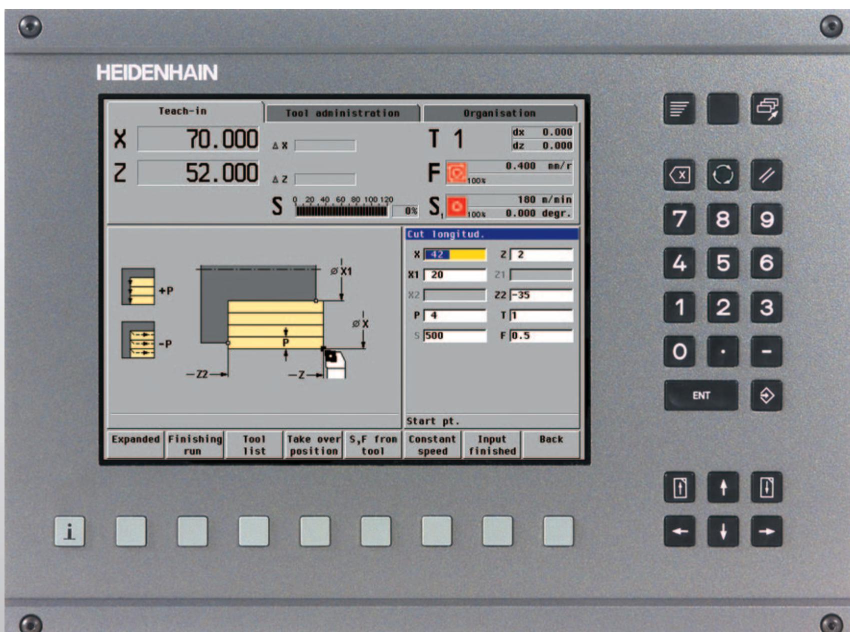
HEIDENHAIN



User's Manual

MANUALplus 4110

NC Software
526 488-xx

English (en)
9/2007

MANUALplus 4110, Software and Functions

This manual describes functions that are available in MANUALplus 4110 controls with NC software numbers 507 807-xx and 526 488-xx.

The machine manufacturer adapts the features offered by the control to the capabilities of the specific machine tool by setting machine parameters. Therefore, some of the functions described in this manual may not be among the features provided by the MANUALplus on your machine tool.

Some of the MANUALplus functions which are not available on every machine are:

- Positioning of spindle (M19) and driven tool
- Machining with the C Axis

Please contact your machine manufacturer for detailed information on the features that are supported by your machine tool.

Many machine manufacturers and HEIDENHAIN offer programming courses for the MANUALplus controls. We recommend these courses as an effective way of improving your programming skills and sharing information and ideas with other MANUALplus users.

HEIDENHAIN also offers the PC software DataPilot 4110 which is designed to simulate the functions of the MANUALplus 4110 control. The DataPilot is suitable for both shop-floor programming as well as off-location program creation and testing. It is also ideal for training purposes. The DataPilot can be run on WINDOWS operating systems.

Intended place of operation

The MANUALplus 4110 complies with EN 55022, Class A, and is intended primarily for operation in industrially zoned areas.

Contents

1 Introduction and Fundamentals 19

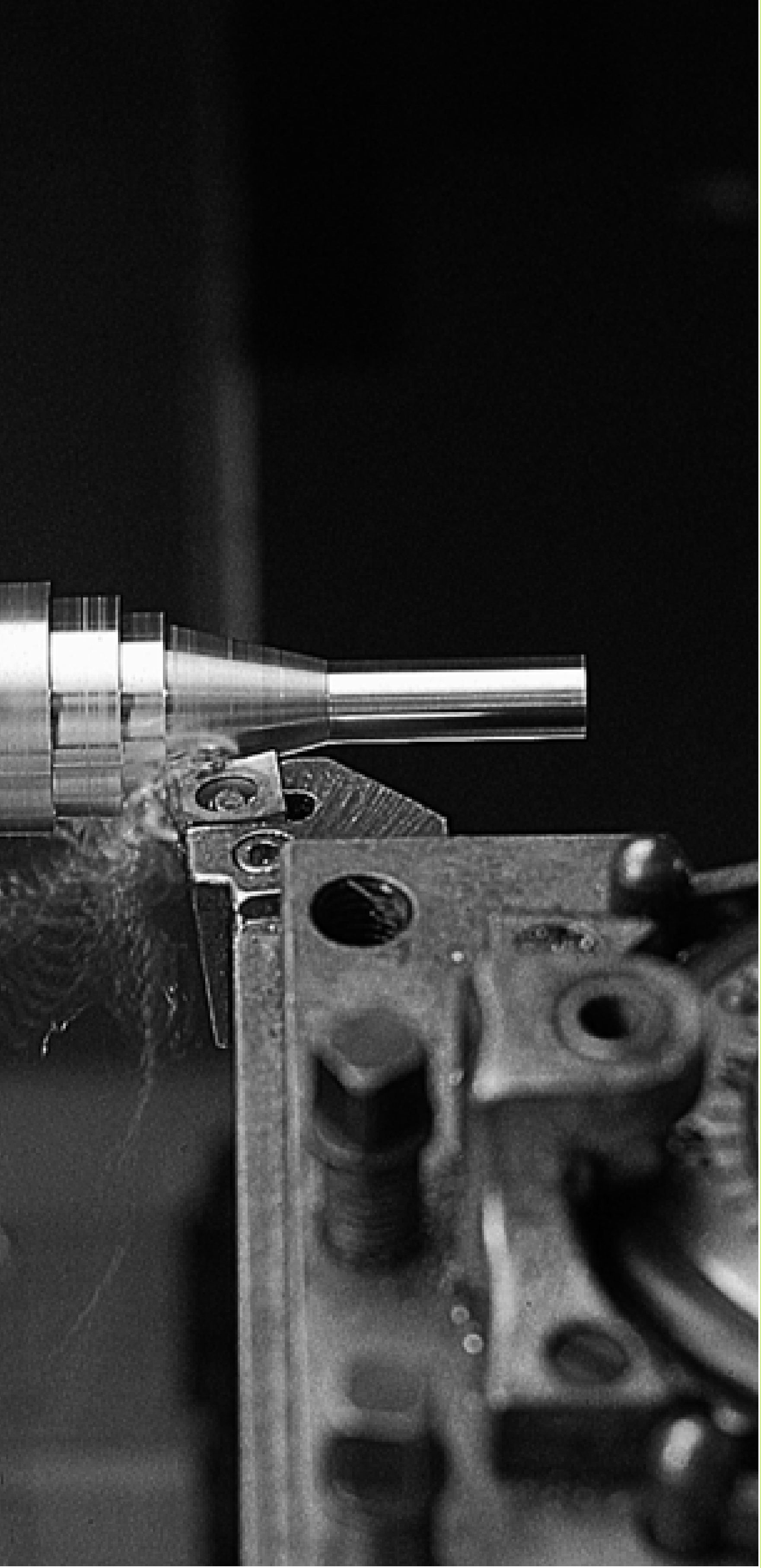
2 Basics of Operation 31

3 Machine Mode of Operation 41

5 ICP Programming 241

6 DIN Programming 277

7 Tool Management Mode 411



1

Introduction and Fundamentals

1.1 The MANUALplus

The MANUALplus control combines modern control and drive technology with the functional features of a hand-operated machine tool. You can run simple machining operations, such as turning or facing, on MANUALplus just like on any conventional lathe. The axes are moved as usual by handwheel or joystick. For machining difficult contours, such as tapers, radii, chamfers, undercuts or threads, MANUALplus offers fixed cycles. These cycles enable you to work faster and produce a higher quality than on a conventional lathe.

In addition, you can teach in a machining sequence and then have MANUALplus rerun the machining operation automatically as often as desired. Each additional part machined saves you time.

MANUALplus offers a wide range of capabilities: From performing simple lathe jobs through to complex workpiece contours, including drilling and milling operations on the face and lateral surface.

MANUALplus lets you choose between manual, semi-automatic and automatic operation. Regardless of whether you are machining a single part, producing a whole batch or repairing a workpiece, MANUALplus always gives you optimum support.

The C axis

With a C axis you can drill and mill a workpiece on its front, back and lateral surfaces.

During use of the C axis, one axis interpolates linearly or circularly with the spindle in the given working plane, while the third axis interpolates linearly.

MANUALplus supports cycle and DIN programming with the C axis.

1.2 Features

The functions of the MANUALplus are grouped into operating modes:

- **Machine mode of operation**
  This operating mode includes all functions for machine setup, workpiece machining, and cycle and DIN program definition.
  - The **cycle programming** functions are available in both manual and automatic modes. You can program cycles for roughing, recessing, thread-cutting and drilling operations.
  - **ICP programming** (**I**nteractive **C**ontour **P**rogramming) enables you to describe complex and even incomplete contours. You need to enter the values for the known elements, MANUALplus then automatically calculates the transitions, intersections, and any other missing data. MANUALplus graphically displays the contour sections entered and calculated. You can usually program a contour with the dimensions given in the workpiece drawing. ICP contour descriptions are included in the machining cycles.
  - The **DIN programming** feature (NC programming in DIN format according to DIN 66025 (ISO 6983)) enables you to run highly complex, technologically sophisticated machining operations. Apart from pure traversing commands, DIN cycles also provide functions for roughing, drilling and milling, for programming schematic contour geometry to calculate missing data, and for programming variables. You can even write separate DIN programs or integrate DIN macros in cycles.
  - **Before** executing a part program, you can run a **graphic simulation** of all machining operations that were programmed with cycles, cycle programs, or DIN programs.
- **Tool management mode**
  MANUALplus stores and manages up to 99 tool definitions. MANUALplus stores all of the tool data required for calculating cutting radius compensation, proportioning of cuts, plunging angle, etc.

  With the tool data, MANUALplus also manages the data for tool life monitoring as well as the cutting data, feed rate and spindle speed.
- **Organization mode of operation**
  The behavior of the MANUALplus system is controlled by parameters. In the Organization mode, you set the parameters to adapt the MANUALplus to your situation.

  Furthermore, you can exchange and save cycles and DIN programs with other systems over a serial data line (PC, host computer, etc).

  This operating mode also provides diagnostic functions for commissioning and checking the system.

1.3 MANUALplus Design

The dialog between machinist and control takes place via:

- Screen
- Soft keys
- Data input keypad
- Machine operating panel

The entered data can be displayed and checked on the **screen.** With the **function keys** directly below the screen, you can select functions, capture position values, confirm entries, and a lot more.

With the **information key** (also found beneath the screen), you can call error and PLC information and activate the PLC diagnostic function.

The **data input keyboard** (operating panel) serves for the input of machine data, positioning data, etc. The MANUALplus does not need an alphanumeric keyboard. Tool descriptions, program descriptions or comments in a DIN program are entered with an on-screen alphanumeric keyboard.

The **machine operating panel** contains all necessary controls for manual operation of the lathe.

The actual control is not accessible to the machinist. You should know, however, that your MANUALplus has an integrated hard disk on which all cycle programs, ICP contours and DIN programs that you enter are stored. This allows you to save a vast number of programs.

For data exchange and data backup, you can use the **serial data interface (RS-232-C)** or the **Ethernet interface.**

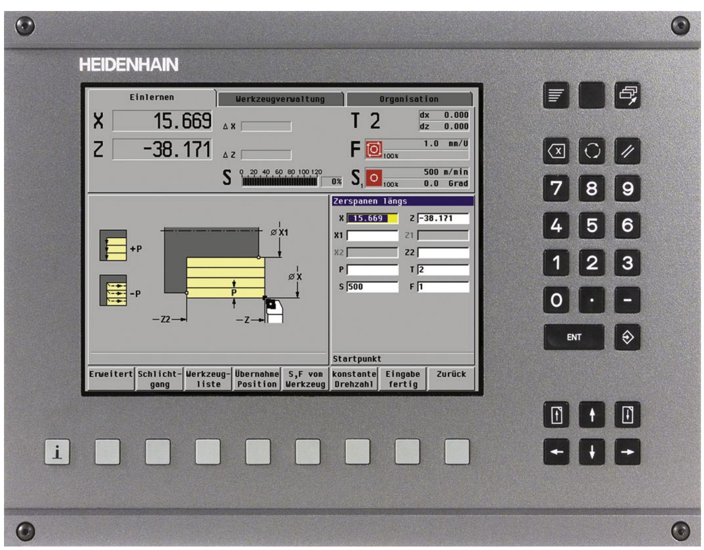
Lathe design

MANUALplus is configured by the machine manufacturer as a vertical boring and turning mill or to machine with tools "in front of" or "behind" the workpiece—depending on the design of the lathe or the position of the tool carrier. The menu symbols, the graphic support windows as well as the graphic representation during ICP and graphic simulation all reflect the configuration of the lathe.

The representations in this User's Manual assume a lathe with tool carrier in front of the workpiece.

Data input keypad	Symbol
Menu Call the main menu.	
Process Select a new mode of operation.	
Backspace Delete the character to the left of the cursor.	
Switching key Switch between help graphics for internal/external machining.	
Clear Delete error messages.	
Numbers (0 to 9) For entering values and selecting soft keys.	0 9
Decimal point	•
Minus Enter the algebraic sign.	–

Data input keypad	Symbol
ENTER Confirm the entered value.	ENT
Store Conclude data input and transfer values.	
Arrow keys Move the cursor in the indicated direction by one position (character, field, line, etc.).	
Page up, Page down (PgUp/PgDn) Show the information of the previous/ next screen page; toggle between two input windows.	
Info Call the error information or PLC status display.	i



Machine operating panel

The machine operating panel is interfaced to the lathe by the machine tool builder. The controls on your machine may deviate slightly from those shown in the illustration. Your machine documentation provides more detailed information.

Controls and displays

1 **Handwheel resolution**
Set the handwheel resolution to 1/10 mm, 1/100 mm or 1/1000 mm per graduation mark—or to other resolutions defined by the machine tool builder.

2 **Handwheel superposition in thread cycles**
Set the handwheel to "superposition for thread cycles."

3 **X handwheel**
Position the cross slide (cross slide axis = X axis).

4 **Feed-rate override**
Change the programmed feed rate.

5 **Speed override**
Change the preset speed.

6 **EMERGENCY STOP button**

7 **Z handwheel**
Position the saddle (saddle axis = Z axis).

8 **Tool change**
Confirm a tool change.

9 **Coolant ON/OFF**
Enable/disable coolant supply.

10 **Joystick**
Move the slide on a linear path at feed rate or rapid traverse; with a built-in switch for enabling rapid traverse.

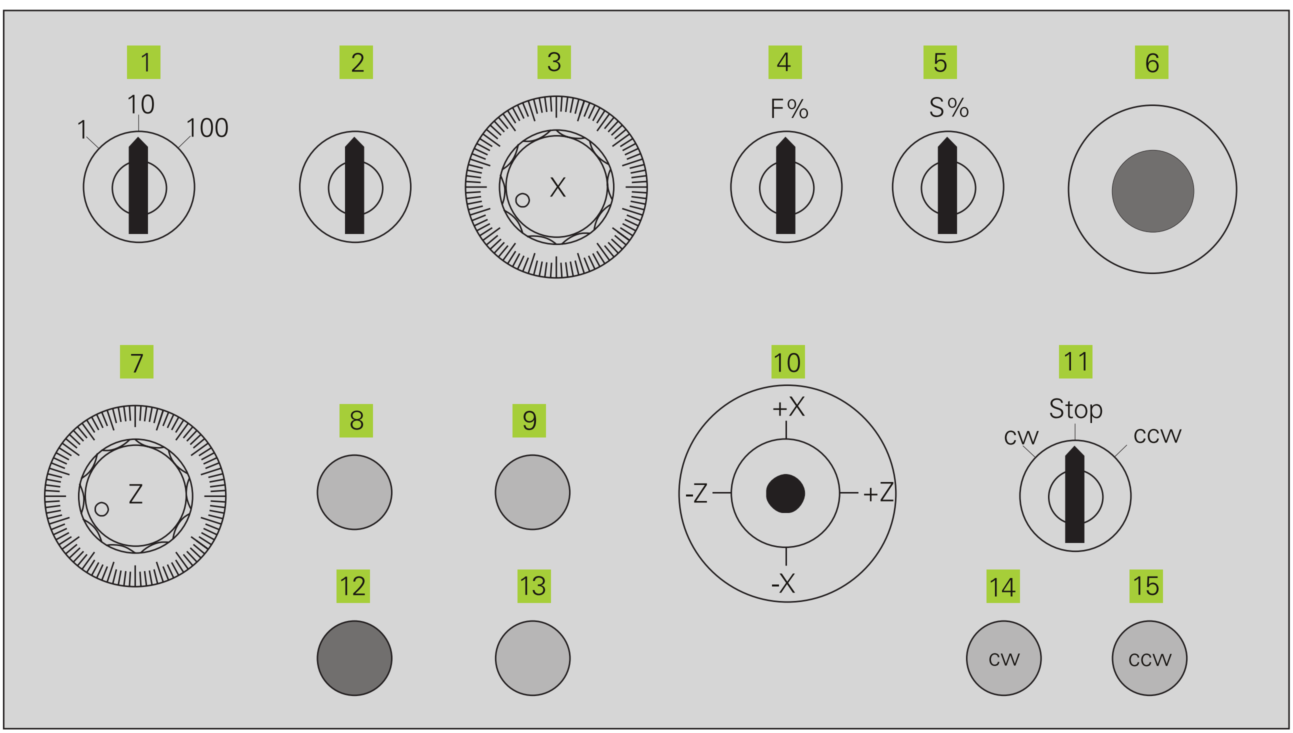
11 **Spindle switch**
Switch spindle to clockwise rotation (cw), counterclockwise rotation (ccw), or spindle stop (M05).

12 **Cycle STOP**
Stop traverse and cycle execution (the spindle remains ON).

13 **Cycle START**
Start a cycle, cycle program or NC program.

14 **Spindle jog cw**
Slowly rotate the spindle clockwise (cw).

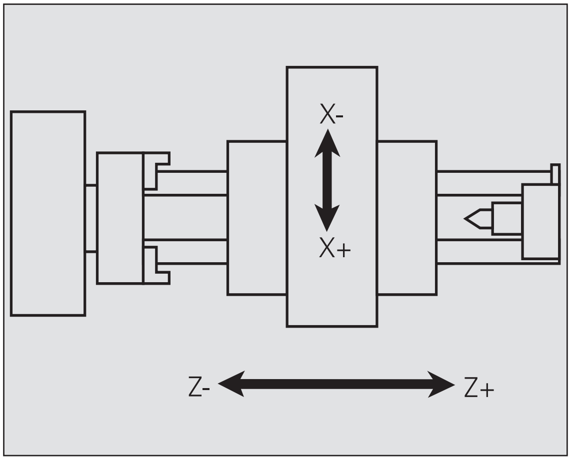
15 **Spindle jog ccw**
Slowly rotate the spindle counter-clockwise (ccw).



1.4 Axis Designations and Coordinate System

Axis designations

The cross slide is referred to as the **X axis** and the saddle as the **Z axis** (see figure at top right).



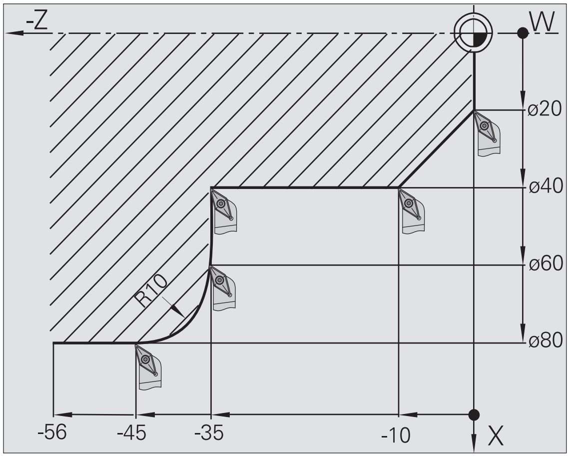
All X-axis values that are displayed or entered are regarded as **diameters.**

When programming paths of traverse, remember to:

- Program a **positive value** to depart the workpiece.
- Program a **negative value** to approach the workpiece.

Coordinate system

The axis designations X and Z describe positions in a two-dimensional coordinate system. As you can see from the figure to the center right, the position of the tool tip is clearly defined by its X and Z coordinates.

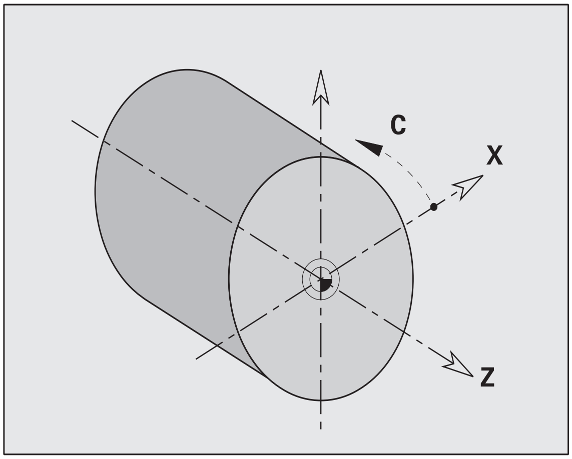


MANUALplus can connect points by linear and circular paths of traverse (interpolations). Workpiece machining is programmed by entering the coordinates for a succession of points and connecting the points by linear or circular paths of traverse.

Like the paths of traverse, you can also describe the complete contour of a workpiece by defining single points through their coordinates and connecting them by linear or circular paths of traverse.

The coordinates entered for the **X axis** and **Z axis** are referenced to the workpiece zero point.

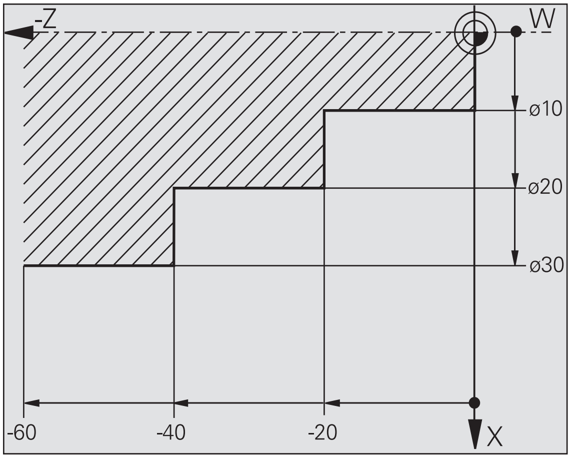
Angles entered for the **C axis** are referenced to the zero point of the C axis (see bottom-right figure).



Positions can be programmed to an accuracy of 1 µm (0.001 mm). This is also the accuracy with which they are displayed.

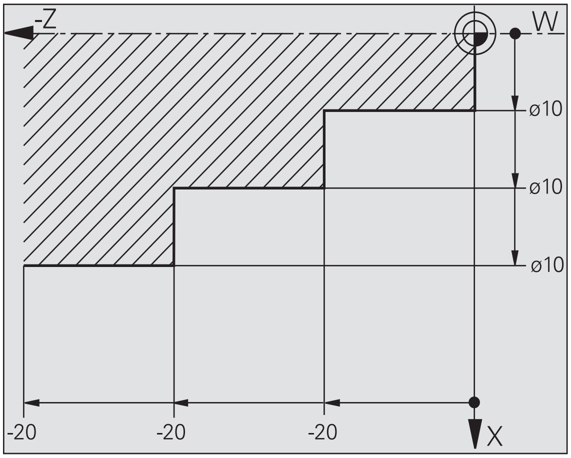
Absolute coordinates

If the coordinates of a position are referenced to the workpiece zero point, they are referred to as absolute coordinates. Each position on a workpiece is clearly defined by its absolute coordinates (see figure at upper right).



Incremental coordinates

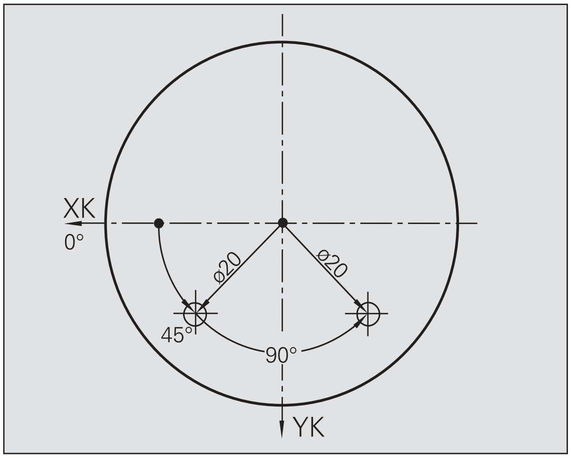
Incremental coordinates are always referenced to the last programmed position. They specify the distance from the last active position and the subsequent position. Each position on a workpiece is clearly defined by its incremental coordinates (see figure at center right).



Polar coordinates

Positions located on the face or lateral surface can either be entered in Cartesian coordinates or polar coordinates.

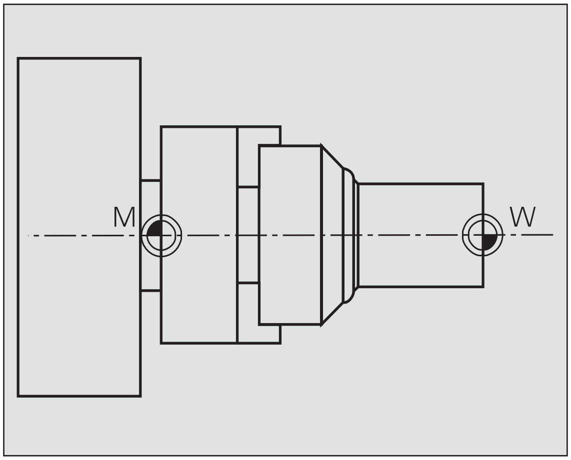
When programming with polar coordinates, a position on the workpiece is clearly defined by the entries for diameter and angle (see figure at bottom right).



1.5 Machine Reference Points

Machine zero point

The point of intersection of the X and Z axes is called the "machine zero point." On a lathe, the machine zero point is usually the point of intersection of the spindle axis and the spindle surface. The machine zero point is designated with the letter "M" (see figure at upper right).

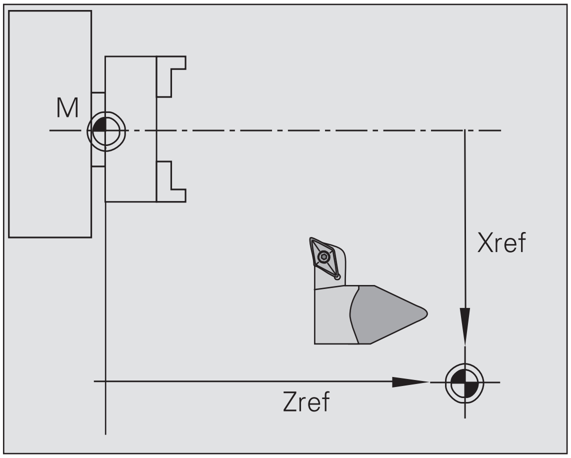


Workpiece zero point

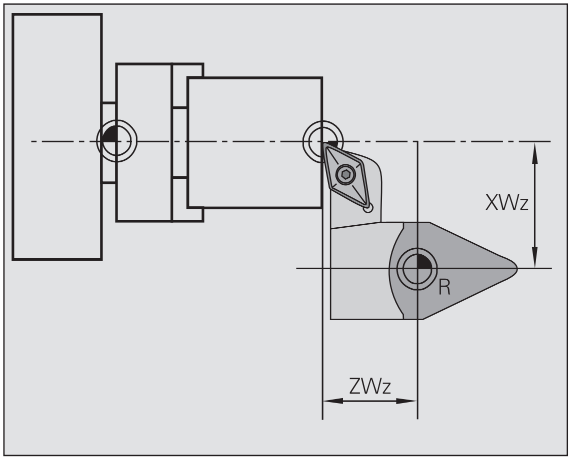
For machining a workpiece, it is easier to reference all input data to a zero point located on the workpiece. By programming the zero point used in the workpiece drawing, you can take the dimensions directly from the drawing, without further calculation. This point is the "workpiece zero point." The workpiece zero point is designated with the letter "W" (see figure at center right).

Reference points

Whether the control "forgets" the positions of the machine axes when it is switched off depends on the position encoders used. If the positions are lost, you must pass over the fixed reference points after switching on the MANUALplus. The control knows the exact distance between these reference marks and the machine zero point (see figure at lower right).



1.6 Tool Dimensions

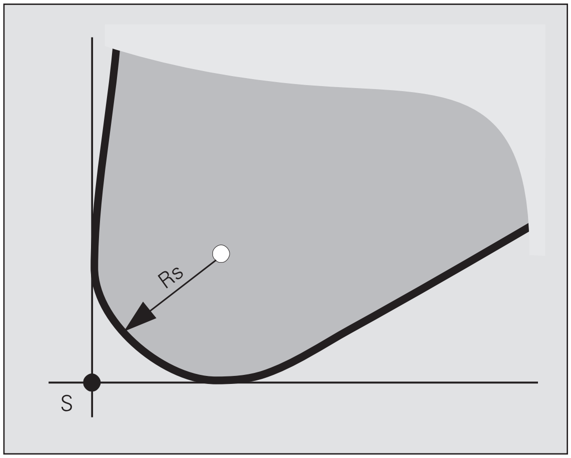


MANUALplus requires data on the specific tools for a variety of tasks, such as positioning the axes, calculating cutting radius compensation or proportioning of cuts.

Tool length

All position values that are programmed and displayed are referenced to the distance between the tool tip and workpiece zero point. Since the control only knows the absolute position of the tool carrier (slide), it needs the dimensions XWz and ZWz to calculate and display the position of the tool tip (see figure at upper right).

Tool compensation



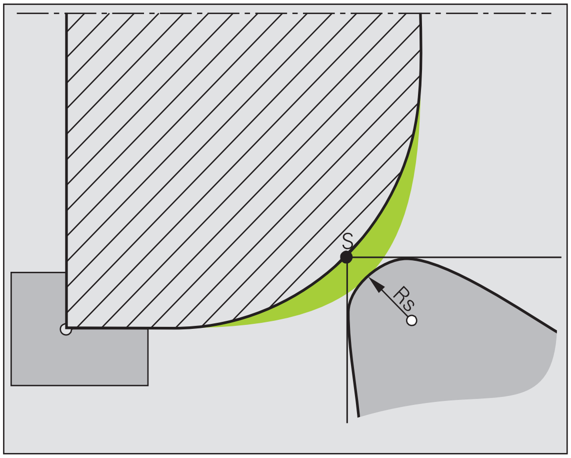
The tool tip is subjected to wear during machining processes. To compensate for this wear, MANUALplus uses compensation values which are managed independent of the values for length. The system automatically adds the compensation values to the values for length.

Tool-tip radius compensation (TRC)

The tip of a lathe tool has a certain radius. When machining tapers, chamfers and radii, this results in inaccuracies which MANUALplus compensates with its cutting radius compensation function.

Programmed paths of traverse are referenced to the theoretical tool tip S (see figure at center right). With non-paraxial contours, this will lead to inaccuracies during machining.

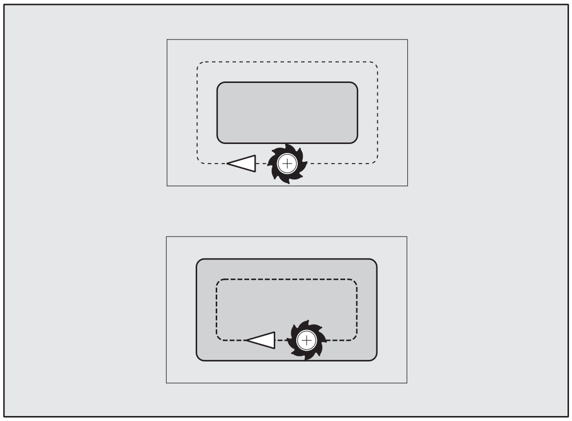
The TRC function compensates this error by calculating a new path of traverse, the **equidistant line** (see figure at bottom right).



MANUALplus calculates the TRC for cycle programming. The DIN programming feature also takes the TRC for clearance cycles into account. During DIN programming with single paths, you can also enable/disable TRC.

Milling cutter radius compensation (MCRC)

In milling operations, the outside diameter of the milling cutter determines the contour. When the MCRC function is not active, the system defines the center of the cutter as reference point. The MCRC function compensates for this error by calculating a new path of traverse, the **equidistant line.**



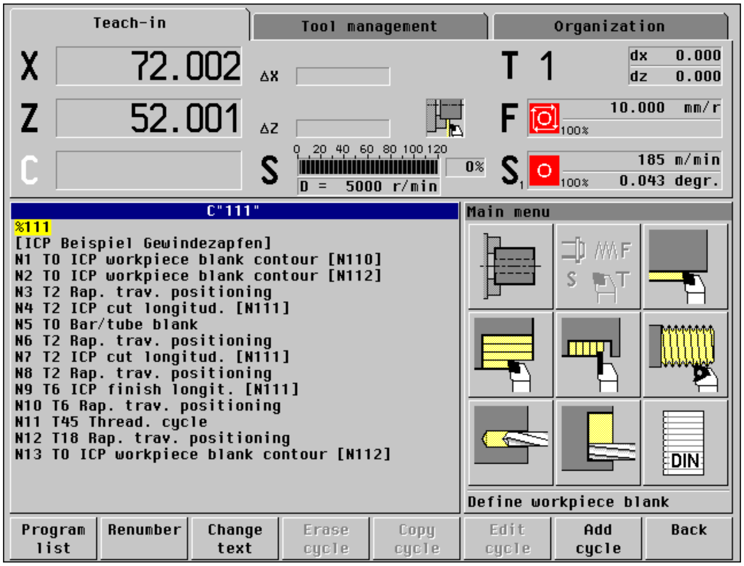
2

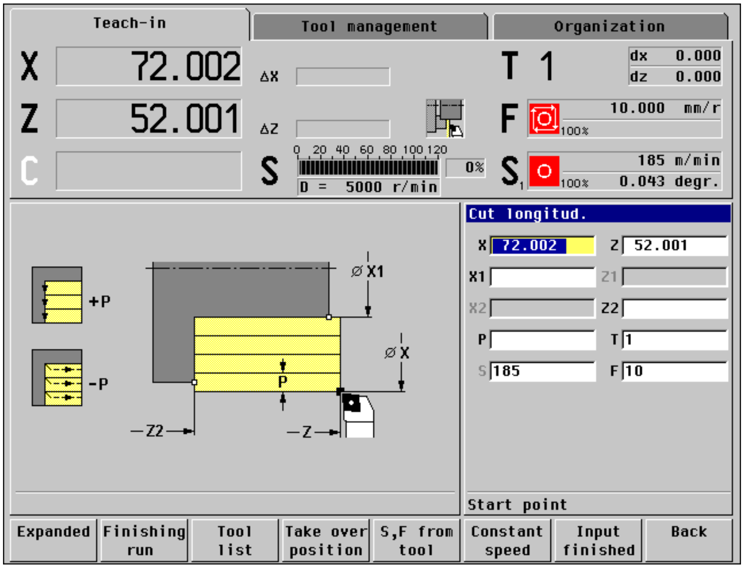
Basics of Operation

2.1 The MANUALplus Screen

MANUALplus shows the data to be displayed in **windows.** Some windows only appear when they are needed, for example, for typing in entries.

In addition, MANUALplus shows the **type of operation** and the **soft-key display** on the screen. Each function that appears in a field of the soft-key row is activated by pressing the soft key directly below it.





Screen windows displayed

- **Machine window**
  Position display, display of machine data, machine status, etc.
- **List and program window**
  Display of program lists, tool lists, parameter lists, etc. To select specific elements from the list, simply move the highlight to the desired element with the arrow keys.
- **Menu window**
  Display of menu symbols. This window only appears on the screen when menu selection is active.
- **Input box**
  For entering the parameters of a cycle, ICP element, DIN command, etc. You can enter data, check already programmed data, and edit and delete data as required. This window is also used to display data.
- **Graphic support window**
  Input data (such as cycle parameters, tool data, etc.) are explained with graphics. The Circle key allows you to switch between the help graphics for internal and external machining.
- **Simulation window**
  The simulation window shows a graphic representation of the contour elements and a simulation of the tool movements. This enables you to check cycles, entire cycle programs, and DIN programs.
- **ICP contour graphics**
  Display of the contour during ICP programming.
- **DIN editing window**
  Display of the DIN program during DIN programming. It is superimposed on the "machine window."
- **Error window**
  Display of encountered errors and warnings.

2.2 Operation and Data Input

Modes of operation

The active mode of operation is highlighted. MANUALplus differentiates between the following operating modes:

- Machine—with the submodes:
  - Manual mode (display: "Machine")
  - Teach-in
  - Program run
- Tool administration (tool management)
- Organization

You can switch between the different operating modes using the **Process key.** Press the Process key once to activate the operating-mode bar. Select the desired mode of operation using the arrow keys and press the Process key again to activate it.

> The Process key can only be used when the main menu of the current operating mode is active. You reach the main menu with ***Back*** or with the "Menu" key.

Menu selection

The numerical keypad is used for activating a menu and for entering data. The menu items are presented as a 9-field box. Each field of this symbol corresponds to the numerical key that is located at the same position on the numerical keypad. The functions, cycles, tools, etc. are displayed as symbols. The meaning of the selected symbol / menu item is described in the footer.

Press the corresponding numerical key, or move the highlight with the arrow keys to the symbol on the screen and press the ENTER key.

Soft keys

- With some system functions, the available functions are arranged on several soft-key levels.
- Some soft keys work like "toggle switches." A function is active when the associated field in the soft-key row is highlighted in color. The setting remains in effect until the function is disabled again.
- With functions like ***Take over position*** you do not have to enter values manually. The data are automatically written into the appropriate input fields.
- Data entries are not concluded until the ***Save*** or ***Input finished*** soft key has been pressed.
- The ***Back*** soft key takes you back to the previous operating level.

Data input

Input windows comprise several **input fields.** You can move the cursor to the desired input field with the vertical arrow keys. The function of the selected field is shown in the bottom line of the window.

Place the highlight on the desired input field and enter the data. Existing data are overwritten. With the horizontal arrow keys, you can move the cursor **within** the input field and place it on the position where you want to delete, copy or add characters.

To confirm the data you entered in a field, press a vertical arrow key or the ENTER key.

If there are more input fields than a window can show, a second input window is used. You will recognize this through the symbol in the bottom line of the input window. To switch back and forth between the windows, press the PgUp/PgDn keys.

Data entry is concluded when you press ***Input finished*** or ***Save.***—If you press the ***Back*** soft key, entries or changes will be lost.

List operations

Cycle programs, DIN programs, tool lists, etc. are displayed as lists. You can scroll through a list with the arrow keys to check data or to highlight elements for operations like deleting, copying, editing, etc.

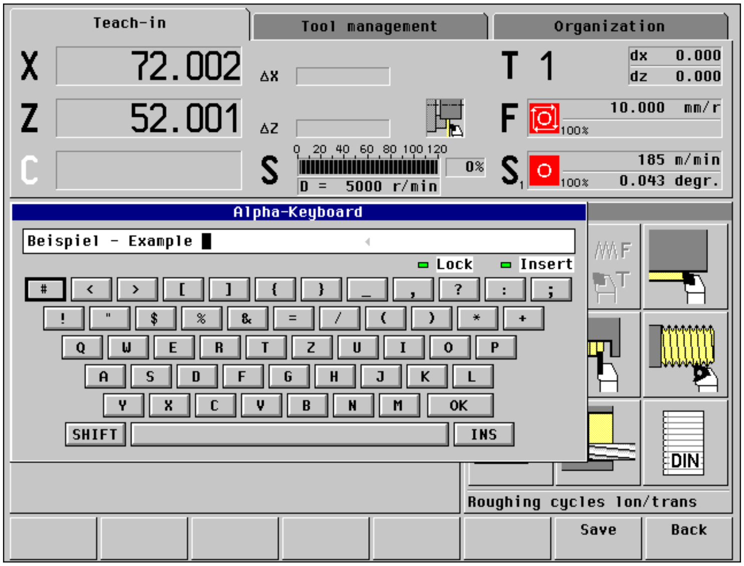
Alphanumeric keyboard

Program descriptions, tool descriptions, comments, etc. are entered with the on-screen alphanumeric keyboard. You select the desired character with the arrow keys and confirm the character with ENTER. You can switch between upper and lower case letters with the SHIFT button.

To edit existing texts, place the cursor on the desired position: Press the Up arrow key repeatedly until the cursor reaches the input line. Then use the horizontal arrow keys to delete, overwrite or add to the text, as required.

