HEIDENHAIN

User manual

MillPlus IT
V530

English (en)
06/2007

Contents

These instructions are a summary of User Manual V520 and its additions up to and including software version V530. Each section starts with its own Table of Contents.

- User Manual V520.
- User Manual Additions V530.

> The search function in the PDF version works throughout the manual.

The additions come in the form of a separate document entitled User Manual Additions V530. This can be downloaded from the MillPlus IT website.

MillPlus IT website

For further information, see:

www.millplus.com or www.millplus.de

625 489-20

HEIDENHAIN

MillPlus IT

NC Software
V5.20

User manual

V1.0
12/2003

Table of contents

1. Introduction

These instructions are intended to assist you in operating and programming the controller.

Please read the information in this manual carefully before you start your new machine. It contains important information on machine operation and safety to enable you to use your machine safely and effectively.

The following advice is important for your safety:

This manual is essential for safe use of the machine.
Please ensure that it is in the vicinity of the machine.

The machine should not be operated, even for a short period, by anyone who has not received suitable training, either in the company, at an Institute of Further Education or in one of the Training Centres.

Please read the general safety regulations issued by your professional association.
If they are not on display in the company, contact your appointed safety representative.

Observe the instructions for proper use of the machine.

The controller and the machine are coordinated using machine constants. Some of these constants are accessible to the user. Caution!
The meaning and function of the constants must be fully understood before any changes are made to these constants. If in doubt, please consult our service department.

The controller is fitted with a backup battery that safeguards the memory content for up to three years after the machine is switched off (but only if the battery is serviceable).

The user should always save the programs and specific data (e.g. technology data, machine constants, etc.) to a PC or to diskette. This will avoid the data becoming irrecoverably lost if the system or backup battery becomes defective.

We reserve the right to make changes to the design, equipment and accessories in the interest of further development. No liability will be accepted for any errors in the data, illustrations or descriptions.

The MillPlus ***IT*** controller is available as a single and dual processor system. Whenever you see this logo, the description refers to the dual processor system.

1.1 MillPlus IT software and functions

This manual describes functions available in MillPlus IT (VME and LE4xx hardware) for the following or higher software versions:
- V420 (LE4xx) Software number 344 198-xx
- V500 (LE4xx) Single processor system software number 349 643-xx .
- V500 (LE4xx) Dual processor system software number 360 476-xx

- V510 (LE4xx) Single processor system software number 358 643-xx
- V510 (LE4xx) Dual processor system software number 358 644-xx

- V520 (LE4xx) Single processor system software number 367.350-xx
- V520 (LE4xx) Dual processor system software number 367.350-xx

The machine builder adapts the versatile capability of MillPlus IT to the machine in question by means of machine parameters. That is why some functions described in this manual are not available with every version of MillPlus IT.
MillPlus IT functions that are not available on every machine include, for example:
- Turning mode expanded
- Tool measurement with TT120/TT130
- Tool measurement with laser system
- Ethernet interface (TCP/IP)
- Autostart (warm machine startup program)

Please contact the machine builder for individual support for the particular machine being controlled.

1.2 Software Version V520

Note

V520 software runs on single and dual processor systems.

Operation:

File management function moved from menu bar to softkey bar

EASYoperate
Menu option Setup for Axis Diagnosis and machine macros added in manual mode

Operation: Dual processor system
Switching off the controller
Diagnostics/Help expansion

Added G functions:

- **G33** Thread-cutting cycle for turning
- **G106** Calculate kinematics: OFF
- **G108** Calculate kinematics: ON
- **G610** Break monitoring TT130
- **G61** Measure turning tools TT130
- **G615** Laser system L/R measurement of turning tools

Measuring cycles

- **G620** Measure angle
- **G621** Measure position
- **G622** Measure outside corner
- **G623** Measure inside corner
- **G624** Measure outside corner and angle
- **G625** Measure inside corner and angle
- **G626** Measure outside rectangle
- **G627** Measure inside rectangle
- **G628** Measure outside circle
- **G629** Measure inside circle

Measuring cycles in the oblique plane (G7):

- **G631** Measure position of inclined plane
- **G640** Determine kinematic turning centre

Drilling cycles

- **G781** Drilling/centring
- **G782** Deep drilling
- **G783** Deep drilling with additional chip breaking
- **G784** Tapping
- **G785** Reaming
- **G786** Hollow boring
- **G790** Reverse countersinking
- **G794** Tapping (interpolating)

Positioning cycles (specimen)

- **G771** Machining in line
- **G772** Machining in a rectangle
- **G773** Machining in a grid
- **G777** Machining in a circle
- **G779** Machining in position

Special cycles

- **G700** Facing
- **G730** Mill lines

Milling cycles

- **G787** Pocket milling
- **G788** Slot milling
- **G789** Circular pocket milling
- **G797** Pocket finishing
- **G798** Slot finishing
- **G799** Circular pocket finishing

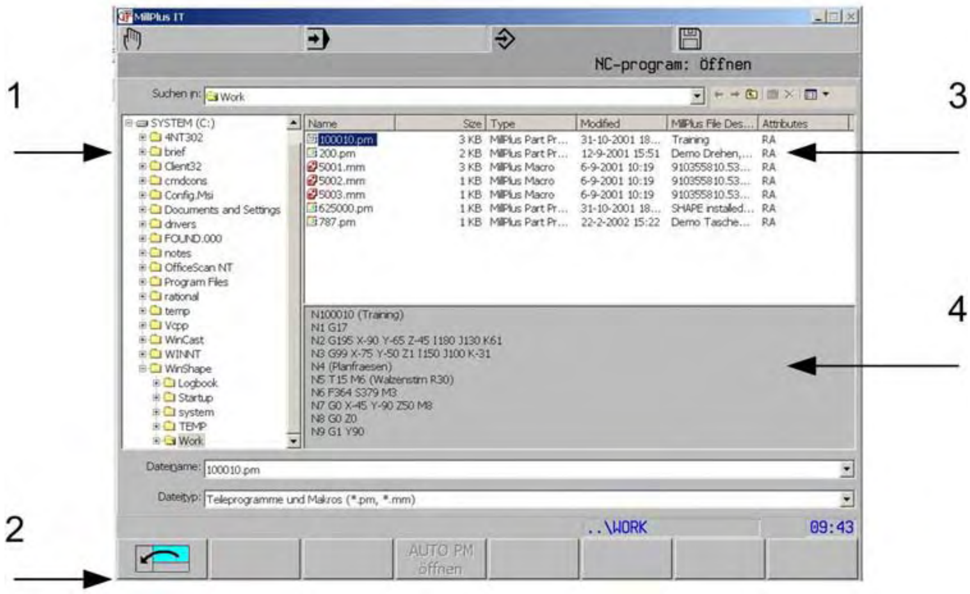
Modified functions:

- **G4** Waiting time in rotations
- **G320** I1=63 up to 65 added
- **G324** I1=29 G106 or G108 added
- **G326** Address D7= added
- **Cycle Design:** Minor additions (INCH)

1.3 Single/dual processor system introduction

Single processor system: SP
Dual processor system: DP
The V500 and its successors can run both on SP/DP systems.
DP-MillPlus ***IT*** has a Windows operating system on the front end.

1.3.1 DP file management



1 List of directories
2 Softkey for window selection
3 Contents of current directory
4 Quick view of current file

Note:

A file can be selected by the left touchpad key. The right touchpad key has the same functions, which can also be called up from the softkeys. Cursor operation and use of double-clicking as with Windows.

1.3.2 Windows Applications

Remarks when installing application software on a dual processor system.

The execution of software applications during MillPlus IT-operation, may lead to fluctuations of the feed during the execution of NC-Programs.

The installed application software:

- May not occupy the windows processor up to its processor limits
- May not be executed with the following Processes Base Priority:
  - above normal
  - high
  - real time

The Processes Base Priority of the application software can be checked via the Windows Task Manager in the column Base Pri(ority) of the Processes window.

To view the Task Manager:

- Click with the right mouse button on a free position of the tool bar
- Click the Task Manager...
- In case the column: Base Pri(ority) is not showed:
- Click in the menu View on Select Columns...
- Mark Base Priority
- Click OK

Programs having one of the above mentioned Base Priorities may not be applied during program execution of the MillPlus IT.

Please pay attention further to the following:

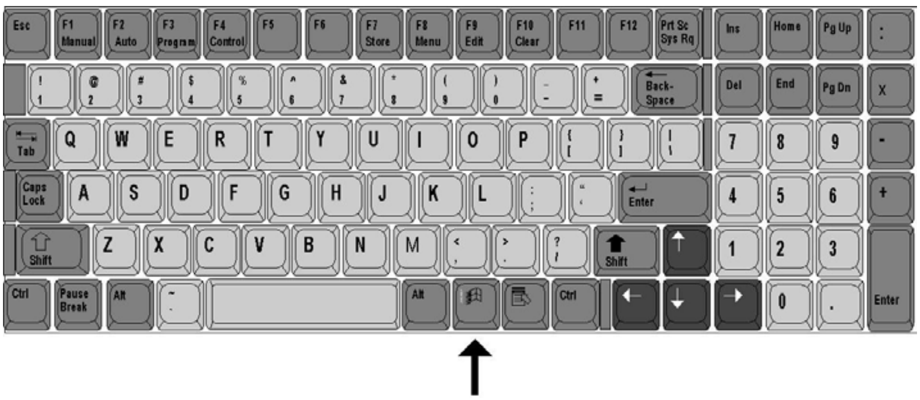
- **HEIDENHAIN cannot offer support at SW-Installation nor can be held responsible for the functions of the Windows Applications.**
- **HEIDENHAIN is not held liable for faulty hard disks, which occur during installation of SW-updates or additional application software.**
- **Costs due to service calls on HEIDENHAIN, required after mentioned program or data changes, will be fully charged.**

1.3.3 Virus protection

Note that the standard installation of Windows and the CNC-software as supplied by HEIDENHAIN, doe not include virus protection programs. The origin of new viruses happens so fast that included virus protection programs will not be up to date. It is therefore the responsibility of the user of the CNC to take care of the installation of adequate virus protection programs.

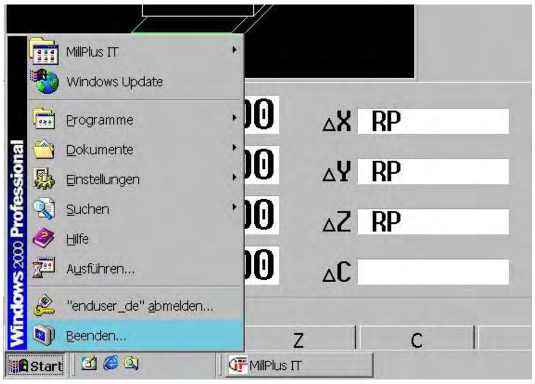
1.3.4 Switching off MillPlus *IT* on a dual processor system

First press the **emergency stop** to ensure the motors are switched off!



Press the Windows key on your MillPlus ***IT*** PC keyboard.

Windows will then open the "START“ menu.



Select "Exit...“

Windows prompts for confirmation.

If you have not used an "Emergency Stop“, the following message is displayed

Note
If you are just switching on the controller, you do not need to wait until the controller software has started up. As soon as the startup procedure is running, you can press Ctrl/Esc, which also takes you to the next procedure.

2. Safety

Meaning of symbols and notices:

Signifies immediate danger to persons.

"LIVE COMPONENTS" Access through authorized personnel only! Indicates danger due to live components, which must be isolated prior to commencing repairs.

ACHTUNG !

Applies to operating or plant procedures which have to be followed precisely to avoid danger or injury to persons and damage to the installation.

WARNUNG !

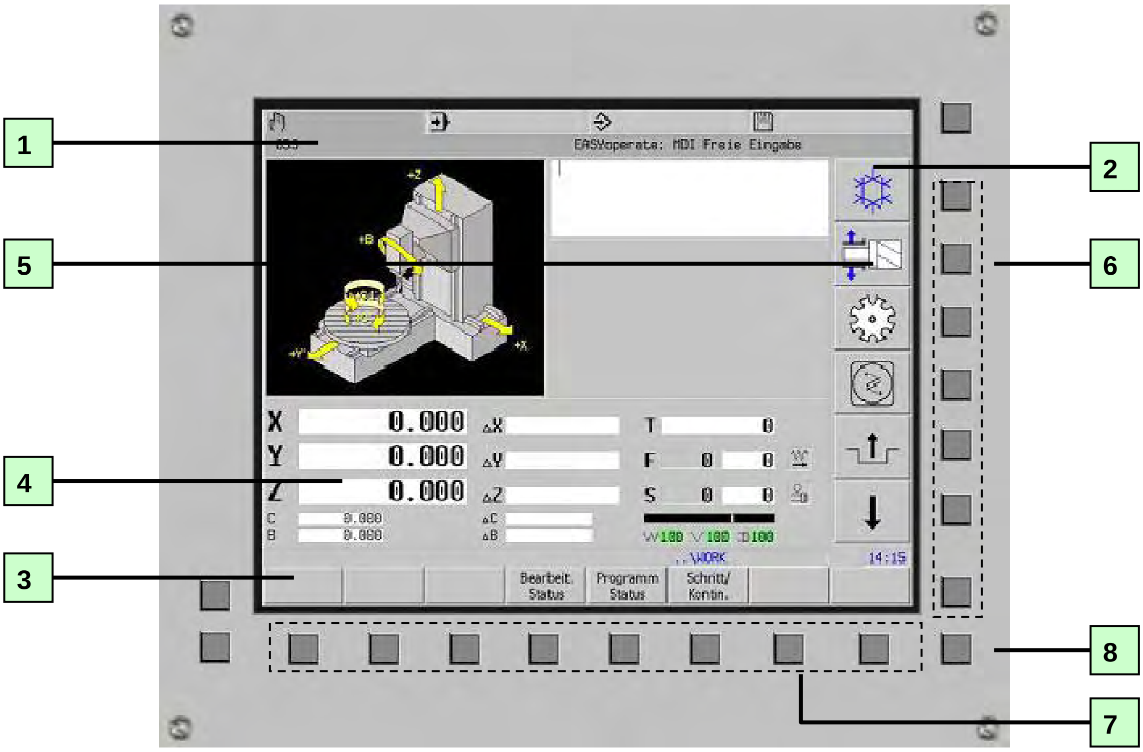
Applies to situations which may pose a danger to persons.

For special technical features which the user must note.

General safety and accident prevention regulations must be heeded as well as the advice given in the operating instructions.

3. Keyboard and screen layout

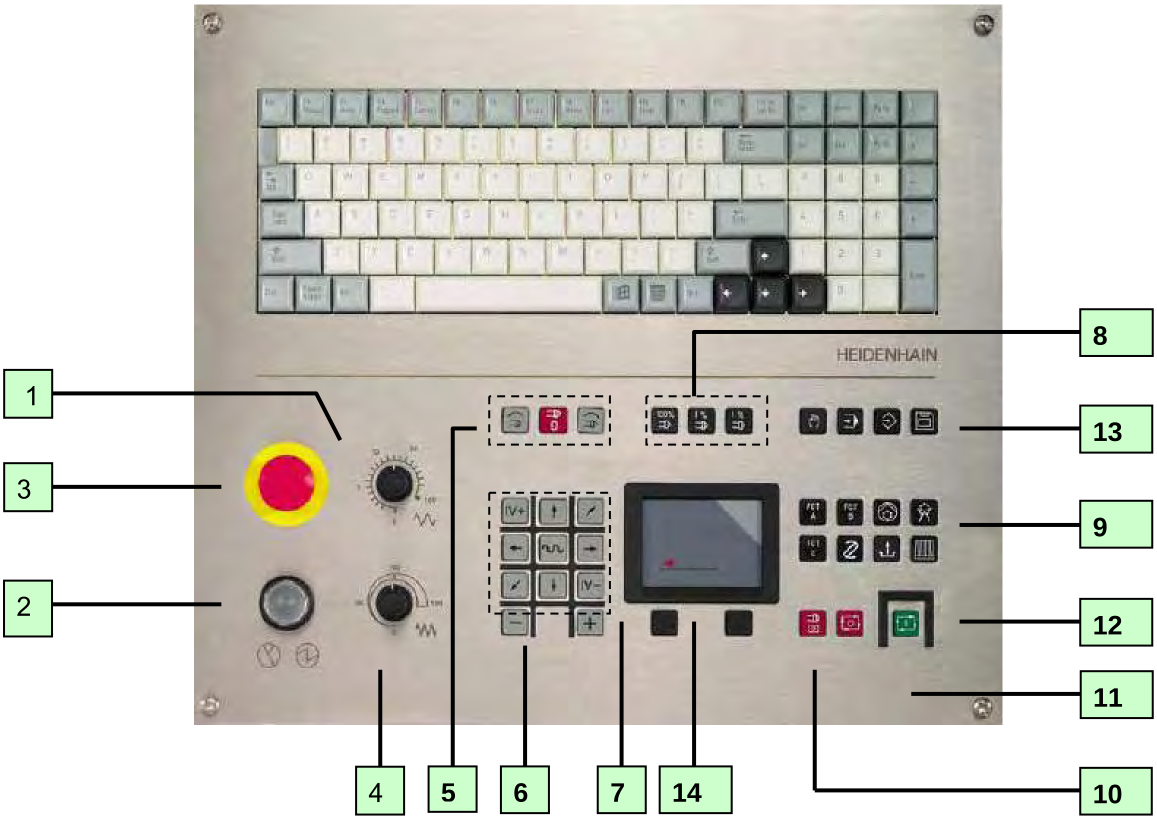
3.1 Screen display



1 Process level
2 Machine function softkeys
3 Softkeys
4 Machine information

5 VGA monitor
6 Machine function softkeys
7 Softkeys
8 Information key

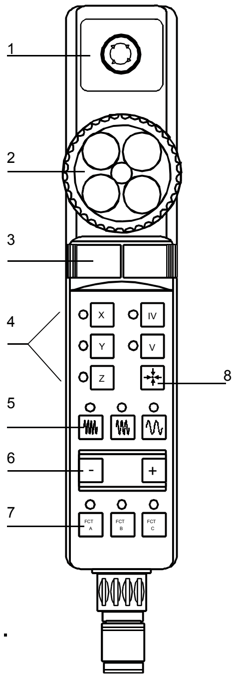
3.2 Control panel



1 Rapid traverse rate control.
2 Machine ON
3 EMERGENCY STOP
4 Feed rate control
5 Spindle On Clockwise Rotation, Stop, On Anticlockwise Rotation
6 Axial movement keys for other axes
7 Axial movement keys and rapid traverse
8 Spindle speed control
9 Machine function keys; the function of the keys is determined by the machine tool builder.
Please refer to your machine tool manual
10 Feed and spindle STOP
11 Feed STOP
12 START
13 Main modes of operation
14 Touchpad

Note The keys (F11, F12, Prt Sc Sys Rq, Pause Break) must not be activated, because no function has been assigned to them.

3.3 Hand wheel HR410 (HCU)



1. Emergency stop button
2. Hand wheel
3. Safety keys
4. Axis selection keys
5. Keys for setting the feed (slow, medium, fast); feed rates are defined by the machine manufacturer
6. Direction into which the CNC moves the selected axis
7. Machine function keys (defined by the machine manufacturer)
8. Key for taking over the actual position
   - setting the actual value
   - tool measurement
   - Program Editor

The red LED displays indicate the axis and feed you selected.

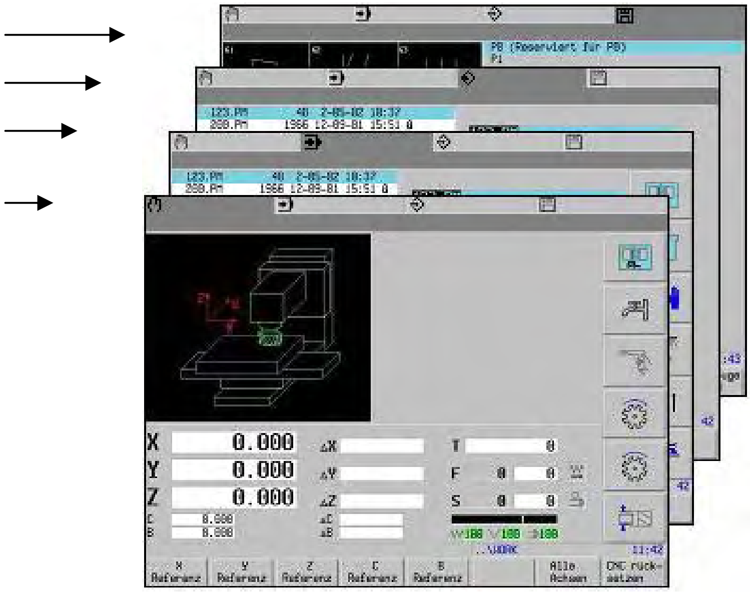
3.3.1 Selecting/deselecting the hand wheel

The hand wheel is selected by pressing the left safety key. In the right top of the display appears HCU. For deselection let go the left safety key.

Note

Operation is defined by the machine manufacturer. Refer to your machine manual.

3.4 The 4 process concept



1. Manual: Manual operation
2. Automatic: Execute program
3. Program: Create program
4. Check: Management of tables, files and communication

Basic principle:
All the 4 process levels function in parallel, with some restrictions.

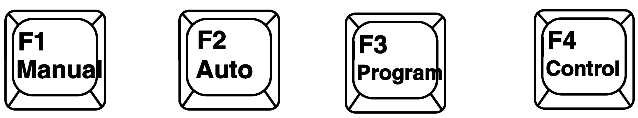
Example of parallel functions:
In the automatic process, a program can be executing while a new program is created simultaneously in the program process.

Example of a restriction:
If the manual process is active, a program cannot be executed in the automatic process.

3.5 Exiting a function

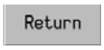
To exit a function or a mode, press "Menu" again,

or



To exit a function, select a different process; when you select the same process level again, the process level will be started at the points at which you left it. To finally exit a function, select a new function within the same process level.

3.6 Return to previous softkey level



Press to return to previous softkey group (if one exists).

3.7 Superimposition of softkey groups

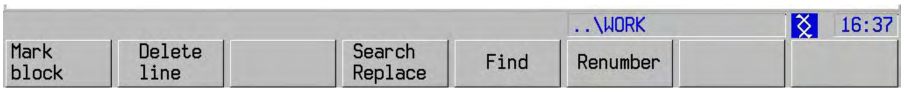
In addition to the current softkey group, other softkey groups may be active in the same mode.

3.7.1 User softkey group

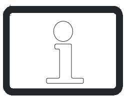
For editing DIN/ISO programs, press a mode key twice:

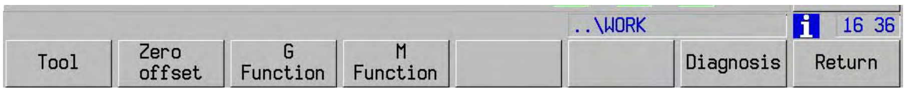
3.7.2 Edit softkey group

Softkey group for editing

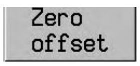


3.7.3 Info softkey group

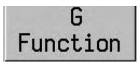




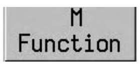
Indication of the tools entered in the tool table.



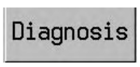
Indication of the zero offset table.



Indication of the list of G functions.



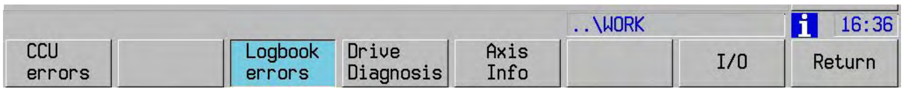
Indication of the list of M functions.



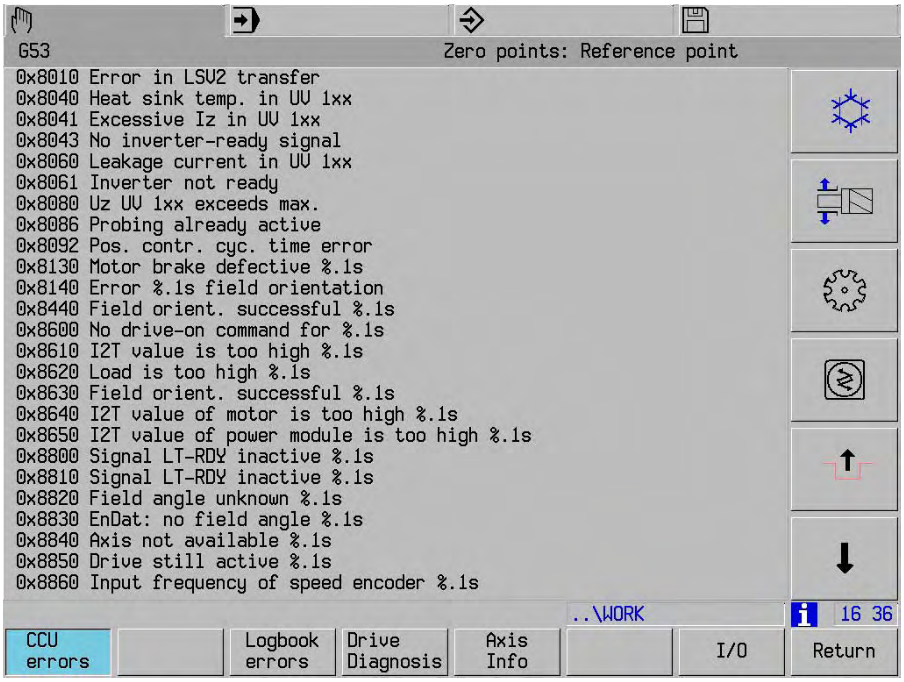
Support for axes and I/O diagnostics

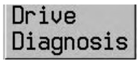
3.7.3.1 Diagnostics

Diagnosis — Support for axes diagnostics and I/O

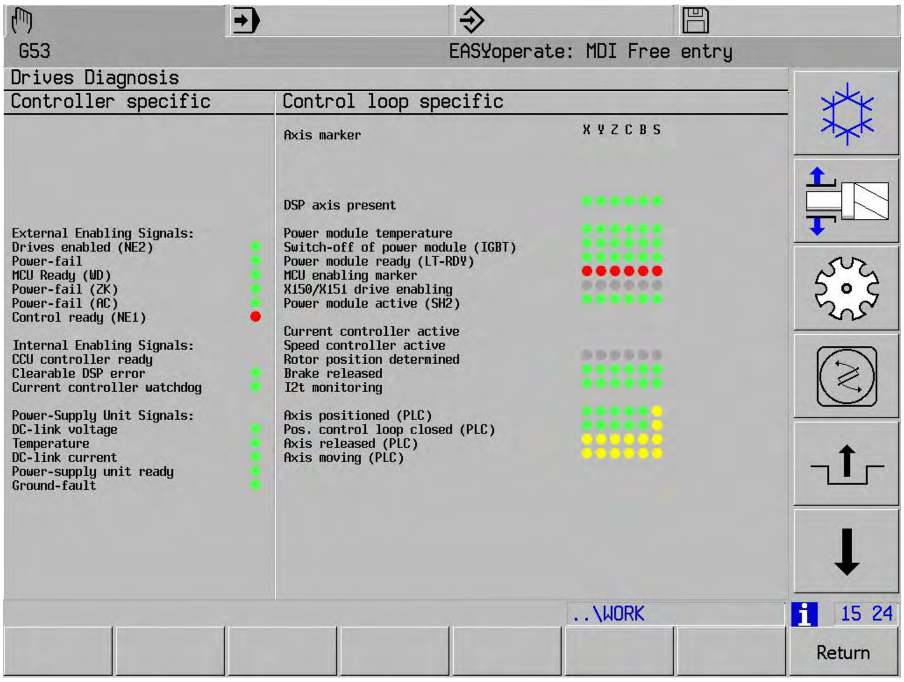


CCU errors — Extended CCU-and axes messages





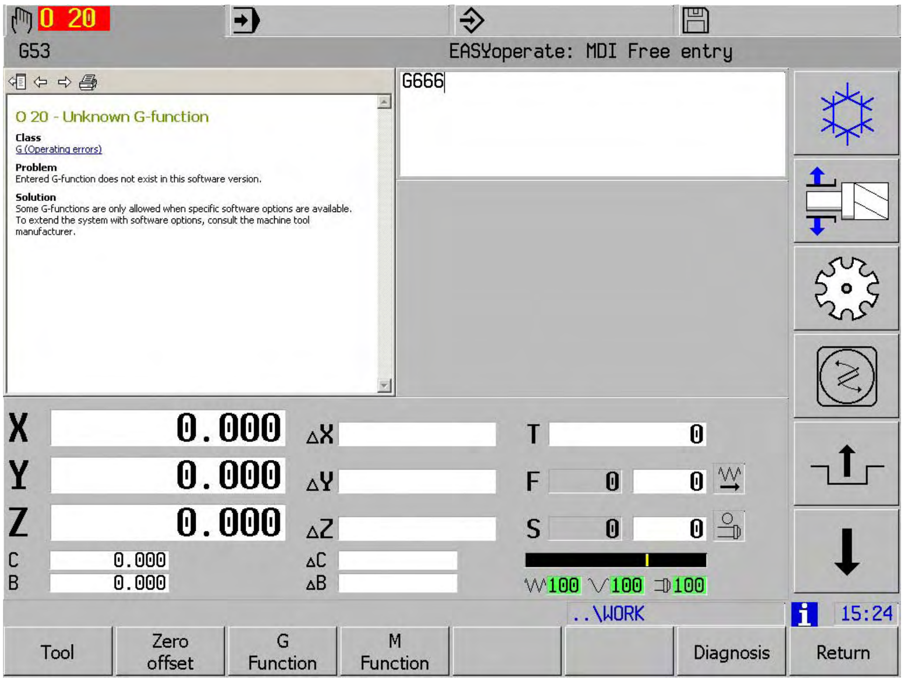
Drive diagnostics



3.7.3.2 Error messages (DP)

If an error is generated, an error code will be displayed. By pressing the Info-Button, an explorer window becomes available with different support levels:

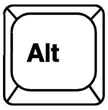
1 Error code description: the error code is explained with a short error code description



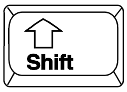
2 Extended error code description: error code description, a more extended error description appears including problem solution

3 Directory: at a mouse click on the explorer icon, the explorer window shows a directory containing files with different descriptions in PDF-format. These files can be selected and opened.

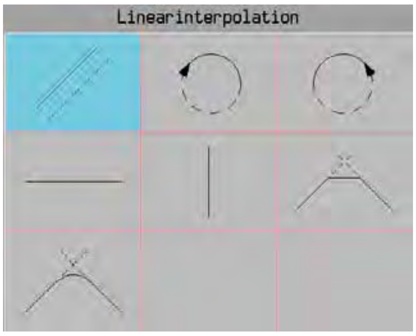
3.8 Switching between upper and lower case characters

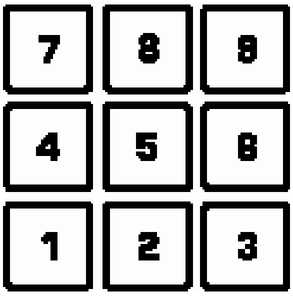


with



3.9 Making selections in the Easy Operate, ICP and IPP menus





1. Use the cursor keys to move left, right, up and down through the menu.
   To choose a menu item, press ENTER

2. or press one of the number keys 1-9. The ENTER key is not used in this case.

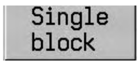
3.10 Quick mode selection

Two-digit mode number. (first digit: menu position, second digit: mode position)

Example: Select clock

3.11 Softkey Status

The status indicator of the softkeys shows the actual condition. For instance:



Softkey grey (Softkey not active)

Softkey blue (Softkey active)

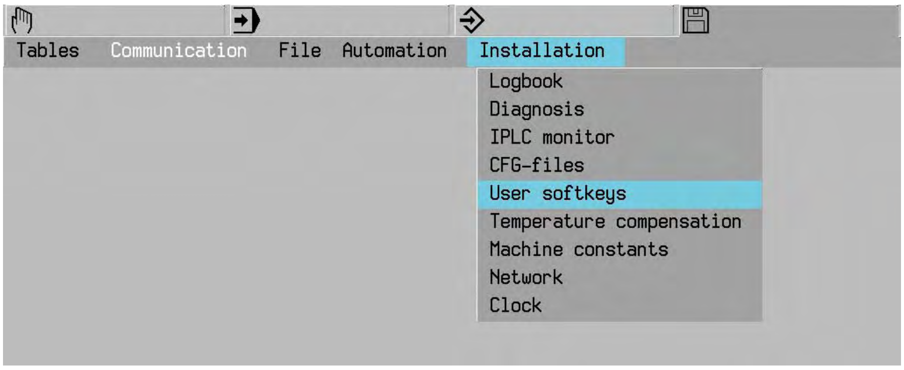
3.12 User softkeys

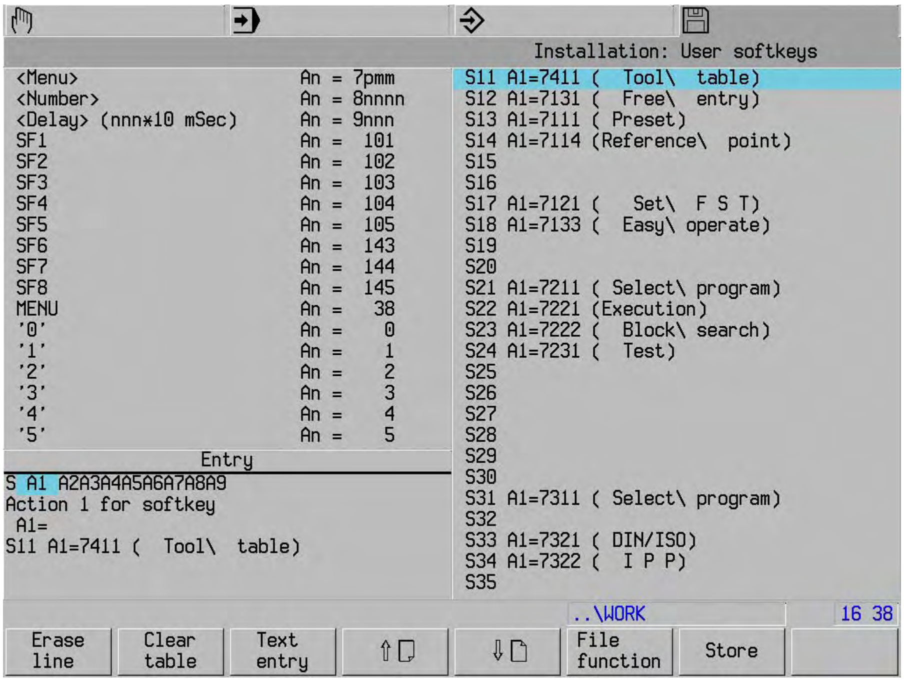
The user softkeys are used to initiate the most common functions quickly.

The user softkeys appear when the current process mode key is pressed a second time.

Pressing the key again causes the user softkeys to disappear. The previous softkey level becomes active again.

3.12.1 Defining the user softkeys





Search auxiliary window

Table with key

<table>
<tr><th>Key command</th><th>Action
value
t</th><th>Key command</th><th>Action
value t</th></tr>
<tr><td>direct menu command</td><td>7000-7499</td><td><-- (Cursor left)</td><td>49</td></tr>
<tr><td>number command</td><td>80000-89999</td><td>^ (Cursor Up)</td><td>51</td></tr>
<tr><td>Delay command</td><td>9000-9999</td><td>v (Cursor Down)</td><td>52</td></tr>
<tr><td>hor. softkey 1</td><td>101</td><td>--> (Cursor right)</td><td>50</td></tr>
<tr><td>hor. softkey 2</td><td>102</td><td>clear</td><td>15</td></tr>
<tr><td>hor. softkey 3</td><td>103</td><td>escape</td><td>166</td></tr>
<tr><td>hor. softkey 4</td><td>104</td><td>back space</td><td>154</td></tr>
<tr><td>hor. softkey 5</td><td>105</td><td>key pad "."</td><td>39</td></tr>
<tr><td>hor. softkey 6</td><td>143</td><td>key pad "="</td><td>40</td></tr>
<tr><td>hor. softkey 7</td><td>144</td><td>key pad "+"</td><td>45</td></tr>
<tr><td>hor. softkey 8</td><td>145</td><td>key pad "-"</td><td>46</td></tr>
<tr><td>menu</td><td>38</td><td>key pad "/"</td><td>47</td></tr>
<tr><td>number "0"</td><td>0</td><td>key pad "*"</td><td>48</td></tr>
<tr><td>number "1"</td><td>1</td><td>help</td><td>153</td></tr>
<tr><td>number "2"</td><td>2</td><td>store/select</td><td>53</td></tr>
<tr><td>number "3"</td><td>3</td><td>tab</td><td>171</td></tr>
<tr><td>number "4"</td><td>4</td><td>ASCII "("</td><td>1044</td></tr>
<tr><td>number "5"</td><td>5</td><td>ASCII ")"</td><td>1045</td></tr>
<tr><td>number "6"</td><td>6</td><td>ASCII "*"</td><td>1046</td></tr>
<tr><td>number "7"</td><td>7</td><td>ASCII "+"</td><td>1047</td></tr>
<tr><td>number "8"</td><td>8</td><td>ASCII ","</td><td>1048</td></tr>
<tr><td>number "9"</td><td>9</td><td>ASCII "-"</td><td>1049</td></tr>
<tr><td>process manual</td><td>139</td><td>ASCII "."</td><td>1050</td></tr>
<tr><td>process automatic</td><td>162</td><td>ASCII "/"</td><td>1051</td></tr>
<tr><td>process program</td><td>140</td><td rowspan="2">ASCII "0"
|
ASCII "9"</td><td rowspan="2">1052
|
1061</td></tr>
<tr><td>process control</td><td>141</td></tr>
<tr><td>store</td><td>53</td><td rowspan="2">ASCII "A"
|
ASCII "Z"</td><td rowspan="2">1068
|
1094</td></tr>
<tr><td>enter</td><td>54</td></tr>
<tr><td>insert</td><td>168</td><td rowspan="2">ASCII "a"
|
ASCII "z"</td><td rowspan="2">1101
|
1127</td></tr>
<tr><td>home</td><td>176</td></tr>
<tr><td>page Up</td><td>170</td><td></td><td></td></tr>
<tr><td>delete</td><td>163</td><td></td><td></td></tr>
<tr><td>end</td><td>165</td><td></td><td></td></tr>
<tr><td>page Down</td><td>169</td><td></td><td></td></tr>
</table>

Process level Manual:	S11 to S18	(Softkey 1-8)
Process level Automatic:	S21 to S28	(Softkey 1-8)
Process level Program:	S31 to S38	(Softkey 1-8)
Process level Monitor:	S41 to S48	(Softkey 1-8)

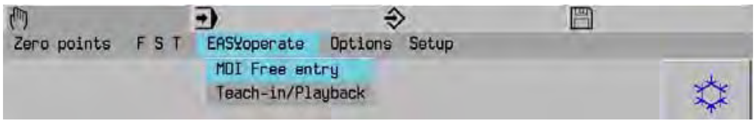
Entering softkey text:

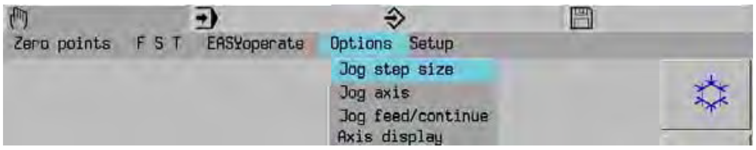
- The softkey text should be in brackets.
- 2 lines, not more than 9 characters per line.
- Character "\" defines the line break.

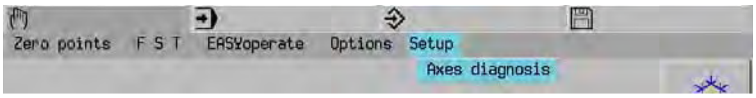
Examples

SF1:	S31 A1=38 A2=1 A3=1	Select file/program
SF3:	S33 A1=38 A2=2 A3=1	DIN/ISO input

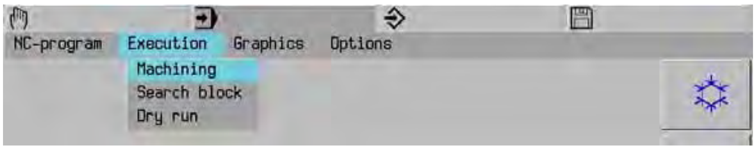
3.13 Process level Manual

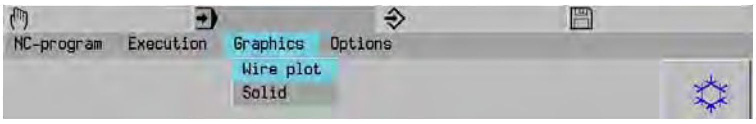




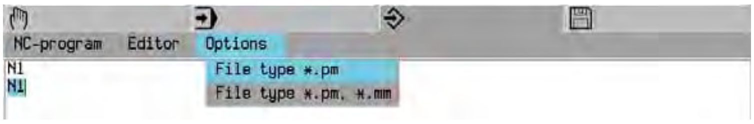


3.14 Process level Automatic

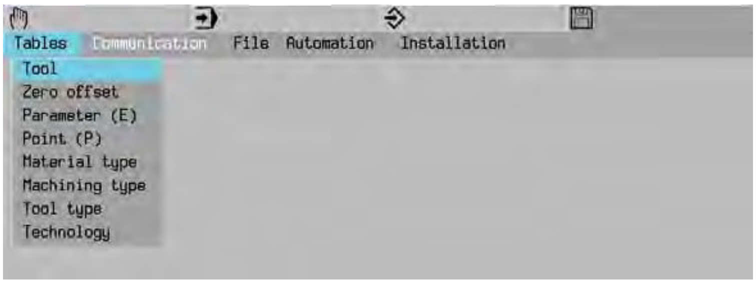


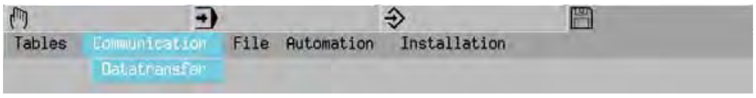


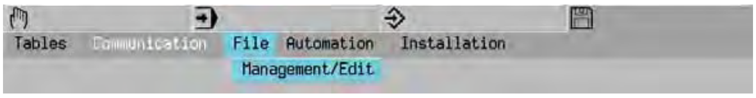
3.15 Process level Program

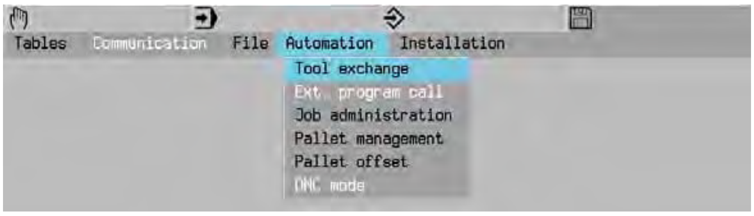


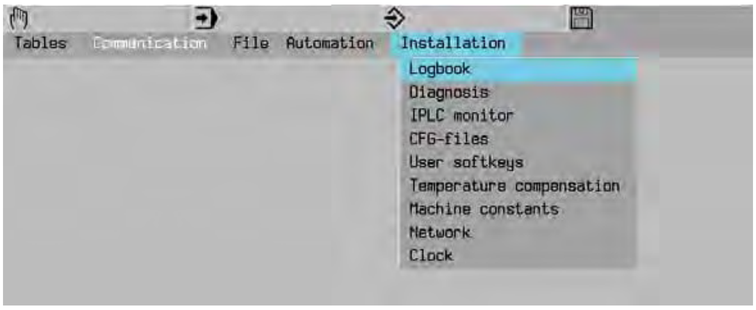
3.16 Process level Monitor

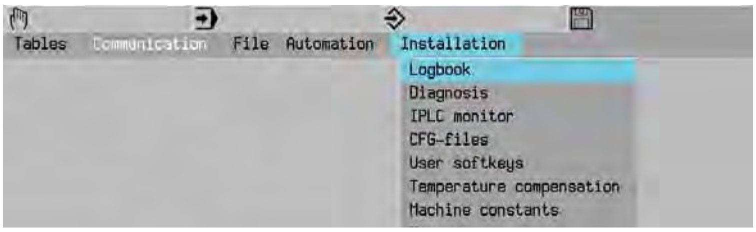






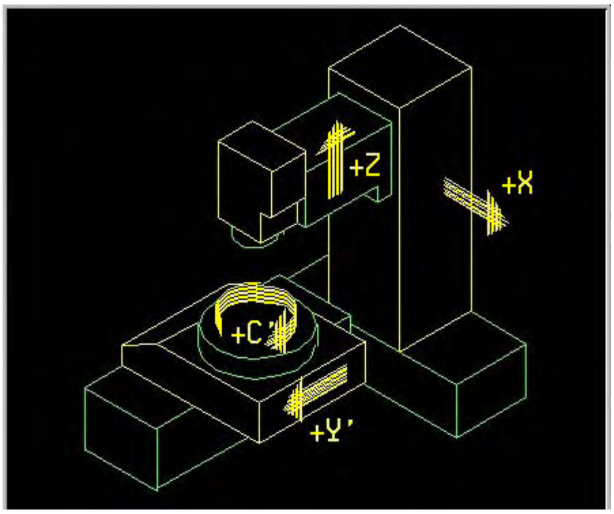




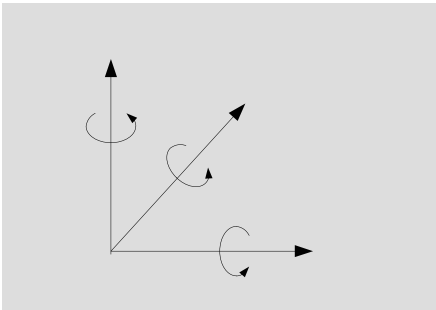


4. Workpiece coordinates

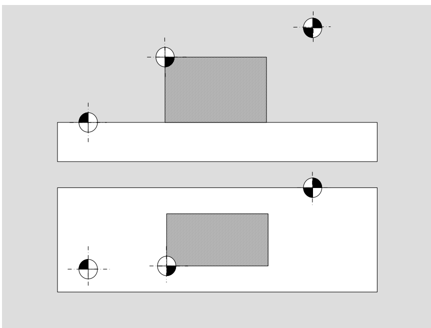
4.1 Coordinate system and direction of movement



4.2 Axes

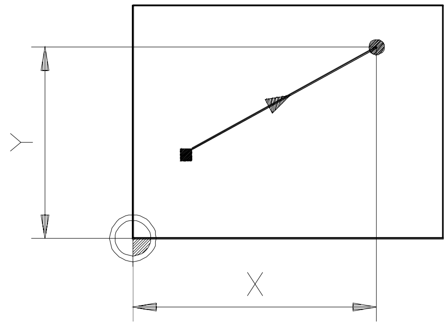


4.3 Zero points

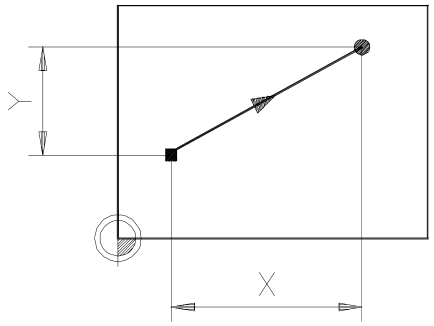


R Referenzpunkt
M Maschinennullpunkt
W Werkstücknullpunkt

Cartesian coordinates



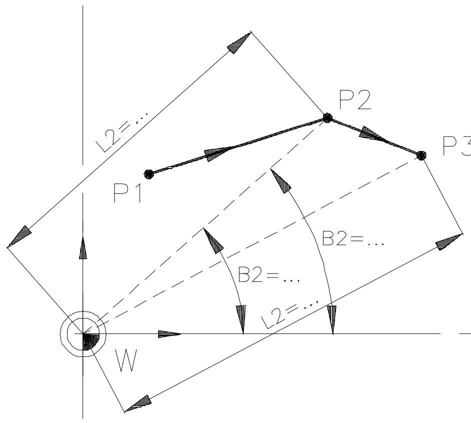
Absolute coordinates (G90)



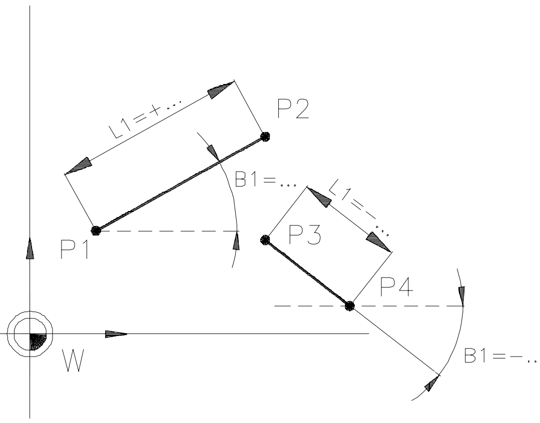
Incremental coordinates (G91)

Wordwise absolute/incremental programming (X90,X91,Y90...) does not depend on the modally valid G90/G91 system of measurement.

4.4 Polar coordinates



Absolute coordinates (G90)



Incremental coordinates (G91)

Programming in polar coordinates is not affected by wordwise absolute/incremental programming.

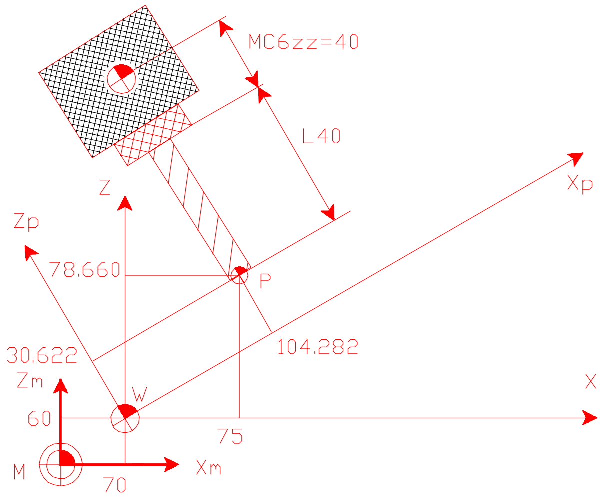
Note

If a pole point has been programmed (see G9), program blocks that use polar programming (angle and length) no longer refer to the zero point, but to the most recently programmed pole point.

4.4.1 Assignment of polar coordinates

Polar coordinates	Angle reference axis	Movement B1=+
X Y G17	+X	+X nach +Y
Z X G18	+Z	+Z nach +X
Y Z G19	+Y	+Y nach +Z

4.5 FSP coordinates



The position display on the screen can change between the position in the G7 plane (Xp,Zp) or in machine coordinates (X,Z).
Both are based on the active null point G52 + G54 + G92/G93.

5. Start machine / reference point

5.1 Start machine (example)

Main switch ON

Power supplied to controller and measuring system.

Danger! High voltage!
Do not touch any exposed components in the switchgear cubicle as they may be live.

Before starting or operating the machine, ensure that noone is likely to be endangered as a result.

ACHTUNG !

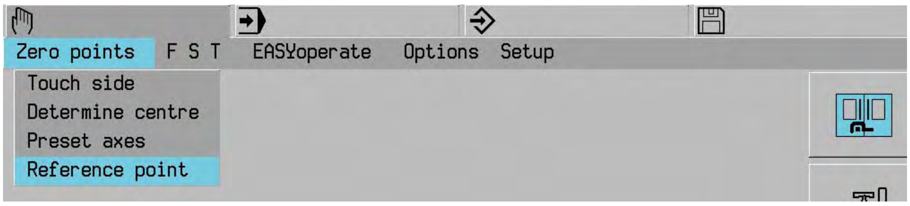
Ensure that only authorised personnel operate the machine!

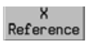
Release the EMERGENCY STOP switch.

Machine ON (keep key depressed) and press CLEAR.

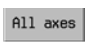
Starting and closing the software on a double processor system, see chapter 3

5.2 Approach reference points

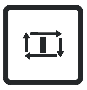




Selection of



one or more axes

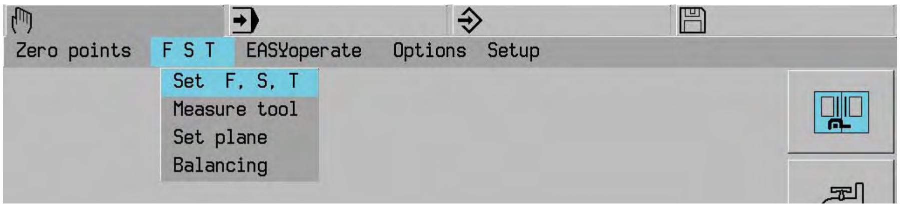

Approach reference point (RPF)

Note

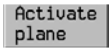
Beware of collision!
The software limit switches are not active prior to "Approach reference points", and the axis slides are able to run up to the mechanical end stop.
Before "Approach reference points", the machine operator should ensure that no collision with the machine will occur when approaching the reference points

5.3 Select level

The active plane can be selected by using the softkey. The functions G17, G18 or G19 are decisive in the machining program and the softkey setting is overwritten.



Selection level

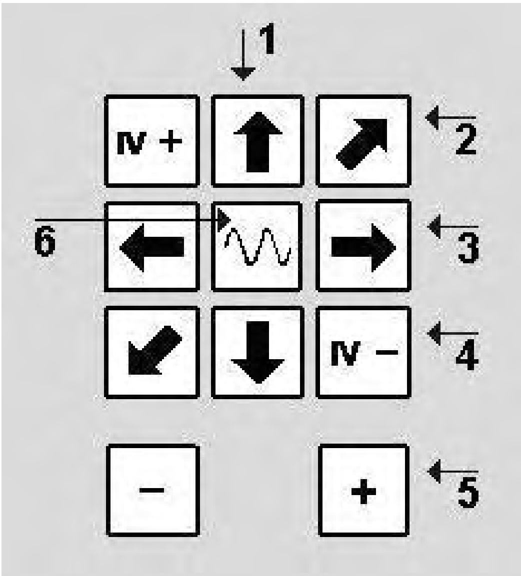


6. Manual operation

The machine axes can be moved continuously and manually by adjustable movement steps. The speed of movement can be regulated using the feed override. It is also possible to move two axes simultaneously. The work spindle may also be moved manually. Other axes, e.g. the fifth axis or spindle, must first be selected.

6.1 Move axes

The axes are moved using the axis movement keys.



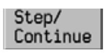
1. Z-axis	2 Y-axis
3 X-axis	4 Axis 4
5 Axis 5	6 Rapid traverse

Note

Select axis 4 with mc153.
Select axis 5 with mc154.

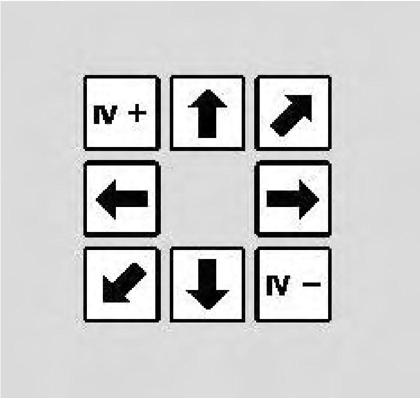
6.1.1 Step movement, continuous movement

It is determined whether the machine axis moves stepwise or continuously when the axis movement key is depressed.

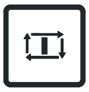


6.1.2 Continuous movement

Kontinuierlich verfahren mit Achsenbewegungstaste und Start. Die Achse verfährt bis sie angehalten wird.

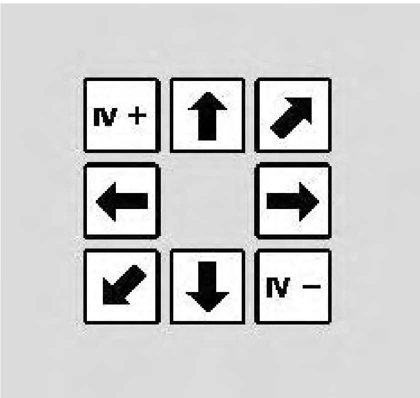


Press at the same time as

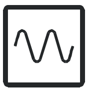


-Feed from MC
-A maximum of 2 axes can be moved at the same time.
-Stop using 'Feed STOP' or 'Feed and Spindle STOP' keys

6.1.3 Rapid traverse motion

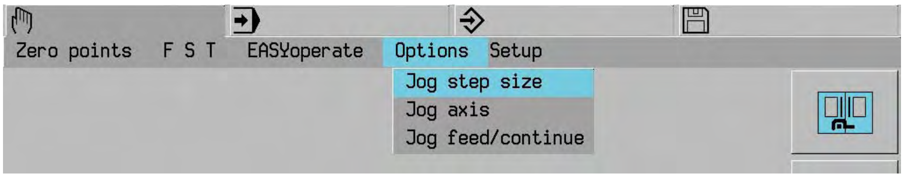


Press at the same time as

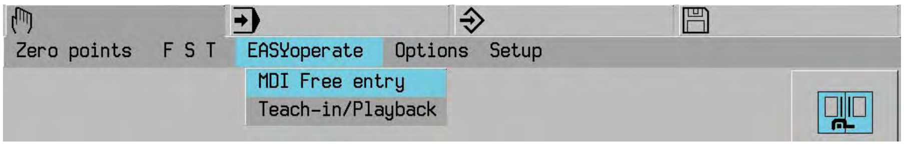


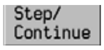
6.1.4 Free step size

The free increment allows you to set the appropriate increment for your machine.

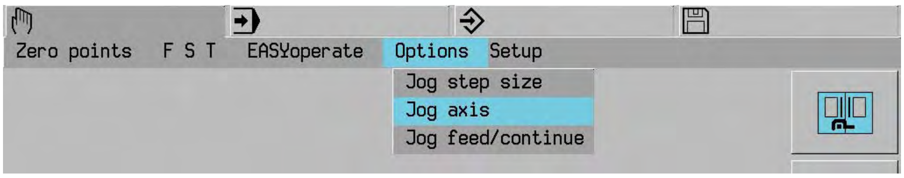


Use free step size:





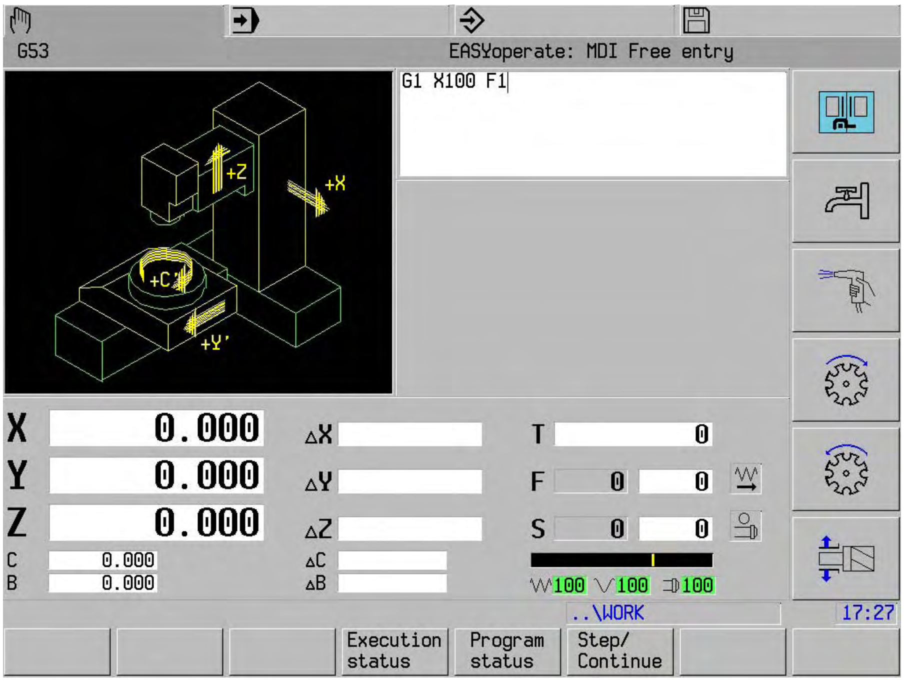
6.1.5 Move spindle and other axes



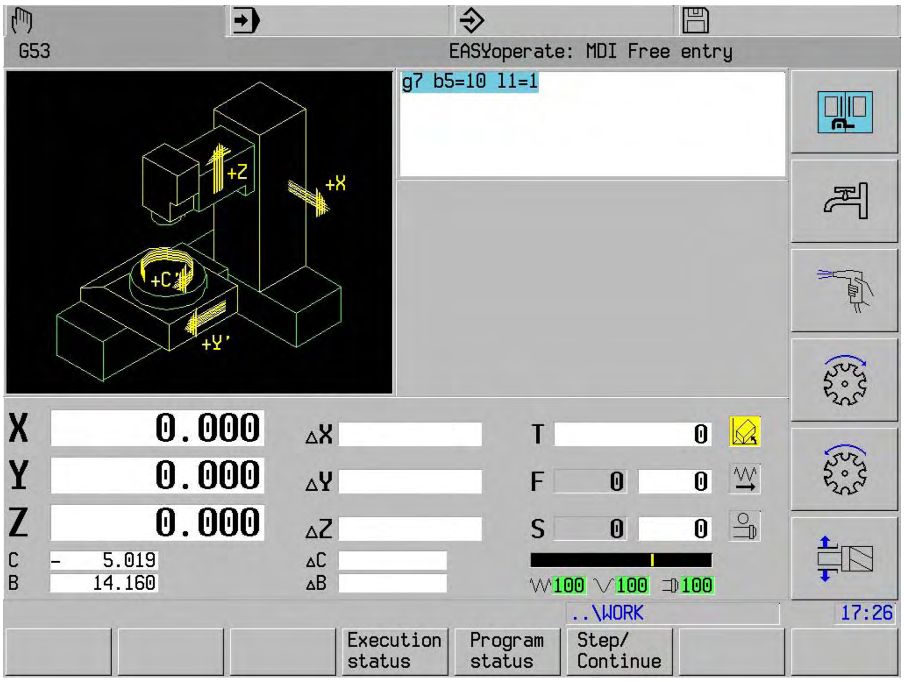
6.2 Procedure in FSP

It is possible to proceed on the FSP level or in the machine axes after enabling the "Free process level"

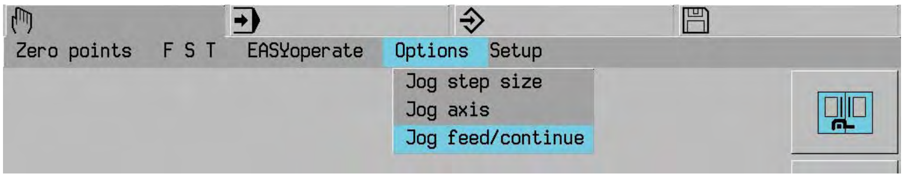
Procedure on the free process level.

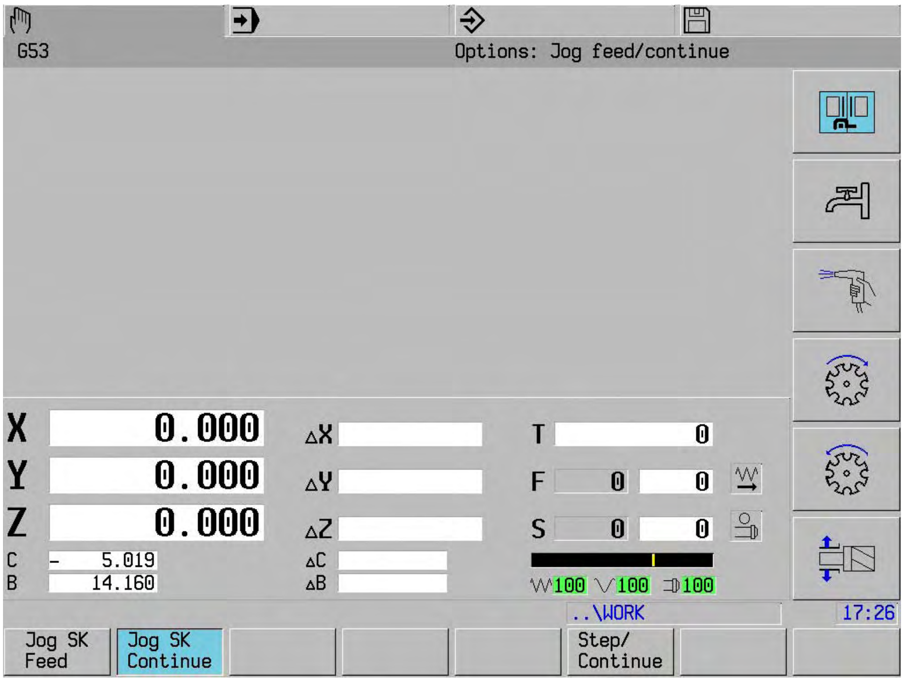


Procedure in the machine axes.



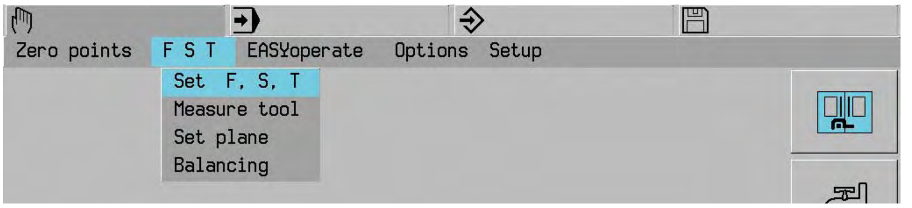
6.3 Switch over rate of advance/continue procedure

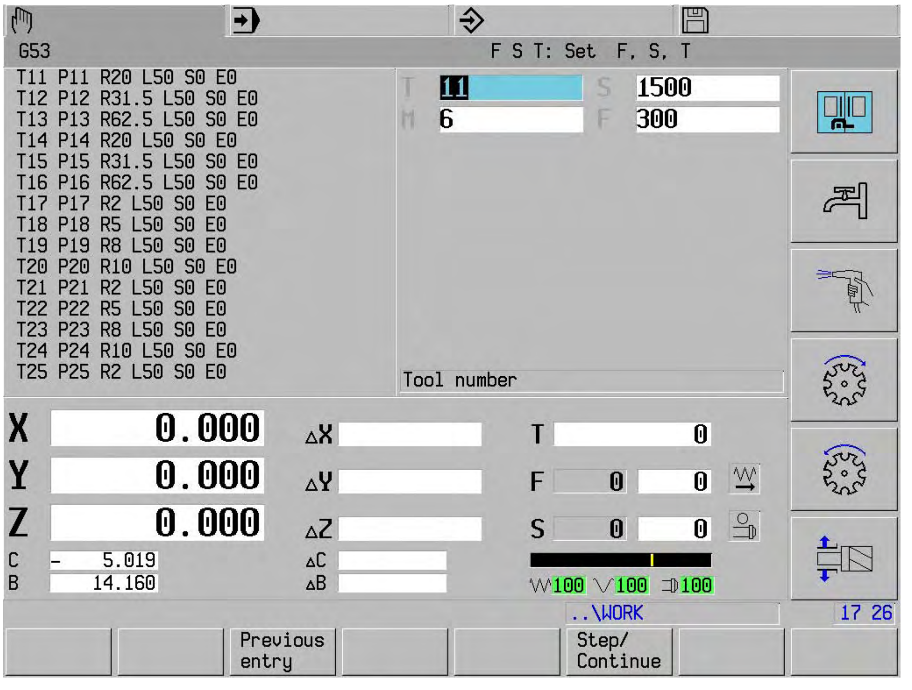


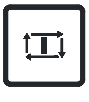


6.4 F, S, T input

Entry of tool number, spindle speed, feed and M-function.







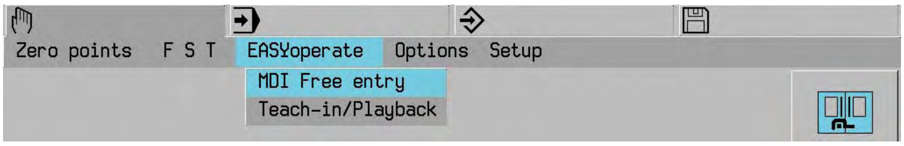
Activate tool change

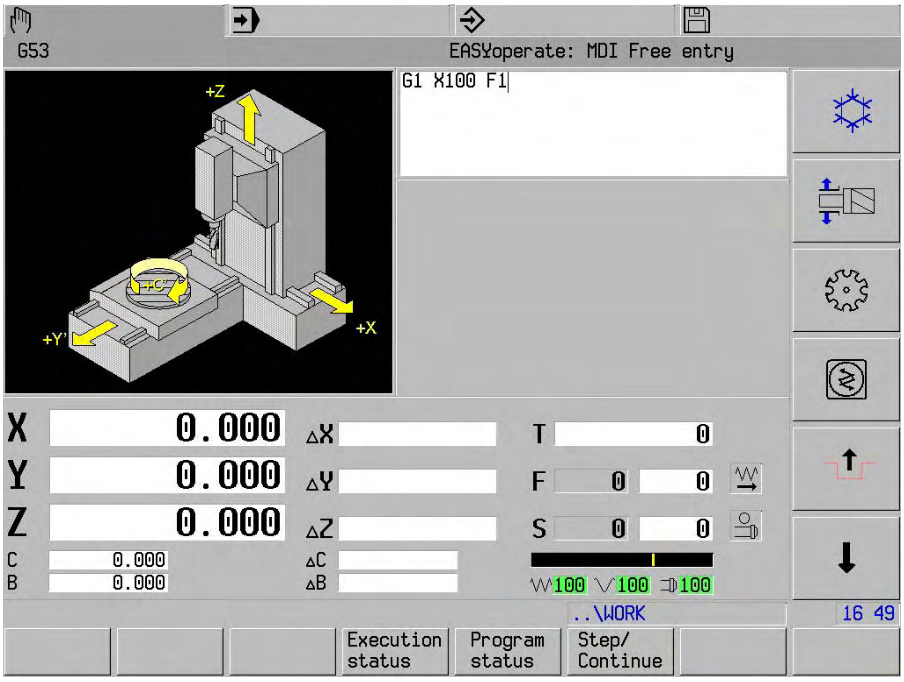
Start spindle (M3 or M4)

7. Free entry (MDI)

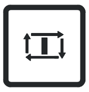
7.1 Free entry

Entry of an instruction in the command line followed by execution.





Enter address and address value from keyboard.



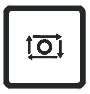
Execute program block.

When execution of the block has been completed, the Free Entry mode remains active.

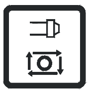
Note

When a free entry is started, this entry is stored in the MDI buffer.
Previously started entries can be reached with cursor ⇩ or ⇧.
The MDI buffer has a maximum of 15 entries. Further new entries will push the oldest entries out of the buffer.
The last MDI buffer place is always empty.
Please refer to the chapter Easy Operate.

7.2 Cancel block (MDI)



or

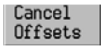


Interrupt program block run

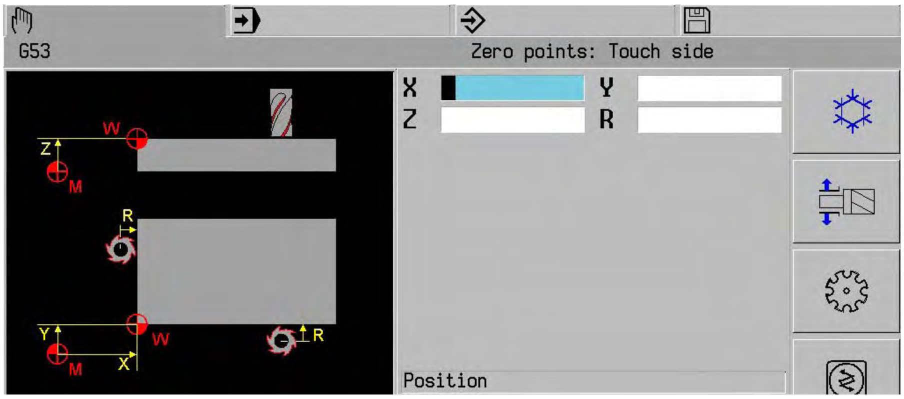
The current block is interrupted.

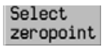
8. Set axis value

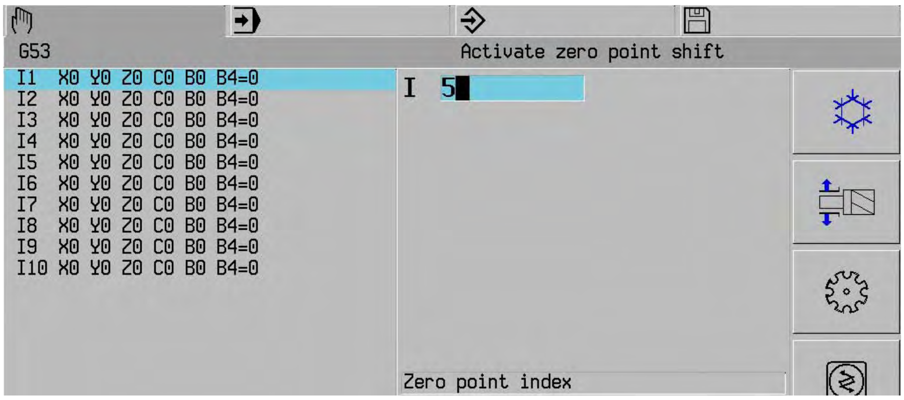
With "Touch side", "Determine centre" and "preset axes" it is possible, after selection of softkey "Select zeropoint", to undo the current zero offset.



8.1 Determine side







Input W activate zero offset

Activate ZPS no.

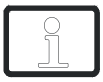
Approach side

Enter offset value (X, Y, Z, R)

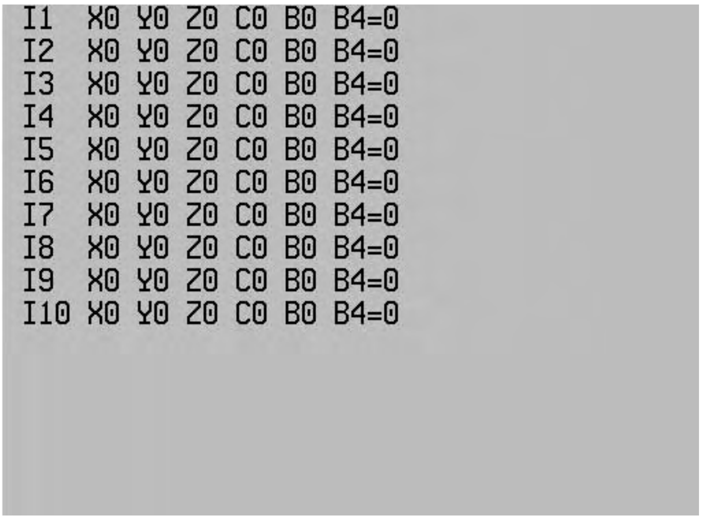
Press the softkey to indicate from which direction the side was approached. The zero offset for the selected axis and direction is calculated and stored in the zero offset memory. The offset value in the current axis screen is also updated.

to

Display of zero offset memory.



Zero offset

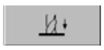


```
I1  X0 Y0 Z0 C0 B0 B4=0
I2  X0 Y0 Z0 C0 B0 B4=0
I3  X0 Y0 Z0 C0 B0 B4=0
I4  X0 Y0 Z0 C0 B0 B4=0
I5  X0 Y0 Z0 C0 B0 B4=0
I6  X0 Y0 Z0 C0 B0 B4=0
I7  X0 Y0 Z0 C0 B0 B4=0
I8  X0 Y0 Z0 C0 B0 B4=0
I9  X0 Y0 Z0 C0 B0 B4=0
I10 X0 Y0 Z0 C0 B0 B4=0
```

8.2 Determine centre

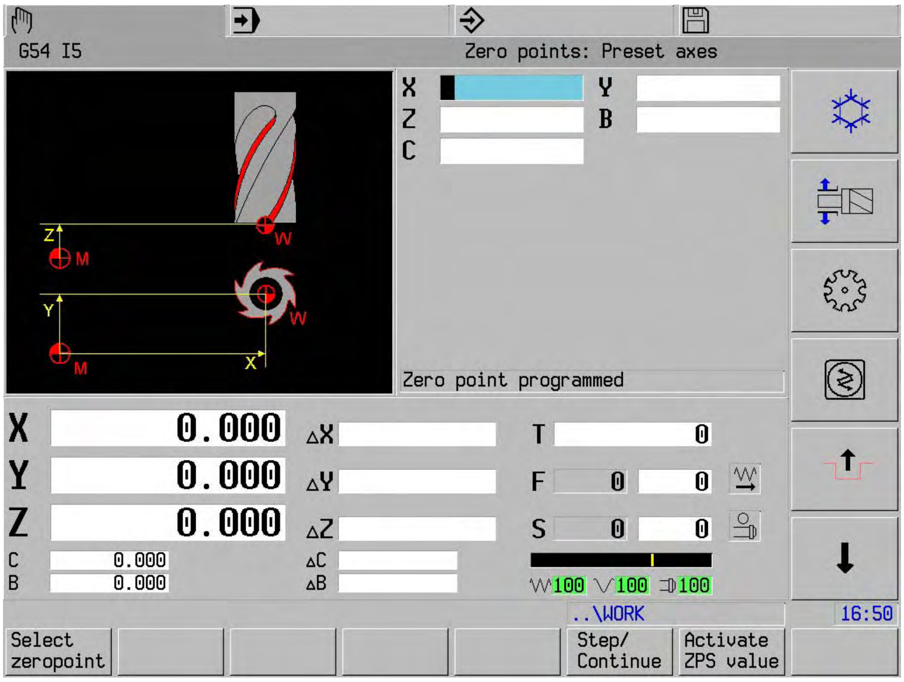
Procedure: as for Determine side.

Activate ZPS value — Activate values in main plane

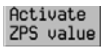
 Activate values in tool axis

8.3 Set actual value

To machine a workpiece, the machine zero points must be synchronised with each other. The workpiece zero point is determined by the machine operator and passed to the controller via the zero offset.



- Select zero point
- Approach position using axis movement keys
- Enter the actual axis values



Update the axis display with the present axis values and add the zero point to the zero offset table.

8.4 Measure side

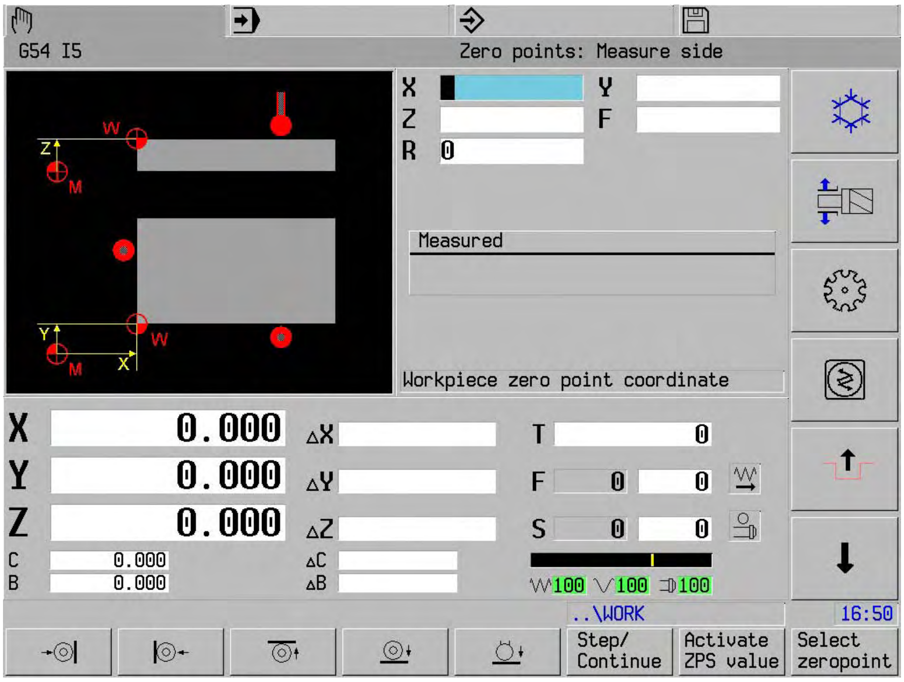
Introduction

With the function "measure side" the workpiece zero offset coordinates can be determined on an arbitrary position of a workpiece clamped on the machining table. This zero offset coordinates are measured by moving a measuring probe against the workpiece sides. The measured positions can be stored in the zero offset table as zero offset coordinates. The measurement is carried out with a 3D-measuring probe, which is mounted in the spindle. The function "measure side" is operable in the normal as well as in a tilted machining plane (G7).

Measuring conditions:

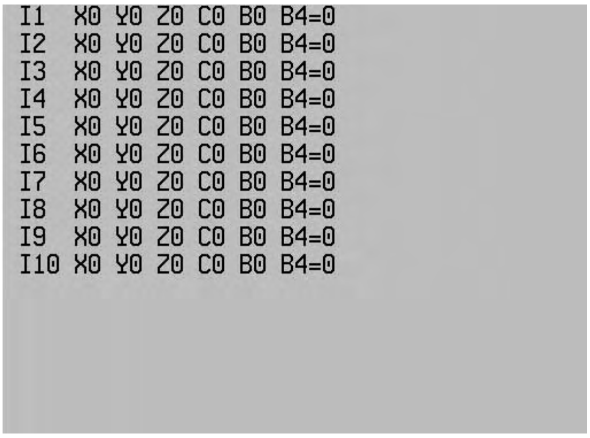
- The axes must be referenced previously
- The measurements are executed in the active coordinate system
- The workpiece sides to be measured are to be aligned and clamped, axes-parallel with the coordinate system of the machining table
- The workpiece sides to be measured are rectangular to another

Before executing the measurement procedure, the measuring probe must be positioned manually, with the jog direction keys, to the measuring start position of the workpiece side to be measured. If G07 is active, the measuring probe can be positioned, either axes-parallel in its basic coordinate system, or in accordance with the tilted machining plane. This is determined by a soft key.
The measuring probe is moved with the jog direction keys from the start position towards to the workpiece side with measuring feed. The axes movement direction must be selected with the soft keys F1 to F5 previously. If the measuring probe touches the workpiece side, the movement stops and the measured position is displayed in the "measured" window.

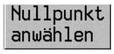


Measuring procedure

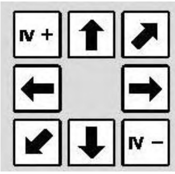
The function „Measure side“ functions only when the machine tool is prepared for measuring probes. Refer to your machine tool manual.



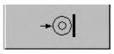
1. The measuring probe is activated by its tool number via MDI and by selecting the function „ Measure side “ in the zero points menu



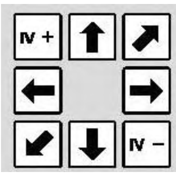
2. Select the zero offset shift number (G54—G59) or select a zero offset index number (I1—I10) in the active zero point shift number (G54—G59). (See picture right)



3. Move the measuring probe with the jog direction keys or the HR410 to the measuring start position of a workpiece side to be measured

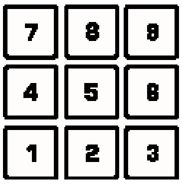


4. Select the axis movement direction of the particular axis X, Y or Z with the soft keys F1 to F5. The measurement procedure now is activated.

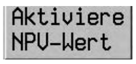


5. Move the axis with the jog direction keys, with programmed measuring feed (F), to the workpiece side. If the measuring probe touches the workpiece side and switches, the movement stops and the measured position of the selected axis X, Y, or Z is displayed in the measuring window.

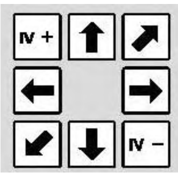
Attention: The axis must be moved perpendicularly to the workpiece side.



6 Enter the zero point coordinate X, Y or Z



7. The zero point coordinate is calculated and stored. The actual position is displayed.



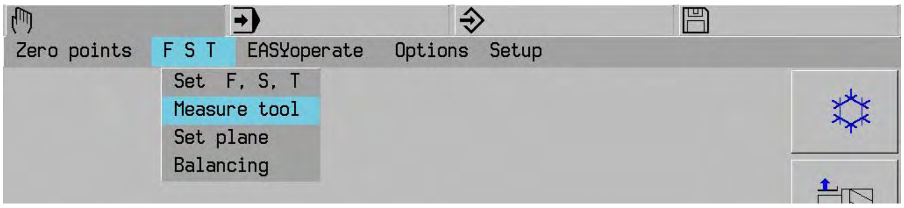
8. Move the axis with the jog direction keys away from the workpiece side.

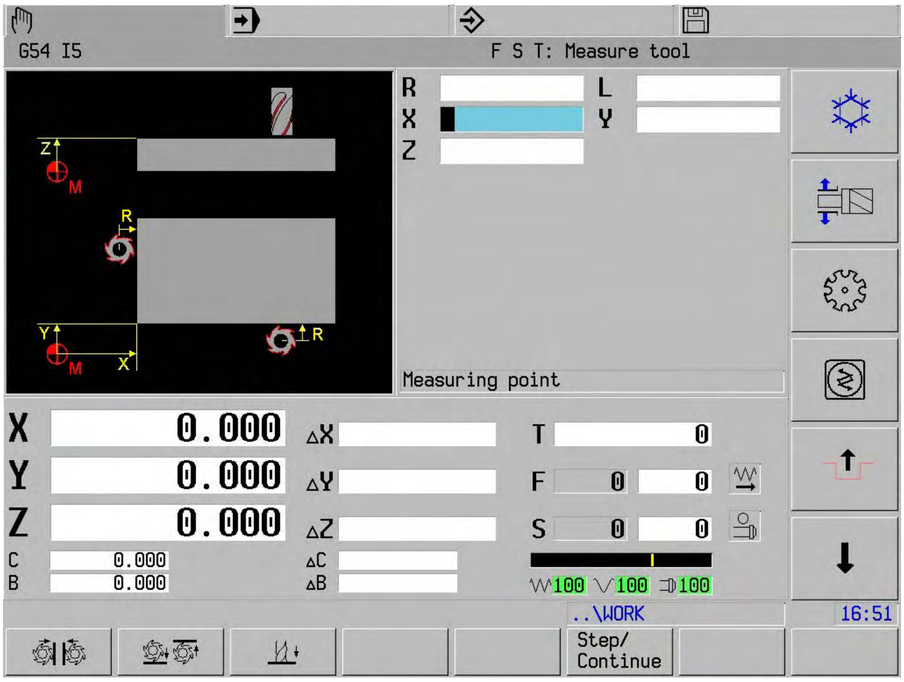
Attention: The axis can only be moved away from the workpiece side.

8.5 Measure tool

The Measure tool function is used to determine tool offset values (radius and length) for the active tool. The offset values are added to the Tool table.

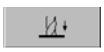
Example: tool length measurement.
- Activate machining levels (e.g. G17)
- Activate zero offset (e.g. G54 or G54 I10)
- Change tool in the spindle (e.g. T1)





The actual tool values are shown under R and L
Radius measurement:
- Enter reference position (e.g. X20).
- Approach reference position.
- Establish tool radius, using softkeys

Length measurement:
- Enter reference position (e.g. Z0).
- Approach reference position.
- Establish tool length, using softkey

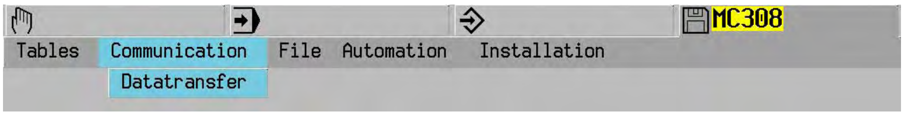


Note Please refer to the chapter Tools.

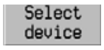
9. Data input / output and file management

Data transfer is not active on a double processor system. Files from a table are transferred to by means of Windows Explorer.

9.1 Data transfer



9.2 Coordinate controller with peripheral device



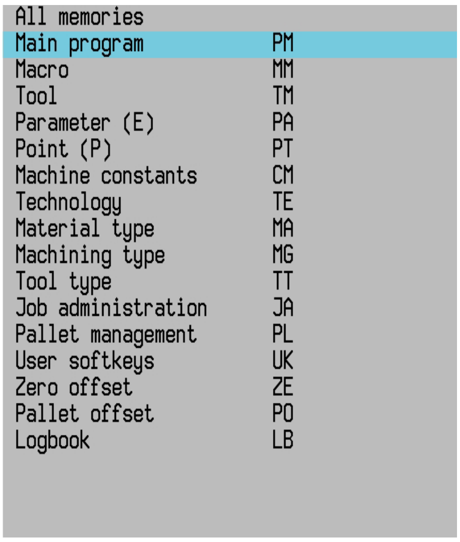
Note

Machine constants for units:

900-908	910-918	920-928	780-783 930-936	790-795	797-799

Block number > 9000, refer to user machine constant list (MC772-774).

9.3 Memory name abbreviations

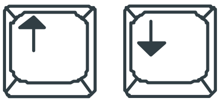


Note

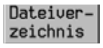
- At mc84=0 the zero offset identifier is ZO.ZO and at mc84>0 ZE.ZE.

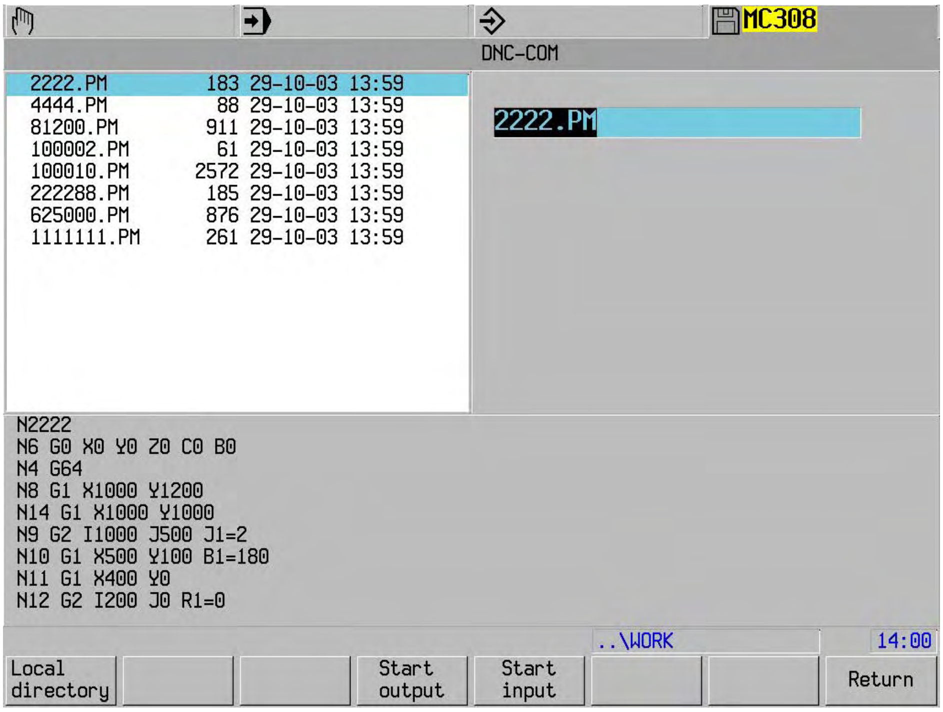
9.4 Reading

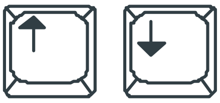
9.4.1 Reading in the program (PM,MM)



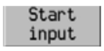
Select PM or MM.



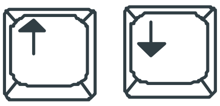




Select the main program or macro from the list



9.4.2 Reading in tables (TM..PO)



Select a 4able from the list.

Note

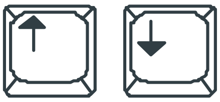
When the technology tables have been read in, they must be saved on the hard disk enabling them to be activated after the controller has been switched on/off (the CNC always saves in the startup directory)..

9.5 Output program

9.5.1 Data back-up

The user should regularly save his programs (PM and MM) and important data (e.g. technology data, machine constants, tools, etc.) onto his PC to prevent irretrievable data loss.

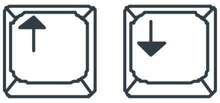
9.5.2 Reading out the program (PM,MM)



Select the main program or macro from the list.

			Start output	Start input		ID List	Select device

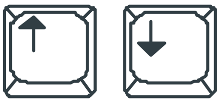
ID List



Select a program

Start output

9.5.3 Reading out a table (TM-LB)



Select a table from the list.

			Start output	Start input		ID List	Select device

Start output

9.6 Mini-PC

3,5" disk drive

			Start output	Start input		ID List	Select device

Select device

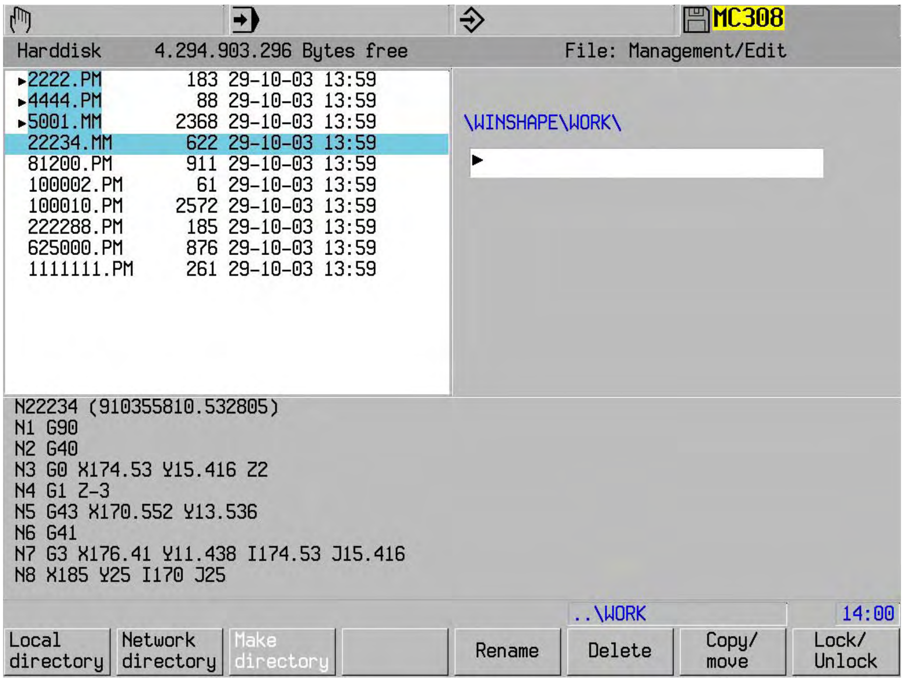
Device 1	Device 2	Device 3	DNC (COM)	DNC (TCP/IP)	Mini-PC		Return

Mini-PC

9.7 Select files

Ctrl	with	Enter	Select a file
Ctrl	with	S	Select a file. A toggle function (between <selected>/<not selected>) for the file where the cursor is currently positioned; the cursor moves to the next file..
Ctrl	with	A	All files in the directory are selected
Ctrl			The current selections are cleared

Files that have been selected have a ➢ character in front of their name



Files can be selected in this way in the following menus:
File Management/Edit:
- Delete file
- File properties

Communication:
- Upload
- Download

Notes:

A destination file that is entered when more than one source file has been selected will be ignored. The destination is then assumed to be the destination directory.
An unselected file at the current cursor position will not be included in the operation.

9.8 File management

A hierarchical directory structure is present on the hard disk supplied. The structure looks like this:

\STARTUP
- WORK
- TEMP

The technology tables and subprograms in the startup directory are loaded into the CNC DRAM when the CNC is initialised.

Executing a faulty program can lead to dangerous situations.

In the Automatic and Edit operating modes, the programs are always selected from the harddisk. The directory can be changed in the modes of operation.
Selected programs are loaded into the working memory (DRAM).

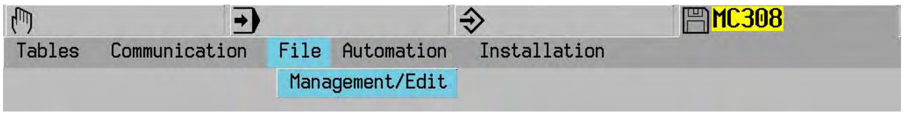
Notes

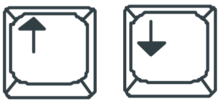
- If a faulty file is found during loading, loading is stopped.
- Programs are checked as they are loaded. If an error occurs during loading, an error message (in brackets) is appended to the relevant program block.
  Example: N.. G301 (O... "Original block contents incorrect")
- The startup directory contains the technology tables and the IPP setup macro. We recommend not to store other programs in the startup directory. The only exceptions are e.g. subprograms which are invoked in several main programs.
- During copying, renaming or loading, the program number in the first program block is adapted to the file name, provided the name of the file matches a valid program number.
- Main programs (invoked with G23) and subprograms (invoked with G22) have to be in the directory of the active main program.
- When leaving the editor, the program will ask whether the changes are to be stored. Changes in the active main program and in the accessory subprograms are stored automatically.
- Large programs that do not fit into the working memory have to be executed with softkey "CAD-Betrieb". However, when in a program that is not executed in "CAD-Betrieb", it is still possible to invoke and execute a large program via G23.

9.8.1 File editing

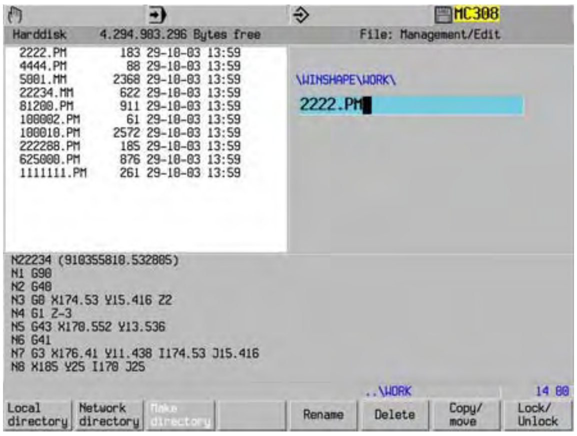
The description of operation is based on the (SP) single processor. If at the (DP) dual processor significant deviations exist, then these will be described/shown on the right site from the dotted line alongside the (SP) single processor.

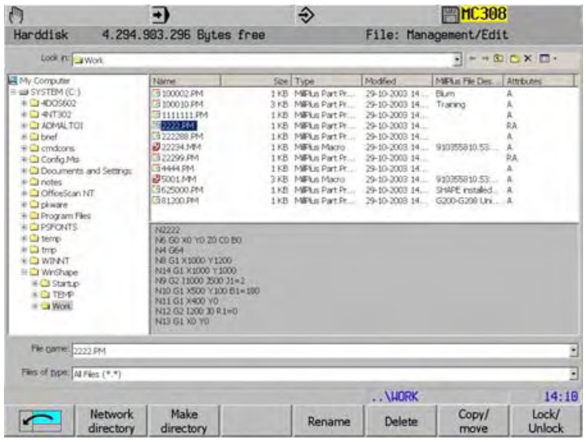
All edit functions are accessible in the control process via File Management/Editor



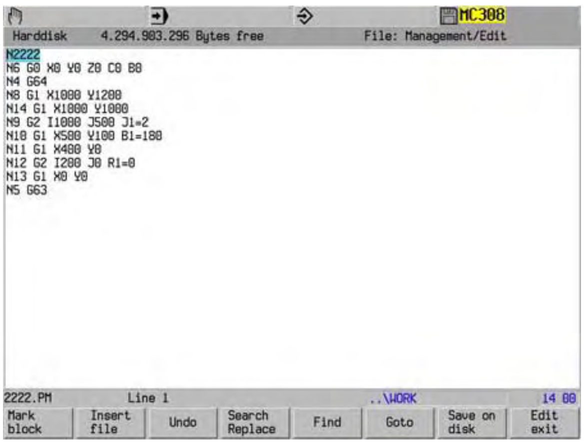


Select program or enter programnumber (e.g. 2222.PM)





Activate file 2222.PM

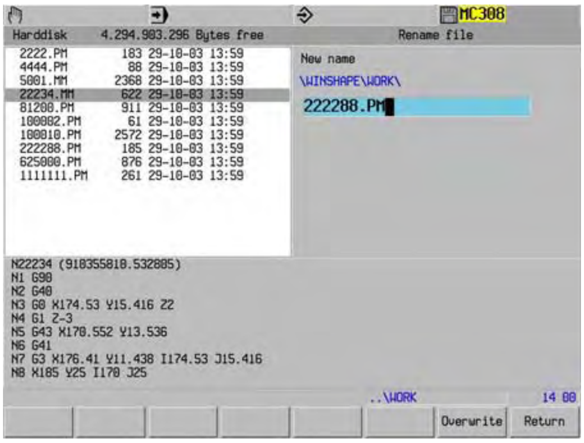


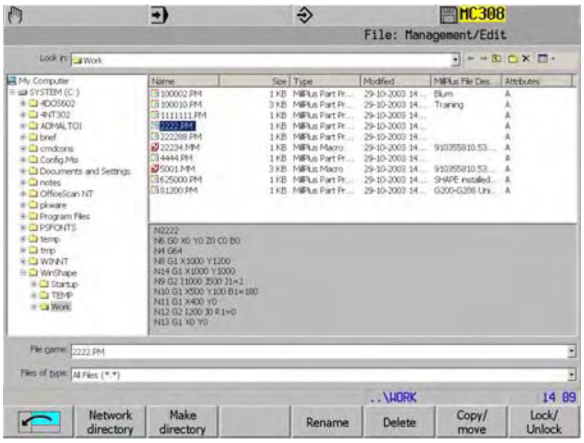
Return to main screen

9.8.2 Rename/shift file


Select file or enter file name (E.g. 20001.PM)

Rename — Enter other file name e.g. 222288.PM)





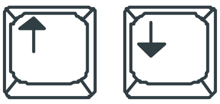
Rename — Overwrite file name (2222288.PM)

Overwrite file name (2222288.PM)

When program 222288.PM already exists, proceed as follows:

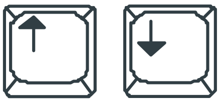
9.8.3 Delete file

Only programs in the current directory can be deleted.
When deleting a complete directory (*.*), the contents of the directory are deleted. The directory itself is not deleted.

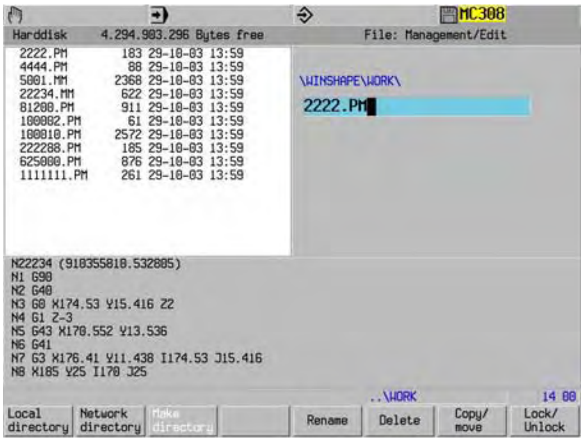

Select program or enter program number

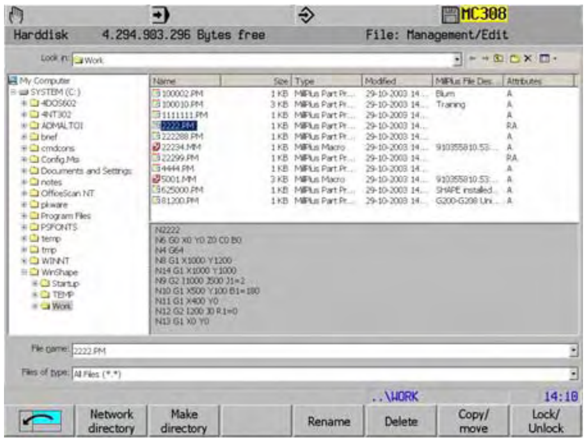
Delete

9.8.4 Attribute file (lock/unlock)



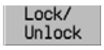
Select program or enter program number



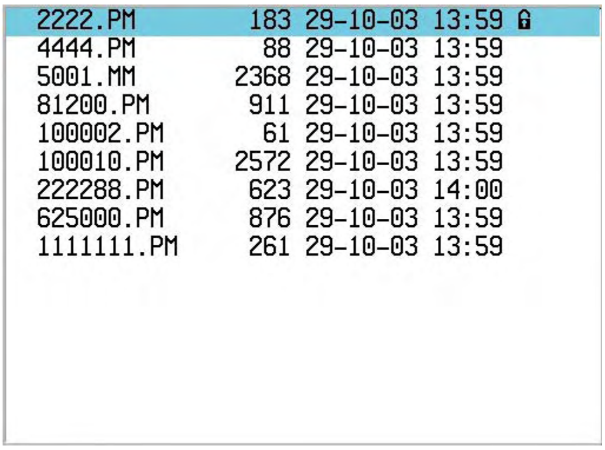


Protected files are marked.

Protected files are marked in the colom "Atributes" with R

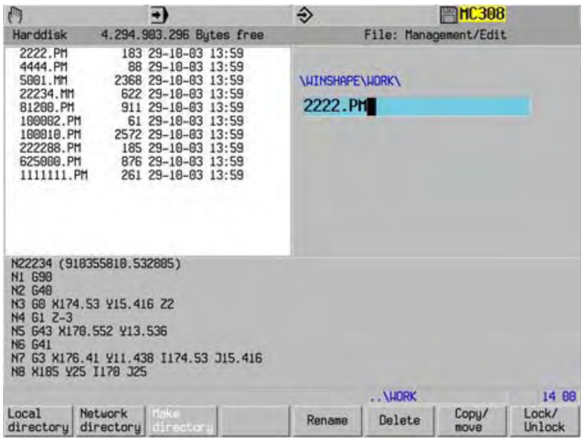


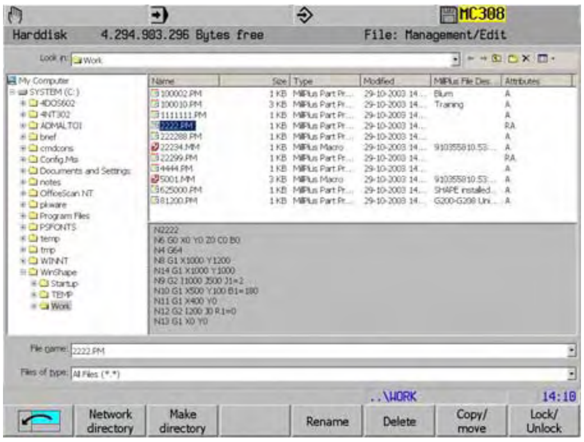
Lock or Unlock file



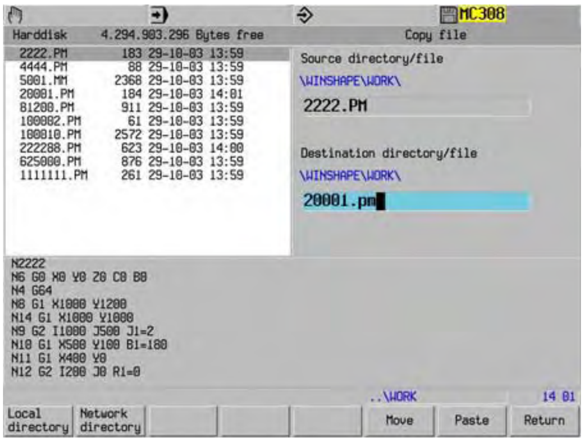
9.8.5 Copy file

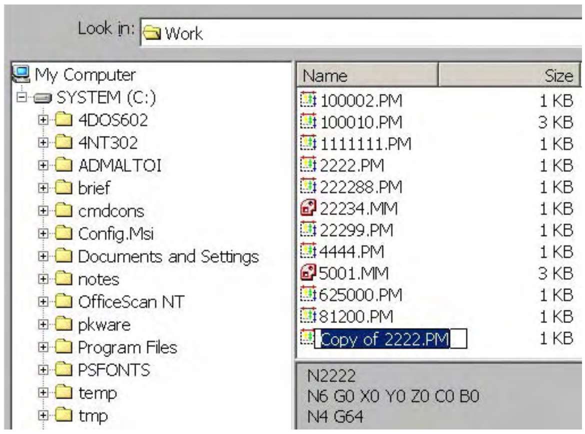
The <File: copy> function is identical regardless of whether the file is being copied across the Ethernet or from one location to another on the local hard disk. The choice of source or target directory determines whether the Ethernet is used or not. One or multiple files can be copied.





Copy/ move — Copy selected file(s)





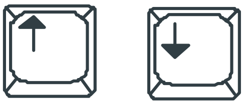
Paste — Enter marked file (e.g.20001.PM) and press enter

Paste — Rename file name (e.g. 20001.PM) and pres ENTER

Enter — Efectuate file 20001.PM

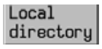
In stead of copying of one file, multiple files can be copied and inserted. To do this mark the files as described in chapter 9.7. In the source directory window the mark symbol [►] is displayed Than, as at one file, insert the marked files in the desination directory.

9.8.6 Copy: local/network directory



Select file

Copy selected file



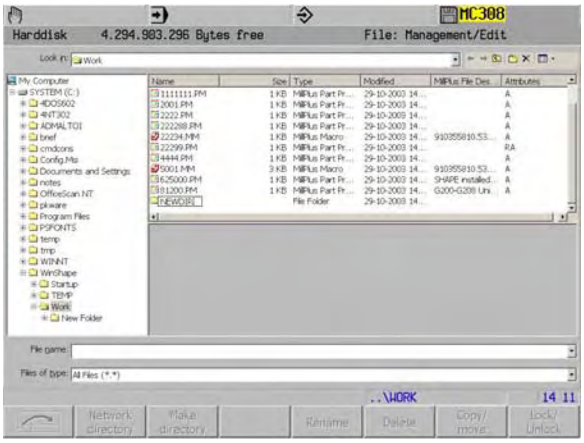
Activate local directory

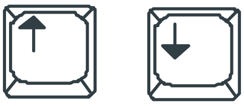
Activate local directory

Or

Activate network directory

Activate network directory





Select directory or enter via entry window

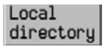
Activate directory

Copy in the directory

Copy file in the directory

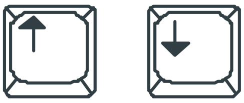
9.8.7 Make directory

This enables you to create a new directory. The name of the directory consists of a maximum of 11 characters (DOS format 8.3 characters). The directory can have up to 5 levels.



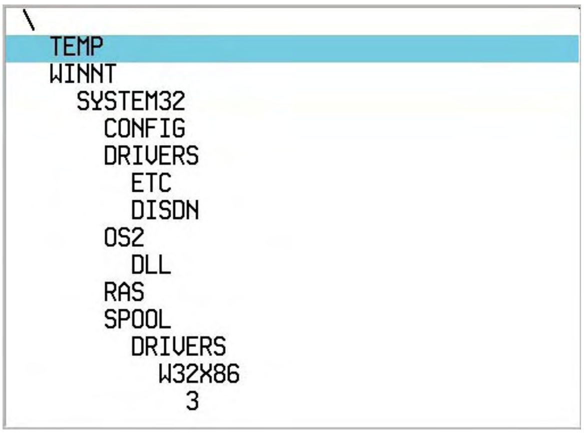
Activate Local directory

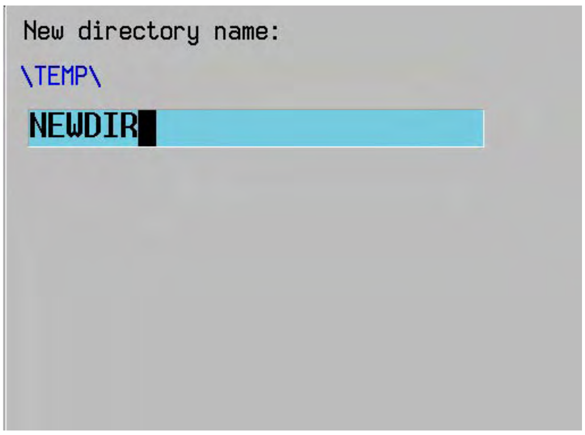
Activate Local directory

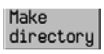


Select directory

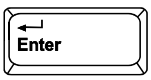
Activate directory







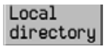
Make directory



Activate directory

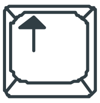
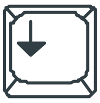
9.8.8 Remove directory

The directory must be empty. The actual directory cannot be removed.



Activate local directory

Activate local directory

Select directory

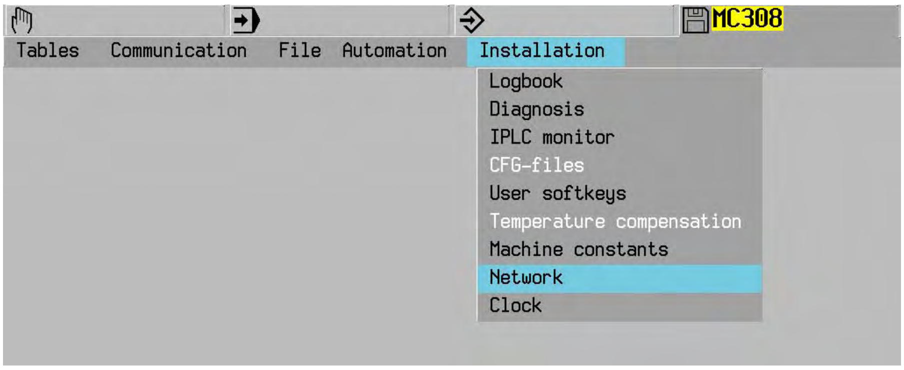
Remove directory

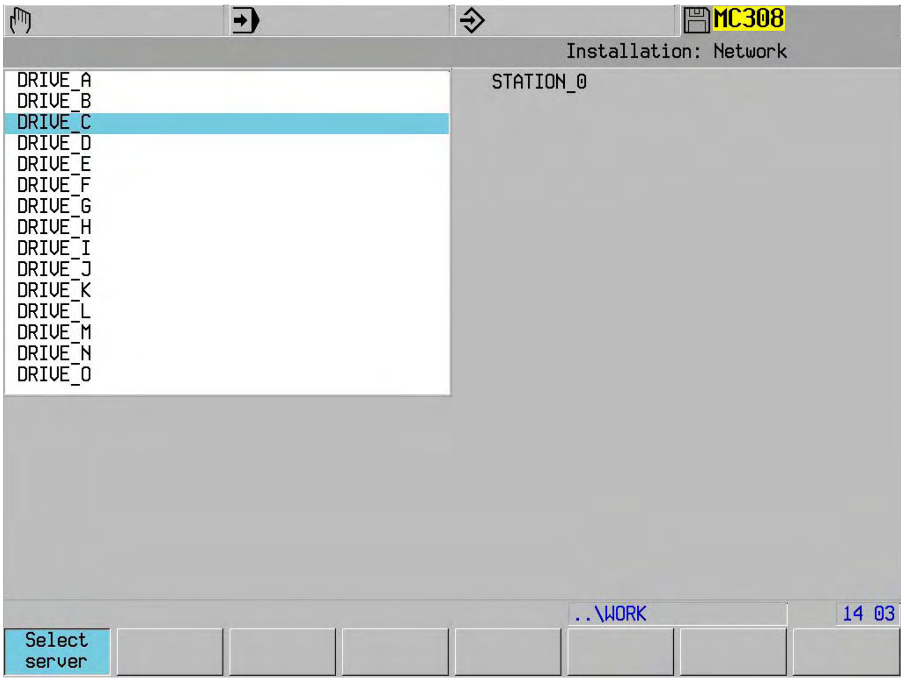
9.9 Ethernet interface

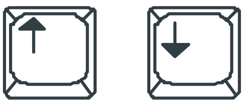
Additional disk drives become available if MillPlus is connected to a network. The Copy File function is the only one that can also be used on network drives.
For details on how to set up the interface, refer to the chapter entitled Miscellaneous.

9.9.1 Select server

The server is the network node that is used to transfer data. Only one server can ever be active at a time. The configuration file contains a list of possible servers. The server that is selected must be an active server.







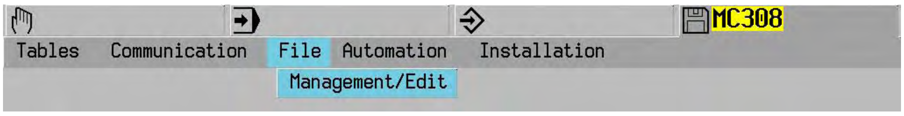
Select server

Make server active

Note

Ethernet provides no way of preventing two clients from accessing the same file on the server at the same time. This may result in corruption of one of the files.

9.9.2 Write to server



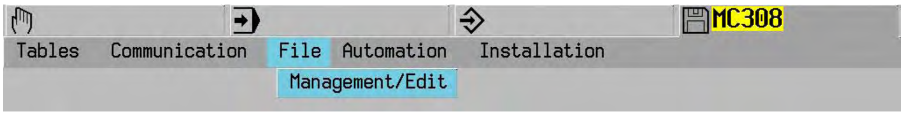
Send the files from the current directory on the CNC hard disk to the specified directory on the server.

-Select source directory on CNC

-Select target directory on server
-Select or enter file name

Write file to server

9.9.3 Read from server



Copy the files from the server to the current directory on the CNC hard disk.

-Select source directory on server

Read file from server

-Select target directory on CNC
-Enter target file name

Write file to CNC

10. Enter / edit program

10.1 DIN/ISO Editor

To edit DIN/ISO programs.

10.2 IPP Editor

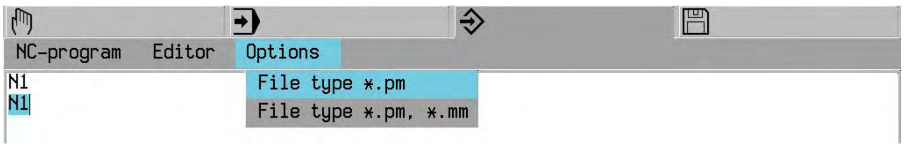
To edit IPP programs.

10.3 Input help

The following are available:

Interactive parts programming (IPP)
Interactive contour programming (ICP)
Support for G-Functions

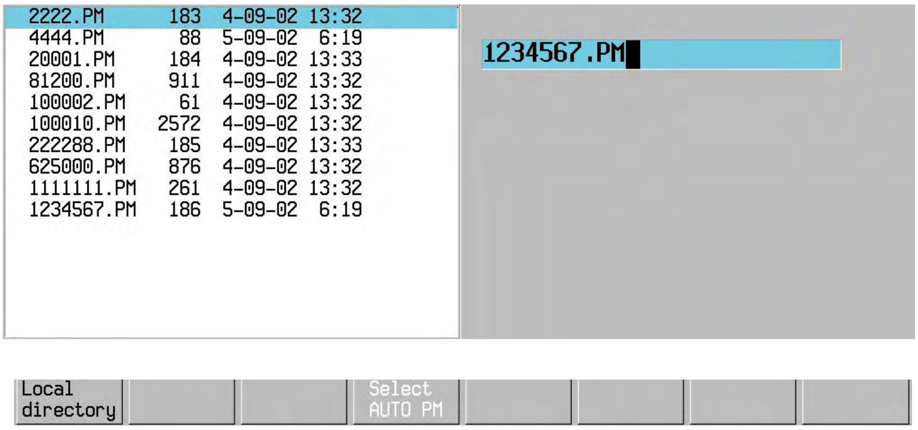
10.4 Enter new program number (main program / macro)



Selecting file type *.pm, *mm: Program window with main programs and macros together.

Example: 10002.PM

10.5 Entering new program numbers (main program / macro)



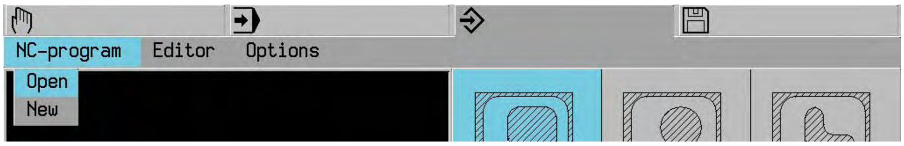
Enter program number (1-999 999 9)
Example: 10002.PM

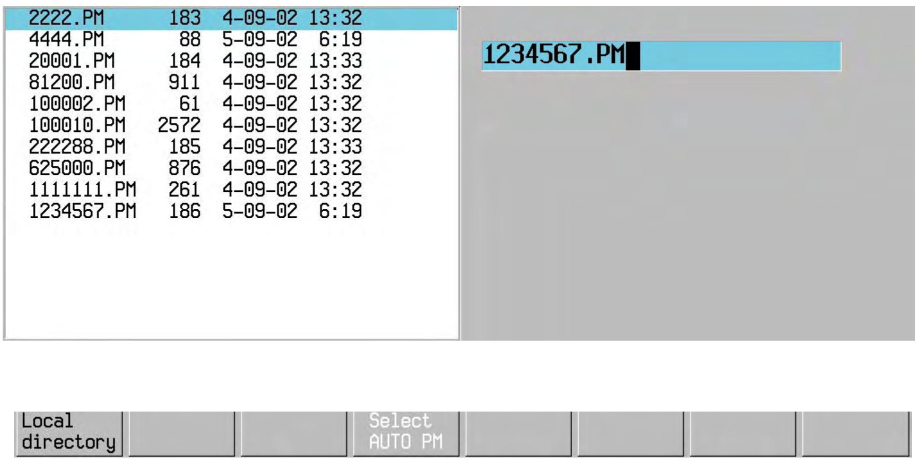
Start the active editor with the new program number.

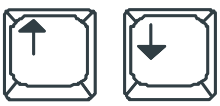
Note

Main programs (invoke with G23) and subprograms (invoke with G22) should be in the directory of the active main program.

10.6 Program selection (main program / macro)



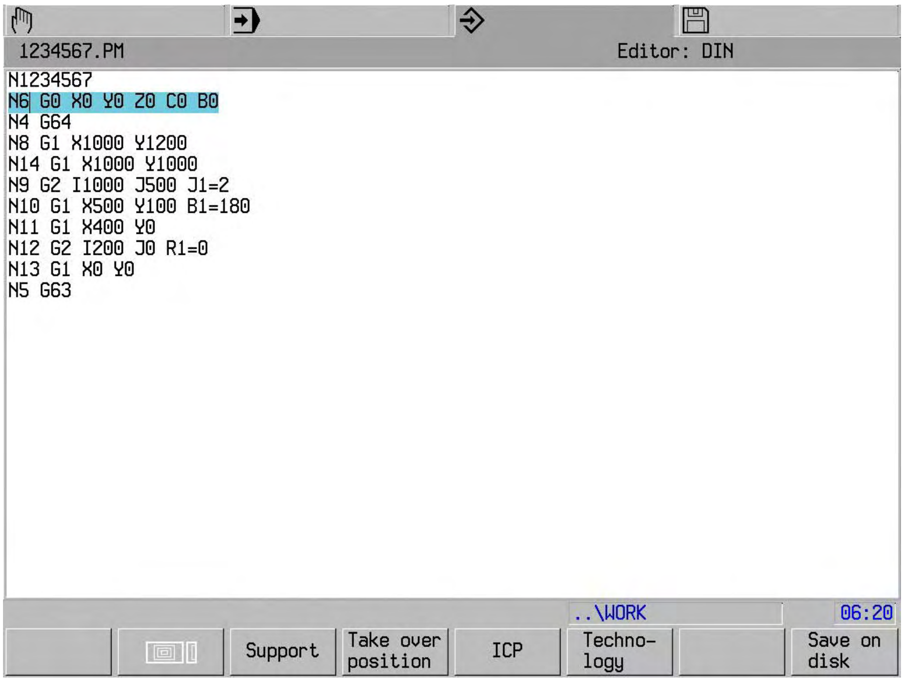




Select program e.g. 1234567.PM.
When entering the program number it is not necessary to enter the extension .PM or .MM.

Activate the program that must be edited

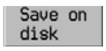
.



Storage request after editing and selection of edited NC program via the menu.

Changes in the active main program and in the accessory subprograms are stored automatically.

10.7 Save to hard disk .

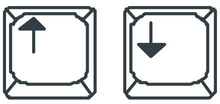


ave program to hard disk.

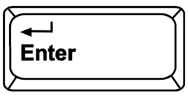
10.8 Enter program block

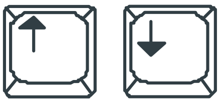
Directly at cursor position using ASCII keyboard

10.9 Insert program block



Select sentence number after which a sentence is to be added.





Edit block and close.

10.10 Text entry.

Text in brackets after parameters, maximum length 124 characters.

Example:
G1 X50 Y83 M13 (turn on coolant)

v

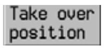
10.11 Mathematical entry

The functions sin(..) cos(..) tan(..) asin(..) acos(..) atan(..) sqrt(..) abs(..) int(..) may only be written in lower case.

Spaces are not permitted in functions.

Maximum size of on one line: 248 Characters.

10.12 Position transfer in the program (DIN editor))

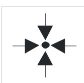


Select the axes to be transferred

Transfers the current position of the selected axes into the program

to the DIN editor

Transfer position using HR410.
Select the axes which should be transferred..



Transfers the current position of the selected axes into the program at the cursor point. Afterwards an <Enter> is automatically executed.

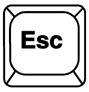
The position can also be transferred while the machine is in motion.

Note

If G0 X100 is written in the line and position X121 Y122 is transferred, the final line entry is G0 X100 X121 Y122. Subsequently the programmer must delete one of the two X addresses.

10.13 Delete address

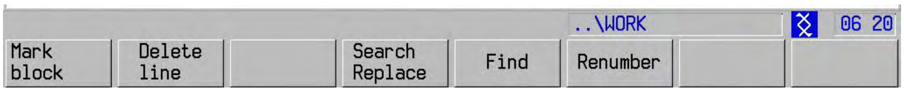
Deletes character to left of cursor.



Undelete the most recently deleted addresses in a block.

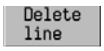
10.14 Block (Move, Copy)

Activate the EDIT softkeys..



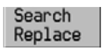
Quit the EDIT function.

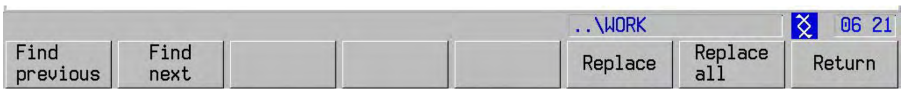
10.14.1 Erase line



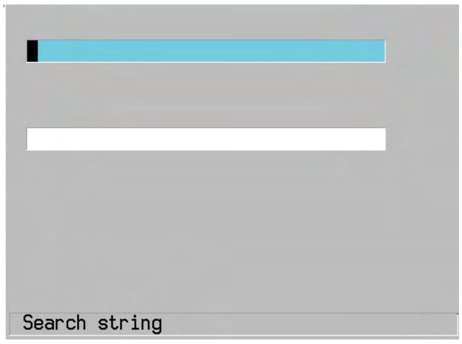
This enables you to erase the active line (indicated by the cursor)

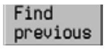
10.14.2 Search & replace

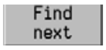


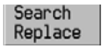


Enter character string

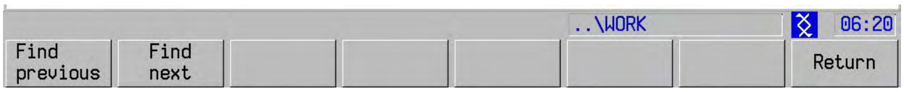




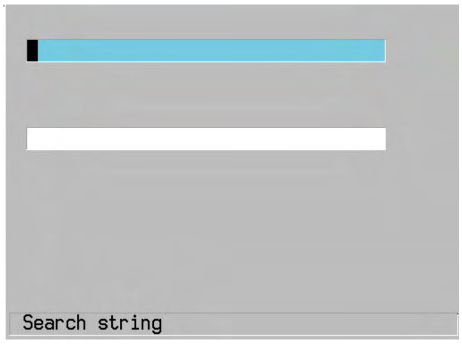


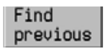


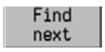
10.14.3 Find



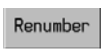
Enter character string







10.14.4 Renumber

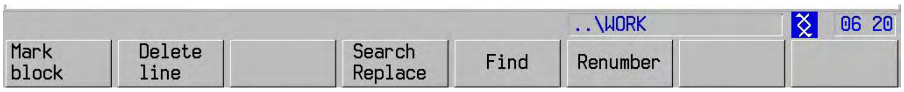


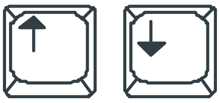
The blocknumbers of the program blocks are renumbered.

Note

The new numbering begins with the sentence number of the first (marked) sentence.

10.14.5 Block (Delete, Renumber)





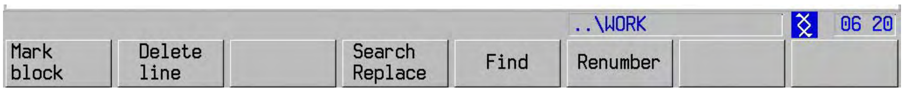
Mark a program record/block.

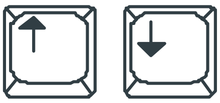
Perform operation

Note

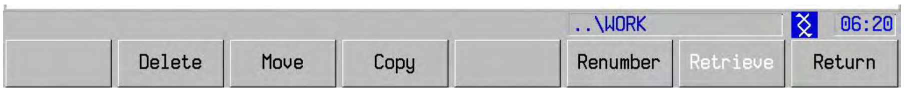
The new numbering starts with the block number of the first highlighted block.

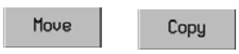
10.14.6 Block (Move, Copy)



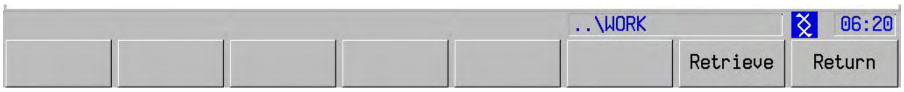


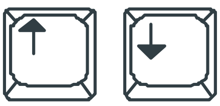
Mark a program record/block





Save program record/block in intermediate storage

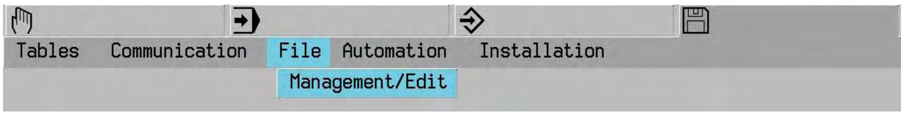




Select record number

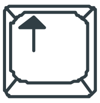
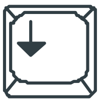
Save record/block in program

10.15 File editor

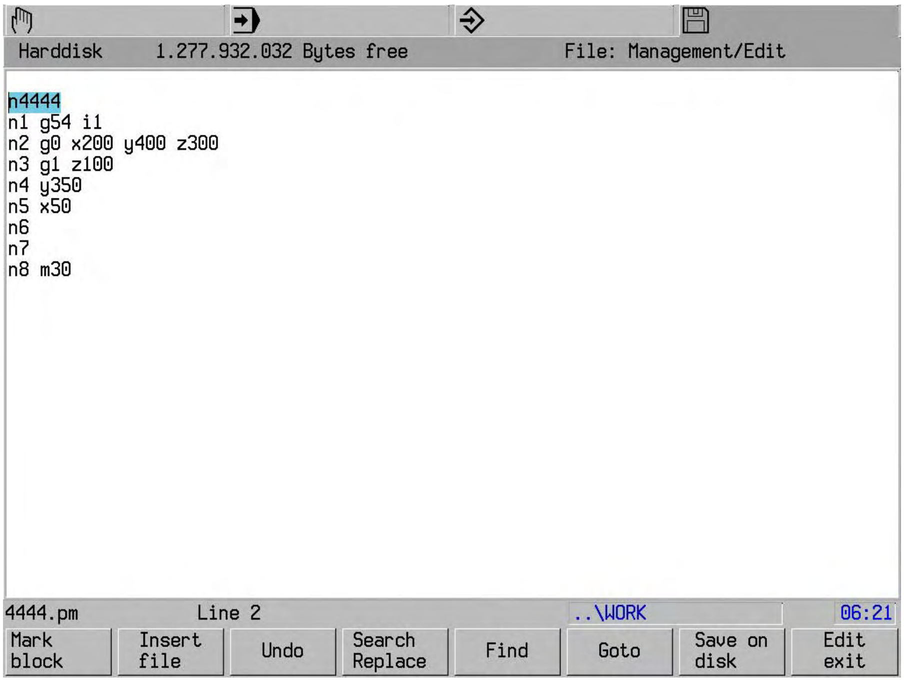


Enter program number, example: 4444.pm

Or

Select blocknumber



Changes take effect immediately.
The file editor does not check blocks as they are entered and saving. Test the program using the graphic test run function.
The graphic test, support, ICP and technology functions are not supported by the file editor.

Features:

For editing programs larger than 1 Mbyte
Blocks are not checked as they are entered and saved
Editing of active programs not possible
No NC language support while editing

10.15.1 Undo)

Undo

Up to 100 operations can be undone.

The following operations cannot be undone:
-Select, Delete, Move, Copy Block
-Write Block/Insert File
-Search & Replace

10.15.2 Go to line number

Goto

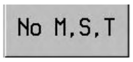
Note:
The line number refers to the line number in the file, not the record number N in a program.

11. Program dry run

11.1 Dry run mode

The test run takes place at an increased feed rate (MC 741).
Activate the program.

11.1.1 Select dry run option



M,S and T not output

Note: Lock axis
MC 100 C3 (1st axis)
MC 105 C3 (2nd axis)
MC 110 C3 (3rd axis)
MC 115 C3 (4th axis)

11.1.2 Perform dry run

Start dry run

11.2 Graphics dry run

Activate the program.

11.2.1 Graphic functions

Single block	/N Block delete	Optional stop	Execution status			Graphic-function	Cancel program

Graphic-function

2D Wireplot	2.5D Wireplot	3D Wireplot			Crosshair	Overview	Return

Select 2/2.5/3D view

3D Wireplot — e.g. 3D view

2D Wireplot	2.5D Wireplot	3D Wireplot		Rotate		Overview	Return

Rotate

						Overview	Return

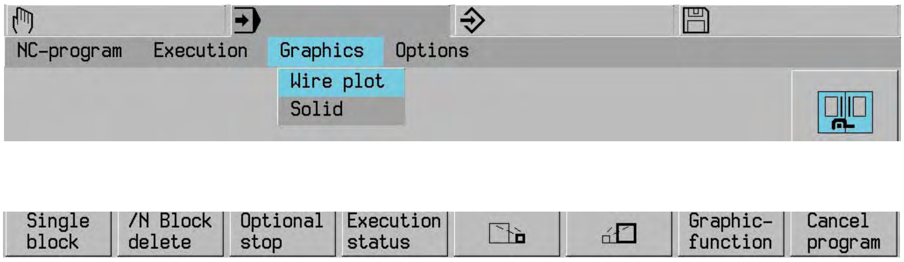
11.2.2 Graphical representation

Rotate	Graphical representation
	Enlarge drawing step by step
	Reduce drawing step by step

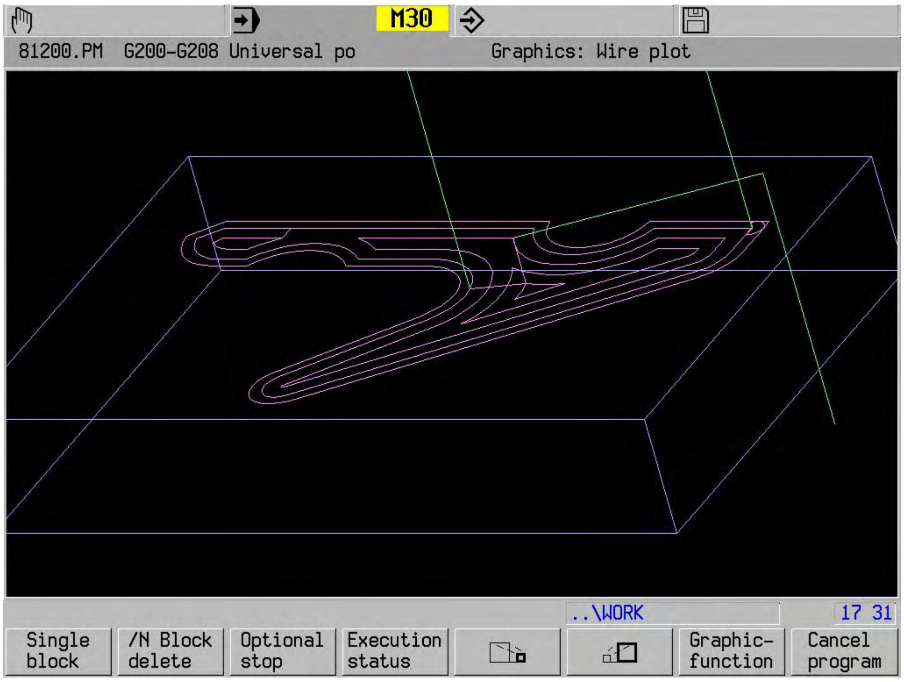
11.2.3 Graphic options

Toolpath	Rapid	Contour		Actual position	Runtime tools		Collision detection

11.2.4 Wire plot



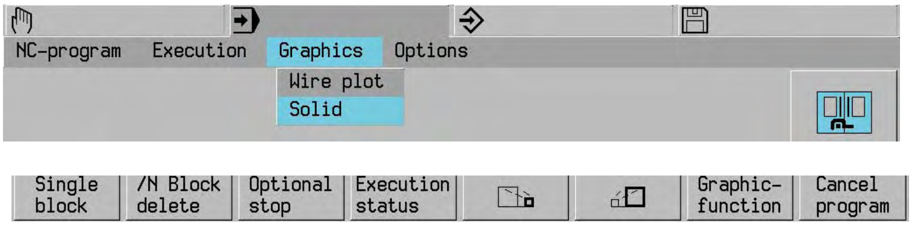
Start graphics solid



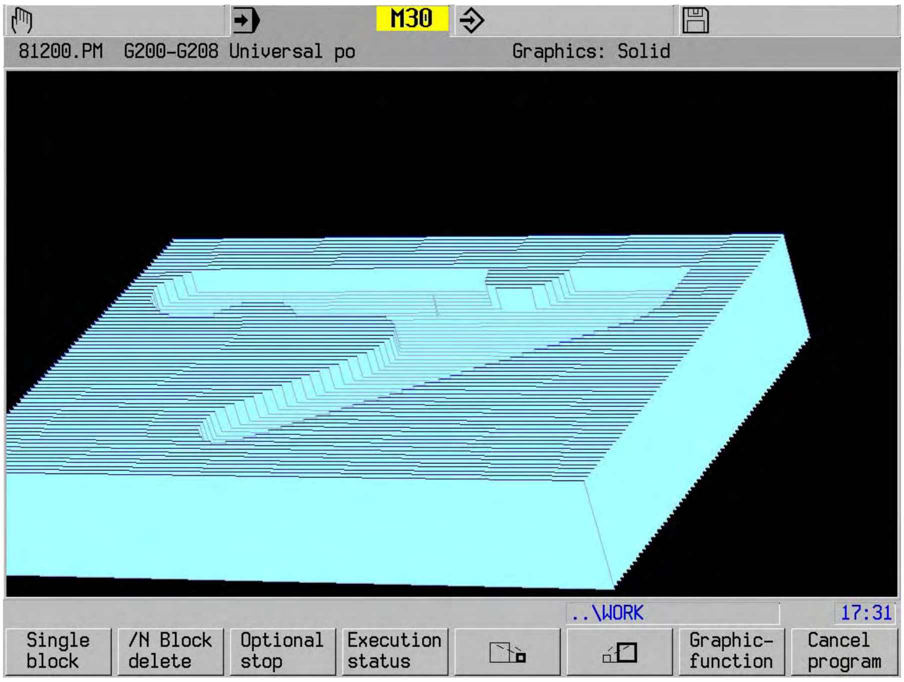
11.2.5 Working with graphics (example)

- Activate the program.
- Select the Graphics option.
- Select Wire or solid graphic.
- Start the program.

11.2.6 Solids



Start graphics solid



11.3 Estimation of run time using graphics operation

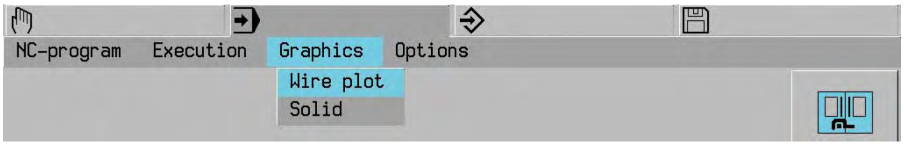
During graphics operation the graphics run time is displayed in the operating status.
The run time is calculated from the tool length and the feed rate (correction = 100%). 10% is added to this calculated time to allow for braking and accelerating in the corners. During programmes at high rates of advance the estimated run time is less than the actual run time, because the machine cannot track the program.

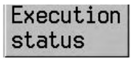
Note

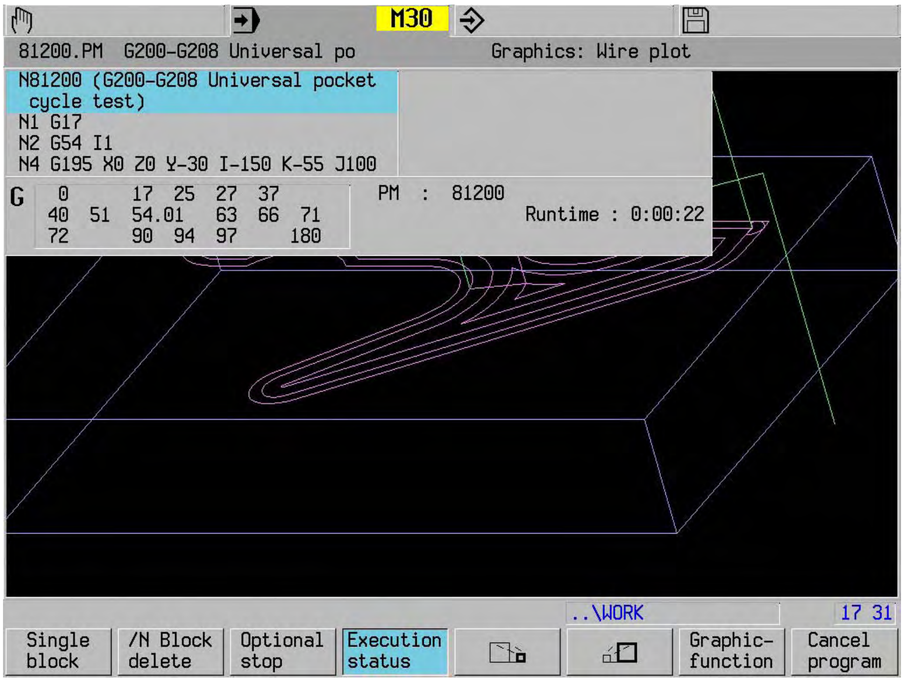
Time taken by the M functions is not taken into account in the estimation.

11.3.1 Time for each tool

The estimation of operating time is also calculated for each tool. In the course of this, only the time that elapses with the set rate of advance is taken into account.

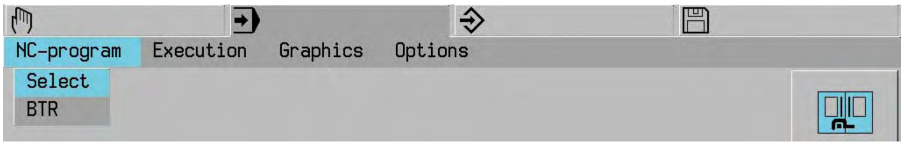


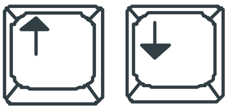




12. Activate / execute program

12.1 Activate program



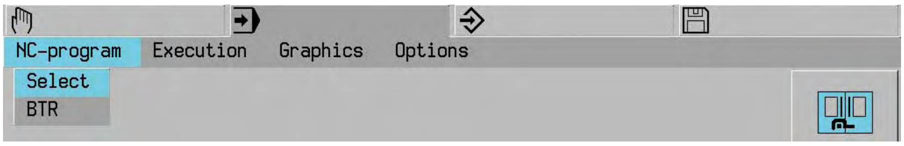


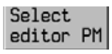
Position the cursor on the desired program or enter program number.

Operating mode "Execution: Machining" is automatically activated.

12.2 Direct activation of an edited program

Editing a program

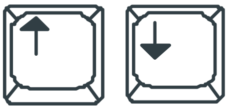




12.3 CAD mode

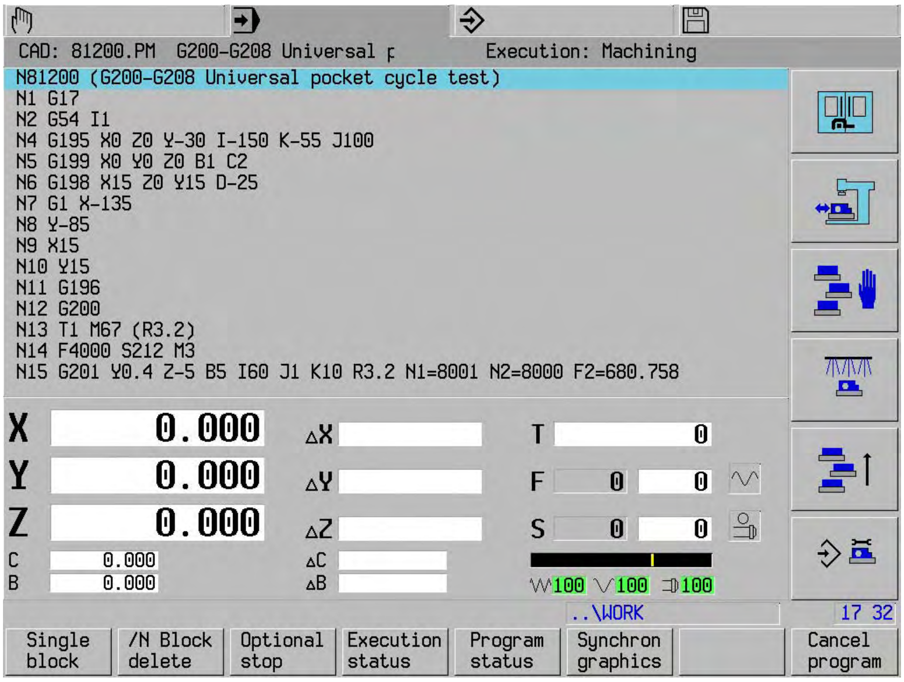
The "CAD mode" function enables you to process programs that require more memory than is available in the CNC-RAM. The size of BTR memory is defined in MC93. (Example 128kbyte).

CAD mode



Position the cursor on the desired program or enter program number.

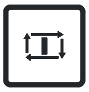
Operating mode "Execution: Machining" is automatically activated.



Note:

The main programs must not contain G23, G14, G29 functions or E0 parameters.
"Satz suchen" backwards is not possible.

12.4 Execute program



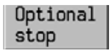
12.5 Single block operation

12.6 Delete block

Note:

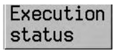
The program block must start with a '/', e.g.: /N5 G1 X100

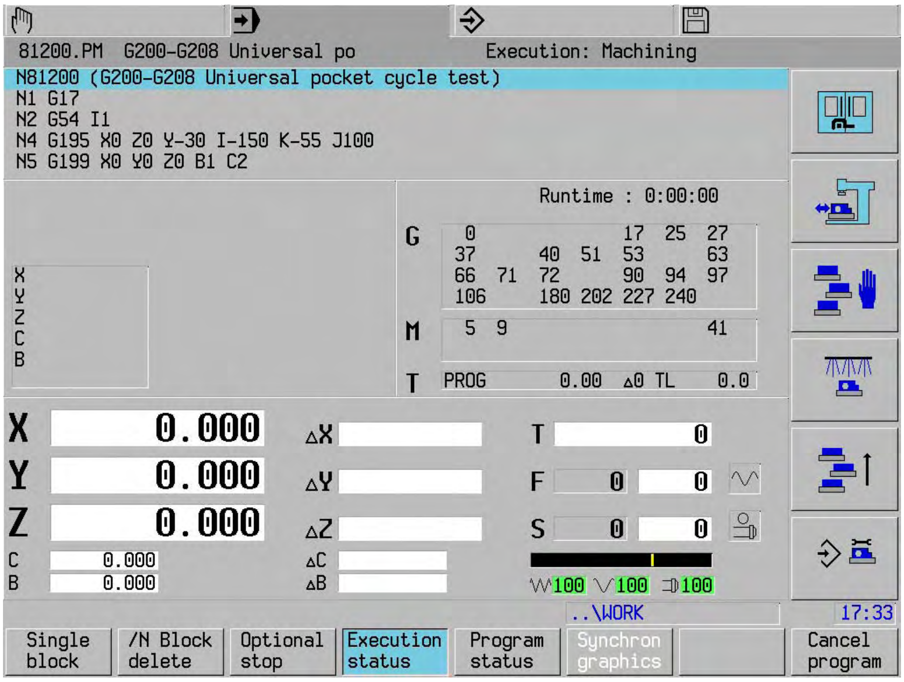
12.7 Optional halt



Halt following execution of M1.

12.8 Execution status





The overlap depth is indicated in the operating status behind MM:

Notes

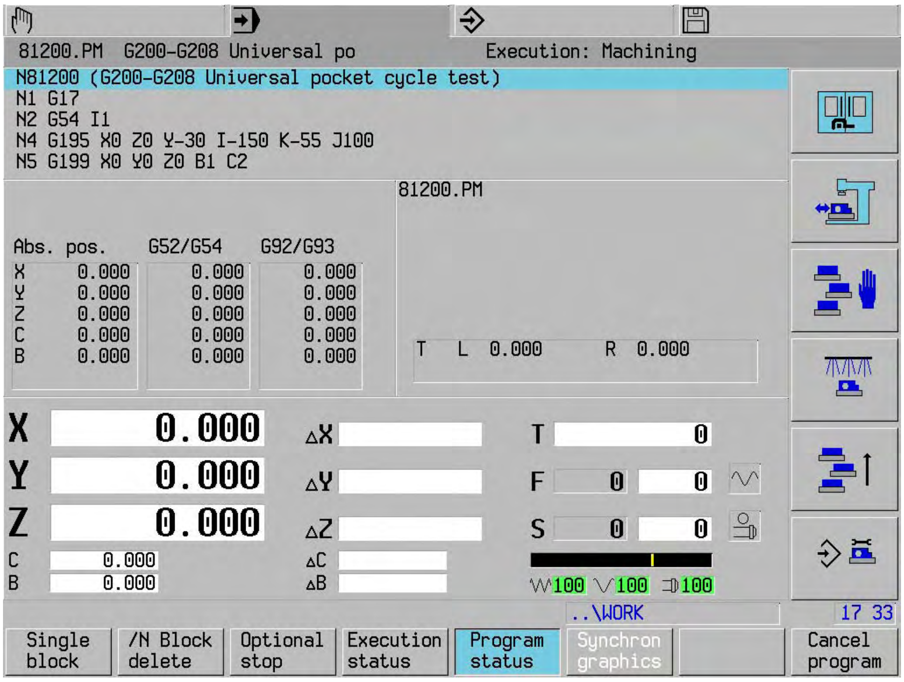
- During BTR and CAD operation the overlap depth is not calculated by the BTR macro.
- The first overlap or repeat depth is "1" and is not displayed.

Im Bearbeitungsstatus wird die Schachtelungstiefe hinter MM angegeben:

12.9 Program status

The following elements are displayed concurrently:

- current tool length (L+L4=) and tool radius (R+R4=).
- current tool oversize G39 L and R
- the position with reference to the machine null point
- the current G52, G54 (Inn or G54-G59) null point displacement
- the current G92 and/or G93 null point displacement
- the complete "overlap tree" of the main programs, the macros and the repetitions

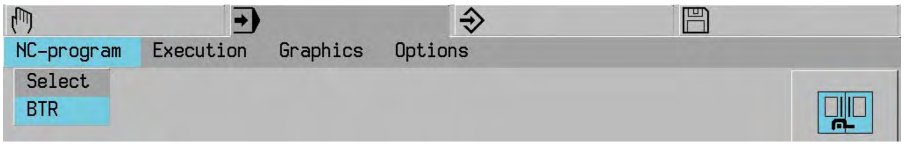


Notes

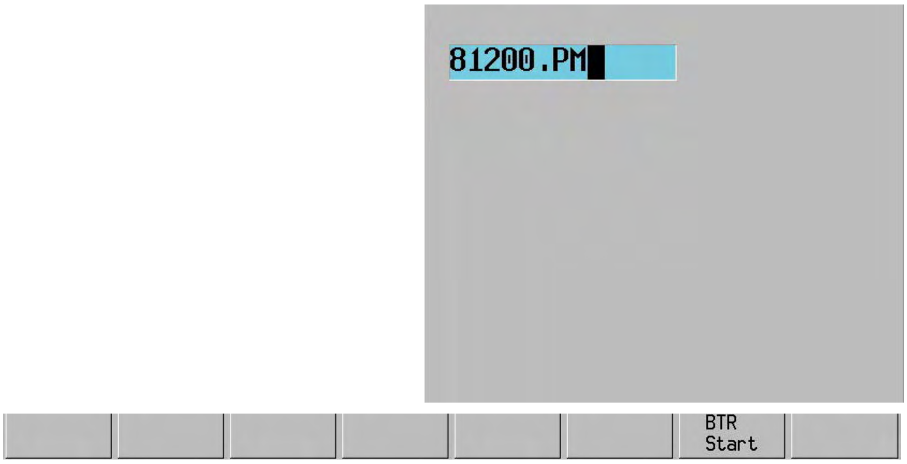
- The overlap tree can hold a maximum of two main programs, eight secondary programs and four repetitions. It "scrolls" automatically in the window as necessary.
- During repetition only the number "still to run" is displayed in Repetitions.
- The command <Program status> cannot be selected during graphics operation.
- Jumps in the program are not displayed in the overlap tree.

12.10 Reload (BTR)

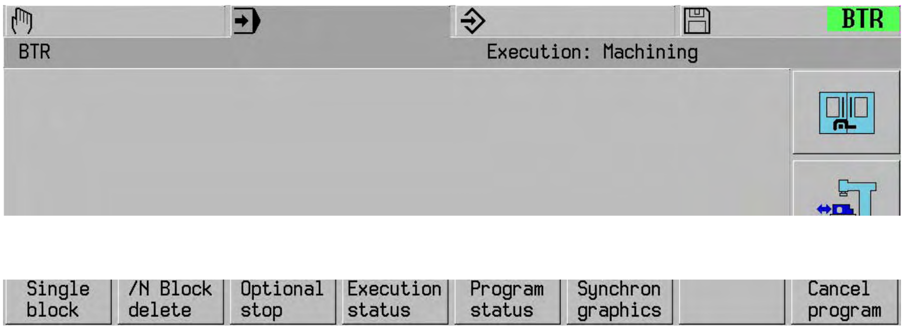
The Reloading function is used to execute programs that need a larger storage volume than the CNC working memory directly from external devices. The size of BTR memory is defined in MC93. Programs from external equipment may be executed by reloading.
Provide data transfer peripheral. (external device with DNC link)

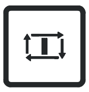


Input program number or select program using the cursor keys.



From external equipment





The program will be executed.

Note:

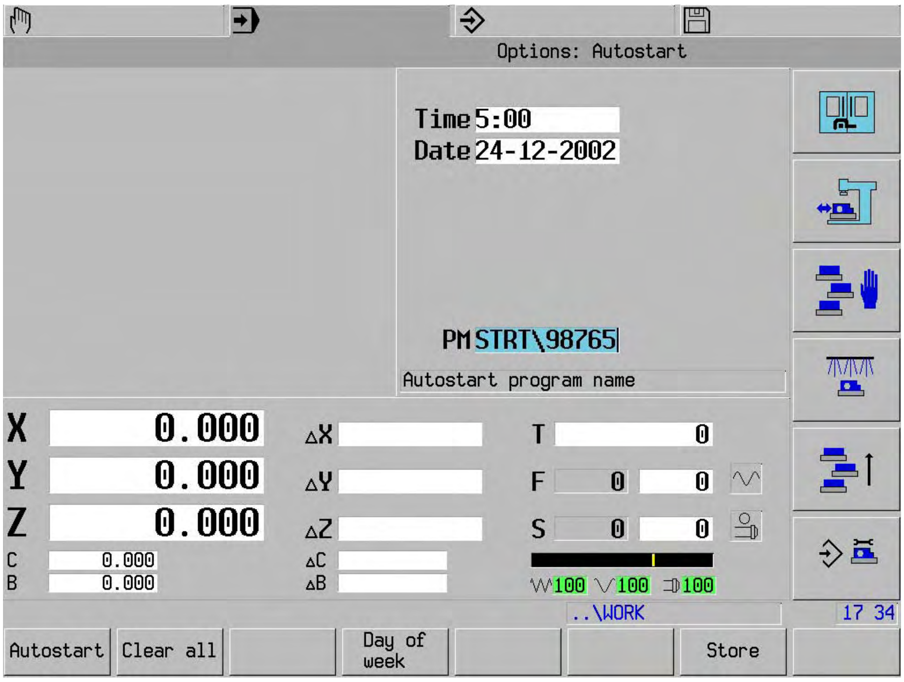
Main programs may not contain any G23,G14,G29 functions or the E0 parameter. A "Search block" is not possible.

12.11 Autostart

The machine should be at operating temperature before machining the first workpiece each morning. The machine is run up to operating temperature by starting a running in program that, for example, lets the spindle rotate for a while. This running in program should be started automatically some time before starting work.

The operator is responsible for ensuring that the machine is in the correct operating mode when the <Autostart> is issued. At this moment, always the actual block or the actual program is started. It can happen, for example, that the operator is running a program in single block mode at the same time that the Autostart issues a <Start>. In such a case the active block will be 'unexpectedly' executed.

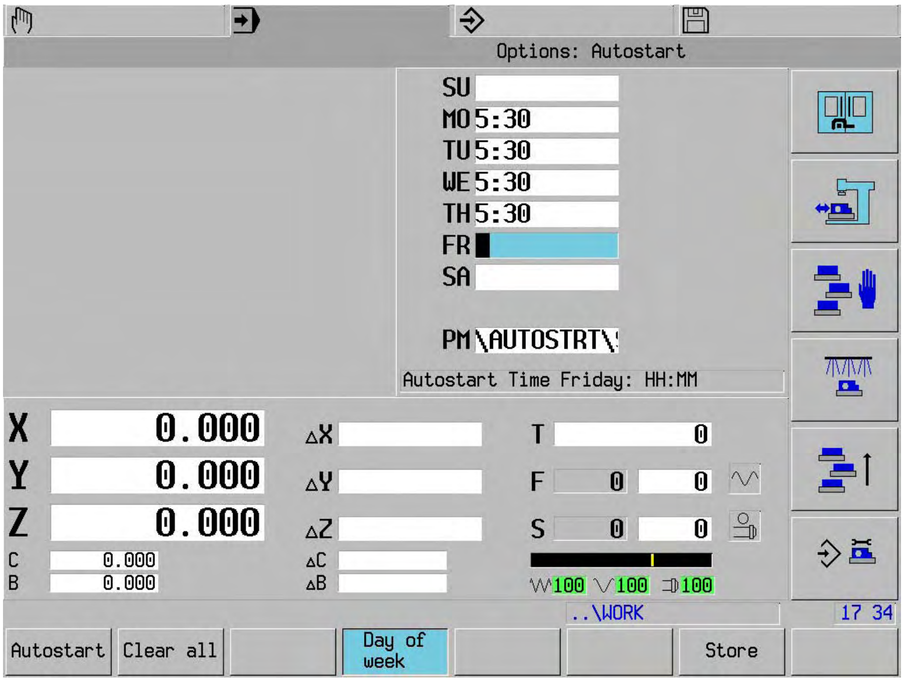
12.11.1 Setting up Autostart



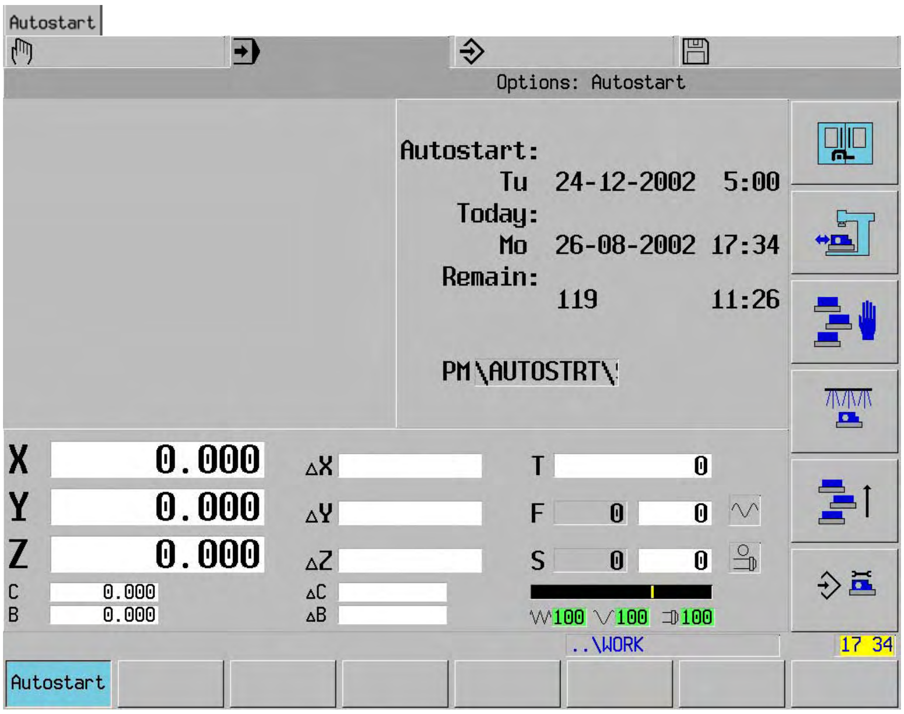
Store   Validates and saves the entered values

Clear all   Input fields on this page are to be deleted

Day of week



12.11.2 Activate Autostart



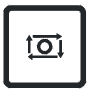
Note:

The CNC and machine tool must be left in the correct operating mode.
If no program is entered, the active program is started.
Autostart active is indicated by a yellow background to the timers

13. Interrupt/cancel program, search block

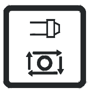
13.1 Interrupt/cancel program execution

Program execution may be aborted at any time during machining and in single block mode.



Feed STOP

or



Feed and Spindle STOP

"Interrupt program" enables programmed feed movement, using the axis movement keys (except Threadcutting).

Possibilities during program interruption
During program interruption the following functions are possible at the interruption point:

- Continue program.
  With the START button the program execution continues.
- Tool movement away from the workpiece.
  In "Manual operation" the axes can be moved manually with the jog direction keys, with programmed feed, away from the workpiece. If the machining plane (G7) is active, a softkey F7 "Jog in G7-Plane" facilitates the axis movement in accordance with (G7) or to the standard coordinate system.
- Cancel program.
  With the softkey "Cancel Program", the progrm execution will be aborted.

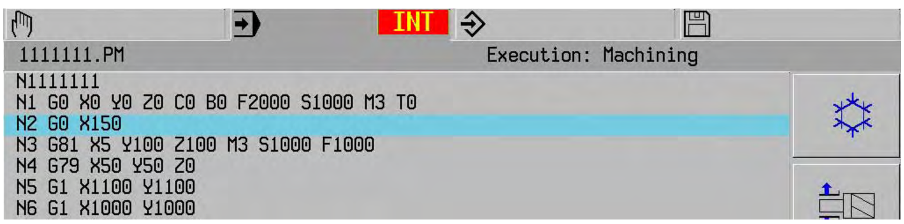
- The procedures with external jog direction keys are machine tool depending. Refer to your machine tool manual!
- Pay attention, that the axes, when repositioned to the interruption point, are moved in a straight line from the actual position. **Collision damage!!**

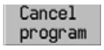
13.2 Erase errors and messages on the screen

Erase errors and messages on the screen. The program is not cancelled.

13.3 Cancel program

Interrupt program execution

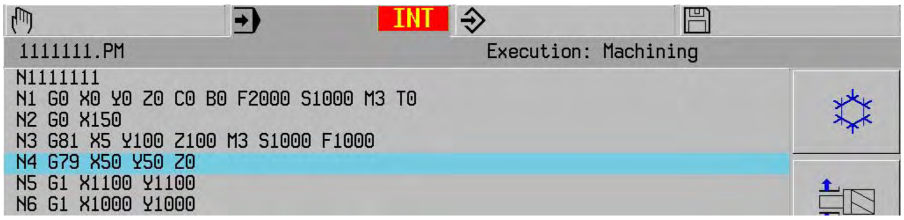


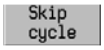


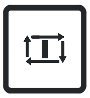
Return to start of program. Only the offset of the current tool, the machining level and the zero offsets remain active. Activated errors and messages will be deleted.

13.4 Interrupt cycle

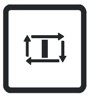
Interrupt the cycle program run.







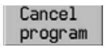
Cancel cycle and movement to starting point.



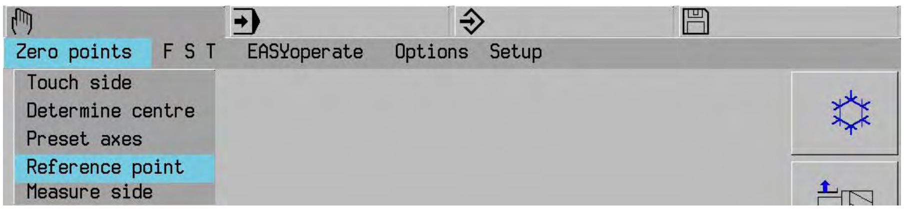
Continue the program from the next block.

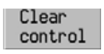
13.5 Reset CNC

Reset all functions (predefined values still apply) and clear all modal parameters.



Cancel program.





13.6 Search block

Find block (e.g. program resumed after program interruption)

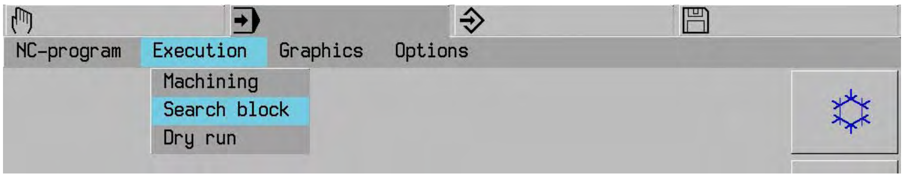
With the function "Search block" the program can be executed from any free block selected in the program. The workpiece machining from program begin up to this block is taken into account and calculated mathematically by the MillPlus. During block search the defined positions of every M-function is calculated. After block search the defined positions of the last defined M-function are actualized and repositioned as safety position.

ACHTUNG !

At start after program search the program continues to run onwards from the searched block in the program. At this block machine tool actions can occur according to the program, which may lead to collision. These machine tool actions are e.g. a tool change (movement to tool change position), rotation of the swivel head and/or machining table, a change or tilting of the active machining plane, workpiece positioning in a linear movement and continue machining etc.

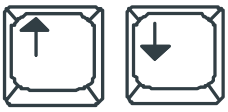
Because of that it is strongly recommended:

- To position manually the (turning)-machining table and (swivel)-head including tool to a safe position before program search. At this safe position the mentioned actions before can be carried out safely without any problems and without danger of possible collisions.
- Better is to prepare the machine tool at this safe position manually, so that after block search the program can be continued fluently.

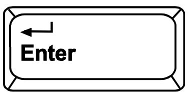


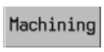
Entry of block number

Or



select block





Return to program

Note

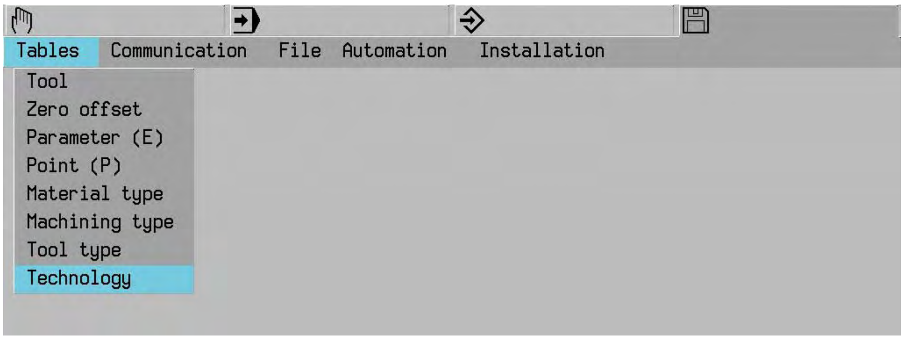
Search for block in repetition part (G14) or subprogram (G22):
- search for program block G14 or G22
- execute G14 or G22 block (single block)
- search for block in repetition part or subprogram

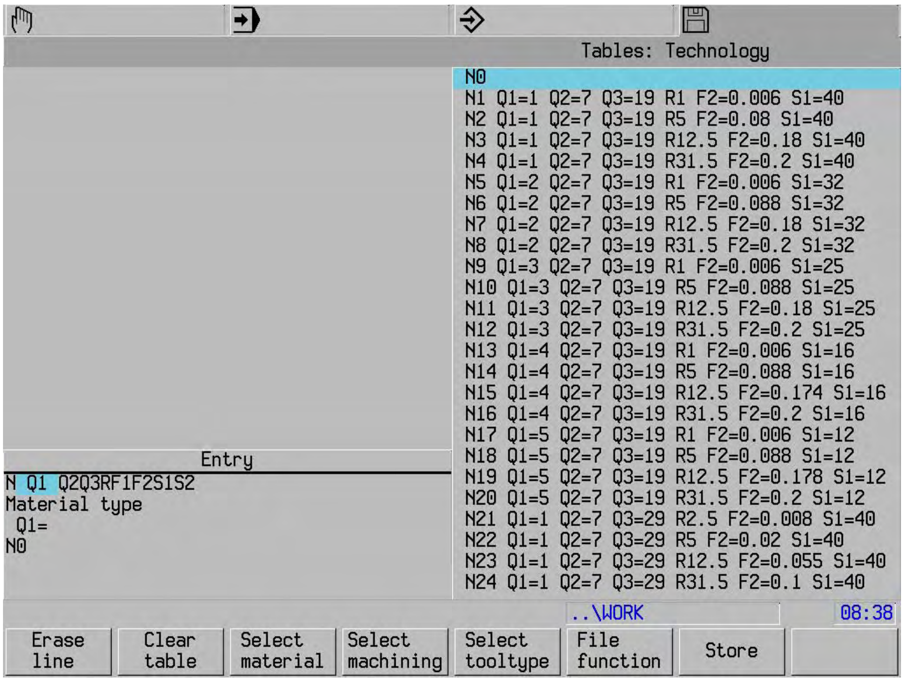
Search in macros:
It is only possible to search for blocks, not for characters.

14. Technology

Establishing the cutting values in a practice-oriented fashion is a most comprehensive task because of the various tools, materials to be cut, coatings, cutting geometries, range of applications, workpiece materials etc.
The feed and speed values suggested by the cutting value calculator may therefore not suit all conditions. Where appropriate, the user should optimise these values.
The cutting values recommended by the tool manufacturer may be useful.

14.1 Technology table





Q1= Material code, which is taken from the file for the material texts.
Q2= Machining process code, which is taken from the file for the machining texts
Q3= Tool type code, which is taken from the file for the tool type texts.
R Tool radius (in mm). If R = 0 is entered, you will be asked to enter the workpiece radius in case the feed rate or spindle speed has to be calculated in a unit of measurement other than that specified in the technology table (the programmed data is specified in rpm, for example, while in the technology table it is given in m/min).

F1 Feed rate in mm/rev. The feed rate for the combination of material, machining process, tool type and tool radius given in the other parameters must be taken from special tables or calculated.

F2 Feed rate per tooth in mm/rev. Refers to tool types with more than one cutting surface. The feed rate for the combination of material, machining process, tool type and tool radius given in the other parameters must be taken from special tables or calculated.

S1 Cutting rate in m/min. This value should be taken from the appropriate documents of the tool manufacturer or empirical values should be used.

S2 Spindle speed in rpm. This value should be taken from the appropriate documents of the tool manufacturer or empirical values should be used.

14.1.1 Tools with more than one radius

In situations where identical tools can have different radii, it is not necessary to enter values in the table for each tool. If the combination of material, machining process and tool type stays the same, only two values need to be entered in the table, one for the smallest tool raadius and one for the largest. The system then uses these two values to interpolate the feed rate and speed and puts forward suggestions for F1 and S1.

14.1.2 Table values for tapping

In some cases, interpolation of the values in the table is not desirable or is not possible, e.g. when tapping. In such situations the feed rate (F1) must be identical to the thread pitch. Interpolation is not possible in such cases.

14.1.3 Relationship between F1 and F2

Both F1 and F2 are used to specify the feed rate. Generally, F1 is used to define the feed rates used in tapping or for drilling using a milling machine. Milling machines usually have more cutting surfaces (teeth). F2 is normally used to specify the feed rate for milling work.

F1 = F2 x number of cutting surfaces

14.1.4 Relationship between S1 and S2

S1 is specified in meters/minute. S2 is displayed in rpm.

S1 = (S2 x 2 x π x R) / 1000

R is the tool radius.

Note

A value is assigned either to parameter F1 or F2 but not to both. The same applies to parameters S1 and S2.

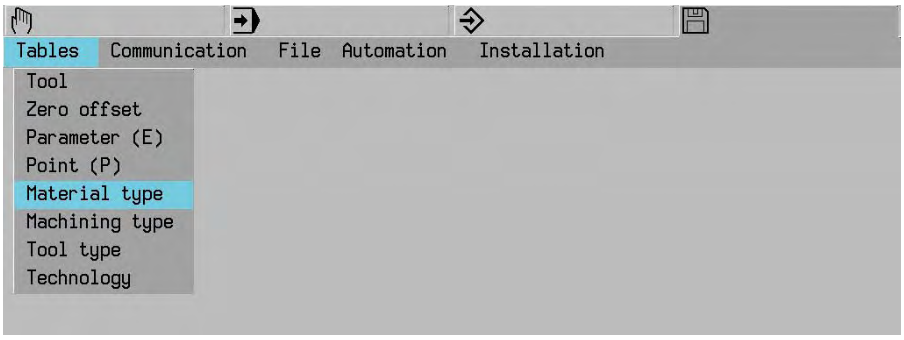
14.2 Storing the technology tables

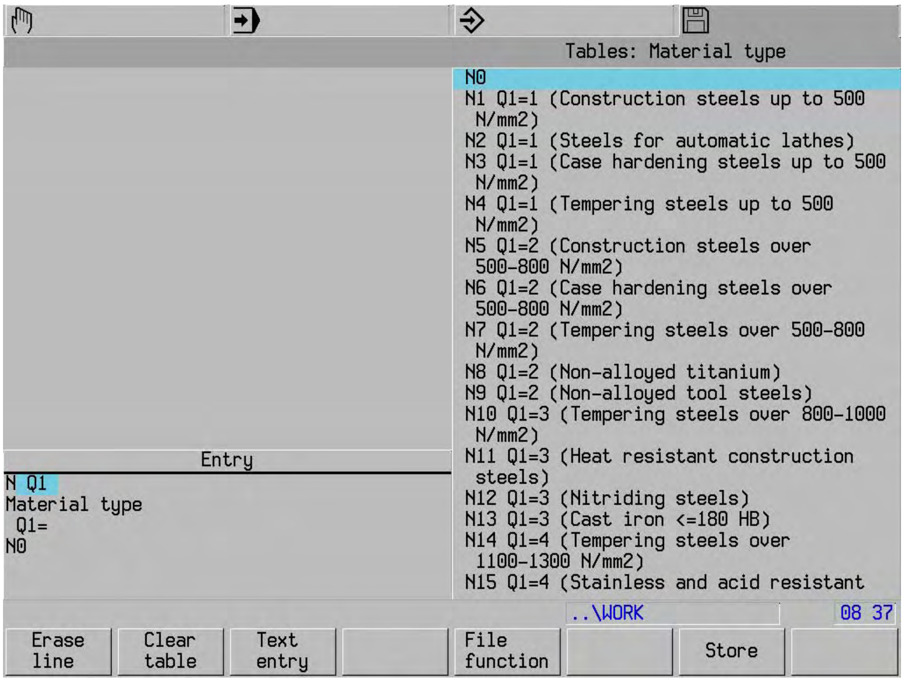
Save on disk — Storing technology tables on hard disk.

Store — Storing technology tables in CNC_RAM.

14.3 Material type

Defining the materials to be machined.





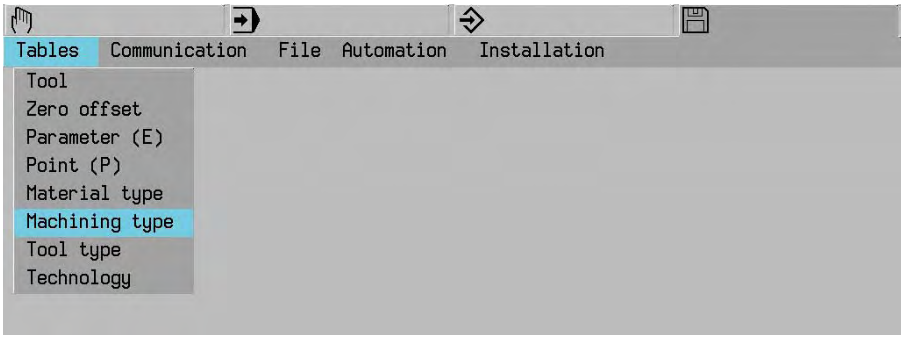
Q1= Material code

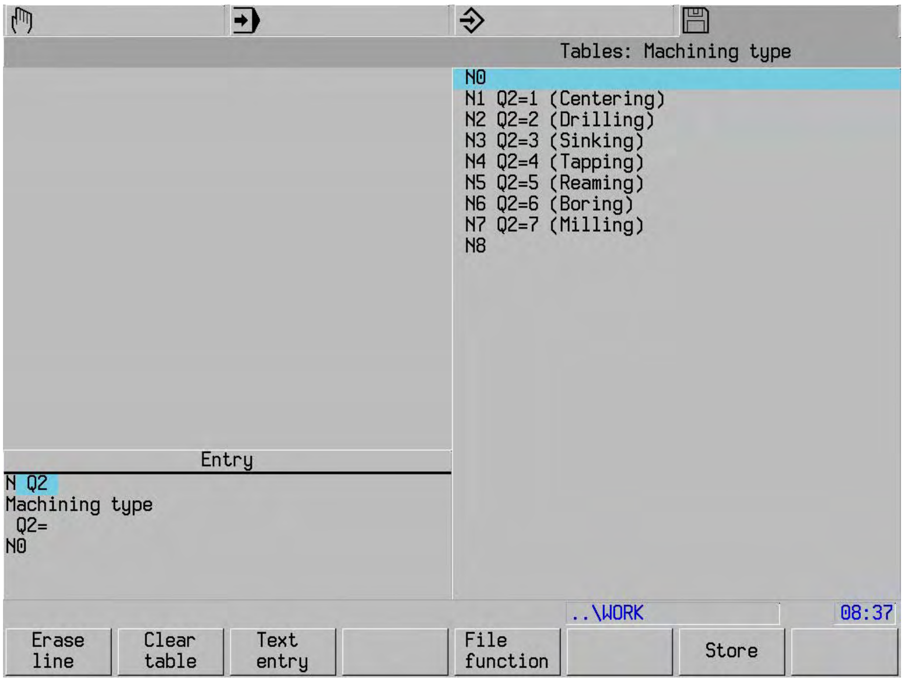
Materials having the same machining properties may be assigned the same material codes.

Text entry — The texts on the material must be in brackets.

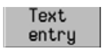
14.4 Machining type

Defining the machining process.





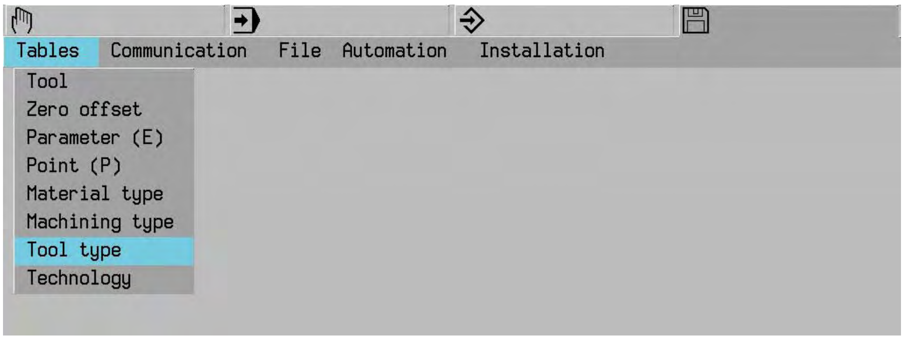
Q2= Machining operation



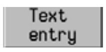
The texts on the material must be in brackets.

14.5 Tool type

Defining the tools.



Q3= Tool type

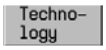


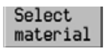
The texts on the material must be in brackets.

14.6 Using the technology

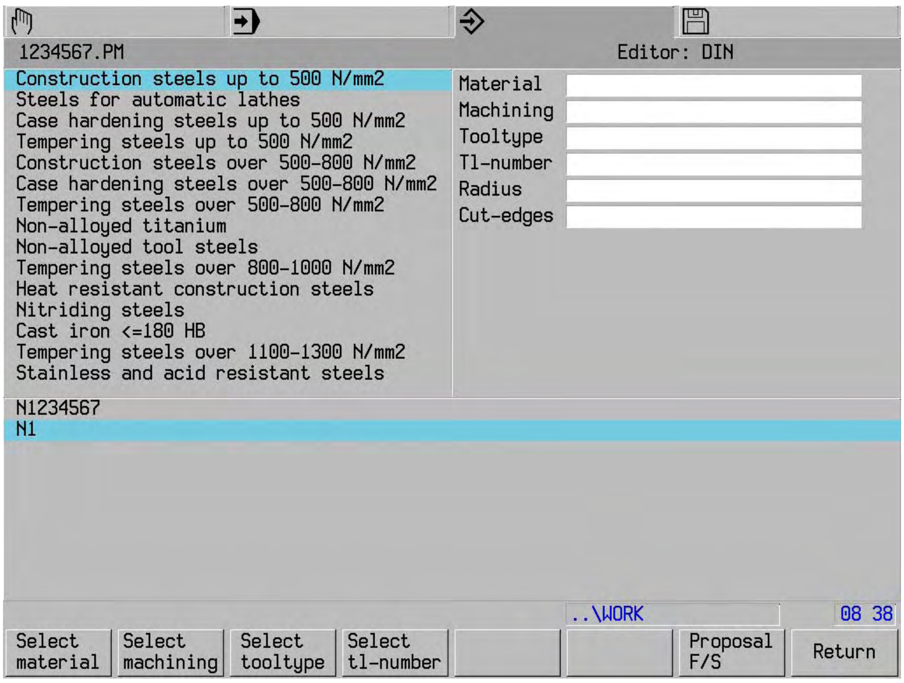
Select program process level and program

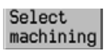
The proposed feed rate and spindle speed can be generated using the following key sequence:



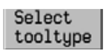


Select the desired material.

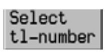




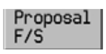
Select the desired machining operation.



Select the type of tool.

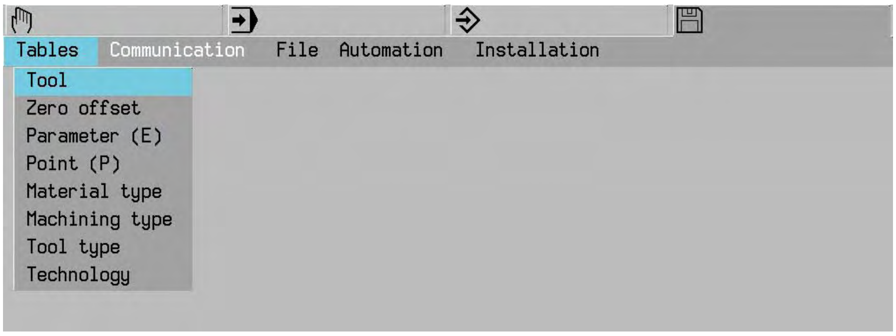


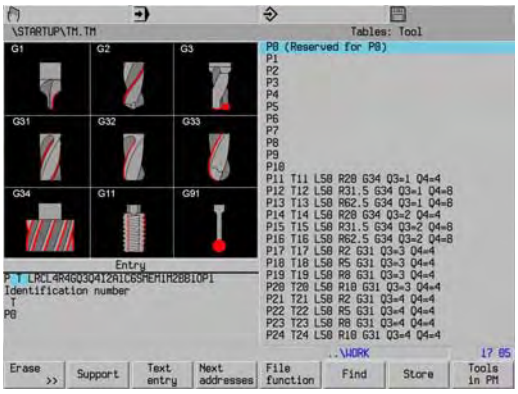
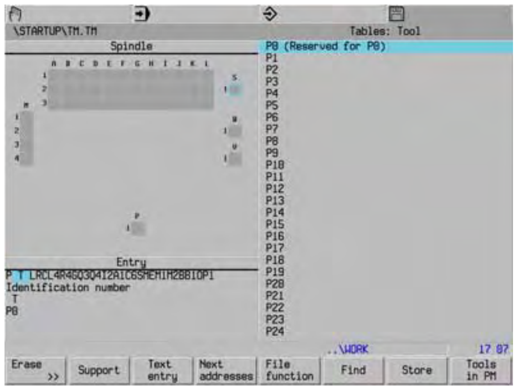
Select the desired tool identification number.



The proposed F, S and T values are transferred to the selected program block

15. Tool

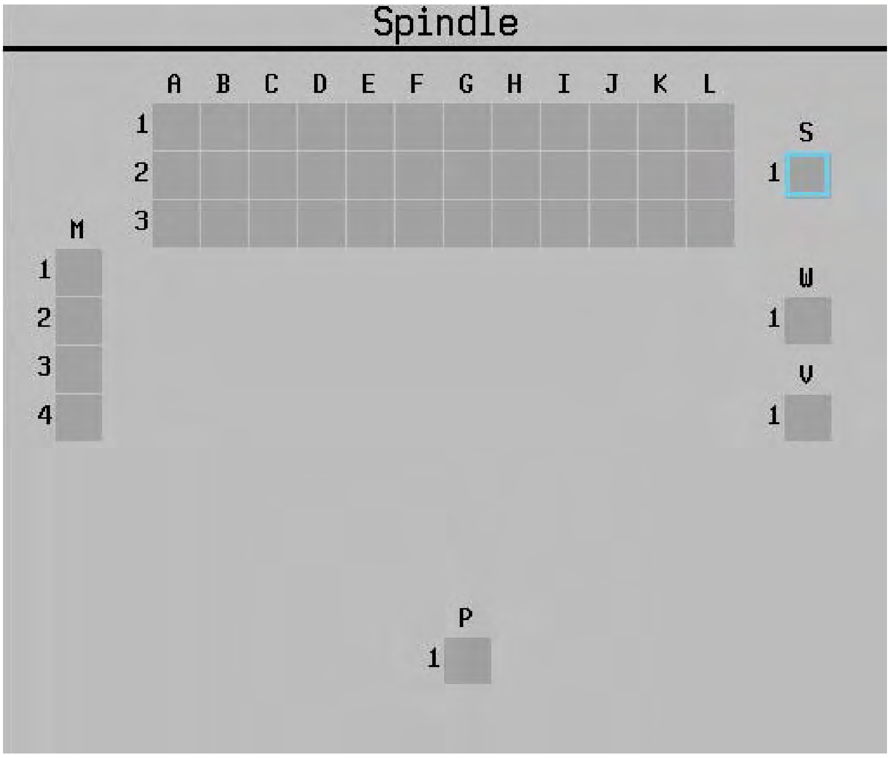


Screen display with tool menu (Standard)	Screen display with tool shelf changer (Optional)
	

Tools in PM	Tool used in current program
Text entry	Plaintext input in table. Enter text in brackets.
File function	File function.
Support	Displays the tool menu and tool pictures for milling and turning tools during edit of or entries in the tool table.
Erase >>	File function.

15.1 Tool shelf changer

At activating the tool table, the support picture of the tool shelf changer is displayed. This support picture is interactive. For example when the tool shelf changer is loaded or unloaded, the actual tool position is displayed by moving the cursor through the tool list. Also you can monitor, during exchanging the tools, the tool position in the various tool shelf sections. During machining the support picture shows clearly were the tools are located in the tool shelf changer.



The picture displayed above shows as an example a support picture of a tool shelf changer including the different sections:

- S: Main spindle
- L: Main storage magazine (3x12 tool places)
- M: Tool transfer station (4 positions)
- W: Changer
- P: Gripper

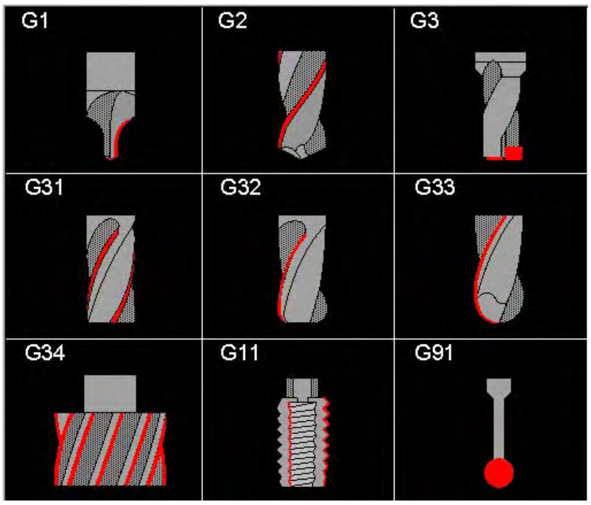
By means of colours the actual status of the tool places is displayed:

- Yellow: Empty, however reserved for another tool or oversized tool
- Green: Released tool is present(Status (E) >= 0)
- Red: Blocked tool os present (Status (E) < 0)
- Blue border: Cursor position

With the soft key „Support“ you switch to the basic tool table display.

15.2 Tool entry/Edit

The tool table screen shows the tool list, tool menu and tool editor for milling and turning tools. The tool menu shows the different tool type images including the corresponding graphic (Gxx)-number. By moving the cursor through the tool list for checking purposes or at tool edit (entry or change), a more detailed tool support picture is displayed.



Tool support

Support

With the soft key „Support“, the different tool menus and support pictures become available.

The tool support function consists of:

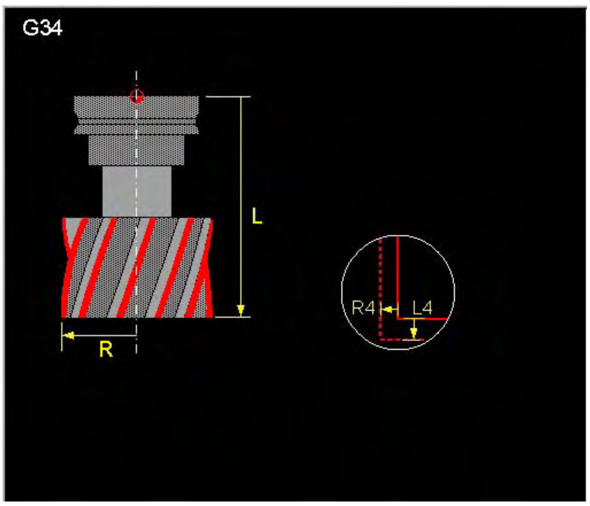
- Tool menus,
  to select the tool type and shape.
- Tool support pictures,
  a support picture available during tool entry and check. These pictures are determined by the G-code in the address selector.

Tool menu pictures
There are two tool menus for milling and turning tools. Initially the milling tool menu is displayed

Next Tool type

With the soft key „Next tool type" the tool menu changes between the tool menus for milling and turning tools. (See picture on the right middle)

Tool support pictures
When the tool support function is activated and the actual block in the tool list or tool editor contains a G-number, the corresponding tool support picture is displayed. (See picture on the right below)



15.3 Tool addresses

P	Magazine pocket. Location of the tool in the tool magazine (if present). Location P0 is reserved for the new tool and cannot be used to store tool parameters. Pocket 1 is indicated by P1, pocket 2 by P2, etc. The actual number of tool pockets in the magazine is saved as a machine constant.
T	Identification number, e.g. T 12345678.00
L	Length
R	Radius
C	Corner radius
L4=	Length allowance
R4=	Radius allowance L and/or R are adjusted when measuring. L4= and/or R4= are set to zero. L and R are not adjusted when checking. Only L4= and/or R4= will be modified.
G	Graphics. Define the tool shape in graphics mode.
Q3	Type. The numbers to identify the tool type are entered in this parameter. Measuring probe Q3=9999: no spindle rotation possible and rapid motion (MC) restricted.
Q4	Number of cutting tips
I2=	Cutting direction 3 clockwise M3 4 anti-clockwise M4
A1	Heel angle (0.1-15 degrees)
S	Size (0=normal, 1=oversize). The maximum tool dimensions and diameter above which a tool is classified as oversize are specified in the supplied machine tool manual. The control keeps a magazine pocket in front of and behind an oversize tool free.
E	Status. The normal setting is E0 (tool enabled, not measured). When the specified tool life is exceeded, E-1 is set automatically. When the tool has been enabled or measured, E1 is set. E-2,-3,-4 tool disabled (new as of V321). The machine tool builder may define other negative status values. Refer to your machine tool manual.
M	Tool life (mins.)
M1	Current tool life (mins.)
M2	Tool life monitoring (0 = off, 1 = on)
B	Break tolerance (0 = MC value) (maximum 255)
B1	Break monitoring (0 = off, 1 = on)

Next addresses — Next address selector.

L1	First additional length
R1	First additional radius
C1	First additional corner radius
L2	Second additional length
R2	Second additional radius
C2	Second additional corner radius
Q5	Break monitoring cycle (0-9999)
L5=	Wear tolerance length (mm)
R5=	Wear tolerance radius (mm) A fault is signalled if the deviation is greater than the values specified here.
L6=	Offset length (mm) Displacement (>=0) of measuring position compared to tool tip.
R6=	Offset radius (mm) Displacement (>=0) of measuring position compared to centre of tool.

15.4 Tool identification

The tool identification number may contain up to eight digits for the tool number plus 2 decades (00) for identifying the tool (original tool or replacement tool). The decade entry may be omitted for the original tool. If a replacement tool is entered for a tool, e.g. T1, this is indicated by the information in the decades (e.g. T1.01, T1.02 etc, i.e. these tools are replacements for T1).

15.5 Calling tool data

The T address and M-function are used to call a tool in the machining program.

Example of calling a tool:	
Tool number T.. [Format 8.2] (255 tools max.)	N.. T1 M..
Original tool (T1-T99999999)	N.. T1
Replacement tool (Tx.01-Tx.99)	N.. T1.01
Activation:	
Automatic tool change	N.. T.. M6
Manual tool change	N.. T.. M66
Activate tool data	N.. T.. M67
First additional tool offset	N.. T.. T2=1 M6/M66/M67
Second additional tool offset	N.. T.. T2=2 M6/M66/M67
Tool life T3=..[0-9999,9min]	N.. T.. T3=x M6/M66
Cutting force control T1=..[1..99]	N.. T.. T1=x M6/M66
Deactivate (T1=0 or T1= not programmed)	N.. T1=0

Modal parameters T, T1=, T2=

Tool preselection in the machining program:
The next tool to be used is preselected by programming the tool number T without a tool change command.

15.6 Input of the tool memory

Options during input of the tool memory. The options are changed by means of MC774:

0 The input addresses are loaded or overwrite the existing addresses.
1 First, the tool memory is cleared. Subsequently, the new addresses are loaded.
2 The existing tools are not changed and, during input, are missed without a fault report.
3 Tool without P overwrites the tool currently existing.

The input addresses are loaded or overwrite the existing addresses.

MC774 = 0	Existing TM	TM to read in	Result
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P1 T1 L1 P2 T2 L2 P3 T3 R3
Without T	P1 T1 L1 P2 T2 L2	P3 R3	Error O/D 61
Without P	P1 T1 L1 P2 T2 L2	T3 R3	P1 T1 L1 P2 T2 L2 P25T3R3 (outside magazine)
T already exists	P1 T1 L1 P2 T2 L2	P3 T1 R1	Error O/D 60
No P T already exists	P1 T1 L1 P2 T2 L2	T1 R1	Error O/D 62

First, the tool memory is cleared. Subsequently the new addresses are loaded.

MC774 = 1	Existing TM	TM to read in	Result
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P3 T3 R3
Without T	P1 T1 L1 P2 T2 L2	P3 R3	Error O/D 61
Without P	P1 T1 L1 P2 T2 L2	T3 R3	P25T3 R3 (outside magazine)
T already exists	P1 T1 L1 P2 T2 L2	P3 T1 R1	P3 T1 R1
No P T already exists	P1 T1 L1 P2 T2 L2	T1 R1	P25T3 R3 (outside magazine)

The existing tools are not changed and, during input, are missed without a fault report.

MC774 = 2	Existing TM	TM to read in	Result
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P1 T1 L1 P2 T2 L2 P3 T3 R3
Without T	P1 T1 L1 P2 T2 L2	P3 R3	Error O/D 61
Without P	P1 T1 L1 P2 T2 L2	T3 R3	P1 T1 L1 P2 T2 L2 P25 T3 R3 (outsode magazine)
T already exists	P1 T1 L1 P2 T2 L2	P3 T1 R1	Error O/D 60
No P T already exists	P1 T1 L1 P2 T2 L2	T1 R1	skip

Tool without P overwrites the currently existing tool.

MC774 = 3	Existing TM	TM to read in	Result
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P1 T1 L1 P2 T2 L2 P3 T3 R3
Without T	P1 T1 L1 P2 T2 L2	P3 R3	Error O/D 61
Without P	P1 T1 L1 P2 T2 L2	T3 R3	P1 T1 L1 P2 T2 L2 P25 T3 R3 (outside magazine)
T already exists	P1 T1 L1 P2 T2 L2	P3 T1 R1	Error O/D 60
No P T already exists	P1 T1 L1 P2 T2 L2	T1 R1	P1 T1 R1 P2 T2 L2

15.7 Tool time monitoring

If the tool time of a tool (M) or the required parking time (T3=..) of a tool is reached, during the next tool change the replacement tool is loaded automatically.

Addresses in the tool memory:
M tool parking time in minutes
M1 residual tool time (only an indication)
M2 tool parking time monitoring (0 = OFF 1 = ON).

The tool time M1 remaining =... can be queried with the function G149 and changed in the tool memory using the function G150.

15.8 Tool breakage monitoring

Machines can be fitted out with a tool breakage monitoring. This function can only be programmed with the help of macros.

The following addresses are used by the tool memory:
B breakage tolerance in mm
R6= radius position for breakage control

In the event of the breakage tolerance being exceeded, tool status E=4 is set and a fault is output in addition.
Also, if at the start of the cycle the tool status is E=1, the breakage control is implemented.
Default value for tolerance is input in MC33.
The breakage monitoring is activated by means of MC32.

The tool breakage monitoring is a machine dependent function. Please consult your user handbook.

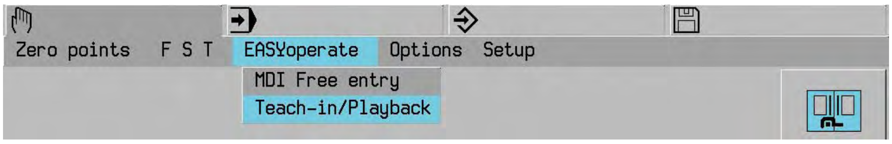
Note

If an original tool is blocked, a replacement tool is automatically loaded in its place (if available).
See G604

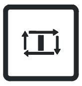
15.9 Manual change of tool (Example)

Change of tool is a machine dependent function. Please consult your user handbook.

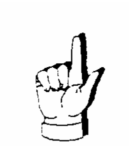
Call-up tool change:



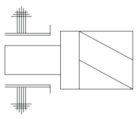
T... M66
Report: int T..



The working area door is unlocked.
Open the working area door..



Please look up the notes on general security



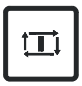
Press "Select tool holder”

Take hold of the tool and use the rotating key or the foot lever "Release tool holder" to halt the machine. The tool holder is released.

Take the tool out of the tool holder.
Insert a new tool.

Release the rotating key or the foot lever and support the tool holder while inserting the tool.

Close the working area doors.



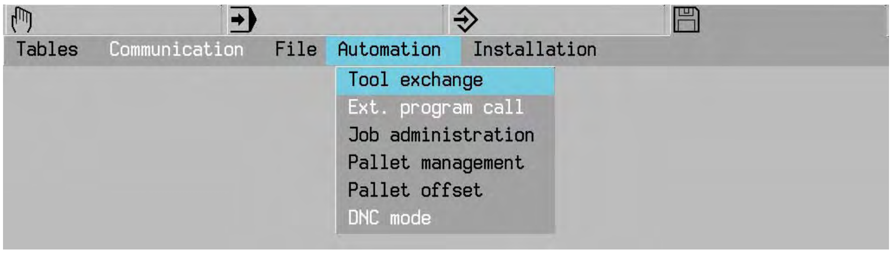
The working area doors are locked

15.10 Tool management

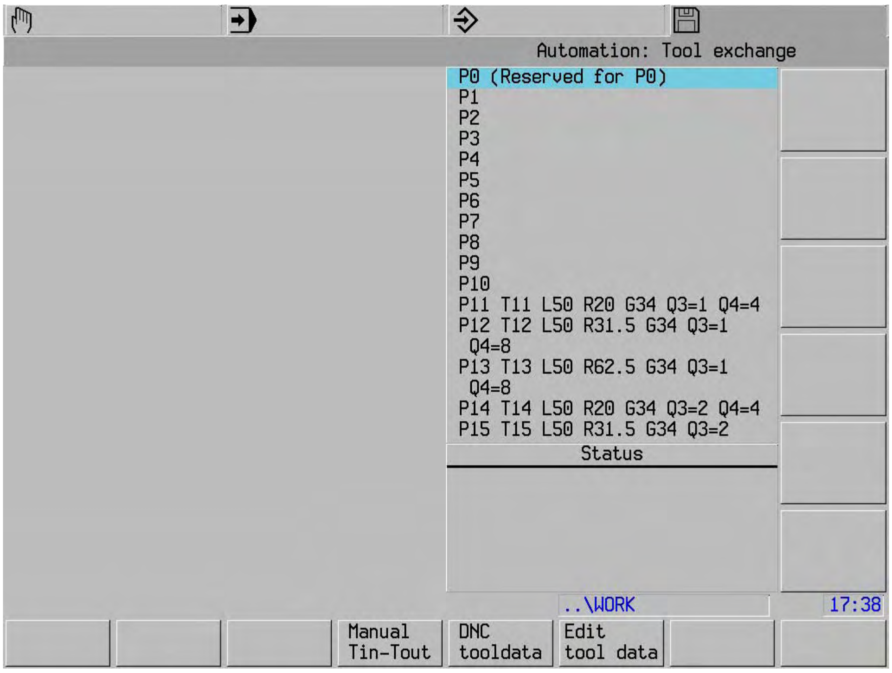
Tool management allows you to input or remove tools from the tool magazine while simultaneously updating the tool data in tool memory.

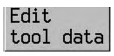
15.10.1 Tool correction

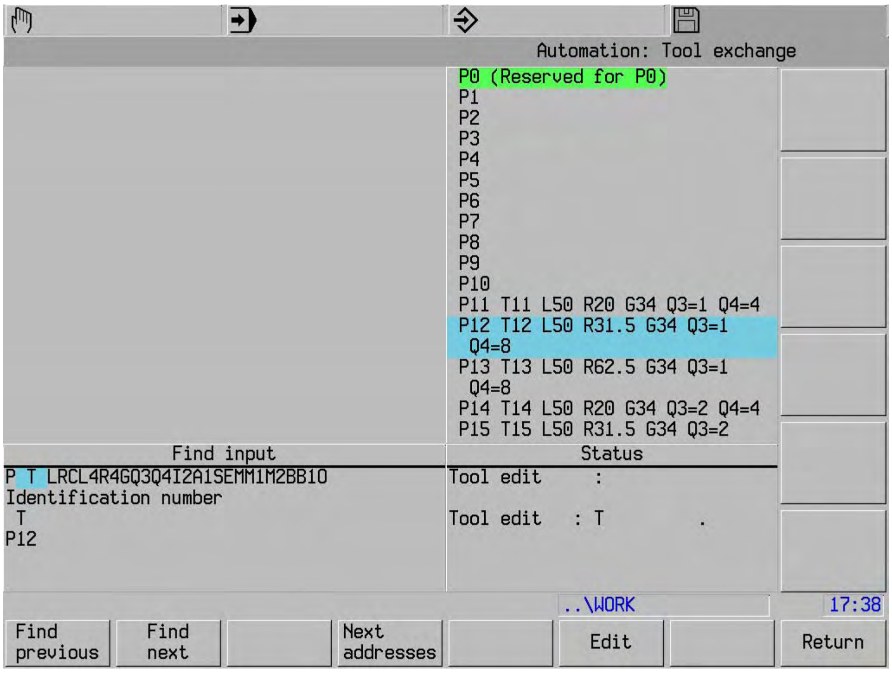
During machining, all the tool data can be edited, apart from the spindle tool.



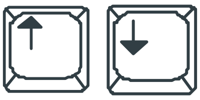
.







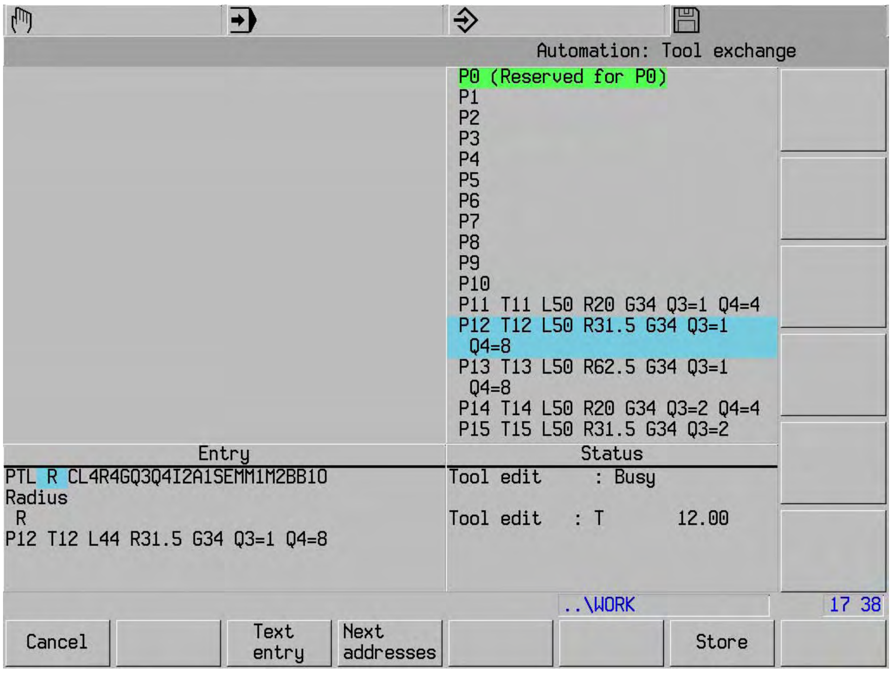
Select block



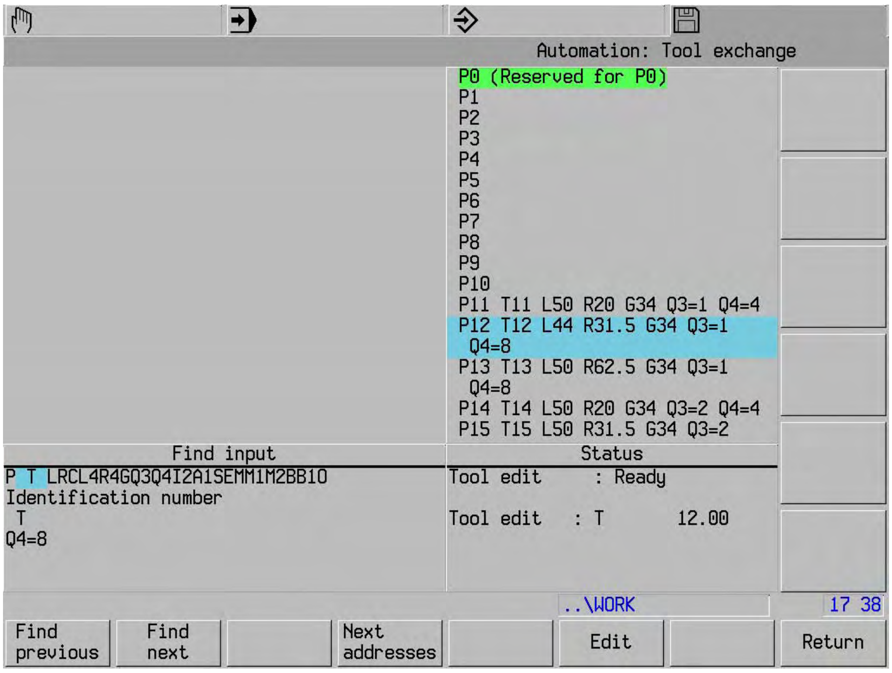
or

Input P12

Input L44


Automation: Tool exchange
P0 (Reserved for P0)
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11 T11 L50 R20 G34 Q3=1 Q4=4
P12 T12 L50 R31.5 G34 Q3=1 Q4=8
P13 T13 L50 R62.5 G34 Q3=1 Q4=8
P14 T14 L50 R20 G34 Q3=2 Q4=4
P15 T15 L50 R31.5 G34 Q3=2
Entry
PTL R CL4R4GQ3Q4I2A1SEMM1M2BB10
Radius
R
P12 T12 L44 R31.5 G34 Q3=1 Q4=8
Status
Tool edit : Busy
Tool edit : T 12.00
..\WORK
17 38
Cancel
Text entry
Next addresses
Store


Store


Automation: Tool exchange
P0 (Reserved for P0)
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11 T11 L50 R20 G34 Q3=1 Q4=4
P12 T12 L44 R31.5 G34 Q3=1 Q4=8
P13 T13 L50 R62.5 G34 Q3=1 Q4=8
P14 T14 L50 R20 G34 Q3=2 Q4=4
P15 T15 L50 R31.5 G34 Q3=2
Find input
P T LRCL4R4GQ3Q4I2A1SEMM1M2BB10
Identification number
T
Q4=8
Status
Tool edit : Ready
Tool edit : T 12.00
..\WORK
17 38
Find previous
Find next
Next addresses
Edit
Return

15.10.2 To take a tool out of the tool magazine (example)

Manual Tin-Tout

	Exchange tool	Insert tool	Remove tool		Automatic position		Return

Remove tool

Cancel	Select tool-out		Start			Find	Return

Select tool-out — Select the tool or enter the tool number..

Cancel			Start			Find	

Start — The tool magazine is in position.

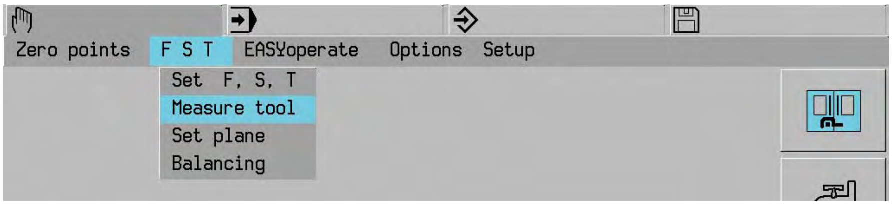
```
Status
Tool removal  : Busy
Positioned at       : P 14

Tool remove : T      14.00
```

Cancel				Tool removed		Find	

Remove tool — Confirmation that the tool has been removed.

```
Status
Tool removal  : Ready
```

15.11 Manual measuring



The machine and MillPlus must be prepared by the machine manufacturer for the TT120/TT130 measuring key system or the laser measurement system. Please consult your user handbook.

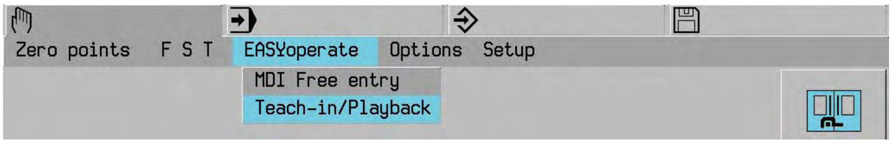
Mit dem TT120/TT130 oder dem Lasermeßsystem und den Werkzeug-Vermessungszyklen der MillPlus vermessen Sie Werkzeuge automatisch: Die Korrekturwerte für Länge und Radius werden von der MillPlus im Werkzeugspeicher abgelegt und beim nächsten Werkzeug-Aufruf verrechnet.

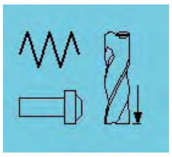
The menu and the corresponding machine settings are updated using the following machine settings:

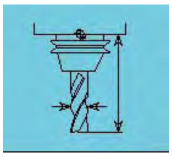
MC261 >0: measurement cycle functions
MC254 >0: tool measurement
MC840 =1: present measuring key
MC854 =1: tool measuring equipment type (0=none, 1=laser, 2=TT120/TT130)

15.12 Tool measurement using the laser measurement system

You can measure tools automatically using the laser measurement system and the MillPlus tool measurement cycle. The correction values for length and radius are placed in the tool memory.



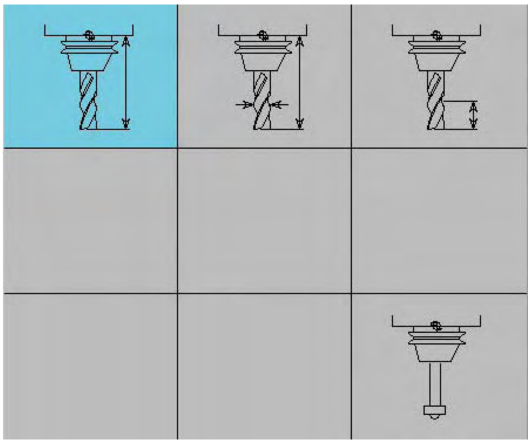




After selecting "Measure tool" the following menu screen appears (MC254=1):

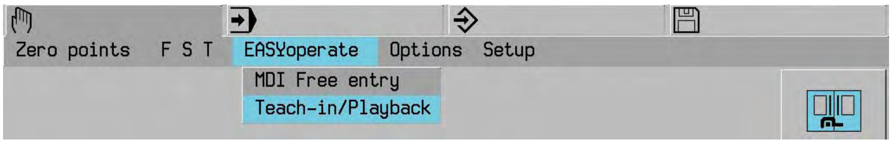
The following cycles are available:

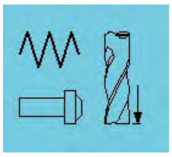
Measurement of the tool length of concentric tools	G953
Measurement of tool length and radius of eccentric tools	G954
Individual cutting controlSF	G955
Individual cutting controlKF	G957
Measurement of tool length, radius and tool nose raddius	G985
Calibration of the laser measurement system	G985

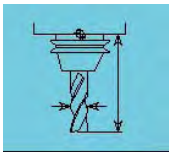


15.13 Tool measurement using the TT120/TT130

Using the TT130 and the MillPlus tool measurement cycles you can measure tools automatically. The correction values for length and radius are placed in the tool memory.



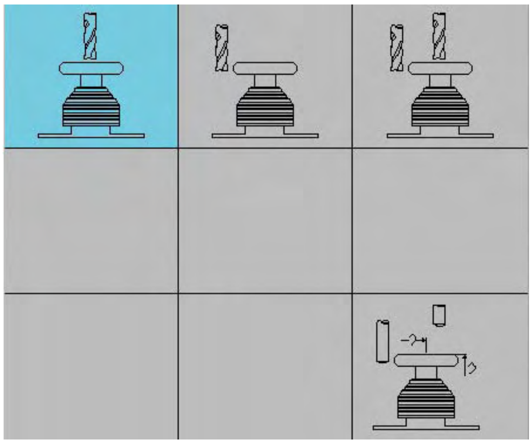




After selecting "Measure tool", the following menu screen appears (MC854=2):

The following cycles are available

Tool length measurement	G606
Tool radius measurement	G607
Tool length and radius measurement	G608
Calibration of the TT120/TT130	G605



Tool length and radius
Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.
Measurement results
During the initial measurement, MillPlus overwrites the tool radius (R10 with R10.012) and the tool length (L100 with L99.456) in the tool memory and sets the oversizes R4 and L4 = 0.
Checking the tool
In the event that you check a tool, the measured tool data is compared with the tool data extracted from the tool memory. MillPlus calculates the mathematically correct variances and enters these as oversizes (R4=0.015 and L4=0.06) in the tool memory.
Radial axis approach direction
The approach direction depends on the position of the masuring key system. It is automatically keyed from the direction in which the greatest travel range is available.

15.14 Update machine settings

MillPlus employs the approach rate from MC394 for measurement with the spindle stationary.

MillPlus automatically calculates the spindle speed in rpm and the approach rate during measurement with the tool rotating. The spindle speed in rpm is calculated in the following way:

$$n = \frac{MC399}{r \cdot 0.0063}$$

where:

n	= turning speed in revolutions/min
MC399	= maximum permissible turning speed in metres per minute [m/min]
R	= active tool radius [mm]

The approach rate is calculated from:

V	= measurement tolerance • n

where:

V	= approach rate [mm/min]
measurement tolerance	= measurement tolerance [mm], dependent on MC391
N	= speed in revolution per minute [1/min]

where:
You enter in MC391 the calculation for the approach rate:

MC391=0:
the measurement tolerance remains constant - independent of the tool radius. However, with very large tools the approach rate is reduced to zero. The lower you choose the maximum turning speed (MC399) and the permissible tolerance (MC392), this effect is produced in order to make it perceptible earlier.

MC391=1:
The measurement tolerance is altered with increasing tool radius. This guarantees an approach rate which is still appropriate for large radius tools. MillPlus alters the measurement tolerance in accordance with the following table:

Tool radius	Measurement tolerance
up to 30 mm	MC392
30 to 60 mm	2 • MC392
60 to 90 mm	3 • MC392
90 to l20 mm	4 • MC392

MC391=2:
The approach rate remains constant; however, the measurement error increases linearly as the tool radius becomes larger:

$$\text{measurement tolerance} = \frac{r \cdot MC392}{5mm}$$

where:

r	= tool radius [mm]
MC392	= maximum permissible measurement error

Synopsis of machine settings:
The TT120/TT130 function can be activated by means of MC854. Following a reboot of the CNC the following machine settings are then available.

MC NUMBER	FUNCTION	INPUT
MC391	Calculating scanning feed.	0 Calculating scanning feed with constant tolerance. 1 Calculating scanning feed with variable tolerance. 2 Scanning feed calculation
MC392	Maximum permissible measuring error during tool gauging with rotating tool	2 ? 1000 ?m
MC394	Scanning feed for tool gauging with non-rotating tool	10 ? 3000 mm/min
MC395	Distance between the lower edge of the tool and the upper edge of the stylus during tool radius gauging.	1 ? 100000 ?m
MC396	Diameter or edge length of the TT120/TT130 stylus.	1 - 100000 ?m
MC397	Safety zone around the TT120/TT130 stylus for pre-positioning.	1 ? 10000 ?m
MC398	Rapid motion in the scanning cycle for the TT120/TT130..	10 ? 10000 mm/min
MC399	Maximum permissible rotational speed at the tool tip.	1 ? 120 m/min
MC854	Type of tool gauging	0=none,1=Laser,2=TT120/TT130
MC350 MC352 MC354	Coordinates of the mid-point of the TT120/TT130 stylus relative to the machine reference point.	-max - +max ?m

15.15 TT120/TT130 measurement cycles for automatic operation

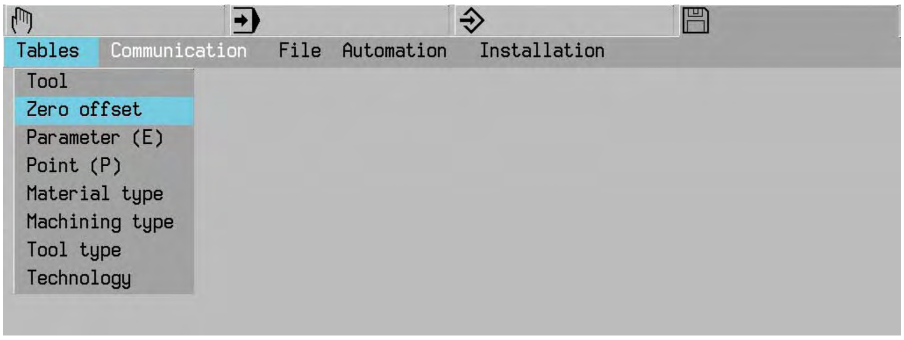
15.15.1 Example

```
N66666
N1 G54 I1
N100 T1 M6 ... (mill D50)
... \
...   milling operation
... /
N191 G609 (measurement of length, radius wear)
N200  T2 M6 ... (drill D4)
... \
...   drill operation
... /
N291 G607 (measurement of length, breakage monitoring)
N300 M30
```

Tool memory at program start-up.
Tools are measured beforehand using the measurement cycles.
The mill is blocked (E-1) if the parking time has elapsed or if the wear limit has been exceeded.
The drill is blocked (E-1) if the parking time has elapsed. During a breakage the drill is blocked (E-4) and a program stoppage with fault executed.

50mm diameter milling with replacement tool:
P.. T1.01 L102.023 R24.978 L4=0 R4=0 E1 M15 M2=1
P.. T1.02 L102.167 R24.986 L4=0 R4=0 E1 M15 M2=1
4mm diameter drill with replacement tool:
P.. T2.01 L85.467 L4=0 E1 B1 M15 M2=1 R6=0
P.. T2.02 L85.246 L4=0 E1 B1 M15 M2=1 R6=0

16. Tables



16.1 Zero offset

Display and entry

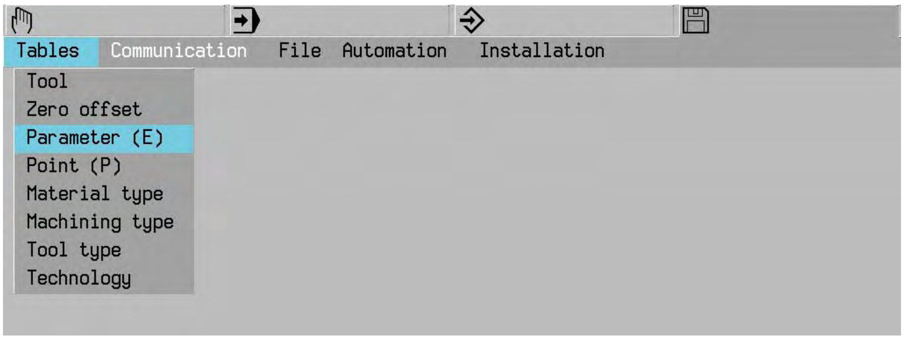
Note

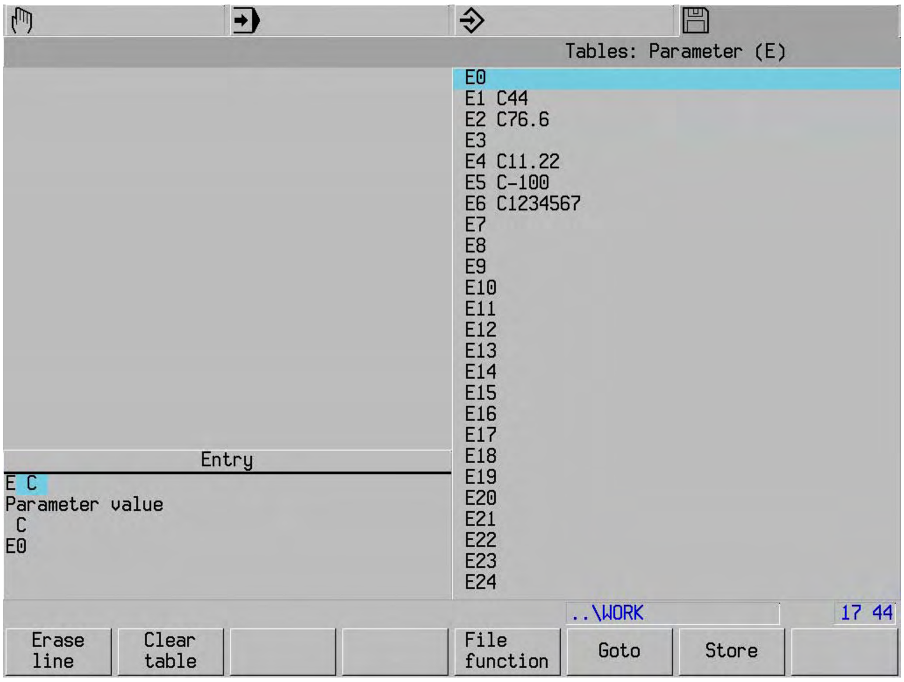
mc84>0
Zero offset G54 I1-I99
Storage name ZE.ZE

mc84=0
Zero offset G51-G59
Storage name ZO.ZO

16.2 Parameter(E)

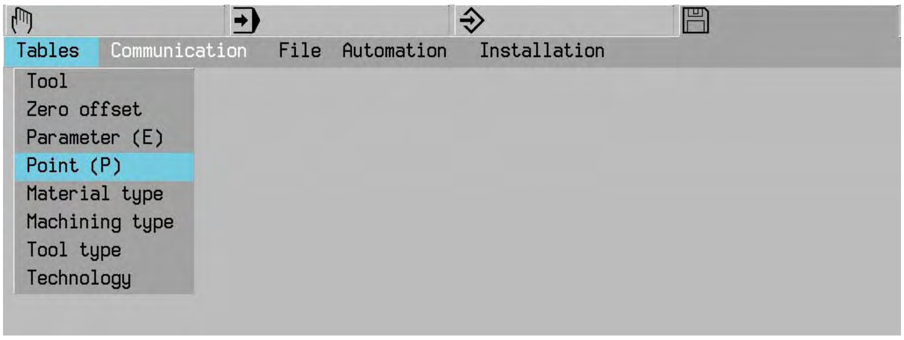
Display and entry of the E parameters

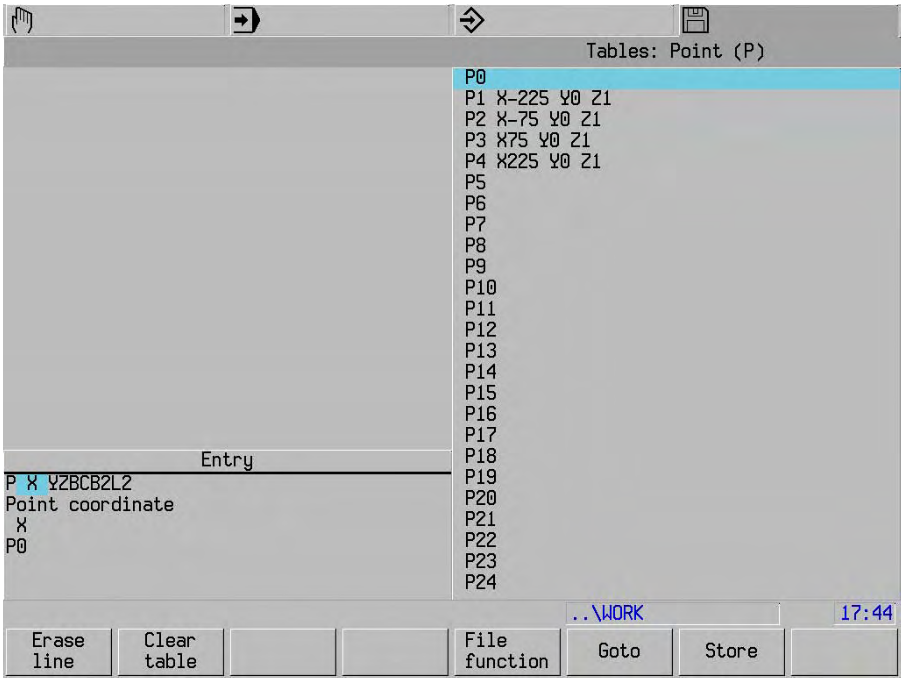




16.3 Point (P)

Display and entry of the point definitions

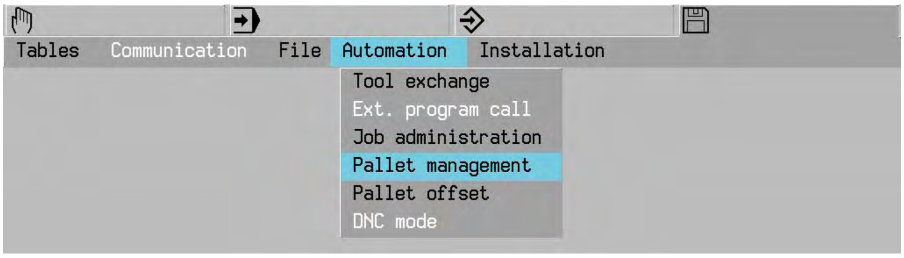


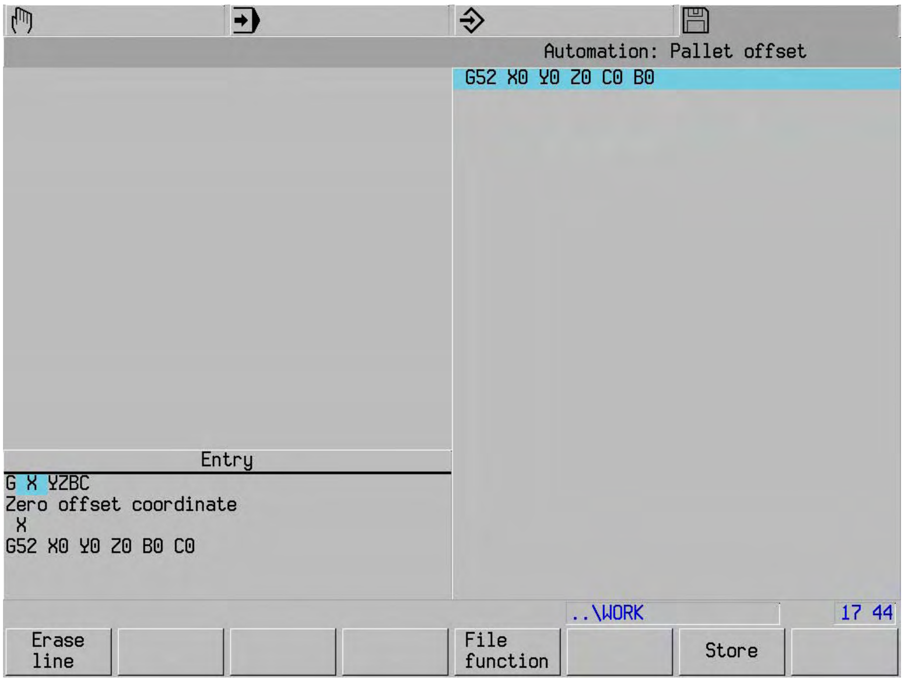


16.4 Pallet zero point

Only with activated ZE.ZE-memory: (see zero offset).

Storing the pallet zero point.

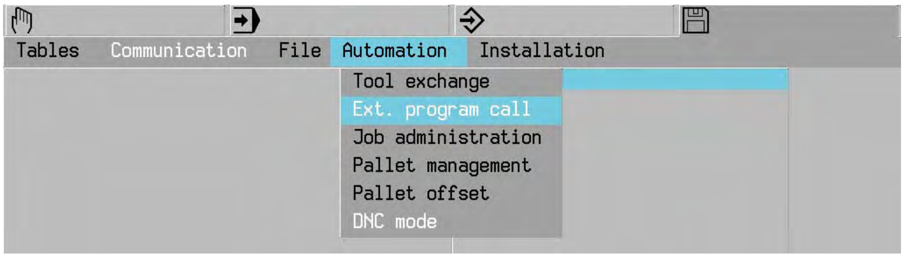




Note:

See Technical Handbook for more information.

17. Automation

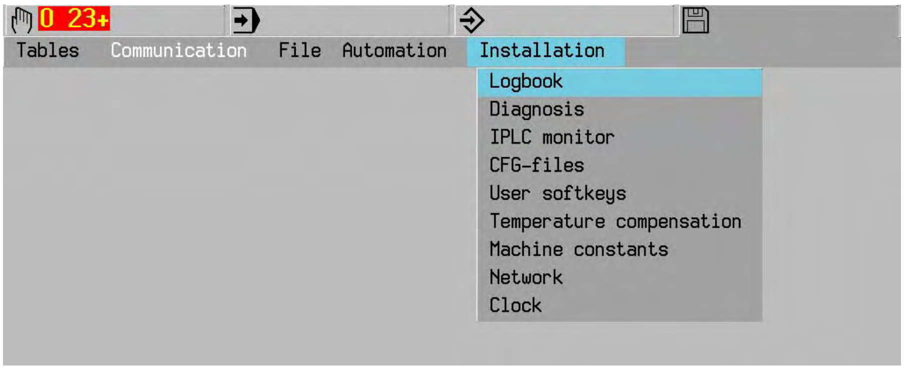


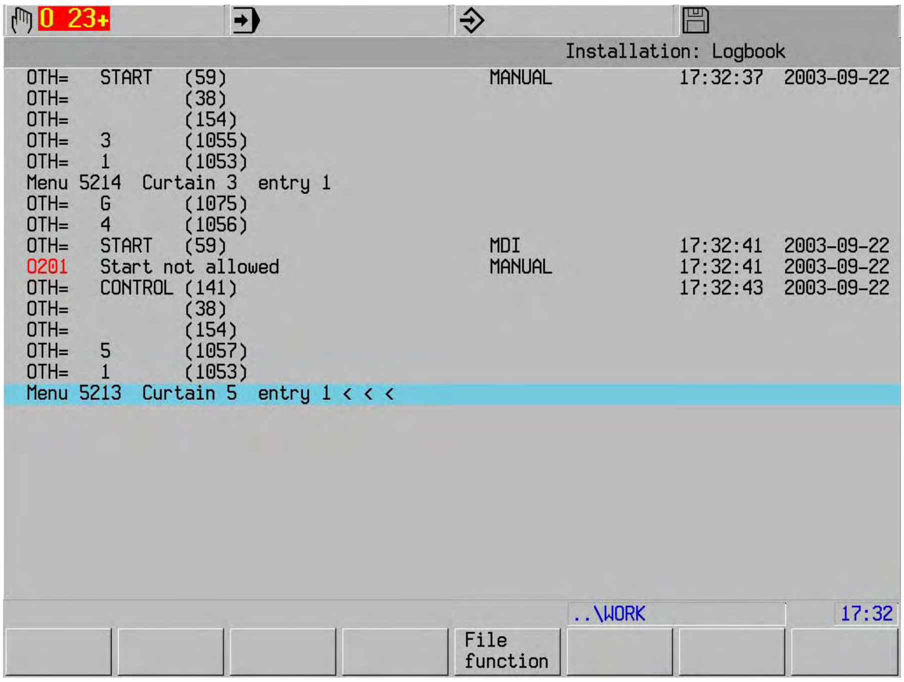
Refer to the documentation provided by the machine builder for information regarding the Ext. program call, job administration, palette management and DNC mode functions.

18. Installation

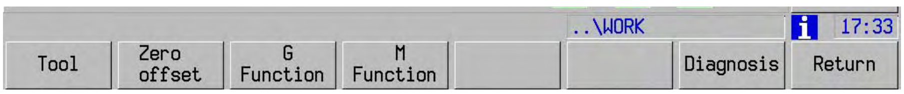
18.1 Logbook

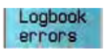
The most recent inputs from the keyboard are stored in the logbook.





18.1.1 Error log

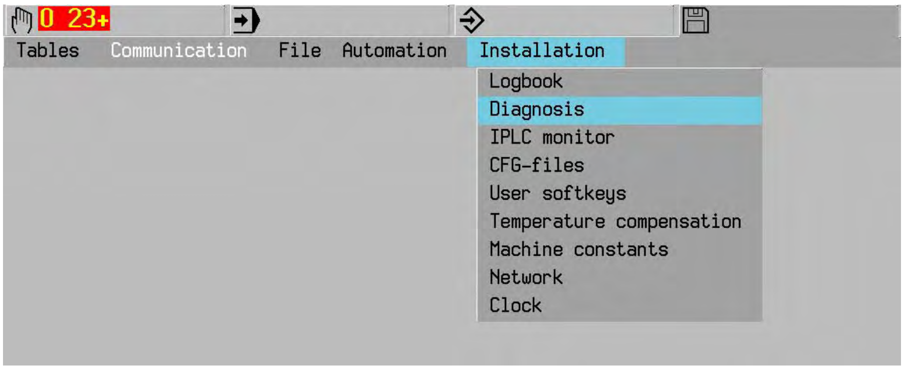


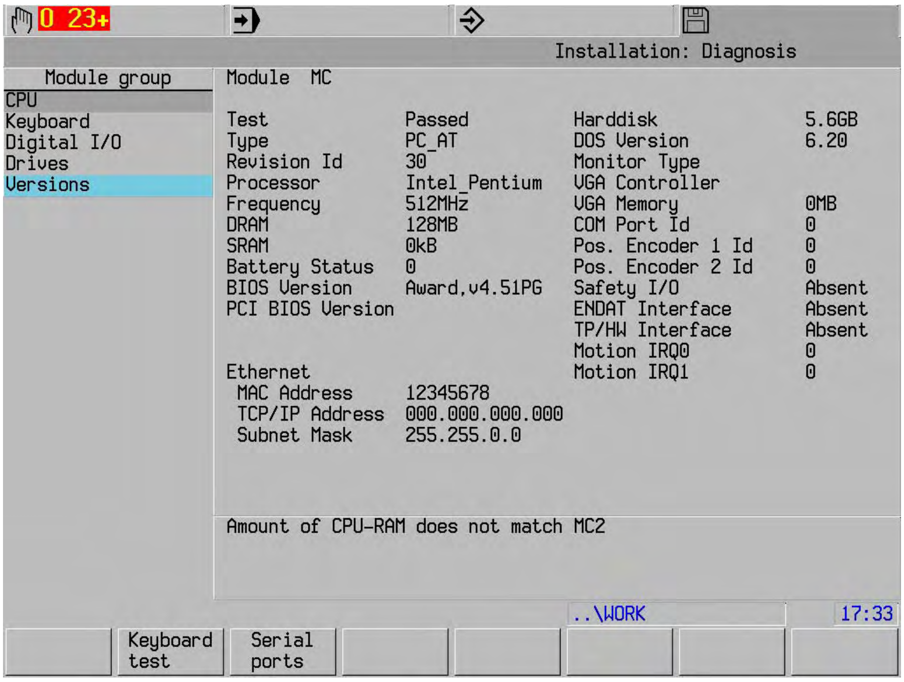


Display of the last error messages (only in Manual and Automatic operating modes).

18.2 Diagnostics

System information may be displayed in Diagnosics.



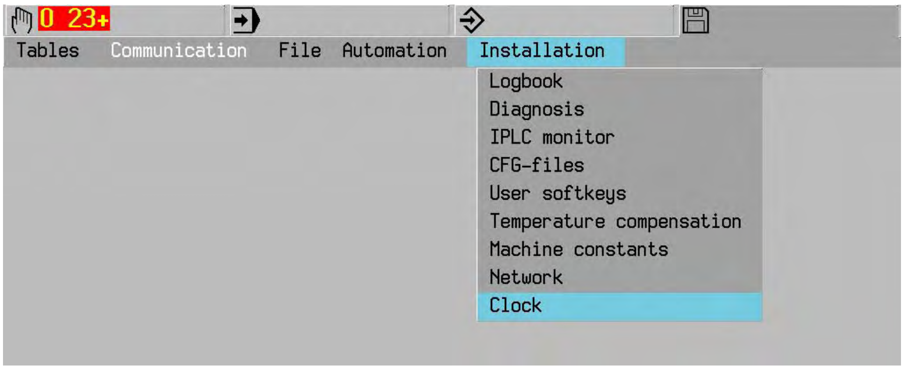


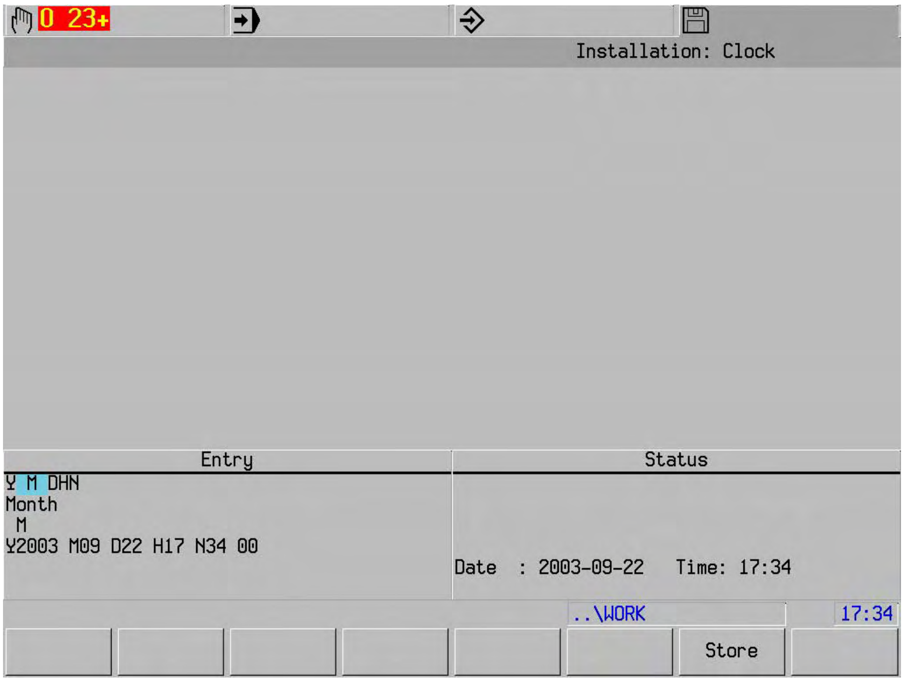
18.2.1 Remote diagnosis

Setting the CNC up for remote diagnosis. The display changes to black/white.

18.3 Clock

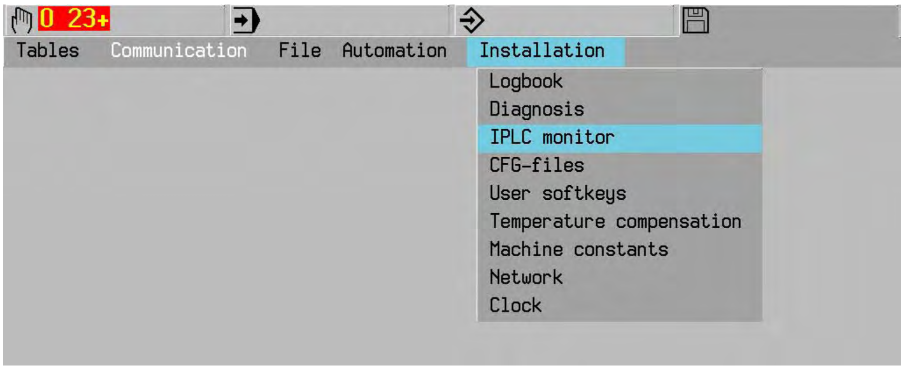
Entering and storing the real time.

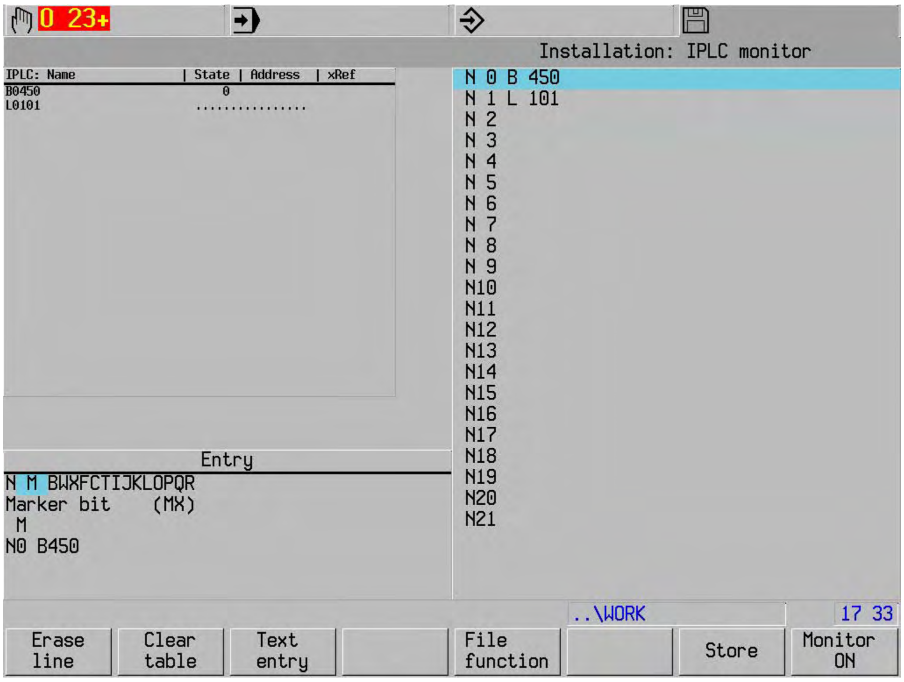




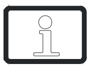
18.4 IPLC monitor

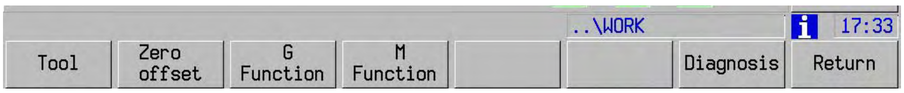
This function only to be used by maintenance/customer service personnel.





18.4.1 I/O layout

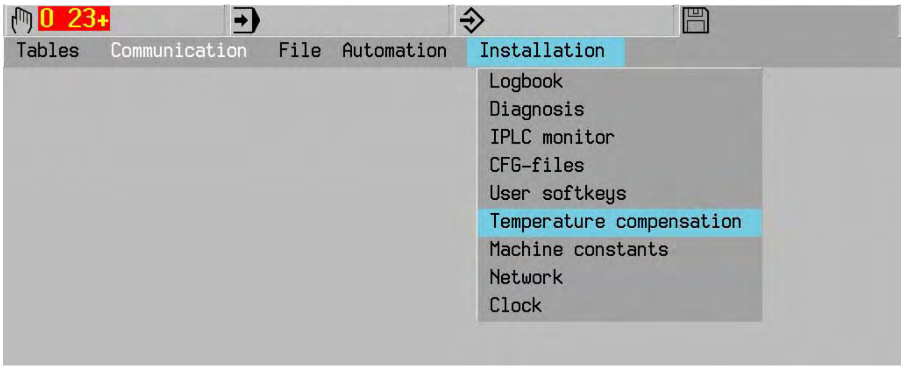




Status indication of I / O layout (only in Manual and Automatic operating modes)

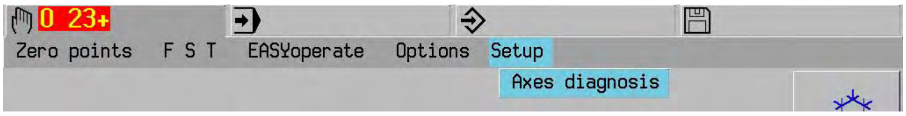
18.5 Temperature compensation

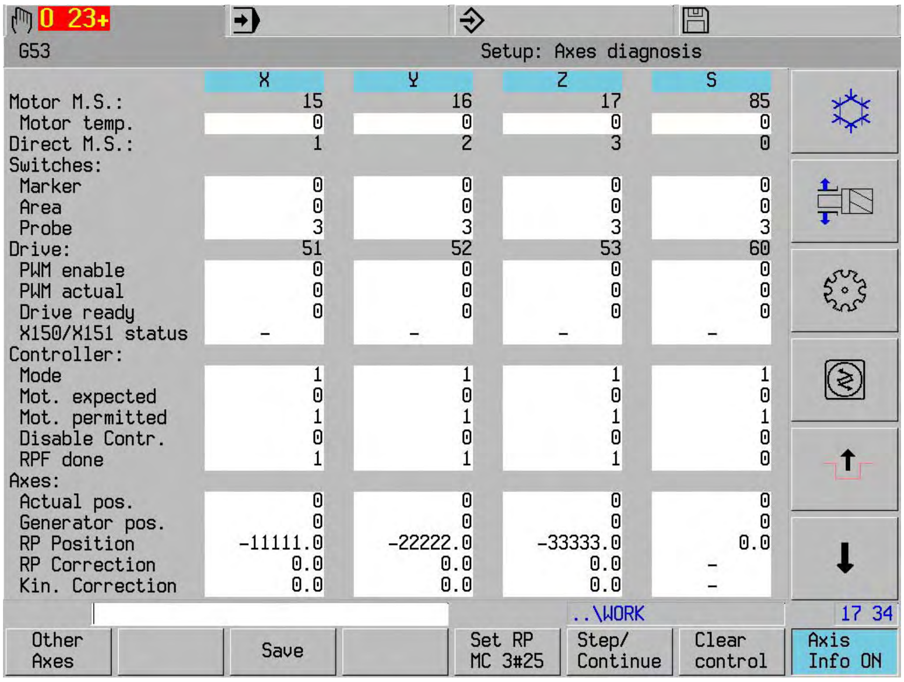
This function only to be used by maintenance/customer service personnel.



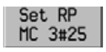
18.6 Axes diagnose

This function only to be used by maintenance/customer service personnel.





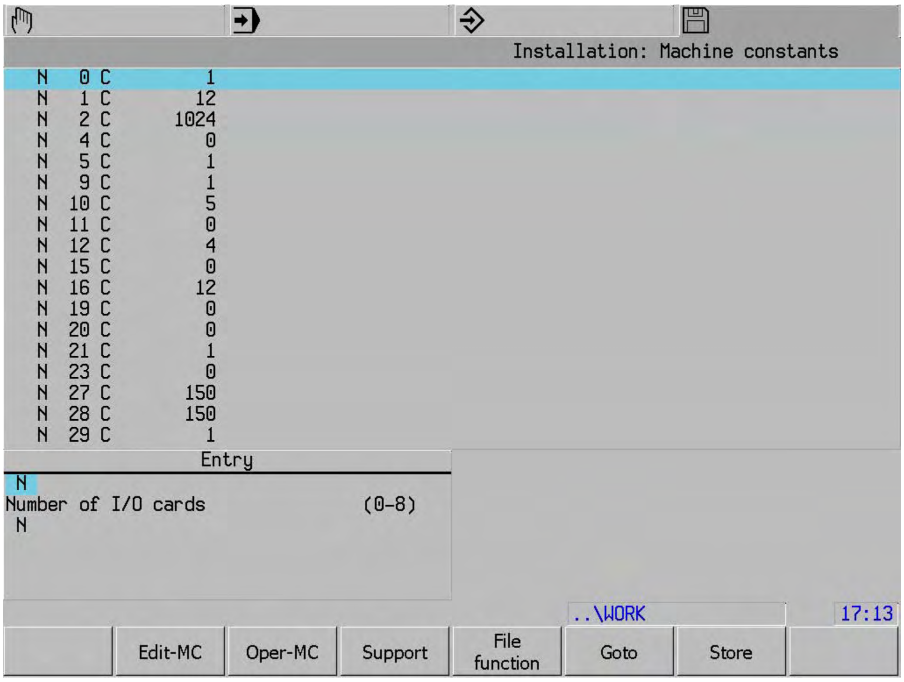
Note



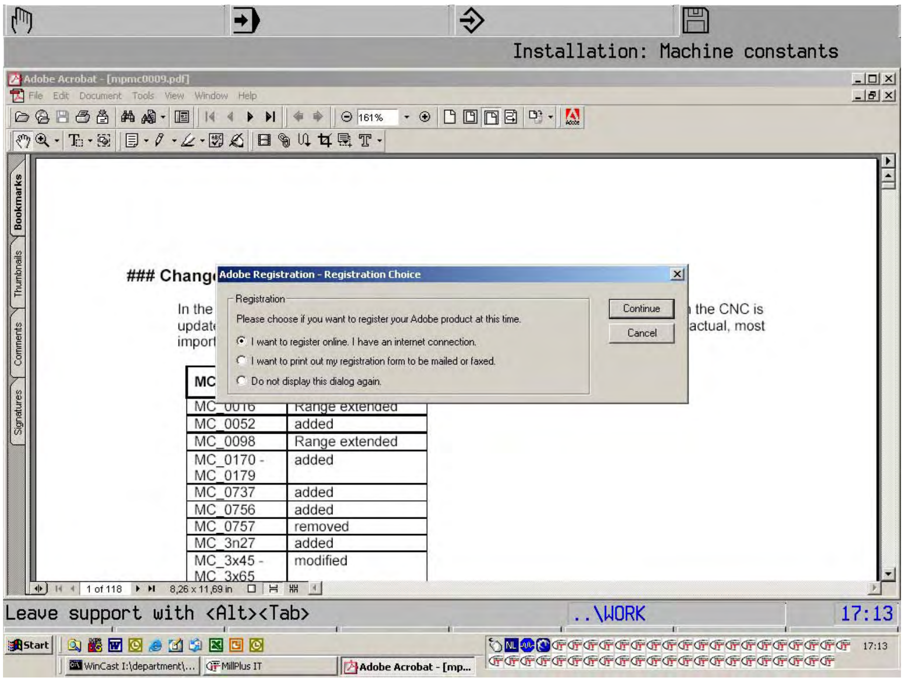
Displayed only when diagnosis switch on.

18.7 Machine constants on-line help (Only DP-system)

With this function the operator has access to on-line information of machine parameters via a PDF-file.



By pressing the soft key, the PDF-file with the MC-description is opened. At repressing the soft key, the focus is back to the CNC-screen.



19. EASYoperate

In EASYoperate, cycles and user-defined input are executed directly on the machine. A graphical menu allows cycles to be selected and offers assistance with input. These entries can be saved in a list (apart from workpiece measurement). If the saved cycles and the user-defined input have the required operational sequence, you can use the repeat start to replay this sequence.
Before machining can begin, F, S and T must be activated and the spindle switched on (not for graphics).

EASYoperate in manual mode:

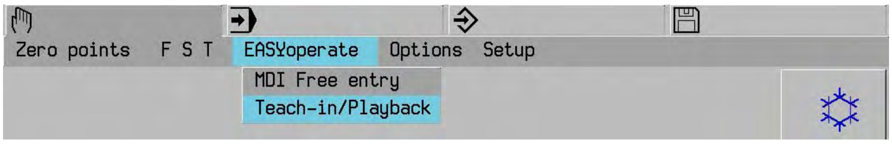
- When setting up complex machines, certain actions can be executed in a direct and simple way. These include measuring and setting up the workpiece.
- Ease of operation is required for executing the simple machining processes that often precede a machining program. Machining processes include roughing and finishing the surface, making the seating or the holes, etc.
- Replaying saved cycle inputs (teach-in/play-back).

Note:

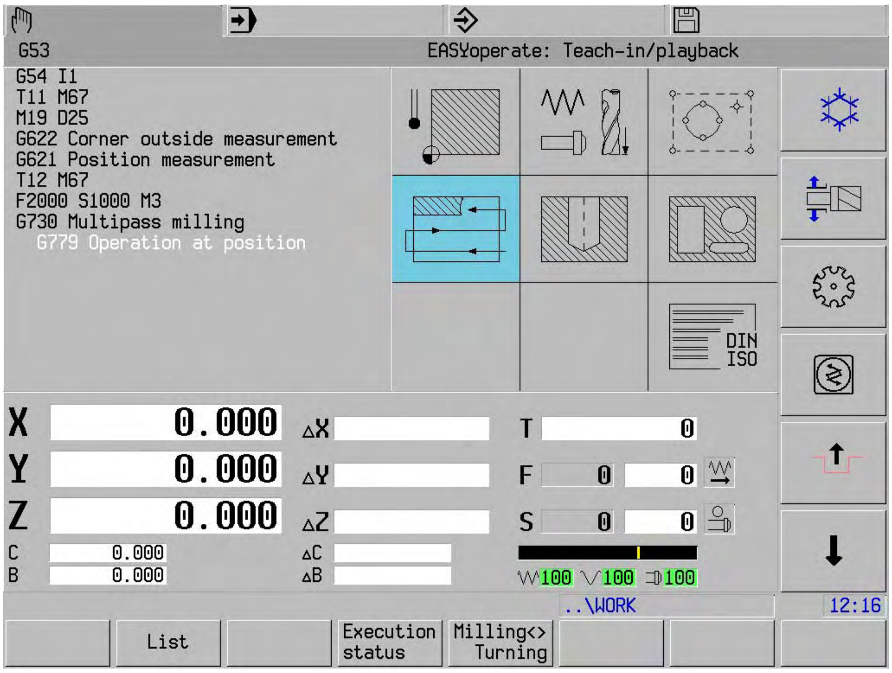
The G functions used in the cycles are described further in the G functions section.

19.1 Accessing EASYoperate mode

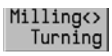
In manual mode, the EASYoperate function is called up from the menu line. First the main menu with the basic functions is displayed.



EASYoperate is used for programming simple machining steps on the machine.
In EASYoperate mode you can select a cycle directly and then execute it. After execution, the cycle is closed and you are returned to the main menu, or with the "Save" softkey, to the list.



Note:



If MillPlus has a turning mode available (activated via machine constant MC314), the "Mill <> Turn“ softkey is displayed. Use this to toggle between milling mode and turning mode. In turning mode, the relevant turning cycles and functions are shown in the menu. See the EASYoperate chapter on the turning main menu.

19.1.1 Exiting EASYoperate

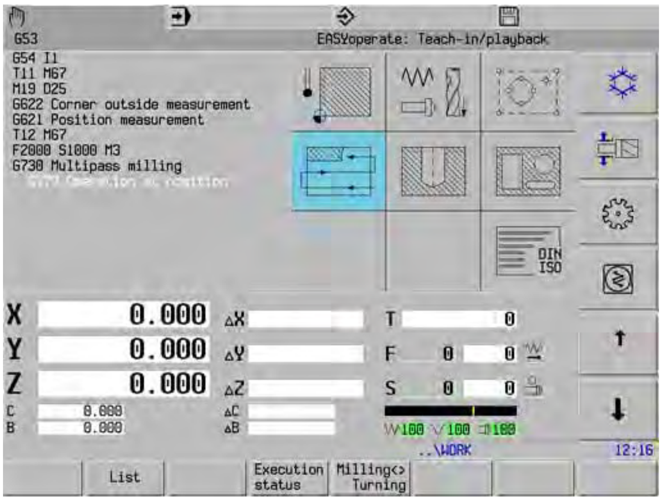
To leave EASYoperate temporarily, select a different process. When you select the "Manual" process level again, EASYoperate will start from the point where you left it.
To close EASYoperate, select the menu key.

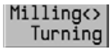
19.2 Basic functions of EASYoperate.

In EASYoperate mode, the screen has 2 windows:
a list on the left and the main menu on the right.

List:
Saved inputs (cycles and user-defined inputs). Der The cursor shows the current position in the list.

Main menu:
Graphical selection of available cycles. Assistance is provided for programming the selected cycle and this can then be executed directly and/or saved to the list.





Toggling between milling mode and turning mode. (machine-dependent)

19.2.1 List function

The list is activated: The cursor in the list turns blue and can be moved using the cursor keys. Detailed information associated with the cursor line is displayed in the right window.

The actions "Change, Copy and Delete“ are executed on the current cursor line or cursor block (marked in blue).

Marking a block within the List function:
Positioning the cursor on the required line. Press "Shift“ (keep it pressed) and move the cursor up or down. The required block is now marked (blue background).
To remove the marking, press the ESC key or any other softkey apart from "Copy or Delete".

As well as a milling process, a turning process can also be described in a list.
You can only add to the list in the correct turning or milling mode.
Changes can be carried out per block and error messages are only issued if the block cannot be executed.
There are no restrictions when deleting or copying blocks.

A status window is displayed over the list in the left window. This is where the modal functions are displayed.

The line indicated by the cursor can be edited. Changes are made using the same input options that were available for the original entry.

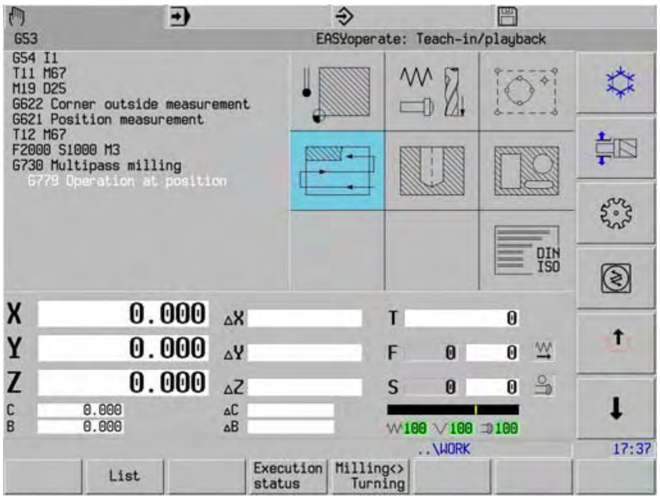
If the "Marked. Delete" softkey is activated, the marked lines are deleted immediately.
If the softkey "Delete list" is activated, then a new softkey appears with the question "Yes/No". If you answer "Yes", the entire list is deleted.

Once the "Copy" softkey has been pressed, the softkey is given a new function: “Insert".
Move the cursor to the position where you want to insert the copy (behind the cursor) and press "Insert".
To cancel the copy function, press the ESC key.

Jump to the main menu

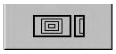
19.2.2 Formular input EASYoperate

During data entry mathematical functions can be input in the dialog entry window. The formular is input via the ASCII-Keyboard. If the mathematical expression does not fit in the dialog window, than the complete formular is showed in the explenation area



19.3 Select, start and/or save cycle/user-defined input.

Once a cycle has been selected (or user-defined input) and your entries have been made, the following functions become available:



A 2.5D graphics simulation starts up. A new softkey bar shows the additional functions.

Previous entry

The previous input associated with this cycle (that has been started or saved) is retrieved.

Store

The cycle (or user-defined input) is saved to the list and control passes back to the main menu (with the list on the left).

Return

The cycle (or user-defined input) is NOT saved in the list and control passes back to the main menu (with the unchanged list on the left).

If an execution cycle (specimen) has been chosen, more softkey functions are available:

Take over position

The current position is copied to the input fields

Inc <> abs

The position can be entered incrementally or absolutely for each input field.

Step/ Continue

Jog movement can be controlled.

If a definition cycle is entered, pressing the "Save" or "Back" softkeys automatically returns you to the Specimen menu. With the remaining cycles, the cursor remains on the last selection in the main menu.

19.3.1 Starting without saving, saving without starting

Starting without saving
In all cases, apart from menu selection, the values entered in the input field may be used directly for starting.
Important: The controller loses the entered values if these were not saved first.

Saving without starting
It is possible to save the entered values without starting.
Important: Saved cycles and user-defined inputs are not tested for the required operational sequence.

Once they have been saved to the list, the cycles and the user-defined inputs can be re-executed by performing the start again.

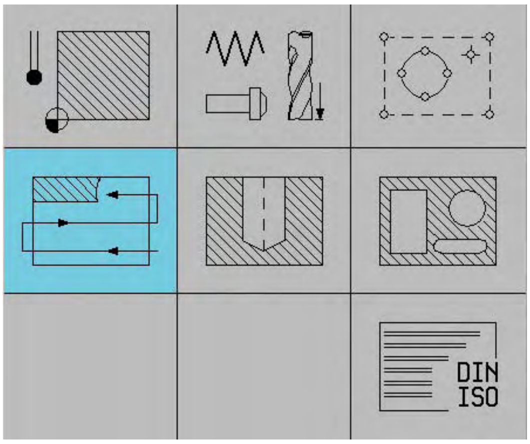
19.4 Milling mode main menu:

Selection options:

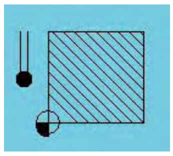
Measuring the material with the probe
FSTM input and tool measuring
Defining specimen positions

Executing a pass
Drilling processes
Pockets

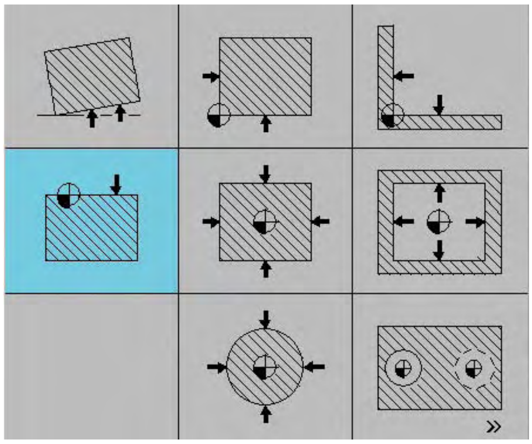
MDI user-defined input (DIN/ISO)



19.5 Menu: Measuring the workpiece zero point



Measuring the workpiece zero point:



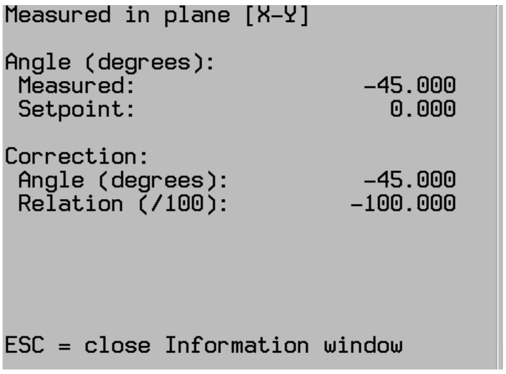
Angle measurement	G620
Measurement outside a workpiece	G622
Measurement inside a workpiece	G623
Workpiece position measurement	G621
Measurement outside a rectangle	G626
Measurement inside a rectangle	G627
Measurement outside a circle	G628
Measurement inside a circle	G629

Note:
For further information, see the Tools chapter.

19.5.1 G62x measurement information window

When a G62x function is called, you can enter the I5= address.
When the cycle starts, an information window appears on the left side (over the support image): Measured values are shown.
To close the window, use the ESC key: The support image will again be visible.

Note for address I5= for G620:
I5=0 Measured values are only displayed on the screen.
I5=1 Measured values are saved for an axis transformation.
I5=2 Measured values are saved for a rotary axis rotation

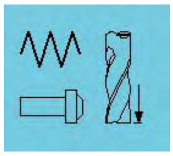


Plane in which measurement is to occur

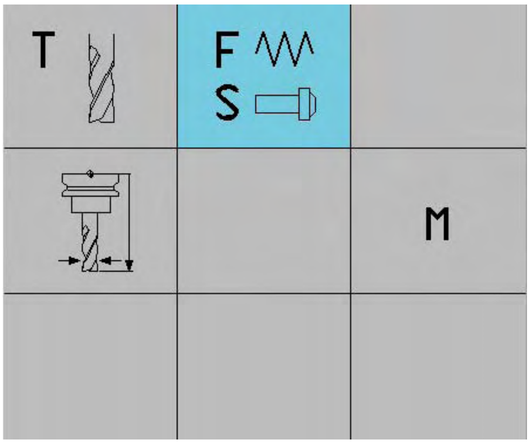
Measured angle value
Entered setpoint value

Difference between the measured value and the setpoint value
in degrees or mm/100mm

19.6 Menu: FST



Selection options:



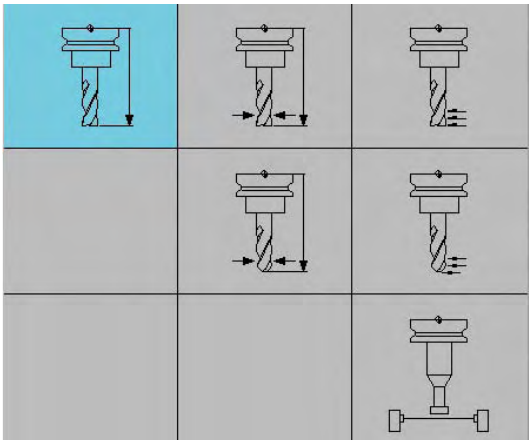
Tool number with associated M-function (with tool overview list)

Feed and cutting speed with associated M function.

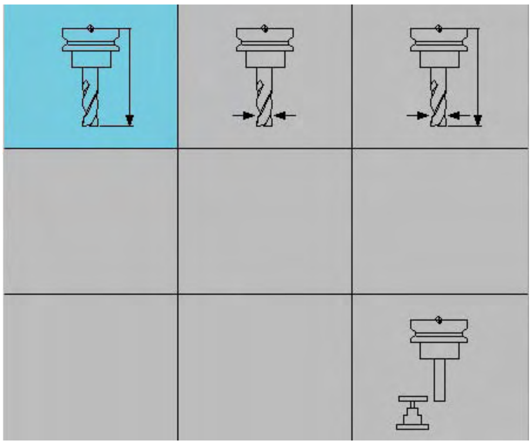
Laser or TT130 measurements (selectable via MC854)

M function (With M function overview list).

Tool gauging:



Laser measurement (MC854=1)

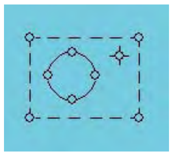


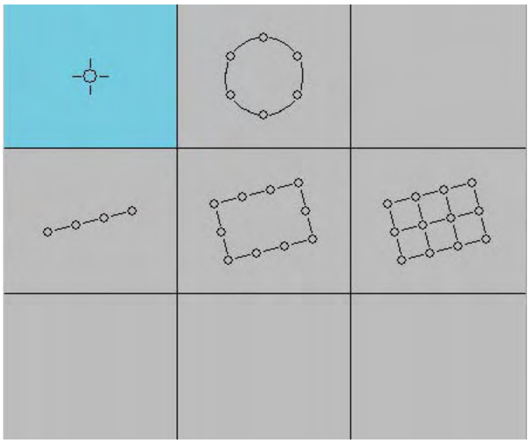
Heidenhain TT130 (MC854=2)

Note

For further information, see the Tools chapter.

19.7 Menu: Specimens





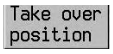
Auswahlmöglichkeiten:

Execution in position.	G779
Execution in a circle.	G777
Execution in a line	G771
Execution in a rectangle	G772
Execution in a grid	G773

Note on all execution cycles:
Only available in EASYoperate.

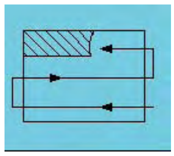
19.7.1 Absolute and incremental inputs

Only in execution cycles can you use the "Inc/Abs" softkey to decide for each position value that you enter whether the value has to be calculated incrementally or absolutely.
If the value is operated incrementally, a delta character is shown next to the address.



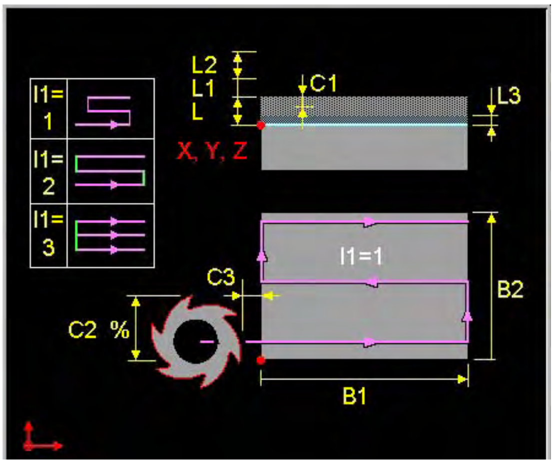
If the "Adopt actual pos." softkey is used to enter a value in the X, Y or Z input field, then this value is automatically absolute.

19.8 Menu: Surface milling



Selection options:

Executing a pass	G730



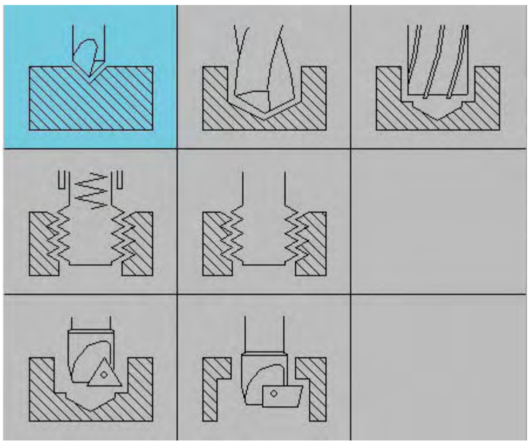
Note:

If C2 is not programmed, the feed width is 67% * tool diameter.
You can use the I1= address to define the machining strategy: meander, with rapid intermediate movements or with parallel paths.

19.9 Menu: Hole machining processes

Selection options:

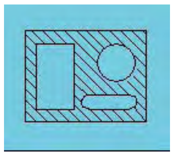
Drilling/centring	G781
Deep drilling	G782
Hollow boring	G786
Tapping with compensating chuck. Only available in EASYoperate.	G784
Tapping without compensating chuck. Only available in EASYoperate.	G794
Reaming	G785
Reverse countersinking	G790



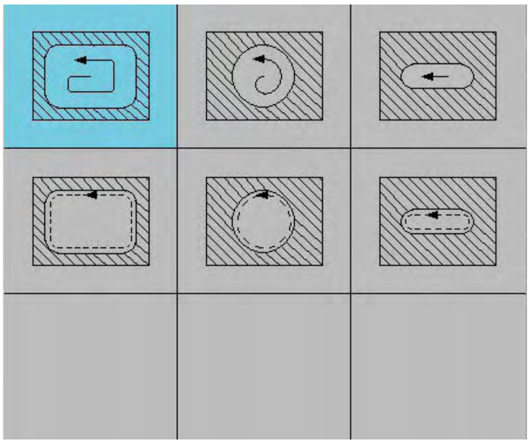
Note:

Tapping: if pitch (F1) is not programmed, the feed is F.

19.10 Menu: Pocket machining



Selection options:



Pocket roughing	G787
Circular pocket roughing	G789
Slot roughing	G788
Pocket finishing	G797
Circular pocket finishing	G799
Slot finishing	G798

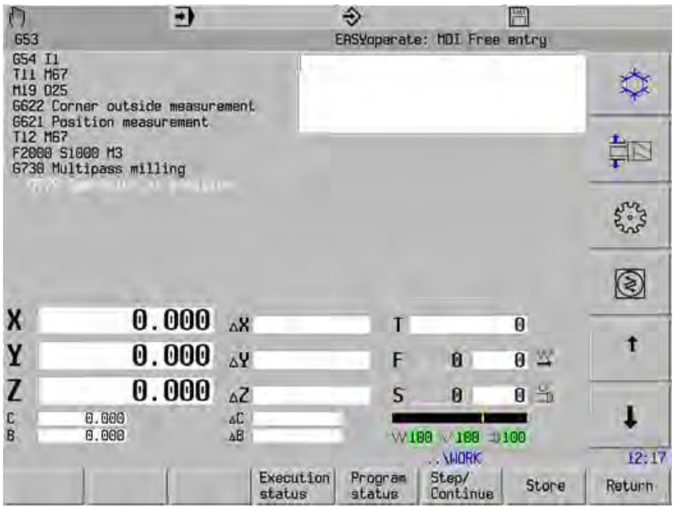
Note:

For further information, refer to the G function for the selection options.
If C2 is not programmed, the feed width is the same as machine constant MC720.

19.11 Menu: DIN/ISO

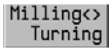
As with direct MDI input, a G, M, FST, etc. entry can be made here. This entry can now be saved to the list.

Comments are placed in the list using bracketed text.



19.12 Turning mode main menu

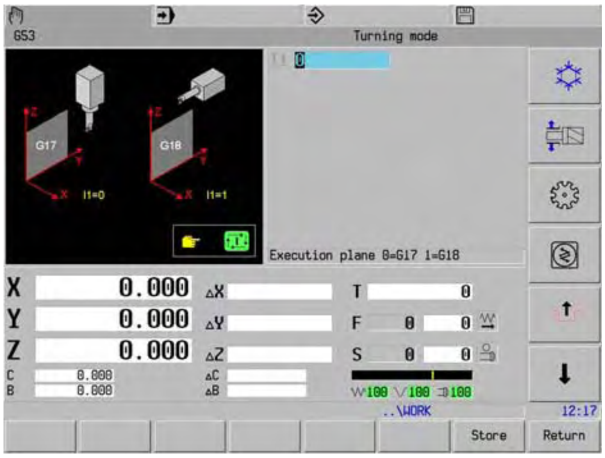
19.12.1 Enable turning mode



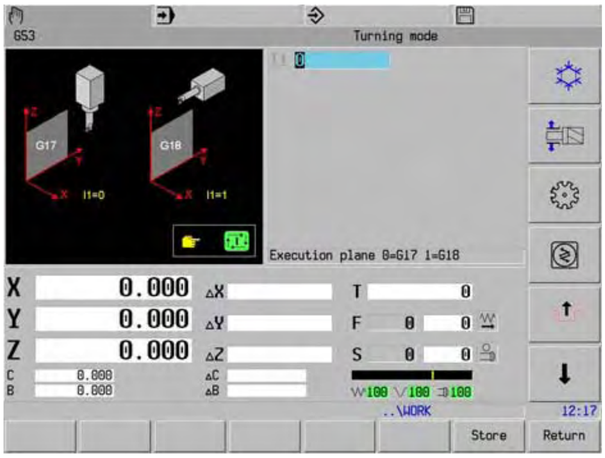
Toggle between milling and turning.

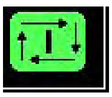
This displays a new menu:

Select turning mode.



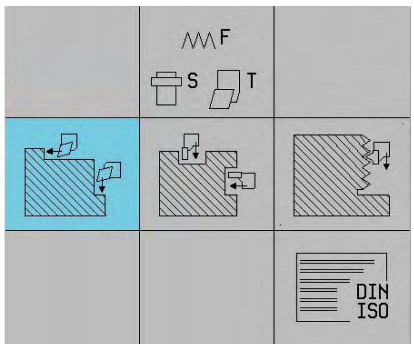
When turning mode is enabled, the machining plane must be selected:
G17 (basic setting) or G18.



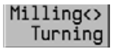


A start must now be performed. This puts the machine into turning mode.

In turning mode, three turning cycles are available



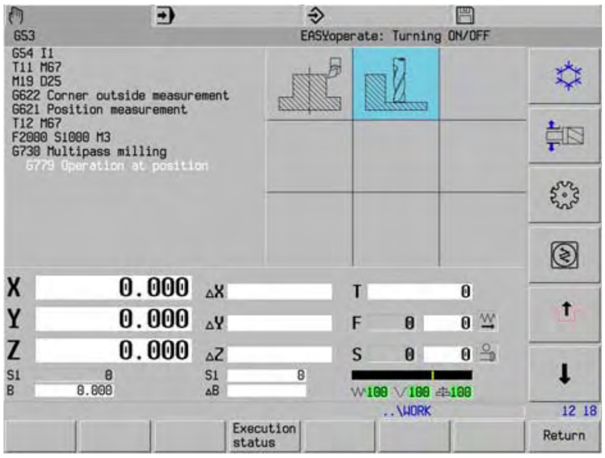
19.12.2 Enable milling mode



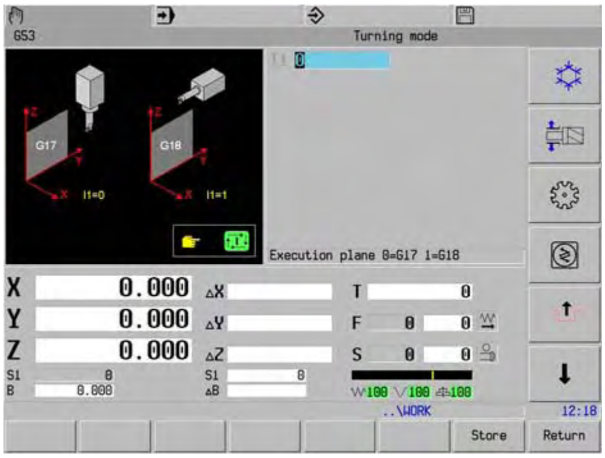
Toggle between turning and milling.

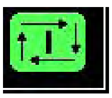
This displays a new menu:

Select milling mode.



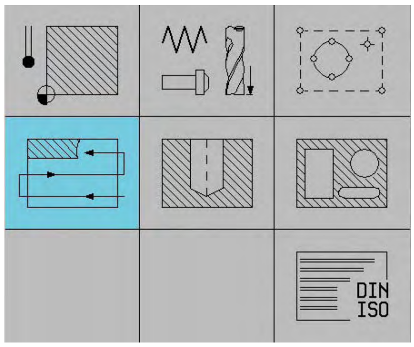
When milling mode is enabled, the machining plane must be selected:
G17 (basic setting) or G18.





A start must now be performed. This puts the machine into milling mode.

In milling mode, three milling cycles are available



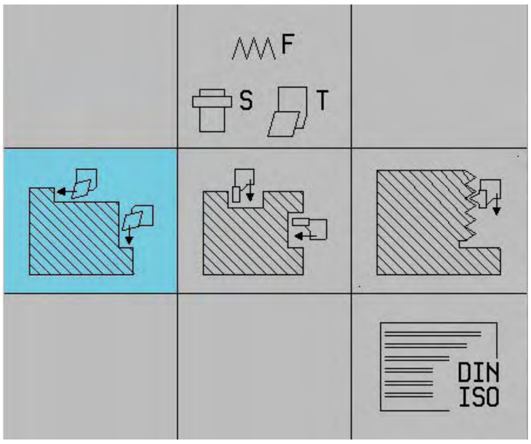
19.13 Menu: Turning mode main menu:

Selection options:

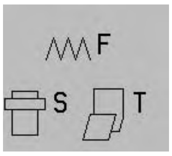
FST input

Machining
Plunge cutting
Treadcutting/Undercuts

MDI user-defined input (DIN/ISO)



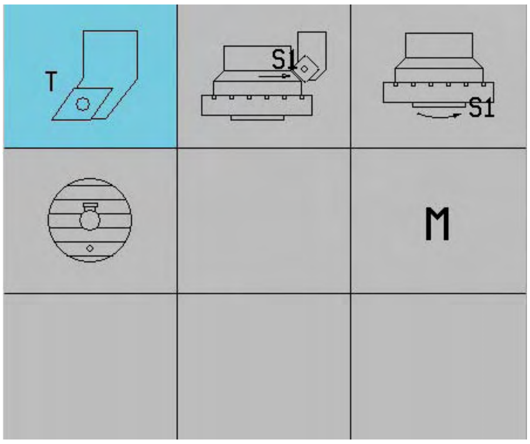
19.14 Menu: FST



Selection options:

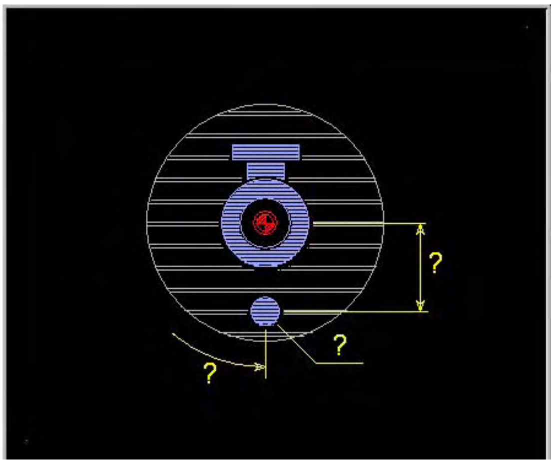
Tool change
Set cutting speed, feed
Set table speed, feed

Unbalance detection
Machine functions

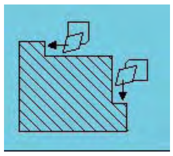


The inputs for the tool (with M function), constant cutting speed and table speed can now be entered.

The workpiece unbalance can be determined. (G691)

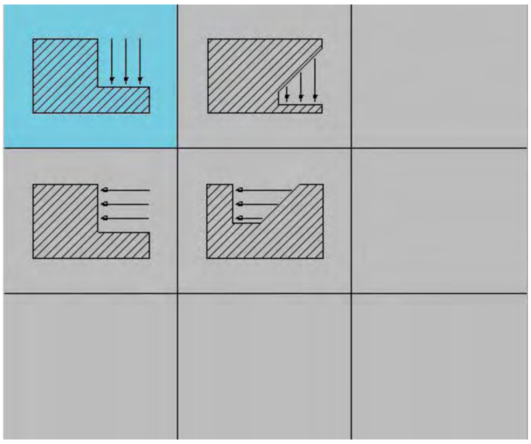


19.15 Menu: Machining

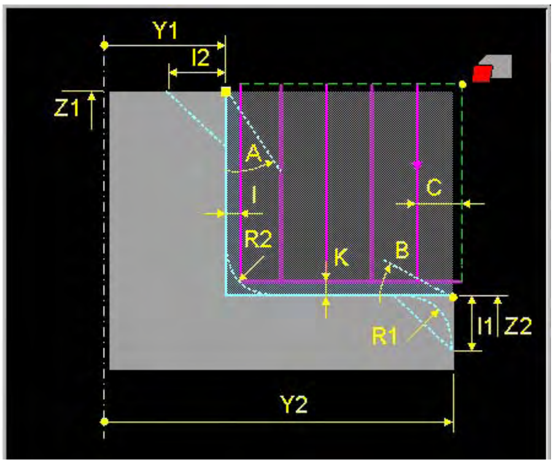


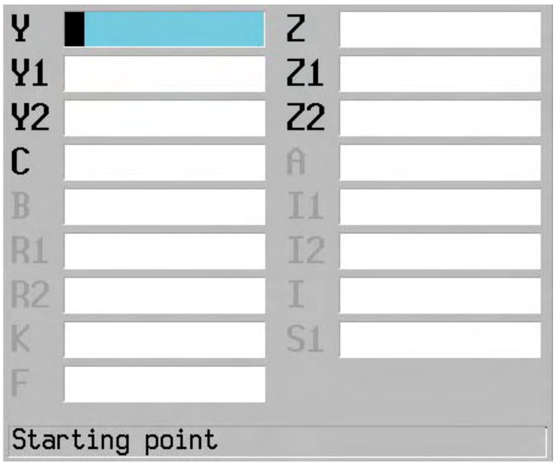
Selection options:

Longitudinal cut	G822/G826
Longitudinalreverse boring	G832/G836
Cutting plan	G823/G827
Reverse boring plan	G833/G837

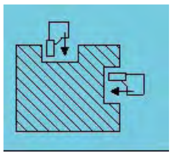


Example:
Cycle: Longitudinal cut (G822)

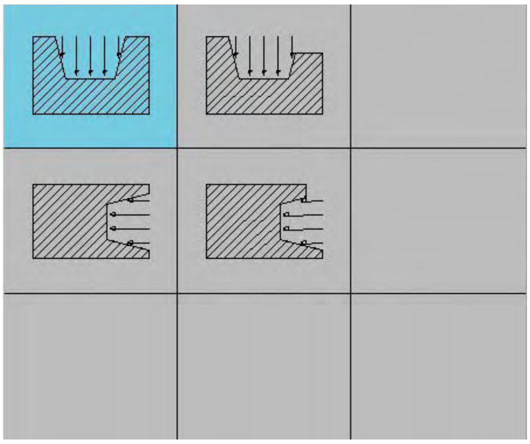




19.16 Menu: Plunge cutting

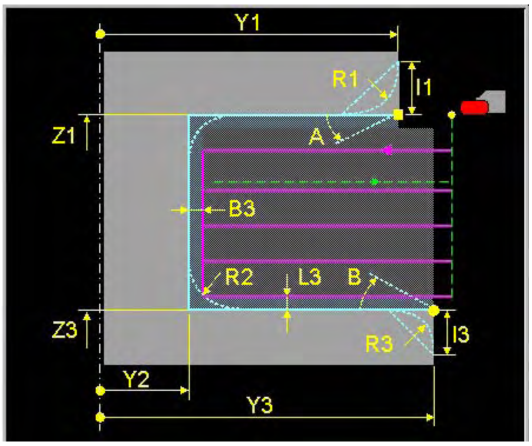


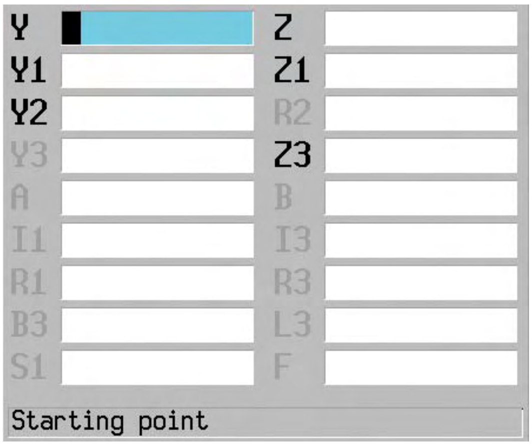
Selection options:



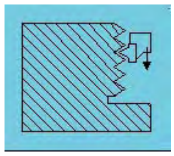
Axial plunge cutting	G842/G846
Radial plunge cutting	G843/G847
Axial plunge cutting universal	G844/G848
Radial plunge cutting universal	G845/G849

Example:
Cycle: Axial plunge cutting universal (G845)



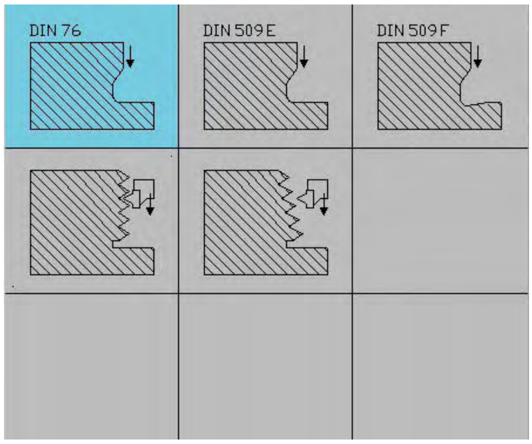


19.17 Menu: Thread cutting/Undercuts

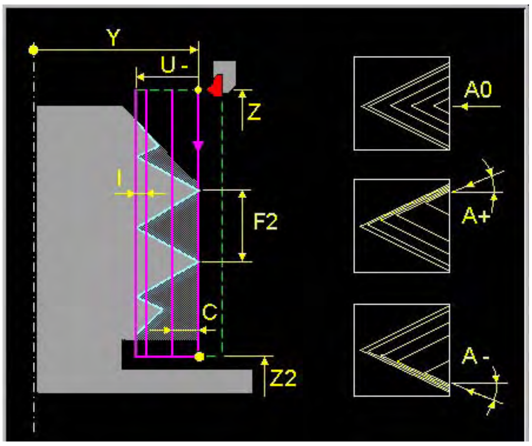


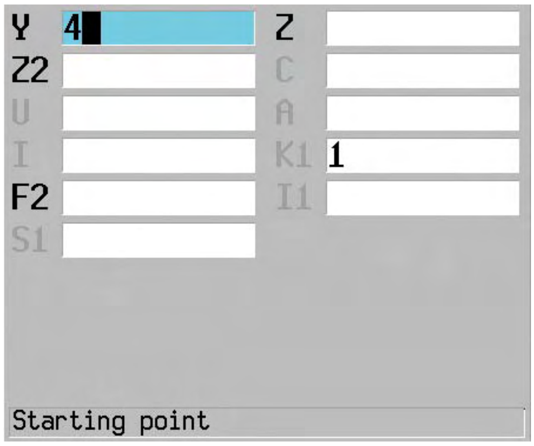
Selection options:

Undercut DIN 76	G850
Undercut DIN509 E	G851
Undercut DIN 509 F	G852
Threadcut cylinder	G861
Threadcut conical	G862

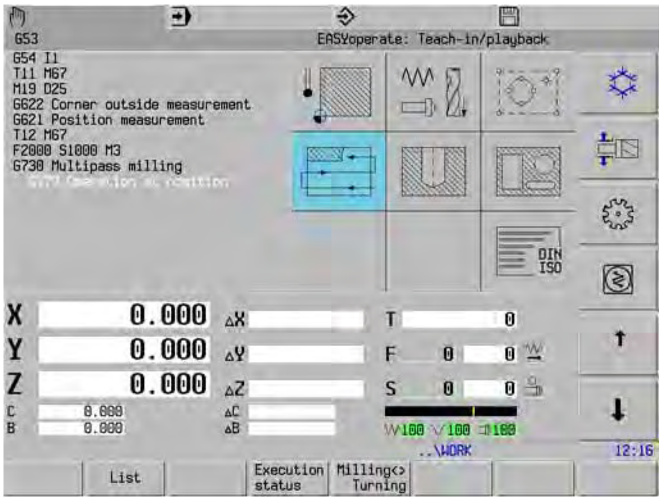


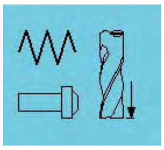
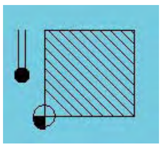
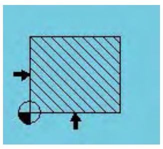
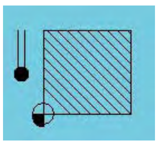
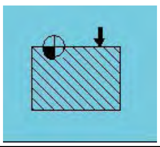
Example:
Cycle: Threadcut cylinder (G861)

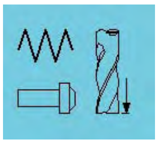
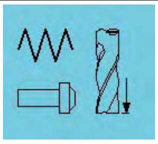
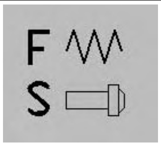
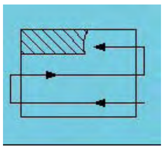
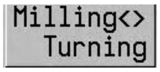




19.18 Example in a list



Operation via menu:	List:	Comments:	
	G54 I1	Activate zero point	
 	T150 M67	Change the probe	
	M19 D25	Orientate the probe	
	(Measure zero point with probe)		
 	G622 measure outside corner	I4=1	Corner number
		B3=10	Distance to the corner
		C1=10	Gauge length
		I5=1	Do not save measured value
 	G621 measure position	I1=-3	Measuring direction=tool axis
		C1=10	Gauge length
		I5=1	Do not save measured value

	(Surface milling)		
 	T12 M67	Change the milling tool	
 	F2000 S1000 M3	Feed, speed and direction of rotation	
	G730 multipass	B1=200, B2=100	Length of side
		L5, L1=1	Height and safety distance
		C2=67	Percentage cutting width
		C3=5	Radial setup clearance
		I1=1	Radial setup clearance
	G779 Machining in position	X0 Y0 Z0	Start position of multipass milling

20. Interactive contour programming (ICP)

20.1 General

ICP can be used with new or existing main programs or macros.

ICP can be used with DIN/ISO and IPP.

The programmer begins at a certain point of the contour and processes the entire workpiece in either a clockwise or anticlockwise direction, each contour being described as a linear or circular movement. Following this initial selection other options are offered until the movement is defined. You are then requested to enter positional data.

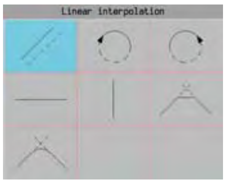
Using ICP each contour is drawn as soon as its position is known, to be precise once the Store key is pressed. However, this does not always have to be the case. If a contour cannot be classified immediately it is joined to the following contour until sufficient positional data is available to calculate its exact position.

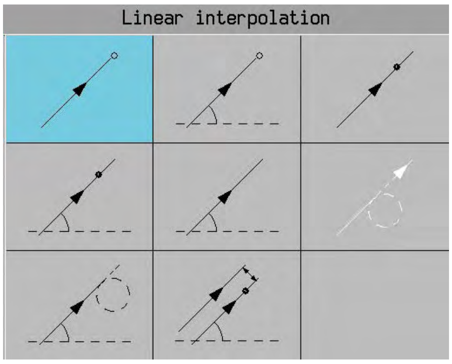
20.2 ICP graphic symbol menu

ICP has a dynamic menu structure. Options are enabled or blocked depending on the previous option selected.

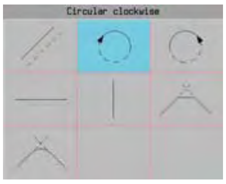
- □ Center point
- ● End point
- ○ Support point

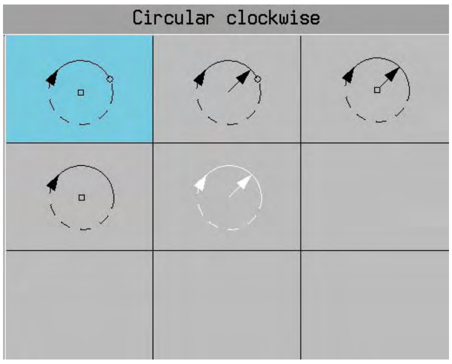
Main menu level **Menu for linear movement**



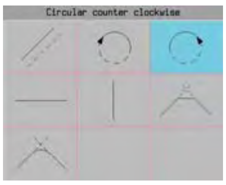


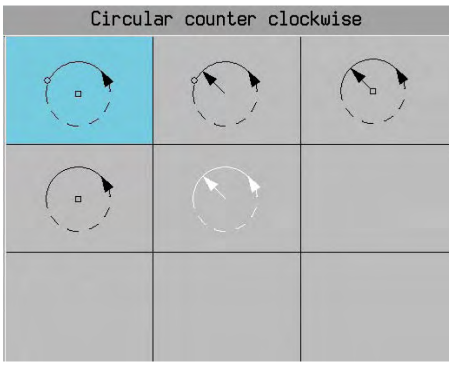
Menu for circular movement in a clockwise direction



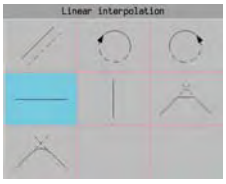


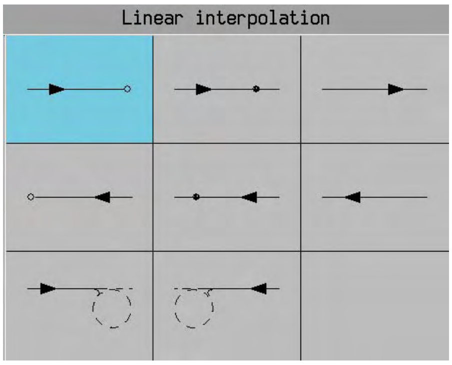
Menu for circular movement in an anti-clockwise direction



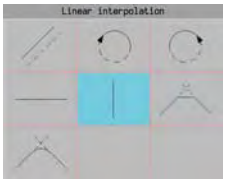


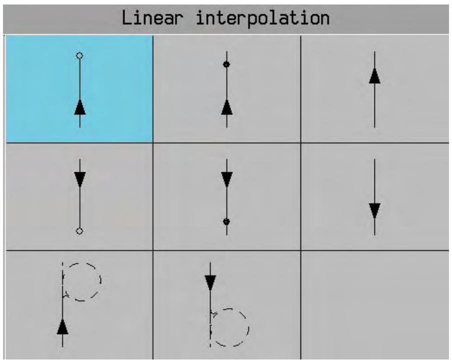
Menu for linear movement horizontally



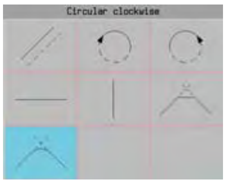


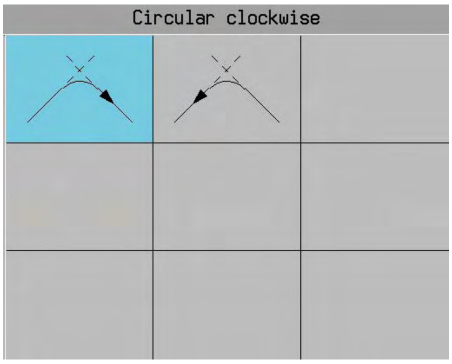
Menu for linear movement vertically



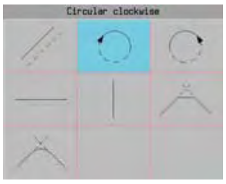


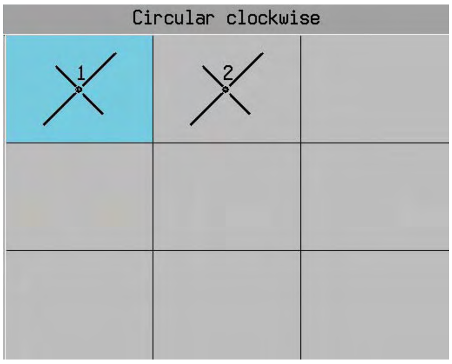
Menu for rounding





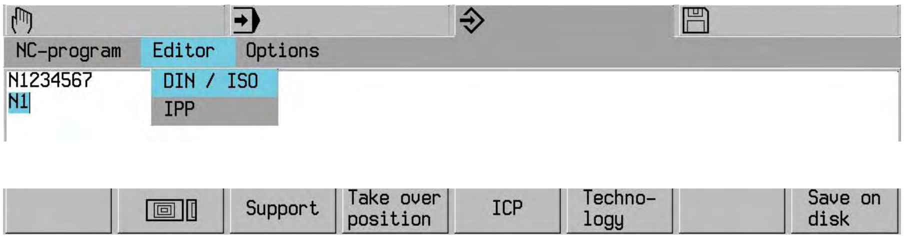
Menu for point of intersection



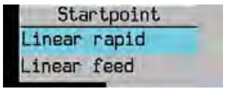


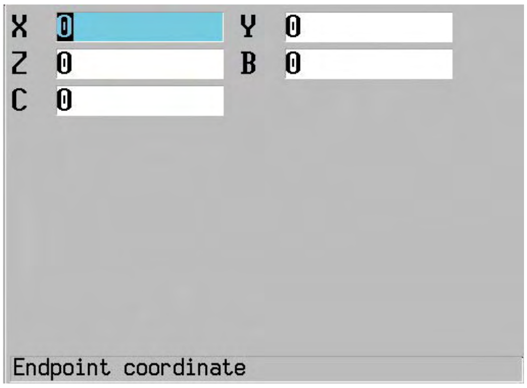
20.3 New ICP programs

20.3.1 Entering the ICP-Mode

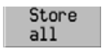


New programs may be completely empty apart from the header line. In this case the programmer is prompted to enter a starting point.





Enter a value for all parameters specified, even if this value is 0.



Note

No allowance is made in ICP for a pole position predefined with G9. G9 must be deselected before ICP

20.3.2 Exiting ICP

Return

			Change element	Add elements	Delete element	Freeze coordin.	ICP exit

ICP exit Ending ICP by confirmation via softkey.

or

F8 Menu

It is possible to exit the ICP INPUT mode at any time during data input, although exiting ICP during contour programming may result in an error message when ICP is re-entered.
The program line or lines affected must then be found and deleted.

20.4 Editing existing programs

When using an existing program, the cursor is positioned at that point in the program at which ICP is to commence.

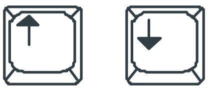
Using the cursor keys, scroll upwards and downwards through the program. The relevant contour section is shown white in the graphic window.

ICP scans the program section before the cursor position for a G64 function without G63 (the cursor is in an ICP section in the program). If the cursor is located outside a G64-G63 area, these ICP G-functions are situated in successive program lines.

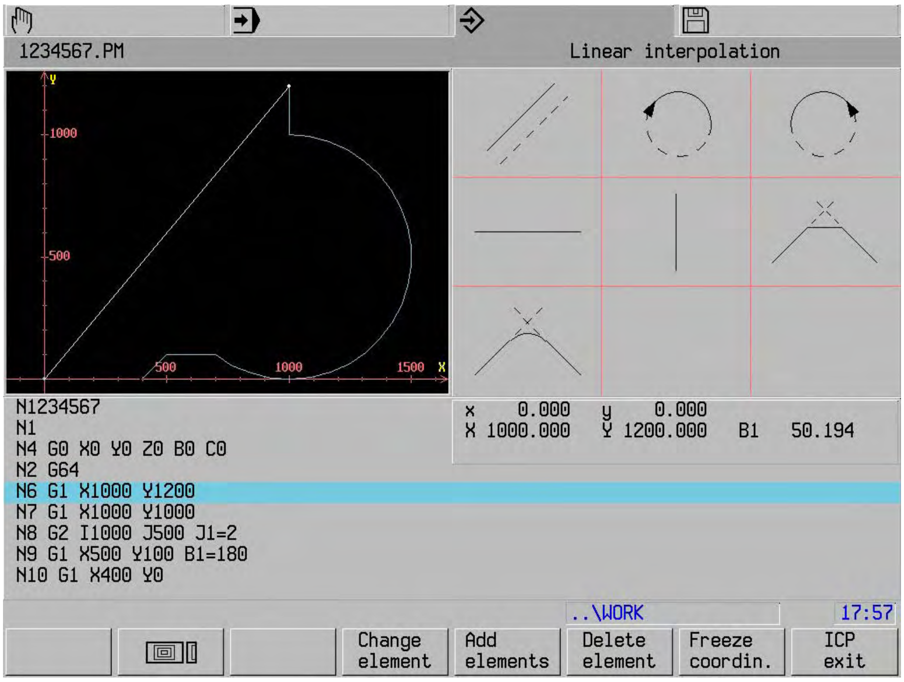
The program is checked in advance as to whether at least a feed movement has been programmed for the addresses of the main plane. If not, the user is requested to enter a traversing movement.

20.4.1 Edit element

ICP Select ICP.

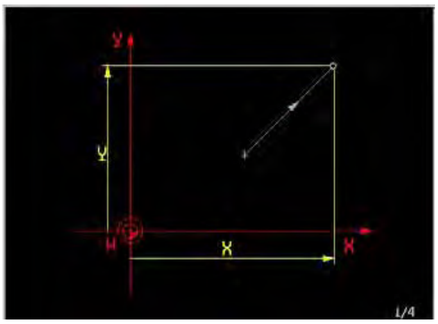


Select program block, e.g. N8.



Change element

The contour element can be defined differently,
e.g. an address value can now be changed.
Enter addresses.

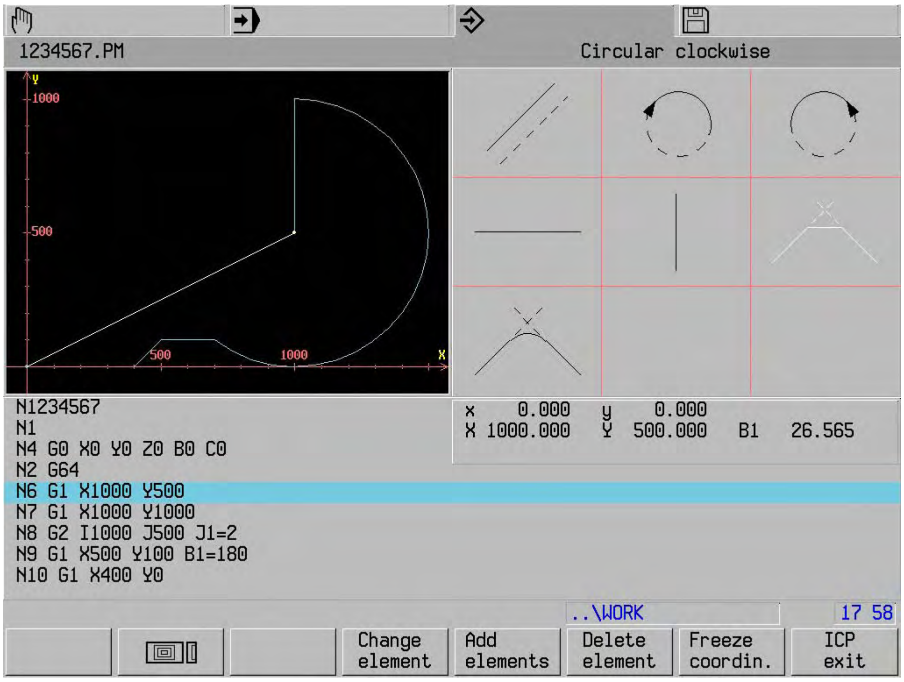


Store

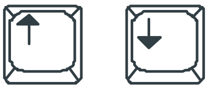
or

F7 Store

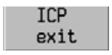
The element is stored and the contour recalculated and displayed.



Have all the changes in change mode been implemented?
No?



Next element..

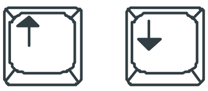


Yes?

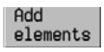
Note

For specific elements (rounding circles) there are additional solution variants. These variants can only

20.4.2 Insert element



Insert contour element / select block



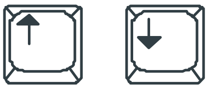
Note:

For specific elements there are several input options:

Next picture — Selection of the options

20.4.3 Delete element

			Change element	Add elements	Delete element	Freeze coordin.	ICP exit



Select the contour element / block to be deleted

Delete element

Note

By deleting, changing or inserting elements, it is possible to create non-continuous contours. The modified element or subsequent elements are shown as dotted lines.

20.4.4 Graphial display of the contour

			Change element	Add elements	Delete element	Freeze coordin.	ICP exit

							Return

Zooming out

Zooming in

Original size

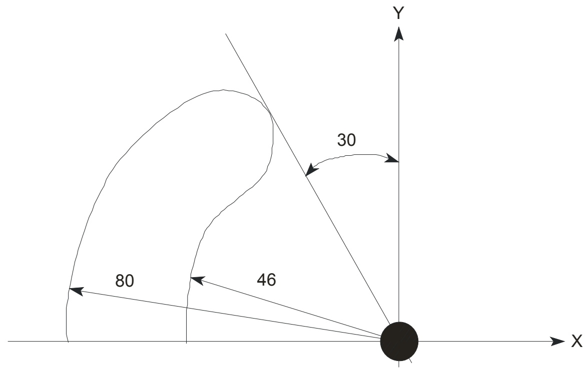
20.5 ICP programming notes

20.5.1 Auxiliary elements in ICP

Lines and circles may be defined by auxiliary elements, for instance tangents or circles. Missing coordinates or angles can be calculated with the auxiliary elements. These calculated values are always displayed for every element.

These calculated values are saved by the "Freeze coordin." softkey. The auxiliary elements may then be deleted and the required circle or straight line can be re-entered.

Example:



N100 G0 X-80 Y0	Starting point
N101 G64	Select ICP
N102 G2 I0 J0	Circle with centre
N103 G2 R17	Rounding (clockwise)
N104 G1 X0 Y0 B1=-60	Auxiliary line with end point and angle, select intersection 2

- Place cursor on block N103.

- Display:	x -57.211	y 55.918		Starting point (lower case)
	X -30.332	Y 52.536		End point (capital)
	I -45.054	J 44.036	R17	Centre and radius

- Save these coordinates by pressing F7 "Freeze coordin.".

- Delete auxiliary line N104 and circle N103.

- Re-enter program blocks N103 (circle with centre) and N104:

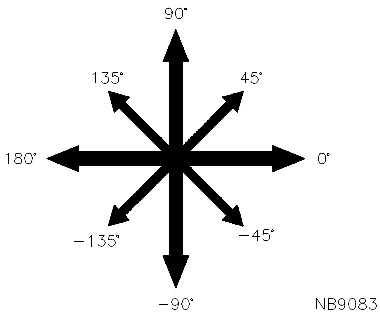
N103 G2 I-45.054 J44.036	Circle (clockwise) with centre
N104 G3 X-46 Y0 R46	Circle (counterclockwise) with end point and radius
N105 G63	

20.5.2 Help points

The "Help point" programming option in ICP offers a simple solution to the problem of defining axis end points in complex contours. The option is used when the axis end point is unknown. As soon as the axis end point is determined by the next movement or those following on from it, it is classified.

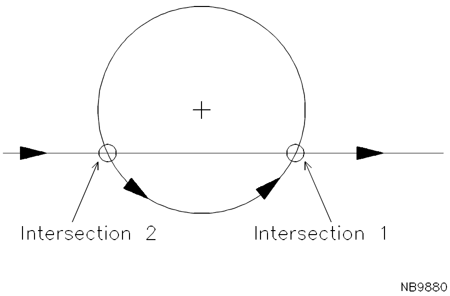
20.5.3 Required angle parameters

Some of the linear interpolation movements call for an angle parameter (specified relative to the horizontal).



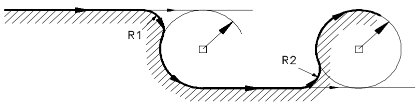
20.5.4 Line intersects circle

ICP draws the line which passes through the circle and the points of intersection (1 and 2) are marked. The programmer is requested to select the correct point of intersection.



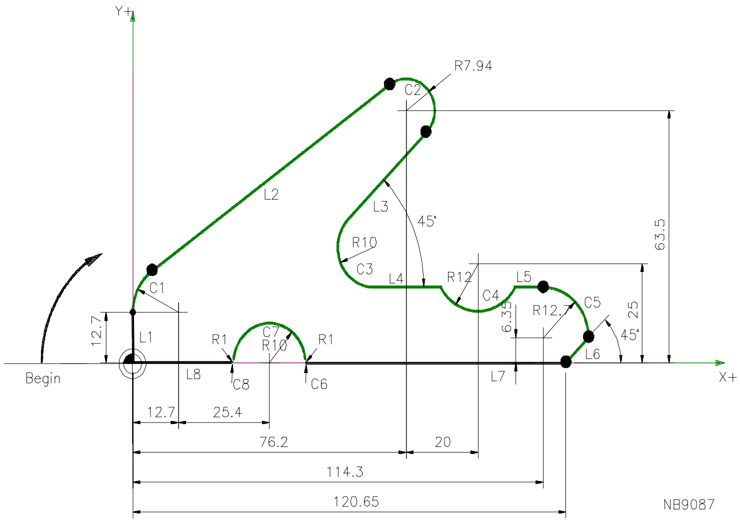
20.5.5 Rounding

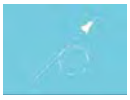
The movement preceding the rounding may be designed in any manner, including with an endpoint. The rounding is specified purely as a radius. Its position and its start and end point are calculated by ICP as soon as sufficient data is available to classify it.



20.6 ICP programming example

First create a new program N111111 with the start point X0, Y0, Z0.



L1			X0 Y=12.7	Enter, Store
C1			I=12.7 J=12.7	Enter, Store
L2				
C2			I = 76.2 J = 63.5 R = 7.94	Enter, Store
L3			B1 = -135	Enter, Store
C3			R = 10	Enter, Store
L4			X = 120 Y = 19.05	Enter, Store
C4			I = 96.2 J = 25 R = 12	Enter, Store

L5	X = 120 Y = 19.05	Enter, Store
C5	I = 114.3 J = 6.35 R = 12.7	Enter, Store
L6	X = 120.65 Y = 0 B1 = -135	Enter, Store
L7		
C6	R = 1	Enter, Store
C7	I = 38.1 J = 0 R = 10	Enter, Store
C8	R = 1	Enter, Store
L8	X = 0 Y = 0	Enter, Store

20.6.1 ICP-generated program

N111111 (ICP-generated program)
N1 G0 X0 Y0 Z0
N2 G64
N4 G1 X0 Y12.7
N5 G2 I12.7 J12.7 R1=0
N6 G1 R1=0
N7 G2 I76.2 J63.5 R7.94 R1=0
N8 G1 B1=-135
N9 G3 R10
N10 G1 X120 Y19.05 B1=0 I1=0 J1=2
N11 G3 I96.2 J25 R12 J1=1
N12 G1 X120 Y19.05 B1=0 I1=0 J1=2
N13 G2 I114.3 J6.35 R12.7 J1=1
N14 G1 X120.65 Y0 B1=-135
N15 G1 B1=180 J1=1
N16 G2 R1
N17 G3 I38.1 J0 R10 J1=1
N18 G2 R1
N19 G1 X0 Y0 B1=180
N3 G63

20.6.2 Alternative ICP programming methods

The previous example showed only one possible method of programming the individual movements. The same result can be achieved in several ways. The various options for programming a line 1 and circle 1 are shown below::

		X = 0 Y = 12.7	N4 G1 X0 Y12. N5 G2 I12.7 J12.7 R1=07
		I = 12.7 J = 12.7	
1 Line as tangent		I = 12.7 N4 G1 R1=0 J = 12.7N5 G2 I12.7 J12.7 R12.7 R1=0 R = 12.7	
			
2. Line with help point		X = 0 Y = 10	N4 G1 X0 Y10 I1=0 J1=2 N5 G2 I12.7 J12.7 R12.7 R1=0
		I = 12.7 J = 12.7 R = 12.7	
			
3. Line with angle		B1 = 90 N4 G1 B1=90 J1=2	N5 G2 I12.7 J12.7 R12.7 R1=0
		I = 12.7 J = 12.7 R = 12.7	
			
4. Line, vertical		Y12.7	N4 G1 Y12.7 B1=90 N5 G2 I12.7 J12.7
		I = 12.7 J = 12.7	

21. Interactive part programming (IPP) / GRAPHIPROG

21.1 General

21.1.1 Introduction to Interactive Parts Programming (IPP)

When using interactive parts programming you must select from a number of features and machining strategies to create a program. For the most part no knowledge of DIN programming is assumed.

IPP technology proposals are derived from the information in the technology database. The information stored therein is based on your own experience in the workplace. Please refer to the chapter on technology.

Each feature begins with a block, which contains the feature description and an identification. You can switch from IPP to DIN programming at any time.

The machining sequence can be simulated at any time during the creation of a program.

21.1.2 Preparation for IPP programming

- The technology tables must contain suitable data.
- The IPP start macro must contain the right data (see 21.8)

Note

- Always make sure that the retract movement of the tool axis in parameter E714 is large enough to avoid a collision between tool and workpiece or fixture.

- The tool table must display a list of the tools used most frequently.

- If there is no suitable tool in the tool table, IPP will generate a new tool in this table.
  All tools created with the aid of IPP should be entered in the tool table.
  During simulation M6 is changed to M67.

21.1.3 IPP programming sequence

The procedure for programming a new program in IPP is described below:

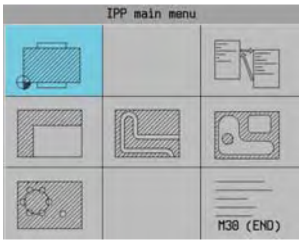
1. First define a blank.

2. You also have the option of defining the type of workpiece clamping device to be used.

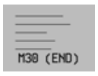
3. Program the workpiece with the aid of IPP features.

4. Select the M30 feature to conclude the program.

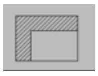
21.2 IPP graphic main menu symbols



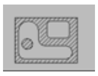
 Drilling operations

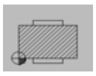


End of program

 Facing and edge milling

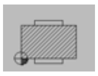
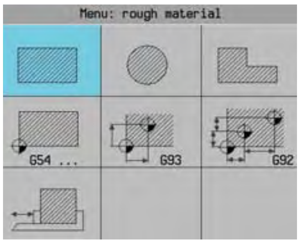
 Contour input, thread milling

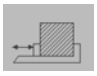
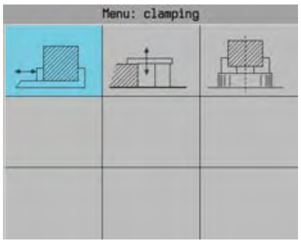
 Pocket with or without islands

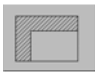
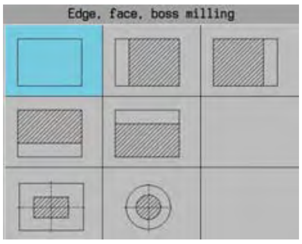
 Invoke macro or main program

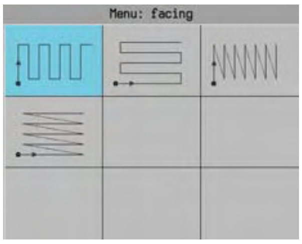
 Set-up (material, zero points and clamping)

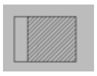
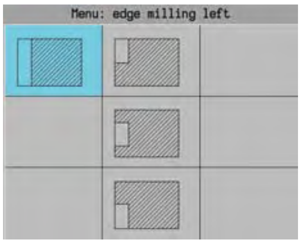
21.3 IPP graphic symbol menu

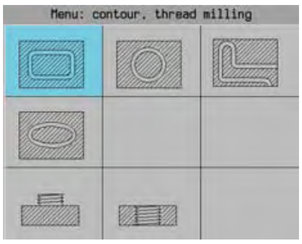



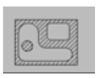
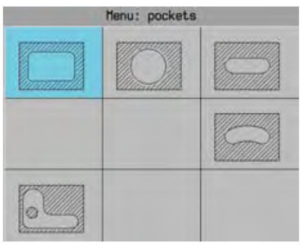



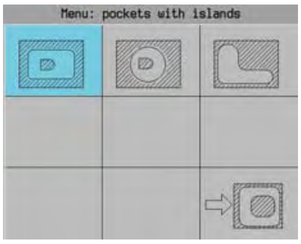



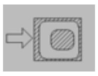


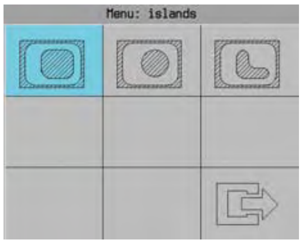



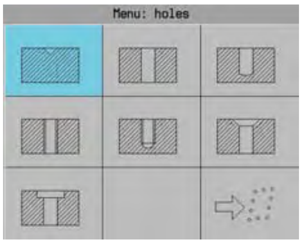



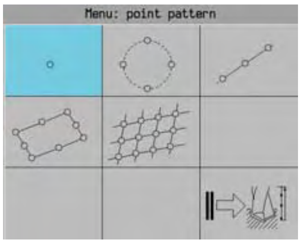



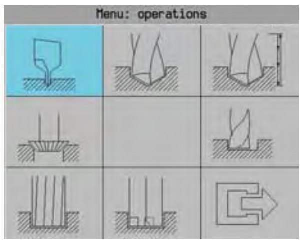

Menu: pockets with islands




Menu: islands


Menu: holes

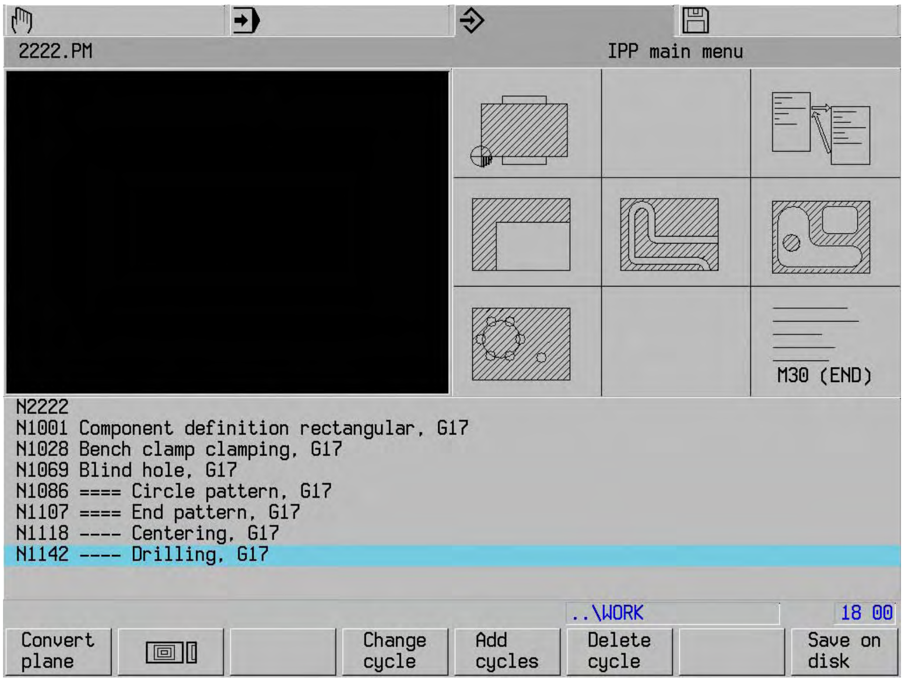

Menu: point pattern


Menu: operations

21.4 New IPP-program

21.4.1 Entering IPP mode

Select program



Note

If it is impossible to access IPP, it should be checked whether the reference point has been approached in all axes or whether G19, G91, G182, G201, G64 or G199 is active.

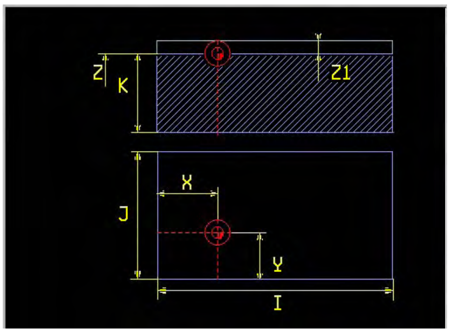
21.4.2 Exiting IPP

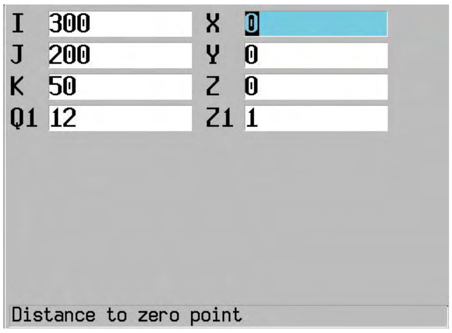
Exit IPP.

Note

Exiting IPP during programming will result in an incomplete program.

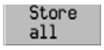
21.4.3 Entering program data





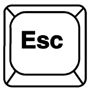
Once a machining operation has been defined by means of Feature, the data entry window appears with the addresses required for complete definition.
A value must be entered for each address. A value is already suggested for many addresses.

Storing the input values and displaying the next data entry.



Storing the input values and exiting the data entry.

Note

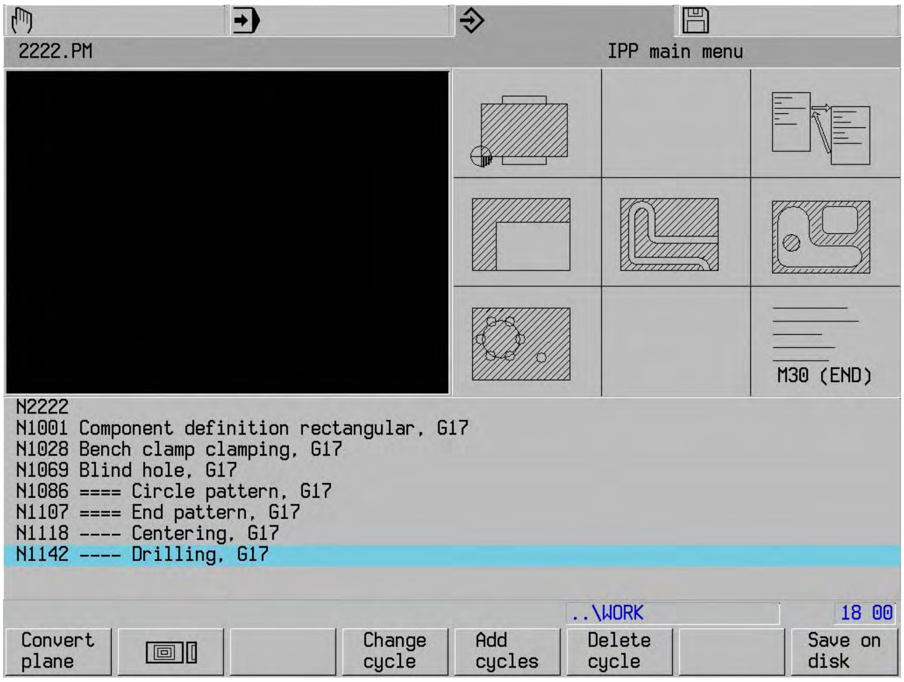


Return without saving data.

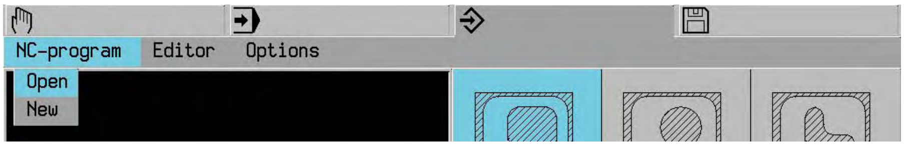
Exiting the data input mode during programming often results in an incomplete program.
The relevant feature must then be deleted and reprogammed.