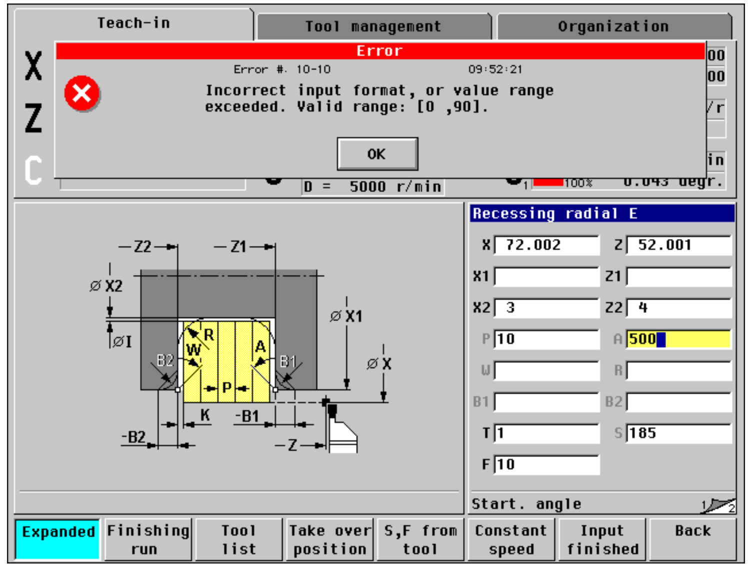
With the INS key (on the alphanumeric keyboard) you can determine whether to insert or overwrite characters. Which mode of the INS key is presently active (insert mode or overwrite mode) is indicated below the input line.

Numbers are entered with the data input keypad.



2.3 Error Messages

The appearance and effect of a MANUALplus error message depend on the current operation.

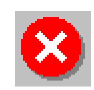


Direct error messages

The MANUALplus uses direct error messages whenever immediate error correction is possible and advisable, for example if the input value of a cycle parameter exceeds the valid input range. Confirm the message with ENTER and correct the error (see figure to the upper right).

Information of direct error messages:

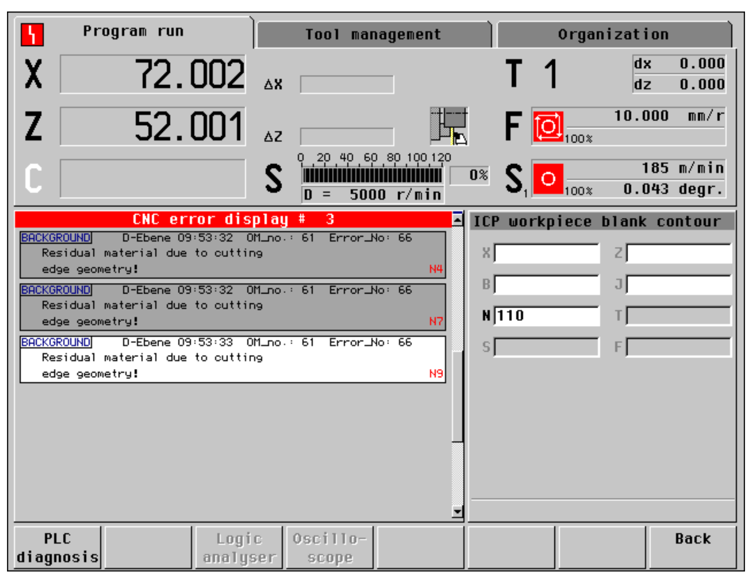
- The **error description** explains the error that has occurred.
- The **error number** is needed whenever you contact the machine manufacturer about a specific error message.
- The **time** shows you when the indicated error occurred.

Meaning of the symbols	
Warning: The program run / operation continues. MANUALplus points out the problem.	
Error: The program run/operation is stopped. You must correct the error before being able to continue.	

Error display

The control temporarily stores any errors or messages that appear during system start, operation or program run, and sets the error symbol in the top line. Using the **Info** key, open the error window to view the messages.

If more error messages have occurred than can be shown in one screen page of the error window, you can scroll through the error display with the arrow keys and PgUp/PgDn to check all messages.



Clearing an error message

You can cancel the error message on which the cursor is located with the "Backspace" key, or cancel all of the error messages with the "Clear" key.

The error symbol remains set in the top line until all of the errors have been canceled.

You can exit the error window without clearing any error messages by pressing ***Back.***

Information in the error message:

- The **error description** explains the error that has occurred.
- The **error number,** level indication (D level, C level) and "OM no." are needed whenever the supplier needs to be contacted.
- The **time** shows you when the indicated error occurred.
- The **error class** is indicated in the framed field (to the top left of the message). A message without this field represents a warning.
  - **Background:** This message serves as information, or merely a "small" error has occurred.
  - **Abort:** The current operation (execution of a cycle, traverse command, etc.) was aborted. You can resume operation once the error has been cleared.
  - **Emergency stop:** An error condition has caused all traverse to be stopped and the abortion of cycle program and DIN program execution. You can resume operation once the error has been cleared.
  - **Reset:** An error condition has caused all traverse to be stopped and the abortion of cycle program and DIN program execution. Switch off the control for a moment, then restart. Contact your machine manufacturer if the error occurs again.

System error, internal error

In the unlikely event that a system error or an internal error occurs, write down all information on the displayed message and inform your machine manufacturer. You cannot correct these errors. Switch off the control and restart.

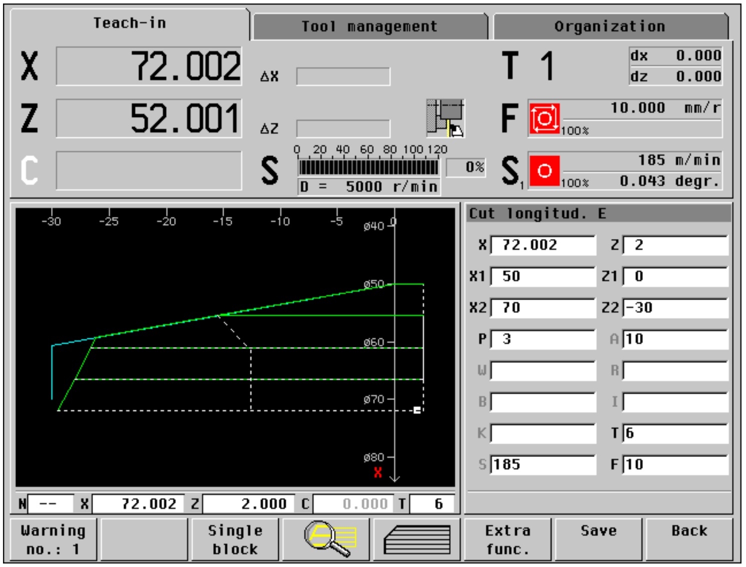
PLC error, PLC status display

Using the soft keys ***PLC diagnosis*** and ***CNC diagnosis,*** you can switch between the error information and the PLC window.

The PLC window is used for PLC messages and the PLC diagnosis. Please refer to your machine manual for more information.

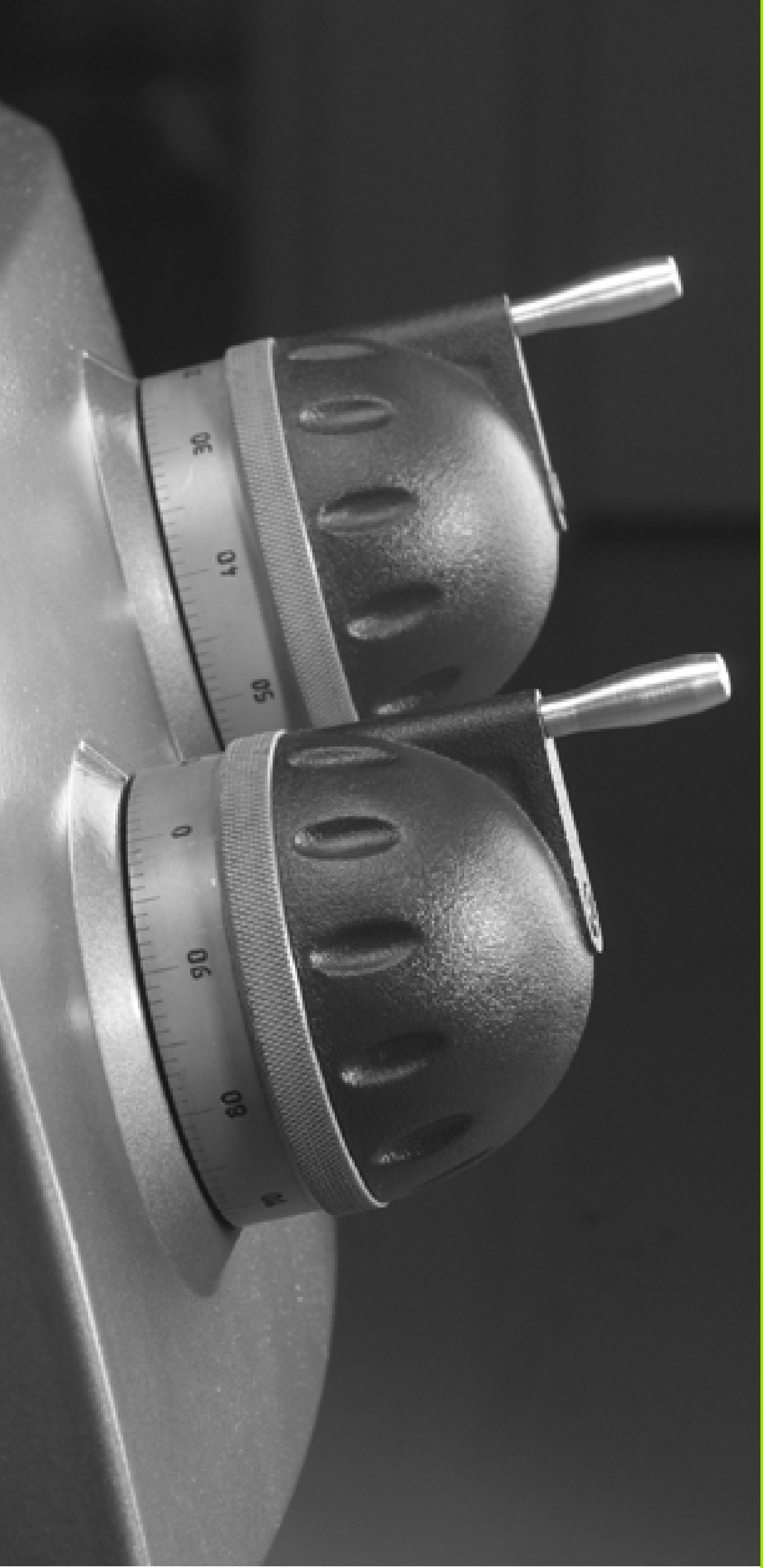
Warnings during simulation

If during simulation of a cycle, an entire cycle program or a DIN program MANUALplus detects problems, it displays a warning in the soft key to the extreme left (see figure to the lower right). Press the soft key to call these messages.



2.4 Explanation of Terms

- **Cursor:** In lists, or during data input, a list item, an input field or a character is highlighted. This "highlight" is called a **cursor.** Entries and operations, like copying, deleting, inserting a new item, etc., refer to the current cursor position.
- **Arrow keys:** The cursor is moved with the horizontal and vertical arrow keys and with the PgUp/PgDn keys.
- **Page keys:** The PgUp/PgDn keys are also called "Page keys."
- **Navigate:** Within a list or an input box, you can move the cursor to any position you would like to check, change, delete or add to. In other words, you "navigate" through the list.
- **Active/ inactive windows, functions, menu items:** Of all windows that are displayed on the screen, only one is **active.** That means, any data you type on the keyboard or keypad are entered in the active window only. In the active window the title bar is shown in color. In the inactive windows, the title bar appears dimmed. Inactive function keys or menu keys also appear dimmed.
- **Menu, menu key:** MANUALplus arranges the available functions and function groups in a 9-field box. This box is called a menu. Each symbol in the menu is a menu key.
- **Editing:** Editing is changing, deleting and adding to parameters, commands, etc. within programs, tool data or parameters.
- **Default value:** If the parameters of cycles or DIN commands are preassigned values, these values are referred to as default values. These values are used if you do not enter the parameters.
- **Byte:** The capacity of a storage disk is indicated in bytes. Since MANUALplus features a hard disk, the individual program lengths are expressed in bytes.
- **Extension:** File names consist of the actual name and the extension. The name part and the extension part are separated by ".". The extension indicates the type of file. Examples:
  - „*.NC“DIN programs
  - „*.NCS“DIN subprograms (DIN macros)
  - „*.MAS“Machine parameters



3

Machine Mode of Operation

3.1 Machine Mode of Operation

The Machine mode of operation includes all functions for machine setup, workpiece machining, and cycle and DIN program definition.

- **Machine setup**
  For preparations like setting axis values (defining workpiece zero point), measuring tools or setting the protection zone.
- **Manual operation**
  Machine a workpiece manually or semi-automatically.
- **Teach-in**
  "Teach-in" a new cycle program, change an existing program, or graphically simulate cycles.
- **DIN programming**
  Creating, editing, deleting DIN programs.
- **Program run**
  Graphically simulate existing cycle programs or DIN programs and use them for the production of parts.

With MANUALplus, you produce a part in the usual manner by moving the axes with the handwheels and jog controls, just like on a conventional lathe. In most cases, however, it is much more convenient to use the cycles offered by MANUALplus.

A **cycle** is a machining step that has already been programmed for you. This can be any machining operation from a single cut through to a complex machining task like thread cutting. In any case, a cycle is always a complete machining step that is immediately executable once you have defined a few parameters that describe the workpiece to be machined.

In Manual mode, the cycles that you program are **not stored.** In Teach-in mode, each machining step is executed with a cycle and then stored and integrated into a complete **cycle program.** You can subsequently use this cycle program in parts production by repeating it as often as desired in the Program run mode.

In **ICP programming,** any contour can be defined using linear/circular elements and transition elements (chamfers, roundings, undercuts). The contour descriptions are included in ICP cycles (see “ICP Contours” on page 242).

The **DIN programming** feature provides commands for simple traversing movements, DIN cycles for complex machining tasks, switching functions, mathematical operations and programming with variables.

You can either create "independent" programs that already contain all necessary switching and traversing commands and are executed in the Program run mode, or program **DIN macros** that are integrated in cycles. The commands that you use in a DIN macro depend on the job at hand. DIN macros support the complete range of commands that is available for DIN programs.

You can also **convert** cycle programs to DIN programs. This enables you to make use of straightforward cycle programming, and then convert the part program to DIN format for subsequent optimization or completion.

3.2 Switch-On / Switch-Off

Switch-on

In the screen headline, MANUALplus displays the individual steps that are performed during system start. When the system has completed all tests and initializations, it switches to the Machine mode of operation. The tool display shows the tool that was last used. Whether a **reference run** is necessary depends on the encoders used.

If errors are encountered during system start, MANUALplus displays the **error symbol** on the screen. You can check these error messages as soon as the system is ready (see "Error Messages" on page 36).

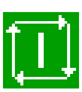
> After system start, MANUALplus assumes that the tool which was last used is still inserted in the tool holder. If this is not the case, you must inform MANUALplus of the tool change.

Traversing the reference marks

Reference run

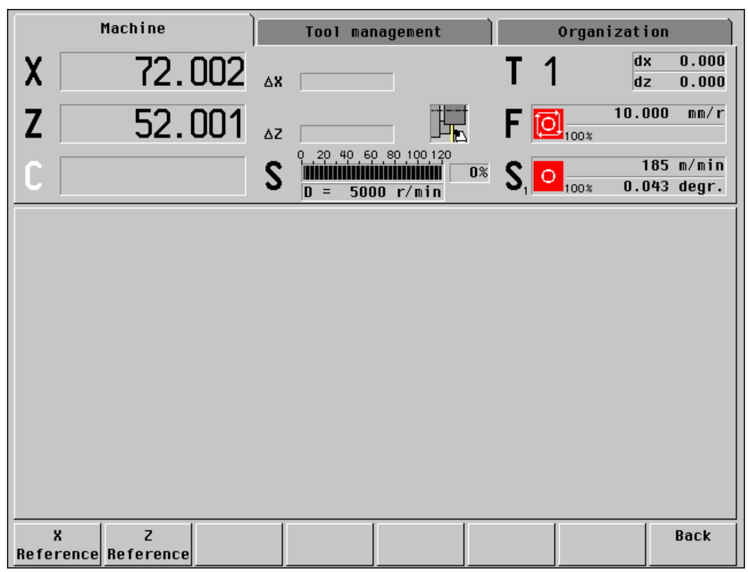
X Reference — Select ***X reference.***

Z Reference — Select ***Z reference.***



Press Cycle START for the control to traverse the reference marks.

MANUALplus activates the position display and switches the menu and the soft-key row to the main menu.



Whether a reference run is necessary depends on the encoders used:

- EnDat encoder: Reference run is not necessary.
- Distance-coded encoders: The position of the axes is ascertained after a short reference run.

- Standard encoder: The axes move to known, machine-based points. As soon as a reference mark is traversed, a signal is transmitted to the control. The control knows the distance between the reference mark and the machine zero point and can now establish the precise position of the axis.

- In case you traverse the reference marks separately for the X and Z axes, you only traverse in either the X or the Z axis.

Monitoring EnDat encoders

If EnDat encoders are used, the control saves the axis positions during switch-off. During switch-on, the MANUALplus compares for each axis the position during switch-on with the position saved during switch-off.

If there is a difference, one of the following messages appears:

- "Axis was moved after the machine was switched-off."
  Check the current position and confirm it if the axis was in fact moved.
- "Saved encoder position of the axis is invalid."
  This message is correct if the control has been switched on for the first time, or if the encoder or other control components involved were exchanged.
- "Parameters were changed. Saved encoder position of the axis is invalid."
  This message is correct if configuration parameters were changed.

The cause for one of the above listed messages can also be a defect in the encoder or control. Please contact your machine supplier if the problem recurs.

Switch-off

Proper switch-off is recorded in the error log file.

Switch-off

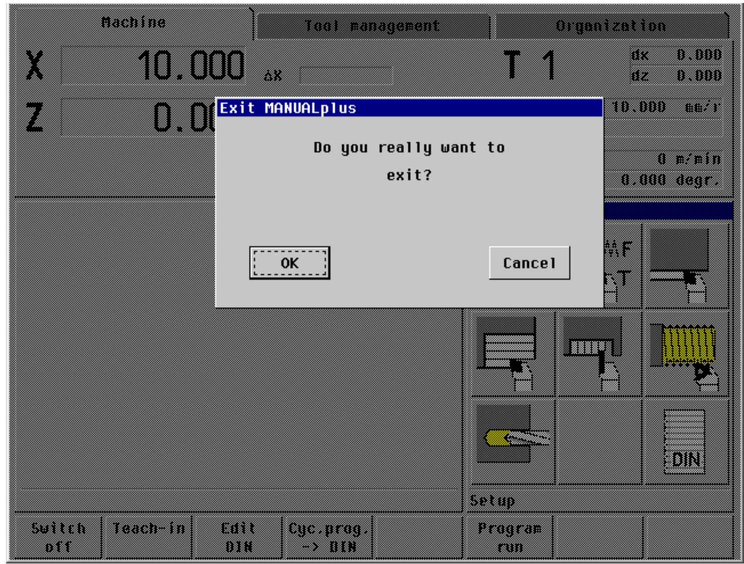
Go to the main level of the Machine mode of operation.

Switch off — Press the ***Switch off*** soft key.

MANUALplus displays a confirmation request.

ENT — Press ENTER to terminate the control.

Wait until MANUALplus requests you to switch off the machine.



3.3 Machine Data

Input and display of machine data

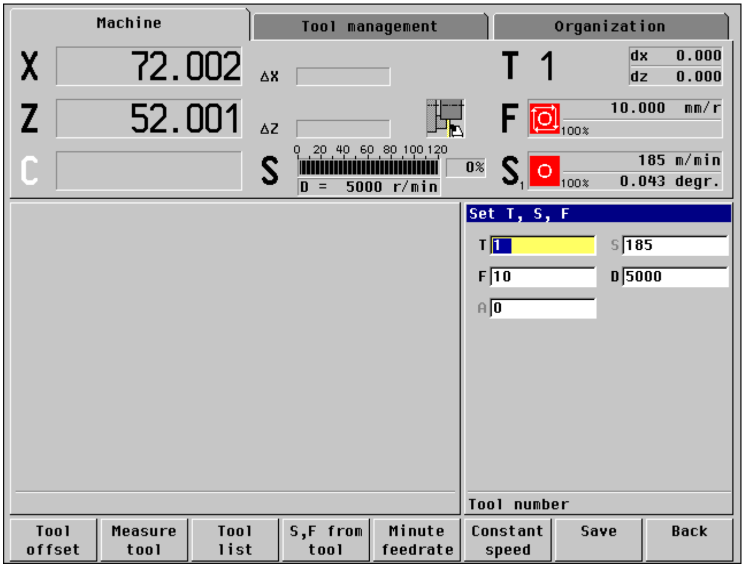
In Manual mode, the machine data for tool, spindle speed and feed rate are entered in **"Set T, S, F ."** In cycle programs the machine data are included in the cycle parameters, and in DIN programs they are part of the NC program.

In "Set T, S, F" you also define the "maximum speed" and the "stopping angle."

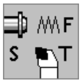
You can store the cutting data (spindle speed, feed rate) together with the tool data and transfer them with the ***S, F from tool*** soft key (see “Tool Data—Supplementary Parameters” on page 426).

Machine data display

The machine data display is configurable. The machine data that appear on your screen may therefore deviate from the example shown.



Entering the machine data



Select "Set T, S, F" (only available in Manual mode).

Define the parameters.

Save

Conclude data input.

Elements of machine data display

Position display X, Z: Distance between tool tip and workpiece zero point

- Letter designating the axis appears in white: Axis "disabled"

Position display C: Position of the C axis

- Empty box: C axis is not active
- Letter designating the axis appears in white: Axis "disabled"

Distance-to-go display X, Z, C: The distance remaining from the current position to the target position of the active traversing command

Distance-to-go Z and protection zone status: Distance-to-go display and display of status of protective zone monitoring

Spindle utilization: Utilization of the spindle motor relative to the rated torque

Spindle utilization and maximum speed: Utilization of the spindle motor and additional display of valid **maximum speed**

Elements of machine data display
T display ■ T number of the inserted tool ■ Tool compensation values ■ "T" highlighted in color: Machining of "mirrored contour" active
S display ■ Symbol of spindle status ■ Upper field: Programmed value ■ Lower field: Setting of override control and actual spindle speed—with position control (M19): spindle position ■ Gear range (figure beside "S") ■ "S" highlighted in color: Display applies to driven tool
F display ■ Symbol of cycle status ■ Upper field: Programmed value ■ Lower field: Setting of override control and actual feed rate

Tool call

T is the identification letter for the tool data. Depending on the tool carrier used, "T" is followed by 2 or 4 characters.

- **One tool holder** (e.g. Multifix): Call: "Tdd"
- **More than one tool holder** (e.g. turret): Call: „Tddpp“
- dd: Position in the tool file (tool list)
- pp: Position on the tool carrier (turret location)

In Manual mode, the T number is entered in "Set T, S, F"—in Teach-in mode, "T" is a cycle parameter.

Power-driven tools

- Driven tools are defined in the tool description.
- If the active tool is driven, the displayed spindle data refer to the tool.
- The following input parameters refer to spindle 1 when a driven tool is active:
  - Spindle speed / Constant cutting speed
  - Maximum speed
  - Feed per revolution in "Set T, S, F"

Soft keys for "Set T, S, F"	
Tool offset	See “Tool compensation” on page 58
Measure tool	See “Setting up Tools” on page 54
Tool list	Call the tool list—Transfer of T number from the tool list possible
S,F from tool	Transfer of spindle speed and feed rate from the tool data
Minute feedrate	**On:** Feed per minute (mm/min) **Off:** Feed per revolution (mm/rev)
Constant speed	**On:** Constant speed (rpm) **Off:** Constant cutting speed (m/min)

- If a driven tool is active, the spindle speed and speed limitation refer to the tool.
- Your machine documentation provides information on whether the driven tool can be operated with feed per revolution.

Tools with more than one cutting edge

If you use special tools with more than one cutting edge, different tool parameters apply (set-up dimensions, cutting radius, etc.). Enter more than one tool definition to define these tools. If "T" is programmed with four digits (Tddpp), program a new "dd" ("pp" remains the same) when another cutting edge of the special tool is used.

Tools in different quadrants

Example: The principal tool carrier of your lathe is in front of the workpiece (standard quadrant). An additional tool holder is behind the workpiece.

When MANUALplus is configured, it is defined for each tool holder whether the X dimensions and the direction of rotation of circular arcs must be mirrored. In the above-mentioned example the additional tool holder is assigned the attribute "Mirrored."

If this method is used, all machining operations are programmed as usual—regardless of which tool holder executes the operation. The simulation also shows all machining operations in the standard quadrant.

The tools are also described and dimensioned for the standard quadrant—even if they are inserted in the additional tool holder.

Mirroring does not become effective until the machining of the workpiece, i.e. when the additional tool holder is executing the machining operation.

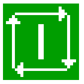
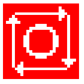
Feed rate

"F" is the identification letter for feed data. Depending on which mode of the ***Feed rate*** soft key is active, data is entered in:

- Millimeters per spindle revolution (feed per revolution)
- Millimeters per minute (feed per minute).

On the screen, you can tell the type of feed rate from the unit of measure in the input field.

You can change the feed value with the **feed compensation controller** (feed override) (range: 0% to 150%).

Feed symbols (F display)	Symbol
Status "Cycle ON" Cycle or program execution is active.	
Status "Cycle OFF" Cycle or program execution is not active.	

Spindle

"S" is the identification letter for spindle data. Depending on which mode of the ***Constant speed*** soft key is active, data is entered in:

- Revolutions per minute (constant speed)
- Meters per minute (constant cutting speed).

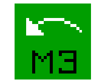
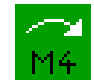
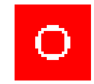
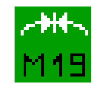
The input range is limited by the maximum spindle speed. You define the speed limitation in "Set T, S, F", in machine parameters 805/855, or in DIN programming with the G26 command.

The speed limit remains in effect until a new speed limit value is programmed.

The speed compensation controller (speed override) allows you to change the spindle speed (range: 50% to 150%).

The subscript number after the identification letter "S" indicates the **gear range.**

- If you are machining with a constant cutting speed, MANUALplus calculates the spindle speed from the position of the tool tip. The smaller the diameter of the tip, the higher the spindle speed. The maximum spindle speed, however, is never exceeded.
- The spindle symbols indicate the direction of spindle rotation as seen from the point of view of the machinist.

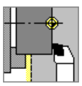
Spindle symbols (S display)	Symbol
Direction of spindle rotation M3	
Direction of spindle rotation M4	
Spindle stopped	
Spindle position-controlled (M19)	

3.4 Machine Setup

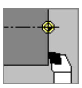
The machine always requires a few preparations, regardless of whether you are machining a workpiece manually or automatically. In Manual mode the following functions are subitems of the "Setup" menu item:

- Setting the axis values (defining workpiece zero point)
- Setting the protection zone
- Defining the tool change position
- Setting C-axis values

Defining the workpiece zero point



Select "Setup."



„Select "Set axis values."

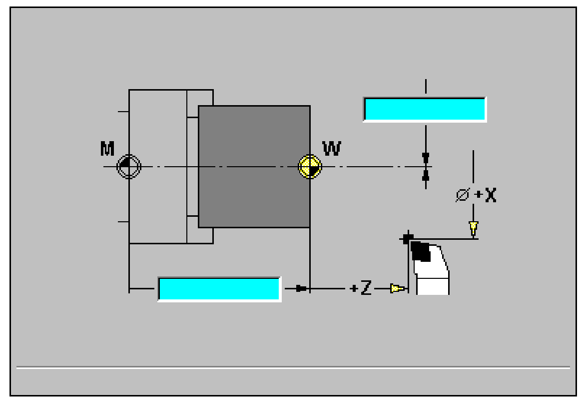
Touch the workpiece zero point (end face).

Z=0	Define this point as the workpiece zero point Z.

Enter the distance between the tool and the workpiece zero point as "measuring point coordinate Z."

Save	MANUALplus calculates the "workpiece zero point Z."

Delete Z offset	Machine zero point Z = workpiece zero point Z (offset = 0).
Delete X offset	Machine zero point X = workpiece zero point X (offset = 0).



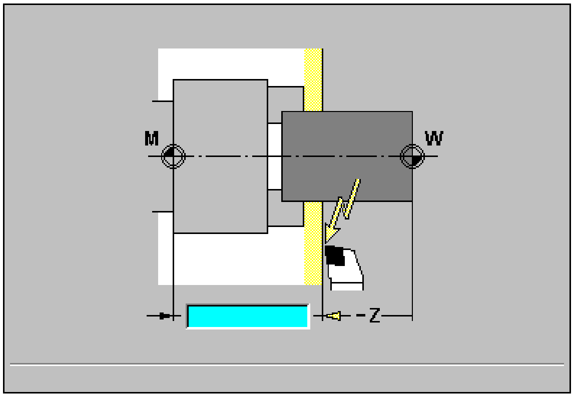
In the graphic support window, MANUALplus illustrates the distance between the machine zero point and the workpiece zero point (also referred to as "offset").

If the workpiece zero point is changed, the display values will be changed accordingly.

> If you want to change the workpiece zero point in X, enter the diameter value as "Meas. pt. coordin. X." The graphic display shows the distance "Machine zero point X to workpiece zero point" as a radius value.

Setting the protection zone

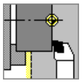
Whenever the tool is moved, MANUALplus checks whether the "protection zone" is violated (in the negative Z direction). If it detects such a violation, it stops the axis movement and generates an error message.



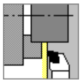
The graphic support window shows the current setting for the protection zone:

- Distance between machine zero point and protection zone.
- "-99999.000" means: Protection zone (in the negative Z direction) is not monitored.

Setting the protection zone/switching off the monitoring function



Select "Setup."



Select the function for setting the protection zone.

Move the tool with the jog keys or handwheel until it reaches the protection zone.

Take over position — Define this position as protection zone.

Enter the position of the protection zone relative to the workpiece zero point (field: "Meas. pt. coordin.–Z").

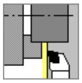
Save — Transfer the entered position as protection zone.

Protectn off — Switch off protective zone monitoring.

Display of the status of protective zone monitoring

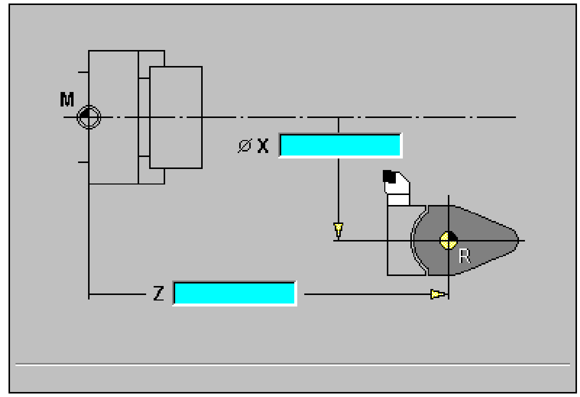
Display symbol 9 of the machine display shows the **current status of protective zone monitoring** (see "Configuration parameters" on page 435 – control parameter 301).

- Protective zone monitoring is not active if the input window "Set protect. zone" is open.
- In DIN programming, protective zone monitoring can be deactivated with M417 and reactivated with M418.

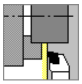
Protection zone status	Symbol
Protective zone monitoring active	
Protective zone monitoring not active	

Defining the tool change position

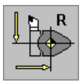
With the cycle "Move to tool change position" or the DIN command G14, the slide moves to the tool change point. Always program the tool change point as far from the workpiece as possible to avoid damage to the workpiece during tool change.



Defining the tool change position



Select "Setup."



Press "Tool change point."

Approach the tool change position.
Move to the tool change point using the jog keys or the handwheel.

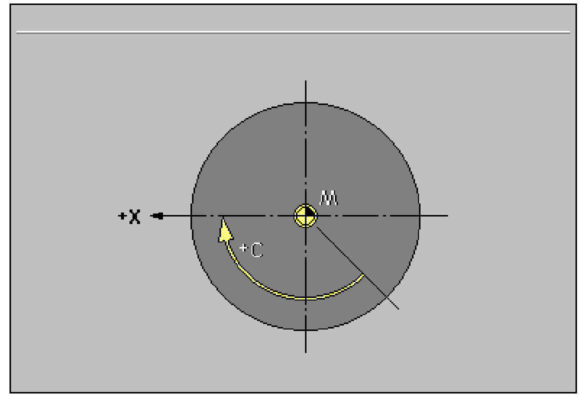
Take over position

Define this position as tool change point.

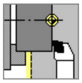
The coordinates of the tool change position are entered and displayed as distance between machine zero point and tool carrier zero point. As these values are not displayed, it is advised to approach the tool change point and then to define the parameters using ***Take over position.***

Setting C-axis values

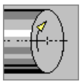
The zero point for the C axis can be defined as follows:



Defining the zero point of the C axis



Select "Setup."



Press "Set C-axis values."

Position the C axis.

C=0 — Define the position as the zero point of the C axis.

Enter the zero point shift of the C axis.

Save — Confirm entry for MANUALplus to calculate the zero point of the C axis.

Delete C offset — Delete zero point shift of the C axis.

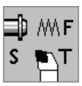
3.5 Setting up Tools

MANUALplus offers functions for measuring tools by touching the workpiece with the tool or by using a touch probe or an optical gauge. Set the measuring method in machine parameter 6.

If the tool dimensions are already known, you can enter the setup dimensions directly in the "Tool management" mode of operation.

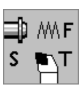
Finding the tool dimensions by touch-off with the tool

In the tool table, enter the tool you want to measure (see “Tool Data” on page 418).



Insert the reference tool and enter the T number in "Set T, S, F."

Turn an end face and define this coordinate as the workpiece zero point.



Return to "Set T, S, F", insert the tool to be measured and enter the associated T number.

Measure tool

Activate ***Measure tool.***

Touch the end face with the tool. Enter the value "0" for the "measuring point coordinate Z" (workpiece zero point).

Take over Z

Save the tool dimensions (the compensation value is deleted).

Turn a measuring diameter. Enter the diameter value as "measuring point coordinate X."

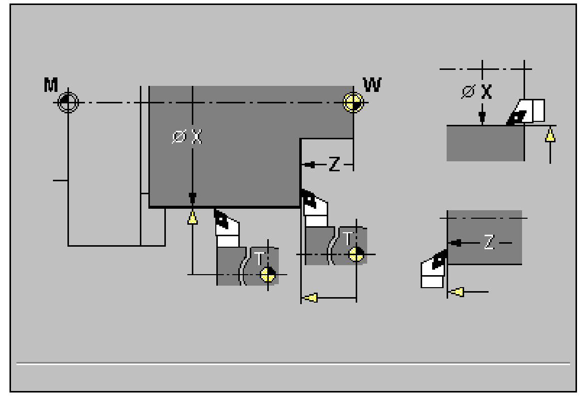
Take over X

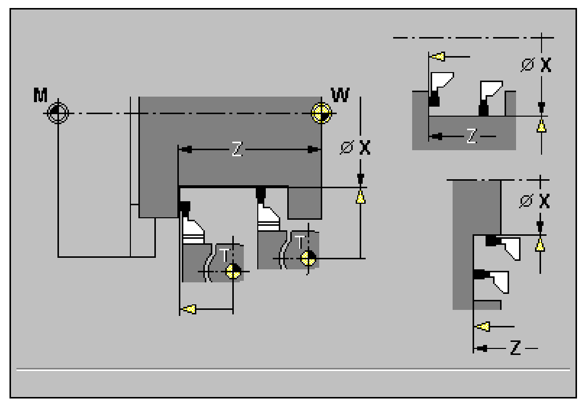
Save the tool dimensions (the compensation value is deleted).

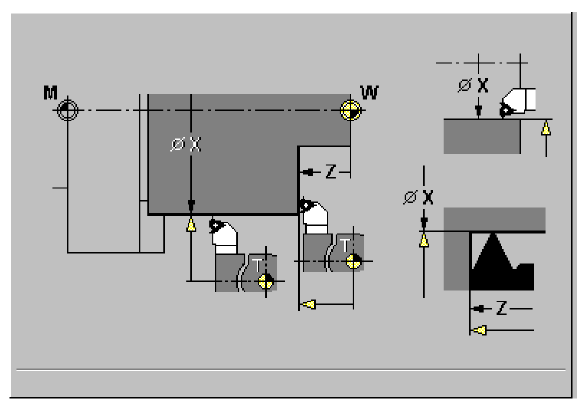
Enter the cutting radius.

Save radius

Transfer the cutting radius to the tool table.

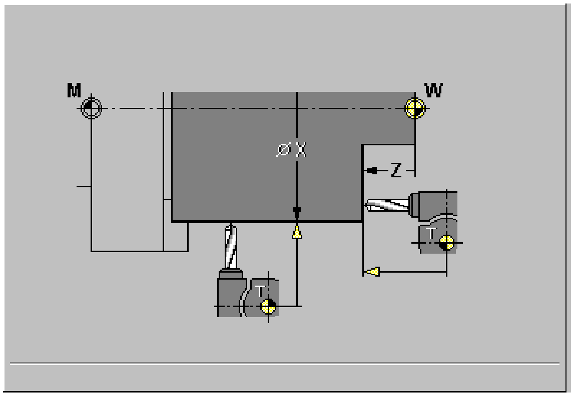


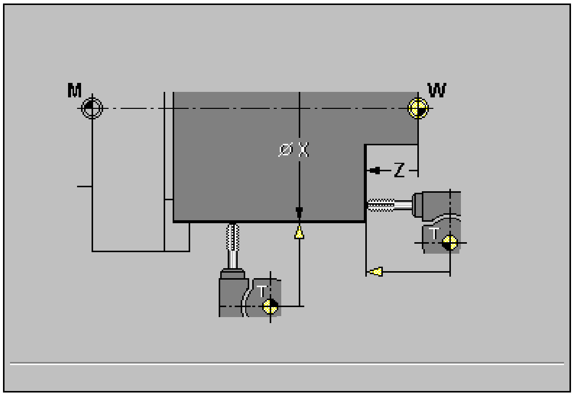


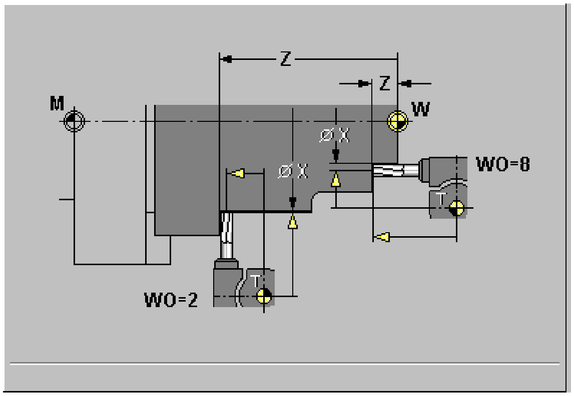


There are several ways to determine tool dimensions. The following method describes how the dimensions are determined by comparing a tool with an already **measured tool.**

The graphic support window shows the details of the tool measurement process, taking the selected tool type and tool orientation into account.

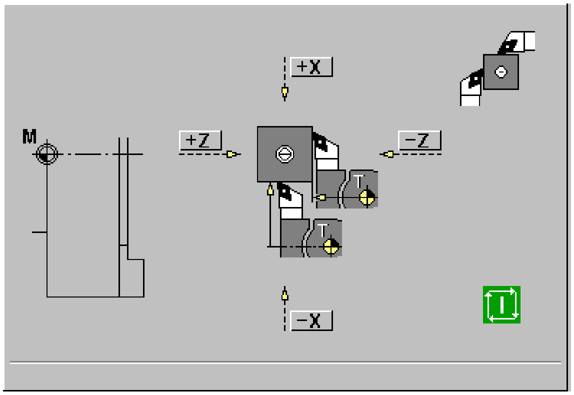






Finding the tool dimensions by using a touch probe

In the tool table, enter the tool you want to measure (see "Tool Data" on page 418).