21.4.4 IPP program list

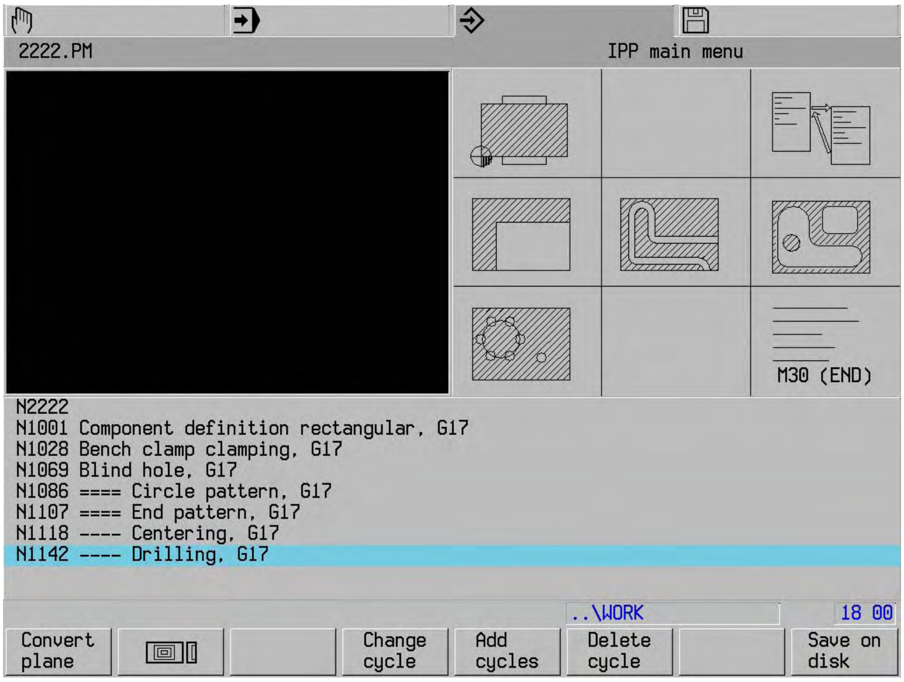
The program window only displays the names of the features used in the parts program

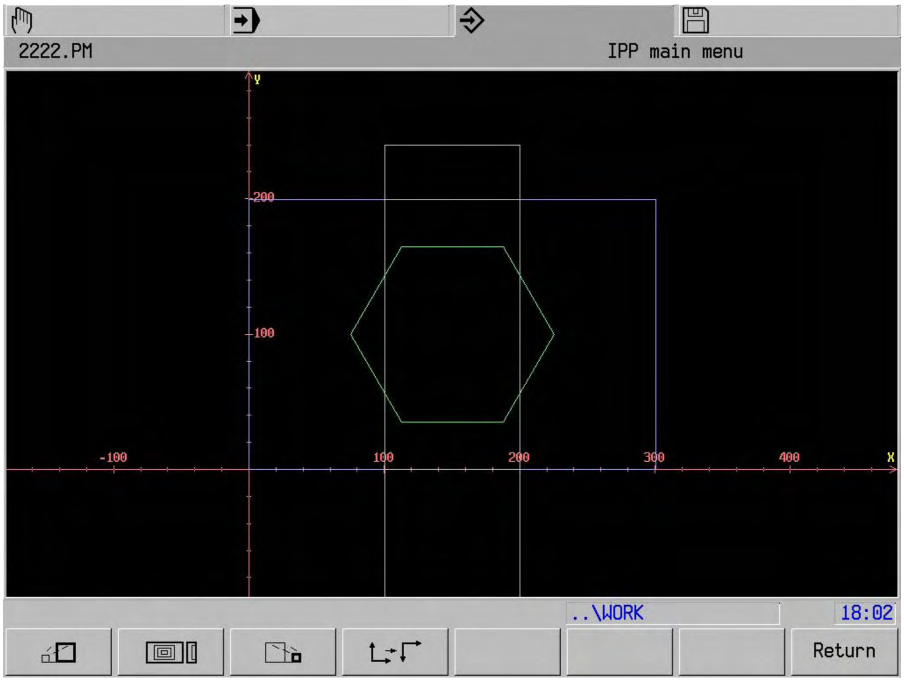


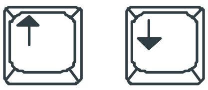
21.5 Editing IPP programs (change line



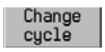
21.5.1 Change features





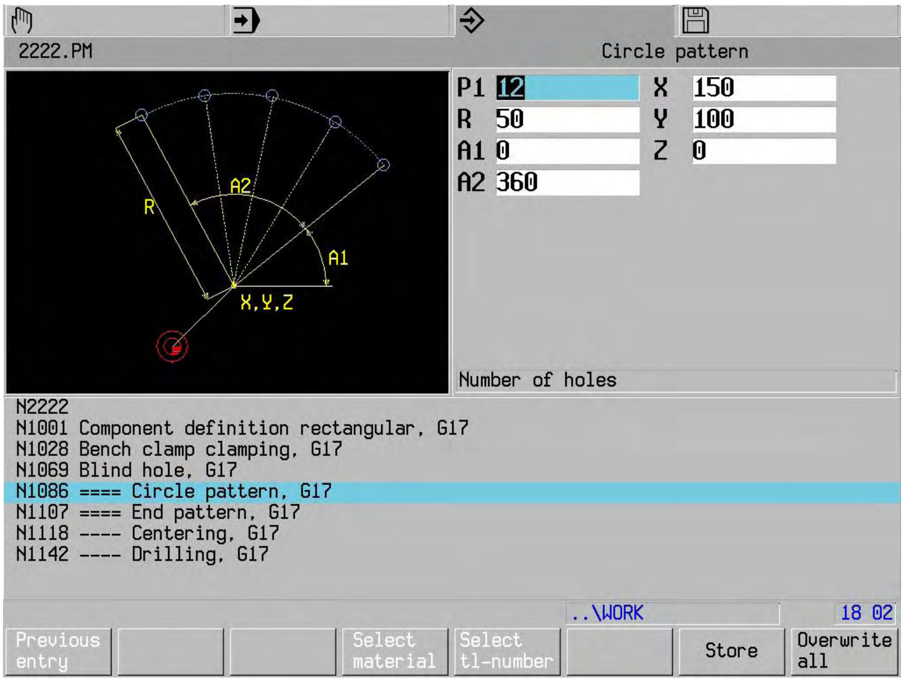


Select the feature to be changed...

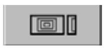


The feature can be redefined.

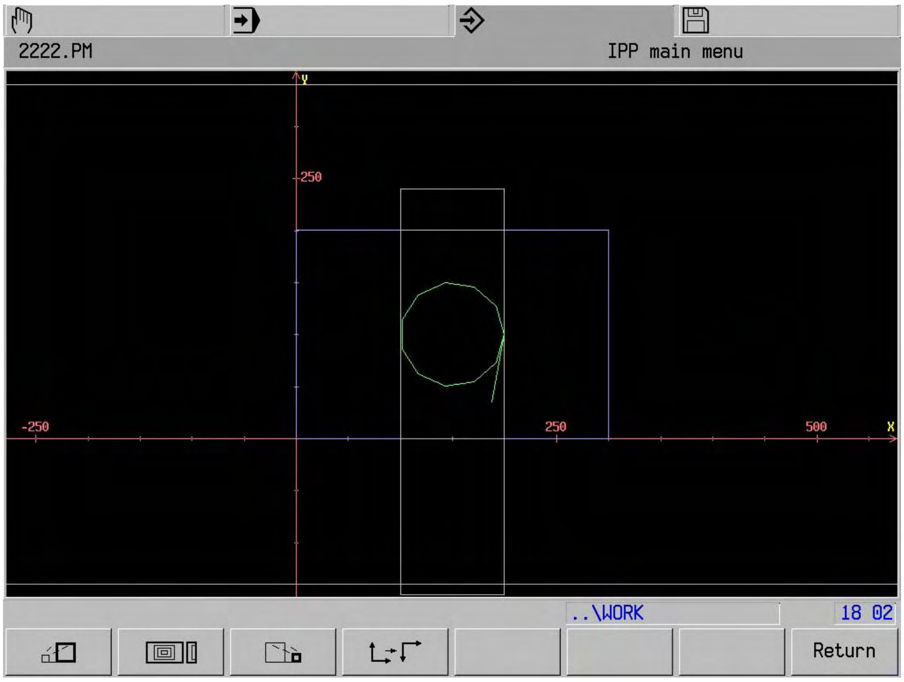
For example, it is now possible to change an address value.
Enter address values.



The feature is directly generated.

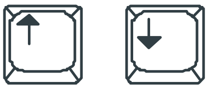


Checking changes with graphic.



Have all changes in the program been made?

If not, select next feature.



Next feature.

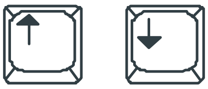
Note

If a feature is changed within an IPP program block, the entire IPP program block

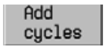
must be run through. Changes that have been made will be accepted in the subsequent features of the IPP program block.

21.5.2 Inserting a feature

When an IPP feature is inserted, the feature is inserted after the position indicated.



Select insert position of feature.



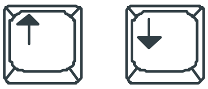
Defining a feature and entering program data.

Note

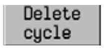
Macro number 8000 is suggested for pocket milling. If the macro number already exists, change the number.

21.5.3 Delete feature

When deleting an IPP feature, all the accompanying instructions in the program are deleted.

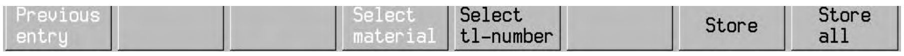


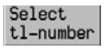
Select the feature to be deleted.

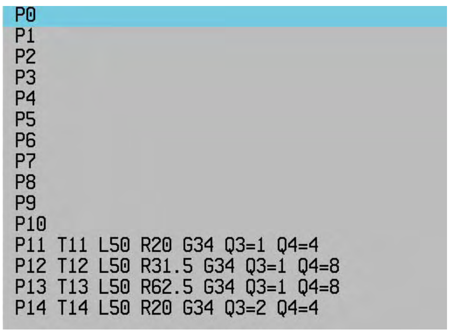


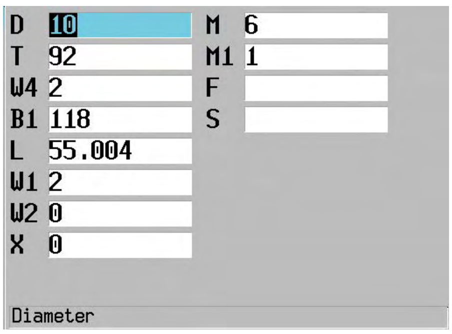
The feature to be deteted is cleared immediately.

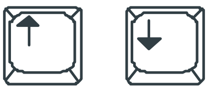
21.5.4 Select tool during editing











Select tool.

Enter

Store — Copy tool in dialogue box.

21.5.5 Graphic display of contour (test run)

Check the parts program briefly for the right sequence and to see whether it is correct.

Convert plane			Change cycle	Add cycles	Delete cycle		Save on disk

							Return

Return — Return to input.

21.5.6 Executing IPP programs

Before executing a parts program the operator must:

Enter all the tools created with the aid of IPP into the magazine and the current tool table.

21.5.7 Converting active plane G17 <-> G18

Programs in IPP are basically made in active plane G17 (XY plane). If the machining operation on the machine has to be carried out in active plane G18 (XZ plane), the program must first be converted from G17 to G18. Reconversion is possible.
Editing is also only possible in G17.

Convert plane

Convert G17->G18

Convert G18->G17

21.6 IPP programming tips

21.6.1 Using ICP to define contours

Once one of the options for the freely designed pocket contour or the contour recess has been selected, ICP is loaded automatically.

The program is checked beforehand to see whether a traversing movement has been programmed at least for the X- and Y-axes. If not, the user is requested to enter a traversing movement.

21.6.2 IPP proposals

The proposals generated during data entry in IPP are based on the table data stored in the CNC (tool and technology tables) and on a special IPP start macro. The proposals made in the IPP start macro can be adapted to suit individual requirements.

21.6.3 Maximum feed rates and spindle speeds

The feed rates and spindle speeds proposed in IPP operation are calculated from the data contained in the technology tables. If the limitations of the machine tool used are not taken into account in this process, the possibility exists that the proposed feed rates and spindle speeds will exceed the maximum values applicable to this machine tool.

For this reason the data stored in the technology tables should take account of the limitations of the machine tool used.

The machine constant memory contains the maximum allowable values of the feed speeds and spindle speeds for this machine tool.

21.6.4 Optimizing programming and machining times

1. Centre drill hole, change tool and drill. Repeat operation for each drill hole.
2. Centre all drill holes, change tool and finish all drill holes.

Note

Always decide on the optimization strategy **before** IPP programming, never afterwards!

21.6.5 Changing IPP programs using the DIN editor

We would like to advise you to change all IPP-generated programs with the aid of IPP. If this should not be possible or desirable, then it is easy to change programs manually thanks to the standard DIN code programs generated by IPP.

Program changes which are made manually are lost if a feature which has been changed manually is later modified in the IPP mode "Change line", to be precise for the reason that IPP deletes the entire feature and regenerates it.

22. Program structure and block format

22.1 Program extract

```
%PM9001
N9001
N1 G17 S630 T1 M6
N2 G54
N3 G0 X60 Y30 Z-8 M3
N4 G1 Z-10 F50
N5 G43 X80 F100
N6 G42
:
M30
```

22.2 Memory identifier

Main program: program number.PM or %PM
Subprogram: program number.MM or %MM

22.3 Program number

N1 - N9999999

22.4 Program block

A program block consists of a several program words (max. 255 characters). Each address can only be used once in the program block.

1	Block number	N1
2	Geometric commands	G17 S630
3	Technological commmands (S,F,T,M)	T1 M3
Total		N1 G17 S630 T1 M3

22.5 Block number

N1 - N9999999
The sequence of the block numbers is unimportant.
The blocks are executed in the programmed sequence.

22.6 Program word

Address, sign, number
(Positive sign can be deleted)

Positive word	X21.43
Negative word	Y-13.8
Indexed word	X1=15.3
Calculated word	=12.5+30
	Y=2^5
	Y=sqrt(25)

22.7 Input formats for axis addresses

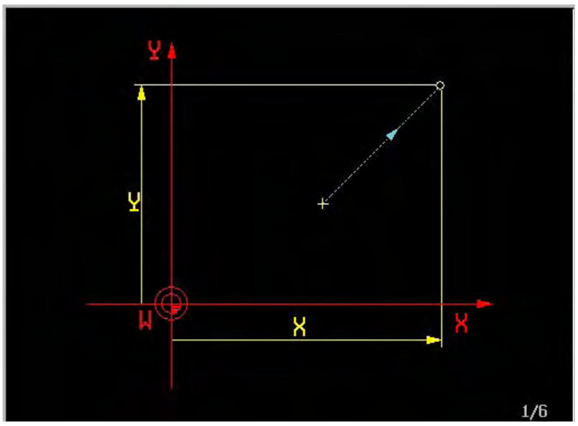
Metric	6.3	X123456.789
Inch	5.4	X12345.6789

23. G-Functions

23.1 Rapid traverse G0-

N... G0 [axis coordinates]

Parameters

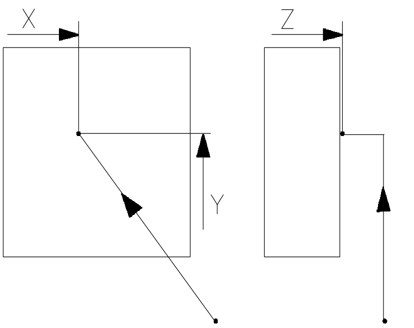


G	Rapid traverse
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
B	Endpoint angle
C	Endpoint angle
B1=	Angle
B2=	Polar angle
?90=	Endpoint abs. (X,Y,Z..)
?91=	Endpoint incr. (X,Y,Z..)
L1=	Path length
L2=	Polar length
P1=	Point definition number

Example

N... G0 X25 Y15 Z30

Simultaneous movement in main plane XY, followed by movement in tool axis Z



Notes

At the start of a program and following each change of tool or swivel head, each active axis must be programmed in a program block for traversing movements. Every axis is thus in the start position.
The positioning logic determines the sequence of the traverse movements in rapid traverse.

Tool movement:	to workpiece G17,18,19			away from workpiece G17,18,19		
1st axis movement	4.+5	4.+5	4.+5	Z	Y	X
2nd axis movement	X+Y	X+Z	Y+Z	X+Y	X+Z	Y+Z
3rd axis movement	Z	Y	X	4.+5.	4.+5.	4.+5.

23.2 Linear interpolation G1

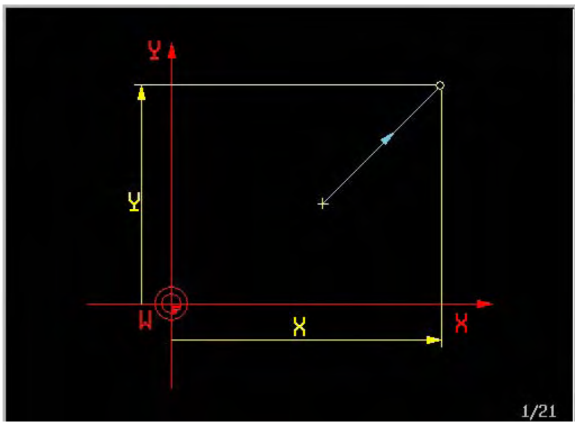
Linear interpolation in the main level:
N.. G1 {X..} {Y..} {Z..} {F..}

3 D interpolation:
N.. G1 X.. Y.. Z.. {F..}

One axis of rotation:
N.. G1 {A..} {B..} {C..} {A40=..} {B40=..} {C40=..} {F...}

Several axes:
N... G1 {X..} {Y..} {Z..} {A..} {B..} {C..} {A40=..} {B40=..} {C40=..} {F...}

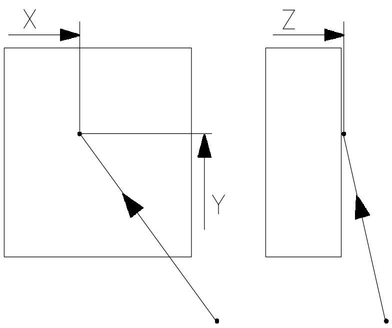
Parameters



G	Linear interpolation
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
B	Endpoint angle
C	Endpoint angle
B1=	Angle
B2=	Polar angle
?90=	Endpoint abs. (X,Y,Z..)
?91=	Endpoint incr. (X,Y,Z..)
L1=	Path length
L2=	Polar length
P1=	Point definition number
P2=	Point definition number
P3=	Point definition number
P4=	Point definition number

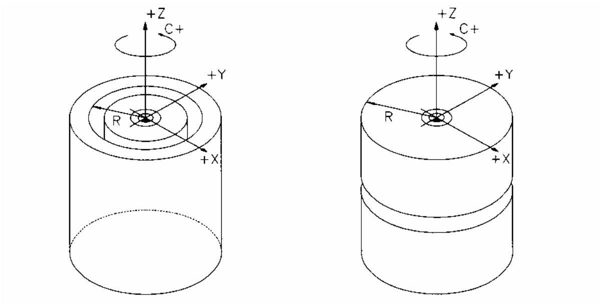
Examples

3 D interpolation

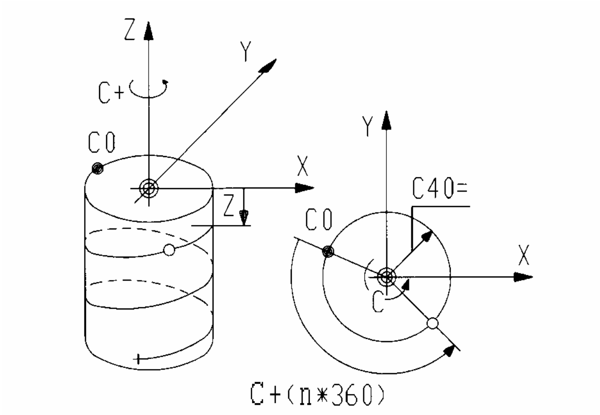


N14 G0 X10 Y5 Z20
N15 G1 X20 Y10 Z40 F100 Simultaneous movement of the axes

Programming rotary axes, with and without linear axis

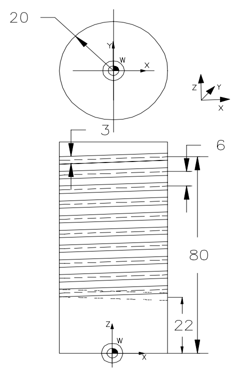


One rotary axis and one linear axis:



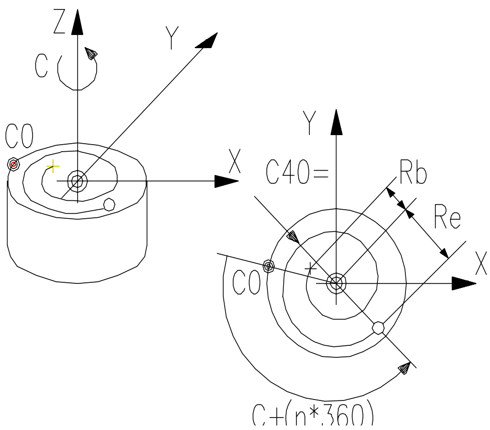
Z and C axis
(X and A axis)
(Y and B axis)

Thread on a cylindrical surface:



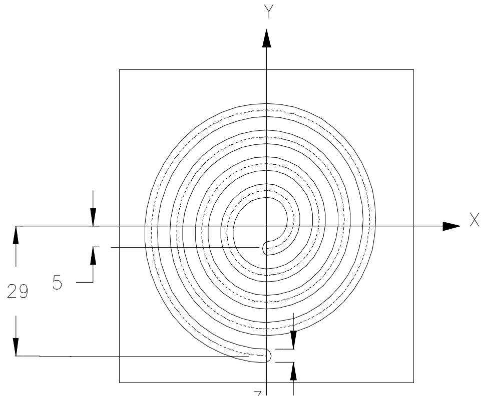
:
N10 G18
N11 T1 M6 S2000 F200 Change tool
N12 G0 X0 Z80 Y22 C0 M3
N13 G1 Y18 Set tool to required position
N14 Z20 C3600 C40=18 Mill spiral, 10 turns
N15 G0 Y25
:

Linear axis with additional rotary axis:



C40=..(central path radius)
C40=(Rb+Re)2
Rb(start radius)
Re(end radius)

Spiral:



:

N10 G17 T1 M6	Change tool
N11 G54	Zero offset
N12 G0 X0 Y-5 Z3 C0 S200 M3	
N13 G1 Z-2 F100	Approach start position
N14 Y-29 C1440 C40=17 F200	Mill spiral, 4 turns
N15 G0 Z100	
:	

Note:

MACHINES WITH KINEMATIC MODEL
The radius of the rotating axis is calculated automatically in machine tools with a kinematic model. A40=, B40= or C40= no longer need to be programmed. The new option is programmed with G94 F5=1.

23.3 Circular clockwise / circular counter clockwise G2/G3

Full circle:

N.. G2/G3 [centre point]

Circular arc less than or equal to 180:

N.. G2/G3 [end point] R..

Circular arc greater than 180 :

N.. G2/G3 [centre point] [end point]

N.. G2/G3 [centre point] B5=..

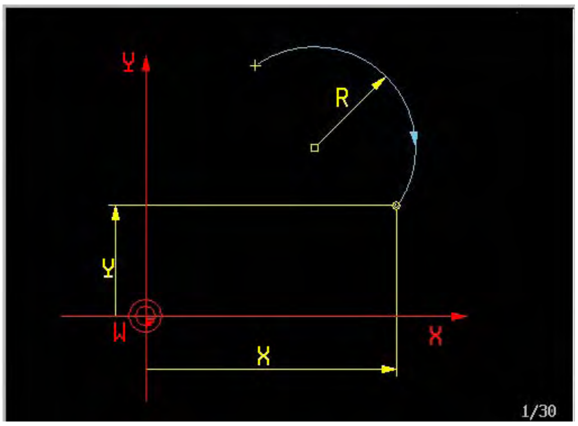
2.5D-Interpolation:

N... G2/G3 [centre point] [end point of arc] [end point on the linear or rotary axis]

Spiral:

N... G2/G3 [centre point] [end pointof arc] [end point on the linear or rotary axis] [pitch]

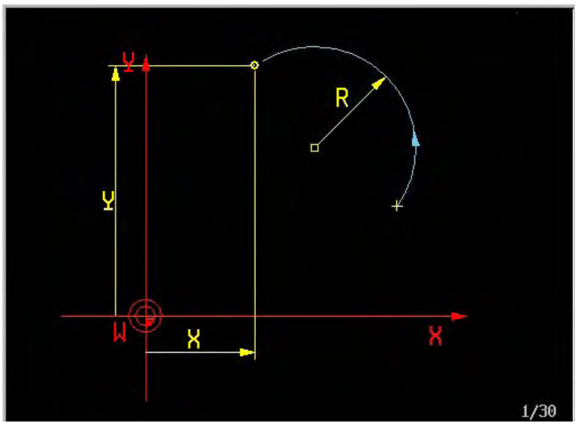
N... G2/G3 [centre point] [pitch] B5=...



G	Circular counter clockwise
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
B	Endpoint angle
C	Endpoint angle
I	Center point in X / pitch in X
J	Center point in Y / pitch in Y
K	Center point in Z / pitch in Z
R	Circle radius
B1=	Angle
B2=	Polar angle
B3=	Polar angle for center
B5=	Angle of arc
?90=	End-/centrep. abs. (X,Y,Z..I,J,K)

G2

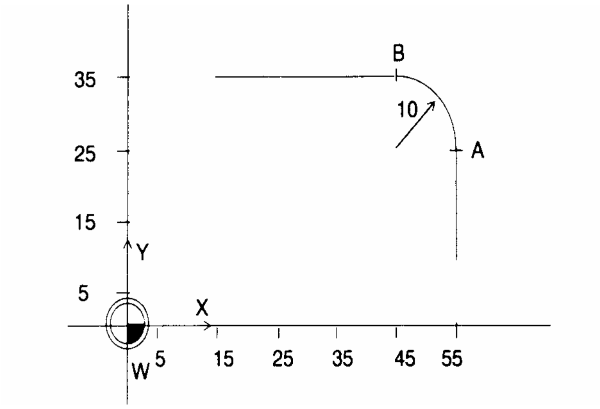
?91=	End-/centrep. incr.(X,Y,Z..I,J,K)
L1=	Path length
L2=	Polar length
L3=	Polar length for center
P1=	Point definition number



G3

Examples

Circular arc less than or equal to 180°

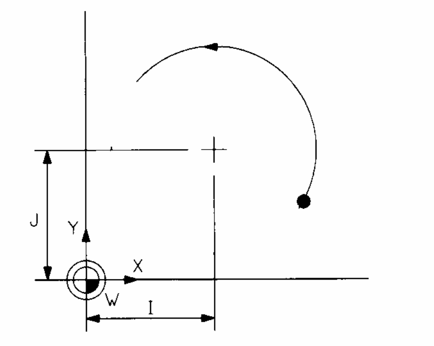


N10 G1 X55 Y25 F100 Linear movement
N20 G3 X45 Y35 R10 Circular counterclockwise movement

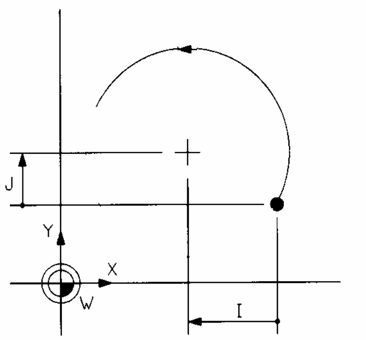
Circular arc greater than 180°

Centre point coordinates:
G17
N.. G2/G3 I.. J..
G18
N.. G2/G3 I.. K..
G19
N.. G2/G3 J.. K..

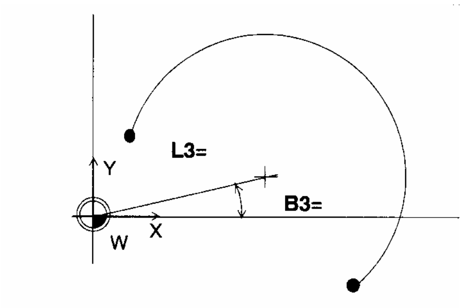
Absolute centre point coordinates (G90):
Centre point coordinates in relation to program zero point



Incremental centre point coordinates (G91):
Centre point coordinates in relation to the start point

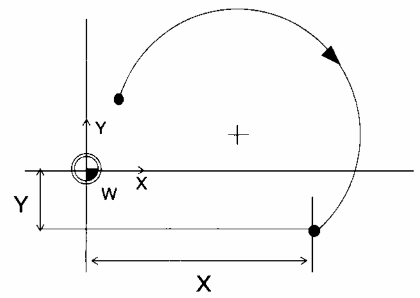


Polar centre point coordinates
N.. G2/G3 L3=.. B3=..   (G17/G18/G19)

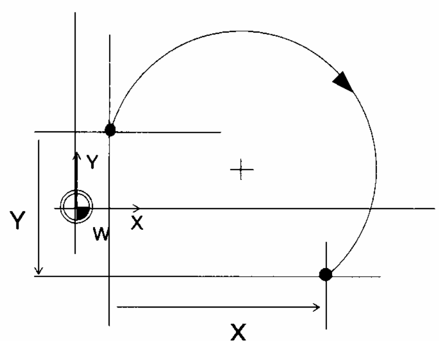


End point coordinates:
Cartesian end point coordinates
G17
N.. G2/G3 X.. Y..
G18
N.. G2/G3 X.. Z..
G19
N.. G2/G3 Y.. Z..

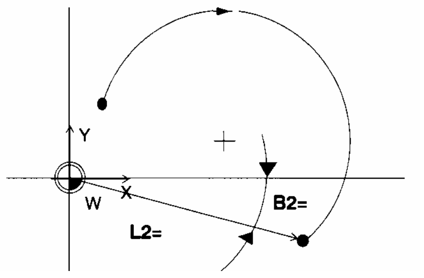
Absolute end point coordinates (G90):
End point coordinates in relation to the program zero point



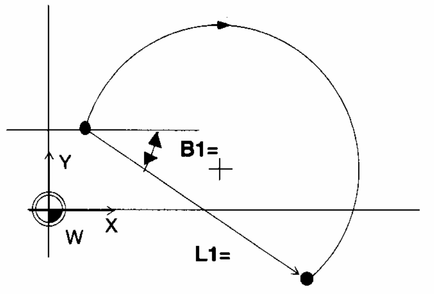
Incremental end point coordinates (G91):
End point coordinates in relation to the start point



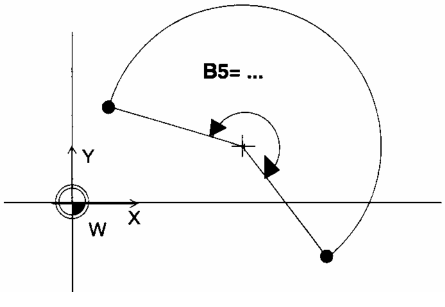
Polar end point coordinates:
End point coordinates in relation to the program zero point
N.. G2/G3 L2=.. B2=..   (G17/G18/G19)



End point coordinates in relation to the start point
N.. G2/G3 L1=.. B1=..   (G17/G18/G19)



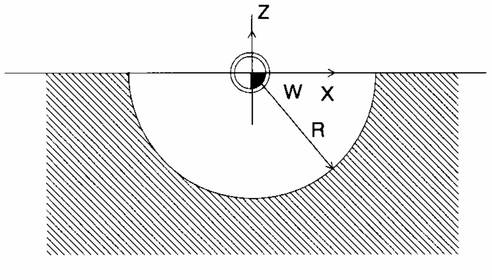
Angle of circular arc:
N2.. G2/G3 B5=..   (G17/G18/G19)



Circular movement not in the main level
Circular arc smaller than or equal to 180°:
- N2.. G2/G3 [end point coordinates of the linear axes] R..
- N2.. G2/G3 [cartesian coordinates of circle centre point]

Circular arc greater than 180°:
- N2.. G2/G3 [cartesian coordinates of the end point andcircle centre point]

Use of the radius correction is not possible.



Circular movement with simultaneous movement in a third axis (2.5D)

Circle in the main level:
N.. G2/G3 [circle definition][tool axis]

Level	G17	G18	G19
Tool axis	Z	Y	X

Circle not in the main level:
N.. G2/G3 [cartesian coordinates of end point and circle centre point] [tool axis]

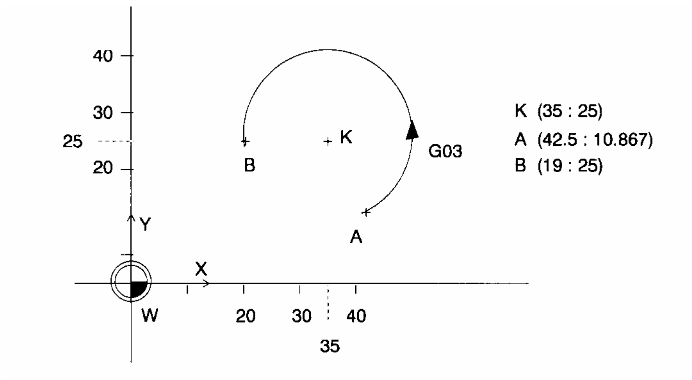
Level	G17	G18	G19
End point	X..Y..	X..Z..	Y..Z..
Centre point	I..J..	I..K..	J..K..
Tool axis	Z	Y	X

Spiral interpolation

Level	G17	G18	G19
Tool axis	Z	Y	X
Centre point	I..J..	I..K..	J..K..
	/	/	/
	B3=..L3=..	B3=..L3=..	B3=..L3=..
Circular arc angle	B5=..	B5=..	B5=..
Spiral pitch	K	J	I

The value of (B5=) can lie between 0 und 999999 degrees (approx. 2777 rotations)

Level	G17	G18	G19
Tool axis	Z	Y	X
Circle end point	X..Y..	X..Z..	Y..Z..
Centre point	I..J..	I..K..	J..K..
Spiral pitch	K	J	I



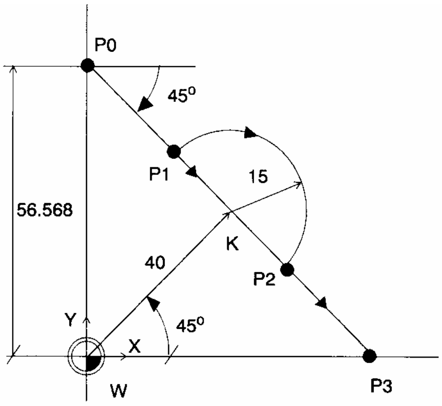
Absolute coordinates
N82000
N1 G17
N2 G98 X0 Y0 Z10 I60 J60 K-30
N3
N4 G0 X0 Y0 Z-10
N5
N6 G1 X42.5 Y10.867 F200 Linear movement
N7 G3 X19 Y25 I35 J20 Circular counterclockwise movement (absolute)
N8
N9 G0 Z100 M30

Incremental coordinates

N82001
N1 G17
N2 G98 X0 Y0 Z10 I60 J60 K-30
N3
N4 G0 X0 Y0 Z-10
N5

N6 G1 X42.5 Y10.867 F200	Linear movement
N7 G91	Incremental size programming
N8 G3 X-23.5 Y14.133 I-7.5 J9.133	Circular counterclockwise movement (incremental)

N9
N10 G0 Z100 M30



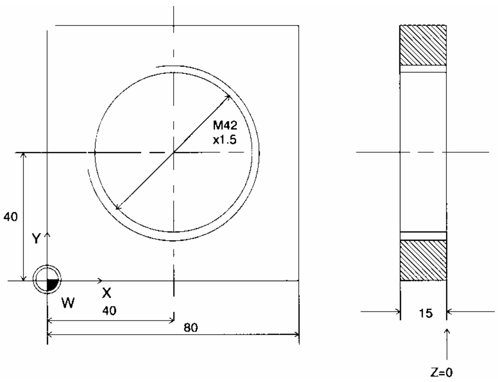
N82030
N1
N2 G17

N3 G98 X-10 Y-10 Z10 I80 J80 K-30	Definition of graphic window

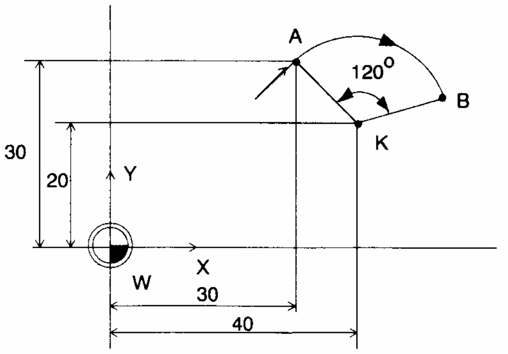
N4
N5 G0 X0 Y56.568 Z0
N6 G1 F200 B1=-45 L1=25

N7 G2 B1=-45 B3=45 L1=30 L3=40	Circular clockwise movement

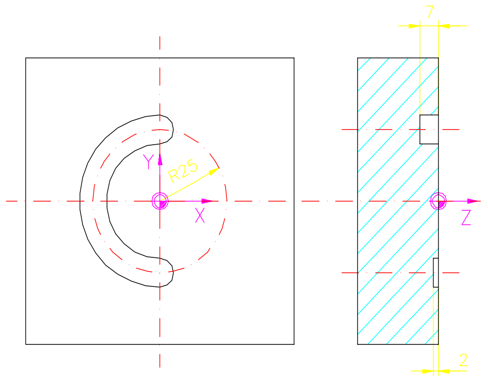
N8 G1 B1=-45 L1=25
N9
N10 G0 Z100 M30



```
N82040
N10 G17 T1 M6                   Active plane, change tool
N11 G0 X40 Y40 Z1.5 S400 M3
N12 G1
N13 G43 Y61 F120                Tool radius compensation to end point
N14 G42                         Tool radius compensation to the right
N15 G2 I40 J40 K1.5 B5=4320     Circular clockwise movement (thread)
N16 G40                         Cancel tool radius compensation
N17 G1 Y40
N18 G0 Z100 M30
```




```
N10  G1  X30  Y30  F500
N11  G2  I40  J20  B5=120       Circular clockwise movement
```



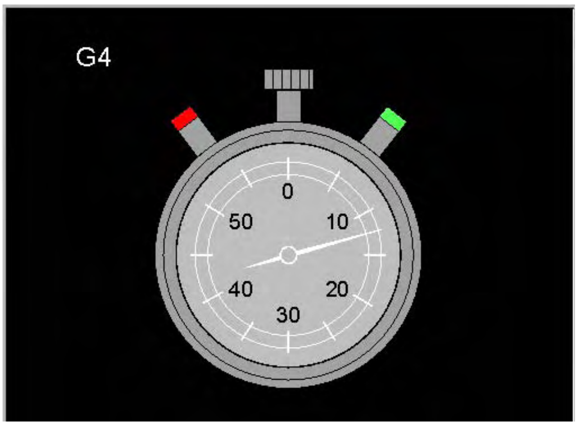

```
N85770
N1 G17
N2 G54
N3 G98 X20 Y50 Z10 I-100 J-100 K-20
N4
N5
N6 S650 T1 M6                   Change tool
N7 G0 X0 Y-25 Z5 M3             Spindle ON clockwise; rapid traversing movement
N8 G1 Z-2 F100                  Move to machining depth
N9 G2 X0 Y25 Z-7 I0 J0 F200     Circular clockwise movement
N10 G1 Z5                       Retract tool from material
N11
N12
N13 M30
```

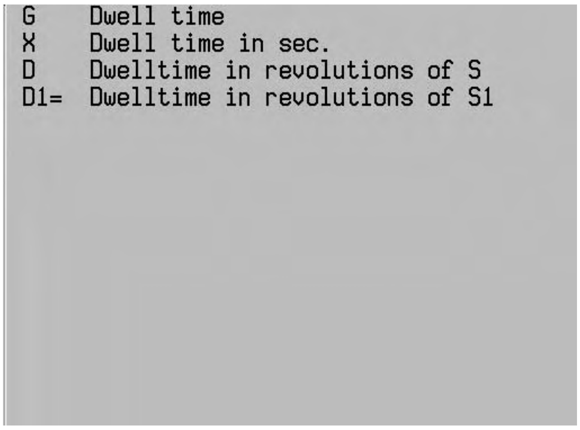
23.4 G4 Dwell time

During the execution of a program inserting a dwell period (time or number of revolutions).

Format

G4 X.. or D.. or D1=..




G Dwell time
X Dwell time in sec.
D Dwelltime in revolutions of S
D1= Dwelltime in revolutions of S1

Notes and usage

Input values

Dwell period (D):	0,1 - 900 Seconds (15 Minutes).
Revolutions (D1=):	0 - 9.9

Example

N50 G4 X2.5	The above block causes a dwell of 2.5 seconds between two operations
N60 G4 D2	The above block causes a dwell of 2 revolutions of the spindle between two operations

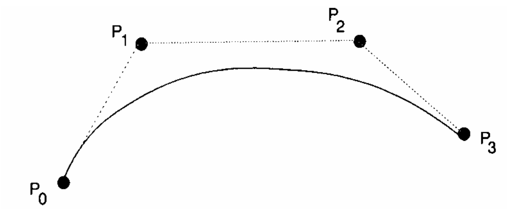
23.5 Spline Interpolation G6

Spline interpolation allows the programmer to create a uniform and smooth curve by entering a few points.

Formats with Bezier splines

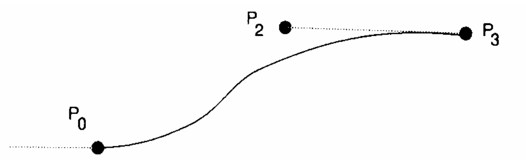
Spline with three vertices:

G6 X61=.. Y61=.. Z61=.. X62=.. Y62=.. Z62=.. X.. Y.. Z..



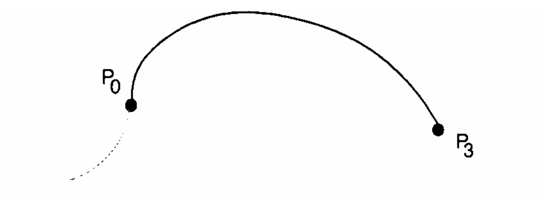
Spline with two vertices and constant tangents to the spline:

G6 X62=.. Y62=.. Z62=.. X.. Y.. Z..

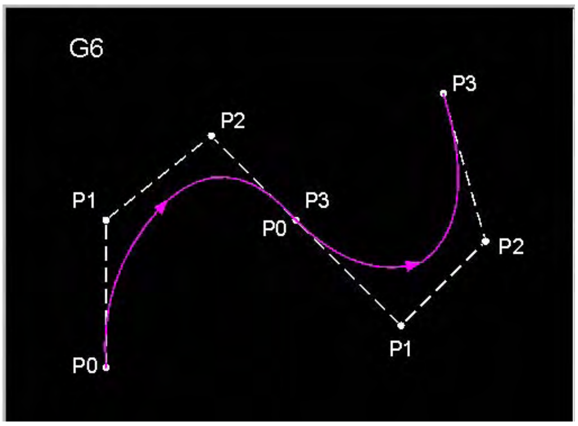


Spline with constant curvature to the previous spline:

G6 X.. Y.. Z..



Parameters Bezier-Splines



G	Spline interpolation
X	Endpoint (X-axis)
Y	Endpoint (Y-axis)
Z	Endpoint (Z-axis)
X51=	First spline coefficient
Y51=	First spline coefficient
Z51=	First spline coefficient
X52=	Second spline coefficient
Y52=	Second spline coefficient
Z52=	Second spline coefficient
X53=	Third spline coefficient
Y53=	Third spline coefficient
Z53=	Third spline coefficient
X61=	First support point (X-axis)
Y61=	First support point (Y-axis)
Z61=	First support point (Z-axis)
X62=	Second support point (X-axis)
Y62=	Second support point (Y-axis)
Z62=	Second support point (Z-axis)

Formats with cubic splines

Spline with all coefficients defined:

G6 X51=.. Y51=.. Z51=.. X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant tangents to the previous spline:

G6 X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant curvature to the previous spline:

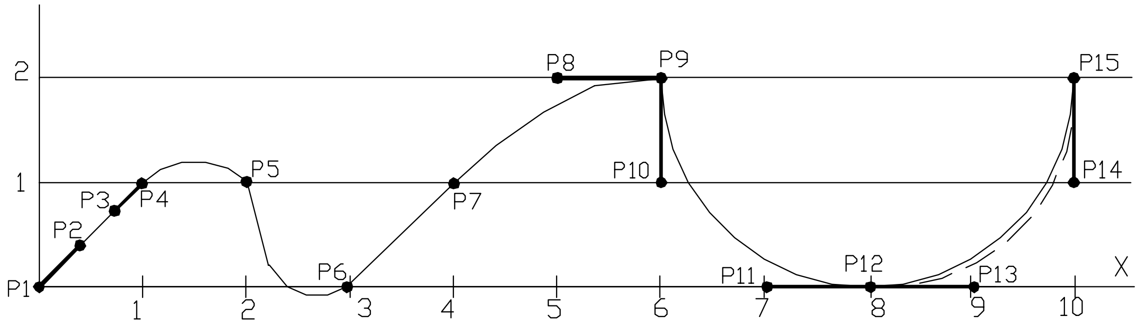
G6 X53=.. Y53=.. Z53=..

Parameters

Cubic splines

X51=, Y51=, Z51=	First spline coefficient
X52=, Y52=, Z52=	Second spline coefficient
X53=, Y53=, Z53=	Third spline coefficient

Example: Bezier splines



N17001 (Spline Kurve)
N1 G98 X2 Y-6 Z-2 I10 J10 K10
N2 G17
N101 G0 X0 Y0 Z0 F500
N102 G6 X1 X61=0.3 X62=0.7 Y1 Y61=0.3 Y62=0.7 Z0.001 Z61=0 Z62=0
N103 X2 Y1.001 Z0
N104 X3 Y0 Z0.001
N105 X4 Y1 Z0
N106 X6 X62=5.7 Y2 Y62=2 Z0.001 Z62=0
N107 X8 X61=6 X62=7.5 Y0 Y61=1.5 Y62=0 Z0 Z61=0 Z62=0.001
N108 X10 X61=8.5 X62=10 Y2 Y61=0 Y62=1.5 Z0.001 Z61=0.001 Z62=0
N109 G0 X0 Y0 Z0
N110 M30

N101: Approach start position (P1)
N102: First element. Straight line. Touches P1-P2 and P3-P4. End point is P4. All coordinates must be entered. For this purpose, select a straight line.
N103: Curve passes through P5
N104: Curve passes through P6
N105: Curve passes through P7. If the curve does not have the required shape, add more points.
N106: Curve passes through P9 and touches line P8-P9.
N107: New curve with sharp transition is defined. First curve element starts in P9 and touches P9-P10 and P11-P12. End point is P12.
N108: New curve with tangential transition is defined. First curve element starts in P12 and touches P12-P13 and P14-P15. End point is P15. The radius of curvature can be adjusted in P15 by changing distance P14-P15.

Note:

In G6 identical coordinates must be different in two blocks (Z0 and Z0.001)

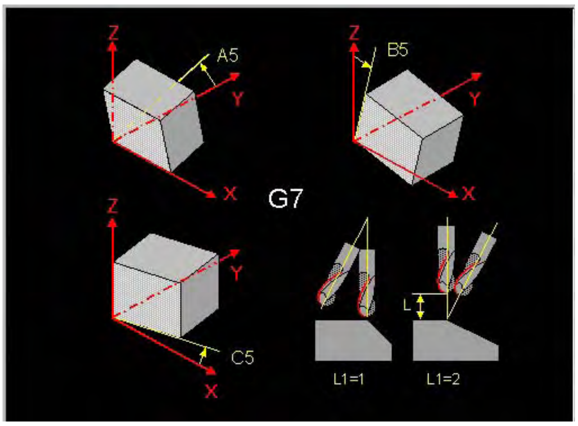
23.6 Tilt operating planes G7

Programming of a tilt operating plane for four or five axis machines.
The position of the operating plane can be tilted using the function "Tilt operating plane". The operation which has then been programmed in the principal plane (G17, G18) can then be implemented within the tilt operating plane. The tool axis is then orientated vertically in the new plane.
The tilt of the operating planes is defined and implemented using the G7 function.

Format

N.. G7 {A5=.. | A6=..} {B5=.. | B6=..} {C5=.. | C6=..} {A7=..} {B7=..} {C7=..} {B47=..} {L1=..} {L2=..} {L..}

Parameters




```
G    Tilting working plane
L    Tool length offset
B47= E-par. for rotation mainplane
A5=  Angle of rotation absolute
B5=  Angle of rotation absolute
C5=  Angle of rotation absolute
A6=  Angle of rotation incremental
B6=  Angle of rotation incremental
C6=  Angle of rotation incremental
B7=  E-par. for position in B
C7=  E-par. for position in C
L1=  0=No move.,1=rot.axes,2=tooltip
L2=  -/+1,2,3 = Neg/Pos A,B,C angle
```

Ancillary function

G FUNCTIONS WHICH ARE NOT PERMITTED, IF G7 IS SWITCHED ON
If G7 is switched on, the following (modal) G functions are not allowed to be active:
G6, G9, G19, G41, G42, G43, G44, G61, G64, G73, G141, G182, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208
If G7 is switched on, the following (modal) G functions with the mentioned addresses are not allowed to be active:
G54 I1 B4=... and G93 B4=...

G FUNCTIONS WITHIN G7 WHICH ARE NOT PERMITTED
If G7 is active, the following G functions are not permitted:
G6, G19, G182

G FUNCTIONS WHICH ARE NOT PERMITTED, IF G7 IS SWITCHED OFF
If G7 is switched off, the following (modal) G functions are not allowed to be active:
G9, G41, G42, G43, G44, G61, G64, G73, G141, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208
If one of these G functions which are not permitted is active, the fault report P77 "G function and Gxxx are not permitted" is generated.

Type of function

modal

Notes and employment

G7 FUNCTION
The freely programmable operating plane is defined using the new G7 function:
The new plane becomes active with the original null point.
The tool is orientated vertically in the new plane. The axes which move depend upon the machine configuration and the programming.
The display shows the coordinates in the new (tilt) plane.
The manual operation is orientated in accordance with the new plane.

SPACE ANGLE

A5=, B5=, C6= defines the absolute angle, by which the operating plane is rotated about the corresponding positive axis.

A6=, B6=, C6= defines the incremental angle, by which the operating plane is rotated about the corresponding positive axis.

Value falls between –359.999 and 359.999 degres.

DEFINITION OF NEW OPERATING PLANES

Tilting of the operational plane can be defined in two ways:

- Programming with A5=, B5= or C5= parameters. In this way, the absolute tilts about the corresponding positive axes are defined. The tilts are implemented as follows:
  1. the active G7 tilt is raised
  2. C5= tilt about the machine fixed positive Z axis
  3. B5= tilt about the positive Y axis
  4. A5= tilt about the positive X axis

- Programming with A6=, B6= or C6= parameters. The incremental tilts about the current corresponding positive axes are defined in this way. The tilts are implemented as follows:
  1. C6= tilt about the current G7 positive Z axis
  2. B6= tilt about the current G7 positive Y axis
  3. A6= tilt about the current G7 positive X axis

The programming is independent of the machine configuration. The plane tilt is calculated with reference to the current null point. The movement is dependent upon the machine configuration.

QUERY A CALCULATED ANGLE POSITION

A7=, B7=, C7= Holds the number of the E-Parameters, in which the computed angle of the corresponding rotary axis is set.

B47= Contains the number of the E-Parameter, in which the computed angle of the main plane is set.

ALTERNATIVE TILTING POSSIBILITIES WITHIN MOVING RANGE OF THE MACHINE

The CNC checks, which tilting possibilities within the moving range of the rotary axes are possible (to the left or to the right).

- No tilting possibilities, than error message is given (P307)
- By only one tilting possibility this will be executed.
- By two tilting possibilities, those with the shortest movement will be executed (L2=0 or not programmed). The shortest movement is not always possible.

With the address L2= can be controlled, which tilting possibilities must be executed. By L2=1/2/3 the A/B/C-axis is positioning so, that a positive angle will be reached. By negative L2= a negative angle will be reached.

TOOL VERTICAL ON THE DEFINED TILT PLANE

The G7 tilt movement takes place interpolating with the power traverse. It tilts the tool axis to the defined plane. The axes which are moved depend upon the type of movement L1=:

- L1=0 The rotary axes do not move (start position).

Comment:

The tilt movement can then be implemented, using the E parameters loaded into A7=, B7= or C7=. This movement must then be programmed manually.

- L1=1 Interpolate only the rotary axes, which do not move the linear axes.
- L1=2 Interpolate the rotary axes and to that end execute a "compensatory movement". In this way the tool tip remains in the same position with respect to the workpiece.

TOOL LENGTH ALLOWANCE
If the tilting motion takes place about the tool tip (L1=2), I defines an allowance in the tool direction between the programmed endpoint and the tool tip.

SWITCHING OFF THE G7 FUNCTION
The operation of G7 remains active until G7 is switched off. G7 is switched off by the programming of G7 without parameters or by G7 L1=1 positioning of the rotary axes on the workpiece null point.

G7 is not switched off by M30 or <Program interrupt>. After switching on the control G7 is permanently active. Travel in the G7 plane is then possible. G7 is switched off in accordance with reference point travel or <Reset CNC>.

Note:

It is recommended that, at the start of every G7 program, that a G7 without parameters is programmed. In this way, during the start-up of the program (interrupt within the tilted plane and the new start), the plane is always reset. Without this G7 at the start, the first part of the program will be implemented in the tilted plane rather than in the untilted plane.

This programming is similar to programming with G17/G18 - different null points or different tools.

ROTARY AXES
Rotary axes can be programmed in the tilted planes in the normal way. It is the programmer's responsibility to ensure compatibility of the rotary axes with the G7 tilt.

ABSOLUTE POSITION G74
If G7 is active, the "Absolute position" G74 is referred to the machine coordinates. This is the same as in V3.3x.

GRAPHICS
The graphics display the G7 plane as the main picture. The screen is refreshed whenever G7 becomes active.
If G7 is active, the position between tool and workpiece is displayed.

DISPLAY
If G7 is active, a yellow icon is displayed on the screen behind the tool number. By means of a small "p" on the right next to the "axes characters", an indication is given as to whether the display relates to the tilted operating planes or to the machine coordinates. The operating status has been enhanced with the current reading of the programmed G7 space angle.

A new soft key (Jog to the G7 plane) appears in the "Jog operation type" soft key group. This soft key is used to switch between the tilted operating planes and the machine coordinates. If the position is displayed in machine coordinates, the actual position of the tool tip is shown.

CHANGE OF TOOL
If G7 is active, a tool change is not permitted (fault report). G7 must first be deselected. G7 must then be selected again, in order to resume work in the tilted plane following the tool change.

Example:

```
N100 G7 B5=45 L1=1        (plane is set)
N110 T14                  (tool preselect)
..
N200 G0 Z200              (the tool axis is withdrawn)
N210 G7 B5=0 L1=1         (deselect G7)
N220 M6                   (tool change)
N230 G0 X.. Y.. Z..       (power traverse to the new start position)
N240 G7 L1=1 B5=45        (face is rotated again in the G7 plane)
```

PALETTE, TILT FACE OR TOOL CHANGE
While G7 is active a change of palette, tilt face or tool cannot be implemented. A fault is displayed and the program must be interrupted. Before such a change, G7 must be deactivated.

TILT OPERATING PLANE WITH M53/M54
During mixed operation with G7 and M53/M54, the tilt face positioning M53/M54 with M55 must be relinquished before programming G7. In this way, the face offset which is active under these conditions is relinquished.

M FUNCTIONS, WHICH ARE NOT PERMITTED IF G7 IS SWITCHED ON
If G7 is switched on, the following M functions are not permitted to be active:
M53, M54

M FUNCTIONS, WHICH ARE NOT PERMITTED WITHIN G7
If G7 is active, the following M functions are not permitted:
M6, M46, M53, M54, M60, M61, M62, M63, M66

FAULT REPORTS

P77 G-function and Gxxx not allowed.
This error message indicates, that a combination of G-functions is not allowed. For example: When G7 will be programmed and G41 is already active, an error message P77 is given.

P306 Plane not clearly defined
The G7 plane is defined with a mix of absolute (A5=, B5=, C5=) and incremental (A6=, B6=, C6=) angles.
Solution: Use only absolute or incremental angles. If required, several G7 definitions with incremental angles behind one another can be defined.

P307 Programmed plane not attainable
The defined G7 tilt position, owing to the restricted range of the rotary axis, cannot be attained.

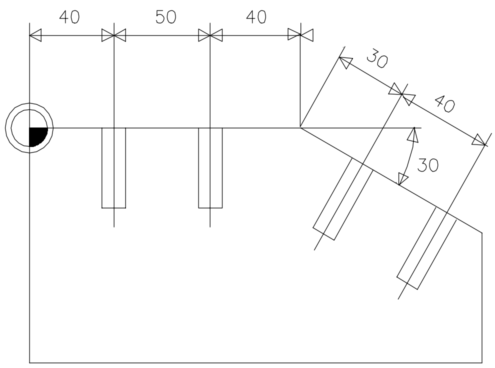
MACHINE SETTINGS

MC312 free operating plane (0=off, 1=on)
activates the free operating plane. The G7 function can be programmed.

MC755 free operating plane: rotation (0=coordinates cross,1=axes)
If the desired rotation of the operating plane is compatible with the rotation of a rotary axis, an adjustment may be made to determine whether the relevant rotary axis or the coordinates cross is rotated.

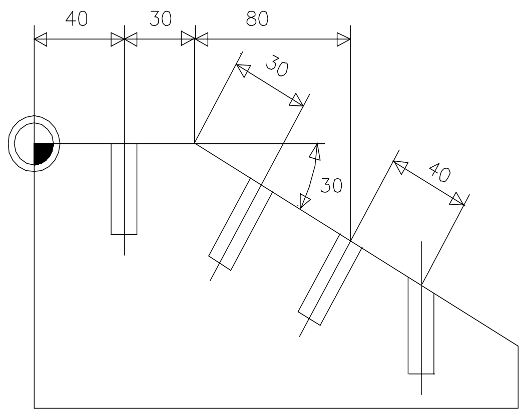
e.g. on a machine with (real C axis) the programming G7 C5=30 and MC755=0 generates a rotation of the coordinates cross through -30° and, if MC755=1, a rotation of the C axis through 30°.

Example 1 Workpiece with tilted operating plane.



N10 G17	define operating plane
N20 G54	null point insertion
N30 M55	deselection of M53/M54
N40 G7 L1=1	reset G7
N..	
N100 G81 Y1 Z-30	drill cycle definition
N110 G79 X40 Z0	drill the first hole in the horizontal plane
N120 G79 X90	drill the second hole in the horizontal plane
N..	other movements in the horizontal plane
N200 G0 X130 Z50	tool is set at the safety distance.
N210 G93 X130	null point is set at the start of the tilted operating plane.
N220 G7 B5=30 L1=2 L50	G7 define new operating plane B5=30 angle of rotation L1=2 tool/table is rotated about the tool tip L50 extra oversize in the direction of the tool. In this way, the tool is rotated about the null point. The distance from the tool tip to the null point is 50 mm.
N230 G79 X30 Z0	drill the first hole in the tilted operating plane
N240 G79 X70	drill the second hole in the tilted operating plane
N..	other movements in the tilted operating plane
N300 G7 L1=2 L50	reverse rotation in the horizontal plane.

Example 2 Workpiece using tilted operating plane.



N10 G17	define operating plane	
N20 G54	null point insertion	
N30 M55	deselection of M53/M54	
N40 G7 L1=1	reset G7	
N..		
N100 T1 M6	change the drill	
N110 G81 Y1 Z-30	define the drilling cycle	
N120 G79 X40 Z0	drilling of a hole in the horizontal plane	
N..	other movements in the horizontal plane	
N200 T2 M6	change the mill	
N210 X70 Z50	tool is set at the safety distance	
N220 G93 X70	null point insertion	
N230 G7 B5=30 L1=2 L50	define new operating plane	
	B5=30	angle of rotation
	L1=2	tool/table is rotated about the tool tip
	L50	extra oversize in the direction of the tool. In this way the tool is rotated about the null point. The distance of the tool tip from the null point is 50 mm.
N240 G1 X0 Z0	positioning of the mill in the tilted plane	
N250 X150	mill in the tilted plane	
N..	other movements in the tilted operating plane	
N300 T1 M6	change the drill	
N310 G79 X30 Z0	drill the first hole in the tilted operating plane	
N320 G93 X=80:cos(30)	null point insertion	
N330 G79 X0 Z0	drill the second hole in the tilted operating plane	
N..	other movements in the tilted operating plane	
N400 G93 X=40	null point insertion	
N410 G0 X0 Z50	tool is set at the safety distance	
N420 G7 B5=0 L1=2 L50	deselect "Tilt operating plane" reverse rotation in the horizontal plane	
	B5=0	angle of rotation
	L1=2	tool/table is rotated about the tool tip
	L50	extra oversize in the direction of the tool. In this way, the tool is rotated about the null point. The distance of the tool tip from the null point it 50 mm.
N430 G79 X0 Z0	drill the third hole in the horizontal operating plane	
N..	other movements in the horizontal operating plane	
N500 M30	end of program.	

23.7 Tilting of the operating plane

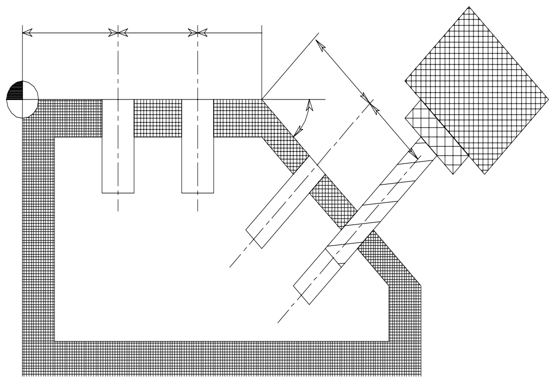
23.7.1 Introduction

The control supports the tilting of operating planes on tool machines with tilting faces or tilting tables. Please consult your user handbook.

Typical applications, for example, are oblique drilling or contours which lie obliquely in the operating area. In this way, the operating plane is always tilted about an active null point. Normally, the operation is programmed in a principal plane, e.g. X/Y plane; however, it is executed in the plane which was tilted to the principal plane.

Consult the description of the G7 function for the programming of the freely programmable operating plane.

The tilting of the operating plane is defined and implemented using the G7 function. The G7 function is made up of two components:
- definition of new operating planes, rotation of the coordinate system.
- in the event that it is programmed, tilt the tool vertically to the defined operating plane.



An operation on an oblique workpiece plane is programmed in local coordinates. In this way, the local X and Y coordinates lie in the oblique plane and the Z coordinate is fixed vertically in the plane.
The machine recognizes the link between the local coordinates and the true machine axes and calculates this. The control calculates the tool correction factor.

Millplus distinguishes two machine types during tilting of the operating plane:

1) Machine with a tilting table
The location of the transformed machine axis is not changed with reference to the coordinates system fixed in the machine. If you rotate your table, and thus the workpiece, through 90°, for example, the coordinate system is not rotated through 90° with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+.
2) Machine with a tilting face
the location of the tilted (transformed) machine axis is changed with reference to the coordinates system fixed in the machine:
If you rotate the tilting face of your machine and hence the tool, e.g. in the B axis about +90°, the coordinates system is rotated with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+ and X+ of the coordinates system fixed in the machine.

Using the G7 function you define the location of the operating plane by the input of tilt angles. The angles entered describe the angular components of a space vector.

If you program the angular components of the space vector, the control automatically calculates the angular position of the tilt axes. MillPlus calculates the location of the space vector and thus the location of the spindle axis, by means of rotation about the coordinates system fixed in the machine. The sequence of rotations for the calculation of the space vector is fixed: MillPlus turns the A axis first, next the B axis and finally the C axis.

The G7 function is effective from the start of its definition in the program.

MillPlus can only position controlled axes automatically.
In the G7 definition, you can, in addition, input a safety distance to the tilt angles, with which the tilt axes are positioned.
Use only pre-set tools (full tool length in the tool table).
During the tilting process the position of the tool tip opposite the workpiece remains virtually unchanged (depending on the type of movement L1=).

MillPlus implements the tilting process using the power traverse.

23.7.2 Machine types

Milling machines with four or five axes can be used for the oblique machining of a workpiece.
Depending on the plane which is tilted, other types of machine are needed for the working. At least two rotary axes and three linear axes are needed, in order to reach all sides and planes (except the under surface) without the need for remounting.

The possible types of machine are:

90° tilting face and turntable
: The tilting face can be in two states. The upper and reverse sides can be worked by means of the tilting face. The four side surfaces can be worked using the turntable (C axis). The machine is only suited to all oblique operating planes if the tilting face can also be set in the oblique position manually.

Double turntable
: The tables (A and C axes) are stacked. In this way, all sides and oblique operating planes can be worked.

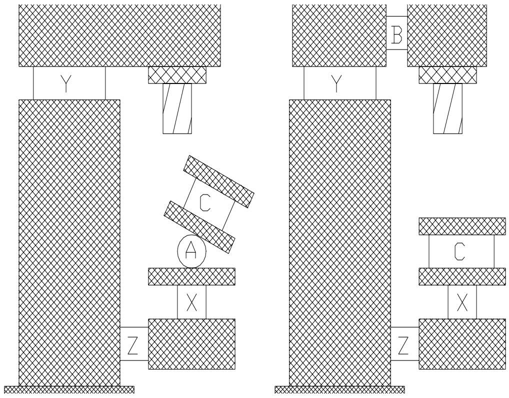
Double turntable and 45° tilting face
: The tables (A and C axes) are stacked. The A axis is limited in its angular movement. In conjunction with the two tilting face states all sides and oblique operating planes can be worked.

45° double turntable
: The tables (B and C axes) are stacked. The B axis is fixed in this way at an angle of 45°. All sides and oblique operating planes can be worked.

Turntable and rotating face
: The face (B axis) can be freely positioned. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

Turntable and 45° rotating face
: The face (B axis) is set at an angle of 45°. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

Outline of the most suitable machine types for use with oblique operating planes.

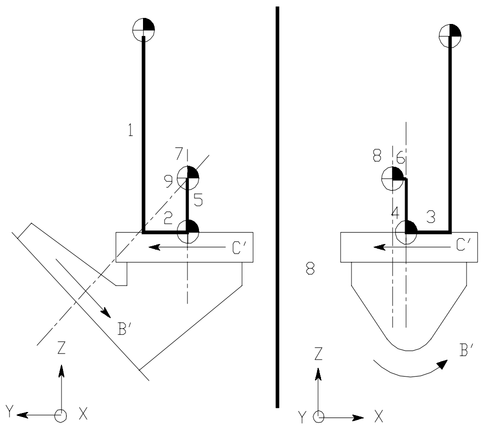


23.7.3 Kinematic model

The control needs a kinematic model of the machine in order to reset the programmed local coordinates in the oblique plane to the movements of the machine axes. A kinematic model describes the "construction" of the axes and the exact position of the different turning points on the rotary axes.

As an example, a kinematic model of the DMU 50 V machine is shown.The kinematic model is made up of a chain stretching from the work piece to the machine frame. It is not necessary to describe the chain from the workpiece to the machine frame, because it includes no rotary axes.

Kinematic model for the DMU 50 V



Explanation of the drawing:

-1,2,3	three elements in the X, Y, and Z directions in order to fix the (absolute) centre position of the workpiece table with reference to the marker positions.
-4	element for definition of the C axis. It is only necessary to describe the rotating axis of a rotary axis, not the centre point.
-5,6	two elements in order to obtain the rotating axis of the second (incremental) rotary axis.
-7	element for definition of the (incremental) direction of the second rotating axis. This direction is -45° in the A axis (all around the X axis).
-8	element for definition of the B axis.
-9	element in order to raise the -45° tilt (Element 7) again. In this way, the kinematic chain ends without rotation.

The kinematic model is entered by means of the machine settings MC600 to MC699.

To determine the connection between the operating planes and the positions of the axes, the stacking and the exact position of the different rotating points of the rotary axes are needed. A description of this stacking is called the kinematic model. The kinematic model is defined in the form of two "chains". One chain defines the axis stacking of the tool as far as the machine frame, the other chain from the workpiece to the machine frame. In this way, it is only necessary to describe a chain if it contains rotary axes.
A kinematic chain defines, by means of displacements and tiltings, the way in which the rotary axes lie with respect to one another. Every displacement or tilting is determined as an element of the kinematic chain in three machine settings. Thus, a total of 25 elements of the kinematic chain can be determined. All rotary axes and positioning axes which are present should be described.

Only the machine types with rotary axes in the X, Y or Z direction, in which the sequence of rotary axes from workpiece to tool is as follows, are supported:

- A C
- C A
- C B
- C A fixed B -A fixed (DMUxxV and DMCxxU in which A is fixed = -45°)
- C
- Changed axis variants (C becomes B, and B becomes C) are also possible.

If other types of machine are entered, the error report 0256 "machine type not recognized" is generated.

23.7.4 Manual operation

The axes are used along the local coordinates within the tilted G7 plane. E.G. Jogging skip of the Z axis moves the tool vertically in the plane. All true linear machine axes can move in this way.

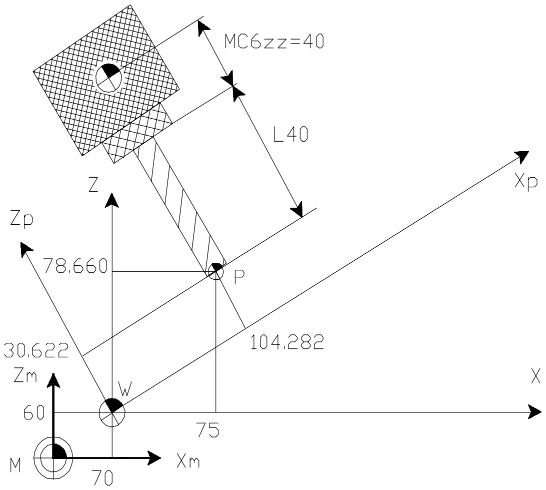
By means of a soft key, operation can be switched to the use of the true machine axes. The display then changes to show the true machine axes.

The user keys and the hand wheels for the linear axes can be assigned, according to choice, to the G7 plane or to the machine axes. The display is then implemented also in the G7 or machine axes plane. A new soft key in the soft key group <Step/continue> is used to implement the choice between G7 level or machine axes. For this purpose, this new soft key offers a "pause" option between the choices of jogging skip <advance> and <continue> soft key.

23.7.5 Display

If G7 is active, a yellow icon is displayed on the screen behind the tool number. By means of a small "p" on the right next to the "axis characters" an indication is given as to whether the display relates to the tilted operating planes or to the machine coordinates. The operating status has been enhanced with the current state of the programmed G7 space angle.

It is possible, using a new soft key in the "Jog operation type" soft key group, to change the display at the same time as the jog direction. If the position is displayed in machine coordinates, the position of the true tool tip is shown. See the next screen:



The position display on the screen can change between the position in the G7 plane (Xp,Zp) or in machine coordinates (X,Z).
Both are based on the active null point G52 + G54 + G92/G93.

23.7.6 Axis selection/positioning axis

An axis which is not regulated must be set to the correct position by hand. However, either before or after this, the oblique setting of the tool must also be entered by means of G7. Otherwise it will not be checked.

Comment: The expected position of the rotary axes is set parametrically in G7 using n7=<parameter number>. An axis selection or a positioning axis can be set manually using this information.

The axis selection or the positioning axis should also be followed in the kinematic model.

23.7.7 Reference point

If the reference point is approached during G7, the rotary axes remain in their reference position following the approach. The G7 plane is deactivated and the G17 plane is activated.

After running up the machine, but before approaching the reference point, the G7 plane is still active.
After < reset CNC> the G7 plane is deactivated.

23.7.8 Interruption

If the G7 movement is interrupted, the exact position of the tool tip is displayed on the screen. Following interruption, the axes can be used in manual operation [mode].

Following <Start> a move in position back to the interrupted point is effected. At the same time the axes run with positional logic corresponding with the G7 plane. Concurrently, the rotary axes rotate to the initial position.

23.7.9 Fault reports

P306 plane not clearly defined

The G7 plane is defined using a mix of absolute angles (A5=, B5=, C5=) and incremental angles (A6=, B6=, C6=).

Solution: Use only absolute or incremental angles. If necessary, several G7 definitions with incremental angles can be defined, one behind the other.

P307 program plane not attainable

The G7 oblique setting defined cannot be attained, on account of the limited range of the rotary axes.

Solution: Machines with a tilting face should tilt the face (by means of the M function) from the instantaneous position (horizontal or vertical) following the other position

O256 Machine type not recognized

The kinematic model in MC600 to MC699 is defining a type of machine which is not supported for the oblique operating plane (G7). Only machine types with the following sequence of rotary axes, as viewed from the workpiece to the to tool, are supported:

- A C
- C A
- C B
- C A fixed B -A fixed
  (A fixed is a fixed tilt in the direction of the A axis, as, for example, the DMU50V has with -45°)
- C
- Axis change variants (C becomes B, and B becomes C) are also possible.

Solution: The kinematic model should be entered in detail, with at least a description of those rotary axes present. The control must be run up once more.

23.7.10 Machine settings

MC312 free operating plane (0=off, 1=on)

activates the free operating plane. The G7 function can be programmed.

MC600 - MC699

There are 100 new machine settings (MC600 – MC699) for the description of the kinematic model. The model is described using a maximum of 25 elements, in which each element is described by means of four machine settings.
The following machine settings are used:

MC600 Kinematic chain (0=end,1=tool,2=workpiece)

MC601 Element (0,1=X,2=Y,3=Z,4=A,5=B,6=C)

MC602 Element type (0=incremental,
1=absolute)

MC603 Element insertion [:m/mdegrees]

MC604, 608, 612, 616, 620, …. , 696 as MC600
MC605, 609, 613, 617, 621, …. , 697 as MC601
MC606, 610, 614, 618, 622, ... , 698 as MC602
MC607, 611, 615, 619, 623, …. , 699 we MC603

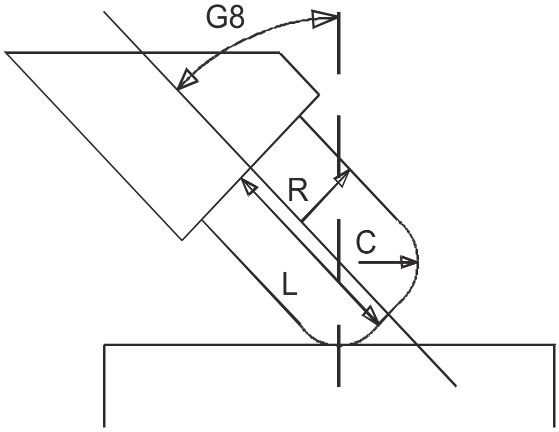
MC755 Free operating plane: rotation (0=coordinates cross,1=axes)

If the desired rotation of the operating plane corresponds with the rotation of a rotary axis, the control has the choice between rotten using the relevant rotary axis or rotation using the coordinates cross. This choice is made with MC755.
E.G on a machine with a (true) C axis the program instruction G7 C5=30 and MC755=0 produces a rotation of the coordinates cross over -30° and MC755=1 a rotation of the C axis over 30°.

23.8 Swivel tool G8

To program a swivelled tool for four or five-axis machines.

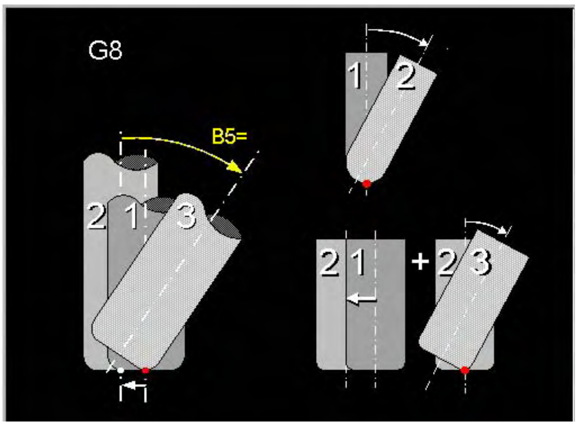
With the function "Swivel tool" the tool axis can be set obliquely relative to the machining plane. This makes angle milling possible and substantially improves cutting conditions and thus surface quality. The programming of G8 is identical to G7. G7 should therefore also be read.



L, R and C from the tool table.

N.. G8 {A5=.. | A6=..} {B5=.. | B6=..} {C5=.. | C6=..} {A7=..} {B7=..} {C7=..} {L} {L1=..} {L2=..} {L3=..} {F}

Parameters



G	Tilting tool orientation
L	Tool length offset
A5=	Angle of rotation absolute
B5=	Angle of rotation absolute
C5=	Angle of rotation absolute
A6=	Angle of rotation incremental
B6=	Angle of rotation incremental
C6=	Angle of rotation incremental
B7=	E-par. for position in B
C7=	E-par. for position in C
L1=	0=No move.,1=rot.axes,2=tooltip
L2=	-/+1,2,3 = Neg/Pos A,B,C angle
L3=	Radius compensation (0=on, 1=off)
F6=	Block feed

Associated function

The following G functions are not permitted when G8 is active:
G6, G19, G40, G41, G42, G43, G44, G141, G180, G182

The rotation of the tool direction can be defined in two ways:
absolute

- Programming with A5=, B5= or C5= parameters. The absolute rotations about the corresponding positive axes are defined by this. The rotations are calculated as follows:
  1. the active G8 rotation is cancelled
  2. C5= rotation about the positive Z axis fixed with respect to the machine
  3. B5= rotation about the positive Y axis
  4. A5= rotation about the positive X axis

incremental

- Programming with A6=, B6= or C6= parameters. The incremental rotations about the corresponding current positive axes are defined by this. The rotations are calculated as follows:
  1. C6= rotation about the current G8 positive Z axis
  2. B6= rotation about the current G8 positive Y axis
  3. A6= rotation about the current G8 positive X axis

Programming is independent of the machine configuration. The plane rotation is calculated relative to the current zero point. The motion is dependent on the machine configuration.

SCANNING A CALCULATED ANGULAR POSITION

A7=, B7=, C7= Contains the number of the E parameter in which the calculated angle of the corresponding rotary axis is set.

ALTERNATIVE TILTING POSSIBILITIES WITHIN MOVING RANGE OF THE MACHINE

The CNC checks, which tilting possibilities within the moving range of the rotary axes are possible (to the left or to the right).

- No tilting possibilities, than error message is given (P307)
- By only one tilting possibility this will be executed.
- By two tilting possibilities, those with the shortest movement will be executed (L2=0 or not programmed). The shortest movement is not always possible.

With the address L2= can be controlled, which tilting possibilities must be executed. By L2=1/2/3 the A/B/C-axis is positioning so, that a positive angle will be reached. By negative L2= a negative angle will be reached.

SWIVEL MOTION

The G8 swivel motion is performed by interpolating with feed (F6=). It swivels the tool axis onto the defined plane. Which axes move depends on the type of motion L1= :

- L1=0 The rotary axes do not move (default).

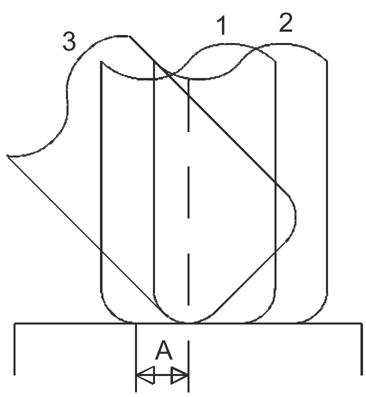
G8 SELECTABLE TOOL RADIUS CORRECTION

L3=0 with radius correction (standard value)
L3=1 no radius correction

Note: The swivel motion can be programmed or performed manually by means of the E parameters that are loaded with A7=, B7= or C7=.

- L1=1 Only the rotary axes swivel, while the linear axes do not move.
- L1=2 The rotary axes swivel and the linear axes perform a movement. This means that the contact point position remains X, Y, Z.

If the contact point lies on the tool corner radius, the movement is only a rotation.
If the contact point is the tool tip and the corner radius (C) is smaller than the tool radius (R), a compensating movement occurs so that the contact point is shifted from the tool tip to the corner radius.



With cylindrical cutters (with corner radius C < cutter radius R), the following particular point applies:
When swivelling from the vertical (1) to the oblique (2--> 3) position or vice versa, the contact point moves from the centre of the cutter to the corner radius (A) and vice versa. A compensating movement at the tool tip ensures that the current contact position X, Y, Z remains unchanged.

TOOL LENGTH_ALLOWANCE
If the swivel motion takes place about the tool contact point (L1=2), L defines an extra allowance in the tool direction between the rotation point and the tool tip.

TOOL CORRECTION
During the function "swivel tool" (G8) the values L, R and C for the tool are corrected.
This G8 tool correction is independent of G40, G41, G42, G43, G44 and is always effective.
At the beginning and end of the tool correction, a compensation movement is carried out if the corner radius (C) is smaller than the tool radius (R).
If the tool dimensions (L,R,C) change with G8 active, the current position of the linear axes is re-calculated.

TURNING OFF THE G8 FUNCTION
G8 remains active until it is cancelled. G8 is cancelled by programming G8 without angle parameters.

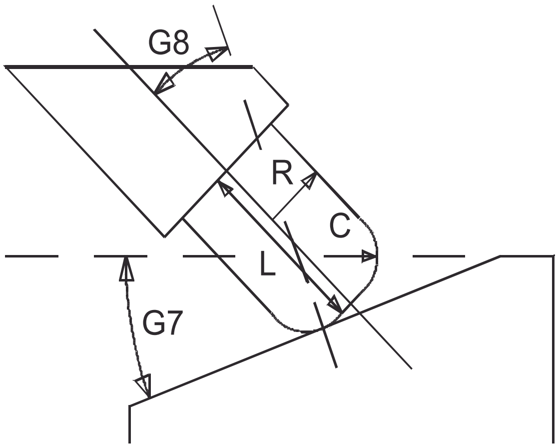
G8 is not cancelled by M30 or <program abort>. After the control is turned on, G8 is still active. After search for reference points or <CNC reset> G8 is cancelled.

Note: At the start any program that uses G8, we recommend that a G8 is programmed with no parameters. This ensures that the tool direction is always reset as the program is starting up (abort with swivelled tool and new start). Without this G8 at the beginning, the first part of the program is executed in the swivelled instead of the unswivelled plane.
The programming is similar to programming with G7/G17/G18 - different zero points or different tools.

CONFIGURATION
Swivel tool (G8) can be used for machines where a kinematic model is defined and entered. See description of the kinematic model.

DISPLAY
When G8 is active, a yellow icon is displayed in the display behind the tool number.
A small 'p' on the right, next to the 'axis letters', is used to display whether the position of the tool tip is displayed or the position in machine coordinates.

Example 1 Workpiece with oblique machining plane and oblique tool.



N10 G17
N20 G54
N30 M55
N40 G7 L1=1
N50 G8 L1=1
..
N100 G0 X130 Z50
N110 G93 X130
N120 G7 B5=-30 L1=2
N130 G8 B5=30 L1=2
..
N200 G8
N210 G7 L1=2

Key:

N10	Define machining plane	
N20	Zero point offset	
N30	Deselect M53/M54	
N40	Reset G7	
N50	Reset G8	
N100	Tool set to safety distance	
N110	Zero point set to the beginning of the swivelled machining plane.	
N120	G7	Define new oblique position of the tool.
	B5=-30	Angle of rotation
	L1=2	Tool/table revolves about the tool tip
N130	G8	Define new oblique position of the tool.
	B5=30	Angle of rotation
	L1=2	Tool rotates about the tool tip and a compensation movement is performed.
N200	Turn tool perpendicular to the machining plane again (rotary and compensation movement).	
N210	Rotate back to the horizontal plane.	

23.9 Defining polar point (measurement reference point) G9

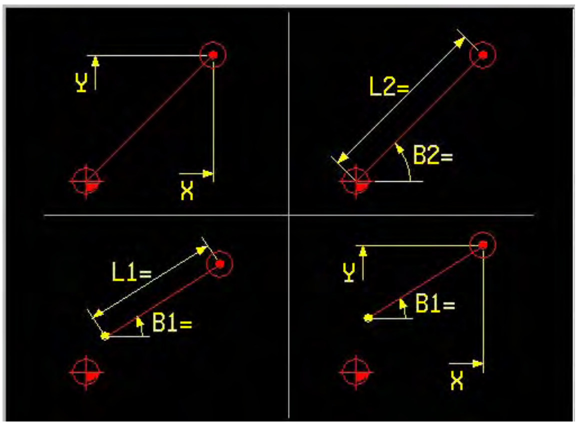
Programming a polar point. If a polar point has been programmed, program blocks with polar programming (angle and length) no longer relate to the zero point but to the last programmed polar point.

N.. G9 X.. Y.. {X90=...} {X91=...} {Y90=...} {Y91=...} {Z90=...} {Z91=...}

N.. G9 X0 Y0 Deactivate pole (same as workpiece zero point)

N.. G9 B2=.. L2=.. {B1=..} {L1=..} (polar point in polar coordinates)

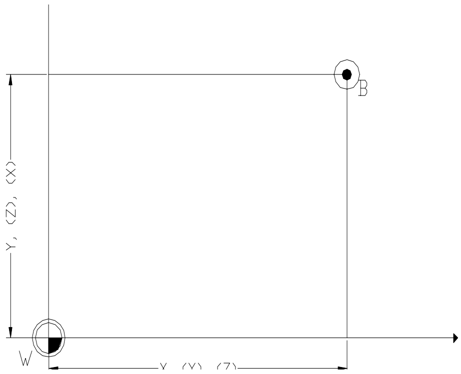
Parameters



G	Define pole position
X	Pole coordinate
Y	Pole coordinate
Z	Pole coordinate
B1=	Angle
B2=	Polar angle
?90=	Pole coordinate abs. (X,Y,Z..)
?91=	Pole coordinate incr. (X,Y,Z..)
L1=	Path length
L2=	Polar length

Notes and usage

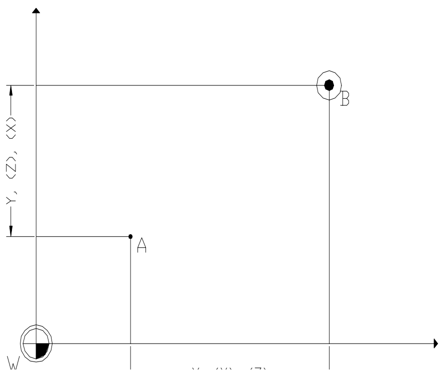
Polar point in absolute coordinates:



B = polar point

N.. G9 X.. Y..

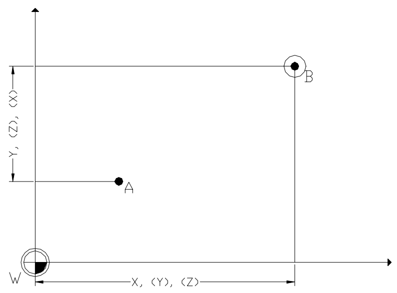
Polar point in incremental coordinates:

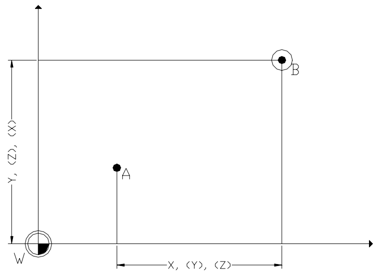


A = existing polar point
B = new polar point

N... G9 X91=... Y91=...

Polar point in combined absolute/incremental coordinates:

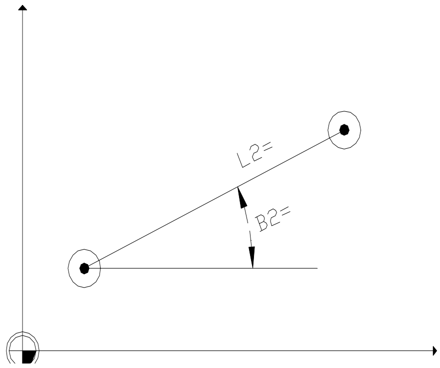




N... G9 X... Y91=...

A = existing polar point B=new polar point
N.. G9 X91=.. Y..

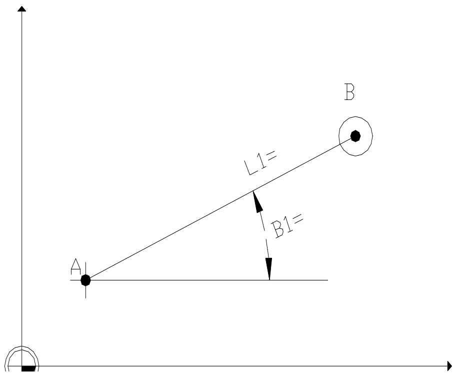
Polar point in absolute polar coordinates:



A = existing polar point
B = new polar point

N.. G9 B2=.. L2=..

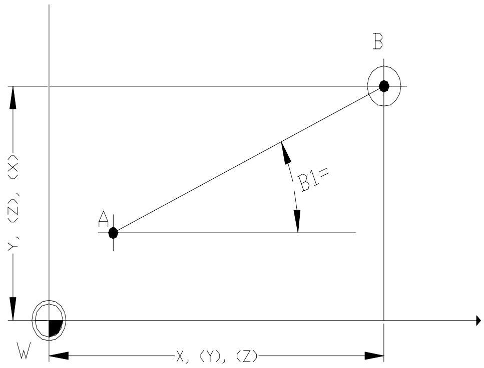
Polar point in incremental polar coordinates:



A = end point of last movement
B = new polar point

N.. G9 B1=.. L1=..

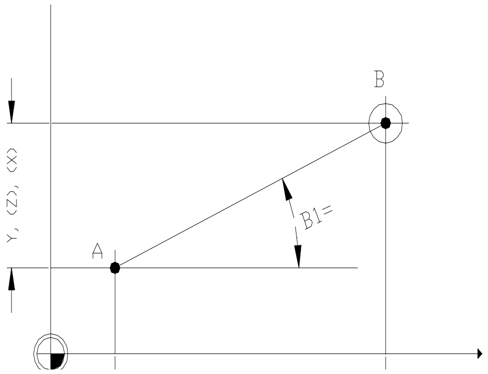
Combined programming: Cartesian absolute/polar:



A = existing polar point
B = new polar point

N.. G9 X.. B1=..

Combined programming: Cartesian incremental/polar:



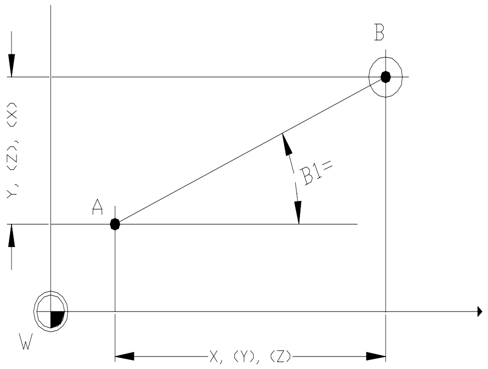
A = existing polar point
B = new polar point

N.. G9 X91=.. B1=..

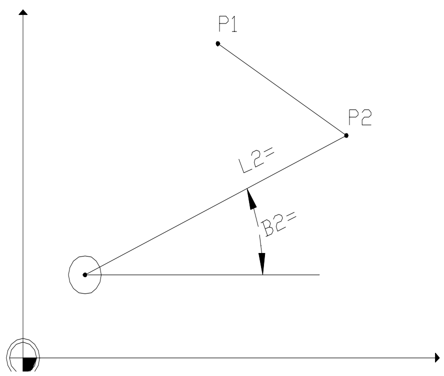
- pole definitions are only permitted in the active working plane
- before the G9 block is called, the polar point is located at the workpiece zero point (polar point = 0)
- when the plane is changed using G17, G18, G19, the polar point is zeroed (0).

Polar end point definition:
During absolute polar programming polar length L2= or L3= and polar angle B2= or B3= no longer relate to the zero point, but to the polar point.

Polar point definition

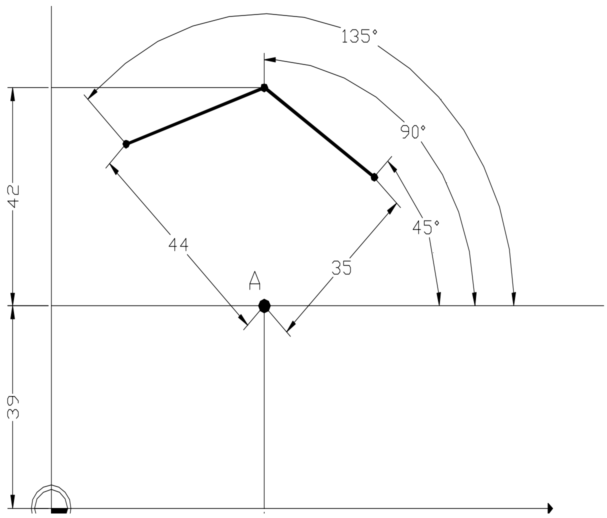


Polar circle definition
In G2 and G3 blocks polar programming with polar point of centre and end point is possible.



ICP/geometry calculation G64
G1, G2 and G3 blocks with B2=, B3= and L3= programming may be programmed in G64 and ICP. They relate to the active polar point. However, the polar point itself can only be changed in G64, **not** in ICP.

Example



A = new polar point

N30 G9 X48 Y39	Definition of new polar point
N40 G1 B2=135 L2=44	Definition of end point coordinate related to the new polar point
N50 G1 B2=90 L2=42	
N60 G1 B2=45 L2=35	

23.10 Linear Chamfer or Rounding Cycle G11

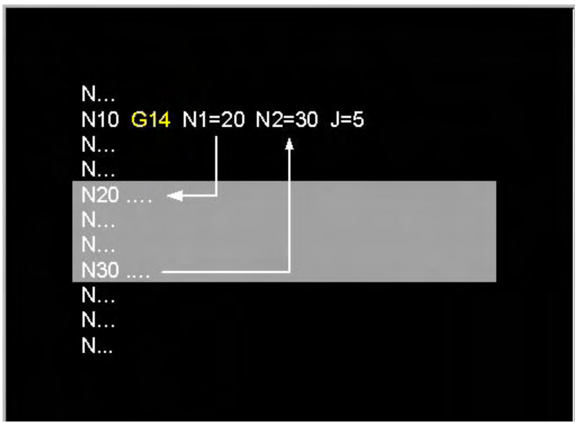
The use of the function is limited to programs that have been created with previous types of controller.

Programs that require geometric calculations can be easily created by the user with the Interactive Contour Programming (ICP).
(See chapter on Interactive Contour Programming)

23.11 Repeat function G14

N... G14 N1=.. {N2=..} {J..} {K..} {E..}

Parameters




```
G    Repeat function
J    Number of repeats
K    Repeat decrement
N1=  Repeater begin block
N2=  Repeater end block
```

Example

Repear program blocks N12-N19 four times. (2 methods)

:
N12
:
N19
:
N90 G14 N1=12 N2=19 J4
Repeat program blocks N12-N19 four times
:

:
N5 E2=4
:
N12
:
N19
:
N90 G14 N1=12 N2=19 E2
Repeat program blocks N12-N19 four times
:

Note

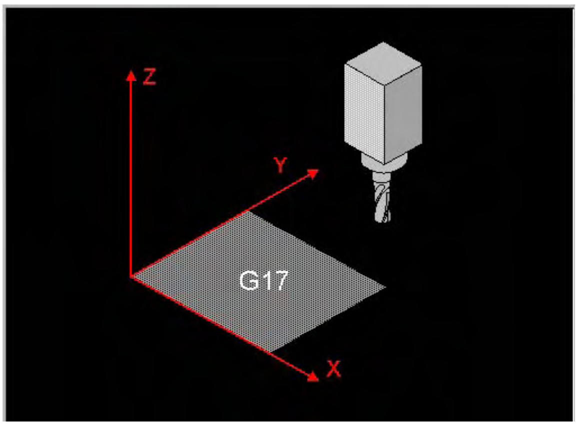
The block numbers of N1=.. and N2=.. must both be in the same part program or subprogram.
If N2=is not programmed, only the block marked N1= will be repeated.

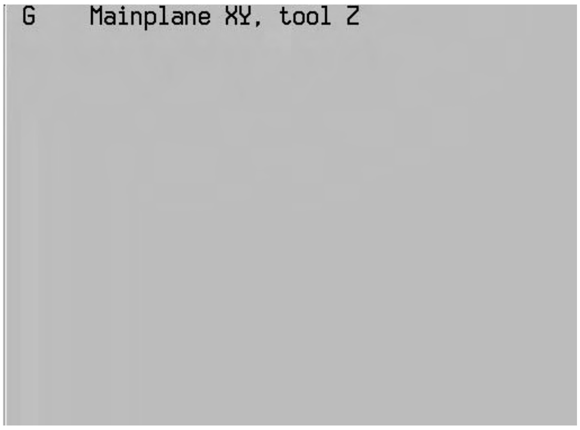
If parameters J and E are not programmed, the block sequence will only be repeated once. A repeating block sequence can be contained in another repeating block sequence (can be nested four times).

A repeat only takes place in a G14 block if E>0. If the K parameter is not programmed, the CNC uses the standard value K1.

23.12 Main plane XY, tool Z G17

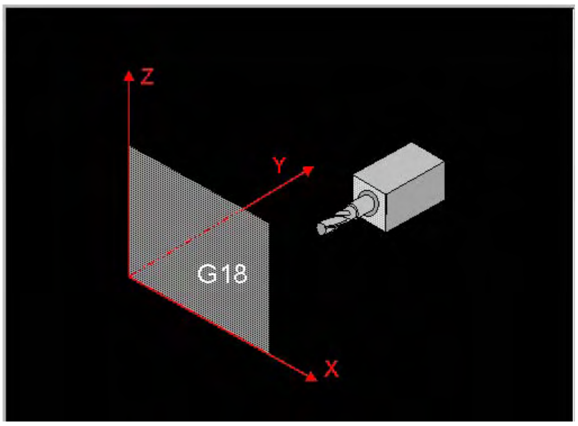
N... G17





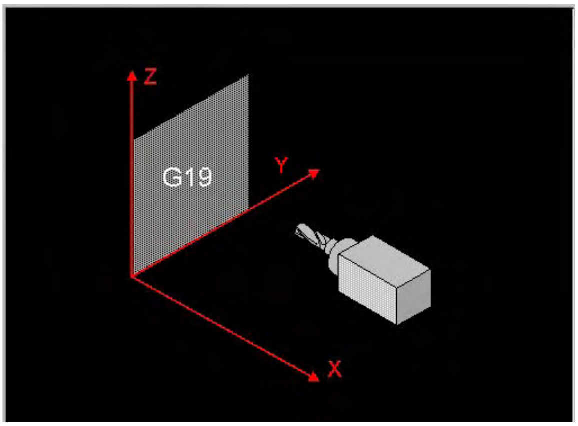
23.13 Main plane XZ, tool Y G18

N... G18



23.14 Main plane YZ, tool X G19

N... G19

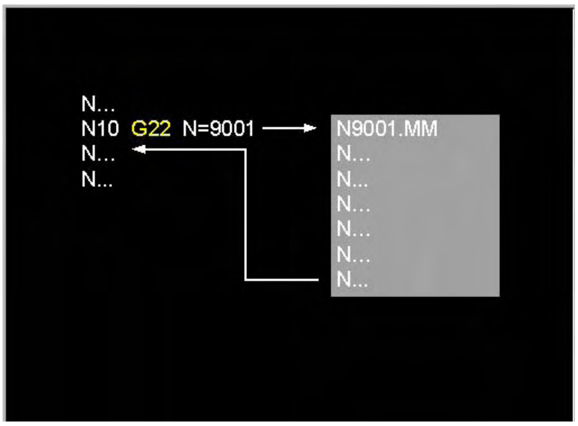


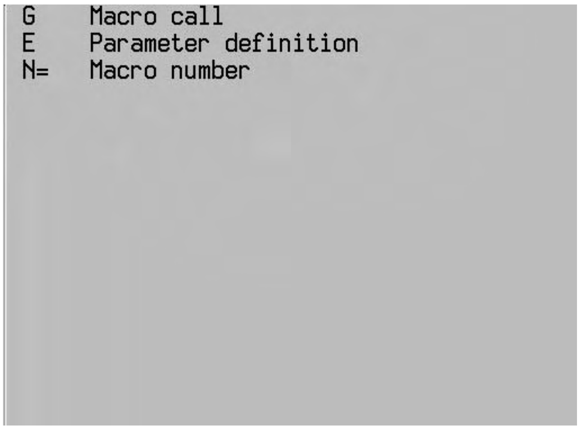
23.15 Macro call G22

Call subprogram:
N... G22 N=..

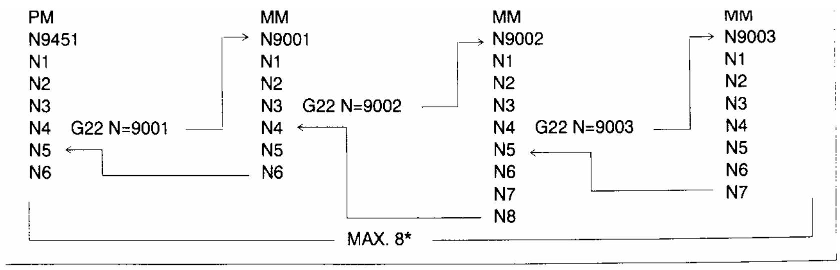
Call subprogram if E..>0:
N... G22 E.. N=.. {E..=..}

Parameters





Example



Note

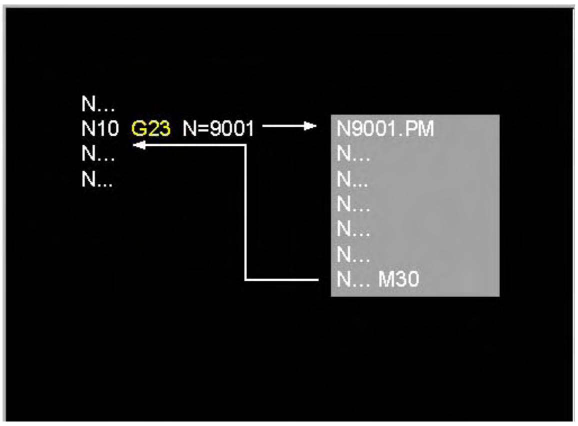
A subprogram can be called from another subprogram (can be nested eight times).

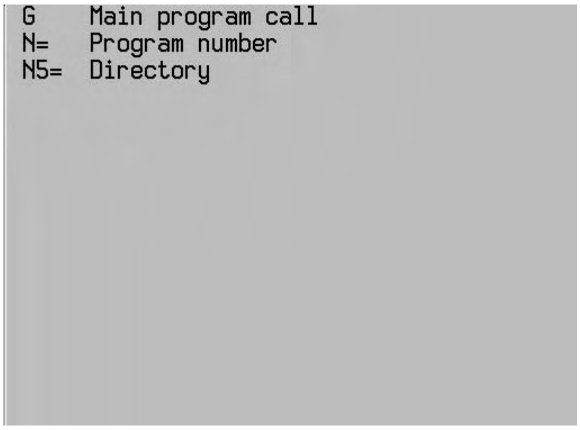
23.16 Main program call G23

This G-function calls another program (*.PM) with the address N=, which is placed in the same directory. When the program is placed in another directory, this program can be called with the directory defined in address N5=.

N.. G23 N=.... N5= "......"

Parameters





Definition of the path (N5=)

In the **SP**-version the total length of the path (N5=) and program number (N) has a maximum of 75 characters. In the **DP**-version this is a maximum of 115 characters.

In the **SP**-version programs only can be called via NFS (Network File System: See Technical Manual). In the **DP**-version programs can be called via the Windows network.

The definition of the path of programs in the CNC is:

G23 N1007	Program N1007 is called from the work directory. Mostly D:\work.
G23 N1007 N5= “test1\“	Program N1007 is called from the sub-directory "test1" from the work directory. Mostly D:\work
G23 N1007 N5= “\test1\“	Starting with \ means calling program N1007 from the subdirectory "test1" on the root directory of the hard disk. Mostly the root directory is D:. Only local drives except C: are allowed

The definition of the path of programs on a network (only **DP**-version) is:

G23 N1007 N5= “\\server1\test1\“	Starting with \\ means calling program N1007 via a network from directory \\server1\test1 on an external hard disk.
G23 N1007 N5= “S:\test1\“	Direct calling program N1007 via a network from directory "test1"on the drive S:. Local drives [C: | D: | {E:} | {F:}] are not allowed.

Notes

The main program or subprogram must not contain any G23 functions, so it may not be nested.

Programs larger than 100 KByte must not contain any jump commands.

23.17 Enable/disable feed and spindle override G25/G26

Activates (G25) or deactivates (G26) the feed and spindle override, for the purpose of the programmed feed and spindle movements. This is fixed at 100% with the feed and spindle override deactivated.

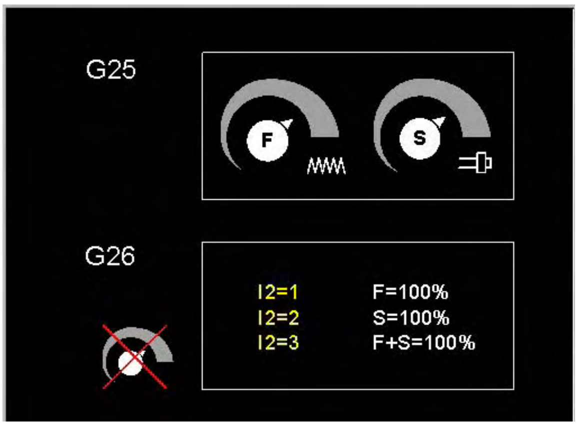
Enable feed and spindle override:
N... G25

Disable feed override (F=100%):
N... G26 I2=1 or without I2=

Disable spindle override (S=100%):
N... G26 I2=2

Disable feed and spindle override (F and S= 100%):
N... G26 I2=3

Parameters



Example

N66 G26 I2=1	Deactivate feed override, i.e. fix at 100%
:	
N70 G25 I2=2	Activate feed override
:	
N68 G26 I2=3	Deactivate feed and spindle override that is to say F and S fixed at 100 %
:	
N70 G25	Activate feed override and spindle override

Note

Reactivate feed override and spindle override using G25, M30, soft key Cancel program or soft key Clear control.

23.18 Reset/activate positioning functions G27/G28

23.18.1 Look Ahead Feed

Look Ahead Feed is used for precalculation on the programmed tool path, while taking account of the dynamics of all axes involved. The path speed is adjusted to achieve the highest contour accuracy at the highest possible speed. The programmed feed is, however, never exceeded.

Taking the programmed feed and actual feed override settings into account, special high-performance algorithms ensure a homogeneous feed at fast processing times.

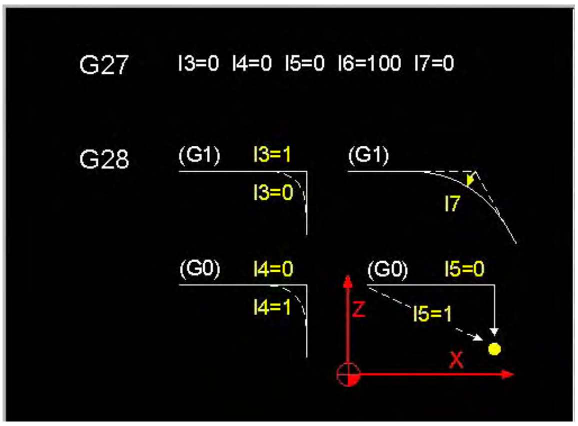
With respect to Look Ahead Feed users need not pay attention to anything else.
This function cannot be influenced.

Existing programs need not be adjusted. They can be run as before.
During Look Ahead Feed the end point and centre of a circle should match to within 64 µm. In this case the centre is corrected automatically. Unlike V310 there is no "compensation movement" at the end point. An error message follows if the end point and centre do not match to within 64 µm. The same applies also to helix interpolation.

The running speed of CAD-generated programs is increased substantially.

23.18.2 Positioning functions G27/G28

Parameters




```
G    Positioning functions
I3=  Feed movement 0=inpos, 1=inpod
I4=  Rapid movement 0=inpod, 1=inpos
I5=  Position logic: 0=with, 1=without
I6=  Reduction acceleration/jerk   [%]
I7=  Contour tolerance
```

1. G28 without parameter
   G1,G2,G3 without In-Position — G28

2. Movement with feed
   G1,G2,G3 without In-Position (initial setting) — G28 I3=0
   G1,G2,G3 with In-Position — G28 I3=1

3. Rapid traverse movements G0
   G0 with In-Position (initial setting) — G28 I4=0
   G0 without In-Position — G28 I4=1

4. Positioning logic with G0
   G0 with positioning logic (initial setting) — G28 I5=0
   G0 with positioning logic — G28 I5=1

5. Acceleration and jerk reduction

Acceleration reduction	G28 I6=...
Resetting	G28 I6=100

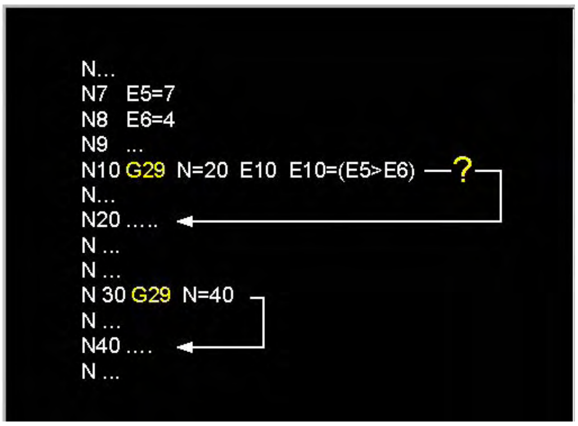
6. Movements with user-definable contour accuracy

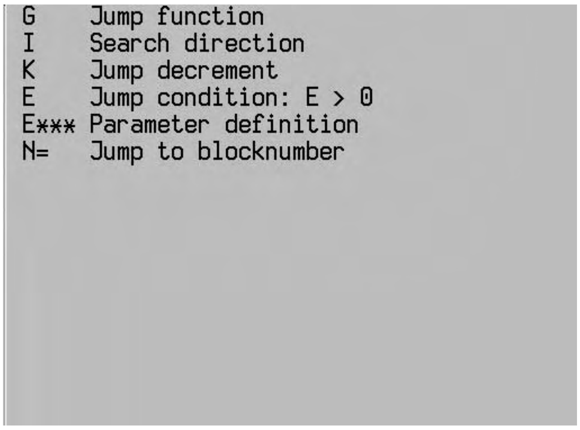
G0,G1,G2,G3	
-contour accuracy (MC765)	
-user-definable contour accuracy	
I7=... (0-10000 mm)	G28 I7=...

23.19 Jump function G29

N.. G29 {E..} N=.. {K..} {I..}

Parameters





Example

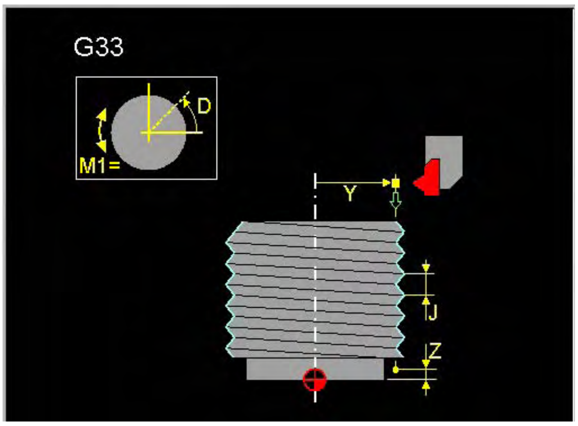
:	
N50 E2=3	Parameter E2 will have value 3
N51	
:	
N100 G29 N=51	Jump to N51
:	
N100 G29 E2 N=51	At E2 > 0 there will be a jump to N51; E2 is reduced by 1. At E2=0 the program run is continued after N100.
:	

Note

The value of the E parameter is reduced by the value of the K address. The E parameter is used as the new branch condition.
If the K address is not programmed, the E parameter is reduced by 1 after each branch.

Forward and backward jumping is possible in a subprogram or program. This is controlled by parameter I. If I=1 or I=0, searching will only be in forward direction. If I=-1 or no value is shown, there will first be a jump backwards to the beginning of the subprogram or program, which is followed by forward searching for the block number.

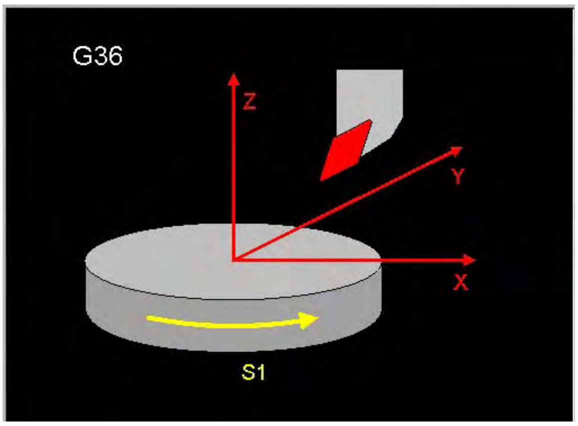
23.20 G33 Basic Threadcutting movement

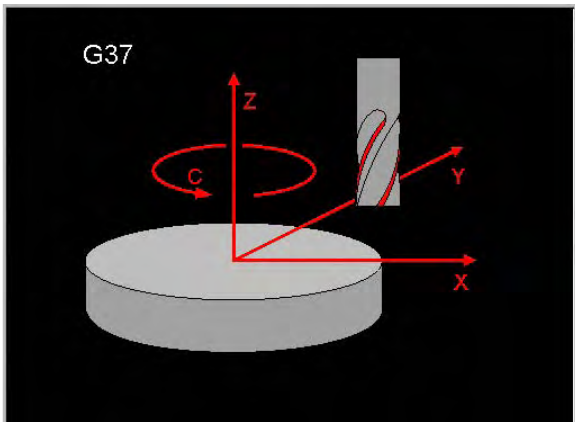



```
G     Single threadcutting movement
X     Endpoint coordinate
Y     Endpoint coordinate
Z     Endpoint coordinate
J     Pitch
D     Start angle threadcutting
?90=  Endpoint abs. (X,Y,Z..)
?91=  Endpoint incr. (X,Y,Z..)
```

Refer to Chapter "Turning mode".

23.21 G36/G37 Activate/ Deactivate turning mode





Refer to Chapter "Turning mode".

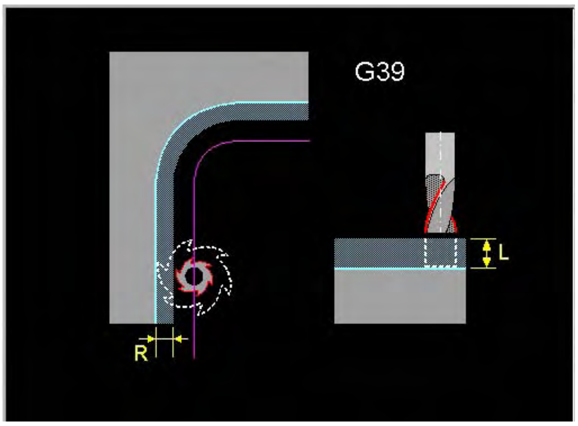
23.22 Activate/deactivate offset G39

The programmed contour may be changed by offset.

Activate offset:
N... G39 {R...} {L...}
R: tool radius offset
L: tool length offset

Deactivate:
N... G39 L0 and/or R0

Parameters



G	Activate tool offset
L	Toollength offset
R	Toolradius offset

Notes and usage

Changes made to the tool length offset will be activated with the next in-feed movement.

The tool radius offset is only active with active cutter radius compensation.

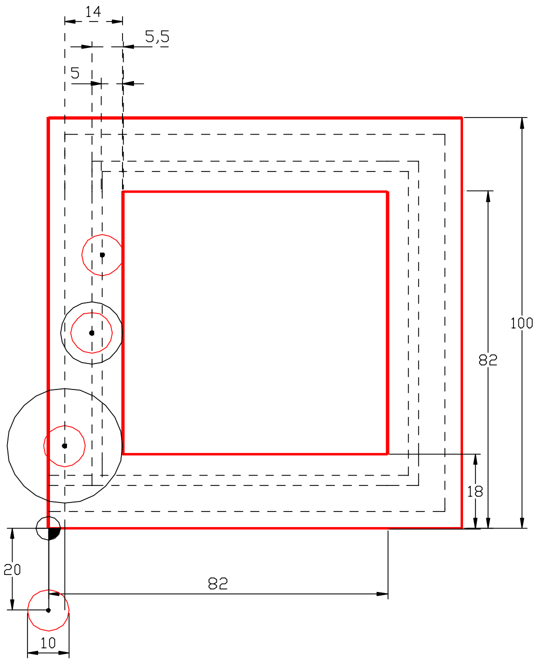
With inactive cutter radius compensation, tool radius offset changes will be activated when cutter radius compensation (G41/G42, G43/G44) has been activated.
With activated cutter radius compensation, tool radius offset changes will be corrected linearly over the entire path in the next movement block.

Note:

Tool radius offset is suppressed when the following functions have been activated: G6, G83-G89, G141, G182. Length offset remains active. Offset programming should be deactivated before these functions.

Example

Rectangular milling by roughing twice and finishing once

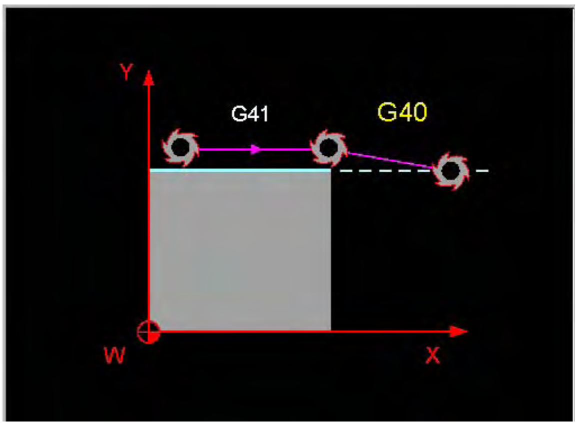


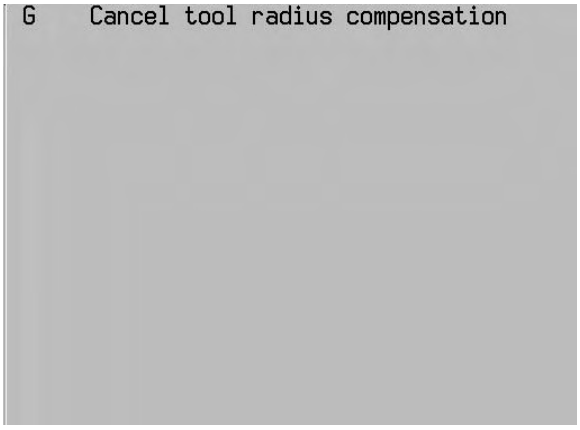
N39001	
N1 G98 X-10 Y-10 Z10 I120 J120 K-60	
	Define the graphic window
N2 G99 X0 Y0 Z0 I100 J100 K-40	
	Define the material
N3 T1 M6	Change tool (cutter radius: 5 mm)
N4 G39 L0 R9	Activate tool radius offset. The offset is 9 mm. (cutter radius for radius compensation is (5+9 =) 14 mm).
N5 F500 S1000 M3	Activate feed and spindle speed
N6 G0 X0 Y-20 Z5	Approach start position
N7 G1 Z-10	Move to depth
N8 G43 X18	Approach contour with radius compensation
N9 G41 Y82	First roughing of the rectangle
N10 X82	
N11 Y18	
N12 X0	
N13 G40	Turn off radius compensation
N14 G39 R0.5	Change tool radius offset. The offset is 0.5 mm. (cutter radius for radius compensation is (5+0.5 =) 5.5 mm)
N15 G14 N1=8 N2=13	Repeat the rectangle (2nd roughing movement)
N16 G39 R0	Change tool radius offset. The offset is 0 mm. (cutter radius for radius compensation is 5 mm)
N17 G14 N1=8 N2=13	Finish the rectangle
N18 G0 Z10	Retract tool from material
N19 M30	End of program

23.23 Cancel tool radius compensation G40

N.. G40

Example





N9 G42	Activate radius compensation to the right
N10 G1 X..	
N11 X.. Y..	
N12 G40	Cancel radius compensation
N13 G0 Y..	
:	

Notes

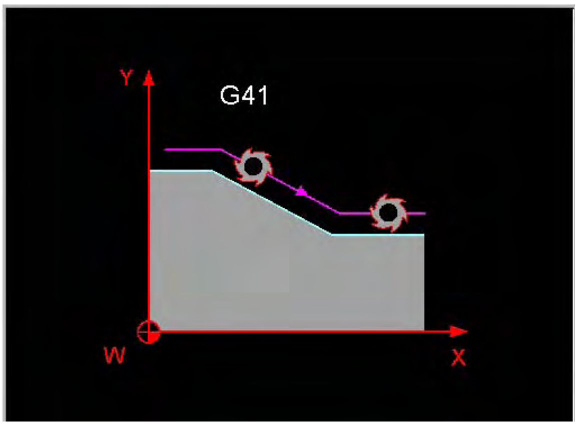
G40 comes in to effect automatically after:

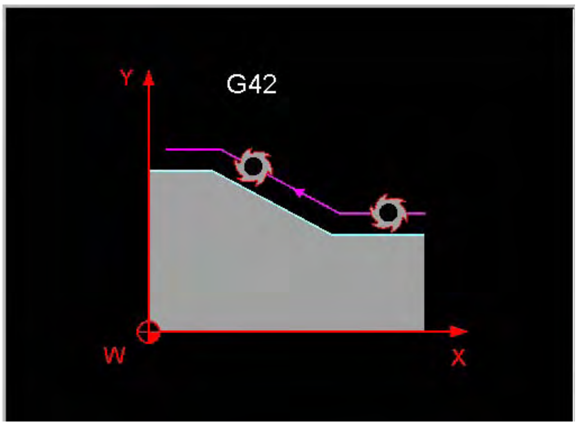
- Switching the controller on
- Softkey Clear control
- Softkey Cancel program
- M30

23.24 Tool radius compensation (left/right) G41/G42

N.. G41/G42

In both cases, the view direction is the direction of tool movement.

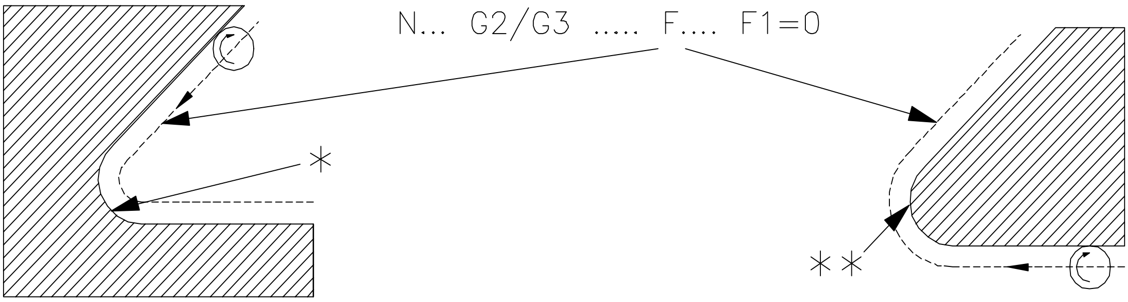




Constant feed for radius compensation of circles

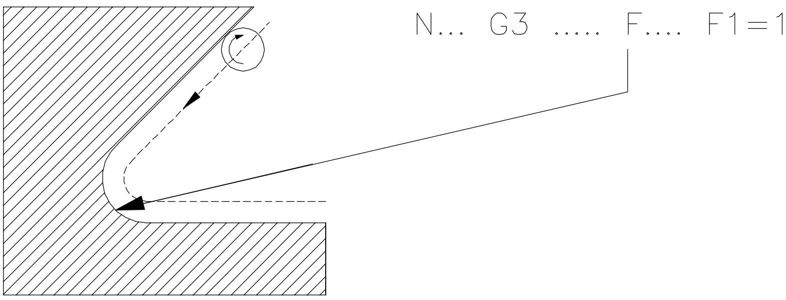
Parameter F1= is used to keep the programmed feed on the workpiece contour constant regardless of the cutter radius and contour shape.

F1=0 feed not constant (switch-on condition, M30, Cancel program softkey or after Clear control softkey). The programmed feed should represent the tool tip speed.

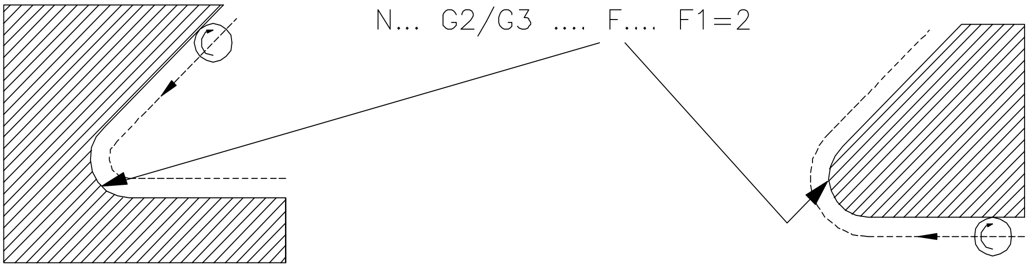


* = feed too large ** = feed too small

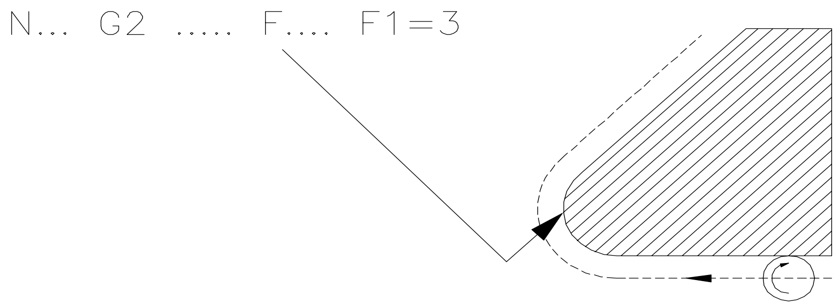
F1=1 constant feed only on the inside of circular arcs. The programmed feed is reduced to ensure that the tool tip moves along the inside of a circular arc at reduced speed.



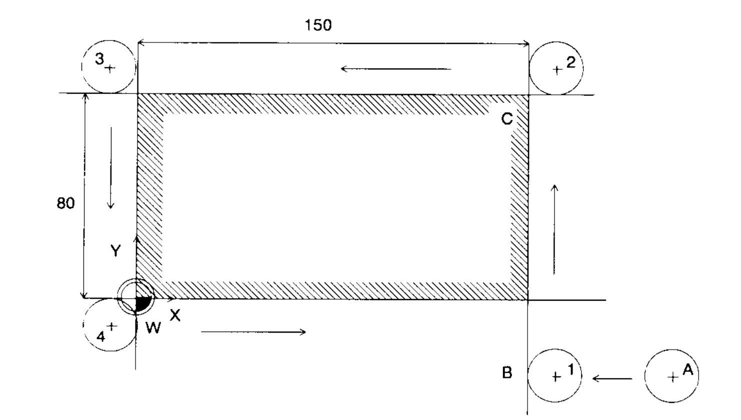
F1=2 constant feed on the inside and outside of circular arcs. The programmed feed is reduced (inside of circular arcs) or increased (outside of circular arcs) to ensure that the tool tip moves at the calculated new speed. If the increased speed exceeds the maximum feed defined by a machine constant, the maximum feed is used.



F1=3 constant feed only on the outside of circular arcs. The programmed feed is increased to ensure that the tool tip moves along the outside of a circular arc at the increased speed.



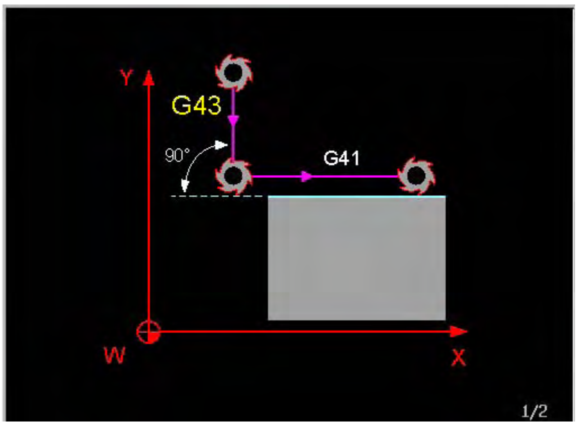
Example

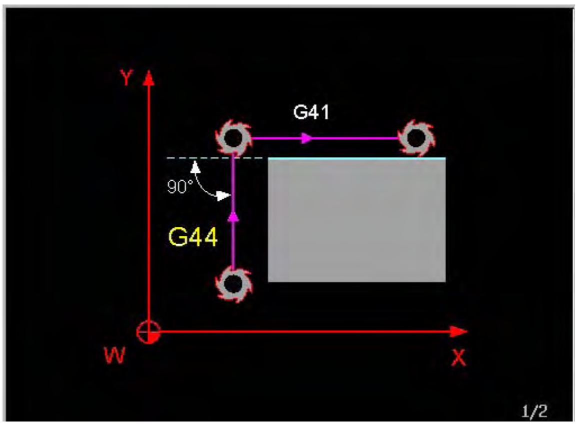


N9999	
N1 G17	
N2 G54	
N3 T1 M6	Change tool
N4 G0 X200 Y-20 Z-5 S500 M3	Spindle start, move tool to X120,Y-20 at traversing speed
N5 G43	Radius compensation to end point
N6 G1 X150 F150	
N7 G42 Y80	Activate radius compensation to the right
N8 X0	
N9 Y0	
N10 X150	
N11 G40	Cancel radius compensation
N12 G0 X200 Y-20	

23.25 Tool radius compensation to end point G43/G44

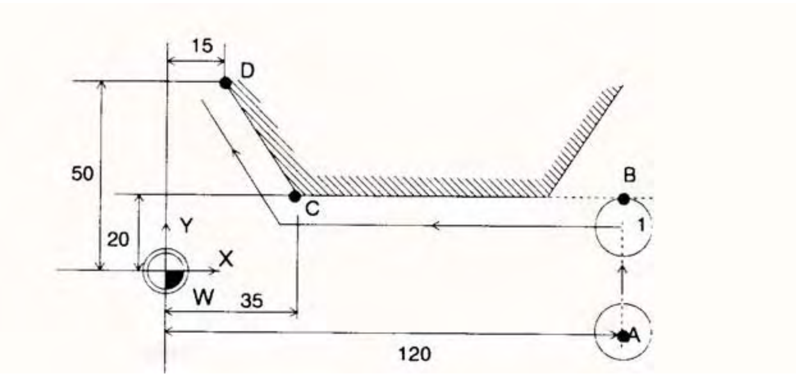
N.. G43/G44





G43 G44

Example



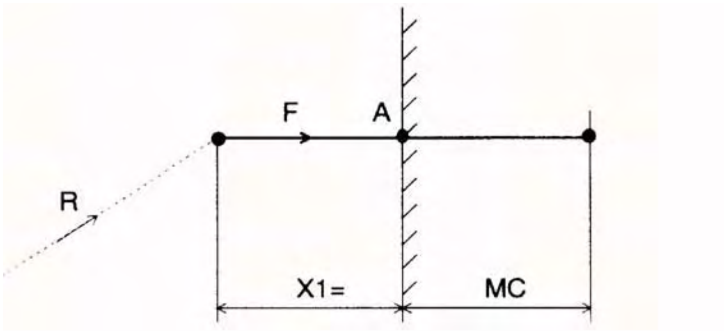

```
:
N40  G0  X120  Y-15  Z10
N41  G1  Z-10  F500
N42  G43  Y20              Radius compensation to end point
N43  G41  X35              Activate radius compensation to the left
N44  X15  Y50
:
```

23.26 Measuring a point G45

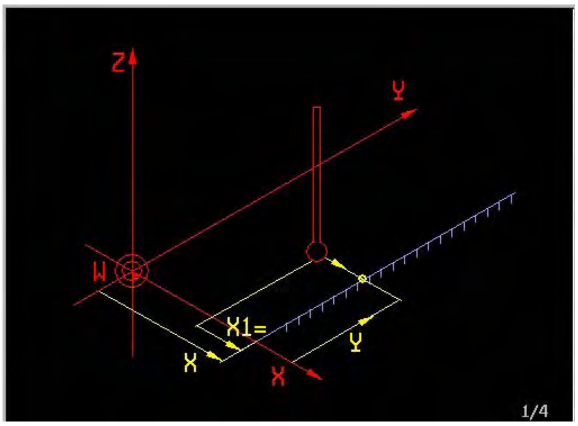
Determines coordinate values with the measurement probe. The clamping position of the work piece and the work piece dimensions can be measured. The measurement results can be further processed by G49 and G50. The freely programmable measurement cycle G145-G150 can be used as an alternative to G45.

N.. G45 [measurement position] {I+/-1} {J+/-1} {K+/-1} {L+/-1} {X1=..} {N=..} {P1=..}

The plane of the rotary table is determined by definition of the 4th axis in the machine constant list. (MC117 should be 4 and MC118 should be B(66) or C(67)). L relates to the 4th axis B or C. Rotary axis A is not allowed.

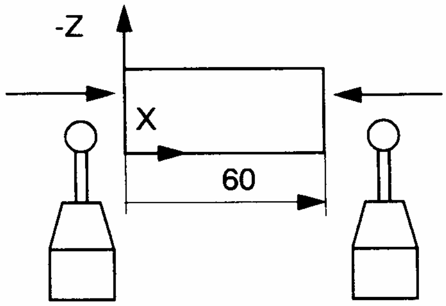


Parameters



G	Measuring a point
X	Measurement target coordinate
Y	Measurement target coordinate
Z	Measurement target coordinate
B	Measurement point angle
C	Measurement target angle
I	Measurement direction for X axis
J	Measurement direction for Y axis
K	Measurement direction for Z axis
L	Measurement direction rotary-axis
M	M25 for tool measurement
E	Parameter-nr measured coordinate
N=	Point-nr.for measured coordinate
X1=	Measurement path length
?90=	Measurement target abs. (X,Y,Z..)
?91=	Measurement target incr.(X,Y,Z..)
P1=	Point definition number

Examples



Measuring a point in the X axis :

Measuring in the positive direction
N.. G45 X0 Y20 Z-10 I1 E1 N=1
Measure point, calculate measurement position, store in point memory N= or in parameter E1.

Measuring in the negative direction
N.. G45 X60 Y20 Z-10 I-1 E1 N=1

Notes

- Only one axis coordinate can be measures with a G45 block.
- Measurements can only be made in the negative direction in the tool axis.
- The spindle speed must not be activated or switched on.
- Locate block.

N105 ...
N110 G148 E20
N115 G29 E21=E20=2 E21 N=125
N120 G45/G46
N125 ...

The tool type Q3=9999 can be entered for the measurement key.
M27 Activate measuring probe.
M28 Turn off measuring probe.

Example: P5 T5 Q3=9999 L150 R4
When tool T5 is called, the controller recognises that this tool is the measuring probe. The “spindle on” function (M3, M4, M13, M14) is suppressed and a fault message is issued.

Function G45 operates only parallel to the axis. The function of G145 has improved and now includes measurement unparallel to the axis. We therefore recommend that you should use the new basic measurement movement G145.

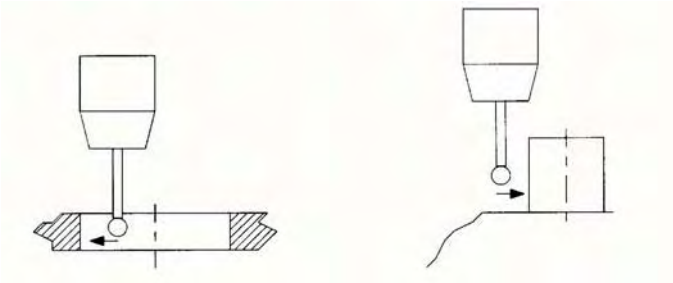
The difference between the measured and the programmed coordinate is calculated and stored internally to be used in operation with G49 or G50.

23.27 Measuring a circle G46

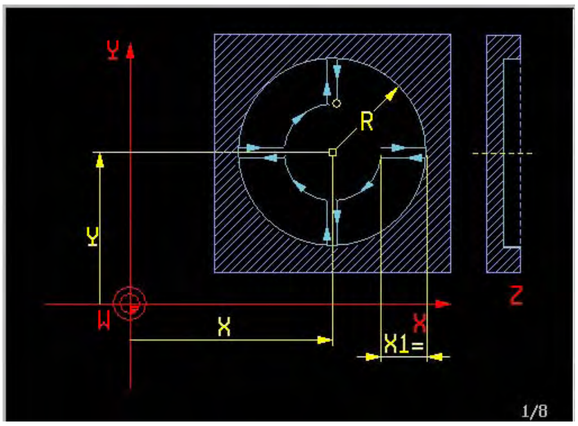
Measures a full circle (internally or externally) with a 4-point measurement. The measurement can be processed further by G49 or G50.

Measure inner circle:
N.. G46 [circle centre point coordinates] R.. {I+1 J+1} {I+1 K+1} {J+1 K+1} {F..} {X1=..} {P1=..} N=.. E..

Measure outer circle:
N... G46 [circle centre point coordinates] R.. {I-1 J-1} {I-1 K-1} {J-1 K-1} {F..} {X1=..} {P1=..} N=.. E..



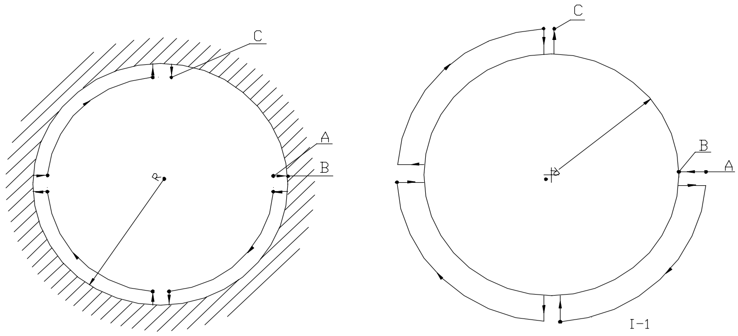
Parameters



G Measuring a circle
X Center point coordinate
Y Center point coordinate
Z Center point coordinate
B Measurement target angle
C Measurement target angle
I Measurement direction for X axis
J Measurement direction for Y axis
R Circle radius
M M26: probe radius measurement
E Parameter-nr. measured radius
N= Point-nr.measured centre point
X1= Measurement path length
?90= Centre point abs. (X,Y,Z..)
?91= Centre point incr. (X,Y,Z..)

P1= Point definition number

Example



Measurement of an internal and an external circle in the XY level:

Internal circle:
N... G46 X30 Y25 Z20 I+1 J+1 R12.5 F3000 N=59 E24

Measure circle, store centre in point memory N=59 and radii in parameter memory E24.

External circle:
N... G46 X30 Y25 Z20 I-1 J-1 R20 F3000 N=58 E23

Level	Internal circle		External circle	
XY (G17)	I+1	J+1	I-1	J-1
XZ (G18)	I+1	K+1	I-1	K-1
XZ (G19)	J+1	K+1	J-1	K-1

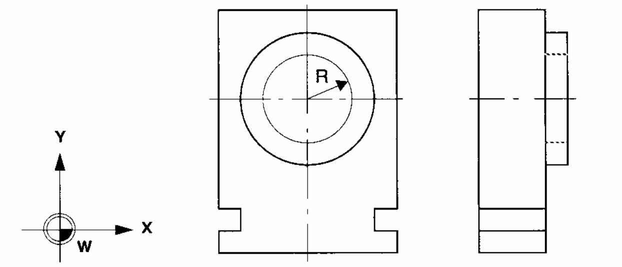
23.28 Calibrating the measuring probe G46 + M26

The measuring probe radius is determined by touching the calibration ring. The controller calculates the probe radius from the measured radius of the calibration ring and the programmed radius. The new radius value is stored in tool memory.

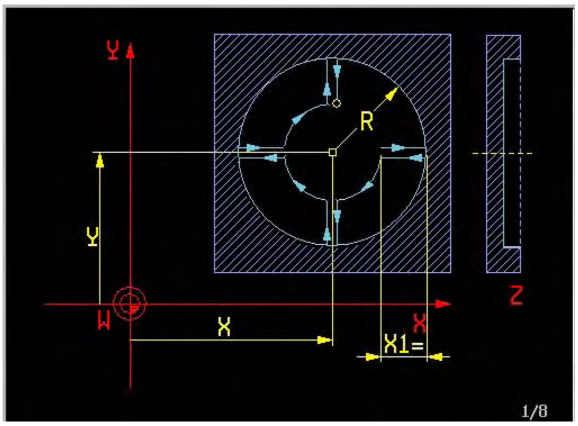
The centre point coordinates and the radius of the calibration ring are entered as machine constants.

Measuring the internal gauge ring:
N... G46 {I+1 J+1} {I+1 K+1} {J+1 K+1} {F...} {X1=...} M26

Measuring the external gauge ring:
N... G46 {I-1 J-1} {I-1 K-1} {J-1 K-1} {F...} {X1=...} M26



Parameters



G	Measuring a circle
X	Center point coordinate
Y	Center point coordinate
Z	Center point coordinate
B	Measurement target angle
C	Measurement target angle
I	Measurement direction for X axis
J	Measurement direction for Y axis
R	Circle radius
M	M26: probe radius measurement
E	Parameter-nr. measured radius
N=	Point-nr.measured centre point
X1=	Measurement path length
?90=	Centre point abs. (X,Y,Z..)
?91=	Centre point incr. (X,Y,Z..)

P1=	Point definition number

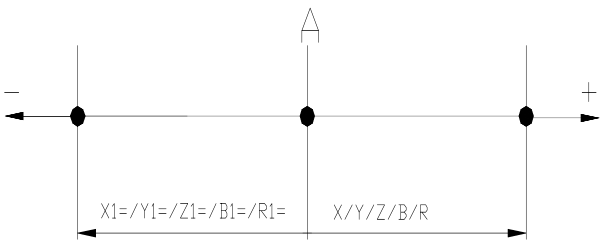
Example

N46002
N1 G17
N2 T1 M6
N3 D207 M19 defined spindle stop
N4 G46 I1 J1 M26 F3000 calibrate measuring probe, store measuring probe radius for T1 in tool memory
N5 Z200 M30

23.29 Checking on tolerances G49

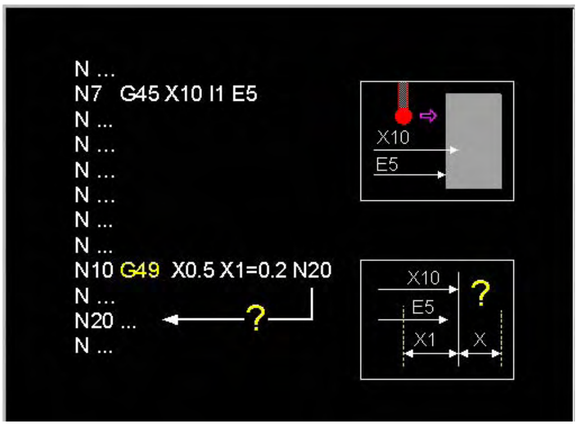
Compares whether the difference between the programmed value and the value measured during the G45 or G46 block lies within the dimensional tolerance limits.
If the difference lies within the tolerance limits, program processing continues.
If the difference lies outside the tolerance limits, the following options are available:

Repetition of the program section:
N.. G49 {X.., X1=..} {Y.., Y1=..} {Z.., Z1=..} {B.., B1=..} {R.., R1=..} N1=.. N2=.. {E..}
Conditional branch:
N.. G49 {X.., X1=..} {Y.., Y1=..} {Z.., Z1=..} {B.., B1=..} {R.., R1=..} N=.. E..



The measurement point must lie between the upper limit dimension (X/..) and the lower limit dimension (X1=/..) of the tolerance range.

Parameters



G	Checking on tolerances
X	Positive tolerance value in X
Y	Positive tolerance value in Y
Z	Positive tolerance value in Z
B	Positive tolerance value in B
C	Positive tolerance value in C
R	Positive tolerance circle radius
E	Jump condition: E > 0
N=	Jump to blocknumber
N1=	Repeater begin block
N2=	Repeater end block
X1=	Negative tolerance value in X
Y1=	Negative tolerance value in Y
Z1=	Negative tolerance value in Z
B1=	Negative tolerance value in B
C1=	Negative tolerance value in C
R1=	Negative tolerance circle radius

Example

N10 G49 R.02 R1=2 E1 N=13
N11 G49 R2 R1=.02 N1=1 N2=6

N10 1st tolerance comparison:
If the upper tolerance limit (R0.02) is exceeded (bore too large), a branch to block N13 takes place. The lower tolerance limit must not be reached (conditional branch).

N11 2nd tolerance comparison:
If the lower tolerance limit (R1=0.02) is exceeded (bore too small), the program section between N1 and N6 is repeated. The upper tolerance limit must not be reached (repetition of program section)

Note

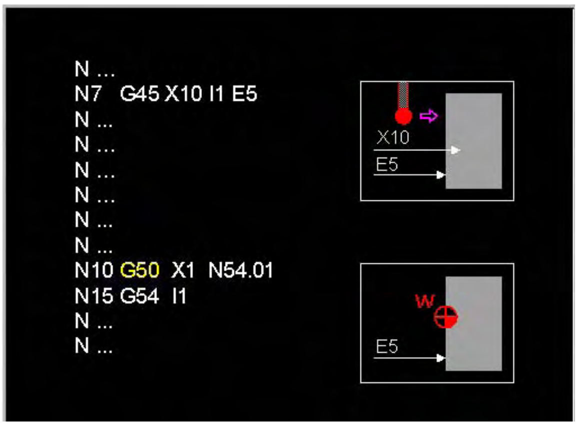
Where there are two consecutive G49 blocks in the program, it must be ensured that the conditional branch is in the first block and the program section repetition is in the second block (otherwise an error message will appear!)

23.30 Processing measuring results G50

Changes the zero point offset or tool dimensions according to the correction values derived from the measured difference values.

Zero point offset correction:
With standard zero points or MC84=0:
N.. G50 {X1} {I..} {Y1} {J..} {Z1} {K..} {B1} {C1} {C2} {B1=} {C1=} {L..} N=..
With extended zero points and MC84>0:
N.. G50 {X1} {I..} {Y1} {J..} {Z1} {K..} {B1} {C1} {C2} {B1=} {C1=} {L..} **N=54.00 .. 54.99**
Tool length correction:
N.. G50 T.. L1=1 {I..} {J..} {K..} {T2=..}
Tool radius correction:
N.. G50 T.. R1=1 {X1=..} {T2=..}

Parameters

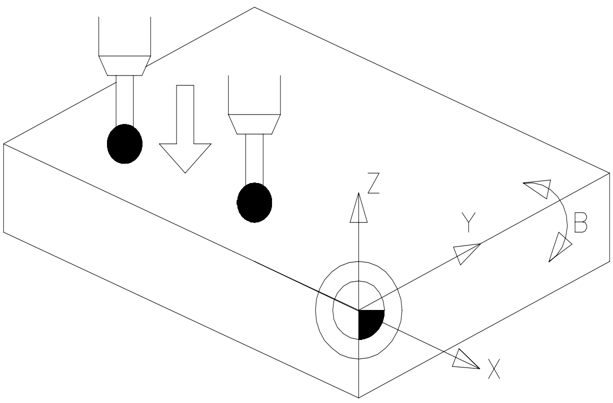


G	Processing measuring results
X	1=zero point shift in X
Y	1=zero point shift in Y
Z	1=zero point shift in Z
B	1=zero point shift in B
C	1=zero point shift in C
I	Multiplication factor for X
J	Multiplication factor for Y
K	Multiplication factor for Z
L	Multipl. factor for rotary-axis
T	Tool dimensions to be corrected
N=	Offset-nr for correction (52-59)
X1=	Multiply factor for tool radius
B1=	Prog.angle in B after calculation
C1=	Prog.angle in C after calculation

L1=	1=correction of tool length
R1=	1= correction of tool radius

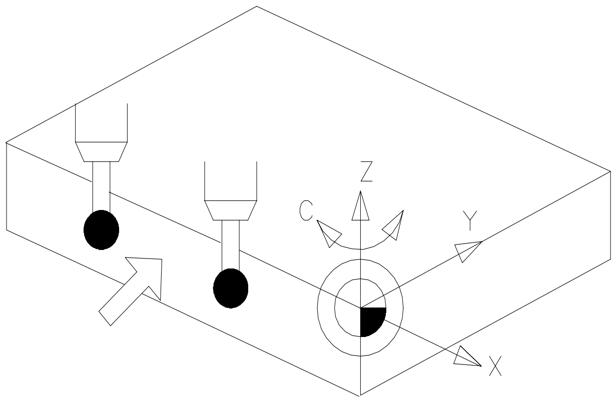
Notes

Machine configurations (B1,C1,C2)
B-axis B1: All that is required to align a clamped work piece with a rotary table (B-axis) rotating about the Y-axis is to measure two points on the X-axis:
-the angle of rotation is relative to the X-axis.
-the tool rotates about the Y-axis.
-the tool axis with the probe is the Z-axis or the Y-axis.

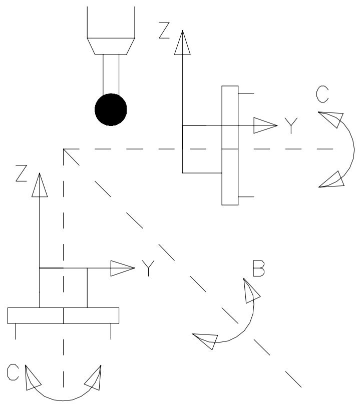


C-axis C1: All that is required to align a clamped work piece with a rotary table (C-axis) rotating about the Z-axis is to measure two points on the X-axis:

-the angle of rotation is relative to the X-axis.
-the tool rotates about the Z-axis.
-the tool axis with the probe is the Z-axis.

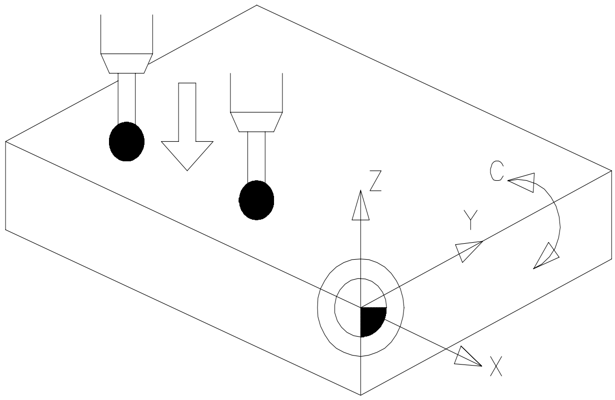


C-axis C2: This is an extension of the C1 option:



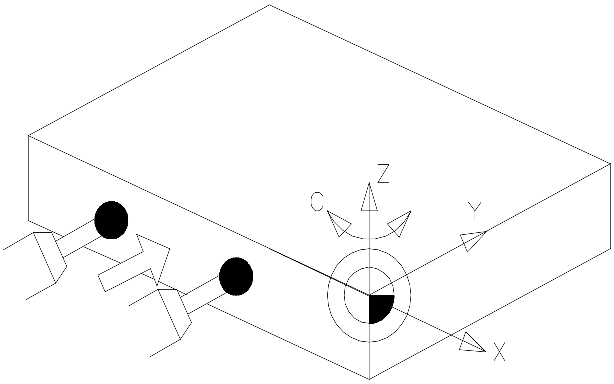
1. The C-axis is rotated through 90 degrees and rotates about the Y-axis instead of the Z-axis.
All that is required to align a clamped work piece with a rotary table (C-axis) rotating about the Y-axis is to measure two points on the X-axis:

-the angle of rotation is relative to the X-axis.
-the tool rotates about the X-axis.
-the tool axis with the probe is the Z-axis.



2. All that is required to align a clamped work piece with a rotary table (C-axis) rotating about the Z-axis is to measure two points on the X-axis:

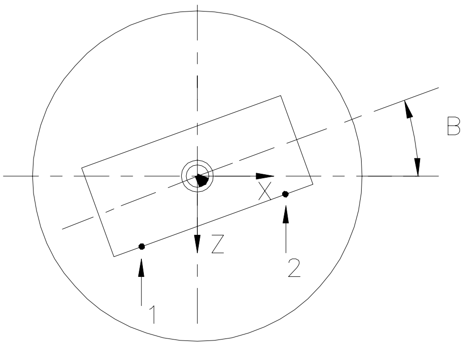
-the angle of rotation is relative to the X-axis.
-the tool rotates about the X-axis.
-the tool axis with the probe is the Y-axis.



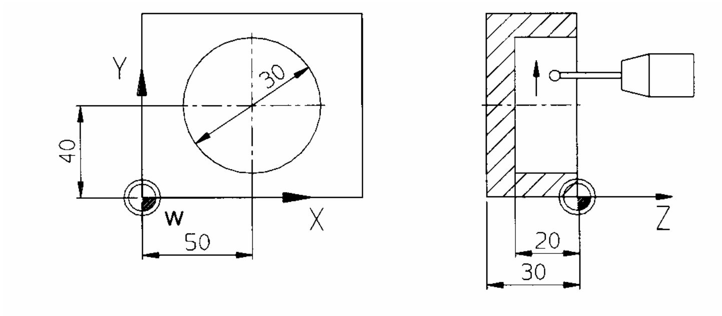
Examples

N.. G50 X1 I0.8 N=54
Change the X coordinates of the G54 offset by multiplying the correction value by 0.8 and enter the new X coordinate value of G54 in the zero point memory.

N.. G50 T5 L1=1 K0.97 R1=1
Correct the length of tool 5 by multiplying the difference in Z (tool in Z axis) by 0.97, and enter the new dimension in the tool memory.



N50003	
N1 G17 T1 M6	
N2 G54	
N4 G45 X-50 Z0 Y-20 C0 J1 N=1	Measurement at point 1
N5 G45 X50 Z0 Y-20 J1 N=2	Measurement at point 2
N6 G50 C1 N=54	Settlement of zero offset
N7 G54	Reactivate zero offset
N8 G0 Z100 B0	



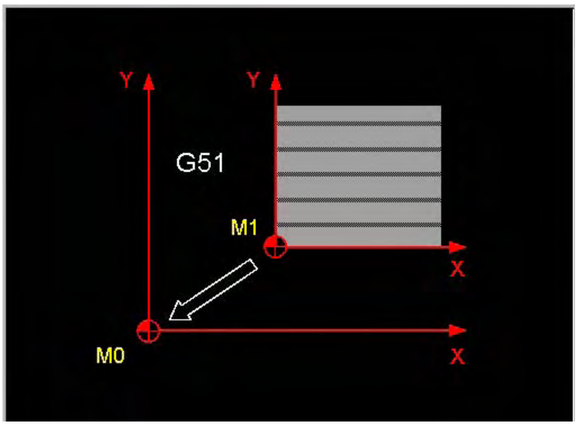
N50006
N1 G54
N2 G17 T1 M67 (Mill R5)
N3 G89 Z-20 B2 R15 F1000 S50 M3
N4 G79 X0 Y0 Z0
N5 G0 Z50 M5
N6 T31 M67 (Measuring probe)
N7 M19
N8 M27 Activate measuring probe
N12 G46 X50 Y40 Z-5 R15 I1 J1 F500 E5 Measuring a full circle
N13 G0 Z50
N14 G49 R0.02 R1=2 N=21 E5 (bore > (15+0.02) jump-> N=21) Tolerance comparison
N15 G49 R2 R1=.02 N=17 E5 (bore < (15-0.02) jump-> N=17) Tolerance comparison
N16 G29 E10 E10=1 N=23 Conditional jump to end of program
N17 G50 T1 R1=1 Settlement of tool radius
N18 M28 Turn off measuring probe
N19 G14 N1=2 N2=5
N20 G29 E1 E1=1 N=23
N21 M0
N22 (Bore outside tolerance area)
N23 M30

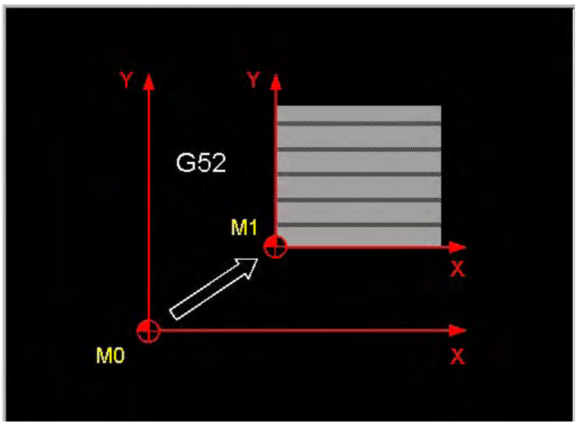
23.31 Cancel/activate G52 zero point shift G51/G52

Determines the work piece zero point with the stored values.

Activate:
N... G52

Cancel:
N... G51





Note

The use of the functions is limited to programs that were created with previous types of controllers.

Function G52 is cancelled by the Clear control softkey or by programming G51.

Functions G51 and G52 remain active after Cancel program and M30.
If a G54 .. G59 zero point offset is already active, G52 is effective from the point of this offset. If G52 ia active, G54 .. G59 are effective from the point of this offset.

FROM V320

If **MC84 = 0**, G52 is in the ZO.ZO (zero point) memory.
If **MC84 > 0**, G52 is in the PO.PO (pallet offset) memory.

The zero points can be edited in both memories.

23.32 Cancel/activate zero point shift G53/G54...G59

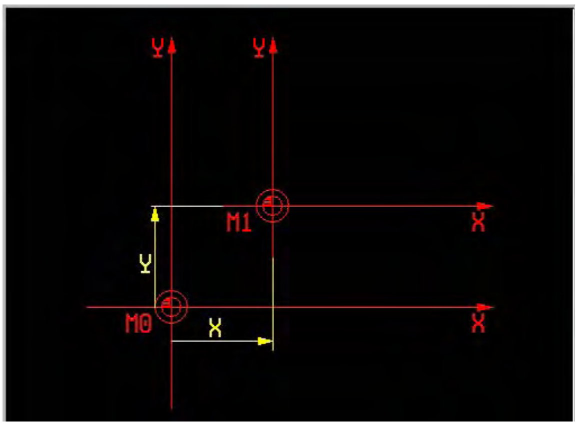
Offsets the work piece zero point to a new position, the coordinate values of which are stored in the zero point memory (under the relevant number).

Activate:

N.. G54 {X..} {Y..} {Z..} {A..} {B..} {C..}
N.. G55
N.. G56
N.. G57
N.. G58
N.. G59

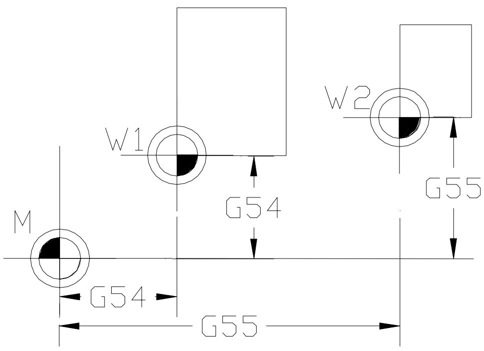
Delete:

N.. G53



G	Activate zero point shift
X	Zero point coordinate
Y	Zero point coordinate
Z	Zero point coordinate
A	Zero point angle
B	Zero point angle
C	Zero point angle

Example



:
N60 G54 Activate zero offset G54
:
N600 G55 Activate zero offset G55. The coordinates relate to the new zero point.
:

23.33 Extended zero offset G54 MC84>0

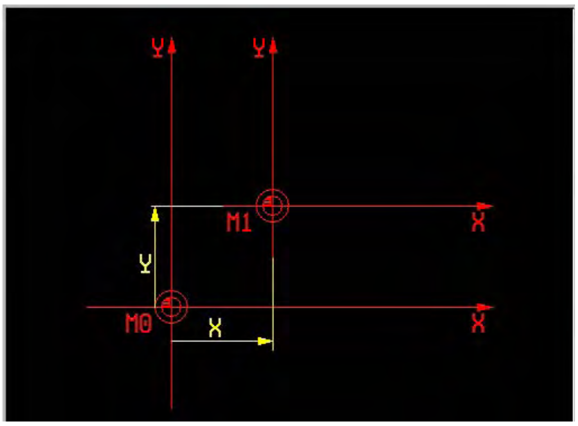
Additionally to the current zero offset table G54..G59 there is another zero offset table G54 I[nr] with a maximum of 99 zero offsets. The relevant zero offset is selected by machine constant MC84.

- Identifier of zero offset memory Ze.Ze (MC84 > 0)
- Programming (offset values) of zero offset in NC program
- Programming an angle of rotation (B4=) in zero offset
- Entering comment in zero offset memory

Define and call zero offset:
G54 I[nr] [axis coordinates] {B4=..}

Call zero offset:
G54 I[nr]

Parameter



G	Activate zero point shift
X	Zero point coordinate
Y	Zero point coordinate
Z	Zero point coordinate
A	Zero point angle
B	Zero point angle
C	Zero point angle
I	Zero point index
B4=	Angle of rotation absolute

Notes and use

The zero offset table is adjusted during scaling (MC84 > 0). The existing zero points are maintained. Extended zero points are initialised to zero.

Attention: **If MC84 is zeroed, the table is changed (ZE.ZE to ZO.ZO). The new zero point table is initialised to zero.**

Offset values may be entered in the zero point memory in two different ways:

- The values of zero offsets G54 I[nr] are entered in the zero offset memory prior to execution of the program, via the control panel or from a data carrier.
- The values of zero offset G54 I[nr] X.. Y.. Z.. A.. B.. C.. B4=.. are programmed in an NC program block. During execution of the program, the programmed values are accepted and activated in the zero offset memory.

Attention: **If no new zero offset values have been programmed in the program block, the zero offset values already existing in the memory must not be overwritten or deleted. The unprogrammed axis coordinates are taken from the memory. Collision hazard!**

Every zero offset in the table may also involve comments.

Every zero offset in the table may also involve axis rotation. First the offset is carried out, then the coordinate system is rotated through angle B4=.

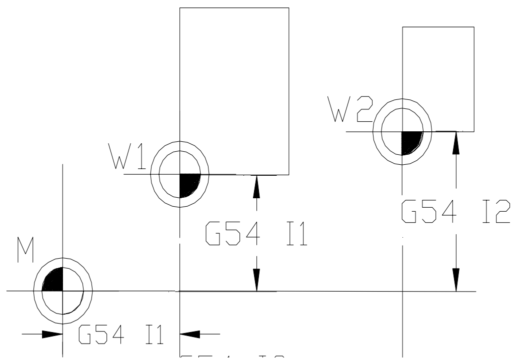
G52 does not affect the functions G53...G59. If G52 is active, G54..G59 will be active from this offset.

A programmed zero offset (G92 or G93) will be cancelled by any of the G54 I[nr] functions.

G54 I[nr] is automatically cancelled by the Clear control softkey or by programming G53. G54. I[nr] is not cancelled by the Cancel Program softkey or M30.

Example

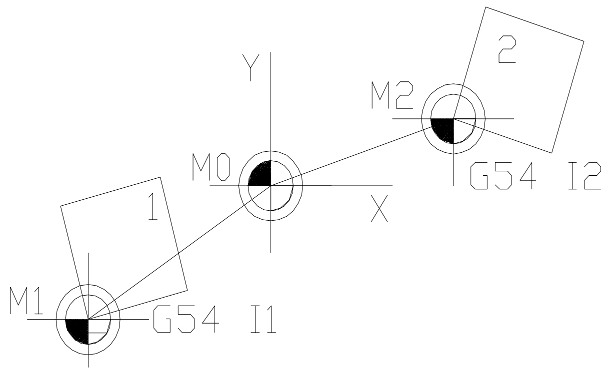
1.



N60 G54 I1	Selection of zero point W1. Its coordinates (X40,Y100,Z300) are taken from the zero offset memory. All programmed coordinates are measured from W1.
N600 G54 I2	Selection of zero point W2. Its coordinates (X200,Y100,Z100) are taken from the zero offset memory. Zero point W1 is cancelled and W2 activated. As a result, all programmed coordinates are measured from W2.
N700 G53	Turn off zero point W2. The coordinates (X0,Y0,Z0) are taken from the G53 zero offset memory. Zero point W2 is cancelled and M is activated. As a result, all programmed coordinates are measured from M.

2.

Axis rotation



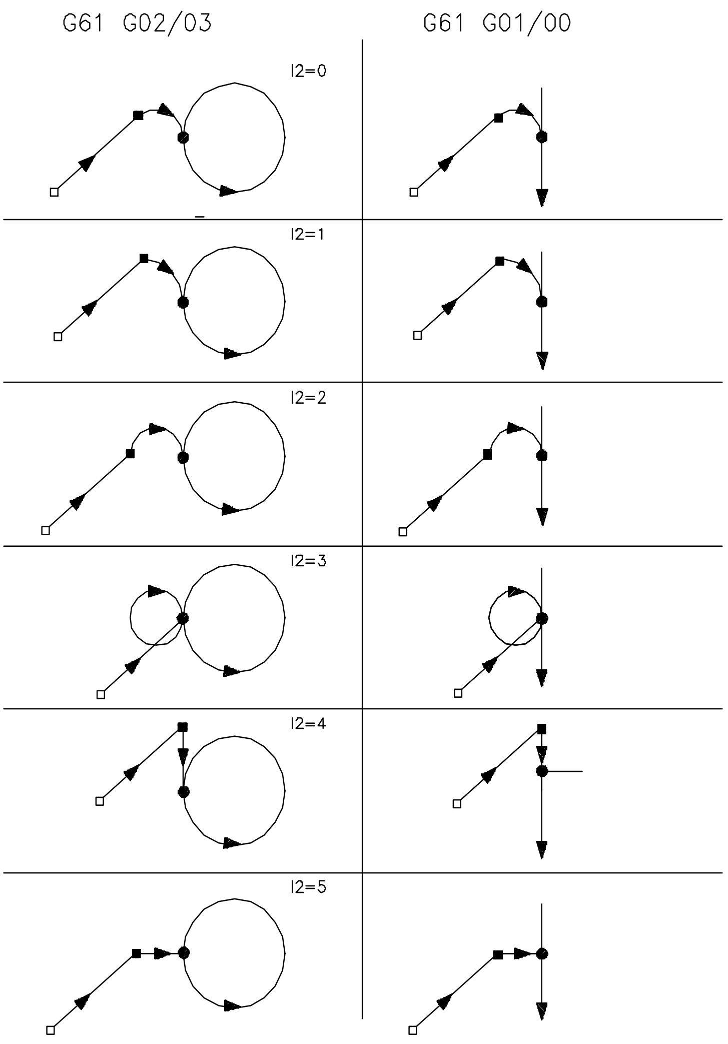
1 Workpiece 1
2 Workpiece 2
3 Machine table

Entry in the zero point table and calling:

N60 G54 I1 X-42 Y-15 B4=14 (Z0 C0)	The zero offset values are entered in the zero offset table. Machine workpiece 1. All programmed coordinates are measured from M1.
N120 G54 I2 X10 Y24 B4=-17	Machine workpiece 2. All programmed coordinates are measured from M2.