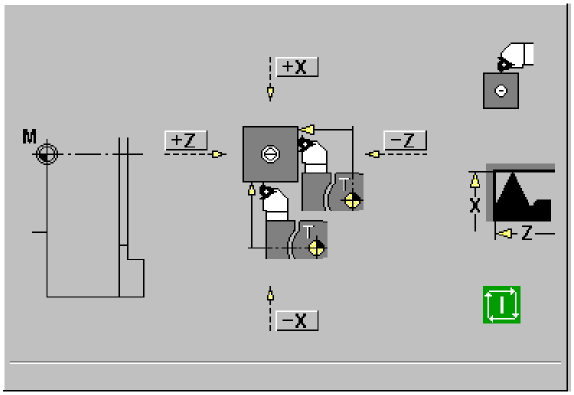


Insert the tool and enter the T number in "Set T, S, F."

Measure tool — Activate ***Measure tool.***

Pre-position the tool for the first direction of measurement.

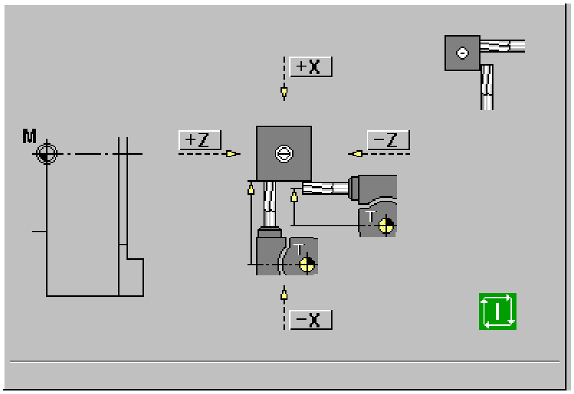
-Z / **+Z** — Press the soft key for this direction (e.g. Z direction).



Press Cycle START. The tool moves in the direction of measurement. When it contacts the touch probe, the control calculates and saves the set-up dimensions. The compensation value is deleted.

Pre-position the tool for the second direction of measurement.

-X / **+X** — Press the soft key for this direction (e.g. X direction).



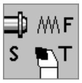
Press Cycle START. The tool moves in the direction of measurement. When it contacts the touch probe, the control calculates and saves the set-up dimensions. The compensation value is deleted.

Enter the cutting radius.

Save radius — Transfer the cutting radius to the tool table.

Finding the tool dimensions by using an optical gauge

In the tool table, enter the tool you want to measure (see “Tool Data” on page 418).



Insert the tool and enter the T number in "Set T, S, F."

Measure tool

Activate ***Measure tool.***

Position the tool at the cross hairs of the optical gauge by using the jog keys or the handwheel.

Take over Z

Save the tool dimension in Z (the compensation value is deleted).

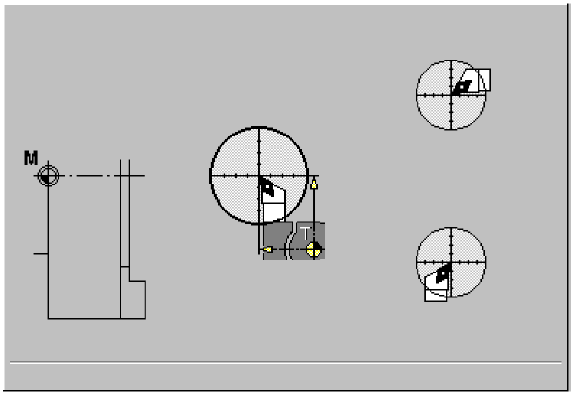
Take over X

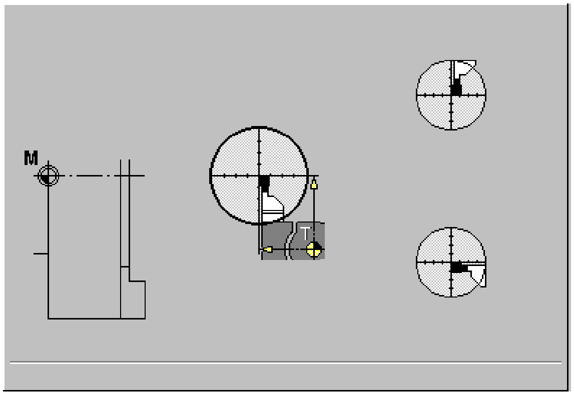
Save the tool dimension in X (the compensation value is deleted).

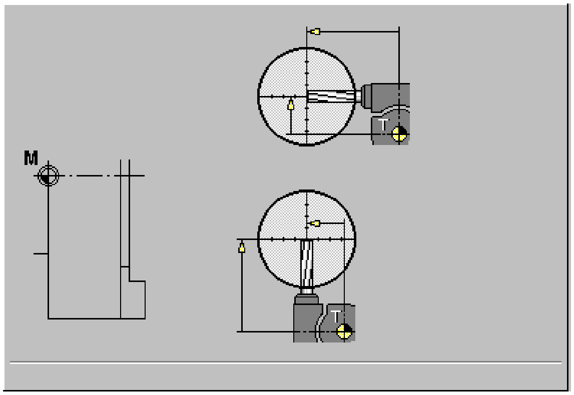
Enter the cutting radius.

Save radius

Transfer the cutting radius to the tool table.





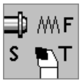


Tool compensation

The tool compensation in X and Z as well as the special compensation for recessing tools compensate for wear of the cutting edge.

> A compensation value must not exceed 99 mm.

Defining tool compensation



Select "Set T, S, F" (only available in Manual mode).

Tool offset

Press ***Tool correct.***

X offset f. tool

Select ***X offset for tool.***

The compensation values that you determine per handwheel are now shown in the "Distance-to-go" display.

Save

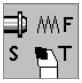
Transfer the compensation value to the tool table.

The T display shows the new compensation value.

The distance-to-go display is cancelled.

Repeat this procedure for the tool compensation Z and the special compensation.

Deleting tool compensation values



Select "Set T, S, F" (only available in Manual mode).

Tool offset

Select ***Tool correct.***

Erase X-offset

Cancel the compensation value entered in X.

Repeat this procedure for the tool compensation Z and the special compensation.

Tool life monitoring

If desired, you can have MANUALplus monitor tool life or the number of parts that are produced with a specific tool.

The tool life monitoring function adds the times a tool is traversed at the machine feed rate and counts the number of finished parts. The count is compared with the entry in the tool data.

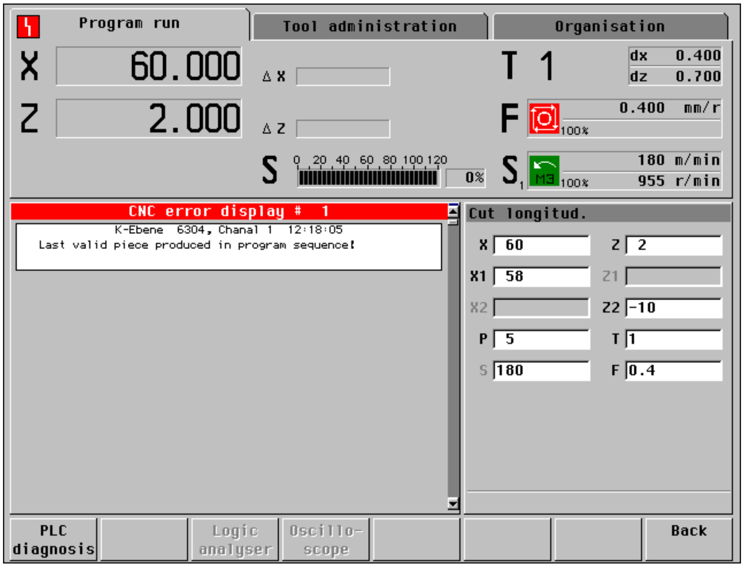
As soon as the tool life expires or the programmed quantity is reached, MANUALplus generates an error message and stops program execution **after** the end of the program. If you are working with program repeats (M99 in DIN programs), the system is stopped after execution of the current repeat.

Tool life monitoring should be carried out for each tool used.

The tool life monitoring data (type of monitoring, maximum tool life / remaining tool life and the maximum number of pieces / remaining number of pieces) are managed in the tool data. The tool life monitoring data are also edited and displayed in the tool data (see "Tool Data— Supplementary Parameters" on page 426).

Tool life monitoring is enabled and disabled in "Current parameters—Setup parameters—Tool monitoring."

You must update the data on tool life and number of pieces in the Tool management mode when you replace the cutting edge of a tool.



3.6 Manual Mode

With **manual workpiece machining,** you move the axes with the handwheels or jog controls. You can also use cycles for machining complex contours (semi-automatic mode). The paths of traverse and the cycles, however, are **not stored.**

After switch-on and traversing the reference marks, MANUALplus is always in Manual mode. This mode remains active until you select ***Teach-in*** or ***Program run.*** You can return to Manual mode with the "Menu" key. "Machine" displayed in the header indicates that you are in Manual mode.

Define the workpiece zero point (see“Machine Setup” on page 50) and enter the machine data (see “Machine Data” on page 46) before you start machining.

Tool change

Enter the **T number** and check the tool parameters.

"T0" does not define a tool. This also means that T0 does not contain any data on tool length, cutting radius, etc.

Spindle

The spindle speed is entered in "Set T, S, F." To start and stop spindle rotation, press the buttons on the machine operating panel. **Position the spindle** by defining the "Stopping angle A" in the "Set T, S, F" menu.

Pay attention to the maximum speed (can be defined with "Set T, S, F").

Handwheel operation

You set the traverse per handwheel increment with the **handwheel resolution** selector switch on the machine operating panel.

Jog operation (joystick)

With the jog controls, you can move the axes at the programmed feed rate or at rapid traverse. The feed rate is programmed in "Set T, S, F." The rapid traverse speed is set in "Current parameters—Machine parameters—Feeds."

Cycles in Manual mode

- Set the spindle speed.
- Set the feed rate
- Insert tool, define T number and check tool data (T0 is not permitted).
- Approach cycle start point.
- Select the cycle and enter cycle parameters.
- Graphic control of cycle run.
- Run the cycle.

3.7 Teach-In Mode

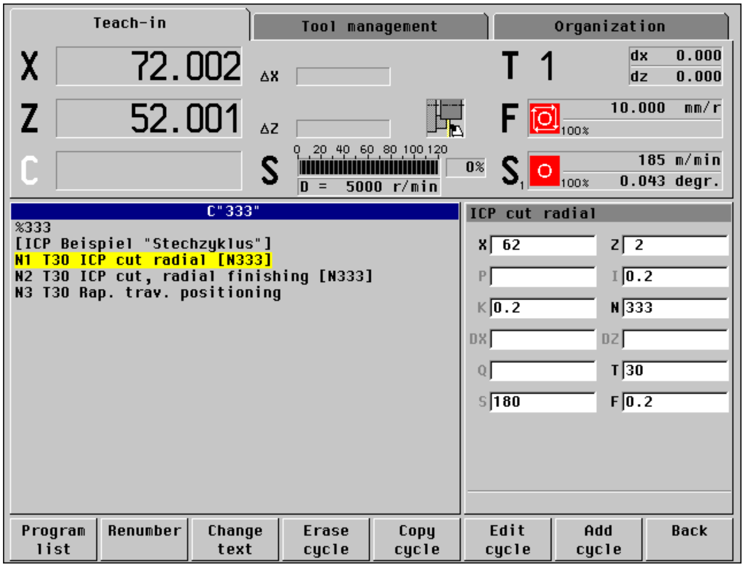
In **Teach-in mode** (cycle mode), you machine a workpiece step by step with the help of cycles. MANUALplus "memorizes" how the workpiece was machined and stores the necessary working steps in a cycle program, which you can call up again at any time.

The Teach-in mode can be switched on by soft key and is displayed in the header.

Each **cycle program** is given a number and a short description. The individual cycles of a cycle program are listed as blocks and are numbered in ascending order. The block number has no meaning for the program run. The cycles are run after each other. When the cursor is located on a cycle block, MANUALplus displays the cycle parameters.

The cycle block includes:

- Block number
- Tool used
- Cycle designation
- Number of ICP contour or of DIN macro (in [...])



Cycle programming

When creating a new cycle program, you program each cycle in the following sequence of actions "Enter—Simulate—Execute—Save." The individual cycles form the cycle program.

You can change cycle programs by simply editing the necessary cycle parameters, and delete or add cycles as required.

When you exit the Teach-in mode or switch off the machine, the cycle program remains as it was programmed or edited.

When you call an ICP cycle, MANUALplus displays a soft key for switching to the ICP contour editor (see “Editing ICP Contours” on page 243).

DIN macros are programmed in the DIN editor and then integrated in a DIN cycle. You can call the DIN editor by soft key when you select the DIN cycle or when you are in the "Main menu" (see “DIN Programming” on page 278).

Soft keys

Program list	Switch to the "Select cycle programs" function (see “Program Management” on page 75).
Renumber	Renumber the block numbers of the cycles.
Change text	Call the alphanumeric keyboard to enter or edit the program description.
Erase cycle	Delete the selected cycle.
Copy cycle	Copy the cycle parameters into a buffer memory. When you then press "Add cycle," the data is inserted (example: copy parameters of roughing cycle into a finishing cycle).
Edit cycle	Edit cycle parameters or cycle mode (the cycle type cannot be edited).
Add cycle	Insert a new cycle below the highlighted block.

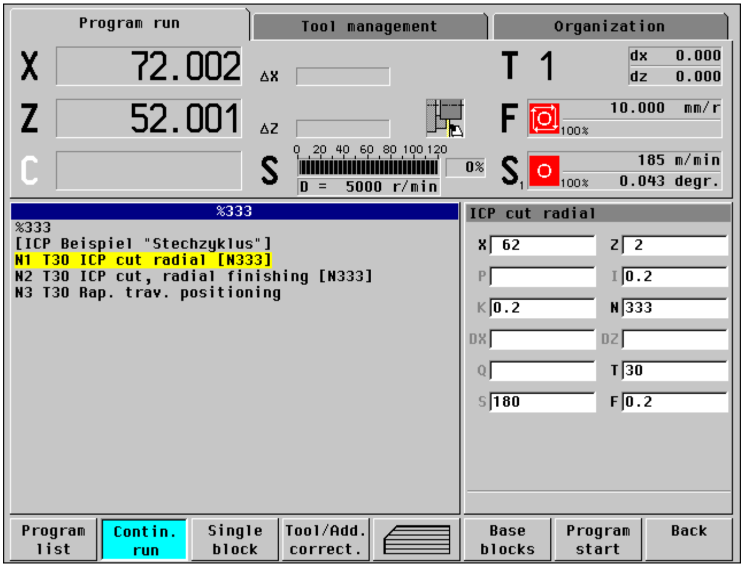
3.8 Program Run Mode

In Program run mode, you use cycle programs and DIN programs for parts production. You cannot change the programs in this mode. The "graphic simulation" feature, however, allows you to check the programs **before** you run them. MANUALplus also offers the "Single block" mode with which you can machine a workpiece, for example, the first of a whole batch, step by step.

You can start a cycle or DIN program at any desired block to resume a machining operation after an interruption.

The program run mode can be switched on with the soft key and is displayed in the header.

If you press ***Program run,*** MANUALplus reads in the program that was last active in this mode or in the editing mode. Alternately, you can select another program with ***Program list*** (see “Program Management” on page 75).



Faulty programs

The MANUALplus checks the programs during loading. If it detects an error (for example, a programmed tool that does not appear in the tool list), it displays the error symbol in the screen headline. You can then press the "Info" key for detailed information on the error.

MANUALplus does not translate faulty cycles, but inserts a "Cycle STOP" at the respective position. All correct cycles of this program are translated.

Danger of collision!

For programs with faulty cycles, ensure that the program can be executed without danger of collision.

Before executing a program

- **Check the cycles and cycle parameters**
  MANUALplus displays the cycle program or the DIN program in the list window. With cycle programs, the parameters of the cycle on which the cursor is placed are displayed.
- **Graphic control**
  You can monitor program run with the graphic simulation feature (see “Graphic Simulation” on page 68).

Start block search and program execution

Preconditions for defining a start block:

- The MANUALplus must be prepared by the machine tool builder for the start block function.
- The start block function must be activated (Organization mode of operation: "Current parameters—NC switches—Settings" or control parameter 1)

MANUALplus starts program run from the cursor position. The starting position is not changed by a previous graphic simulation.

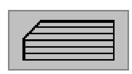
> When selecting the start block in a DIN program, ensure that the control executes all commands that define the machine data (T, S, F) before it reaches the first traversing command.

Program execution

The selected cycle or DIN program is executed as soon as you press "Cycle START." You can interrupt machining at any time by pressing "Cycle STOP."

During program run, MANUALplus highlights the cycle (or DIN block) that is presently being executed. With cycle programs, the parameters of the cycle currently being run are displayed in the input window.

The soft keys allow you to influence the program run—see table.

Soft keys	
Program list	Select cycle program or DIN program (see "Program Management" on page 75).
Contin. run	Cycle program ■ **On:** Cycles are run continuously, one after the other, up to the next tool change. ■ **Off:** MANUALplus stops after each cycle. To start the subsequent cycle, press "Cycle START." DIN program ■ **On:** Program execution without any interruption. ■ **Off:** Stop before command M01.
Single block	■ **On:** MANUALplus stops after each traverse. To start the next path of traverse, press Cycle START. (Recommendation: Single block should be used together with the basic-block display.) ■ **Off:** Cycles / DIN commands are executed without any interruption.
Tool/Add. correct.	Input of tool compensation values or additive correction values.
	Switch the graphic simulation on.
Base blocks	■ **On:** The traversing and switching commands are shown in DIN format (base blocks). ■ **Off:** Cycle or DIN program is displayed.
Program start	The cursor returns to the first block of the cycle program or DIN program.

Entering compensation values during program execution

Compensation values can be entered **during** program execution. Entered values are **added** to the existing compensation values and are effective immediately.

Entering tool compensation values

Tool/Add. correct. | Tool offset

Activate the tool compensation.

Enter the tool number.

Save

Press ***Save*** for the valid compensation data to be displayed in the input window.

Enter the compensation values.

Save

Transfer the compensation values (see "Setting up Tools" on page 54).

Entering additive compensation values

Tool/Add. correct. | Additive correct.

Activate the additive compensation.

Enter the number of the additive compensation.

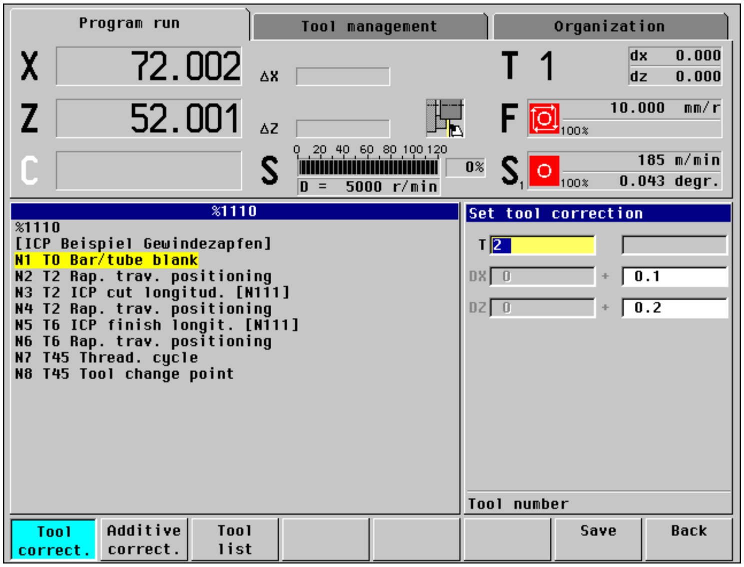
Save

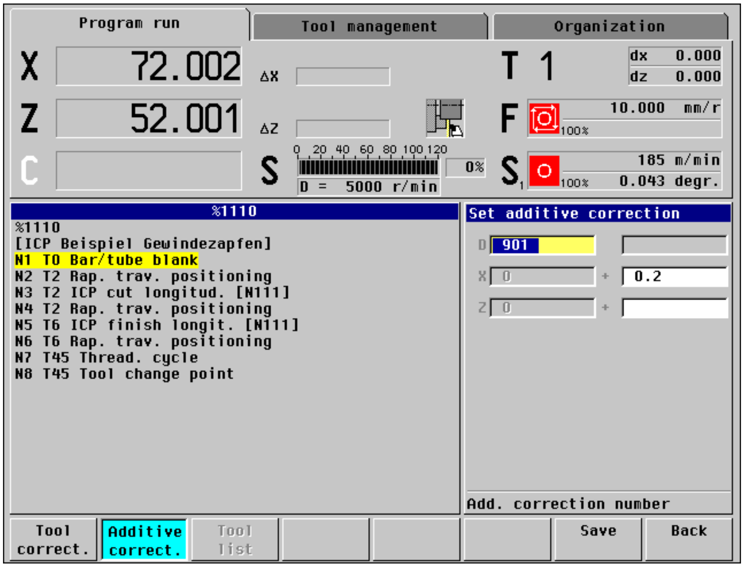
Press ***Save*** for the valid compensation data to be displayed.

Enter the compensation values.

Save

Press ***Save.***





MANUALplus manages 16 additive compensation values as "parameters." You can edit the additive compensation values in the "Organization mode of operation—Current parameters." Additive compensation values need to be activated with "G149" in a DIN program or a DIN macro.

Setting compensation values with the handwheel

The “Compensation values via handwheel” function is only available if **bit 13 of the configuration code** (MP 18 – control configuration) is set.

Entering tool compensation values with the handwheel

Interrupt program run with **Cycle Stop**

Tool offset

Press ***Tool correct.***

X offset f. tool

Select ***X offset for tool*** (or ***Z offset for tool)***.

The compensation values that you determine per handwheel are now shown in the "Distance-to-go" display.

Save

Transfer the compensation value to the tool table.

The T display shows the new compensation value.

The distance-to-go display is cancelled.

Deleting tool compensation values

Interrupt program run with **Cycle Stop**

Tool offset

Select ***Tool correct.***

Erase X-offset

Select ***Erase X offset*** (or ***Erase Z offset***) for the entered compensation value to be deleted.

Program execution in "dry run" mode

The dry run mode is used for fast program execution up to a point at which machining is to resume. The prerequisites for a dry run are:

- The MANUALplus must be prepared by the machine tool builder for dry run (The function is activated with a keylock switch or a key.)
- The Program Run mode must be activated

In dry run, all feed paths (except thread cuts) are traversed at the rapid rate. You can reduce the traversing speed with the feed rate override. Do not use the dry run feature for anything other than "cutting air."

When dry run is activated, the spindle status or spindle speed is "frozen." After deactivation of the dry run, the MANUALplus returns to the programmed feed rates and spindle speeds.

Use the **dry run** feature only to "cut air."

3.9 Graphic Simulation

The graphic simulation feature enables you to check the machining sequence, the proportioning of cuts and the finished contour **before** actual machining.

In the Manual and Teach-in modes, this function simulates the execution of a single cycle—in Program run mode it simulates a complete cycle or DIN program.

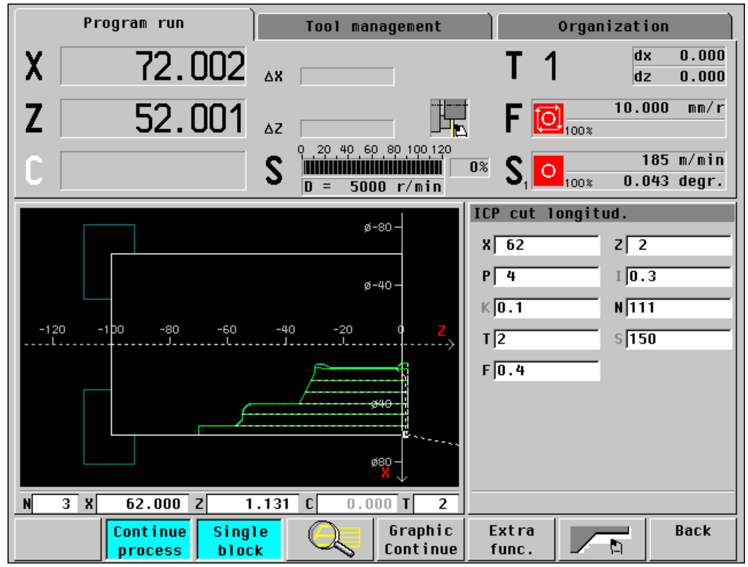
A programmed workpiece blank is displayed in the simulation graphics. MANUALplus also simulates machining operations that are executed with a traversable spindle or the C axis on the face or lateral surface. This allows you to check the complete machining process.

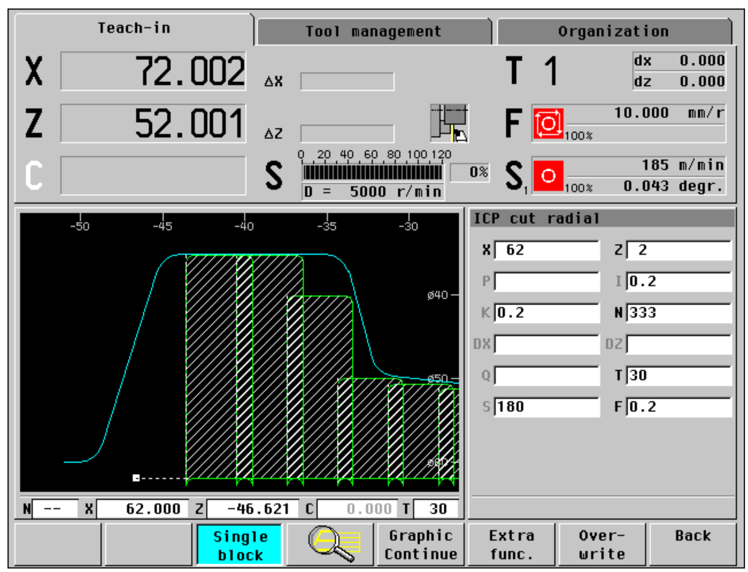
In Manual mode and Teach-in mode, the cycle you are currently working on is simulated. In the Program run mode, the simulation of a cycle program always starts from the cursor position. DIN programs are simulated from the beginning of the program.

You can choose between wire frame graphics and cutting path graphics. In addition, the motion simulation graphics (erasing graphics) is available for displaying turning operations. It is recommended to use this graphic check in the "Program run" mode, since it provides a good overview of the machining process.

The **wire frame graphics** is particularly convenient if you only need a quick overview of the proportioning of cuts. The path of the theoretical tool tip, however, is not identical with the contour of the workpiece. This graphics is therefore not as suitable if you wish to run a thorough check on the machined contour. In the CNC, this "falsification" is compensated by the cutting radius compensation.

The **cutting path graphics** accounts for the exact geometry of the tool tip. Here, you can check whether the contour is machined completely or needs to be reworked, whether the contour is damaged by the tool or overlaps are too large. The cutting path graphics is especially useful for recessing, drilling and milling operations where the tool shape has an essential influence on the accuracy of the resulting workpiece.



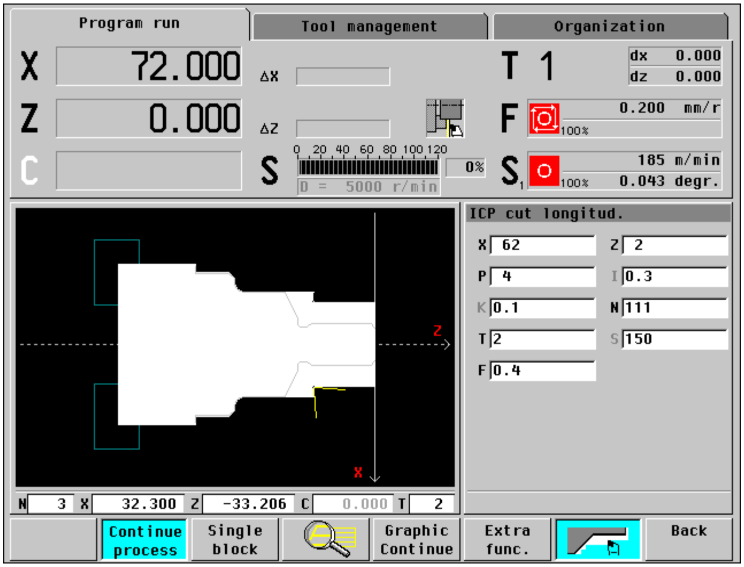


The **motion simulation** depicts the workpiece blank material as a "filled surface" and "machines" it during simulation by "erasing" the material (erasing graphics). The tools move at the programmed feed rate (program-run graphics).

If during running simulation you switch to the motion simulation, it will not become effective until the simulation function is restarted.

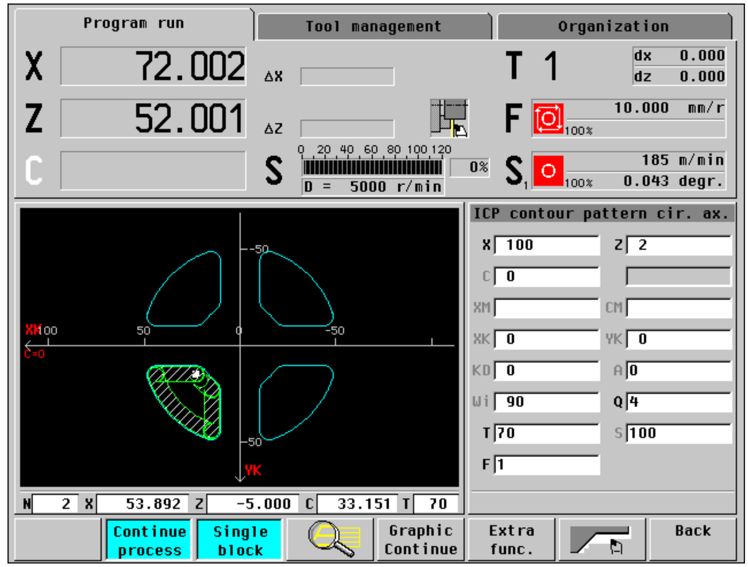
You can interrupt the motion simulation at any time, even during simulation of an NC block. The display below the simulation window indicates the target position of the current path.

When using the motion simulation for checking individual cycles, please note that in some cycles the workpiece blank is not known. The tool movements will then be displayed, but the machining process will not be displayed.

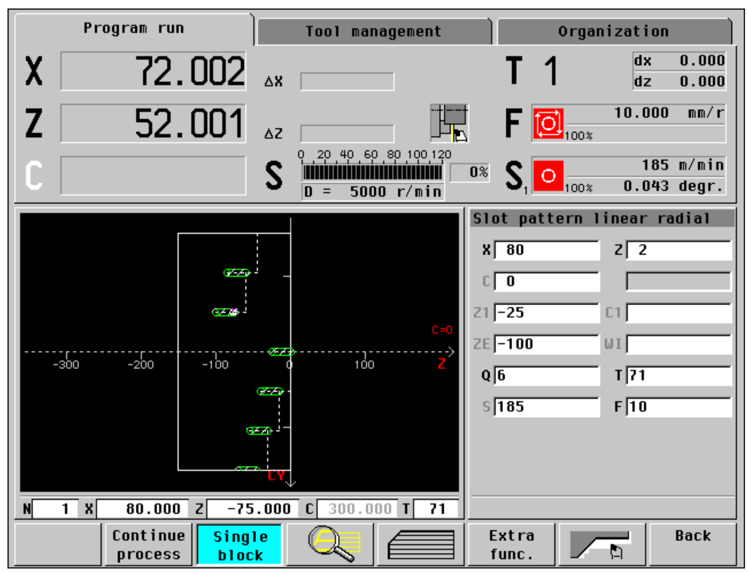


Views

Machining operations with traversable spindle or a C axis can be controlled in the face view or surface view (under "Extra functions"). The "Lathe, Face or Surface view" can be displayed as an alternative.



- **Lathe view**
  Representation in the X/Z plane.
- **Face view**
  Representation in the XK/YK plane. The coordinate system is based on Cartesian coordinates. The origin is located in the turning center, with the angle C=0° positioned on the positive XK axis (see figure at top right).
- **Surface view**
  Representation of the "unrolled lateral surface" (Z/CY plane). The coordinate system is based on Cartesian coordinates. The horizontal line gives the Z axis, the vertical line the CY axis coordinates (see figure at bottom right). The upper and lower lines of this workpiece correspond to the angular positions C=-180°/+180°, respectively. All drilling and milling operations are within the range -180° to +180°.
  - **Cycle program or DIN program:**
    Workpiece simulation is based on the parameter values for "standard blank" (Current parameters—Graphic parameters—Standard blank).
  - **Individual cycle or Teach-in**
    Workpiece simulation is based on the workpiece section defined by the respective cycle (expansion in Z and limiting diameter X).

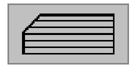
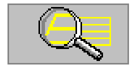
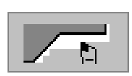


> - The soft keys "Face view / Surface view" are only available when a cycle / cycle program with drilling/milling functions or a DIN program is active.
> - You can check the depth of an axial hole / milled contour in Lathe view, and the depth of a radial hole / milled contour in Face view.

Graphic elements

During simulation, MANUALplus shows the following elements and tool movements in the graphics window:

- **Origin of the coordinate system**
  The workpiece zero point serves as the origin of the coordinate system.
- **Contour**
  At the beginning of a cycle simulation, the programmed contour of that cycle is depicted in cyan. In the Teach-in mode, you can display the preceding contour elements of the cycle program (function "Display contour elements).
- **Light dot**
  The light dot (small white rectangle) represents the theoretical tool point.
- **Feed paths**
  These paths are shown as a continuous green line. They describe the path of the theoretical tool tip (also called "wire frame graphics").
- **Rapid traverse paths**
  These paths are shown as a broken white line.
- **Tool tip** (cutting edge)
  Instead of the light dot, MANUALplus shows the cutting edge of the tool as a continuous yellow line. In this case, you can see the real cutting radius, cutting width and tool-tip position.
  With many machining operations, like recessing or machining an oblique surface or a rounding, you can check the machining sequence a lot more precisely by showing the tool tip instead of the light dot.
  This graphic display is based on the tool data: If the control does not have enough data on the tool, it can only represent the tool tip as a light dot.
- **Cutting path**
  In the "cutting path display," MANUALplus shades the area traversed by the cutting edge of the tool. You can see the area that will actually be machined, with the cutting radius, cutting width and tool-tip position already accounted for. This graphic display is based on the tool data.

Soft keys	
	Switch the graphic simulation on.
	Magnify, reduce, shift view.
Contin. run	Cycle program ■ **On:** Cycles are simulated up to the next tool change. ■ **Off:** MANUALplus stops after each cycle. To start the subsequent cycle, press ***Graphic Continue.*** DIN program ■ **On:** Program execution without any interruption. ■ **Off:** Stop before command M01.
Single block	■ **On:** MANUALplus stops after each traverse. To start the simulation of the next path, press ***Graphic Continue.*** (Recommendation: Single block should be used together with the basic-block display.) ■ **Off:** Cycles / DIN commands are simulated without any interruption.
Graphic Stop	The simulation is interrupted.
Graphic Continue	The simulation is resumed.
	Switch on machining simulation.
Extra func.	Switch to the soft-key row with the soft keys for extra functions.
Track	■ **On:** The tool paths are displayed in the cutting path graphics. ■ **Off:** The tool paths are displayed in the wire frame graphics.
Slide	■ **On:** The tool tip is displayed. ■ **Off:** The light dot represents the tool tip.
Contour view	Displays the workpiece blank contour (if programmed) in Teach-in mode and the contours defined in every cycle from the beginning of the program to the cursor position.

Displays beneath the graphics window:

- **Field "N"**
  Block number of the simulated block.
- **Fields "X" and "Z"**
  Target coordinates of the simulated rapid traverse or feed path.
- **Field "C"**
  Spindle angle for positioned spindle (M19) or C axis.
- **Field "T"**
  Simulated tool (programmed T number).
- **Input box**
  For cycle programs, the cycle designation and the parameters are displayed.

Warnings

The simulation feature checks cycle and DIN programs. Warnings that have occurred are displayed with the soft key to the extreme left (see “Error Messages” on page 36). To read the warnings, press the soft key.

Soft keys	
Process times	Call the "Time calculation" (see “Time Calculation” on page 74).
Face view	Switch to Face view if you have programmed drilling/ milling cycles for the face.
Surface view	Switch to Surface view if you have programmed drilling/milling cycles for the cylindrical surface.

Magnify / Reduce

With cycle programs, the simulation feature selects the area to be simulated in such a way that all paths of traverse can be illustrated. With DIN programs and DIN macros, the area to be simulated is taken from "Current parameters—Graphic parameters—Standard window size / Standard blank." This is also the case for the face and lateral-surface views.

MANUALplus offers two menu options, one for magnifying and reducing the displayed graphic, and one for isolating a detail:

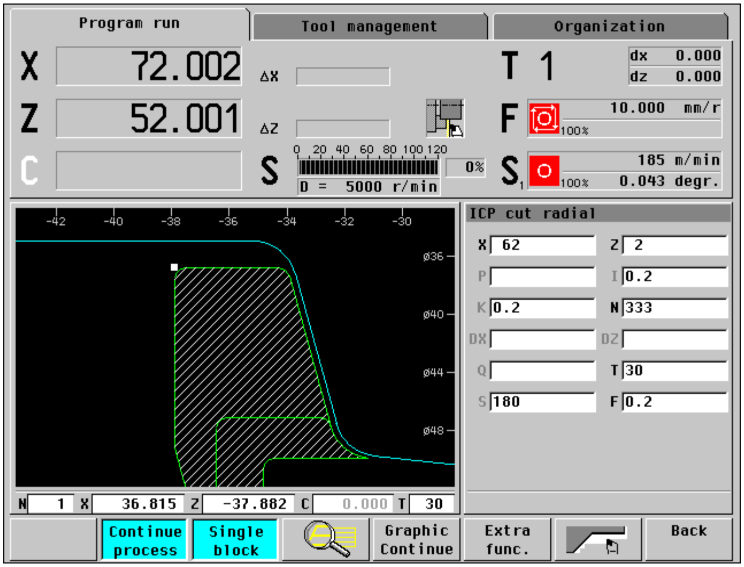
1 When you press "Zoom," a "red frame" appears on the screen with which you can select the detail you wish to isolate. The frame is moved with the arrow keys, enlarged with PgUp and reduced with PgDn. To display the selected detail, press ***Take over.***

The following functions also exist:

- **Extend view:** By reducing the size of the workpiece, a larger area of the working space can be displayed. You can use this function if you would like to isolate an area of the workpiece with the "red frame," which is not displayed in the graphics window.
- **Zoom off:** All defined contour areas (the "workpiece") and the paths of traverse are shown as large as possible.
- **Last zoom:** Return to the last setting of Zoom.

2 You can magnify/reduce the displayed graphics with the PgUp/PgDn keys, and pan the detail with the arrow keys. These functions are always available during simulation.

With operation **1** you can isolate and display a precise detail, while operation **2** allows you to pan, magnify or reduce the graphic spontaneously. It may require several steps until the desired detail is displayed in the correct size.



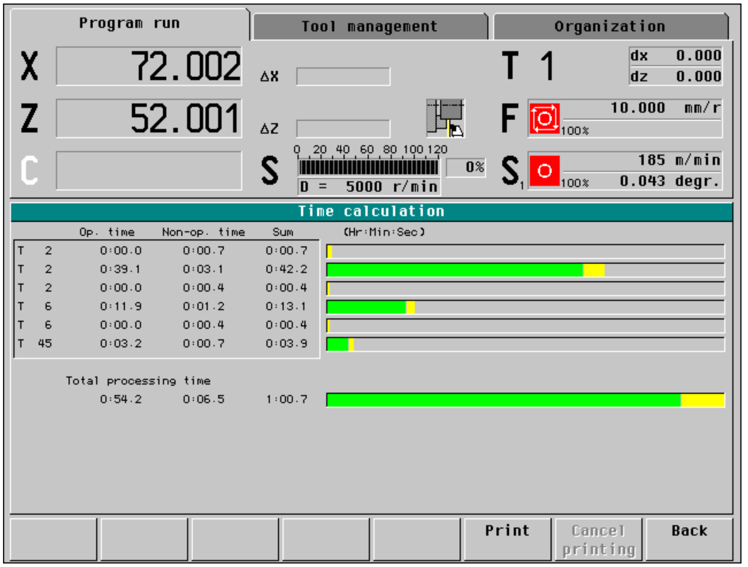
3.10 Time Calculation

During simulation, the machining and idle-machine times are calculated. MANUALplus shows the machining times under the menu item "Extra functions—Process times (machining times)."

The machining times, idle times and total times are shown in the table "Time calculation" (green: machining times; yellow: idle times). If you are working with cycle programs, each cycle is shown in a separate line. In DIN programs, each line represents the use of a new tool (for each tool call with T).

If there are more table entries than fit on a screen page, you can call further time data with the arrow keys and PgUp/PgDn. The arrow symbols in the title bar indicate that more entries are available.

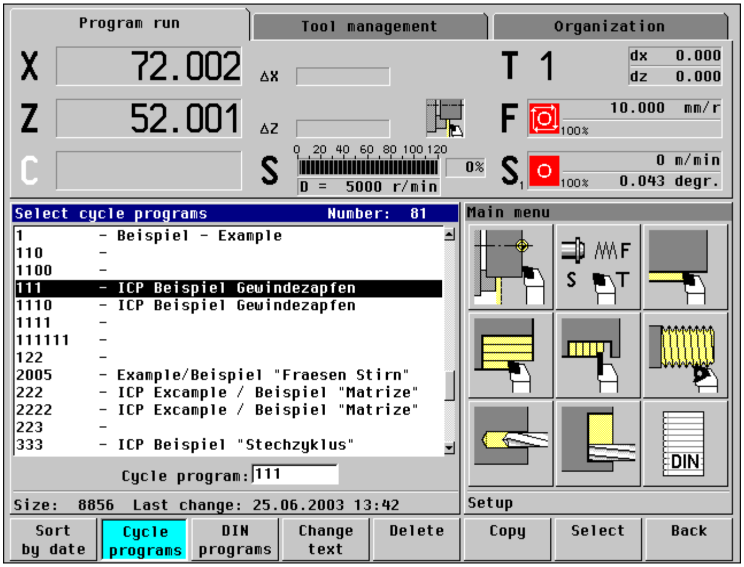
It is also possible to print the time calculation.



3.11 Program Management

MANUALplus differentiates between the following program groups:

- Cycle programs
- ICP contours
- DIN programs
- DIN macros



Program information

- **Program number**
  The program number (1 to 8 characters) serves to identify the program within a program group. Completing zeros are part of the program number.
- **Program description**
  You can describe a program by a short text of up to 35 alphanumeric characters. This text is displayed in the program list.
- **Date, time**
  The control stores the time the program was last changed and displays this in "Sort by date."
- **Program length**
  With this information you can estimate the size of a program. The program length is given in bytes. By rule of thumb, one cycle or ICP contour element needs about 165 bytes, and each character of a DIN program or DIN macro takes up 1 byte.

Functions for program management

First select the desired program and then press the corresponding function key. The selected program is displayed in the "Program number" field.

- **Sort** program list
  The programs of a program group can be listed in alphabetical order or by date.
- **Select program**
  You can select the desired program from the list or enter the program number.
- **Activate program**
  When you press ***Select program,*** the control returns to the previous operating environment. The program that is displayed in the "Program number" field is activated.
- **Create new program**
  Enter the "new program number" and press ***Select program.***
- **Copy program**
  The selected program is copied. The copied program then needs to be assigned a new program number. The other program data is not changed. Neither are the contents.
- **Delete program**
  The selected program is deleted from the system.
- **Change the program description**
  With ***Change text*** you can call the alphanumeric keyboard to change a program description or to enter a new one.

If the cursor is located in the "program number" field, you can enter the number of the desired program. If you do not know the exact program number, simply enter the "incomplete" number and switch into the program list with ENTER. The cursor is then located on the first program number that matches your entry.

When the cursor is in the program list window, you can scroll through the list to find the desired program. Enter the first character of the program number for the highlight to jump to the **next** program that starts with this number.

You can switch from "Program number" to "Program list" with ENTER (or the vertical arrow keys). And ENTER (or the horizontal arrow keys) will take you back to "Program number" again.

If you wish to change a program number, you must copy the program, assign it a new program number, and then erase the original program.

3.12 Conversion into DIN Format

The "Convert to DIN" function enables you to convert a cycle program to a DIN program with the same functionality. You can then optimize, expand such a DIN program, etc.

Conversion into DIN format

Cyc.prog. -> DIN

Press ***Cyc.prog. --> DIN.***

Select the program to be converted.

Create DIN prog.

Press ***Create DIN prog.***

The generated DIN program has the same program number as the cycle program.

Should MANUALplus encounter any errors during conversion, it generates an error message and aborts conversion.

3.13 Inch Mode

You can operate MANUALplus in the metric or inch system (for inch mode, see illustration to the right).

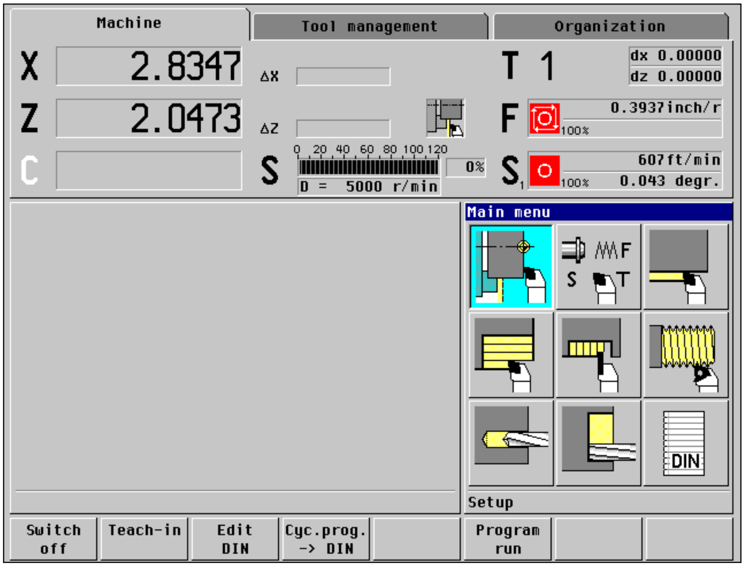
Units in inch mode:

- Coordinates, lengths, path data in inches
- Feed rate in inch/revolution or inch/min
- Cutting speed in ft/min (feet/min)

The inch/metric setting is also evaluated for the displays and entries in tool management and parameters.

For accuracies for displays and entries, see the table at right.

The metric/inch setting is selected in the parameter "Current parameters - NC switches - Settings." Changed metric/inch settings do not become effective until the control is restarted.

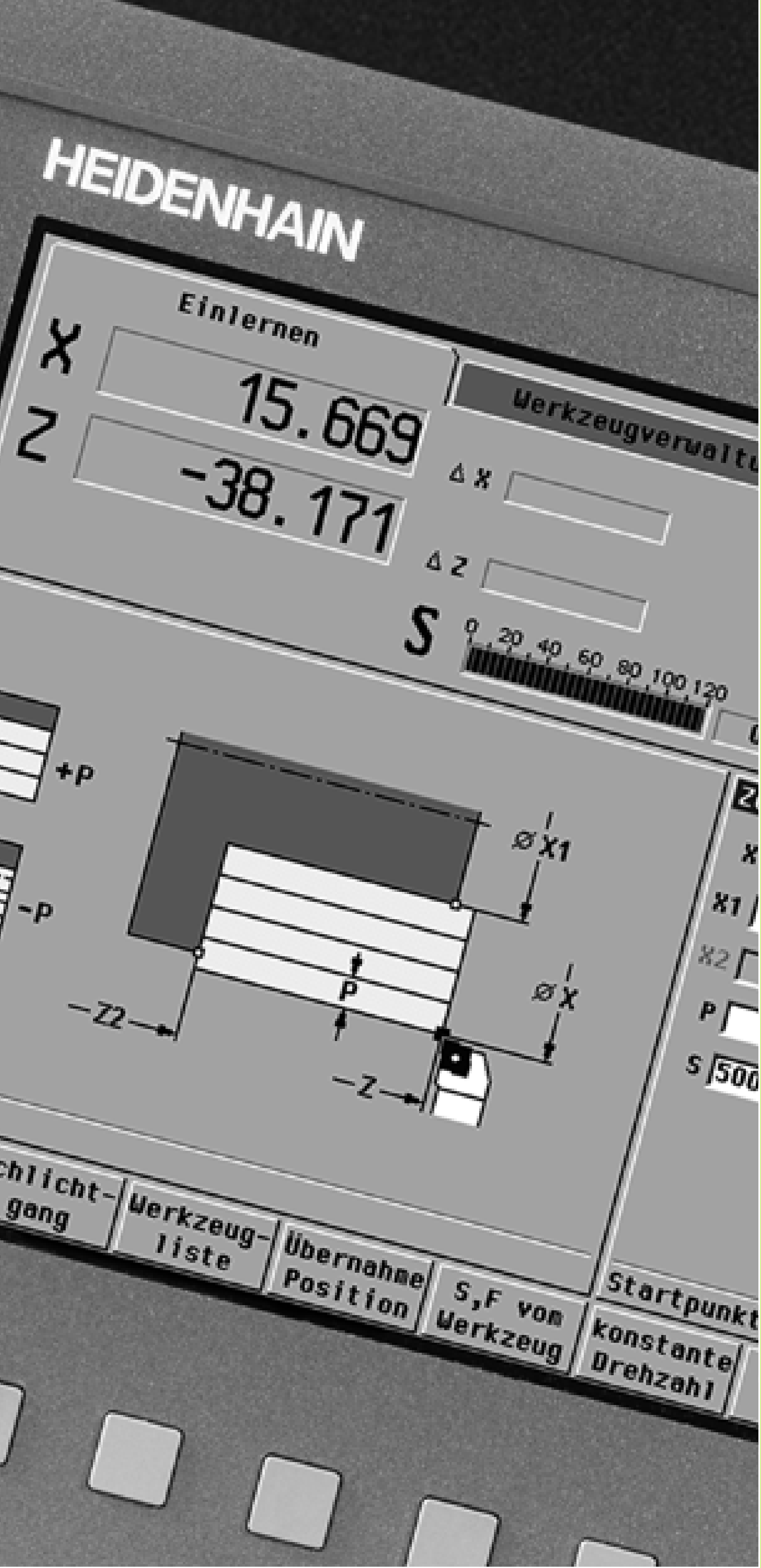


Number of decimal places	Metric	Inch
With coordinate data and path data:	3	4
With compensation values:	3	5

Cycle programs

Cycle programs are always stored in the metric system, regardless of whether they were written in metric or inch mode. If you load a cycle program while in inch mode, MANUALplus automatically converts the cycle parameters to inches. The cycle parameters are then displayed and entered in inches.

- **DIN programs** that were written in metric mode may only be executed in this mode. The same applies for inch mode. When executing a DIN program, MANUALplus **does not check** whether it was written in inch or metric mode.
- Refer to your machine manual if you want to know whether and how the **handwheel resolution** can be set to inches.



4

Cycle Programming

4.1 Working with Cycles

Before you can use the cycles, you must set the workpiece zero point and ensure that the tools you are going to use are described. The machine data (tool, feed rate, spindle speed) are entered with the other cycle parameters in Teach-in mode. In Manual mode, you must program these machine data before calling a cycle.

Define the individual cycles as follows:

- Position the tool tip with the handwheel or the jog keys to the starting point of the cycle (only in Manual mode).
- Select and program a cycle.
- Graphically test the cycle.
- Execute the cycle.
- Save the cycle (only in Teach-in mode).

Starting point of cycles

In Manual mode, cycles are executed from the "current tool position."

In Teach-in mode, you enter the starting point as one of the parameters. MANUALplus moves to this position in rapid traverse by the shortest path (diagonal) **before executing the cycle.**

Danger of collision!

If the tool cannot approach the next starting point on the shortest path without colliding with the workpiece, you must define an auxiliary position with the "Rapid traverse positioning" cycle.

Cycle transitions

Finishing cycles in expanded mode are interrupted at the contour end point. This enables you to sequentially link several finishing cycles and thus finish a selected part of the contour in a single uninterrupted cut.

MANUALplus, however, knows only the contour area of the cycle that is presently being machined. After finishing this contour area, the tool is positioned for a subsequent horizontal contour element. If the subsequent element is not horizontal, the tool is positioned to the contour starting point before the defined contour area is finished. All positioning is done at feed rate speed.

Help graphics

The functionalities and parameters of the cycles are illustrated in the graphic support window. These graphics usually show an external machining operation. The Circle key allows you to switch to the help graphics for internal machining, and

- to switch between the help graphics for internal and external machining.

Elements used in the graphic support window:

- Broken line: Rapid traverse path
- Continuous line: Feed path
- Dimension line with arrow head on one side: "Directional dimension"—the algebraic sign defines the direction.
- Dimension line with arrow head on both sides: "Absolute dimension"—the algebraic sign has no effect.

DIN macros

You can integrate DIN macros in cycle programs. Make sure that the DIN macros do not contain any zero point shifts.

Danger of collision!

Cycle programming: With DIN macros, the zero point shift is reset at the end of the cycle. Therefore, do not use any DIN macros with zero point shifts in cycle programming.

Graphical test run (simulation)

Before executing a cycle, you can graphically test the contour details and the machining sequence (see “Graphic Simulation” on page 68).

Cycle keys

A programmed cycle is not executed until **Cycle START** is pressed. **Cycle STOP** interrupts a running program. With thread cutting, the current cut will be completed before cycle execution is interrupted.

During a cycle interruption you can:

- Resume cycle execution with "Cycle START." The control will always resume execution of the cycle at the point of interruption—even if you have moved the axes in the meantime.
- Move the axes using the jog keys or handwheels.
- Terminate the machining process with the ***Cancel*** soft key.

Switching functions (M functions)

Whether switching functions are triggered automatically or must be commanded manually depends on your specific machine. MANUALplus generates all switching functions that are necessary for running a cycle.

The direction of spindle rotation must be defined in the tool parameters. Using the tool parameters, the cycles generate spindle trigger functions (M3 or M4).

Your machine manual provides further information on automatically triggered switching functions.

Comments

You may assign a comment to an existing cycle. The comment is inserted in brackets "[...]" below the cycle.

Adding or editing comments

Create/select a cycle.

Change text — Press the ***Change text*** soft key.

Enter the comment with the on-screen alphanumeric keyboard.

Save — Transfer the comment.

Cycle menu

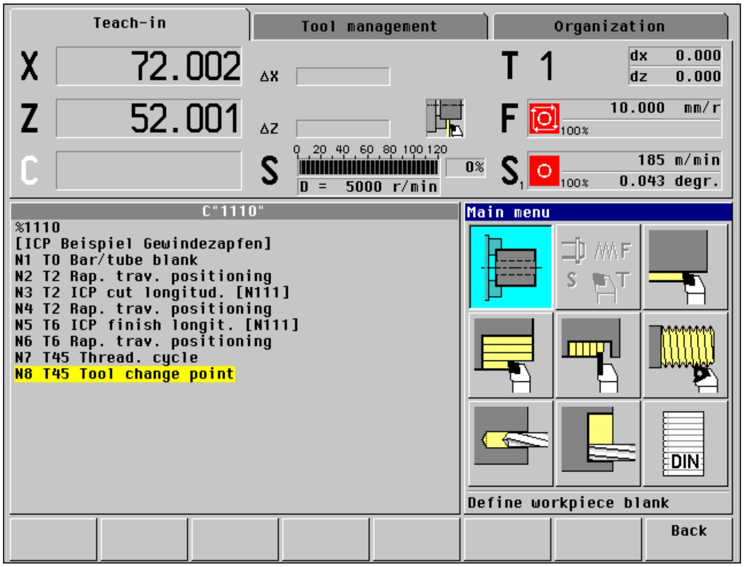
The main menu shows the cycle groups. Once a cycle group has been selected, the soft keys for the individual cycles appear.

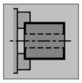
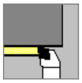
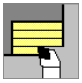
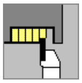
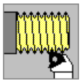
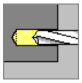
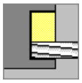
You can use **ICP cycles** for complex contours, and **DIN macros** for technologically sophisticated machining operations (see “ICP Contours” on page 242 and “DIN Programming” on page 278). In cycle programs, the numbers of the ICP contours or DIN macros are at the end of the line of the cycle.

Some cycles offer **optional parameters.** That means, specific contour elements will only be machined if you set the corresponding parameters. The identification letters for optional parameters and parameters that are preassigned default values are displayed in gray.

The following parameters are only required in **Teach-in mode:**

- Starting point X, Z
- Machine data S, F and T



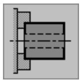
Cycle groups	Menu key
Workpiece blank Define standard workpiece blank or workpiece blank with ICP.	
Single cuts Position at rapid traverse, linear and circular single cuts, chamfers, and rounding arcs.	
Roughing cycles, longitudinal/transverse Roughing and finishing cycles for turning and facing.	
Recessing and recess-turning cycles Cycles for recessing, contour recessing, undercuts and parting.	
Thread cutting Thread cycles, relief turns and thread repair.	
Drilling Drilling cycles and patterns for face and lateral-surface machining.	
Milling Milling cycles and patterns for face and lateral-surface machining.	
DIN macros Integrate program sections written in DIN format into the cycle program.	

Soft keys in cycle programming

Depending on the type of cycle, you define the functions for the cycle by soft key. The table to the right lists the soft keys used in cycle programming.

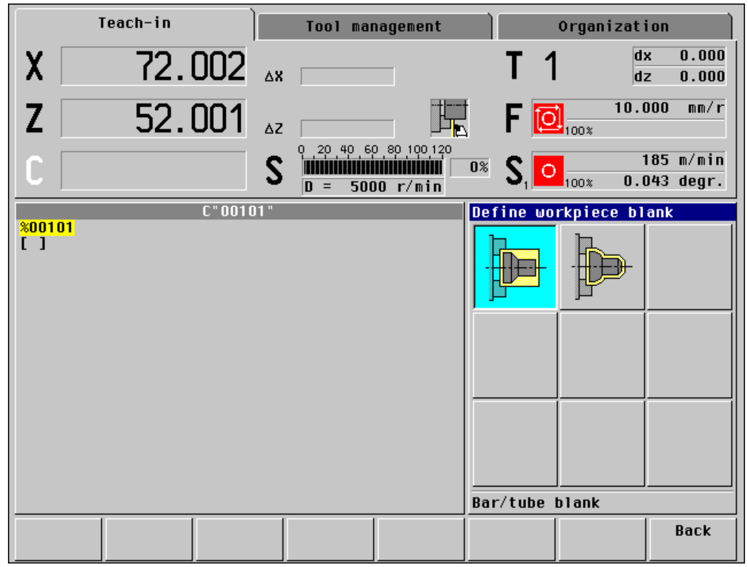
Soft keys in cycle programming	
Edit ICP	Call the ICP editor.
T-Change approach	Approach the tool change position.
Spindle Stop M19	Activate spindle positioning (M19).
With return	■ **On:** Tool returns to the cycle start point. ■ **Off:** Tool remains at cycle end position.
Pattern linear	Linear hole pattern on face or lateral surface.
Pattern circular	Circular hole pattern on face or lateral surface.
Re-cut	■ **On:** Rework existing thread. ■ **Off:** Cut new thread.
Last cut	Repeat the last thread cut.
Tool list	Call the "Tool list." You can transfer the T number from the tool list.
Take over position	Transfer the actual position X, Z.
S,F from tool	Transfer the spindle speed and feed rate from the tool data.
Constant speed	■ **On:** Constant speed (rpm) ■ **Off:** Constant cutting speed (m/min)
Inner thread	■ **On:** Internal thread ■ **Off:** External thread
Input finished	Transfer entered/changed values.
Back	Cancel the current dialog.

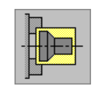
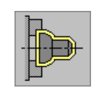
4.2 Workpiece Blank Cycles



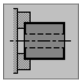
The workpiece blank cycles describe the workpiece blank and the setup used. The workpiece blank cycles do not influence the machining process.

This information is evaluated during the simulation of the machining process.

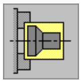


Workpiece blank	Symbol
Blank—bar/tube Define the standard blank.	
ICP workpc. blank contour Define workpiece blank contours with ICP.	

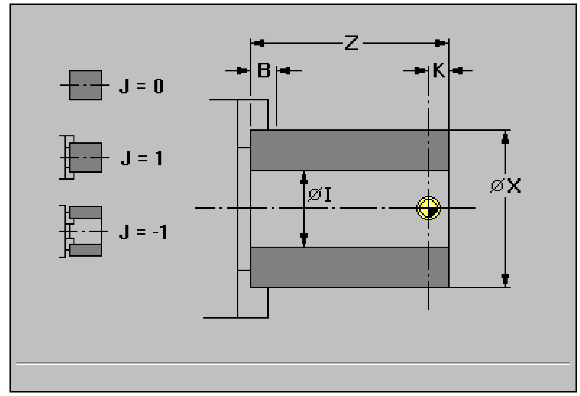
Blank—bar/tube



Select the function for defining a workpiece blank.



Select "Blank—bar/tube."

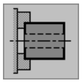


The cycle describes the workpiece blank and the setup used. This information is evaluated during the simulation.

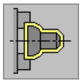
Cycle parameters

- **X outside diameter**
- **Z length**—including transverse allowance and clamping range
- **I inside diameter** for workpiece blank "tube"
- **K right edge** (transverse allowance)
- **B clamping range**
- **J type of clamping**
  - 0: Not clamped
  - 1: Externally clamped
  - 2: Internally clamped

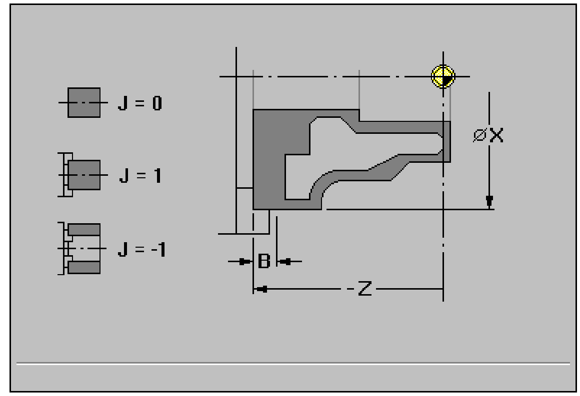
ICP workpc. blank contour



Select the function for defining a workpiece blank.



Select "ICP workpiece blank contour."

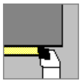


The cycle integrates the workpiece blank defined with ICP and describes the setup used. This information is evaluated during the simulation.

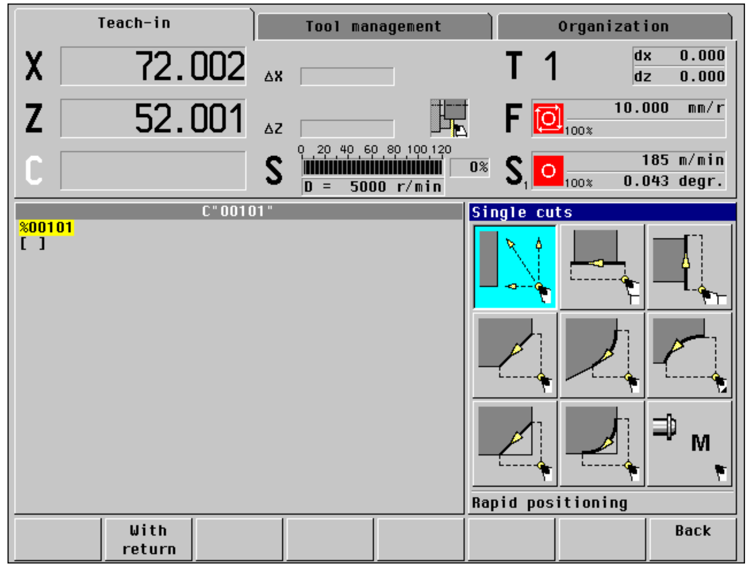
Cycle parameters

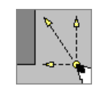
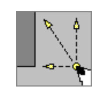
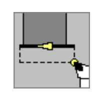
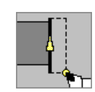
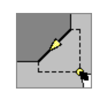
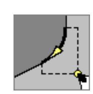
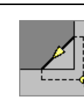
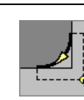
- **X clamping diameter**
- **Z clamping position**
- **B clamping range**
- **J type of clamping**
  - 0: Not clamped
  - 1: Externally clamped
  - 2: Internally clamped
- **N ICP contour number**

4.3 Single Cut Cycles

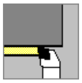


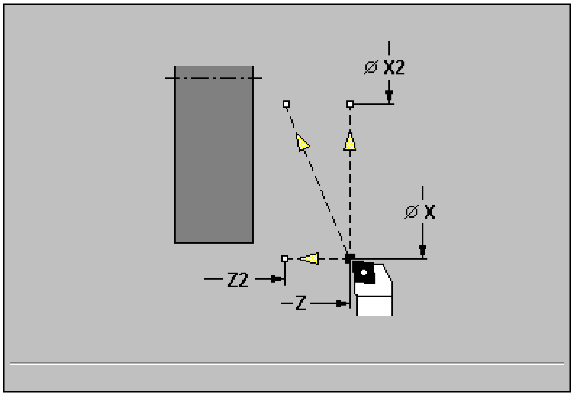
In the single cut cycles you position the tool in rapid traverse, perform linear or circular cuts, machine chamfers or rounding arcs, and enter M functions.



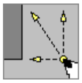
Single cuts	Symbol
Rapid traverse positioning	
Approach the tool change position	
Linear machining, longitudinal/transverse Single longitudinal/transverse cut.	 
Linear machining at angle Single oblique cut.	
Circular machining Single circular cut (for cutting direction, see menu key).	 
Machine a **chamfer.**	
Machine a **rounding.**	
Call an **M function.**	

Rapid traverse positioning





Call the single-cut menu.



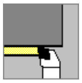
Select the "Rapid traverse positioning" cycle.

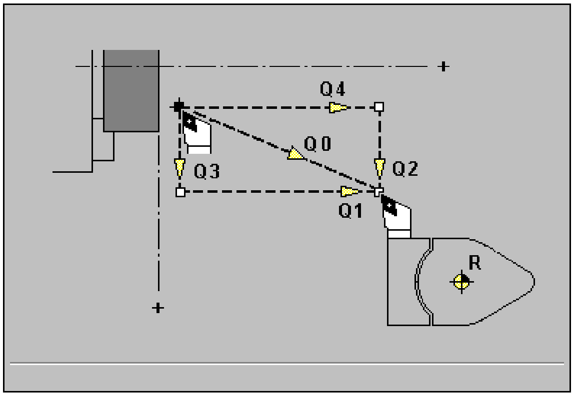
The tool moves at rapid traverse from the starting point to the target point.

Cycle parameters

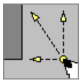
- **`X, Z starting point`**
- **`X2, Z2 target point`**
- **`T tool number`**

Approach the tool change position





Call the single-cut menu.



Select the "Rapid traverse positioning" cycle.

`T-Change approach`

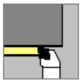
Activate the ***T-Change approach*** function

The tool moves at rapid traverse from the current position to the tool change position (see “Defining the tool change position” on page 52).

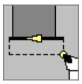
Cycle parameters

- **`Q sequence`**—default: 0
  - 0: Diagonal path of traverse
  - 1: First X, then Z direction
  - 2: First Z, then X direction
  - 3: X direction only
  - 4: Z direction only
- **`T tool number:`** After reaching the tool change position, MANUALplus switches to the tool indicated in "T."

Linear machining, longitudinal



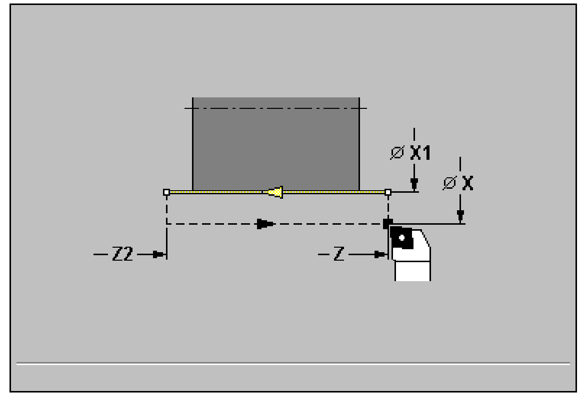
Call the single-cut menu.

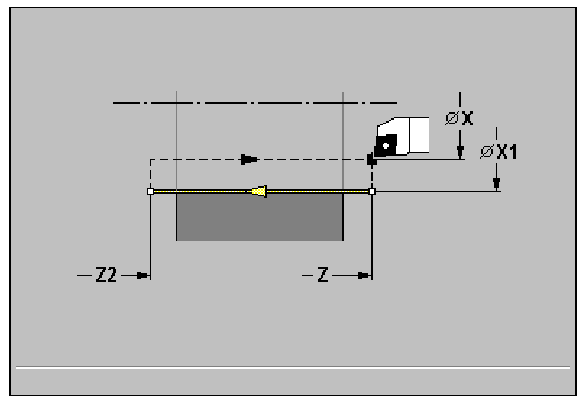


Select the "Longitudinal linear machining" cycle.

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position.
On: Tool returns to the starting point.





Longitudinal linear machining
The tool moves from the starting point to the contour end point at the programmed feed rate and remains at the cycle end position.

Contour linear, longitudinal ("With return")
The tool approaches the workpiece, executes the longitudinal cut and returns to the starting point at the end of cycle (see figures at right).

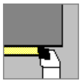
Cycle parameters

- **X, Z starting point**
- **X1 contour starting point**
- **Z2 contour end point**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

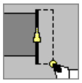
Cycle run (if "With return" is active)

1. Move from X, Z to contour starting point X1.
2. Move to contour end point Z2 at programmed feed rate.
3. Retract and return to starting point on paraxial path.

Linear machining, transverse



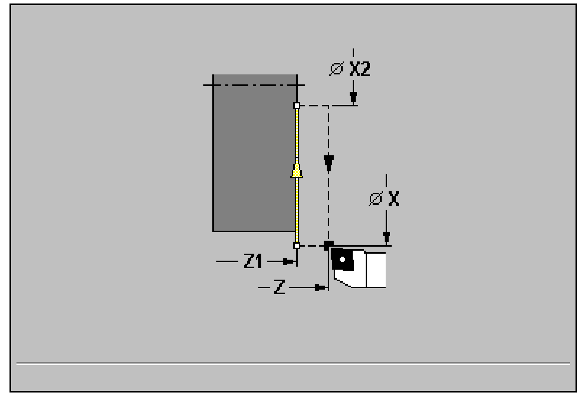
Call the single-cut menu.

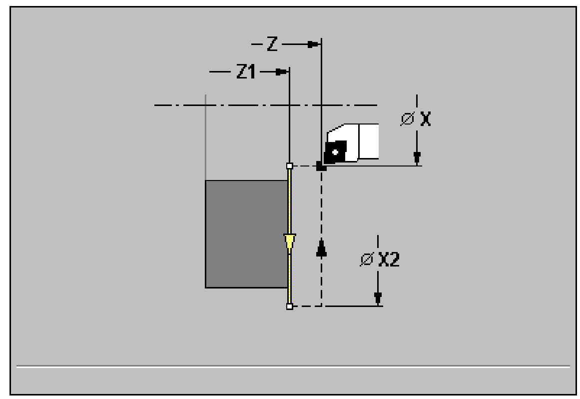


Select the "Transverse linear machining" cycle.

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position.
On: Tool returns to the starting point.





Transverse linear machining
The tool moves from the starting point to the contour end point at the programmed feed rate and remains at the cycle end position.

Contour linear, transverse ("With return")
The tool approaches the workpiece, executes the transverse cut and returns to the starting point at the end of cycle (see figures at right).

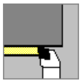
Cycle parameters

- **`X, Z starting point`**
- **`Z1 starting point of contour`** (if "With return" is active)
- **`X2 contour end point`**
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

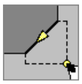
Cycle run (if "With return" is active)

1. Move from X, Z to contour starting point Z1.
2. Move to contour end point X2 at programmed feed rate.
3. Retract and return to starting point on paraxial path.

Linear machining at angle



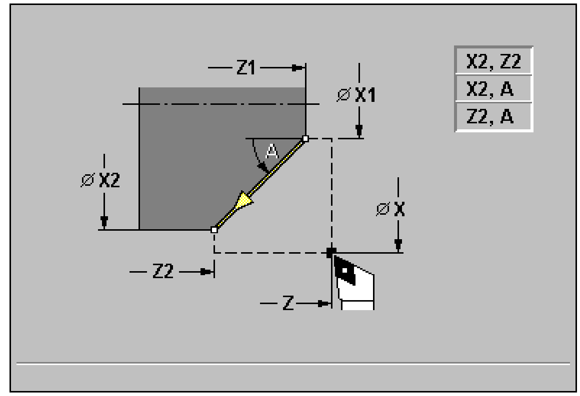
Call the single-cut menu.

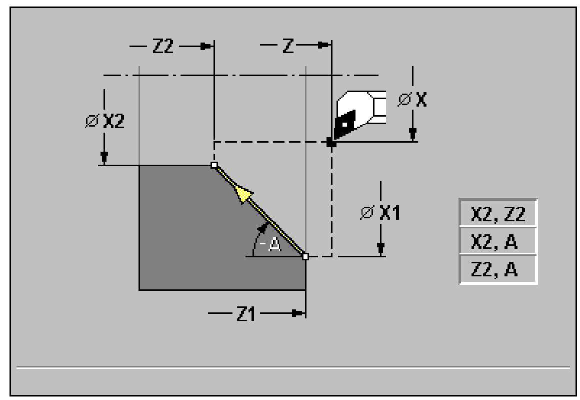


Select the "Linear machining at angle" cycle.

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position.
On: Tool returns to the starting point.





Linear machining at angle
MANUALplus calculates the target position and moves the tool on a straight line from the starting point to the target position at the programmed feed rate. When the cycle is completed, the tool remains at the cycle end position.

Contour linear, at angle ("With return")
MANUALplus calculates the target position. The tool then approaches the workpiece, executes the linear cut and returns to the starting point at the end of cycle (see figures at right). Cutter radius compensation is taken into account.

Cycle parameters

- **X, Z starting point**
- **X1, Z1 starting point of contour** (if "With return" is active)
- **X2, Z2 contour end point**
- **A starting angle**—range: –180° < A < 180°
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

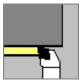
Parameter combinations for defining the target point:

- X2, Z2
- X2, A
- Z2, A

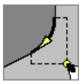
Cycle run (if "With return" is active)

1. Calculate the target position.
2. Move on a straight line from X, Z to contour starting point X1, Z1.
3. Move to target position at programmed feed rate.
4. Retract and return to starting point on paraxial path.

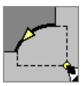
Circular machining



Call the single-cut menu.



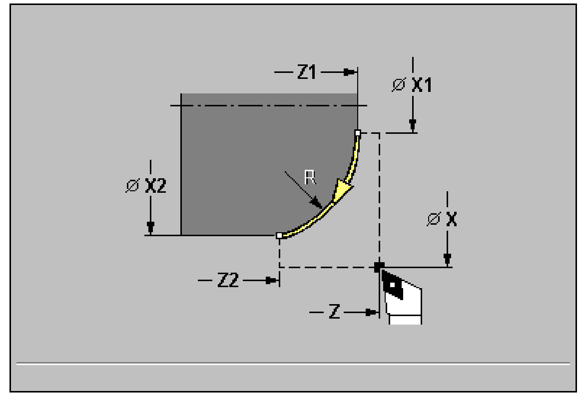
Select the "Circular machining cycle" (clockwise).

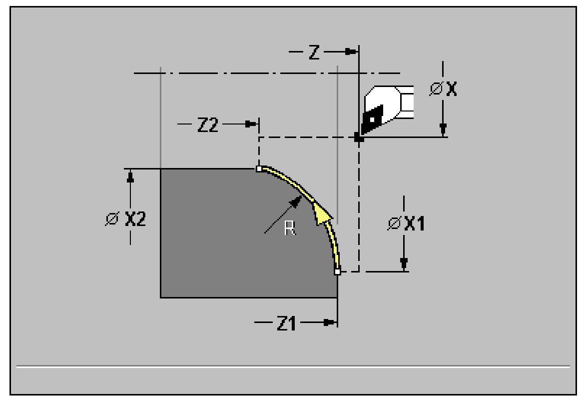


Select the "Circular machining cycle" (counterclockwise).

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position.
On: Tool returns to the starting point.





Circular machining
The tool moves in circular arc from the starting point to the contour end point at the programmed feed rate and remains at the cycle end position.

Contour circular ("With return")
The tool approaches the workpiece, executes the circular cut and returns to the starting point at the end of cycle. Cutter radius compensation is taken into account (see figures at right).

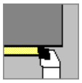
Cycle parameters

- **X, Z starting point**
- **X1, Z1 starting point of contour** (if "With return" is active)
- **X2, Z2 contour end point**
- **R radius** of rounding
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

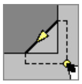
Cycle run (if "With return" is active)

1. Move on paraxial path from X, Z to contour starting point X1, Z1.
2. Move in circular arc to contour end point X2, Z2 at programmed feed rate.
3. Retract and return to starting point on paraxial path.

Chamfer



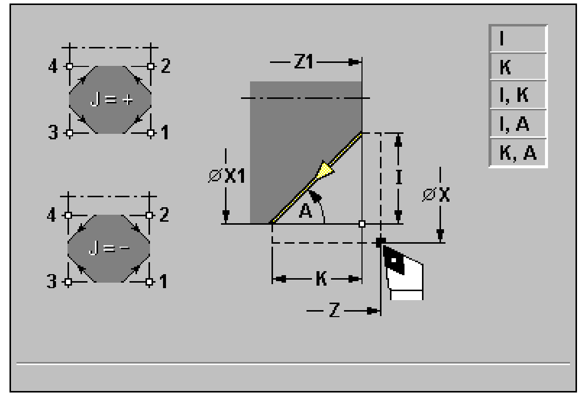
Call the single-cut menu.

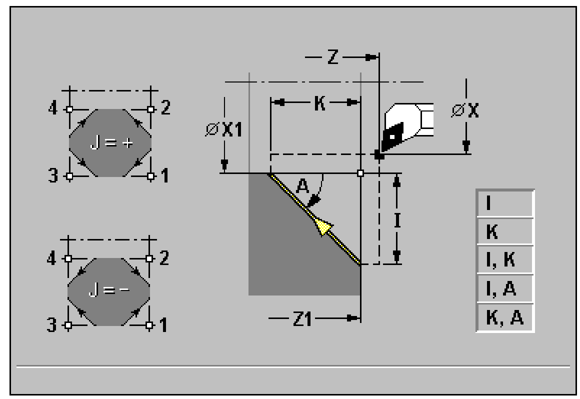


Select the "Chamfer" cycle.

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position.
On: Tool returns to the starting point.





Chamfer
The cycle produces a chamfer that is dimensioned relative to the corner of the workpiece contour. When the cycle is completed, the tool remains at the cycle end position.

Contour chamfer ("With return")
The tool approaches the workpiece, machines the chamfer that is dimensioned relative to the corner of the workpiece contour and returns to the starting point at the end of cycle. Cutter radius compensation is taken into account (see figures at right).

Cycle parameters

- **X, Z starting point**
- **X1, Z1 contour corner**
- **A starting angle:** Angle of the chamfer, range: 0°< A < 90°
- **I, K chamfer width** (in X, Z)
- **J element position** (see graphic support window)—default: 1
  - Position is relative to X1, Z1
  - Algebraic sign determines cutting direction
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

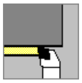
Parameter combinations for defining the chamfer:

- I (45° chamfer)
- K (45° chamfer)
- I, K
- I, A
- K, A

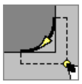
Cycle run (if "With return" is active)

1. Calculate starting point and end point of chamfer.
2. Move on paraxial path from X, Z to starting point of chamfer.
3. Move to end point of chamfer at programmed feed rate.
4. Retract and return to starting point on paraxial path.

Rounding



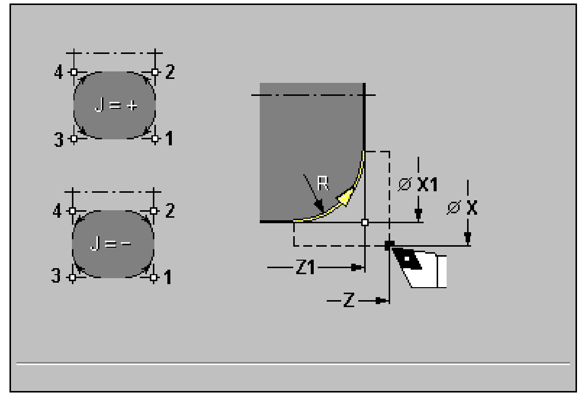
Call the single-cut menu.