23.34 Tangential approach G61

Programs a tangential approach movement between a starting point and the start of a contour.



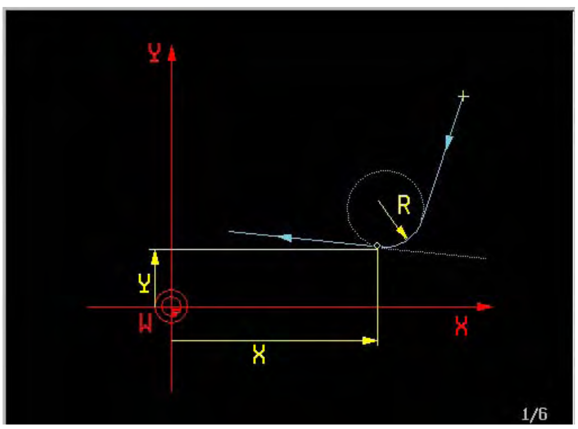
Actual position.

Calculated starting position in the plane.
Tool axis can be programmed.
Z1 in G17)

Contour starting position (X, Y, Z).

TANGENTIAL CONTOUR APPROACH G61
N... G61 {I2=..} X... Y... Z... R... [{X1=..} {Y1=..} {Z1=}] {I1=} {F2=}
N... G61 {I2=..} B2=... L2=... Z... R... [{X1=} {Y1=}] {Z1=} {I1=} {F2=}

Parameters




```
G    Tangential approach
X    Endpoint tangential approach
Y    Endpoint tangential approach
Z    Endpoint tangential approach
P    Point definition number
R    Radius
Z1=  Startpoint in Z
B2=  Polar angle
?90= Endpoint abs. (X,Y,Z..)
?91= Endpoint incr. (X,Y,Z..)
I1=  Linear movement 0=rapid,1=feed
I2=  Tangential approach definition
L2=  Polar length
```

I2=0 with line and circle
I2=1 with quarter circle
I2=2 with semicircle
I2=3 Helix for feeding (for pockets)
I2=4 Parallel to contour
I2=5 Vertical

Notes

The control system itself calculates a starting position. The first movement is a positioning movement to the calculated starting point. The approach movement starts from this point.

The approach movement consists of two different movements. First, the rapid traversing or feed movement (determined by I1=) to the (calculated) starting point of the approach movement. Secondly, a feed movement along the approach contour to the starting point of the contour.

The approach side is determined by the active function G41/G42. When G40 is active, there will be an approach movement, similarly to G41.

If radius compensation (G41/G42 without travel in the program block) is activated directly before the G61 block, compensation takes place during linear movement. Depending on the actual position, the movement will be closer to or farther away from the approach circle.
If radius compensation is already active, both the linear and circular movements will be carried out with radius compensation.

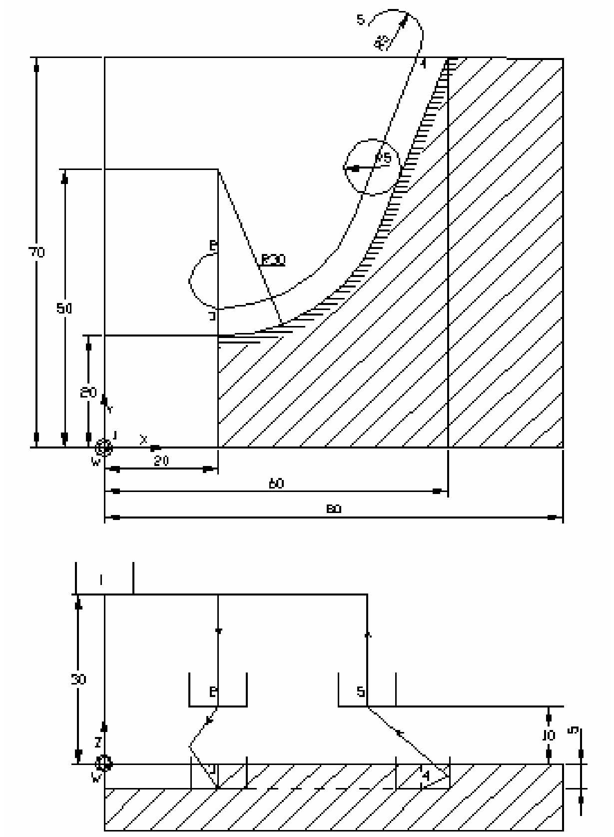
If no G function has been programmed after the G61 block, G1 is not active automatically. The last movement of the G61 function may be G1, G2 or G3.

If the distance between the actual position and the approach circle exceeds the milling radius (I2=0), the approach movement consists of a line and circular arc.
If the distance between the actual position and the approach circle is smaller than the milling radius, I2=0 is changed to I2=1, and the approach movement will be a quarter circle.

G61 programming is subject to the following limitations: G61 is not allowed in the ICP and G64 modes, in the MDI mode and in the G182 mode.

Specific limitations apply to the blocks immediately following the approach movement (G61). Only the following functions G64, G0, G1, G2 and G3 with movements in the active plane are allowed.
Falls nach dem G61-Satz keine G-Funktion programmiert worden ist, wird G1 nicht automatisch wirksam. Die letzte Bewegung der G61 Funktion kann G1, G2 oder G3 sein.

Example



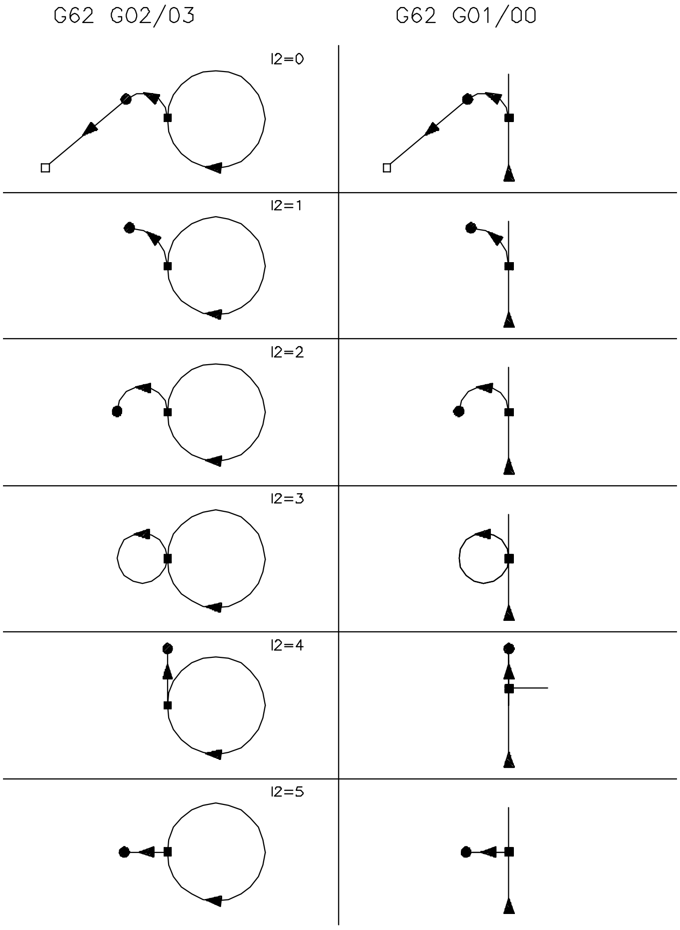
N1 G17
N2 T1 M6 (cutter R5)
N3 F500 S1000 M3
N4 G0 X0 Y0 Z30 — Approach starting position. (position 1: X0 Y0 Z30).
N5 G41 — Radius compensation to the left.
N6 G61 I2=2 X20 Y20 Z-5 Z1=10 R5 I1=0 F2=200

Tangential approach movement (I2=2) with semicircle. The initial part of the approach movement is a rapid traversing movement with positioning logic to the starting point of the semicircle (position 2: X.. Y.. Z10). Radius compensation is activated for this movement. The circular arc is made as a helix. The contour starts at position X20 Y20 Z0 (position 3: X20 Y25 Z-5)

N7 G64
N8 G3 I20 J50 R1=0
N9 G1 X60 Y60
N10 G63
N11 G62 I2=2 Z1=10 R5 — Tangential exit (I2=2) with semicircle. The semicircle is made as a helix. Starting height of Z axis -5. The end height is 10. (position 5: X.. Y.. Z10).
N12 G40
N13 G0 X0 Y0 Z30
N14 M30

23.35 Tangential exit G62

Programs a tangential exit after the end of the contour.



End position of the contour..

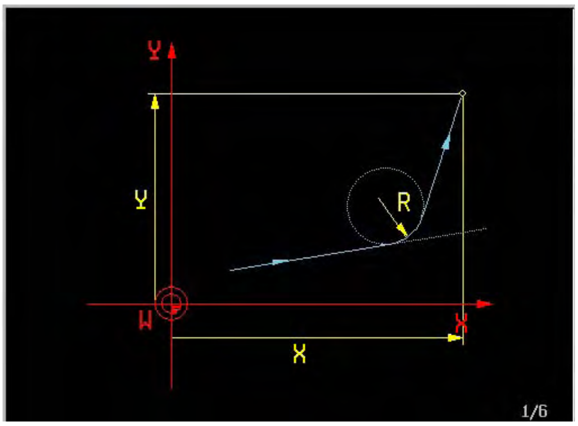
Calculated end position the plane. Tool axis Z (G17). Z1 can be programmed. The height remains unchanged when Z1 has not been programmed.

Programmed end position of the exit movement (X, Y, Z) (only I2=0).

TANTENTIAL EXIT FROM THE CONTOUR G62:

N... G62 I2>0 Z1=... R... {I1=} {F2=}
N... G62 I2=0 X... Y... Z... Z1=... R... {I1=} {F2=}
N... G62 I2=0 B2=... L2=... Z... R... {I1=} {F2=}

Parameters



G	Tangential exit
X	Endpoint tangential exit
Y	Endpoint tangential exit
Z	Endpoint tangential exit
P	Point definition number
R	Radius
Z1=	Startpoint in Z
B2=	Polar angle
?90=	Endpoint abs. (X,Y,Z..)
?91=	Endpoint incr. (X,Y,Z..)
I1=	Linear movement 0=rapid,1=feed
I2=	Tangential exit definition
L2=	Polar length

I2=0 with line and circle
I2=1 with quarter circle
I2=2 with semicircle
I2=3 Helix for feeding (for pockets)
I2=4 Parallel to contour
I2=5 Vertical

Notes

First read G61 to understand G62.

Notes and usage

If radius compensation (G40 without travel in the program block) is turned off immediately before the G62 block, compensation will be deactivated during the tangential exit movement. If radius compensation with G40 is not deactivated, both the circular and the linear movement will be with radius compensation.

Limitations

Programming of G62 is subject to the following limitations:

- G62 is not allowed in the ICP and G64 modes
- G62 is not allowed in the MDI mode
- G62 is not allowed in the G182 mode

Specific limitations apply to the blocks immediately following the approach movement (G61). Only the following functions are allowed:

- G64
- G0, G1, G2, G3 with movements in the active plane

Example

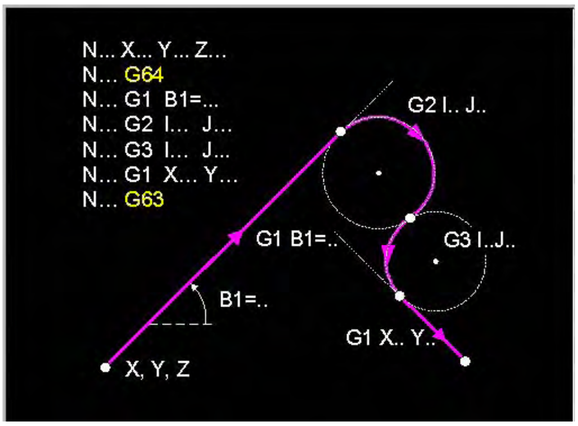
Refer to example of G61.

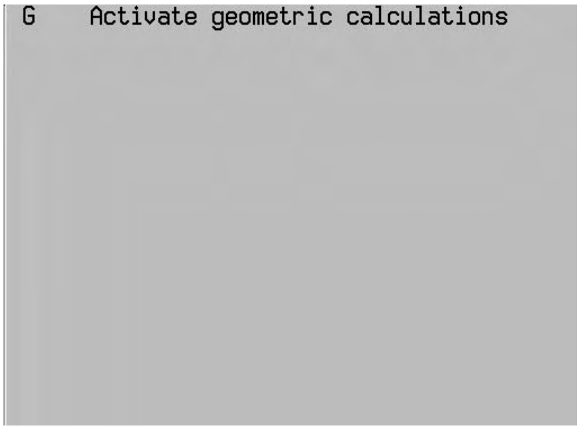
23.36 Cancel / activate geometric calculations G63/G64

G63: Cancels the geometry calculation

G64: Activates the geometry calculation

Parameters: G64 active

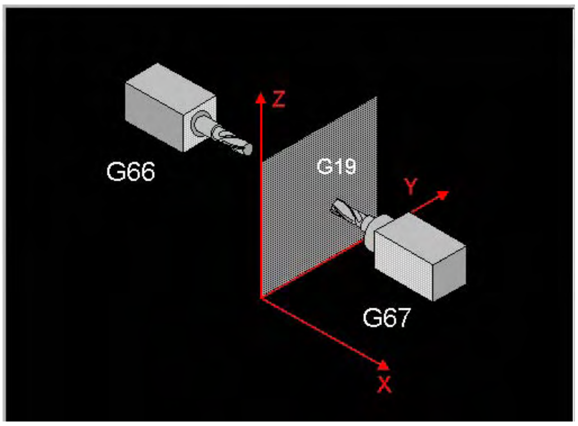


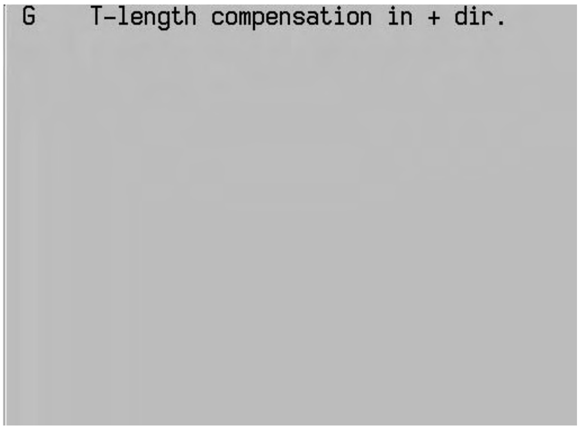


Note

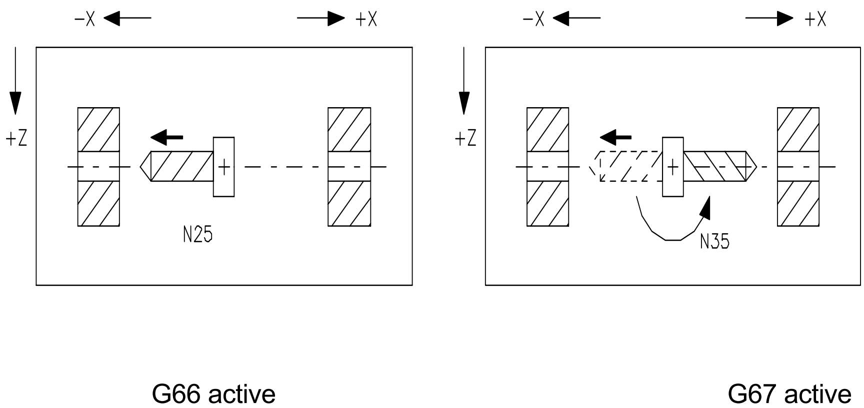
Programs that require geometry calculation can be easily created by the user with Interactive Contour Programming (ICP).
(see chapter on Interactive Contour Programming)

23.37 G66/G67 Select negative/positive tool direction





Example



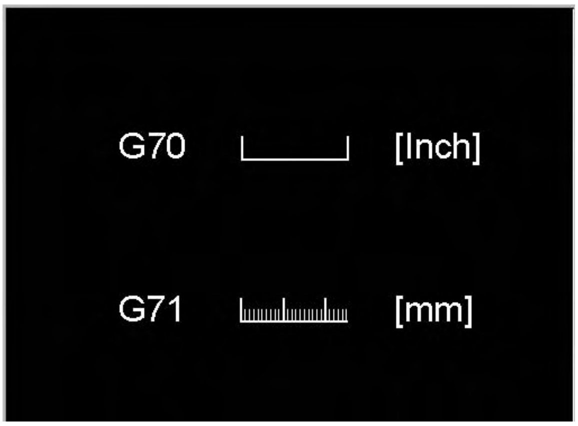
N25 G1 [End point coordinates]	First hole is drilled.
N30 G67	Select tool to point in the positive direction of the tool axis.
N35 G1 [End point coordinates]	Second hole is drilled.

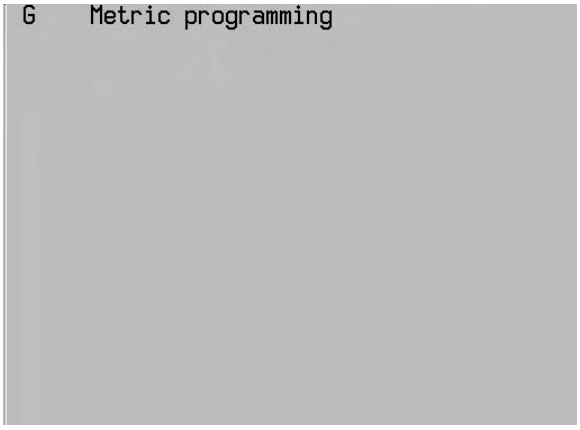
23.38 INCH / METRIC programming G70/G71

Loads and calls part programs that are described in the alternative unit of measurement to that specified in the CNC (unit of measurement defined in the machine constants)

Inch programming:
N... (PROGRAM NAME) G70

Metric programming:
N... (PROGRAM NAME) G71





Examples

1st unit of measurement: CNC: Metric Program: Inch

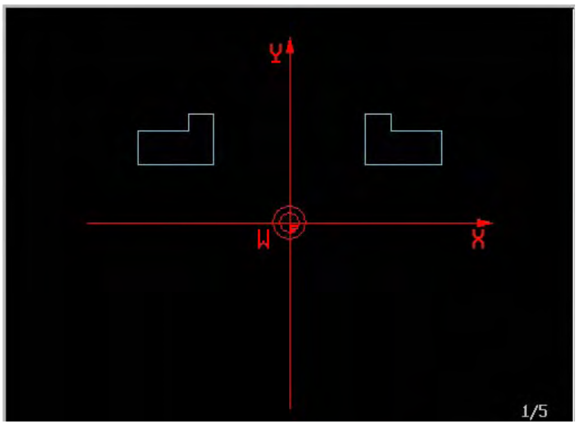
9001.PM
N9001 G70
:
N50 G1 X2 Y1.5 F8 Read-in ensures storage of X50.8 Y38.1 and F203.2.
:

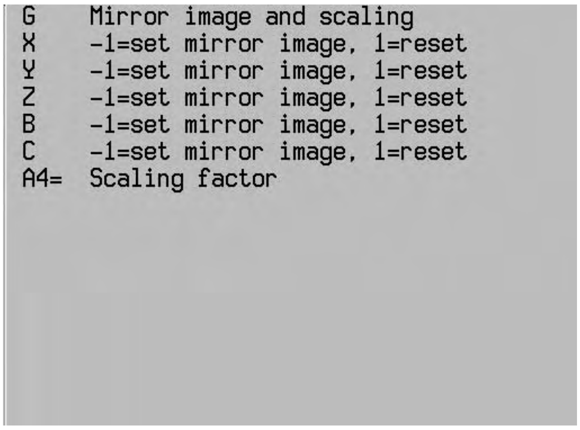
2nd unit of measurement: CNC: Inch Program: Metric

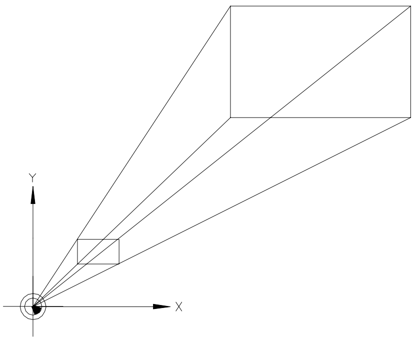
9001.PM
N9002 G71
:
N50 G1 X50.8 Z38.1 F203.2 Read-in ensures storage of X2 Y1.5 and F8.
:

23.39 Cancel / activate mirror image and scaling G72/G73

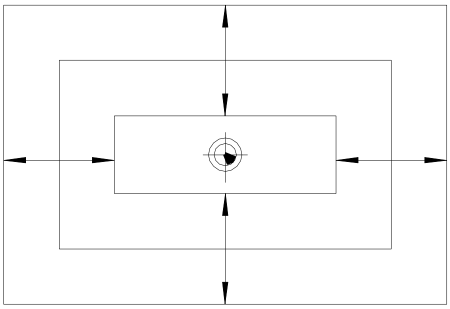
Enable zoom/reduce:
N.. G73 A4=.. (factor or percentage, setting in machine constants)
Disable zoom/reduce:
N.. G73 A4=1 (factor)
N.. G73 A4=100 (percentage)
Mirroring about an axis or sign change per axis:
N.. G73 {X-1} {Y-1} {Z-1} {A-1} {B-1} {C-1}
Disable mirroring/sign change per axis:
N.. G73 {X1} {Y1} {Z1} {A1} {B1} {C1}
Disable zoom/reduce and mirroring:
N.. G72



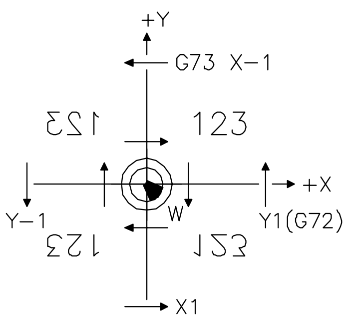




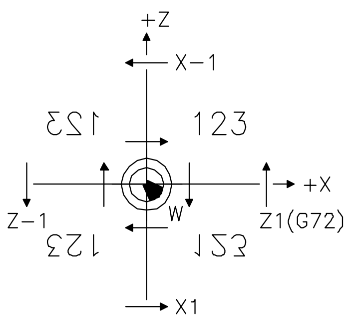
G73 A4=2



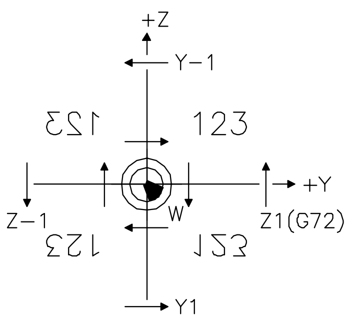
G73 A4=0.5



XY-Ebene (G17)



XZ-Ebene (G18)

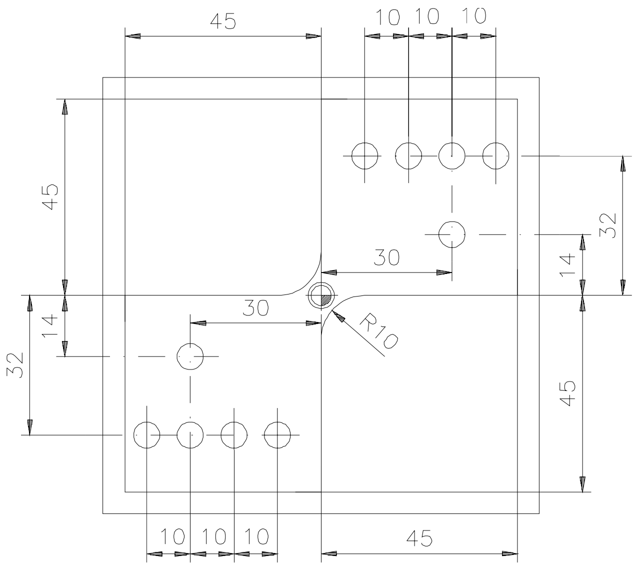


YZ-Ebene (G19)

Parameters

G72	No parameters	
G73	Zoom / reduce	
	Mirroring / sign change	
	A4=	Scaling factor

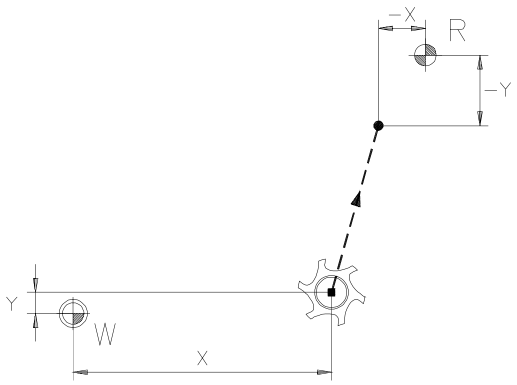
Example




```
N7273  (MIRROR AN ISLAND)
N1 G17
N2 G54
N3 T1 M6 S2000 F200          Change tool
N4 G0 X-60 Y20 Z0 M3
N5 G1  Z-9
N6 G43 Y0
N7 G41 X-10
N8 G3 X0 Y10 R10
N9 G1 X0 Y45
N10 G1 X45 Y45
N11 G1 X45 Y-10
N12 G40
N13 G1 Z10
N14 G73 X-1 Y-1              Mirror coordinates around X and Y axes
N15 G14 N1=4 N2=13           Repeat the blocks 4-13
N16 G72                      Cancel mirroring
N17 S1000 F100 T6 M6         Change tool 6
N18 G81 Y5 Z-20
N19 G79 X30 Y14
N20 G79 X10 Y32
N21 G79 X20 Y32
N22 G79 X30 Y32
N23 G79 X40 Y32
N24 G73 X-1 Y-1              Mirror coordinates around X and Y axes
N25 G14 N1=19 N2=23          Repeat the blocks 19-23
N26 G72                      Cancel mirroring
N27 G0 Z50 M30
```

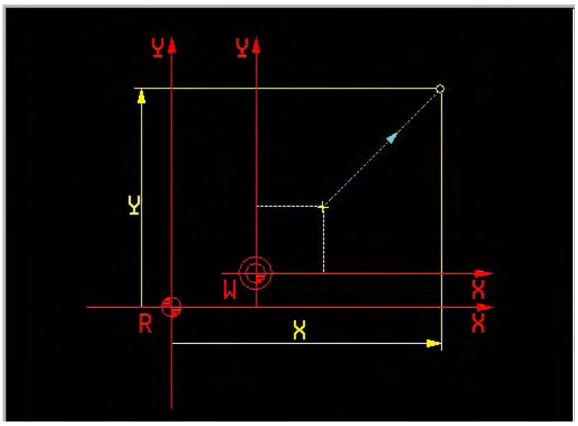
23.40 Absolute position G74

Rapid movement to a position whose coordinates refer to the reference point



N... G74 X.. Y.. Z.. {X1=..} {Y1=..} {Z1=..} {K...} {L...} {K2=...)

Parameter



G	Absolute position
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
B	Endpoint angle
C	Endpoint angle
K	0=inpod / 1=inpos / 2=MC or K2=
L	0:with toollength, 1=without
X1=	Absolute position MC (1-18)
Y1=	Absolute position MC (1-18)
Z1=	Absolute position MC (1-18)
B1=	Absolute position MC (1-18)
C1=	Absolute position MC (1-18)
K2=	Inpoc window (0: MC, 1-32767 µm)

Notes and usage

The G74 function is mainly used in programming cycles for tool changers, pallet stations etc., if the programmed coordinates should not depend on the coordinates used to define machining of the workpiece.

The end point coordinate may be defined in two different ways:

1) X100: relative position to reference point
2) X100 X1=2: relative position to the absolute position of the machine constant

Machine positions 1 to 9 and 10 10 18 for the first axis can be determined using the machine settings MC3145 -- MC3154 and MC3158 – MC3165. The machine settings MC3245 -- MC3254 and MC3258 – MC3265 etc are used for the second axis.
If the value in the machine setting being used is zero, no drive movement is implemented.

With G74 there will be travel simultaneously in all programmed axes. The next travel does not start until the required position is reached in all axes.

K0: Allowance is made for an (accurate) stop between the movement of block G74 and the movement in the next block, as is usual for rapid traversing movements.
(K0 is the switch-on position).

K1: No allowance is made for a stop between the movement of block G74 and the movement in the next block (rounding). The next movement starts when the required position has almost been reached in all axes.

K2: No allowance is made for a stop between the movement of block G74 and the movement in the next block. The next movement starts when the required position has almost been reached in all axes. This position is defined by the machine constant (MC136) (K2=0) or the window size (K2=...) for the corner release distance.
K2= window size in mm (0-32.766 mm)

If an incremental movement is programmed after a G74 movement, the coordinates relate to the position indicated in the G74 block.

Tool length compensation is usually not applied in G74 (L0 is switch-on position). L1 must be programmed for tool length compensation.

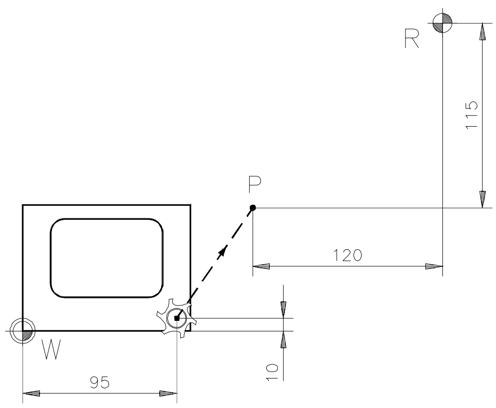
Radius compensation (G41...G44) should be cancelled before activation of the G74 function.

Geometry function G64 must not be active during G74.

The active zero offset is ignored for the G74 block.

The travel immediately preceding G74 should be programmed with G0 or G1. The travel immediately following G74 is automatically executed with the same G function.

Example



The coordinates of P in relation to R are known. P is programmed as follows:
N10 G0 X95 Y10
N11 G74 X-120 Y-115 Movement from X95 Y10 to P

Example of block:
N20 G74 X100 X1=1 Y123.456 Z1=10 K2 K2=25.2

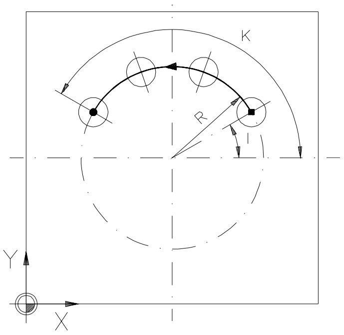
X100 X1=1	Relative position to the absolute position of the machine constant.
Y123.456	Relative position to the reference point.
Z1=10 (Z0)	Position related to the absolute position of the machine constant.
K2	No allowance is made for a stop between the movement of block G74 and the movement in the next block. The next movement starts when the required position has almost been reached in all axes. This position is defined by the window size (K2=...) for the corner release distance.
K2=	Window size in mm

23.41 Bolt hole cycle G77

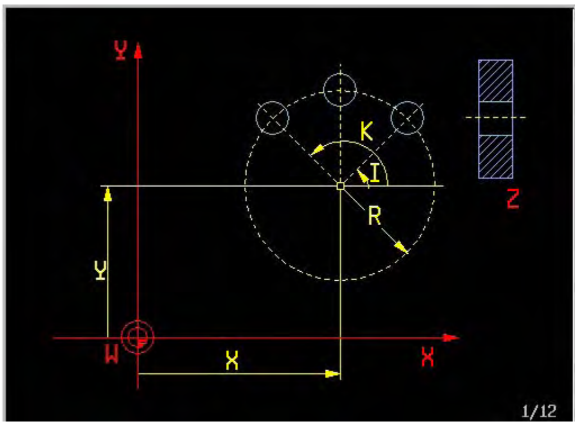
Executes a previously programmed drilling or milling cycle at points spaced equally on a circular arc or full circle.

Points on a circular arc:
N.. G77 [centre point] R.. J.. I.. K.. {B1=..}

Points on a full circle:
N... G77 [centre point] R.. J.. I.. {B1=..}



Parameter

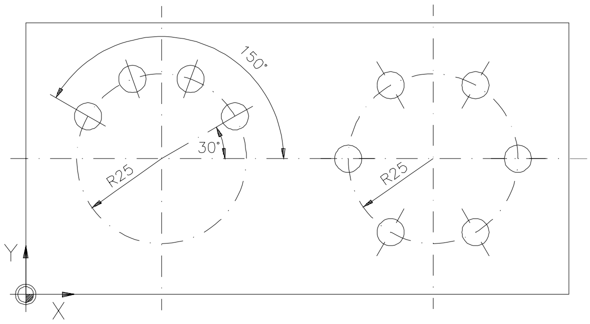


G	Bolt hole circle
X	Center point coordinate
Y	Center point coordinate
Z	Center point coordinate
B	Endpoint angle
C	Endpoint angle
I	Angle to first point
J	Number of points
K	Angle to last point
R	Circular pattern radius
B1=	Angle
B2=	Polar angle
?90=	Centre point abs. (X,Y,Z..)
?91=	Centre point incr. (X,Y,Z..)
L1=	Path length
L2=	Polar length
P1=	Point definition nr.for centre

Hinweis

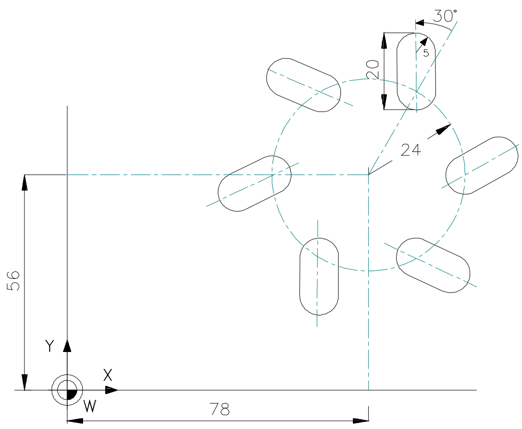
B1= hat zwei Bedeutungen:
Es stellt den Winkel für das Drehen einer Tasche bzw. Nute dar, oder die Lage des Kreismittelpunktes (B1= mit L1=, oder X/Y mit B1=).

Exemples



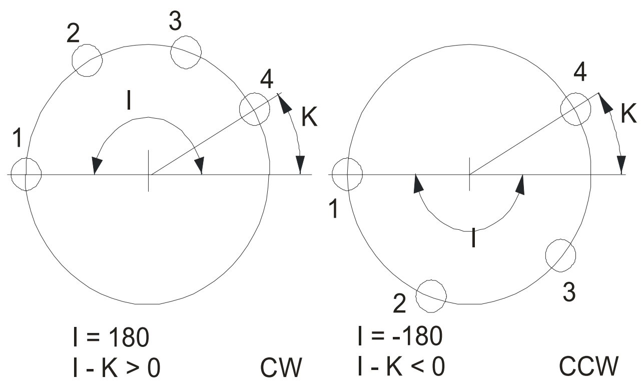
N40 G78 P2 X.. Y.. Z..	Second defined point
:	
N50 G81 Y1 Z-10 F100 S1000 M3	Define cycle
:	
N60 G77 P2 R25 I30 K150 J4	Repeat cycle four times on circular arc
N41 G78 P1 X.. Y.. Z..	First defined point
:	
N50 G81 Y1 Z-10 F100 S1000 M3	Define cycle
:	
N60 G77 P1 R25 I0 J6	Repeat cycle six times on full cycle

Turned grooves.



N60 T1 M6	Change tool 1 (cutter with radius of 4.8 mm)
N65 G88 X20 Y10 Z-10 B1 F100 S1000 M3	Define the groove as if the sides were parallel to the X and Y axes.
N70 G77 X78 Y56 Z0 R24 I0 J6 B1=30	The turned grooves are milled.

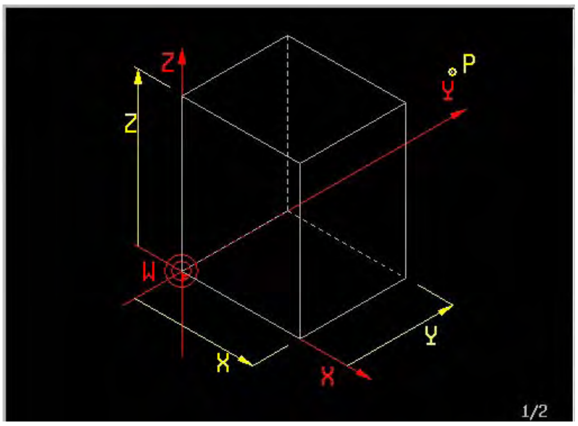
Direction of the bores on a circular arc



N50 G81 Y1 Z-10 F100 S1000 M3	Define cycle
N60 G77 X0 Y0 Z0 R25 **I180 K30** J4	Repeat cycle four times on the circular arc; start at 180 degrees, end at 30 degrees in clockwise direction (CW).
N70 G77 X0 Y0 Z0 R25 **I-180 K30** J4	Repeat cycle four times on the circular arc; start at -180 degrees, end at 30 degrees in counter-clockwise direction (CCW).

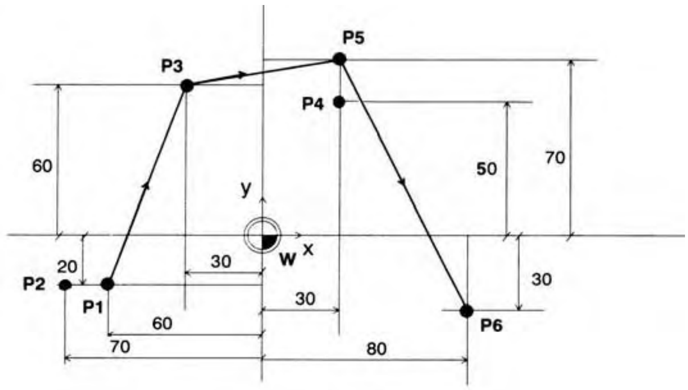
23.42 Point definition G78

Defines the coordinates of a point once in a program. For subsequent travel to this point, only its number needs to be programmed.
N... G78 P... [Point coordinates]




```
G    Point definition
X    Point coordinate
Y    Point coordinate
Z    Point coordinate
B    Point angle
C    Point angle
P    Point definition number
B2=  Polar angle
L2=  Polar length
```

Example



N10 G78 X-60 Y-20 P1	Define point 1
N11 G78 X-70 Y-20 P2	
N12 G78 X-30 Y60 P3	
N13 G78 X30 Y55 P4	
N14 G78 X30 Y70 P5	
N15 G78 X80 Y-30 P6	
:	
N90 G0 P1=1	Move tool in rapid traversing to the position defined by P1.
:	
N91 G1 P1=3 P2=5 P3=6 F1000	Move tool with programmed feed to P3, P5 and then to P6.
:	

Note

Only one point can be defined in each G78 block. All the point coordinates are in relation to the active work piece zero point W.

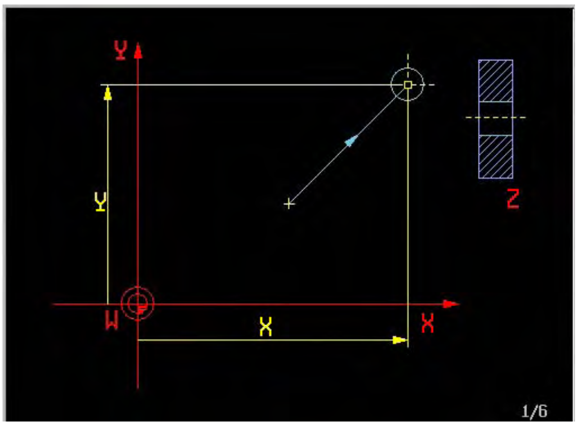
Program blocks with G1 or G79 can contain up to 4 points. In all other cases, there can only be one point in a program block.
Example: N.. G1 P1=9 P2=1 P3=3 P4=8
P adresses with index:
The index value (1-4) determines the priority for the machining sequence (1=highest priority, 4=lowest priority). The entry after the equals sign gives the number of points in the points memory.
Another possibility is to enter the point definitions as parameters, the index again denoting the priority.
P-Adresse mit Index:

23.43 Activate cycle G79

Executes previously programmed drilling cycles (G81, G83-G86) or milling cycles (G87-G89) at defined positions.

N... G79 [point coordinates] {B1=..}

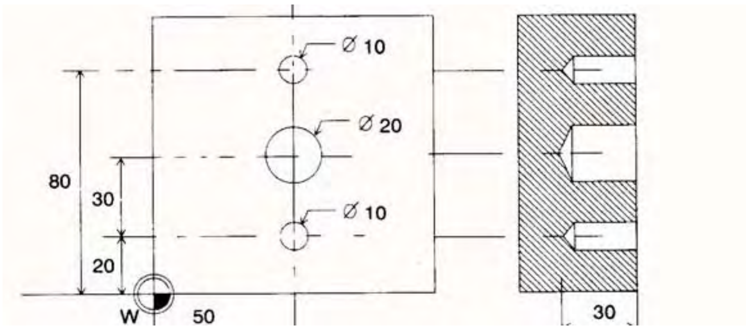
Parameters



G	Activate cycle
X	Point coordinate
Y	Point coordinate
Z	Point coordinate
B	Point angle
C	Point angle
B1=	Angle
B2=	Polar angle
?90=	Point abs. (X,Y,Z..)
?91=	Point incr. (X,Y,Z..)
L1=	Path length
L2=	Polar length
P1=	Point definition number
P2=	Point definition number
P3=	Point definition number
P4=	Point definition number

Example

Three holes are to be drilled



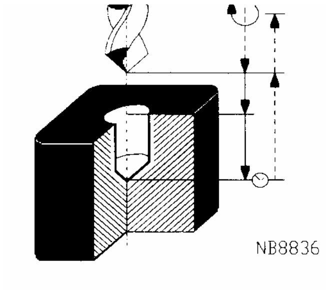
N50 G78 P1 X50 Y20 Z0	Define point
N55 G78 P2 X50 Y80 Z0	
N60 T1 M6	
N65 G81 Y1 Z-30 F100 S1000 M3	Define drilling cycle
N70 G79 P1 P2	Drill holes at positions 1 and 2
N75 T2 M6	
N80 G79 X50 Y50 Z0 M3	Drill hole

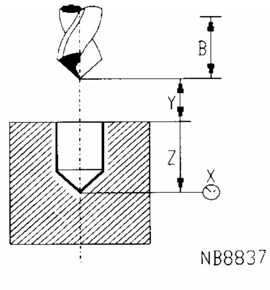
Note

B1= has two meanings:
It represents the angle for machining a pocket or slot, or the position of the circle centre point (B1= with L1=, or X/Y with B1=).
See example G77 "Turned grooves"

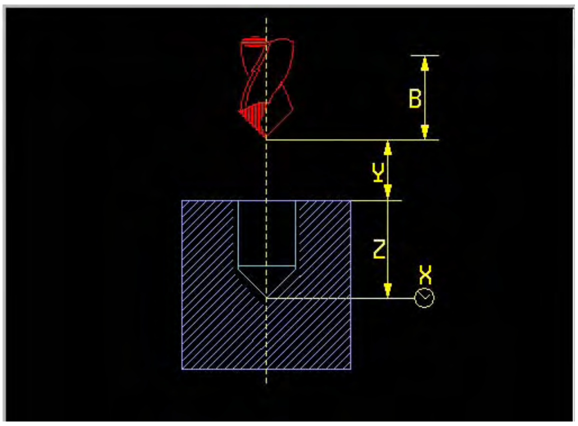
23.44 Drilling cycle G81

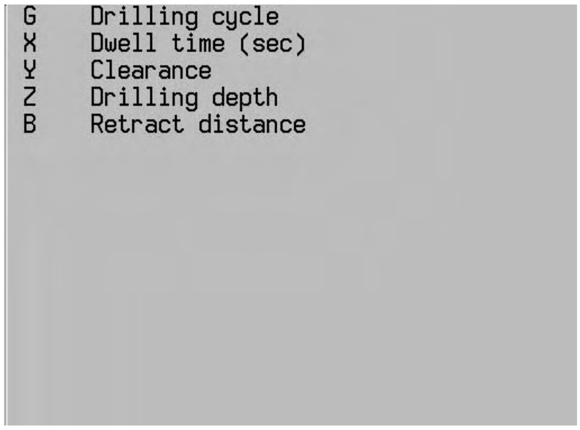
N.. G81 Z.. {X..} {Y..} {B..}



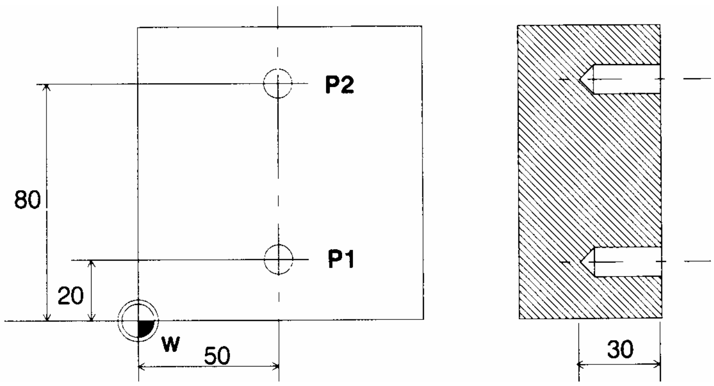


Parameter





Example



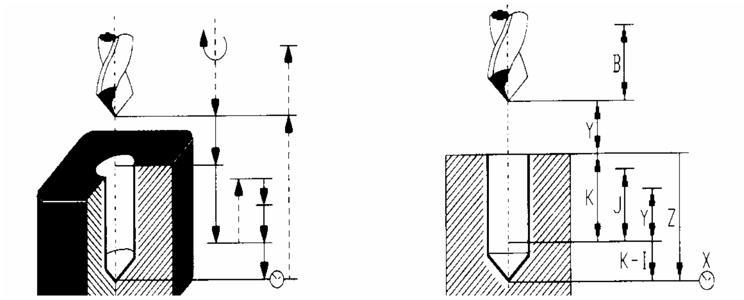
N50 G78 P1 X50 Y20 Z0	Define point 1
N55 G78 P2 X50 Y80 Z0	Define point 2
N60 G0 Z10 T1 M6	
N65 G81 X1.5 Y1 Z-30 F100 S500 M3	Define cycle
N70 G79 P1 P2	Execute cycle at positions 1 and 2

Note

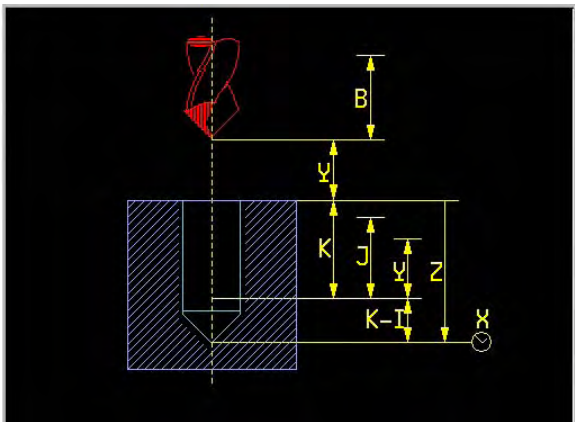
A machining cycle (G81-G89) is executed with G77 or G79.

23.45 Deep hole drilling cycle G83

N.. G83 Z.. {X..} {Y..} {B..} {I..} {J..} {K..} {K1=..}

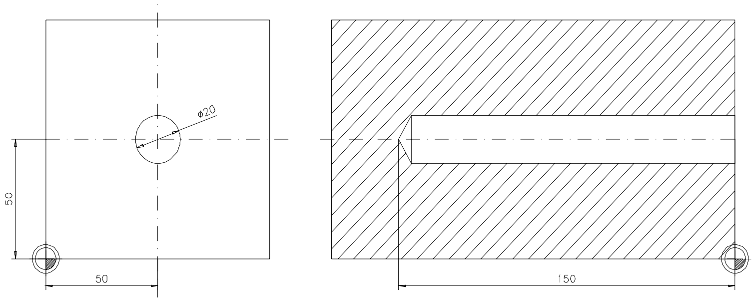


Parameter



G	Deep hole drilling cycle
X	Dwell time (sec)
Y	Clearance
Z	Overall drilling depth
B	Retract distance
I	Drilling depth decrement
J	Retract distance after step
K	Drilling depth first movement
K1=	Number of retract distances

Examples



1.

N5 T1 M6	
N10 G83 Y4 Z-150 I2 J6 K20 F200 S500 M3	Define cycle
N20 G79 X50 Y50 Z0	Execute cycle

2.

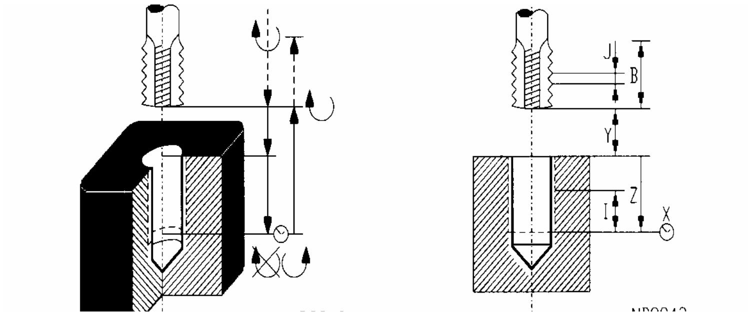
N.. G83 Y4 Z-150 I2 J6 K20 K1=3	Define cycle
N20 G79 X50 Y50 Z0	Execute cycle

Note

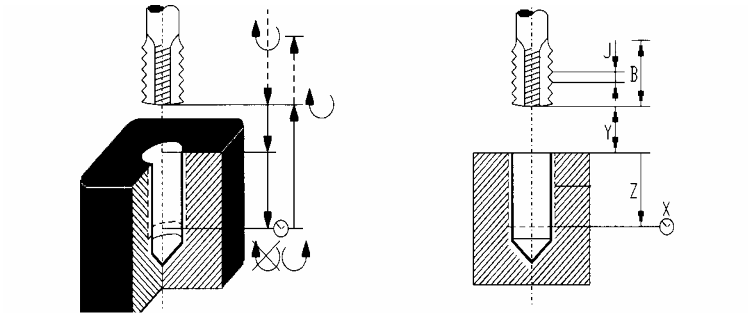
A machining cycle (G81-G89) is executed with G77 or G79.

23.46 Tapping cycle G84

N... G84 Z... {Y...} {B...} {J...} {X...}
or
N... G84 **I1=0** Z... {Y...} {B...} {J...} {X...}

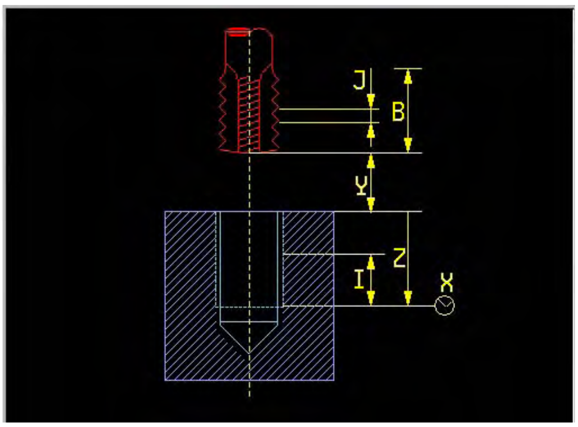


from V400:
Tapping can also be implemented in a closed control loop, as an interpolation between the tool axis and the spindle. The accelerating power of the spindle is taken up in this interpolation. In this way, the running of the spindle in the desired position and at the correct speed in rpm is guaranteed ("synchronous tapping").



N... G84 **I1=1** Z... {Y...} {B...} {J...} {X...}

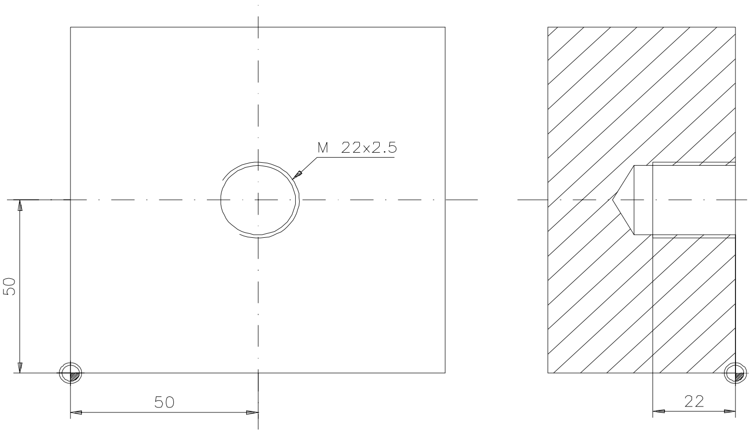
Parameter



G	Tapping cycle
X	Dwell time (sec)
Y	Clearance
Z	Tapping depth
B	Retract distance
I	Speed ramp (rev)
J	Pitch
I1=	Interpolation (0=without,1=with)

F(feed) = J(pitch) * S(speed)

Example



N14 T3 M6
N15 G84 Y9 Z-22 J2.5 S56 M3 F140 Define cycle
N20 G79 X50 Y50 Z0 Execute cycle

Note

A machining cycle (G81-G89) is executed with G77 or G79.

During call-up of a G84 cycle by means of G79 the soft key Clear control must be set for G94 operation (feed in mm/min) and not for G95 operation (feed in mm/360 degree turn). G94 is always to be programmed before G84.

From V400:
Tapping can be programmed with or without interpolation.
I1=0 guided (basic position, open position control loop)
I1=1 interpolating (closed position control loop)
An active "Process level G7 traverse" can only be operated with interpolation (I1=1)

As of V410,
if "Swivel working plane (G7)" is active, and the head has not been swivelled (tool axis is same as the Z axis), guided tapping can also be performed (I1=0).

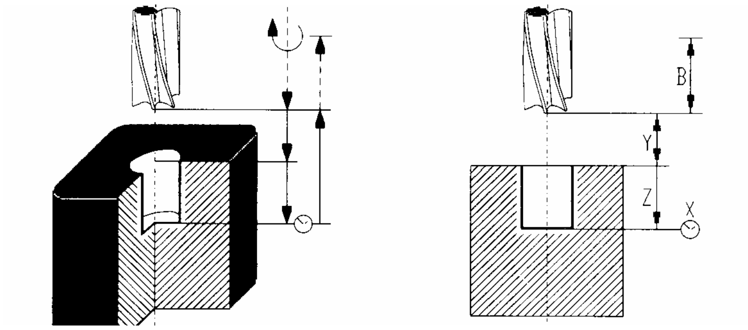
Tapping with start angle
By machines with interpolation (I1=1) the programming of an oriented Spindle stop (M19), with D-parameter 'Spindle angle value', gives the possibility for tapping with start angle.

Remark After the interpolated tapping with start angle (I1=1) the modal M-function (M3,M4) is not active more. This M-function will be overwritten by M19.

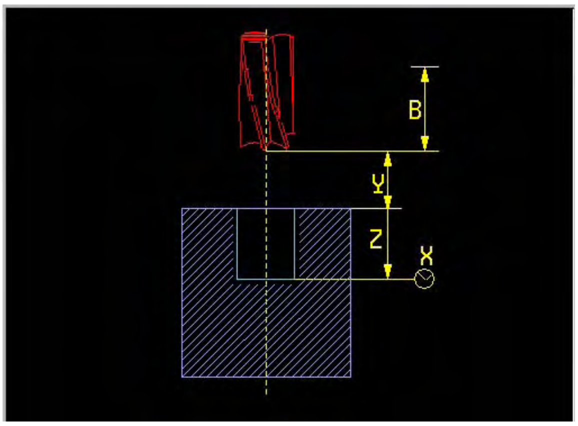
Machine settings
MC723 and MC727 are no longer required during interpolation. The machine settings of the spindle should be set correctly during tapping. The spindle acceleration is calculated for every operation with the help of MC2491, 2521, 2551, 2581 and MC2495, 2525, 2555, 2585. In every case, MC4430 should be active for good control.

23.47 Reaming cycle G85

N.. G85 Z.. {X..} {Y..} {B..} {F2=..}

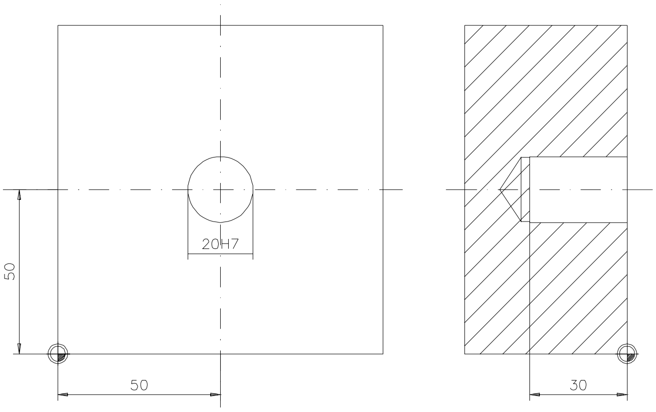


Parameter



G	Reaming cycle
X	Dwell time (sec)
Y	Clearance
Z	Reaming depth
B	Retract distance
F2=	Feedrate to startpoint

Example



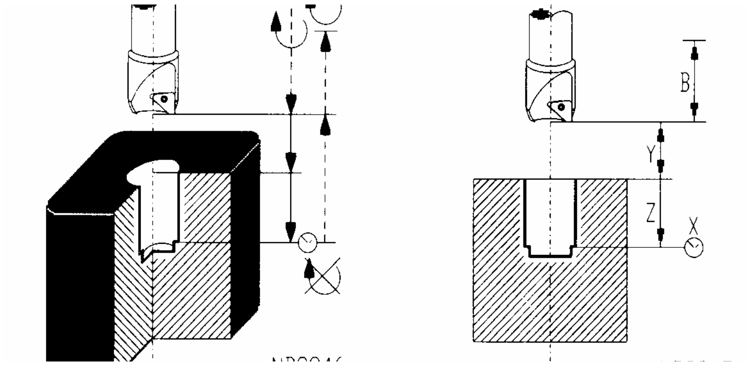
N25 T4 M6
N30 G85 X2 Y3 Z-30 F50 S100 F2=200 M3 Define cycle
N35 G79 X50 Y50 Z0 Execute cycle

Note

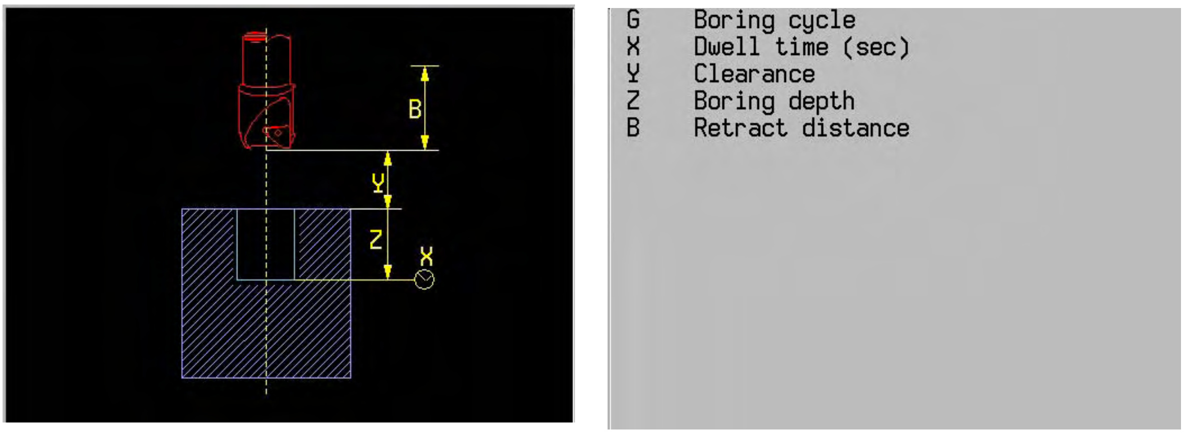
A machining cycle (G81-G89) is executed with G77 or G79.

23.48 Boring cycle G86

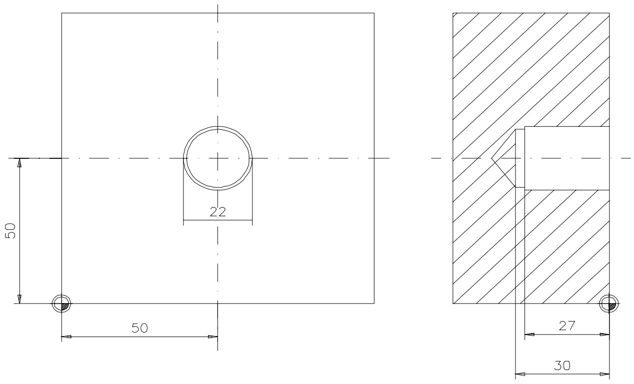
N.. G86 Z.. {X..} {Y..} {B..}



Parameter



Example



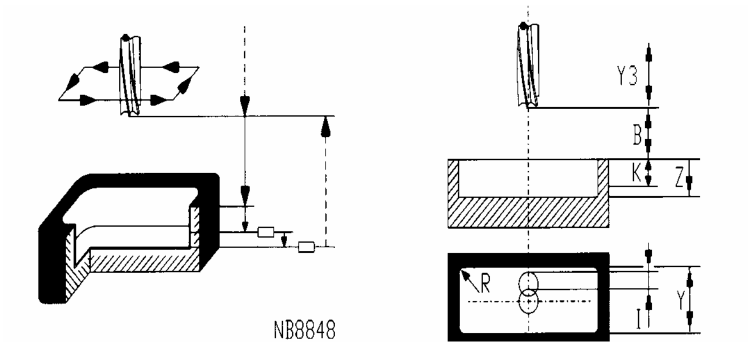
N45 T5 M6
N50 G86 X1 Y9 Z-27 B10 F20 S500 M3 Define cycle
N55 G79 X50 Y50 Z0 Execute cycle

Note

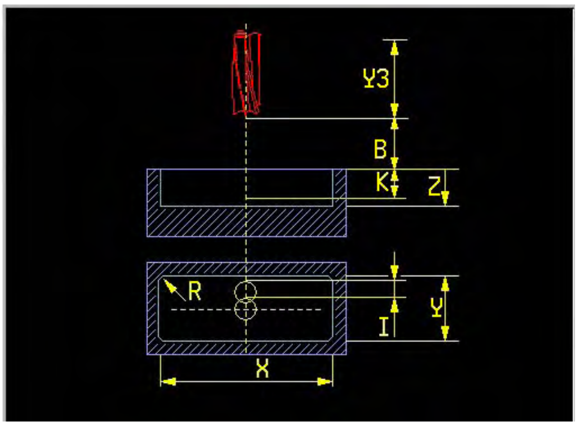
A machining cycle (G81-G89) is executed with G77 or G79.

23.49 Rectangular pocket milling cycle G87

N.. G87 X.. Y.. Z.. {R..} {B..} {I..} {J..} {K..} {Y3=..} {F2=..}

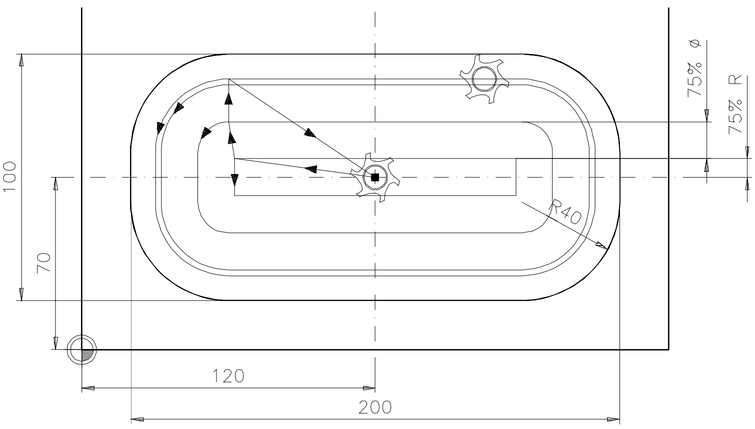


Parameter



G	Rectangular pocket milling cycle
X	Dimension parallel to X
Y	Dimension parallel to Y
Z	Total pocket depth
B	Clearance
I	Cutting width mill in %
J	1=climb, -1=conventional
K	Cutting depth
R	Corner radius
Y3=	Special retract distance
F2=	In depth feed (only this block)

Example



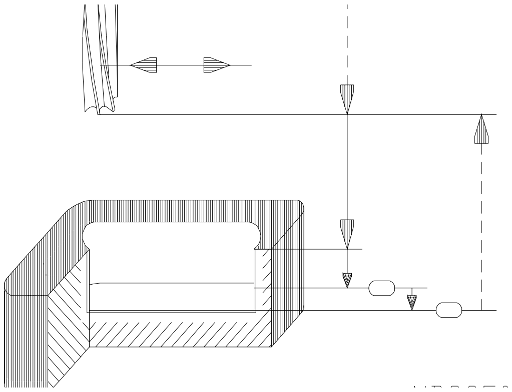
N10 T1 M6	
N20 G87 X200 Y100 Z-6 J+1 B1 R40 I75 K1.5 F200 S500 M3	Define cycle
N30 G79 X120 Y70 Z0	Execute cycle

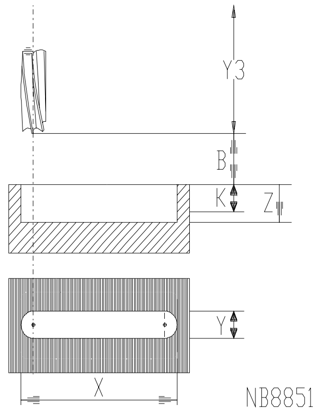
Note

A machining cycle (G81-G89) is executed with G77 or G79.

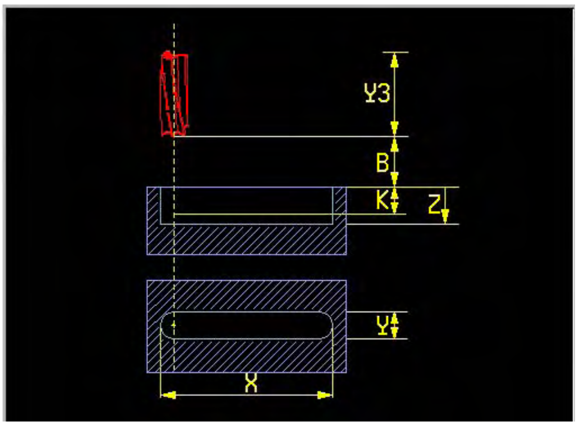
23.50 Groove milling cycle G88

N.. G88 X.. Y.. Z.. {B..} {J..} {K..} {Y3=..} {F2=..}



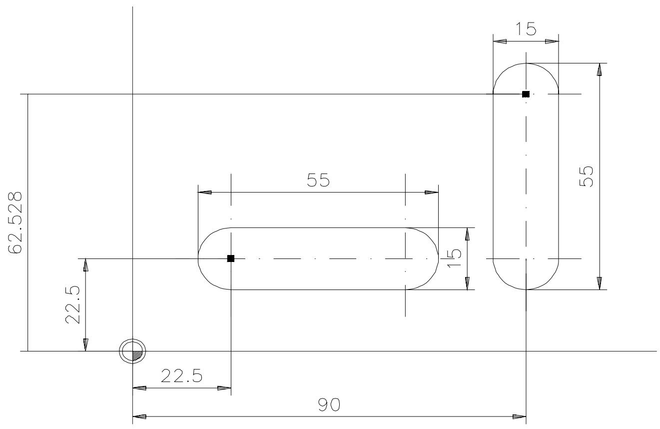


Parameter



G	Groove milling cycle
X	Dimension parallel to X
Y	Dimension parallel to Y
Z	Total groove depth
B	Clearance
J	1=climb, -1=conventional
K	Cutting depth
Y3=	Special retract distance
F2=	In depth feed (only this block)

Example



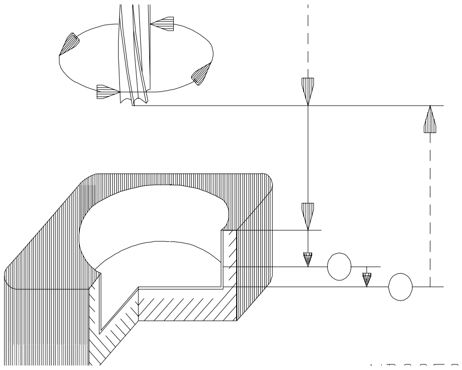

```
N10  S500  T1  M6
N20  G88  X55  Y15  Z-5  B1  K1  F350  Y3=10  F2=200  M3      Define cycle
N30  G79  X22.5  Y22.5  Z0                                     Execute cycle
N40  G88  X15  Y-55  Z-5  B1  K1  Y3=10  F2=200
N50  G79  X90  Y62.528  Z0
```

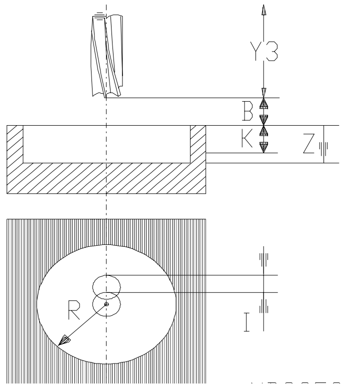
Notes

A machining cycle (G81-G89) is executed with G77 or G79.
The signs of X and Y determine the direction of the slot from the start point S.

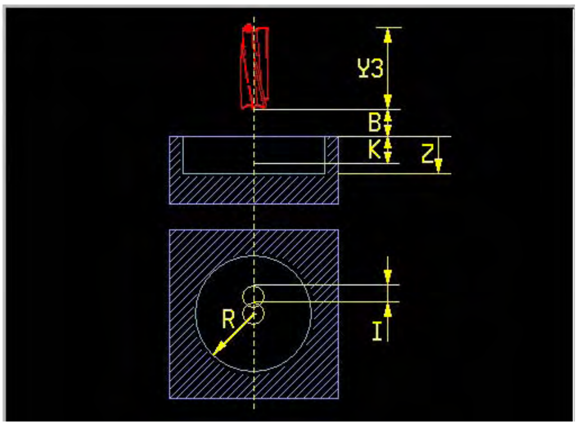
23.51 Circular pocket milling cycle G89

N.. G89 Z.. R.. {B..} {I..} {J..} {K..} {Y3=..} {F2=..}



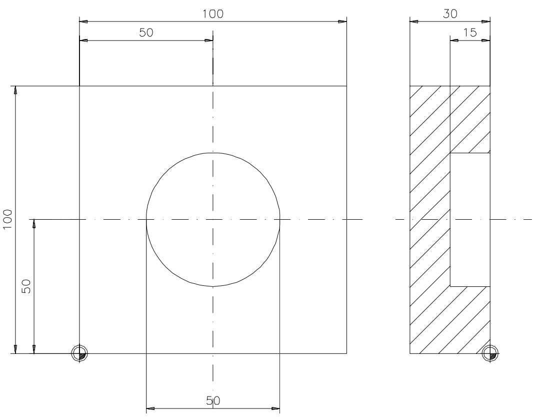


Parameter



G	Circular pocket milling cycle
Z	Total pocket depth
B	Clearance
I	Cutting width mill in %
J	1=climb, -1=conventional
K	Cutting depth
R	Radius circular pocket
Y3=	Special retract distance
F2=	In depth feed (only this block)

Example



N10 T1 M6	
N20 G89 Z-15 B1 R25 I75 K6 F200 S500 M3	Define cycle
N30 G79 X50 Y50 Z0	Execute cycle
N40 G0 Z200	

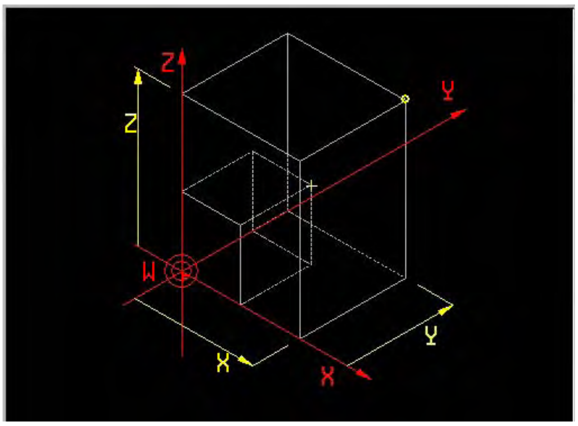
Note

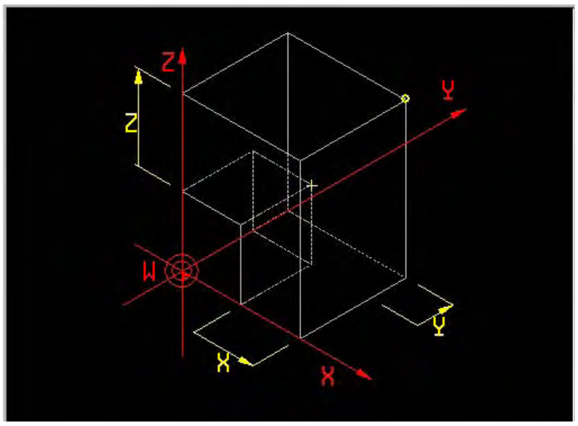
A machining cycle (G81-G89) is executed with G77 or G79.

23.52 Absolute/incremental programming G90/G91

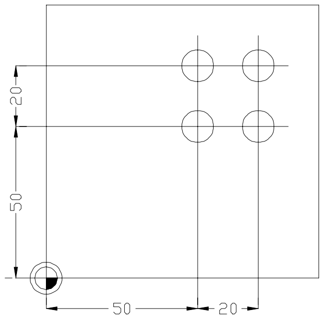
G90: Absolute coordinates, measured from the program zero point W.
G91: Incremental coordinates, relative to the last position.

N.. G90/G91





Example




```
N88550
N1 G17
N2 G54
N3 G98 X0 Y0 Z60 I100 J100 K-80    Graphic window definition
N4 S1300 T1 M6
N5 G81 Y2 Z-10 F200 M3              Define cycle
N6 G79 X50 Y50 Z0                   Execute cycle
N7 G91                      Change to incremental size programming
N8 G79 Y20                          Execute cycle
N9 G79 X20
N10 G79 Y-20
N11 G90                             Change to absolute size programming
```

Note

An absolute position must be programmed before the entry of the incremental dimension G91.

23.53 Wordwise absolute/incremental programming

Wordwise absolute/incremental programming independently of G90/G91.

absolute programming:
N.. G.. [axis name]90=...

incremental programming:
N.. G.. [axis name]91=...

Parameter

Achsname: X, Y, Z, U, V, W, I, J, K, A, B, C

Hinweise und Verwendung

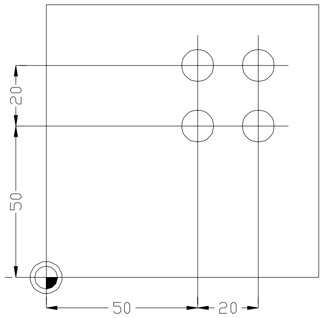
Kartesische Koordinaten:

Die wortweise Absolut-/Inkremental-Programmierung ist unabhängig vom modal gültigen Maßsystem G90/G91.

Polarkoordinaten:

Die Programmierung in Polarkoordinaten wird nicht beeinflußt.

Example



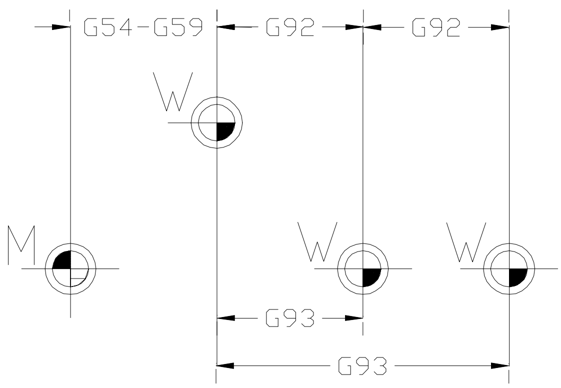
N88550	
N1 G17	
N2 G54	
N3 G195 X0 Y0 Z60 I100 J100 K-80	Define graphic window
N4 S1300 T1 M6 (drill bit R5)	Change tool 1
N5 G81 Y2 Z-10 F200 M3	Define drilling cycle
N6 G79 X50 Y50 Z0	Cycle call for 1st drilling
N7 G79 Y91=20	Cycle call for 2nd drilling, incremental movement
N8 G79 X91=20	Cycle call for 3rd drilling, incremental movement
N9 G79 Y91=-20	Cycle call for 4th drilling, incremental movement
N10 M30	

23.54 Zero point shift incremental / rotation or absolute rotation G92/G93

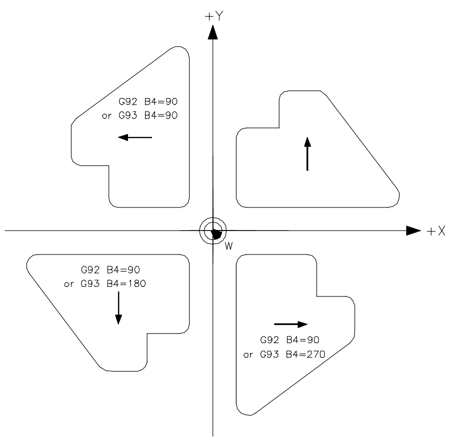
Zero point offset:
N.. G92 [incremental coordinate(s), in relation to the last program zero point]
N.. G93 [absolute coordinates, in relation to the zero point that was defined by G54-G59]

Rotation of the coordinates system:
N... G92/G93 B4=..

Zero point offset:



Rotation of the coordinate system:



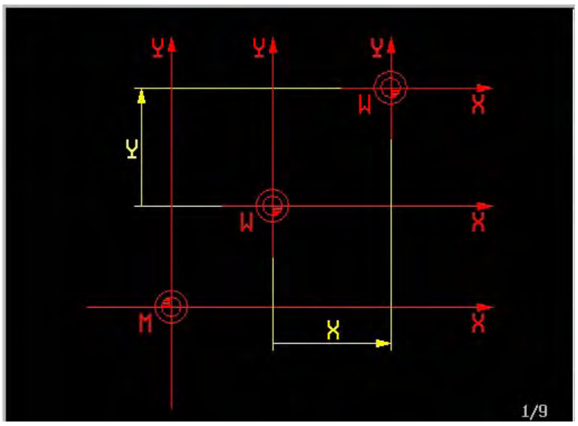
FSP: Driving up from the traverse position by the shortest route

FSP now always outputs an angle between -180 and +180 degrees. This is changed so that an angle is output between the end switches. This angle is then the shortest route. The disadvantage is that the position of the circular axis can climb to very high values which should be reset to a moment.
The disadvantage of very high positions is resolved with a separate function with which the (internal) position is reset to a value between 0 and 360 degrees.

G93 {X},{Y},{Z},{A},{B},{C},{B2=},{L2=},{P},{P1=},{B4=},{A3=1},{B3=1},{C3=1}

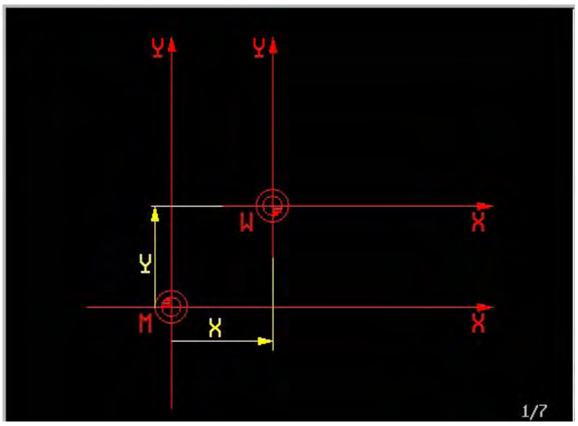
in which:
A3=1, B3=1, C3=1
The corresponding axial position is reset to a value between 0 and 360 degrees.

Parameter bei G92



G	Zero point shift incr./ rotation
X	Zero point coordinate
Y	Zero point coordinate
Z	Zero point coordinate
B	Zero point angle
C	Zero point angle
B1=	Angle
B4=	Angle of rotation incremental
L1=	Path length

Parameter bei G93

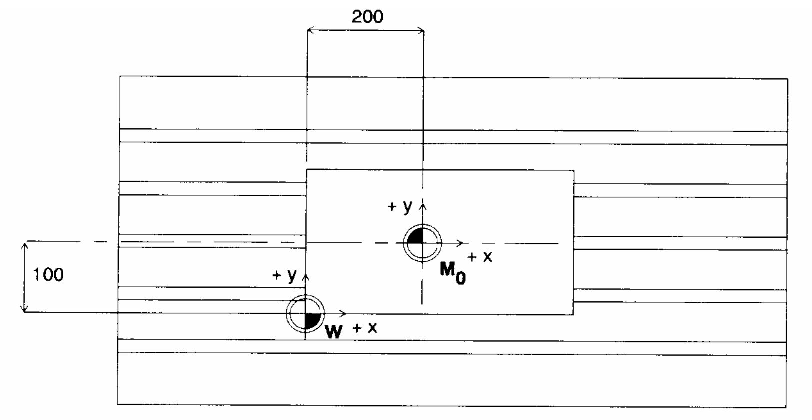


G	Zero point shift abs./ rotation
X	Zero point coordinate
Y	Zero point coordinate
Z	Zero point coordinate
B	Zero point angle
C	Zero point angle
B2=	Polar angle
B3=	1=Reset position 0-360 degrees
B4=	Angle of rotation absolute
L2=	Polar length
P1=	Point definition number
C3=	1=Reset position 0-360 degrees

Reset function **(V400 and higher)**
A3=,B3=,C3= Reset parameter
G93 A3=1 resets the position of the corresponding rotary axis to a value between 0 and 360 degrees.
Example: an A axis with a position of 370 degrees is modified to 10 degrees by entering G94 A3=1.

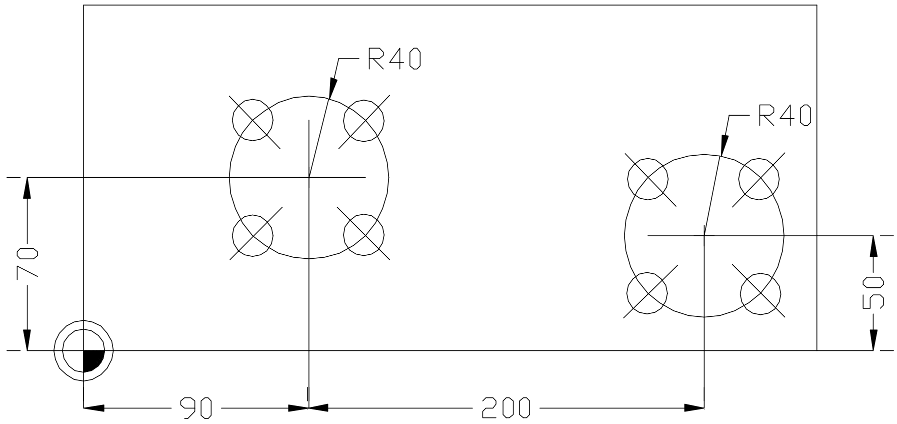
Examples

1. The centre point of the work piece coincides with the machine centre point (M). The program zero point (W) is placed in the left corner of the work piece.



N30 G93 X-200 Y-100

2. The four holes around points A and B are to be drilled. In the program, the program zero point (W) lies in A and B.



Program with G92
N79560
N1 G17
N2 G54
N3 G98 X-10 Y-10 Z10 I420 J180 K-30
N4 G99 X0 Y0 Z0 I420 J160 K-10
N5 F200 S3000 T1 M6
N6 G92 X90 Y70 Incremental zero offset
N7 G81 Y1 Z-12 M3 Define cycle
N8 G77 X0 Y0 Z0 I45 J4 R40 Call cycle
N9 G92 X200 Y-20 Incremental zero offset
N10 G14 N1=8 Repeat function
N11 G93 X0 Y0 Cancel incremental zero offset
N12 G0 Z100 M30

Program with G93
The program appears as follows in relation to the clamping zero point:
N79561
N1 G17
N2 G54
N3 G98 X-10 Y-10 Z10 I420 J180 K-30
N4 G99 X0 Y0 Z0 I420 J160 K-10
N5 F200 S3000 T1 M6
N6 G93 X90 Y70 Absolute zero offset
N7 G81 Y1 Z-12 M3
N8 G77 X0 Y0 Z0 I45 J4 R40
N9 G93 X290 Y50 Absolute zero offset
N10 G14 N1=8
N11 G93 X0 Y0 Cancel absolute zero offset
N12 G0 Z100 M30

Notes

If no G54-G59 has previously been activated, G92/G93 is effective from the machine zero point.

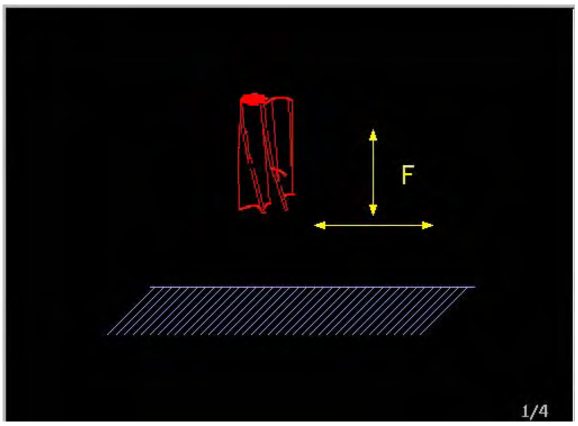
If rotation of the coordinate system (G92/G93 B4=..) is active, a zero point offset programmed with G92/G93 is no longer allowed.

23.55 Feed in mm/min (inch/min) / mm/rev (inch/rev) G94/G95

Information to the controller about how the programmed feed is to be evaluated.

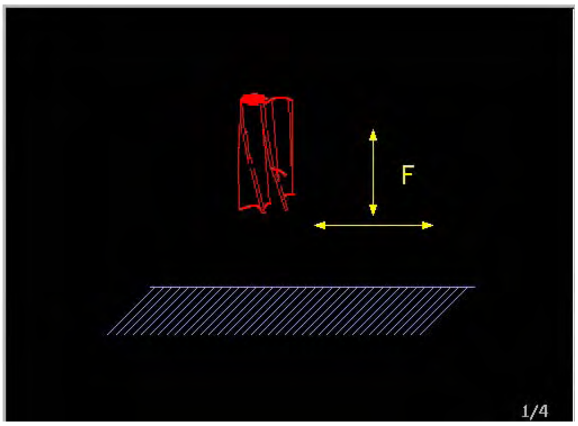
G94 : Feed in mm/min or inch/min
G95 : Feed in mm/rev or inch/rev
G94 F5= : Feed of rotary axes F5=0 degrees/min (default)
F5=1 mm/min or inches/min

Parameters G94



G	Feed in mm/min (inch/min)
F	Feed
F1=	F-adaptation:1=red.,2=r/h,3=high
F3=	In depth feed
F4=	In plane feed
F5=	Feed rotary Axes

G95



G	Feed in mm/rev (inch/rev)
F	Feed
F1=	F-adaptation:1=red.,2=r/h,3=high
F3=	In depth feed
F4=	In plane feed

Notes:

MACHINES WITH KINEMATIC MODEL
The G94 F5= function is only present if a kinematic model has been defined for the machine (MC312 must be active).

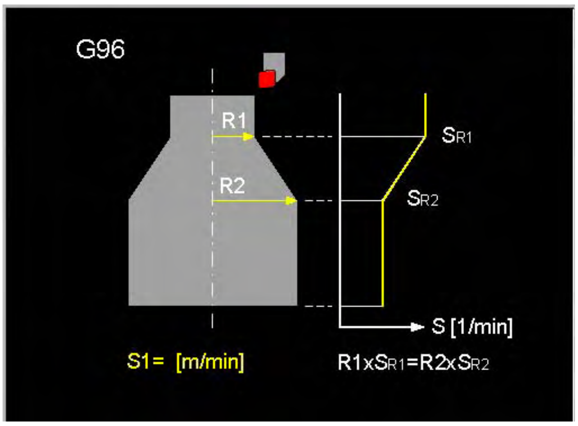
CALCULATION OF RADIUS OF ROTARY AXIS G94 F5=1
In machines with a kinematic model, the radius of the rotary axis between the centre point of the rotary axis and the workpiece can be calculated. This means that A40=, B40= or C40= no longer need to be programmed.

CANCEL G94 F5=1
G94 F5=1 is cancelled by: G94 F5=0, G95, programming with A40=, B40= or C40= in G0 or G1, M30, <Program abort> or <Reset CNC>.

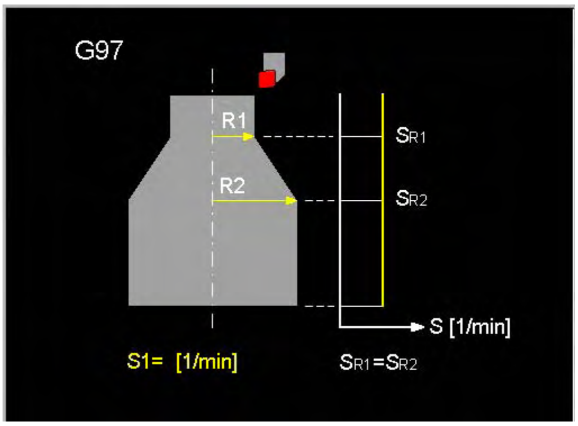
Examples

N.. G94	Feed in mm/min
N.. G1 X.. Y.. F200	Move to X.. Y.. at a feed of 200 mm/min
N.. G95	Feed in mm/rev.
N.. G1 X.. Y.. F.5	Move to X.. Y.. at a feed of 0.5 mm/rev.

23.56 G96/G97 Constant cutting speed



G	Constant cutting speed
D	Upper speed limit (rev/min)
F	Feed
S	Cutting speed (m(feet)/min)
M	Machine function
S1=	Cutting speed (m(feet)/min)
M1=	Machine function



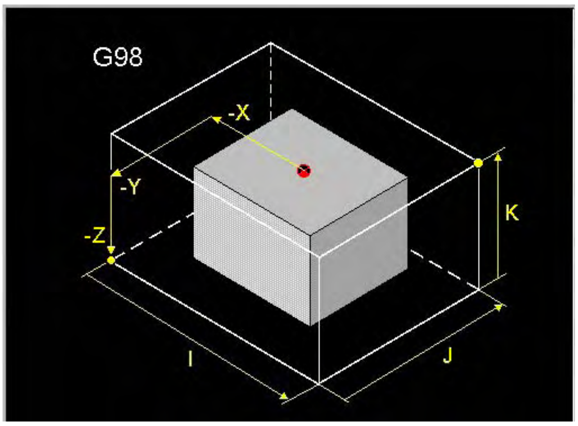
G	Spindle speed
S	Speed (rev/min)
M	Machine function
S1=	Speed (rev/min)
M1=	Machine function

Refer to chapter "Turning mode".

23.57 Graphic window definition G98

Defines the position relative to the program zero point W and the dimensions of a 3D graphics window in which the machining of the work piece is to be simulated graphically.

N.. G98 X.. Y.. Z.. I.. J.. K.. {B..} {B1=..} {B2=..}



G	Graphic window definition
X	Start point coordinate
Y	Start point coordinate
Z	Start point coordinate
B	Rotation around hor. axis (3D)
I	Dimension parallel to X
J	Dimension parallel to Y
K	Dimension parallel to Z
B1=	Rotation around vert. axis (3D)
B2=	Rotation around third axis (3D)

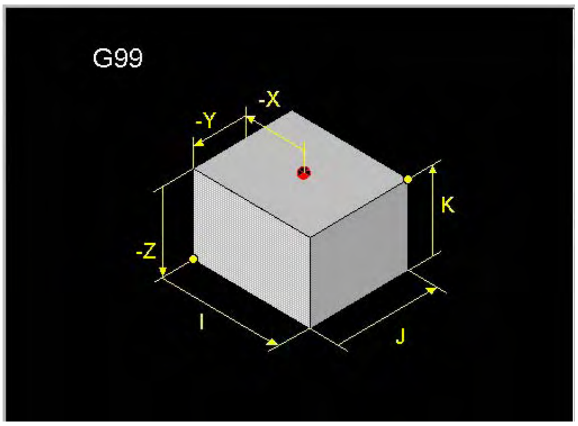
Example

N9000	
N1 G98 X-20 Y-20 Z-75 I140 J90 K95	Starting point and dimensions of the 3D graphic window
N2 G99 X0 Y0 Z0 I100 J50 K-55	Define blank as 3D area
:	

23.58 Graphic: material definition G99

Defines a three-dimensional blank and its position in relation to the program zero point W. The dimensions are needed for the graphical simulation.

N... G99 X... Y... Z... I... J... K...



G	Graphic: material definition
X	Start point coordinate
Y	Start point coordinate
Z	Start point coordinate
I	Dimension parallel to X
J	Dimension parallel to Y
K	Dimension parallel to Z

Example

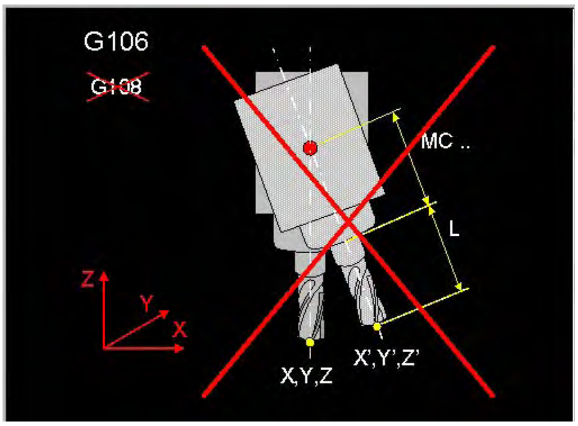
N9000	
N1 G98 X-20 Y-20 Z-75 I140 J90 K95	Starting point and dimensions of the 3D graphic window
N2 G99 X0 Y0 Z0 I100 J50 K-55	Define blank as 3D area
:	

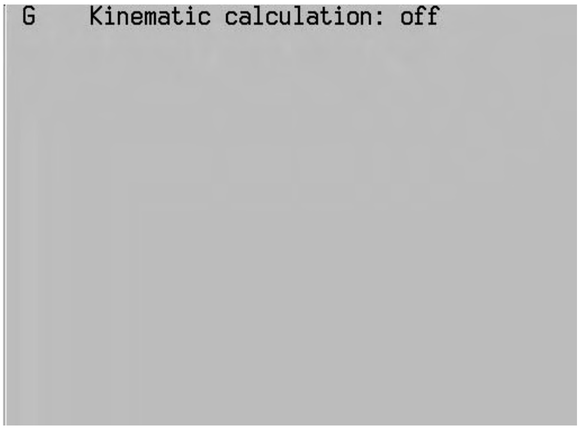
23.59 G106 Kinematic Calculation: OFF

Switches off G108 (Calculate kinematics: ON).

Format

G106





Notes and application

Modality

This function is modal with G108.

Execution

G106 waits with all actions until the movement in the preceding block is finished with <INPOD>. G106 deactivates calculation of the kinematics. The active offset in the linear axes is cancelled.

Note: G106 has the same effect as G108 I1=0 or MC756=0 (no calculation of kinematics).

Display

The G106/G108 functions remain in processing status in the modal G series. There is no separate symbol (as with G7/G8/G141) for the status with G108 active.

Example

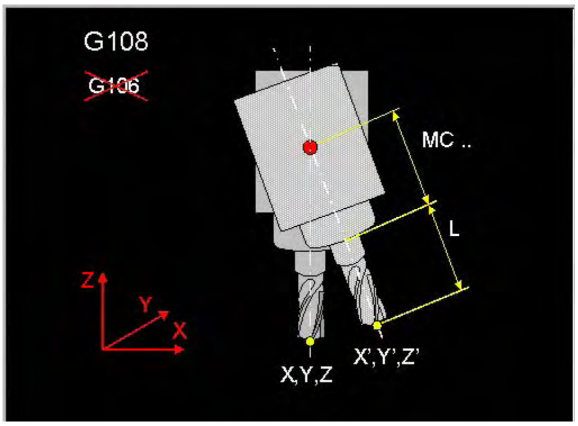
N10 G106 Switch off G108.

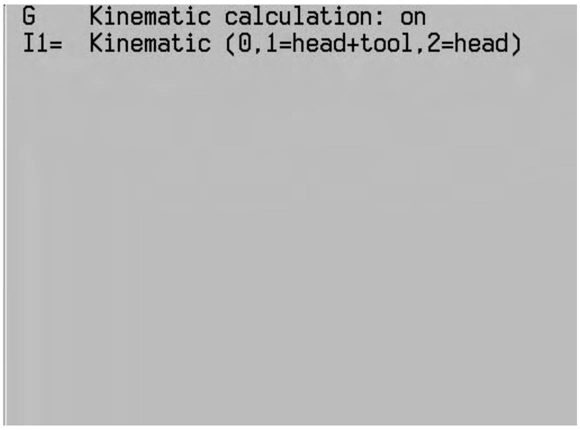
23.60 G108 Kinematic calculation: ON

Function whereby, with rotated circular axes, the position of the tool tip is calculated using the kinematic model. G108 activates calculation of the kinematics.
The status of the tool head is calculated at the end of a positioning movement into the position of the linear axes. The linear axes are not included.
The position display of MillPlus ***IT*** takes account of a change in the machine kinematics, such as would occur when a head is tilted. The offset caused is compensated for by an absolute programmed movement of the axes concerned.

Format

G108 {I1=..}





I1= 0 = same as G106
1= tool head and tool length is compensated
2= only tool head is compensated

Basic settings

Depending on MC756. This setting is active again after <Clear Control> and M30
If G108 is programmed without parameter, I1=1 is default

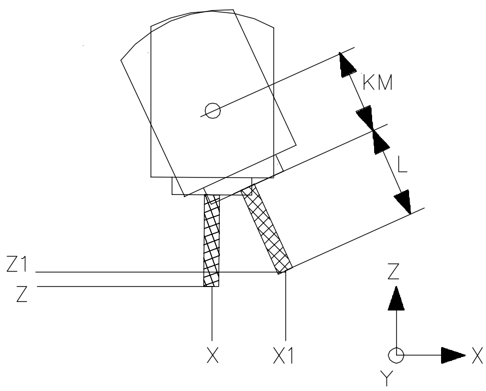
Notes and application

Modality

This function is modal with G106.

Execution

G108 waits with all actions until the movement in the preceding block is finished with <INPOD>.



KM = calculation with the kinematic model.

X, Z is the starting position. Tool length compensation is calculated in the Z direction.

X1, Z1 is the display position when G108. The head position is calculated in the rotated direction and if I1=1, tool length compensation is calculated in the Z direction (depending on G17).

Warning: **If G108 is active, the position of the tool tip in intermediate positions of this rotary axis is different from what it was previously (The PLC program has been adapted for this and the calculation is no longer compatible). This could make existing NC programs cause collisions.**

Warning: **If G108 is calculating the tool length (I2=1) the tool direction is no longer defined by G17/G18/G19 or G66/G67. This could make existing NC programs cause collisions.**

Switch off G108

G106 switches the G108 function off. G108 is reactivated in the MC basic setting (MC756) after <Program Cancel>, M30 <Clear Control> or switching on the CNC.

Machine zero point

It is assumed in the function G108 that the zero point is defined in the vertical position of the tool head. In the horizontal position (or in-between positions the position is corrected.

Rotary axis movement

When G108 is active the linear axis display is updated at the end of every positioning movement of the rotary axes defined in G108. <INPOD> then rapidly stops movement.

Interruption

When a rotary axis movement is interrupted the linear axis display is not updated. During an interruption the linear axis display is only updated to show the rotary axis status after <Emergency stop>, <Cancel program> or <Manual> has been pressed.

Manual

The G108 function remains active after M30 and is active during manual operation. The linear axis display is updated when rotary axis movement stops.

Kinematic model

The function is active for all machine tool types with rotary axes in the tool head.

Machine constants

MC 756 Calculate Kinematics (0 = no,1 = with tool length,2=without tool length)

Defines whether the function G108 is activated automatically after switching on the CNC and <Clear Control> and M30. With G108 is defined whether the rotary axes positions are processed in the display of the linear axes.

0 = G106 is active after switching on
G108 can be programmed, but after <Program Cancel> or M30 G106 is active again.

1 = G108 is active after switching on. The rotary axes in the tool head and the tool length are processed in the kinematic model.

2 = G108 is active after switching on. The rotary axes in the tool head are processed in the kinematic model.

Warning: **When MC756 is activated existing NC programs could cause collisions.**

Example Kinematic model permanently active.

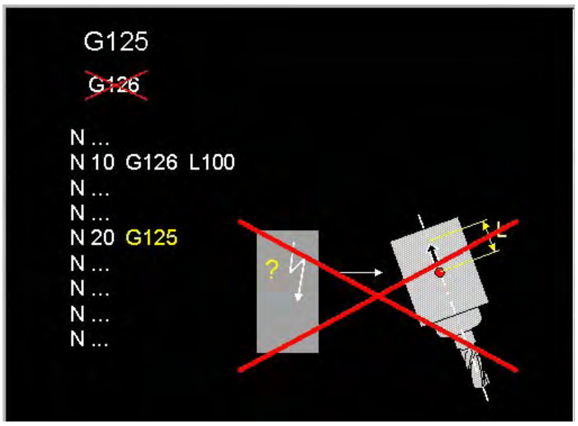
Program example	Description
N10 G108	Calculation of rotary axes in the tool head.

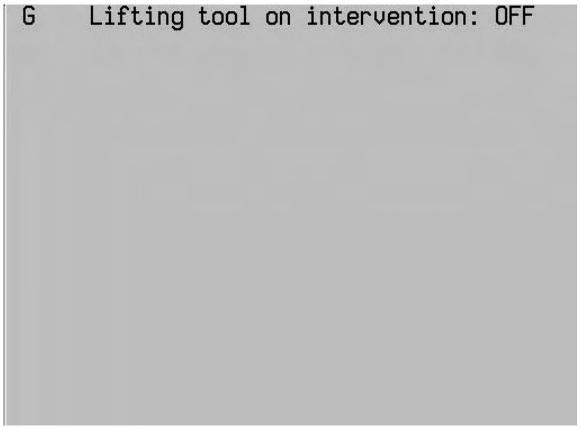
23.61 G125 Lifting tool on intervention: OFF

Deactivating the tool lifting movement.

Format

G125





Notes and application

Modality

This function is modal with G126

Execution

G125 resets the modal <Tool lifting enabled status> of the G126 function. After this no tool lifting movement can occur.

G125 is identical to G126 I1=0 I2=0 I3=0

G125 causes <INPOD>.

Display

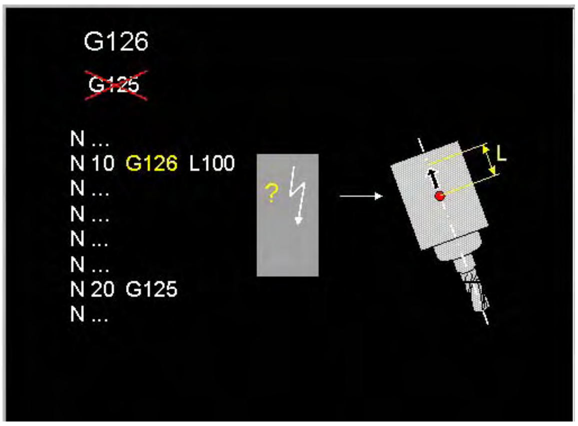
The function G125/G126 are listed in the modal G-group in the operating status.

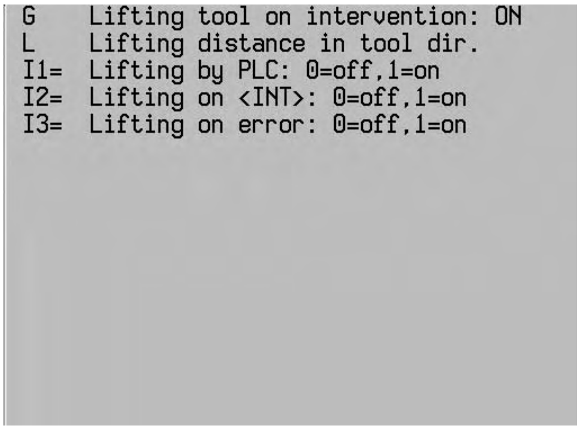
23.62 G126 Lifting tool on intervention: ON

G126 is a function to lift the tool from the work piece under certain conditions (coolant failure, intervention and errors).

Format

G126 {I1=..} {I2=..} {I3=..} {L..}





I1=	Tool lifting by PLC (Coolant failure): 0= no lifting, 1= lifting
I2=	Tool lifting at intervention <INT>: 0= no lifting, 1= lifting
I3=	Tool lifting at errors: 0= no lifting, 1= lifting
L=	Lifting distance in the tool direction
L	Defines the distance in the tool direction or tool orientation direction (G36 turning) over which is lifted. Default value in 'MC758 tool lifting distance'. Value between 0.001 and 99999.999 [mm] or 0.0001 and 9999.9999 [inch]

Basic settings

I1=1, I2=0, I3=0, L=MC758

Notes and application

Modality

This function is modal with G125.

Execution

G126 causes <INPOD>. After this a modal <tool lifting enabled status> is set.

The tool lifting movement is activated when:

- An event as described in I1 - I3 (coolant failure, intervention or error) occurs.
- The G126 Modal <tool lifting enabled status> is activated.
- A feed is active. In case the feed override is set to zero, no tool lifting takes place.
- During fixed cycles also when rapid is active.
- Certain G functions are activated.

Remark: Also when the tool lifting movement was not activated, the movement stops. When e.g. WOX_RETRACT_TOOL is set during rapid, the movement stops without a tool lifting movement.

The tool lifting movement occurs:

- in the programmed direction
- in the tool direction (G37 'milling', G126 L parameter or basic setting), or until the programmed tool lifting height or the SW end switch is reached.

After the tool lifting movement, the program execution and the spindle is stopped with an (additional) error message 'I264 Machining stopped with lifted tool'.

Remark: When the tool lifting movement is activated by an error (G126 I3=1) which also causes emergency stop, the servo's are already switched off before the tool lifting movement has ended.

Movement sequence

Before the tool lifting movement starts , the MillPlus decelerates until the correct (jerk free) angle velocity is reached.

During the following G functions, even when the G126 function is active, the tool lifting movement is not possible:

Movements	0, 6, 31, 33	Depending on the G28 setting for the feed movements
Planes	7, 182	
Measuring cycles	45, 46, 49, 50, 145, 148, 149, 150	
Positioning	74, 174	
Fixed cycles	84, 86	
New cycles	784, 786, 790, 794	
Graphics	98, 99, 195, 196, 197, 198, 199	
Pocket cycle	200, 201, 203, 204, 205, 206, 207, 208	

Switching off G126

At <M30>, <Program cancel>, G125 active and <Clear control> G126 '(Tool lifting on intervention: ON)' is deactivated.

Status display

The G125 / G126 status is shown in the modal G-group display.

Manual block search

During manual block search the functions G125 and G126 are maintained. The last one is executed before repositioning and output.

Interrupt of the tool lifting movement

The tool lifting movement itself can be interrupted. However, after interruption it is not completed. A new <Start> causes repositioning.

Repositioning

After the tool lifting movement the normal possiblities during intervention are available. Repositioning occurs with positioning logic.

Machine constants

MC 756 Tool lifting movement distance
Value between 1 and 99999999 [um].

With G320 the status of G126/G125 and the programmed distance can be requested:

I1=72 Programmed status
0 = G125
1 = PLC (G126 I1=1)
2 = INT (G126 (I2=1)
3 = PLC + INT (G126 I1=1 I2=1)
4 = ERR (G126 I3=1)
5 = PLC + ERR (G126 (I1=1 I3=1)
6 = INT + ERR (G126 I2=1 I3=1)
7 = all (G126 I1=1 I2=1 I3=1)
I1=73 Programmed distance

Example Activate tool lifting function.

Programming example	Description
N10 G126 I1=1 I2=1	Activating the tool lifting function by IPLC or intervention.

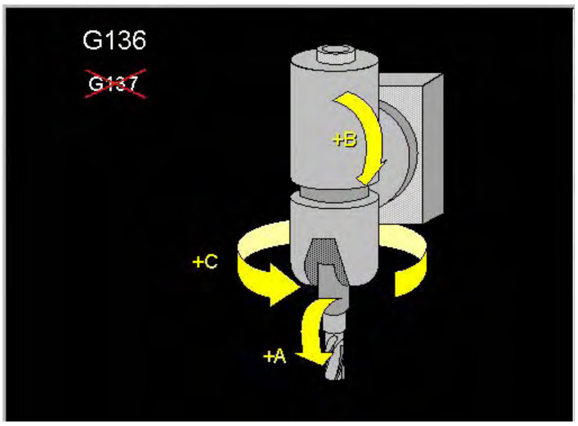
23.63 G136 Second axes configuration for fork head: ON

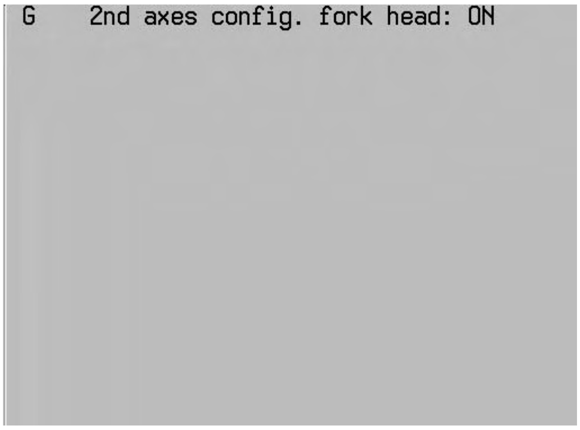
With G136 a -by the machine tool builder fixed implemented- function is activated **(e.g. a fork head moved into position)** Doing so a second axes configuration is activated. See the machine tool manual for the possibilities. In case your machine tool is not equipped with such kind of device the functions G136 and G137 have no meaning.

General description of the moveable fork head

The machine tool is delivered with a moveable fork head. In this case the machine tool has two configurations: 1 Normal head
2 fork head

With a continuous controlled fork head (B-axis, second C-axis and A-axis) it is possible to machine surfaces with five axes.





Moving the fork head into position must be started by an M-function (see machine tool manual).
By activating the moveable fork head by G136, the main axis C (rotary table) is exchanged with the fourth auxiliary axis. The fourth auxiliary axis controls the C-axis in the fork head.
The fork head, activated by G136 is de-activated by G137 and the C-axis is changed back from C-axis-head to C-axis-table.

Actions when using the fork head:

1 Output of an M-function (defined in MC_1063) to move the fork head into position. The kinematic model, defined by the machine tool builder is exchanged.
2 Output of the G-function (G136) to activate the fork head. The C-axis in the table is exchanged with the C-axis in the head.

Example: activating the fork head

In this example is assumed that M153 and M154 are used to move the fork head into position:

M153: Move the normal head into position (default)
M154: Move the fork head into position

Program example	Description
N9000 (smart fräsen)	
N10 G17	Select plane XY
N20 G7	Switch off G7
N30 M153	Move the normal head into position
N40 M55	Move the milling head (C-axis) into the vertical position
N50 G54 I33	Zero point with X, Y, Z, C-table and B
...	
N100 T203 M6	Change tool in the normal spindle
N110 G0 X1000 Y2000 Z1000 C0 B0	G137 C-table active (always after M153)
N120 S3000 M3	Start the normal spindle
N130 M7	Coolant 2

N140 G7 B5=-30 L1=1	B-axis to 30 degrees
N150 G1 Z990 F3000	
...	
N370 G7	Switch off G7
N380 G174	Tool retract movement
N390 T0 M6	Normal spindle is empty
N400 M154	Move fork head into position (G137 C-table is active). C-table 90. (Zero point in C-table is 180 => real position is C270)
N410 G54 I60 C180	Set zero point C-axis
N420 G0 X1000 Y2000 Z1000	
N430 C90 A0	Position C-table and A
N440 G136	Activate C-head (fork head)
N450 T405 M6	Tool change in fork head. Only possible in G136 (C-head)
N460 G54 I60 C0.002	Set zero point C-head
N470 G0 C0 A0	C-head rotates
N480 S30000 M3	Start fork head spindle
N490 M8	Coolant 1
N500 G141 F1=5000	Activate 3D tool correction
N510 G1 Z999 F10000	
N520 X999 Y1999 Z998 I1=0 J1=0.098 K1=988.987	
...	
N10000 G40	Switch off tool correction
N10010 G174	Tool retract movement
N10020 T0 M6	Fork head spindle is empty
N10030 G137	Activate C-table. In G54 I60 is C-table 180 reactivated Position of the C-table is 90 degrees again
N10040 M153	Move the fork head out of position
N10050 M30	

General description of the second axes configuration

Format

G136

Modality

G136 and G137 are mutual modal.

Switching of the axes

G136 and G137 activate the exchange of the axes configuration.
G137 switches off the axes configuration of G136 (fork head).

Kinematic model.

The (auxiliary) axes used by G136 must be present in the kinematic model.
The machine tool needs two kinematic models for the fork head (with and without fork head)

Movement of the programmed axes

Moving to the programmed "main axes positions" in the NC-program is now done by the exchanged auxiliary axis. This is also valid for the jog buttons of the axes.

Allowed G-functions when G136 is activated:

G136 may not be programmed when G7, G8, G36, G41-G44, G64, G141, G182, G19x or G20x is active
When G136 is active, all G-functions are allowed.

Switching off G136

The function G136 is switched off with G137. G136 is not switched off by <program interrupt>, M30 or <Clear control>.
After switching on the CNC, G137 is always active. When the fork head is in position it must be therefore moved out of position or be activated by G136.

Actions

G136 and G137 refrain from all actions until the movements of the previous block are ended with <INPOD>.

Display

When G136 is active the main axes, which are exchanged by auxiliary axes become a <2> behind the relevant axes characters in front of the actual position.
During G137 the axes characters are displayed normal (without <1>).

Zero points

When an axis is exchanged by G136, resp. G137, the relevant zero point values (G52, G54, G92, G93) of these axes are also exchanged. During this the values of the switched off axes (invisible) are saved. When these axes are changed back the zero point shifts are reactivated.
The saved zero point shifts are cleared in the following cases:

- Saved value for G52 is cleared when a new pallet zero point shift or another pallet function is activated.
- Saved value for G54 Inn is cleared when a new zero point shift G54 Inn is programmed.
- Saved values for G92/G93 are cleared after programming of new G92/G93 and after M30, <cancel program> or <Clear Control>.

Note: The saved G52/G54 zero point shift values for the switched off axes are saved in the stand-by memory and are retained also after switching off the CNC.

23.64 G137 Second axes configuration for fork head: OFF

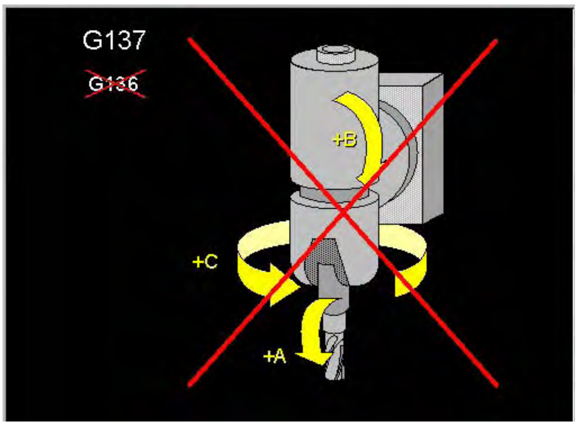
With G136 a -by the machine tool builder fixed implemented- function is deactivated **(e.g. a fork head moved into position)**. The machine tool is reset to the normal axes configuration. For the possible options see the machine tool manual.

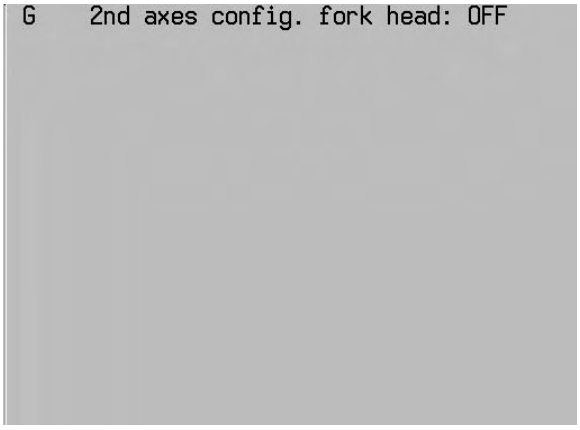
General description of the moveable fork head

The fork head activated by G136 is deactivated by G137 and the C-axis is switched back from C-head to C-table.

Format

G137





General notes and usage

Read the description of G136 first.

Modality

G136 and G137 are mutual modal.

Exchanging the axes

G137 Switches back the axes configuration set by G136.
G137 refrains from all actions until the movements in the previous block ended with <INPOD>.

Allowed G-functions when G137 is activated:

G137 may not be programmed when G7, G8, G36, G41-G44, G64, G141, G182, G19x or G20x is active.
When G137is active, all G-functions are allowed.

Switching off G137

The function G137 is switched off with G136. G137 is not switched off by <cancel program>, M30 or <Clear Control>.
After switching on the CNC, G137 is always active.

23.65 G141 3D-Tool correction with dynamic TCMP

Permits the correction of tool dimensions for a 3D tool path that is programmed in these points by its end point co-ordinates and normalised vectors perpendicular to the surface.

Format

To activate 3D-tool correction

G141 {R..} {R1 =..} {L2=}

To program straight-line movements

G141

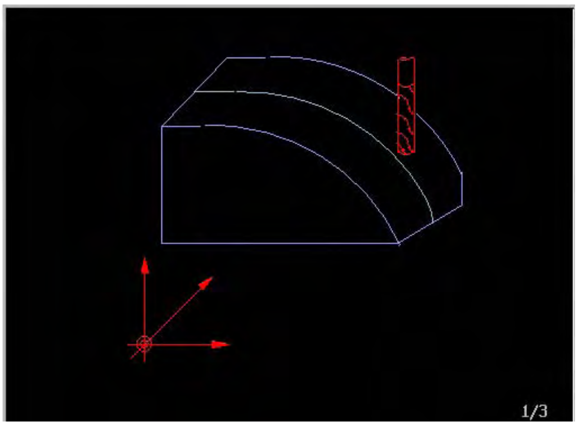
G0/G1 [end point coordinates] [I.. J.. K..]

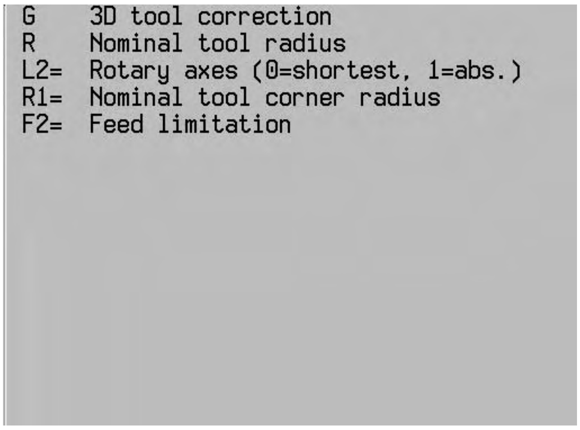
TCPM with active kinematical model

G0/G1 [end point coordinates] {I.. J.. K..} {I1=.. J1=.. K1=..} {A, B, C} {F..}

To delete 3D-tool correction

G40





With G141

R	Nominal tool radius
R1=	Nominal tool corner radius
L2=	Circular axes (0=shortest, 1=absolute)

With G0/G1

X, Y, Z	Linear end point coordinates
I, J, K	Axis components of surface normal vector.
I1=, J1=, K1= **(TCPM)**	Axis components of tool vector
A, B, C **(TCPM)**	Circular axis components of tool vector
F	Feed along the path

Associated functions

G40 and G412 to G44 for radius correction in a plane

For TCPM G8

General principles of G141

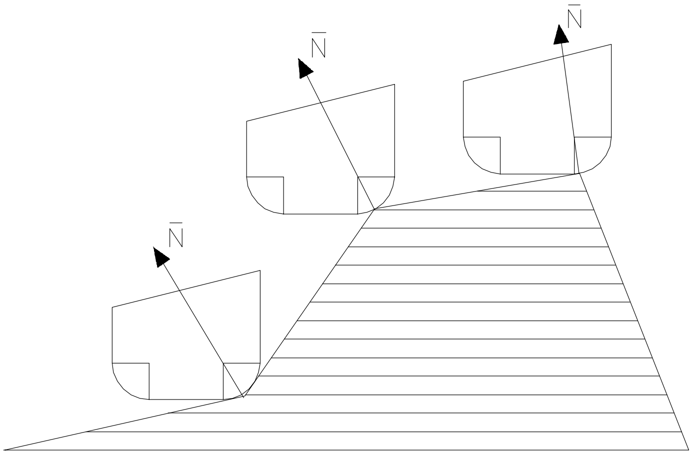
When milling a 3D surface, a given tool is moved along the surface in straight-line movements with a particular tolerance.

The calculation of the tool path on a 3D surface requires many calculations that are usually carried out by an NC programming system or a CAD system.

The calculated tool path depends on the shape of the tool, the dimensions of the tool and the tolerance to the surface.

When executing the appropriate program **without G141**, the milling tool used must have the same dimensions as in the calculations, i.e. a standard milling tool must be used.
If a new tool is required while machining a 3D surface, this tool must also have the same dimensions as the **standard tool**.
If dimensional deviations are detected on the workpiece, a new calculation must be made using the programming system.

The 3D tool correction **(G141)** allows the use of tools whose dimensions differ from the dimensions of the standard milling tool. The corrections are carried out with the help of the direction vectors that are created by the programming system together with the end point co-ordinates.
In addition, the workpiece dimensions can be calculated by the programming system and the tool path by the CNC from the normalised vectors and the tool dimensions.



$\overline{N}$ = Surface normal vector (I, J, K)

Notes and application

Radius (R, R1=)

The R.. and R1=.. values should be the same as the nominal tool dimensions used by the **programming system** for calculating the toolpath. These values are set equal to zero, if not programmed.

R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.
R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

General principles of TCPM

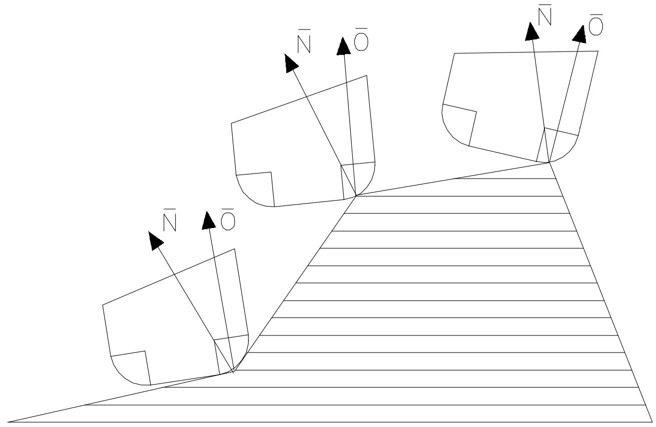
Maintaining position of tool tip when positioning swinging axes (TCPM)

(TCPM stands for "Tool Centre Point Management").

With **G141 '3D tool correction without TCPM'**, a curved (CAD) surface can be travelled taking the current tool dimensions into account. In this case, the path is described by end point co-ordinates and vectors perpendicular to the surface. The G141 function only guides the three linear axes but not the circular axes. In this way, the tool is always used in the same direction and is not guided over the workpiece surface at the optimum angle.

With **G8 'Tool orientation'** (static TCPM), the tool can be placed on the surface of the workpiece at an optimum angle. The G8 function is a feed movement and cannot be used continuously on a curved surface during a path movement.

In the case of **G141 with dynamic TCPM**, the tool is guided on a curved workpiece surface at an optimum angle. The current workpiece dimensions are taken into account. Dynamic TCPM is used for 5-axis milling. Dynamic TCPM also controls the circular axes. The tool is guided on the curved workpiece surface either vertically or at a programmed orientation.



$\bar{N}$ = Surface normal vector (I, J, K)

$\bar{O}$ = Tool vector (I1=, J1=, K1=) or rotary axes coordinates of the tool vector (A, B, C)

The programming format of the linear blocks within G141 is expanded to include the option of programming a tool vector. Possible combinations are surface normal vectors and/or tool vectors. If only the tool vector is used, the tool correction must be calculated in the CAD system.

G7 may be active. In this case, the surface normal vectors and the tool vectors are defined in the G7 level.

Notes and application

Addresses (R, R1=, L2=, F2=) **(TCPM)**

R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

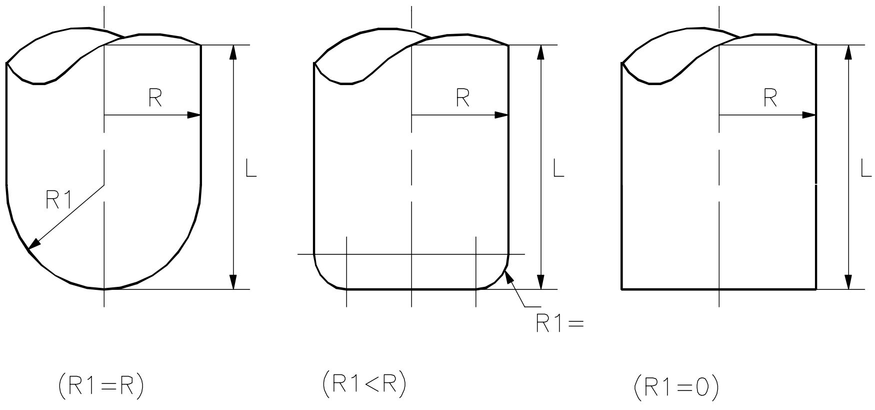
R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

L2= 0 Circular axes travel the shortest distance (basic setting)

1 Circular axes travel to their absolute position (with circular axis programming).

F2= Feed limitation on highly curved surfaces. When radiusing an outside edge the machine may suddenly move at maximum feed. F2= limits this maximum feed. Feed override is active. F2= can only be programmed in the G141 block but it is also effective within G141 movements until the block with G40.

Possible tools



Tools used for the G141 function

Tool memory

The following dimensional details must be loaded into the tool memory to enable different types of tools to be used:

Radius milling tool	:	R (tool radius), L (tool length), C (=tool radius)
Radius end milling tool	:	R (tool radius), L (tool length), C (=rounding radius)
End milling tool	:	R (tool radius), L (tool length), C0

If no value of C is entered, C automatically becomes 0.
The standard milling tool is thus an end milling tool.

Note: The rounding radius in the G141 block is programmed with the word R1=. The rounding radius is stored in the tool memory with the C word.

Created tool path

When the programming system creates the tool path (surface normal vector is programmed), the dimensions of the nominal tool (R.. and R1=) are programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to correct the tool path.

Workpiece dimensions

When the programming system creates the workpiece dimensions (surface normal vector and tool vector are programmed), the R.. and R1= words are not programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to calculate the tool path.

Activating G141

In the first block after G141, the milling tool travels from the current tool position to the corrected position in this block.

End point coordinates

Only absolute or incremental (X, X90, X91) Cartesian dimensional data can be used.
Up to V420, the co-ordinates in the first G141 block must be absolute and are measured from the programming zero point W.

G90/G91

The functions G90 and G91 are used for programming absolute (G90) or incremental (G91) dimensions. These functions must be alone in their own block.

Mirroring

If the mirroring function (G73 and axis co-ordinates) is active before G141 is activated, the mirrored co-ordinates are used during the 3D tool correction.
Mirroring is possible as before once G141 is activated. Mirroring is cancelled by the G73 function.

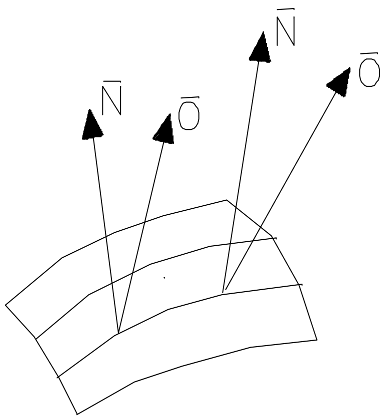
Radius correction G41...G44

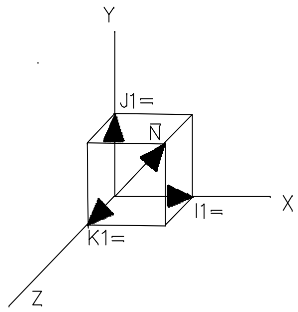
After activating a G141 block, the effective radius correction programmed with G41...G44 is deleted.

Surface normal vector (I, J, K) **(TCPM)**

Defines the surface normal vector perpendicular to the surface.

The surface normal vector is perpendicular to the workpiece surface. The tool is positioned so that this vector always passes through the centre point of the tool corner rounding. This vector controls the positioning of the linear axes within G141. .





Vector components

The vector components of the axes are independent of the level selected.
If vector components are not programmed in a block, the components not programmed are set at zero.

Dimension factor

The input format of the vectors (I, J, K, I1=, J1=, K1= words) is limited to three decimal places. The surface normal and tool vectors do not, however, have to have the length 1. To increase the dimensional accuracy, the values in question can be multiplied by a dimension factor between 1 and 1000. With the factor 1000, for example, the input accuracy of the vector components is increased to six significant figures.

Back cutting

Back cutting or collisions between tool and material at points not to be machined are not detected by the CNC.

Kinematic model **(TCPM)**

The kinematic model is used for calculations within G141.
If no kinematic model is active (MC312 ‘Free machining level’ = 0), G141 remains compatible with the G141 functions in older CNC versions.

Tool vector **(TCPM)**

I1=, J1=, K1=	axis components of tool vector
or	
A, B, C	circular axis components of tool vector

The tool vector or the circular axis co-ordinates indicate the direction of the tool axis. The tool is turned so that it is parallel to this vector. This vector controls the positioning of the circular axes (and the associated compensation movement with linear axes) within G141.

Deleting

Function G141 is deleted by G40, M30, the program interrupt softkey or the CNC reset softkey. The milling tool stops at the last corrected position. The circular axes are not turned back automatically.

Functions to be deleted

When working with G141, functions G64, scale change (G73 A4=..), axis rotation (G92/G93 B4=..) and G182 must be deleted.

The following G functions are permitted if G141 **(TCPM)** is switched on:

Basic motions	0, 1, 7
Levels	17, 18
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	39, 40, 141
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	70, 71, 90, 91
Graphics	195, 196, 197, 198, 199

If a G function that is not permissible is programmed, error message P77 ‘G function and Gxxx not permitted’ is issued.

The following G functions are permitted if G141 **(TCPM)** is active:

Basic motions	0, 1 Parameters of G0 and G1 are limited G0 without positioning logic
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	40, 141 G40 switches G141 off
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	90, 91

If a G function that is not permissible is programmed, error message P77 ‘G function and G141 not permitted’ is issued.

Programming limitations

G functions that are not listed above may not be used.
Point definitions (P) and E parameters may not be used.
No tool change may be made after activating G141.

Notes and application for TCPM

Risk of collision

When G141 is switched on, compensation movements similar to those in G8 may occur.

In the case of the switch-on movement, the tool tip must not be resting on the surface of the workpiece and should be programmed with a distance from the material at least equal to the tool diameter.

Remark: If G141 is switched off via G40, M30 or program cancel, there is no compensation movement and the circular axes remain in their last positions.

When approaching the contour, it may happen that the table with the workpiece is turned through 180 degrees to achieve the programmed tool direction. **ATTENTION! RISK OF COLLISION!**

Undercutting

If the tool direction changes within a G1 block, this tool direction change is carried out interpolating with the movement to the end point. In doing this, the path between the start and end points is corrected for undercutting.

Undercutting is not detected during block transitions. This undercutting should be corrected by inserting a block without an end point and with only one change of the tool vector by the CAD system. In this case, the tool turns about the tool contact point until the new tool direction is reached.

Display

When G141 is active, a yellow icon is displayed behind the tool number and the programmed G141 tool vectors (I1, J1, K1) can be seen in the machining status (on the G7/G8 positions).

Remark: If G7 and G141 are active at the same time, the G7 angle or vector can be seen.

A small ‘p’ at the bottom right, near the ‘axis letters’, shows whether the position of the tool contact point or the position is in machine co-ordinates. The display changes with the same softkey as with G7.

Feedrate

The programmed feedrate applies to the contact point between the surface and the tool. The tool head may make other movements.

Error messages

P341 Tool vector incorrect

The tool vector (I1=, J1=, K1=) is incorrect. This error message is generated if all the components of the vector are zero.

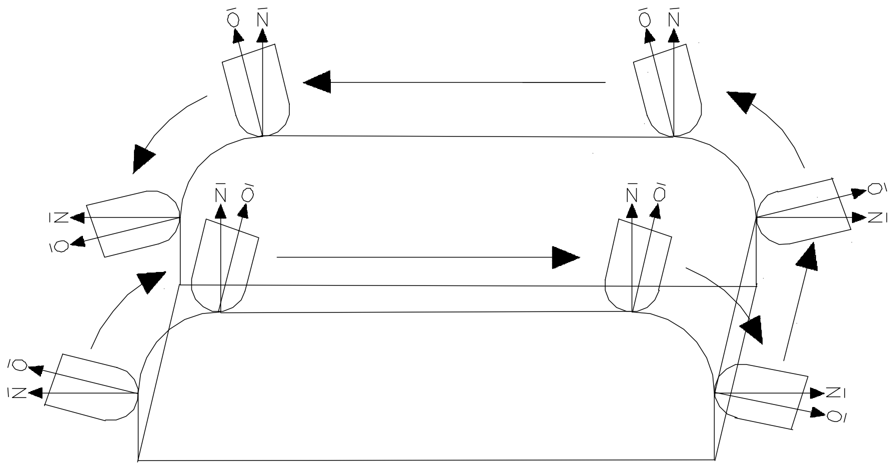
P342 Surface normal vector incorrect

The surface normal vector (I, J, K) is incorrect. This error message is generated if all the components of the vector are zero.

Example

Example 1 G141 and TCPM

Tool vector with (I1=, J1=, K1=)
This program is independent of the machine.



N113 (square material with top rounding (R4) and swung tool (5 degrees)
N1 G17
N2 T6 M67 (10 round spherical milling tool: T6 R5 C5 in tool table)
N3 G54 I10
N4 G0 X0 Y0 Z0 B0 C0 S6000 M3
N5 F50 E1=0
N6 G141 R0 R1=0 L2=0 (all basic settings, do not need to be programmed)
N7 (R in CAD System is 0 mm)
N8 (R1 in CAD System is 0 mm)
N9 (L2=0 circular axes move shortest distance)
N10
N11 G0 X-1 Y=E1 Z0 I1=-1 K1=0
N12 (generated in CAD System)
N13 (front left arc)
N14 G1 X=0 Y=E1 Z=-4 I1=-0.996194698 K1=0.087155743
N15 G1 X=0.000609219 Z=-3.930190374 I1=-0.994521895 K1=0.104528463
N16 G1 X=0.002436692 Z=-3.860402013 I1=-0.992546152 K1=0.121869343
N17 G1 X=0.005481861 Z=-3.790656175 I1=-0.990268069 K1=0.139173101

N… (Each degree a point)

N100 G1 X=3.790656175 Z=-0.005481861 I1=0.034899497 K1=0.999390827
N101 G1 X=3.860402013 Z=-0.002436692 I1=0.052335956 K1=0.998629535
N102 G1 X=3.930190374 Z=-0.000609219 I1=0.069756474 K1=0.99756405
N103 G1 X=4 Z=0 I1=0.087155743 K1=0.996194698
N104 (front right arc)
N105 G1 X=36 Z=0 I1=0.087155743 K1=0.996194698
N106 G1 X=36.06980963 Z=-0.000609219 I1=0.104528463 K1=0.994521895
N107 G1 X=36.13959799 Z=-0.002436692 I1=0.121869343 K1=0.992546152

N…

N194 G1 X=39.99756331 Z=-3.860402013 I1=0.998629535 K1=-0.052335956
N195 G1 X=39.99939078 Z=-3.930190374 I1=0.99756405 K1=-0.069756474
N196 G1 X=40 Z=-4 I1=0.996194698 K1=-0.087155743

```
N197 G40
N1971 (back right arc)
N1972 (move up to next cut)
N1973 G174 I100 (tool withdrawal)
N1974 G0 B0 C0 (rotate circular tables to original coordinates system)
N198 E1=E1+0.25
N1981 G1 Y=E1 (movement in normal X, Y, Z coordinates system)
N1982 G141
```

OR without deactivation G141

```
N197 (back right arc)
N198 E1=E1+0.25 (move up to next cut)
```

```
N199 G1 X=40 Y=E1 Z=-4 I1=0.996194698 K1=0.087155743
N200 G1 X=39.99939078 Z=-3.930190374 I1=0.994521895 K1=0.104528463
N201 G1 X=39.99756331 Z=-3.860402013 I1=0.992546152 K1=0.121869343

N…

N287 G1 X=36.13959799 Z=-0.002436692 I1=-0.052335956 K1=0.998629535
N288 G1 X=36.06980963 Z=-0.000609219 I1=-0.069756474 K1=0.99756405
N289 G1 X=36 Z=0 I1=-0.087155743 K1=0.996194698
N290 (back left arc)
N291 G1 X=4 Z=0 I1=-0.087155743 K1=0.996194698
N292 G1 X=3.930190374 Z=-0.000609219 I1=-0.104528463 K1=0.994521895
N293 G1 X=3.860402013 Z=-0.002436692 I1=-0.121869343 K1=0.992546152

N…

N379 G1 X=0.002436692 Z=-3.860402013 I1=-0.998629535 K1=-0.052335956
N380 G1 X=0.000609219 Z=-3.930190374 I1=-0.99756405 K1=-0.069756474
N381 G1 X=0 Z=-4 I1=-0.996194698 K1=-0.087155743
N382 E1=E1+0.25

N383 G14 N1=10 N2=389 J40

N384 G40
N385 G174 I100 (tool withdrawal movement)
N386 G0 B0 C0 (rotate circular tables to original coordinates system)
N387 M30
```

Example 2 G141 and TCPM

Identical workpiece
Tool vector with (A, B, C)
This program is machine dependent.

This program is for a machine with on the table a B-Axes under 45°, with upon a C-axes.

```
N114 (Rectangle block with rounding on top (R4) and tilting tool position (5 degrees))
N1 G17
N2 T6 M67 (Ball cutter round 10: In tool table T6 R5 C5)
N3 G54 I10
N4 G0 X0 Y0 Z0 B0 C0 S6000 M3
N5 F50 E1=0
```

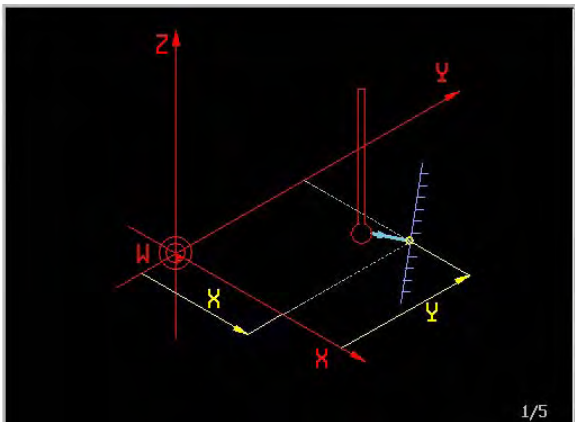
```
N6 G141 R1=0 L2=0 (all default, so not necessary to program)
N7 (R in CAD System is 0 mm)
N8 (R1 in CAD System is 0 mm)
N9 (L2=0 Rotary axes moves shortest way)
N10
N11 G0 X-1 Y=E1 Z0 B180 C-90
N12 (generated in CAD System)
N13 (front arc left)
N14 G1 X=0 Y=E1 Z=-4 B145.658 C-113.605
N15 G1 X=0.000609219 Z=-3.930190374 B142.274 C-115.789
N16 G1 X=0.002436692 Z=-3.860402013 B139.136 C-117.782
N17 G1 X=0.005481861 Z=-3.790656175 B136.191 C-119.624

N... (Each degree a point)

N100 G1 X=3.790656175 Z=-0.005481861 B2.829 C1
N101 G1 X=3.860402013 Z=-0.002436692 B4.243 C1.501
N102 G1 X=3.930190374 Z=-0.000609219 B5.658 C2.001
N103 G1 X=4 Z=0 B7.073 C2.502
N104 (front arc right)
N105 G1 X=36 Z=0 B7.073 C2.502
N106 G1 X=36.06980963 Z=-0.000609219 B8.489 C3.004
N107 G1 X=36.13959799 Z=-0.002436692 B9.906 C3.507

N...

N194 G1 X=39.99756331 Z=-3.860402013 B206.449 C108.384
N195 G1 X=39.99939078 Z=-3.930190374 B210.629 C111.170
N196 G1 X=40 Z=-4 B214.342 C113.605
N197 (back arc right)
N198 E1=E1+0.25    (now translation)
N199 G1 X=40 Y=E1 Z=-4 B145.658 C66.395
N200 G1 X=39.99939078 Z=-3.930190374 B142.274 C64.211
N201 G1 X=39.99756331 Z=-3.860402013 B139.136 C62.218

N...

N287 G1 X=36.13959799 Z=-0.002436692 B4.243 C-178.499
N288 G1 X=36.06980963 Z=-0.000609219 B5.658 C-177.999
N289 G1 X=36 Z=0 B7.073 C-177.498
N290 (back arc left)
N291 G1 X=4 Z=0 B7.073 C-177.498
N292 G1 X=3.930190374 Z=-0.000609219 B8.489 C-176.996
N293 G1 X=3.860402013 Z=-0.002436692 B9.906 C-176.493

N...

N379 G1 X=0.002436692 Z=-3.860402013 B206.449 C-71.616
N380 G1 X=0.000609219 Z=-3.930190374 B210.629 C-68.830
N381 G1 X=0 Z=-4 B214.342 C-66.395
N382 E1=E1+0.25

N383 G14 N1=14 N2=382 J40

N384 G40
N385 G174 L100 (Retract tool)
N386 G0 B0 C0 (turn rotary tables to original coordinates system)
N387 M30
```

23.66 Linear measuring movement G145

Executes a freely programmable linear measurement movement to determine axis positions.

N... G145 [Meesuring point coordinates] [(axis address) 7=..] {S7=..} E.. {F2=..} {K..} {I3=..}

Parameter

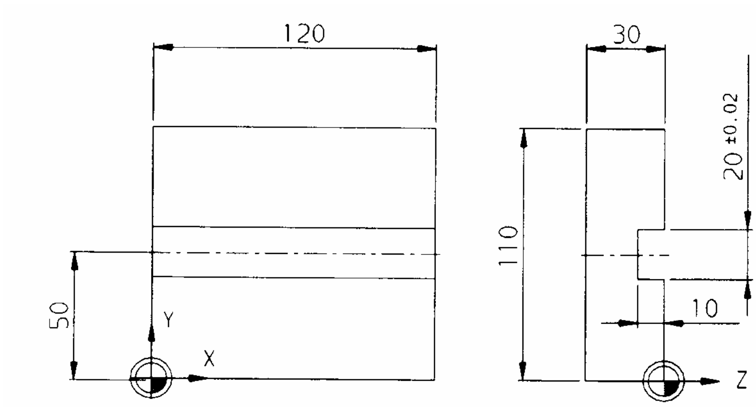


G	Linear measuring movement
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
B	Endpoint angle
C	Endpoint angle
K	0=tool correction on, 1=off
E	E-parameter for measuring status
B1=	Angle
B2=	Polar angle
X7=	E-par. for measured value in X
Y7=	E-par. for measured value in Y
Z7=	E-par. for measured value in Z
B7=	E-par. for measured value in B
C7=	E-par. for measured value in C

?90=	Endpoint abs. (X,Y,Z..)
?91=	Endpoint incr. (X,Y,Z..)
I3=	Status control (0=on, 1=off)
I4=	Air supply (0=off, 1=on)
L1=	Path length
L2=	Polar length
P1=	Point definition number
F2=	Measuring feed
S7=	E-par. for measured value in S

Example

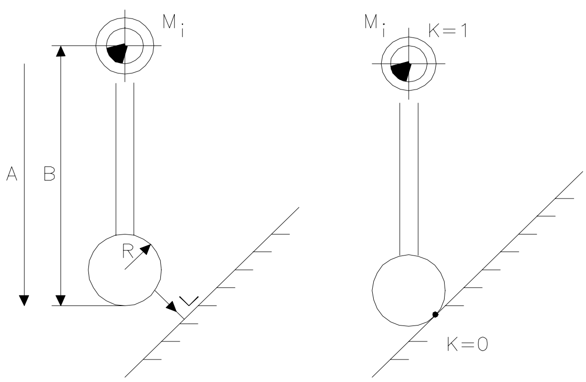
A slot is to be milled and its width measured. If the slot width is too small, the milling radius must be corrected and the slot re-worked.



N14504 (Milling and measuring a slot)
N1 G17
N2 G54
N3 E15=20.02 (Maximal slot width)
N4 E16=19.98 (Minimum slot width)
N5 E3=(E15+16):2

```
N6 S1000 T1 M6  (Milling tool d=18 mm)
N7 G0 X-25 Y50 Z-10 B0 F400 M3
N8 G1 X140
N9 G43
N10 G1 Y60
N11 G41
N12 X-25
N13 Y40
N14 X140
N15 G40
N16 Y50
N17 G0 Z50 M5
N18 G149 T0 E30
N19 T30 M6  (Measuring probe)
N20 M19                    (D address optional)
N21 M27
N22 G0 X60 Y50 Z-8 B0
N23 M29
N24 G145 Y65 E10 Y7=1 F2=500
N25 GO Y50
N26 G29 E11=E10=0 E11 N=30
N27 M29
N28 G145 Y35 E10 Y7=2 F2=500
N29 G0 Y50
N30 M28
N31 G29 E11=E10=0 E11 N=41
N32 E5=E1-E2
N33 E6=(E5-E3):2
N34 G29 E20=E5>E15 E20 N=44
N35 G29 E20=E5>E16 E20 N=46
N36 G149 T=E30 R1=4
N37 G150 T=E30 R1=E4+E6
N38 S1000 T1 M6  (Milling tool d=18 mm)
N39 G0 X140 Y50 Z-10 B0 F400 M3
N40 G29 E20 E20=1 N=9
N41 M0
N42 (Measuring probe has not made contact, no measurement carried out)
N43 G29 E20 E20=1 N=46
N44 M0
N45 (Slot width too large)
N46 M30
```

Notes



Tool correction:

K0: Tool correction on.
Measuring positions are corrected to take account of tool length and tool radius. Measuring positions in rotational axes do not take tool data into account.

K1: Tool correction off. Measuring positions will not be corrected.

The following assumptions are made if the measuring positions are corrected to take account of the size of the measuring probe:

- the measuring probe lies parallel to the tool axis
- the measuring probe is completely spherical
- the measuring probe moves vertically in relation to the surface being examined.

E parameter:

The number of the E parameter in which the measured axis position is stored (e.g. X7=2 indicates that the measured value in the X axis will be stored in parameter E2. X7=E1 (E1=5) signifies that the measured value will be stored in E5.

Measuring probe status:

E...=0: the programmed end position has been reached, but no measuring point has been detected. The associated E parameters containing the measured values remain unchanged.

E...=1: a measuring point was detected during the measuring operation. The measuring position has been saved in the E parameters.

Status monitoring (I3= 0=on, 1=off) (Status of the turning aside of the probe)

Monitoring of the measuring key status within G145 can be disabled for certain equipments (laser). The standard value is zero.

The functions G145 to G150 must not be used when G182 is being used.

In all the specified operating modes, a value of 2 is allocated to the E parameter for the status of the measuring probe. The use of parameters without measuring data can be prevented by testing the value of this parameter in the measuring macros.

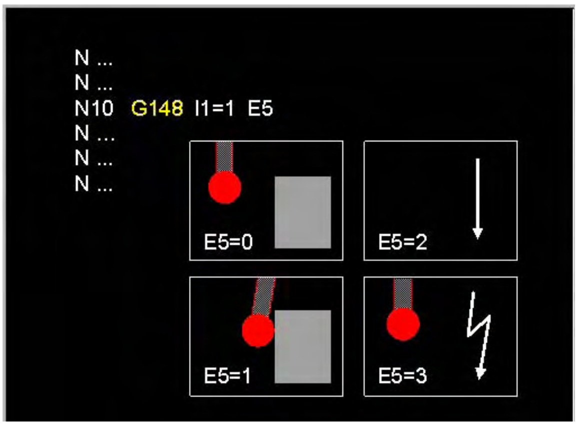
Air supply (I4=) (0=no 1=yes)

The air supply time before the measuring is stored in Machine constant (MC842). (Default is 0)

23.67 Reading measuring probe status G148

N... G148 {I1=...} E...

Parameters




```
G    Read measure probe status
E    E-parameter for probe status
I1=  Status group (1-3)
```

Example

:
N110 G148 E27
N115 G29 E91=E27=2 E91 N=300
:
N300 M0 (Present mode: block search, test run, demo)
:
N400 M30

Note

Measuring probe status:
I1=1 or is not programmed (standard value)
E...=0: Probe not deflected.
E...=1: Probe deflected.
E...=2: The G145 block was executed during a block search, test run or demo.
E...=3: Measuring probe error; no measurement possible.

The priority for the measuring key status codes is as follows:
1 : Code 2 (active mode)
2 : Code 3 (measuring key error)
3 : Code 0 or 1 (measuring key contact)

I1=2
E...= 0: no measuring point has been determined during the measurement
E...= 1: a measuring point has been determined during the measurement

I1=3
E...= 0: information from the IPLC (information program logic control): key/laser not enabled
E...= 1: information from the IPLC (information program logic control): key/laser enabled

See the probe system documentation.

During operation with G182 the use of the functions G148 to G150 is not permitted.

23.68 Reading tool or offset values G149

Interrogate current tool :
    N.. G149 T0 E..

Interrogate tool dimensions:
    N.. G149 T.. {T2=..} {L1=..} {R1=..} {M1=}

Interrogate tool status:
    N.. G149 T.. E..

Interrogate current zero point offsets:
    N.. G149 N1=0/1 E..

Interrogate pallet offset values:
    N.. G149 N1=0/1 E..

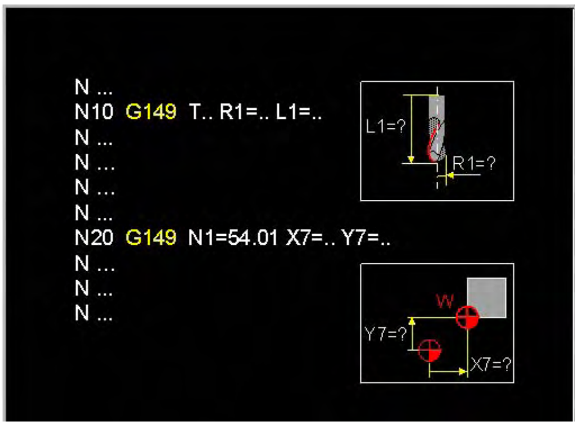
Interrogate saved zero point offsets:
With standard zero points or MC84=0:
    N.. G149 **N1=51..59** [(axis address)7=..] {(axis address)7=..}

With extended zero points and MC84>0:
    N.. G149 **N1=54.[NR]** [(axis address)7=..] {(axis address)7=..} **{B47=...}**
    N.. G149 N1=51..59 [(axis address)7=..] {(axis address)7=..}

Interrogate programmable zero point offsets:
    N... G149 N1=92 {93} [(axis address)7=...] {(axis address)7=...}

Interrogate current position values of the axes.
    N... G149 [(Axis address)7=...]{(Axis address)7=...}

Parameters



G	Read tool or zero offset values
T	Tool number
E	E-parameter
N1=	Zero offset shift
B47=	E-parameter for rotation in B4=
X7=	E-par. for offset/position in X
Y7=	E-par. for offset/position in Y
Z7=	E-par. for offset/position in Z
B7=	E-par. for offset/position in B
C7=	E-par. for offset/position in C
L1=	E-parameter for toollength
R1=	E-parameter for tool radius
T2=	Tool offset index
M1=	E-parameter for toollife

Notes

The tool status can be loaded from the tool memory into the stated E-parameter.
The tool status can have the following values:

E... = 1	Tool has been released and measured
E... = 0	Tool has been released, but has not yet been measured
E... = -1	Tool is blocked
E... = -2	Tool life has been reached
E... = -4	Tool fracture error
E... = -8	Tool cutting force reached
E... = -16	Tool life programmed shorter than T3

A combination of error messages is possible as well:

E... = -13 means: error message -8 and -4 and -2 and 1.

Examples

1: Interrogate number of current tool.
N100 G149 T0 E1
E1 contains the number of the current tool

2: Interrogate dimensions of the current tool.
N100 G149 T12 L1=5 R1=6
E5 contains the tool length
E6 contains the tool radius

3: Interrogate the active zero point offset function
N100 G149 N1=0 E2
N110 G149 N1=1 E3
E2 contains the current zero point offset (51 or 52)
E3 contains the current saved zero point offset (53...59) **or G54.[nr]**

4: Interrogate the offset G54
N100 G149 **N1=54** X7=1 Z7=2
or
N100 G149 **N1=54.[nr]** X7=1 Z7=2

E1 contains the offset in X
E2 contains the offset in Z

5: Calling G54 offset with rotary angle **(MC84>0)**
N100 G149 **N1=54.[nr]** X7=1 B47=2

E1 contains offset in X
E2 contains rotary angle of coordinate system

6: Call up the remaining tool life M1=:
N100 G149 T1 M1=3 (Store remaining tool life of T1 in parameter E3)
E3 enthält die aktive gespeicherte Nullpunktverschiebung (53...59) oder **G54.[nr]**

Notes

The tool correction index 0, 1 or 2 can be specified. The default value is T2=0.

From V400:
T2=0: Tool radius = radius (R) + dimension (R4=).
Tool length = length (L) + dimension (L4=).
It is better to use G321.

23.69 Change tool or offset values G150

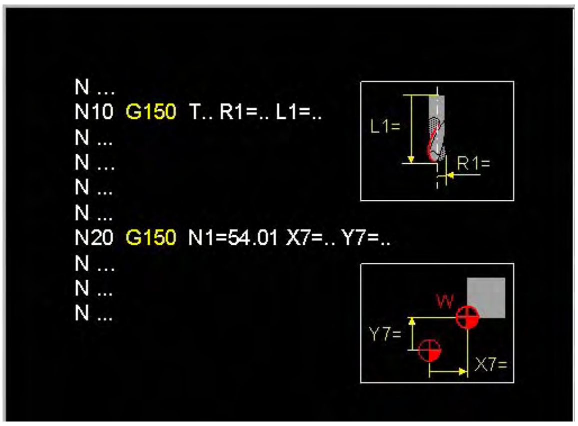
Modify tool data in tool memory:
N.. G150 T.. {T2=..} L1=.. R1=.. M1=..

Modify tool status in tool memory:
N.. G150 T.. E..

Modify zero point offset data in tool memory:
With standard zero points or MC84=0:
N.. G150 **N1=51..59** [(axis address)7=..] {(axis address)7=..}

With extended zero points and MC84>0:
N.. G150 **N1=54.[NR]** [(axis address)7=..] {(axis address)7=..} **{B47=...}**

Parameters



G	Change tool or offset values
T	Tool number
E	E-parameter
N1=	Zero offset shift
B47=	Angle of rotation in B4=
X7=	Offset in X
Y7=	Offset in Y
Z7=	Offset in Z
B7=	Offset in B
C7=	Offset in C
L1=	Toollength value in T
R1=	Tool radius value in T
T2=	Tool offset index
M1=	Tool life value in T

Notes

The tool status can be loaded from the tool memory into the stated E-parameter.
The tool status can have the following values:

E... = 1	Tool has been released and measured
E... = 0	Tool has been released, but has not yet been measured
E... = -1	Tool is blocked
E... = -2	Tool life has been reached
E... = -4	Tool fracture error
E... = -8	Tool cutting force reached
E... = -16	Tool life programmed shorter than T3

A combination of error messages is possible as well:
E... = -13 means: error message -8 and -4 and -2 and 1.

Examples

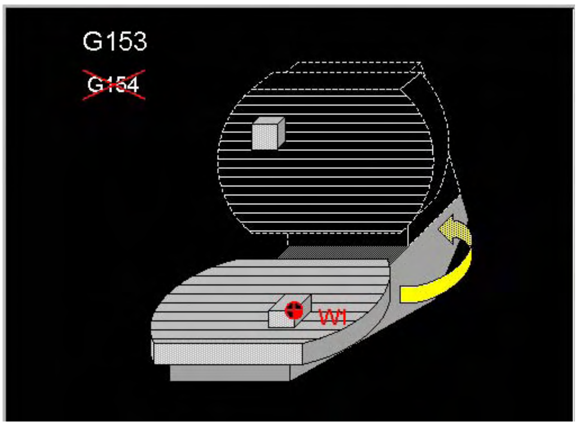
1. Modify tool data in tool memory:
   N50 G150 T1 L1=E2 R1=4
2. Modify zero point offset data in tool memory:
   N70 G150 **N1=57** X7=E1 Z7=E6
   or
   N70 G150 **N1=54.[nr]** X7=E1 Z7=E6
3. Changing a zero offset with rotary angle of the coordinate system:
   N70 G150 **N1=54.[nr]** X7=E1 B47=E2
4. Change the remaining tool life M1=:
   N110 G150 T1 M1=10 (Change the new remaining tool life of T1 to 10 minutes)

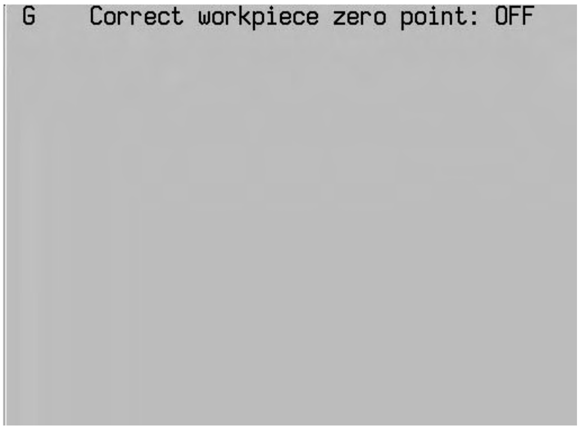
23.70 G153 Correct workpiece zero point: OFF

G153 deactivates the zero point displacement. The active offset in the linear axes is cancelled.

Format

G153





Notes and usage

Modality

This function is mutual modal with G154.

Execution

G153 resets the modal status of the G154 function. The work piece zero point displacement is switched off.

G153 refrains from all actions until the movement in the previous block has ended (<INPOD>).

Display

The functions G153/G154 are displayed in the modal G row in the machining status display.

23.71 G154 Correct workpiece zero point: ON

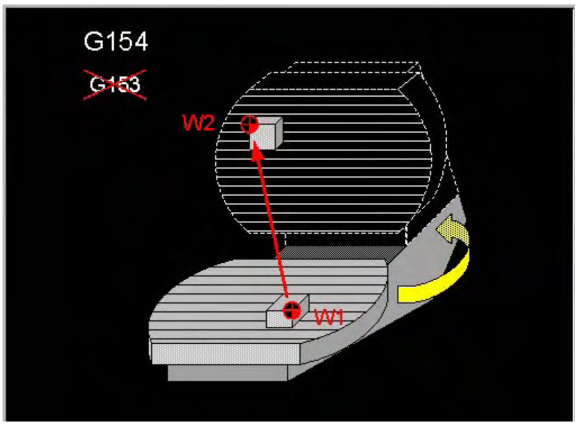
When the rotary axis rotates, the zero point from the work piece rotates with the work piece. The difference with G7 is, that the axes directions are not rotated also.

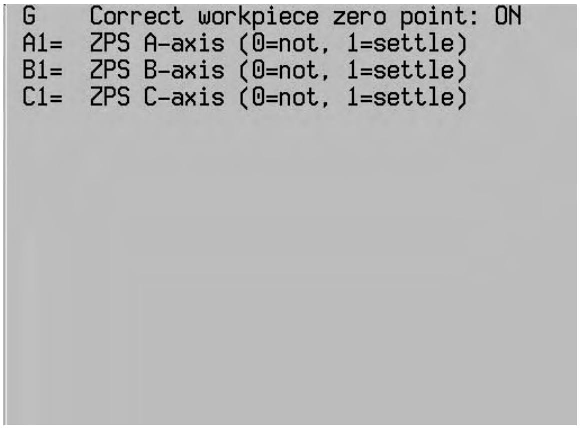
The G154-function activates the displacement of the work piece zero point by means of calculations in the kinematics. This can only be activated for rotary axes in the table. When active, the position of the programmed rotary axis is calculated in the position of the linear axes. The linear axes are not dragged along.

Note: The offset in the linear axes because of G108 is independent of G154/G153 and remains active. G108 has the same function, however is only active for the head.

Format

G154 {A1=..} {B1=..} {C1=..}





A1= Defines whether the position of the A-axis in the table is calculated in the linear axes.
0 = not calculated (default)
1 = calculated
This address is only allowed when there is an a-axis in the table.

B1= and C1= for the B-axis and C-axis.

Default settings

When no address is programmed all axes in the table are activated.

Notes and usage

Modality

This function is mutual modal with G153.

Execution

When G154 is active, the display of the linear axes at the end of every positioning of the axes defined in G154 is adapted.
G154 refrains from all actions until the movement in the previous block has ended (<INPOD>).

Switching off G154

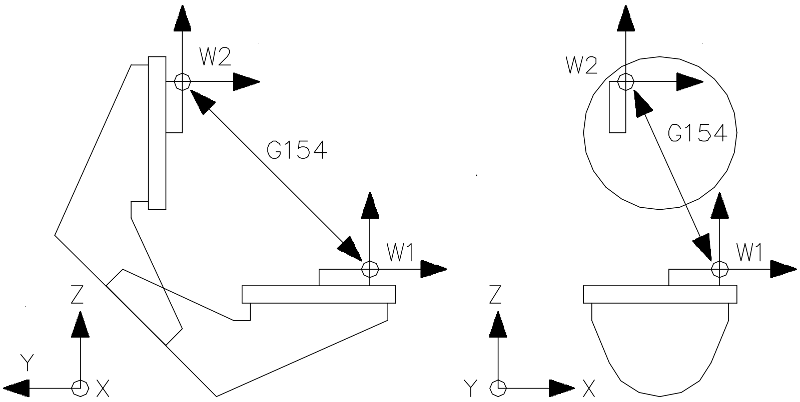
The function G154 is switched off by G153.
After <cancel program>, M30, <Clear Control> or switching on the CNC, the function G154 remains active. The programmed rotary axis is saved in the stand-by memory.

Interrupt

When a rotary axis movement is interrupted, the display of the liner axes is not adapted.
Only after <Emergency Stop>, <cancel program> or <manual mode> during program interrupt, the display of the linear axes is updated to the state of the rotary axis.

Manual mode

The function G154 remains active after M30 and is active in manual mode. The display of the linear axes is updated when the rotary axis movement is stopped.



W1 = Work piece zero point in position 1
W2 = Work piece zero point in position 2.
In this case the table is rotated 180° around the B-axis.
G154 is the zero point displacement caused by the axis rotation.

Zero point shift

A zero point shift (G54, G92, G93) or IPLC-shift in the relevant rotary axis is taken into account. This means that the new zero point of the rotary axis is taken as the zero position for the kinematic calculations.

Status-display

The G153- / G154-status is displayed in the modal G-group display.

Example Activating zero point displacement.

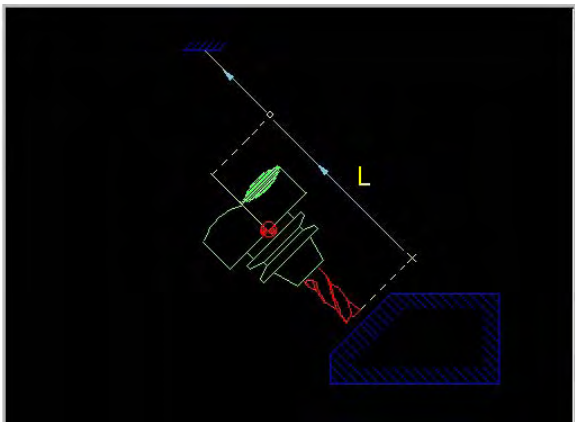
Programming example	Description
N10 G154 B1=1	Work piece zero point is corrected after the table rotation.