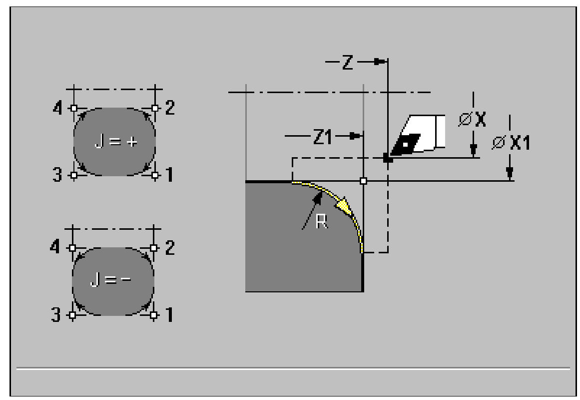


Select the "Rounding" cycle.

`With return`

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position.
On: Tool returns to the starting point.





Rounding
The cycle produces a rounding that is dimensioned relative to the corner of the workpiece contour. When the cycle is completed, the tool remains at the cycle end position.

Contour rounding ("With return")
The tool approaches the workpiece, machines the rounding that is dimensioned relative to the corner of the workpiece contour and returns to the starting point at the end of cycle. Cutter radius compensation is taken into account (see figures at right).

Cycle parameters

- **`X, Z starting point`**
- **`X1, Z1 contour corner`**
- **`R rounding:`** Radius of rounding
- **`I, K chamfer width`** (in X, Z)
- **`J element position`** (see graphic support window)—default: 1
  - Position is relative to X1, Z1
  - Algebraic sign determines cutting direction
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

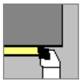
Cycle run (if "With return" is active)

1. Calculate starting point and end point of rounding.
2. Move on paraxial path from X, Z to starting point of rounding.
3. Move in circular arc to end point of rounding at programmed feed rate.
4. Retract and return to starting point on paraxial path.

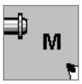
M functions

Machine commands (M functions) are not executed until Cycle START has been pressed. For the meaning of the M functions, refer to your machine manual (see “M Functions” on page 408).

M function



Call the single-cut menu.

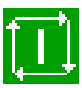


Select "M function."

Enter the number of the M function.

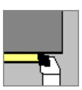
Input finished

Conclude entry.

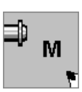


Press Cycle START.

Spindle stop M19 (spindle positioning)



Call the single-cut menu.



Select "M function."

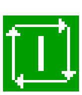
Spindle Stop M19

Switch M19 on.

Enter the stopping angle.

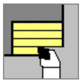
Input finished

Conclude entry.



Press Cycle START.

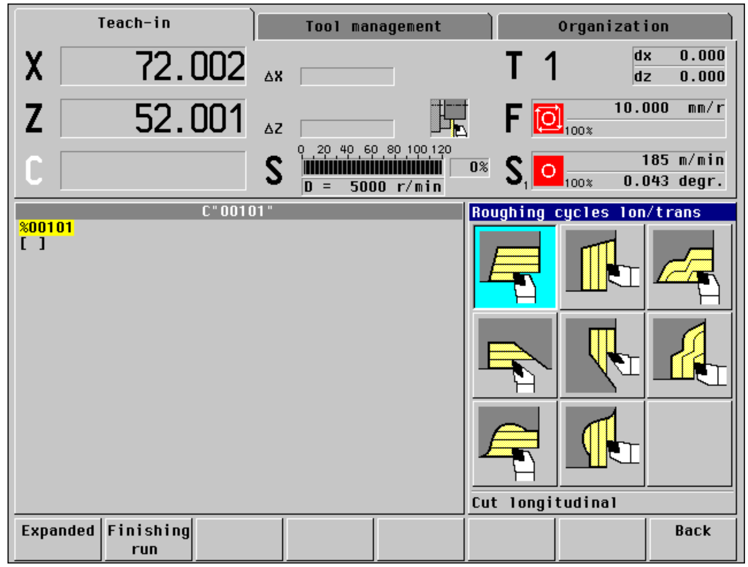
4.4 Roughing Cycles



Roughing cycles rough and finish simple contours in "basic mode" and complex contours in "expanded mode."

With ICP cutting cycles, you can machine contours defined with ICP (see “Editing ICP Contours” on page 243).

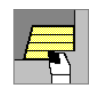
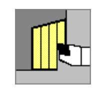
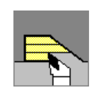
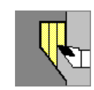
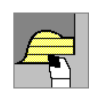
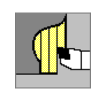
- **Proportioning of cuts:** MANUALplus calculates an infeed that is <= infeed depth P. An "abrasive cut" is avoided.
- **Oversizes:** In "expanded" mode.
- **Cutter radius compensation:** Active.
- **Safety clearance** after each step:
  - Basic mode: 1 mm
  - Expanded mode: The safety clearance is set separately for internal and external machining under "Current parameters—Machining—Safety distances."



Cutting and infeed directions for roughing cycles

MANUALplus automatically determines the cutting and infeed directions from the cycle parameters.

- **Basic mode:** The parameters for starting point X, Z (Manual mode: current tool position) and contour starting point X1 / contour end point Z2 determine these directions.
- **Expanded mode:** The parameters for contour starting point X1, Z1 and contour end point X2, Z2 determine these directions.
- **ICP cycles:** The parameters for contour starting point X, Z (Manual mode: current tool position) and starting point of the ICP contour determine these directions.

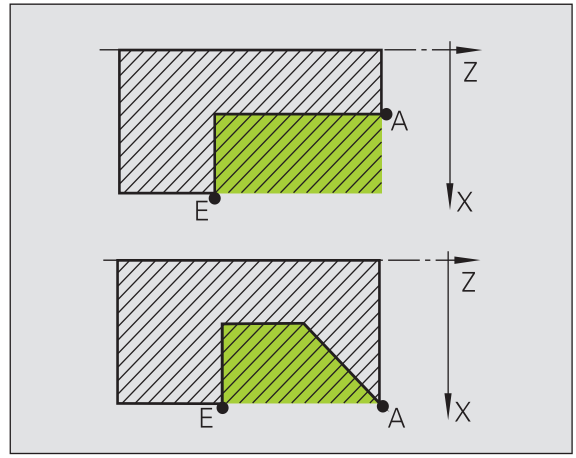
Roughing cycles	Symbol	
Roughing, longitudinal/transverse Roughing and finishing cycle for simple contours		
Plunge-cutting, longitudinal/transverse Roughing and finishing cycle for simple contours		
ICP contour-parallel, longitudinal/ transverse Roughing and finishing cycle for any type of contour		
ICP roughing, longitudinal/transverse Roughing and finishing cycle for any type of contour		

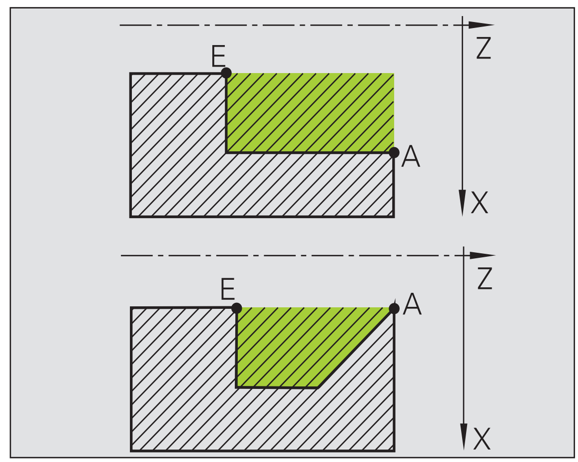
Tool position

It is important that you observe the tool positions (starting point X, Z) before executing any of the roughing cycles in expanded mode. However, they also apply for all cutting and infeed directions as well as for roughing and finishing (see examples of linear cycles in figures at right).

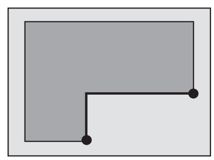
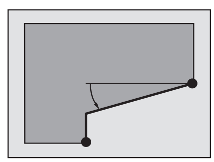
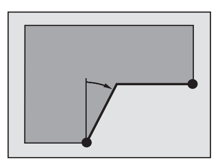
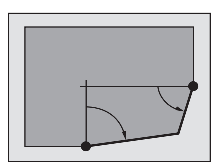
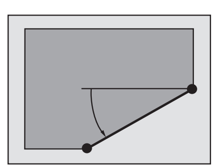
- The starting point must not be located in the shaded area.
- The area to be machined starts at the starting point X, Z if the tool is positioned before the contour area. MANUALplus will otherwise only machine the contour area defined.
- If the starting point X, Z for internal machining is located above the turning center, only the contour area defined will be machined.

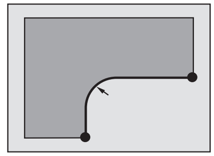
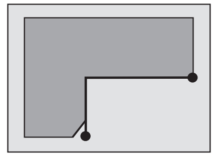
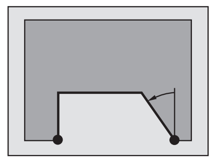
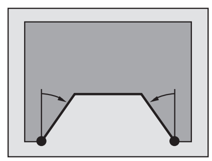
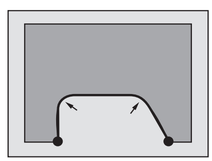
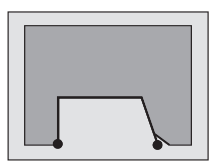
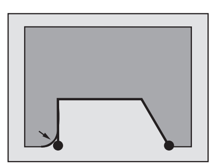
(A = contour starting point X1, Z1; E = contour end point X2, Z2)



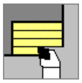


Contour elements

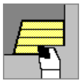
Contour elements in roughing cycles	
Basic mode Machining a rectangular area	
Expanded mode Oblique cut at contour start	
Expanded mode Oblique cut at contour end	
Expanded mode Oblique cuts at contour start and end with angles > 45°	
Expanded mode **One** oblique cut (by entering the starting point of contour, end point of contour **and** starting angle)	

Contour elements in roughing cycles	
Expanded mode Rounding	
Expanded mode Chamfer (or rounding) at contour end	
Basic mode Machining with descending contour	
Basic mode Oblique cut at contour end	
Expanded mode Rounding in contour valley (in both corners)	
Expanded mode Chamfer (or rounding) at contour start	
Expanded mode Chamfer (or rounding) at contour end	

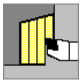
Roughing, longitudinal/transverse



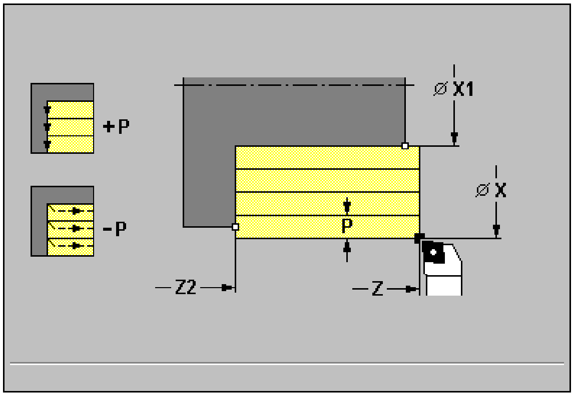
Call the "Roughing, longitudinal/transverse" cycles.

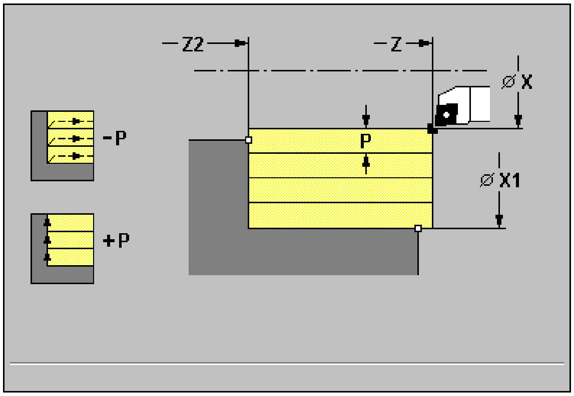


Select "Cut longitudinal" (see figures at right).



Select "Cut transverse" (see figures on the following page).





The "Cut longitudinal" cycle machines the rectangular area defined by X, Z and X1, Z2.

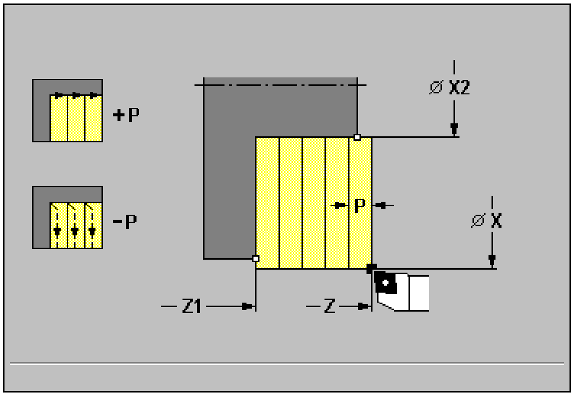
The "Cut transverse" cycle machines the rectangular area defined by X, Z and X2, Z1.

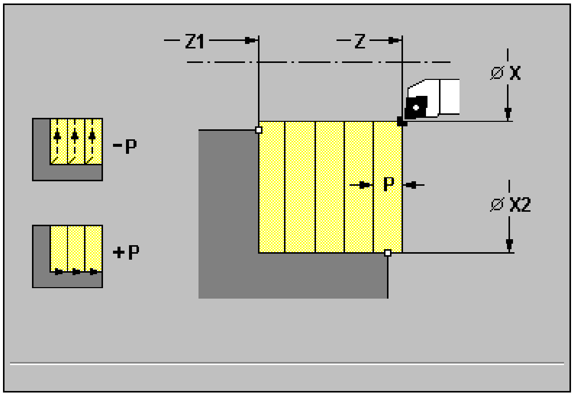
Cycle parameters

- **`X, Z starting point`**
- **`X1 contour starting point`** (roughing longitudinal)
- **`Z2 end point of contour`** (roughing longitudinal)
- **`Z1 contour starting point`** (roughing transverse)
- **`X2 end point of contour`** (roughing transverse)
- **`P infeed depth:`** Maximum infeed depth
  - P>0: Machine contour outline.
  - P<0: Retract by 1 mm at 45°.
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

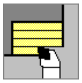
Cycle run

1. Calculate the proportioning of cuts (infeed).
2. Approach workpiece for first pass from X, Z.
3. Move to end point Z2 at programmed feed rate.
4. Depending on algebraic sign of infeed depth P:
   - P>0: Machine contour outline.
   - P<0: Retract at angle of 45°.
5. Retract and approach for next pass.
6. Repeat steps 3 to 5 until X1 or Z1 is reached.
7. Return to starting point on diagonal path.

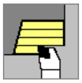




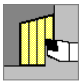
Roughing, longitudinal/transverse—Expanded



Call the "Roughing, longitudinal/transverse" cycles.



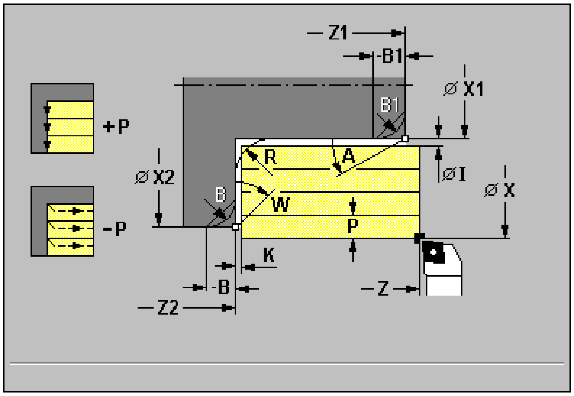
Select "Cut longitudinal" (see figures at right).

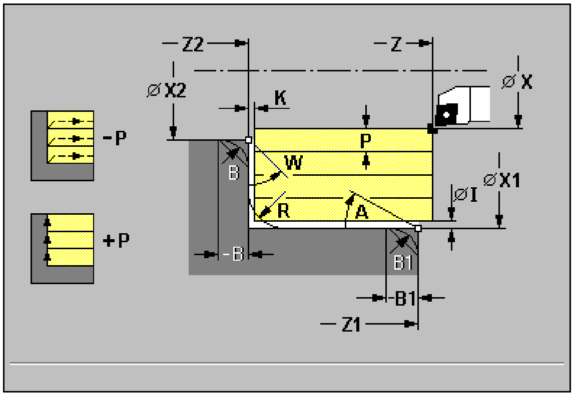


Select "Cut transverse" (see figures on the following page).

Expanded

Press the ***Expanded*** soft key.





The "Cut longitudinal" cycle machines the area defined by X, Z and X1, Z2, taking the oversizes into account.

The "Cut transverse" cycle machines the area defined by X, Z and Z1, X2, taking the oversizes into account.

Cycle parameters

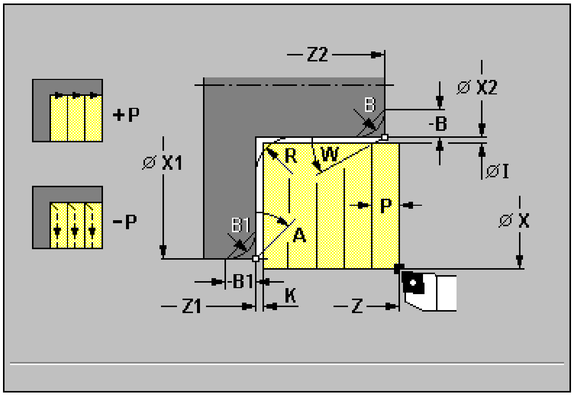
- **`X, Z starting point`**
- **`X1, Z1 contour starting point`**
- **`X2, Z2 contour end point`**
- **`P infeed depth:`** Maximum infeed depth
  - P>0: Machine contour outline.
  - P<0: Retract by the safety clearance at 45°.
- **`A starting angle:`** Range: 0° <= A < 90°
- **`W end angle:`** Range: 0° <= W < 90°
- **`R rounding`**
- **`B, B1 chamfer/rounding`** (B contour end; B1 contour start)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **`I, K oversize X, Z`**
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

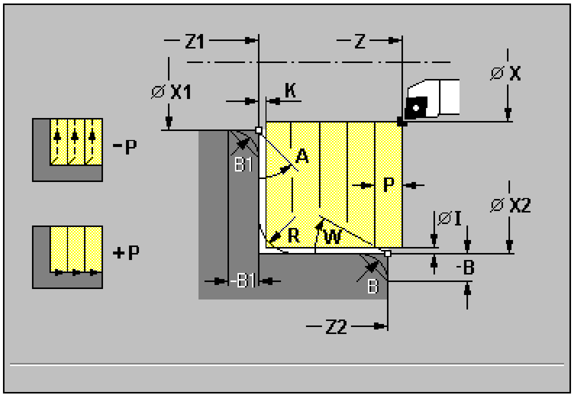
By setting the following **optional parameters,** you can define additional contour elements:

- A: Oblique cut at contour start
- W: Oblique cut at contour end
- R: Rounding
- B: Chamfer/Rounding at contour end
- B1: Chamfer/Rounding at contour start

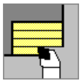
Cycle run

1. Calculate the proportioning of cuts (infeed).
2. Approach workpiece for first pass from X, Z.
3. Move to contour end point Z2 or contour end point X2, or if defined, to one of the optional contour elements at programmed feed rate.
4. Depending on algebraic sign of P:
   - P>0: Machine contour outline.
   - P<0: Retract at angle of 45°.
5. Retract and approach for next pass.
6. Repeat steps 3 to 5 until X1 or Z1 is reached.
7. Return to starting point on paraxial path.

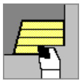




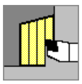
Finishing cut, longitudinal/transverse



Call the "Roughing, longitudinal/transverse" cycles.



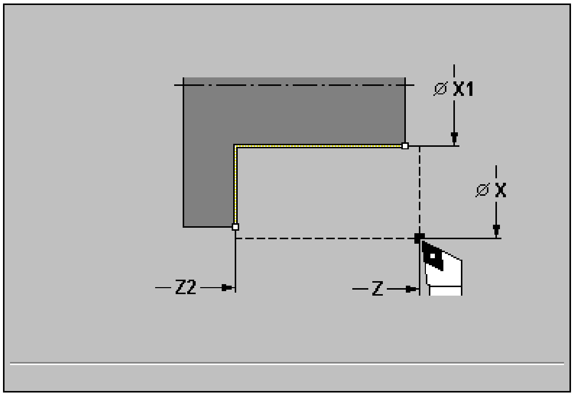
Select "Cut longitudinal" (see figures at right).

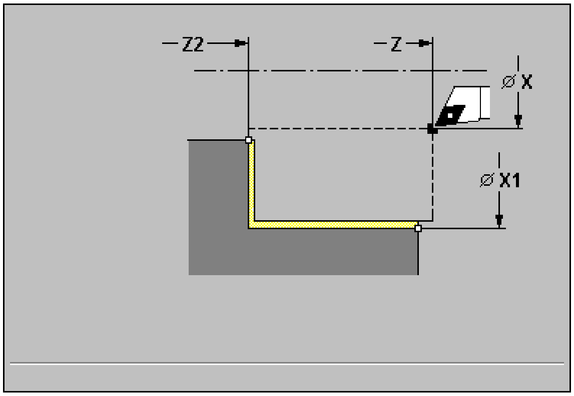


Select "Cut transverse" (see figures on the following page).

Finishing run

Press the ***Finishing run*** soft key.





The cycle "Finishing cut, longitudinal" finishes the contour area from X1 to Z2.

The cycle "Finishing cut, transverse" finishes the contour area from Z1 to X2.

At the end of cycle, the tool returns to the starting point.

Cycle parameters

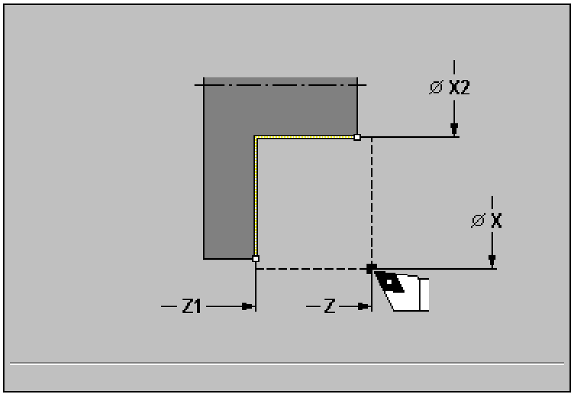
- **X, Z starting point**
- **X1 contour starting point** (finishing cut, longitudinal)
- **Z2 end point of contour** (finishing longitudinal)
- **Z1 contour starting point** (finishing transverse)
- **X2 end point of contour** (finishing transverse)
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

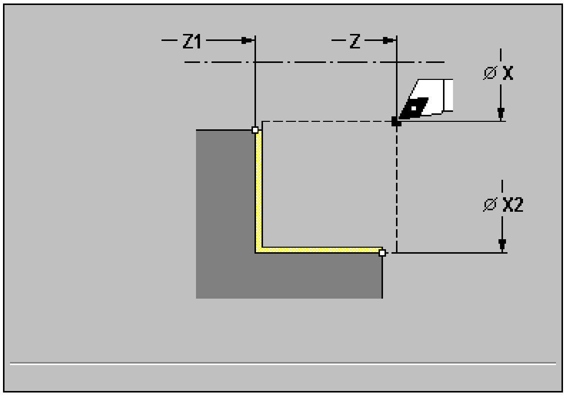
Execution of "Finishing cut, longitudinal" cycle

1. Move in transverse direction from X, Z to contour starting point X1.
2. Finish first in longitudinal direction, then in transverse direction.
3. Return in longitudinal direction to starting point.

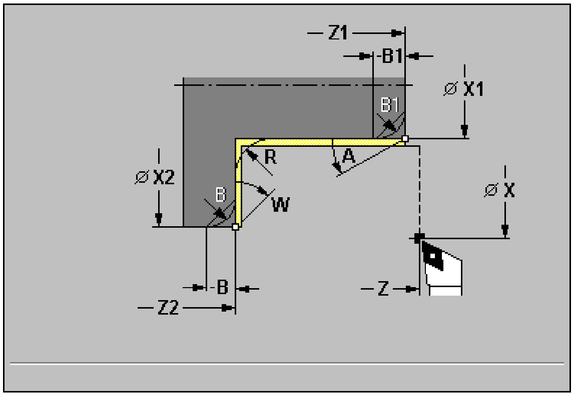
Execution of "Finishing cut, transverse" cycle

1. Move in transverse direction from X, Z to contour starting point Z1.
2. Finish first in transverse direction, then in longitudinal direction.
3. Return in transverse direction to starting point.





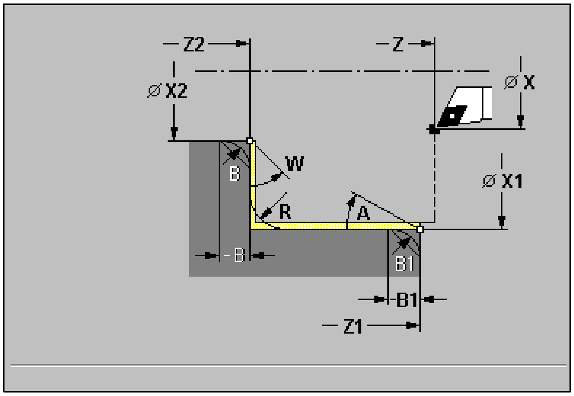
Finishing cut, longitudinal/transverse—Expanded



Call the "Roughing, longitudinal/transverse" cycles.

Select "Cut longitudinal" (see figures at right).

Select "Cut transverse" (see figures on the following page).



Expanded — Press the ***Expanded*** soft key.

Finishing run — Press the ***Finishing run*** soft key.

The cycle finishes the contour area from X1, Z1 to X2, Z2. When the cycle is completed, the tool remains at the cycle end position.

Cycle parameters

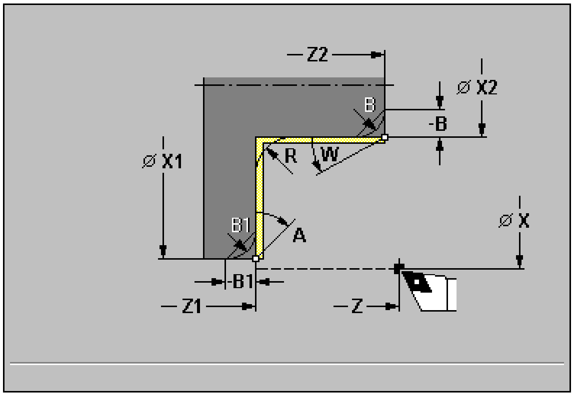
- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **A starting angle:** Range: 0° <= A < 90°
- **W end angle:** Range: 0° <= W < 90°
- **R rounding**
- **B, B1 chamfer/rounding** (B contour end; B1 contour start)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

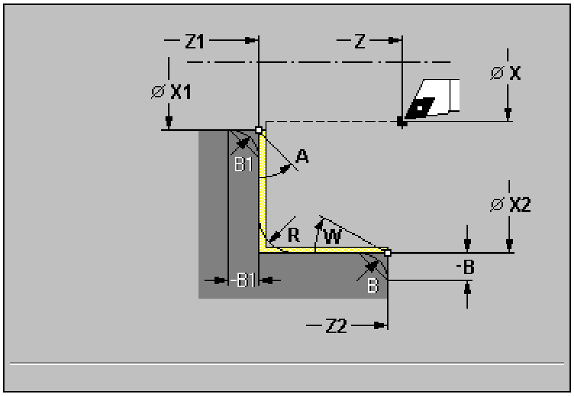
By setting the following **optional parameters,** you can define additional contour elements:

- A: Oblique cut at contour start
- W: Oblique cut at contour end
- R: Rounding
- B: Chamfer/Rounding at contour end
- B1: Chamfer/Rounding at contour start

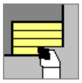
Cycle run

1. Move in transverse direction from X, Z to X1, Z1.
2. Finish contour area from X1, Z1 to X2, Z2, taking optional contour elements into account.

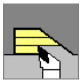




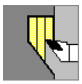
Plunge longitudinal/transverse



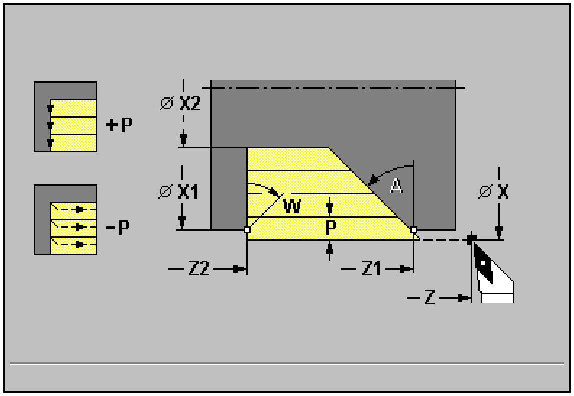
Call the "Roughing, longitudinal/transverse" cycles.

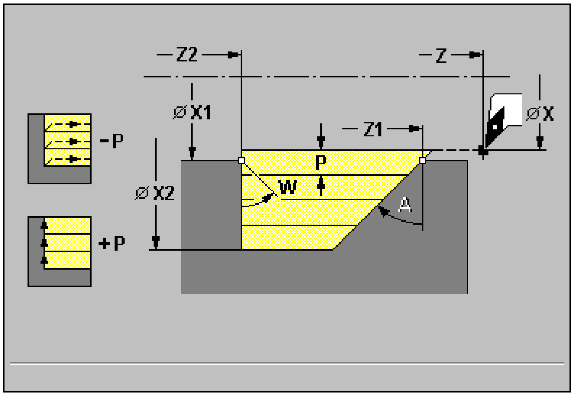


Select "Plunge longitudinal" (see figures at right).



Select "Plunge transverse" (see figures on the following page).





This cycle machines the area defined by X1/Z1, X2/Z2 and plunging angle A.

- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%).
- Pay attention to the dimensions of facing tools (see “Facing tools” on page 419).

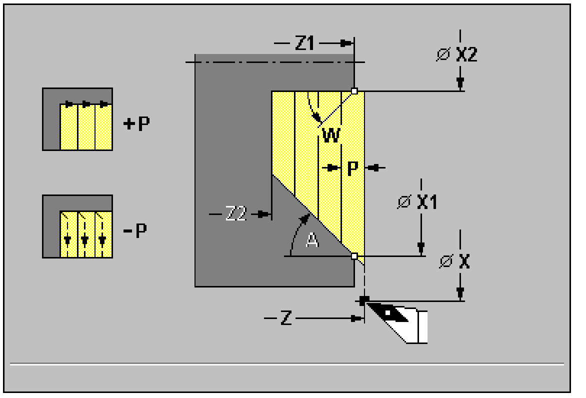
Danger of collision!

If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.

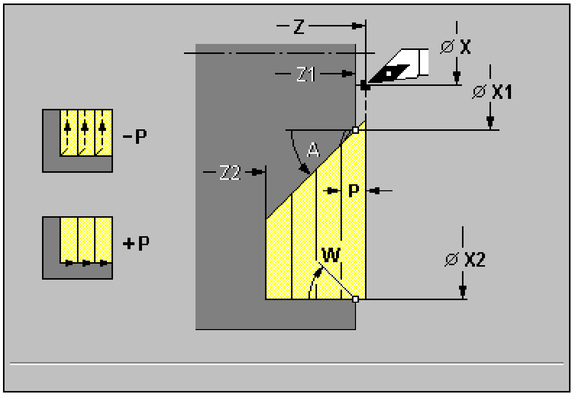
Cycle parameters

- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **P infeed depth:** Maximum infeed depth
  - P>0: Machine contour outline.
  - P<0: Retract by 1 mm at 45°.
- **A plunging angle** (default: 0°): Range: 0° <= A < 90°
- **W end angle:** Oblique cut at contour end—Range: 0° <= W < 90°
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

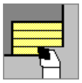
Cycle run



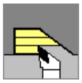
1. Calculate the proportioning of cuts (infeed).
2. Approach workpiece on paraxial path for first pass from X, Z.
3. Plunge-cut at plunging angle A with reduced feed.
4. Move to contour end point Z2 or X2 or, if programmed, to oblique contour element defined by W at programmed feed rate.
5. Depending on algebraic sign of P:
   - P>0: Machine contour outline.
   - P<0: Retract at angle of 45°.
6. Return and approach again for next pass.
7. Repeat steps 3 to 5 until X2 or Z2 is reached.
8. Return to starting point on paraxial path.



Plunge, longitudinal/transverse—Expanded



Call the "Roughing, longitudinal/transverse" cycles.

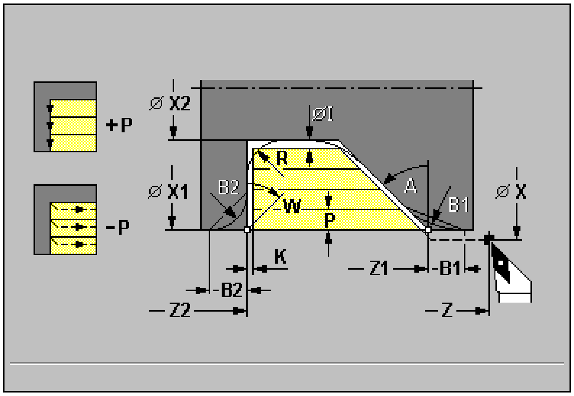


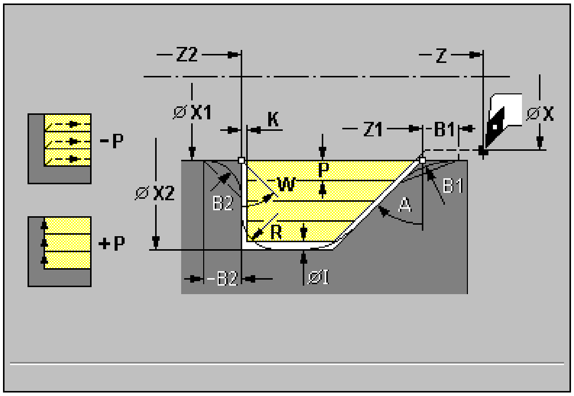
Select "Plunge longitudinal" (see figures at right).

Select "Plunge transverse" (see figures on the following page).

Expanded

Press the ***Expanded*** soft key.





This cycle machines the area defined by X1/Z1, X2/Z2 and plunging angle A, taking the oversizes into account.

- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%).
- Pay attention to the dimensions of facing tools (see “Facing tools” on page 419).

Danger of collision!

If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.

Cycle parameters

- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **P infeed depth:** Maximum infeed depth
  - P>0: Machine contour outline.
  - P<0: Retract by the safety clearance at 45°.
- **A plunging angle** (default: 0°): Range: 0° <= A < 90°
- **W end angle:** Range: 0° <= W < 90°
- **R rounding**
- **B1, B2 chamfer/rounding** (B1 contour start; B2 contour end)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **T tool number**

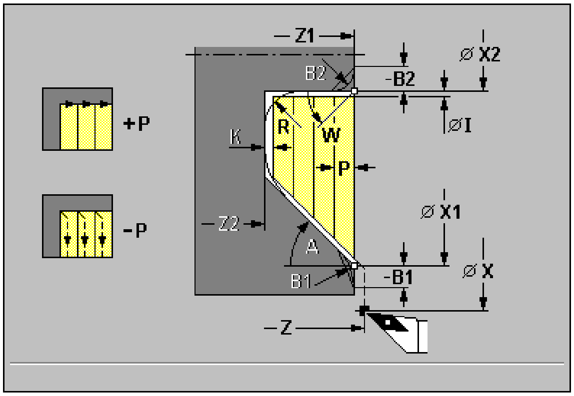
▶ **`S spindle speed / cutting speed`**

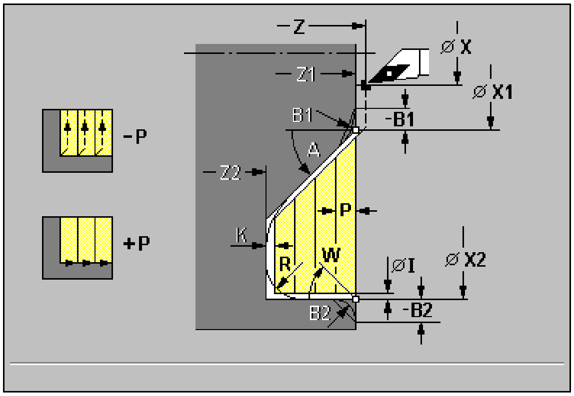
▶ **`F feed per revolution`**

▶ **`I, K oversize X, Z`**

By setting the following **optional parameters,** you can define additional contour elements:

- W: Oblique cut at contour end
- R: Rounding (in both corners of the contour valley)
- B1: Chamfer/Rounding at contour start
- B2: Chamfer/Rounding at contour end

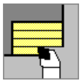


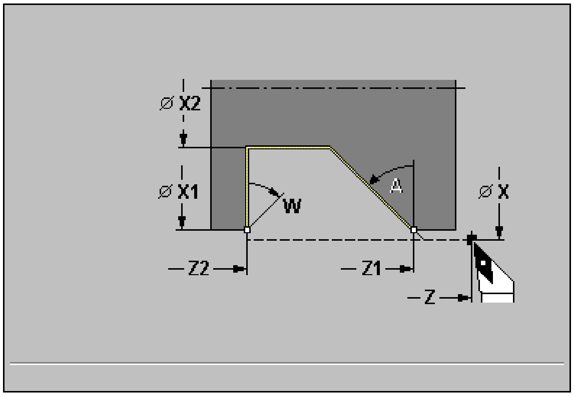


Cycle run

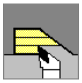
1. Calculate the proportioning of cuts (infeed).
2. Approach workpiece on paraxial path for first pass from X, Z.
3. Plunge-cut at plunging angle A with reduced feed.
4. Move to contour end point Z2 or contour end point X2, or if defined, to one of the optional contour elements at programmed feed rate.
5. Depending on algebraic sign of P:
   - P>0: Machine contour outline.
   - P<0: Retract at angle of 45°.
6. Return and approach for next pass.
7. Repeat steps 3 to 6 until contour end point X2 or Z2 is reached.
8. Return to starting point on paraxial path.

Finishing plunge, longitudinal/transverse





Call the "Roughing, longitudinal/transverse" cycles.

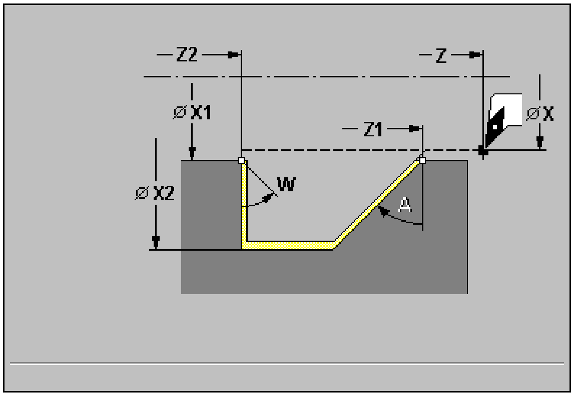


Select "Plunge longitudinal" (see figures at right).

Select "Plunge transverse" (see figures on the following page).

Finishing run

Press the ***Finishing run*** soft key.



The cycle finishes the contour area from X1, Z1 to X2, Z2. At the end of cycle, the tool returns to starting point X, Z.

- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%).
- Pay attention to the dimensions of facing tools (see "Facing tools" on page 419).

Danger of collision!

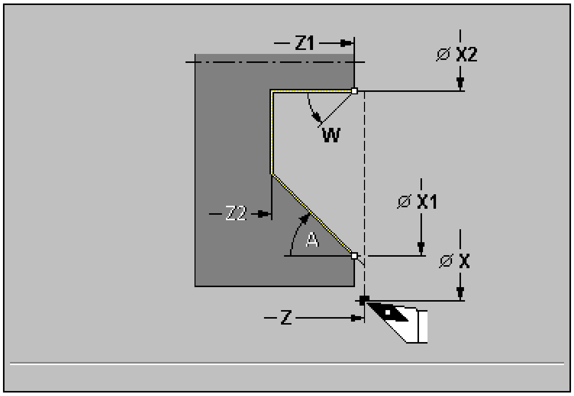
If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.

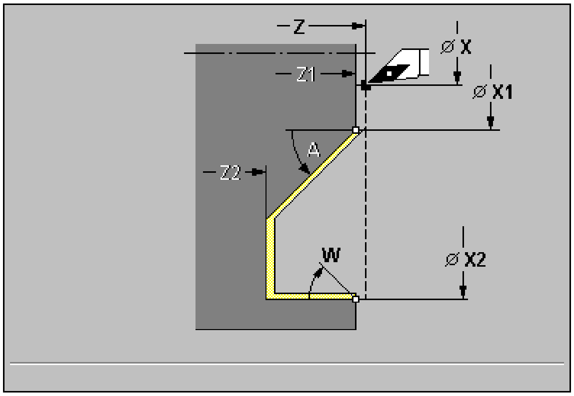
Cycle parameters

- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **A plunging angle** (default: 0°): Range: 0° <= A < 90°
- **W end angle:** Oblique cut at contour end—Range: 0° <= W < 90°
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Cycle run

1. Move in transverse direction from X, Z to contour starting point X1, Z1.
2. Finish defined contour area.
3. Return to starting point on paraxial path.