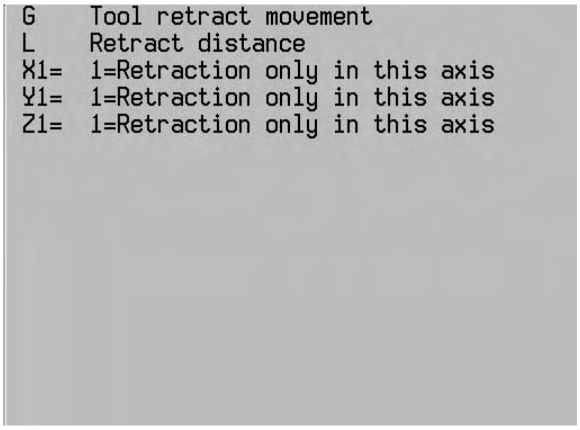
23.72 G174 Tool withdrawal movement

Movement to move the tool axis clear during 5-axis milling.

Format

G174 {L....} {X1=.. or Y1=.. or Z1=..}





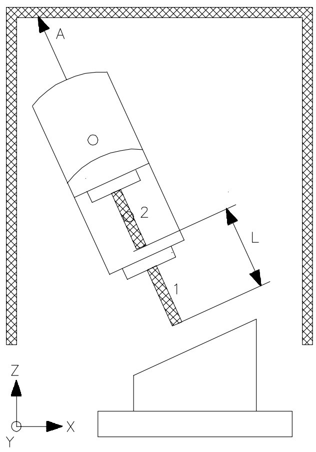
Notes and usage

Execution

With this function, you are always able to move away in the direction of the tool axis. The tool is withdrawn until the ‘first’ SW limit switch is reached. The direction of movement is determined by the position of the milling head. In the direction of the tool head the tool is withdrawn.

Execution (X1= or Y1= or Z1=)

With programming an X1= or Y1= or Z1= will be fixed, which machine axis will be moved. During G7 the machine axis can be different from the programmed axis. A combination of X1=, Y1= und Z1= is not allowed (P414). **The movement is not in the tool direction**. X1=1 means, that the X-axis will be moved.



1	Starting position
L	Withdrawal distance
2	End position
A	Limitation by software limit switch

Withdrawal distance (L)

The withdrawal distance (L > 0) defines the distance travelled in the direction of the tool.
An error message is given, when L is bigger than the distance to the software limit switch (Z31).
Without programming the withdrawal distance (L) the movement is limited by the software limit switch.

Execution (G0)

G174 is executed in rapid. If F6= is programmed this feed is taken.
Following G107, G0 or G1 from the previous block is modally active again.

Example Tool withdrawal movement.

N10 G174 L100	Tool retracts 100 mm.
N..	
N30 G174 L100 X1=1	Tool moves 100 mm in the X-axis.

23.73 Cancel cylinder interpolation or activate basic coordinate system G180

Cancel the cylindrical coordinate system or define the main plane and tool axis (basic coordinate system).

N... G180 [main axis 1] [main axis 2] [tool axis]   Basic coordinate system

Parameters

```
G    Cancel cylinder interpolation
X    1=allocate axis to coord.system
Y    1=allocate axis to coord.system
Z    1=allocate axis to coord.system
B    1=allocate axis to coord.system
C    1=allocate axis to coord.system
```

General principles

The normal expression is G180 X1 Y1 Z1

Only the following configurations are possible:

Main axis 1	X
Main axis 2	Y
Tool axis	Z or W

The correct procedure depends on three different items of information:

1) G17/G18/G19 determines the tool axis (G17 Z).
2) G180 determines the axes to be changed (G17 W in Z)
3) The machine constants for the tool axis definition should be OK. (Tool axis W belongs to Z).

Example

N12340	
N1 G17 T1 M6	
N2 G54	
N3 F1000 S1000 M3	
N4 G180 X1 Y1 Z1	Activate main plane XY and tool axis Z.
N5 G81 Y2 B10 Z-22	Define cycle.
N6 G79 X0 Y0 Z0	Drill with the feed movement in the Z axis.

Notes and usage

Functions G41...G44, G64, axis rotation (G92/G93 B4=) and G141 should be cancelled before G180 is activated.

Tool length compensation is active in the defined tool axis. Radius compensation is active in the main plane.

The machine constants must be used correctly. If the tool axis is the fourth axis, MC117 should be 3 (MC117 = 3) (same as Z axis). MC3401 = 0 (tool axis is a linear axis).

Only Cartesian coordinates can be used.

If G180 is programmed and radius compensation is still active, it will be cancelled by G180.
We recommend to cancel radius compensation, using G40, and to change to the basic coordinate system.

23.74 Cancel / activate cylinder interpolation G182

Selection of the cylindrical coordinate system. This system simplifies the programming of contours and positions on the curved cylinder surface.

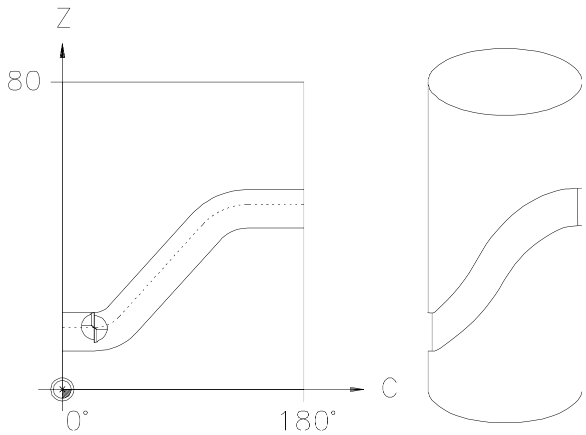
Activate the cylindrical coordinate system:
N.. G182 [cylinder axis] [rotational axis] {tool axis}

Rapid feed when G182 in effect:
N.. G0 [cylinder axis] [rotational axis] (tool axis}

Linear feed movement:
N.. G1 [cylinder axis] [rotational axis] (tool axis} {F..}

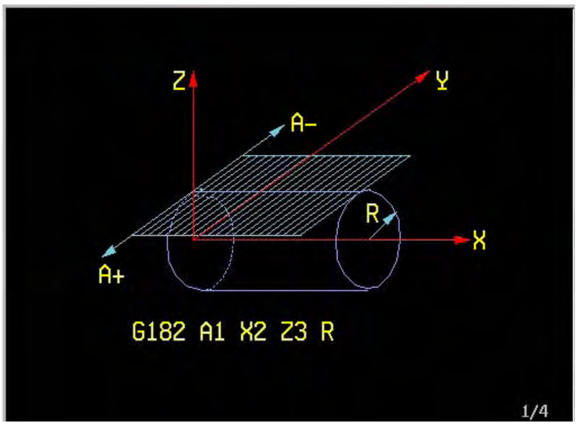
Circular feed movement:
N.. G2/G3 [cylinder axis] [rotational axis] R..

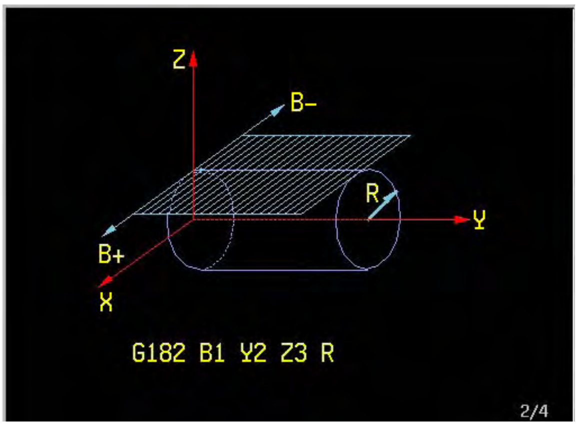
Return to basic coordinate system:
N.. G180
or
M30, Cancel program softkey, Clear control softkey.



Parameters

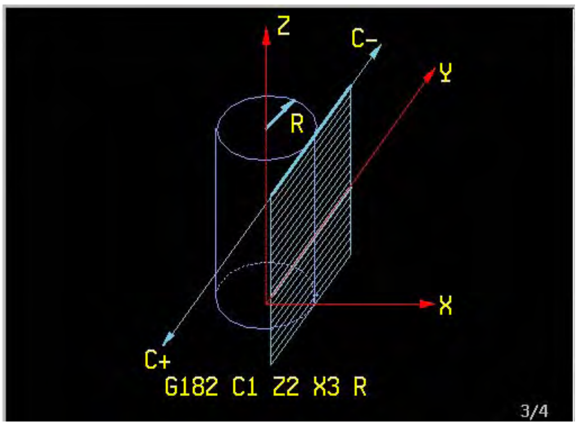
G	Activate cylinder interpolation
X	Cylinder plane:2 / Tool axis:3
Y	Cylinder plane:2 / Tool axis:3
Z	Cylinder plane:2 / Tool axis:3
B	Cylinder plane:1
C	Cylinder plane:1
R	Cylinder radius

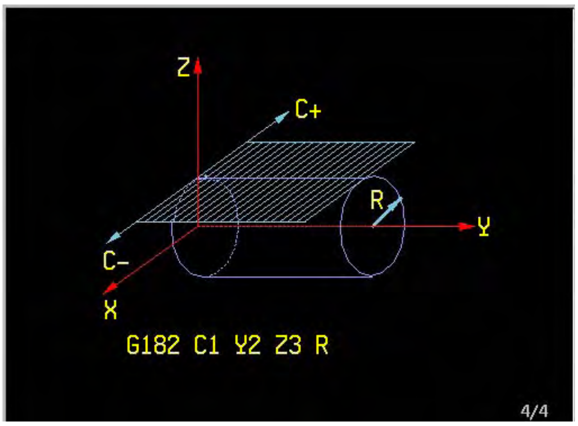




G182 A1 X2 Z3 R..
or (as until now)
G182 A1 X1 Z1 R..

G182 B1 Y2 Z3 R..
or (as until now)
G182 B1 Y1 Z1 R..





G182 C1 Z2 X3 R..
or (as until now)
G182 C1 X1 Z1 R..

G182 C1 Y2 Z3 R..

Specification of the cylinder plane

Notes

The words X,Y,Z,A,B,C must not be programmed without a value.
The configuration for the cylinder interpolation is programmed in block G182:

- standard configuration

rotational axis	A1	B1	C1
cylinder axis	X1	Y1	Z1
tool axis	Y1/Z1	X1/Z1	X1/Y1
cylinder radius	R	R	R

- enhanced configuration (V321)

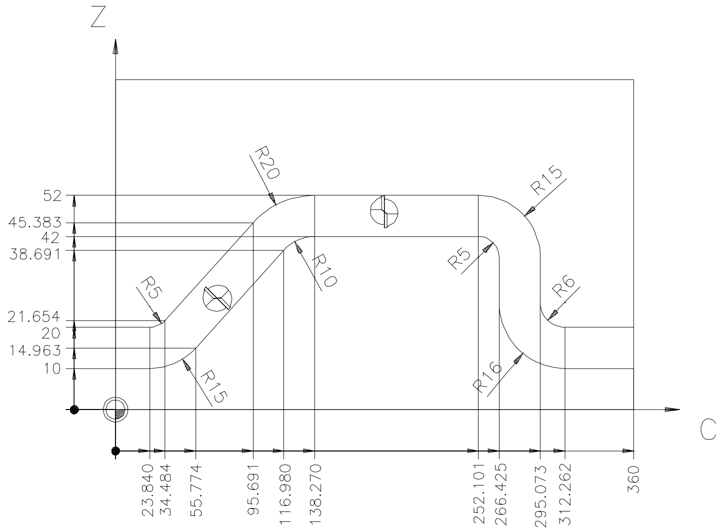
rotational axis marked 1	A1	B1	C1
cylinder axis marked 2	X2/Y2/Z2	X2/X2/Z2	Z2/X2/Y2
tool axis marked 3	Y3/Z3/X3	X3/Z3/Y3	X3/Y3/Z3
cylinder radius	R	R	R

Machine constants

The machine constants for the axis definitions have to be correct.

MC 102 = 1, MC103 = 88 (X-axis)
MC 107 = 2, MC108 = 89 (Y-axis)
MC 112 = 3, MC113 = 90 (Z-axis)
MC 117 = 4 belongs to axis 1 (4-3), MC118 = 65 (A-axis turning around X-axis)
MC 122 = 6 belongs to axis 3 (6-3), MC123 = 67 (C-axis turning around Z-axis)

Example



The recess on the curved surface of a cylinder (diameter 40 mm) is to be milled using a dual-point endmilling cutter (diameter 9.5 mm). The working depth is to be 4 mm. The horizontal working of the workpiece is to be performed on the rotational axis C, the cylinder axis Z and the tool axis Y.

```
N12340
N1  G18  S1000  T1  M66
N2  G54
N3  G182  Y1 C1  Z1  R20
N4  G0  Y22 C0  Z15  M3
N5  G1 Y16  F200
N6  G43 Z10
N7  G41
N8  G1 C23.84
N9  G3 Z14.963 C55.774  R15
N10  G1 Z38.691 C116.98
N11  G2 Z42 C138.27  R10
N12  G1 C252.101
N13  G2 Z37 C266.425  R5
N14  G1 Z26
N15  G3 Z10 C312.262  R16
N16  G1 C365
```

```
N17  G40
N18  G41 Z20
N19  G1 C312.262
N20  G2 Z26 C295.073  R6
N21  G1 Z37
N22  G3 Z52 C252.101  R15
N23  G1 C138.27
N24  G3 Z45.383 C95.691  R20
N25  G1 Z21.654 C34.484
N26  G2 Z20 C23.84  R5
N27  G1 C0
N28  G40
N29  G180
N30  G0 Y100  M30
```

Notes

Only cartesian coordinates can be used.

The following functions must not be active when G182 is active:
G41-G44, G64, G92/G93 B4=, G141

The following cannot be programmed when G182 is active:
G25/G26, G27/G28, G51-G59, G61/G62, G70/G71, G73, G92/93.

The selected tool radius should only be fractionally smaller than the width of the recess (undercutting !)

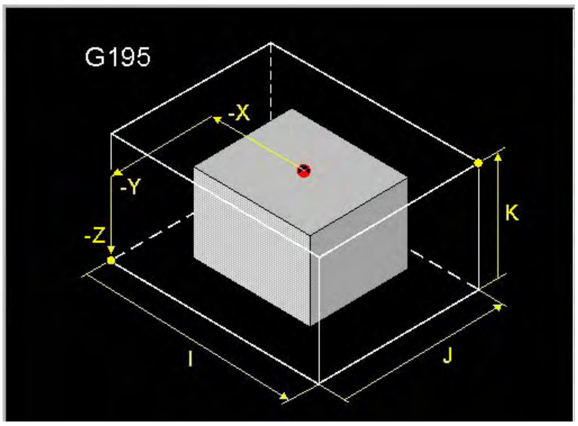
Limitation:

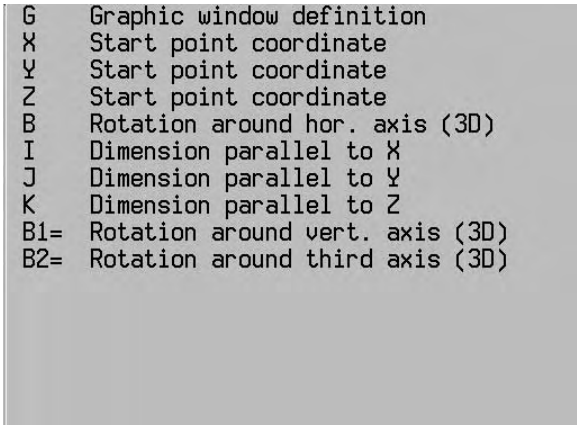
Cylinder radius >5mm <500mm

23.75 Graphic window definition G195

Specify the dimensions and length of a 3D graphics window with reference to the zero point W.

N.. G195 X.. Y.. Z.. I.. J.. K.. {B..} {B1=..} {B2=..}





Example

N9000	
N1 G17	
N2 G195 X-30 Y-30 Z-70 I170 J150 K100	Graphic window definition
N3 G199	Start of graphic contour description

23.76 End graphic model description G196

N.. G196

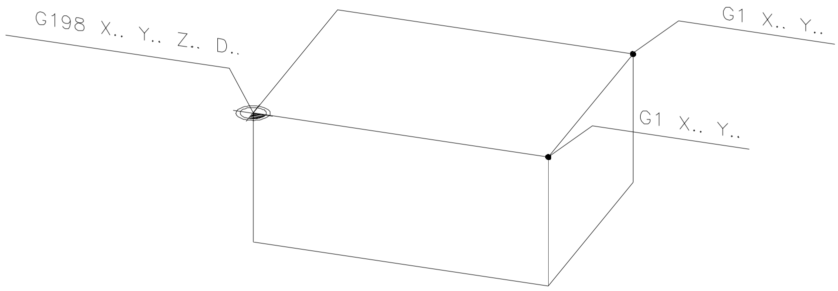
Example

:	
N2 G195 X... Y... Z... I... J... K...	Graphic window definition
N3 G199 X... Y... Z.. B.. C..	Start of graphic contour description
N4 G198 X.. Y.. Z.. D..	Start of outside contour description
:	
:	
N25 G197 X.. Y.. D..	Start of inside contour description
:	
:	
N35 G196	End of graphic contour description

23.77 Begin inside / outside contour description G197/G198

Define the start point of an internal contour:
N.. G197 X.. Y.. {Z..} D.. {I1=..}.

Define the start point of an external contour:
N.. G198 X.. Y.. {Z..} D.. {I1=..}.



Parameters

G	Begin inside contour description
X	Startpoint coordinate
Y	Startpoint coordinate
D	Depth inside contour
?90=	Startpoint abs. (X,Y,Z..)
?91=	Startpoint incr. (X,Y,Z..)
I1=	Colour

G	Begin outside contour description
X	Startpoint coordinate
Y	Startpoint coordinate
Z	Startpoint coordinate
D	Depth outside contour
?90=	Startpoint abs. (X,Y,Z..)
?91=	Startpoint incr. (X,Y,Z..)
I1=	Colour

Example

See G199

Possible colours (I1=):

1	red	11	light red
2	green	12	light green
3	yellow	13	light yellow
4	blue	14	light blue
5	grey	15	light magenta
6	cyan	16	light cyan
7	white	17	bright white
8	black	18	black
9	foreground	19	foreground
10	background	20	background

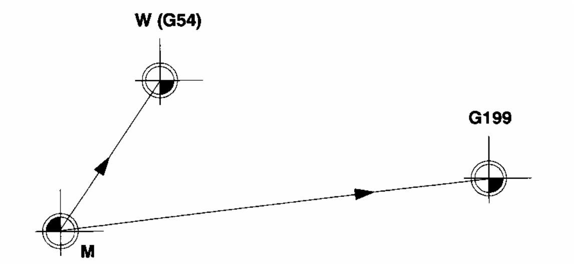
Notes

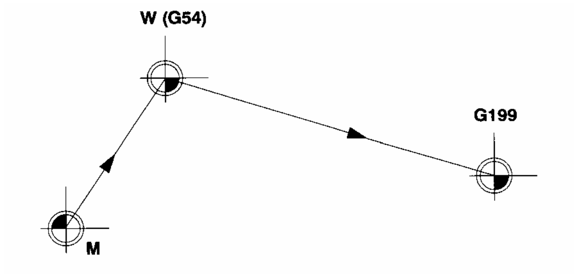
The start point of the contour is based on the offset in G199.
The contour must be complete.
The internal contour must lie within the external contour.
An internal contour cannot lie within another internal contour.

23.78 Begin graphic model description G199

Define the position of a blank contour or a machine part (e.g. chucking equipment) that the tool could collide with. A collision can be detected during the graphical simulation.

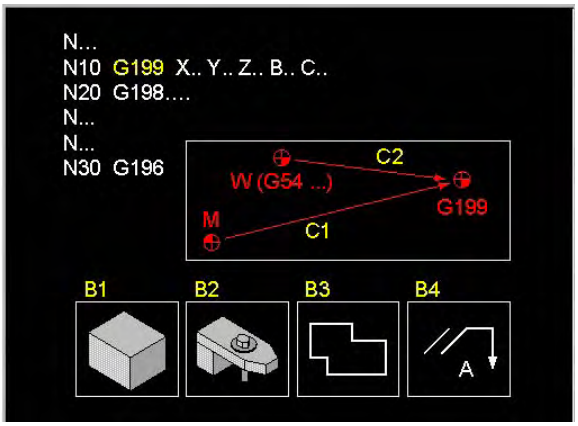
Define a blank contour:
N.. G199 [start coordinates] B1 {C1} {C2}
Define a machine part contour:
N... G199 [start coordinates] B2 {C1} {C2}
Drawing a contour during simulation of the wire plot graph.
N... G199 [initial co-ordinates] B3 {C1} {C2}
Draw one or more geometry elements (line or circle) during the wire model graphic simulation.
N... G199 [co-ordinates of position] B4 {C1} {C2} Drahtmodellgrafik.

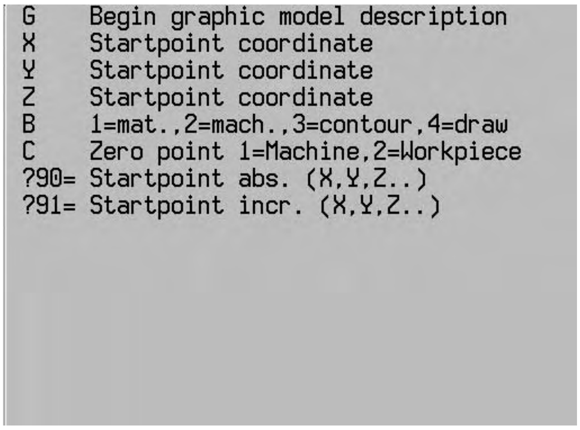




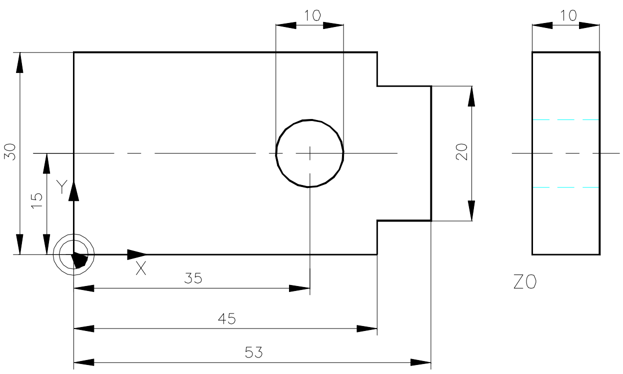
C1 Description related to M

C2 Description related to W





Example



Each chucking tool is described in its own macro. The start point of the chucking tool contour is programmed using two parameters:

E1: X coordinate of the contour start point, in relation to the program zero point
E2: Y coordinate of the contour start point, in relation to the Program zero point

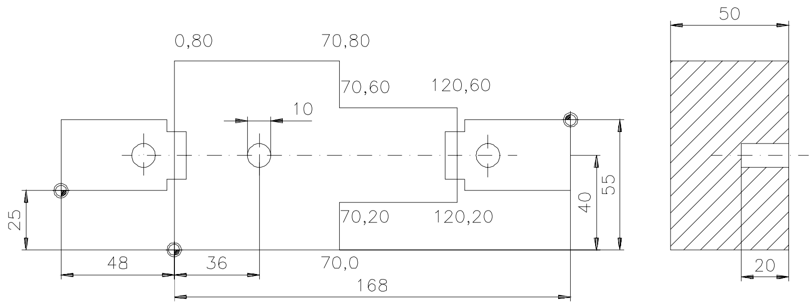
Macro for the left chucking tool:

```
N1991
N1  G92  X=E1  Y=E2
N2  G199  X0  Y0  Z0  B2  C2    Start of graphic contour description
N3  G198  X0  Y0  Z0  D10       Start of outside contour description
N4  G1  X45
N5  Y5
N6  X53
N7  Y25
N8  X45
N9  Y30
N10  X0
N11  Y0
N12  G197  X30  Y15  D-10       Start of inside contour description
N13  G2  I35  J15
N14  G196                       End of graphic contour description
N15  G92  X=-E1  Y=-E2
```

Macro for the right chucking tool (top figure, rotated 180°)

```
N1992
N1  G92  X=E1  Y=E2
N2  G199  X0  Y0  Z0  B2  C2
N3  G198  X0  Y0  Z0  D10
N4  G1  X-45
N5  Y-5
N6  X-53
N7  Y-25
N8  X-45
N9  Y-30
N10  X0
N11  Y0
N12  G197  X-30  Y-15  D-10     Start of inside contour description
N13  G2  I-35  J-15
N14  G196                       End of graphic contour description
N15  G92  X=-E1  Y=-E2
```

Graphical section of the part program:




```
N199000 (Main program)
N1  G17
N2  G54
N3  S1200  T1  M6
N4  G195  X-20  Y-20  Z-60  I180  J110  K70
N5  G199  X0  Y0  Z0  B1  C2           Start of inside contour description
N6  G198  X0  Y0  D-50                 Start of outside contour description
N7  G1  X70
N8  Y20
N9  X120
N10  Y60
N11  X70
N12  Y80
N13  X0
N14  Y0
N15  G197  X31  Y40  D-20              Start of inside contour description
N16  G2  I36  J40
N17  G196                              End of graphic contour description
N18  G22  N=1991  E1=-48  E2=25        Macro call - LH fixture
N19  G22  N=1992  E1=168  E2=55        Macro call - RH fixture
:
N200  M30
```

23.79 Universal pocket milling cycle G200- G208

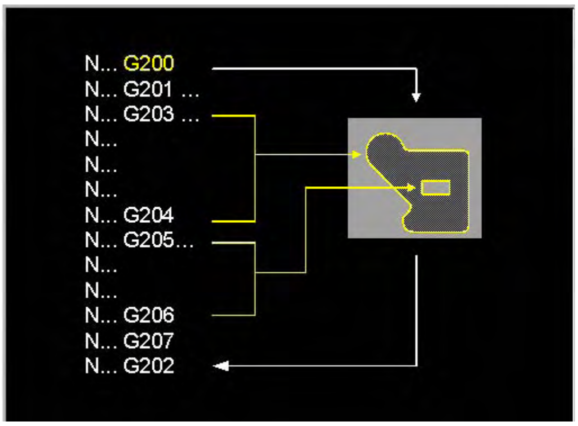
The universal pocket cycle is an easy and fast way of making CNC-programs for milling pockets of any shape with or without islands.

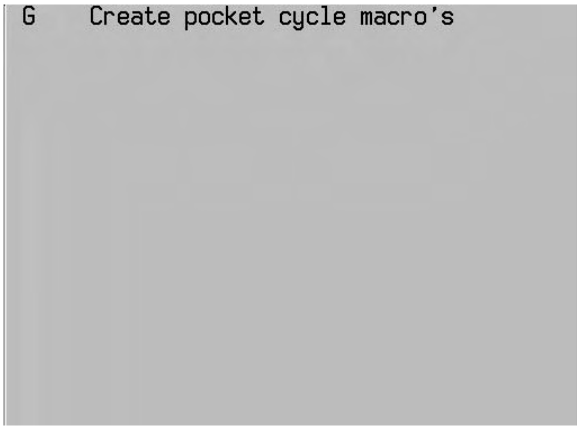
Program format:

```
N99999
N1  G17
N2  G54
N3       \
:         >                                  Normal working
N96      /
N97  G200
N98  G81
N99  G22  N=..                               Rough drill start point
N100 G201 N1=.. N2=..                        Start of pocket description for milling the pocket
N101  G203  N1=..                            Start of pocket contour description
N102    \
:         >                                  Pocket contour description
N109    /
N110  G204                                   End of pocketncontour description
N111  G205  N1=..                            Start of island contour description
N112    \
:         >                                  Contour description island 1
N118    /
N119  G206                                   End of island contour description
N120  G205  N1=..                            Start of island contour description
N121    \
:         >                                  Contour description island 2
:        /
N130  G206                                   End of island contour description
N220  G207 X..  Y..  N=..  N1=..             Call island contour macro
N221  G203 / G205                            Start pocket / island contour description
N222  G208                                   Contour description for parallelogramm
N223  G204 / G206                            End of pocket / island contour description
N131  G202                                   End of pocket contour cycle
N350  G22  N=..                              Rework the contour
N351  G22  N=..                              Rework island 1
N352  G22  N=..                              Rework island 2
:
N500  M30
```

23.80 Create pocket cycle macro's G200

N.. G200





This function must be programmed before the universal pocket cycle and indicates that:

- the coordinates for the milling paths must be calculated (if they have not already been calculated).
- the milling paths are programmed in a macro generated by the CNC; the number (N1=..) of this macro is specified in a G201 block.
- if necessary (indicated by N2=.. in a G201 block) a second macro to drill the start point will be generated.
- if necessary (indicated in a G203 or G205 block) the macros (N1=..) for reworking the contours will be generated.

All operating conditions, such as processing level, zero point offsets and tool corrections should be specified before the G200 function is executed.

Point definitions (G78), which are used to specify the pocketn contour, should be defined before the G200 block.

A G200 block can be incorporated in a macro; the pocket will, however, only be searched for in macros nested at a lower level.

The CNC expands the macros before the program is executed. Blocks between G200 and G201 are therefore ignored at first. These blocks are processed once the macros have been generated.

All universal pocket cycles programmed between a block G200 and G202 or M30 are calculated simultaneously.

The machining level (G17/G18/G19) has to be defined before G200 or after G202 was programmed.

Note

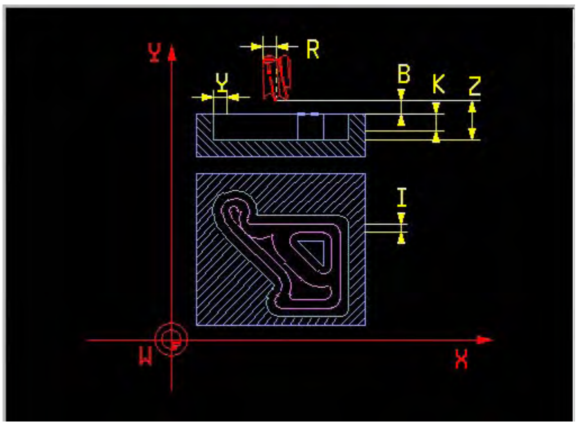
As of V321 the user can no longer see generated macros in the macro memory. If a macro is to be used in another program, the relevant macro number must first be entered in the macro memory. Only then will the macro be visible in the macro memory and can it be read in/out.

23.81 Start contour pocket cycle G201

Start of the definition of a pocket (possibly including islands). The block contains the technological data required for calculating the milling paths. The milling of the pocket starts from the G201 block.

N... G201 Y... Z... N1=.. N2=.. {B...} {I..} {J..} {K..} {R..} {F..} {F2=..}

Parameters



G	Start contour pocket cycle
Y	Stock removal
Z	Total pocket depth
B	Clearance
I	Cutting width mill in %
J	1=climb, -1=conventional
K	Cutting depth
R	Tool radius for calculation
N1=	Milling macro number
N2=	Startpoint macro number

Notes

The addresses (especially Y and Z) are independent of the active level.

The functions G90, G40 and G63 are activated automatically when the G201 function is executed.

The functions G201/G202, G203/G204 and G205/G206 must all be in the same program/macro.

The only blocks that may appear between G201 and G202 are: G203/G204, G205/G206 and G207.

The only blocks that may appear between G203/G204 and G205/G206: G1, G2/G3, G208, G63/G64, G90, G91.
The movements G1, G2/G3 are limited to the main level. Tool axes and rotary axis coordinates are not permitted.

The program is to be continued from an absolute position after describing the pocket.

E parameters can be used for contour descriptions. Calculations must be performed before G200 is executed.

23.82 End contour pocket cycle G202

Termination of the full pocket description.

N.. G202

Note

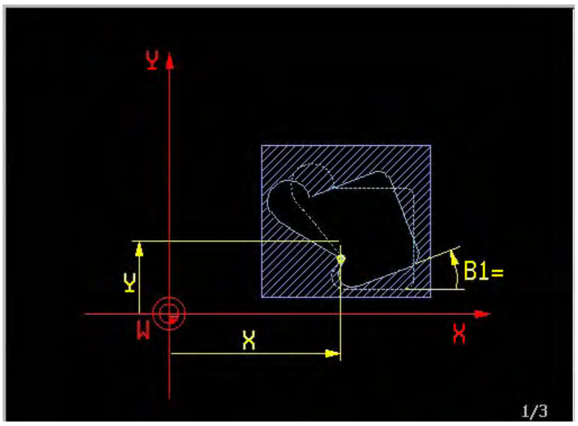
The functions G0, G40, G63 and G90 are activated automatically when the G202 function is executed.

With G202 the calculation of universal pocket cycles is stopped. Calculation is continued with the next G200.

23.83 Start pocket contour description G203

N.. G203 X.. Y.. Z.. N1=.. {P..} {B1=..} {B2=..} {L2=..} {P1=..}

Parameters



G	Start pocket contour description
X	Startpoint in X
Y	Startpoint in Y
Z	Startpoint in Z
P	Point definition number
N1=	Finishing macro number
B1=	Rotation angle of contour pocket
B2=	Start point polar angle
L2=	Start point polar length
P1=	Point definition number

The tool axis coordinates must always be in the G203 block.

Notes

G1, G63 and G90 are activated automatically when G203 is executed.

The first point of a contour description must be specified in a G203 block. Reworking of the contour also starts at this point.

The bottom of the pocket must lie parallel to the processing level.

The edges of the pocket must be at right angles to the bottom of the pocket.

Two elements of the same pocket must not intersect or touch.

During finish-cutting, the programmer must ensure that a tool diameter is selected that is smaller than the narrowest section in the pocket of the work piece. Contour violations during finishing are not detected by the controller.

23.84 End pocket contour description G204

This function terminates the description of the pocket contour.

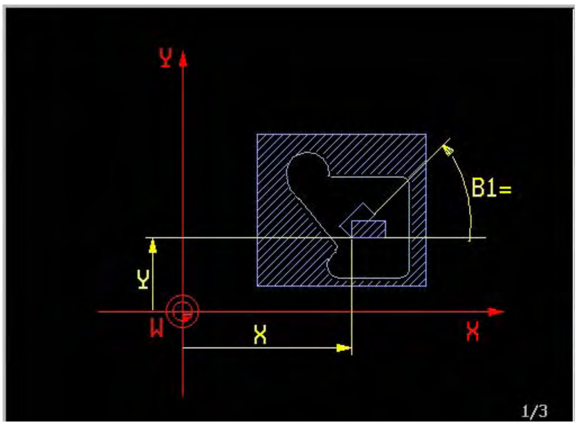
N.. G204

23.85 Start island contour description G205

The contour of an island is described in the same way as the contour of a pocket. The description begins with G205 and the absolute start position of the island.

N.. G203 X.. Y.. N1=.. {Z..} {P..} {B1=..} {B2=..} {L2=..} {P1=..}

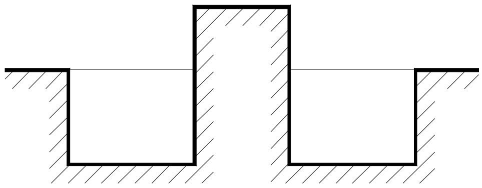
Parameter



G	Start island contour description
X	Startpoint in X
Y	Startpoint in Y
P	Point definition number
N1=	Finishing macro number
B1=	Rotation angle of island contour
B2=	Start point polar angle
L2=	Start point polar length
P1=	Point definition number

Notes

The CNC assumes that the tops of the island and the pocket are at the same level.



If the island is higher than the top of the pocket, the B word in the G201 block can be used to prevent a collision between the milling tool and the work piece while moving from one starting point to the next.

G205 activates G1, G63 and G90.

The tool axis must not be programmed.

The contour of an island must be complete.
Two islands may not intersect or touch.
Islands must be situated in the pocket and must not intersect or touch the edges.
The sides of an island must be at right angles to the bottom.

23.86 End island contour description G206

The contour description is terminated by G206. The description for pocket contours applies equally to island contours.

N.. G206

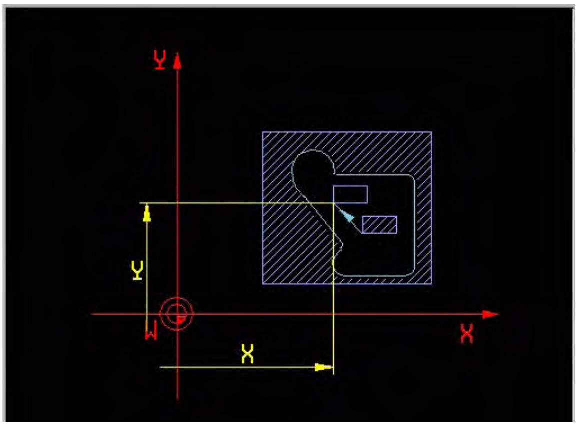
23.87 Call island contour macro G207

N... G207 X.. Y.. Z.. N=.. N1=..

There are three possibilities:
1. The same island appears at another place in the same pocket contour.
2. The same island contour appears in another pocket contour.
3. The same island contour appears in another program.

As the island contour is included in a macro, the three options can all be handled in the same way.

Parameters

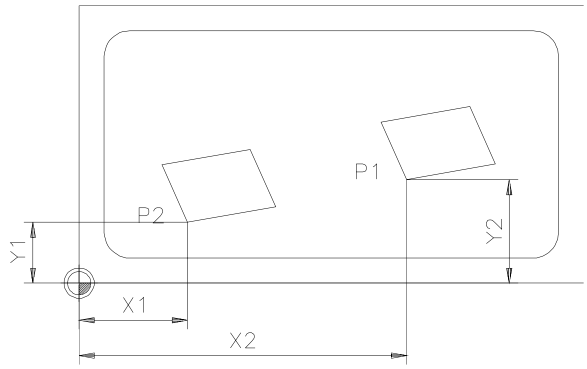


G	Call island contour macro
X	Shift along in X
Y	Shift along in Y
Z	Shift along in Z
N=	Macro with islandcontour
N1=	Finishing macro number

Das Makro der Inselkontur lautet:
N9xxx G205 X=X2 Y=Y2 N1=..
N.. \
: > Inselkontur
N.. /
N.. G206

N9xxx stellt hier die Makrokennzeichnung dar.
Das Makro wird mit der Funktion G207 aufgerufen.
N.. G201
:
N.. G207 N=9xxx
N.. G207 N=9xxx X=(X1-X2) Y=(Y1-Y2)
N.. G202

Example



1 : An island whose contours are programmed as a macro
    P1 : Starting point of the contour description (G205 block)
2 : Desired position of the island
    P2 : Starting point of the moved contour
    X..: Distance parallel to the X axis of P1 to P2
    Y..: Distance parallel to the Y axis of P1 to P2

Notes

The subprogram called up in the G207 block must not contain any references to G63/G64.
The best procedure is to start an island contour with the coordinates X0,Y0 (zero offset).The starting point can be programmed in the G207 block without calculation.

The identical macro of the island contour will then be as follows:

```
N9xxx G205 X0 Y0 N1=..
N..       \
:          > Island contour with zero offset
N..       /
N.. G206
```

N9xxx represents the macro identification.
The macro is called with the G207 function.

```
N.. G201
:
N.. G207 N=9xxx X=X2 Y=Y2
N.. G207 N=9xxx X=X1 Y=Y1
N.. G202
```

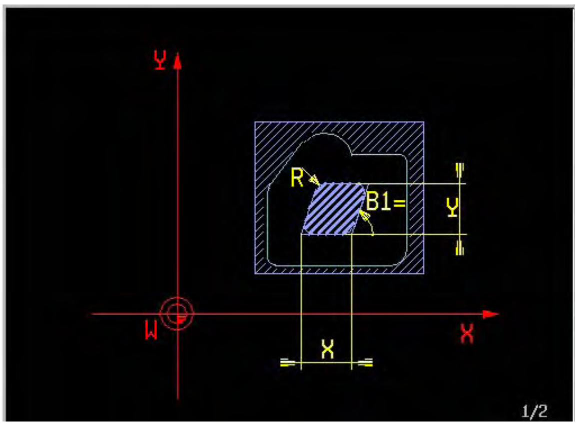
The subprogram for the island contours can be programmed in absolute or incremental dimensions.

23.88 Quadrangle contour description G208

The G208 function enables a regular quadrangle, particularly a rectangle or a parallelogram, to be programmed very easily.

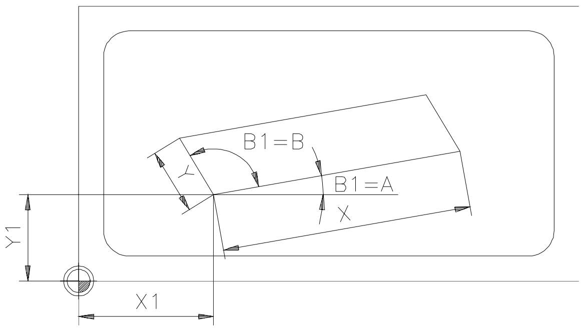
N... G208 X.. Y.. {Z..} {I..} {J..} {R..} {B1=..}

Parameter



G	Quadrangle contour description
X	Length in X
Y	Length in Y
Z	Length in Z
I	Chamfer length
J	1=climb, -1=conventional
R	Rounding radius
B1=	Angle quadrangle contour

Example



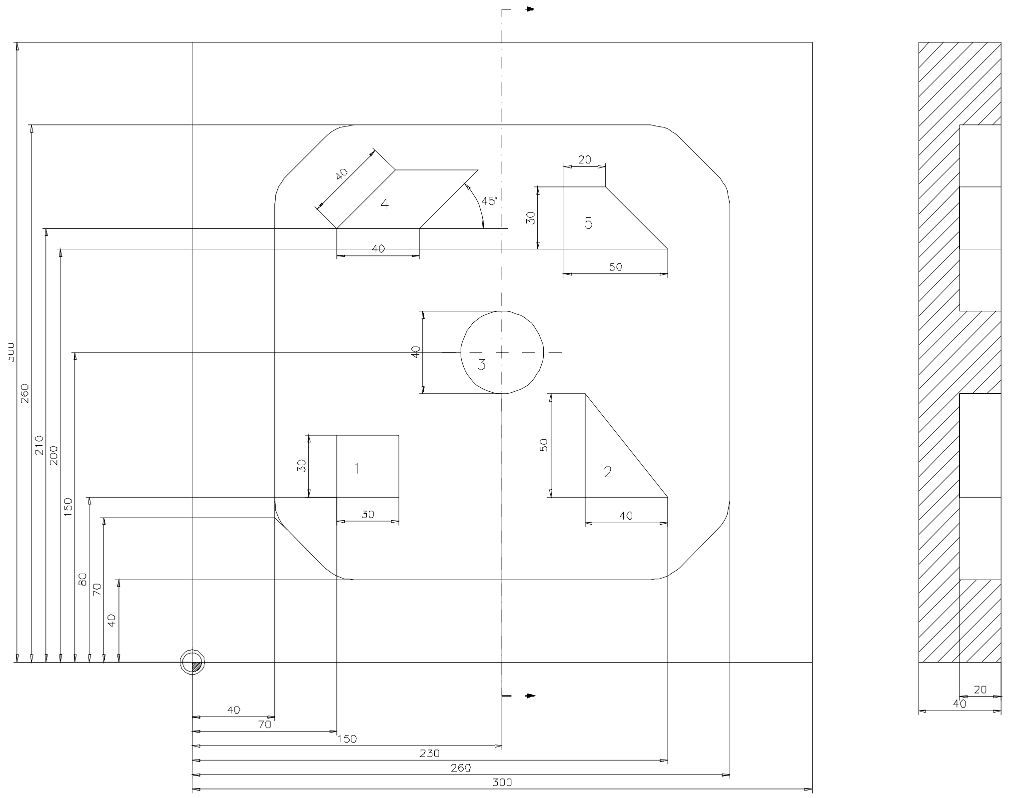
G203 X (=X1) Y (=Y1) Z (=Z1) B1= (=A)
G208 X (=X) Y (=Y) B1= (=B)
G204

Note

The bottom of the pocket must always be parallel to the main level.

Example of a pocket contour

Pocket with islands. Rough drilling of the starting point and reworking of the contours are taken into account.




```
N82150
N1 G17
N2 G54
N3 G98 X-10 Y-10 Z10 I320 J320 K-60
N4 G99 X0 Y0 Z0 I300 J300 K-40
N5 F200 S3000 T2 M6
N6 G200
N7 G81 Y2 Z-20 M3                         (Predrill start point)
N8 G22 N=9992
N9 S2500 T3 M6                            (Evacuate pocket)
N10 G201 Y0.1 Z-20 B2 I50 R10 F200 N1=9991 N2=9992 F2=100
N11 G203 X70 Y40 Z0 N1=9993               |
N12 G64                                   |
N13 G1 X260 B1=0 I1=0                     |
N14 G1 I30                                |
N15 G1 X260 Y260 B1=90 I1=0               |(Pocket contour)
N16 G1 I30                                |
N17 G1 X40 Y260 B1=180 I1=0               |
N18 G1 I30                                |
N19 G1 X40 Y70 B1=270                     |
N20 G63                                   |
N21 G204                                  |
N22 G205 X100 Y80 N1=9994
N23 G208 X-30 Y30 J-1                     (Island 1)
N24 G206
N25 G205 X190 Y80 N1=9995
```

```
N26 G91
N27 Y50                                  (Island 2)
N28 X40 Y-50
N29 G90
N30 G206
N31 G205 X150 Y130 N1=9996
N32 G2 I150 J150                         (Island 3)
N33 G206
N34 G205 X110 Y210 N1=9997
N35 G208 X-40 Y40 J-1 B1=135             (Island 4)
N36 G206
N37 G205 X180 Y200 N1=9998
N38 G91
N39 G1 Y30
N40 X20                                  (Island 5)
N41 X30 Y-30
N42 G90
N43 G206
N44 G202
N45 F200 S2200 T4 M6                     |
N46 G22 N=9993                           |
N47 F200 S2500 T5 M6                     |
N48 G22 N=9994                           |(Reworking)
N49 G22 N=9995                           |
N50 G22 N=9996                           |
N51 G22 N=9997                           |
N52 G22 N=9998                           |
N53 G0 Z100 M30
```

23.89 G217/G218 Deactivate/Activate angular head

With G218 an angular head is activated. With this it is possible, also in a slanted plane (G7), to define correctly the dimensions and direction (plane) of an angular head with tool.

Format

G217
G218 {X} {Y} {Z} {A5=} {B5=} {C5=}

```
G    Activate angular head
X    Offset angular head
Y    Offset angular head
Z    Offset angular head
A5=  Rotation tool direction X-axis
B5=  Rotation tool direction Y-axis
C5=  Rotation tool direction Z-axis
```

X, Y, Z — Defines the offset without tool in X, Y, Z-direction of the angular head [mm].
A5=, B5=, C5= — Defines the rotation around the X, Y, Z-axis (space angle) of the tool direction (degr.). If no angle is programmed, a default value of A5= -90 [degr.] is taken. This corresponds with an angular head in negative Y-direction.

Notes and usage

Modality

G217 and G218 are mutual modal.

Deactivation

The function G218 is deactivated by G217.
G217 deactivates the allowances of G218. The normal tool length of the active tool is reactivated.
G217 and G218 refrain from all actions until the movements in the previous block are stopped with <INPOD>.

Data, used when activating the angular head.

- Dimensions of the angular head in X, Y, Z and tool direction in A5=, B5=, C5=.
- Tool length, radius and corner radius from the tool table. Also additional lengths and radii from the tool table are used.
- Depending on the IPLC, the angular heads have their own Q3= coding in the tool table.

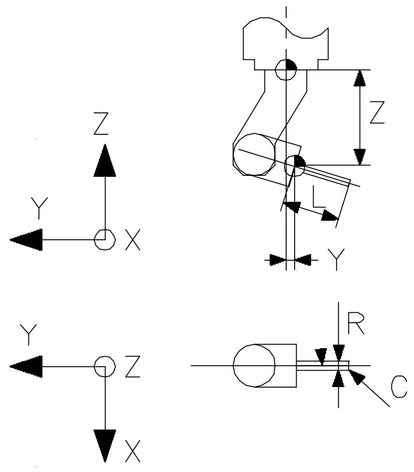
Note: For the measures of the angular head, it is assumed that the angle setting and the tool are fixed. The angle and the tool cannot be changed without measuring the complete system again.

Data of the angular head (Array).

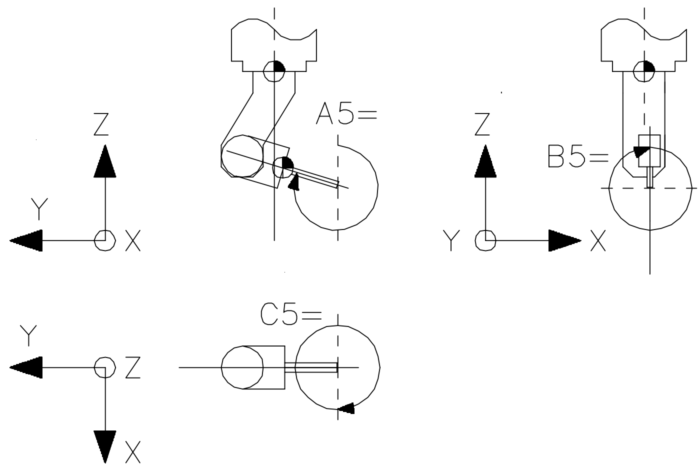
The data of the angular head are stored in arrays.
During measuring the angular head, the cycle writes the measured dimensions in an array.

Note: These cycles and the basic function G128 can also be used for a feed spindle.

Dimensional notations: Angular head reference point



Dimensional notations: Angular head direction = G7 plane



The offsets of the angular head are defined without a tool. The dimensions are defined in the positive direction, which means that the Z-offset is positive in any case and the X- and Y-offsets are depending on the situation (in this example positive).

The angles are defined as space angle. This means as a positive rotations around the positive normalised linear axes XYZ (as in G7). Herewith the rotation around the C-axis is executed first, then around B- and finally around the A-axis.

In this example applies: A5 = 290 or -70 [degr.]
B5 = 0 [degr.]
C5 = 0 [degr.]

Note: The angle C5= is measured from the positive X-axis. A default rotation between this positive X-axis and the M19 D0 position (and angle setting on the angular head) is set in a machine constant.

G7 Plane

When G218 is active, the plane must be set separately with G7. Herewith the G7 can be programmed with the same angles as defined for the angular head. In this case the rotary axes do not turn.

When required, the main plane (Xp, Yp) can be turned with G7 C6=.

Turning mode G36

In turning mode it is also possible to activate an angular head (anyhow in theory). In this case the tool radius R is also compensated with the angles of the angular head in the turning planes G17 Y1=1 Z1=2 and G18 Y1=1 Z1-=2.

Tool length allowance

When G218 is activated, the G39 "allowance programming" and the measuring cycle allowance L4= in the tool table are also compensated with the angles of the angular head.

Tool retract movement

When G218 is active, the G174 "tool retract movement" is executed in the direction of the angular head.

Note: When G174 is programmed with axis information, the real axis is moved as usual.

Start up of the CNC

G218 is immediately active after starting up the CNC. The function G218 is stored with parameters in the stand-by memory.

Display

The function G218 is not visible in the display

Kinematic model

The function is operative for all machine tool types.

Example: activating angular head

Program example	Description
N1 G218 X0.01 Y-25 Z150 A5=-60 B5= 0 C5= 0	Activating angular head
N2 G217	Deaktivieren Winkelkopf

23.90 G227/G228 Unbalance Monitor: ON/OFF

G227 Switch off Unbalance Monitor.

G228 Switch on Unbalance Monitor.

For the description, please refer to the chapter "Turning mode".

23.91 G240/G241 Contour check: OFF/ON

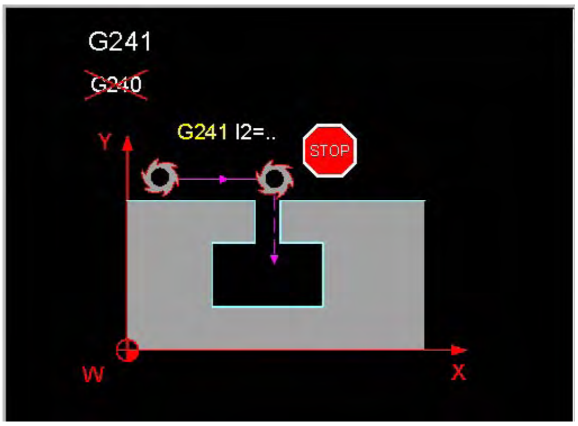
A contour can be checked in two ways with this function:

1. With the reverse check (I1=1 I2=0) is checked whether the compensated straight line (G0/G1) or circle or the programmed straight line (G0/G1) or circle are running in the same direction.
2. With the look ahead check (I1=0 I2=nnn) is checked whether the tool collides with the programmed contour.

These functions are only valid for programs with G41 and/or G42.

Format

G240
G241 {I1=} {I2=...}




```
G    Contour check: ON
I1=  Reverse check: 0=off, 1=on
I2=  Look ahead check: 0=off,>0=number
```

I1= Reverse check:
- 0 = no reverse check (compatible with previous versions).
- 1 = all movements with radius compensation are checked on "reverse".

I2= Defines whether this contour is checked with look ahead check:
- 0 = no check
- nnn = Number of blocks for look ahead check. When nnn > 0, the look ahead check is active.
  Value lies between 0 and 400 (Default: nnn=5)

Note: In version V510, G241 without a parameter is the same as G241 I1=1.
In version V520, G241 without a parameter is the same as G241 I1=1 I2=5.

Note and usage

Refer also to G41/G42

Modality

G240 and G241 are mutual modal.

Deactivating

G241 will be deactivated with G240, M30, < CANCEL PROGRAM > or < CLEAR CONTROL>

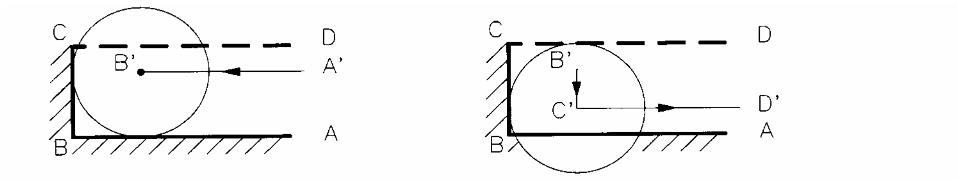
Programming errors

If an inversion of the direction is detected, an error message P412 is given.
<Corrected contour in wrong direction>

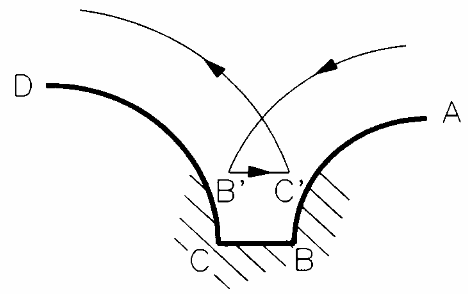
Direction inversion

When the radius of the tool is too big, an inversion of the direction can take place and the workpiece can be damaged. After activating G241 an error message is generated in this case.

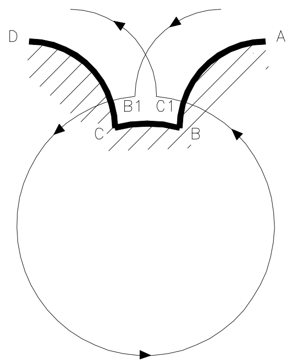
a. The contour from AB to BC is programmed. With active radius compensation the tool retracted along CD. If BC is smaller than two times the tool radius, the tool collides with the workpiece during the movement from B' to C' and from C' to D'.



b. A contour of the shape given in the illustration below is programmed. If the straight line is smaller than two times the tool radius, the tool collides with the workpiece during machining.



c A contour of the shape given in the illustration below is programmed. The tool moves to point B1, then from B1 to C1 and then parallel along CD. The movement from B1 to C1 takes place in the same direction as programmed on the circle BC. If the circular movement BC is too small, the tool will make almost a complete circle before it arrives at C1.



I2= Checking the contour with look ahead check:

Starting the look ahead check

G241 with parameter I2 > 0 sets a modal status. Herewith a look ahead check is started for every next block with G41 or G42.

stopping look ahead check

The look ahead check is stopped by:

- A block with G40, G240 or M30
- A block that switches the radius compensation off automatically (e.g. G79)
- A block with a programming error or a G function that is not allowed (error message)
- End of program or end of an internal read in macro (CAD-mode or BTR)
- Detected collision

Only when no collision is detected the machining of the contour is started.

Interrupt

The calculations for the G241 function can be interrupted.
After interrupting a checked contour, changing the program or tool measures and restarting, the changed contour is not checked again.

Programming errors

When the contour to be checked is faulty, the corresponding error message is already generated during contour checking, together with a P34 error message for the block number.

When during execution a collision is recognised, the error following error message is displayed:

P416 Collision N@@@@@@@ with N@@@@@@@

Example: P416 Collision N24 with N16
When milling block N16 block number N24 is damaged.

Performance

The calculation time for the algorism of G241 I2= is proportional to the total number of the movement elements and to the number of movement elements (I2= parameter) that are checked against each other. The look ahead check of a contour of 100 blocks where 20 blocks must be checked against each other (I2=20) must be ready within 10 [sec.]

Display

The G241 function is shown in the modal G-group display.
During the calculations for the G241 function the "yellow clock" is displayed.

Graphics

When the G241 I2= function finds a collision during a graphical test run, the contour is drawn up to the colliding blocks. With the wire plot graphics the blocks are drawn with the block number and the erroneous block in yellow. The error message P416 is displayed in the last drawn block.

Note: The display of block numbers in the wire plot graphics can also be turned on for "normal" cases. To activate this the softkey F4 <Block numbers> is added to the process <Execution>, menu <Options: Graphics>.

Manual block search

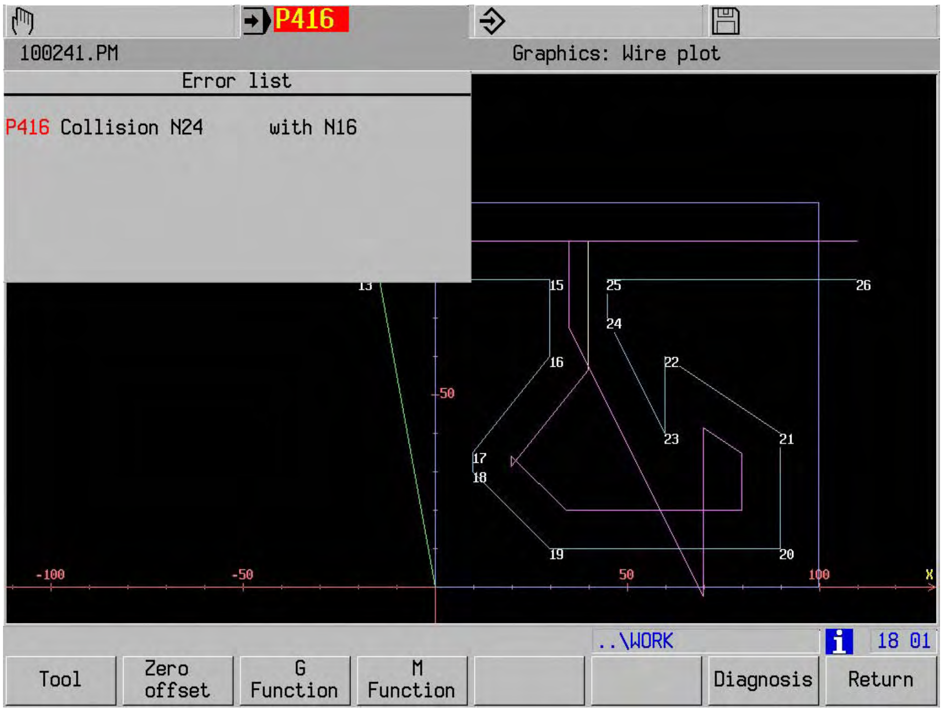
During manual block search the checking of the G241 function are carried out normally.

Example: Contour with radius compensation is checked with look ahead check

Programming example	Description
N100241	Program number
N1 G195 X-5 Y-5 Z5 I110 J110 K-30	Graphics window
N2 G199 X0 Y0 Z0 B1 C2	Material description
N3 G198 X0 Y0 Z0 D20	
N4 G1 X100	
N5 Y100	
N6 X0	
N7 Y0	
N8 G196	End of material description
N9 T20 M6 (Radius 10)	Tool definition radius 10 mm
N10 F1000 S1000 M3	Set feed and spindle speed
N11 G241 I1=0 I2=15	Switch on contour look ahead check (15 blocks)
N12 G0 X-20 Y110 Z-5	Starting position
N13 G43 X-20 Y80	
N14 G41	Switch on radius compensation
N15 G1 X30	Contour description
N16 Y60	
N17 X10 Y35	

N18 Y30	
N19 X30 Y10	
N20 X90	
N21 Y40	
N22 X60 Y60	
N23 Y40	
N24 X45 Y70	
N25 Y80	
N26 X110 Y80	Endposition Kontur
N27 G40	Radiuskorrektur ausschalten
N28 G240	Kontur vorausberechnen ausschalten
N29 M30	Programm Ende

The function G241 I2= builds internally a material contour of all the elementary movements, including the possible generated interconnection circles. After that is checked whether the tool wrap of every elementary movement is not colliding with the programmed number (I2=) of blocks of the look ahead check in the material contour.
The G241 I2= function is programmed modally and works only when the radius compensation is activated. The look ahead check is executed in every block with G41 or G42.



At the first found collision an error message is generated.

In this example 3 collisions are programmed.
The first collision is reported as error: P416 Collision N24 with N16. The other errors are not reported. These are collision N19 with N17 and collision N20 with N23.

In this case all collisions are avoided by reducing the cutter radius to 5 mm.

24. Specific G-Functions for macros

24.1 Overview G-Functions for macros:

Error message functions

G300	Programming error messages
G301	Error in a program or macro

Executable functions

G302	Overwriting radius compensation parameters.
G303	M19 with programmable direction
G310	Store table on disk
G311	Load table from disk

Query functions

G318	Read pallet or job table data
G319	Query actual technology data
G320	Query actual G-data
G321	Query tool data
G322	Query machine constant memory
G324	Query G-group
G325	Query M-group
G326	Query actual position
G327	Query operation mode

Write functions

G331	Write tool data

Calculation functions:

G341	Calculation of G7-plane angles

Formatted write functions

G350	Display window
G351	Write to file

Array functions

24.2 Error message functions

24.2.1 G300 Programming error messages

Setting error messages during the execution of universal programs or macros.

Format

G300 [{D...}|{D1=...}] =...

```
G    Program error call
D    P Error message number
D1=  R Error message number
```

Notes and usage

D are general milling error messages (P), D1= are error messages (R) in turning mode (G36)

The error messages only cover the existing P and R-errors (refer to Machine Manual).

Example Setting an error message if a programmed angle is not allowed.

N9999 (Macro for calculation of table rotations)	
N11 (input parameter: E4: phi)	
N100	
N110 G29 I1 E30 N=180 E30=(E4>360)	Compare if E4 > 360 degrees. If so, jump to N180
N120 G29 I1 E30 N=210 E30=(E4<0)	Compare if E4 < 0 degree. If so, jump to N210
N150 G29 I1 E30 N=290 E30=1	Jump to 290 (0 <= E4 <= 360 degrees)
N160	
N170 (error message: phi>360)	
N180 G300 D190 (programmed value > maximum value)	
	Error message: programmed value > maximum value Program should be ended and a modified E4 be entered
N190	
N200 (error message: phi<0)	
N210 G300 D191 (programmed value < minimum value)	
	Error message: programmed value < minimum value Program should be ended and a modified E4 be entered
220	
N290	Normal program

24.2.2 G301 Error in program or macro that just has been read in.

Error in program or macro block that just has been read in.

Format

G301 (O... Wrong original block)

Notes and usage

When the controller retrieves a program block or macro block and discovers an error it activates G301 Function G301 can only be active in an error stopped program or macro.

This function cannot insert in MDI.

The error texts are O errors. (Refer to Machine Manual).

Example

The program is stored on hard disk.
Program is made with a MC84=0.

```
N9999 (Program)
N1 G17
N2 G57
N3 T1 M6
N4 F200 S1000 M3
..
N99 M30
```

Error stops program in RAM.
Zero point shift extension MC84 > 0 is active.

```
N9999 (ERR*)(Program ...)
N1 G17
N2 G301 (O138 G57)
N3 T1 M6
N4 F200 S1000 M3
..
N99 M30
```

N2 G301 (O138 G57): G301 explains that the program is false. G57 must be G54 I3

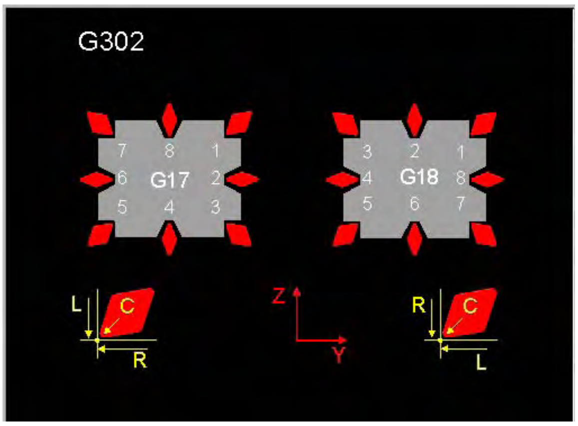
Note The false program can be activated. When passing the block G301 the controller stops and gives the following error text P33 (Modify block converted to connect). The block containing G301 must be changed before restarting.

24.3 Executable functions

24.3.1 G302 Overwriting radius compensation parameters

The G302 function overwrites the active tool parameters during execution. The tool parameters in the tool memory are not changed.

In this version, only the O parameter for tool orientation can be overwritten.




```
G     Overrule radius comp. parameters
O     Tool Orientation
```

For description refer to chapter "Turning mode".

24.3.2 G303 M19 with programmable direction

M19 with programmable direction (CW or CCW).

Format

G303 M19 D... I2=...

```
G     M19 with programmable direction
```

Notes and usage

Only M19 can be programmed.

Default for I2=3

Example Stop spindle with M19.

N100 G303 M19 D75 I2=4	N100:Orientates spindle stop	Angle 75 degrees	CCW

24.3.3 G310 Store table on disk

Storing of user files such as parameter tables or tool data on hard disk.

The maximum number of lines in the user tables is limited by a maximum value allowed in machine constants. By saving to (G310) and reloading from (G311) hard disk of a part of or the complete table the number can be virtually increased.
For tool tables the management is improved. All data of the tools can be stored centrally (presetting device) and still be reached by the CNC.

Format

G310 N5= {I1=} {I2=}

```
G    Store table on disk
I1=  First record in table
I2=  Last record in table
N5=  File name of table
```

N5= Defines the filename and path with which the table must be stored on the internal hard disk or on an external PC.
The complete file name <path + name + type> must be entered between quotation marks ("").

I1= Defines the starting address of a file section.
Value lies between 0 and the end of the relevant user file.
If I2= is not programmed all data lines are stored from I1= onwards.

I2= Defines the end address of a file section.
Value lies between the starting address and the end of the relevant user file.
If I1= is not programmed, all data lines are stored up to and including I2=.

Path definition (N5=)

Work directory is D:\work\

The definition of the path on the internal hard disk is:

- N5= “param.pa“	Data is written to the work directory as param.pa.
- N5= “test1\param.pa“	Data is written to the subdirectory "test1" of the work directory as param.pa.
- N5= “\test2\param.pa“	Starting with \ means that the data is written directly to the directory D:\test2 as param.pa.
- N5= “C:\test3\param.pa“	Error message.

The definition of the path on a network:

- N5= “Z:\test4\param.pa“	**SP**-version: The user file is stored via NFS (Network File System: See Technical Manual) in the directory Z:\test4 as param.pa. **DP**-version: The user file is stored via the windows network in the directory Z:\test4\ as param.pa.
- N5= “\\server1\test5\param.pa“	**SP**-version: Error message. **DP**-version: Starting with \\ means that the user file is stored via the windows network in directory \\server1\test5 as param.pa

The total length of the path and name is:

SP-version: 80 Characters
DP-version: 120 Characters
A local path may be only 5 directories deep in both versions

Note: The path definition is the same for WinShape as for the **DP**-version. The work directory however, depends on the installation, normally it is <c:\winshape\>.

Notes and usage of G310 and G311

Table type:

The following file types are allowed.

PA	E-parameter	Depending on MC83 (Number of E-parameters).
PT	Points	Depending on MC82 (Number of point definitions).'
TM	Tools	Only tool data outside the tool magazine. Depending on MC27 (Number of tools) and MC28 (Number of tool places in the magazine).'
Other tables		Only for service purposes. See Technical Manual.

Execution

G310 and G311 refrain from all actions until <INPOD>. G310 stores the specified section of the table on the hard disk

G311 reads the specified section of the table and stores it in memory. In the remaining program execution the new stored data is used immediately.

When reading the tool memory (G311), MC 774 (Tool in (0,1=clear, 2=protect, 3=replace) is taken into consideration.

Allowed G-functions

G310 and G311 are not allowed with G41, G42, G64 and G141.

Operation and display

When G310 or G311 are executed, the sofkey operation concerning the file functions of the tables is possible. On the other hand the functions G310 and G311 are executed when the file functions are used.

During the execution of G310 or G311 a "yellow clock" is displayed.

Graphics, test run

In the operation modes graphics and test run the functions G310 and G311 are executed.

Manual block search

During manual block search the functions G310 and G311 are executed.

Interrupt

G310 and G311 can be interrupted by <Feed Hold> and <Feed Speed Hold>.

Example:

Programming example	Description
N9000 (Loading/storing data)	
N1 E2=50	nnn=50 enter value
N2 E(E2)=E2	Ennn =nnn
N3 E2=E2+1	increase nnn with 1
N4 G29 I-1 N=2 E0=(E2<=250)	When nnn is equal to or smaller than 250 jump back to N2
N10 G310 N5="datei1.pa" I1=50 I2=250	Storing E-parameters 50 up to 250 on the directory D:\work in the file datei1.pa
N20 G311 N5="\\Server\MillPlus\Param.pa"	Adding of E-parameters in SRAM via network from the file Param.pa on the external directory "\\Server\MillPlus"

24.3.4 G311 Load table from disk

Loading user files such as parameter table or tool data from hard disk.

Note: Please read the description of G310 (Store table on disk) also.

Format

G311 N5= {I1=} {I2=}

```
G    Load table from disk
I1=  First record in table
I2=  Last record in table
N5=  File name of table
```

N5= File name and path, with which the table is stored on the hard disk. The complete file name <path + name + type> must be entered between quotation marks ("").

I1= Defines the starting address of a file section.
Value lies between 0 and the end of the relevant user file.
If I2= is not programmed all data lines are read from I1= onwards.

I2= Defines the end address of a file section.
Value lies between the starting address and the end of the relevant user file.
If I1= is not programmed, all data lines are read up to I2=.

Path definition (N5=)

Work directory is D:\work\

The definition of the path on the internal hard disk is:

- N5= “param.pa“ — Data is read from the work directory as param.pa.
- N5= “test1\param.pa“ — Data is read from the subdirectory "test1" of the work directory as param.pa.
- N5= “\test2\param.pa“ — Starting with \ means that the data is read directly from the directory D:\test2 as param.pa.
- N5= “C:\test3\param.pa“ — Error message.

The definition of the path on a network:

- N5= “Z:\test4\param.pa“ — **SP**-version: The user file is read via NFS (Network File System: See Technical Manual) from the directory Z:\test4 as param.pa.
  DP-version: The user file is read via the windows network from the directory Z:\test4 as param.pa.
- N5= “\\server1\test5\param.pa“ — **SP**-version: Error message.
  DP-version: Starting with \\ means that the user file is read via the windows network from directory \\server1\test5 as param.pa

Example:

Programming example	Description
	Work directory is D:\WORK\
N10 G311 N5="test1\param.pa"	File from D:\WORK\TEST1\ is loaded
N20 G311 N5="\test2\param.pa"	File from D:\TEST2\ is loaded
N30 G311 N5="c:\test3\param.pa"	Error message
N40 G311 N5="z:\test4\param.pa"	**SP**: File from NFS-directory Z:\TEST4\ is loaded. **DP** and WinShape: File from windows network directory Z:\TEST4\ is loaded
N50 G311 N5="\\server1\test5\param.pa"	**SP**: Error message. **DP** and WinShape: File from windows network directory \\SERVER1\TEST5\ is loaded

24.4 Query functions

24.4.1 G318 Read pallet or job table data

Query pallet data or job table data.

Format

G318 I1=.. I2=.. I3=.. E...

```
G    Read pallet or job table data
E    E-parameter
I1=  1=Pallet manag. 2=Job admin.
I2=  Index number of table record
I3=  Table address 1-5=PQSP1L1/SFDRx
```

Possible function:

I1=1 Pallet management
- I2=.. Index number in pallet table. (PO.PO)
  - I3=1 Pallet number
  - I3=2 Priority
  - I3=3 Workpiece status (0= empty, 1=blank, 2=cutting, 3=ready, 4=reject)
  - I3=4 Pallet type
  - I3=5 Location type

I1=2 Job administration
- I2=.. Index number in job table. (JA.JA)
  - I3=1 Order size
  - I3=2 Finished products
  - I3=3 Defect products
  - I3=4 Blanks

Notes and usage

Reading of addresses without data

If the address not exist, the E-parameter contains the number -999999999.

Example Query job administration and storing the data in E-parameter 10.

N... G318 I1=2 I2=5 I3=2 E10 — I1=1 I2=5 I3=2 query of the number of finished products. E10 contains the number of finished products.

24.4.2 G319 Query actual technology data

Query active F (Feed), S (Speed), S1 (Cutting speed/rotational speed) or T (Tool number).

Format

G319 I1=.. E... {I2=..}

```
G    Read actual technology data
E    E-parameter
I1=  1-7 (F,S,T,S1,F1,F3,F4)
I2=  0=programmed, 1=actual
```

Possible function:

- I1=1 Feed (F)
- I1=2 Speed (S)
- I1=3 Tool number (T)

I1=4 Cutting speed/speed (S1=) (only turning)
I1=5 Constant cutting feed (F1= by G41/G42)
I1=6 In depth feed (Infeed F3=)
I1=7 In plane feed (F4=)

I2=0 Programmed value (default)
I2=1 Actual value.

Notes and usage
Reading of addresses without data
If the address not exist, the E-parameter contains the number -999999999.

Example query active feed and storing the data in E-parameter 10.
N... G319 I1=1 E10 I2=0 I1=1 query feed.
E10 then contains the value

24.4.3 G320 Query current G data

Query address value of current modal G function and save this value in the E parameter provided for this purpose.

Format
G320 I1=.. E...

```
G    Read actual G data
E    E-parameter
I1=  Selection number
```

Notes and usage
Defaults
All values are initialised when the machine is started. Most parameters are set on zero.

Reading active modal g-functions
G324 can be used to query whether a G function is active.
Particular information can always be queried with G320.

Result dimension
The unit of the result is mm or inches. Degrees for angles.

Selection number

I1=selection number	G-function	result min—max.	default
	G7 Tilting working plane		
1	Angle of rotation A-axis	-180--180°	0
2	Angle of rotation B-axis	-180--180°	0
3	Angle of rotation C-axis	-180--180°	0
	G8 Tilting tool orientation		
4	Angle of rotation A-axis	-180--180°	0
5	Angle of rotation B-axis	-180--180°	0
6	Angle of rotation C-axis	-180--180°	0

	G9 Defining pole position point		
7	Pole coordinate X-axis		0
8	Pole coordinate Y-axis		0
9	Pole coordinate Z-axis		0
	Result from G17, G18, G19, G180 and G182		
10	First main axis	1--6	
11	Second main axis	1--6	
12	Tool axis	1--3	
		1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C	
	G25 Feed- and speed override active		
13	Feed- and speed override active	0	
	G26 Feed- and speed override not active		
13	Feed- and speed override not active	1--3	
		1=F=100%, 2=S=100%, F und S=100%	
	G27 Positioning functions		
14	Feed movement (I3=)	0	
15	Rapid movement (I4=)	0	
16	Positioning logic (I5=0	0	
17	Acceleration reduction (I6=)	100%	
18	Contour tolerance (I7=0)	MC765	
	G28 Positioning functions		
14	Feed movement (I3=)	0--1	
15	Rapid movement (I4=)	0--1	
16	Positioning logic (I5=0	0--1	
17	Acceleration reduction (I6=)	5—100%	
18	Contour tolerance (I7=0)	0—10.000µm or MC765	
	G39 Activate tool offset		
19	Tool length offset (L)		0
20	Tool radius offset (R)		0
	G52 Palettes zero point shift		
21	Zero point shift in X-axis		0
22	Zero point shift in Y-axis -		0
23	Zero point shift in Z-axis -		0
24	Zero point shift in A-axis -		0
25	Zero point shift in B-axis -		0
26	Zero point shift in C-axis -		0
	G54 Standard zero point shift		
27	Zero point shift in X-axis -		0
28	Zero point shift in Y-axis -		0
29	Zero point shift in Z-axis -		0
30	Zero point shift in A-axis -		0
31	Zero point shift in B-axis -		0
32	Zero point shift in C-axis -		0
33	Angle of rotation		0

	G92/G93 incremental or absolute zero point shift	
34	Zero point shift in X-axis -	0
35	Zero point shift in Y-axis -	0
36	Zero point shift in Z-axis -	0
37	Zero point shift in A-axis -	0
38	Zero point shift in B-axis -	0
39	Zero point shift in C-axis -	0
40	Angle of rotation	0
	United zero point shift (G52 + G54 + G92/G93)	
41	Zero point shift in X-axis -	0
42	Zero point shift in Y-axis	0
43	Zero point shift in Z-axis -	0
44	Zero point shift in A-axis	0
45	Zero point shift in B-axis -	0
46	Zero point shift in C-axis -	0
47	Angle of rotation	0
	G72 Mirror image and scaling not active	
48	Scaling factor plane (A4=)	1
49	Scaling factor tool axis (A4=)	1
50	Mirror image in X-axis	1
51	Mirror image in Y-axis	1
52	Mirror image in Z-axis	1
53	Mirror image in A-axis	1
54	Mirror image in B-axis	1
55	Mirror image in C-axis	1
	G73 Mirror image and scaling active	
48	Scaling (factor or %) plane (A4=)	1
49	Scaling (factor or %) tool axis (A4=)	1
	MC714	0= Machining plane (factor) 1= Machining plane (percent eel) 2= all linear axes (factor) 3= all linear axes (percent)
50	Mirror image in X-axis	-1--1
51	Mirror image in Y-axis	-1--1
52	Mirror image in Z-axis	-1--1
53	Mirror image in A-axis	-1--1
54	Mirror image in B-axis	-1--1
55	Mirror image in C-axis	-1--1
	System axes number determinate by machine constants (MC103, MC105, etc.).	
56	First main axis	0--6 =not active, 1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C
57	Second main axis	0--6
58	Tool axis	0--6
59	First rotation axis	0--6
60	Second rotation axis	0--6
61	Third rotation axis	0--6
	Information of actual tools (Value is zero, when T0 is active or no value is given):	
62	Actual tool length	(L/L1=/L2= + L4= + G39 L)
63	Actual tool radius	(R/R1=/R2= + R4= + G39 R)
64	Actual tool corner radius	(C)
65	Actual tool orientation	(O or G302 O)

Actual spindle position angle after tool head rotation (G7 or manual)
66 Projected actual spindle position angle on the XY-plane after automatic (G7) or manual tool head rotation.

G106 and G108 Kinematics calculations
67 Total shift in X (Rotary axis position - compensation + Kin. compensation - MC3x14,
- without programmable offsets
G108-offsets (head and table)
IPLC-offsets
68 Total shift in Y
69 Total shift in Z
70 Value from I1= address from G108
0 = G106 active
1 = G108 active (in the head and possibly in the table)

G153 und G154 work piece zero point tracking
71 Programmed status
0 = G153
1 = G154

G125 and G126 Programmed tool lifting
72 Programmed status
0 = G125
1 = PLC (G126 I1=1)
2 = INT (G126 I2=1)
3 = PLC + INT (G126 I1=1 I2=1)
4 = ERR (G126 I3=1)
5 = PLC + ERR (G126 I1=1 I3=1)
6 = INT + ERR (G126 I1=1 I3=1)
7 = PLC + INT + ERR (G126 I1=1 I2=1 I3=1)
73 Programmed distance

Kinematic position of the rotary axis
74 Returns the kinematic position of the A-rotary axis
75 Returns the kinematic position of the B-rotary axis
76 Returns the kinematic position of the C-rotary axis
- 0 = not present
- 10 = controlled axis in the tool head
- 11 = controlled axis 45° in the tool head
- 12 = manual axis in the tool head (MC501 = 10n)
- 13 = manual axis 45° in the tool head (MC501 = 10n)
- 14 = swivel axis in the tool head (MC501 = 20n)
- 15 = swivel axis 45° in the tool head (MC501 = 20n)
- 20 = controlled axis in the work piece table
- 21 = controlled axis 45° in the work piece table
- 22 = manual axis in the work piece table (MC501 = 10n)
- 22 = manual axis 45° in the work piece table (MC501 = 10n)
- 23 = swivel axis in the work piece table (MC501 = 20n)
- 23 = swivel axis 45° in the work piece table (MC501 = 20n)

Software endswitch
77 returns the distance to the positive SW-endswitch in X
78 returns the distance to the positive SW-endswitch in Y
79 returns the distance to the positive SW-endswitch in Z
80 returns the distance to the negative SW-endswitch in X
81 returns the distance to the negative SW-endswitch in Y
82 returns the distance to the negative SW-endswitch in Z

	G106 and G108	Kinematic calculations
83	G108 Offset in the X-axis	
84	G108 Offset in the Y-axis	
85	G108 Offset in the Z-axis	
	G153 and G154	work piece zero point tracking
86	G154 Offset in the X-axis	
87	G154 Offset in the Y-axis	
88	G154 Offset in the Z-axis	
	G218 activate angular head:	
89	G218 Offset in the X-axis	
90	G218 Offset in the Y-axis	
91	G218 Offset in the Z-axis	
92	G218 Rotation (space angle) in the A-direction	
93	G218 Rotation (space angle) in the B-direction	
94	G218 Rotation (space angle) in the C-direction	

Example Query of Address of G-function (I1=) and store of the value in E-parameter 10.

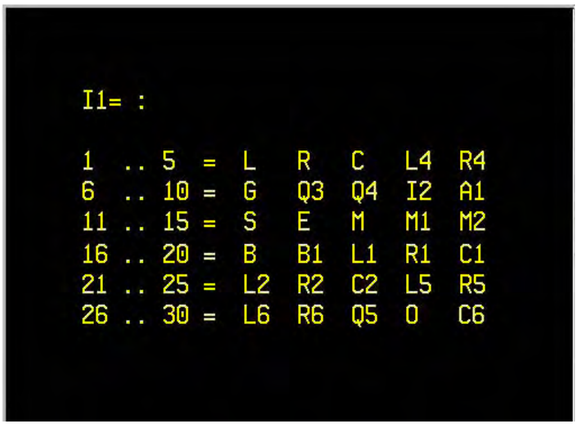
Programmbeispiel	Beschreibung
N11 G320 I1=10 E11	I1=10 Query first main axis E11 contains the result E11=1 X-axis is first main axis.
N12 G320 I1=11 E12	I1=11 Query second main axis E12 contains the result E12=2 Y-axis is second main axis.
N13 G320 I1=12 E13	I1=12 Query tool axis E13 contains the result E13=3 Z-axis is tool axis.

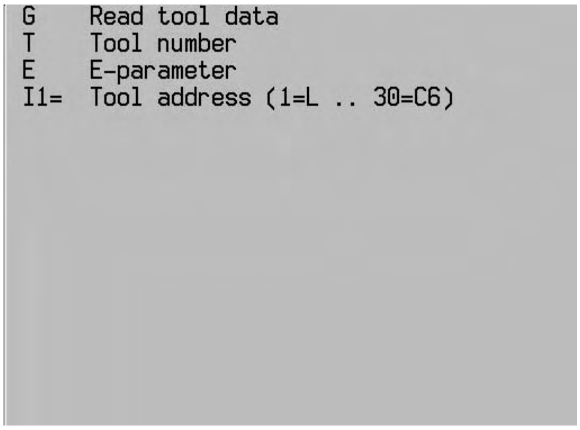
24.4.4 G321 Query tool data

Query tool table.

Format

G321 T.. I1=.. E...





Notes and usage

Tool number and position

The Tool number (T) must be known. The position (P) in the tool table cannot be queried.

Reading of the tool table values without data

If The E-Parameter contains the number -999999999, the address in the tool table is empty.

Classification

I1=1	L	Length
I1=2	R	Radius
I1=3	C	Corner radius
I1=4	L4=	Length oversize
I1=5	R4=	Radius oversize
I1=6	G	Graphics
I1=7	Q3=	Type
I1=8	Q4=	Number of cutting edges
I1=9	I2=	Cutting direction
I1=10	A1=	Approach angle
I1=11	S	Size
I1=12	E	Status
I1=13	M	Initial tool life
I1=14	M1=	Actual tool life
I1=15	M2=	Tool life monitoring
I1=16	B	Breakage tolerance
I1=17	B1=	Breakage monitoring
I1=18	L1=	First extra length
I1=19	R1=	First extra radius
I1=20	C1=	First extra corner radius
I1=21	L2=	Second extra length
I1=22	R2=	Second extra radius
I1=23	C2=	Second extra corner radius
I1=24	L5=	Wear tolerance length
I1=25	R5=	Wear tolerance radius
I1=26	L6=	Offset length
I1=27	R6=	Offset radius
I1=28	Q5=	Breakage monitoring cycle (0-9999)
I1=29	O	Tool orientation (only turning)

Example Program queries the tool table.

N30 G321 T10 I1=1 E1	G321 Read command T (tool number) I1= Information about the tool address E1 is E-parameter L (tool length) is set in E-parameter 1
N40 G321 T10 I1=2 E10	R (tool radius) is set in E-parameter 10
N50 G321 T10 I1=3 E20	C (corner radius) is set in E-parameter 20 (If C has no value, E20=-999999999 is set)
N60 G321 T10 I1=4 E2	L4 (length oversize) is set in E-parameter 2
N70 G321 T10 I1=5 E11	R4 (radius oversize) is set in E-parameter 11
N80 E3=E1+E2	The correct tool length (E3) is L+L4 (E1+E2)
N90 E12=E10+E11	The correct tool radius (E12) is R+R4 (E10+E11)

24.4.5 G322 Query machine constant memory

To read out a machine constant value and store it in the appropriate E-parameters.

Format

G322 E.. N1=...

```
G    Read machine constant memory
E    E-parameter
N1=  Machine constant number
```

Notes and usage

Reading out a machine constant without value

When invisible addresses are read from the machine constant table, the E-parameter remains unchanged.

Examples Universal program blocks, which can be used for both zero point, table types.

N50 G322 N1=84 E10	Machine constant 84 is set in E10
N60 G29 E1 N=90 E1=E10>0	Compare if MC84 > 0. If so, jump to N90
N70 G150 N1=57 X7=E1 Z7=E6	Store the zero point shift table ZO.ZO
N80 G29 E1 N=100 E1=1	Jump to N100
N90 G150 N1=54.3 X7=E1 Z7=E6	Store the zero point shift table ZE.ZE
N100 ..	

24.4.6 G324 Query G-group

Query current **modal** G-function and stores with this value in the E-Parameters preprogrammed for this purpose.

Format

G324 I1=.. E...

```
G    Read G-group
E    E-parameter
I1=  G-group (1,2,etc.)
```

Notes and usage

Read out of group without data

If the group or the G-function not exists, the E-parameter is unchanged.

Group classification

I1=	G-function
1	G0, G1, G2, G3, G6, G9
2	G17, G18, G19
3	G40, G41, G42, G43, G44, G141
4	G53, G54, G54_I, G55, G56, G57, G58, G59
5	G64, G63
6	off, G81, G83, G84, G85, G86, G87, G88, G89, G98
7	G70, G71
8	G90, G91
10	G94, G95
11	G96, G97 (only turning)
12	G36, G37 (only turning)
13	G72, G73
14	G66, G67
15	off, G39
16	G51, G52
17	G196, G199
19	G27, G28
20	G25, G26, G26_S, G26_F_S
21	off, G9
22	G202, G201
24	G180, G182, G180_XZC
27	off, G7
28	off, G8
29	G106, G108

Result

In general is the result equal to the value of the modal G-function.

For example: G324 I1=3 gives, when G40 is active, as result the value 40.

Exceptions are:

Off gives value 0. G26_S, G26_F_S gives 26.

G54_I gives 54.nn, where nn is the index. G180_XYZ gives 180.

Example selection of the G-function (I1=2) and storage of the value in E-parameter 10.

N... G324 I1=2 E10 I2=2: query group 2 G-function

E10 holds the result

E10 =17 G17 is active.

24.4.7 G325 Query M group

Query current modal M-function and store this value in the E-Parameter pre-programmed for this purpose.

Format

G325 I1=.. E...

```
G    Read M-group
E    E-parameter
I1=  M-group (1,2,etc.)
```

Notes and usage

Read out of group without data

If the group or the M-function does not exist, the E-parameter is unchanged.

Meaning M-functions

Some of these M-functions are basis M-functions and are described in the paragraph "M-functions" of chapter "Technological instructions". The other are machine dependent M-functions. Please refer to the machine builder handbook for a description.

Combined M-functions (M13 and M14)

M13 and M14 are combined M-functions. (M13=M3 + M8). These functions are determinate by two blocks.

N... G325 I1=1 E10.
N... G325 I1=3 E11
When E10=3 and E11=8, than M13 is active.

Group classification

Group I1=	M-function
1	off, M5, M3, M4, M19
2	off, M40, M41, M42, M43, M44
3	M9, M7, M8
4	off, M17, M18, M19
5	off, M10, M11
6	off, M22, M23
7	off, M32, M33
8	off, M55
9	off, M51, M52
10	off, M53, M54
11	off, M56, M57, M58
12	off, M72, M73
13	off, M1=..

Result

In general is the result equal to the value of the modal M-function.

For example: G324 I1=2 gives, when M40 is active, as result the value 40.

Exceptions are: Off gives value 0.

Example: selection of the M-function (I1=1) and storage of its value in E-parameter 10.

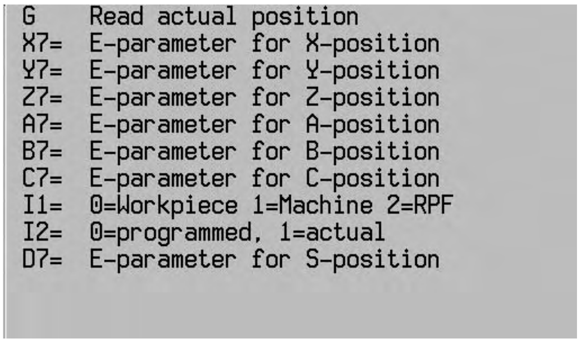
N... G325 I1=1 E10 — I2=1: query group 1 M-function
E10 holds the result
E10 =5 M5 is active.

24.4.8 G326 Query actual position

To read out the actual axes-positions values and store it in the appropriate E-parameters.

Format

G326 {X7=..} {Y7=..} {Z7=..} {A7=..} {B7=..} {C7=..} {D7=..} {I1=..} {I2=..}



I1=	0	Position to work piece zero point (Default)
	1	Position to machine zero point
	2	Position to reference point
	3	Total zero point shift (without IPLC shift).
I2=	0	Programmed value (default)
	1	Current value

Notes and usage

Reading out of not existent axes

When an axis not exist the contents of the E-parameter is filled with -999999999.

Reading out by graphical simulation

By graphical simulation only the X, Y and Z can be read out. The E-parameters for the rotating axes stays zero.

Reading out of spindle position (D7=):

When I1=0 is, is the result, the programmed spindle position of M19 or the programmed spindle position in G700.

Examples

Example 1: Read out actual axes-position von X, Y and Z and store the values in E-parameters 20, 21 and 22.

N... G326 X7=20 Y7=21 Z7=22	E20 contains the actual X-axis-position.

Example 2: Program continuation after a universal pocket cycle.

N30 G202	End pocket cycle
N40 G326 X7=20 Y7=21	Unknown actual End-position von X and Y
N50 G29 E1 N=90 E1=E20>100	Actual X-position >100, then jump to N90
N60 G29 E1 N=90 E1=E20<-100	Actual X-position <-100, then jump to N90
N70 G0 X-110	G0 movement to X-110, if the actual X-position is situated between 100 and -100. On this manner for example an obstacle can be rounded.
N80 G0 Y 100	Further turn aside movement

24.4.9 G327 Query operation mode

To scan the current operating mode and store this value in the E parameter provided.

Format

G327 I1=.. E...

```
G    Read operation mode
E    E-parameter
I1=  Active mode (1-6)
```

Notes and usage

Arrangement of group

Group

I1=	Operating mode	
1	EASYoperate	0 = not active, 1=active
2	Single record	0 = not active, 1=active
3	Graphic	0 = not active, 1=active
4	Test run	0 = not active, 1=active
5	Search	0 = not active, 1=active
6	Demo	0 = not active, 1=active

Example Fetch operating mode (I1=1) and store the value in E parameter 10.

N... G327 I1=1 E10 — I1=1: Check whether EASYoperate is active.
E10 contains the result: 0= not active, 1=active.

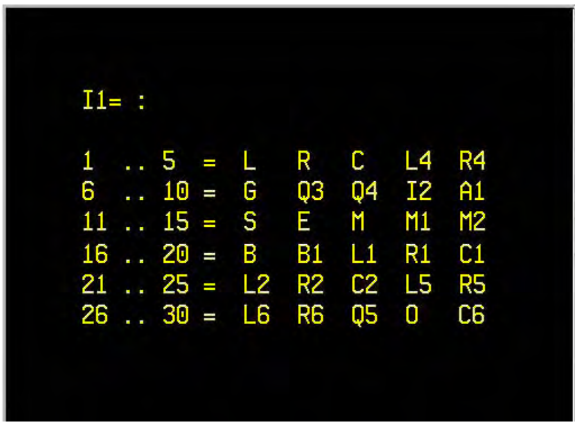
24.5 Write functions

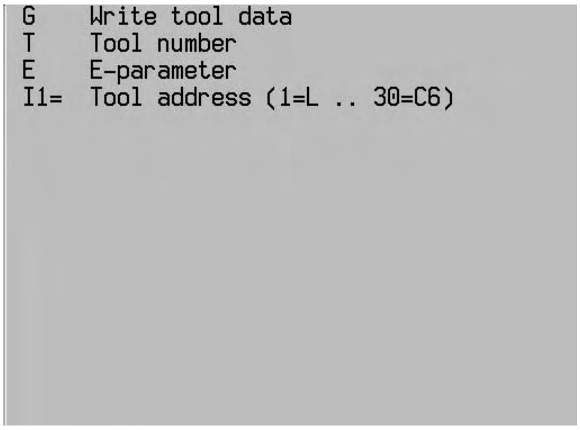
24.5.1 G331 Write tool data

Write from values in the tool table.

Format

G331 T.. I1=.. E...





Notes and usage

Tool number and position

The tool number (T) must be known. The position (P) in the tool table cannot be changed.

Writing in the tool table without data

If the E-parameter contains the value -999999999, the address in the tool table becomes empty.

New information activating

The changed tool information must be activated again following the writing. (T.. M67)

Classification

I1=1	L	Length
I1=2	R	Radius
I1=3	C	Corner radius
I1=4	L4=	Length oversize
I1=5	R4=	Radius oversize
I1=6	G	Graphics
I1=7	Q3=	Type
I1=8	Q4=	Number of cutting edges
I1=9	I2=	Cutting direction
I1=10	A1=	Approach angle
I1=11	S	Size
I1=12	E	Status
I1=13	M	Initial tool life
I1=14	M1=	Actual tool life
I1=15	M2=	Tool life monitoring
I1=16	B	Breakage tolerance
I1=17	B1=	Breakage monitoring
I1=18	L1=	First extra length
I1=19	R1=	First extra radius
I1=20	C1=	First extra corner radius
I1=21	L2=	Second extra length
I1=22	R2=	Second extra radius

I1=23	C2=	Second extra corner radius
I1=24	L5=	Wear tolerance length
I1=25	R5=	Wear tolerance radius
I1=26	L6=	Offset length
I1=27	R6=	Offset radius
I1=28	Q5=	Breakage monitoring cycle (0-9999)
I1=29	O	Tool orientation (only turning)

The tool commentary cannot be changed.

Example

N10 E5=100 (Tool length)	L (tool length) is set in E-parameter 5
N11 E6=10 (Tool radius)	R (tool radius) is set in E-parameter 6
N12 E7=-999999999 (Tool corner radius)	C (tool corner radius) will be stored in E-parameter 7 (If C has no value, E7= must be set to -999999999)
N13 E8=0 (Length oversize)	L4 (length offset) is set in E-parameter 8
N14 E9=0 (Radius oversize)	R4 (radius offset) is set in E-Parameter 9
N..	
N20 G331 T10 I1=1 E5	L (tool length) writing of E-parameter 5 in the tool table
N21 G331 T10 I1=2 E6	R (tool radius) writing of E-parameter 6 in the tool table
N22 G331 T10 I1=3 E7	C (tool corner radius) writing of E-parameter 7 in the tool table
N23 G331 T10 I1=4 E8	L4 (length offset) writing of E-parameter 8 in the tool table
N24 G331 T10 I1=5 E9	R4 (radius offset) writing of E-parameter 9 in the tool table
N30 T10 M67	The tool must be activated once more with the changed information.
N..	
N40 E8=0.3 (Length oversize)	L4 (length offset) E-parameter 8 is set to 0.3
N41 G331 T10 I1=4 E8	L4 (length offset) writing of E-parameter 8 in the tool table
N50 T10 M67	Tool must be activated once more with the changed information.

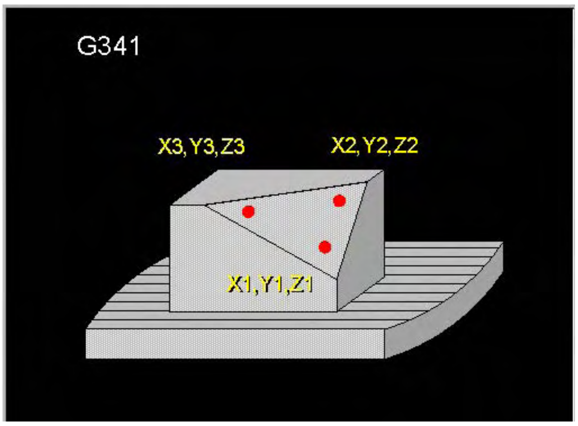
24.6 Calculation functions

24.6.1 G341 Calculation of G7-plane angles

G341 is used to calculate the solid angles A5=, B5= and C5= from 3 defined points. These angles are used in G7 to set up the plane.

Format

G321 {X1=.. Y1=.. Z1=.. X2=.. Y2=.. Z2=.. X3=.. Y3=.. Z3=..} O1=.. O2=.. O3=..



G	Calculation of G7 plane angles
X1=	E-parameter number of plane point
Y1=	E-parameter number of plane point
Z1=	E-parameter number of plane point
X2=	E-parameter number of plane point
Y2=	E-parameter number of plane point
Z2=	E-parameter number of plane point
X3=	E-parameter number of plane point
Y3=	E-parameter number of plane point
Z3=	E-parameter number of plane point
O1=	E-parameter number plane angle A5
O2=	E-parameter number plane angle B5
O3=	E-parameter number plane angle C5

Notes and usage

X1= to Z3= are E parameter numbers with axis position values of 3 points that define the machining plane [mm or inches]. If one of these addresses X1= to Z3= is programmed, all the addresses must be programmed. The 3 points do not have to be identical, nor do they need to be in a line. If the E parameters are not entered, G341 calculates A5=, B5= and C5= from the rotated plane that is set.

O1= to O3= are the numbers of the E parameters where the calculated solid angles A5=, B5= and C5= are stored [in degrees]. O1=, O2= and O3= must be programmed.

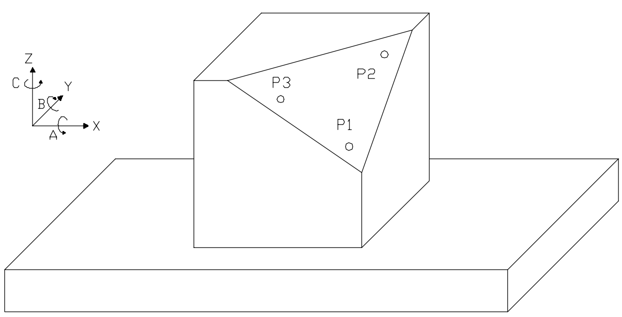
If G7 or G8 is active the input values must be defined in the active co-ordinate system.

G341 is not allowed if G19 is active.

Note

If the G341 inputs are determined in G7, G8, G17, or G18, the calculation by G341 must be carried out in the same mode.

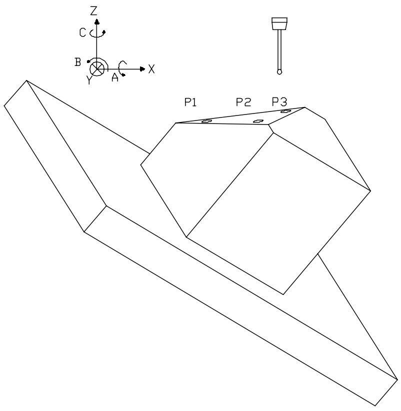
Example: Flattening an oblique face.



Therefore the oblique face must be defined by 3 points: (P1 (X,Y,Z), P2 (X,Y,Z)and P3 (X,Y,Z)). Because the face is too oblique to get accurate measure points, first the workpiece is turned until the oblique face has approximately been flattened (the round axes have been jogged and are not equal to zero anymore).

Next, the 3 points are determined with a measure probe and are saved in E-parameters E10 up to and including E18:

P1 (X, Y, Z) = E10, E11 and E12
P2 (X, Y, Z) = E13, E14 and E15
P3 (X, Y, Z) = E16, E17 and E18

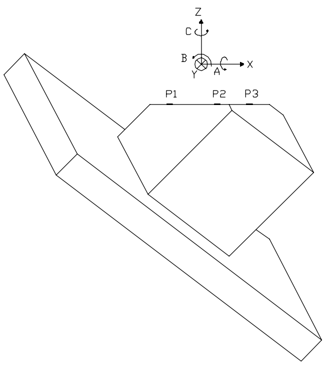


Next, G341 determines the round axes positions, which can be used by G7 to flatten the oblique face. The round axes positions are written in E-parameters E20, E21 and E22.

G341 X1=10 Y1=11 ……Z2=17 Z3=18 O1=20 O2=21 O3=22

Finally the oblique face is flattened by G7:

G7 A5=E20 B5=E21 C5=E22



24.7 Formatted write functions

24.7.1 Introduction formatted write functions:

The formatted write function, can be used for:
- to write to the screen
- to write to the file on the hard disk

Configuration file to define a file or window (display/input).

Configuration files are required to describe how and where to write.
These configuration files are saved on the hard disk:
D:\STARTUP\CYCLES\FORMnnnn.CFG.
nnnn is the file number from 1 to 8999.

Configuration files are activated when the system is started.
End users can define files themselves.
The file size is unlimited.

Description of configuration file:

```
:Commentary starts with a ';'
;
;Sections:
Only for one window:
;[window]                           defines present window
;number=       windowId             where windowId = 1…4   See G350

;[file]                             defines file (only for G351')
;name   =      file name            where filename is 8.3 ASCII characters
;                                   The directory is always D:\STARTUP\

;[string]                           defines position and content of the block
;line     =    line number          where line number = [1|...|n]   basic setting = 1
;position=     position number      where   position number = [1|...|n]   basic setting = 1
;gb       =    "string"             where block is <n> ASCII characters
;d        =    "string"
;                                   Texts are defined for various languages
;                                   Code gb=, d=, f=

;                                   dependency condition (IF)
conditioneparam= E-Parameter number [1|...|MC83]       (e.g. 240)
conditionvalue   = Values                              (e.g. 3)
```

When the 'conditioneparam' (E240) has a 'conditionvalue' of 3, this instruction is executed. In this case the text "string" is written in the window or file.

```
;
;[value]                            defines position, format and E parameter of the value
;line     =    line number
;position=     position number
;eparam=       E parameter          where E parameter number = [1|...|MC83]
;form     =    Determines the input format (default 6.3). 6.3 means: 6 figures before the decimal
               point and 3 after.
```

When the address dimension [mm], [degr], [mm/min] or [diam] is, the number of digits behind the decimal point depends of MC705 and MC707.
MC705 (Decimal digits behind the decimal point) is 3 or 4. The number or digits before and after the decimal point will be adapted.

MC707 (Inch/Metric). is 70 (metric) or 71 (Inch). When MC707=71 the number of digits behind the decimal point will be increased by one and the number of digits before the decimal point will de decreased by one.

Overview:	Metric		Inch	
MC707	71	71	70	70
MC705	3	4	3	4
Dimensions				
[mm] Linear axis	6.3	5.4	5.4	4.5
[degr] Rotation axis	6.3	5.4	6.3	5.4
[mm/min] Feed	6.3	6.3	5.4	5.4
[diam] Diameter programming in mm	6.3	5.4	5.4	4.5

;dimension= Only [mm], [degr], [mm/min] and [diam] are allowed. Addresses with these dimensions are depending of MC705 and MC707.

[mm] mm for linear axis
[degr] Degree for rotation axis
[mm/min] mm pro minute for feed
[diam] Diameter programming in mm
Default: no dimension

;sign = yesNo where yesNo = y = space for sign
; n = no space for sign

; Dependency condition (IF)
conditioneparam= E-Parameter number [1|...|MC83] (e.g. 240)
conditionvalue = Values (e.g. 3)
When the 'conditioneparam' (E240) has a 'conditionvalue' of 3, this instruction is executed. In this case the text "string" is written in the window or file.
;
Only for input window:
;[input] defines position, format and E parameter before an input field
; only for G350 and windowId = 2
; only one [input] section is allowed
;line = line number
;position= position number
;eparam= E parameter number where E parameter number = [1|...|MC83]
;form = digitDecimal where digitDecimal = <digits>.<decimals>
;sign = yesNo where yesNo = y =space for sign
; n = no space for sign

24.7.2 G350 Writing to a window

Specific lines and values can be written to a window using E parameters and a configuration file. In addition, a particular input can be expected. For unbalance detection, the result can be displayed to the operator in this way.

Format

G350 N1=.. {I1=...} {I2=...}

```
G    Write to window
N1=  Configuration file number
I1=  Window (0=closed, 1=open)
I2=  Window (0=no interv, 1=interv)
```

N1= Defines the configuration file 'D:\STARTUP\CYCLES'\FORMnnnn.CFG> that is used for the format, lines and E parameters that are written. File number between 1 and 8999.

I1= 0 = window not visible. Setting on switch-on:
1 = window visible.

I2= 0 = Program do not stop.
1 = Program stops like “intervention“ and waits for <Start>

Notes and application

G350 can be used to make a previously defined window visible. The texts in the window are fixed, and the values are continuously updated according to the defined E parameters.
When I2=1 is programmed, the program waits until <Start> is pressed. Only one entry window can be active at any one time.

4 windows are defined:

Number	Window type	Mode	Position	Size
1	Display	Manual Automatic	Right side of screen Top 'Dashboard'	15 lines, 37 characters
2	Input	Manual Automatic	Right side of screen Top 'Dashboard'	5 lines, 37 characters
3	Graphics	Manual Automatic	Left side of screen Top 'Dashboard'	
4	Display	Manual Automatic	Left side of screen Top 'Dashboard'	15 lines, 37 characters

The window also appears in graphics, but not during block search.
The window becomes invisible following M30 and <Cancel program>.

24.7.2.1 Writing to a window

N1 E11=45 Hole number
N2 E12=6 Number
N10.. G350 N1=3501 I1=1 I2=1 Write to window
File D:\STARTUP\CYCLES\FORM3501.CFG is used

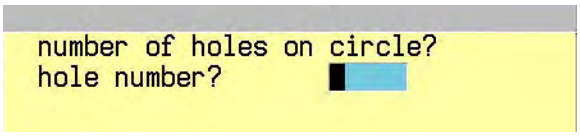
```
 Drilling pattern
----------------------------
 Maximum number of holes  45
 hole number               6
----------------------------
```

Display window configuration file

```
;FORM3501.CFG

[Window]
number      = 1          ;Uses window number 1 of the available windows.

[string]
line        = 2
gb          = "drilling pattern"

 [string]
line        = 4
position    = 1
gb          = "Maximum number of holes"

[value]
line        = 4
position    = 27         ;Print value in field at position 8 and onwards
eparam      = 11         ;E parameter E300 is given the value
form         = 3.0       ;format 3 figures and 0 decimals
sign         = n         ;No space reserved for sign
 [string]
line        = 5
position    = 1
gb          = "Actual hole number"

[value]
line        = 5
position    = 27         ; Print value at position 27 and onwards
eparam      = 12
form        = 3.0
sign        = n
```

24.7.2.2 Writing to a window and asking for information

```
N10.. G350 N1=3502 I1=1       Write to window
                              File D:\STARTUP\CYCLES\FORM3502.CFG is used
```



Display window configuration file

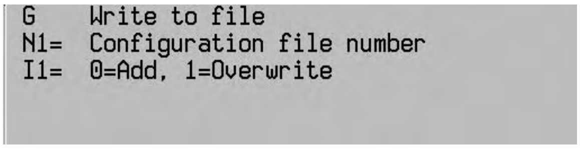
```
;FORM3502.CFG
[window]
number      = 2    ; Uses window number 2 of the available windows.
[string]
line        = 1
position    = 1
gb          = "number of holes on circle"
[string]
line        = 2
position    = 1
gb          = "number of holes"
[input]
eparam      = 10         ;E parameter E10 contains an input value received the operator input
form        = 3.0        ;format 3 figures and 0 decimals
sign        = n          ; No space reserved for sign
```

24.7.3 G351 Writing to a file

Specific lines and values can be written to a text file in D:\Startup\ using E parameters and a configuration file. This can be used to create the calibration curves for unbalance detection.

Format

G351 N1=.. {I1=...}



N1= Defines the configuration file <'Directory'\FORMnnnn.CFG> that is used for the format, lines and E parameters that are written. File number between 1 and 8999.
The directory can be any 'Cycle Design' directory.
The configuration file is the same as for writing to a window, but ‘section’ [window] and [input] are ignored.

I1= States whether the data is to be inserted at the end of an existing file or whether a file that may exist is to be overwritten. Basic setting <0> for insertion.

Notes and application

G351 writes the lines and values of the configuration file and E parameters to the hard disk.
A maximum of 50 lines of 255 characters can be written at the same time.
The file is not written during graphics or block search.

Example Listing measurement data and writing to a file.
The radius of a pocket is measured in the program

The following data available in the E parameters must be listed:

N10 (measurement programmed in blocks N12 to N16)
N11 (in this case as example of just the results from e.g. measurement cycle G145)

N12 E50=34.1	(setpoint)	(entered)
N13 E51=34.05	(lower tolerance)	(entered)
N14 E52=34.15	(upper tolerance)	(entered)
N15 E53=34.108	(actual value)	(measured)
N16 E54=0.008	(difference)	(calculated)

N20 G351 N1=0002 I1=0 (write file)
File D:\STARTUP\CYCLES\FORM0002.CFG is used.
I1=0 is insert

The file messdat.txt is:

```
Radius
Setpoint =          34.1
Lower tolerance =34.5
Upper tolerance =34.5
Actual value =     34.108
Difference =          0.008
****************************
```

Configuration file for listing measurement data

FORM0002.CFG

```
;*****************************************************************
; CFG file for writing measurement data
;*****************************************************************

;---- Name of file to be written to startup\ --------
[file]
name            = Messdat.txt

;---- Type of measurement ------------------------------
[string]
line            = 1
position        = 1
d               = Radius

;---- Setpoint --------------------------
[string]
line            = 2
position        = 1
d               = Sollwert =

[value]
line            = 2
position        = 20
eparam          = 50
form            = 6.3
dimension       = mm
sign            = y

;---- Lower tolerance --------------------------
[string]
line            = 3
position        = 1
d               = Untere Toleranz =

[value]
line            = 3
position        = 20
eparam          = 51
form            = 6.3
dimension       = mm
sign            = y

;---- Upper tolerance --------------------------
[string]
line            = 4
position        = 1
d               = Obere toleranz =

[value]
line            = 4
position        = 20
eparam          = 52
form            = 6.3
dimension       = mm
sign            = y
```

```
;---- Actual value --------------------------
[string]
line            = 5
position        = 1
d               = Istwert =

[value]
line            = 5
position        = 20
eparam          = 53
form            = 6.3
dimension       = mm
sign            = y

;---- Difference --------------------------
[string]
line            = 6
position        = 1
d               = Differenz =

[value]
line            = 6
position        = 20
eparam          = 54
form            = 6.3
dimension       = mm
sign            = y

;---------------------------------------
[string]
line            = 7
d               = ***************************************************
```

24.8 Array functions

24.8.1 Introduction to array functions:

Array functions can be used for handling two-dimensional numeric arrays. An array exists of rows and columns. A row number and a column number define an element (a value).

These functions allow you to interact with and manipulate arrays in various ways. Arrays are essential for storing, managing, and operating on sets of (a big number of) variables. For example: Storing of a great number of measuring positions and calculating of a centre position.

The advantage of the new features of the array functions is:
- Make it easier to define the format of two-dimensional arrays.
- Load array data from hard disk directly into CNC memory (during a program-run).
- Store complete array data on hard disk in one storing procedure and not one value after another.
- Manipulate array data directly in CNC memory.
- Store array data in a clear table format, so it can be easily examined.

Automatic Array Deletion

All arrays in memory, except the arrays which are loaded during start-up, will be deleted automatically from memory after: <Clear control>, <Cancel program>, <Cancel block> during EASYoperate, M30 and CNC system restart.

24.8.2 Overview array functions:

Function	Description	Source	Target
arrayNew()	Create a two-dimensional array in memory.		M
arraySave()	Store an array from memory on hard disk.	M	HD
arrayOpen()	Load an array from hard disk into memory.	HD	M
arrayExist()	Test the existence of an array on hard disk or in memory.	HD / M	
arraySize()	Determine the number of rows or columns in an array.	M	
ArrayFind()	Find data in an array.	M	
arrayWrite()	Add data to an array.		M
arrayRead()	Extract data from an array.	M	
arrayFilter()	Filter an array.	HD / M	M
arraySort()	Sort an array by column.	HD / M	M
arrayDelete()	Delete an array	HD / M	

Remarks:
- The third and fourth column describes the place where an array is stored (‘HD’ = hard disk, ‘M’ = memory).
- Several array functions are operating with arrays both on hard disk as well as in memory. Furthermore, due to a large amount of array data, it might be necessary to manipulate arrays directly on hard disk instead of loading the source data in memory first.
- The return values are stored in an E-parameter. E.g. E10=arrayExist( ).

24.8.2.1 arrayNew (format)

The goal of the function arrayNew() is to create a two-dimensional array in the memory of the CNC system.

<format> column names or number of columns
If an array with column names must be created, these column names must be programmed between double quotes and delimited by the symbol '|'. If no column names are required, <format> must be programmed as a number.
The length of each column name makes up the column width.

Returns: 0 if the array is not created.
nnn an internal array identification number is given, when the array is created.
E.g.: 1= first array, 2=second array, etc.

Example
This example creates an array in memory for tool data. The array contains three columns with the column names 'Tool', 'Length' and 'Radius'.

```
N1 E10=arrayNew(" Tool | Length | Radius ")
```

The return value (internal array identification number) is e.g. E10=xxx.

This example creates an array in memory, that contains three columns and no column names.

```
N1 E10=arrayNew(3)
```

24.8.2.2 arraySave (filename, internal array identification number)

The goal of the function arraySave() is to store an array from CNC memory on hard disk.

<filename> array name on hard disk.
The filename must be programmed between double quotes.

< internal array identification number > array name in CNC memory.
The array name must be programmed as a number or as an E-parameter (return value of arrayNew() or arrayOpen()).

Note: If the array <filename> already exists on hard disk, the contents of this array is overwritten.

Returns: 0 if the array is not save on hard disk.
1 if the array is saved.

Format on hard disk.
The array written to the hard disk has the following format. This file can be edit with the editor.
For example an array with 3 columns. Each information is separated by"|".

```
[BEGIN]
 Tool |Length.|Radius |
     1|   20.7|      5|
     2|    2.3|    5.7|
    10|   35,3|    5.8|
[END]
```

Example
This example saves an array file with tool data and with machine data.

```
N1 E1=xxx        internal array identification number from arraynew
N2 E10=arraySave("\Work\Tool.arr", E1)
N3 E11=arraySave("\Work\Machine.arr", 2)
```

24.8.2.3 arrayOpen (filename)

The goal of the function arrayOpen() is to load an array from hard disk into the memory of the CNC system.

<filename> array name on hard disk (entered between double quotes).

Returns: 0 if the array is not opened.
nnn The array is loaded in memory.
nnn is the unique internal identification number of the array (arrayNew).

Example

The following example opens an array file with tool data and with machine data. If these files are opened and successfully loaded, then arrays are created in memory.

N1 E10=arrayOpen(“\Work\Tool.arr”)
The return value (internal array identification number) is for example E10=xxx.
N2 E11=arrayOpen(“\Work\Machine.arr”)

24.8.2.4 arrayExist (name)

The goal of the function arrayExist() is to test the existence of an array on hard disk or in CNC memory.

<name> array name on hard disk or in memory.
hard disk: string (between double quotes).
memory: number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

Returns: 0 if the array does not exist.
1 if the array exists.

Example

This example tests the existence of the array file ‘Tool.arr’ on hard disk.

N1 E10=arrayExist(“\Work\Tool.arr”)

This example tests the existence of two arrays in memory.

N1 E1=9700 (internal array identification number)
N2 E10=arrayExist(E1)
N3 E11=arrayExist(9701)

24.8.2.5 arraySize (internal array identification number, rowcol)

The goal of the function arraySize() is to return the number of rows or columns in an array.

<internal array identification number > array name in memory.
number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).
<rowcol> 1=determine the number of rows
2=determine the number of columns.

Note: The number of rows in the array <name> equals
- The highest row number of a non-empty row, if this row is written by arrayWrite().
- The number of rows, if these rows are written by arrayOpen(), arraySort() or arrayFilter().

Returns: The number of rows in the array <name> if <rowcol> equals ‘1’.
The number of columns in the array <name> if <rowcol> equals ‘2’.

Example

This example determines the number of columns in the array in memory.

N1 E10=arrayOpen(“\Work\Tool.arr”)
N2 E11=arraySize(E10, 2)

24.8.2.6 arrayFind (internal array identification number, column, value)

The goal of the function arrayFind() is to return the number of the row in which the first occurrence of a value is found.
< internal array identification number >   array name in memory.
<column>   column number.
<value>   value that must be found.

Returns
The row number in which the value <value> is found. If this value is not found in the programmed column, then the value ‘0’ must be returned.

Example
The following array is stored in memory with internal array identification number stored in E40.

Id	Unbalance	Speed	Amplitude
10	100,000	25	0.00345
11	100,000	50	0.00862
20	200,000	25	0.00710
21	200,000	50	0.01992

N8 E41=arrayFind(E40, 1, 20)   Find value= 20 in column= 1. The result E41= 3.

Remark: With arrayFilter an array with the desired value can be generated. On this manner the next row can be found.

24.8.2.7 arrayWrite (internal array identification number, row, column, value)

The goal of the function arrayWrite() is to add data to an array in CNC memory.
<internal array identification number >   array name in memory.
number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).
<row>   row number.
<column>   column number.
<value>   value to be written in the array.
The array element(<row>,<column>) will be made empty,. If the <value> is programmed as ‘-999999999’

Returns: 0   if the value is not written in the array.
1   if the value is written.

Example

Tool	Length	Radius
1	20.7	5
2	42.3	5.7
10	35.5	5.8

This example loads the array in memory and after that it adds a complete new row to this array in memory.

```
N1 E10=arrayOpen("\Work\Tool.arr")      E10= internal array identification number
N2 E20=arrayWrite(E10, 4, 1, 11)
N3 E21=arrayWrite(E10, 4, 2, 46.0)
N4 E22=arrayWrite(E10, 4, 3, 10.6)
```

Tool	Length	Radius
1	20.7	5
2	42.3	5.7
10	35.5	5.8
11	46.0	10.6

Note that the changed array must be saved to harddisk with arraySave.

24.8.2.8 arrayRead (internal array identification number, row, column)

The goal of the function arrayRead() is to extract data from an array in CNC memory and store it in an E-parameter.

< internal array identification number > array name in memory.

number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

<row> row number.

<column> column number.

Returns

The value in array element(<row>,<column>). If this element in the array is empty, then the value '-999999999' must be returned.

Example

Tool	Length	Radius
1	20.7	5
2	42.3	5.7
10	35.5	5.8

This example first loads the array in memory. After that it reads the element in the third row of the first column from this array in memory.

N1 E10=arrayOpen("\Work\Tool.arr") E10= internal array identification number
N2 E20=arrayRead(E10, 3, 1)

Parameter E20 contains now the value 10

24.8.2.9 arrayFilter (name, column, criteria)

The goal of the function arrayFilter() is to return a filtered array. This filtered array consists of the rows that contains the value to filter on.

<name> array name on hard disk or in memory.

hard disk: string (between double quotes).

memory: number or E-parameter (return value of arrayNew*()* or arrayOpen()).

<column> column number.

<criteria> criteria expression be used for filtering.

For the parameter <criteria>, all expressions are allowed, which are also allowed for DIN programming. An example is the following expression: (<=sin(90)).

It returns a filtered array with all values smaller than and equal to *sin(90)*.

Returns: 0 if the array is not filtered.

nnn internal array identification number

Example

Unbalance	Speed	Amplitude
100000	25	0.00345
100000	50	0.00862
200000	25	0.00710

This example filters the first column of the array on hard disk and stores the result in memory.

N1 E10=arrayFilter("\Work\Balance.arr", 1, 100000) E10= internal array identification number

Unbalance	Speed	Amplitude
100000	25	0.00345
100000	50	0.00862

24.8.2.10 arraySort (name, column, order)

The goal of the function arraySort() is to return a column sorted array.

<name> array name on hard disk or in memory.
hard disk: string (between double quotes).
memory: number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

<column> column number.

<order> sort order; 1=ascending and 2=descending

Note: If the non-sorted array contains empty rows, the number of rows in the sorted array must be less than the number of rows in the non-sorted array.

Returns: 0 if the array is not sorted.
nnn internal array identification number.

Example

Unbalance	Speed	Amplitude
100000	25	0.00345
100000	50	0.00862
200000	25	0.00710

This example sorts the third column of the array on hard disk ascending and stores the result in memory.

N1 E10=arraySort(“\Work\Balance.arr”, 3, 1) E10= internal array identification number

Unbalance	Speed	Amplitude
100000	25	0.00345
200000	25	0.00710
100000	50	0.00862

24.8.2.11 arrayDelete (name)

The goal of the function arrayDelete() is to delete an array from hard disk or from CNC memory.

<name> array name on hard disk or in memory.
hard disk: string (between double quotes).
memory: number or E-parameter (internal array identification number) (return value of arrayNew() or arrayOpen()).

Returns: 0 if the array is not deleted.
1 if the array is successfully deleted.

Example

This example deletes an array from hard disk.

N1 E10=arrayDelete(“\Work\Tool.arr”)

This example deletes an array from memory.

N1 E10=arrayOpen(“\Work\Tool.arr”) E10= internal array identification number
N2 E11=arrayDelete(E10)

24.8.3 Method with Configuration file (previous versions)

In the previous versions the following a restricted possibility was implemented.
It is advisable to use only the new functionality.

Configuration file

Configuration files are required to describe how and where to write or read.
These configuration files are saved on the hard disk:
D:\STARTUP\CYCLES\ARRnnnnn.CFG
nnnnn is the file number from 1 to 89999.

File to define an array and fill it with basic settings

An array is defined with a configuration file.
This is activated when the system is started.
A maximum of 10 arrays can be defined.
End users can define files themselves.
The maximum size for all arrays together is 5000 elements.

Description of an array configuration file:

```
;Comments start with ';'
;
;Sections:
;[element]                            defines an element in the array
;row     =     row number             where row number = [1|...|9999]
;col     =     column number          where column number = [1|...|9999]
;                                     row * column <= 5000
;val     =     value                  where value = real number (double)
;
```

Filling a configuration file

The configuration file can be filled with values (arrays). These arrays can be read (arrayRead) during execution like E-parameters. There is no function to write values in the array during execution

Example: Array configuration file:

ARRnnnnn.CFG

```
[element]
row     =     1
col     =     1
val     =     0                ; element (1,1).=.0

[element]
row     =     3
col     =     66
val     =     397.01           ; element (3,66) = 397.01

 [element]
row     =     9999             ;maximum row size
col     =     9999
val     =     -123456789.123456789
```

arrayread (arraynumber, row, column)

arraynumber	is the number of the array. Every array has its own configuration file. Arraynumber between 1 and 89999.
Row	is the row number in the array that is to be read. Row between 1 and 999999.
Column	is the position in the row of the array that is to be read. Column between 1 and 999999.

Fixed arrays can be read with the arrayread function. The arrays are filled from a configuration file D:\STARTUP\CYCLES\ARRnnnnn.CFG).

Empty ‘elements’ in the array have the value <–999999999>.

Example arrayread

E300 = arrayread(100,1,2)
E300 has the value of array 100, row 1, column 2.

25. Tool measuring cycles for laser measuring

25.1 General remarks for laser measuring

Laser measuring is extended with the following G-functions:

G951	Calibration.	replaced G600
G953	Measure tool length	replaced G601
G954	Measure length, radius	replaced G602
G955	Cutter control shank	replaced G603
G956	Tool breakage control	replaced G604
G957	Cutter control shape.	
G958	Tool setting length, radius, corner radius.	

For the explanation of these G-functions, see: Manual Blum.

For laser measuring of turning tools: see G615 in chapter: Turning.
For laser measuring of temperature compensation: see G642 in chapter: Measuring cycles.

26. Measuring system “Table-Probe“ (TT)

26.1 General notes measuring system “Table-Probe“ (TT)

Remark: TT means “Table Probe“, for example TT130 or a similar instrument.

Availability

The machine manufacturer for the measuring instrument must prepare the machine and MillPlus ***IT***. If not all the G functions described here are available on your machine, consult your machine handbook.

Programming

Before calling one of the G600-G609 functions a M24 (active measuring system) must be programmed, so that the measuring system is set in the measuring position.
After measuring a M28 (deactivate measuring system) must be programmed, so that the measuring system is retract.

Machine constants

The G function and associated machine constants are activated via the following machine constants.

MC 261 >0	measurement cycle functions
MC 254 >0	measure tool
MC 840 =1	measurement probe present
MC 854 =2	tool measuring instrument type (0=none, 1=laser, 2=TT)

MC 350	Probe position 1st axis µm
MC 352	Probe position 2nd axis µm
MC 354	Probe position 3rd axis µm

Coordinates of the TT stylus centre point relative to the machine zero point G51 and G53 (-max - +max µm)

After calibration the exact positions is written in MC350 – Mc355.

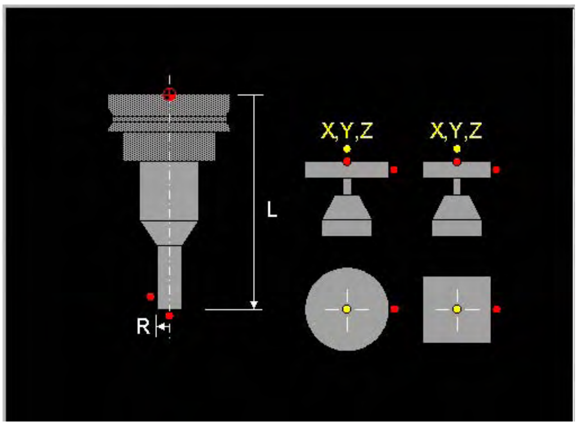
MC 356	axis number for radial measurement: 1=X, 2=Y, 3=Z
MC 357	tool axis number for measuring: 1=X, 2=Y, 3=Z
MC 358	measuring: 3rd axis 0=no, 1=yes
MC 359	radial probe contact side: -1=negative, 0=automatic, 1=positive

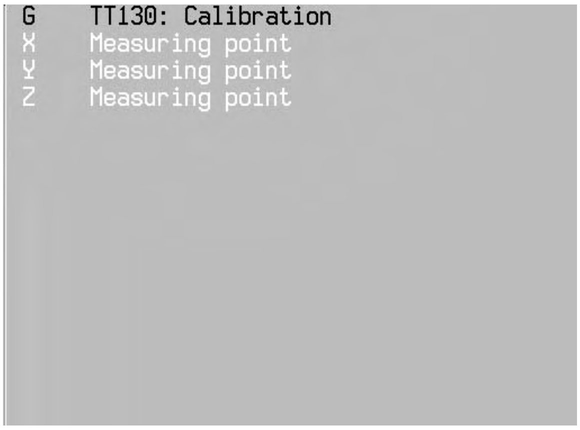
MC 360 -- MC 369 are for the second laser measurement system a second work area or an attachment spindle. Witch area will be used, is determinates by the IPCL.

MC 392	maximum permitted measurement error for tool measurement with rotating tool (2 - 1000 µm)
MC 394	probe measuring feed with tool measurement with non-rotating tool (10 - 3000 mm/min)
MC 395	distance from tool underside to stylus top for tool radius measurement (1 - 100000 µm)
MC 396	diameter or side length of the stylus of the TT. (1 - 100000 µm)
MC 397	safety zone around the stylus of the TT for pre-positioning. (1 - 10000 µm)
MC 398	rapid in measuring cycle for TT. (10 - 10000 mm/min)
MC 399	maximum permitted rotational speed at tool edge (1 - 120 m/min).

26.2 G606 TT: Calibration

To determine the position of the measuring instrument and store this position value in the machine constants provided.





Notes and use

Calibration tool

Before you calibrate, you must enter the exact radius and the exact length of the calibration tool in the tool table.

Sequence

The calibration process runs automatically. MillPlus ***IT*** also determines the centre offset of the calibration tool automatically. For this, MillPlus ***IT*** rotates the spindle after half of the calibration cycle by 180°. As a calibration tool, use an exactly cylindrical part, e.g. a cylindrical pin. MillPlus ***IT*** stores the calibration values in the machine constants and takes them into account in the subsequent tool measurements.

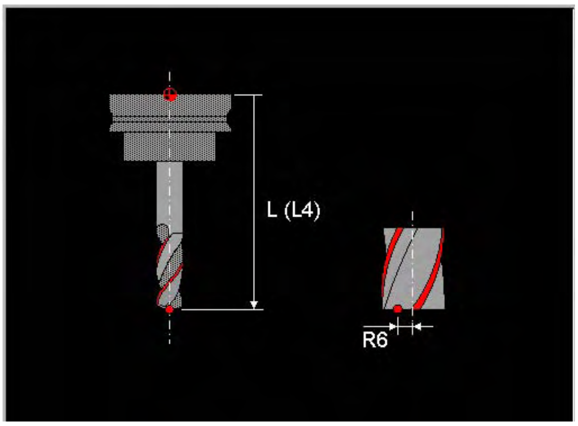
In MC 350, MC 352, MC 354 the position of the TT in the work area of the machine must be stipulated.
If you change one of MC 350, MC 352, MC 354, you must re-calibrate.

Position

Input in the X, Y and Z-axes, the position in which the possibility of collision with workpieces or clamping fixtures is excluded. If the position height input is so small that the tip of the tool would be below the plate surface, MillPlus ***IT*** positions the calibration tool above the plate non-automatically.

26.3 G607 TT: Measuring tool length

To measure the tool length.



Notes and use

Tool length and radius

Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.

Addresses of the tool memory

The following addresses of the tool memory are used:

L	tool length
L4=	length allowance
L5=	length wear tolerance
R	tool radius
R4=	radius allowance
R6=	measurement offset radius
E	tool status

Sequence

The tool length can be determined in three different ways:

1. If the tool diameter is greater than the diameter of the measurement surface of the TT, measure with tool rotating.
2. If the tool diameter is smaller than the diameter of the measurement surface of the TT or if you determine the length of drills or radius cutters, measure with tool stationary.
3. With the parameter I2=1 all teeth are measured. The measurement is carried out with stationary spindle. The greatest tooth length is entered in the tool table.

Measuring with tool rotating

To determine the longest edge, the tool to be measured is offset to the probe system centre point and moved, rotating, onto the measurement surface of the TT. Program the offset in the tool table under tool offset; radius (R).

Measuring with tool stationary (e.g. for drills).

The tool to be measured is moved to be concentrically above the measurement surface. Then it travels with the spindle stationary onto the measurement surface of the TT. For this measurement enter the tool offset: radius (R6=0) in the tool table.

Individual edge measurement

MillPlus ***IT*** pre-positions the tool to be measured to the side of the probe. The end face of the tool is then located below the probe top as laid down in MC 395. In the tool table, you can stipulate an additional offset under tool offset; length (L). MillPlus ***IT*** applies the probe radial with the tool rotating, to determine the start angle for the individual edge measurement. It then measures the length of all edges by changing the spindle orientation. For this measurement, select the Softkey all teeth.

Measure tool (E=0 or no value)

During the initial measurement, MillPlus ***IT*** overwrites the tool radius (R10 with R10.012) and the tool length (L100 with L99.456) in the tool memory and sets the oversizes R4 and L4 = 0.

Check tool (E=1)

During the initial measurement, MillPlus ***IT*** overwrites the tool length L in the tool memory and sets the oversize L4=0. In the event that you are checking a tool, the actual length measured is compared with tool length L extracted from the tool table. MillPlus ***IT*** calculates the mathematically correct variance and enters this as the oversize L4 in the tool table. If this oversize is greater than the permissible wear or breakage tolerance for the tool length, then a fault report is made.

Safe height (I1=):

Enter a position in the spindle axis, by means of parameters from the entry dialog (I1 = safety distance), such that a crash with pieces of work or their supporting holders is excluded. The safe height refers to the reference point for the active piece of work. If the safe height entered is so small that the tool tip would lie below the top surface of the plate, MillPlus ***IT*** does not automatically place the tool over the plate (security zone from MC397)

Cut measurement (I2=):

switch on or off individual cut measurement (Parameter I2=)
With I2=0 or no value, individual edge measurement is carried out.

Difference EASYoperate and DIN.

In EASYoperate is parameter edge measurement (I2=) replaced by a Softkey "all Teeth".

Stationary spindle

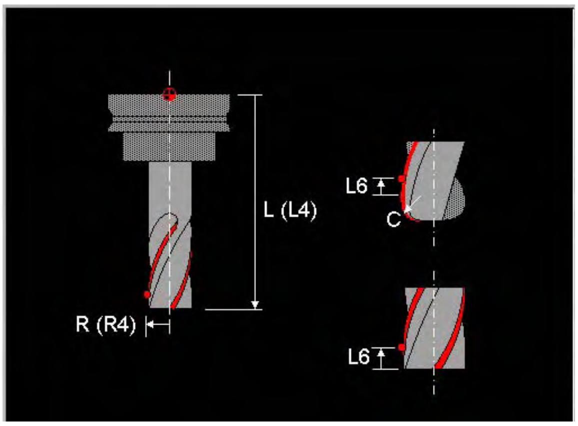
MillPlus ***IT*** uses the probe measuring feed from MC 394 for the measurement with stationary spindle.

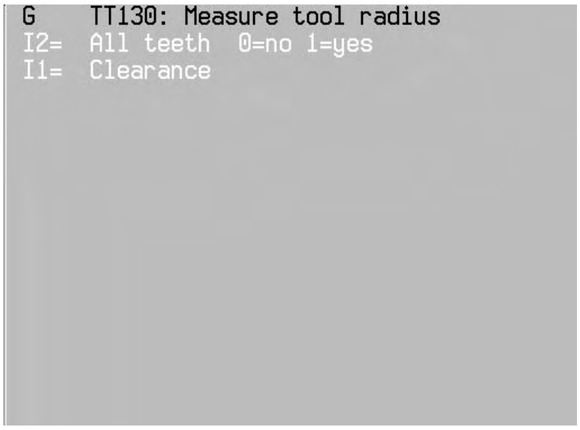
Calculation of the spindle Speed

When measuring with a tool, MillPlus ***IT*** calculates the spindle speed and the probe measuring feed automatically.

26.4 G608 TT: Measuring tool radius

To measure the tool radius.





Notes and use

Tool length and radius

Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.

Addresses of the tool memory

The following addresses of the tool memory are used:

L	tool length
L4=	length allowance
R	tool radius
R4=	radius allowance
R5=	radius wear tolerance
E	tool status

Measure tool (E=0 or no value)

During the initial measurement, MillPlus ***IT*** overwrites the tool radius (R10 with R10.012) and the tool length (L100 with L99.456) in the tool memory and sets the oversizes R4 and L4 = 0.

Measurement sequence

You can determine the tool radius in two ways:

1) Measurement with rotating tool
2) Measurement with rotating tool and subsequent individual edge measurement

With individual edge measurement, the radius is first measured roughly and the position of the largest tooth determined. After that, the other teeth are measured.

MillPlus ***IT*** pre-positions the tool to be measured to the side of the probe. The milling cutter end face is then below the top of the probe, as laid down in MC 395. MillPlus ***IT*** applies probe measuring radial with rotating tool. If an individual edge measurement is also to be carried out, the radii of all edges are measured by means of spindle orientation.

Check tool (E=1)

If you check a tool, the measured radius is compared with the tool radius R from the tool table. MillPlus ***IT*** calculates the difference with correct sign and enters this as allowance R4 in the tool table. If the allowance is greater than the permitted wear (R5=) or breakage tolerance for the tool radius, an error message is output.

Clearance (I1=)

Enter a position in the spindle axis, by means of parameters from the entry dialog (I1 = safety distance), such that a crash with pieces of work or their supporting holders is excluded. The safe height refers to the active workpiece reference point. If the safe height entered is so small that the tool tip would lie below the top surface of the plate, MillPlus ***IT*** does not automatically place the tool over the plate (security zone from MC397)

Edge measurement (I2=)

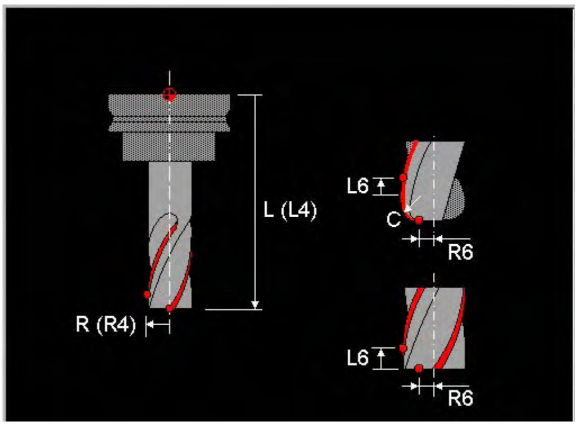
With parameter I2=1 all teeth are measured.
With I2=0 or no value, an individual edge measurement is carried out.

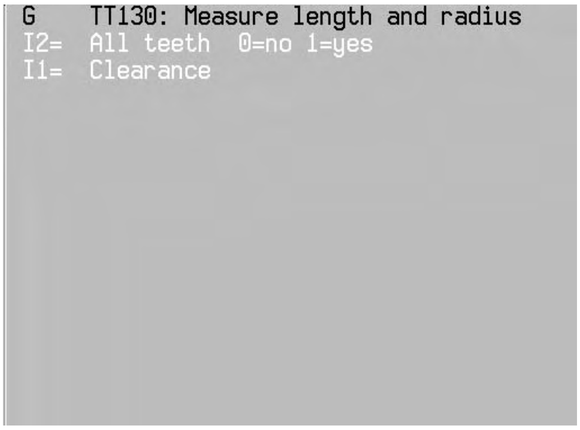
Difference EASYoperate and DIN.

In EASYoperate is parameter edge measurement (I2=) replaced by a Softkey "all Teeth".

26.5 G609 TT: Measuring length and radius

To measure tool length and radius.





Notes and use

Tool length and radius

Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.

Addresses of the tool memory

The following tool memory addresses are used:

L	tool length
L4=	length allowance
L5=	length wear tolerance
R	tool radius
R4=	radius allowance
R5=	radius wear tolerance
E	tool status

Measurement sequence

MillPlus ***IT*** measures the tool according to a fixed, programmed sequence. First the tool radius and then the tool length are measured.

You can determine the tool radius in two ways:

1) Measurement with rotating tool
2) Measurement with rotating tool and subsequent individual edge measurement

Measure tool (E=0 or no value)

The function is especially suitable for the first measurement of tools since, compared with the individual measurement of length and radius, there is a considerable time advantage.
With the first measurement, MillPlus ***IT*** overwrite the tool radius R and tool length L in the tool memory and sets the allowance R4 and L4 = 0.

Check tool (E=1)

If you check a tool, the measured tool data are compared with the tool data from the tool table. MillPlus ***IT*** calculates the differences with correct signs and enters these as allowance R4 and L4 in the tool table. If an allowance is greater than the permitted wear (L5= and R5=) or breakage tolerance for the tool radius, an error message is output.

Clearance (I1=)

The clearance (I1=) in the direction of the spindle axis, excluded the possibility of a collision with workpieces or clamping fixtures. The clearance relates to the top of the measuring device. Default I1=MC397

Edge measurement (I2=)

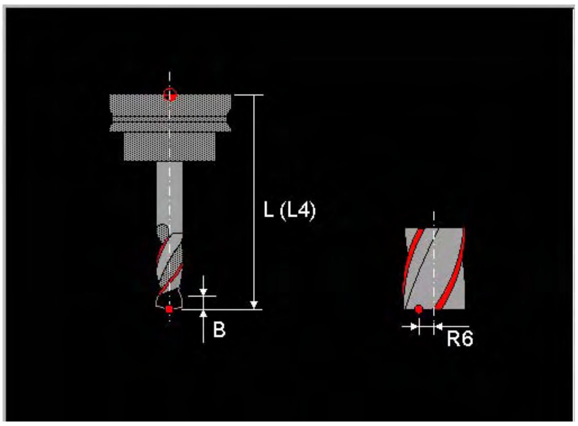
With parameter I2=1 all teeth are measured.
With I2=0 or no value, an individual edge measurement is carried out.

Difference EASYoperate and DIN.

In EASYoperate is parameter edge measurement (I2=) replaced by a Softkey "all Teeth".

26.6 G610 TT: Tool breakage control

Monitoring tool length. Mainly used for monitoring tools that are liable to break, such as drills. The measured wear is not corrected.




G TT130: Tool breakage control
I2= All teeth 0=no 1=yes
I1= Clearance
I3= 0=Error/Pallet 1=no error
O1= E-param. number for tool status

Hinweise und Verwendung

Tool data

Tool data must be entered in the tool table beforehand. No measurement is done where the tool status is -1 or -4.

Addresses of tool memory

The following addresses of the tool memory are used:

L	Tool length
L4=	Length allowance
R6=	Radius position for breakage check
B	Breakage tolerance in mm (also in inch mode)
E	Tool status

For individual cutting measurement:

R	Tool radius
R4=	Radius allowance
L6=	Length position for breakage check

Differences between EASYoperate and DIN:

This function is not available in EASYoperate.

Sequence

Tool breakage, like tool length, can be determined in three different ways.

1 If the tool diameter is greater than the measuring surface of the TT, then measure with the tool rotating.

2 If the tool diameter is less than the measuring surface of the TT, then measure with the tool stationary. The same applies if you wish to determine the length of drills or radiusing mills.

3 All teeth are measured using parameter I2=1. This measurement is carried out with the spindle stationary.

Measuring with a rotating tool

The tool to be measured is offset to the sampling system centre and brought to the TT measuring surface while rotating. You must program the offset in the tool table under tool offset radius (R6=).

Measurement with stationary tool (e.g. drill):

The tool to be measured is centred above the measuring surface. Then it advances with a stationary spindle to the TT measuring surface. For this measurement, enter the tool offset radius (R6=0) in the tool table.

Individual cutting measurement

The MillPlus ***IT*** positions the tool to be measured at the side of the probe. The front surface of the tool is then below the top edge of the probe, as laid down in MC395. You can define an additional offset in the tool table under tool offset length (L6=). MillPlus ***IT*** scans radially with the tool rotating in order to determine the starting angle for the individual cutting measurement. It then measures the length of all cuts by changing the spindle orientation. For this measurement, you select I2=1"

Safety distance (I1=)

The setup clearance (I1=) in the direction of the spindle axis must be sufficient to prevent any collision with the workpiece or clamping devices. The setup clearance is with respect to the top edge of the stylus. Basic setting I1=MC397

Cutting measurement (I2=)

If I2=1 an individual cutting measurement is carried out.
If I2=0 or no value, individual cutting measurement is deselected.

Error evaluation (I3=)

If a break is detected, various actions can follow:
13= 0 error message or reject pallet (basic setting)
I3= 1 no error message
If I3=0 is selected, function M105 (tool break detected) is issued in the case of tool breakage. The IPLC switches the TT off and the controller issues an error message.
If, however, a pallet system is present, the pallet is rejected if possible, the current program is interrupted and a new pallet is brought in.
If I3=1 is selected, no error message is issued on tool breakage. Every action must be programmed in the part program. To achieve this, the tool status (value E from the tool memory) can be written directly to an E parameter. See address O1.

Tool status output to E parameter (O1=)

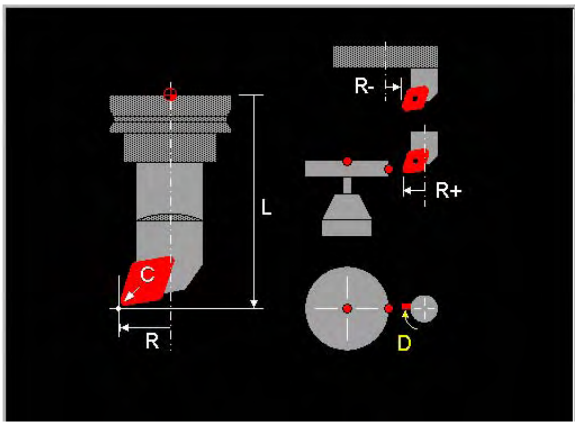
The tool status (definition E in the tool memory) is written to the specified E parameter. Based on this parameter, the program can determine whether a tool breakage has been detected (status 4). This is only meaningful if the error message has been switched off with I3=1.

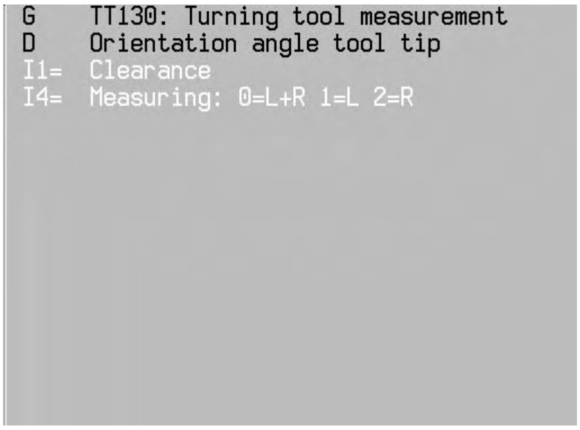
Stationary spindle

For measurement with a stationary spindle, MillPlus ***IT*** uses the scanning feed from MC394.

See G607 for calculation of the spindle speed or scanning feed.

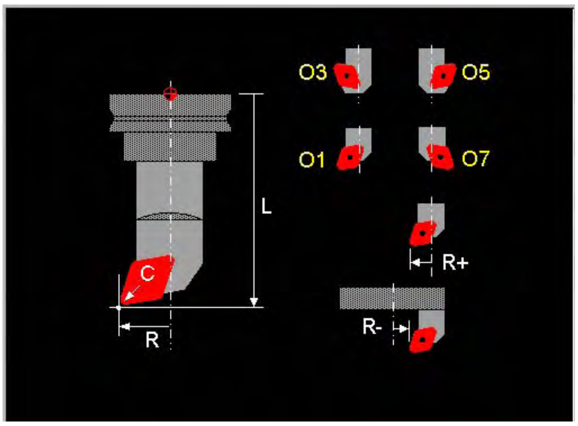
26.7 G611 TT: Measuring turning tools

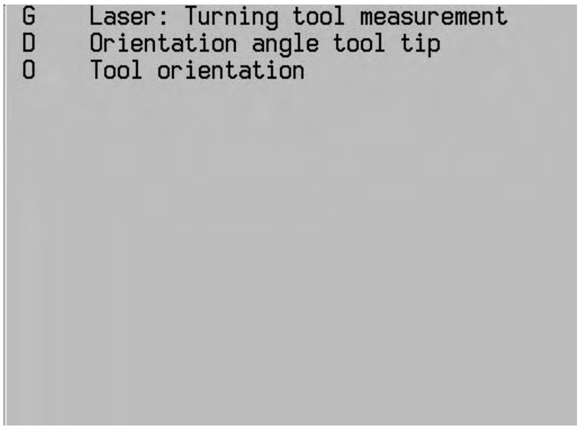




Refer to Chapter "Turning mode".

26.8 G615 Laser: Measuring turning tools





Refer to Chapter "Turning mode".

27. Measuring cycles

27.1 Introduction to measuring cycles

Measuring cycles in the main plane:

G620 Angle measurement
G621 Position measurement
G622 Corner outside measurement
G623 Corner inside measurement
G626 Datum outside rectangle
G627 Datum inside rectangle
G628 Circle measurement outside
G629 Circle measurement inside

Special measuring cycles:

G631 Measure the inclination of a plane (G7)
G633 Angle measurement 2 holes
G634 Measurement center 4 holes
G640 Rotary table center offset.
G642 Laser: temperature compensation

Comments

Comments are not allowed in a block with a machining cycle.

Results of activating a measuring cycle:

- G91 is deactivated.
- Radius correction is deactivated (G40 is active)
- Scaling with G72 is deactivated
- L and R in G39 are zeroed.

	G17	G18	G19
Main axis	X	X	Y
Secondary axis	Y	Z	Z
Machining plane	XY	XZ	YZ
Tool axis	Z	Y	X or –X (G66/G67)

In some cycles the direction of measurement is determined by the address (I1=).

Zero point

Measured values (I5>0) can be stored in the zero offset table where an offset is currently active and/or in an E parameter.
Restriction with G7: measured values can only be written in an E parameter. (I5= must only be zero).

Differences between EASYoperate and DIN/ISO

Certain addresses are not available in EASYoperate. The measured values are displayed in a window.

Comments

Comments are not allowed in a block with a machining cycle.

Results of activating a measuring cycle:

- G91 is deactivated.
- Radius correction is deactivated (G40 is active)
- Scaling with G72 is deactivated
- L and R in G39 are zeroed.

Machine constants that are important for measuring cycles

MC261 >0: Measuring cycle functions active
MC312 =1: Free machining plane active (G631)
MC840 =1: Measuring probe present
MC843: Measuring feed
MC846 >0: Angle of orientation of measuring probe
MC849 : Probe 1. angle of orientation

Functions that are not allowed when a measuring cycle is called.

G36, rotations (B4=) in G92/G93 and G182.
G7 must not be active if the measured values are stored in zero point offset (I5>0).
Tool T0 is not allowed.

Warning: Pre-position the tool so that there can be no collision with the workpiece or clamping devices.

27.2 Description of addresses

Mandatory addresses

Mandatory addresses are shown in black.
If a mandatory address is not entered an error message is issued.

Optional addresses

Optional addresses are shown in light grey.
If this address is not entered it is ignored or given the basic setting that has already been entered.

Explanation of addresses.

The addresses described here are used in most cycles. Specific addresses are described in the cycle.

X, Y, Z: Starting point

Starting point of measuring motion. The measuring cycle starts here. If all the starting point coordinates are not entered, the current position of the tool is adopted.

Execution
Unlike a milling cycle, a measuring cycle is carried out directly from the starting point (X, Y, Z).
The probe moves to the first starting point (X, Y, Z) in rapid motion and depending on G28, using positioning logic.

C1= Maximum measured length

Maximum distance between the starting and finishing points of the measuring stroke. (Basic setting 10). Movement stops once the wall of the workpiece or the end of the measured length is reached.

Note:
If there is no contact with material within the measuring stroke (C1=) an error message is issued.

L2= Safety distance

During (if I3=1) and at the end of measurement, the probe moves at the safety distance (default setting 0 for measurement on the outside of the workpiece or 1mm for measurements in pockets and holes). Safety distance (L2=) is with respect to the current starting point X, Y, Z.

B3= Distance to the corner
The distance between the first starting point and the corner of the workpiece.

Distance to next measurement about the corner of the workpiece.
The path traced by the probe around the corner of the workpiece to the starting point of the 2nd measurement is the same length in both directions. For each direction the distance is the sum of B3= and the first measuring distance travelled.

I1= Direction of probe movement with respect to workpiece
I1=±1 Main axis
I1=±2 Secondary axis
I1=-3 Tool axis
The angular reference axes are always perpendicular to the direction of scanning

I3= Movement between measuring strokes.
I3= is used to determine whether the positioning movements between measurements take place at the measuring height or the safety distance (L2=).
I3=0 The positioning movement between measuring strokes is at the measuring height and parallel to the main axis.
In the case of circular movement the positioning movement is circular and at the feed rate.
I3=1 The positioning movement between measuring strokes is at the measuring height and in a line between measurement points.

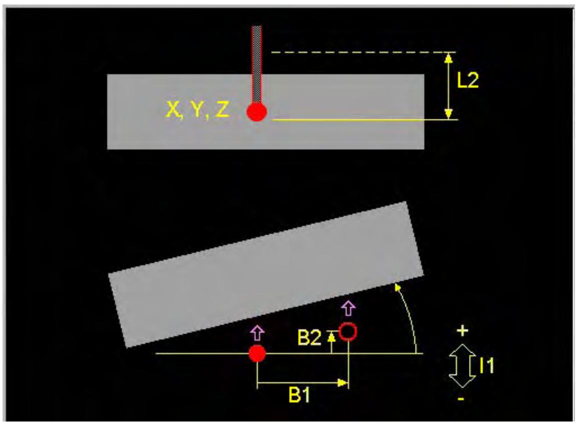
I4= Corner number (1 - 4)
Defines the corner where the first measurement should take place (default setting 1).
The first measurement is always perpendicular to the main axis.
The second measurement is always perpendicular to the secondary axis.

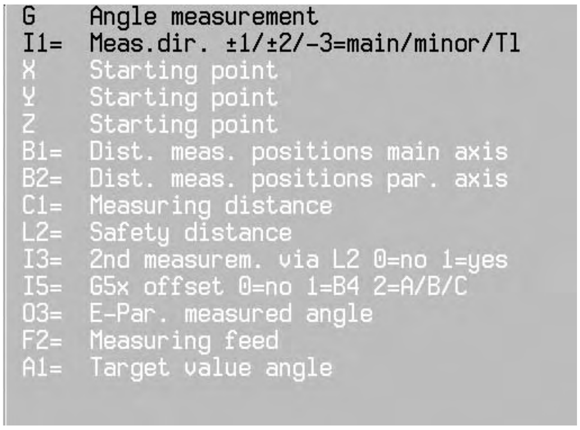
O1= to O6= Save measured values
The measured values can be written in the E parameters.. The number of the E parameter must be entered. If no number is entered, nothing is saved.
Example: O1=10 means that the result is stored in E parameter 10.

F2= Measuring feed The basic setting is MC843.

27.3 G620 Angle measurement

Measuring the inclined position of a clamped workpiece.





B1= Distance with direction along the main axis.
If I1=±2, B1= must be programmed (B1= must not equal zero).
If I1=-3, B1= and B2= do not both need to be programmed at the same time.

B2= Distance with direction along the secondary axis.
If I1=±1, B2= must be programmed (B2= must not equal zero).
If I1=-3, B1= and B2= do not both need to be programmed at the same time.
The following is not allowed: B1= B2= 0

I5= Save measured values in a zero point offset.
I5=0 Do not save
I5=1 Save in the active zero point offset in the angle of rotation (G54 B4=).
I5=2 Save in the active zero point offset in the axis of rotation (A/B/C).
On saving, the measured values are added to the active zero point offset.

A1= If the measured angle is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured position thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

B1=0, B2=0, C1=10, L2=0, I3=0, I5=0, F2=MC843, A1=0.

Notes and application

Depending on the plane selected (G17, G18 or G19), the parameter I1= determines the direction of measurement and this defines the meanings of B1= and B2=.

	G17				**G18**				**G19**			
Direction of measurement	I1=±1	I1=±2	I1=3		I1=±1	I1=±2	I1=3		I1=±1	I1=±2	I1=3	
			B1=	B2=			B1=	B2=			B1=	B2=
Angle plane	XY	XY	XZ	YZ	XZ	XZ	XY	ZY	YZ	YZ	YX	ZX
Axis of rotation	C	C	B	A	B	B	C	A	A	A	C	B

EASYoperate ⇔ DIN/ISO

The addresses O3= and F2= are not available in EASYoperate.

The cycle

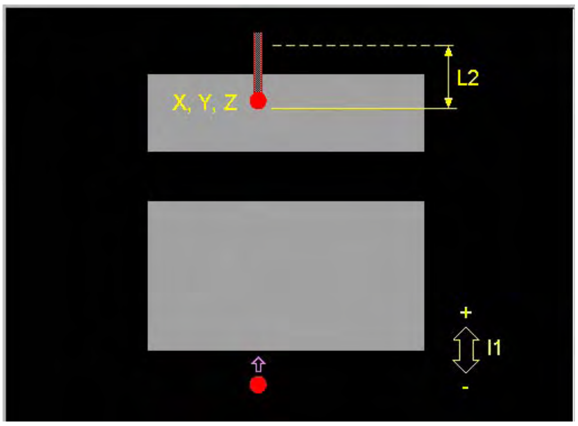
1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. At the end there is rapid movement to the safety distance (L2=).
7. The measured value is stored as per I5=.

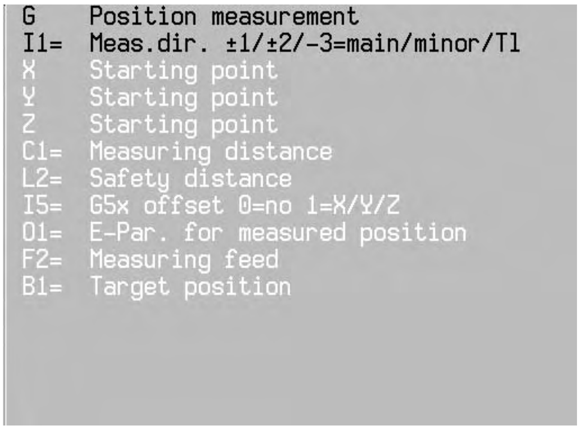
Example: Setting up a workpiece

N40 G17	Set the surface plane
N50 G54 I3	Set zero
N60 G620 X-50 Y-50- Z-5 I1=2	
B1=100 L2=10 I3=1 I5=2	Define and execute the measuring cycle
	After the cycle G54 I3 is recalculated
N70 G0 C0	Rotary table is positioned at zero (G17).

27.4 G621 Position measurement

Measurement of a coordinate on the wall of a workpiece.





I5= Save measured values in a zero point offset.
I5=0 Do not save
I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point offset.

B1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

C1=10, L2=0, I5=0, F2=MC843, B1=0

Notes and application

Address I1= determines the direction of measurement, depending on the plane selected (G17, G18 or G19).

EASYoperate ⇔ DIN/ISO

The addresses O1= and F2= are not available in EASYoperate.

The cycle

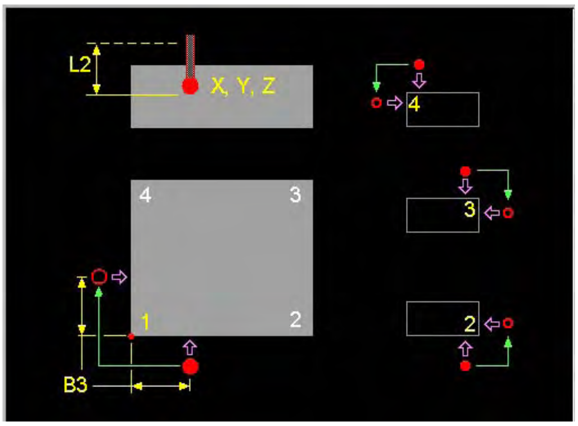
1 Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2 First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3 Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4 At the end, rapid movement back to the safety distance (L2=).
5 The measured value is stored as per I5=.

Example: Measuring a position.

N60 G621 X40 Y40- Z-5 I1=2
L2=20 O1=300 — Define and execute the measuring cycle
After the cycle the result is written in E parameter (E300).

27.5 G622 Corner outside measurement

Measure the corner position (outside) of an aligned workpiece.



G	Corner outside measurement
I4=	Corner number
X	Starting point
Y	Starting point
Z	Starting point
B3=	Distance to corner
C1=	Measuring distance
L2=	Safety distance
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. position main axis
O2=	E-Par. meas. position minor axis
F2=	Measuring feed
X1=	Target position corner
Y1=	Target position corner
Z1=	Target position corner

I5= Save measured values in a zero point offset
I5=0 Do not save
I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Check:
- the sides must be parallel to the axes
- the angle of the workpiece must be 90 degrees
- the measured plane is at right angles to the axis of the workpiece.

Direction of approach to measurements
- the first measurement is always perpendicular to the main axis.
- the second measurement is always perpendicular to the secondary axis.

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2= and F2= are not available in EASYoperate.

The cycle

1 Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2 First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3 Rapid movement back to the first starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).

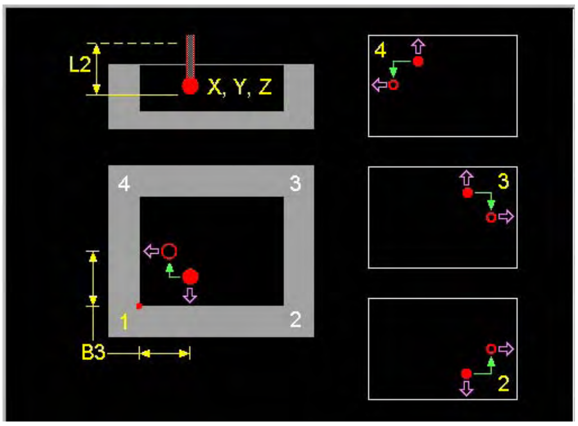
4 Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5 Second measurement (as points 2 and 3).
6 At the end, rapid movement back to the safety distance (L2=).
7 The measured value is stored as per I5=.

Example: Setting up an outside corner of a workpiece

N40 G1 X.. Y.. Z-5	Locate the probe 10mm to the right of corner 1 and 8mm away from the front.
N50 G54 I3	Set zero
N60 G622 L2=20 B3=25 I3=1	
I5=1 X1=-50 Y1=-50	Define and execute the measuring cycle After the measuring cycle the zero point offset is overwritten so that the coordinates of corner 1 are equal to X1= and Y1=.

27.6 G623 Corner inside measurement

Measure the corner position (inside) of an aligned workpiece.




```
G     Corner inside measurement
I4=   Corner number
X     Starting point
Y     Starting point
Z     Starting point
B3=   Distance to corner
C1=   Measuring distance
L2=   Safety distance
I3=   2nd measurem. via L2 0=no 1=yes
I5=   G5x offset 0=no 1=X/Y/Z
O1=   E-Par. meas. position main axis
O2=   E-Par. meas. position minor axis
F2=   Measuring feed
X1=   Target position corner
Y1=   Target position corner

Z1=   Target position corner
```

I5= Save measured values in a zero point offset
I5=0 Do not save
I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=10, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Check:
- the sides must be parallel to the axes
- the workpiece angle must be 90 degrees
- the measured plane is at right angles to the axis of the workpiece.

Direction of approach to measurements
- the first measurement is always perpendicular to the main axis.
- the second measurement is always perpendicular to the secondary axis.

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2= and F2= are not available in EASYoperate.

The cycle

1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the first starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).

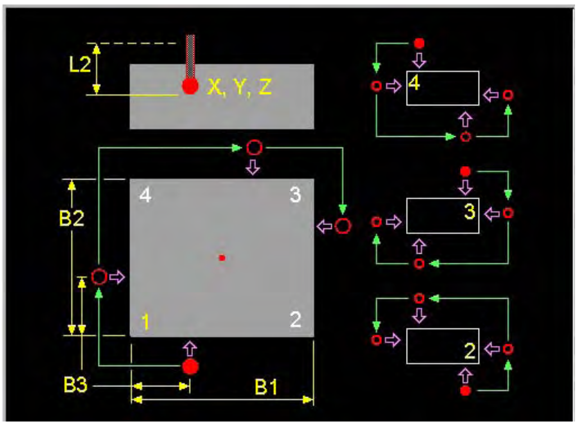
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. At the end, rapid movement back to the safety distance (L2=).
7. The measured value is stored as per I5=.

Example: Setting up an inside corner of a workpiece

N40 G1 X.. Y.. Z-5	Locate the probe 10mm to the right of corner 1 and 8mm away from the front.
N50 G54 I3	Set zero.
N60 G623 L2=20 B3=25 I3=1 I5= 1 X1=-50 Y1=-50	Define and execute the measuring cycle After the measuring cycle the zero point offset is overwritten so that the coordinates of corner 1 are equal to X1= and Y1=.