Finishing plunge, longitudinal/transverse—Expanded

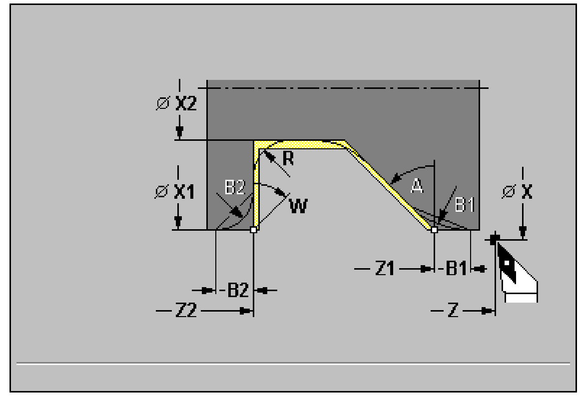
Call the "Roughing, longitudinal/transverse" cycles.

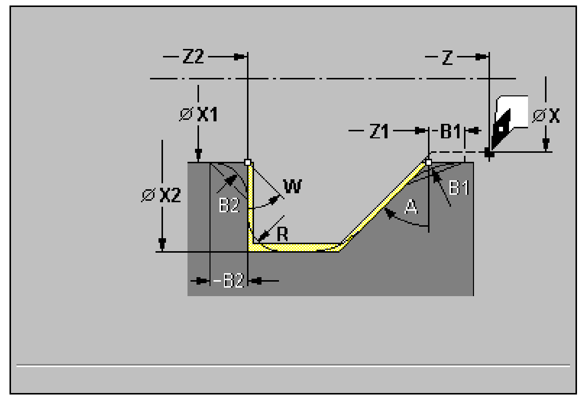
Select "Plunge longitudinal" (see figures at right).

Select "Plunge transverse" (see figures on the following page).

Expanded — Press the ***Expanded*** soft key.

Finishing run — Press the ***Finishing run*** soft key.





The cycle finishes the contour area from X1, Z1 to X2, Z2. When the cycle is completed, the tool remains at the cycle end position.

- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%).
- Pay attention to the dimensions of facing tools (see “Facing tools” on page 419).

Danger of collision!

If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.

Cycle parameters

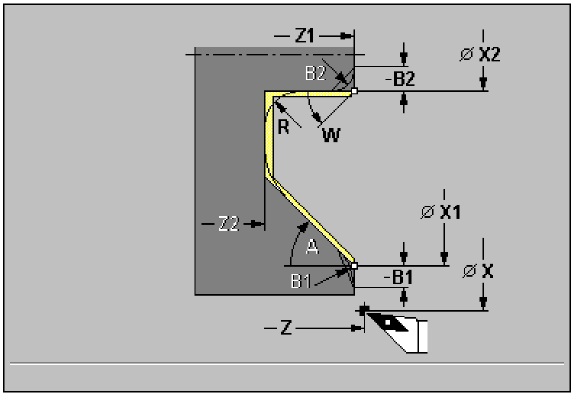
- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **A plunging angle** (default: 0°): Range: 0° <= A < 90°
- **W end angle:** Range: 0° <= W < 90°
- **R rounding**
- **B1, B2 chamfer/rounding** (B1 contour start; B2 contour end)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

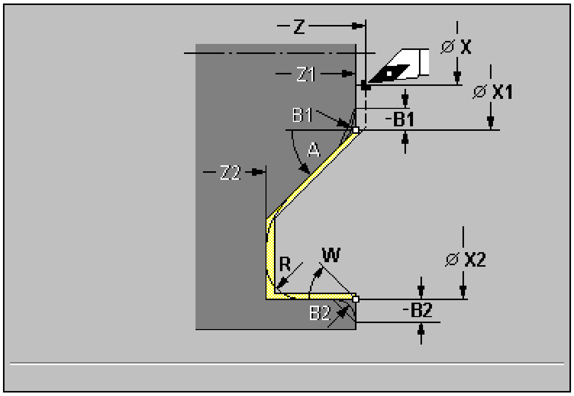
By setting the following **optional parameters,** you can define additional contour elements:

- W: Oblique cut at contour end
- R: Rounding (in both corners of the contour valley)
- B1: Chamfer/Rounding at contour start
- B2: Chamfer/Rounding at contour end

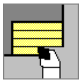
Cycle run

1. Move on paraxial path from X, Z to contour starting point X1, Z1.
2. Finish defined contour area, taking optional contour elements into account.





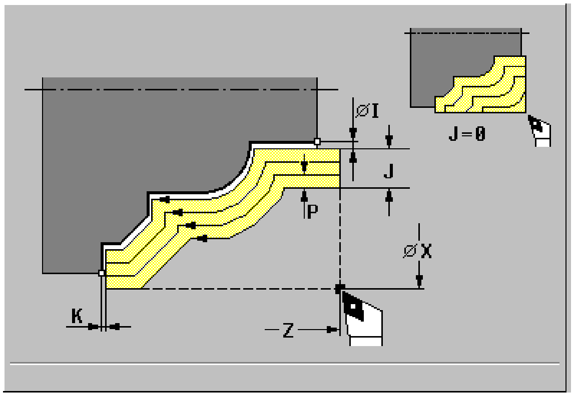
ICP contour-parallel, longitudinal/transverse

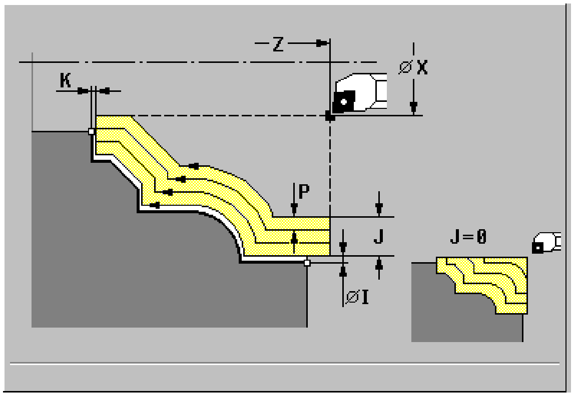


Call the "Roughing, longitudinal/transverse" cycles.

Select "ICP contour-parallel, longitudinal" (see figures at right).

Select "ICP contour-parallel, transverse" (see figures on the following page).





The cycle machines **parallel to the contour,** depending on the J parameter:

- J=0: The area defined by X, Z and the ICP contour, taking the oversizes into account.
- J>0: The area defined by the ICP contour (plus oversizes) and the "workpiece blank oversize J."

Danger of collision!

- If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.
- For **workpiece blank oversize J>0**: Set the "infeed depth P" to the smaller infeed, if the maximum infeed differs for the longitudinal and transverse directions due to the cutting geometry.

Cycle parameters

- **X, Z starting point**
- **P infeed depth**—the infeed depth is determined taking J into account:
  - J=0: P is the maximum infeed depth. The cycle reduces the infeed depth if the programmed infeed is not possible in the transverse or longitudinal direction due to the cutting geometry.
  - J>0: P is the infeed depth. This infeed is used in the longitudinal and transverse directions.
- **I, K oversize X, Z**
- **N ICP contour number**
- **J workpiece blank oversize**—the cycle machines:
  - J=0: From the current tool position.
  - J>0: The area defined by the workpiece blank oversize.
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Cycle run

1 Calculate the proportioning of cuts (infeed), taking the parameter J into account.
   - J=0: The cutting geometry is taken into account. This may result in the use of different infeeds for the longitudinal and transverse directions.
   - J>0: The same infeed is used for both the longitudinal and the transverse direction.

2 Approach workpiece on paraxial path for first pass from X, Z.

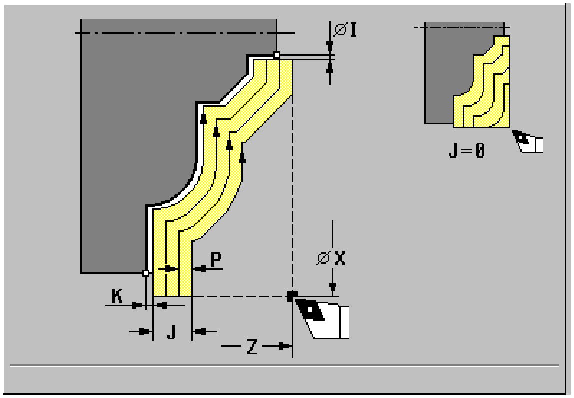
3 Machine the workpiece according to the calculated proportioning of cuts.

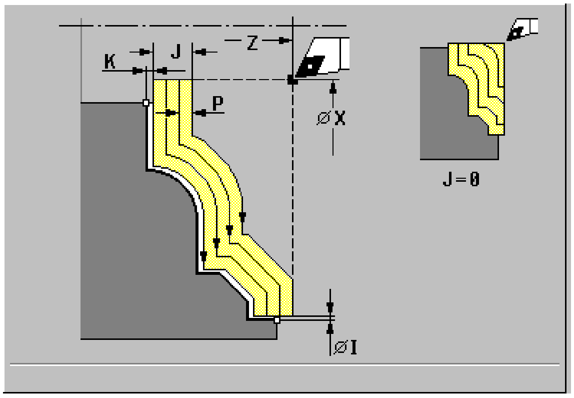
4 Return and approach for next pass.

5 Repeat 3 to 4 until the defined area has been machined.

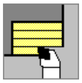
6 Return to starting point on paraxial path.

The cycle parameter **`workpiece blank oversize J`** is available as of NC software versions 507 807-16 and 526 488-08. With earlier software versions, the cycle starts the machining operation from the current tool position.





ICP contour-parallel finishing, longitudinal/ transverse



Call the "Roughing, longitudinal/transverse" cycles.

Select "ICP contour-parallel, longitudinal" (see figures at right).

Select "ICP contour-parallel, transverse" (see figures on the following page).

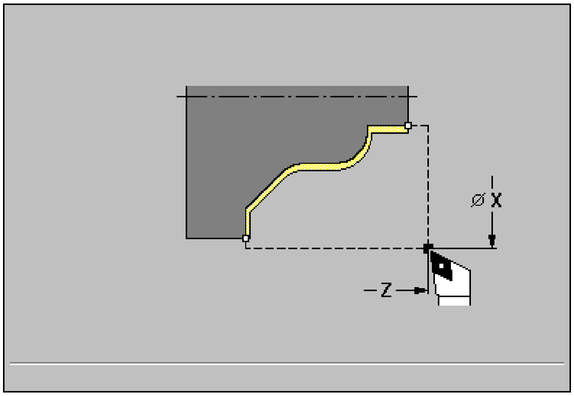
Finishing run

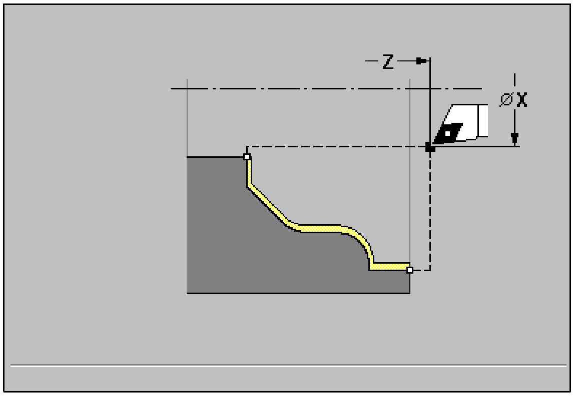
Press the ***Finishing run*** soft key.

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.

Danger of collision!

If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.



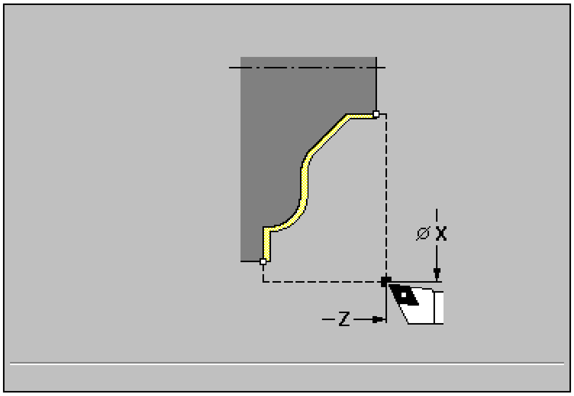


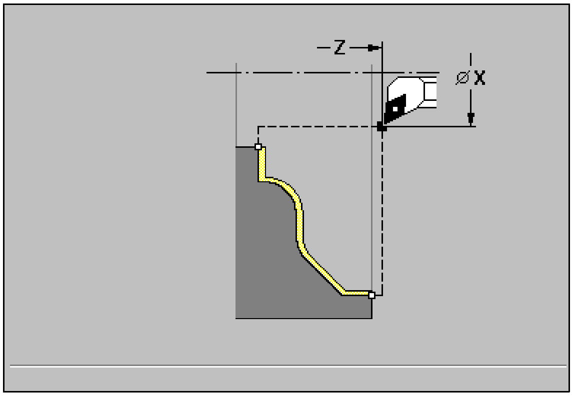
Cycle parameters

- **X, Z starting point**
- **N ICP contour number**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

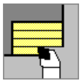
Cycle run

1 Move on paraxial path from X, Z to contour starting point.
2 Finish defined contour area.





ICP roughing, longitudinal/transverse



Call the "Roughing, longitudinal/transverse" cycles.

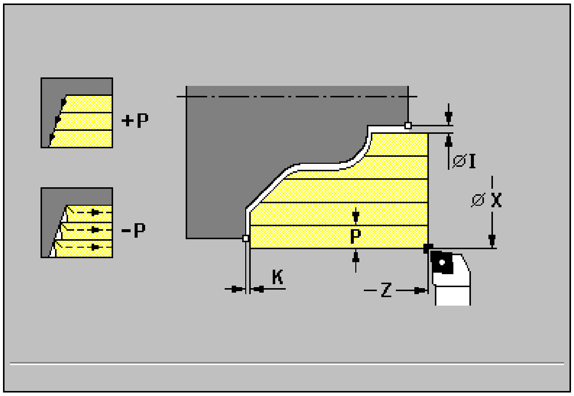
Select "ICP cut, longitudinal" (see figures at right).

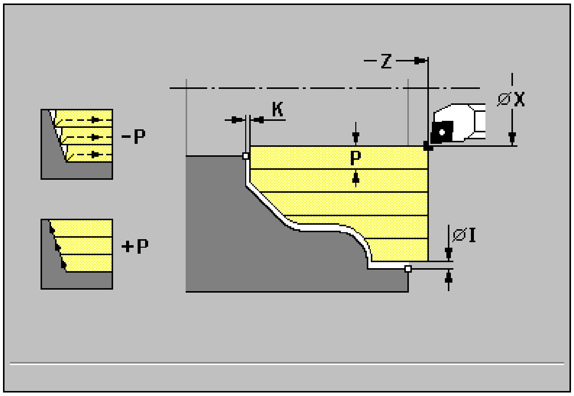
Select "ICP cut, transverse" (see figures on the following page).

The cycle machines the area defined by X, Z and the ICP contour, taking the oversizes into account.

Danger of collision!

If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.



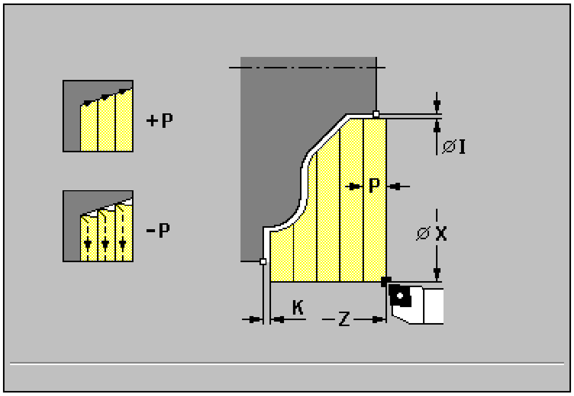


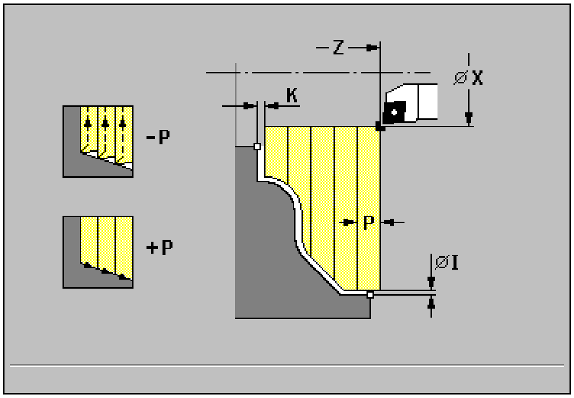
Cycle parameters

- **X, Z starting point**
- **P infeed depth:** Maximum infeed depth
  - P>0: Machine contour outline.
  - P<0: Retract by the safety clearance at 45°.
- **I, K oversize X, Z**
- **N ICP contour number**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Cycle run

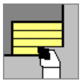
1. Calculate the proportioning of cuts (infeed).
2. Approach workpiece on paraxial path for first pass from X, Z.
3. For sloping contours, plunge into the material at reduced feed rate.
4. Machine the workpiece according to the calculated proportioning of cuts.
5. Depending on algebraic sign of P:
   - P>0: Machine contour outline.
   - P<0: Retract by the safety clearance at 45°.
6. Return and approach for next pass.
7. Repeat 3 to 6 until the defined area has been machined.
8. Return to starting point on paraxial path.





- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%).
- Pay attention to the dimensions of facing tools (see “Facing tools” on page 419).

ICP finishing, longitudinal or transverse



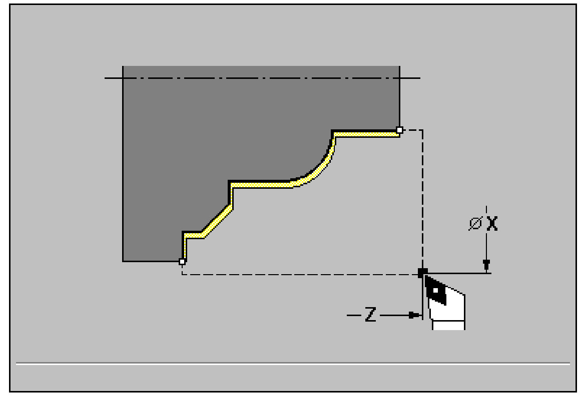
Call the "Roughing, longitudinal/transverse" cycles.

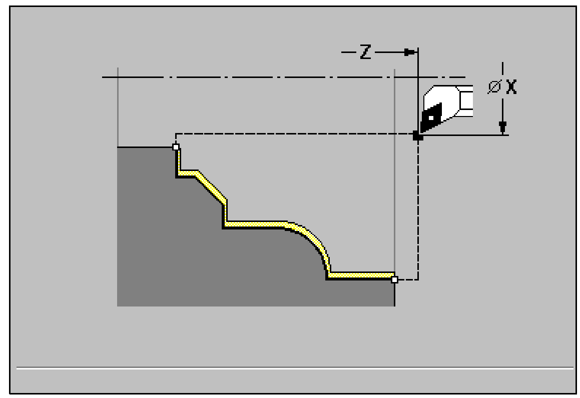
Select "ICP cut, longitudinal" (see figures at upper and center right).

Select "ICP cut, transverse" (see figures on the following page).

Finishing run

Press the ***Finishing run*** soft key.





The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.

Danger of collision!

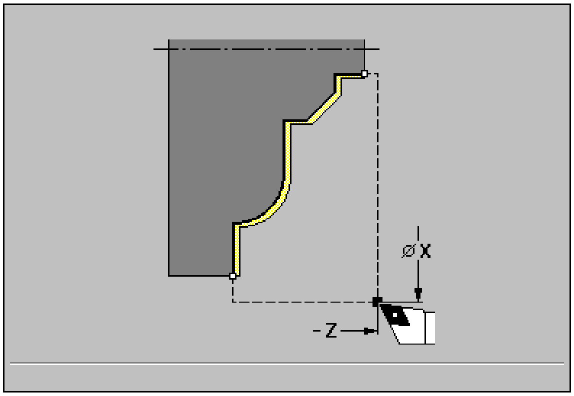
If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.

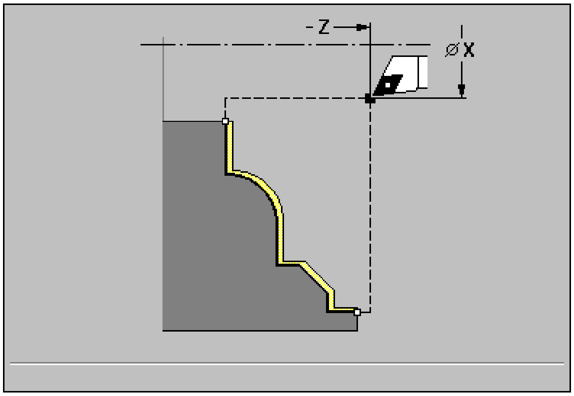
Cycle parameters

- **X, Z starting point**
- **I, K oversize X, Z**
- **N ICP contour number**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Cycle run

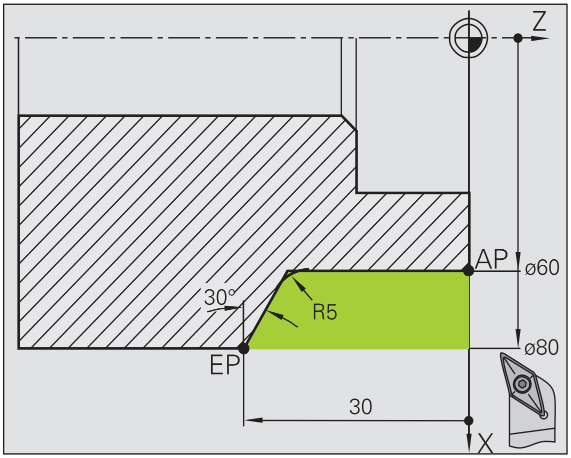
1. Move on paraxial path from X, Z to contour starting point.
2. Finish defined contour area.

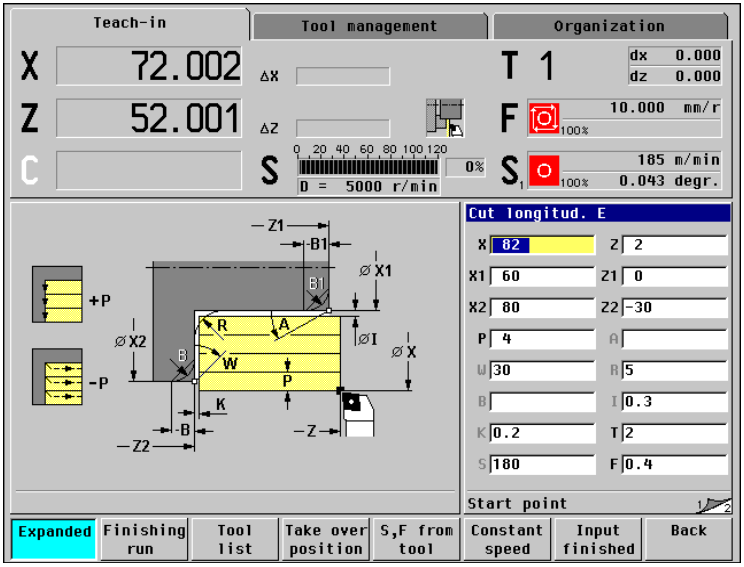


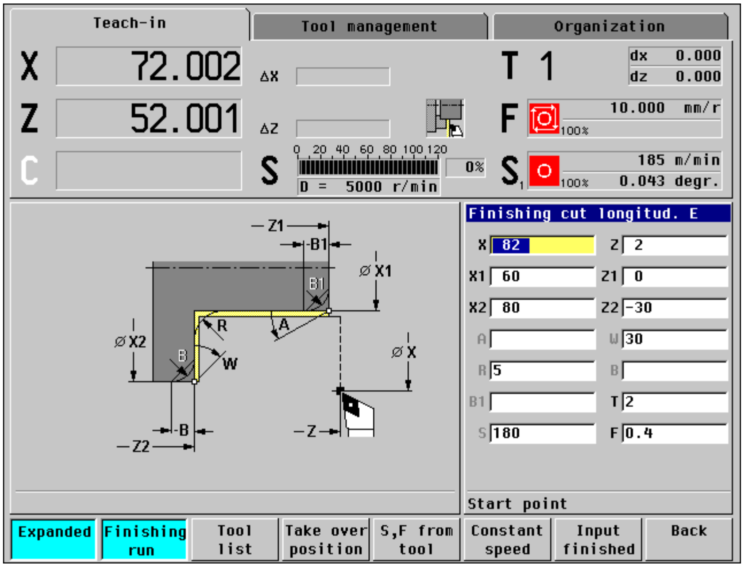


Examples of roughing cycles

Roughing and finishing an outside contour







The shaded area from "AP" (starting point of contour) to "EP" (contour end point) is first rough-machined with the cycle "Cut longitudinal—Expanded," taking oversizes into account (see figure at upper right). This contour area is to be finished subsequently with the cycle "Finishing cut longitudinal—Expanded" (see figure at lower right).

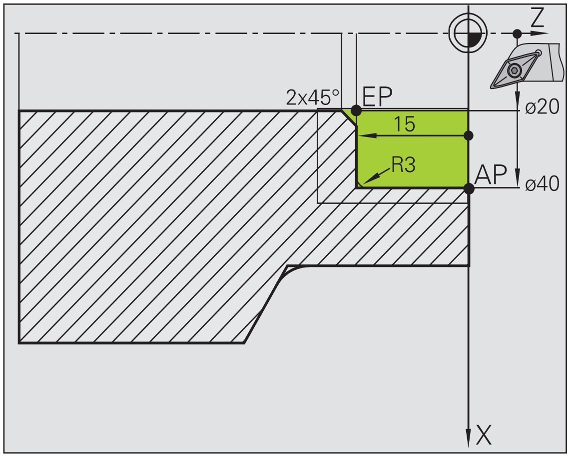
The rounding and the oblique cut at the contour end are also machined in "expanded mode."

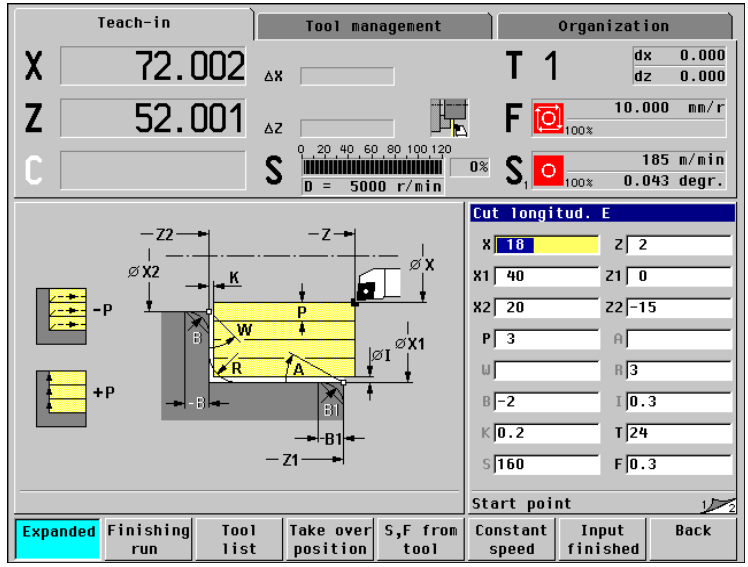
The parameters for contour starting point X1, Z1 and contour end point X2, Z2 determine the cutting and infeed directions—in this example, external machining and infeed in negative X-axis direction.

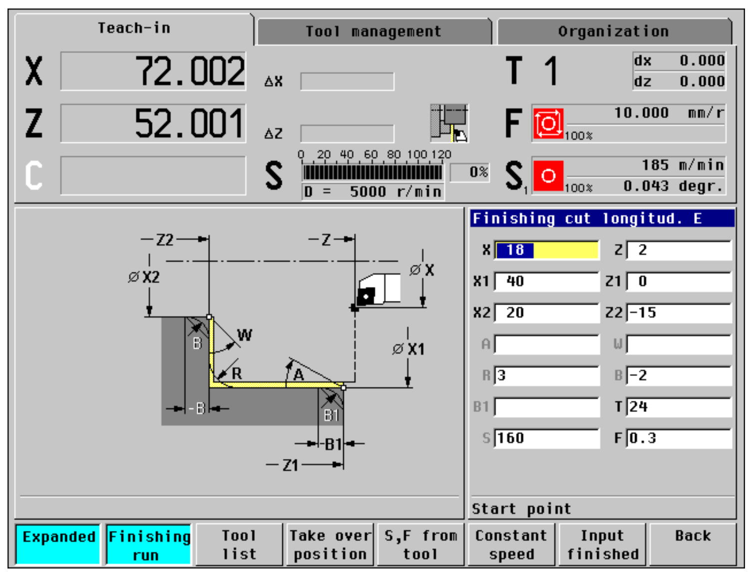
Tool data

- Lathe tool (for external machining)
- WO = 1—Tool orientation
- A = 93°—Tool angle
- B = 55°—Point angle

Roughing and finishing an inside contour







The shaded area from "AP" (starting point of contour) to "EP" (contour end point) is first rough-machined with the cycle "Cut longitudinal—Expanded," taking oversizes into account (see figure at upper right). This contour area is to be finished subsequently with the cycle "Finishing cut longitudinal—Expanded" (see figure at lower right).

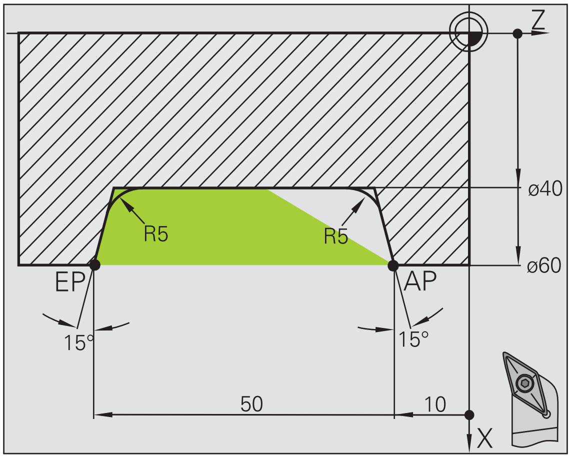
The rounding and the chamfer at the contour end are also machined in "expanded mode."

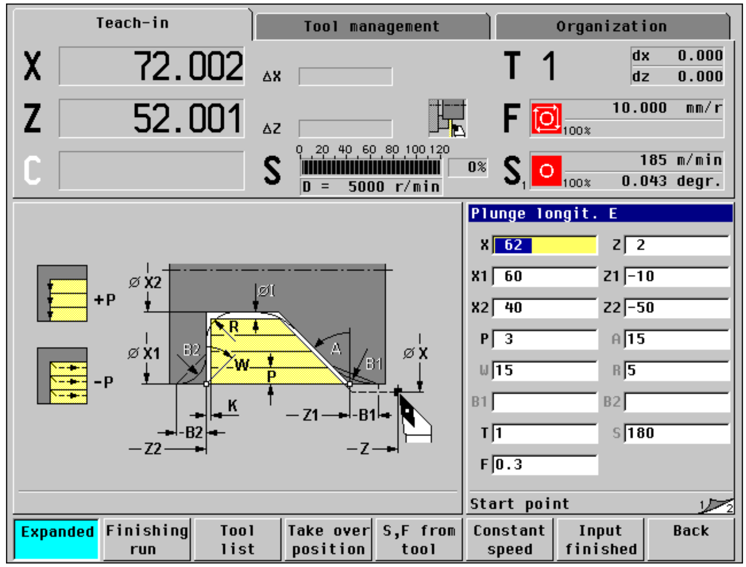
The parameters for contour starting point X1, Z1 and contour end point X2, Z2 determine the cutting and infeed directions—in this example, internal machining and infeed in negative X-axis direction.

Tool data

- Lathe tool (for internal machining)
- WO = 7—Tool orientation
- A = 93°—Tool angle
- B = 55°—Point angle

Roughing (recess clearance) with plunge cycle





The tool to be used cannot plunge at the required angle of 15°. The roughing process for the area therefore requires two steps.

First step:

The shaded area from "AP" (starting point of contour) to "EP" (contour end point) is rough-machined with the cycle "Plunge longitudinal—Expanded," taking oversizes into account.

The "starting angle A" is defined with 15°, as specified in the workpiece drawing. From the tool parameters, MANUALplus automatically calculates the maximum plunging angle that can be achieved with the programmed tool. The resulting contour will not be complete and will be reworked in the second step.

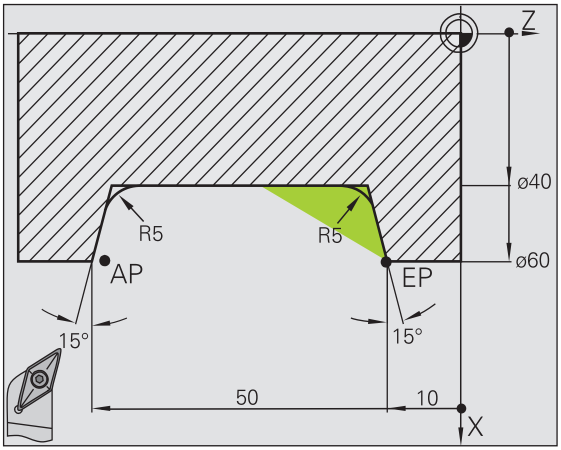
The rounding arcs in the contour valley are also machined in "expanded mode."

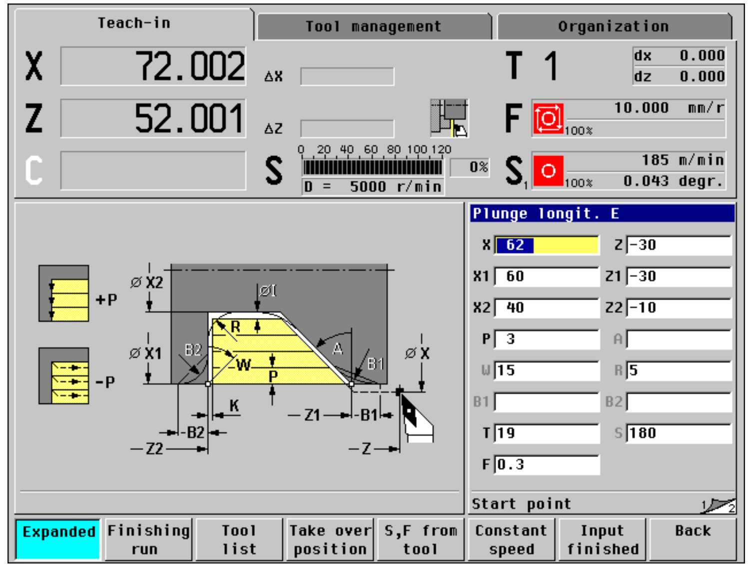
Be sure to enter the correct values for the parameters "contour starting point X1, Z1" and "contour end point X2, Z2." These parameters determine the cutting and infeed directions—in this example, external machining and infeed in negative X-axis direction.

Tool data

- Lathe tool (for external machining)
- WO = 1—Tool orientation
- A = 93°—Tool angle
- B = 55°—Point angle

Second step:





The area that was left out in the first step (shaded area in top left figure) is machined with the cycle "Plunge, longitudinal—Expanded." Before executing this step, you must change tools.

The rounding arcs in the contour valley are also machined in "expanded mode."

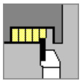
The parameters for contour starting point X1, Z1 and contour end point X2, Z2 determine the cutting and infeed directions—in this example, external machining and infeed in negative X-axis direction.

The parameter for contour starting point Z1 was determined during simulation of the first step.

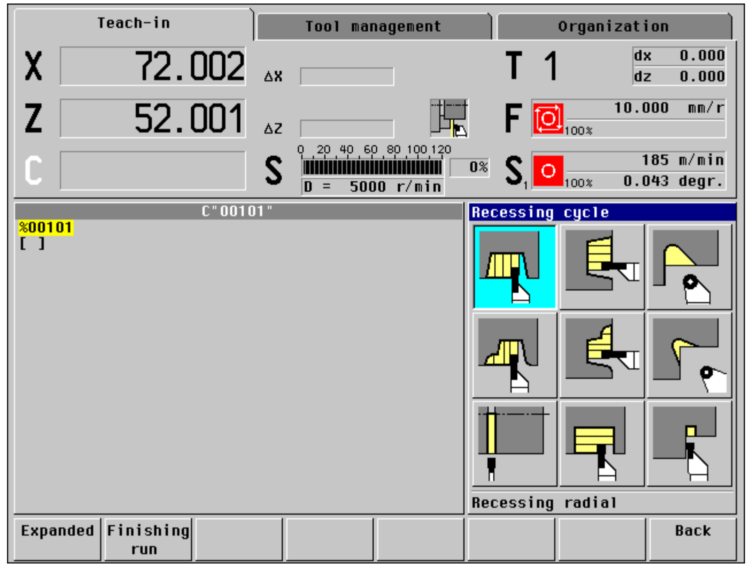
Tool data

- Lathe tool (for external machining)
- WO = 3—Tool orientation
- A = 93°—Tool angle
- B = 55°—Point angle

4.5 Recessing cycles



The recessing cycle group comprises recessing, recess turning, undercut and parting cycles. Simple contours are machined in "basic mode," complex contours in "expanded mode." With ICP recessing cycles, you can machine any type of contour defined with ICP (see “ICP Contours” on page 242).



- **Proportioning of cuts:** MANUALplus calculates an infeed that is <= infeed depth P.
- **Oversizes:** In "expanded" mode.
- **Cutter radius compensation:** Active (exception: Undercut type K).

Cutting and infeed directions for recessing cycles

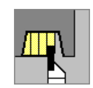
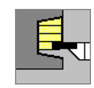
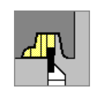
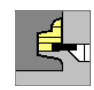
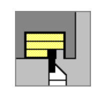
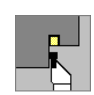
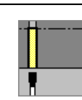
MANUALplus automatically determines the cutting and infeed directions from the cycle parameters. The decisive ones are:

- **Basic mode:** The parameters for starting point X, Z (Manual mode: current tool position) and contour starting point X1 / contour end point Z2.
- **Expanded mode:** The parameters for contour starting point X1, Z1 and contour end point X2, Z2.
- **ICP cycles:** The parameters for starting point X, Z (Manual mode: current tool position) and starting point of the ICP contour.

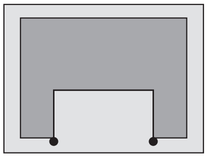
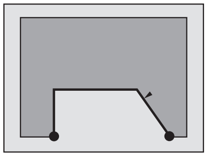
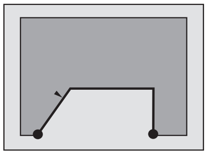
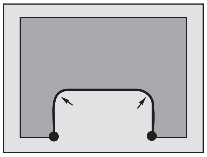
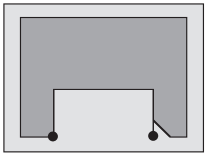
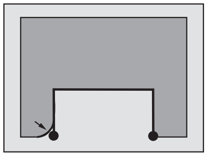
Undercut position

MANUALplus determines the position of an undercut from the cycle parameters for starting point X, Z (current tool position in Manual mode) and corner point of contour X1, Z1.

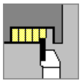
Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.

Recessing cycles	Symbol
Recessing radial/axial Recessing and finishing cycles for simple contours	 
ICP recessing, radial/axial Recessing and finishing cycles for any type of contour	 
Recess turning, radial/axial Recess-turning and finishing cycles for simple contours and any type of contour	
Undercut H Undercut type H	
Undercut K Undercut type K	
Undercut U Undercut type U	
Parting Cycle for parting the workpiece	

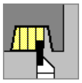
Contour elements

Contour elements in recessing cycles	
Basic mode Machining a rectangular area	
Expanded mode Oblique cut at contour start	
Expanded mode Oblique cut at contour end	
Expanded mode Rounding arc in both corners of contour valley	
Expanded mode Chamfer (or rounding) at contour start	
Expanded mode Chamfer (or rounding) at contour end	

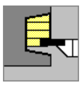
Recessing, radial/axial



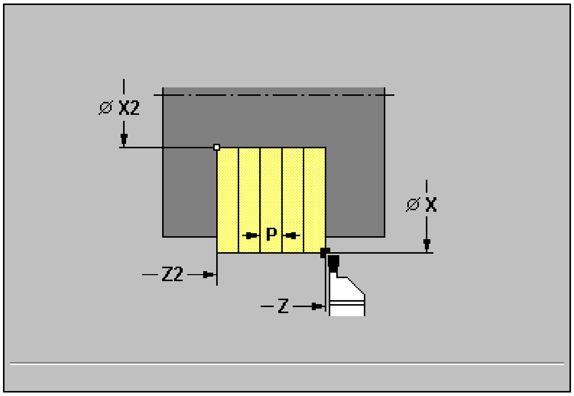
Call the recessing menu.