27.7 G626 Datum outside rectangle

Measuring the centre of an axially parallel rectangle.



G	Datum outside rectangle
I4=	Corner number
X	Starting point
Y	Starting point
Z	Starting point
B1=	1st Side length
B2=	2nd Side length
B3=	Distance to corner
C1=	Measuring distance
L2=	Safety distance
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. centre main axis
O2=	E-Par. meas. centre minor axis
O4=	E-Par. meas. length main axis
O5=	E-Par. meas. length minor axis
F2=	Measuring feed
X1=	Target centre point
Y1=	Target centre point
Z1=	Target centre point

I5= Save measured values in a zero point offset
- I5=0 Do not save
- I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).

On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Two opposite corners of the workpiece are measured (1+3 or 2+4)

Direction of approach to the first corner measurement
- the first measurement is always perpendicular to the main axis.
- the second measurement is always perpendicular to the secondary axis

Direction of approach to the second corner measurement
- clockwise from corner number 1 → 3 or 3 → 1
- anticlockwise from corner number 2 → 4 or 4 → 2

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O4=, O5= and F2= are not available in EASYoperate.

The cycle

1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.

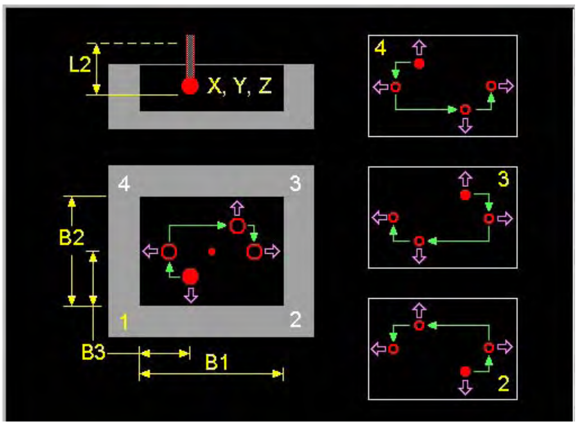
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. The opposite corner is measured using 3rd and 4th measurements (as points 2 and 3).
7. At the end, rapid movement back to the safety distance (L2=).
8. The measured value is stored as per I5=.

Example: Save the centre of a rectangle in the zero point offset.

N50 G54 I3	Set zero
N60 G626 X-45 Y-3 Z-5 B1=100	
B2=20 B3=5 I3=1 I5=1	Define and execute the measuring cycle
	After the cycle X and Y are recalculated in G54 I3

27.8 G627 Datum inside rectangle

Measuring the centre of an axially parallel rectangular hole.



G	Datum inside rectangle
I4=	Corner number
X	Starting point
Y	Starting point
Z	Starting point
B1=	1st Side length
B2=	2nd Side length
B3=	Distance to corner
C1=	Measuring distance
L2=	Safety distance
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. centre main axis
O2=	E-Par. meas. centre minor axis
O4=	E-Par. meas. length main axis
O5=	E-Par. meas. length minor axis
F2=	Measuring feed
X1=	Target centre point
Y1=	Target centre point
Z1=	Target centre point

I5= Save measured values in a zero point offset
- I5=0 Do not save
- I5=1 Save in the active zero point offset in the linear axes (X/Y/Z). On saving, the measured values are added to the active zero point offset.

X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

I4=1, B3=10, C1=10, L2=10, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

Two opposite corners of the workpiece are measured (1+3 or 2+4)

Direction of approach to the first corner measurement
- the first measurement is always perpendicular to the main axis.
- the second measurement is always perpendicular to the secondary axis.

Direction of approach to the second corner measurement
- clockwise from corner number 1 → 3 or 3 → 1
- anticlockwise from corner number 2 → 4 or 4 → 2

Remark: The support picture is in G17. By a machine with exchanged axis (G18) the picture is not correct. The angle 1 will be exchanged with 2 and 3 with 4.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O4=, O5= and F2= are not available in EASYoperate.

The cycle

1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.

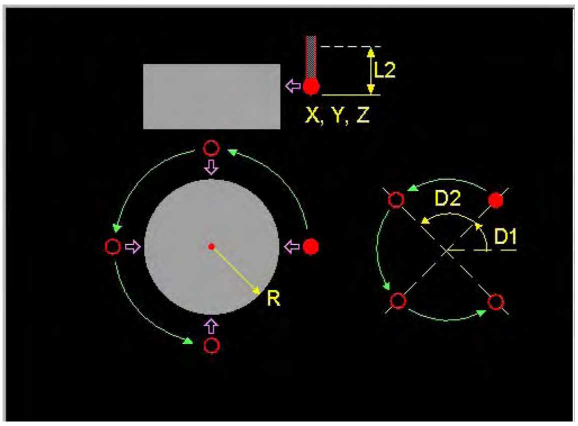
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second measurement (as points 2 and 3).
6. The opposite corner is measured using 3rd and 4th measurements (as points 2 and 3).
7. At the end, rapid movement back to the safety distance (L2=).
8. The measured value is stored as per I5=.

Example: Save the centre of a rectangle in the zero point offset.

N50 G54 I3	Set zero
N60 G627 X-45 Y-3 Z-5 B1=100	
B2=20 B3=5 I3=1 I5=1	Define and execute the measuring cycle
	After the cycle X and Y are recalculated in G54 I3

27.9 G628 Circle measurement outside

Measuring the centre of a circle.



G	Circle measurement outside
R	Circle radius
X	Starting point
Y	Starting point
Z	Starting point
D1=	Starting angle
D2=	Second angle
C1=	Measuring distance
L2=	Safety distance
I2=	Probe orientat. 0=no 1=180 2=yes
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. centre main axis
O2=	E-Par. meas. centre minor axis
O6=	E-Par. measured diameter

F2=	Measuring feed
X1=	Target centre point
Y1=	Target centre point
Z1=	Target centre point

D1= Angular offset of the circle measurement with respect to the main axis.
I2= Probe orientation in the direction of measurement:
0= measurement without rotation
1= measurement using 2 measurements with 180º rotation.
First measurement with standard orientation (MC849).
Second measurement with 180º rotation
The measured value is the average of these two.
2= measurement with orientation in the direction of measurement.
Only possible with an infra-red probe with all-round emitter.
The orientation option for the probe is defined in MC486.
I5= Save measured values in the zero point offset
0 Do not save
1 Save in the active zero point offset in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point offset.
X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.
The other addresses are described in the introduction to the measuring cycles.

Basic settings

D1=0, D2=90, C1=20, L2=10, I2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

The starting point selected for circle measurement should be such that the first measurement moves as exactly as possible in the direction of the centre of the circle.

Circle measurement is executed anticlockwise.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O6= and F2= are not available in EASYoperate.

The cycle

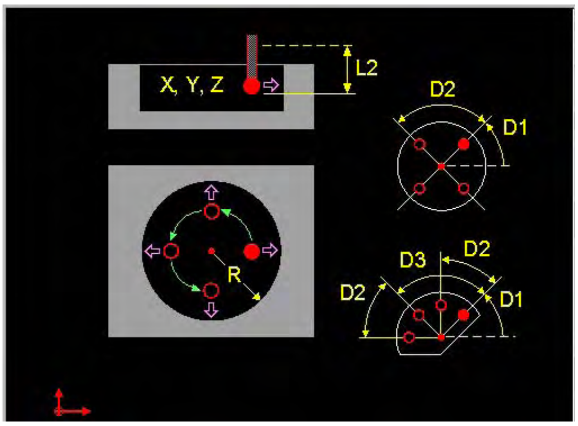
1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second, 3rd and 4th measurements (as points 2 to 4).
6. At the end, rapid movement back to the safety distance (L2=).
7. The measured value is stored as per I5=.

Example: Save the centre of a circular projection in the zero point offset.

N50 G54 I3	Set zero
N60 G628 X-45 Y-3 Z-5 R50 I3=1 I5=1	Define and execute the measuring cycle
	After the cycle X and Y are recalculated in G54 I3

27.10 G629 Circle measurement inside

Measuring the centre of a circular hole.



G	Circle measurement inside
R	Circle radius
X	Starting point
Y	Starting point
Z	Starting point
D1=	Starting angle
D2=	Second angle
D3=	Third angle
C1=	Measuring distance
L2=	Safety distance
I2=	Probe orientat. 0=no 1=180 2=yes
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. centre main axis
O2=	E-Par. meas. centre minor axis
O6=	E-Par. measured diameter
F2=	Measuring feed
X1=	Target centre point
Y1=	Target centre point
Z1=	Target centre point

D1= Angular offset of the circle measurement with respect to the main axis.
D2= Angle between the first and the second measurement and between the third and fourth measurement. The lowest value is 5°.
D3= Angle between the first and the third measurement. D3 must be at least 5° bigger than D2. When D3 and D2 are equal, a 3-points measurement is executed.

Remark: The highest accuracy will be reached by a symmetrical measuring with default values D2=90 and D3=180.

I2= Probe orientation in the direction of measurement:
0= Measurement without rotation
1= measurement using 2 measurements with 180º rotation.
First measurement with standard orientation (MC849).
Second measurement with 180º rotation
The measured value is the average of these two.
2= measurement with orientation in the direction of measurement.
Only possible with an infra-red probe with all-round emitter.
The orientation option for the probe is defined in MC486.
I5= Save measured values in the zero point offset
I5=0 Do not save
I5=1 Save in the active zero point offset in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point offset.
X1=, Y1=, Z1= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.
The other addresses are described in the introduction to the measuring cycles.

Basic settings

D1=90, D2=90, D3=180, C1=10, L2=10, I2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Notes and application

The starting point selected for circle measurement should be such that the first measurement moves as exactly as possible in the direction of the centre of the circle.

Circle measurement is executed anticlockwise.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O6= and F2= are not available in EASYoperate.

The cycle

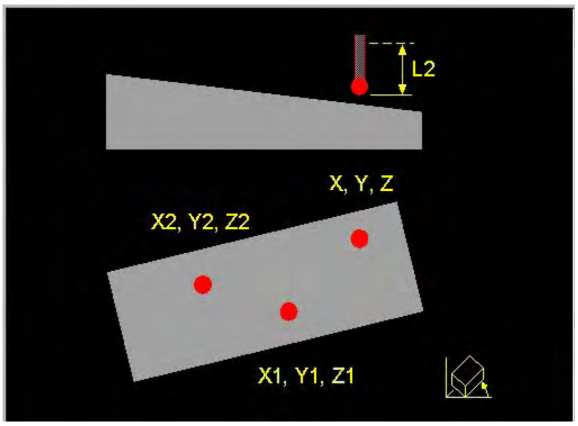
1. Rapid motion to the first starting point (X, Y, Z). If X, Y or Z is not programmed, the current position is taken as the starting point.
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Rapid motion, depending on I3= over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Third and 4th measurements (as points 2 to 4).
6. At the end, rapid movement back to the safety distance (L2=).
7. The measured value is stored as per I5=.

Example: Save the centre of a circle in the zero point offset.

N50 G54 I3	Set zero
N60 G629 X-45 Y-3 Z-5 R50 I3=1 I5=1	Define and execute the measuring cycle
	After the cycle X and Y are recalculated in G54 I3

27.11 G631 Measure position of inclined plane

Measure the inclination of a workpiece plane surface (g7) using 3-point measurement.



G	Obliqueness measurement
I1=	Meas.dir. ±1/±2/-3=main/minor/Tl
X	Starting point (meas. point 1)
Y	Starting point (meas. point 1)
Z	Starting point (meas. point 1)
X1=	Measuring point 2
Y1=	Measuring point 2
Z1=	Measuring point 2
X2=	Measuring point 3
Y2=	Measuring point 3
Z2=	Measuring point 3
O1=	E-Par. Angle of abs. rotation A5=
O2=	E-Par. Angle of abs. rotation B5=
O3=	E-Par. Angle of abs. rotation C5=
C1=	Measuring distance
L2=	Safety distance
I3=	Measur. 2 and 3 via L2 0=no 1=yes
F2=	Measuring feed

L2= The safety measurement is related to each starting point of a measurement and is in the measuring direction.

The other addresses are described in the introduction to the measuring cycles.

Basic settings

C1=20, L2=0, I3=0, F2=MC843

Notes and application

The measured inclination can be set exactly with the G7 function.

EASYoperate ⇔ DIN/ISO

The addresses O1=, O2=, O3= and F2= are not available in EASYoperate.

The cycle

Rapid movements always take place with positioning logic in the active (and possible already tilted) machining plane.

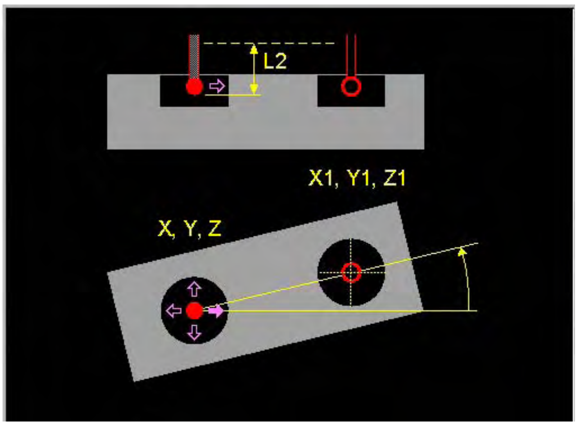
1. Rapid motion to the first starting point (X, Y, Z).
2. First measurement with measuring feed (F2=) until the end of the workpiece or the maximum measuring distance (C1=) is reached.
3. Rapid movement back to the starting point. An error message is issued if the probe has not switched within the maximum measuring distance (C1=).
4. Movement, depending on I3=, over the safety distance (L2=) to the starting point for the 2nd measurement.
5. Second and 3rd measurements (as points 2 to 4).
6. At the end there is rapid movement to the safety distance (L2=).
7. The measured values are stored.

Example: Set up the machining plane and rotate

N3416	Measure the machining plane and rotate
N1 G17	Set the surface plane
N2 G54 I1	
N3 T35 M66	Change the probe
N4 G0 X50 Y20 Z100	
N5 G631 X18 Y0 Z-16 X1=18 Y1=10	
Z1=-16 X2=10 Y2=0 Z2=-6 C1=15	
L2=20 O1=10 O2=11 O3=12 F2=150	Measure position of inclined plane
N10 G0 Z100	Go to a safe height (G17)
N11 G7 A5=E10 B5=E11 C5=E12 L1=1	Turn the machining plane

27.12 G633 Angle measurement 2 holes

Measuring the skew of a work piece set-up.
The probe measures the centre points of two cylindrical holes. Next the MillPlus calculates the angle between the main axis of the working plane and the connection line between the centre points of the holes.




```
G    Angle measurement 2 holes
X    Starting point (meas. point 1)
Y    Starting point (meas. point 1)
Z    Starting point (meas. point 1)
X1=  Measuring point 2
Y1=  Measuring point 2
Z1=  Measuring point 2
L2=  Safety distance
C1=  Measuring distance
I5=  G5x offset 0=no 1=B4 2=A/B/C
O3=  E-Par. measured angle
F2=  Measuring feed
A1=  Target value angle
```

X, Y, Z Starting point of the measurement of the first cylindrical hole (or the actual position)
X1=, Y1=, Z1= Starting point of the measurement of the second cylindrical hole (all three coordinates must be entered)
C1= Maximum measuring distance
L2= Safety distance
O3= Number of the E-parameter in which the angle is stored.
I5= Storing the measuring values in a zero point shift:
I5=0 Do not store
I5=1 Store in the active zero point shift of the rotation angle (B4=).
I5=2 Store in the active zero point shift of the rotary axis (A/B/C).
During storing the measuring values are added to the active zero point shift.
A1= If the measured angle is stored in the active zero point shift (I5>0), it is calculated in the command position.
For the remaining programming the measured position gets the command position.

The description of the remaining addresses can be found in the introduction to measuring cycles.

Default settings

C1=20, I5=0, F2=MC_0843, A1=0.

Notes and usage

The starting position must be programmed inside the cylindrical hole.

EASYoperate ⇔ DIN/ISO

In EASYoperate the addresses O3= and F2= are not available.

Cycle sequence

1. Movement in rapid to the first starting point (X, Y, Z) in the first cylindrical hole. When X, Y, Z are not programmed, the actual position is taken as the starting point.
2. Measuring movement with measuring feed (F2=) to the hole side or till the maximum measuring distance (C1=) is reached. The centre point is first measured roughly and than exactly
3. Movement in rapid back to the starting position. An error message is given when the measuring probe was not triggered within the maximum measuring distance (C1=). Retract movement to the safety distance (L2=)
4. Movement in rapid with regard to the safety distance to the starting point of the 2nd hole.
5. The second hole is measured in the same way.

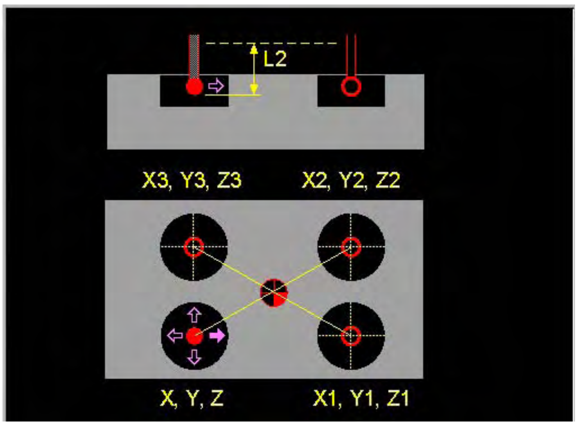
6. At the end a movement in rapid follows to the safety distance (L2=).
7. Depending on I5= the measured value is stored.

Example: Aligning a work piece

N40 G17	Set the plane
N50 G54 I3	Set the zero point
N60 G633 X-100 Y-50 Z-5	Define the measuring cycle with the starting point of the 1st cylindrical hole
X1=-10 Y1=-50 Z1=-5	Starting point of the 2nd hole
L2=30 I5=2	Safety distance = 30 and the measured value is stored in the zero point shift of the rotary table (C)
N70 G0 C0	Rotary table is positioned to zero (G17)

27.13 G634 Measurement center 4 holes

This measurement cycle calculates the intersection point of the connection lines of two cylindrical hole center points and sets this interconnection point as a centre point. At choice the MillPlus can store the interconnection point also in a zero point table.




```
G     Measurement center 4 holes
X     Starting point (meas. point 1)
Y     Starting point (meas. point 1)
Z     Starting point (meas. point 1)
X1=   Measuring point 2
Y1=   Measuring point 2
Z1=   Measuring point 2
X2=   Measuring point 3
Y2=   Measuring point 3
Z2=   Measuring point 3
X3=   Measuring point 4
Y3=   Measuring point 4
Z3=   Measuring point 4
L2=   Safety distance
C1=   Measuring distance
```

```
I5=   G5x offset 0=no 1=X/Y/Z
O1=   E-Par. meas. centre main axis
O2=   E-Par. meas. centre minor axis
F2=   Measuring feed
X4=   Target centre point
Y4=   Target centre point
Z4=   Target centre point
```

X, Y, Z Starting point of the measurement of the 1st hole (or the actual position)
X1=, Y1=, Z1= Starting point of the measurement of the 2nd hole (all 3 coordinates must be entered)
X2=, Y2=, Z2= Starting point of the measurement of the 3rd hole (all 3 coordinates must be entered)
X3=, Y3=, Z3= Starting point of the measurement of the 4th hole (all 3 coordinates must be entered)
C1= Maximum measuring distance
L2= Safety distance
I5= Storing measuring values in a zero point shift:
I5=0 Do not store
I5=1 Store in the active zero point shift of the linear axes (X/Y/Z).
During storing the measuring values are added to the active zero point shift.
X4=, Y4=, Z4= If the measured coordinate is saved in the active zero point offset (I5>0), it is used to calculate the target value.
The measured coordinate thus becomes the target value for subsequent programming.
O1= Number of the E-parameter in which the measured centre point in the main axis is stored.
O2= Number of the E-parameter in which the measured centre point of the minor axis is stored.

The description of the remaining addresses can be found in the introduction to measuring cycles.

Default settings

C1=20, I5=0, F2=MC_0843.

Notes and usage

The starting position must be programmed inside the cylindrical hole.

EASYoperate ⇔ DIN/ISO

In EASYoperate the addresses O1=, O2= and F2= are not available.

Cycle sequence

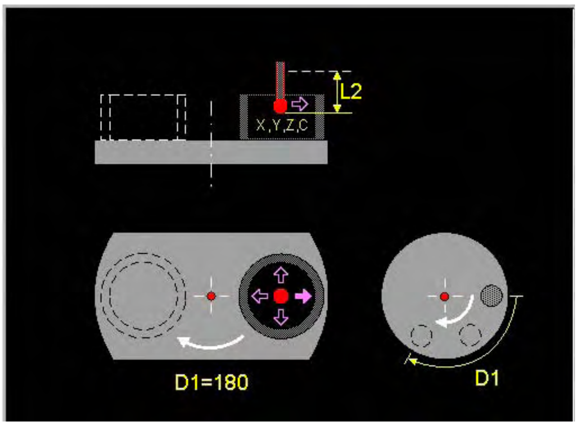
1. Movement with rapid to the first starting point (X, Y, Z) in the 1st cylindrical hole. When X, Y, Z are not programmed the actual position is taken as starting point.
2. Measuring movement with measuring feed (F2=) to the hole side or till the maximum measuring distance (C1=) is reached. The centre point is first measured roughly and than exactly.
3. Movement in rapid back to the starting position. An error message is given when the measuring probe was not triggered within the maximum measuring distance (C1=). Retract movement to the safety distance (L2=)
4. Movement in rapid with regard to the safety distance to the starting point of the 2nd hole.
5. The second hole is measured in the same way.
6. To measure the 3rd and 4th hole the steps 3 and 4 are repeated.
8. At the end a movement in rapid follows to the safety distance (L2=).
9. Depending on I5= the measured value is stored.

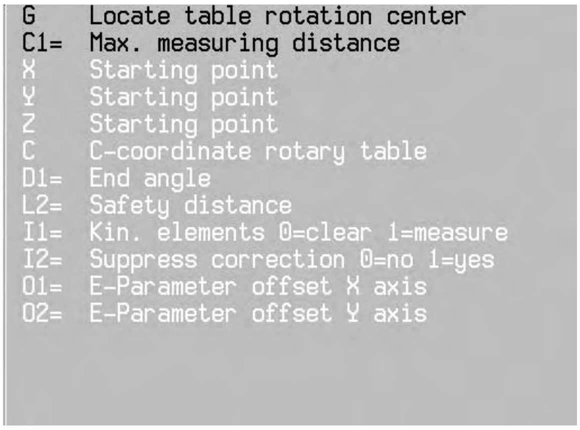
Example: Determine the centre point of 4 cylindrical holes of a work piece

N40 G17	Set the plane
N50 G54 I3	Set the zero point
N60 G634	Define the measuring cycle with
X-10 Y-20 Z-5	Starting point of the 1st hole
X1=-100 Y1=-40 Z1=-5	Starting point of the 2nd hole
X2=-100 Y2=-100 Z2=-5	Starting point of the 3rd hole
X3=-10 Y3=-120 Z3=-5	Starting point of the 4th hole
L2=30 I5=1	Safety distance is 30. After the measuring cycle X and Y in G54 are updated.

27.14 G640 Locate table rotation center.

Measuring and correction of temperature dependant (or small mechanical) table displacements with the help of a measuring probe. (TPC= Table Position Control)
For this measurement a hole in the table or work piece must be present. The probe measures the hole, the table is rotated 180 degrees and the measurement is repeated. The cycle G640 corrects the, from the measurement calculated turning center in both axes.





D1 End angle.
This end angle is necessary by C-axis with limited reach (Z.B. set up table).
When D1 between -180 and +180, the measuring will be done on 3 positions.
When D1 equal –180 or +180 is, the measuring will be done on 2 positions.
When the measuring happens on 3 positions, which are not lying on a circle, but on an arc, the calculation of the table rotation centre is not so precise as with 2 opposite holes.

Basic settings
I1=1, I2=0, L2=0, D1=180

Notes and application
Remarks
- C Axis must be present.
- The starting position must be programmed inside the hole.
- The deviation measured in the X and Y axis, is corrected in the first correction element of the relevant axis in the active kinematics model.
- When G7 is active, X, Y, Z und C must be entered.

It is not allowed to program G640 when:
- G18, G19, G36, G182 are active.
- G54 up to G59 B4= does not equal 0.
- G93 B4= is programmed with A or B or C.
- Tool number T0 is programmed.

G640 activates: G90, G40, G39 L0 R0, G72
G640 deactivates: G7
All measurement movements are performed with the default measuring feed (MC842).

Conditions
- The kinematics model of the machine tool must be entered and must contain the correction elements for X and Y.
- The maximum correction per axis is ± 0.200mm.

Switching on:
The correction elements of the kinematics model are set to zero when switching on the machine tool.

Cycle sequence

1 When G7 is active or the rotary axes are not at the zero point position:
- Retract movement with rapid to the SW-end switch
- G7 is switched off
- B axis and A axis are moved to the zero point position and the tool axis is moved again to the SW end switch

In all other cases:
- Retract movement with rapid to the SW end switch or when programmed to the safety distance (L2=). If the measuring probe is already in the start position (X,Y,Z and C not programmed), this movement is skipped.

2 Movement with rapid to the start position in the hole. Measurement of the center point.
3 Second measurement to measure the center point exactly (sequence depends on the probe type).
4 Retract movement with rapid to the SW end switch or when programmed to the safety distance (L2=). When the hole in the turning center is used, no retract movements occur.
5 The rotary table is rotated over 180°.
6 The hole is measured in the new position in the same way.
7 Retract movement with rapid to the SW end switch or when programmed to the safety distance (L2=).
8 The rotary table is positioned to its original position.
9 The calculated turning center displacement is corrected in the correction elements.
The difference between the old and new correction values is stored in E parameter (O1=, O2=).

When for D1 a value between –180 and +180 is given,
- The hole will be measured on 3 different positions of an arc. First on position C, after that on position C+D1:2 and latest on position C+D1.
- The table rotation centre will be calculated of the 3 centre points of the measured holes.
- When D1 equal 180 or +180 is, than the cycle sequence is equal to 2 measuring points.

Measuring result
The measuring results are written to a text file G640RESU.TXT at D:\startup.
In manual mode (MC320) a window is shown, e.g.:

```
Measured in:        [X]       [Y]

Offset old:        0.015     0.010

Rotat.center:    300.648  -480.043
Offset new:        0.010     0.012
                ---------------------
Sum:             300.658  -480.031



Temperature:      22.3

ESC = close information window
```

Error messages
P421 No correction element available
This error message appears when the relevant correction elements are not entered in the kinematics model.

Machine constants

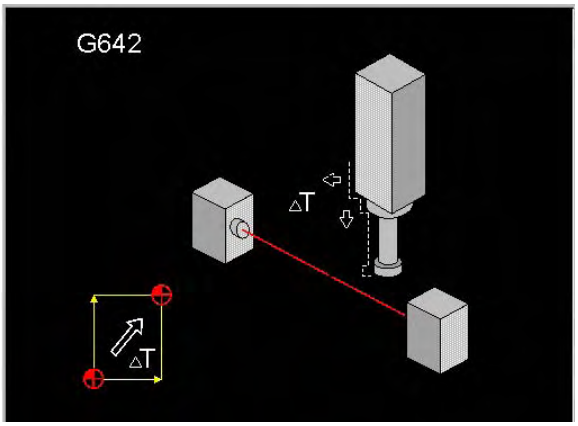
MC843	Measuring feed rate [(µm,mDeg)/min]
MC846	Measuring probe: orientation angles (0,1,2,3=all)
MC849	Measuring probe 1st orientation angle [Deg]

Example

N1 G17	set the surface plane
N2 T2 M6	Change the probe
N3 G0 X.. Y.. X..	Position the probe in the rotary table hole
N4 G640 C1=50 I1=1	Determine turning center The correction elements are corrected

27.15 G642 Laser: Temperature compensation

Measuring and correction of the spindle temperature expansion in 2 axes (HPC, Head Position Control) with the aid of a calibration tool and laser measuring system.
G642 corrects small temperature dependant axes errors. It corrects the radial axis (with respect to the laser), the tool axis and the head kinematics. An advantage is that the measurement is executed with rotating spindle so that the temperature remains stable.



G	Laser: temperature compensation
S	Speed
I2=	Suppress correction 0=no 1=yes
I3=	Textfile 0=overwrite 1=add
O1=	E-Par. temp. deviation rad. axis
O2=	E-Par. temp. deviation tool axis

O1=, O2= Output of the difference between the old and new correction values.

Basic settings
I2=0, I3=0

Notes and application
General

This cycle, used at higher accuracy demands, executes a temperature compensation for the NC-axes with the laser measuring system. The temperature dependant position change, mainly caused by the tool head, is compensated in the radial and axial axes and in the tool head. The errors occur because the automatic temperature compensation with sensor and correction table is calibrated for an average temperature development.
The cycle measures with the aid of a calibration tool the radial and axial positions of the laser beam. The difference with the calibrated laser position is stored in the kinematics chain machine constants to correct these axes.

Notes:

The incorporation of the temperature compensation measurement in the machining sequence should follow the schedule shown below:

1 Establish the turning center of the table with G640. Herewith the kinematics position of the table is corrected. For machine tools without rotary tables this measurement is skipped.
2 Next, calibrate with the calibration tool the laser measuring system (G600) to establish the actual machine kinematics as reference.
3 After this normal operation can take place: Measuring of the tools with the laser measuring system, setting the zero point by hand or with a measuring probe, work piece machining, etc.
4 Execute G642 regular. Depending on the thermal expansion of the machine tool and the required accuracy, the temperature compensation cycle can be executed before every n-th work piece or before a critical machining part.

Remark:
Measuring the kinematics and calibrating (item 1 and 2) is not required when the machine tool is switched on again in a batch production and the previous calibration is still valid.

Conditions:

- The measurement in the temperature compensation cycle G642 must be executed in vertical position. Doing so, the radial axis (in reference to the laser) and the tool axis are measured and corrected. The axis parallel to the laser beam **cannot** be corrected.
- The kinematics model of the machine tool must be entered and must contain correction elements for X, Y and Z. In case a rotary axis or swivel head in the tool head is present, also a correction element for the tool axis in the head must be available.
- The maximum correction per axis is ±0.200mm

Measuring result

The measuring results are written to a test file G642RESU.TXT at D:\startup, e.g.:

Temp	d-Rad	d-Tl	Date	Time
22.3	0.013	0.034	10- 2-2003	10:05
22.4	0.014	0.036	10- 2-2003	10:06

Meaning:
Temp : Temperature of the sensors [ºC].
d-Rad : Deviation, measured in the radial axis [mm|inch]
d-Tl : Deviation measured in the tool axis [mm|inch]

Overwriting or adding the text file (I3=)

When during the cycle call overwrite is selected (I3=0), two lines, head and measuring data are re-written. When add (I3=1) is selected only one line with the measuring data is added. In this way a table is originated where the result of several measurements is visible.

Switching on:

The correction elements are set to zero when switching on the CNC.

Correction of the kinematics model

The deviation measured in the radial axis and tool axis, is corrected in the first correction element of the relevant axis from the table in the active kinematics model.
This correction element behaves like a zero point shift in the relevant axis.

The measures caused by swiveling are corrected separately via a correction element in the head. This measure is not directly measured, but is derived from the correction element in the table in the tool axis with the formula:

head correction = total head correction * MC470 / 100,

where MC470: 'Temperature compensation: head lengthen/ distance [%]'.

Error message

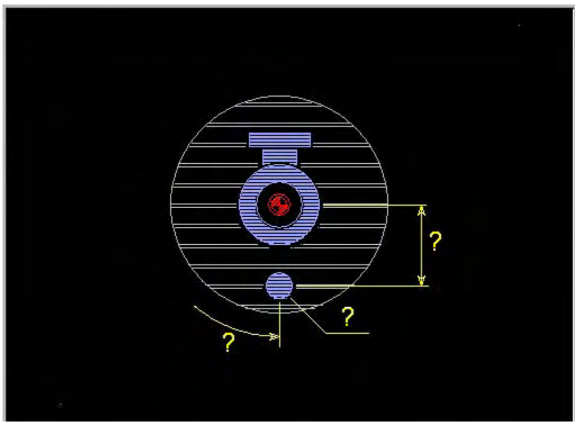
P421 No correction element available

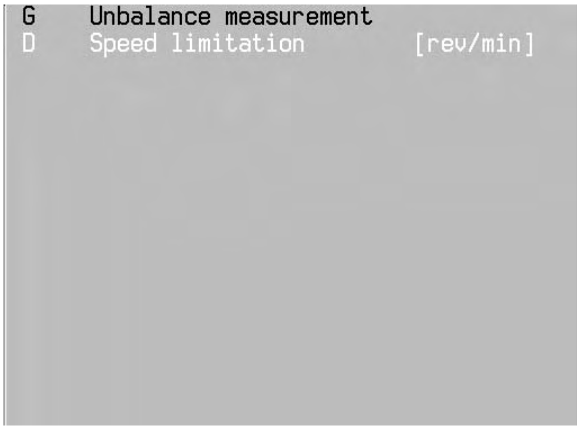
This error message appears when the relevant correction elements are not entered in the kinematics model. When this happens, this G function cannot be used.

28. Specific cycles

G691 Measure unbalance
G692 Unbalance checking

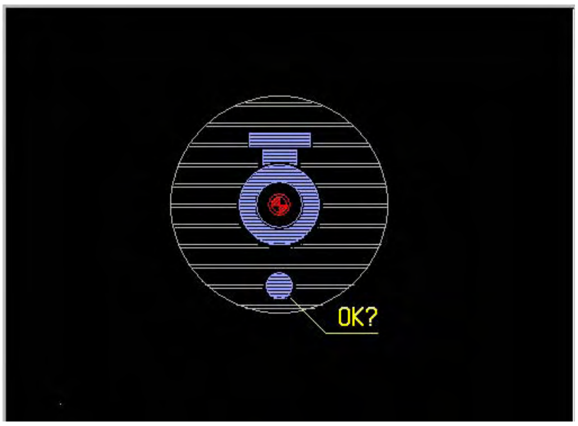
28.1 G691 Measure unbalance.

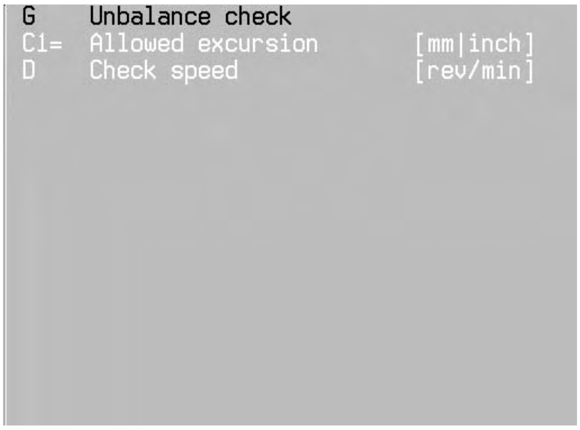




For description see chapter: "Turning".

28.2 G692 Unbalance checking.





For description see chapter: "Turning".

Reserved

29. Machining and positioning cycles

The machining cycle defines a machining departure point. A separate positioning cycle defines execution of the machining cycle at a position.

29.1 Summary of machining and positioning cycles:

Special cycle:

1	G700	Facing	(only in DIN/ISO)
2	G730	Executing a pass	

Positioning cycles (Pattern) (only in EASYoperate):

1	G771	Machining on a line	
2	G772	Machining on a rectangle	
3	G773	Machining on a grid	
4	G777	Machining on a circle	extension of G77
5	G779	Machining at a position	extension of G79

Drilling cycles:

1	G781	Drilling / centering	extension of G81
2	G782	Deep drilling	extension of G83
3	G783	Deep drilling (chip break)	extension of G83 (only in DIN/ISO)
4	G784	Tapping with compensating chuck	extension of G84 (only in EASYoperate)
5	G785	Reaming	extension of G85
6	G786	Hollow boring	extension of G86
7	G790	Reverse countersinking	
8	G794	Interpolating tapping	extension of G84 (only in EASYoperate)

Milling cycles:

1	G787	Pocket milling	extension of G87
2	G788	Slot milling	extension of G88
3	G789	Circular pocket milling	extension of G89
4	G797	Pocket finishing	
5	G798	Slot finishing	
6	G799	Circular pocket finishing	

29.2 Introduction

Machining plane

Cycle programming is independent of the machining plane (G17, G18, G19 and G7).

Tool axis and machining plane

The cycles are carried out in the current main plane G17, G18, G19 or in the inclined plane G7. The working direction of the cycle is determined by the tool axis. The direction of the tool axis can be reversed with G67.

Procedure in EASYoperate:

The machining cycles (special cycle, drilling cycle and milling cycle) are carried out on the patterns defined by the position cycles G77, G79, G771, G772, G773, G777 or G779.

General example:

Machining cycle (drilling cycle):	N... G781 ......
Positioning cycle:	N... G779 X... Y.... Z...

Cycle G781 is carried out in this position, determined by G779.

Procedure in DIN:

The new machining cycles (special cycle, drilling cycle and milling cycle) are only carried out by positioning cycle G79 in one position. Points (P1-P4) are not allowed.

Positioning logic

The tool moves in rapid motion, and depending on G28, using the positioning logic and the 1st setup clearance, to the position (X, Y, Z,) defined by the positioning cycle.

Mirroring and scaling

Mirroring and scaling are not allowed to be activated between a drilling/milling cycle and a positioning cycle.

Deleting cycle data

Cycle data is deleted by M30, the <Cancel program> softkey, the <Reset CNC> softkey or by defining a new cycle.

Switch on spindle

The spindle must be switched on for the cycle to start. F and S in the cycle definition can be overwritten.

Mirroring

If you are only mirroring one axis, the direction of rotation of the tool changes. This does not apply during machining cycles.

Comments

Comments are not allowed in a block with a machining cycle.

Before calling up the cycle, you must program radius correction G40.

Warning

Pre-position the tool so that there can be no collision with the workpiece or clamping devices.

29.3 Description of addresses

Mandatory addresses

Mandatory addresses are shown in black. If a mandatory address is not entered an error message is issued.

Optional addresses

Optional addresses are shown in light grey. If these addresses are not entered they are ignored or given the basic setting that has already been entered.

Explanation of addresses.

The addresses described here are used in most cycles. Specific addresses are described in the cycle.

X, Y, Z: Position of the defined machining geometry

Machining is carried out in this position. If X, Y or Z is not entered, the current position of the tool is adopted.

Execution

The tool moves to the starting point in rapid motion and depending on G28, using positioning logic. If X, Y or Z is not programmed, the current position is taken as the starting point. The first setup clearance (L1=) is taken into account in the tool axis. When going down the lines (G730) the other axes are also displaced.

L Depth (greater than 0) When going down the lines (G730) this is the machining depth: distance between programmed workpiece surface and surface of unmachined part.

R Radius of the circular pocket

L1= 1st setup clearance at start of cycle.

L2= 2nd setup clearance: height above the 1st setup clearance.
At the end of the cycle the tool moves to the 2nd setup clearance (if entered).

C1= Feed depth (> 0): dimension used to adjust the tool each time. The depth (L) or machining depth (L) does not necessarily have to be a multiple of the feed depth (C1=). The CNC moves to the depth in one work pass if the feed depth is the same as or greater than the depth (C1=>L-L3).

Note:

If a feed depth (C1=) is programmed for milling or machining, there is usually a residual cut that is smaller than the programmed feed depth.

For drilling, the last 2 cuts are distributed equally if the residual cut >0. This avoids having a very small last cut.

D3= Dwell time: Number of revolutions for which the tool stays at the base of the hole for free cutting. (Minimum is 0 and maximum is 9.9.)

F2= Rapid plunging motion: traverse speed of tool when moving from setup clearance to the milling depth.

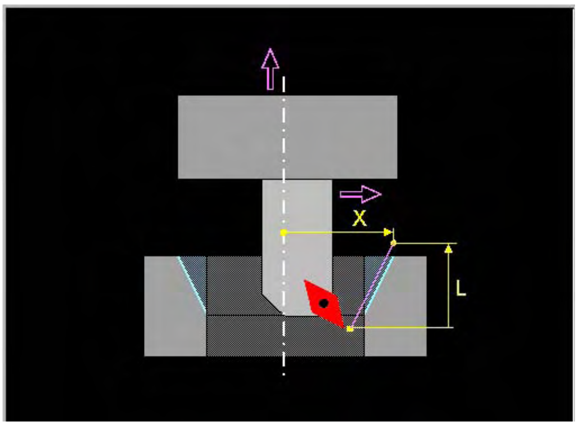
F5= Rapid retraction movement: traverse speed of tool when moving out of the hole.

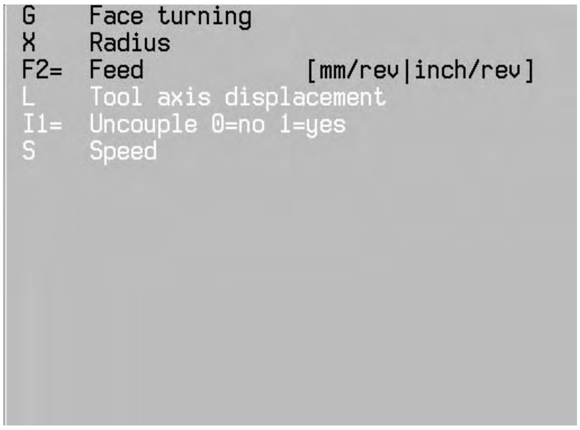
F and S

The addresses F and S are not available in machining cycles within EASYoperate. They must be programmed in the FST menu.

29.4 G700 Facing cycle

Der Plandrehzyklus führt eine einzelne flache oder konische Drehbearbeitung aus.





Basic settings

L0, I1=0

EASYoperate ⇔ DIN/ISO

G700 is not available in EASYoperate.

The following addresses in the tool memory are used by the cycle:

R Adjustment radius. Is automatically overwritten with the current radius after facing.
A1 Orientation angle for engaging. Is automatically overwritten with the current angle (0-359.999 degrees) after facing.
R1 Minimum diameter (optional)
R2 Maximum diameter (optional)

Notes and application

G700 must not be programmed if:

- G36 and/or G182 are active.
- tool T0 is programmed.
- the spindle orientation at an angle is not allowed to be zero.

Resetting the radial facing slide:

The maximum speed allowed can be used to reset the radial facing slide to the starting diameter.

Actual diameter reached:

The programmed diameter is rounded so that it exactly matches one of the 72 indexing positions of the clamp. The maximum difference that this causes is < (feed/72)/2, i.e. 0.001mm deviation for 0.15mm feed/rev.

Note:

G40, G72, G90 and G94 remain active after G700

Block approach

In a block approach the head must be in the correct position before a G700 cycle starts. Therefore the radius R and angle A1 must be correctly entered in the tool table.

Speed and feed correction switch:

The speed correction switch is not active. The feed correction switch is active.

Display:

During movement the speed is displayed in the current S field. At the end the spindle position is always displayed in the range 0-359.999 degrees.
The programmed feed remains unchanged. The current feed displays zero or the feed of the traverse in the tool axis.

The cycle automatically indexes movement in and out:

M82 indexing of **outward** movement (in the facing head). M80 indexing of **inward** movement

Example:

Programming example	Description
	Tool memory: tool radius R20 Tool memory: orientation angle A1=0
N120 G700 X50 L5 F=0.05 S600	Chamfer 5mm from diameter 40 to 50
N140 G700 X70	Facing movement at diameter 70
N130 G0 Z100	Lift off
N140 G700 X40 I1=1 S1200	Return to diameter 40 and disengage

Facing head

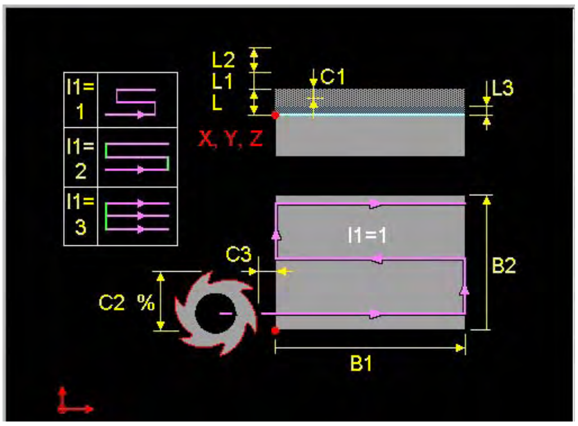
The facing head can be turned into the spindle and then used as a hollow boring head. The bracket is fixed by the indexing device built into the machine and at the same time the locking device between the bracket and facing head is loosened. When the spindle is rotating a mechanical gearing of e.g. 0.1mm per rev causes the radial facing slide to move. The transverse feed is determined by the rotary speed of the spindle. Synchronised movement of the spindle and tool axis (Z) enables cones and chamfers to be turned. Rotate the spindle anticlockwise to reset.

The cycle

1 Set the facing head adjustment radius and enter it into the tool memory.
2 Turn the facing head round in the spindle (the first time, check the engagement angle).
3 Check the orientation and indexing and run out if necessary.
4 The spindle turns, thus carrying out a facing movement.
5 Angle positions in multiples of 5 degrees are approached.
6 The adjustment radius and angle of orientation are automatically written into the tool memory

29.5 G730 Multipass milling

Define a single pass milling cycle in a single program block.




```
G    Multipass milling
B1=  1st Side length
B2=  2nd Side length
L    Height
L1=  1st Setup clearance
L2=  2nd Setup clearance
L3=  Finishing allowance
C1=  Plunging depth
C2=  Proportional cutting width
C3=  Radial setup clearance
I1=  1=meander 2=M.+rapid 3=parallel
F    Feed
S    Speed
F2=  Rapid for plunging
```

B1= Length of 1st side in the main axis (with direction prefix)
B2= Length of 2nd side in the secondary axis (with direction prefix)
L Machining height (>0)
C2= Percentage cutting width: maximum percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections. On the last cut 10% of the diameter of the mill goes over the edge of the material.
C3= radial setup clearance
I1= Method:
- I1=1 Meander
- I1=2 meander and transverse movement out of the material
- I1=3 Machining in the same direction. The directions of B1= and B2= are used to determine whether to mill using forwards or reverse rotation.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=0, C1=L-L3, C2=67%, C3=5, I1=1

The cycle

Method: meander

1 Rapid motion to the 1st setup clearance above the surface of the workpiece. The starting point is the radius of the tool plus the radial setup clearance (C3=) in addition to the programmed position.
2 Rapid plunging movement (F2=) by the feed depth (C1=) to the next depth.
3 After this the tool mills one line in the main axis. The end point of this movement is in the material by the cutting width (C2= maximum 50% of the milling cutter radius). In the last cut the tool travels outside the material by the amount of the radial clearance.
4 The tool moves with transverse milling advance to the starting point of the next pass. In the last pass it moves outside the material by 10% of the milling cutter radius.
5 Repeat steps 3 and 4 until all of the surface that has been defined has been machined.
6 Repeat steps 1 to 6 until the depth (L) has been reached.
7 At the end there is rapid movement to the 1st plus 2nd setup clearances (L1= plus L2=).

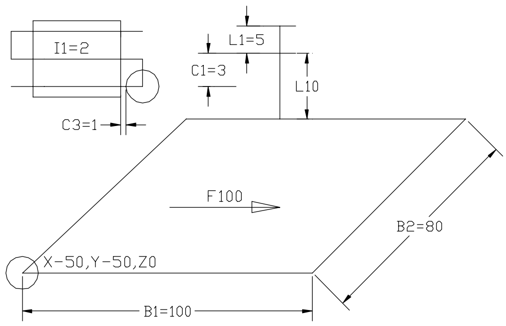
Method: meander and transverse movement out of the material

In this method the end point of each pass is outside the material by the amount of the radial setup clearance. The tool executes the transverse movement rapidly.

Method: milling in the same direction.

In this method the tool mills in the same direction on each pass (forward or reverse rotation). The end point of each pass is outside the material by the amount of the radial setup clearance. The CNC retracts the tool by the 1st setup clearance (L1=) at the end of a line. The tool then moves rapidly back to the main axis and then executes the transverse movement.

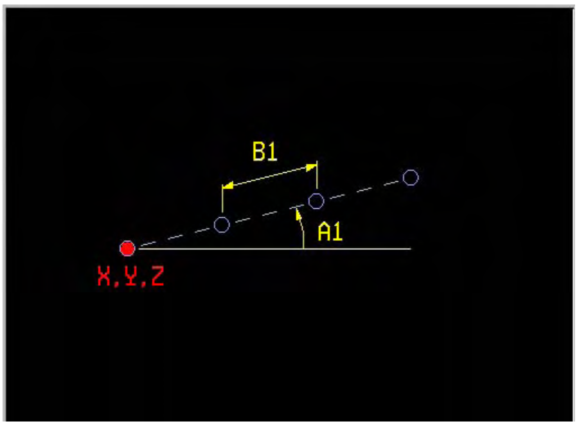
Example

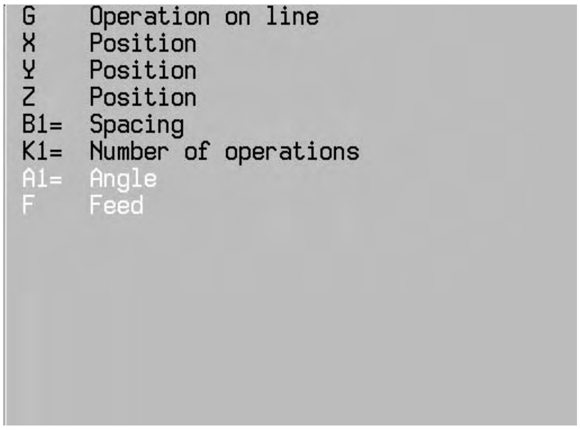


Programming example	Description
N55 T1 M6	Change tool
N60 S500 M3	Switch on spindle
N65 G730 I1=2 B1=100 B2=80 L10 L1=5 C1=3 C2=73 C3=1 F100	Define multipass milling cycle
N70 G79 X-50 Y-50 Z0	Carry out multipass milling cycle

29.6 G771 Machining on a line

Execution of a machining cycle on points that are equally spaced out along a line.





Basic settings

A1=0

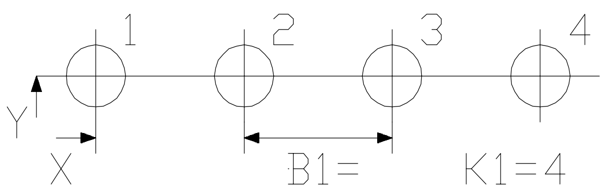
EASYoperate ⇔ DIN/ISO

G771 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position.
4. Repeat steps (2-3) until all positions (K1=) have been machined.

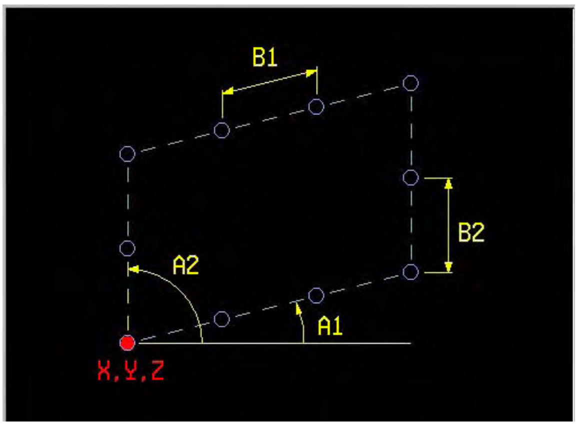
Example

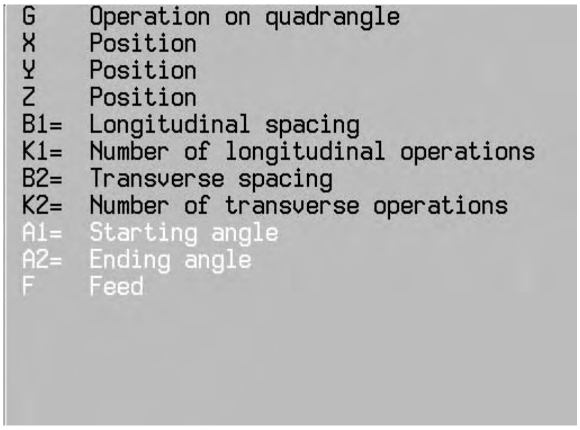


Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G771 X50 Y20 Z0 B1=40 K1=4	Carry out drilling cycle at 4 points

29.7 G772 Machining on a rectangle

Execution of a machining cycle on points that are equally spaced out on a rectangle.





Basic settings

A1=0, A2=90

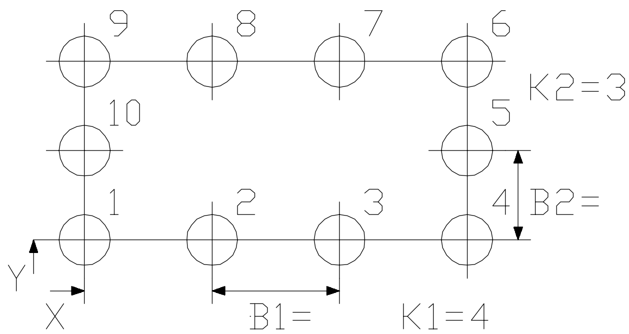
EASYoperate ⇔ DIN/ISO

G772 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position. The direction of the rectangle is determined by the angle A1=.
4. Repeat steps (2-3) until all positions (K1=, K2=) have been machined.

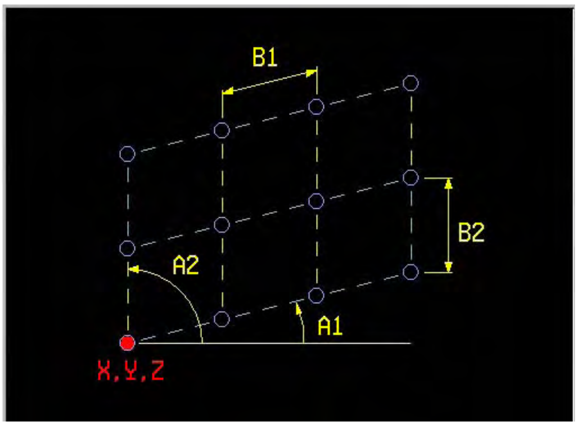
Example

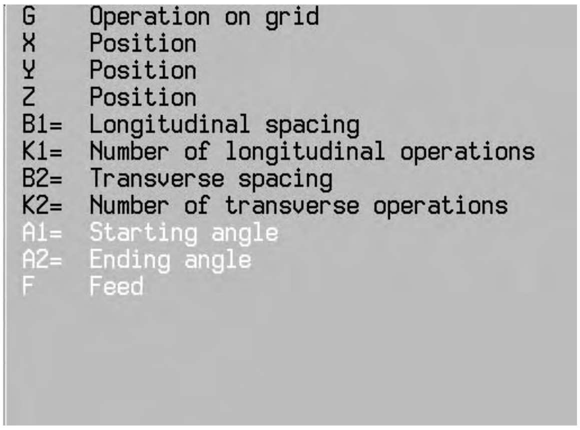


Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G772 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3	Execute the drilling cycle at 10 points on the rectangle

29.8 G773 Machining on a grid

Execution of a machining cycle on points that are equally spaced out on a grid.





Basic settings

A1=0, A2=90

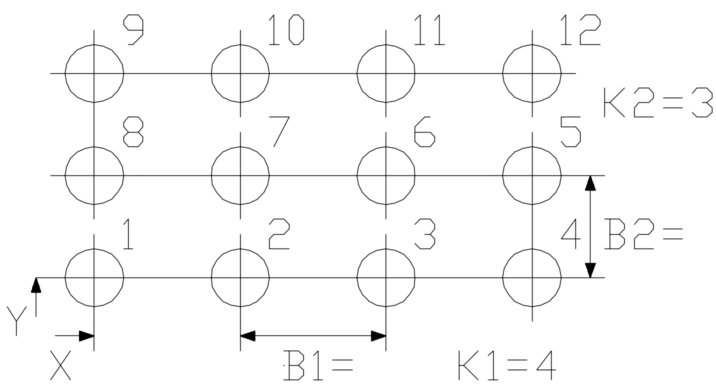
EASYoperate ⇔ DIN/ISO

G773 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position. The tool advances in the initial direction to the positions using a zigzag movement, determined by the angle A1.
4. Repeat steps (2-3) until all positions (K1=, K2=) have been machined.

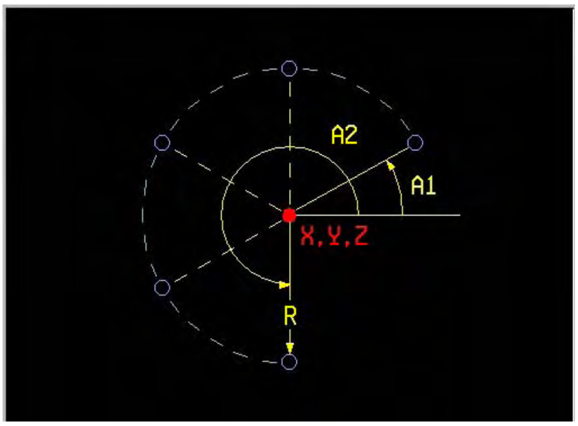
Example

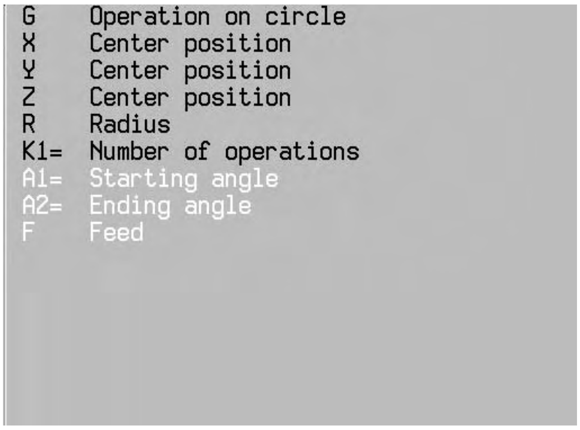


Programming example	Description
N60 T1 M6	Insert tool 1
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G773 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3	Execute the drilling cycle at 10 points on the grid

29.9 G777 Machining on a circle

Execution of a machining cycle on points that are equally spaced out on an arc or a full circle.





Basic settings

A1=0, A2=360

EASYoperate ⇔ DIN/ISO

G777 is only available in EASYoperate.

Note

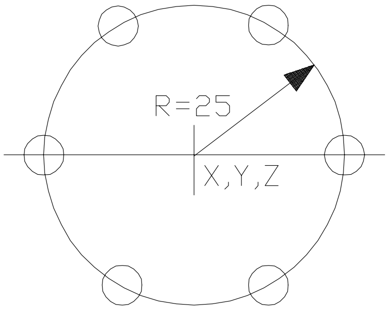
Direction:
If A1= is greater than A2=, the holes are made clockwise.
If A1= is less than or equal to A2=, the holes are made anticlockwise.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.
3. The tool then advances to the next position. The direction of the positions is determined by A1= and A2=.
4. Repeat steps (2-3) until all positions (K1=) have been machined.

Examples

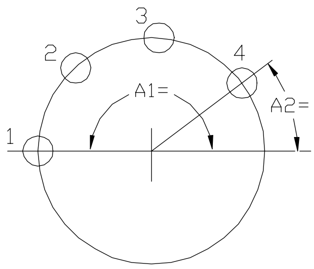
Example 1: Cycle on a full circle



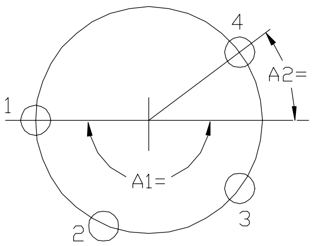
Programming example	**Description**
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G777 X50 Y20 Z0 R=25 K1=6 A1=0 A2=300	Execute the drilling cycle at 6 points on the circle K1=6 Number of holes =6 A1=0 Starting angle = 0 degrees A2=300 Stopping angle = 300 degrees
or	
N75 G777 X50 Y20 Z0 R=25 K1=7 A1=0, A2=360	Execute the drilling cycle at 6 points on the circle K1=7 Number of holes entered =7 Number of holes machined =6 A1=0 Starting angle = 0 degrees A2=360 Stopping angle = 300 degrees

Note: In this case 6 holes are drilled instead of 7, the number entered. The first and last holes in the cycle are in the same position. If an operation has to be carried out a second time in the same position during the cycle, the second operation is not executed.

Example 2 Direction of drilling on an arc



A1 = 180
A1 – A2 > 0 CW

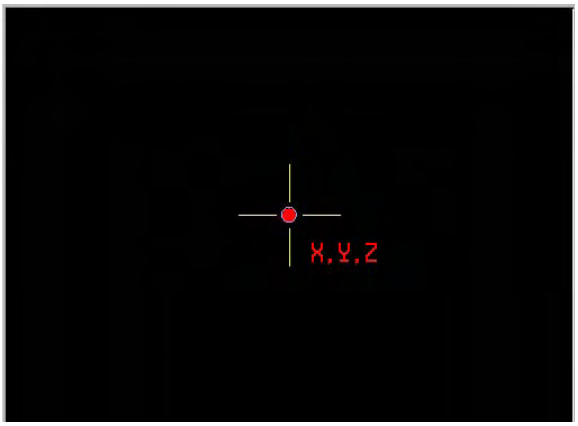


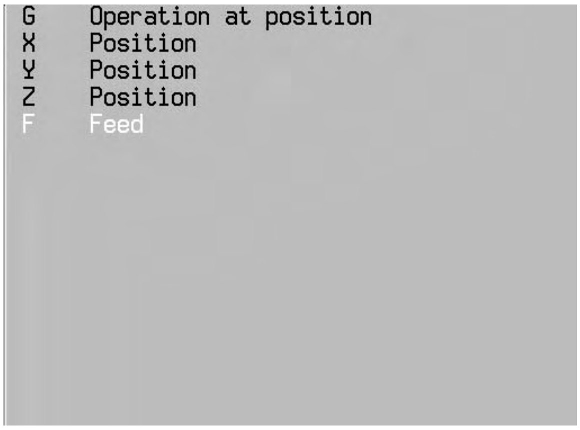
A1 = -180
A1 – A2 < 0 CCW

Programming example	**Description**
N50 G81 Y1 Z-10 F100 S1000 M3	Define cycle
N60 G77 X0 Y0 Z0 R25 **A1=180 A2=30** J4	Repeat the cycle four times on the arc; start at 180 degrees, end at 30 degrees going clockwise (CW).
N70 G77 X0 Y0 Z0 R25 **A1=-180 A2=30** J4	Repeat the cycle four times on the arc; start at 180 degrees, end at 30 degrees going anticlockwise (CCW).

29.10 G779 Machining at a position

Ausführen eines Bearbeitungszyklus auf einer Position.





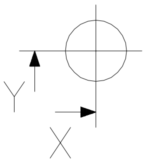
EASYoperate ⇔ DIN/ISO

G779 is only available in EASYoperate.

The cycle

1. Rapid movement into position.
2. The predefined machining cycle is executed at this point.

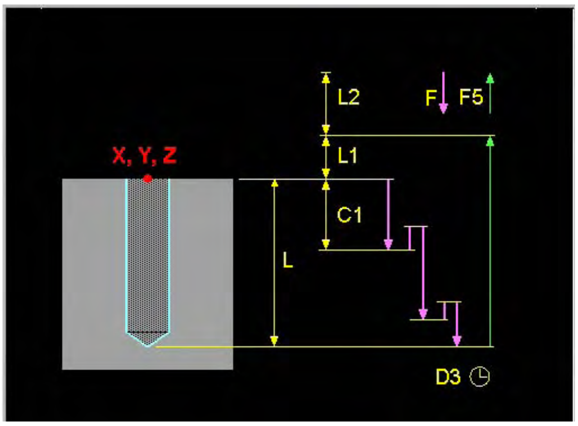
Example

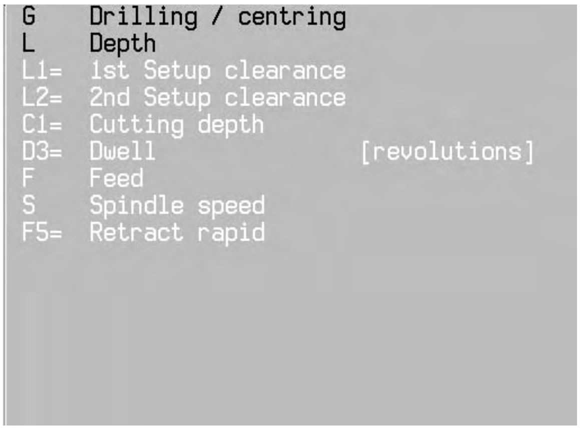


Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L-30 F100 F5=6000	Define drilling cycle
N75 G779 X50 Y20 Z0	Carry out drilling cycle at the point

29.11 G781 Drilling / centring

Define a simple drilling or centring cycle with possible chip break in a single program block.





Basic settings

L1=1, L2=0, C1=L, D3=0

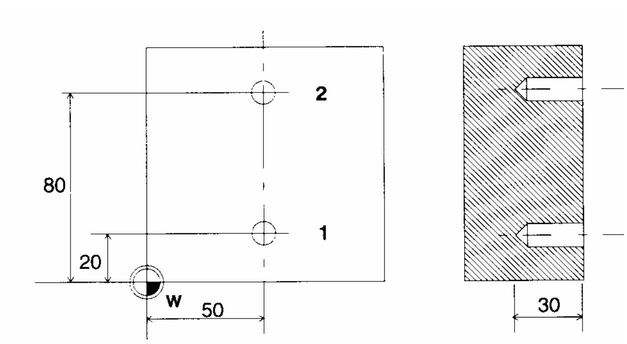
EASYoperate ⇔ DIN/ISO

The addresses D3=, F and S are not available in EASYoperate.

The cycle

1. Rapid motion to the 1st setup clearance (L1=).
2. Drilling with drilling advance by the cutting depth (C1=) or depth (L).
3. Rapid retraction (F5=) of 0.2mm
4. Repeat steps 2 to 3 until the drilling depth (L) has been reached.
5. At the bottom of the hole, dwell (D3=) for free cutting.
6. Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

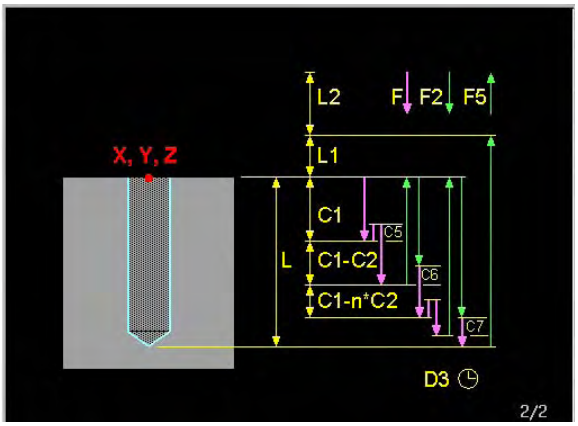
Example



Programming example	Description
N60 T1 M6	Change tool
N65 S500 M3	Switch on spindle
N70 G781 L30 F100 F5=6000	Define drilling cycle
N75 G79 X50 Y20 Z0	Carry out drilling cycle at point 1
N76 G79 X50 Y80 Z0	Carry out drilling cycle at point 2

29.12 G782 Deep hole drilling

Define a deep hole drilling cycle with reducing feed depth for chip break and regular chip removal in a single program block.




```
G     Deep-hole drilling
L     Depth
L1=   1st Setup clearance
L2=   2nd Setup clearance
C1=   Cutting depth
C2=   Cutting depth reduction
C3=   Minimum cutting depth
C5=   Retract distance for chip break.
C6=   Safety distance after retract
C7=   Safety dist. after last retract
K1=   Number of steps before retract
D3=   Dwell                 [revolutions]
F     Feed
S     Spindle speed
F2=   In depth rapid
```

```
F5=   Retract rapid
```

If the cutting depth (C1=) is not programmed or C1= is greater than or equal to the depth (L), the addresses C2=, C3=, C5=, C6=, C7= and K1= are meaningless.
If the number of steps to retraction (K1=) is not programmed or K1=1, the addresses C6= and C7= are meaningless.

With distributed cuts for chip break and/or chip removal.

C2= Value by which the feed depth reduces after every advance. (C1 = C1 - n * C2). The feed depth (C1=) is always greater than or equal to the minimum feed depth (C3=).

C5= Retraction distance for chip break (incremental): distance by which the tool retracts for chip breaking.

Chip removal after a number of cuts:

K1= Number of advance movements (C1=) before the tool moves out of the hole for chip removal. For chip breaking without removal, the tool retracts each time by the retraction distance (C5=). If K1=0 chip removal takes not place.

C6= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the first advance.

C7= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the last advance. If C6= is not equal to C7=, the safety distance between the first and last cuts is gradually reduced.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C1=L, C2=0, C3=C2, C5=0.1, C6=0.5, C7=0.5, K1=1, D3=0

EASYoperate ⇔ DIN/ISO

The addresses C5=, C6=, C7=, K1=, D3=, F and S are not available in EASYoperate.

Notes and application

Rules for distribution of cuts.

1. The cutting depth is always limited by the hole depth (L).
2. If C3 is programmed and there are 2 cuts, the first drilling cut can be reduced.
3. Every cut is smaller than or equal to the preceding one.
4. If there are more than 2 cuts plus a final cut, the final cut and the one preceding it are executed in 2 equal steps. This avoids having a very small final cut.

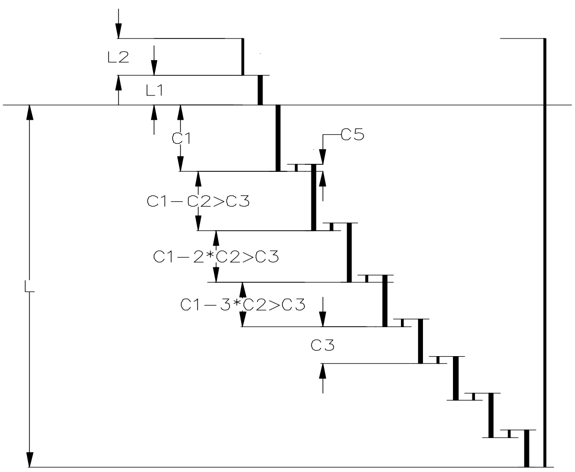
Examples of distribution of cuts.

Programming	Drilling cuts	Instructions or rules
One or two drilling cuts:		
G782 L10 C1=15	**10**	Rule 1
G782 L10 C1=9	**9 1**	
G782 L10 C1=9 C3=2	**8 2**	Rule 2
G782 L10 C1=7 C3=6	**5 5**	Rules 2 and 3
More than 2 drilling cuts		
G782 L25 C1=7	**7 7 5.5 5.5**	Rule 4
G782 L25 C1=7 C2=2	**7 5 3 2 2 2 2 2**	
G782 L24 C1=7 C2=2	**7 5 3 2 2 2 1.5 1.5**	Rule 4
G782 L29 C1=7 C2=2 C3=3	**7 5 3 3 3 3 2.5 2.5**	Rule 4

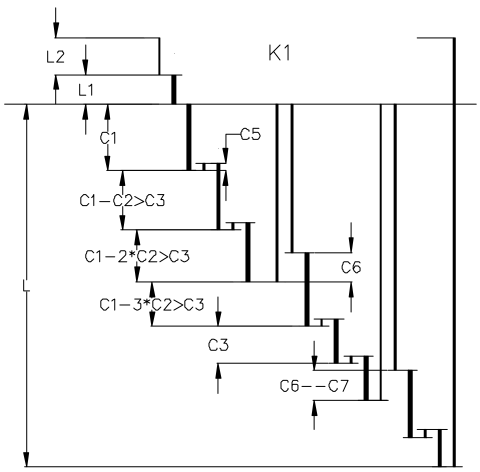
The cycle

1 Rapid motion to the 1st setup clearance (L1).
2 Drilling with drilling advance by the cutting depth (C1=).
3 For chip breaking: reverse movement by the retraction value (C5=).
For chip removal: Rapid retraction (F5=) followed by rapid plunging (F2=) as far as the safety distance (C5= up, to C7= down).
4 The feed depth (C1=) then reduces by the cutting depth reduction (C2=). The minimum feed depth is equal to C3=.
5 Repeat steps 2 to 4 until the drilling depth (L) has been reached.
6 At the bottom of the hole, dwell (D3=) for free cutting.
7 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

Machining sequence

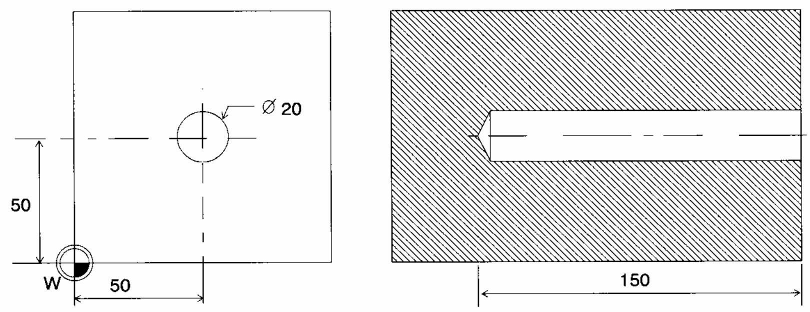


Input: C1=..., K1=large



Input: C1=..., K1=3

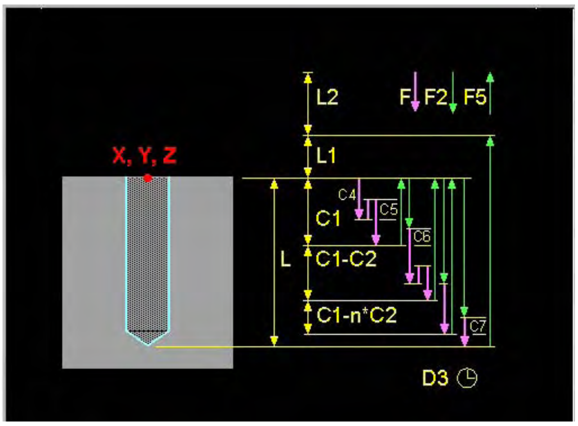
Example



Programming example	Description
N5 T1 M6	Change tool
N10 S500 M3	Switch on spindle
N15 G782 L150 L1=4 C1=20 C2=3 C3=6	Define deep hole drilling cycle
N20 G79 X50 Y50 Z0	Execute deep hole drilling cycle

29.13 G783 Deep drilling (chip breaking)

Define a deep hole drilling cycle with reducing feed depth for chip removal and a fixed chip break distance in a single program block.



G	Deep-hole drill. add. chip break.
L	Depth
L1=	1st Setup clearance
L2=	2nd Setup clearance
C1=	Cutting depth
C2=	Cutting depth reduction
C3=	Minimum cutting depth
C4=	Drilling depth before chip break.
C5=	Retract distance for chip break.
C6=	Safety distance after retract
C7=	Safety dist. after last retract
D3=	Dwell [revolutions]
F	Feed
S	Spindle speed
F2=	In depth rapid
F5=	Retract rapid

If the cutting depth (C1=) is not programmed or C1= is greater than or equal to the depth (L), the addresses C2=, C3=, C4=, C5=, C6= and C7= are meaningless.
If the drilling depth before chip break (C4=) is not programmed or C4= is greater than or equal to the hole depth (L), the addresses C6= and C7= are meaningless.

C4= Advance after which a chip break is performed. If C4>C1 or is not programmed there is no chip break.

C6= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the first advance.

C7= Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the last advance.
If C6= is not equal to C7=, the safety distance between the first and last cuts is gradually reduced.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C1=L, C2=0, C3=C1, C4=C1, C5=0.1, C6=0.5, C7=C6, D3=0

Notes

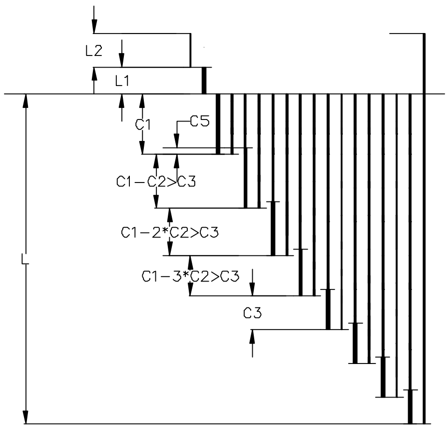
Cutting depth:
If more than 2 cuts are required the final cut and the one preceding it are executed in 2 equal steps. This avoids having a very small final cut.

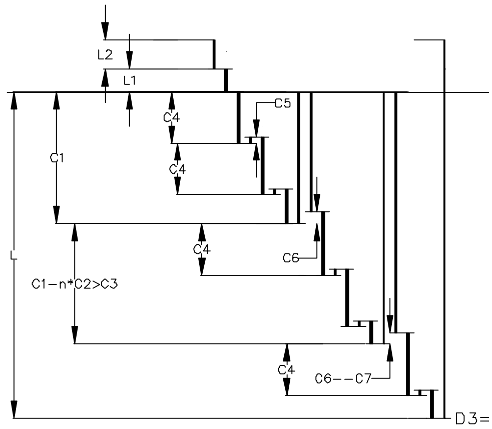
The cycle

1 Rapid motion to the 1st setup clearance.
2 No chip break (C4>C1 or C4 not programmed: drilling with drilling advance by the cutting depth (C1=).
With chip break (0 < C4 < C1): drill to depth (C4=). After this, retract by the retraction distance (C5=).
Repeat until the cutting depth (C1=) is reached.
3 Rapid retraction (F5=) followed by rapid plunging (F2=) as far as the safety distance (C5= up, to C7= down).
4 The feed depth (C1=) then reduces by the cutting depth reduction (C2=). The minimum feed depth is equal to C3=.
5 Repeat steps 2 to 4 until the drilling depth (L) has been reached.
6 At the bottom of the hole, dwell (D3=) for free cutting.

7 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

Machining sequence

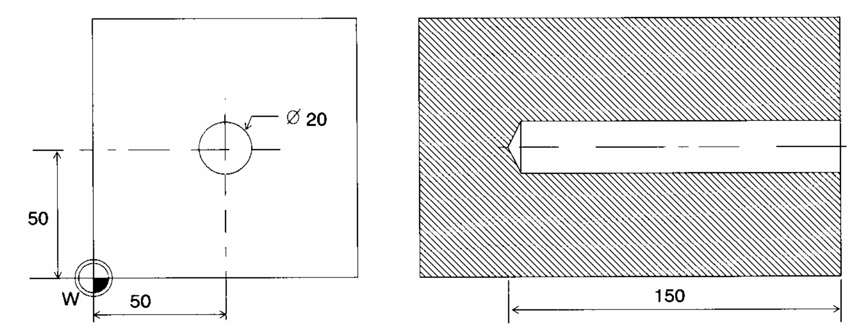




Input: C1=.., C4=C1

Input: C1=.., C4<C1

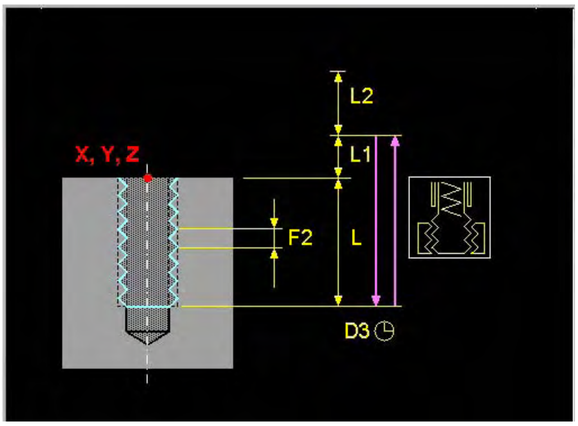
Example

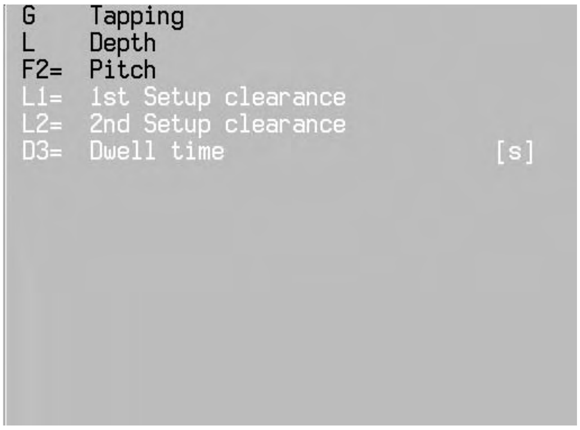


Programming example	Description
N5 T1 M6	Change tool
N10 S500 M3	Switch on spindle
N15 G783 L150 L1=4 C1=20 C4=5 C2=2 C3=6 C5=0.5 F200	Define deep hole drilling cycle
N20 G79 X50 Y50 Z0	Execute deep hole drilling cycle

29.14 G784 Tapping with compensating chuck

Define a tapping cycle in a single program block.





L Depth (> 0)
L1= Guideline value: 4x pitch
D3= Length of time in seconds that the tool dwells at the bottom of the hole.

Basic settings
L1=1, L2=0, D3=0

EASYoperate ⇔ DIN/ISO
G784 is only available in EASYoperate.

Notes and application:
The tool must be clamped in a linear compensation chuck. A linear compensation chuck compensates for the advance and speed tolerances during machining.

At the end of the cycle the coolant and spindle are restored to their status before the cycle.

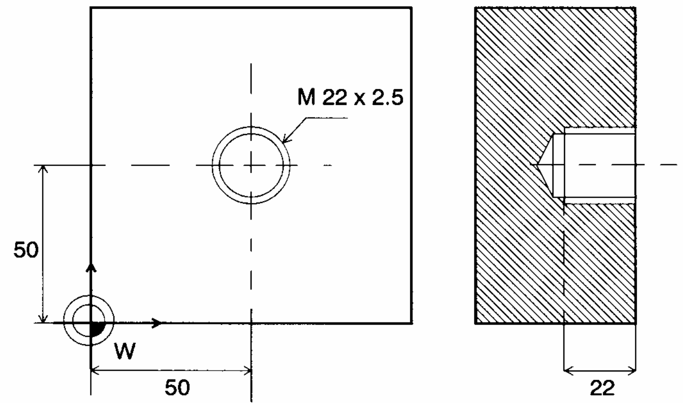
The advance is determined by the speed. Speed override is active during tapping. Feed override is not active.

When a G784 cycle is called up using G79 the CNC must be set to G94 mode (advance in mm/min), not G95 (advance in mm/rev).

Machine and CNC must be prepared for the G784 cycle by the machine builder.

The cycle
1. Rapid motion in the spindle axis to the 1st setup clearance (L1=).
2. Tapping with pitch (L3=) to depth (L).
3. After the dwell time (D3=) the direction of spindle rotation is reversed.
4. The tool is retracted with the pitch (L3=) to the 1st setup clearance (L1=) and then rapidly retracted to the 2nd setup clearance (L2=).
5. At the end the direction of spindle rotation is reversed once more.

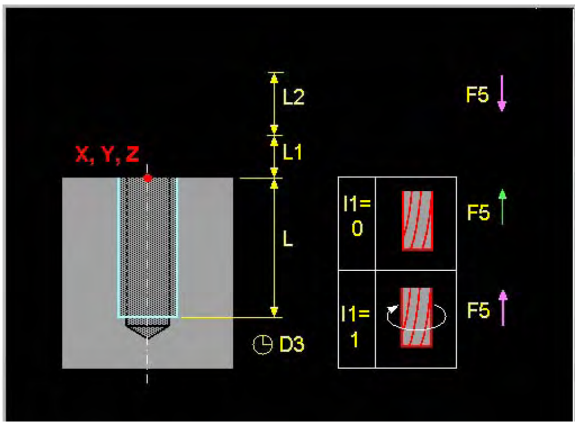
Example

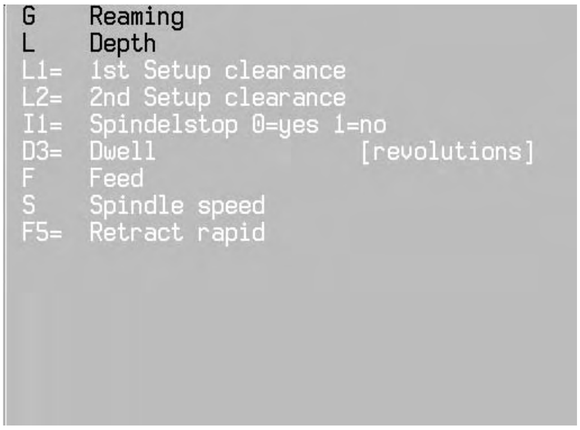


Programming example	Description
N13 T3 M6	Insert tool 3
N14 S56 M3	Switch on spindle
N15 G784 L22 L1=9 L3=2.5	Define the tapping cycle A linear compensation chuck must be used.
N20 G79 X50 Y50 Z0	Execute the cycle at the programmed position

29.15 G785 Reaming

Define a single pass reaming cycle in a single program block.





I1= 0: Retraction with rapid movement and stationary spindle
1: Retraction with advance and rotating spindle

F5= Rapid movement (I1=0) or advance (I1=1) retraction: Traverse speed of tool when moving out of the hole in mm/min.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, I1=0, D3=0

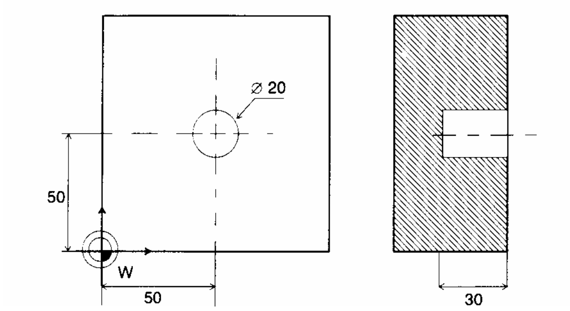
EASYoperate ⬄ DIN/ISO

The addresses D3=, F and S are not available in EASYoperate.

The cycle

1 Rapid motion to the 1st setup clearance (L1=).
2 Reaming with advance F down to depth (L).
3 At the bottom of the hole, dwell (D3=).
4 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

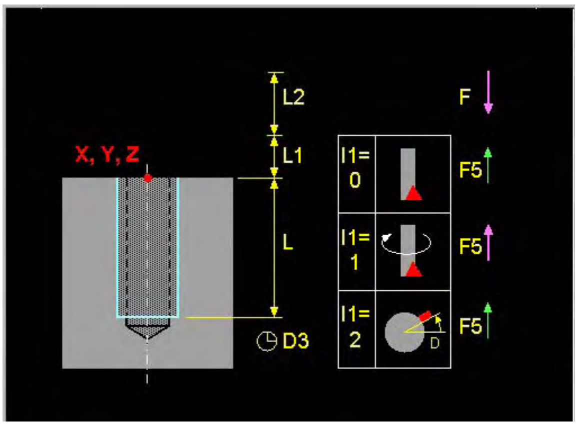
Example

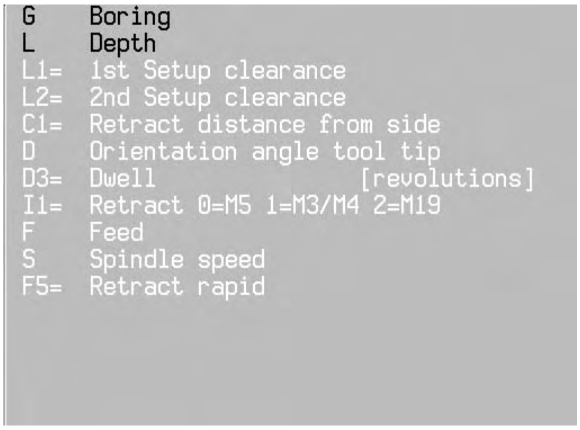


Programming example	Description
N25 T4 M6	Change tool
N30 S1000 M3	Switch on spindle
N35 G785 L29 D3=2 F100 F5=2000	Define reaming cycle
N34 G79 X50 Y50 Z0	Execute the reaming cycle at the programmed position

29.16 G786 Boring

Define a cycle with the option to move clear with an oriented spindle in a single program block.





C1= Distance by which the tool is retracted from the wall when moving clear.
I1= 0: retract with rapid movement and stationary spindle without moving clear.
1: retract with advance movement and rotating spindle without moving clear.
2: with oriented spindle (M19) and rapid retraction.
D Angle (absolute) at which the tool positions itself before moving clear (I1=2 only). The direction of moving clear is –X in G17/G18 and –Y in G19.
F5= Rapid movement (I1=0 or I1=2) or advance (I1=1) retraction: Traverse speed of tool when moving out of the hole in mm/min.
The other addresses are described in the introduction to the machining cycles.

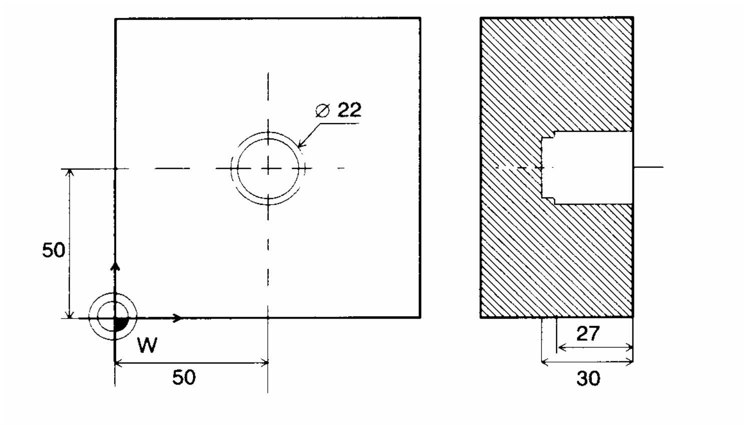
Basic settings
L1=1, L2=0, C1=0.2, D=0, D3=0, I1=0, F5=rapid motion (I1=0 or I1=2) or F5=F (I1=1)

Notes and application
At the end of the cycle the spindle status that was active before the cycle is reactivated.

Risk of collision
The direction of the tool tip (MDI) should be such that it points to the positive main axis. The angle displayed should be entered as the orientation angle (D) so that the tool moves away from the edge of the hole in the direction of the negative main axis. The direction of moving clear is –X in G17/G18 and –Y in G19.

The cycle
1 Rapid motion to the 1st setup clearance (L1=).
2 Reverse boring with advance (F) down to depth (L).
3 At the bottom of the hole, dwell (D3=) with running spindle for free cutting.
4 With I1=2 there is spindle orientation (D=) and a reverse movement along the main axis to the retraction distance (C1=).
5 Rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

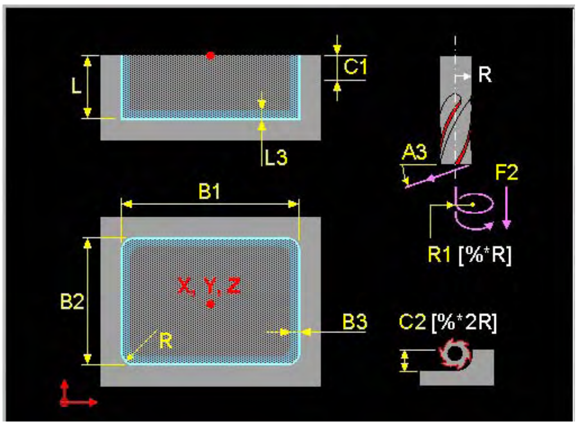
Example



Programming example	Description
N45 T5 M6	Change tool
N50 S500 M3	Switch on spindle
N55 G786 L27 L1=4 L2=10 D3=1 F100	Define reverse boring cycle
N60 G79 X50 Y50 Z0	Execute the cycle at the programmed position

29.17 G787 Pocket milling

Define a pocket milling cycle for rough machining of rectangular pockets in a single program block. This cycle allows oblique plunging and mills in a continuous spiral path.




```
G    Pocket milling
B1=  1st Side length
B2=  2nd Side length
L    Depth
L1=  1st Setup clearance
L2=  2nd Setup clearance
L3=  Finishing allowance bottom
B3=  Finishing allowance sides
C1=  Plunging depth
C2=  Proportional cutting width
R    Rounding radius
R1=  Proportional helix radius
A3=  Plunging angle
I1=  Milling 1=climb -1=conventional
F    Feed
```

```
S    Speed
F2=  Feed for plunging
```

B1=	Length of the pocket in the main axis.
B2=	Width of the pockets in the secondary axis.
C2=	Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
R	Radius for the corners of the pocket. Where radius R=0, the rounding radius is the same as the tool radius.
R1=	Percentage of the tool diameter to be used as the cutting width (>0) on oblique plunging.
A3=	Angle (0 to 90º) at which the tool can plunge into the workpiece. The plunging angle is adjusted so that the tool always plunges with a whole number of rectangular movements. It only plunges vertically at 90º.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=0, B3=0, C1=L, C2=67%, R= tool radius, R1=80%, A3=90, I1=1, F2=0.5*F for vertical plunging F2=F for oblique plunging.

Notes and application

B1= and B2= must be greater than 2*(tool radius + finishing allowance for sides B3).

For finishing, the dimensions L3 and B3 must be entered.

The cycle

1 Rapid motion to the 1st setup clearance (L1=) above the centre of the pocket.

2 If the plunging angle A3=90º, the tool advances with feed (F2=) to the first feed depth (C1=).
If the plunging angle A3<90º, the tool advances obliquely, using a whole number of rectangular movements, to the first feed depth (C1=) with plunging feed (F2=).

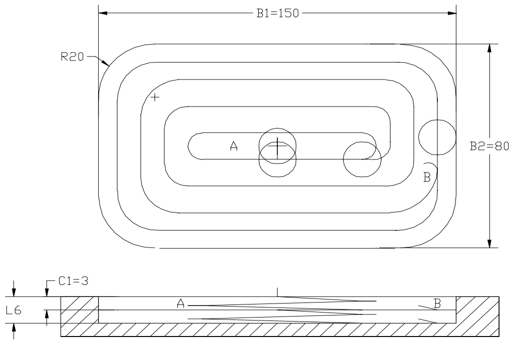
3 Machining with feed (F) in the positive direction of the long side, in a flowing movement from inside to outside.

4 At the end of this process the tool is retracted from the wall and the floor in a tangent to the helix and brought rapidly to the centre.

5 Repeat steps 2 to 4 until the depth (L) has been reached.

6 At the end there is rapid movement to the 1st plus 2nd setup clearances (L1= plus L2=).

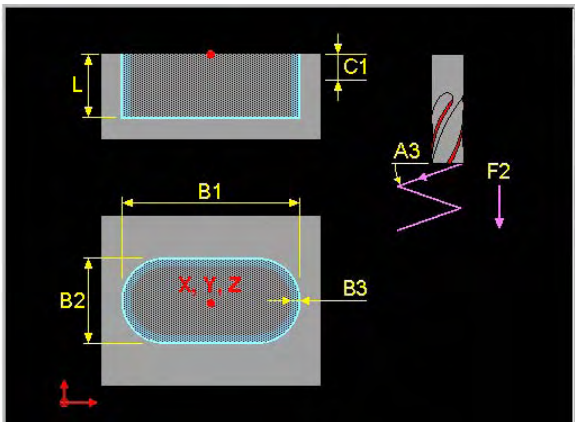
Example

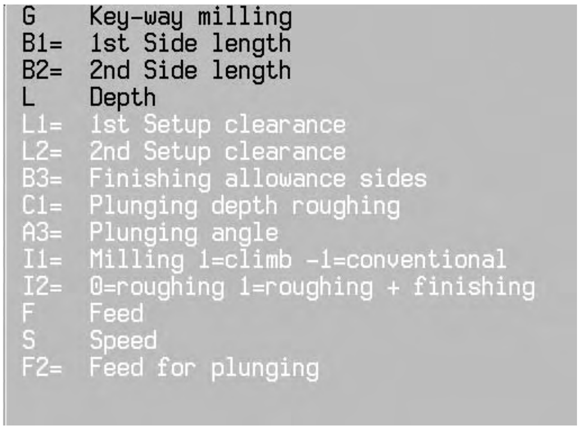


Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3	Switch on spindle
N30 G787 B1=150 B2=80 L6 L1=1 A3=5 C1=3 C2=60 R20 I1=1 F200	Define pocket milling cycle
N40 G79 X160 Y120 Z0	Execute the cycle at the programmed position

29.18 G788 Key-way milling

Define a pocket milling cycle for rough machining and/or finishing of a slot in a single program block. This cycle allows oblique plunging.





B1= Length of slot in the main axis
B2= Width of the slot in the secondary axis. If the slot width is the same as the tool diameter it is only roughed.
A3= Maximum angle (0 to 90º) at which the tool can plunge into the workpiece. It only plunges vertically at 90º.
I2= 0: Roughing only.
1: Roughing and finishing.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, B3=0, C1=L, A3=90, I1=1, I2=0, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

Notes and application

- When roughing with oblique plunging, there is a pendulum effect as the tool plunges into the material from one end of the slot to the other. There is thus no need to pre-drill.
- Vertical plunging always takes place into the end of the slot on the negative side. Pre-drilling is required at this point.
- Choose a milling cutter whose diameter is no greater than the width of the slot and no smaller than a third of the slot width.
- The diameter of the milling cutter chosen must be less than half the length of the slot, otherwise the CNC cannot use the pendulum effect for plunging.
- For finishing the dimension (B3=) must be entered.

The cycle

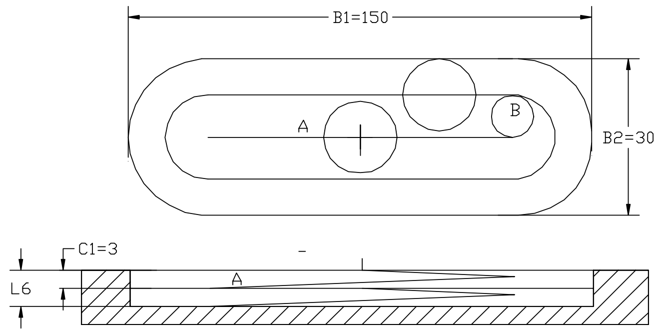
Roughing:

1. Rapid motion to the 1st setup clearance (L1=) and into the centre of the left circle.
2. If the plunging angle A3=90º, the tool advances with feed (F2=) to the first feed depth (C1=) and then with feed F into the centre of the right circle.
   If the plunging angle A3<90º, the tool advances obliquely, with plunging feed (F2=), using oblique motion, into the centre of the right circle. The tool then moves back to the centre of the left circle, again plunging obliquely. These steps are repeated until the cutting depth (C1=) is reached. .
3. At the milling depth, the tool moves to the other end of the slot and then machines the slot shape until the finishing dimension is reached.
4. Repeat steps 2 to 3 until the programmed depth (L) has been reached.

Finishing:

5. The tool moves tangentially in the left or right circle of the slot at the contour and finishes it in forwards rotation (I1=1).
6. At the end of the contour the tool moves tangentially away from the contour and floor to the centre of the slot.
7. At the end there is rapid movement to the 1st plus 2nd setup clearances (L1= plus L2=).

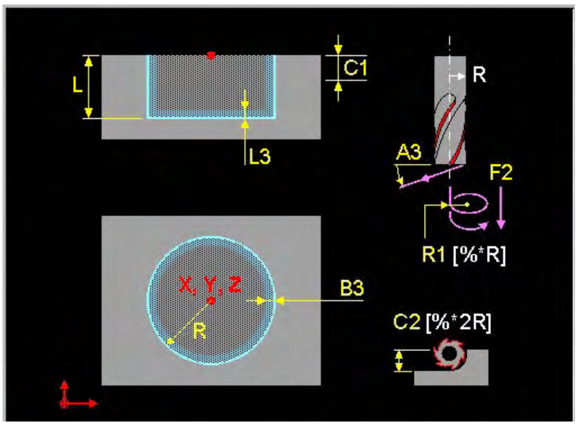
Exemple



Programming example	Description
N10 T1 M6 (R10 milling cutter)	Change tool
N15 S500 M3	Switch on spindle
N20 G788 B1=150 B2=30 L6 L1=1 A3=5 C1=3 I1=1 I2=0 F200	Define the slot milling cycle, parallel to the X axis
N30 G79 X20 Y20 Z0	Execute the cycle at the programmed position

29.19 G789 Circular pocket milling

Define a pocket milling cycle for rough machining of circular pockets in a single program block. This cycle allows oblique plunging and mills a continuous spiral path.




```
G    Circular pocket milling
R    Radius
L    Depth
L1=  1st Setup clearance
L2=  2nd Setup clearance
L3=  Finishing allowance bottom
B3=  Finishing allowance sides
C1=  Plunging depth
C2=  Proportional cutting width
R1=  Proportional helix radius
A3=  Plunging angle
I1=  Milling 1=climb -1=conventional
F    Feed
S    Speed
F2=  Feed for plunging
```

C2= Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
R1= Percentage of the tool diameter to be used as the cutting width (>0) on oblique plunging.
A3= Angle (0 to 90º) at which the tool can plunge into the workpiece. It only plunges vertically at 90º.

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=0, B3=0, C1=L, C2=67%, R1=80%, A3=90, I1=1, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

Notes and application

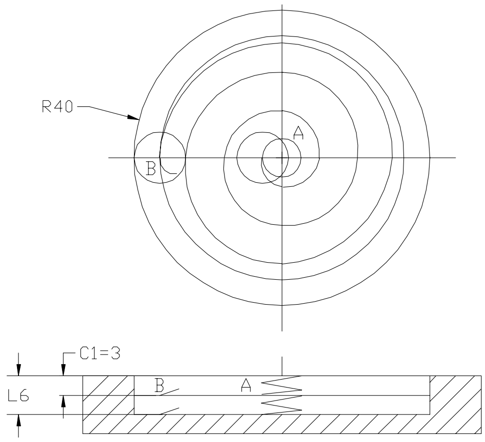
R must be greater than 2*(tool radius + finishing allowance for sides B3=).

For finishing, the dimensions L3 and B3 must be entered.

The cycle

1. Rapid motion to the 1st setup clearance (L1=) above the centre of the pocket.
2. If the plunging angle A3=90º, the tool advances with feed (F2=) to the first feed depth (C1=).
   If the plunging angle A3<90º, the tool advances obliquely with plunging feed (F2=), using a number of circular movements, to the first feed depth (C1=).
3. Machining with feed (F) in an outwards-moving spiral.
4. At the end of this process the tool is retracted from the wall and the floor in a tangent to the helix and brought rapidly to the centre.
5. Repeat steps 2 to 4 until the depth (L) has been reached.
6. At the end there is rapid movement to the 1st plus 2nd setup clearances (L1= plus L2=).

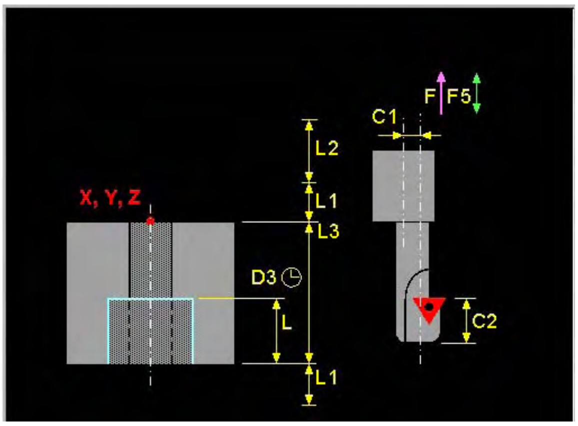
Example

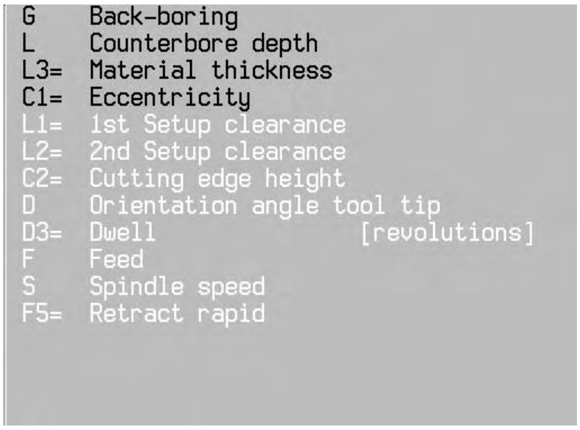


Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3	Switch on spindle
N30 G789 R40 L=6 L1=1 A3=5 C1=3 C2=65 I1=1 F200	Define pocket milling cycle
N40 G79 X160 Y120 Z0	Execute the cycle at the programmed position

29.20 G790 Back-boring

Define a reverse countersinking cycle in a single program block.
The cycle only operates with reverse boring bars to create countersinks on the underside of the workpiece.





L3= Thickness of workpiece
C1= Eccentricity of the boring bar (to be taken from the tool data sheet)
C2= Distance from bottom edge of boring bar to main cutter (to be taken from the tool data sheet)
D Angle (absolute) at which the tool positions itself before plunging and before moving out of the hole. The direction of moving clear is –X in G17/G18 and –Y in G19.
The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C2=0, D=0, D3=0.2, F5=rapid motion

Notes and application

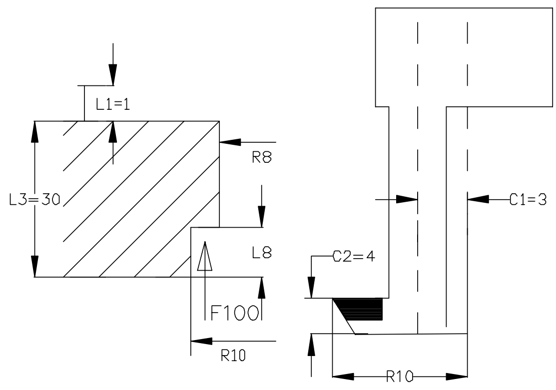
Enter the tool length so that the cutting edge of the boring bar is dimensioned.
The CNC takes the height of the cutting edge (C2=) into account when calculating the starting point.
At the end of the cycle the spindle status that was active before the cycle was called up is reactivated.

Risk of collision
The direction of the tool tip (MDI) should be such that it points to the positive main axis. The angle displayed should be entered as the orientation angle (D) so that the tool moves away from the edge of the hole in the direction of the negative main axis. The direction of moving clear is –X in G17/G18 and –Y in G19.

The cycle

1 Rapid motion to the 1st setup clearance (L1=).
2 Spindle orientation to the D position and tool offset by the eccentricity dimension (C1=).
3 Rapid retract (F5=) plunging into the pre-drilled hole until the cutting edge is at the 1st setup clearance (L1=) below the bottom of the workpiece.
4 Movement to the centre of the hole, switch on spindle and coolant and machine at countersinking feed to the depth that has been entered.
5 At the bottom of the hole, the tool dwells with running spindle for free cutting.
6 The tool then moves out of the hole, performs spindle orientation and is once again displaced by the eccentricity dimension (C1=).
7 At the end, rapid retraction (F5=) to 1st setup clearance (L1=) followed by rapid movement to 2nd setup clearance (L2=).

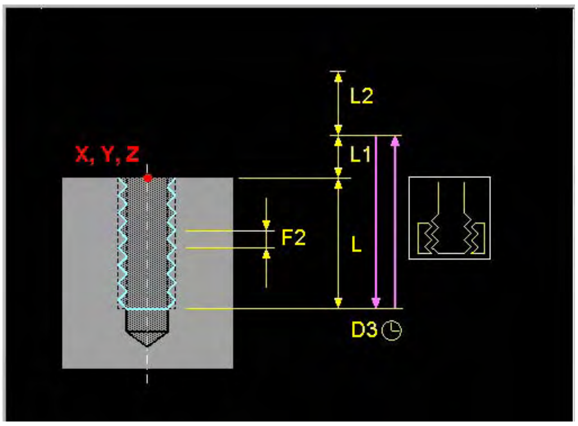
Example

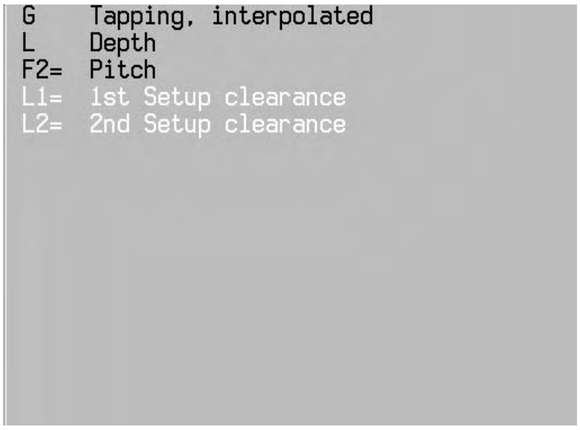


Programming example	Description
N60 T1 M6	Change tool (Tool radius R10, eccentricity C1=3, cutting edge height C2=4, angle for spindle orientation D0)
N65 S500 M3	Switch on spindle
N70 G790 L3=30 L8 L1=1 C1=3 C2=4 F100	Define reverse countersinking cycle
N75 G79 X30 Y40 Z0	Carry out defined cycle at the point

29.21 G794 Interpolated tapping

Define a tapping cycle with interpolation in a single program block.





Basic settings

L1=1, L2=0

EASYoperate ⇔ DIN/ISO

G794 is only available in EASYoperate.

Notes and application:

At the end of the cycle the coolant status and spindle status that were active before the cycle are reactivated.

The advance is determined by the speed. Speed override is active during tapping. Feed override is not active.

When a G794 cycle is called up using G79 the CNC must be set to G94 mode (advance in mm/min).

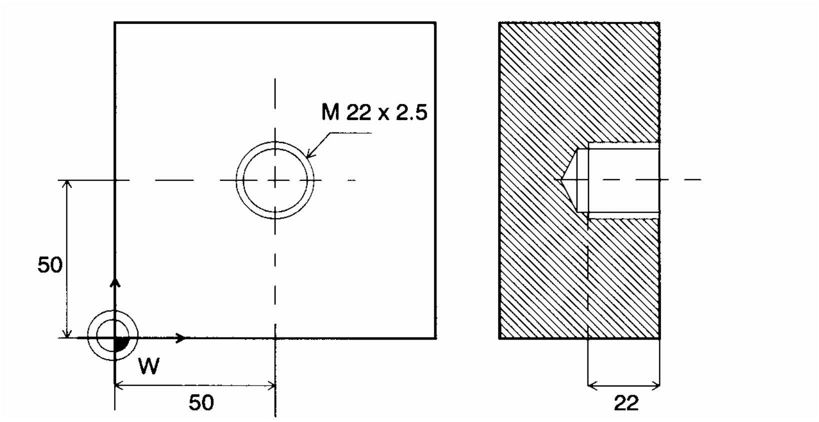
The spindle machine constants for interpolation should be correctly set during tapping. The spindle acceleration for each gear is calculated using MC2491, 2521, 2551, 2581 and MC2495, 2525, 2555, 2585. MC4430 should also be active in all cases to ensure proper adjustment.

Machine and CNC must be prepared for the G794 cycle by the machine builder.

The cycle

1 Rapid motion in the spindle axis to the 1st setup clearance (L1=) and spindle orientation once there.
2 Tapping with pitch (L3=) to depth (L).
3 The direction of spindle rotation is then reversed once more.
4 The tool is retracted with the pitch (L3=) to the 1st setup clearance (L1=) and then rapidly retracted to the 2nd setup clearance (L2=).
5 The spindle is stopped here.

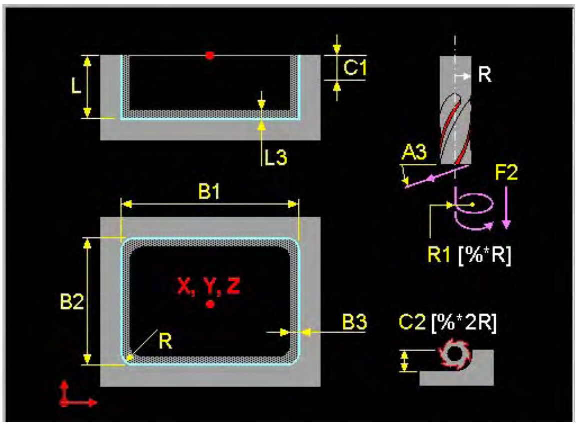
Example



Programming example	Description
N13 T3 M6	Insert tool 3
N14 S56 M3	Switch on spindle
N15 G794 L22 L1=9 L3=2.5	Define the tapping cycle
N20 G79 X50 Y50 Z0	Execute the cycle at the programmed position

29.22 G797 Pocket finishing

Define a rectangular pocket milling cycle for finishing the wall and floor of rectangular pockets in a single program block. The sides can be machined in a number of advances. This cycle allows oblique plunging into the floor and mills in a continuous spiral path.



G	Pocket finishing
B1=	1st Side length
B2=	2nd Side length
L	Depth
L1=	1st Setup clearance
L2=	2nd Setup clearance
L3=	Allowance bottom
B3=	Allowance sides
C1=	Plunging depth
C2=	Proportional cutting width
R	Rounding radius
R1=	Proportional helix radius
A3=	Plunging angle
I1=	Milling 1=climb -1=conventional
I2=	Finishing 0=complete 1=sides

F	Feed
S	Speed
F2=	Feed for plunging

B1= Length of the pocket in the main axis.
B2= Width of the pocket in the secondary axis
B3= Allowance sides, which will be removed by finishing.
L3= Allowance bottom, which will be removed by finishing.
C2= Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
R Radius for the corners of the pocket. Where radius R=0, the rounding radius is the same as the tool radius.
R1= Percentage of the tool diameter to be used as the helix radius (>0) on oblique plunging.
A3= Angle (0 to 90º) at which the tool can plunge into the workpiece. The plunging angle is adjusted so that the tool always plunges with a whole number of rectangular movements. It only plunges vertically at 90º.
I2= 0: Finishing wall and floor
1: Finish machining of wall only

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=0, B3=1, C1=L, C2=67%, R= tool radius, 0, R1=80%, A3=90, I1=1, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

Notes and application

B1= or B2= must be greater than 2*(tool radius + finishing allowance for sides B3=).

The cycle

1 Rapid motion to the 1st setup clearance (L1=) above the centre of the pocket.

Finishing the floor:

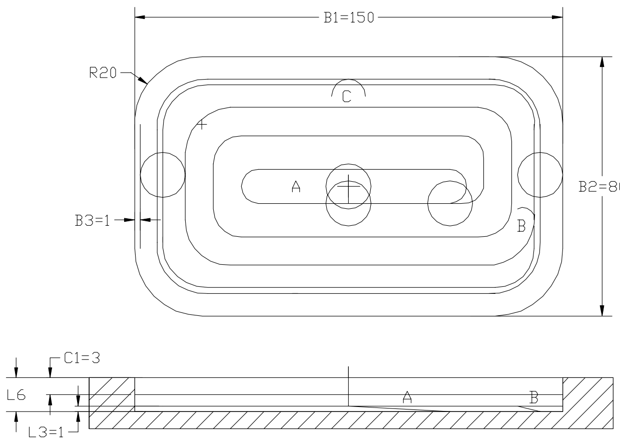
2 If the plunging angle A3=90º, the tool advances with drilling feed (F2=) to the depth (L).
If the plunging angle A3<90º, the tool advances obliquely, using a whole number of rectangular movements, to the depth (L).

3 Machining with feed (F) in the positive direction of the longer side, in a flowing movement from inside to outside.
4 At the end of this process the tool is retracted from the wall and the floor in a tangent to the helix.

Finishing the side:
5 Rapid motion to the plunging depth (C1=).
6 The starting position is the first plunging depth and at least the finishing allowance (B3=) from the side. The tool moves in tangentially, mills the contour and moves away tangentially.
7 Repeat steps 5 to 6 until the depth (L) has been reached.

8 At the end of the cycle the tool moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=) and then into the centre of the pocket.

Example

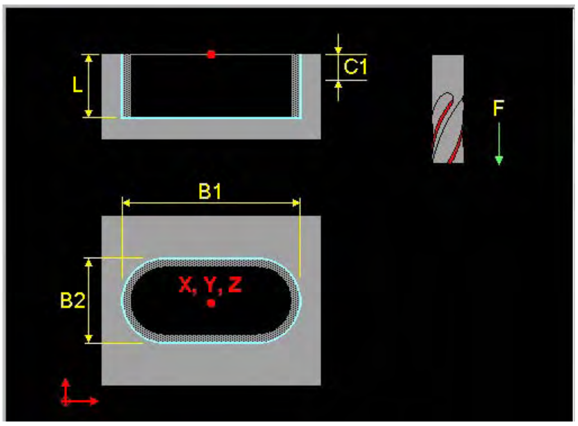


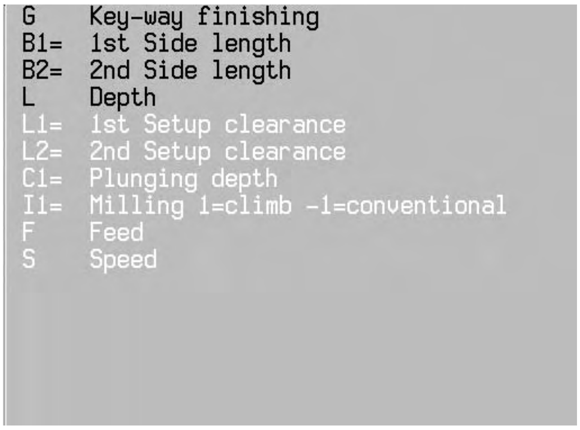
A is go obliquely to the depth. Then continuous movement.
B is move away tangentially.
C is move away tangentially.
C is advance tangentially for side finishing.

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3 F200	Switch on spindle
N30 G787 B1=150 B2=80 B3=1 L6 I1=1 L3=1 R20 A3=5 C2=65 C1=3	Define pocket milling roughing cycle
N40 G79 X160 Y120 Z0	Execute the roughing cycle at the programmed position
N50 G797 B1=150 B2=80 B3=1 L6 L3=1 A3=5 C1=3 C2=60 R20	Define pocket milling finishing cycle
N60 G79 X160 Y120 Z0	Execute the finishing cycle at the programmed position

29.23 G798 Key-way finishing

Define a slot milling cycle for finishing in a single program block.





B1= Length of the slot in the main axis.
B2= Width of the slot in the secondary axis.
The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, C1=L, I1=1

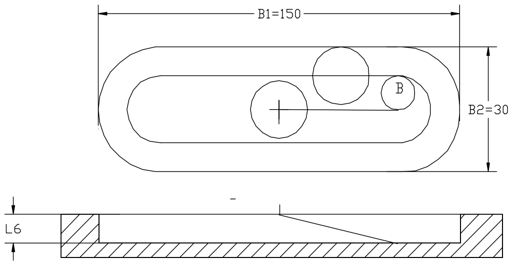
Notes and application:

Choose a milling cutter whose diameter is no greater than the width of the slot and no less than a third of the slot width.

The cycle

1 Rapid motion to the 1st setup clearance (L1=) above the centre of the slot.
2 The tool moves tangentially to the contour from the centre of the slot and finishes it in forwards rotation (I1=1).
3 At the end of the contour the tool moves tangentially away from the contour and floor to the centre of the slot.
4 The tool then moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=).

Example

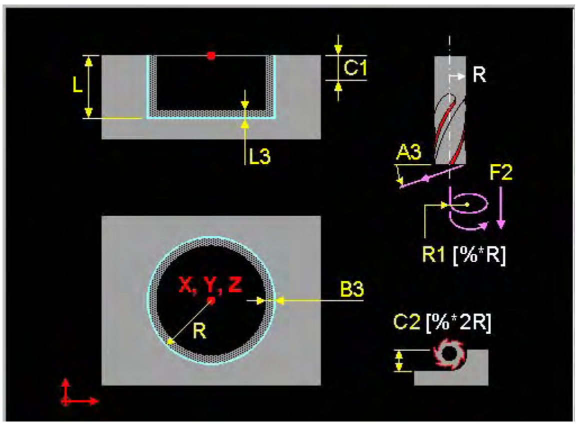


B is tangential approach and retraction. Then continuous movement.

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N15 S500 M3	Switch on spindle
N20 G788 B1=150 B2=20 B3=1 L6 L1=1 A3=10 C1=3 I1=1 I2=0 F100 F2=200	Define slot milling roughing cycle parallel to the X axis
N30 G79 X20 Y20 Z0	Execute the roughing cycle at the programmed position
N40 G798 B1=150 B2=30 L6 L1=1 I1=1 F200	Define the slot milling finishing cycle, parallel to the X axis
N50 G79 X20 Y20 Z0	Execute the finishing cycle at the programmed position

29.24 G799 Circular pocket finishing

Define a circular pocket milling cycle for finishing the wall and floor of rectangular pockets in a single program block. The sides can be machined in a number of advances. This cycle allows oblique plunging into the floor and mills in a continuous spiral path.



G	Circular pocket finishing
R	Radius
L	Depth
L1=	1st Setup clearance
L2=	2nd Setup clearance
L3=	Finishing allowance bottom
B3=	Finishing allowance sides
C1=	Plunging depth
C2=	Proportional cutting width
R1=	Proportional helix radius
A3=	Plunging angle
I1=	Milling 1=climb -1=conventional
I2=	Finishing 0=complete 1=sides
F	Feed
S	Speed
F2=	Feed for plunging

B3= Allowance sides, which will be removed by finishing.
L3= Allowance bottom, which will be removed by finishing.
C2= Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
R1= Percentage of tool radius (>0).
A3= Angle (0 to 90º) at which the tool can plunge into the workpiece. It only plunges vertically at 90º.
I2= 0: Finishing wall and floor
1: Finish machining of wall only

The other addresses are described in the introduction to the machining cycles.

Basic settings

L1=1, L2=0, L3=1, B3=1, C1=L, C2=67%, R1=80%, A3=90, I1=1, I2=0, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

Notes and application:

The minimum size of the pocket (R) is 2*(tool radius + finishing allowance for sides B3=).

The cycle
Finishing the floor:

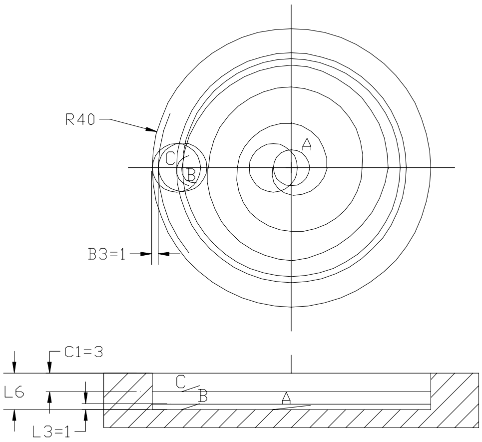
1. Rapid motion to the centre of the pocket and stay at the 1st setup clearance (L1=) above the centre of the pocket.
2. If the plunging angle A3=90º, the tool advances with feed (F2=) to the depth (L).
If the plunging angle A3<90º, the tool advances obliquely, using a whole number of circular movements, to the depth (L).
3. The tool then moves in a spiral path (direction depends on forward rotation (I1=1) with M3) and then clears the floor of the pocket from inside to outside.

Finishing the side:

4. Rapid motion to the plunging depth (C1=).
5. The side is then machined in a number of sections. The starting position is the first plunging depth and at least the finishing allowance (B3=) from the side. The tool then moves in tangentially, mills the contour and moves away tangentially.

6. Repeat steps 4 to 5 until the depth (L) has been reached.
7. At the end of the cycle the tool moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=) and then to the centre of the pocket.

Example



A is go obliquely to the depth. Then continuous movement over the floor
B is move away tangentially.
C is advance tangentially for side finishing.
C is move away tangentially.

Programming example	Description
N10 T1 M6 (R8 milling cutter)	Change tool
N20 S500 M3	Switch on spindle
N30 G789 R40 L6 B3=1 I1=1 L1=1. L3=1 A3=5 C2=65 C1=3 F200	Define circular pocket milling roughing cycle
N40 G79 X160 Y120 Z0	Execute the roughing cycle at the programmed position
N50 G799 R40 B3=1 L6 L1=1 L3=1 A3=5 C1=3 C2=65 I1=1 F200	Define pocket milling finishing cycle
N60 G79 X160 Y120 Z0	Execute the finishing cycle at the programmed position

30. Cycles in the G800 series (Turning).

30.1 General description.

The machine and MillPlus ***IT*** must be prepared by the machine manufacturer for these G-functions. If not all the G functions described here are available on your machine, consult your machine handbook.

For description of these G-functions, see: chapter turning.

30.2 G822 Clearance axial.

30.3 G823 Clearance radial.

30.4 G826 Clearance axial finishing.

30.5 G827 Clearance radial finishing.

30.6 G832 Roughing axial.

30.7 G833 Roughing radial.

30.8 G836 Roughing axial finishing.

30.9 G837 Roughing radial finishing.

30.10 G842 Grooving axial.

30.11 G843 Grooving radial.

30.12 G844 Grooving axial universal.

30.13 G845 Grooving radial universal.

30.14 G846 Grooving axial finishing.

30.15 G847 Grooving radial finishing.

30.16 G848 Grooving axial universal finish.

30.17 G849 Grooving radial universal finish.

30.18 G850 Undercut (DIN 76).

30.19 G851 Undercut (DIN 509 E)..

30.20 G852 Undercut (DIN 509 F)..

30.21 G861 Threadcutting axial.

30.22 G862 Threadcutting taper.

31. Cycles in the G900 series.

31.1 General description.

The machine and MillPlus ***IT*** must be prepared by the machine manufacturer for these G-functions. If not all the G functions described here are available on your machine, consult your machine handbook.

For description of these G-functions, see: Manual Blum

31.2 G951 Calibration.

31.3 G953 Measure tool length.

31.4 G954 Measure length, radius.

31.5 G955 Cutter control shank.

31.6 G956 Tool breakage control.

31.7 G957 Cutter control shape.

31.8 G958 Tool setting length, radius, corner radius.

32. Turning

32.1 Introduction

The turning mode has been developed for machines with a C axis that can turn continuously. In this way, turning operations can be carried out on a milling machine.

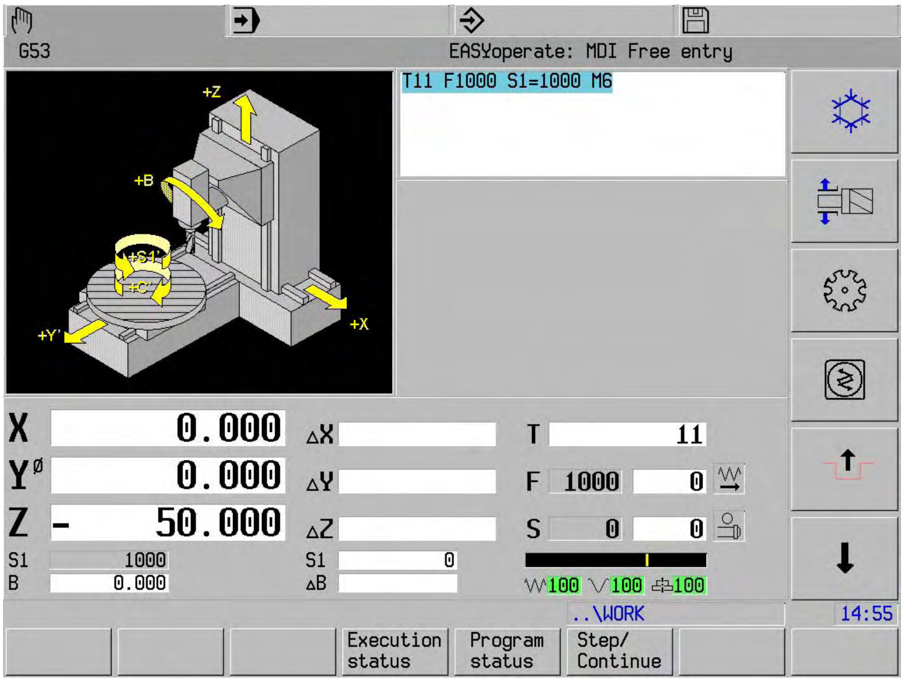
The C axis can be switched to turning mode. The C axis is then programmed as a turning spindle via S1= and M1=. The turning tools are mounted in the milling spindle and clamped at the desired orientation.

In special cases, the milling spindle can be programmed parallel to the turning spindle via S and M. A second milling spindle is not possible on machines with turning mode.

Notes and application

AVAILABILITY	Machine and CNC must be prepared for turning mode by the machine builder. If your machine is not equipped with all the G functions described here, please refer to your machine manual.
GRAPHICS	The graphic is not displayed symmetrically to the rotation.
DISPLAY	If G36 is active, the display of the C axis position changes to display S1=. S1= is the spindle revolution (G97) or constant cutting speed (G96). The axes display for the axes X and Y can optionally be changed to diameter via: manual operation mode, options and axes display. The programming remains in radius. Only when the turning mode is activated, the axes display is changed from radius to diameter The machining status is expanded with G36/G37.
REFERENCE POINT	When the controller runs up, it is always in milling mode G37. The C axis can only be switched to turning mode after the reference points have been approached.
ZERO POINT	In turning mode, the workpiece zero point in X should lie in the centre of rotation of the S1 axis. It is recommended that the workpiece zero point in Y should also lie in the centre of rotation of the S1 axis.
SPINDLE OVERRIDE	Spindle override is effective for both spindles in turning mode (G36).

Screen in turning mode



32.2 Machine constants

Machine constants for turning

Machine constants	Description
MC 268	Second Spindel (0=no, 1=yes)
MC 314	Turning mode (0=off, 1=on) Activated: - G functions G36 and G37 - Turning cycles - Machine constants MC2600 - MC27xx, MC45xx
MC 450	Balancing: measurement axis (1=X, 2=Y, 3=Z) This MC determines the axis on which the rotary table is installed. Unbalance is easiest to measure in this axis. Normally, 2 = Y axis The MC is used in the 'unbalance calibration' (installation), G691 'unbalance detection' and G692 'unbalance checking' cycles.
MC 451	Balancing: maximum amplitude [μm] This MC specifies the permissible residual amplitude in the measuring axis. The measurement is cancelled if the measured amplitude is greater than MC451 at a particular speed. Normally 5 [μm]. The MC is used in the 'unbalance calibration' (installation), G691 'unbalance detection' and G692 'unbalance checking' cycles. The C1 parameter can be superimposed on this in the G691 and G692 cycles
MC 452	Balancing: initial radial position [μm] This MC specifies the radial position (distance from centre point) of the rotary table (S1 axis) at which a balancing mass is normally mounted to compensate for unbalance. The MC is used in the G691 'unbalance detection' cycle.
MC 453	Balancing: rotary table displacement [mGrad] This MC specifies the 0 position of the rotary table and the position (door) where the operator fits the mass to compensate (and calibrate) the unbalance. The MC is used in the 'unbalance calibration' (installation) and G691 'unbalance detection' cycles.
MC2600 - MC2799, MC4500 - MC4599	Second spindle

32.3 G36/G37 Switching turning mode on and off

G36 Switches the machine from milling mode on the C axis to turning mode with turning spindle S1.

G37 Terminates turning mode. Switches the machine back to milling mode

Format

N... G36 or N... G36

Parameters

none.

Type of function

modal

Notes and application

G36

The CNC switches the C axis to turning mode.
In turning mode, the circular axis is programmed as a second spindle using S1= and M1=. C parameters can no longer be programmed.
The display of C (setpoint and actual value) on the screen is switched to S1. If the turning spindle is stationary, the position (0-359.999 degrees) is displayed.
G95 is active, assigned to the second spindle.
All G functions can be programmed, but not all the G functions are meaningful. For instance, a pocket has no meaning in turning mode. The C parameters and certain other parameters can no longer be programmed in certain G functions.
A survey of permitted G-Functions can be found in section 14
The effect of G36 remains active until it is cancelled by G37, runup or <CNC reset>. G36 is not cancelled by M30 or <Cancel program>.

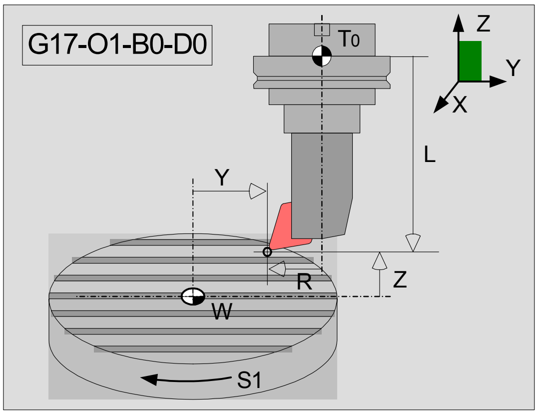
G37

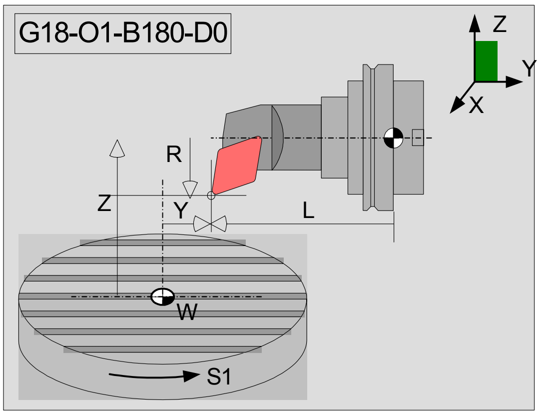
The CNC switches the C axis on again.
If the rotary spindle is still turning at the start of G37, it is first stopped.
The position of the circular axis is displayed on the screen with a value between 0 and 359.999 degrees.
G94 becomes active.
The effect of G37 remains active until it is cancelled by G36. G37 is not cancelled by M30 or <Cancel program>. G27 is always active following runup or <CNC reset>.

Program examle	Description
N9000 (C-Axies operation)	
N1 T.. M06	ActivateTurning tool
N2 G0 Y.. Z..	Tool positioning
N3 G74 X1=1 Y1=1	Rapid movement to table center
N4 G54 I1	Zero point table center X0, Y0
N5 G36	Activate turning mode
N6 G17 Y1=1 Z1=2	Activate working plane
N7 G96 M1=3 S1=200	Constant cutting speed and spindel direction
N8 G302 O7	Tool orientation override
N9 G..	Turning machining
N10 G37	Switch-off turning mode
N11 G..	Milling machining
N12 M30	Program end

32.4 G17/G18: Machining planes for turning mode

In the turning mode the machine tool can machine work pieces in the different machining planes. The machining plane is defined in the turning mode (G36), with:
- G17 Y1= 1 Z1=2, tool axis Z (vertical) or
- G18 Y1= 1 Z1=2, tool axis Y (horizontal)





The function G17/G18 defines, in which axes (Y/Z) the tool corrections for length (L) and radius (R) are calculated:
- G17: L in Z-direction, R in Y-direction
- G18: L in Y- direction, R in Z- direction

In the turning mode machining can be performed in both the YZ or XZ- machining surface as individual DIN-commands. With the machining cycles however, machining can be performed only in the YZ-machining surface.

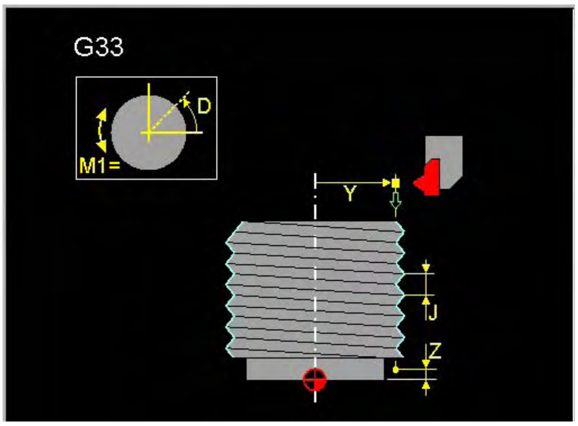
Remark:
- Y1=1 (first main axis); Z1=2 (second main axis)
- The angle (positive) and circular direction (CW) are defined from the Y-axis to the Z-axis.
- The G37 switches the actual G17/G18-plane in the turning mode back to its G17/G18-plane in the milling mode.
- The tool radius (R) is calculated in the different G17/G18-planes as a shift. Depending of the tool orientation (O) the compensation is calculated in the relevant Y or Z-axis.

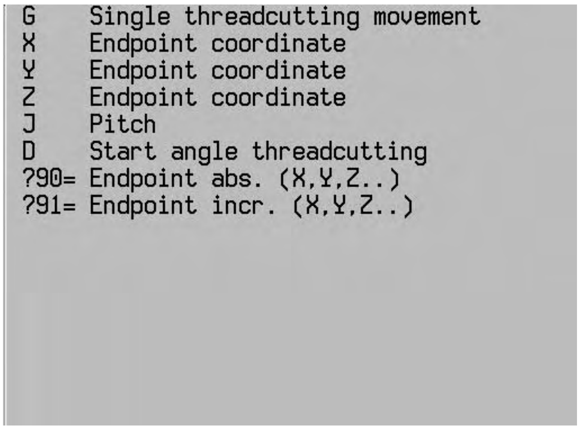
32.5 G33 Thread cutting

G33 is a thread-cutting movement. In a single pass it cuts a thread with feed and fixed pitch. The feed is determined by the spindle speed and the pitch.
Characteristics:

- Thread cutting is carried out with an open positioning control loop. Possible thread types: cylindrical and conical
- Spindle and feed override are ineffective during G33
- A number of thread movements can be programmed in sequence (e.g. oblique entry and exit)
- The lead angle of the thread can be programmed.
- The speed (S1=) and direction of rotation (M1=) must be pre-programmed

G33 is signalled to the IPLC (WIX thread movement)





G	Single threadcutting movement
X	Endpoint coordinate
Y	Endpoint coordinate
Z	Endpoint coordinate
J	Pitch
D	Start angle threadcutting
?90=	Endpoint abs. (X,Y,Z..)
?91=	Endpoint incr. (X,Y,Z..)

Notes and application

USE

G33 movement commences:
- when the actual and programmed spindle speeds are equal (actual N=target N) and
- after the marker and the calculated lead angle D

G33 carries out a single thread cutting movement from the current position to the programmed point.
The programmed speed (G97 S1=) and lead (J) determine the axial feed rate.
G33 stops at the end of the movement with an accurate stop and G1 is modally active.

Notes: If the pitch or speed is not programmed, there is no G33 movement; the axis remains stationary:
- if the pitch J or speed S1= is not programmed, an error message (P02/P26) is issued
- the direction of spindle rotation M1= 3 or 4 has no effect on the direction of movement
- Speed and Feed override are not effective during G33 movement and are switched to 100%

INTERRUPTION

It is possible to interrupt thread cutting by:

- stopping the feed: Movement stops at the end of a G33 movement.
- stopping the feed/spindle: Spindle and movement stop at the end of a G33movement.

Notes: If a number of G33 movements are programmed in sequence, the machine stops after the last G33 movement.

MACHINING PLANE

G33 can only be executed within one turning plane

MODES

- G33 is inoperative in MDI mode: Error code P77.
- In single block operation a number of G33 movements are executed in sequence.

TEST RUN / GRAPHICS

In graphics and in the test run without MST, G33 runs like G1.

PROGRAMMING EXAMPLE

Programming example	Description
N9000 (thread cutting)	
N1 T.. M06	Change thread cutting tool
N1 G0 Y.. Z..	Position the tool
N2 G36	Switch on turning mode.
N3 G17 Y1=1 Z1=2	Activate machining plane
N4 G97 M1=3 S1=100	Speed and direction
N7 G0 Y.. Z..	Advance to starting position
N8 G0 Y..	Adjust to cutting depth
N9 G33 J2 Z91=..	Thread cutting to end point
N10 G0 Y..	Retract
N11 G0 Z..	Return to starting position
N7 G37	Switch on milling mode
N6 M30	Program end

32.6 G94/G95 Expanded choice of feed unit

Informs the CNC how to evaluate the programmed speed (S).
This function is expanded for turning mode.
The spindle and the circular table must be programmed for turning.

Notes and application

In addition, the rotary table (second spindle) must be programmed with S1= and M1= for turning.

In milling mode (G37): N... G95 F.. {S..} {M..}
In turning mode (G36): N... G95 F.. {S1=..} {M1=..}

S and M refer to the spindle
S1= and M1= refer to the second spindle

PRIORITY

The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

MAXIMUM SPEED

The value of the second spindle speed (S1=) lies between 0 and ‘Max. output voltage speed’ (MC2691).

MACHINE FUNCTION

Second spindle machine functions:
- M1=3 second spindle clockwise
- M1=4 second spindle anticlockwise
- M1=5 second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.
The S1= and M1= addresses can also be programmed in the following G functions: G0, G1, G2, G3, G94.
The G95 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.

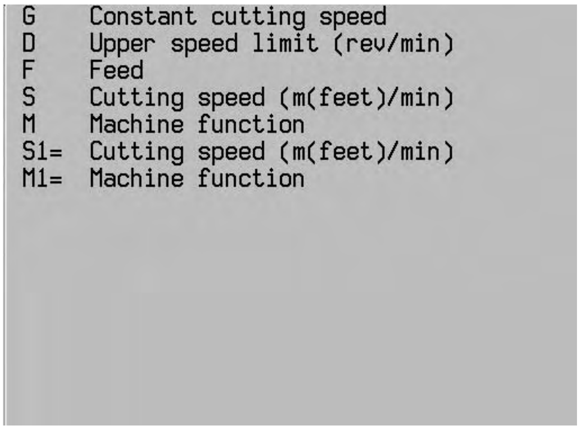
32.7 G96/G97 Constant cutting speed

G96 Programming constant cutting speed.
G97 Switching off constant cutting speed..

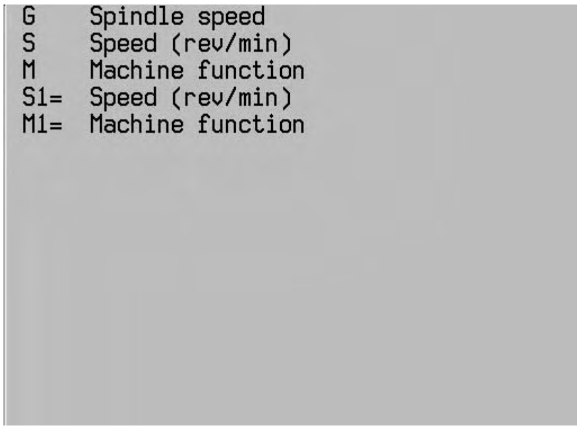
Format

N... G96 F.. D.. {S..} {M..} {S1=..} {M1=..}
N... G97 F.. {S..} {M..} {S1=..} {M1=..}

Parameters



G96



G97

S and M refer to the spindle
S1= and M1= refer to the second spindle (rotary table)

Type of function

modal

Notes and application

MAXIMUM SPEED (D)

The value of the second spindle speed lies between 0 and ‘Max. output voltage speed’ (MC2691).

MACHINE FUNCTION

Second spindle machine functions:
- M1=3 second spindle clockwise
- M1=4 second spindle anticlockwise
- M1=5 second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.
The G96 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.
The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

32.8 Turning tools in the tool table

Tool data

The most relevant tool data that is stored in the the tool table for turning tools is listed below:

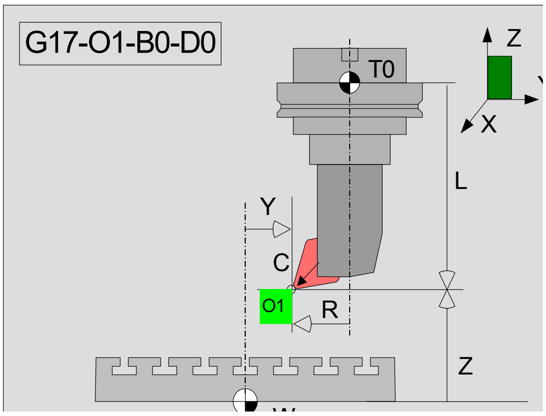
L	Length
R	Radius
C	Corner radius
Q3	Tool type
G	Graphics
O	Orientation

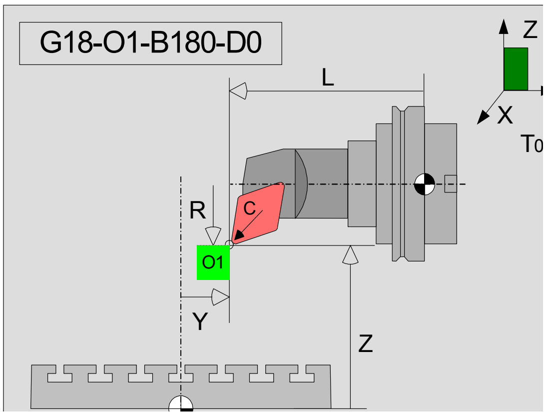
Tool correction

The tool dimensions are stored in the tool table as tool length L and tool radius R. How these dimensions are calculated in the relevant axes, depends on the actual plane (G17/G18) and tool nose position (orientation O):

- G17: Tool length L in the Z-Axis; tool radius R in the Y-Axis
- G18: Tool length L in the Y-Axis; tool radius R in the Z-Axis

The radius (R) is considered to be a shift and is calculated, depending on the tool orientation (O) with sign (+/-) in the relevant axis.





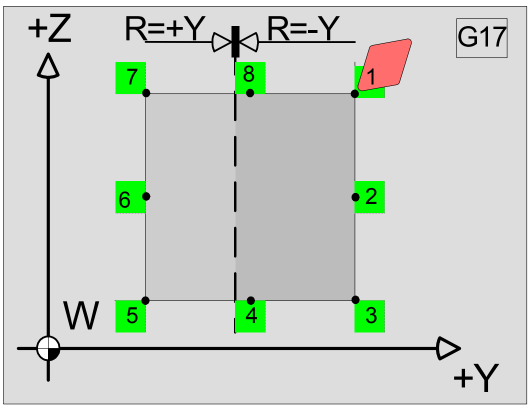
Tool orientation (O)

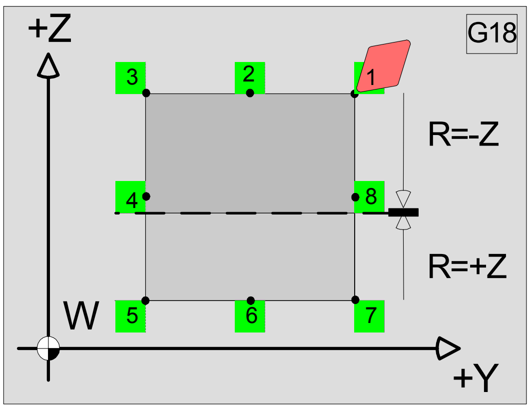
The tool orientation (O) determines in which direction the tool nose cutting edge is positioned. It calculates and compensates the tool path in the respective axes with two parameters for:

- Tool radius (R)
- Tool nose radius (C)

Tool radius compensation (R)

The pictures below show in which axis the tool radius in the G17/G18-plane is calculated.



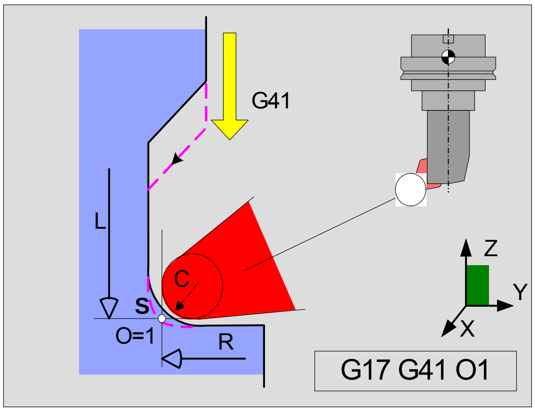


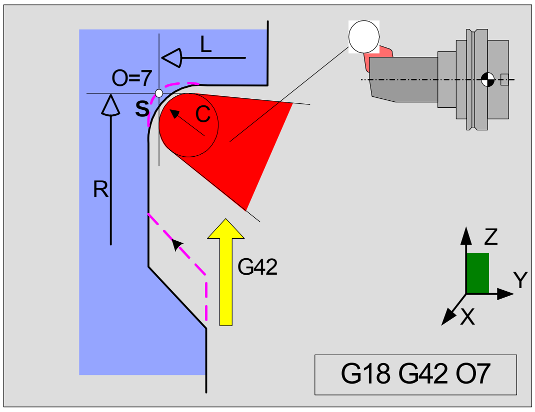
The table below shows the relation between G17/G18, R, C and the way the radius is calculated.

	Plane	Orientation	Radius correction	Radius as shift
G17	G17	Not active	R	Not active
	G17 Y1=1 Z1=2	1, 2, 3, 4, 8	C and O	R in negative Y-direction
		5, 6, 7	C and O	R in positive Y- direction
G18	G18	Not active	R	Not active
	G18 Y1=1 Z1=2	1, 2, 3, 4, 8	C and O	R in negative Z- direction
		5, 6, 7	C and O	R in positive Z- direction

Remark: - Tool nose radius compensation refers to the tool tip corner radius C.
- Radius compensation refers to the tool radius R.
- The tool orientation O is taken from the tool table but can be overwritten by the G-function (G302 Ox) in the program.

Tool nose radius compensation (TNR)
Turning tools have a nose radius (C) on the cutting edge. During machining of e.g. conicals, phases and radii, inaccuracy problems occur which can be corrected by the tool nose radius compensation **TNR**.
Programmed movements are related to theoretical tool cutting point (S). Contour errors appear at contours that are not axes parallel. The **TNR** calculates a compensated tool path, **equidistant**, to correct this error.





The pictures above show a turning tool in the different machining planes G17/G18.
The turning tool is performing a single cut with G1/G3 and is situated:

- At the left side of the contour (G41) with orientation O1 (picture on the left) and
- At the right side of the contour (G42) with orientation O7 (picture on the right)

Considered is the tool nose. The tool nose tip, with its radius (C), is considered to be as a circular plate, whereby its backside is able to cut the contour. The clearance angle of the tool (back side of the plate) must be appropriate to prevent the contour from damage during cutting.

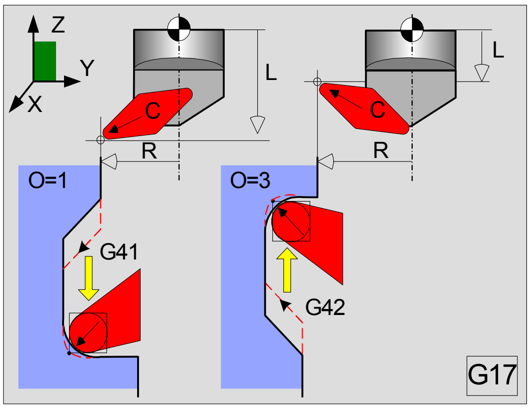
Tool nose radius correction (TNR) switching on/off
The **TNR** is calculated at all clearance- and grooving cycles.
At DIN-programming (G1/G2/G3) the **TNR** can be switched on/off additionally. The **TNR** is switched on/off with the following G-functions:

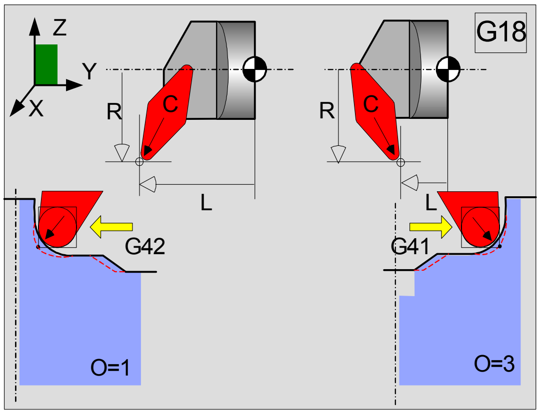
- G40: **TNR** is switched off
- G41: **TNR-on,** the turning tool is on the left from the contour side
- G42; **TNR-on,** the turning tool is on the right from the contour side

Examples of TNR in **G41 and G42**
In the pictures below two examples are shown of a turning application.

- The left picture shows a turning application in the axial axis in G17:
  - G41 and O1 (Left side)
  - G42 and O3 (Right side)
- The right picture shows a turning application in the radial axis in G18:
  - G42 and O1 (Left side)
  - G41and O3 (Right side)

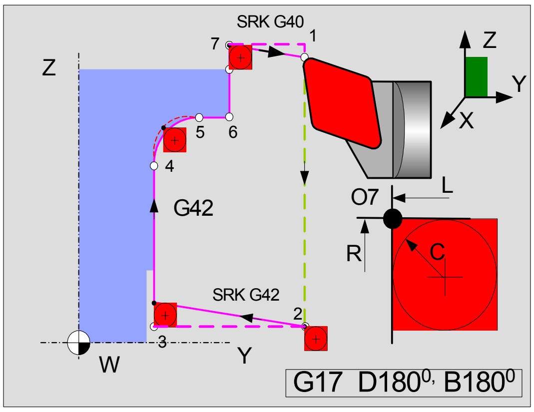
Note in the pictures:
- The swivel head position
- The different cutting edges





TNR Start/Stop
The picture below shows, as an example of the DIN-program N171842.PM, the way **TNR** is switched on and -off.

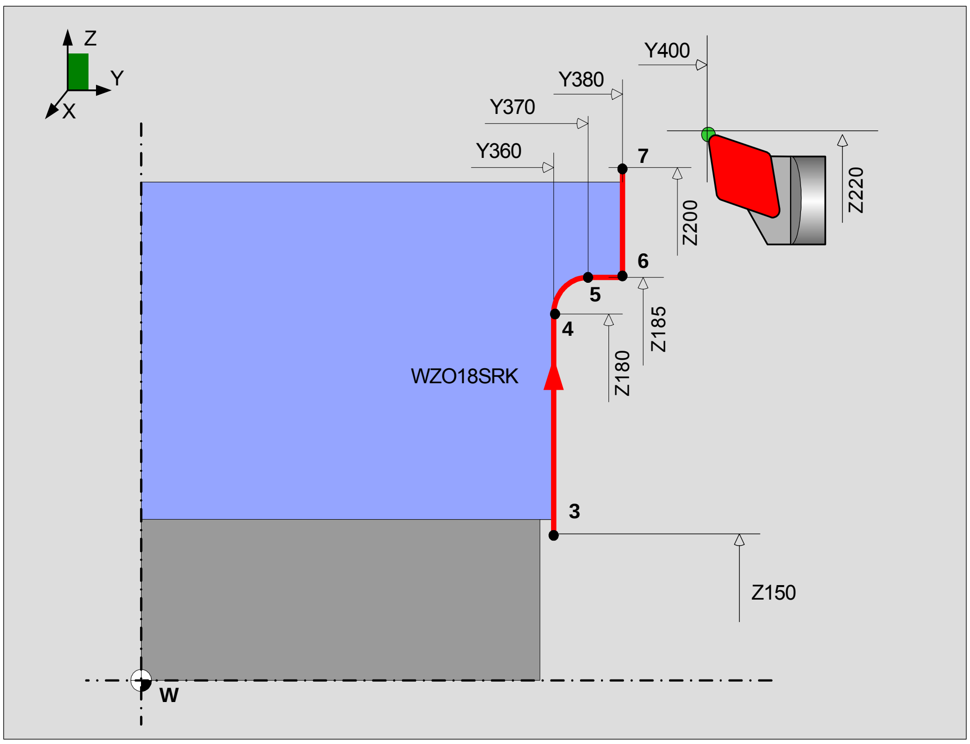
Note: - The tool must have enough lead and trail cut at switching on and -off **TNR** in order to cut the complete contour.
- Switching on and -off **TNR** must be programmed perpendicular to the contour side



Example DIN-Program

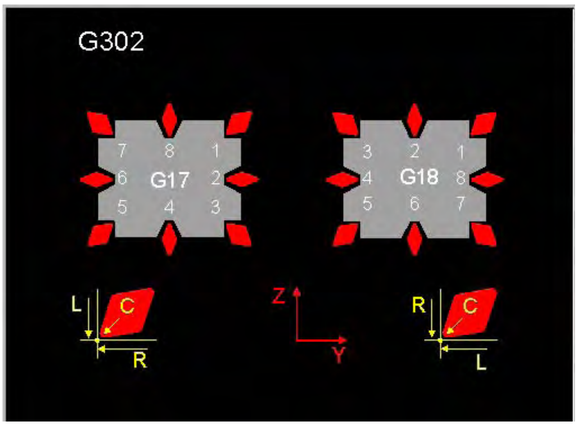
Program example	Description
N171842 (Contour cutting)	
N1 G195 X0 Y0 Z0 I0 J300 K300	Graphic window definition
N2 G54 I10	Zero point shift to table centre
N3 G0 X0 Y450:2 Z250	Tool displacement
N4 T10 M06	Tool exchange turning tool
N5 G36	Turning mode active
N6 G17 Y1=1 Z1=2	Machining plane G17 active
N7 B180	Tool head swivel
N8 G0 Y400:2 Z220	(1) Tool positioning
N9 G96 M1=3 S1=200 D500	Constant cutting speed and table direction
N10 G302 O7	Tool orientation O7
N11 M52	Main spindle release
N12 M19 D0	Tool orientation
N13 M51	Main spindle clamp
N14 G0 Z150	(1→2) Positioning
N15 **G42**	**TNR** switching on G42
N16 G1 Y360:2	(2→3) Contour side approach with G42
N17 G1 Z180	(3→4 Contour side cutting
N18 G2 Z185 Y370:2 R5	(4→5) Radius cutting
N19 G1 Y380:2	(5→6) Contour side cutting
N20 G1 Z200	(6→7) Contour side cutting
N21 **G40** Y400:2 Z220	(7→1) Positioning with G40 (**TNR** switching off)
N22 G97 S1=100	Turning table in G97-mode
N23 G37	Milling-mode active
N24 M30	Program end

Work piece drawing

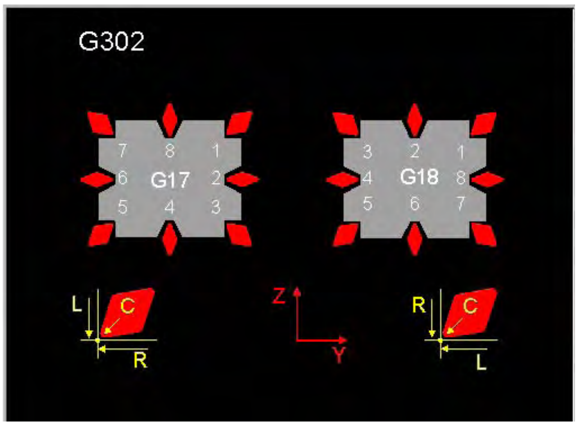


32.9 G302 Overrule radius comp. parameters

The G302 function determines the tool orientation during execution. The tool parameters in the tool memory are not changed.



G17



G18

```
G     Overrule radius comp. parameters
O     Tool Orientation
```

O Defines the tool orientation used during execution.
The value lies between 0 and 8.

Type of function

Non-modal

Notes and application

Remarks:

If the active tool orientation is overwritten, the direction of the R displacement may also change.
In G18, the active tool orientation is already changed by the CNC. See chapter 'Tool correction'.

USE

The G302 function should be used if. for example, the main spindle has been turned through 180 degrees with M19 D90. In this case, the orientation is mirrored compared with the status with M19 D90. The orientation should also be mirrored when turning takes place 'across the centre'.

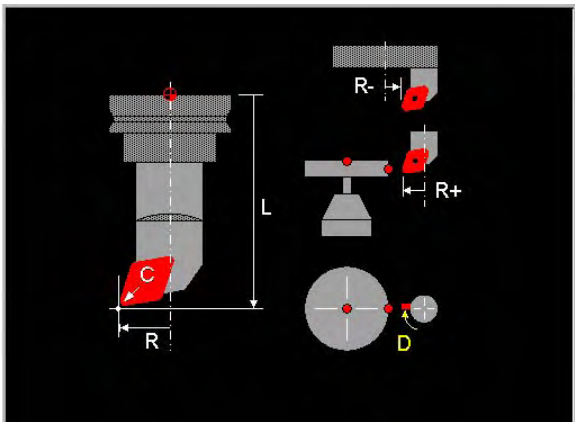
Note: In these cases, the direction of rotation of the 2nd spindle should also be reversed.

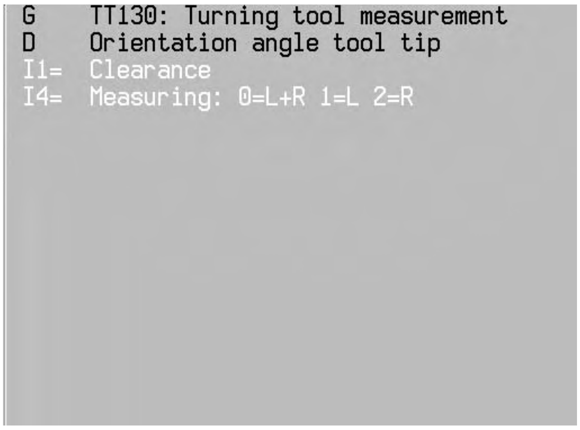
DELETING

G302 is switched off again with G302 without parameter, set plane (G17, G18, G19), tool change, M30 and <Cancel program>

32.10 G611 TT130: Measure turning tools

This cycle measures the length and radius of turning tools. Only tools in the G17 machining plane are measured.





Notes and application

INPUT PARAMETERS

D The tool tip must always be located in the correct position before measuring, i.e. with its tip parallel to the axis and perpendicular to the measuring device. Since the turning tool can be at any angle during machining, depending on the type of work, the operator decides whether the tool measuring position (D) is programmed into the measuring cycle.

I1= Safety distance (I1=)
The safety distance in the direction of the spindle axis must be sufficient to prevent any collision with the workpiece or clamping devices. The safety distance is with respect to the top edge of the stylus. Basic setting (I1=0)

I4= Measuring: 0=L+R 1=L 2=R (as desired)
The tool length and radius are measured as standard

Notes: Both the position and direction of the tool are reset after measuring.
- If the angle of orientation is not known (no spindle reference run) error message P339 is issued.
- If neither the orientation nor the position of the tool i known, error message P334 is issued.
- Only tool orientations (O1 and O7) are allowed for measurement with TT-120. If a different tool orientationis given, error message R326 (tool orientation not allowed) is issued.

TOOL PARAMETERS FROM THE TOOL TABLE

The measuring cycle uses the following parameters from the tool table.

Parameters	Description
L*	Tool length
R*	Tool radius
C	Cutting radius of tool
L4=	Length allowance
R4=	Radius allowance
L5=	Length tolerance
R5=	Radius tolerance
E	Tool status
O	Tool orientation

Important: Make sure that the length (L) and radius (R) entered are within the tolerance (MC397), otherwise there will be an error message.

Note: - Before measuring the tool for the first time, enter the estimated radius, the estimated length and the tool orientation of the tool concerned in the tool table.
- The measuring cycle adopts the current O from the tool table or from G302

THE CYCLE

MillPlus ***IT*** measures the tool in accordance with a fixed programmed sequence:

1. The machining plane for measurement is set
2. The tool axis moves to the safety distance (I1=)
3. The current tool position is checked and reset if it is not correct for measurement
4. Both axes advance to the measuring position of the probe
5. The tool axis advances to the probe
6. The tool length is measured first, followed by the radius
7. The tool axis moves up to the safety distance
8. The R/L measured values (first measurement) or the tolerance R4=/L4= (check measurement) are saved
9. The original working plane, tool position and tool orientation are reset

MEASURE TOOL (E=0 or no value)

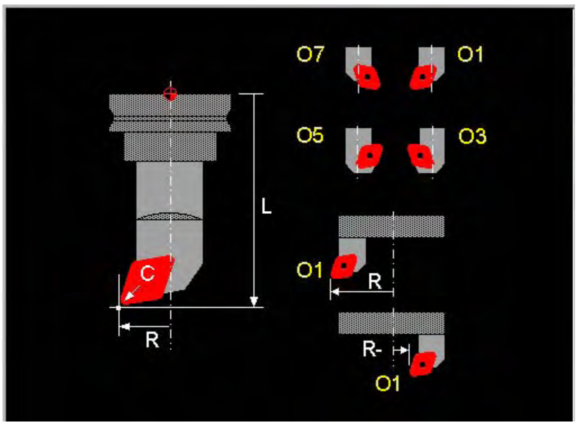
On the first measurement MillPlus ***IT*** overwrites the tool radius R and the tool length (L) in the tool memory and sets the allowance R4 and L4=0.

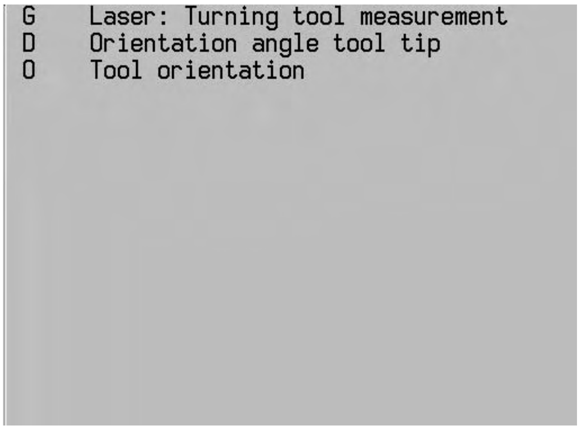
CHECK TOOL (E=1)

If you are checking a tool, the measured tool data is compared with the data in the tool table. MillPlus ***IT*** calculates the deviations with the correct sign and enters these in the tool table as allowances R4 and L4. If one of the dimensions is greater than the allowable wear (L5= and R5=) or breaking tolerance an error message is issued.