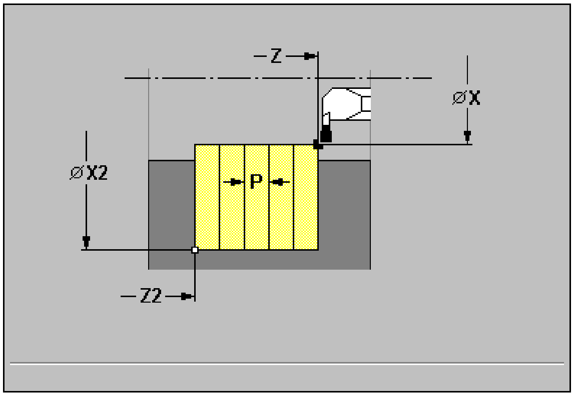


Select the "Recessing, radial" cycle (see figures at right).



Select "Recessing axial" (see figures on the following page).





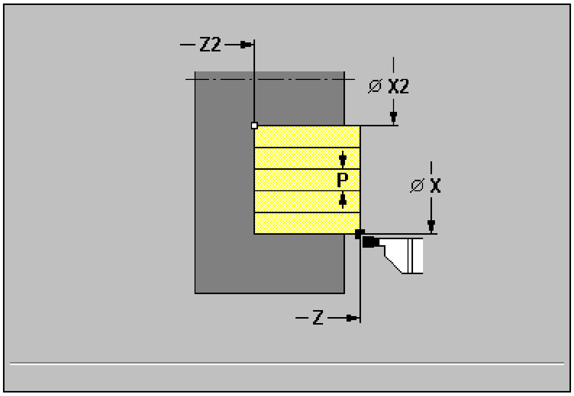
The cycle machines the number of recesses defined in Q. The parameters X/Z to X2/Z2 define the first recess (position, recess depth and recess width).

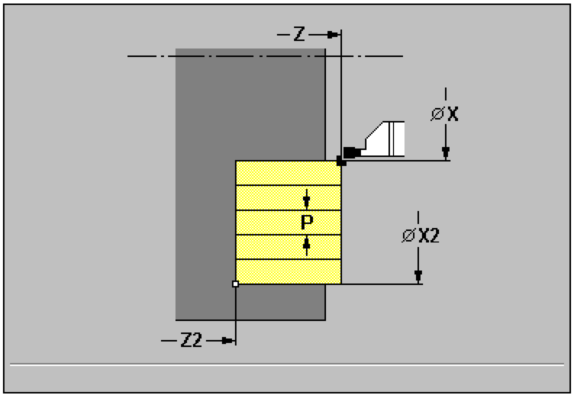
Cycle parameters

- **`X, Z starting point`**
- **`X2, Z2 contour end point`**
- **`P recessing width:`** Infeeds <= P
  - No input: P = 0.8 * cutting width of the tool
- **`E dwell time`** (for chip breaking)—default: length of time for two revolutions
- **`DX, DZ distance to subsequent recess`** with respect to the preceding recess
- **`Q number of recess cycles`**—default: 1
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

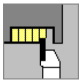
Cycle run

1. Calculate the recess positions and the proportioning of cuts.
2. Approach workpiece for next recess from starting point or from last recess on paraxial path.
3. Move to end point X2 or end point Z2 at programmed feed rate.
4. Remain at this position for dwell time "E."
5. Retract and approach for next pass.
6. Repeat 3 to 5 until the complete recess has been machined.
7. Repeat 2 to 6 until all recesses have been machined.
8. Return to starting point on paraxial path.

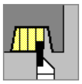




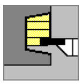
Recessing, radial/axial—Expanded



Call the recessing menu.



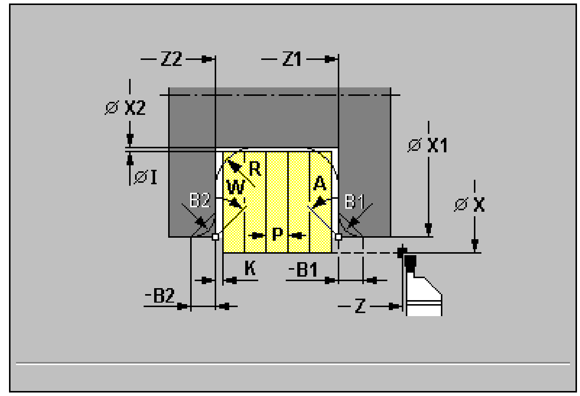
Select the "Recessing, radial" cycle (see figures at right).

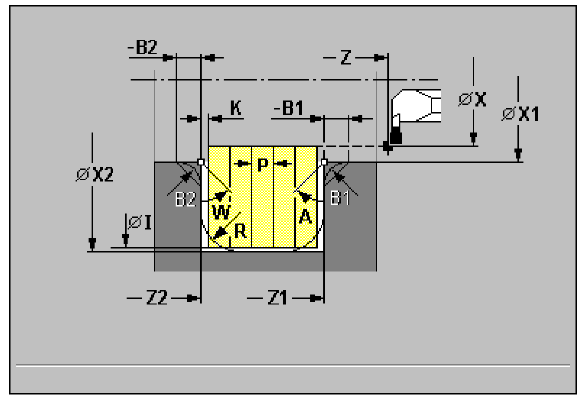


Select "Recessing axial" (see figures on the following page).

Expanded

Press the ***Expanded*** soft key.





The cycle machines the number of recesses defined in Q. The parameters X1/Z1 to X2/Z2 define the first recess (position, recess depth and recess width).

Cycle parameters

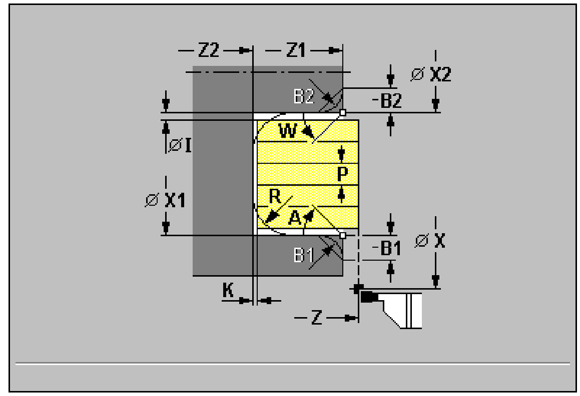
- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **P recessing width:** Infeeds <= P
  - No input: P = 0.8 * cutting width of the tool
- **A starting angle:** Range: 0° <= A < 90°
- **W end angle:** Range: 0° <= W < 90°
- **R rounding**
- **B1, B2 chamfer/rounding** (B1 contour start; B2 contour end)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**
- **I, K oversize X, Z**
- **DX, DZ distance to subsequent recess** with respect to the preceding recess
- **Q number of recess cycles**—default: 1

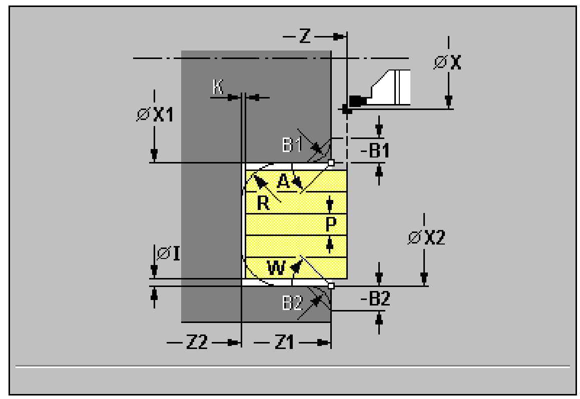
By setting the following **optional parameters,** you can define additional contour elements:

- A: Oblique cut at contour start
- W: Oblique cut at contour end
- R: Rounding (in both corners of the contour valley)
- B1: Chamfer/Rounding at contour start
- B2: Chamfer/Rounding at contour end

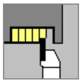
Cycle run

1 Calculate the recess positions and the proportioning of cuts.
2 Approach workpiece for next recess from starting point or from last recess on paraxial path.
3 Move to contour end point X2 or contour end point Z2, or if defined, to one of the optional contour elements at programmed feed rate.
4 Remain at this position for a dwell time of two revolutions.
5 Retract and approach for next pass.
6 Repeat 3 to 5 until the complete recess has been machined.
7 Repeat 2 to 6 until all recesses have been machined.
8 Return to starting point on paraxial path.

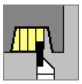




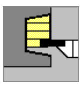
Recessing radial/axial, finishing



Call the recessing menu.



Select the "Recessing, radial" cycle (see figures at right).

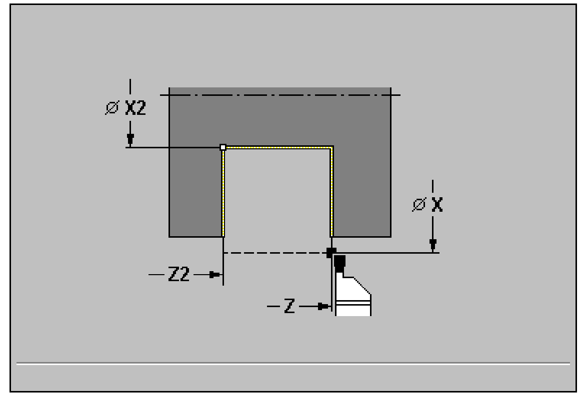


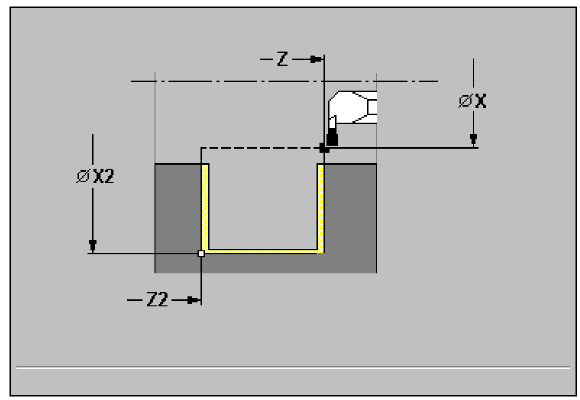
Select "Recessing axial" (see figures on the following page).

Finishing run

Press the ***Finishing run*** soft key.

The cycle finishes the number of recesses defined in Q. The parameters X/Z to X2/Z2 define the first recess (position, recess depth and recess width).



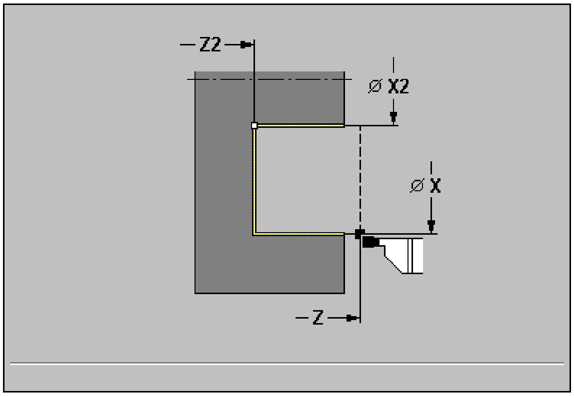


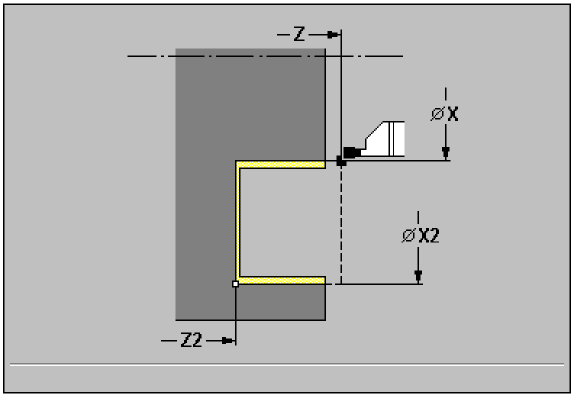
Cycle parameters

- **`X, Z starting point`**
- **`X2, Z2 contour end point`**
- **`DX, DZ distance to subsequent recess`** with respect to the preceding recess
- **`Q number of recess cycles`**—default: 1
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

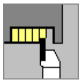
Cycle run

1. Calculate the recess positions.
2. Approach workpiece for next recess from starting point or from last recess on paraxial path.
3. Finish first side and the contour valley up to position just before recess end point.
4. Approach workpiece for finishing the second side on paraxial path.
5. Finish the second side and the remainder of the contour valley.
6. Repeat 2 to 5 until all recesses have been machined.
7. Return to starting point on paraxial path.

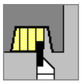




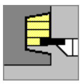
Recessing radial/axial, finishing—Expanded



Call the recessing menu.



Select the "Recessing, radial" cycle (see figures at right).



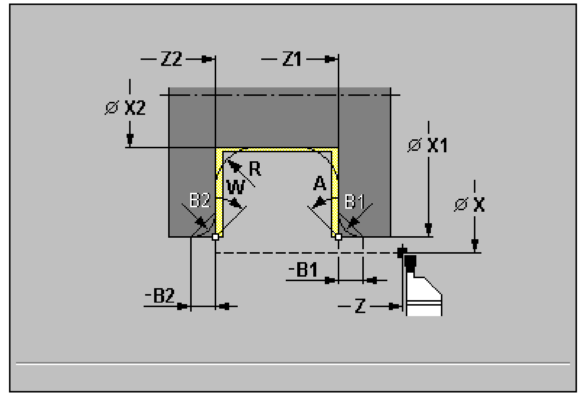
Select "Recessing axial" (see figures on the following page).

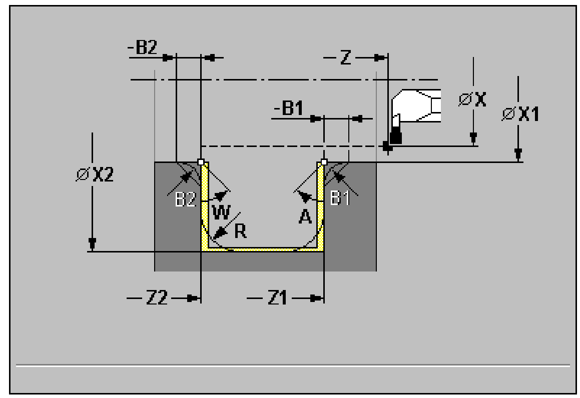
Expanded

Press the ***Expanded*** soft key.

Finishing run

Press the ***Finishing run*** soft key.





The cycle machines the number of recesses defined in Q. The parameters X/Z to X2/Z2 define the first recess (position, recess depth and recess width).

Cycle parameters

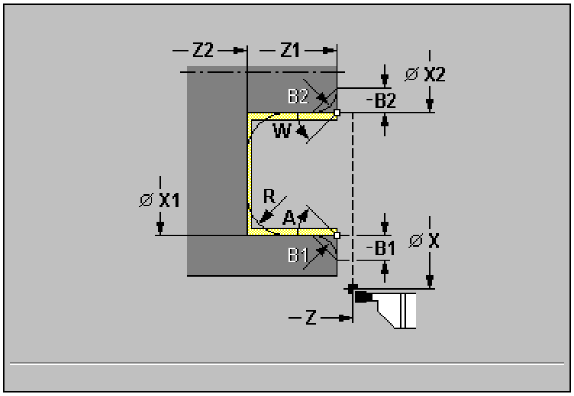
- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **A starting angle:** Range: 0° <= A < 90°
- **W end angle:** Range: 0° <= W < 90°
- **R rounding**
- **B1, B2 chamfer/rounding** (B1 contour start; B2 contour end)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**
- **DX, DZ distance to subsequent recess** with respect to the preceding recess
- **Q number of recess cycles**—default: 1

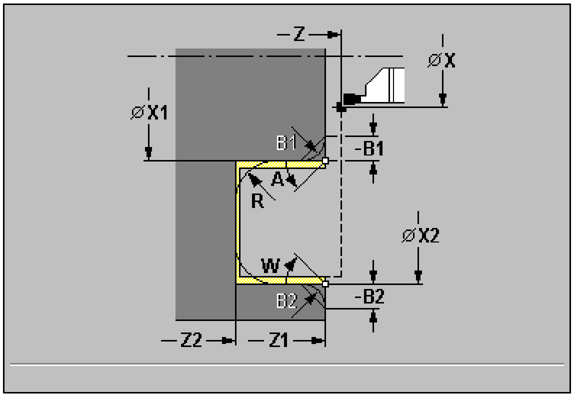
By setting the following **optional parameters,** you can define additional contour elements:

- A: Oblique cut at contour start
- W: Oblique cut at contour end
- R: Rounding (in both corners of the contour valley)
- B1: Chamfer/Rounding at contour start
- B2: Chamfer/Rounding at contour end

Cycle run

1. Calculate the recess positions.
2. Approach workpiece for next recess from starting point or from last recess on paraxial path.
3. Finish first side, taking optional contour elements into account; then finish contour valley up to position just before recess end point.
4. Approach workpiece for finishing the second side on paraxial path.
5. Finish second side, taking optional contour elements into account; then finish remainder of contour valley.
6. Repeat 2 to 5 until all recesses have been finished.
7. Return to starting point on paraxial path.



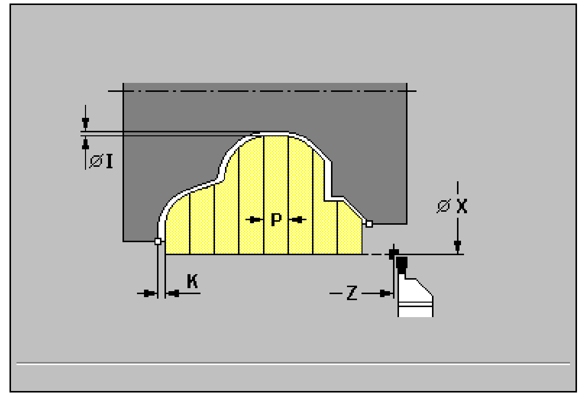


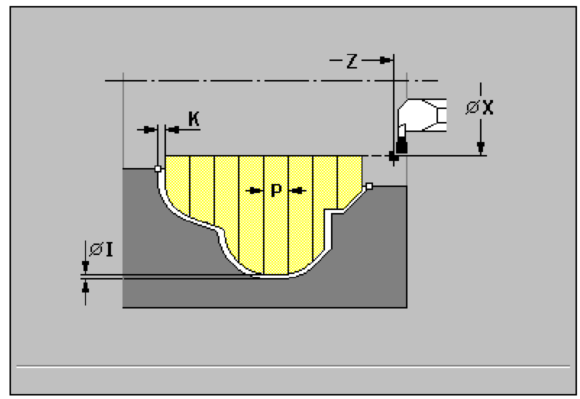
ICP recessing cycles

Call the recessing menu.

Select the "ICP recessing, radial" cycle (see figures at right).

Select "ICP recessing axial" (see figures on the following page).





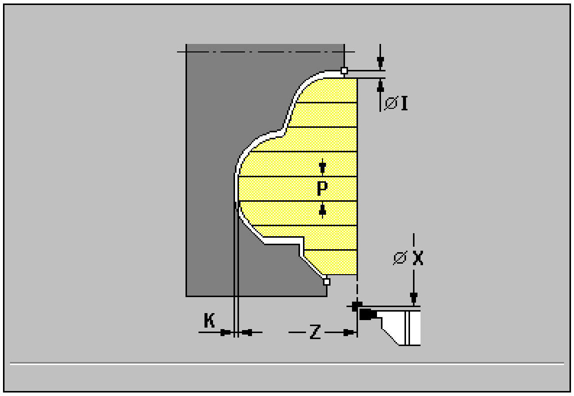
The cycle machines the number of recesses defined in Q with the ICP recessing contour. The parameters "X, Z" define the position of the first recess.

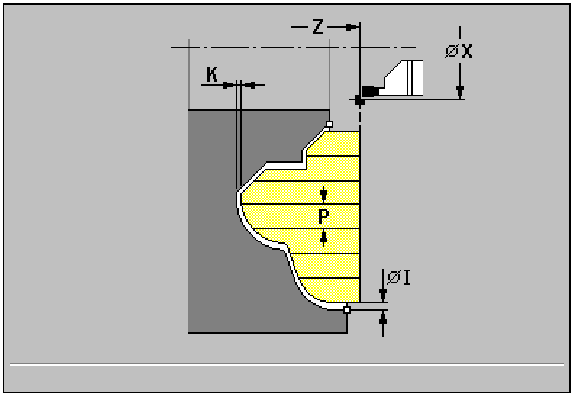
Cycle parameters

- **X, Z starting point**
- **P recessing width:** Infeeds <= P
  - No input: P = 0.8 * cutting width of the tool
- **I, K oversize X, Z**
- **N ICP contour number**
- **DX, DZ distance to subsequent recess** with respect to the preceding recess
- **Q number of recess cycles**—default: 1
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

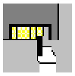
Cycle run

1. Calculate the recess positions and the proportioning of cuts.
2. Approach workpiece for next recess from starting point or from last recess on paraxial path.
3. Machine along the defined contour.
4. Return and approach for next pass.
5. Repeat 3 to 4 until the complete recess has been machined.
6. Repeat 2 to 5 until all recesses have been machined.
7. Return to starting point on paraxial path.

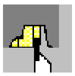




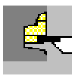
ICP recessing radial/axial, finishing



Call the recessing menu.



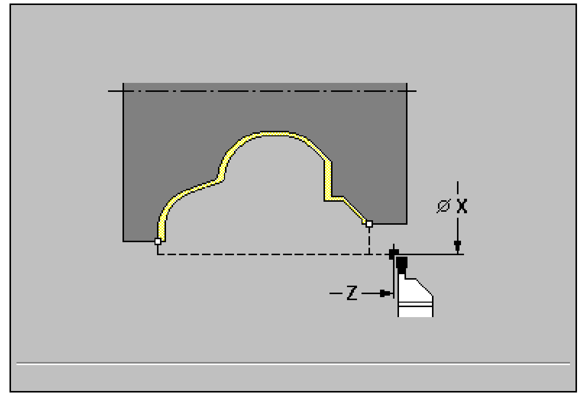
Select the "ICP recessing, radial" cycle (see figures at right).

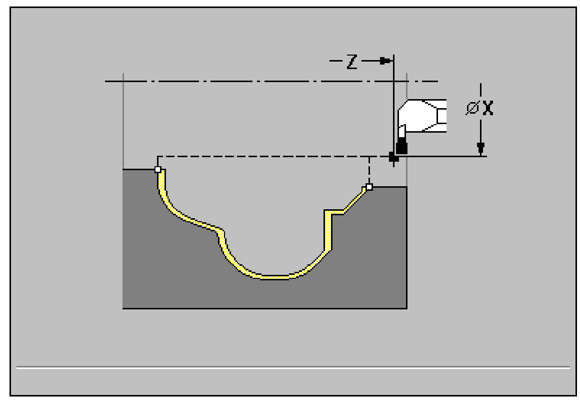


Select "ICP recessing axial" (see figures on the following page).

Finishing run

Press the ***Finishing run*** soft key.





The cycle finish-machines the number of recesses defined in Q with the ICP recessing contour. The parameters "X, Z" define the position of the first recess.

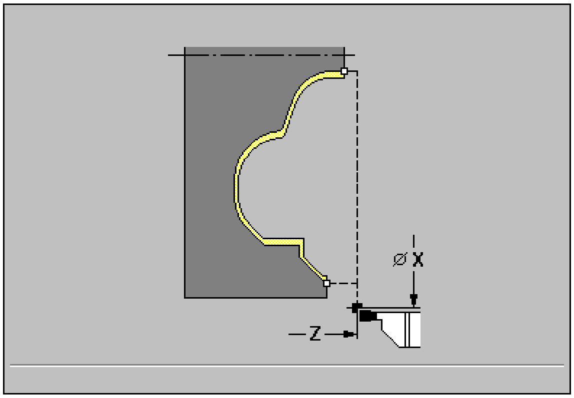
At the end of cycle, the tool returns to the starting point.

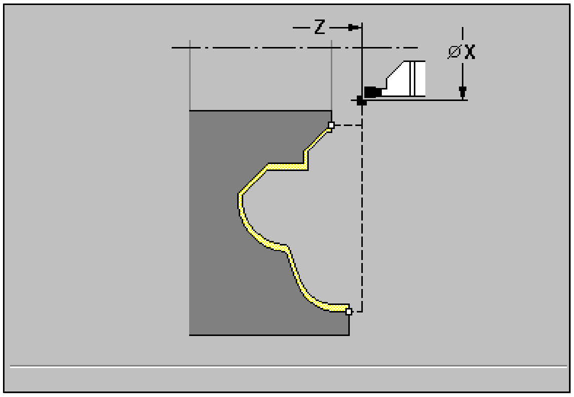
Cycle parameters

- **X, Z starting point**
- **I, K oversize X, Z**
- **N ICP contour number**
- **DX, DZ distance to subsequent recess** with respect to the preceding recess
- **Q number of recess cycles**—default: 1
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Cycle run

1. Calculate the recess positions.
2. Approach workpiece for next recess from starting point or from last recess on paraxial path.
3. Finish the recess.
4. Repeat 2 to 3 until all recesses have been machined.
5. Return to starting point on paraxial path.





Recess turning

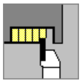
The workpiece is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements.

To influence recess-turning operations, use the following parameters:

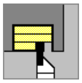
- Recessing feed rate O: Feed rate for recessing movement.
- Turning operation, unidirectional/bidirectional U: You can perform a unidirectional or bidirectional turning operation. With **radial** recess-turning cycles, unidirectional turning operations are always performed in the direction of the spindle. With **axial** ICP recess-turning cycles, the machining direction corresponds to the direction of contour definition.
- Offset width B: After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by "offset width B." Each time the system switches from turning to recessing on this side, the path is reduced by "B"—in addition to the previous offset. The total offset is limited to 80% of the effective cutting width (effective cutting width = cutting width – 2*cutting radius). If required, the MANUALplus reduces the programmed offset width. After precutting, the remaining material is removed with a single cut.
- Depth compensation RB: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. The resulting infeed error can be compensated with "depth compensation RB" during finishing. The depth compensation factor is usually determined empirically.

These cycles require the use of **recess-turning tools.**

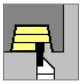
Recess turning, radial/axial



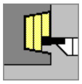
Call the recessing menu.



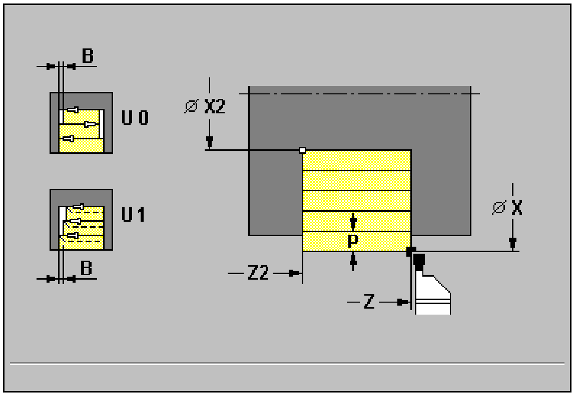
Select the "Recess turning" cycle.

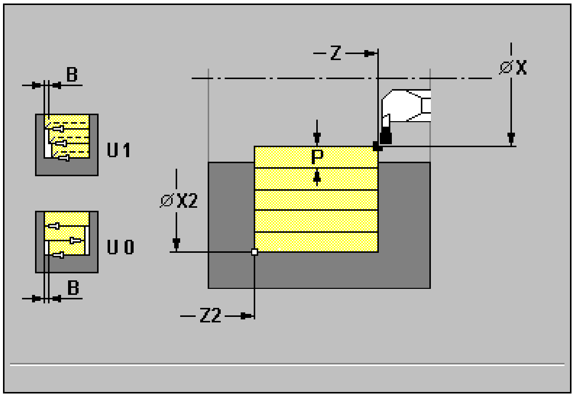


Select the "Recess turning, radial" cycle (see figures at right).



Select "Recess turning, axial" (see figures on the following page).





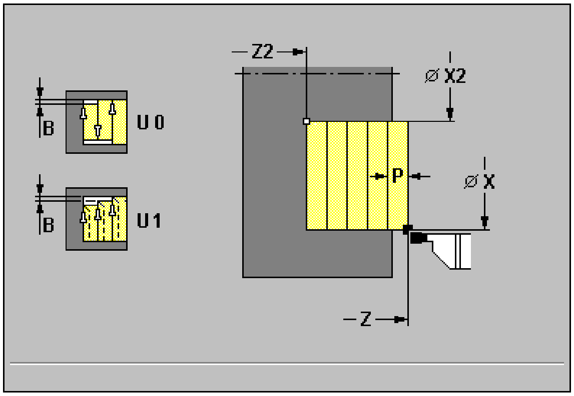
The cycle machines the rectangular area defined by X, Z and X2, Z2 (see also “Recess turning” on page 143).

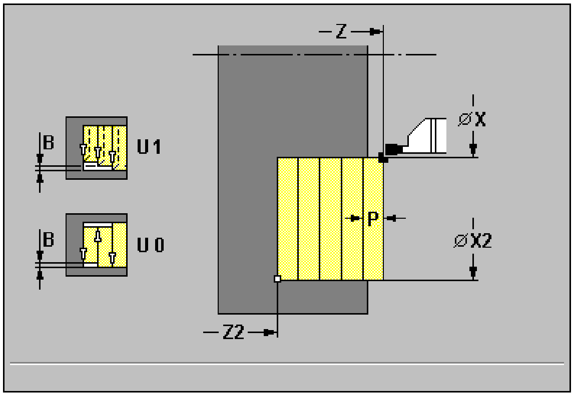
Cycle parameters

- **X, Z starting point**
- **X2, Z2 contour end point**
- **P infeed depth:** Maximum infeed depth
- **O recessing feed rate**—default: Active feed rate
- **B offset width**—default: 0
- **U unidirectional turning**—default: 0
  - U=0: bidirectional
  - U=1: unidirectional (direction: see graphic support window)
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

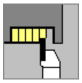
Cycle run

1. Calculate the proportioning of cuts.
2. Approach workpiece for first pass from X, Z.
3. Execute the first cut (recessing).
4. Machine perpendicularly to recessing direction (turning).
5. Repeat 3 to 4 until contour end point Z2/X2 is reached.
6. Return to starting point on paraxial path.

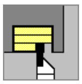




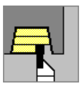
Recess turning, radial/axial—Expanded



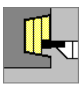
Call the recessing menu.



Select the "Recess turning" cycle.



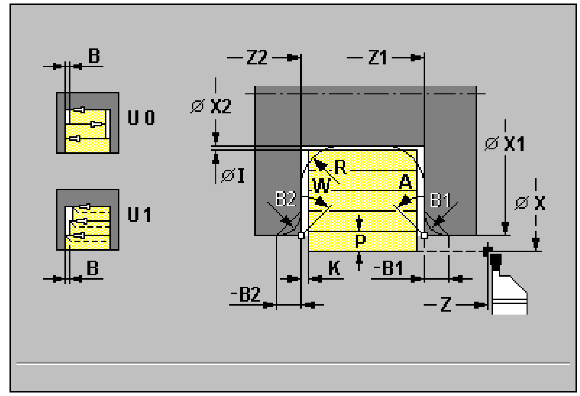
Select the "Recess turning, radial" cycle (see figures at right).

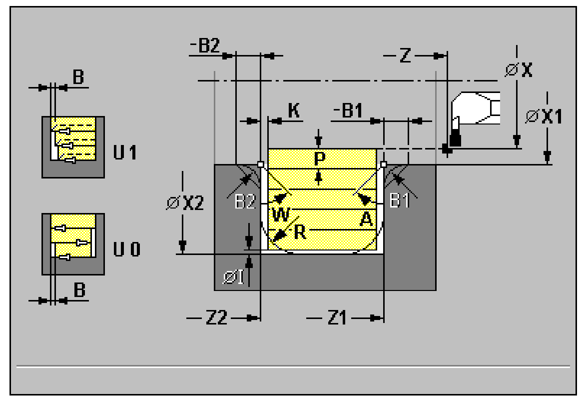


Select "Recess turning, axial" (see figures on the following page).

Expanded

Press the ***Expanded*** soft key.





The cycle machines the area defined by X/Z1 and X2, Z2, taking the oversizes into account (see also "Recess turning" on page 143).

Cycle parameters

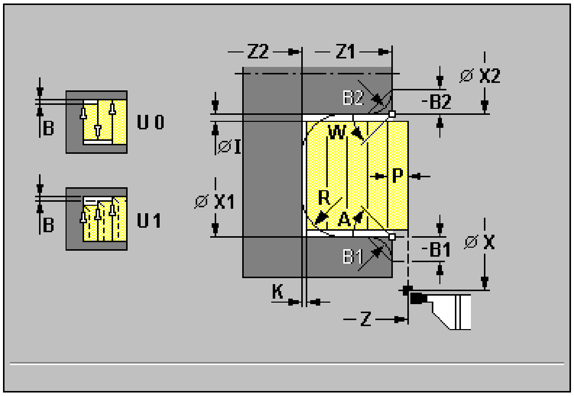
- **`X, Z starting point`**
- **`X1, Z1 contour starting point`**
- **`X2, Z2 contour end point`**
- **`P infeed depth:`** Maximum infeed depth
- **`O recessing feed rate`**—default: Active feed rate
- **`A starting angle:`** Range: 0° <= A < 90°
- **`W end angle:`** Range: 0° <= W < 90°
- **`R rounding`**
- **`B1, B2 chamfer/rounding`** (B1 contour start; B2 contour end)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**
- **`B offset width`**—default: 0
- **`U unidirectional turning`**—default: 0
  - U=0: bidirectional
  - U=1: unidirectional (direction: see graphic support window)
- **`I, K oversize X, Z`**

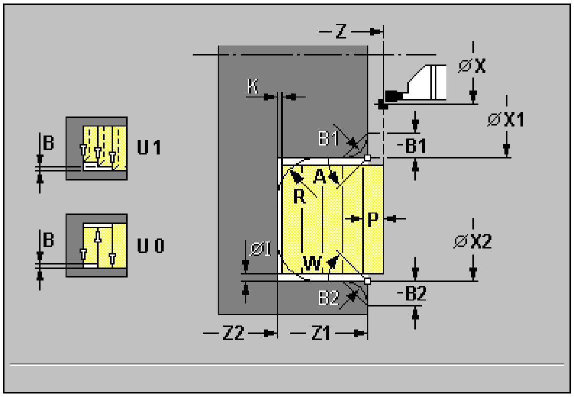
By setting the following **optional parameters,** you can define additional contour elements:

- A: Oblique cut at contour start
- W: Oblique cut at contour end
- R: Rounding (in both corners of the contour valley)
- B1: Chamfer/Rounding at contour start
- B2: Chamfer/Rounding at contour end

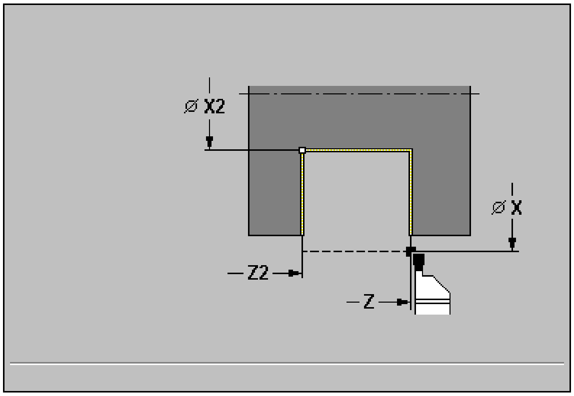
Cycle run

1. Calculate the proportioning of cuts.
2. Approach workpiece for first pass from X, Z.
3. Execute the first cut (recessing).
4. Machine perpendicularly to recessing direction (turning).
5. Repeat 3 to 4 until contour end point Z2/X2 is reached.
6. Machine chamfer/rounding at contour start / contour end if defined.
7. Return to starting point on paraxial path.





Recess turning radial/axial, finishing

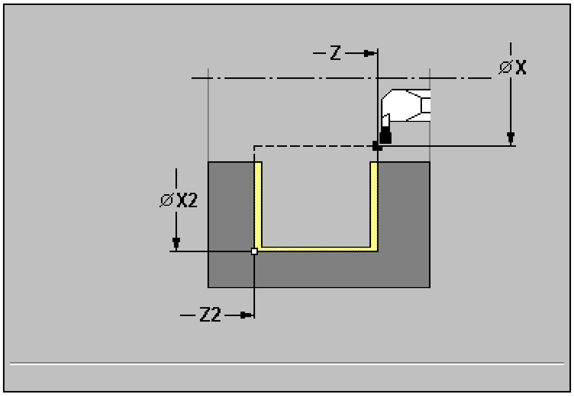


Call the recessing menu.

Select the "Recess turning" cycle.

Select the "Recess turning, radial" cycle (see figures at right).

Select "Recess turning, axial" (see figures on the following page).



Finishing run

Press the ***Finishing run*** soft key.

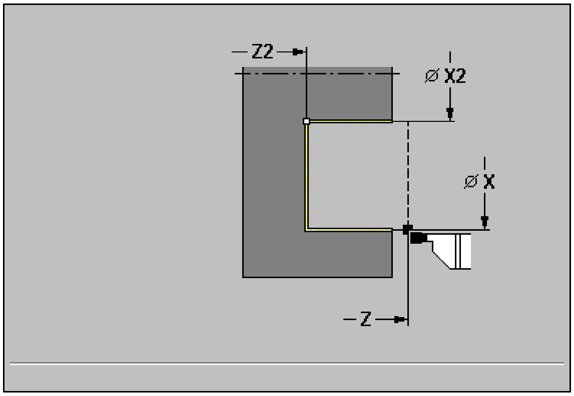
The cycle finishes the contour area from X, Z to X2, Z2 (see also "Recess turning" on page 143).

With "oversizes I, K" for the workpiece blank, you define the material to be machined during the finishing cycle. It is therefore absolutely necessary to define the oversizes for recess turning—finishing.

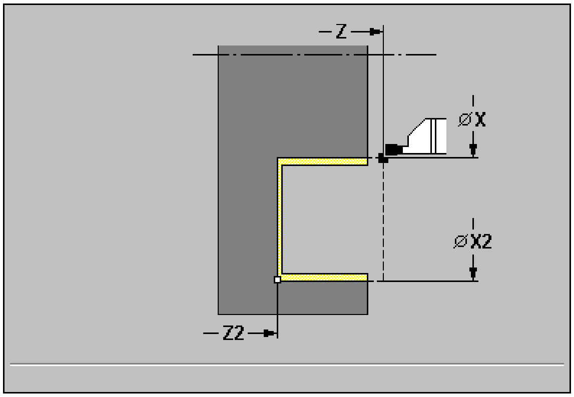
Cycle parameters

- **X, Z starting point**
- **X2, Z2 contour end point**
- **I, K workpiece-blank oversize X, Z**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

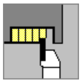
Cycle run



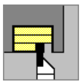
1 Approach contour area from X, Z.
2 Finish first side, then finish contour valley up to position just before contour end point Z2/X2.
3 Move on paraxial path:
   - radially to X/Z2.
   - axially to Z/X2.
4 Finish second side, then finish remainder of contour valley.
5 Return to starting point on paraxial path.



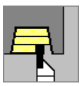
Recess turning radial/axial, finishing—Expanded



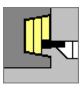
Call the recessing menu.



Select the "Recess turning" cycle.



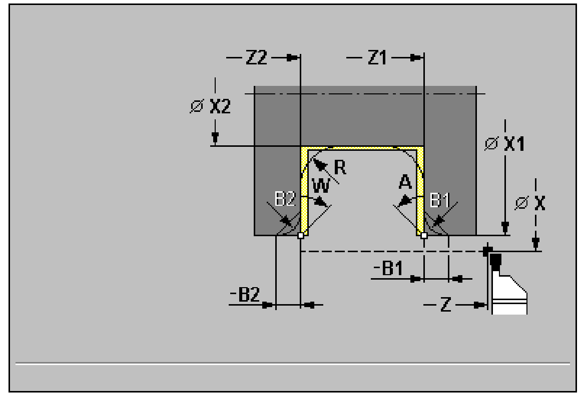
Select the "Recess turning, radial" cycle (see figures at right).