32.11 G615 laser system: L/R measurement of turning tools

This cycle measures the length and radius of turning tools The turning tool is measured when stationary in both the G17 and G18 planes. Only turning tools with tool orientation 1 or 7 can be measured.





Notes and application

INPUT PARAMETERS

D Tool position for measuring position
In the safety position, the tool is oriented to the programmed position (D). The tool tip must then be parallel to the axis and at right angles to the laser.

O Tool orientation
The orientation (O) of the tool tip determines whether measurement takes place in front of the laser or behind it. Only values 1 or 7 are allowed.

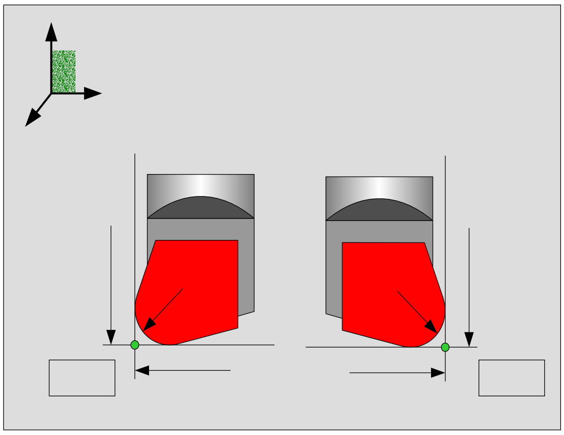
TOOL PARAMETERS FROM THE TOOL TABLE

Parameters	Description
L	Tool length
R	Tool radius
C	Cutting radius of tool
L4=	Length allowance
R4=	Radius allowance
L5=	Length tolerance
R5=	Radius tolerance
L6=	Length measurement offset
R6=	Radius measurement offset
E	Tool status
O*	Tool orientation

Note: -The tool length (L) and radius(R) must be entered accurate to +/- 5mm
- The tool cutting radius (C) should preferably be entered
- The orientation O is not used in the measuring cycle

TOOL TYPES

Turning and plunging tools can be measured with the main and secondary cutter to the rear (see illustrations on the right)



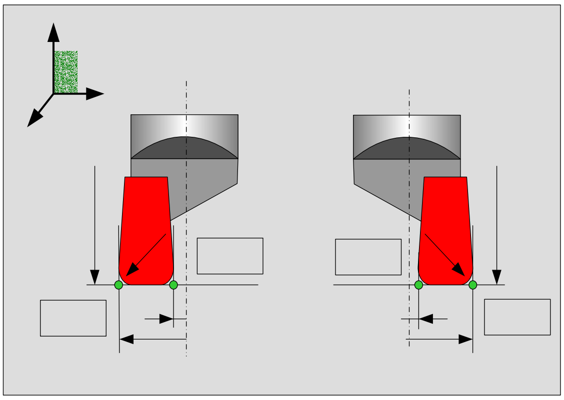
LENGTH AND RADIUS MEASUREMENT

The tool length (L) and radius(R) must be stored in the tool memory.
Before the first measurement the approximate length and radius must be entered (max. deviation +/-5mm).

Note*:* incorrect input can lead to error messages or even collision with the laser light cabinet.

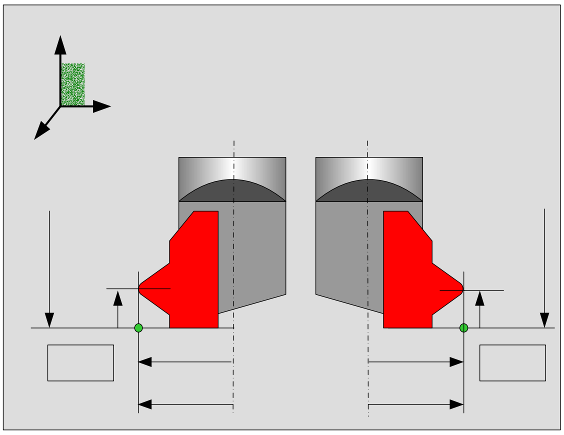
CORNER RADIUS

We recommend always entering a corner radius (C) in the tool memory.
The cycle then runs faster.



ACTIONS

- Measure tool (E=0 or no value)
  On the first measurement the tool length
  (L) and radius R are overwritten, the allowance
  L4=0/ R4=0 and the tool status E=1 are
  set. If a corner radius C is entered, this is
  also corrected.
- Check tool (E=1)
  The measured deviation is added to L4=/R4= in the tool table



THE CYCLE

1. At the start of the cycle the axes move rapidly to the safety position using positioning logic.
2. In the safety position, the tool is orientated to the programmed position (D) and clamped there.
3. The tool moves into the measuring position at measuring speed.
4. The measurement is carried out.
5. After the measuring process the Z axis moves back to the safe position

Notes: The cycle can be called in milling mode and in turning mode.
- The tool can be measured both in front of and behind the laser. The greatest accuracy is reached when the tool is measured in the machining position.

After completing the cycle, the spindle remains in the programmed position (D) and the orientation before measurement (O) is active.

32.12 Unbalance cycles

32.12.1 General information

To machine workpieces to be turned on an FP machine, both the machine (rotary table) and workpiece must be balanced, otherwise the life of the machine, the quality of the workpiece or even the safety of the operator cannot be guaranteed.

First, the unbalance properties of the rotary table must be determined. Usually, this unbalance calibration takes place when the machine is handed over or during servicing.

To determine the unbalance of the clamped workpiece, a new cycle has been introduced: **G691 unbalance detection**.
This cycle can be called up directly in manual mode under the FST menu.
The result is a suggestion for compensating for the measured unbalance: what mass should be attached at what radial position from the turning centre. The rotary table is automatically turned to the position where the mass should be attached.
The radial position for an available compensating mass can be calculated in the dialog window. The relationship between mass and position are shown graphically.

To ensure that no turning operations take place in automatic mode with too great an unbalance, a new G function can be called in the program: **G692 unbalance check**.
This G function checks the unbalance present against the permissible unbalance. If this is exceeded, an error message is issued, following which the operator can cancel the automatic mode and carry out a new unbalance detection with correction in manual mode

32.12.2 Description of unbalance

When working in turning mode, centrifugal forces occur if the clamped part (e.g. a pump housing) has an unbalance. This influences concentric accuracy because the second spindle (= circular axis C) is configured on the Y axis.

Unbalance U = m . R
where:

m	= mass	[g]
R	= distance form centre of mass to centre of table	[mm]

The unbalance is given in [gmm} (grammes*mm). This means that 500 [grammes] at 300 [mm] (=150000 [gmm]) has the same effect as 1000 [grammes] at 150 [mm].

The centrifugal force is proportional to the unbalance and rises quadratically with rising speed.

Centrifugal force Fc = m . R : 1000000 . (S . 2 . PI : 60) ^ 2
where:

Fc	= centrifugal force	[N]
m	= mass	[g]
R	= distance form centre of mass to centre of table	[mm]
S	= speed	[rpm]

The unbalance must be compensated by a balance weight. For this purpose, the available measuring systems of the circular axis C and the linear axis Y are used to detect the unbalance that exists.

32.12.3 (G227/G228) Unbalance monitor

This function monitors the unbalance that occurs during machining when a part that has not been balanced is being turned on a milling lathe. If a defined limit is exceeded machining stops. There are two such limits, one fixed limit that can be set and one programmable limit. The fixed limit is set by the machine manufacturer and is always active. It is set ‘higher’ with the purpose of protecting the machine. The programmable limit is ‘lower’ and is switched on as required, for example not during feed movements.

Note: - The current unbalance value is displayed in the ‘Spindle performance display’.
- The unbalance monitor function can be switched on and off in the program.

SWITCHING ON THE UNBALANCE MONITOR (G228 I1=, I2=, I3=)

I1= Defined when the MillPlus ***IT*** generates an error message n28 'Unbalance monitor 1: Excessive unbalance '

0 =	Feed movement:	no error message (Basic setting).
	Rapid movement:	direct error message
1 =	Feed movement:	error message at end of contour
	Rapid movement:	direct error message
2 =	Feed movement:	error message at end of block
	Rapid movement:	error message at end of block
3 =	Feed movement:	direct error message
	Rapid movement:	direct error message

I2= Defines which value is still allowed for the maximum unbalance If this is not programmed the value in MC454 'Unbalance monitor 1: limit’ is taken. The value lies between 0 and 100 [μm].

I3= Defines the maximum sum (of unbalances exceeding the limit) before an alarm is issued. If this is not programmed the value in MC454 'Unbalance monitor 1: sum over limit’ is taken. The value lies between 0 and 1000 [μm].

Note: - G228 is only present when MC314 'milling and turning mode' is active.
- G228 activates the first unbalance monitor. The setting of the 1st unbalance monitor is taken from the machine constants MC454 and MC455 or, if programmed, from parameters I2= and I3=. Depending on parameter I1=, an error message is issued.

SWITCHING OFF THE UNBALANCE MONITOR (G227)

Note: - G227 switches off G228 and therefore the 1st unbalance monitor.
- G227 is automatically activated after <Reset control>, <Cancel program> or M30.
- The 2nd unbalance monitor cannot be switched off.

OPERATOR INTERFACE

The current unbalance value is displayed in the Spindle performance display. Here the 1st programmable limit is marked in yellow and the second fixed limit is marked in red. The highest unbalance value that has occurred since the start of the program or programming of G228 is shown in green.
The display is only present when one of the unbalance monitors is active. The red marking is always 90% along the total length.

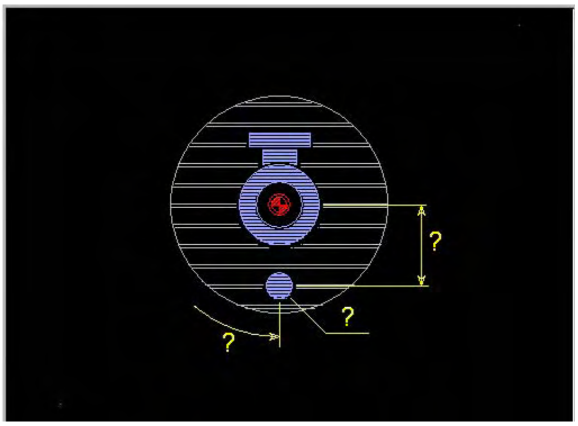
ERROR MESSAGES

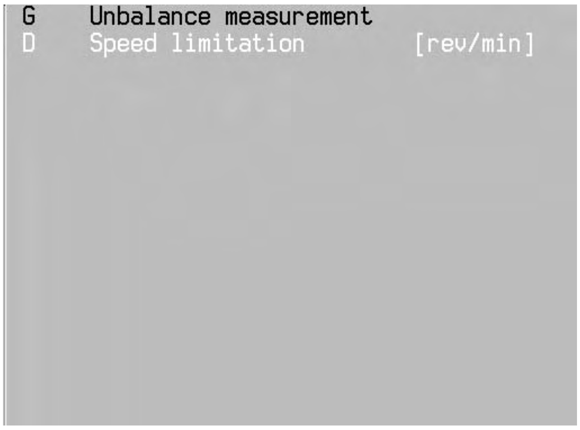
S228 Unbalance monitor 1: Excess unbalance **Class: D**
The 1st unbalance monitor generates an alarm. Whether and when this error occurs depends on the machine constants MC454 and MC455 and/or can be programmed in G228 ‘Unbalance monitor: ON’

S229 Unbalance monitor 2: Excess unbalance **Class: D**
The 2nd unbalance monitor generates an alarm. Whether and when this error occurs depends on the machine constants MC456 and MC457.

32.12.4 G691 Measure unbalance

This cycle calculates the instantaneous unbalance. It gives the operator a suggestion how to compensate for the unbalance. This cycle should be called after each clamping operation and after milling mode..





D Maximum speed for terminating the measurement
Basic setting MC2691 ‘maximum speed
Minimum value 50 [rpm]
The speed limit should be at least as high as the programmed speed for turning machining.

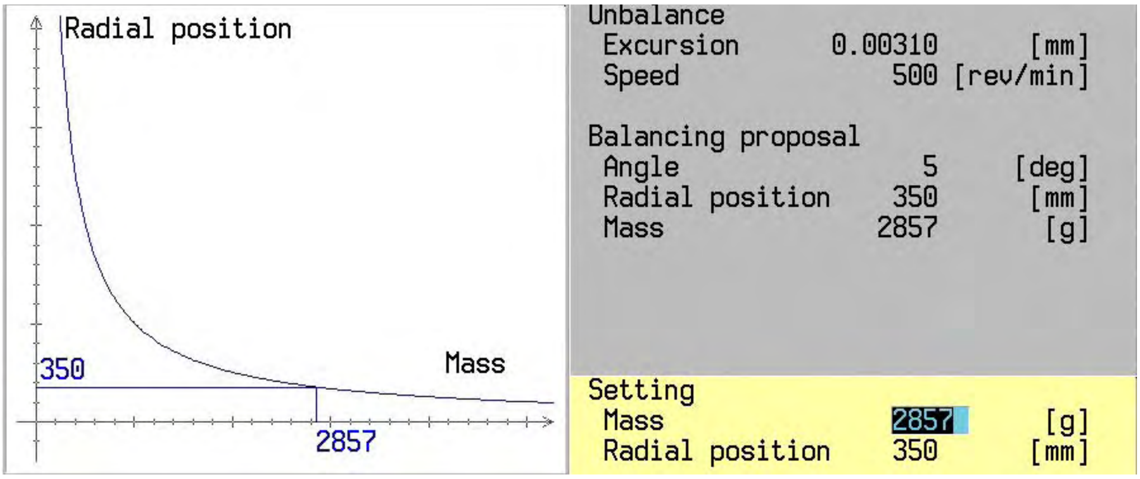
Notes and application

When detecting unbalance, the position error of the linear axis is measured with rising speed. The speed is increased in steps of 25 rpm. When the position error has reached the maximum value (MC451) or the maximum speed has been reached, the measurement is terminated. The unbalance is calculated from the measured error and the stored calibration data.
The unbalance (gmm) and compensation position (degrees) are displayed. This position is approached at the end of the cycle.

Example: Balancing a workpiece
Nxx G691 D500
Explanation:
1. Start balancing cycle with maximum speed of 500 rpm.
2. Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically positioned.
3. Enter the weight of an available mass in the window.
4. The CNC displays the new radial distance for the available mass.
5. Fit the mass at the radial position (distance and angle). Terminate with start.
6. Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.

Representation of measurement results

Once the unbalance detection measurement is terminated, the measurement results are displayed instead of the input and support fields. This image is created by G350.



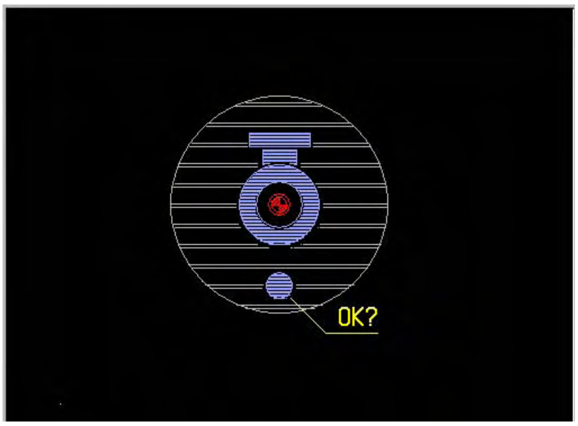
Left:
The relationship between mass and position are shown graphically.
Top right:
The measured unbalance causes a deflection at the speed displayed. This unbalance can be compensated in accordance with the balancing suggestion.

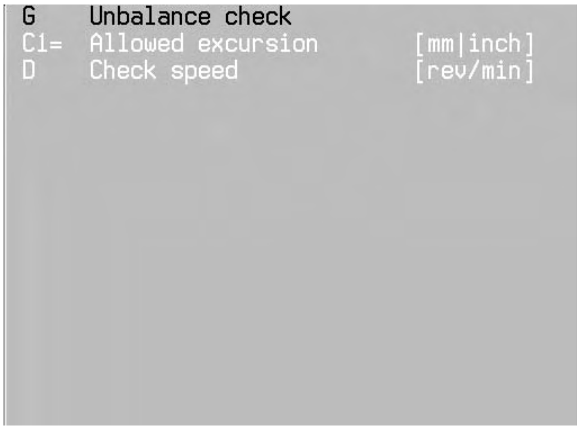
Bottom right:
The radial position for a selected mass is calculated in the dialog window. The calculation takes place after pressing the <ENTER> key. The START key terminates the cycle and closes this window.

In **automatic mode**, the left graphical window is not shown so that the program pointer remains visible.

32.12.5 G692 Unbalance checking

This cycle checks that the unbalance does not exceed a particular value. It should be called at the start of every turning operation to ensure that the concentric error does not exceed the desired tolerance or the specified limit.





C1= Maximum unbalance for message
Basic setting MC451 “maximum deflection”.

D Programmed speed for checking
Basic setting MC2691 “maximum speed”

Notes and application

When checking unbalance, the deflection of the linear axis is measured at a specified speed. If the deflection reaches the value C1=, an error message is issued.

Example: Checking unbalance.

G692 C1=0.003 D500 The CNC detects whether the deflection of the table is within the limit of 0.003 mm at a speed of 500 rpm. If the deflection is greater than the value entered (C1=), the program is stopped.

Unbalance example

Program example	Description
N9999	
N1 G691 D500	1 Start balancing cycle with maximum speed of 500 rpm. 2 Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically located. 3 Enter the weight of an available mass in the window. 4 The CNC displays the new radial distance for the available mass. 5 Fit the mass at the radial position (distance and angle). Terminate with start.
N2 G691 D500	Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.
N...	Milling Unbalance may change due to milling processes or changes in the clamping.
N30 G37	Start turning mode
N31 G692 D500	Check whether unbalance is still correct
N...	Turning

32.13 Turning cycles

AVAILABILITY

The machine builder must prepare machine and CNC for turning operations. If your machine is not equipped with all the G functions described here, please refer to your machine manual.

The tuning cycles are executed as macros, every block can be seen in the display and each block is processed as a single block.

General notes and application

STARTING POINT

The starting point determines the place where the tool starts machining. The cutting steps start from this position. If the tool is a long distance away, several cutting steps take place. If the tool is between Y1= and Y2=, cutting will start there and the cutting may not all be carried out.

If the co-ordinate of the starting point Y is smaller than the co-ordinate of the machining starting point Y1, the machine first travels to co-ordinate Z1.

TOOL MEMORY ADDRESSES

The following addresses are used in the tool memory:

C Tool tip radius

O Tool orientation

C6 Tool width (Grooving cycles)

If no O is entered in the tool memory, a standard orientation is assumed depending on the machining.

RADIUS COMPENSATION

Tool tip radius compensation is carried out automatically in this G function.

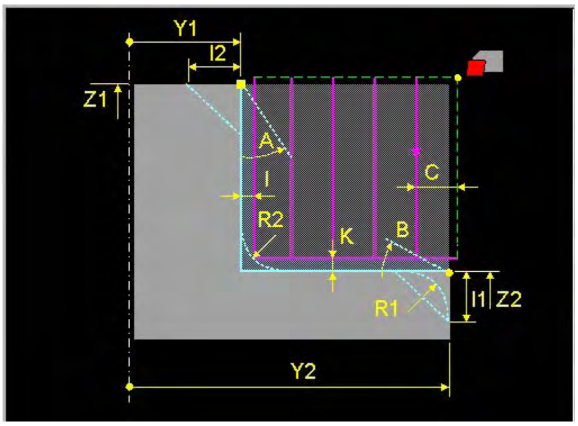
Cycle survey

Clearance, Grooving, Undercut and threading cycles

The control system offers several clearance- and grooving cycles. The clearance cycles are divided into two groups: clearance- and roughing cycles

Cycles	Cycle	G-Function
Clearance	Clearance axial	G822
	Clearance radial	G823
	Clearance axial finishing	G826
	Clearance radial finishing	G827
Roughing	Roughing axial	G832
	Roughing radial	G833
	Roughing axial finishing	G836
	Roughing radial finishing	G837
Grooving (Standard)	Grooving axial	G842
	Grooving radial	G843
	Grooving axial finishing	G847
	Grooving radial finishing	G846
Grooving (Universal)	Grooving axial -Universal	G844
	Grooving radial -Universal	G845
	Grooving axial Finishing -Universal	G848
	Grooving radial Finishing -Universal	G849
Undercut	Undercut DIN 76	G850
	Undercut DIN 509 E	G851
	Undercut DIN 509 F	G852
Threading	Threading Axial	G861
	Threading Conical	G862

32.13.1 G822 Clearance axial



G	Clearance axial
Y	Starting point
Z	Starting point
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Cutting depth
A	Angle 1
B	Angle 2
I1=	Chamfer length 1
R1=	Radius 1
I2=	Chamfer length 2
R2=	Radius 2
I	Finishing

K	Finishing
S1=	(Cutting) Speed
F	Feed

Y	Starting point.	Position of tool n radial direction. This position is the starting point for machining. Y is reduced with C until Y1= is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z until Z2 is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0.Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0.Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0.Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding between angles A and B.
I and K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, I2=0, R2= Tool nose radius, I=0, K=0

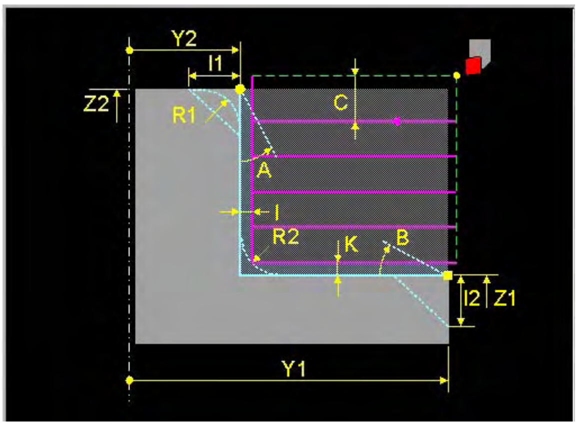
Associated functions

G827 for finish machining

Notes and application

Cutting takes place first, then finish machining.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius.

32.13.2 G823 Clearance radial



G	Clearance radial
Y	Starting point
Z	Starting point
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Cutting depth
A	Angle 1
B	Angle 2
I1=	Chamfer length 1
R1=	Radius 1
I2=	Chamfer length 2
R2=	Radius 2
I	Finishing
K	Finishing
S1=	(Cutting) Speed
F	Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Z is reduced with C until Z1= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension (incremental: by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0.Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0. Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding between angles A and B.
I and K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, I2=0, R2= Tool nose radius, I=0, K=0

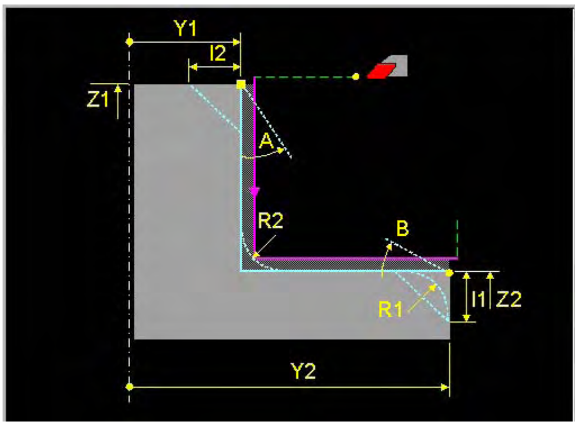
Associated functions

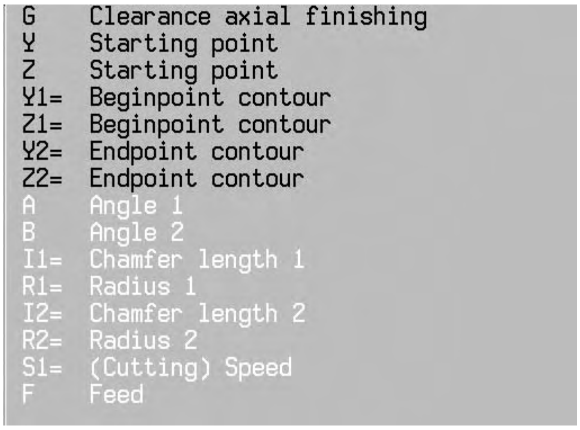
G827 for finish machining

Notes and application

Cutting takes place first, then finish machining.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius.

32.13.3 G826 Clearance axial finishing





Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining. Finish machining starts at Y.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0. Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding between angles A and B.

Basic settings

A=0, B=0, I1=0, R1=0, I2=0, R2= Tool nose radius

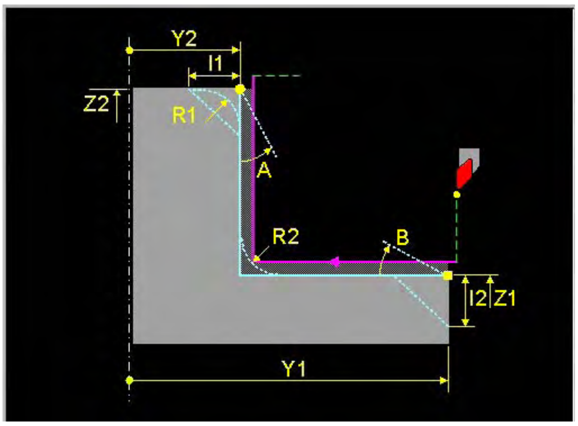
Associated functions

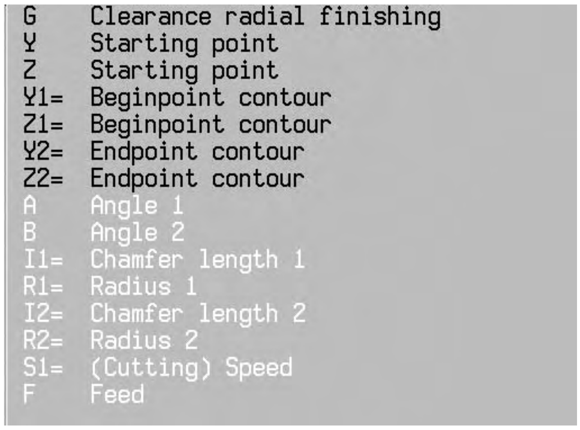
G822 for rough machining

Notes and application

Finish machining goes from Y1/Z1 to Y2/Z2.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius.

32.13.4 G827 Clearance radial finishing





Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point.
I1=	Chamfer length	Basic setting I1=0. Chamfer length at contour end point. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at contour end point.
I2=	Chamfer length	Basic setting I2=0. Chamfer length at contour starting point.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding between angles A and B.

Basic settings

A=0, B=0, I1=0, R1=0, I2=0, R2= Tool nose radius

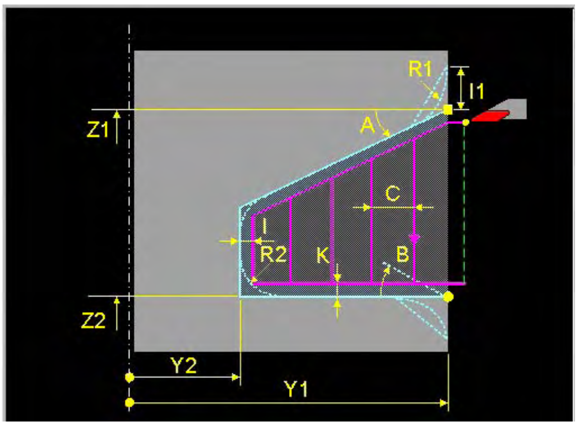
Associated functions

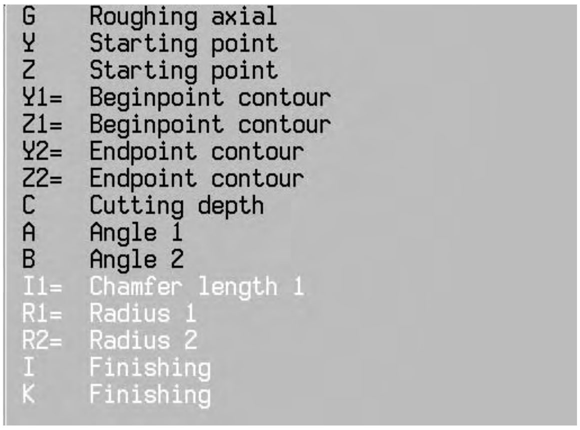
G823 for rough machining

Notes and application

Finish machining goes from Y1/Z1 to Y2/Z2.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius

32.13.5 G832 Roughing axial





Y	Starting point.	Position of tool in Radial direction. This position is the starting point for machining. Machining starts at Y and is reduced with C until Y2= is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z1= until Z2= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=) Angles A and B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
I and K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0, K=0

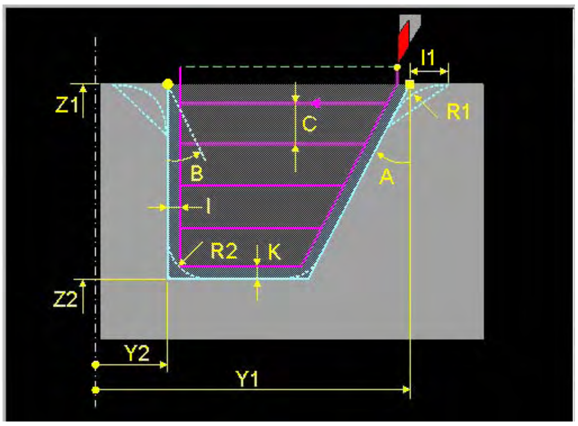
Associated functions

G837 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

32.13.6 G833 Roughing radial



G	Roughing radial
Y	Starting point
Z	Starting point
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Cutting depth
A	Angle 1
B	Angle 2
I1=	Chamfer length 1
R1=	Radius 1
R2=	Radius 2
I	Finishing
K	Finishing

S1=	(Cutting) Speed
F	Feed

Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y1= until Y2= is reached.
Z	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Z and is reduced with C until Z2= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Radial feed depth	Dimension (incremental) by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=) Angles A and B must be chosen so that the tool does not undercut.
B	Angle:	Basic setting B=0. Angle (>0) at contour end point. (Y2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
I and K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0 K=0

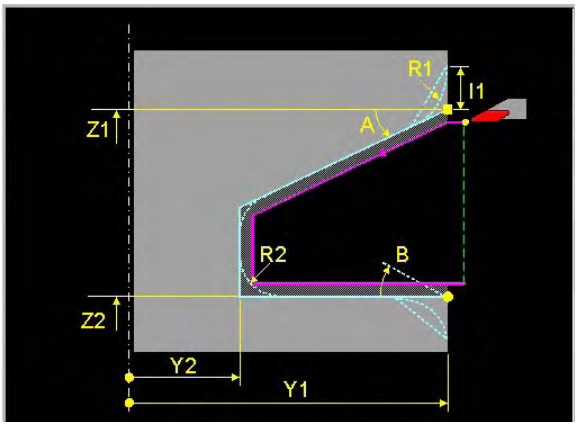
Associated functions

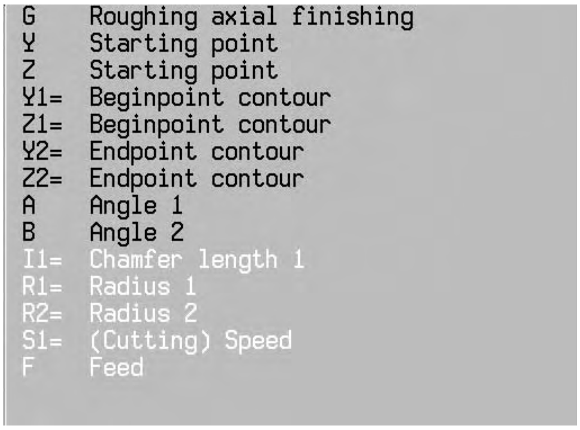
G837 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius.

32.13.7 G836 Roughing axial finishing





Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining. Finish machining starts at Z1= until Z2= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=) Angles A and B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius

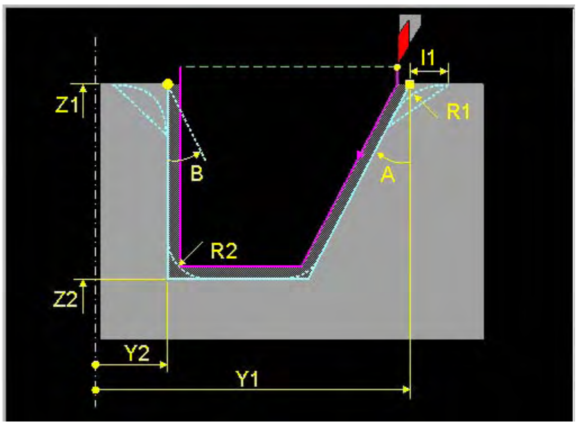
Associated functions

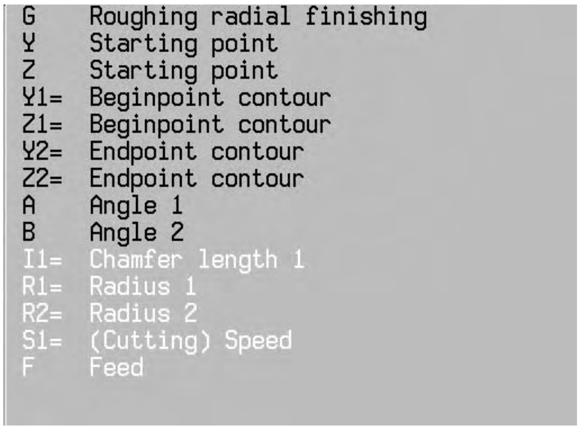
G832 for finish machining

Notes and application

Finish machining goes from Y1/Z1 to Y1/Z2.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

32.13.8 G837 Roughing radial finishing





Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y1= until Y2= is reached.
Z	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=) Angles A and B must be chosen so that the tool does not undercut.
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Y2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius

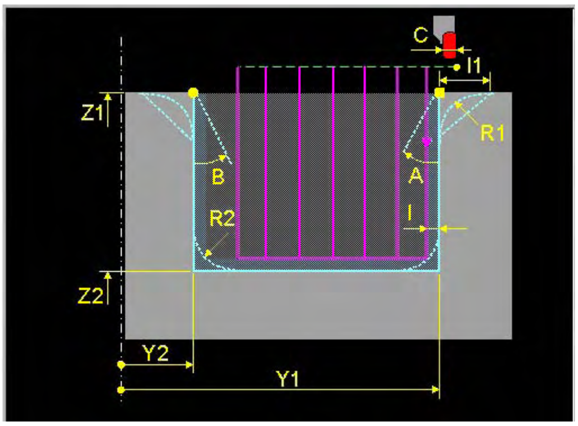
Associated functions

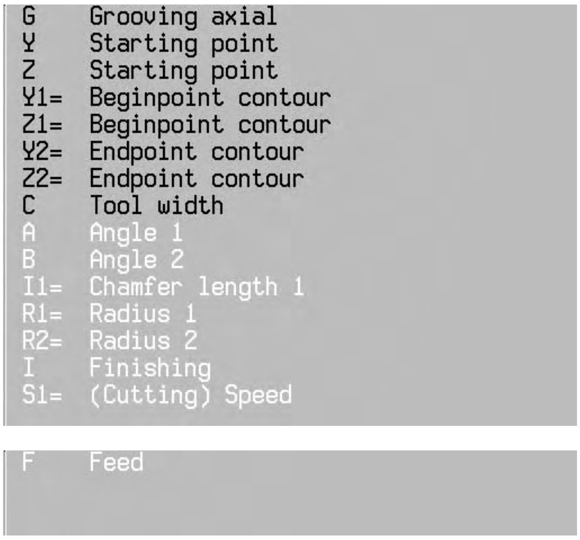
G833 for finish machining

Notes and application

Finish machining goes from Y1/Z1 to Y2/Z1.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius..

32.13.9 G842 Grooving axial





Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y1= with the feed width until Y2= is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the tip radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Y2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool corner radius. Rounding at the bottom of the contour. Finish machining allowance: basic setting I=0.
I		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0

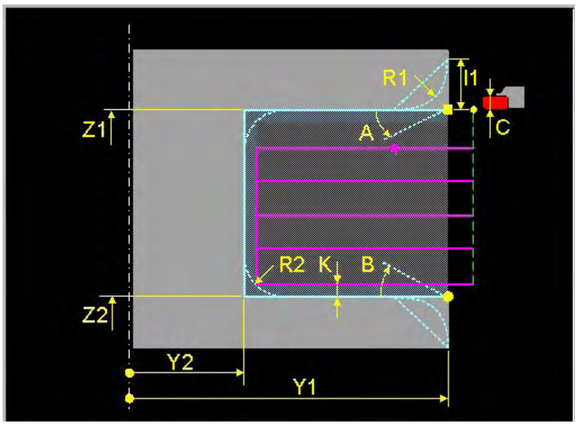
Associated functions

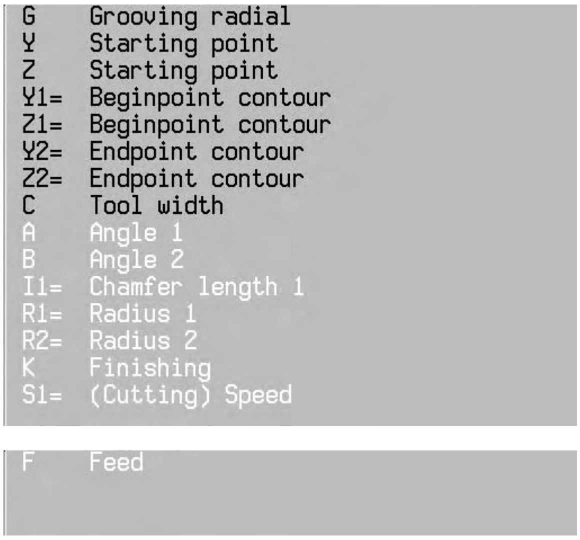
G846 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius..

32.13.10 G843 Grooving radial





Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z2= with the feed width until Z1= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the tip radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, K=0

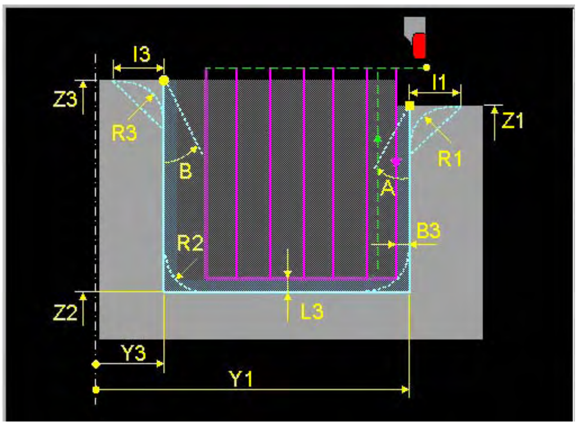
Associated functions

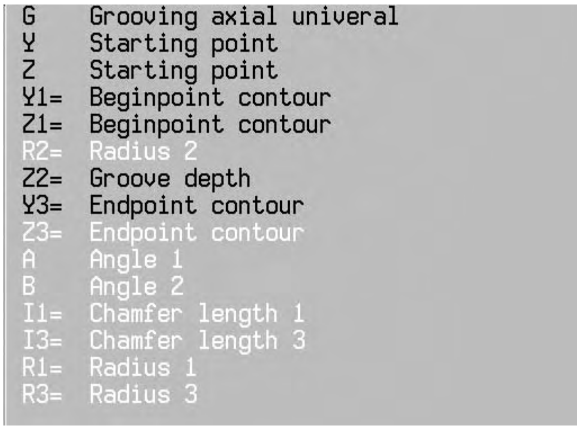
G847 for finish machining

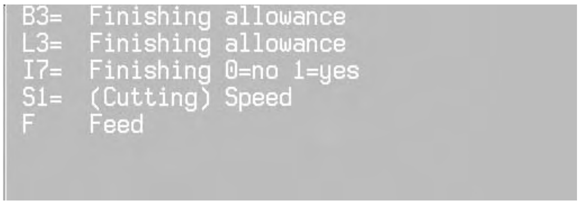
Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

32.13.11 G844 Grooving universal axial roughing







Y, Z	Starting point grooving cycle.
Y1=, Z1=	Contour Starting point
Z2=	Contour bottom
Y3=, Z3=	Contour end point. If Z3 is not programmed then (Z3=Z1)
A	Angle (0-89^0) at groove starting point (Y1, Z1)
B	Angle (0-89^0) at groove end point (Y3, Z3)
I1=	Chamfer length at groove starting point (Y1, Z1)
I3=	Chamfer at groove end point (Y3, Z3)
R1=	Rounding at groove starting point. (Y1, Z1)
R2=	Rounding at both sides of groove bottom.
R3=	Rounding at groove end point (Y3, Z3)
B3=	Finishing allowance along the Z-Axis
L3=	Finishing allowance along the Y-Axis
I7=	Finishing included 0=No 1=Yes

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0, I7=0, B3=0, L3= 0

Associated functions: G848 for finishing

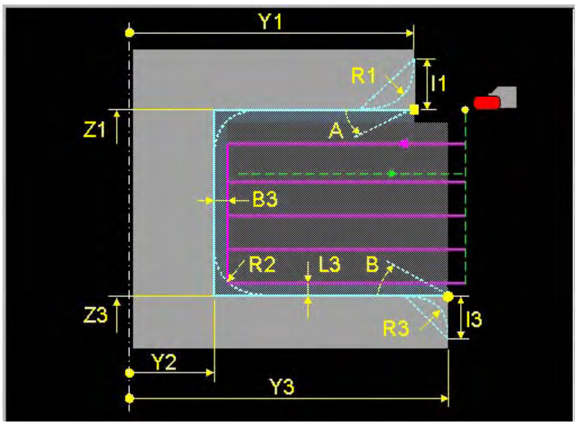
Notes and application

- First grooving (roughing) than, depending on (I7), finishing.
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Groove displacement is (C6-2xC). Maximum displacement is (C6)
- Tool orientation (O):
  . The tool orientation is stored in the tool table
  . With the G-function G302, the tool orientation can be overwritten in the program
  . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45^0 and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

32.13.12 G845 Grooving universal radial roughing



G Grooving radial universal
Y Starting point
Z Starting point
Y1= Beginpoint contour
Z1= Beginpoint contour
Y2= Groove depth
R2= Radius 2
Y3= Endpoint contour
Z3= Endpoint contour
A Angle 1
B Angle 2
I1= Chamfer length 1
I3= Chamfer length 3
R1= Radius 1
R3= Radius 3

B3= Finishing allowance
L3= Finishing allowance
I7= Finishing 0=no 1=yes
S1= (Cutting) Speed
F Feed

Y, Z	Starting point grooving cycle.
Y1=, Z1=	Contour Starting point
Y2=	Contour bottom
Y3=, Z3=	Contour end point. If Y3 is not programmed then (Y3=Y1)
A	Angle (0-89^0) at groove starting point (Y1, Z1)
B	Angle (0-89^0) at groove end point (Y3, Z3)
I1=	Chamfer length at groove starting point (Y1, Z1)
I3=	Chamfer at groove end point (Y3, Z3)
R1=	Rounding at groove starting point. (Y1, Z1)
R2=	Rounding at both sides of groove bottom.
R3=	Rounding at groove end point (Y3, Z3)
B3=	Finishing allowance along the Y-Axis
L3=	Finishing allowance along the Z-Axis
I7=	Finishing included 0=No 1=Yes

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0, I7=0, B3=0, L3= 0

Associated functions: G848 for finishing

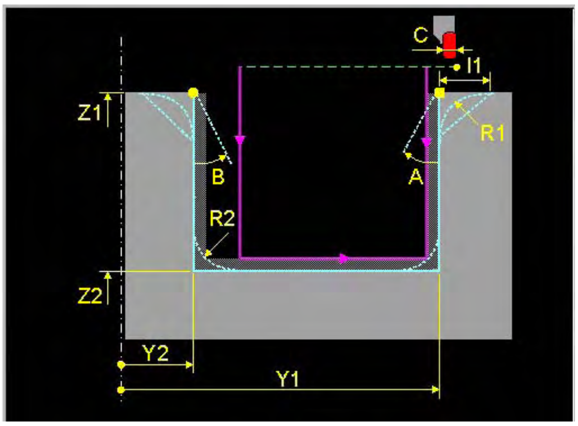
Notes and application

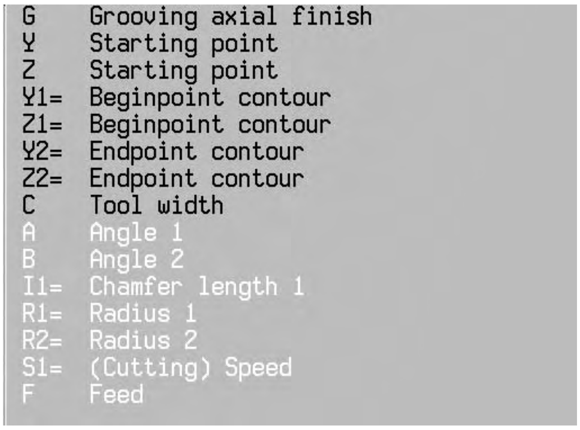
- First grooving (roughing) than, depending on (I7), finishing.
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Groove displacement is (C6-2xC). Maximum displacement is (C6)
- Tool orientation (O):
  . The tool orientation is stored in the tool table
  . With the G-function G302, the tool orientation can be overwritten in the program
  . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45^0 and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

32.13.13 G846 Grooving axial finishing





Y	Starting point.	Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for at Z2= until Z1= is reached.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the corner radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Y1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Y2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
I		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, I=0

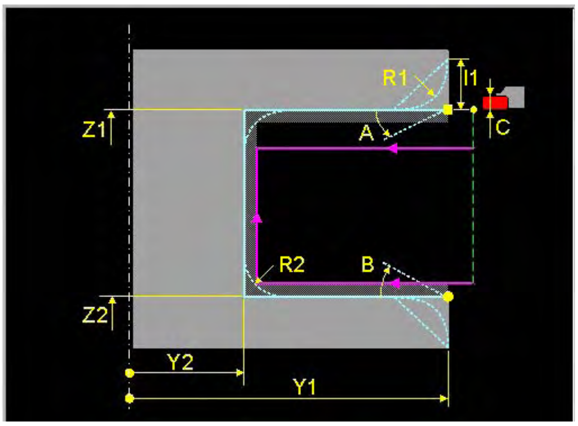
Associated functions

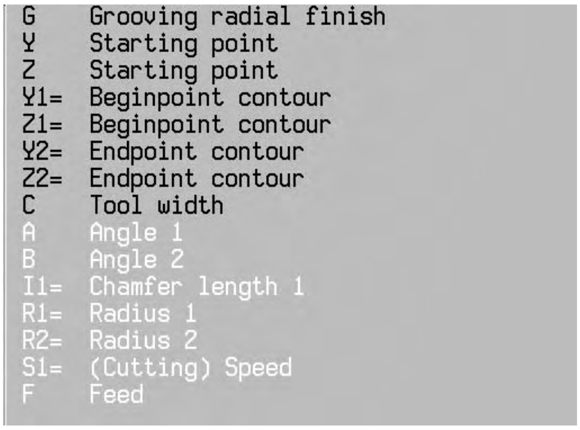
G842 for finish machining

Notes and application

Finish machining goes from Y1/Z1 to Y1/Z2.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius.

32.13.14 G847 Grooving radial finishing





Y	Starting point.	Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y until Y2 is reached.
Z	Starting point.	Position of tool in axial direction. This position is the starting point for finish machining.
Y1=	Contour starting point	Starting point of the contour to be machined.
Z1=	Contour starting point	Starting point of the contour to be machined.
Y2=	Contour end point	End point of the contour to be machined.
Z2=	Contour end point	End point of the contour to be machined.
C	Chisel width	Width of tool. The feed width is C minus twice the corner radius
A	Angle	Basic setting A=0. Angle (>0) at contour starting point. (Z1=)
B	Angle	Basic setting B=0. Angle (>0) at contour end point. (Z2=)
I1=	Chamfer length	Basic setting I1=0. Chamfer length at start and end of contour. Only I1= or R1= may be programmed.
R1=	Rounding	Basic setting R1=0. Rounding at start and end of contour.
R2=	Rounding	Basic setting R2= tool tip radius. Rounding at the bottom of the contour.
K		Stock removal

Basic settings

A=0, B=0, I1=0, R1=0, R2= Tool nose radius, K=

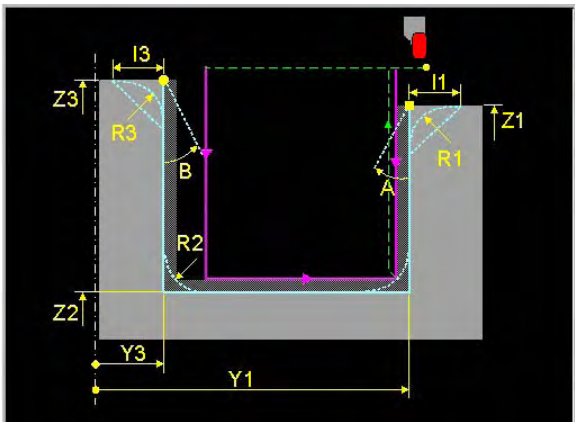
Associated functions

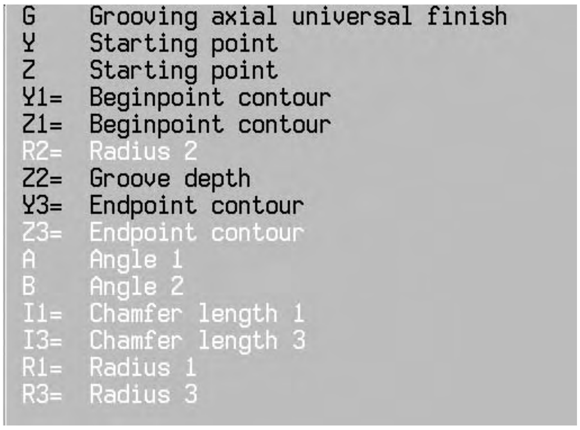
G843 for rough machining

Notes and application

Finish machining goes from Y1/Z2 to Y1/Z1.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius

32.13.15 G848 Grooving universal axial, finishing





Y, Z	Starting point grooving cycle.
Y1=, Z1=	Contour Starting point
Z2=	Contour bottom
Y3=, Z3=	Contour end point. If Z3 is not programmed then (Z3=Z1)
A	Angle ($0-89^0$) at groove starting point (Y1, Z1)
B	Angle ($0-89^0$) at groove end point (Y3, Z3)
I1=	Chamfer length at groove starting point (Y1, Z1)
I3=	Chamfer at groove end point (Y3, Z3)
R1=	Rounding at groove starting point. (Y1, Z1)
R2=	Rounding at both sides of groove bottom.
R3=	Rounding at groove end point (Y3, Z3)

.

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0

Associated functions: G844 for roughing

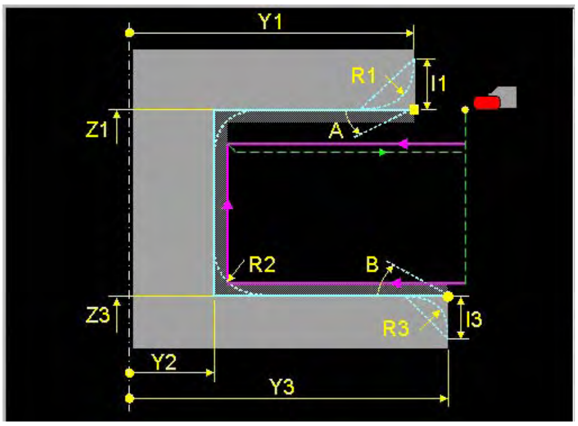
Notes and application

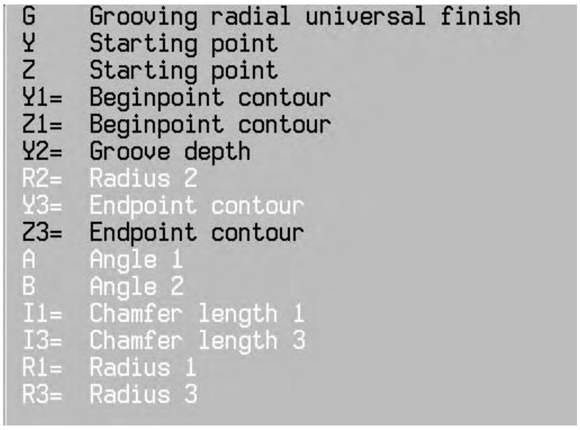
- First the opposite groove side is cut, than the adjoining groove side followed by the groove bottom
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Tool orientation (O):
  . The tool orientation is stored in the tool table
  . With the G-function G302, the tool orientation can be overwritten in the program
  . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45^0 and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

32.13.16 G849 Grooving universal radial, finishing





Y, Z	Starting point grooving cycle.
Y1=, Z1=	Contour Starting point
Y2=	Contour bottom
Y3=, Z3=	Contour end point. If Y3 is not programmed then (Y3=Y1)
A	Angle (0-89^0) at groove starting point (Y1, Z1)
B	Angle (0-89^0) at groove end point (Y3, Z3)
I1=	Chamfer length at groove starting point (Y1, Z1)
I3=	Chamfer at groove end point (Y3, Z3)
R1=	Rounding at groove starting point. (Y1, Z1)
R2=	Rounding at both sides of groove bottom.
R3=	Rounding at groove end point (Y3, Z3)

Basic settings: A=0, B= 0, I1=0, R1= 0, I3=0, R3=0, R2=0

Associated functions: G845 for roughing

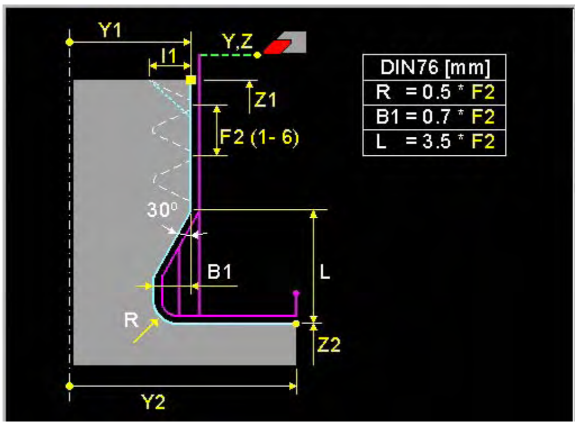
Notes and application

- First the opposite groove side is cut, than the adjoining groove side followed by the groove bottom
- The tool width (C6) is taken from the tool table. An error code appears if the tool width is not available.
- Tool orientation (O):
  . The tool orientation is stored in the tool table
  . With the G-function G302, the tool orientation can be overwritten in the program
  . If there is no tool orientation available, the tool orientation will be calculated from the cycle (sense of machining).
- At the end of the groove, the tool is retracted at an angle of 45^0 and 0.5 mm away from the groove side

Remark:

Make sure that the tool orientation physically corresponds with the actual tool position: Left/Right or In/Outside cutting edge.

32.13.17 G850 Undercut DIN76



G	Undercut (DIN 76)
Y	Starting point
Z	Starting point
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
F2=	Pitch
I1=	Chamfer length 1
S1=	(Cutting) Speed
F	Feed

Y, Z	Starting point undercut cycle.
Y1=, Z1=	Contour starting point
Y2=, Z2=	Contour endpoint.
F2=	Pitch (1-6)
I1=	Chamfer length

Basic settings

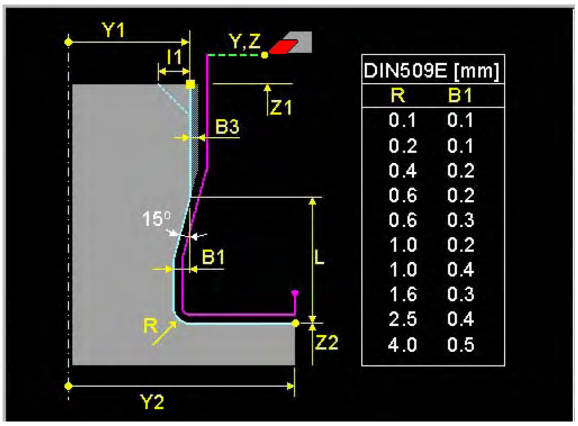
I1=0

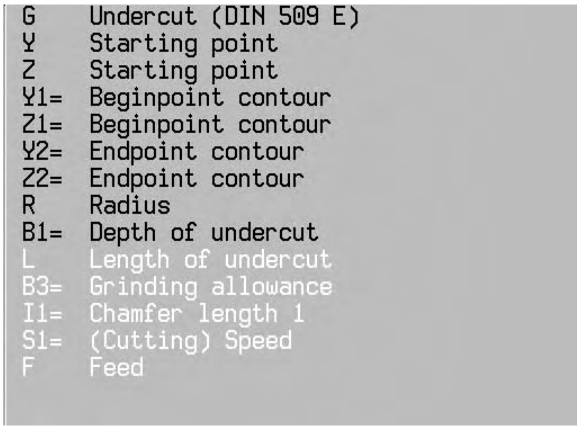
Notes and application

- The undercut contour consists of the elements: Chamfer (Optional), Cylinder, Undercut geometry, Face surface on a pre-cut contour shape.
- Only undercuts conform the DIN-norm can be programmed.
- Undercut (DIN-norm):
  . Length is F2 x 3.5
  . Depth is F2 x 0.7
  . Radius is F2 x 0.5
  . Angle is 30° fixed
- Sequence:

▪ Start motion axis parallel from starting point (Y, Z) to contour starting point (Y1=, Z1=)
▪ Roughing movement of the undercut shape to contour endpoint (Y2=, Z2=).
  Depending on the pitch (F2), the undercut shape will be cut in multiple cuts.
▪ Finishing of the complete undercut shape
▪ At the contour endpoint, the Z-axis retracts 0.1 mm from the contour

32.13.18 G851 Undercut DIN 509 E





Y, Z	Starting point undercut cycle.
Y1=, Z1=	Contour starting point
Y2=, Z2=	Contour endpoint
R	Radius of the undercut shape
B1=	Undercut depth
L	Undercut length
B3=	Finishing allowance.
I1=	Chamfer length

Basic settings

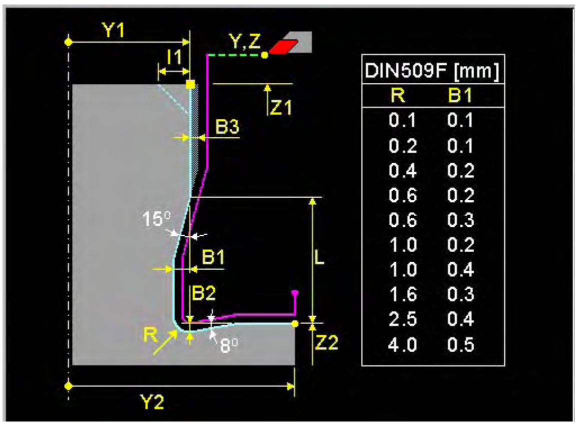
I1=0

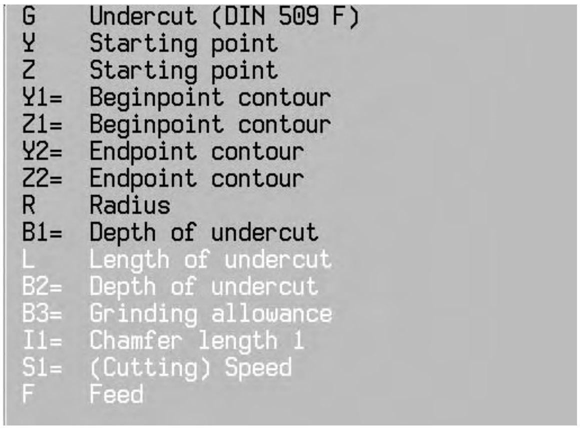
Notes and application

- The undercut contour consists of the elements: Chamfer (Optional), Cylinder, Undercut geometry, Face surface on a pre-cut contour shape.
- Undercuts can be programmed conform the DIN-norm or free-form
  . For DIN-norm undercut values for depth (B1) and radius (R) can be taken from the table.
  . For free form undercuts (B1) and (R) are free programmable
- Sequence:

▪ Start motion axis parallel from starting point (Y, Z) to contour starting point (Y1=, Z1=)
▪ Roughing movement of the undercut shape to contour endpoint (Y2=, Z2=).
▪ Finishing of the complete undercut shape
▪ At the contour endpoint, the Z-axis retracts 0.1 mm from the contour

32.13.19 G852 Undercut DIN 509 F





Y, Z	Starting point undercut cycle.
Y1=, Z1=	Contour starting point
Y2=, Z2=	Contour endpoint
R	Radius of the undercut shape
B1=	Undercut depth
L	Undercut length
B2=	Undercut depth
B3=	Finishing allowance.
I1=	Chamfer length

Basic settings

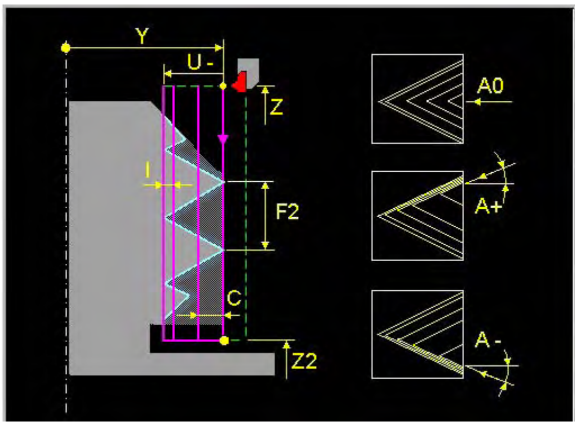
I1=0

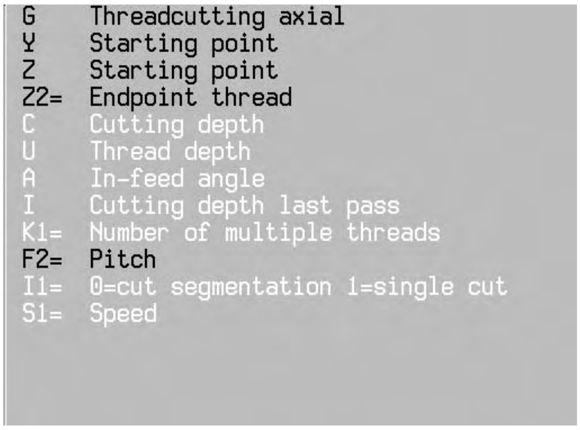
Notes and application

- The undercut contour consists of the elements: Chamfer (Optional), Cylinder, Undercut geometry, Face surface on a pre-cut contour shape.
- Undercuts can be programmed conform the DIN-norm or free-form
  . For DIN-norm undercut values for depth (B1) and radius (R) can be taken from the table.
  . For free form undercuts (B1) and (R) are free programmable
- Sequence:

▪ Start motion axis parallel from starting point (Y, Z) to contour starting point (Y1=, Z1=)
▪ Roughing movement of the undercut shape to contour endpoint (Y2=, Z2=).
▪ Finishing of the complete undercut shape
▪ At the contour endpoint, the Z-axis retracts 0.1 mm from the contour

32.13.20 G861 Threading axial





Y, Z	Starting point threading cycle.
Z2=	End point. At the end point the Y-axis will be retracted at an angle of 90º to (Y) and the Z-axis moves in rapid traverse back to (Z)
C	In-feed depth is calculated from: in-feed angle (A), threading depth (U) and finishing allowance (I). Minimum in-feed depth: 0.002
U	Threading depth (+/- U) is calculated from pitch (F2): Outside thread U = - 0.6495 x F2; Inside thread U = 0.6403 x F2 U –999: Outside thread with calculation (Default) U 999: Inside thread with calculation
A	In-feed angle (Default 28º) A = - 45º < A < 45º ; In-feed along the thread edge. A = 0º ; In-feed only in Y-direction
I	Last cut at thread depth. Minimum value (Default): 0.010
K1=	Number of thread cuts. (Default 1). 1 < K1= < 99
F2=	Pitch in mm/revolution.
I1=	Single cut. The thread will be cut in one pass to depth. (Thread finish)
S1=	Spindle revolution Rev./Min (G97)

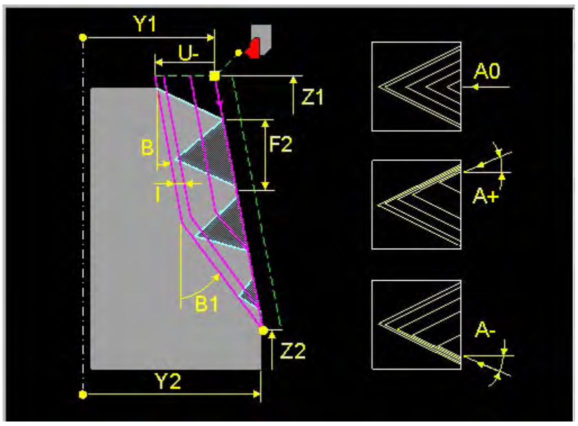
Basic settings

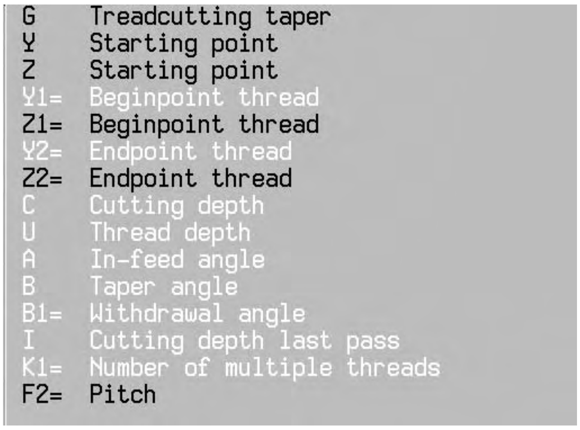
U=+ 999, A=28º, I=0.010, K1=1, I1=0,

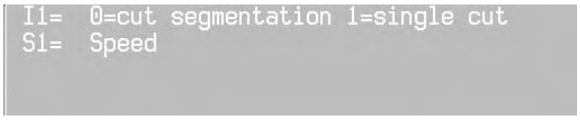
Notes and application

- The turning table should be programmed in revolution/min (S1) (G97).
- Regard the maximum feed (feed is S1= x F2).
- Feed and spindle revolution override is not operational during thread cutting
- The turning table speed is fixed during thread cutting.
- Threading can be interrupted but stops only at the end of the thread cut.
- Regard the turning table direction (M1=03/04) and the tool orientation (O)

32.13.21 G862 Treading conical







Y, Z	Starting point threading cycle.
Z2=	End point. At the end point the Y-axis will be retracted at an angle of 90º to (Y) and the Z-axis moves in rapid traverse back to (Z)
C	In-feed depth is calculated from: in-feed angle (A), threading depth (U) and finishing allowance (I). Minimum in-feed depth: 0.002
U	Threading depth (+/- U) is calculated from pitch (F2): Outside thread U = - 0.6495 x F2; Inside thread U = 0.6403 x F2 U –999: Outside thread with calculation (Default) U 999: Inside thread with calculation
A	In-feed angle (Default 28º) A = - 45º < A < 45º ; In-feed along the thread edge. A = 0º ; In-feed only in Y-direction
I	Last cut at thread depth. Minimum value (Default): 0.010
K1=	Number of thread cuts. (Default 1). 1 < K1= < 99
F2=	Pitch in mm/revolution.
I1=	Single cut. The thread will be cut in one pass to depth. (Thread finish)
S1=	Spindle revolution Rev./Min (G97)
B	Cone angle in relation with the Z-axis (-45º < B < 45º). (B/Y1=) or B/Y2=) has to be programmed.
B1=	Run-out angle at the end of thread (Default 45º) (0º < B1= < 90º)
I	Last cut at thread depth. Minimum value (Default): 0.010
K1=	Number of thread cuts. (Default 1). 1 < K1= < 99
F2=	Pitch in mm/revolution.
I1=	Single cut. The thread will be cut in one pass to depth. (Thread finish)
S1=	Spindle revolution Rev./Min (G97)

Basic settings

U=+ 999, A=28º, I=0.010, K1=1, I1=0,

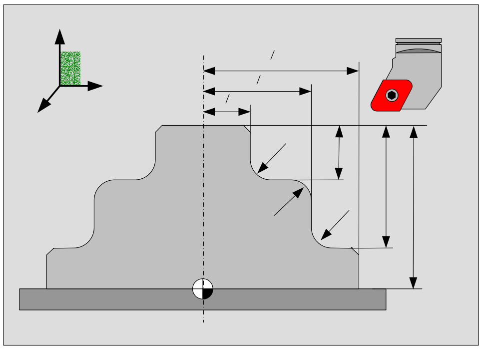
Notes and application (see G861)

32.14 Examples

Example 1

Program	Description
N9999	
N1 G17	Set planes for milling. Length compensation in Z direction.
N2 G37	Milling mode
N3 M54	Head is in the Z direction
N4 T1 M6	Insert milling tool
N5 S1000 F1000 M3	Start Spindle
N...	Milling
N100 G17 Z1=1 Y1=2	Set planes for turning. Main axis 1 is Z, main axis 2 is Y. Radius correction in ZY plane.
N101 G36	Turning
N102 T7 M6	Insert tool
N103 S1=100 M1=3	Start rotary table for continuous turning
N104 G0 X0 Y100 Z100	Position turning tool
N105 G822	Start longitudinal cutting cycles
N...	Turning
N200 G17	Set planes for milling. Length compensation in Z direction.
N201 G37	Milling mode
N203 T1 M6	Insert milling tool
N204 S1000 M3	Start Spindle
N205	Position milling tool
N300 M30	Program end

Example 2: Workpiece drawing Example 2:



Program	Description
N9999	
N1 G17	Set planes for milling. Length compensation in Z direction
N2 G37	Milling mode
N3 G54 I1 Z8	Zero point displacement for Z direction. Upper edge of material is zero
N4 G36	Turning
N5 M54	Head is in the Z direction
N6 G17 Z1=1 Y1=2	Set planes for turning. Main axis 1 is Z, main axis 2 is Y. Radius correction in ZY plane
N7 G195 X-1 Y-1 Z1 I2 J12 K-11.	Set graphics window
N8 G199 X0 Y0 Z0 B4 C2	Start of material graphical contour description. B4 means automatic drawing.
N9 G198 I1=14 X0 Y8 Z0	Start of contour description. I1=14 is light blue colour
N10 G2 X0 Y8 I0 J0	Upper circle of cylinder
N11 G1 X0 Y8 Z-8	Line
N12 G2 X0 Y8 I0 J0	Lower circle of cylinder
N13	End of graphical contour description
N14 T1 M6 (L100 R5 C0.3 Q3=800)	Insert turning tool (length, radius, corner radius and type)
N15 S1=1000 M1=3	Start rotary table for continuous turning
N16 G0 X0 Y8 Z3 F1000	Position turning tool
N17	
N18 G823 Y8 Z0.3 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5 C0.2	G823 start cutting plan cycles. Turn upper part
N19 G823 Y8 Z-2.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5 C0.2	G823 start cutting plan cycles. Turn lower part
N20	
N21 G827 Y8 Z-6.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5	G827 start finish machining cutting plan cycles. Finish machine lower part
N22 G827 Y8 Z-2.7 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5	G827 start finish machining cutting plan cycles. Finish machine upper part
N23 G0 Z10	Move tool clear
N24 T0 M6	Reset tool
N25 G37	Milling mode
N26 G53	Deactivate zero point displacement
N300 M30	Program end

32.15 Survey of permitted G-Functions in the turning mode.

The permitted G-Functions applicable in the turning mode are listed in the tabel underneath. For more information about the G-Functions refer to the control system user manual.

G-Funktions in Turning mode	Explanation
G00	Rapid traverse
G01	Linear interpolation
G02/G03	Circular clockwise/Circular counter clockwise
G04	Dwell time
G14	Repeat function
G17/G18	Main plane
G22	Macro call
G23	Main program call
G25/G26	Enable/Disable feed and spindel override
G27/G28	Reset/Activate positioning functions
G29	Conditional jump
G33	Basic threatcutting movement
G36/G37	Switching turning mode on and off
G39	Activate/Deactivate offset
G40-G41/G42,G43/G44	Tool radius compensation
G45- -50	Measuring cycles
G53/G54- -G59	Cancel/Activate zero point shift
G63/G64	Cancel/Activate geometric calculations
G70/G71	Inch/Metric Programming
G90/G91	Absolute/Incremental programming
G92/G93	Zeropoint shift incremental/absolute
G94/G95	Feed in mm/min or mm/rev
G96/G97	Constant cutting speed
G98/G99, G195, G196, G197/G198, G199	Graphic functions
G227/G228	Unbalance monitor
G300- -G351	Special functions for macros
G611- -G615	Measuring cycles
G691/G692	Unbalance cycles
G822- -G823- -G826- -G827	Clearance cycles
G832- -G833- -G836- -G837	Roughing cyles
G842- -G843- -G846- -G847	Grooving cyles

33. G-functions produced by cycle design

33.1 Cycle Design

Cycle Design allows the user to define his own G functions and integrate them in the control. These G functions can be programmed within part programs using graphics support.

Note

Refer as well to your Programming manual.

34. List of G- and M-functions

34.1 G-functions

G..	Description	Modal
G0	Rapid traverse	√
G1	Linear interpolation	√
G2 **G3**	Circular clockwise Circular counter clockwise	√
G4	Dwell time	-
G6	Spline Interpolation	√
G7	Tilt operating planes	√
G8	Swivel tool	-
G9	Defining polar point (measurement reference point	√
G11	Polar coordinate, Rounding, Chamfering	-
G14	Repeat function	-
G17 **G18** **G19**	Main plane XY, tool Z Main plane XZ, tool Y Main plane XZ, tool Y	√
G22 **G23**	Macro call Main program call	-
G25 **G26**	Enable feed and spindle override Disable feed and spindle override	√
G27 **G28**	Reset positioning functions Activate positioning functions	√
G29	Conditional jump	-
G33 **G36** **G37**	Basic Threadcutting movement Activate turning mode Deactivate turning mode	√
G39	Activate/deactivate offset	√
G40 **G41** **G42** **G43** **G44**	Cancel tool radius compensation Tool radius compensation left Tool radius compensation right Tool radius compensation to end point Tool radius compensation to end point	√

G..	Description	Modal
G45 **G46** **G46 + M26** **G49** **G50**	Measuring a point Measuring a circle Calibrating the measuring probe Checking on tolerances Processing measuring results	-
G51 **G52**	Cancel G52 zero point shift Activate G52 zero point shift	√
G53 **G54** **G55** **G56** **G57** **G58** **G59** **G54 I1 ..** **G54 I99**	Cancel zero point shift (G54-59) Activate zero point shift Activate zero point shift Activate zero point shift Activate zero point shift Activate zero point shift Activate zero point shift Extended zero offset	√
G61 **G62**	Tangential approach Tangential exit	-
G63 **G64**	Cancel geometric calculations Activate geometric calculations	√
G70 **G71**	INCH programming METRIC programming	√
G72 **G73**	Cancel mirror image and scaling Activate mirror image and scaling	√
G74	Absolute position	-
G77	Bolt hole cycle	-
G78	Point definition	-
G79	Activate cycle	-
G81 **G83** **G84** **G85** **G86** **G87** **G88** **G89**	Drilling cycle Deep hole drilling cycle Tapping cycle Reaming cycle Boring cycle Rectangular pocket milling cycle Groove milling cycle Circular pocket milling cycle	√
G90 **G91**	Absolute programming Incremental programming	√
G92 **G93**	Zero point shift incremental rotation Zero point shift absolute rotation	√

G..	Description	Modal
G94 **G95** **G96** **G97**	Feed in mm/min (inch/min) Feed in mm/rev (inch/rev) Constant cutting speed switches off constant cutting speed	√
G98 **G99**	Graphic window definition Graphic: material definition	-
G106 **G108** **G125** **G126** **G136** **G137**	Kinematic Calculation: OFF Kinematic Calculation: ON Lifting tool on intervention: OFF Lifting tool on intervention: ON Second axes configuration for fork head: ON Second axes configuration for fork head: OFF	√
G141	3D-Tool correction with dynamic TCMP	√
G145 **G148** **G149** **G150**	Linear measuring movement Reading measuring probe status Reading tool or offset values Change tool or offset values	-
G153 **G154**	Correct workpiece zero point: OFF Correct workpiece zero point: ON	√
G174	Tool withdrawal movement	-
G180 **G182**	Cancel cylinder interpolation Activate cylinder interpolation	√
G195 **G196** **G197** **G198** **G199**	Graphic window definition End graphic model description Begin inside contour description Begin outside contour description Begin graphic model description	-
G200 **G201** **G202** **G203** **G204** **G205** **G206** **G207** **G208**	Create pocket cycle macro's Start contour pocket cycle End contour pocket cycle Start pocket contour description End pocket contour description Start island contour description End island contour description Call island contour macro Quadrangle contour description	√
G217 **G218** **G227** **G228** **G240** **G241**	Deactivate angular head Activate angular head Unbalance Monitor: OFF Unbalance Monitor: ON Contour check: OFF Contour check: ON	√

34.2 List of G-functions for macros

G..	Description	Modal
G300	Program error call	-
G301	Program halt	
G302	Overwriting radius compensation parameters.	
G303	M19 with programmable direction	
G310	Store table on disk	
G311	Load table from disk	
G318	Read pallet or job table data	
G319	Read actual technology data	
G320	Read actual G-data	
G321	Read tool data	
G322	Read machine constant memory	
G324	Read G-group	
G325	Read M-group	
G326	Read actual position	
G327	Query operation mode	
G329	Query offset from kinematics model	
G331	Write tool data	
G339	Write offset in kinematics model	
G341	Calculation of G7-plane angles	
G350	Display window	
G351	Write to file	

34.3 List of G-functions measurement cycles

G..	description	Modal
G600	Laser: Calibration	-
G601	Laser: Measure tool length	
G602	Laser: Measure length and radius	
G603	Laser: Check of individual edge	
G604	Laser: Tool breakage control	
G606	TT130: Calibration	
G607	TT130: Measuring tool length	
G608	TT130: Measuring tool radius	
G609	TT130: Measuring length and radius	
G610	TT130: Tool breakage control	
G611	TT130: Measuring turning tools	
G615	Laser: Measuring turning tools	
G620	Angle measurement	
G621	Position measurement	
G622	Corner outside measurement	
G623	Corner inside measurement	
G626	Datum outside rectangle	
G627	Datum inside rectangle	
G628	Circle measurement outside	
G629	Circle measurement inside	
G631	Measure the inclination of a plane (G7)	
G633	Position measurement	
G634	Position measurement	
G640	Rotary table center offset.	
G642	Laser: Temperature compensation	
G691	Measure unbalance	
G692	Unbalance checking	

34.4 List of G-functions milling cycles

G..	description	Modal
G700	Facing	
G730	Executing a pass	
G771	Machining on a line	
G772	Machining on a rectangle	
G773	Machining on a grid	
G777	Machining on a circle	
G779	Machining at a position	
G781	Drilling / centring	
G782	Deep drilling	
G783	Deep drilling (chip break)	
G784	Tapping with compensating chuck	
G785	Reaming	
G786	Hollow boring	-
G787	Pocket milling	
G788	Slot milling	
G789	Circular pocket milling	
G790	Reverse countersinking	
G794	Interpolating tapping	
G797	Pocket finishing	
G798	Slot finishing	
G799	Circular pocket finishing	
G691	Measure unbalance	
G692	Unbalance checking	

34.5 List of G-functions turning cycles

G..	description	Modal
G822	Clearance axial	
G823	Clearance radial	
G826	Clearance axial finishing	
G827	Clearance radial finishing	
G832	Roughing axial	
G833	Roughing radial	
G836	Roughing axial finishing	
G837	Roughing radial finishing	
G842	Grooving axial	
G843	Grooving radial	-
G844	Universal grooving axial roughing	
G845	Universal Grooving radial roughing	
G846	Grooving axial finishing	
G847	Grooving radial finishing	
G848	Universal Grooving axial roughing	
G849	Universal Grooving radial roughing	
G850	Undercut (DIN 76)	
G851	Undercut (DIN 509 E)	
G852	Undercut (DIN 509 F)	
G861	Treadcutting cylinder	
G862	Treadcutting taper	

34.6 List of G-functions Laser measurement

G..	Description	Modal
		-
G951	Calibration**.**	
G953	Measure tool length.	
G954	Measure length, radius.	
G955	Cutter control shank	
G956	Tool breakage control.	
G957	Cutter control shape.	
G958	Tool setting length, radius, corner radius	

34.7 Basic M-functions

M..	Early	Late	Description	Modal with:
M0		X	Program stop	-
M1	X		Optional stop	-
M30		X	Program end	-
M3	X		Spindle ON, clockwise rotation	M4,M5,M14,M19
M4	X		Spindle ON, counter-clockwise rotation	M3,M5,M13,M19
M5		X	Spindle STOP	M3,M4,M13,M14
M19		X	Spindle STOP in defined angle position.	M3,M4,M13,M14
M6	X		Automatic tool change	-
M66	X		Manual tool change	-
M7	X		Internal cooling lubrication ON	M9
M8	X		External cooling lubrication ON	M9
M9		X	Coolant OFF	M7,M8,M13,M14
M13	X		Spindle ON – right rotation (M3) and External cooling lubrication ON (M8)	M9
M14	X		Spindle ON – right rotation (M3) and External cooling lubrication ON (M8)	M9
M25	X		Tool measurement activated	-
M26	X		Calibrate measuring calipers	-
M27	X		Activate measuring calipers	M28
M28	X		De-activate touching system	M27
M24			Touch system activated position	
M29			Blow air ON	
M41	X		1.gear step spindle drive.	M42,M43,M44
M42	X		2. gear step spindle drive	M41,M43,M44
M43	X		3. gear step spindle drive	M41,M42,M44
M44	X		4. gear step spindle drive	M41,M42,M43
M67	X		Activate/alter tool correction value	-

34.8 Machine dependent M-functions

M..	Early	Late	Description	Modal with:
M10 **M11** **M22** **M23** **M32** **M33**	 x x x	x x x 	Clamping 4.- or 5. axis ON OFF Clamping 4.- or 5. axis ON OFF Clamping 6. axis ON OFF	-
M16 **M18**	 x	x 	Chip flushing / work piece cleaning OFF Work piece cleaning ON	-
M20	x		Free allocatable NC exit	-
M46	x		Automatic tool exchange - (Axes not participating in the tool exchange are not released.)	-
M53/M54	x		Swivel milling head for horizontal machining	-
M55	x		Release NC cutter head (B axis) -	-
M56 **M57** **M58**			Release 1. travel radius for X axis (B axis) (Modal) Release 2. travel radius for X axis (B axis) Release 3. travel radius for X axis (B axis))	-
M60/M61/ M62	-		Exchange pallets	-
M68			Load/unload tool hopper in the operating area	-
M70 **M71**	x 	 x	Chip conveyor ON Chip conveyor OFF	-
 M74 **M75** **M76** **M77**	 - - - -		Emergency functions: Save function Pallet hopper Save function pallet changer Save function swivel milling head Save function tool changer	-
M80-M89	-		Free M functions	-

35. E Parameters and arithmetic functions

35.1 E parameters

Parameter E..	N.. E..
Format:	
Integer	E1=20
Fixed-point number	E1=200.105
Floating point number (exponent value: -99 - +99)	E1=1.905e5

Change unit of measurement G70 <--> G71:
All values are converted. In this case information such as spindle speed, feed rate, etc., should not be defined as parameters.

E parameters are modal.

Note

The address 'E' (parameter) must be entered into the program as an upper case character.

35.2 Arithmetic functions

Standard arithmetic functions (Blanks not permitted in functions)	E1=E2 E1=E2+E3 E1=E2-E3 E1=E2*E3 E1=E2:E3
Exponentiation	E1=E2^2 E1=(-3)^E3
Reciprocal values	E1=E2^-2(E1=1:E2^2)
Square root (value must be positive)	E1=sqrt(E2)
Absolute values	E1=abs(E2)
Integers	E1=int(E2)
Angle definition Format: Degree/Minutes/Seconds (cannot be entered directly) Input formats	44° 12' 33.5":
Decimal format	E1=44.209303
Angular conversion	E1=44+12:60+33.5:3600
(gives an angle of)	E1=44.209303

Circle constants 'pi' or π (3.14)	E1=(E2*pi):2
Radian format	E1=44+12:60+33.5:3600 E2=((E1:360)*2*pi)rad
Trigonometric functions	sin(E..) cos(E..) tan(E..) asin(E..) acos(E..) atan(E..)
Comparison functions (Condition satisfied --> E..=1) (Condition not satisfied --> E..=0)	E1=E2=E3 --> E1=1 E1=E2<>E3 --> E1=1 E1=E2>E3 --> E1=1 E1=E2>=E3 --> E1=1 E1=E2<E3 --> E1=1 E1=E2<=E3 --> E1=1

Evaluation priority of arithmetic expressions and comparison functions

1. sin, cos, tan, asin, acos, atan, sqrt, abs, int
2. Exponentiation (^), reciprocal values (^-1)
3. Multiplication (*), division (:)
4. Addition (+), subtraction (-)
5. Relational expressions (=, <>, >, >=, <, <=)

If a block contains operations of the same priority, they are executed in sequence from the start of the block to the end.

35.3 Expanded calculation operations

35.3.1 E parameters

Format:

Arc sine E1=asin(E2,E3)
Arc cosine E1=acos(E2,E3)
Arc tangent E1=atan(E2,E3)
Whole number conversion with large value E1=ceil(E2)
Whole number conversion with small value E1=floor(E2)
Rounding E1=round(E2,n) (n is no. of decimal places)
Remainder of division E1=mod(E2,E3)
Sign E1=sign(E2)

Remark: The integer function is changed with the floor function in V420 and higher.

35.3.2 Whole numbers

When using the integer function, the numerical value is rounded, i.e. all figures after the decimal point are ignored.
E1=int(E2)

Example: E2=8.9 results in 8, E2=-8.9 results in –8

35.3.3 Whole numbers with largest value

When using the integer function with the largest value, the numerical value is rounded according to the largest argument.
E1=ceil(E2)

Example: E2=8.9 results in 9, E2=-8.9 results in –8

35.3.4 Whole numbers with smallest value

When using the integer function with the smallest value, the numerical value is rounded

according to the smallest argument.
E1=floor(E2)

Example: E2=8.9 results in 8, E2=-8.9 results in –9

35.3.5 Rounding

When the rounding function is used, the numerical value is rounded according to the number of decimal places.
E1=round(E2,n) (n is number of decimal places)
Remark: If the number of decimal places is not entered, zero is assumed.

Example: n=1 and E2=8.94 results in 8.9, n=1 and E2=-8.94 results in -8.9
n=1 and E2=8.96 results in 9.0, n=1 and E2=-8.96 results in -9.0

35.3.6 Remainder of division

When the remainder function is used, the remainder is returned by the argument.
E1 =mod(E2,E3)
Remarks:
-E1=E2-int(E2:E3)*E3
- If E3 is 0, E2 is returned.
- If E3 is not entered, 1 is assumed.
- The sign is the same as the sign of E1.

Example: E2=5 and E3=3 results in 2, E2=-5 and E3=3 results in –2

35.3.7 Sign

When the sign function is used, the sign is returned.
E1 =sign(E2)

Example: E2=8.9 results in 1, E2=0 results in 0, E2=-8.9 results in -1
Also possible (V429 and higher):
E1=asin(E3,E4) E1=acos(E3,E4) E1=atan(E3,E4) where E2=E3:E4
Remark: - abs(E2) must be less than or equal to 1 for acos and asin.
- the angle created lies between 0° and +360°

35.3.8 Variable parameter no.:

E(value or expression)=<value or expression>

Examples:
E(1)=
E(1.2e1)
E(E1)=
E(E1+E2)=
E(sin(45)*100)=

36. Technological commands

36.1 Feed rate

Feed rate F.. [mm/min | inch/min]
N.. F100

Constant feed rate:

F1=0 Feed rate relative to equidistant. (Starting position)
N.. F.. F1=0

F1=1 Feed rate relative to workpiece contour. The feed is reduced in the case of inside radii.
N.. F.. F1=1

F1=2 Feed rate relative to workpiece contour. The feed is reduced in the case of inside radii and increased for outside radii.
N.. F.. F1=2

F1=3 Feed rate relative to workpiece contour. The feed is increased in the case of outside radii.
N.. F.. F1=3

F2=... Retract feed at G85, infeed at G86/G89, G201 or measuring feed at G145.

F3=... Feed for (negative) infeed movement (insertion).

F4=... Feed for plane movement.

F5=... Feed unit for rotating axes
F5=0 degrees/min (default)
F5=1 mm/min or inches/min

F6=... Local feed within a block

Tool axis:	axis perpendicular to plane of operation (G17, G18, ...).
radial milling direction:	milling in the plane of operation
axial milling direction:	milling in direction of tool axis (only in infeed direction)

Modal parameters F, F1=

36.2 Spindle speed

Spindle speed S.. [rpm]

S parameters are modal.

N.. S600

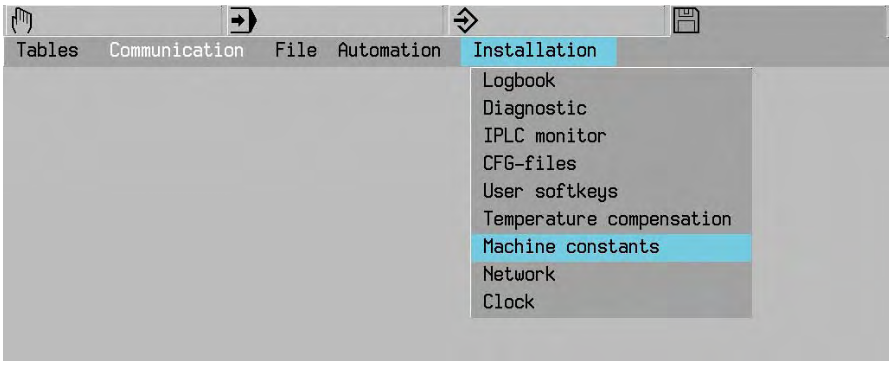
36.3 Tool number

Tool number T.. [Format 8.2] (255 tools max.)	N.. T1 M..
Original tool (T1-T99999999)	N.. T1
Replacement tool (Tx.01-Tx.99)	N.. T1.01
Activation:	
Automatic tool change	N.. T.. M6
Manual tool change	N.. T.. M66
Activate tool data	N.. T.. M67
First additional tool offset	N.. T.. T2=1 M6/M66/M67
Second additional tool offset	N.. T.. T2=2 M6/M66/M67
Tool life T3=..[0-9999,9min]	N.. T.. T3=x M6/M66
Cutting force control T1=..[1..99]	N.. T.. T1=x M6/M66
Deactivate (T1=0 or T1= not programmed)	N.. T1=0
Modal parameters T, T1=, T2=	

37. Miscellaneous

37.1 Operator machine constants

Refer to the documentation provided by the machine builder.



For user

For maintenance/customer service personnel only

37.2 Machine settings monitoring file

In Edit-MC the machine settings which also exist in the monitoring file are displayed with a lock indicator. These machine settings then cannot be edited.
Release for editing purposes is achieved by means of a password.

Machine settings which exist in the monitoring file can only be overwritten if the password has been entered. In this way, the unintentional changing of machine settings is precluded.

Note

The machine settings 250 to 316 inclusive are used for selection of the available options.

37.2.1 List of user machine constants

20 Axes orientation (0=0,1=-90,2=180,3=90) O
21 Spindle power display (0=off, 1=on) O
22 Display G181 (0=fictive, 1=real) O
24 Screensaver time out (0=off,1-255[min]) O
80 Selection demo mode (0=off,1=on,2=IPLC) O
93 BTR memory size (4-1024)[kB] O
251 Technology entry (0=off, >0 = on) O
252 DNC remote function (0=off, >0 =on) O
254 Tool measurement entry (0=off,1=on) O
255 Int.act.contour prog. (0=off, >0 = on) O
262 BTR function (0=off, >0 = on) O
262 BTR function (0=off, >0 = on) O
263 3D tool function (0=off,1=on) O
264 Cylinder interpolator (0=off,1=on) O
265 G6 (spline) function (0=off,1=on) O
266 Universal pocket cycle (0=off, >0 = on) O
271 Erase graphics (0=off, >0 = on) O
272 Synchrone graphics (0=off, >0 = on) O
292 Memory MEX 1 (0=off,??????=on) O
293 Memory MEX 2 (0=off,??????=on) O
294 Memory MEX 3 (0=off,??????=on) O
295 Memory MEX 4 (0=off,??????=on) O
296 Memory MEX 5 (0=off,??????=on) O
297 Memory MEX 6 (0=off,??????=on) O
350 Probe position 1st axis negative [µm] O
351 Probe position 1st axis positive [µm] O

352 Probe position 2nd axis negative [µm] O
353 Probe position 2nd axis positive [µm] O
354 Probe position 3rd axis negative [µm] O
355 Probe position 3rd axis positive [µm] O
714 Scaling mode (0+2=factor,1+3=%,2+3=3D) O
715 Decimal point scaling (0-6) O
772 DIO: line syntax check (0=off,1=on) O
773 DIO: block numbers > 9000 (0=off,1=on) O
774 Tool in (0,1=clear,2=protect,3=replace) O
782 DNC remote directory (0=no, 1=yes) O
783 DNC disk format function (0=no, 1=yes) O
792 IPC remote directory (0=no, 1=yes) O
793 IPC disk format function (0=no, 1=yes) O
795 IPC %-protocol in file (0=no, 1=yes) O
799 MPC %-protocol in file (0=no, 1=yes) O
847 Width fixed measuring probe [µm] O
848 Radius calibration ring [µm] O
901 Dev1: baudrate (110-57600) O
903 Dev1: number of stopbits (1 or 2) O
904 Dev1: leader/trailer length (0-120) O
905 Dev1:data carrier (0=ASCII,1=ISO,2=EIA) O
906 Dev1:auto code recognition (0=off 1=on) O
907 Dev1: flowcontrol (0=RTS,1=RTS-f,2=XON) O
908 Dev1: check DTR (0=no, 1=yes) O
911 Dev2: baudrate (110-57600) O
913 Dev2: number of stopbits (1 or 2) O
914 Dev2: leader/trailer length (0-120) O
915 Dev2:data carrier (0=ASCII,1=ISO,2=EIA) O
916 Dev2:auto code recognition (0=off 1=on) O
917 Dev2: flowcontrol (0=RTS,1=RTS-f,2=XON) O
918 Dev2: check DTR (0=no, 1=yes) O
921 Dev3: baudrate (110-57600) O
923 Dev3: number of stopbits (1 or 2) O
924 Dev3: leader/trailer length (0-120) O
925 Dev3:data carrier (0=ASCII,1=ISO,2=EIA) O
926 Dev3:auto code recognition (0=off 1=on) O
927 Dev3: flowcontrol (0=RTS,1=RTS-f,2=XON) O
928 Dev3: check DTR (0=no, 1=yes) O
931 LSV/2 baudrate (110-57600) O
932 LSV/2 characterset (0=ASCII,1=ISO) O
933 LSV/2 time out period (0-128)[s O
934 LSV/2 nr.of repeats (0=no limit,1-12) O
935 LSV/2 delay time (0-128)[ms] O
936 LSV/2 check DTR (0=no, 1=yes) O

2455 Position fixed measuring probe 1 O
2456 Position fixed measuring probe 2 O
2457 Position calibration ring O
2655 Position fixed measuring probe 1 O
2656 Position fixed measuring probe 2 O
2657 Position calibration ring O
2855 Position fixed measuring probe 1 O
2856 Position fixed measuring probe 2 O
2857 Position calibration ring O
2955 Position fixed measuring probe 1 O
2956 Position fixed measuring probe 2 O
2957 Position calibration ring O
3055 Position fixed measuring probe 1 O
3056 Position fixed measuring probe 2 O
3057 Position calibration ring O
3155 Position fixed measuring probe 1 O
3156 Position fixed measuring probe 2 O
3157 Position calibration ring O
3255 Position fixed measuring probe 1 O
3256 Position fixed measuring probe 2 O
3257 Position calibration ring O
3355 Position fixed measuring probe 1 O
3356 Position fixed measuring probe 2 O
3357 Position calibration ring O
3455 Position fixed measuring probe 1 O
3456 Position fixed measuring probe 2 O
3457 Position calibration ring O
3555 Position fixed measuring probe 1 O
3556 Position fixed measuring probe 2 O
3557 Position calibration ring O
3655 Position fixed measuring probe 1 O
3656 Position fixed measuring probe 2 O
3657 Position calibration ring O
3755 Position fixed measuring probe 1 O
3756 Position fixed measuring probe 2 O
3757 Position calibration ring O
3855 Position fixed measuring probe 1 O
3856 Position fixed measuring probe 2 O
3857 Position calibration ring O
3955 Position fixed measuring probe 1 O
3956 Position fixed measuring probe 2 O
3957 Position calibration ring O
4055 Position fixed measuring probe 1 O
4056 Position fixed measuring probe 2 O
4057 Position calibration ring O
4155 Position fixed measuring probe 1 O
4156 Position fixed measuring probe 2 O
4157 Position calibration ring O
4255 Position fixed measuring probe 1 O
4256 Position fixed measuring probe 2 O
4257 Position calibration ring O

37.3 Connecting cable for data interfaces

Client must ensure that an external interface cable is being used which is shielded on either side.

If a T-switch is being used, the signal ground and shield must not be connected. Mechanical switch-over is only permitted to signal lines.

Should any problems be encountered with the data interface, check for the following:
Is a shielded data cable being used?
Does the length of the data line not exceed 15 metres?
Is the machine connected to the machine socket?

37.4 Configuring the Ethernet interface

Note

The MillPlus should be configured by a network specialist.

The MillPlus is fitted with an Ethernet interface to allow the control to be integrated into your network as a client. The MillPlus transfers data across the Ethernet interface using the TCP/IP protocol (Transmission Control Protocol/Internet Protocol) and the NFS (Network File System). TCP/IP and NFS are widespread in UNIX systems, so you should normally be able to integrate MillPlus into the UNIX world without having to use additional software.
The PC world with its Microsoft operating systems also uses TCP/IP for networking, but not NFS. You will therefore need to install some additional software to enable MillPlus to be integrated into a PC network.
NFS Client in the CNC has been tested with the following network software:

Operating system	**Network software**
Windows NT 4.0	Diskshare NFS server for Windows NT, version 03.02.00.07 (Intergraph, web site: www.intergraph.com).
	Maestro NFS server for Windows NT, version 6.10 (Hummingbird Communications, web site: http:\\www.hummingbird.com). e-mail: support@hummingbird.com
Windows 95	Solstice NFS server, a component from the Solstice Network Client for Windows package, version 3.1 (Sun Microsystems, web site: www.sun.com).
Windows 95/98, NT4.0	Omni-NFS server, (Xlink Technologies Inc., web site: http:\\www.xlink.com).
	CimcoNFS server, (CIMCO Integration, web site: http:\\www.cimco.dk).

37.4.1 Ethernet interface connection options

You can connect the MillPlus Ethernet interface to your network using the RJ45 connector (10BaseT). The connector is galvanically isolated from the control electronics.

RJ45 connector (10BaseT)
Use twisted-pair cables to connect the MillPlus to your network if using the 10BaseT connector.
If using screened cables, the maximum cable run between MillPlus and a node is 400m.

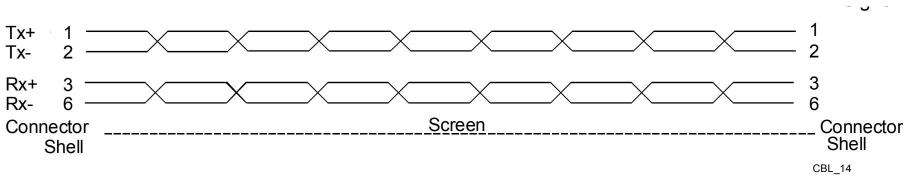
Note

If you connect the MillPlus directly to a PC, crossover cables must be used.

37.4.2 Connecting cable for Ethernet interface

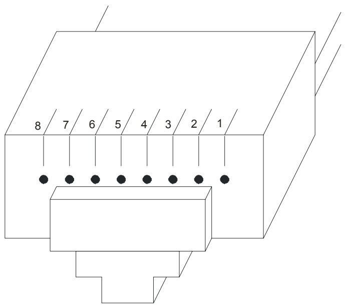
Ethernet interface RJ45 socket

Maximum screened cable run	:400 m
Maximum transmission rate	:200 kBaud to 1 MBaud



Pin	Signal description
1	TX+ Transmit Data
2	TX– Transmit Data
3	REC+ Receive Data
4	spare –
5	spare –
6	REC– Receive Data
7	spare –
8	spare –

Front view of connector



The interface complies with the safe mains isolation requirements of IEC 742 EN 50 178.

37.4.3 Configure MillPlus Ethernet interface (file tcpip.cfg)

Note

The MillPlus should be configured by a network specialist.

Setting up machine constants:

Mc311=0	DNC Plus	(0=off, on=??????)
Mc313=Password	NFS Server	(0=off, on=??????)
??????=Password		

The data connection can be configured using the tcpip.cfg file. The tcpip.cfg file must always be on the C:\ hard disk. A maximum of one local, four hardware, one service, ten NFS servers and ten DNC servers can be defined and managed. The language is always English.

The tcpip.cfg file can be modified from the "HEIDENHAIN NUMERIC Service Menu". The Service menu can be called up while the CNC system is being initialised by pressing the S key on the ASCII keyboard. Select the tcpip.cfg editor using "TCP/IP configuration". A line should have no more than 128 characters. No distinction is made between upper and lower case characters. A comment line is indicated by a semicolon ‘;’. Configuration sections can be repeated. A section is defined by a name in square brackets. ‘[Name]’

Hardware section

This is indicated by the section name [Hardware] and contains the network device parameters. The configuration file may have a number of hardware sections containing settings for several network devices. The 'local' section determines which network device is to be used.

Parameter		Meaning
Type	= <device name>	Name of the network device, e.g. SMC, NE2000, i8255x or AT-lantic
i0	= <irq number>	The parameters i0 to i3 control the allocation of the four interrupt outputs of the network device to the IRQ lines of the CPU. This is determined by the CNC hardware. See "Sample tcpip.cfg file".
i1	= <irq number>	
i2	= <irq number>	
i3	= <irq number>	
Irq	= <irq number>	Defines which IRQ the driver software uses. This number must be one of the numbers defined through i0 to i3.
Iobase	= <iobase address>	Setting of the I/O base address of the network device.

Local section

[local] contains the local parameters for the TCP/IP data communications protocol. There may only be one local section.

Parameter		Meaning
Type	= <device name>	Defines the type of network device in the CNC. The name must correspond to the device name specified in one of hardware sections under Type_Parameter.
Connector	= 10baseT | 10base2	Defines which connection is to be used, 10BaseT (RJ45) or 10Base2 (BNC).
HostName	= < network name>	The name that the MillPlus uses to log on to the network. Network name: must contain no more than 17 letters. If you do not enter a name, the MillPlus will use the Null authentication and not the normal Unix authentication; the parameters UserId, GroupID, DirCreateMode and FileCreateMode will be ignored.
IpAddress	= <IP address>	Address that your network administrator must assign for the MillPlus. Input: four integer values separated by decimal points (0 to 255). Ask your network administrator for the number, e.g. 192.168.0.17
SubnetMask	= <IP adress mask>	The subnet mask used to save on addresses within your network. This defines how many bits of the 32 bit Internet address are to be used for the Subnet-ID and how many for the station ident number e.g. 255.255.255.0 defines 24 bits for the subnet number and 8 bits for the station ident number. Again, ask your network administrator what value to enter.
DefaultRouter	= < Router addr>	Internet address of your default router. Only to be specified if your neetwork consists of various subnets. Input: four integer values separated by decimal points (0 to 255). Ask your network administrator for the number. Enter 0.0.0.0 if your network does not have a router.
Protocol = rfc | ieee		Definition of the transmission protocol. rfc: Ethernet protocol according to RFC 894 ieee: IEEE 802.2/802.3 protocol according to RFC 1042 Default value is 'rfc'.
Timezone= <time zone>		The time parameter of files addressed by NFS. Shown in UTC (Universal Time Coding), commonly known as GMT (Greenwich Mean Time). The Timezone parameter indicates the difference between local time and UTC, e.g. in Frankfurt the local time is UTC+1 (hours), in other words Timezone = -1. Default value is -1.
DncPort	= <port number>	Defines the port number for the DNC service in both the Mill Plus CNC and the DNC service of a remote system. Default port number = 19000
SummerTime	= y | n	The SummerTime parameter determines whether the system is to switch automatically from summer to winter time and vice versa. Default value is y.

NFS server section

[NFS server] marks the start of the NFS server section. This section contains the remote parameter values for the NFS server. The configuration file may have a number of remote sections containing settings for several NFS servers.

Parameter		Meaning
IpAddress	= <IP address>	Defines the IP address of your server. Input: four integer values separated by decimal points. Ask your network administrator for the number, e.g. 192.168.0.1
DeviceName	= <server name>	Name of the NFS server as shown in the MillPlus file administration, e.g. Server_NT1.
RootPath = <Path name>		Directory on the NFS server to which you want to link the MillPlus. The MillPlus is only able to access this directory and its sub-directories. Watch out for upper/lower case when typing the path name.
TimeOut	= <Timeout in ms>	Time in ms that the MillPlus allows to elapse before repeating a Remote Procedure Call that the server did not respond to. Input range: 0 to 100 000. The default value '0' corresponds to a timeout of 700 ms. You should only use higher values if the MillPlus has to communicate with the Server via a number of routers, e.g. for Intergraph and Hummingbird Servers, 1000 ms is sufficient; for Sun’s Solstice Server, 5000 ms is necessary. Ask your network administrator for the value.
rwtimeOut	= 30	Timeout before retrying a read/write operation on NFS files (the time is doubled on each retry of the same record until the timeout value is reached).
ReadSize = <packet size>		Packet size in bytes when receiving data. Input range: 512 to 4096. Input 0: MillPlus uses the optimum packet size as reported by the server. Default value is 1300.
WriteSize = <packet size>		Packet size in bytes when sending data. Input range: 512 to 4096. Input 0: MillPlus uses the optimum packet size as reported by the server. Default value is 1300
HardMount	= y | n	Specifies whether the MillPlus should repeat the Remote Procedure Call until the NFS server responds. y: always repeat n: do not repeat Do not use y if there is no active server on the network.
AutoMount	= y | n	Specifies whether MillPlus should be automatically mounted on the network when it is switched on. y: automount n: no automount
UseUnixId	= y | n	Use 'Unix style' authentication for NFS. y: Unix authentication, uses Userid, GroupId, DirCreateMode and FileCreateMode n: no authentication. Userid, GroupId, DirCreateMode and FileCreateMode are not used. Default value is y.
UserId	= <user Id>	User identification (Unix style): used by NFS to identify the user (the CNC) on the server, e.g. 100. Ask your network administrator for the value.
GroupId	= <group Id>	Specifies which group_identification (Unix style) you use to access files on the network, e.g. 100. Ask your network administrator for the value.
DirCreateMode	= <mode>	Specifies the access rights to directories on the NFS server. Enter value in binary format. Example: 111101000 0: access not permitted 1: access permitted Default value is 0777 (octal).
CaseSensitive	= y | n	Uses or ignores the difference between capitals and small letters when comparing directory or file names during directory searching. Defaults to ‘y’. y: Case sensitive searches. E.g. 1234.pm is different from 1234.PM n: Not case sensitive searches. E.g. 1234.pm is equal to 1234.PM
DncPort	= <port number>	Defines the port number for the DNC service in both the Mill Plus CNC and the DNC service of a remote system. Default port number = 19000
FileCreateMode	= <mode>	Specifies the access rights to files on the NFS server. Enter value in binary format. Example: 111101000 0: access not permitted 1: access permitted Default value is 0777 (octal).

```
1 1 1 1 0 1 0 0 0          = 0750 (Oktalzahl)
| | | | | | | | |
| | | | | | | | └──────────  Alle anderen Benutzer: Suchen
| | | | | | | └────────────  Alle anderen Benutzer: Schreiben
| | | | | | └──────────────  Alle anderen Benutzer: Lesen
| | | | | └────────────────  Arbeitsgruppe:    Suchen
| | | | └──────────────────  Arbeitsgruppe:    Schreiben
| | | └────────────────────  Arbeitsgruppe:    Lesen
| | └──────────────────────  Benutzer:         Suchen
| └────────────────────────  Benutzer:         Schreiben
└──────────────────────────  Benutzer:         Lesen
```

DncServer

[DncServer] indicates a DNC remote server section. It contains the parameter settings for a remote DNC server. One or more DNC remote server sections can be present in the configuration file to define one or more DNC servers. The remote section contains the following parameters:

Parameter		Meaning
IpAddress	= <IP address>	Defines the IP address of your server. Input: four integer values separated by decimal points. Ask your network administrator for the address, e.g. 192.168.0.1
DeviceName	= <server name>	Name of the DNC server as shown in the MillPlus file management, e.g. DMG_Service_1.
TimeOut	= <Timeout in sec.>	Defines the connection timeout in s for the connection between local DNC client and remote DNC server. When the remote DNC server is on the local network set TimeOut to zero. Use non-zero values when the remote DNC server is reached through an external connection such as an ISDN router.

Service

[Service] indicates a DNC remote server section. It contains the parameter settings for a remote DNC server. One or more DNC remote server sections can be present in the configuration file to define one or more DNC servers. The remote section contains the following parameters:

Parameter		Meaning
IpAddress	= <IP address>	Specifies the IP address of your server. Input: four integer values separated by decimal points. Ask your network administrator for the address, e.g. 192.168.254.3
serverName	= <server name>	Name of the DNC server as shown in the MillPlus file management, e.g. DMG_Service_1.
port	= <port number>	Default = 19001
repeatTime	= <Time in sec.>	Default = 10 Sec.
idleTimeout	= <Time in min.>	Default = 15 Min.
request	= @<File name> or <Ascii string>	e.g. @c:\OEM\request.txt.

Sample tcpip.cfg file

```
; TCP/IP configuration file
; More sections of [remote] are allowed   --> more NFS servers to choose
; More sections of [hardware] are allowed --> actually used hw is defined in [local] section
; The keywords with an ';' placed in front can be omitted. The value shown is the default
; value
;
;[hardware]                          ; LE412 HARDWARE
;type          = smc                 ; this hw is an smc network device
;irq           = 9                   ; irq used by network device driver
;i0            = 9                   ; hardware connections of network device to irq's
;i1            = 3
;i2            = 10
;i3            = 11
;iobase        = 0x300               ; io base address of network device
;
;[hardware]                          ; LE422 HARDWARE
;type          = i8255x              ; this hw is an i8255x network device
;irq           = 10                  ; irq used by network device driver
;iobase        = 0xE400              ; io base address of network device
;
[hardware]                           ; VMEBUS HARDWARE
type           = at-lantic           ; this hw is a ne2000 compatible network device
                                     ; note: the VMEbus at/lantic is used in ne2000 compatible mode
irq            = 5                   ; irq used by network device driver
i0             = 3                   ; hardware connections of network device to irq's
i1             = 5
i2             = 9
i3             = 15
iobase         = 0x300 0x240         ; io base address of network device
;
[hardware]                           ; dos_shape_pc

type           = ne2000              ; this hw is a ne2000 compatible network device
                                     ; note: the VMEbus at/lantic is used in ne2000 compatible mode
irq            = 5                   ; irq used by network device driver
iobase         = 0x300               ; io base address of network device
;
[local]                              ; configuration of CNC
type           = ne2000              ; the type of network device used:
                                     ; must match a [hardware] type
connector      = 10base2             ; 10baseT: RJ45 (twisted pair), 10base2: bnc (coax)
hostName       = MillPlusshape       ; CNC network name, maximum of 17 characters
ipAddress      = 170.4.100.16        ; internet address of the CNC ==> ask your network
subnetMask     = 255.255.0.0         ;subnet mask of network     ==> administrator for values
defaultRouter  = 0.0.0.0             ;internet address of default router, 0.0.0.0: no router
                                     ;                      ==> ask your network
                                     ;                          administrator for value
;protocol      = rfc                 ; Link layer protocol used rfc: Ethernet, ieee: IEEE 802
;timezone      = -1                  ; + 1 hour of gmt :gmt + tz == local-> gmt=local - tz!!
;summerTime    = y                   ; use automatic summertime correction (daylight saving)
port           = 19000               ; portnumber DNC service
;
[nfsServer]                          ; configuration of a remote server.
                                     ; more than one remote sections allowed
ipAddress      = 170.4.100.140       ; internet address of the server ==> ask your network
                                     ;                          administrator for value
deviceName     = Intergraph          ; Server name used inside CNC
rootPath       = c:\temp             ; server directory to be mounted as network drive on CNC
                                     ; This must be a shared directory on the NFS server
timeOut        = 50000               ; units in milliseconds for timeout in server connection
                                     ; 0..100 000, 0: timeout set to 700 ms
```

```
;rwtimeOut        = 30               ; timeout used for retry at read/write of NFS-files
                                     ; (time is doubled for each retry of same packet until timeOut)
;readSize         = 1300             ; packet size for data reception: 512 to 4096, or 0 = use
                                     ; server reported packet size
;writeSize        = 1300             ; packet size for data transmission
;hardMount        = n                ; yes/no continue mouting until succesfull
                                     ; don't use 'y' if you're uncertain server is running
autoMount         = n                ; yes/no automatically mount when CNC initialises
;useUnixId        = y                ; use UserId/groupId to identify to the server
userId            = 100              ; Unix style user id for Authentication ==> ask your network
groupId           = 100              ; Unix style group id                   ==> administrator
;dirCreateMode    = 0777             ; Unix style access right for dir-create: Octal number
;fileCreateMode   = 0777             ; Unix style access rights for file-create: Octal number
;
[nfsServer]                          ; configuration of a remote server.
                                     ; more than one remote sections allowed
ipAddress         = 170.4.100.171    ; internet address of the server ==> ask your network
                                     ;                                    administrator for value
deviceName        = Hummingbird      ; Server name used inside CNC
rootPath          = c:\NFS_DATA      ; server directory to be mounted as network drive on CNC
                                     ; This must be a shared directory on the NFS server
timeOut           = 1000             ; units in milliseconds for timeout in server connection
                                     ; 0..100 000, 0: timeout set to 700 ms
;rwtimeOut        = 30               ; timeout used for retry at read/write of NFS-files
                                     ; (time is doubled for each retry of same packet until timeOut)
;readSize         = 1300             ; packet size for data reception: 512 to 4096, or 0 = use
                                     ; server reported packet size
;writeSize        = 1300             ; packet size for data transmission
;hardMount        = n                ; yes/no continue mouting until succesfull
                                     ; don't use 'y' if you're uncertain server is running
autoMount         = n                ; yes/no automatically mount when CNC initialises
;useUnixId        = y                ; use UserId/groupId to identify to the server
userId            = 100              ; Unix style user id for Authentication ==> ask your network
groupId           = 100              ; Unix style group id                   ==> administrator
;dirCreateMode    = 0777             ; Unix style access right for dir-create: Octal number
;fileCreateMode   = 0777             ; Unix style access rights for file-create: Octal number
;
;
[NFSserver]                          ; configuration of a remote server.
                                     ; more than one remote sections allowed
ipAddress         = 170.4.100.194    ; internet address of the server ==> ask your network
                                     ;                                    administrator for value
deviceName        = Solstice         ; Server name used inside CNC
rootPath          = C:\solstice      ; server directory to be mounted as network drive on CNC
                                     ; This must be a shared directory on the NFS server
timeOut           = 6000             ; units in milliseconds for timeout in server connection
                                     ; 0..100 000, 0: timeout set to 700 ms
rwtimeOut         = 600              ; timeout used for retry at read/write of NFS-files
                                     ; (time is doubled for each retry of same packet until timeOut)
;readSize         = 1300             ; packet size for data reception: 512 to 4096, or 0 = use
                                     ; server reported packet size
;writeSize        = 1300             ; packet size for data transmission
;hardMount        = n                ; yes/no continue mouting until succesfull
                                     ; don't use 'y' if you're uncertain server is running
autoMount         = n                ; yes/no automatically mount when CNC initialises
;useUnixId        = y                ; use UserId/groupId to identify to the server
userId            = 100              ; Unix style user id for Authentication ==> ask your network
groupId           = 100              ; Unix style group id                   ==> administrator
;dirCreateMode    = 0777             ; Unix style access right for dir-create: Octal number
;fileCreateMode   = 0777             ; Unix style access rights for file-create: Octal number
;
[NFSserver]                          ; configuration of a remote server.
                                     ; more than one remote sections allowed
ipAddress         = 170.4.100.143    ; internet address of the server ==> ask your network
                                     ;                                    administrator for value
deviceName        = pmeSolstice      ; Server name used inside CNC
rootPath          = d:\solstice      ; server directory to be mounted as network drive on CNC
                                     ; This must be a shared directory on the NFS server
timeOut           = 5000             ; units in milliseconds for timeout in server connection
                                     ; 0..100 000, 0: timeout set to 700 ms
rwtimeOut         = 100              ; timeout used for retry at read/write of NFS-files
                                     ; (time is doubled for each retry of same packet until timeOut)
;readSize         = 1300             ; packet size for data reception: 512 to 4096, or 0 = use
                                     ; server reported packet size
;writeSize        = 1300             ; packet size for data transmission
```

```
;hardMount          = n                 ; yes/no continue mouting until succesfull
                                        ; don't use 'y' if you're uncertain server is running
autoMount           = n                 ; yes/no automatically mount when CNC initialises
;useUnixId          = y                 ; use UserId/groupId to identify to the server
userId              = 100               ; Unix style user id for Authentication ==> ask your network
groupId             = 100               ; Unix style group id          ==> administrator
;dirCreateMode      = 0777              ; Unix style access right for dir-create: Octal number
;fileCreateMode     = 0777              ; Unix style access rights for file-create: Octal number
;
[dncServer]
serverName          = Teleservice       ; alias name for this server (PME-pc)
ipAddress           = 170.4.100.143     ; its ip address
;timeOut            = 1000              ; timeout in connection
;port               = 19000             ; port number for dnc services

[Service]                               ; (MAHO) service centre
serverName          = "Maho Service"    ; alias name for this service
ipAddress           = 170.4.100.140     ; its ip address
request             = "here I am"       ; @fileName/tekst to identify yourself
;IdleTimeOut        = 15                ; disconnect after .. minutes
;port               = 19001             ; port number for service
;repeatTime         = 10                ; repeat time in seconds to connect
;
; end of file
```

HEIDENHAIN

User Manual
Changes and additions
starting with V520

MillPlus IT V530

Valid to
V520/00e
V521/00f
V522/00c
V530/00f

English (en)
06/2007

1

Brief overview

1.1 Brief overview

The changes and additions that were realized in a later version of the MillPlus IT software V520/00 are listed.

These additions are available starting with the following software versions:

- V520/00e
- V521/00f
- V522/00
- V530/00f

Therefore, this manual supplements the V520 User Manual.

Please enquire of the machine manufacturer regarding the functional content of each software version.

V520

Description		Valid starting:	Change:
	Positioning logic based on block search, departure, and return (U-head)	V520/00	Function
	Return movement after block search while turning	V520/00	Text
	Axis allocation in the zero point tables (fork head)	V520/00	Text
G17 / G18	Machining levels while turning	V520/00a	Text
	Turning tool data in the tool table	V520/00	Text
	Related tools	V520/00	Text
G23	Call up the main program	V520/00	Text
G77 / G79	Bolt hole cycle and activate cycle	V520/00	Text
G126	Clear tool during interruption	V520/00	Text
G141	3D tool correction with dynamic TCPM	V520/00d	Function
G303	M19 with programmable direction (deactivated)	V520/00	Text
G325	Query modal M function	V520/00e	Function
G350	Write in window	V520/00a	Text
G691	Measure imbalance	V520/00	Text
G321	Query tool data	V520/00a	Function
G331	Write tool data	V520/00	Function
G801	Turning	V520/00	Text
G802	Milling	V520/00	Text

V521

Description		Valid starting:	Change:
	Palette management	V521/00	Function
	Management	V521/00	Function

	Block search	V521/00	Text
	Machine status with pictogram	V521/00	Function
	Manual axis dialog operation	V521/00	Function
G52	Activating palette zero allowance	V521/00	Function
G615	G615 Laser system: L/R measurement of turning tools (chisel width measurement C6)	V521/00c	Function
G740	Inner thread milling	V521/00	Function
G741	Outer thread milling	V521/00	Function
G880	Machining contour, lengthwise	V521/00	Function
G880	Machining contour lengthwise (cutting width correction C6)	V521/00c	Function
G881	Machining contour, planar	V521/00	Function
G881	Machining contour, planar (cutting width correction C6)	V521/00c	Function
G884	Machining contour, lengthwise (finishing)	V521/00	Function
G884	Machining contour, lengthwise (finishing) (cutting width correction C6 and clearance angle A1)	V521/00c	Function
G885	Machining contour, planar (finishing)	V521/00	Function
G885	Machining contour, planar (finishing) (cutting width correction C6 and clearance angle A1)	V521/00c	Function
	ICP Contour programming for turning	V521/00	Function
	U-head	V521/00	Function

V522

Description		Valid starting:	Change:
G28	G28 Positioning functions, I2= path jerk reduction	V522/00	Function
G39	G39 Activate tool allowance	V522/00	Text
G84	G84 I2=1 for fast acc/dec with small thread	V522/00	Function
G141	G141 3D tool correction with dynamic TCPM	V522/00	Function
G151	Cancel G152	V522/00c	Function
G152	Limitation traverse range	V522/00c	Function
G195	Graphic window definition with begin and end block		
G626	G626 expanded by B3= and B4=	V522/00	Function
G627	G627 expanded by B3= and B4=	V522/00	Function
G628	G628 expanded by D3=	V522/00	Function
G628	G628 expanded by R1=, R2= und O7=	V522/00b	Function
G629	G629 expanded by R1=, R2= und O7=	V522/00b	Function
G636	G636 Measure inner circle (CP)	V522/00	Function
G636	G636 expanded by R1=, R2= und O7=	V522/00b	Function
G646	G646 Determine rotary table centre and height	V522/00c	Funktion
G647	G647 Schwenkkopf-Zentrum ermitteln	V522/00c	Funktion
G648	G648 Schwenktisch-Zentrum ermitteln	V522/00c	Funktion
G771	Machining on a line	V522/00	Function
G772	Machining on a rectangle	V522/00	Function
G773	Machining on a grid	V522/00	Function
G777	Machining on a circle	V522/00	Function
G880	Machining contour, lengthwise (reverse contour direction)	V522/00	Function
G881	Machining contour, planar (reverse contour direction)	V522/00	Function

G884	Machining contour, lengthwise (finishing) (reverse contour direction)	V522/00	Function
G885	Machining contour, planar (finishing) (reverse contour direction)	V522/00	Function
	Movement release after block search	V522/00	Function
Introduction to measuring cycles	Zeropoint setting with measuring cycles G620 and G633 is not possible wenn G7 is active	V522/00b	Function
Introduction to measuring cycles	Example: Set zero point in 90° corner of oblique surface	V522/00b	Text

V530

Description:		**Valid starting:**	**Change:**
G330	G330 Read point definition data	V530/00f	Function
G606	Calibration of table probe (TT) or a combination of laser and table probe (TT).	V530/00a	Function
G611	Measuring turning tool with laser or table probe	V530/00a	Function
G621-G636	Probe orientation	V530/00f	Function
G638	G638 Touch probe calibration with ball	V530/00f	Function
G639	G639 Touch probe calibration	V530/00f	Function
G645	Determining table height	V530/00a	Function
G646, G647, G648	3D QuickSet: take over values in kinematic main element	V530/00a	Function
G710, G711, G714, G715	U-head cutting cycles	V530/00a	Function
G645, G646, G648	Measuring B-A-machine with 3D-QuickSet	V530/00c	Function

2

General information

2.1 Small changes

Positioning logic in U-head mode

Positioning logic is **not** active if a rotation level (e.g. G17 U1=1 Z1=2 or G18 U1=2 Y1=1) is activated in U-head mode.

After, e. g. block pre-select in the U-head mode, all axes operate simultaneously.

Comment:

If no rotation level is active during U-head mode G 180 U1 Y1 Z1, the axes move with positioning logic.

Positioning logic while turning

While turning, there is **no** positioning logic wherever a special level (e.g. G17 Y1=1 Z1=2) is active.

After e. g. block pre-select while turning, all axes move simultaneously.

Axes allocation in the zero point tables

If the machine is equipped with a fork head, the C address is replaced by C2 in the zero point tables (ZO, ZE, and PO), if the fork head is activated.

G17 / G18 Machining levels for turning

While turning, the direction of the angle (positive) and circle (CCW) from the Y axis to the Z axis is defined in the coordinate system (G17= Y1=1 Z1=2 and G18=Y1=1 Z1=2) (see Section 32.4).

Note for programs created with a previous software version:

In turning mode (G36), the definition of the B1 and B2 angles in the G17 Y1=1 Z1=2 and G18 Y1=1 Z1=2 levels was incorrect. B1 and B2 are used in geometry (G64) and for polar coordinates. Existing programs should be corrected by subtracting 90 degrees from the programmed B1 and B2 value.

Example: program lines
Software V511: N... G1 B1=120
Software V520: N... G1 B1=30 (120-90 degrees).

Turning tools in the tool table

The Q3= function in the tool table can only be used if the machine manufacturer has prepared it (see Section 32.8).

Related tools

The tool table contains, e. g. tool T1 with spare tools T1.01 and T1.02.

T1 is swapped in (T1 M6) during automatic tool change (M6).

The spare tool log is now active. If T1 is blocked, a spare tool is automatically swapped in. (T1.01).

T1 is swapped in (T1.01 M6) during automatic tool change (M6).

The spare tool log is **not** activated. If T1.01 is blocked, no spare tool is swapped in. Fault P118 is generated.

Comment:

If tool T1.01 is the last to be measured during tool measurements, the operator does not need to swap out this tool until he/she wishes to continue working with T1. If T1.01 is in the spindle, the T1 tool isn't exchanged for T1 M6.

Pallet management

The pallet management system is a function dependent on machines. MillPlus offers operation that supports this function. For a complete function overview, see the machine documentation supplied by your machine builder.

Zero points

1. A softkey (F2) **Clear table** was added to the pallet zero point table. This completely clears the table.
2. When editing pallet zero points G52 Ixx, the active pallet zero point G52 is adjusted to I0.

Management

The overview in the Work piece status window was expanded with S5 and looks like this:

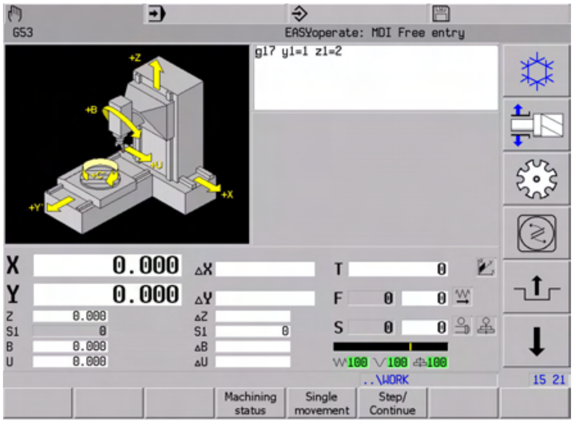
S0	Empty
S1	Blank
S2	Cutting
S3	Ready
S4	Reject
S5	Locked

Block search

Using the block search function, you can search for a block in a machining program and execute the program from this block using the START button.

> Make sure that after the "Block search" the START button is only used in automatic mode immediately after the block search to start the searched-for block.

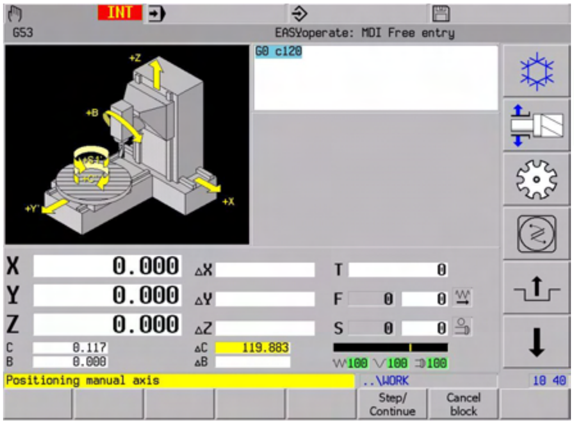
Machine status with pictogram



The machine status display has been expanded with different pictograms for

1. Turntable
   This pictogram appears if G36 is active
2. Machining level
   This pictogram appears if G36 and a machining level are active, e. g.
   - G17 Y1= 1 Z1=2 or G18 Y1= 1 Z1=2
   - G17 U1=1 Z1=2 or G18 U1=2 Y1=1)

Manual axes dialog operation



Introduction

Rotary axes are frequently used as manually adjustable manual axes in addition to the main axes for simple CNC machines. The manual rotary axes are programmed for this in the program and must be brought into position manually.

Sequence

If manual rotary axes must be positioned via the program, the operator is notified on the screen. The program sequence stops, and the following messages appear on the screen:

- INT: Stopping the feed. Stopping the spindle can be manually executed
- The status row shows the message "Position manual axis".
- The display of the distance to go is highlighted in yellow for the affected axes.

The operator turns the manual axis/axes until the remaining distance is set at 0. As soon as the remaining distance is within the tolerance window, the background colour changes to green, and the program can continue to be started. If a manual axis is still not in the tolerance window during a start, the error message "Manual axis not in position" appears.

- If the transversing movement of a manual axis is smaller than the tolerance window, a stop is still executed, and the remaining distance is shown as green.
- Deviations between target and actual positions that are smaller than the programming format (0.001 or 0.0001 degree) are not considered to be a transversing movement and do not cause the program to stop.
- It is not permitted to interpolate NC axes and manual axes. This will result in the "Axis and manual axis not permitted" fault message.

Easyoperate

In EASYoperate mode, the "Ink <>abs" softkey is removed during data entry.

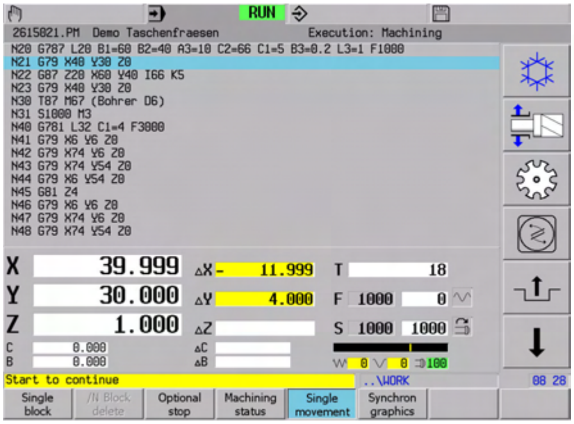
Block search in measuring cycles

When block search in measuring cycles is executed:

- modal Functions G90, G40, G72 and G39 R0 LO are set.
- no correction of zero point shift is done.
- the measuring results in E-Parameters (O1=, O2=, ...) are reset to zero.

In search mode, the handling of measuring results in E-Parameters must be skipped in the part programm. This can be done by checking whether the E-parameter value is equal to zero, or, by checking whether CNC is in search mode by using G148.

2.2 Movement release after block search



The operator him/herself determines the movement release after block search using the "Single movement" softkey.

Attention: the calculated axes movement must be checked for every start. Danger of collision.

Application

The "Movement release after block search" function is activated using MC701 single movement (0:off, 1:on, 2:auto).

0	Option not active: "Single movement" softkey is not available.
1	Option active: "Single movement" softkey is available.
2	The same as 1 except that, after block search, the "Single movement" softkey is automatically selected.

Sequence

1 Controller is on searched block (Default: "Single movement" active).

2 After starting, MillPlus stops with the first movement. The remaining distance of the axis/axes is shown in the machine display field highlighted in yellow. Feed and rapid movement are set to zero.

3 An additional start moves the axes to the next movement. Positioning logic is observed.

4 After deactivation of the "Single movement" softkeys and start, the program continues to run.

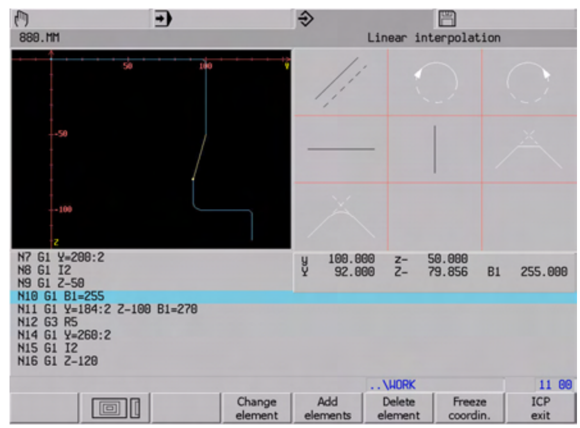
2.3 ICP contour programming for turning

An NC program, e. g. a contour profile, can be created with MillPlus using ICP programming. This NC program is programmed between the geometric functions G63/G64 and can be written either into the main program (*.PM) or into a macro (*.MM).
For contour machining cycles G880 to G885, the ICP program must be written into a macro (*.MM).

Operation

- ICP Milling
  The ICP program is created in the most recently programmed milling level.
- ICP Turning
  The program is created in the rotation level; G17 Y1=1 Z1=2 or G18 Y1=1 Z1=2 (see picture).
  The programming of the geometry program occurs with coordinates Y and Z.

ICP programming can be started in the program editor using the "ICP" softkey and afterwards with the "Mill ICP" or "Turn ICP" softkeys.

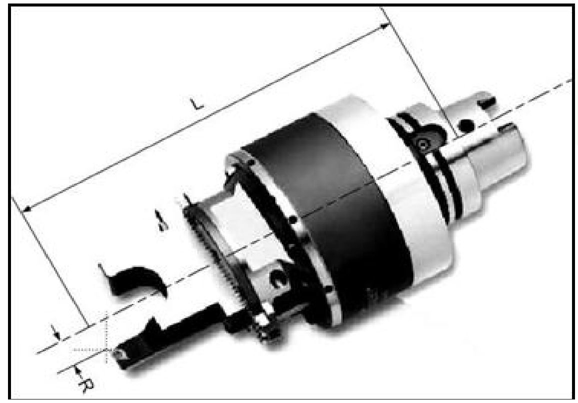


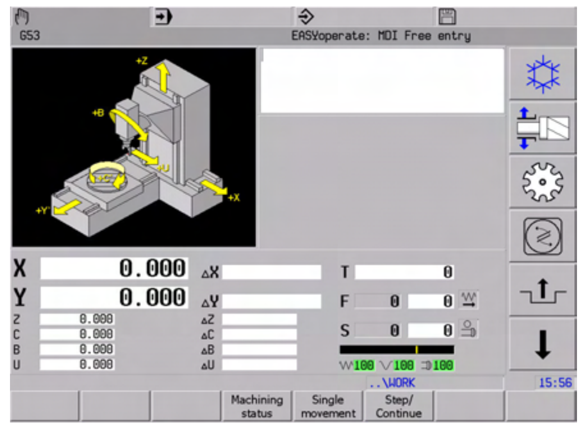
Example: N880.mm (ICP contour macro)

```
N1 G1 Y0 Z0
N2 G64
N3 G1 Y=200:2
N4 G1 I2
N5 G1 Z-50
N6 G1 B1=255
N7 G1 Y=184:2 Z-10 B1=270
N8 G3 R5
N9 G1 Y250:2
N10 G1 I2
N11 G1 Z-120
N12 G63
```

2.4 U-head

The draw bar tool (facing turret, radial facing slide) in the U-axis is used for turning or bore machining (see picture).





Application

Activate tool change, U axis

The draw bar tool is swapped in and out using the usual Txx M6 or M66 command.

- The tool is swapped in and the U axis is automatically referenced with M6.

The U-axis is automatically referenced with M66 after the manual change has ended.

- The M67 function has no effect on the U axis.

Operation

The U axis can only be used if a U-axis tool is in the spindle. If the U axis is used without a U-axis tool, a fault is generated. The U axis can be selected for manual operation (jogging).

U axis coordinate system

The U axis is always present in the display and can only be programmed if the tool is in the spindle. The U axis is defined: G180 U1 Y1 Z1 (U=main axis 1, Y = main axis 2, Z = tool axis). The machining level for tool nose radius compensation is defined with G17 U1=1 Z1=2 or G18 U1=2 Y1=1.

Zero point of the U axis

The position of the U axis must be the true distance to the spindle centre. The zero point displacement U can be useful for e. g. for shape displacement, rough machining, and finishing.

Tool table

The tool is indexed as a special U-axis tool with tool type Q3=9997. The radial compensation of the tool peak is defined by the tool orientation O and the tool radius R (+R4). These addresses are written the same as with turning G36. The difference with respect to turning is that the radius of the U-axis tool is measured in a fixed placement of the U axis. This is the U=R or R=0 position. The cutting radius is entered with the C address. Length L, radius R, and corner radius C are necessary for the CNC.

Tool nose radius compensation

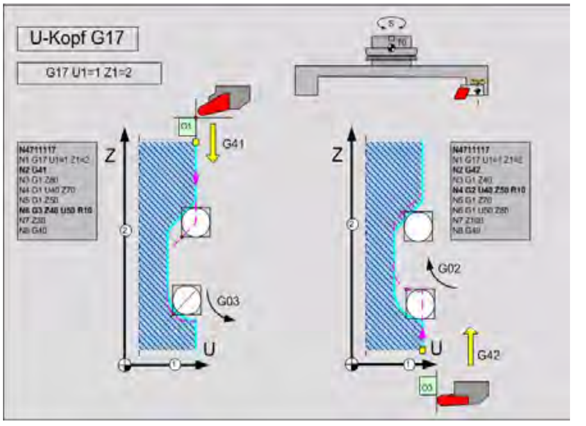
The tool nose radius compensation is programmed with G41 and G42. Before switching on the radius compensation, the level F17 U1 Z1=2 must be programmed. The tool orientation must be programmed with G302 O. The tool is moved in the U-axis direction. Therefore, the radius R is defined as a radius at the position U=0. The effective radius is R+U.

Constant cutting speed

Constant cutting speed is programmed with the G96 S function. The spindle speed for radius is calculated from the actual U-axis position.

Measure tool

The tools can be measured using the BLUM laser system. G615 Laser: Turning tool measurement.

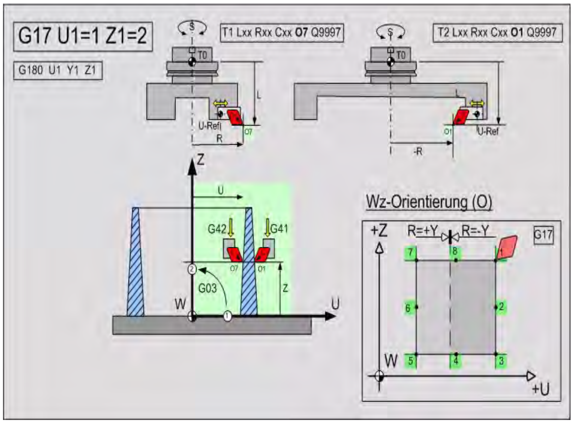


Programming

Coordinate system

In order to define the coordinate system, the G180 function must be used.

An example of a coordinate system: UYZ, G180 U1 Y1 Z1 (see picture).

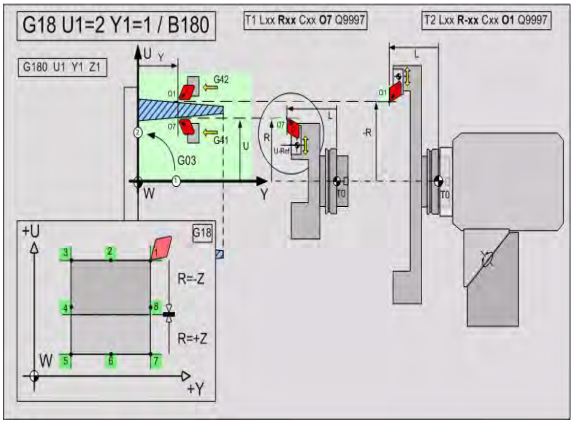


Machining level

As with the other turning tools, the working level is defined by two main axes. The definition of these two axes must be programmed by the G17 or G18 functions and their corresponding arguments. If a U-axis tool is used for turning, a main axis must be defined as the U axis. The other main axis must be vertical to the U axis and parallel to the tool axis.

Example: G17 and G18 configuration

UZ level (G17 U1=1 Z1=2), the U axis as the first main axis and the Z axis as second main axis (or G18 U1=2 Y1=1) (see pictures).



Swivel machining level

The U axis is not a part of the swivelled machining level (G7). Activating G7 therefore has no effect on the positions in the U axis.

Zero allowance

Zero allowance G54, G54 I1 = and G93 U

Absolute and incremental coordinates

Movements in the U axis can be programmed absolutely with G90 or incrementally with G91.

Tool radius compensation

Turning tools have a radius (C) at the tool tip. During machining of conicals, phases, and radii, inaccuracy problems occur, which can be corrected by tool nose radius compensation. Programmed travel paths are based on tool cutting point S. The tool nose radius compensation calculates a new travel path (equidistant) to compensate for this fault.

Switch tool radius compensation on and off

The tool nose radius compensation is switched on and off with the following G functions:

- G40: Switched off

- G41: Tool is to the left of the contour side

- G42: Tool is to the right of the contour side

During switch-on and switch-off, the tool must have sufficient lead and trail cut to completely cut the contour side.

Programming unit

The U axis can be programmed in inches (G70) or metric (G71).

Absolute position

The G74 absolute position function is not permitted in conjunction with U-axis tools!

Contour check

The contour check (G241) produces a fault during production if the programmed shape cannot be manufactured.

Correct tool

Do not use G8 with the U axis

Move reference point

It is not necessary to manually move reference point. The U axis is automatically moved during swap-in. If the tool is in the spindle, it can be **activated with M141 and deactivated with M142**.

Attention
Make sure that the position of the U axis is always referenced.
For example, after changing an MC, starting up the CNC, or programming the G180, the position of the U axis is unknown. Using M141, the draw bar tool must be referenced again.

Constant cutting speed

Constant cutting speed is activated using G96 S. The G96 function calculates the feed in [mm/min (inch/min)] using the programmed feed in [mm/rev], [inch/rev] and the active spindle speed.

Withdrawal movement

The withdrawal movement of the tool must always be executed in the direction of the tool axis. Use G174 for this.
If G126 is being programmed, a FAULT is generated.

Interrupt

Movements in the U axis can be interrupted.

Block search

All active axes, including the U axis, are included in the block search. Movements in the U-axis direction are only valid if a U-axis tool is in the spindle.

Positioning logic after block search, departure, and return

During the return to the contour, the axes move with positioning logic:
1. Rotary axes, minor axes, and main axes
2. U axis

Return movement after block search with active U axis

After block pre-select with active U axis, the linear axes move in a direction towards the return position without positioning logic.

Note:
The return movement is dependent on the current machining level. While turning, there is a special level, e.g. G17 U1=1 Z1=2, G18 U1=2 Y1=1, active, and the special level doesn't use positioning logic.

Jogging and manual wheel

The U-axis tool can be moved manually by jogging or with a manual wheel

Simulation

Simulation is possible in the wiring graphic. Simulation graphic is not possible!

2.5 Introduction to measuring cycles

Zero point

> Wenn G7 is active, it is not possible to store the measured angle in zero point with G620 or G633 I5=2. Program cycles G620 and G633 with I5=0 O3=.. and use the E-parameter in an incremental G7-shift, for example G7 C6=E10 L1=1.

Explanation of addresses

Next adresses are used in most cycles. Specific adresses are described at the cycle.

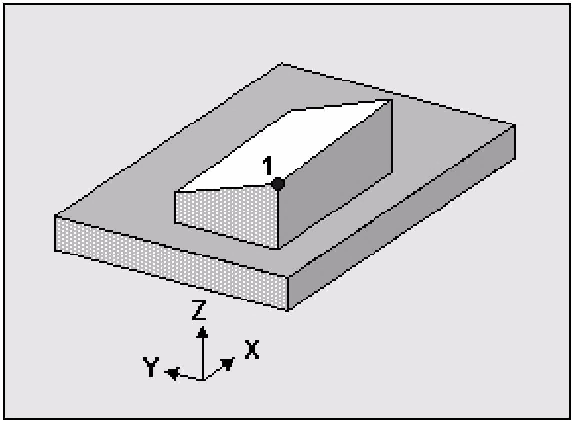
- **B3= Distance to corner in main axis** Distance between startpoint and workpiece corner in main axis.
- **B4= Distance to corner in minor axis** Distance between start-point and workpiece corner in minor axis.
- **O1= untill O7= Storage of measuring value** Measuring values can be stored in an E-parameter. The number of the E-parameter must be entered. In case no number is entered, no value will be stored. Example: programming O1=10 means that the measuring value is stored in E-parameter E10.
- **I2= Probe orientation**
  - **I2=-1** Automatic selection of orientation method depending on probe type (default).
    MC846=0,1: Measuring without orientation, as for I2=0
    MC846=2: Measuring with rotation, as for I2=1
    MC846=3: Measuring with orientation, as for I2=2
  - **I2=0** Measuring without orientation
  - **I2=1** Measuring with rotation
  - **I2=2** Measuring with orientation in measurement direction

> If I2=2 is programmed, the oriented tool radius R1= must be entered in the tool table.

Example: set zero point in 90° corner of oblique surface

In this example a zero point is set in the 90° corner of an oblique surface. The following sequence must be considered:

- The surface must be tilted perpendicular to the tool-axis by:
  - Measuring the space angles of the surface by means of a 3-points measurement (G631).
  - Positioning the surface perpendicular to the tool-axis by executing a G7-movement with the measured space angles.
- Bring the surface parallel to the X-axis by:
  - Measuring the angle between surface and X-axis (G620).
  - Rotating the surface to the X-axis by executing a G7 rotary axis shift using the measured angle.
- Measuring the position of the corner (see 1 in picture) and storing the measured position in the active zeropoint shift (G622).
- Measuring the upper side of the workpiece and storing the measured position in the active zeropoint shift (G621).



`G17`	Activation of G17-plane.
`G54 I1 X0 Y0 Z0 B0 C0 B4=0`	Reset and activation of zero point.
`T1 M6`	Tool change.
`G631 I1=-3 X100 Y5 Z1 X1=165 Y1=5 Z1=15 X2=100 Y2=45 Z2=3 O1=10 O2=11 O3=12`	Measuring obliqueness and storage of absolute space angles A5=, B5= and C5= in E10, E11 and E12.
`G0 X100 Y5 Z100`	Rapid positioning.
`G7 A5=E10 B5=E11 C5=E12 L1=1`	Positioning the surface perpendicular to the tool.
`G620 I1=2 X0 Y0 Z10 B1=20 B2=5 C1=10 L2=100 I5=0 O3=14 F2=150`	Measurement of angle between long side of rectangle and X-axis and storage of this angle in E14.
`G7 C6=E14 L1=1`	Shifting the X-axis parallel to the long side of the rectangle.
`G622 I4=1 X12 Y1 Z18 B3=20 C1=10 L2=100 I5=1 O1=16 O2=17 F2=150`	Measuring corner 1 and storage of this position in both zero point and E16 and E17.
`G621 I1=-3 X10 Y10 Z22 C1=10 L2=100 I5=1 O1=18 F2=150`	Measuring the upper side of the workpiece and storage of the position in both zero point and E18.
`M30`	End of program.

2.6 Tool measurement cycles for Table probe measuring systems (TT)

TT stands for "Tisch-Taster", for example TT130 or a similar device.

Notes for measuring system "Table probe" (TT)

Application

Machine tool and CNC must be prepared for the measurement device by the machine tool manufacturer. If the machine does not provide all described G-functions, please read the machine manual.

Programming

Before one of the G-functions G606-G611 can be executed, an M24 (activating measuring probe) must be programmed, so that the measurement device is set in the right position. After the cycles, M28 (switching off measuring probe) must be programmed, so that the measurement device is retracted again.

Machine constants

G-function and accessory machine constants are activated by the following machine constants:

MC261>0	Measuring cycle function
MC254>0	Tool measurement
MC840=1	Measuring probe connected
MC854=2	Tool measurement device (0=none, 1=laser, 2=probe)

MC350	Probe position 1st axis µm
MC352	Probe position 2nd axis µm
MC354	Probe position 3rd axis µm
	The TT Stylus centre coordinates are based on machine zero point G51 and G53. After calibration, the exact positions are stored in MC350 till MC355.
MC356	Measuring: radial axis (1=X, 2=Y, 3=Z)
MC357	Measuring: tool axis (1=X, 2=Y, 3=Z)
MC358	Measurement: 3. axis 0=no, 1=yes
MC359	Radial touch side: -1=neg, 0=aut, 1=pos
MC360-MC369	Second measurement device in another execution area or a second spindle. Which range is used, is defined by IPLC.
MC391	Calibration radial: 2 sides 0=no,1=yes
MC392	Maximum measuring error for tool measurement with rotating tool (2 - 1000 µm)
MC393	Stylus width in cross direction (µm)

MC394	Touch feed for non-rotating tool. (10 - 3000 mm/min)
MC395	Half of stylus height, used for tool radius measurement. (1 - 100000 µm)
MC396	Stylus width in radial direction (1 - 100000 µm)
MC397	Safety zone around stylus of TT for prepositioning. (1 - 10000 µm)
MC398	Rapid feed in measuring cycle for TT. (10 - 10000 mm/min)
MC399	Maximum surface speed of the tool tip. (1 - 120 m/min)
MC400-409	Defined for a combination of TT and laser.

3

G functions

3.1 G23 Main program call

In the description of the G23 function, it says "N** G23 N1007" in several spots.
This information must read as follows: "N** G23 N=1007".

3.2 G28 Positioning functions

Contour smoothing by path jerk reduction.

Address description

- ▶ **I2= Path jerk reduction [%]**

Default

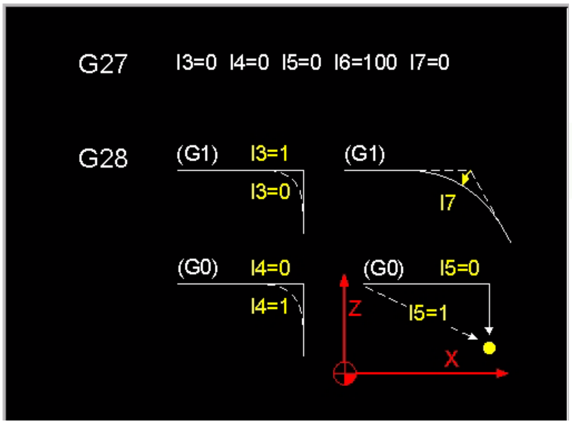
I2=100 to be compatible with existing programs.

Application

With path jerk reduction a contour smoothing is achieved. Corners are rounded and at the same time axis velocity is reduced. Consequently the axis will move more quietly. The difference with corner rounding by contour tolerance (I7=) is the increased axis velocity at the corners if I7= is programmed. The value of I2= lies between 10 and 100 %.

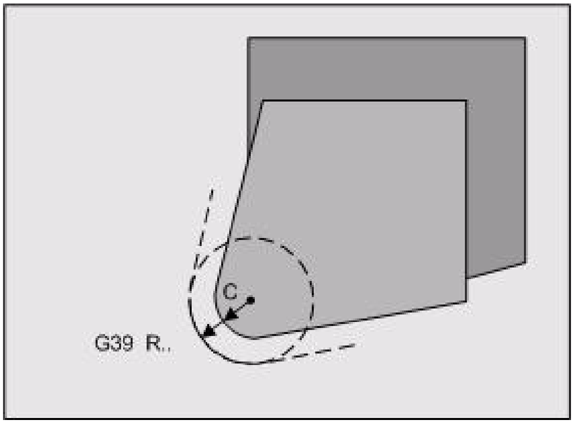
I2=100 (%) corresponds with the tolerance programmed in I7=.

Path jerk reduction (I2=)	**Obtained accuracy**
100	Contour tolerance I7=
50	1.5 * contour tolerance I7=
10	2.0 * contour tolerance I7=




```
G     Positioning functions
I2=   Path jerk reduction             [%]
I3=   Feed movement 0=inpos, 1=inpod
I4=   Rapid movement 0=inpod, 1=inpos
I5=   Position logic: 0=with, 1=without
I6=   Acceleration/jerk reduction     [%]
I7=   Contour tolerance
```

3.3 G39 Activate tool allowance



Tool radius allowance

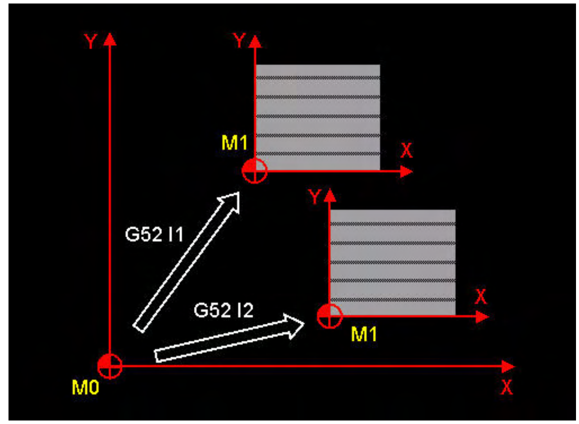
The allowance R has influence on the tool nose radius C during turning (G36) and is only effective with active radius correction.

The allowance of the tool nose radius is added to the centrepoint of the nose radius (the same as orientation 0), and is therefore independent of the active tool orientation.

3.4 G52 Activate pallet zero point

The coordinate values of several pallet zero points can be entered into the pallet zero point table.

> Pallet zero points are used to automate the pallet control. These zero points are activated by the IPLC program using G52 Ixx, wherein xx corresponds to the pallet zero point. In the NC program, the selected zero point can be switched off using G51 and switched back on using G52. The program is thus independent of the pallet number.




```
G     Activate pallet zero point
I     Zero point index
```

Format

Activate pallet zero point with:

G52, I0	Activate zero point value in G52 10 or activate a single pallet zero point.
G52, Ixx	Activate zero point value in G52 Ixx and copy in I0.

Address description

- **I Zero point index** Index number of the zero point that must be activated.

Notes and usage

Modality

G52 is modal with G51.

Associated functions

G51, G52, G52 I [no.], G53, G54... G59, G54 I [no.], G92, G93, G149, G150

Number of zero points

The number of possible zero points in the table is determined by a machine constant (MC26) (0<=99). MC26 is only present if MC84>0.

Change the machine constant MC26

When increasing or decreasing (MC26 > 0), the number of zero points is adapted in the table. The existing zero points are retained. Additional zero points are initialized to zero.

> If MC26 is set to zero, the table (PO.PO) is reduced to one block.
> All entered values are then deleted.
> In addition, no index Ixx can be programmed.

Activating a pallet zero point

When changing palletes (M60/M61), the PLC can activate G52 Ixx using a machine macro.

Comment: G52 Ixx can also be activated in the parts program. While activating, the active zero point allowance is copied into G52 I0.

Enter values into zero point table

A zero point can contain up to 6 axis coordinates.
The coordinate values of the zero points G52 Ixx are entered into the zero point table, before the program execution, using the control panel or from a data carrier.

Comment: If the zero point values of an active allowance are changed, these values are automatically accepted into I0. I0 itself cannot be directly edited or read in or out.

Machine zero points

If a tool machine has several palletes or tables, the information from several zero points is necessary. The zero points always refer to the geometric machine zero point (MO). The distances in the axes measured from the zero point (MO), indicate the position of these zero points and are entered in the pallet zero point table.

G52 Zero point allowance

G52 does not influence function G54 I-[no.]. If G52 is active, G54 I-[no.] of this allowance is effective.

Absolute/incremental zero point allowances G92/G93

A programmed zero point allowance (G92 or G93) is deleted by G52 I-[no.]

Increasing / decreasing, mirroring, and axis rotation (G73, G92/ G93)

Use of G52 I-[no.] in a program section that should be increased/ decreased, mirrored, or rotated is permitted. The zero point allowance occurs in the coordinate system of the tool machine and is not influenced by the programmed coordinate change.

Deactivating a pallet zero point

- G52 I-[no.] is deactivated using the CLEAR CONTROL softkey and via programming of G51.
- 10 is deleted using the CLEAR CONTROL softkey and when the table is deleted.
- G52 I-[no.] is not deactivated using the CANCEL PROGRAM softkey or M30.

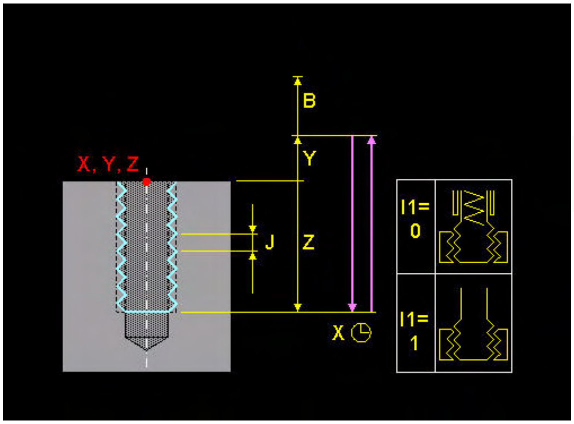
3.5 G77 / G79 Bolt hole circle and activate cycle

Calculate kinematics.
No rotary axes (A, B, C) may be programmed with G77 and G79 (error message O141).

Typically, error message O144 occurs during a block search for G79, if the search was made via a programmed **rotary axis movement in the tool head**. In this case, the tool head must first be placed into the desired position.

Added starting with version V520/00e:
Error message O144 does not occur if G7 and/or G8 are active or if the movement is less than 0.01 degree.

3.6 G84 Tapping cycle



Quick retracting or extending while thread-cutting to avoid tool breakage when thread-cutting with small radii.

Address description

- **I2= fast acc/dec (0=off,1=on)**

Default

I2=0 for compatibility with existing programs.

Application

Only effective with interpolated thread-cutting (I1=1).

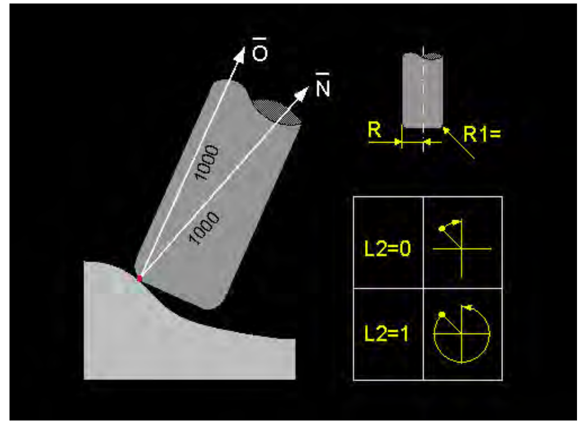
MC726 is maximum jerk for G84.

3.7 G126 Lifting tool on intervention

In the description of the G126 function, it says in one spot: "MC756".
This must read as follows: "MC758".

3.8 G141 3D Tool correction

Allows the tool dimensions to be corrected for a 3D tool path, generated from short, straight sections, with 3-axis and 5-axis machining. Rotary axes can be programmed directly with angle or indirectly with a tool vector. Radius correction occurs if the normal vector is programmed in the endpoint. A typical application is the machining of free-form surfaces.




```
G    3D tool correction
R    Nominal tool radius
L2=  Rotary axes (0=shortest, 1=abs.)
R1=  Nominal tool corner radius
F2=  Feed limitation
```

Address description

With G141

- **R Nominal tool radius**
- **R1= Nominal tool corner radius**
- **L2= Rotary axes (0=shortest, 1=absolute)**
- **F2 Feed limitation**

With G0/G1

- **X, Y, Z Linear endpoint co-ordinates**
- **I, J, K Axis components of the normalised space vector**
- **I1=, J1=, K1= (TCPM) Axis components of the tool vector**
- **A, B, C (TCPM) Rotary axes co-ordinates of the tool vector**
- **F Feed on the path**

Address description

- **R Nominal tool radius** defines the tool radius used to calculate the endpoints of the G0/G1 blocks in the CAD system.
- **R1= Nominal tool corner radius** defines the tool corner radius used to calculate the endpoints of the G0/G1 blocks in the CAD system.
- **L2= Rotary axes**
  - L2=0 Rotary axes traverse the shortest route (default).
  - L2=1 Rotary axes approach their absolute position (with rotary axis programming).
- **F2= Feed limitation** highly-curved surfaces can cause the rotary axes to move abruptly at maximum feed. F2= limits this feed. F2= is programmed in the G141 block and acts for all G0/G1 movements up to the block with G40.

Format

3-axis machining with normalised vector (I,J,K) for radius correction:

- G141 {R...} {R1=...} {L2=...} {F2=...}
- G0/G1 [X..Y.. Z..] {I... J... K...}

5-axis machining with TCPM (Tool Center Point Management).
Normalised vector (I,J,K) for radius correction.

- G141 R.. {R1=..} {L2=..} {F2=..}
- G0/G1 [X..Y.. Z..] [I.. J.. K...] {I1=.. J1=.. K1=..}/{A.. B.. C..}

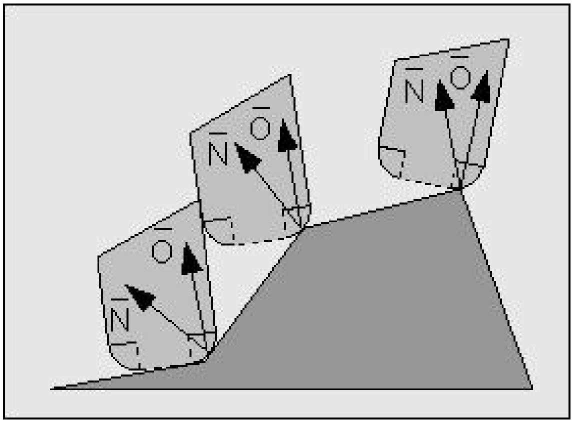
Cancel 3D tool correction:

- G40

Default

G141 L1=0 R1=0 R=0

Application



5-axis machining of a curved workpiece surface involves guiding the tool to the surface at an optimised angle. Dynamic TCPM is used for this 5-axis machining and guides the rotary axes and linear axes, allowing for current tool length and tool radius. In the G0/G1 block, the rotary axes can be programmed directly with angle (A,B,C) or indirectly with a tool vector (I1=, J1=, K1=). The radius correction is calculated by MillPlus if the normalised vector (I, J, K) is programmed in the G0/G1 block.

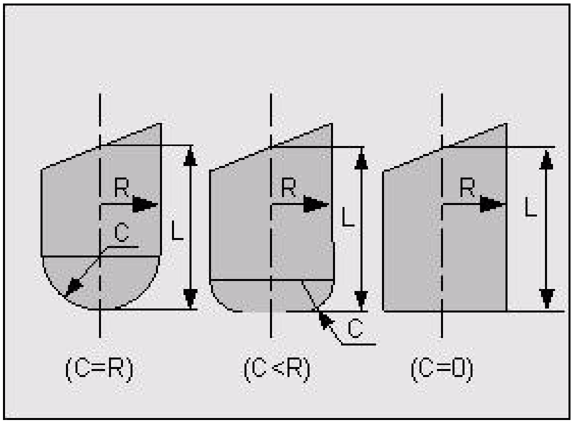
- N = Normalised vector (I, J, K) (see picture)
- O = Tool vector (I1=, J1=, K1=)

G7 may be active. In this case, normalised and tool vectors are defined in level G7.

Potential tools

The following dimensions must be loaded in the tool memory for use of the various tool types (see picture):

Radius milling tool:	R (tool radius) L (tool length) C (rounding radius) C=R
End milling tool	R (tool radius) L (tool length) C (rounding radius) C<R
End milling tool:	R (tool radius) L (tool length) C=0



If no C-value is entered, C automatically becomes 0.

> The rounding radius in the G141 block is programmed with the word R1=. The C-word is for storing the rounding radius in the tool memory.

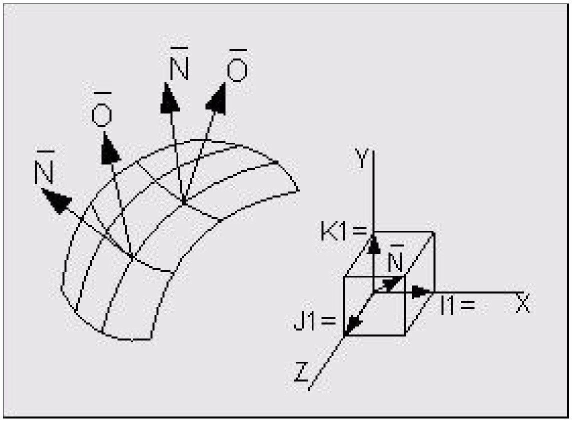
Radius correction

The radius correction is calculated by MillPlus if the normalised vector (I, J, K) is programmed in the G0/G1 block. If no normalised vector is programmed, radius correction is not activated.

The tool is positioned so that this vector always passes through the centre point of the corner rounding. If the endpoints are calculated in the CAD/CAM system with nominal radius and corner radius, this can be defined in the G141 block using R and R1=. The actual radius R and corner radius C will then be entered into the tool table. The controller corrects the difference between the nominal and actual radius.

R Radius defines the tool radius used to calculate the endpoints of the G0/G1 blocks in the CAD system.

R1= Radius defines the tool corner radius used to calculate the endpoints of the G0/G1 blocks in the CAD system.



Normalised vector (I, J, K)

The normalised vector is perpendicular to the workpiece surface. I,J,K are the vector components in directions X,Y,Z. The tool is positioned so that this vector always passes through the centre point of the tool corner rounding. See picture.

Tool vector (I1=, J1=, K1=) (TCPM)

This vector points towards the tool axis. I1=,J1=,K1= are the vector components in directions X, Y, Z. The tool vector instead of the rotary axes can be programmed in the G0/G1 block. During the movement, rotary axes A, B, C and linear axes are interpolated so that a straight line is generated in the machining area. At the endpoint of the movement, the tool points towards this vector.

Vector components

A vector is programmed with at least one component in the G0/G1 block. Unprogrammed components are equal to zero.

Vector accuracy

The input format for the vectors (I, J, K, I1=, J1=, K1= words) is limited to three decimal places. The surface normal and tool vectors do not, however, need to have length 1. To increase the dimensional accuracy, the values in question can be multiplied by a dimension factor of between 1 and 1000. The factor 1000, for example, increases the input accuracy of the vector components to six significant figures.

Activate G141

In the first block after G141, the milling tool traverses from the current tool position to the corrected position in this block. G141 deletes a radius correction programmed using G41..G44.

Cancel G141

The function G141 is cancelled using the G40, M30, cancel program softkey or the CNC reset softkey. The milling tool stops at the last corrected position. The rotary axes are not turned back automatically.

Switch on condition before G141

Before switching on G141, the following functions must be switched-off: Geometry G64, Scale change G73 A4=, Axis rotation B4= with G54-G59, G54 I.. , G92/G93, cylinder co-ordinates G182

The following functions are permitted if G141 is **switched on**:

Basic motions	0, 1
Free machining level	7
Levels	17, 18
Program control	14, 22, 23, 29
Positioning feed	25, 26, 27, 28, 94, 95, 96, 97
Tool allowance	39
Radius correction	40
Zero points	51, 52, 53, 54-59, 54 I.., 92, 93
Geometry	72, 73
Absolute/incremental	90, 91
Graphics	195, 196, 197, 198, 199

The following G functions are permitted if G141 is **active**:

Basic motions	0, 1
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	40
Zero points	51, 52, 53, 54-59, 54 I.., 92, 93
Geometry	72, 73
Absolute/incremental	90, 91

An error message is output if a G function that is not permitted is programmed.

Programming limitations

G functions not listed above may not be used. Point definitions (P) may not be used. No tool change may be made after activating G141.

Endpoint co-ordinates

Absolute or incremental (X, X90, X91) Cartesian dimensions may be used.

G1

When a tool vector I1=,J1=,K1= is being programmed, G0 or G1 must be in the same block.

Mirroring

If the mirroring function (G73 and axis co-ordinates) is effective before G141 is activated, the mirrored co-ordinates will be used during the 3D tool correction operation. Mirroring is re-enabled once G141 is activated. Mirroring is cancelled using Function G73.

Back cutting

Back cutting or collisions between tool and material at points not to be machined are not detected by the CNC.

Modulo function

A rotary axis that can turn continuously should be defined as a modulo axis for use with G141 (MC713=1). The display of the actual position is then limited to 0° to 360°. In addition, L2=0 (rotary axis traverses shortest route) should be programmed for G141.

The modulo function is effective for all rotary axes where the distance between the limit switches exceeds 720°.

The modulo function is deactivated after: G141 L2=1, G40, M30, and the softkeys CANCEL PROGRAM or CNC RESET.

If the modulo function is not switched on, an unwanted direction change on this axis may occur at the continuously rotating rotary axis limit switch.

Behaviour of the rotary axes at the limit switches

If the rotary axes on G141 is programmed directly with A.. B.. C.. an error message will be output if the programmed position lies past the limit switch.

Selecting a solution with vector programming

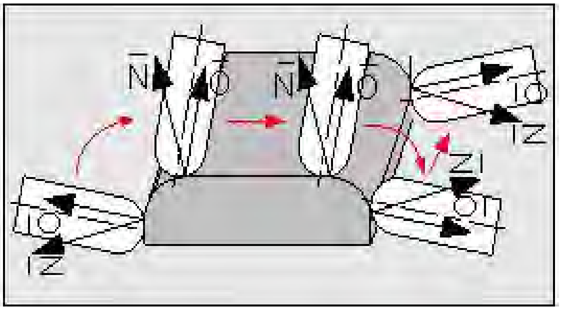
If the rotary axes are programmed via the tool vector I1=, J1=, K1=, there are often two solutions for the rotary axis positions. Selecting a solution:

- The solution that lies past the limit switch is invalid.
- The solution that comes past the limit switch of a linear axis during interpolation is invalid.
- If two solutions are valid, the solution with the shortest route is selected, even when L2=1 (rotary axis absolute).
- If both solutions are invalid, an error message is displayed, indicating that the programmed level cannot be reached.

Endpoint co-ordinates

With endpoint coordinates, only the programmed axes are moved.

Example



Example1: G141 and TCPM with tool vector I1=, J1=, K1=

This programming is **independent** of the machine.

```
N113
G17
T6 M67 (T6 R5 C5)
G54 I10
G0 X0 Y0 Z0 B0 C0 S6000 M3
F50 E1=0
G141 R0 R1=0 L2=0
G0 X-1 Y=E1 Z0 I1=-1 K1=0
G1 X0 Y=E1 Z-4 I1=-996.195 K1=087.156
G1 X0.001 Z-3.930 I1=-994.522 K1=104.528
G1 X0.002 Z-3.860 I1=-992.546 K1=121.869
G1 X0.005 Z=-3.791 I1=-990.268 K1=139.173
G1 X3.791 Z-0.005 I1=34.899 K1=999.391
G1 X3.860 Z-0.002 I1=52.336 K1=998.626
G1 X3.930 Z-0.001 I1=69.756 K1=997.564
G1 X4 Z0 I1=87.156 K1=996.195
G1 X36 Z0 I1=87.156 K1=996.195
G1 X36.070 Z-0.001 I1=104.528 K1=994.522
G1 X36.140 Z-0.002 I1=121.869 K1=992.546
```

Example 2: G141 and TCPM with rotary axes A, B, C

Identical workpiece.

This programming is **dependent** on the machine.

This program is for a machine with a B-axis less than 45° on the table and a C-axis above it.

```
N114
G17
T6 M67 (T6 R5 C5)
G54 I10
G0 X0 Y0 Z0 B0 C0 S6000 M3
F50 E1=0
G141 R0 R1=0 L2=0
G0 X-1 Y=E1 Z0 B180 C-90
G1 X0 Y=E1 Z-4 B145.658 C-113.605
G1 X0.001 Z-3.930 B142.274 C-115.789
```

```
G1 X0.002 Z-3.861 B139.136 C-117.782
G1 X0.005 Z-3.791 B136.191 C-119.624
G1 X3.791 Z-0.005 B2.829 C1
G1 X3.860 Z-0.002 B4.243 C1.501
G1 X3.930 Z-0.001 B5.658 C2.001
G1 X4 Z0 B7.073 C2.502
G1 X36 Z0 B7.073 C2.502
G1 X36.070 Z-0.001 B8.489 C3.004
G1 X36.140 Z-0.002 B9.906 C3.507
```

3.9 G151 Cancel G152

Cancel G152.

Format

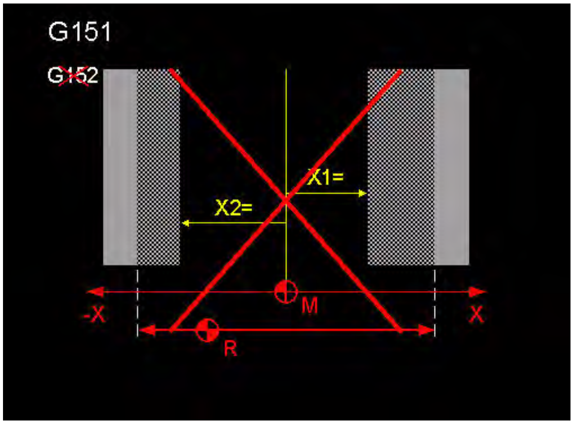
G151.

Address description

No addresses.

Application

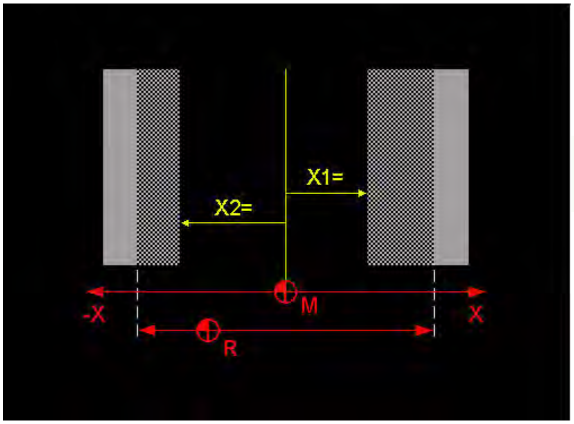
This function can be used to deactivate G152.



Associated functions
G152.

3.10 G152 Limitation traverse range

Limiting the traverse ranges. The programmed positions relate to the reference point.



G	Limitation traverse range
X1=	Range in positive direction
Y1=	Range in positive direction
Z1=	Range in positive direction
B1=	Range in positive direction
C1=	Range in positive direction
X2=	Range in negative direction
Y2=	Range in negative direction
Z2=	Range in negative direction
B2=	Range in negative direction
C2=	Range in negative direction

Format

G152 X1=... Y1=... Z1=... {B1=...} {B2=...} X2=... Y2=... Z2=... {C1=...} {C2=...}

Address description

See picture.

Application

This function enables the traverse range to be limited in the NC program. With G141, for example, it is possible to prevent the C-axis (table) from rotating further around a vector solution than is permitted. It is also possible to program a limit level.

The programmed positions must lie within the range of SW limit switches MC3n18, MC3n19, otherwise an error message is output.

Associated functions

G151

Deactivation

G152 is deactivated by:

- G151
- Program end M30
- Cancel program
- Reset CNC
- Switch on controller

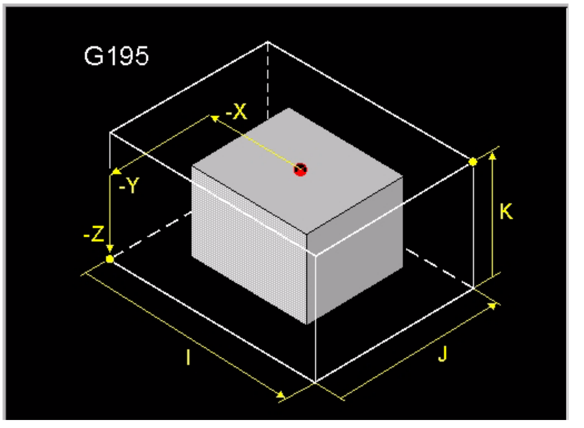
Example

The traverse range of the C-axis is limited by:

```
G152 C1=30.000 C2=-30.000
```

G152	The C-axis is permitted only within the range of +30 to -30 degrees, otherwise an error message is output.

3.11 G195 Graphic window definition with begin and end block



G	Graphic window definition
X	Start point coordinate
Y	Start point coordinate
Z	Start point coordinate
B	Rotation around hor. axis (3D)
I	Dimension parallel to X
J	Dimension parallel to Y
K	Dimension parallel to Z
N1=	Graphic begin block
N2=	Graphic end block
B1=	Rotation around vert. axis (3D)
B2=	Rotation around third axis (3D)

Defines the dimensions of a graphic window and its position relative to zero point W. The optional addresses N1= and N2= can be used to define a program part that is displayed in the graphic simulation.

Format

G195 X.. Y.. Z.. I.. J.. K.. {B..} {B1=..} {B2=..} {N1=..} {N2=..}

Address description

See picture.

Application

In programs with several level definitions, only the machined levels in the last programmed machining level are displayed.

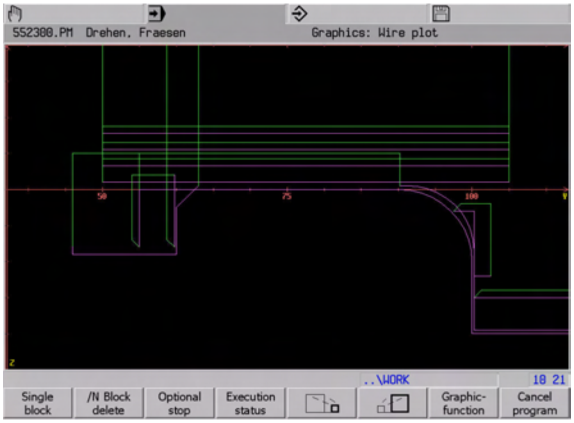
Addresses N1= "Graphic begin block" and N2= "Graphic end block" are used to record the graphic window for a particular program part. All movements in the blocks from address N1= up to and excluding the block number in Address N2= are displayed in the graphics window.

For programs with both turning and milling machining, for example, this allows any program part with turning or milling machining to be displayed.

Example

Define graphic window for the turning program part

```
N1   G195 X0 Y45 Z-25 I45 J60 K45 N1=17 N2=128
N8   G36
N10  G17 Y1=1 Z1=2
N17  (start of turning)
N..
N128 (end of turning)
N135 G37
N138 G17
N150 (start of milling)
N..
N178 (end of milling)
```



G195	Define graphic window and "turning" program part for simulation
G36	Activate turning mode
G17 Y1=1 Z1=2	Activate machining level for turning mode
G37	Activate milling mode
G17	Activate machining level for milling mode

3.12 G303 M19 with programmable direction

This function is deactivated in the V520.

3.13 G321 Read tool data

Read values from the tool table

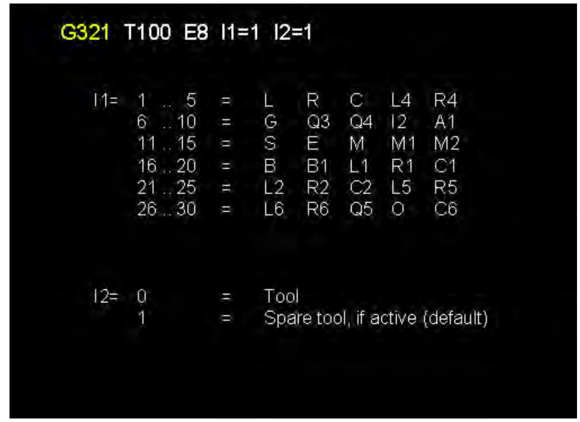
Address description

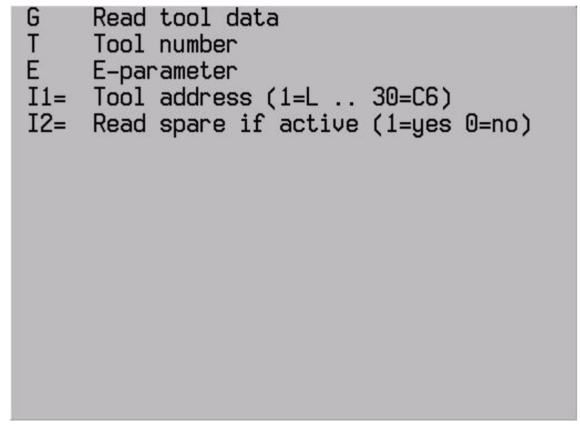
I1=	**Selectable functions**
I1=13	M Tool life (units of time are minutes)
I1=30	C6 Cutting width (only with turning option)

I2=	**Spare tools**
I2=1	Data from tool is queried (default).
I2=0	Data from spare tools is queried.

Read spare tools

With I2=1, the data from the spare tool (e.g. T1000.01) is queried.





3.14 G325 Read modal M function

The address I1 = is expanded to 15.

I1=14 off. M78, M79.
I1=15 off. M130, M131.

3.15 G330 Read point definition data

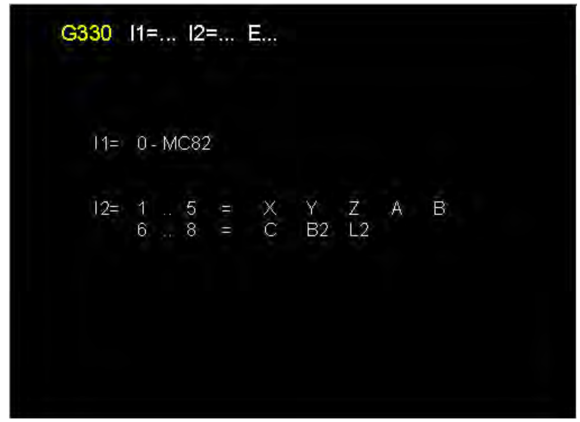
Read values from the point table

Address description

I1= **Point definition number (0-MC82)**

I2= **Address in point table**

1	X
2	Y
3	Z
4	A
5	B
6	C
7	B2
8	L2




```
G    Read point definition data
E    E-parameter
I1=  Point definition number
I2=  1-8 (X,Y,Z,A,B,C,B2,L2)
```

3.16 G331 Write tool data

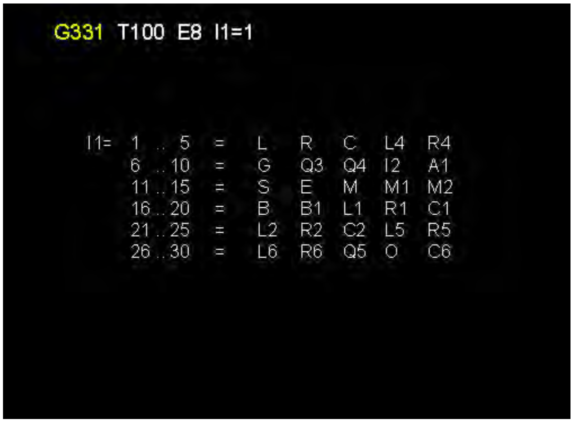
Write values into tool table

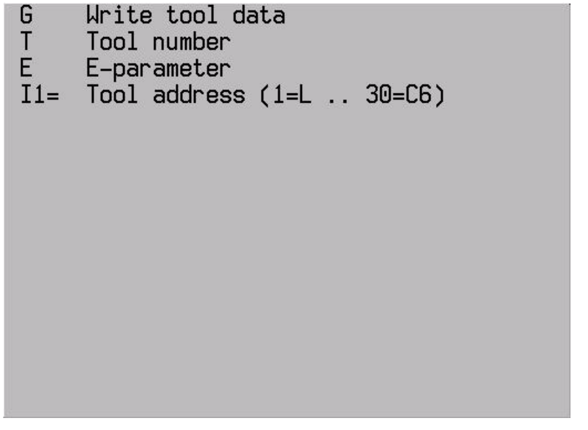
Address description

I1=	**Selectable functions**
I1=13	M Tool life (units of time are minutes)
I1=30	C6 Cutting width (only with turning option)

Tool life

If M (G331 I1=13 E...) is written into the tool memory, M1= is also written into the tool memory simultaneously (G331 I1=14 E...). The time units are minutes.





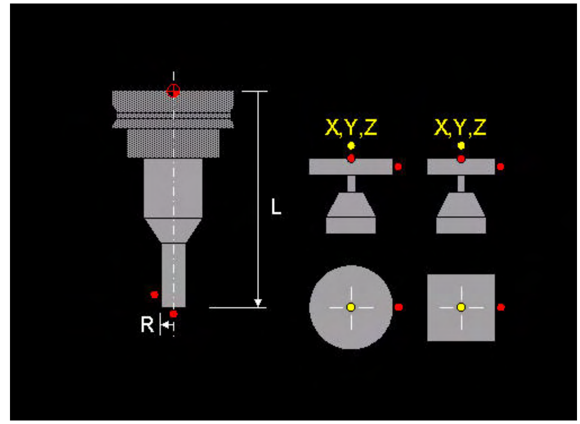
3.17 G350 Write into the window

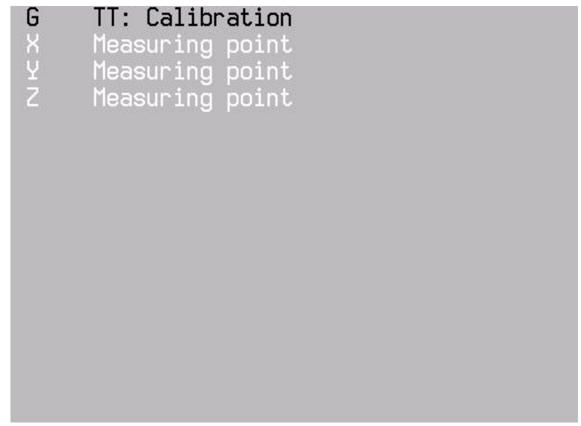
Format:
G350 N1=... I1=... {I2=…}
I1= must be programmed
I2= optional

Default:
I2=0

3.18 G606 TT: Calibration

Determines the position of the measuring system and saves these position values to the machine constants provided.





Address description

- **`X,Y,Z measuring point`**

Application

Measuring system

G606 can be used to calibrate a table probe (TT) or a laser and table probe (TT) combined.

Calibration tool

Before starting the calibration process, you must enter the exact radius and exact length of the calibration tool into the tool table.

Sequence

The calibration process runs automatically. MillPlus ***IT*** also automatically determines the centre offset of the calibration tool. MillPlus ***IT*** does this by rotating the spindle by 180° after half the calibration cycle. Use a precisely cylindrical part, e.g. a cylindrical pin, as a calibration tool. MillPlus ***IT*** saves the calibration values to the machine constants and takes them into account in subsequent tool measurements.

The position of the TT in the machine work area is defined in MC350, MC352, MC354. If you alter one of the MC350, MC352 or MC354, you must re-calibrate.

The position of the probe in the machine work area with a laser and TT combined is defined in MC400-406. If you alter one of the MC400-406, you must re-calibrate.

Position

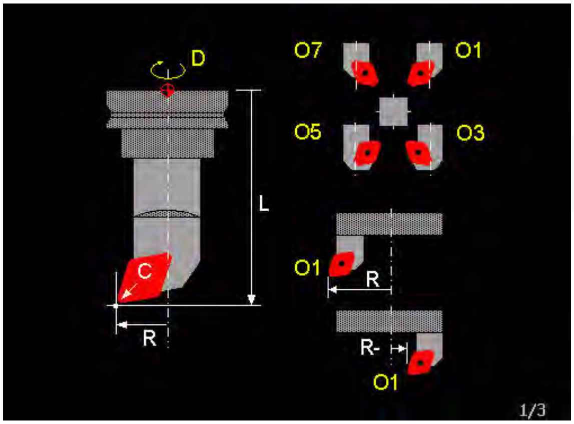
Enter position into X, Y and Z if the measuring system has not been calibrated and hence the positions in the machine constants have not yet been precisely determined.

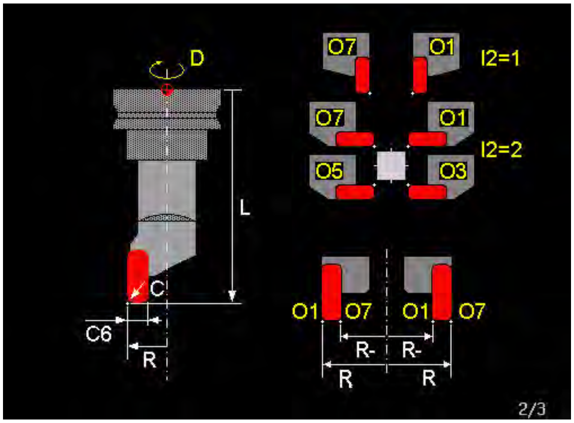
Head position

G606 can be used only when the head is in a vertical position.

3.19 G611 TT: Measuring turning tool

This cycle measures the length, radius, and cutting width of standard turning and plunging tools, as well as turning tool plates that are mounted in a U-head. The turning tool is measured when stationary in the G17 level. Inner and outer tools can be measured.






```
G    TT: Turning tool measurement
D    Orientation angle tool tip
O    Tool orientation
I1=  Clearance
I2=  Measuring width 0=no 1=ax 2=rad
I4=  Measuring: 0=L+R 1=L 2=R
```

Address description

- **D Orientation angle** The tool is oriented in the programmed position (D) at the safety position. The tool tip must be parallel to the axis and perpendicular to the probe.
- **O Tool orientation** The tool orientation (O) of the tool tip determines whether measurements are taken:
  - Before or after the probe
  - Below or above on the tool cutter (plunging tools)
- **I1= Clearance** The clearance towards the spindle axis must be sufficient to prevent a collision with workpiece or clamping fixture. The clearance relates to the top edge of the stylus.
- **I2= Measuring cutter width** The tool cutter width is calculated from two measurements: inner and outer measurement. The machining direction of the bit plunging surface (axial or radial) must be entered.
  - **I2=0** No
  - **I2=1** Axial tool measurement
  - **I2=2** Radial tool measurement
- **I4= Measuring: 0=L+R 1=L 2=R**

> If no tool orientation is entered in the tool table, the programmed tool orientation (O) will be saved. If a tool orientation is entered in the tool table and does not correspond with the programmed one, the cycle is stopped and an error message is output.

Default

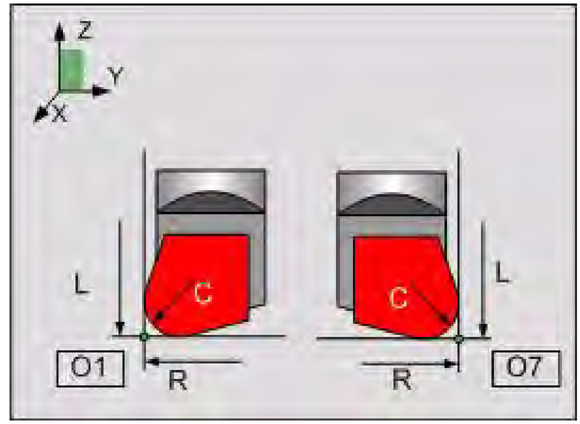
I1=30, I2=0, I4=0

Application

Addresses used by the tool memory:

- **L Tool length**
- **R Tool radius**
- **C Corner radius**
- **C6= Cutter width**
- **L4= Length allowance**
- **R4= Radius allowance**

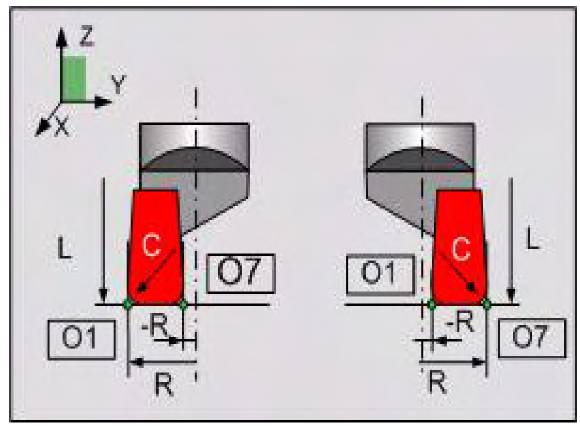
- **L5= Length tolerance**
- **R5= Radius tolerance**
- **L6= Length measurement offset**
- **R6= Radius measurement offset**
- **E Tool status**
- **O Tool orientation**



Ensure that the length (L) and radius (R) are entered within the tolerance (MC397), otherwise an error message will be output.

Tool types

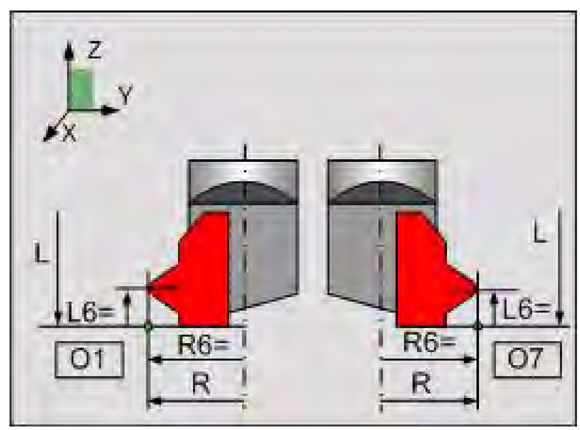
Standard turning tools (fixed in the main spindle) and rotary turning tools (U-head) can be used. Both types of turning tools are measured when stationary and fixed. Turning and plunging tools with a recessed main cutter andsecondary cutter (orientation 1 or 7) can be measured (see pictures).



Measurement of length, radius and width

Tool length (L), tool radius (R) and cutting width (C6=) must be saved to the tool memory. Before the initial measurement, the rough length and radius must be entered (max. deviation +/-MC397).

Incorrect entries can result in error messages or even collision with the probe device.



Corner radius

It is recommended that you always enter a corner radius (C) into the tool memory.

Measure or test tool

- Measure tool (E=0 or no value). During the initial measurement, the tool length (L) and radius (R) are overwritten. The allowance is set to L4=0/R4=0 and the tool status is set to E=1. If a corner radius C is entered, this is also corrected.
- Check tool (E=1):
  The measured deviation is added to L4=/R4= in the tool table.

Cycle sequence

The MillPlus IT measures the tool based on a fixed, programmed sequence:

1. When the cycle starts, the axes move to the safety position by rapid traverse with positioning logic.
2. The tool is oriented and clamped in the programmed position (D) at the safety position.
3. The tool moves to the measuring position with measuring feed.
4. The measurement is executed.
5. After the measurement, the Z axis returns to the safety position.

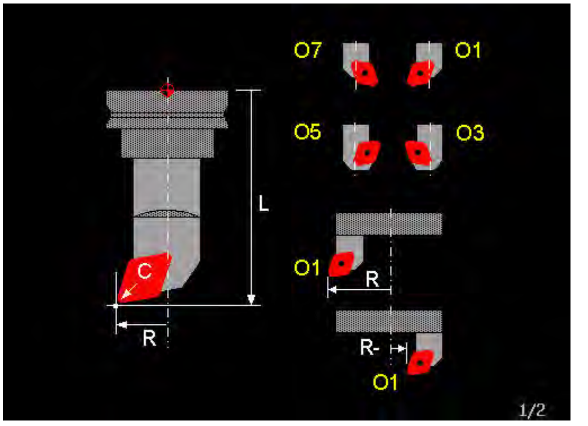
Note

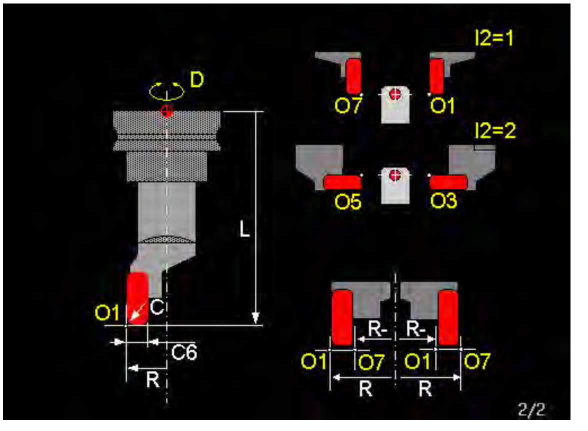
- The cycle can be called up in milling and turning mode.

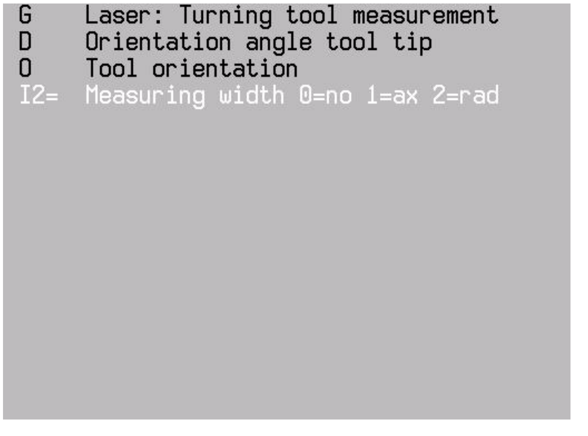
- The tool can be measured before or after the probe. The greatest precision is achieved if the tool is measured in the machining position.
- When U-head tools are measured, the stroke adjustment in the U-axis must be in neutral.
- Measuring the axial cutting width (I2=1) with orientation O3 or O5 is not permitted.
- The probe should be fitted so that it can be scanned from the two radial sides and from the bottom.

3.20 G615 Measuring laser measurements for turning tool

This cycle measures the length, radius, and plunging width of standard turning and plunging tools as well as turning tool plates that are mounted in a U-head. The turning tool is measured when stationary in the G17 and G18 levels. Inner and outer tools can be measured.







Address description

- **D Orientation angle** The tool is oriented in the programmed position (D) at the safety position. The tool tip must be parallel to the axis and perpendicular to the laser.
- **O Tool orientation** The tool orientation (O) of the tool tip determines whether measurements are taken:
  - Before or after the probe
  - Below or above on the tool cutter (plunging tools)
- **I2= Measuring cutter width** The tool cutter width is calculated from two measurements: inner and outer measurement. The machining direction of the bit grooving surface (axial or radial) must be entered.
  - **I2=0** No
  - **I2=1** Axial tool measurement
  - **I2=2** Radial tool measurement

If no tool orientation is entered in the tool table, the programmed tool orientation (O) will be saved. If a tool orientation is entered in the tool table and it does not correspond with the programmed one, the cycle is stopped and an error message is output.

Default

I2=0

Application

Addresses used by the tool memory:

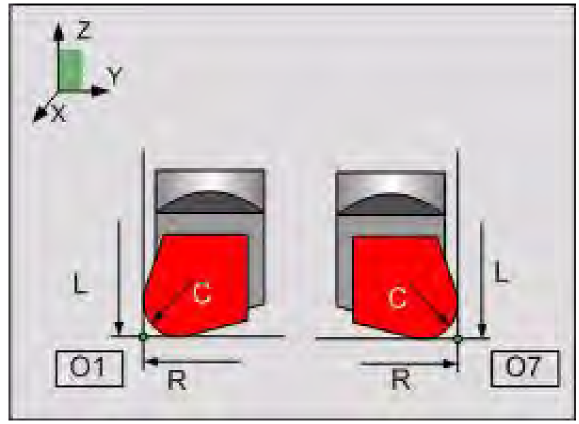
- **L Tool length**
- **R Tool radius**
- **C Corner radius**
- **C6= Cutter width**
- **L4= Length allowance**
- **R4= Radius allowance**
- **L5= Length tolerance**
- **R5= Radius tolerance**
- **L6= Length measurement offset**

- **R6= Radius measurement offset**
- **E Tool status**
- **O Tool orientation**

> Ensure that the length (L) and radius (R) are entered within the tolerance (MC397), otherwise an error message will be output.

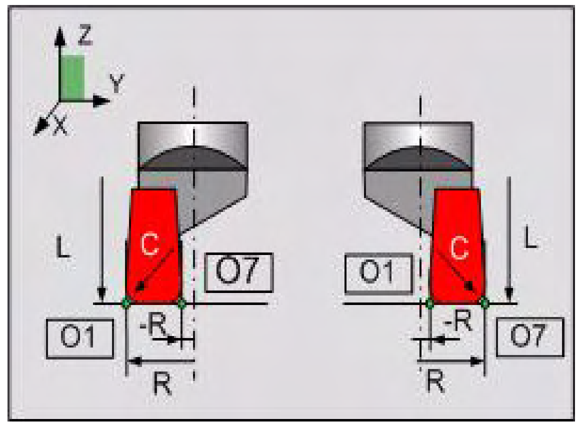
Tool types

Standard turning tools (fixed in the main spindle) and rotary turning tools (U-head) can be used. Both types of turning tools are measured when stationary and fixed. Turning and plunging tools with a recessed main cutter andsecondary cutter (orientation 1 or 7) can be measured (see pictures).



Length, radius and width measurement

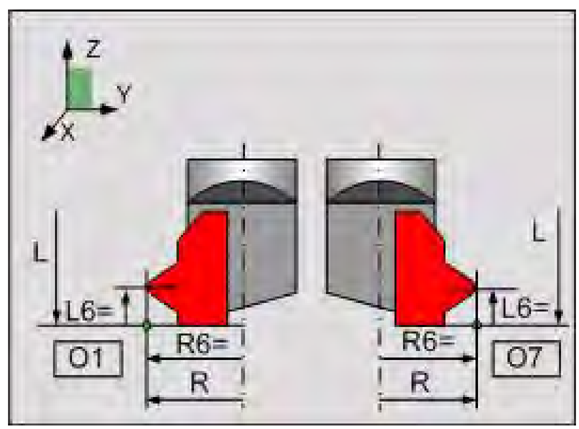
Tool length (L), tool radius (R), and tool width (C6=) must be saved to the tool memory. Before the initial measurement, the rough length and radius (max. deviation +/-5 mm) and the bit width (max. deviation +/- 50 %) must be entered.



> Incorrect entries can result in error messages or even collision with the laser device.

Corner radius

It is recommended that you always enter a corner radius (C) into the tool memory. This speeds up the cycle.



Measure or test tool

- Measure tool (E=0 or no value). During the initial measurement, the tool length (L) and radius (R) are overwritten. The allowance is set to L4=0/R4=0 and the tool status is set to E=1. If a corner radius C is entered, this is also corrected.
- Check tool (E=1):
  The measured deviation is added to L4=/R4= in the tool table.

Cycle sequence

The MillPlus IT measures the tool based on a fixed, programmed sequence:

1. When the cycle starts, the axes move to the safety position by rapid traverse with positioning logic.
2. The tool is oriented and clamped in the programmed position (D) at the safety position.
3. The tool moves to the measuring position with measuring feed.
4. The measurement is executed.
5. After the measurement, the Z axis returns to the safety position.

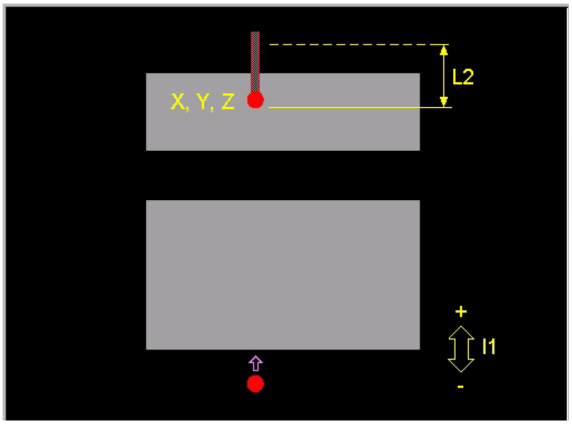
Note

- The cycle can be called up in milling and turning mode.

- The tool can be measured in front of or behind the laser. The greatest precision is achieved if the tool is measured in the machining position.
- After the cycle sequence, the spindle remains in the programmed position (D), and the orientation (O) from before the measurement becomes active.
- When U-head tools are measured, the stroke adjustment in the U axis must be in neutral.
- The cycle can only be used when the head is in a vertical position.

3.21 G621 Position measurement

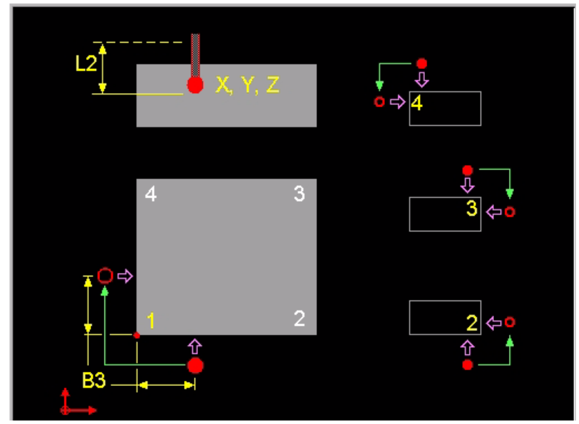
G621 has been expanded with Address I2= for probe orientation. For further information, see introduction to measuring cycles.



G	Position measurement
I1=	Meas.dir. ±1/±2/-3=main/minor/Tl
X	Starting point
Y	Starting point
Z	Starting point
C1=	Measuring distance
L2=	Safety distance
I2=	Orient. -1=auto 0=no 1=180 2=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. for measured position
F2=	Measuring feed
B1=	Target position

3.22 G622 Corner outside measurement

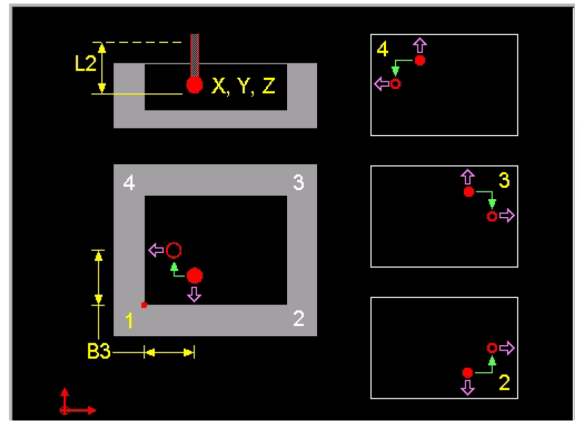
G622 has been expanded with Address I2= for probe orientation. For further information, see introduction to measuring cycles.



G	Corner outside measurement
I4=	Corner number
X	Starting point
Y	Starting point
Z	Starting point
B3=	Distance to corner
C1=	Measuring distance
L2=	Safety distance
I2=	Orient. -1=auto 0=no 1=180 2=yes
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. position main axis
O2=	E-Par. meas. position minor axis
F2=	Measuring feed
X1=	Target position corner
Y1=	Target position corner
Z1=	Target position corner

3.23 G623 Corner inside measurement

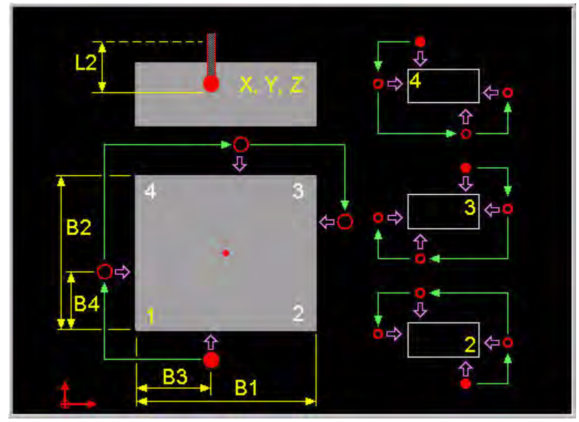
G623 has been expanded with Address I2= for probe orientation. For further information, see introduction to measuring cycles.



G	Corner inside measurement
I4=	Corner number
X	Starting point
Y	Starting point
Z	Starting point
B3=	Distance to corner
C1=	Measuring distance
L2=	Safety distance
I2=	Orient. -1=auto 0=no 1=180 2=yes
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. position main axis
O2=	E-Par. meas. position minor axis
F2=	Measuring feed
X1=	Target position corner
Y1=	Target position corner
Z1=	Target position corner

3.24 G626 Datum outside rectangle

Measure the mid-point of a rectangle parallel to the axis



Address description

- **I5= Save measured values in a zero point allowance** I5=0 do not save, I5=1 save in the active zero point allowance in the linear axes (X/Y/Z). During a save, the measured values are added to the active zero point allowance.
- **X1=, Y1=, Z1= Target centre point** Once the measured coordinate is saved in the active zero point allowance (I5>0), this corrects the target value. The measured coordinate then receives the target value for further programming.
- **B3= Distance to the corner in the main axis**
- **B4= Distance to the corner in the secondary axis** If B4= is not entered, then B4=B3

The description of additional addresses is in the introduction to the measurement cycles

G	Datum outside rectangle
I4=	Corner number
X	Starting point
Y	Starting point
Z	Starting point
B1=	1st Side length
B2=	2nd Side length
B3=	Distance to corner in main axis
B4=	Distance to corner in minor axis
C1=	Measuring distance
L2=	Safety distance
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. centre main axis
O2=	E-Par. meas. centre minor axis

O4=	E-Par. meas. length main axis
O5=	E-Par. meas. length minor axis
F2=	Measuring feed
X1=	Target centre point
Y1=	Target centre point
Z1=	Target centre point

Default

I4=1, B3=10, B4=B3, C1=10, L2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Application

Measurement

Two opposing tool noses are measured (1+3 or 2+4).

Direction of approach of the first corner measurement

- The first measurement is always perpendicular to the main axis.
- The second measurement is always perpendicular to the secondary axis.

Direction of approach of the second corner measurement

- Clockwise from corner number 1 --> 3 or 3 --> 1.
- Counter-clockwise from corner number 2 --> 4 or 4 --> 2.

The support picture is in G17. The picture is not correct for a machine with replaced axes (G18). Angle 1 must be replaced with 2, and 3 with 4.

Sequence

1. Rapid movement to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
2. First measurement with measurement feed (F2=), until the tool or the maximum measured distance (C1=) is achieved.

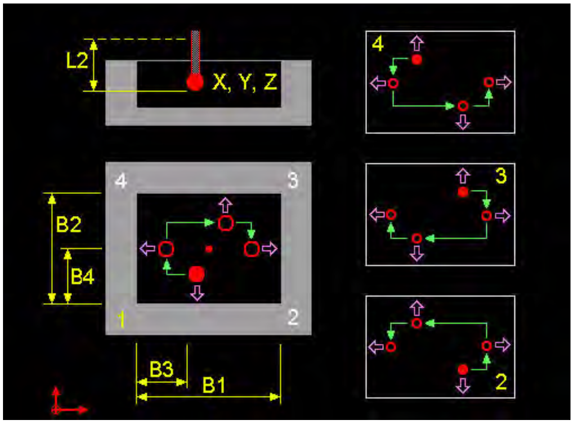
3 Rapid movement back to first starting point. An error message is issued if the measurement probe has not switched within the maximum measured distance (C1=).
4 Rapid movement, dependent on I3=above the safety distance (L2=), to the starting point of the 2nd measurement.
5 Second measurement (the same as described in items 2 and 3).
6 The opposing corner is measured using a 3rd and 4th measurement (the same as described in items 2 and 3).
7 At the end, a rapid movement is executed to the safety distance (L2=).
8 Dependent on I5=, the measured value is saved.

Example: Save centrepoint of a rectangle in the zero point allowance

```
G54 I3
G626 X-45 Y-3 Z-5 B1=100 B2=20 B3=5 I3=1 I5=1
```

G54	Set zero point.
G626	Define and execute measured cycle (B4=B3). After the measured cycle, X and Y are adapted in G54 I3.

3.25 G627 Datum inside rectangle



G	Datum inside rectangle
I4=	Corner number
X	Starting point
Y	Starting point
Z	Starting point
B1=	1st Side length
B2=	2nd Side length
B3=	Distance to corner in main axis
B4=	Distance to corner in minor axis
C1=	Measuring distance
L2=	Safety distance
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z
O1=	E-Par. meas. centre main axis
O2=	E-Par. meas. centre minor axis

O4=	E-Par. meas. length main axis
O5=	E-Par. meas. length minor axis
F2=	Measuring feed
X1=	Target centre point
Y1=	Target centre point
Z1=	Target centre point

Measure the centrepoint of a rectangular hole parallel to the axis

Address description

- **I5= Save measured value in a zero point allowance** I5=0 Do not save I5=1 Save in the active zero point allowance in the linear axes (X/Y/Z). When saving, the measured values are added to the active zero point allowance.
- **X1=, Y1=, Z1= Target centre point** Once the measured coordinate is saved in the active zero point allowance (I5>0), this corrects the target value. The measured coordinate then receives the target value for further programming.
- **B3= Distance to the corner in the main axis**
- **B4= Distance to the corner in the secondary axis** If B4= is not entered, then B4=B3

The description of additional addresses is in the introduction to the measurement cycles

Default

I4=1, B3=10, B4=B3, C1=10, L2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Application

Measurement

Two opposing tool noses are measured (1+3 or 2+4).

Direction of approach of first corner measurement

- The first measurement is always perpendicular to the main axis.
- The second measurement is always perpendicular to the secondary axis.

Direction of approach of second corner measurement

- Clockwise from corner number 1 --> 3 or 3 --> 1.
- Counter-clockwise from corner number 2 --> 4 or 4 --> 2.

> The support picture is in G17. The picture is not correct for a machine with replaced axes (G18). Angle 1 must be replaced with 2, and 3 with 4.

Sequence

1. Rapid movement to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
2. First measurement with measurement feed (F2=) until the tool or the maximum measured distance (C1=) is reached.

3 Rapid movement back to first starting point. An error message is issued if the measurement probe has not switched within the maximum measured distance (C1=).
4 Rapid movement, dependent on I3=above the safety distance (L2=), to the starting point of the 2nd measurement.
5 Second measurement (the same as described in items 2 and 3).
6 The opposing corner is measured using a 3rd and 4th measurement (the same as described in items 2 and 3).
7 At the end, a rapid movement is executed to the safety distance (L2=).
8 Dependent on I5=, the measured value is saved.

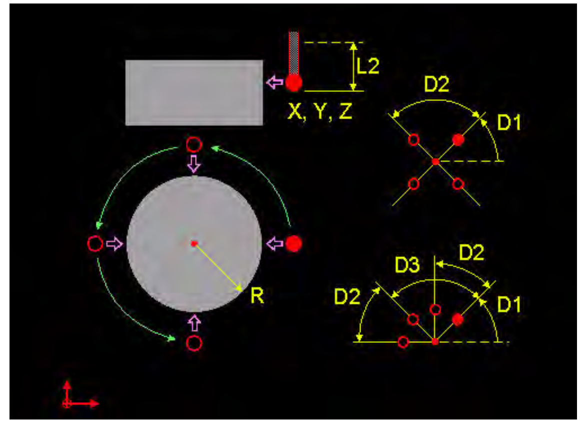
Example: Save centrepoint of a rectangle in the zero point allowance

```
G54 I3
G627 X-45 Y-3 Z-5 B1=100 B2=20 B3=5 I3=1 I5=1
```

G54	Set zero point.
G627	Define and execute measured cycle (B4=B3). After the measured cycle, X and Y are adapted in G54 I3.

3.26 G628 Circle measurement outside

Measure centrepoint of a circle.



G	Circle measurement outside
R	Circle radius
R1=	Minimum circle radius
R2=	Maximum circle radius
X	Starting point
Y	Starting point
Z	Starting point
D1=	Starting angle
D2=	Second angle
D3=	Third angle
C1=	Measuring distance
L2=	Safety distance
I2=	Orient. -1=auto 0=no 1=180 2=yes
I3=	2nd measurem. via L2 0=no 1=yes
I5=	G5x offset 0=no 1=X/Y/Z

O1=	E-Par. meas. centre main axis
O2=	E-Par. meas. centre minor axis
O6=	E-Par. measured diameter
O7=	E-Par. radius difference
F2=	Measuring feed
X1=	Target centre point
Y1=	Target centre point
Z1=	Target centre point

Address description

- **R1= Minimum radius** The measured radius must be equal to or greater than R1, otherwise an error message will be output.
- **R2= Maximum radius** The measured radius must be equal to or smaller than R2, otherwise an error message will be output.
- **D1= Starting angle** Angle allowance of the circle measurement, based on the main axis.
- **D2= 2nd angle** Angle between first and second measurement and between the third and fourth measurement. The smallest entry value is 5°.
- **D3= 3rd angle** Angle between the first and third measurement. D3 must be at least 5° larger than D2. If D3 and D3 are the same, a 3-point measurement is executed.
- **I2= Probe orientation in measurement direction** The orientation option of the probe is established in MC846.
  - **I2=0** Measure without rotation.
  - **I2=1** Measure using 2 measurements with 180° rotation. First measurement with standard orientation (MC849). Second measurement with 180° turning. The measured value is the average value of these two measurements.
  - **I2=2** Measure with orientation in measurement direction. Only possible with infra-red probe with all-around emitter.
- **I5= Save measured values in a zero point allowance**
  - **I5=0** Do not save.
  - **I5=1** Save in the active zero point allowance in the linear axes (X/Y/Z). During a save, the measured values are added to the active zero point allowance.
- **O7= E-Par Radiusdifferenz** Die Differenz zwischen dem gemessenen Radius und dem programmierten Kreisradius R wird in einem E-Parameter gespeichert. Die Nummer des E-Parameters muss eingetragen sein. Wenn keine Nummer eingetragen ist, wird nichts gespeichert.
- **X1=, Y1=, Z1= Target centre point** Once the measured coordinate is saved in the active zero point allowance (I5>0), this corrects the target value. The measured coordinate then receives the target value for further programming.

The greatest precision is achieved with a symmetrical measurement with standard values D2=90 and D3=180.

The description of further addresses is in the introduction to the measurement cycles.

Default

D1=0, D2=90, D3=180 C1=20, L2=10, I2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Application

Starting point

The starting point of the circle measurement must be selected so that the first measurement moves as precisely as possible in the direction of the circle centre.

Measurement direction

The circle measurement is executed counter-clockwise.

Sequence

1 Rapid movement to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
2 First measurement with measurement feed (F2=) until the tool or the maximum measured distance (C1=) is reached.
3 Rapid movement back to first starting point. An error message is issued if the measurement probe has not switched within the maximum measured distance (C1=).
4 Rapid movement, dependent on I3=above the safety distance (L2=), to the starting point of the 2nd measurement.
5 Second measurement (the same as described in items 2 and 4).
6 At the end, a rapid movement is executed to the safety distance (L2=).
7 Dependent on I5=, the measured value is saved.

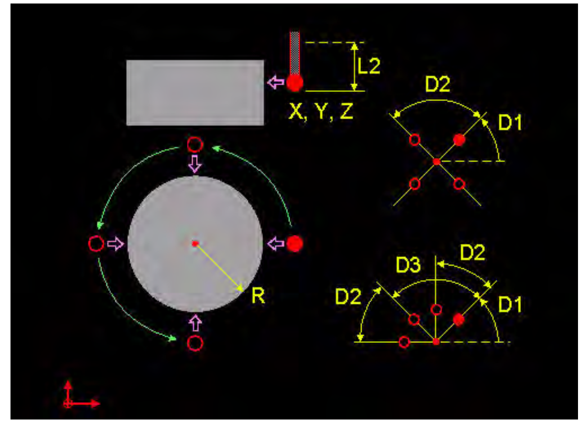
Example

Save centrepoint of a stud in the zero point allowance

```
G54 I3
G628 X-45 Y-3 Z-5 R50 I3=1 I5=1
```

G54	Set zero point.
G628	Define and execute measurement cycle. After the measured cycle, X and Y are adapted in G54 I3.

3.27 G629 Circle measurement inside




```
G     Circle measurement inside
R     Circle radius
R1=   Minimum circle radius
R2=   Maximum circle radius
X     Starting point
Y     Starting point
Z     Starting point
D1=   Starting angle
D2=   Second angle
D3=   Third angle
C1=   Measuring distance
L2=   Safety distance
I2=   Orient. -1=auto 0=no 1=180 2=yes
I3=   2nd measurem. via L2 0=no 1=yes
I5=   G5x offset 0=no 1=X/Y/Z
```

```
O1=   E-Par. meas. centre main axis
O2=   E-Par. meas. centre minor axis
O6=   E-Par. measured diameter
O7=   E-Par. radius difference
F2=   Measuring feed
X1=   Target centre point
Y1=   Target centre point
Z1=   Target centre point
```

Measuring the centre point of a circle.

Address description

- **R1= Minimum radius** The measured radius must be equal to or greater than R1, otherwise an error message will be output.
- **R2= Maximum radius** The measured radius must be equal to or smaller than R2, otherwise an error message will be output.
- **D1= Starting angle** Angle allowance of the circle measurement, based on the main axis.
- **D2= 2nd angle** Angle between first and second measurement and between the third and fourth measurement. The smallest entry value is 5°.
- **D3= 3rd angle** Angle between the first and third measurement. D3 must be at least 5° larger than D2. If D3 and D3 are the same, a 3-point measurement is executed.

> The greatest precision is achieved with a symmetrical measurement with standard values D2=90 and D3=180.

- **I5= Save measured values in a zero point allowance**
  - **I5=0** Do not save.
  - **I5=1** Save in the active zero point allowance in the linear axes (X/Y/Z). During a save, the measured values are added to the active zero point allowance.
- **O7= E-par radius difference** The difference between the measured radius and the programmed circular radius R is saved to an E-parameter. The number of the E-parameter must be entered. If no number is entered, nothing is saved.
- **X1=, Y1=, Z1= Target centre point** Saving the measured co-ordinate to the active zero point allowance (I5>0) corrects the target value. The measured co-ordinate is assigned the target value for further programming.

The description of additional addresses appears in the introduction to measuring cycles.

Default

D1=0, D2=90, D3=180 C1=20, L2=10, I2=0, I3=0, I5=0, F2=MC843, X1=0, Y1=0, Z1=0.

Application

Starting point

The starting point of the circle measurement must be selected so that the first measurement moves as precisely as possible in the direction of the circle centre.

Measuring direction

The circle measurement is executed in an anti-clockwise direction.

Sequence

1 Rapid movement to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is taken as the starting point.

2 First measurement with measuring feed (F2=) until the workpiece or the maximum measured distance (C1=) is reached.

3 Rapid traverse back to first starting point. An error message is output if the touch probe has not switched within the maximum measuring distance (C1=).

4 Rapid traverse, dependent on I3=above the clearance (L2=), to the starting point of the 2nd measurement.

5 Second measurement (as per items 2 and 4).

6 Rapid traverse to clearance (L2=).

7 The measured value is saved depending on 15=.

Example

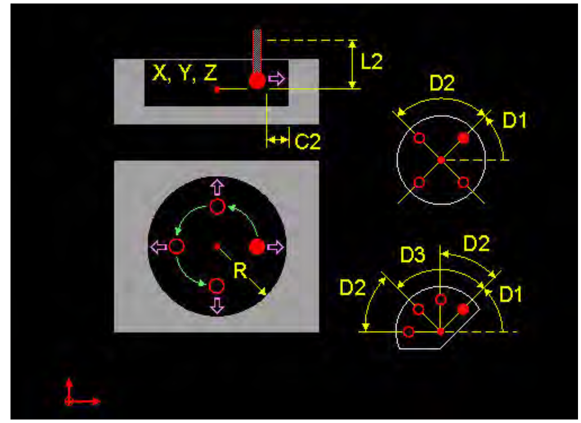
Save centrepoint of a stud in the zero point allowance

```
G54 I3
G629 X-45 Y-3 Z-5 R50 I3=1 I5=1
```

G54	Set zero point
G629	Define and execute measuring cycle. Once the measuring cycle is complete, X and Y are adapted in G54 I3.

3.28 G636 Circle measurement inside (CP)

Measure the centrepoint of a circle.




```
G     Circle measurement inside (CP)
R     Circle radius
R1=   Minimum circle radius
R2=   Maximum circle radius
X     Circle centerpoint
Y     Circle centerpoint
Z     Circle centerpoint
D1=   Starting angle
D2=   Second angle
D3=   Third angle
C2=   Pre distance meas.point
L2=   Safety distance
I2=   Orient. -1=auto 0=no 1=180 2=yes
I3=   2nd measurem. via L2 0=no 1=yes
O1=   E-Par. meas. centre main axis
```

```
O2=   E-Par. meas. centre minor axis
O6=   E-Par. measured diameter
O7=   E-Par. radius difference
F2=   Measuring feed
F5=   Feed circular movement
```

Address description

- **R1= Minimum radius** The measured radius must be equal to or greater than R1, otherwise an error message will be output.
- **R2= Maximum radius** The measured radius must be equal to or smaller than R2, otherwise an error message will be output.
- **X, Y, Z Circle centrepoint** Theoretical centrepoint of the circle to be measured.
- **D1= Starting angle** Angle allowance of the circle measurement, based on the main axis.
- **D2= 2nd angle** Angle between first and second measurement and between the third and fourth measurement. The smallest entry value is 5°.
- **D3= 3rd angle** Angle between the first and third measurement. D3 must be at least 5° larger than D2. If D3 and D3 are the same, a 3-point measurement is executed.

> The greatest precision is achieved with a symmetrical measurement with standard values D2=90 and D3=180.

- **C2= Pre-measurement distance** The distance between the starting point of the measurement movement and the theoretical circle radius. The default is MC844.
- **O7= E-par radius difference** The difference between the measured radius and the programmed circular radius R is saved to an E-parameter. The number of the E-parameter must be entered. If no number is entered, nothing is saved.
- **F5= Feed circle movement** Feed of the circle movements between measurements. The default is MC740.

The description of further addresses is in the introduction to the measurement cycles.

Default

D1=0, D2=90, D3=180, C2=MC844, L2=10, I2=0, I3=0, F2=MC843, F5=MC740

Application

Starting point

Select the starting point of the circle measurement so that the first measurement moves as precisely as possible in the direction of the circle centre.

The starting point of the measurement movement is determined from the circle centrepoint, the pre-measurement distance, and the starting angle. The measurement cycle is executed from this. If all coordinates of the centrepoint are not entered, the current position of the measurement probe is accepted.

Measurement direction

The circle measurement is executed counter-clockwise.

Sequence

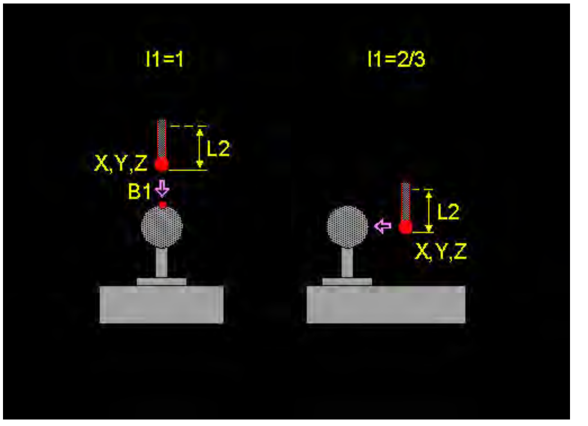
1. Rapid movement to the starting point calculated from X, Y, Z, R, and C2. If X, Y, Z are not programmed, the current position is used as the starting point.
2. First measurement with measurement feed (F2=) until the tool or the maximum measured distance (C2+MC845) is reached.
3. Rapid movement back to first starting point. An error message is issued if the measurement probe has not switched within the maximum measured distance (C2+MC845).
4. Rapid movement, dependent on I3= above the safety distance (L2=) or with a circle movement (F5=), to the starting point of the 2nd measurement.
5. Second measurement (the same as described in items 2 and 4).
6. At the end, a movement is executed in rapid movement to the safety distance (L2=).

Example: Save centrepoint and diameter of a circle in E-parameter

```
G636 X-45 Y-3 Z-5 R5 O1=1 O2=2 O6=3
```

G636	Define and execute measurement cycle. After the measuring cycle, E-parameters E1, E2, and E3 are adapted.

3.29 G638 Touch probe calibration with ball



Calibrating length, radius and oriented radius of a touch probe using a ball.

Address description

- **I1= calibrate 1=length 2=radius 3=both**
- **B1= Target position** When I1= 1 or 3, the measured co-ordinate is compared with the target position. The difference is offset in the new probe length.
- **R ball radius** When I1= 2 or 3, the ball radius must be filled up.

The description of additional addresses appears in the introduction to measuring cycles

G	Touch probe calibration on ball
I1=	Calibr. 1=length 2=radius 3=both
X	Starting point
Y	Starting point
Z	Starting point
C1=	Measuring distance
B1=	Target position
R	Ball radius
L2=	Safety distance
O1=	E-Par. probe length L
O2=	E-Par. probe radius R
O3=	E-Par. oriented probe radius R1

Default

C1=20, L2=0.

Application

General information

The touch probe must be calibrated when:

- Being used for the first time
- The touch probe pin is replaced
- The touch probe pin is bent

Calibrating probe length

To calibrate the probe length, a target position must be entered for Address B1. The new probe length is saved to Address L in the tool table. If the probe has an all-round emitter (MC846=3), the new probe length is also saved to Address L1=.

Calibrating probe radius

When a calibration ring is calculated, the centre probe radius R is determined and automatically saved to the tool table. If the probe has an all-round emitter (MC846=3), the oriented probe radius is also saved to Address R1=.

Machine constants

MC848	Calibration ring radius

Sequence for calibrating probe length (I1=1)

1 Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is taken as the starting point.

2 Measurement in tool axis until the ball or maximum measuring distance (C1=) is reached.
3 Rapid traverse back to starting point. An error message is output if the touch probe has not switched within the maximum measuring distance (C1=).
4 At the end, a rapid traverse movement back to the clearance (L2=) is executed.

Sequence for calibrating probe radius/probe radius+length (I1=2, I1=3)

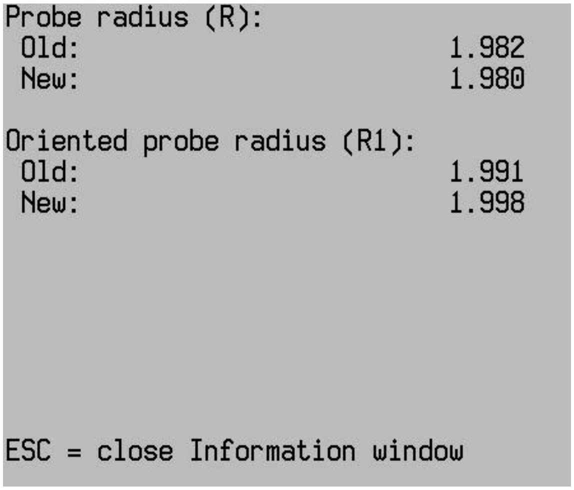
1 Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is taken as the starting point.
2 Rough measurement of centre point. An error message is output if the touch probe has not switched within the maximum measuring distance (C1=).
3 Measurement for precisely measuring the centre point.
4 Only when MC846=3: oriented measurement for determining R1
5 Non-oriented measurement for determining R.
6 At the end, a rapid traverse movement back to the clearance (L2=) is executed.

Example

Calibrate probe radius

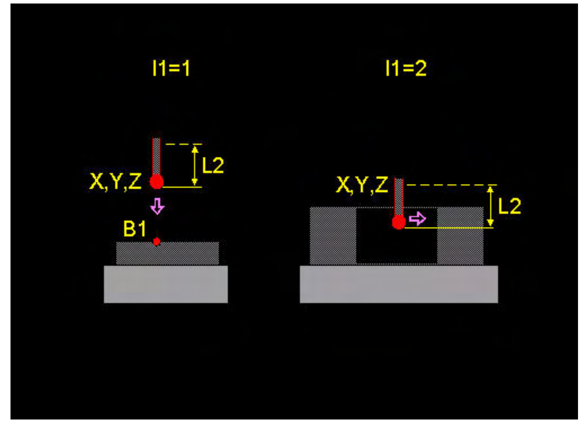
```
G54 X0 Y0 Z0
G638 R10 I1=2 X-45 Y-3 Z342.651 C1=20
```

G54	Delete zero point allowance
G638	Calibrate touch probe radius (R). Addresses R and R1are automatically adapted in the tool table.



3.30 G639 Touch probe calibration

Calibration of length, radius and oriented radius of a touch probe.



G	Touch probe calibration
I1=	Calibration 1=length 2=radius
X	Starting point
Y	Starting point
Z	Starting point
C1=	Measuring distance
B1=	Target position
L2=	Safety distance
O1=	E-Par. probe length L
O2=	E-Par. probe radius R
O3=	E-Par. oriented probe radius R1

Address description

- **I1= calibrate 1=length 2=radius**
- **B1= Target position** If I1= 1, the measured co-ordinate is compared with the target position. The difference is offset in the new probe length.

The description of additional addresses appears in the introduction to measuring cycles

Default

C1=20, L2=0.

Application

General information

The touch probe must be calibrated when:

- Being used for the first time
- The touch probe pin has been replaced
- The touch probe pin is bent

Calibrate probe length

To calibrate the probe length, a target position must be entered for Address B1. The new probe length is saved to Address L in the tool table. If the probe has an all-round emitter (MC846=3), the new probe length is also saved to Address L1=.

Calibrate probe radius

When a calibration ring is calculated, the centre probe radius R is determined and automatically saved to the tool table. If the probe has an all-round emitter (MC846=3), the oriented probe radius is also saved to Address R1=.

Machine constants

MC848	Calibration ring radius

Sequence for calibrating probe length (I1=1)

1. Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is taken as the starting point.
2. Measurement in the tool axis until the table (or measured block) or the maximum measuring distance (C1=) is reached.
3. Rapid traverse back to starting point. An error message is output if the touch probe has not switched within the maximum measuring distance (C1=).

4 At the end, a rapid traverse movement back to the clearance (L2=) is executed.

Sequence for calibrating probe radius (I1=2)

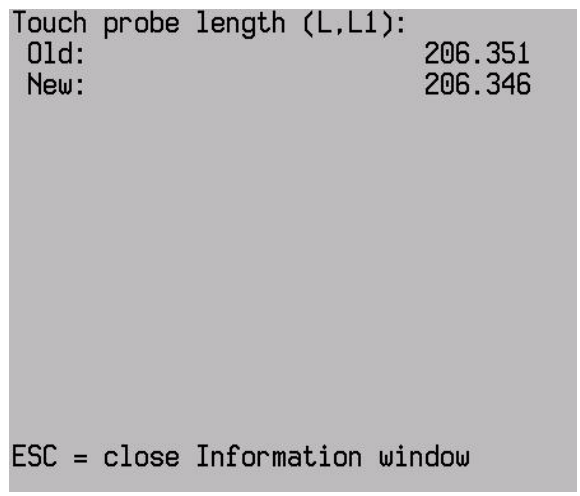
1 Rapid traverse to starting point (X, Y, Z) in calibration ring. If X, Y, Z are not programmed, the current position is taken as the starting point.
2 Rough measurement of centre point. An error message is output if the touch probe has not switched within the maximum measuring distance (C1=).
3 Measurement for precisely measuring the centre point.
4 Only when MC846=3: oriented measurement for determining R1
5 Non-oriented measurement for determining R.
6 At the end, a rapid traverse movement back to the clearance (L2=) is executed.

Example

Calibrate probe length

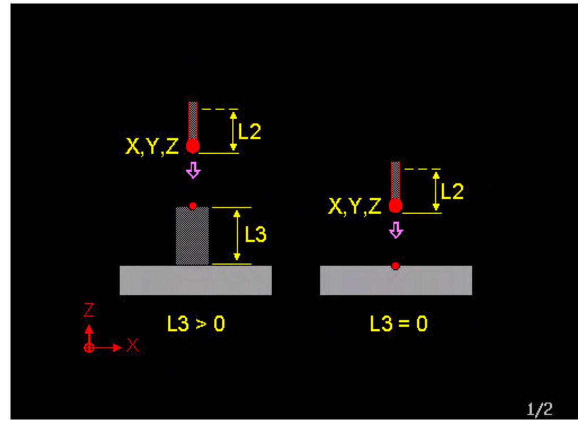
```
G54 X0 Y0 Z0
G639 I1=1 X-45 Y-3 Z342.651 C1=20 B1=309.769
```

G54	Delete zero point allowance
G639	Calibrate touch probe length (L). Address L is automatically adapted in the tool table.



3.31 G645 Determine table height

Measuring and correcting table height in the kinematic model. The active zero point remains unchanged. This cycle is available only with a password in MC342 "3D QuickSet".




```
G     Determine table height
L3=   Gauge block height
X     Starting point
Y     Starting point
Z     Starting point
C1=   Measuring distance
I5=   Correction 0=no 1=yes 2=read-in
L2=   Safety distance
O2=   E-Par. height deviation [mm/inch]
```

Address description

- **L3= Gauge block height**
  - **L3=0** Determines table height.
  - **L3>0** Offsets the length of the gauge block.
- **I5= Correction: 0=No 1=Yes 2=Read-in** The programmable elements or the main elements are corrected depending on MC349.
  - **I5=0** Measures correction value but does not save it to the kinematic model
  - **I5=1** Measures correction value, saves it to the kinematic model and offsets it
  - **I5=2** Imports the correction values to the kinematic model from the array G645RESU.ARR to D:\STARTUP\.
- **O2= `E-par. height deviation [mm/inch]`** The difference between the measured position and the position programmed into the kinematic model is saved to an E-parameter. If no number is entered, nothing is saved.

The description of additional addresses appears in the introduction to measuring cycles

Default

C1=20, I5=0.

Application

Machine constants

MC342	3D QuickSet (0=off,??????=on)
MC349	3D QuickSet mode

Conditions

- All axes must be corrected in advance via axis correction.
- The kinematic model must be entered.

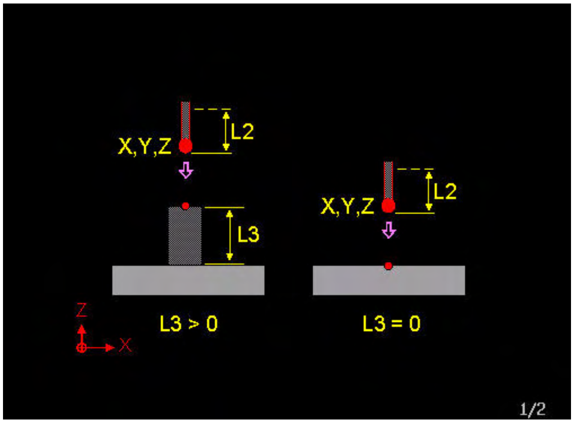
Zero point allowance

- If a zero point allowance is active, it is not deselected but rather offset.
- The active zero point is not corrected, but remains unchanged.

Measurement results

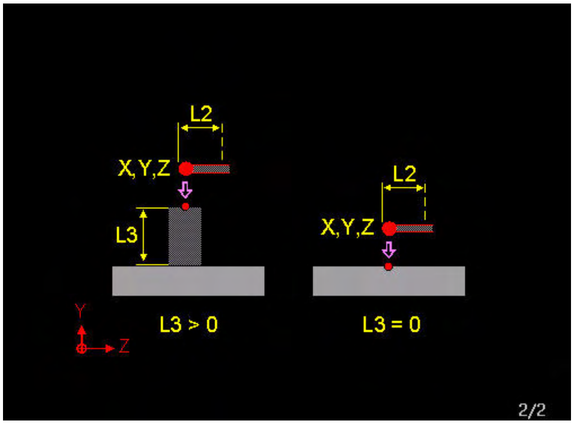
- **I5=0** The last measured values are saved to: D:\STARTUP\G645RESU.TXT and in array G645RESU.ARR. If these files do not yet exist, they will be created by G645. In manual mode, a window appears at the end of the cycle.
- **I5=1** Automatically enters the measured deviations in the elements in the kinematic model (MC_0500-MC_0699) and stores them on the hard drive, see I5=0.
- **I5=2** Imports a saved array file G645RESU.ARR from D:\STARTUP\. The values are entered into the elements in the kinematic model (MC_0500-MC_0699).

Sequence with rotary table C and rigid table



1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. If A or B rotary axes are fitted, they are positioned at zero.
3. The touch probe is positioned at the starting point and scans in a negative Z-direction.
4. The touch probe is retracted to the software limit switch (G174) or, if programmed, up to the clearance (L2=).
5. The cycle calculates the table height and writes it, as defined for I5= , to E-parameter, file or kinematics.

Sequence with rotary table B (horizontal machine)



1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. The A-rotary axis is positioned at zero.
3. The touch probe is positioned at the starting point and scans in a negative Y-direction with orientation.
4. The touch probe is retracted to the software limit switch (G174) or, if programmed, up to the clearance (L2=).
5. The cycle calculates the table height and writes it, as defined for I5= , to E-parameter, file or kinematics.

Example

Determine and automatically correct table height

```
G54 I3
G645 L3=15.000 C1=10 L2=130 X0 Y0 Z0 I5=1 O2=1
```

G54	Set zero point
G645	Determine and automatically correct table height (15=1)

Measurement results

In manual mode, a window displaying the old and new value of the programmed element appears (see picture).

The measurement results are saved to D:\STARTUP\ G645RESU.TXT (see picture).

Array

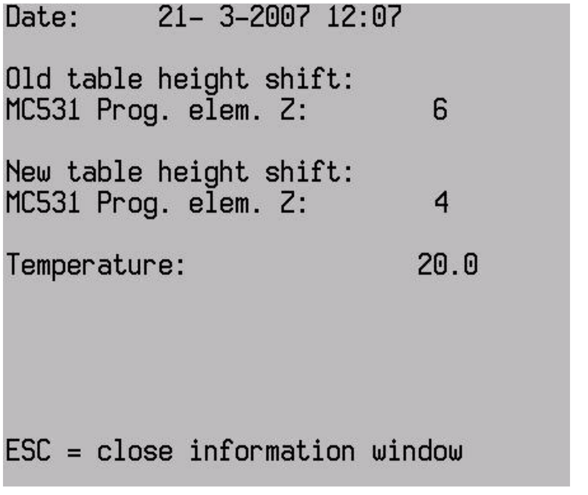
```
[BEGIN]
 MC-nr |    Value    |
    527 |    298647 |
    531 |              4 |
    535 |              0 |
[END]
```

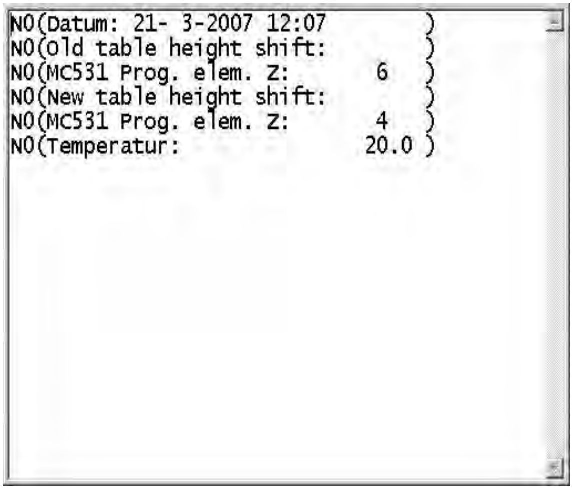
List of machine constants

N531 C4

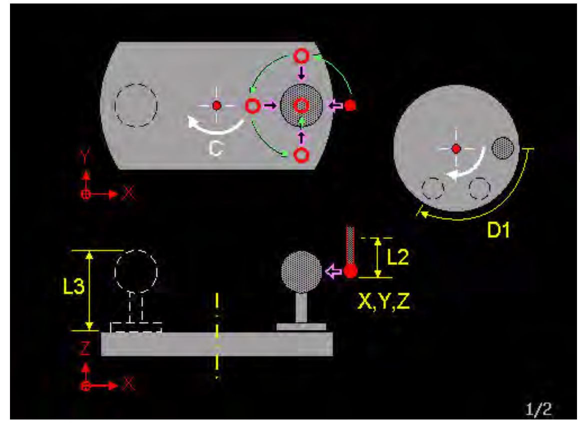
E-parameter list

E1 C-0.002





3.32 G646 Determine rotary table centre and height



G	Determine rot.table center,height
R	Ball radius
L3=	Ball bar length for corr. in Z
X	Starting point
Y	Starting point
Z	Starting point
C1=	Measuring distance
D1=	End angle
D2=	Intermed. angle with ball measur.
I5=	Correction 0=no 1=yes 2=read-in
L2=	Safety distance
O4=	E-Par. X-deviation [mm/inch]
O5=	E-Par. Y-deviation [mm/inch]
O6=	E-Par. Z-deviation [mm/inch]

Measuring and correcting the centre point of a rotary table in the kinematic model. The active zero point remains unchanged. This cycle is available only with a password in MC342 "3D QuickSet".

Address description

- **R Ball radius**
- **L3= Ball bar length** Length of ball bar. If L3= is not entered, the table height is not determined.
- **D1= End angle** Angle between first and last measurement. If D1=180 or -180 is not entered, the ball is measured at two positions, otherwise at 3 positions.

> Maximum accuracy is achieved using a symmetrical measurement with standard value D1=180.

- **D2= Intermediate angle with ball measurement** This address can be used only on a BA table. If the touch probe approaches from the side, D2= is a clearance that prevents a collision with the ball.
- **I5= Correction: 0=No 1=Yes 2=Read-in** The programmable elements or the main elements are corrected depending on MC349.
  - **I5=0** Measures correction value but does not save it to the kinematic model
  - **I5=1** Measures correction value, saves it to the kinematic model and offsets it
  - **I5=2** Imports the compensation values to the kinematic model from the array G646RESU.ARR to D:\STARTUP\.
- **O4=, O5=, O6=** **`E-par. X, Y, Z-deviation [mm/inch]`** The difference between the measured position and the position programmed into the kinematic model is saved to an E-parameter. If no number is entered, nothing is saved.

The description of additional addresses appears in the introduction to measuring cycles

Default

C1=20, D1=180, D2=60 (BA table only), I5=0.

Application

Machine constants

MC342	3D QuickSet (0=off,??????=on)
MC349	3D QuickSet mode

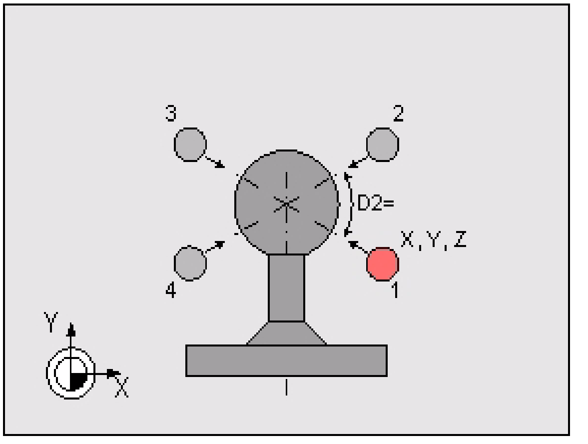
Conditions

- All axes must be corrected in advance via axis correction.

- The kinematic model must be entered.

Starting point

- The starting point of the cycle must be selected so that the first measurement moves as precisely as possible in the direction of the circle centre.
- On a BA machine, the measuring direction is defined by D2=. See picture.



Measuring direction

The circle measurement is executed in an anti-clockwise direction.

Zero point allowance

- If a zero point allowance is active, it is not deselected but rather offset.
- The active zero point is not corrected, but remains unchanged.

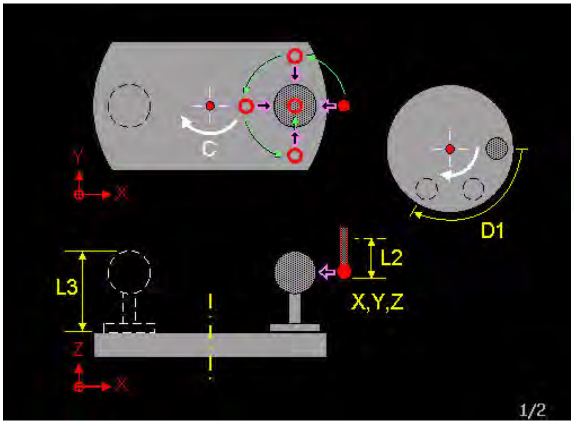
Touch probe type

Touch probes that cannot be rotated must be indexed (no oblique positions) in order to enable a precise measurement.

Measurement results

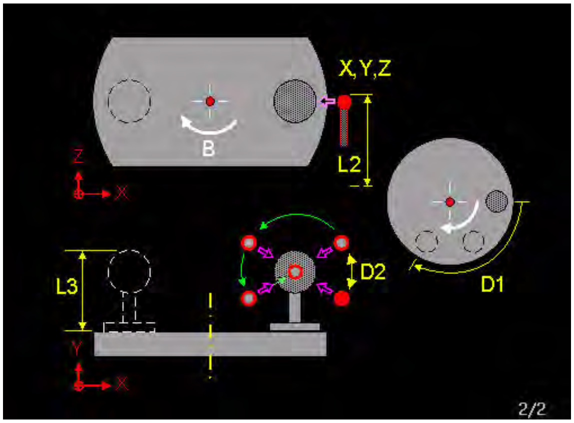
- **I5=0** The last measured values are saved to: D:\STARTUP\G646RESU.TXT and in array G646RESU.ARR. If these files do not yet exist, they will be created by G646. In manual mode, a window appears at the end of the cycle.
- **I5=1** Automatically enters the measured deviations in the elements in the kinematic model (MC_0500-MC_0699) and stores them on the hard drive, see I5=0.
- **I5=2** Imports a saved array file G646RESU.ARR from D:\STARTUP\. The values are entered into the elements in the kinematic model (MC_0500-MC_0699).

Sequence with rotary table C



1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. If A or B rotary axes are fitted, they are positioned at zero.
3. To determine the centre point of the ball, the touch probe is positioned at the starting point. The ball is then scanned parallel to the axis at the four positions opposite and at the upper surface with no orientation of the touch probe.
4. This is repeated with orientation or rotation of the touch probe in order to precisely determine the ball centre.
5. The touch probe is retracted to the software limit switch (G174) or, if programmed, up to the clearance (L2=).
6. The rotary table is rotated.
7. The ball is measured at the new position in the same manner (3–5).
8. If D1= is not equal to 180 and -180, the rotary table is rotated and the ball is measured at a third position.
9. The touch probe is retracted to the software limit switch (G174) or, if programmed, up to the clearance (L2=).
10. The rotary table is retracted to the starting position.
11. The cycle calculates the table centre point and writes it, as defined for I5=, to E-parameter, file or kinematics.

Sequence with rotary table B (horizontal machine)



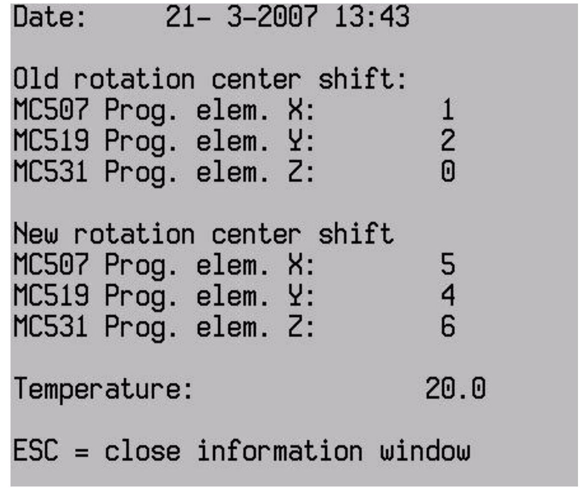
1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. The A-rotary axis is positioned at zero.
3. To determine the centre point of the ball, the touch probe is positioned at the starting point. The ball is then scanned obliquely at the four positions opposite and at the front side with no orientation of the touch probe. The intermediate angle of the measurement is defined by D2= and has a default of 60°
4. This is repeated with orientation or rotation of the touch probe in order to precisely determine the ball centre.
5. The touch probe is retracted to the software limit switch (G174) or, if programmed, up to the clearance (L2=).
6. The rotary table is rotated.
7. The ball is measured at the new position in the same manner (3–5).
8. If D1= is not equal to 180 and -180, the rotary table is rotated for a final time and the ball is measured at a third position.
9. The touch probe is retracted to the software limit switch (G174) or, if programmed, up to the clearance (L2=).
10. The rotary table is retracted to the starting position.
11. The cycle calculates the table centre point and writes it, as defined for I5=, to E-parameter, file or kinematics.

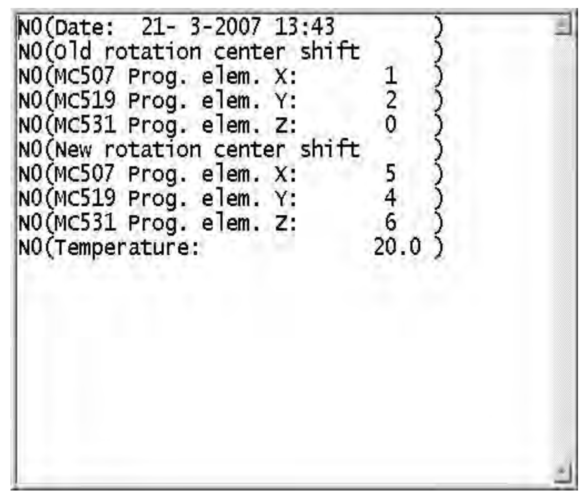
Example

Determine and automatically correct rotary table offset

```
G54 I3
G646 L3=73.448 R9 C1=10 L2=130 X0 Y0 Z0 I5=1 O4=4 O5=5 O6=6
```

G54	Set zero point
G646	Determine and automatically correct rotary table offset (I5=1)





Measurement results

In manual mode, a window displaying the old and new value of the programmed element appears (see picture).

The measurement results are saved to D:\STARTUP\ G646RESU.TXT (see picture).

Array

[BEGIN]

MC-nr	Value
503	298647
507	5
511	0
515	-480046
519	4
523	0
527	-118333
531	6
535	0

[END]

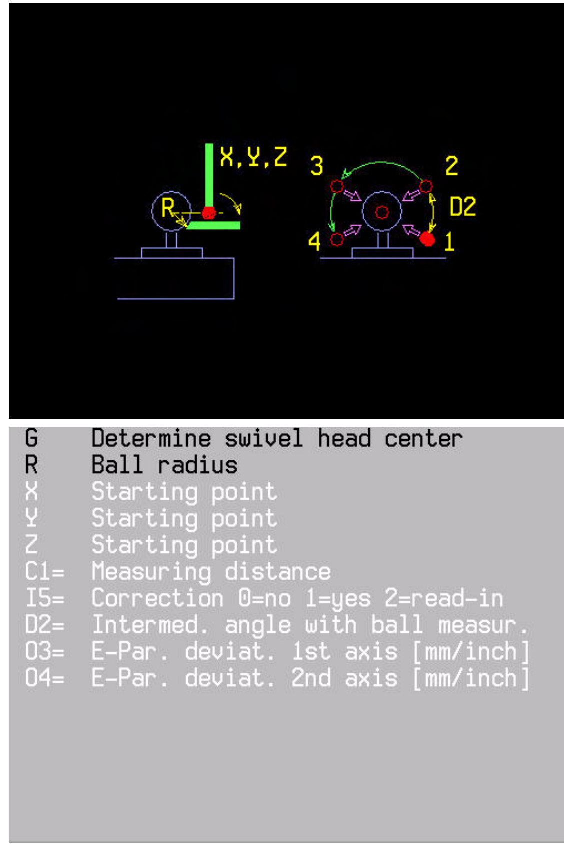
List of machine constants

..	..
N507	C5
..	..
..	..
N519	C4
..	..
..	..
N531	C6
..	..

E-parameter list

E4	C0.004
E5	C0.002
E6	C0.006

3.33 G647 Determine swivel head centre



Measuring and correcting head offset in the kinematic model. The active zero point remains unchanged. This cycle is available only with a password in MC342 "3D QuickSet".

Address description

- **R Ball radius**
- **I5= Correction: 0=No 1=Yes 2=Read-in** The programmable elements or the main elements are corrected depending on MC349.
  - **I5=0** Measures correction value but does not save it to the kinematic model
  - **I5=1** Measures correction value, saves it to the kinematic model and offsets it
  - **I5=2** Imports the correction values to the kinematic model from the array G647RESU.ARR to D:\STARTUP\.
- **D2= Intermediate angle with ball measurement** If the touch probe approaches from the side, D2= is a clearance that prevents a collision with the ball.
- **O3=, O4= E-par. 1st, 2nd axis deviation [mm/inch]** The difference between the measured position and the position programmed into the kinematic model is saved to an E-parameter. If no number is entered, nothing is saved.

The description of additional addresses appears in the introduction to measuring cycles

Default

C1=20, I5=0, D2=60.

Application

Machine constants

MC342	3D QuickSet (0=off,??????=on)
MC349	3D QuickSet mode

Conditions

- All axes must be corrected in advance via axis correction.
- The kinematic model must be entered.

Starting point

The starting point of the cycle must be selected so that the first measurement (in a negative X-direction) moves as precisely as possible in the direction of the circle centre.

Measuring direction

The circle measurement is executed in an anti-clockwise direction.

Zero point allowance

- If a zero point allowance is active, it is not deselected but rather offset.
- The active zero point is not corrected, but remains unchanged.

Touch probe type

Touch probes that cannot rotate must be very accurately aligned (no oblique positions) in order to achieve an acceptable result.

Measurement results

- **I5=0** The last measured values are saved to: D:\STARTUP\G647RESU.TXT. If this file does not yet exist, it will be created by G647. In manual mode, a window appears at the end of the cycle.
- **I5=1** Automatically enters the measured deviations in the elements in the kinematic model (MC_0500-MC_0699) and stores them on the hard drive, see I5=0.
- **I5=2** Imports a saved array file G647RESU.ARR from D:\STARTUP\. The values are entered into the elements in the kinematic model (MC_0500-MC_0699).

Sequence

1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. If fitted, the B-axis and the A-axis will be positioned at zero.
3. To determine the centre point of the ball, the touch probe is positioned at the starting point. The ball is then scanned parallel to the axis at the four positions opposite and at the upper surface with no orientation of the touch probe.
4. This is repeated with orientation or rotation of the touch probe in order to precisely determine the ball centre.
5. The touch probe is retracted to the software limit switch (G174).
6. The head is swivelled horizontally.
7. The ball is measured at the new position in the same manner (3–5). The intermediate angle of the measurement is defined by D2=.
8. The touch probe is retracted to the software limit switch (G174).
9. The tool head is retracted to the starting position.
10. The cycle calculates the head offset and writes it, as defined for I5=, to E-parameter, file or kinematics.

Example

Determine head offset but do not correct automatically

```
G54 I3
G647 C1=10 R9 X0 Y0 Z0 I5=0 D2=60 O3=3 O4=4
```

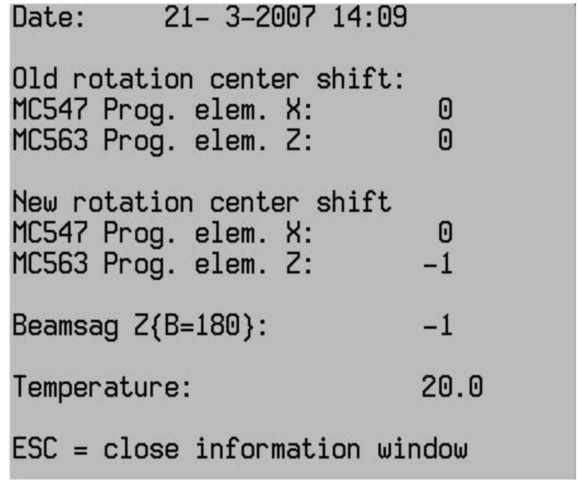
G54	Set zero point

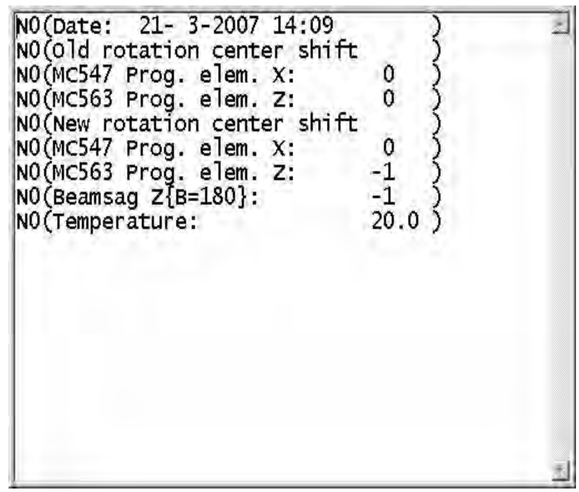
G647 Determine rotary table offset but do not correct automatically (I5=0)

Measurement results

In manual mode, a window displaying the old and new value of the programmed element appears (see picture).

The measurement results are saved to D:\STARTUP\ G647RESU.TXT (see picture).





Array

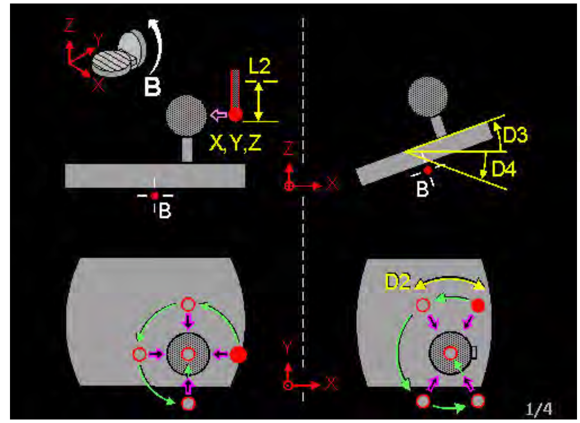
```
[BEGIN]
  MC-nr |  Value   |
    543 |       -8 |
    547 |        0 |
    551 |        0 |
    559 |   -99711 |
    563 |       -1 |
    567 |        0 |
[END]
```

E-parameter list

E3	C0
E4	C-0.001

3.34 G648 Determine swivel table centre

Measuring and correcting swivel table centre in the kinematic model. Before G648 can be used, the table centre must first be corrected via G646. The active zero point remains unchanged. This cycle is available only with a password in MC342 "3D QuickSet".




```
G     Determine swivel table center
R     Ball radius
X     Starting point
Y     Starting point
Z     Starting point
C1=   Measuring distance
I5=   Correction 0=no 1=yes 2=read-in
D2=   Intermed. angle with ball measur.
D3=   2nd angle with tilting table
D4=   3rd angle with tilting table
O3=   E-Par. deviat. 1st axis [mm/inch]
O4=   E-Par. deviat. 2nd axis [mm/inch]
```

Address description

- **R Ball radius**
- **I5= Correction: 0=No 1=Yes 2=Read-in** The programmable elements or the main elements are corrected depending on MC349.
  - **I5=0** Measures correction value but does not save it to the kinematic model
  - **I5=1** Measures correction value, saves it to the kinematic model and offsets it
  - **I5=2** Imports the correction values to the kinematic model from the array G648RESU.ARR to D:\STARTUP\.
- **D2= Intermediate angle with ball measurement** If the touch probe approaches from the side, D2= is a clearance that prevents a collision with the ball.
- **O3=, O4= `E-par. 1st, 2nd axis deviation [mm/inch]`** The difference between the measured position and the position programmed into the kinematic model is saved to an E-parameter. If no number is entered, nothing is saved.

The description of additional addresses appears in the introduction to measuring cycles

Default

C1=20, I5=0, D2=60.

Application

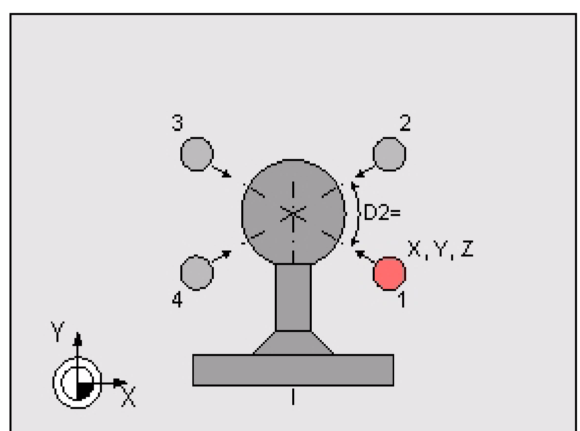
Machine constants

MC342	3D QuickSet (0=off,??????=on)
MC349	3D QuickSet mode

Conditions

- All axes must be corrected in advance via axis correction.
- The kinematic model must be entered.

Before G648 can be used, the table centre must be corrected with G646 and possibly G645.



Starting point

The starting point of the cycle must be selected so that the first measurement moves as precisely as possible in the direction of the circle centre.

On a vertical machine with A-swivel table, the measuring direction is defined by D2=. See picture.

Measuring direction

The circle measurement is executed in an anti-clockwise direction.

Zero point allowance

- If a zero point allowance is active, it is not deselected but rather offset.
- The active zero point is not corrected, but remains unchanged.

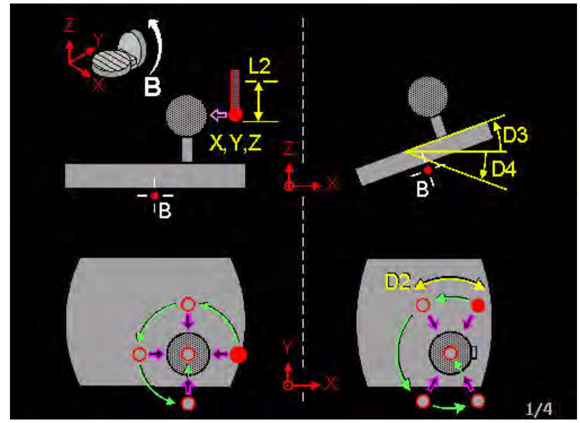
Touch probe type

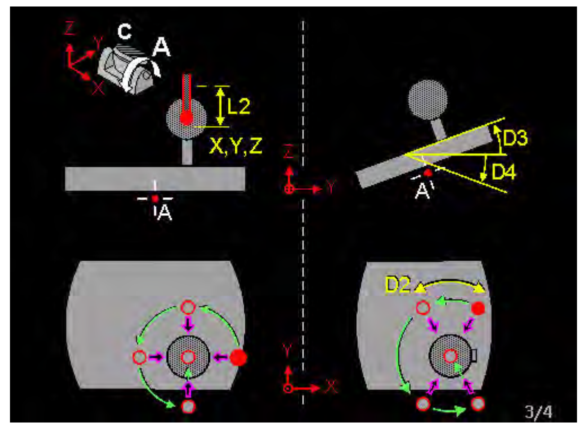
Touch probes that cannot be rotated must be indexed (no oblique positions) in order to enable a precise measurement.

Measurement results

- **I5=0** The last measured values are saved to:D:\STARTUP\G648RESU.TXT. If this file does not yet exist, it will be created by G648. In manual mode, a window appears at the end of the cycle.
- **I5=1** Automatically enters the measured deviations in the elements in the kinematic model (MC_0500-MC_0699) and stores them on the hard drive, see I5=0.
- **I5=2** Imports a saved array file G648RESU.ARR from D:\STARTUP\. The values are entered into the elements in the kinematic model (MC_0500-MC_0699).

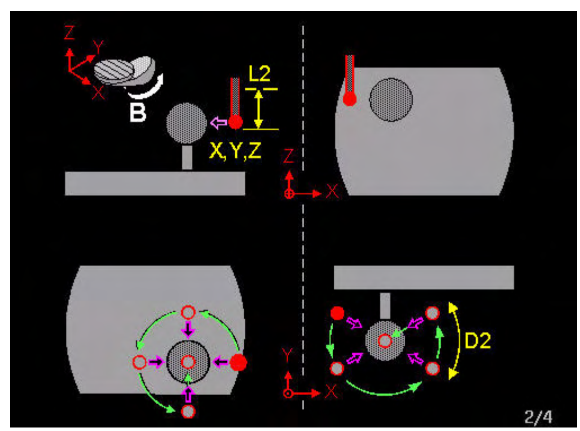
Sequence with tilting table A or B (vertical machine), 3 measured positions





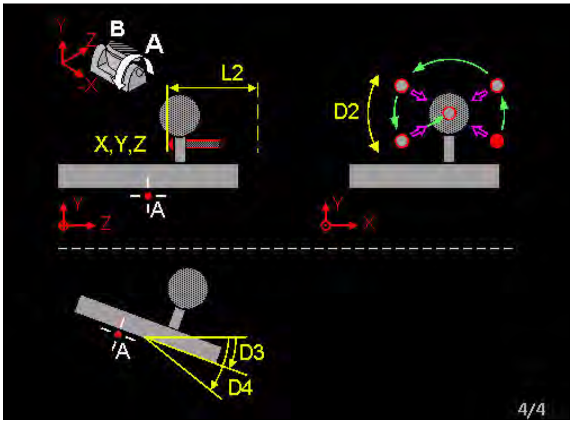
1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. If fitted, the B-axis and the A-axis will be positioned at zero.
3. To determine the centre point of the ball, the touch probe is positioned at the starting point. The ball is then scanned parallel to the axis at the four positions opposite and at the upper surface with no orientation of the touch probe.
4. This is repeated with orientation or rotation of the touch probe in order to precisely determine the ball centre.
5. The touch probe is retracted to the software limit switch (G174).
6. The swivel axis is rotated about angle D3=.
7. The ball is measured at the new position in the same manner (3–5). The intermediate angle of the measurement is defined by D2=
8. The touch probe is retracted to the software limit switch (G174).
9. The swivel axis is rotated about angle D4=.
10. The ball is measured at the new position in the same manner (3–5). The intermediate angle of the measurement is defined by D2=.
11. The touch probe is retracted to the software limit switch (G174).
12. The tilting table is retracted to the starting position.
13. The cycle calculates the table offset and writes it, as defined for I5=, to E-parameter, file or kinematics.

Sequence for swivel table B, 2 measured positions



1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. If fitted, the B-axis and the A-axis will be positioned at zero.
3. To determine the centre point of the ball, the touch probe is positioned at the starting point. The ball is then scanned parallel to the axis at the four positions opposite and at the upper surface with no orientation of the touch probe.
4. This is repeated with orientation or rotation of the touch probe in order to precisely determine the ball centre.
5. The touch probe is retracted to the software limit switch (G174).
6. The table is swivelled vertically.
7. The ball is measured at the new position in the same manner (3–5). The intermediate angle of the measurement is defined by D2=.
8. The touch probe is retracted to the software limit switch (G174).
9. The swivel table is retracted to the starting position.
10. The cycle calculates the table offset and writes it, as defined for I5=, to E-parameter, file or kinematics.

Sequence with tilting table A (horizontal machine), 3 measured positions



1. The touch probe is retracted to the software limit switch (G174). This movement stops once all positions have been transferred.
2. The A-rotary axis is positioned at zero.
3. To determine the centre point of the ball, the touch probe is positioned at the starting point. The ball is then scanned at the four positions opposite and at the upper surface with no orientation of the touch probe. The intermediate angle of the measurement is defined by D2=.
4. This is repeated with orientation or rotation of the touch probe in order to precisely determine the ball centre.
5. The touch probe is retracted to the software limit switch (G174).
6. The swivel axis is rotated about angle D3=.
7. The ball is measured at the new position in the same manner (3–5).
8. The touch probe is retracted to the software limit switch (G174).
9. The swivel axis is rotated about angle D4=.
10. The ball is measured at the new position in the same manner (3–5).
11. The touch probe is retracted to the software limit switch (G174).
12. The tilting table is retracted to the starting position.
13. The cycle calculates the table offset and writes it, as defined for I5=, to E-parameter, file or kinematics.

Example

Determine and automatically correct tilting table position

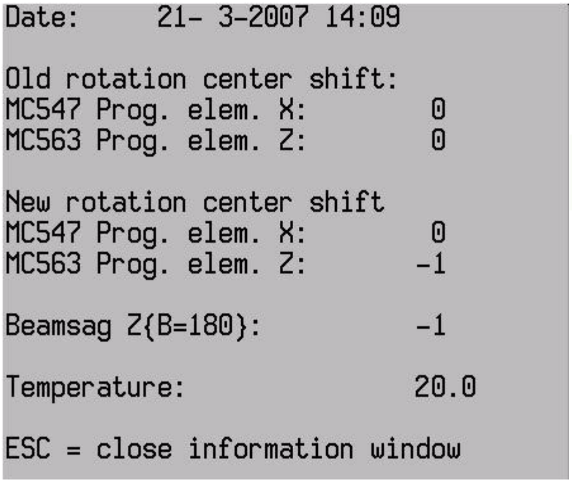
```
G54 I3
G648 R9 X0 Y0 Z0 C1=10 I5=1 D2=60 D3=-45 D4=45 O3=3 O4=4
```

G54	Set zero point
G648	Determine and automatically correct tilting table position (I5=1)

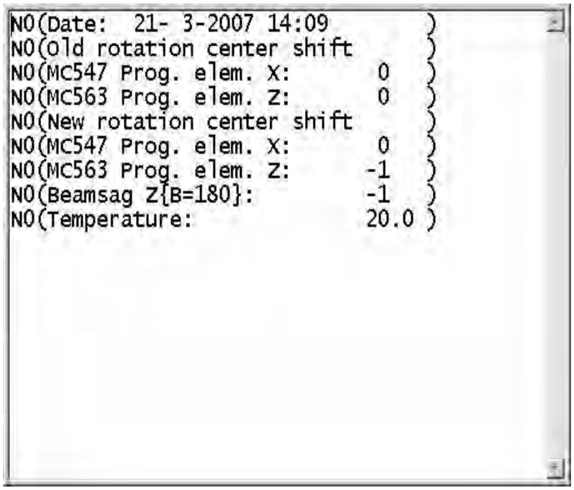
Measurement results

In manual mode, a window displaying the old and new value of the programmed element appears (see picture).

The measurement results are saved to D:\STARTUP\ G648RESU.TXT (see picture).



```
Date:      21- 3-2007 14:09

Old rotation center shift:
MC547 Prog. elem. X:         0
MC563 Prog. elem. Z:         0

New rotation center shift
MC547 Prog. elem. X:         0
MC563 Prog. elem. Z:        -1

Beamsag Z{B=180}:           -1

Temperature:                20.0

ESC = close information window
```



```
N0(Date:   21- 3-2007 14:09       )
N0(Old rotation center shift      )
N0(MC547 Prog. elem. X:       0   )
N0(MC563 Prog. elem. Z:       0   )
N0(New rotation center shift      )
N0(MC547 Prog. elem. X:       0   )
N0(MC563 Prog. elem. Z:      -1   )
N0(Beamsag Z{B=180}:         -1   )
N0(Temperature:              20.0 )
```

Array

```
[BEGIN]
  MC-nr |   Value    |
    543 |         -8 |
    547 |          0 |
    551 |          0 |
    559 |    154970 |
    563 |         -1 |
    567 |          0 |
[END]
```

List of machine constants

```
..      ..
N547    C0
..      ..
..      ..
N563    C-1
..      ..
```

E-parameter list

```
E3      C0
E4      C-0.001
```

3.35 G691 Measure imbalance

Until now, only one radial position could be calculated for a selected mass.
The dialog window has been expanded so that a mass for a selected radial position can be calculated.

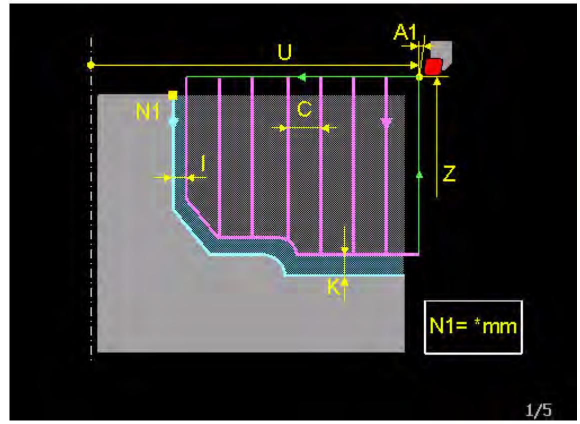
3.36 G710 U-head cutting axial

The "U-head contour cutting cycle axial" machines the workpiece parallel to the axis from the complete material or from the blank allowance up to the programmed contour profile or finishing allowance. The contour description is established in a macro.

The cycle is available only if MC_0343 "U-head cycles" is equal to 1.

See G880 "Contour cutting axial" for a detailed description of the cycle.

A general description of the facing head is provided at the start of the User Manual.



G	U-head cutting axial
U	Starting point
Y	Starting point
Z	Starting point
C	Cutting depth
N1=	Contour macro
I1=	Finish. 0=cont.dir. 1=downwards
I2=	Reverse contour 0=no 1=yes
N2=	Raw contour macro
B	Raw material allowance
A1=	Clearance angle
I	Finishing allowance
K	Finishing allowance
S	(Cutting) Speed
F	Feed

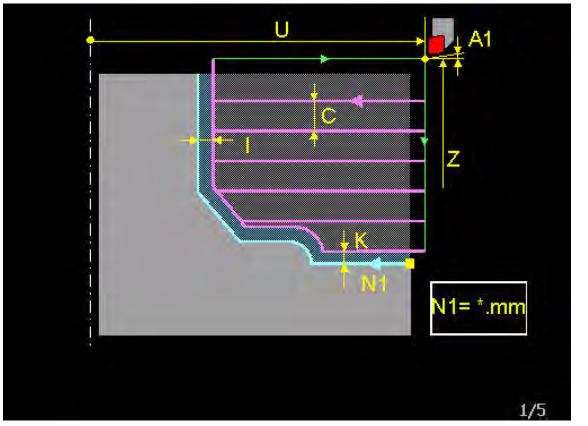
3.37 G711 U-head cutting radial

The "U-head contour cutting cycle radial" machines the workpiece parallel to the axis from the complete material or from the blank allowance up to the programmed contour profile or finishing allowance. The contour description is established in a macro.

The cycle is available only if MC_0343 "U-head cycles" is equal to 1.

See G881 "Contour cutting radial" for a detailed description of the cycle.

A general description of the facing head is provided at the start of the User Manual.



G	U-head cutting radial
U	Starting point
Y	Starting point
Z	Starting point
C	Cutting depth
N1=	Contour macro
I1=	Finish. 0=cont.dir. 1=downwards
I2=	Reverse contour 0=no 1=yes
N2=	Raw contour macro
B	Raw material allowance
A1=	Clearance angle
I	Finishing allowance
K	Finishing allowance
S	(Cutting) Speed
F	Feed

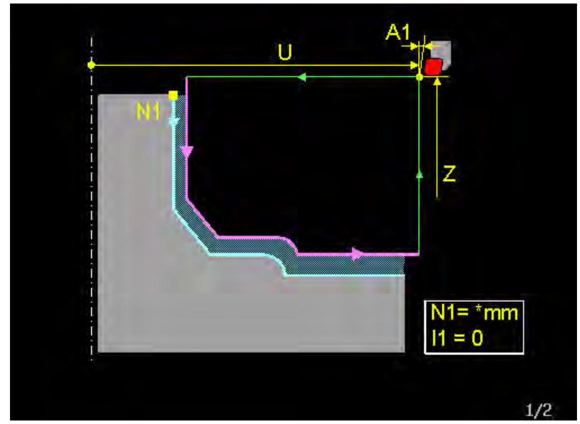
3.38 G714 U-head cutting axial finishing

The "U-head contour cutting cycle axial finishing" machines the workpiece parallel to the axis from the complete material or from the blank allowance up to the programmed contour profile or finishing allowance. The contour description is established in a macro.

The cycle is available only if MC_0343 "U-head cycles" is equal to 1.

See G884 "Contour cutting axial finishing" for a detailed description of the cycle.

A general description of the facing head is provided at the start of the User Manual.



G	U-head cutting axial finishing
U	Starting point
Y	Starting point
Z	Starting point
N1=	Contour macro
I1=	Finish. 0=cont.dir. 1=downwards
I2=	Reverse contour 0=no 1=yes
A1=	Clearance angle
I	Allowance
S	(Cutting) Speed
F	Feed

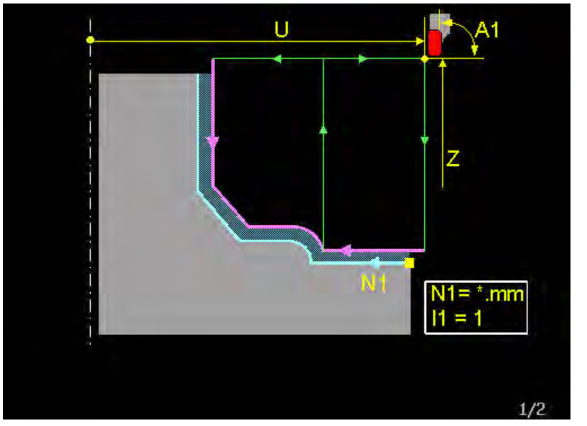
3.39 G715 U-head cutting radial finishing

The "U-head contour cutting cycle radial finishing" machines the workpiece parallel to the axis from the complete material or from the blank allowance up to the programmed contour profile or finishing allowance. The contour description is established in a macro.

The cycle is available only if MC_0343 "U-head cycles" is equal to 1.

See G885 "Contour cutting radial finishing" for a detailed description of the cycle.

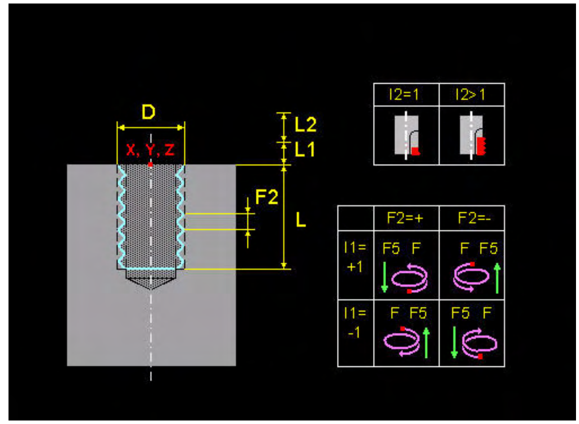
A general description of the facing head is provided at the start of the User Manual.



G	U-head cutting radial finishing
U	Starting point
Y	Starting point
Z	Starting point
N1=	Contour macro
I1=	Finish. 0=cont.dir. 1=downwards
I2=	Reverse contour 0=no 1=yes
A1=	Clearance angle
K	Allowance
S	(Cutting) Speed
F	Feed

3.40 G740 Thread milling inside

An inner thread is milled with this function.




```
G    Thread milling inside
D    Diameter
F2=  Pitch, +/-=thread direction
L    Depth
I2=  Number of threads per step
L1=  1st Setup clearance
L2=  2nd Setup clearance
I1=  Milling 1=climb -1=conventional
F5=  Plunge/Retract rapid
F    Feed
S    Speed
```

Address description

- **D Diameter** Nominal thread diameter.
- **F2= Thread pitch and direction** The sign determines the thread pitch: right thread (+) and left thread (-). Range: +/- 99.9999 mm.
- **L Depth** Distance between tool surface and thread base.
- **I2= Number of thread cuts per step** Number of thread ridges per tool:
  - I2=1 one ridge. Continuous helix over the length of the thread
  - I2>1 several ridges. Several helix paths with start and departure. The tool is pushed between start and departure by I2 times the pitch.
- **L1= Safety distance** 1 Distance between the tool tip and tool surface.
- **L2= Safety distance** 2 Distance in tool direction wherein no collision between tool and clamp can occur.
- **I1= Milling** Type of mill machining: +1 = forwards, –1 = reverse.
- **F5= Rapid movement plunging/retraction** Maximum speed while plunging or retracting. Can be influenced by rapid movement override.
- **F Feed**
- **S Spindle speed**

Defaults

I1=1, L1=F2, L2=0, F5=F

Notes and usage

Tool for thread-milling

The tool for thread-milling requires a specific compensation value, which is entered in the catalogue of the tool manufacturer. This value must be entered in the allowance radius (R4=) in the tool table.

Note that the tool moves beyond the programmed depth during tangential start or departure, and a **collision** can occur with insufficient clearance.

Tangential retracting and extending with G740 and G741 is calculated as follows:

- Tangential retracting and extending is executed with a semicircle where radius = pitch.
- Lead cut/overflow = F2 * F2 / 2 * Helix diameter (helix diameter thread diameter / 2 - tool diameter).
- Usually the helix radius is smaller than the pitch, and the overflow is smaller than half of the pitch.

Mill machining starts in the tool axis at the starting point or at the thread base. This direction is determined by the pitch direction (F2=+/-) and mill direction (I1=).

For tools turning right, the relationship between the entry parameters is:

Inner thread	Pitch (F2=)	Mill direction (I1) +1 Forwards -1 Reverse	Working direction of tool axis
	+ Right thread	I1=+1	Z+
	+ Right thread	I1=-1	Z-
	- Left thread	I1=+1	Z-
	- Left thread	I1=-1	Z+

Outer thread	Pitch (F2=)	Mill direction (I1) +1 Forwards -1 Reverse	Working direction of tool axis
	+ Right thread	I1=+1	Z-
	+ Right thread	I1=-1	Z+
	- Left thread	I1=+1	Z+
	- Left thread	I1=-1	Z-

Cycle sequence

1. The thread mill is positioned at the safety distance above the tool surface in rapid movement.
2. The thread mill moves to the starting position in rapid movement. This position is determined by the thread pitch (F2=), the running direction (I1=), and the number of thread cuts per step (I2=).
3. The mill executes a compensation movement to receive the correct starting position. Then the mill tangentially moves to the thread radius in the helix.
4. Dependent on the entry parameter "Number of thread cuts per step" (I2=), the tool mills the thread in one or more cuts or in a continuous helix movement.
5. At the end, the mill moves away from the tool in the helix tangentially. Then the mill returns to the starting position with increased feed.
6. At the end of the cycle, the tool returns to the 1st, and, if programmed, the 2nd safety distance in rapid movement.

Feed

Normally, the feed is based on the tool centre. In this case, the feed is based on the tool radius (see: F1=, constant cut feed with radius compensation of circles).

Attention

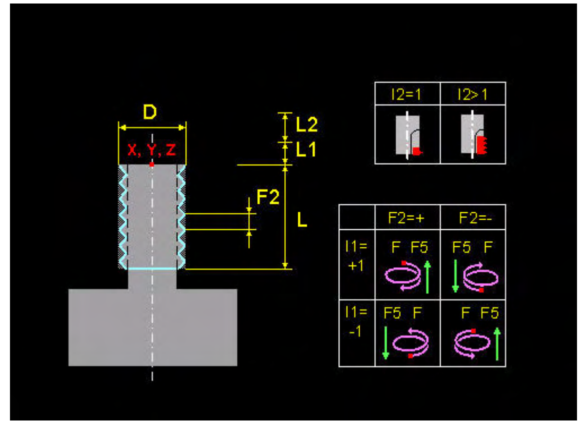
Typically, the mill direction is from bottom to top (see example). The mill direction can also be from top to bottom, depending on the parameters I1=/F2.

Example

```
T2 M6
S800 F120 M3
G740 D=60 F2=5,5 L16 I2=1 F5=1500 I1=1 L1=5 F=200
G79 X0 Y0 Z0
```

3.41 G741 Thread milling outside

An outer thread is milled with this function.



G	Thread milling outside
D	Diameter
F2=	Pitch, +/-=thread direction
L	Depth
I2=	Number of threads per step
L1=	1st Setup clearance
L2=	2nd Setup clearance
I1=	Milling 1=climb -1=conventional
F5=	Plunge/Retract rapid
F	Feed
S	Speed

Address description

- **D Diameter** Nominal thread diameter.
- **F2= Thread pitch and direction** The sign determines the thread pitch: right thread (+) and left thread (-). Range: +/- 99.9999 mm.
- **L Depth** Distance between tool surface and thread base.
- **I2= Number of thread cuts per step** Number of thread ridges per tool:
  - I2=1 one ridge. Continuous helix over the length of the thread.
  - I2>1 several ridges. Several helix paths with start and departure. The tool is pushed between start and departure by I2 times the pitch.
- **L1= Safety distance** 1 Distance between the tool tip and tool surface.
- **L2= Safety distance** 2 Distance in tool direction wherein no collision between tool and clamp can occur.
- **I1= Milling** Type of mill machining: +1 = forwards, –1 = reverse.
- **F5= Rapid movement plunging/retraction** Maximum speed while plunging or retracting. Can be influenced by rapid movement override
- **F Feed**
- **S Spindle speed**

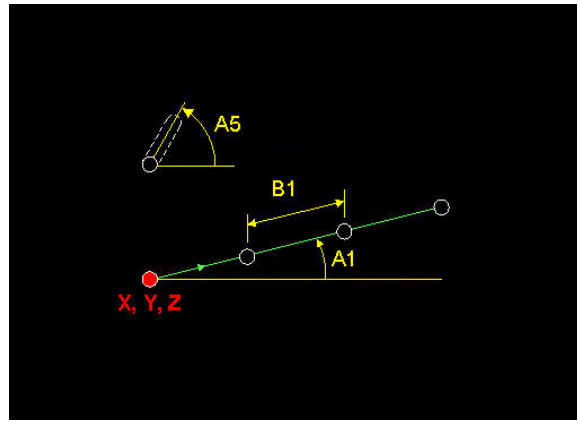
Defaults

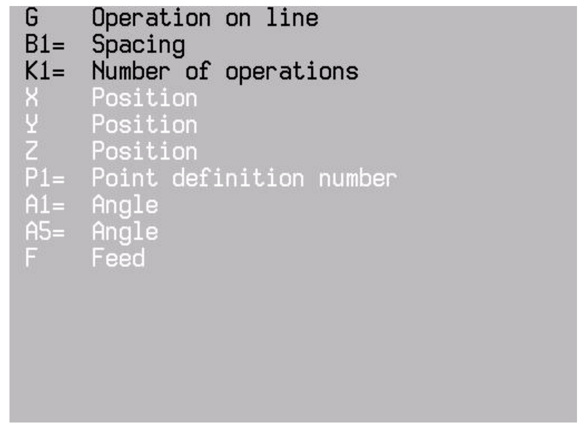
I1=1, L1=F2, L2=0, F5=F

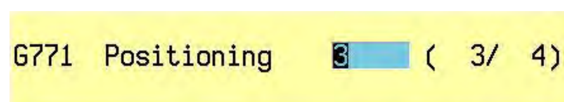
Example

```
T2 M6
S800 F120 M3
G740 D=60 F2=5,5 L16 I2=1 F5=1500 I1=1 L1=5 F=200
G79 X0 Y0 Z0
```

3.42 G771 Operation on line







Execution of a machining cycle at points that are located at fixed equal distances on a line.

Address description

See picture

Default

A1=0, A2=90, A5=0.

Application

Machining position

The machining position is defined using X,Y,Z or point definition number P1=.

Jump in the pattern

In single block mode, it is possible to jump to a specific position (machining) in the pattern. The desired number of the machining is entered in the input window (see picture).

1. After the start, a rapid movement is made to the safety distance via the desired machining position.
2. After restart, machining begins.

Numbering the pattern

The machining at position X,Y,Z is the first one.

Pocket angle

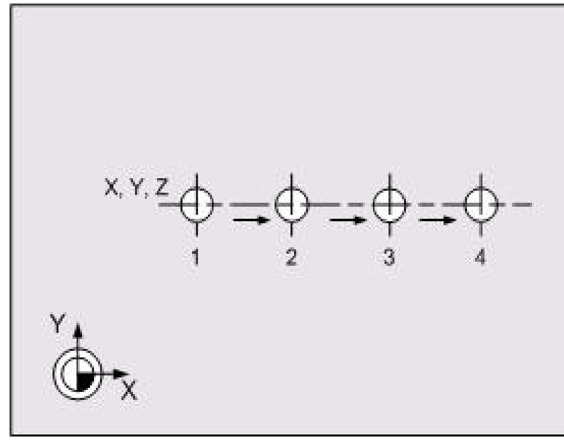
The pocket angle is defined by A5.

Sequence

1. Rapid movement to the position.
2. The machining cycle previously defined is executed at this spot.
3. The next position is approached after execution.
4. Repeat procedure (2-3) until all positions (K1=) have been machined.

Example

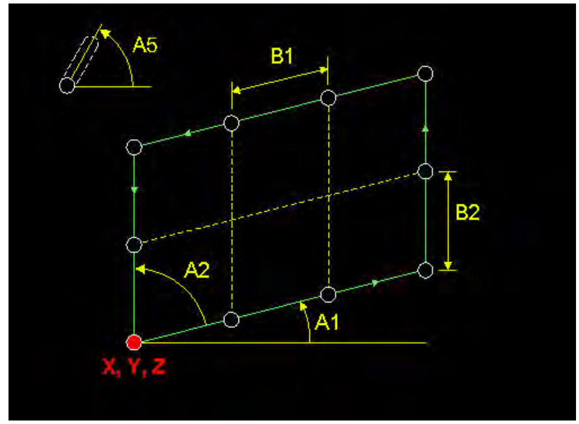
```
G781 L30 F100 F5=6000
G771 X50 Y20 Z0 B1=40 K1=4
```



G781	Define bore cycle
G771	Execute bore cycle at 4 positions

3.43 G772 Operation on quadrangle

Execution of a machining cycle at points that are located in fixed distances on a rectangle.



G	Operation on quadrangle
B1=	Longitudinal spacing
K1=	Number of longitudinal operations
B2=	Transverse spacing
K2=	Number of transverse operations
X	Position
Y	Position
Z	Position
P1=	Point definition number
A1=	Starting angle
A2=	Ending angle
A5=	Angle
F	Feed

G772 Positioning 3 (3/ 10)

Address description

See picture

Default

A1=0, A2=90, A5=0.

Application

Machining position

The machining position is defined using X,Y,Z or point definition number P1=.

Jump in the pattern

In single block mode, it is possible to jump to a specific position (machining) in the pattern. The desired number of the machining is entered in the input window.

1. After the start, a rapid movement is made to the safety distance via the desired machining position.
2. After restart, machining begins.

Numbering the pattern

Numbering starts with position X, Y, Z.

Pocket angle

The pocket angle is defined by A5.

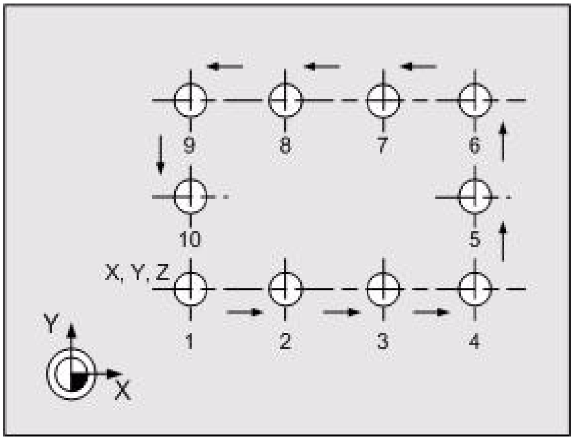
Sequence

1. Rapid movement to the position.
2. The machining cycle previously defined is executed at this spot.
3. The next position is started after execution. The direction of the rectangle is determined by angle A1=.
4. Repeat procedure (2-3) until all positions (K1=, K2=) have been machined.

Example

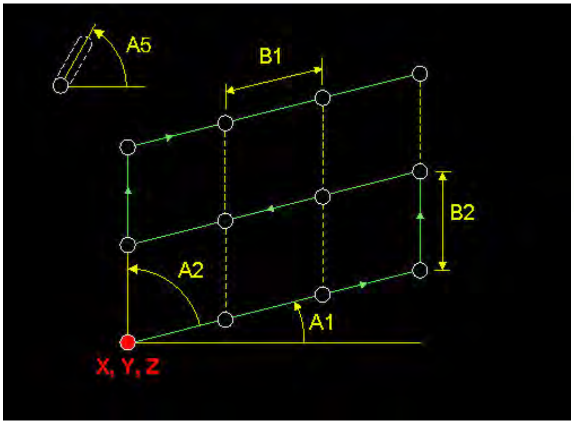
```
G781 L30 F100 F5=6000
G772 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3
```

G781	Define bore cycle
G772	Execute bore cycle on the rectangle with 10 positions



3.44 G773 Operation on grid

Execution of a machining cycle at points that are located in fixed distances on a grid.



G	Operation on grid
B1=	Longitudinal spacing
K1=	Number of longitudinal operations
B2=	Transverse spacing
K2=	Number of transverse operations
X	Position
Y	Position
Z	Position
P1=	Point definition number
A1=	Starting angle
A2=	Ending angle
A5=	Angle
F	Feed

G773 Positioning 3 (3/ 12)

Address description

See picture

Default

A1=0, A2=90, A5=0.

Application

Machining position

The machining position is defined using X,Y,Z or point definition number P1=.

Jump in the pattern

In single block mode, it is possible to jump to a specific position (machining) in the pattern. The desired number of the machining is entered in the input window.

1. After the start, a rapid movement is made to the safety distance via the desired machining position.
2. After restart, machining begins.

Numbering the pattern

Numbering starts with position X, Y, Z.

Pocket angle

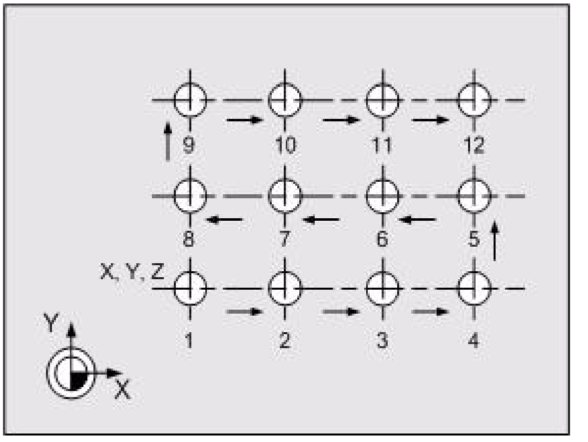
The pocket angle is defined by A5.

Sequence

1. Rapid movement to the position.
2. The machining cycle previously defined is executed at this spot.
3. The next position is started after execution. The positions are advanced in zigzags in the start direction, determined by angle A1=.
4. Repeat procedure (2-3) until all positions (K1=, K2=) have been machined.

Example

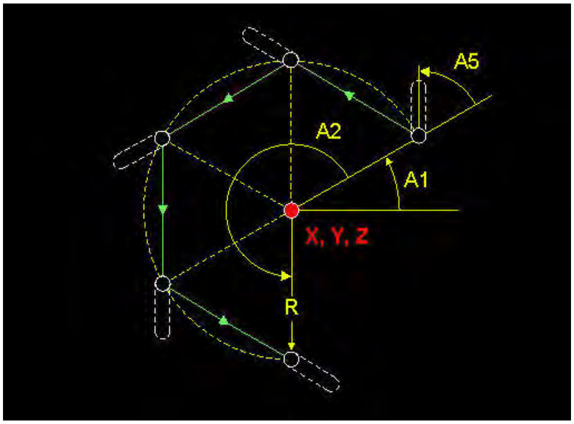
```
G781 L30 F100 F5=6000
G773 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3
```



G781	Define bore cycle
G773	Execute bore cycle on the grid with 10 positions

3.45 G777 Operation on circle

Execution of a machining cycle at points that are located in fixed distances on a semi-circle or circle.



G	Operation on circle
R	Radius
K1=	Number of operations
X	Position
Y	Position
Z	Position
B2=	Polar angle
L2=	Polar length
P1=	Point definition nr. for centre
A1=	Starting angle
A2=	Ending angle
A5=	Angle
F	Feed

G777 Positioning 3 (3/ 6)

Address description

See picture

Default

A1=0, A2=360.

Application

Machining position

The machining position is defined using X,Y,Z,B2,L2 or point definition number P1=.

Machining direction

If A2= negative, the holes are clockwise.
If A2= positive, the holes are counter-clockwise.

Jump in the pattern

In single block mode, it is possible to jump to a specific position (machining) in the pattern. The desired number of the machining is entered into the input window.

1. After the start, a rapid movement is made to the safety distance via the desired machining position.
2. After restart, machining begins.

Numbering the pattern

Numbering starts with starting angle A1 and goes in the direction of A2.

Pocket angle

If A5 is not programmed, the pocket angles are the same opposite the main axis.
If A5=0, then the pocket angle turns with the circle.
If A5 is not equal to 0, an additional rotation is added.

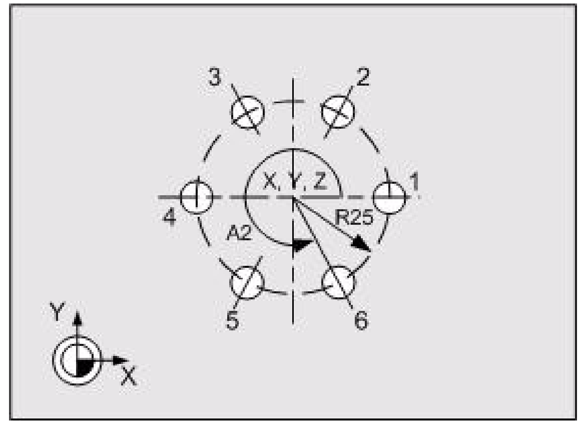
Sequence

1. Rapid movement to the position.
2. The machining cycle previously defined is executed at this spot.
3. The next position is started after execution. The direction of the positions is determined by A1= and A2=.
4. Repeat procedure (2-3) until all positions (K1=) have been machined.

Example

Cycle on a circle

```
G781 L30 F100 F5=6000
G777 X50 Y20 Z0 R=25 K1=6 A1=0 A2=300
```

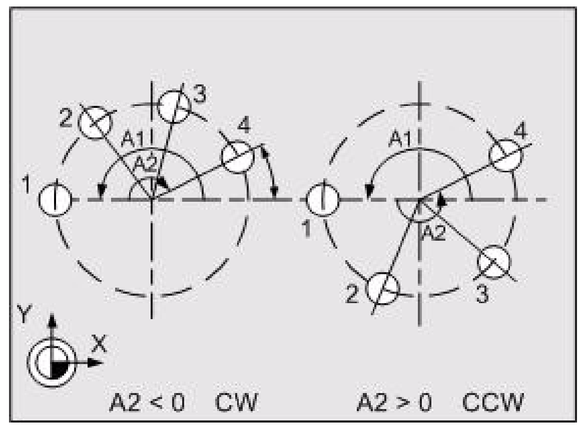


G781	Define bore cycle.
G777	Execute bore cycle on a circle with 6 points.

- K1=6 (Number of holes)
- A1=0 (Starting angle)
- A2=300 (End angle)

Direction of bore holes on a semi-circle

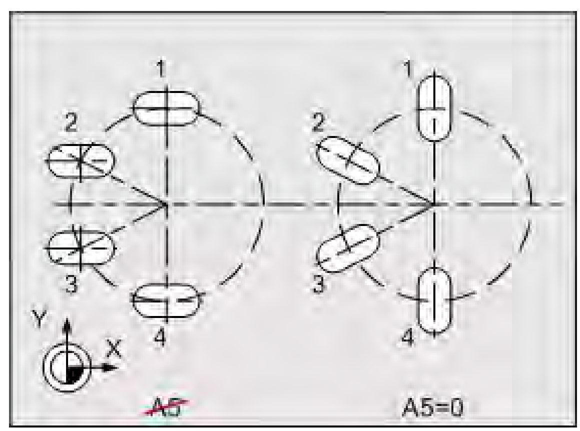
```
G781 L30 F100 F5=6000
G777 X0 Y0 Z0 R25 A1=180 A2=-150 K1=4
G777 X0 Y0 Z0 R25 A1=-180 A2=210 K1=4
```



G781	Define bore cycle.
G777	Repeat cycle four times on the semi-circle, starting with 180 degrees, ending with 30 degrees, clockwise (CW).
G777	Repeat cycle four times on the semi-circle, starting with 180 degrees, ending with 30 degrees, counter-clockwise (CCW).

Angle of the slots on a semi-circle

```
G788 B1=16 B2=8 L5 F5=6000
G777 X0 Y0 Z0 R25 A1=90 A2=180 K1=4
G777 X0 Y0 Z0 R25 A1=90 A2=180 K1=4 A5=0
```

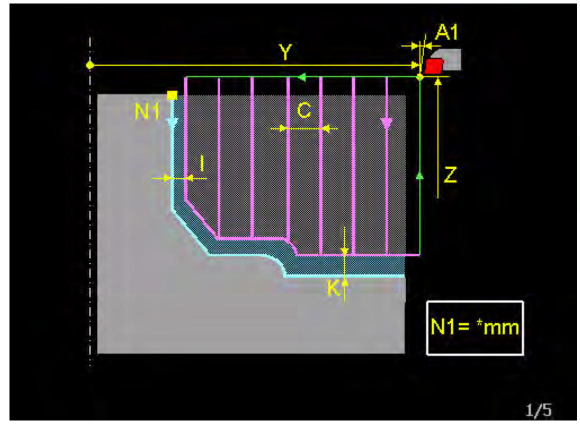


G788	Define slot cycle.
G777	The slots all have the same direction.
G777	The slot angle is dependent on the position on the semi-circle.

3.46 G880 Contour cutting axial

The contour machining cycle machines the tool lengthwise, parallel to the axis, from the complete material or from the blank allowance up to the programmed contour or finishing allowance. The contour description is established in a macro.

Contour machining with grooving tools is executed with consideration of the tool width through both sides of the grooving tool.



G	Contour cutting axial
Y	Starting point
Z	Starting point
C	Cutting depth
N1=	Contour macro
I1=	Finish. 0=cont.dir. 1=downwards
I2=	Reverse contour dir. 0=no 1=yes
N2=	Raw contour macro
B	Raw material allowance
A1=	Clearance angle
I	Finishing allowance
K	Finishing allowance
S1=	(Cutting) Speed
F	Feed

Address description

- **Y, Z Starting point** Starting point for the contour machining cycle.
- **C Feed depth** Dimension by which the tool is fed in each radial direction. The depth may not be a multiple of the feed depth.
- **N1= Contour macro** Macro (*.MM) in which the contour description is saved.
- **I1= Finishing** Machining direction of the last cut: 0: contour direction, 1: flank direction.
- **I2= Reverse contour direction 0=no 1=yes** Reversal of the contour(s) if different than in the support picture.
- **N2= Blank contour macro** Macro (*.MM) in which the blank contour description is saved.
- **B Blank allowance** allowance around the contour (N1=) or blank contour (N2=) (0 to 100 mm).
- **A1= Clearance angle** Clearance angle of the tool. (0 to 90°).
- **I, K Finishing allowance** allowance in the Y and Z axis.

Default

I=0, K=0, B=0, A1=90, I1=0, I2=0

Application

Cycle starting point (Y/Z)

The cycle starting point must lie outside of the contour starting point. Note the tool orientation, dependent on the machining direction.

It is permitted, if necessary, to place the contour end point for the Y axis below or above the contour start point.

Error messages: (dependent on machining direction)

P362 Tool with wrong orientation:Starting point in Y is smaller/larger than contour starting point Y in the macro.

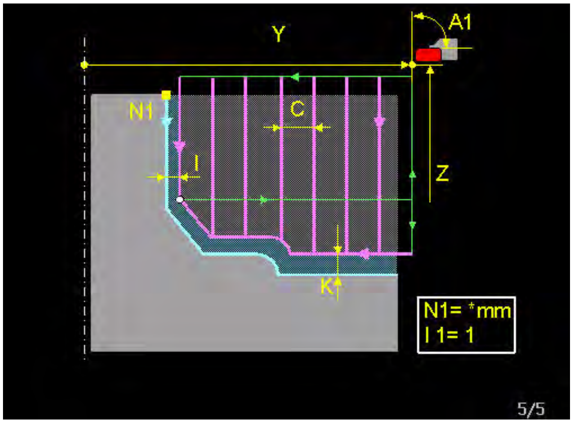
P363 Start point in material: Starting point in Z is smaller/larger than contour starting point Z in the macro.

Clearance angle (A1)

The clearance angle (A1=) detects whether there is residual material when machining infeed contour elements. A "Warning: Rest material" message is issued.

The clearance angle (A1=) must be entered into the cycle or into the tool table.

If A1=0, infeed contour elements are skipped.



Feed (F)

With infeed contour elements, the plunge feed is reduced proportionally by 1/3xF to F with a contour angle between 0° and 30 ° and from 1/3xF to F between 30° and 90°.

Tool orientation (O)

Make sure that the tool orientation (O) corresponds to the machining direction (-/+Z), machining type (inner/outer) and machining level G17/ G18.

If the tool orientation (O) is not present in the tool table or isn't programmed with G302 Oxx, it is derived from the machining direction and machining level.

Tool nose radius compensation (C in tool table)

The tool nose radius compensation is effective during machining.

Contour direction, finishing I1 (see picture)

- I1=0 The machining direction of the last cut is in the contour definition direction (see pictures).
- I1=1 The machining direction of the last cut is along the contour flank in the direction of the deepest point of the contour.

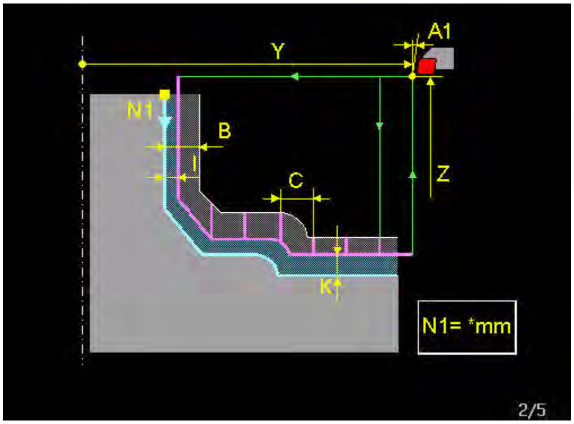
When finishing in the flank direction (I1=1), make sure of the following:

- The measured cut position of the grooving tool must correspond to the current tool orientation.
- The tool cutter width (C6=) must be entered in the tool table for grooving tools. If no value is entered, only the tool cutter radius (C) is corrected.
- If the clearance angle A1=0, the infeed contour sections are skipped.
- If the width of a infeed contour section is smaller than the cutter width (C6=), this is skipped.

Reverse contour direction

- If the contour direction (N1=) is opposite the machining direction, the contour direction is adapted to the machining direction using I2=1.
- I2=0 The contour direction is defined by the contour starting point up to the contour end point and must be described according to the cycle machining direction.
- I2=1 The contour direction was not described according to the cycle machining direction.

The contours from N1 and N2 must be programmed in the same direction.

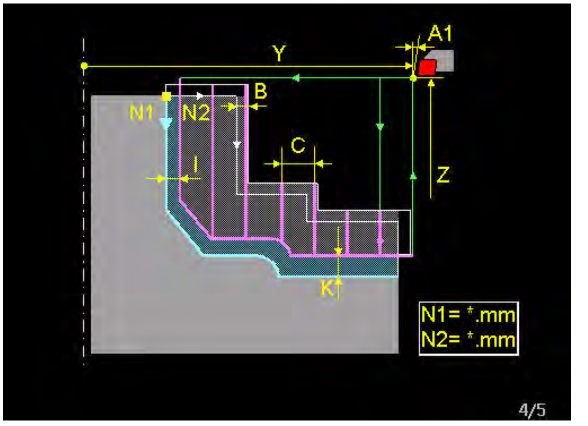


Contour description N1= (see picture)

- The contour starting point must be programmed with G1 Y Z in absolute coordinates.
- The contour description is created with the single G functions: G1 and G2/G3.
- The contour starting point and the contour direction are highlighted in the support picture.
- The contour direction is defined by the contour starting point up to the contour end point. If the contour direction was not described according to the cycle machining direction, address I2 must be programmed equal to 1 (reverse contour direction).
- Plunging contour elements in the (-Z) and (-Y) direction are permitted.

Blank contour description N2= (see picture)

The contour profile has a cast or rough allowance for cast or pre-machined parts. With a blank contour around this allowance, all tool movements occur only in the allowance range with feed, to decrease the machining time:



Blank contour variants

1. The blank contour is derived from the contour profile with an allowance (B) with address B.
2. The blank contour is programmed with N2= and established in a macro (*.MM). The contour description occurs similarly to N1=, but the contour description N2= must be "closing", i.e. contour description N2= must start with starting point N1= and close either with end point N1= or starting point N1=. Plunging contours may be programmed, but are not executed in rapid movement.
3. Blank contour N2= has an allowance (B) with N2= and B.

Sequence

Rough machining

The contour machining cycle machines the work piece parallel to the axis from the complete material up to the programmed contour profile. The contour profile is established in a macro. If the contour profile is cast and has different cast thicknesses, a blank allowance can be placed over the contour profile. The blank shape can be derived from the contour profile or be programmed as a free shape. Machining is only done in the blank allowance region.

Finishing

When finishing in the flank direction (I1=1), the contour is machined lengthwise as usual with depth feed from the cycle starting point up to the contour starting point. The last cut is either executed to the contour profile or allowance value and occurs as described below (see picture).

1. The last cut occurs from the contour starting point in the contour direction up to the first increasing contour section.
2. At this point, a return is executed in rapid movement up to the starting point height and this continues to the contour endpoint.
3. From the contour endpoint, the contour continues to be cut in the turning centre direction up to the contour section, as discussed in point 1.
4. After the clearance movement, a return is executed in rapid movement to the starting point height and back to the cycle starting point.

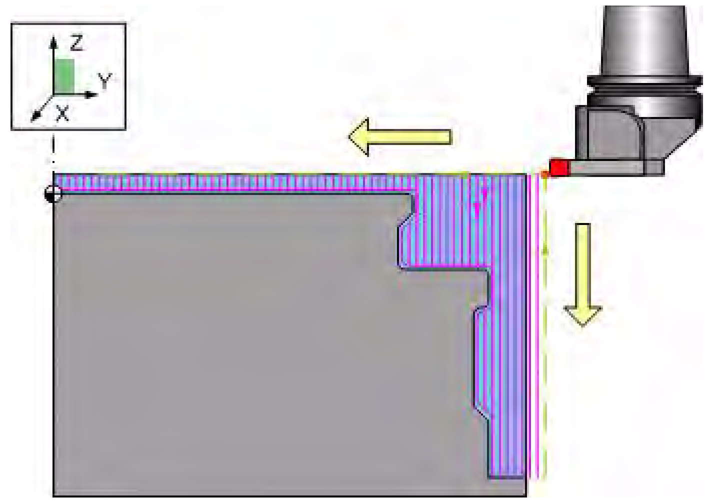
Example program, machining contour (parallel to axis)

Sequence (See drawing)

- From starting point Y/Z, feed by C (-Y direction) and first cut with feed in (Z direction) to the contour end point.
- Return to Y/Z in rapid movement
- Feed by C and next cut with feed to contour end point.

This procedure repeats itself up to the contour starting point. The plunging contour elements are not machined.

- Last cut from contour starting point along the contour to the first plunging contour element.
- Feed by C and machining in the contour shape. Last cut along the contour up to the second plunging contour element.
- Feed by C and machining in the contour shape. Last cut along the contour up to the contour end point. Rapid movement back to the starting point.



Note: machining contour parallels

If the value entered under C (feed depth) is increased by a value (distance between cycle starting point and contour starting point), machining is executed parallel to the contour instead of parallel to the axis. If different contour machining cycles with different allowances I and K are set one after the other, this results in machining parallel to the contour.

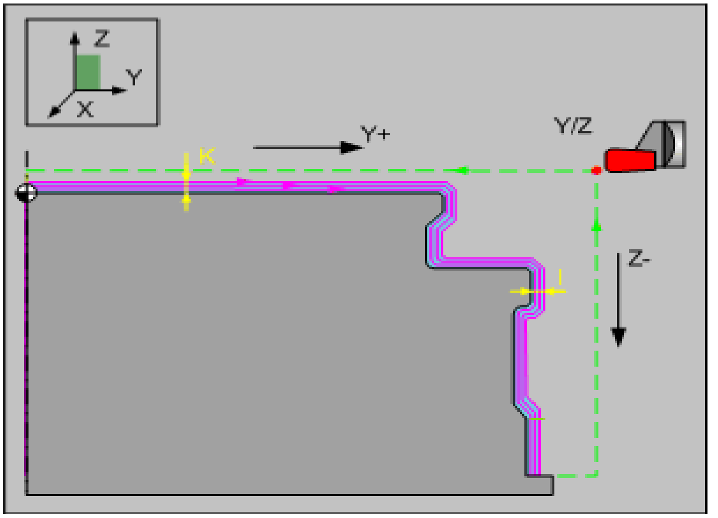
G36	Turning
G17 Y1=1 Z1=2	Turning level G17
G98 X0 Y0 Z100 I0 J50 K-250	Window definition graphic
G99 X0 Y0 Z0 I0 J125 K-100	Blank definition graphic
G0 Y150 Z50	Rapid movement position
T1 M67	Call up tool
G96 S1=200 M1=4 F0.15 D500	Table speed constant cutting speed
G880 N1=88001 Y130 Z5 C0.5 I0.5 K0.5S1=200 F0.15	Rough contour machining
G884 N1=88001 Y130 Z5 S1=300 F0.1	Finish contour machining
G0 Y150 Z50	Rapid movement position
G97 M1=5 S1=0	End constant cutting speed
G37	Milling

Example program, machining contour (parallel to contour)

Sequence (see picture)

- From starting point Y/Z, in rapid movement to contour starting point
- With feed along the contour with allowance I/K to contour endpoint
- Return to starting point Y/Z with rapid movement

Repeat this procedure with adapted allowance I/K



G0 Y150 Z200	Rapid movement position
G36	Turning
G17 Y1=1 Z1=2	Turning level G17
G98 X0 Y0 Z100 I0 J50 K-250	Window definition graphic
G99 X0 Y0 Z0 I0 J125 K-100	Blank definition graphic
G0 Y150 Z50	Rapid movement position
T1 M67	Call up tool
G96 S1=200 M1=4 F0.15 D500	Table speed constant cutting speed
G880 N1=88001 Y130 Z5 C120 I1 K1 S1=200 F0.15	Rough contour machining (I1 / K1)
G880 N1=88001 Y130 Z5 C120 I0.5 K0.5 S1=200 F0.15	Rough contour machining (I0.5 / K0.5)
G884 N1=88001 Y130 Z5 S1=300 F0.1	Finish contour machining
G0 Y150 Z50	Rapid movement position
G97 M1=5 S1=0	End constant cutting speed
G37	Milling

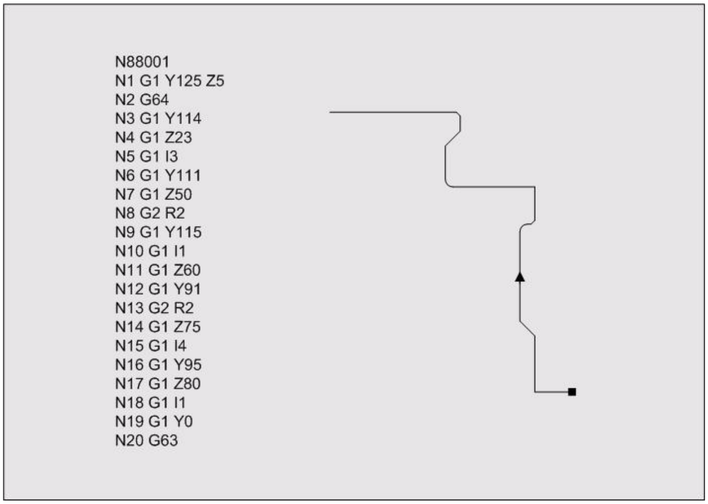
Example program, reverse contour direction

Contour programming

The contour is accepted from the tool drawing. If the contour direction was not described according to the cycle machining direction, address I2=1 must be programmed for the cycle (reverse contour direction)

Example: N88001.mm (ICP contour macro)

In contour macro 88001.mm, the contour of the neighbouring drawing is programmed.



Cycle programming

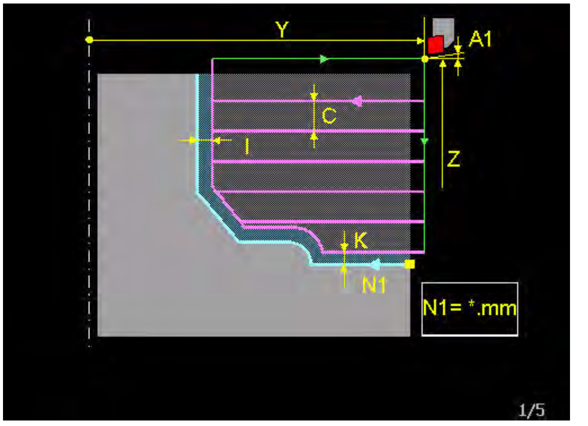
Because the contour direction of contour macro 88001.mm was programmed in the opposite direction with regard to the direction required by the cycle, address I2=1 must be programmed with the cycle (reverse contour direction).

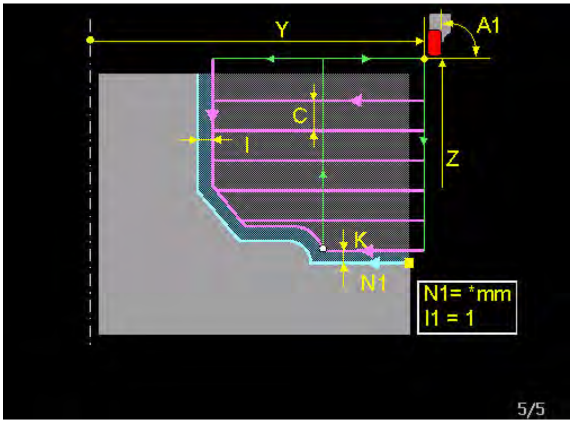
Sequence

The cycle sequence occurs in the same manner as with the example, which is contour machining parallel to the axis.

`G0 Y150 Z200`	Rapid movement position
`G36`	Turning
`G17 Y1=1 Z1=2`	Turning level G17
`G98 X0 Y0 Z100 I0 J50 K-250`	Window definition graphic
`G99 X0 Y0 Z0 I0 J125 K-100`	Blank definition graphic
`G0 Y150 Z100`	Rapid movement position
`T1 M67`	Call up tool
`G96 S1=200 M1=4  F0.15 D500`	Table speed constant cutting speed
`G880 N1=88001 Y130 Z90 C0.5 I2=1 I0.5 K0.5S1=200 F0.15`	Rough machine contour with reverse contour direction
`G884 N1=88001 Y130 Z90 I2=1 S1=300 F0.1`	Finish machine contour with reverse contour direction
`G0 Y150 Z100`	Rapid movement position
`G97 M1=5 S1=0`	End constant cutting speed
`G37`	Milling

3.47 G881 Contour cutting radial





G	Contour cutting radial
Y	Starting point
Z	Starting point
C	Cutting depth
N1=	Contour macro
I1=	Finish. 0=cont.dir. 1=downwards
I2=	Reverse contour dir. 0=no 1=yes
N2=	Raw contour macro
B	Raw material allowance
A1=	Clearance angle
I	Finishing allowance
K	Finishing allowance
S1=	(Cutting) Speed
F	Feed

The contour planar machining cycle machines the work piece parallel to the axis from the complete material or from the blank allowance up to the programmed contour profile or finished allowance. The contour description is established in a macro.

Contour machining with grooving tools is executed with consideration of the tool width through both sides of the grooving tool.

Address description

- **Y, Z Starting point** Starting point for the contour machining cycle.
- **C Feed depth** Measure by which the tool is fed in each radial direction. The depth may not be a multiple of the feed depth
- **N1= Contour macro** Macro (*.MM) in which the contour description is saved.
- **I1= Finishing** Machining direction of the last cut: 0: contour direction, 1: flank direction.
- **I2= Reverse contour direction 0=no 1=yes** Reversal of the contour(s) if different than in the support picture.
- **N2= Blank contour macro** Macro (*.MM) in which the blank contour description is saved.
- **B Blank allowance** Allowance around the contour (N1=) or blank contour (N2=) (0 to 100 mm).
- **A1= Clearance angle** Clearance angle of the tool. (0 to 90°) .
- **I, K Finishing allowance** Allowance in the Y and Z axis.

Default

I=0, K=0, A1=90 I1=0 I2=0

Application

See section "Notes and application G880" with the exception of:

Cycle starting point (Y/Z)

The cycle starting point must lie outside of the contour starting point. Note the tool orientation, dependent on the machining direction.

If necessary, it is permitted to place the contour endpoint for the Z axis below or above the contour starting point.

Error messages: (dependent on machining direction)

P362 tool with wrong orientation:Starting point in Z is smaller/larger that contour starting point Z in the macro.

P363 Start point in material: Starting point in Y is smaller/larger than contour starting point Y in the macro.

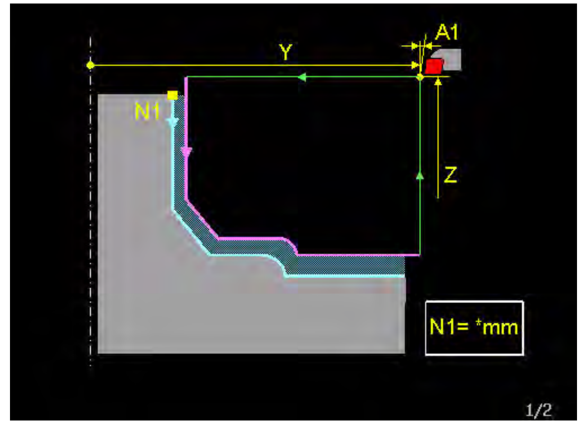
Tool orientation (O)

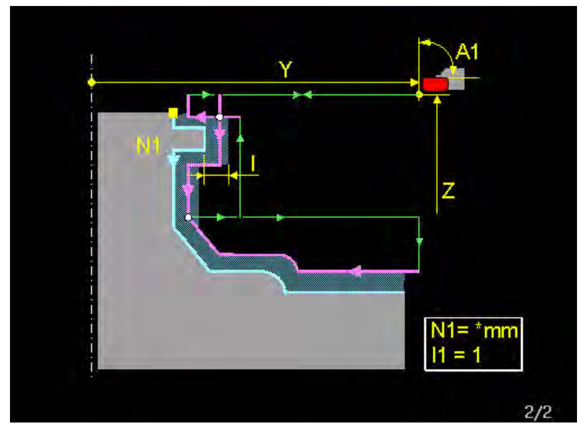
Make sure that the tool orientation (O) corresponds to the machining direction (-/+Y).

3.48 G884 Contour cutting axial finishing

The lengthwise contour machine cycle (finishing) finishes the work piece contour lengthwise. The contour description is established in a macro.

Finish machining with grooving tools is executed with consideration of the tool width through both sides of the grooving tool.





Address description

- **Y, Z Starting point** Starting point for the contour machining cycle.
- **N1= Contour macro** Macro (*.MM) in which the contour description is saved.
- **I1= Finishing** Machining direction of the last cut: 0: contour direction, 1: flank direction.
- **I2= Reverse contour direction 0=no 1=yes** Reversal of the contour(s) if different than in the support picture.
- **A1= Clearance angle** Clearance angle of the tool. (0 to 90°)
- **I allowance** The allowance value forms a safety distance over which the tool can freely move.

If I1 is programmed the same as 1 (finishing in flank direction), allowance I must also be programmed.

G	Contour cutting axial finishing
Y	Starting point
Z	Starting point
N1=	Contour macro
I1=	Finish. 0=cont.dir. 1=downwards
I2=	Reverse contour dir. 0=no 1=yes
A1=	Clearance angle
I	Allowance
S1=	(Cutting) Speed
F	Feed

Default

A1=90 I1=0 I2=0

Application

See section "Notes and application G880" with the exception of:

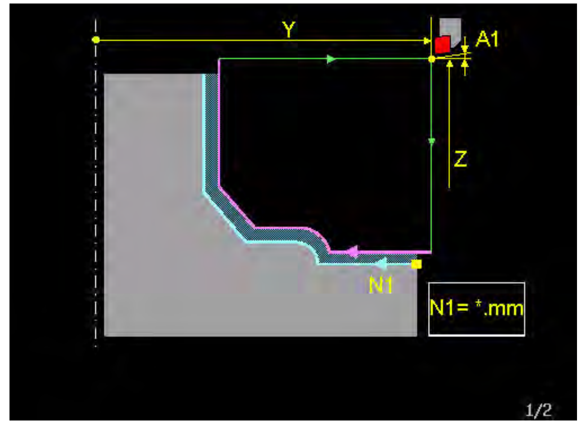
Feed (F)

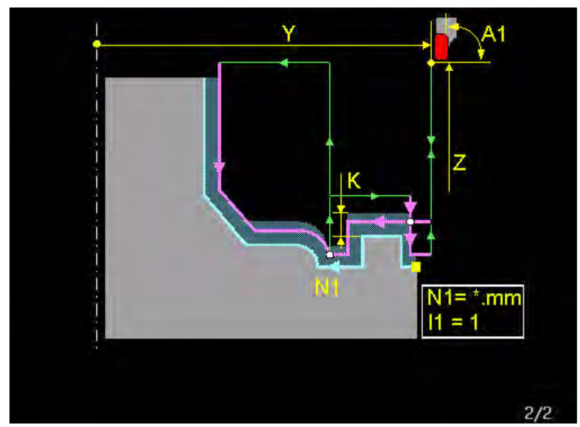
If a contour element with separate feed must be produced, this is programmed with a separate feed (F6=) in the corresponding contour element of the macro.

3.49 G885 Contour cutting radial finishing

The contour planar machining cycle finishes the work piece contour in the planar direction. The contour description is established in a macro.

Finish machining with grooving tools is executed with consideration of the tool width through both sides of the grooving tool.






```
G    Contour cutting radial finishing
Y    Starting point
Z    Starting point
N1=  Contour macro
I1=  Finish. 0=cont.dir. 1=downwards
I2=  Reverse contour dir. 0=no 1=yes
A1=  Clearance angle
K    Allowance
S1=  (Cutting) Speed
F    Feed
```

Address description

- **Y, Z Starting point** Starting point for the contour machining cycle.
- **N1= Contour macro** Macro (*.MM) in which the contour description is saved.
- **I1= Finishing** Machining direction of the last cut: 0: contour direction, 1: flank direction.
- **I2= Reverse contour direction 0=no 1=yes** Reversal of the contour(s) if different than in the support picture.
- **A1= Clearance angle** Clearance angle of the tool. (0 to 90°)
- **K allowance** The allowance value forms a safety distance over which the tool can freely move.

If I1 is programmed the same as 1 (finishing in the flank direction), allowance K must also be programmed.

Default

A1=90 I1=0 I2=0

Application

See section "Notes and application G880" with the exception of:

Feed (F)

If a contour element with separate feed must be produced, this is programmed with a separate feed (F6=) in the corresponding contour element of the macro.

SYMBOLE

W