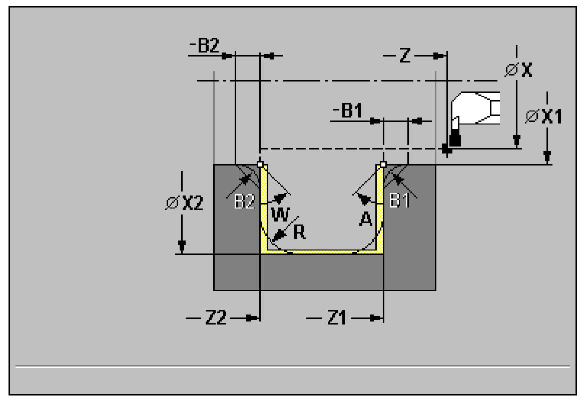


Select "Recess turning, axial" (see figures on the following page).

Expanded — Press the ***Expanded*** soft key.

Finishing run — Press the ***Finishing run*** soft key.





The cycle finishes the contour area from X1, Z1 to X2, Z2 (see also "Recess turning" on page 143).

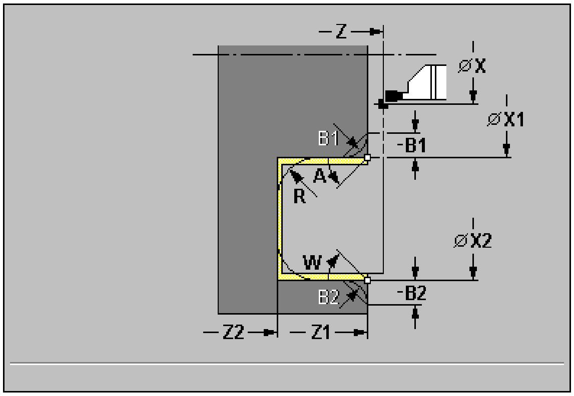
Cycle parameters

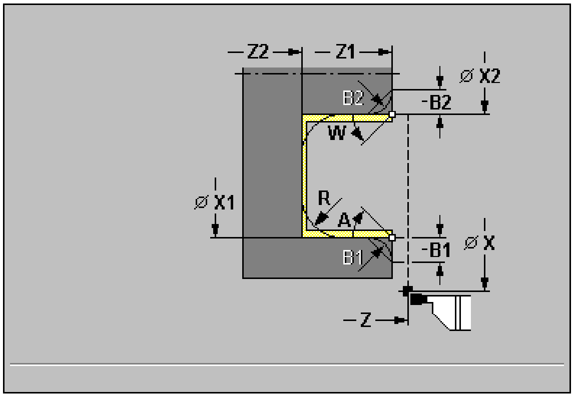
- **X, Z starting point**
- **X1, Z1 contour starting point**
- **X2, Z2 contour end point**
- **O recessing feed rate**—default: Active feed rate
- **A starting angle:** Range: 0° <= A < 90°
- **W end angle:** Range: 0° <= W < 90°
- **R rounding**
- **B1, B2 chamfer/rounding** (B1 contour start; B2 contour end)
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **RB depth compensation**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**
- **I, K workpiece-blank oversize X, Z**

By setting the following **optional parameters,** you can define additional contour elements:

- A: Oblique cut at contour start
- W: Oblique cut at contour end
- R: Rounding (in both corners of the contour valley)
- B1: Chamfer/Rounding at contour start
- B2: Chamfer/Rounding at contour end

> With "oversizes I, K" for the workpiece blank, you define the material to be machined during the finishing cycle. It is therefore absolutely necessary to define the oversizes for recess turning—finishing.





Cycle run

1. Approach contour area from X, Z.
2. Finish first side, taking optional contour elements into account; then finish contour valley up to position just before contour end point Z2/X2.
3. Approach workpiece for finishing the second side on paraxial path.
4. Finish second side, taking optional contour elements into account; then finish remainder of contour valley.
5. Finish chamfer/rounding at contour start / contour end, if defined.

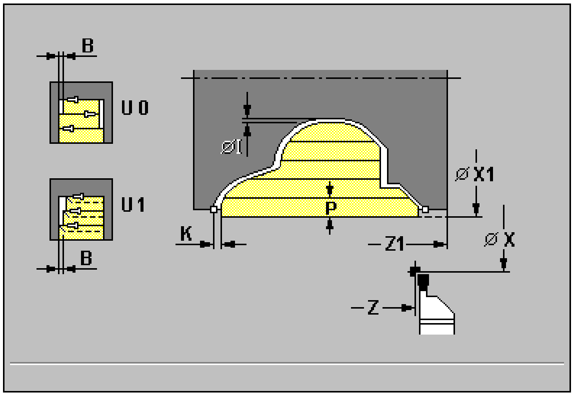
ICP recess turning, radial/axial

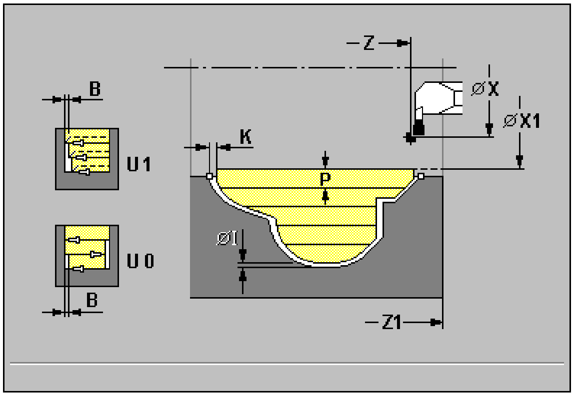
Call the recessing menu.

Select the "Recess turning" cycle.

Select the "ICP recess turning, radial" cycle (see figures at right).

Select "ICP recess turning axial" (see figures on the following page).





The cycle proceeds as follows, taking oversizes into account:

- For descending contours, the area defined by X, Z and the ICP contour is machined.
- For ascending contours, the area defined by X1, Z1 and the ICP contour is machined.

See also “Recess turning” on page 143.

If you are machining:

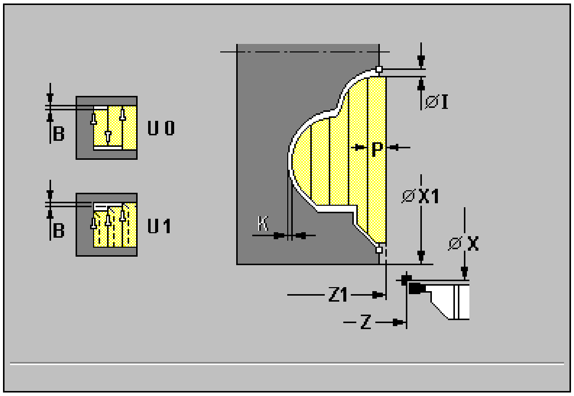
- **Descending contours, define:** Only starting point X, Z—but not contour starting point X1, Z1.
- **Ascending contours, define:** Starting point X, Z and contour starting point X1, Z1.

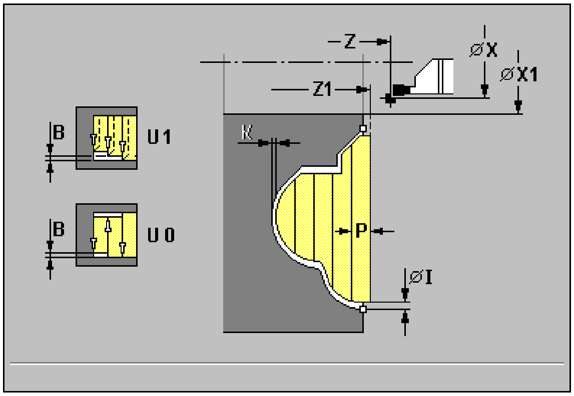
Cycle parameters

- **X, Z starting point**
- **X1, Z1, starting point of workpiece blank**
- **P infeed depth:** Maximum infeed depth
- **O recessing feed rate**—default: Active feed rate
- **B offset width**—default: 0
- **U unidirectional turning**—default: 0
  - U=0: bidirectional
  - U=1: unidirectional (direction: see graphic support window)
- **I, K oversize X, Z**
- **N ICP contour number**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

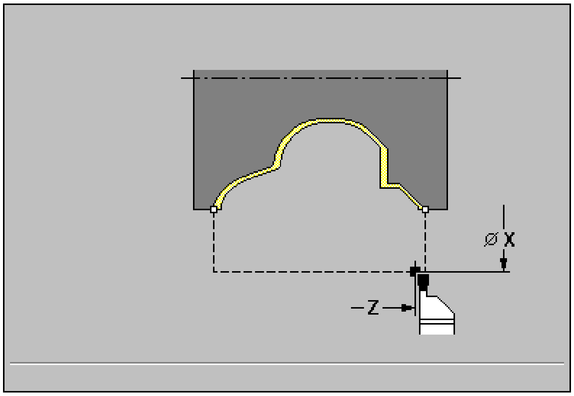
Cycle run

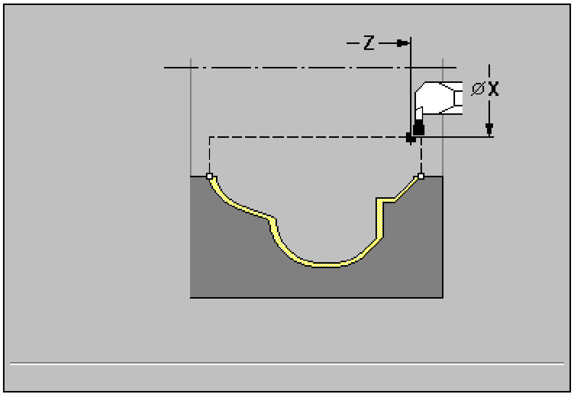
1. Calculate the proportioning of cuts.
2. Approach workpiece for first pass from X, Z.
3. Execute the first cut (recessing).
4. Machine perpendicularly to recessing direction (turning).
5. Repeat 3 to 4 until the defined area has been machined.
6. Return to starting point on paraxial path.





ICP recess turning radial/axial, finishing





Call the recessing menu.

Select the "Recess turning" cycle.

Select "ICP recess turning radial " (see figures at upper and center right).

Select "ICP recess turning axial" (see figures on the following page).

Finishing run

Press the ***Finishing run*** soft key.

The cycle finishes the contour area defined by the ICP contour (see also “Recess turning” on page 143). At the end of cycle, the tool returns to the starting point.

Cycle parameters

- **X, Z starting point**
- **RB depth compensation**
- **I, K workpiece-blank oversize X, Z**
- **N ICP contour number**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

With "oversizes I, K" for the workpiece blank, you define the material to be machined during the finishing cycle. It is therefore absolutely necessary to define the oversizes for recess turning—finishing.

Cycle run

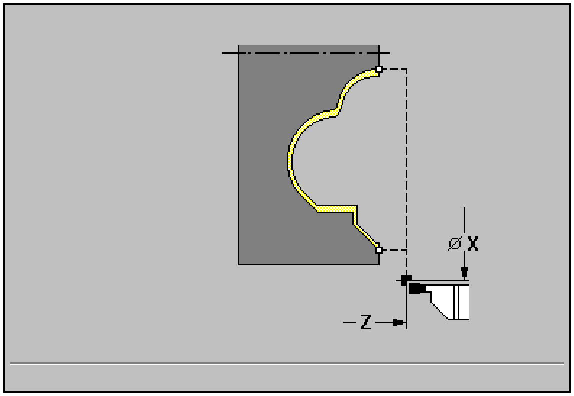
1 Approach contour area from X, Z on paraxial path.

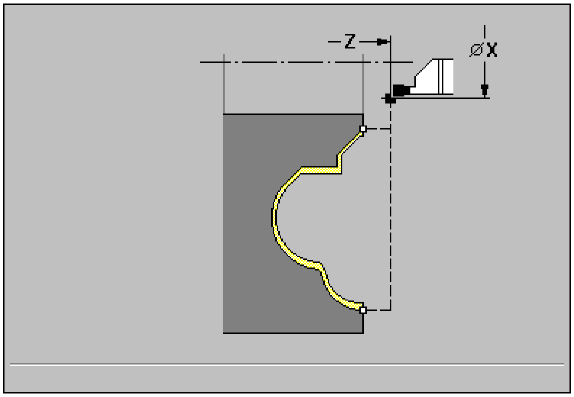
2 Finish first side and contour area up to position just before end point X2/Z2.

3 Approach workpiece for finishing the second side on paraxial path.

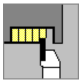
4 Finish second side, then finish remainder of contour valley.

5 Return to starting point on paraxial path.

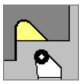




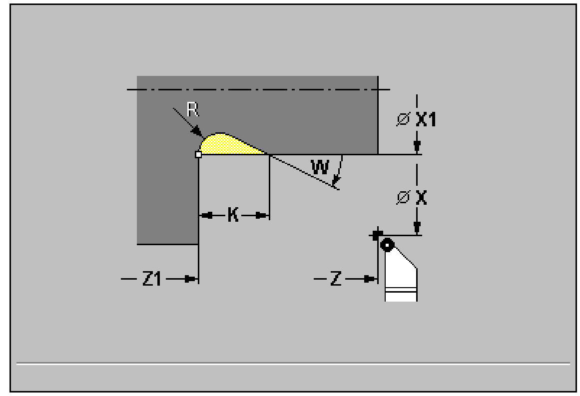
Undercut type H



Call the recessing menu.



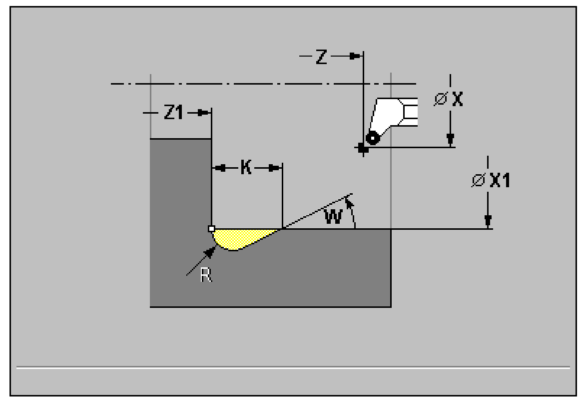
Select the "Undercut H" cycle.



The contour depends on the parameters defined. If you do not define an "undercut radius R," the oblique cut will be executed up to "contour corner Z1" (tool radius = undercut radius).

If you do not define "plunging angle W," it is calculated from "undercut length K" and "undercut radius R." The final point of the undercut is then located at the "contour corner."

The end point of the undercut is determined from the plunging angle in accordance with "Undercut type H."



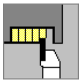
Cycle parameters

- **`X, Z starting point`**
- **`X1, Z1 contour corner`**
- **`K undercut length`**
- **`R undercut radius`**—default: No circular element
- **`W plunging angle`**—default: W is calculated
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

Cycle run

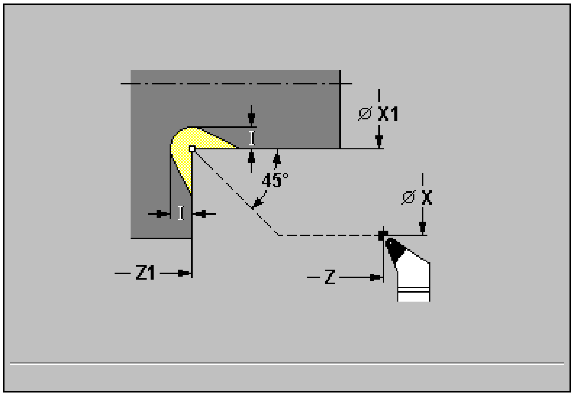
1. Pre-position to safety clearance from X, Z.
2. Machine undercut according to cycle parameters.
3. Return to starting point on diagonal path.

Undercut type K

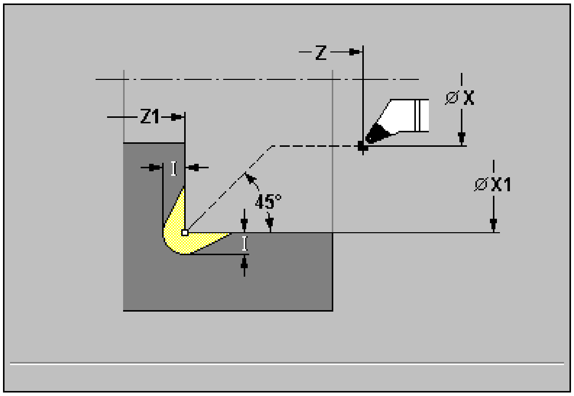


Call the recessing menu.

Select the "Undercut K" cycle.



This cycle performs only one cut at an angle of 45°. The resulting contour geometry therefore depends on the tool that is used.



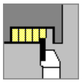
Cycle parameters

- **X, Z starting point**
- **X1, Z1 contour corner**
- **I undercut depth**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

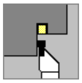
Cycle run

1. Pre-position at an angle of 45° to safety clearance above contour corner point X1, Z1 in rapid traverse.
2. Plunge by "undercut depth I."
3. Retract to starting point X, Z on same path.

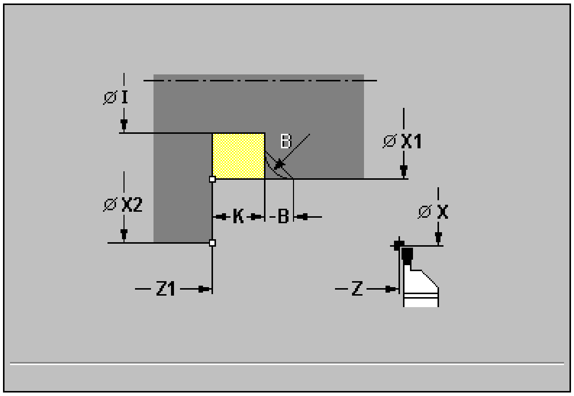
Undercut type U



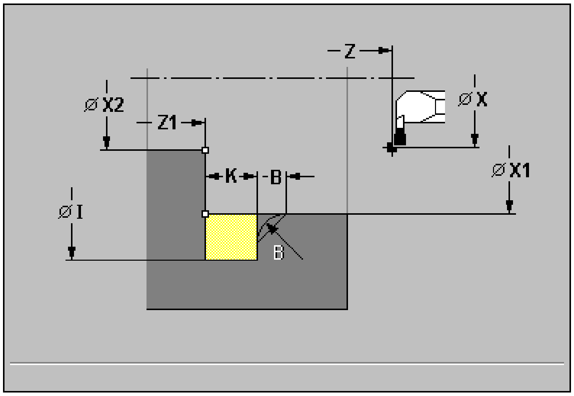
Call the recessing menu.



Select the "Undercut U" cycle.



This cycle machines an "Undercut type U" and, if programmed, finishes the adjoining plane surface. The undercut is executed in several passes if the undercut width is greater than the cutting width of the tool. If the cutting width of the tool is not defined, the control assumes that the tool's cutting width equals K. A chamfer or rounding (optional) is machined.



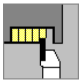
Cycle parameters

- **X, Z starting point**
- **X1, Z1 contour corner**
- **X2 end point on plane surface**
- **I undercut diameter**
- **K width of undercut**
- **B chamfer/rounding**
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

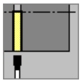
Cycle run

1. Calculate the proportioning of cuts.
2. Pre-position to safety clearance from X, Z.
3. Move at feed rate to "undercut diameter I" and remain at this position for the time of two revolutions.
4. Retract and approach for next pass.
5. Repeat steps 3 to 4 until corner point Z1 is reached.
6. After the last pass, finish adjoining plane surface, starting from end point X2", if defined.
7. Machine chamfer/rounding, if defined.
8. Return to starting point on diagonal path.

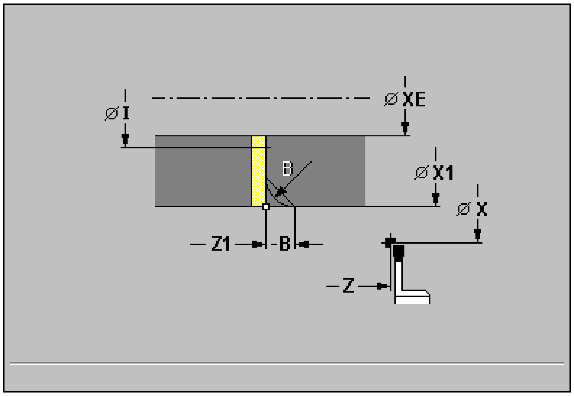
Parting

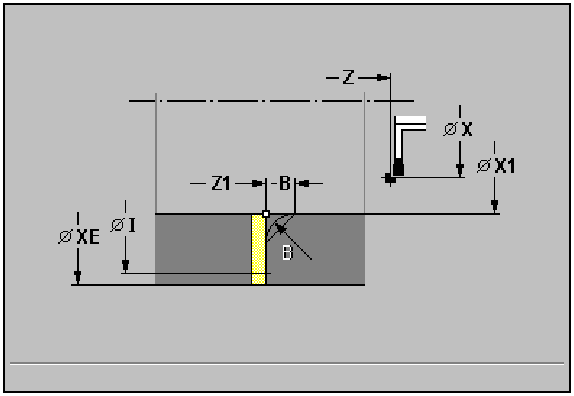


Call the recessing menu.



Select the "Cut-off" cycle.





The cycle parts the workpiece. If programmed, a chamfer or rounding arc is machined on the outside diameter.

Cycle parameters

- **X, Z starting point**
- **X1, Z1 contour corner**
- **XE inside diameter (tube)**
- **I diameter for feed reduction**
- **B chamfer/rounding**
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **E reduced feed rate**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

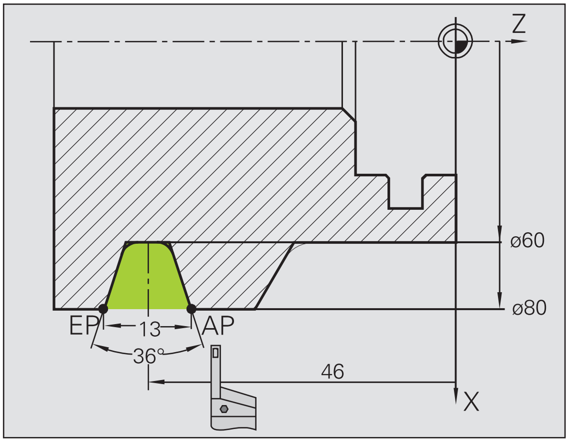
Cycle run

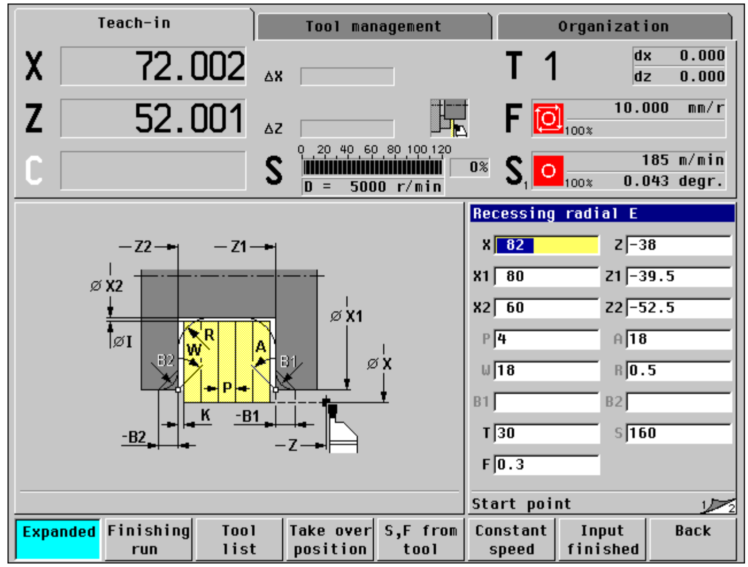
1. Pre-position to safety clearance from X, Z.
2. Cut to depth of chamfer or rounding and machine the chamfer/rounding if defined.
3. Depending on the cycle parameters, move at feed rate to
   - The turning center, or
   - The "inside diameter (tube) XE."

   If you have programmed a feed rate reduction, MANUALplus switches to the "reduced feed E" as soon as the tool reaches the "diameter feed reduction I."
4. Retract at end face and return to starting point.

Examples of recessing cycles

Recess outside

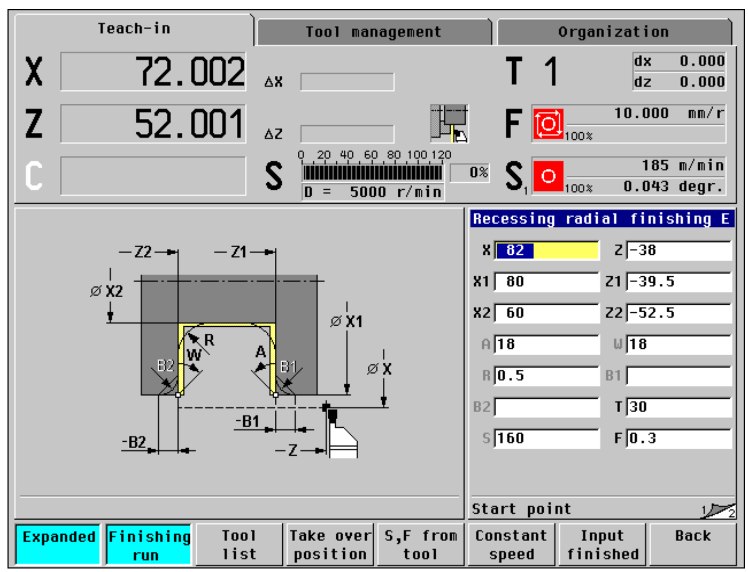




The machining operation is to be executed with the "Recessing, radial—Expanded" cycle, taking oversizes into account (see figure at upper right). This contour area is to be finished subsequently with the "Recessing radial, finishing—Expanded" cycle (see figure at lower right).

The rounding arcs in the corners of the contour valley and the oblique surfaces at the contour start and end are also machined in "expanded mode."

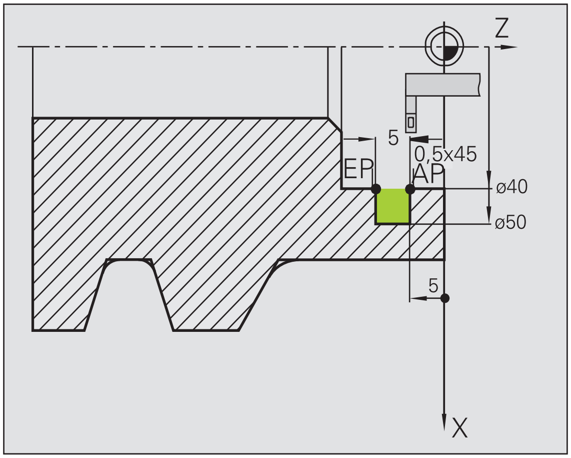
Be sure to enter the correct values for the parameters "contour starting point X1, Z1" and "contour end point X2, Z2." These parameters determine the cutting and infeed directions—in this example, external machining and infeed in negative Z-axis direction.

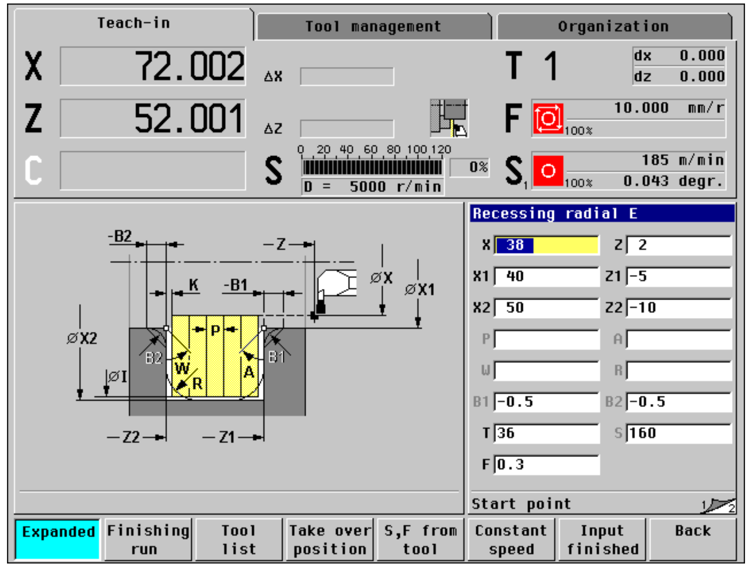


Tool data

- Lathe tool (for external machining)
- WO = 1—Tool orientation
- K = 4—Cutting width (4 mm)

Recess inside



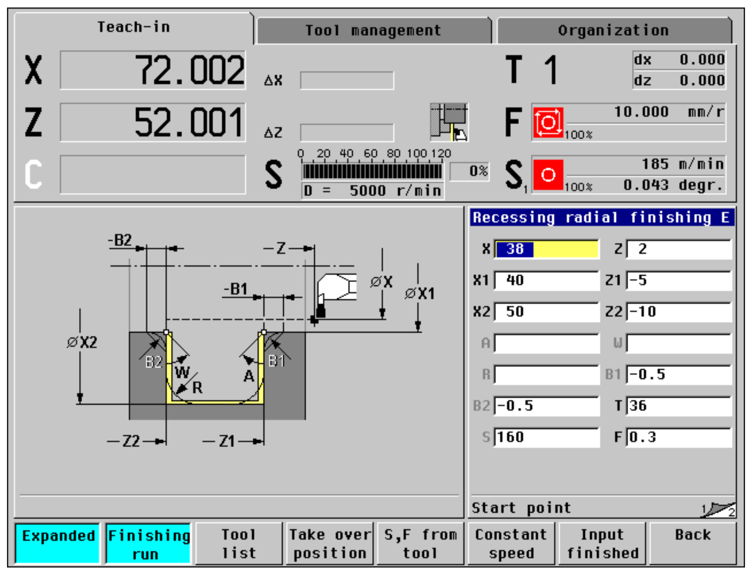


The machining operation is to be executed with the "Recessing, radial—Expanded" cycle, taking oversizes into account (see figure at upper right). This contour area is to be finished subsequently with the "Recessing radial, finishing—Expanded" cycle (see figure at lower right).

As the "plunge width P" is not input, the MANUALplus plunge-cuts with 80% of the plunge-width of the tool.

In expanded mode, the chamfers are machined at the start/end of the contour.

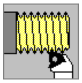
Be sure to enter the correct values for the parameters "contour starting point X1, Z1" and "contour end point X2, Z2." These parameters determine the cutting and infeed directions—in this example, internal machining and infeed in negative Z-axis direction.



Tool data

- Lathe tool (for internal machining)
- WO = 7—Tool orientation
- K = 2—Cutting width (2 mm)

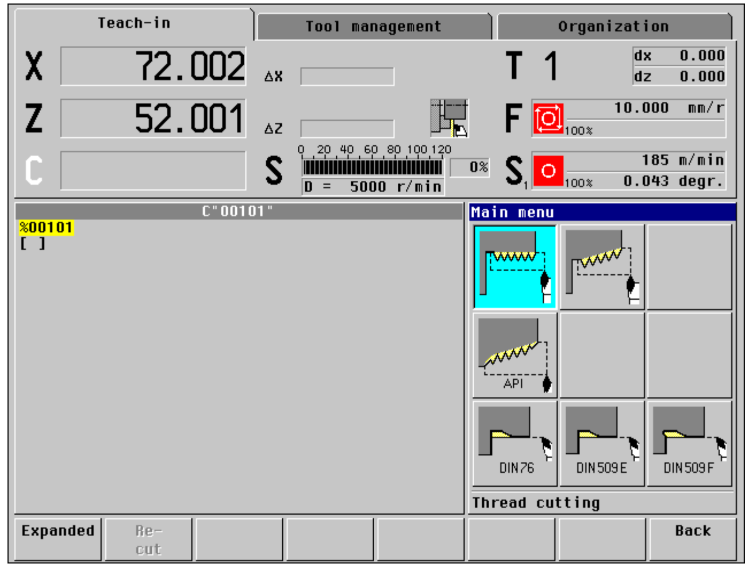
4.6 Thread and Undercut Cycles



These cycles machine single or multi-start longitudinal and tapered threads, as well as thread undercuts.

In Manual mode you can:

- Repeat the last cut to compensate for tool inaccuracies.
- Use the function **Recut** to rework damaged threads.



> - Threads are cut with constant speed.
> - "Cycle STOP" becomes effective at the end of a thread cut.
> - The feed rate and spindle speed overrides are not effective during cycle execution.

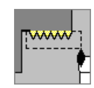
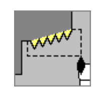
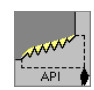
Thread position

MANUALplus determines the direction of the thread from the parameters for starting point Z (or current tool position in Manual mode) and end point Z2. You select internal or external thread by soft key.

Undercut position

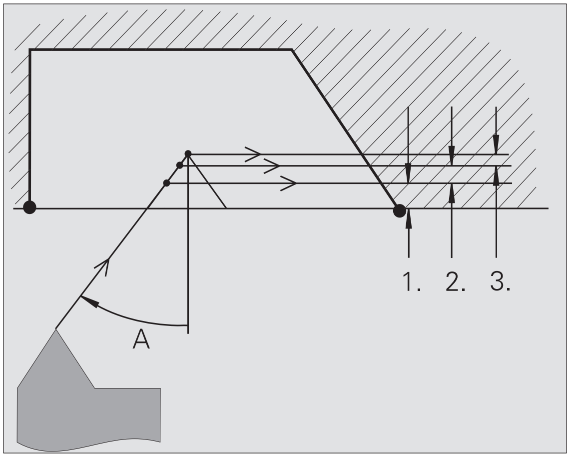
MANUALplus determines the position of an undercut from the parameters for starting point X, Z (current tool position in Manual mode) and cylinder starting point X1 / end point Z2 on plane surface.

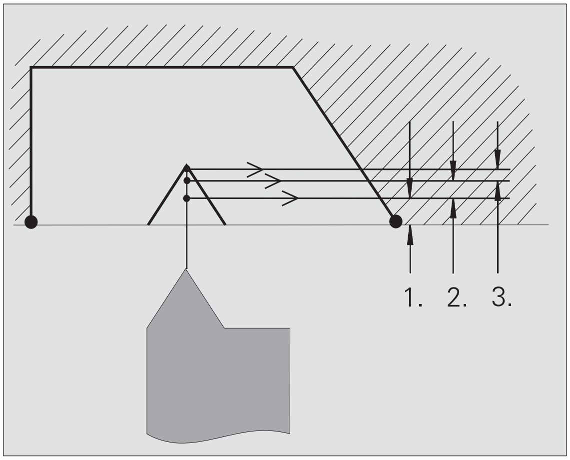
> An undercut may only be machined in a right-angled paraxial contour corner in the linear axis.

Thread and undercut cycles	Symbol
Thread cycle Longitudinal single or multi-start thread	
Tapered thread Tapered single or multi-start thread	
API thread Single or multi-start API thread (API: American Petroleum Institute)	
Undercut DIN 76 Thread undercut and thread chamfer	
Undercut DIN 509 E Undercut and cylinder chamfer	
Undercut DIN 509 F Undercut and cylinder chamfer	

Angle of infeed (thread angle)

With some thread cycles, you can indicate the angle of infeed. The figures to the right show the operating sequence of the MANUALplus at an angle of infeed of –30° (figure at upper right) and an angle of infeed of 0° (figure at center right).





Thread depth, proportioning of cuts

The thread depth is programmed for all thread cycles. MANUALplus reduces the cutting depth with each cut (see figure at center right).

Handwheel superpositioning in threading cycles

As of software version 526 488-09, you can influence the current thread cut via handwheel superpositioning in X and Z, and so optimize the production of the thread. Handwheel superpositioning must be supported by the machine manufacturer, and is activated via a switch on the machine operating panel.

Handwheel superpositioning is restricted as follows:

- X direction (thread depth): Depends on the current cutting depth; the starting and end points of the thread in X are not exceeded.
- Z direction: No more than one turn; the starting and end points of the thread in Z are not exceeded.

Thread run-in / thread run-out

The slide requires a run-in distance to accelerate to the programmed feed rate before starting the actual thread, and a run-out distance at the end of the thread to decelerate again.

Calculation of chamfer (run-in) length:

BA > 0.75 * (F*S)² / a + 0.15

Calculation of run-out length:

BA > 0.75 * (F*S)² / e + 0.15

BA: Minimum run-in length

BE: Minimum run-out length

F: Thread pitch in mm/revolution

S: Speed in **revolutions/second**

a: Acceleration in mm/s² (see “Configuration parameters” on page 435–1105 "Acceleration at start of block")

e: Acceleration in mm/s² (see “Configuration parameters” on page 435 – 1105 "Acceleration at start of block")

If the run-in / run-out length is too short, the thread may not attain the expected quality. In this case, MANUALplus displays a warning.

Last cut

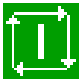
After the cycle is finished, the MANUALplus presents the **Last cut** option. You can use this function to repeat the last thread cut with an updated tool compensation.

Sequence for the “last cut” function:

Initial situation: The thread cut cycle has been performed, and the thread depth is not correct.

- Perform the tool compensation

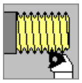
- Press ***Last cut***



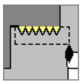
- Activate **Cycle Start**
- Check the thread

> The tool compensation and the “last cut” can be repeated as often as necessary until the thread is correct.

Thread cycle (longitudinal)



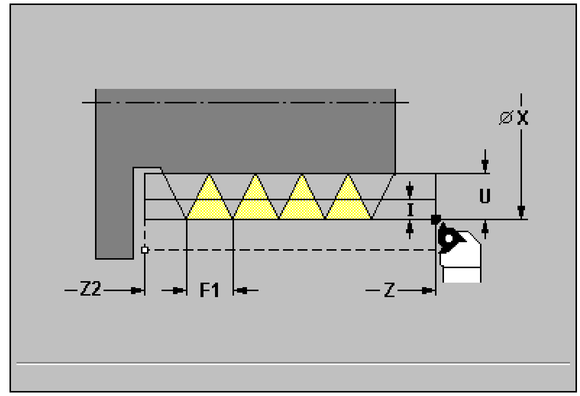
Call the thread-cutting menu.

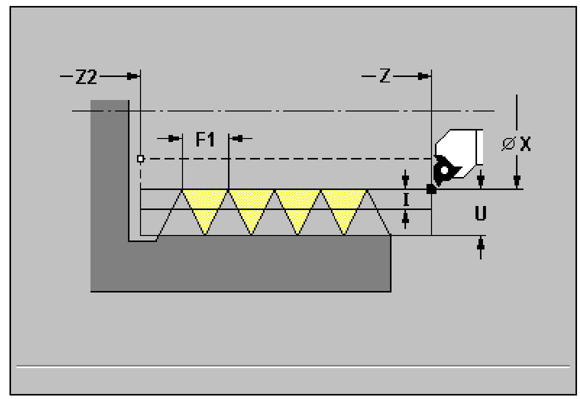


Select the "Thread cycle."

Inner thread

Inner thread soft key
On: Internal thread
Off: External thread





This cycle cuts a single external or internal thread with a thread angle of 30°. Tool infeed is performed in the X axis only.

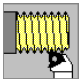
Cycle parameters

- **X, Z starting point** of thread
- **Z2 end point of thread**
- **F1 thread pitch** (= feed rate)
- **U thread depth**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **I 1st cutting depth**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth
  - I=U: One cut
  - No input: Calculation from U and F1
- **T tool number**
- **S spindle speed / cutting speed**

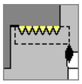
Cycle run

1. Calculate the proportioning of cuts.
2. Start first pass at Z.
3. Move to end point Z2 at programmed feed rate.
4. Return on paraxial path and approach for next pass.
5. Repeat 3 and 4 until "depth U" has been reached.

Thread cycle (longitudinal)—Expanded



Call the thread-cutting menu.



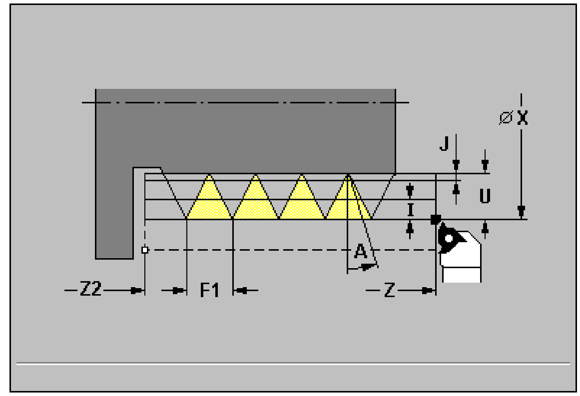
Select the "Thread cycle."

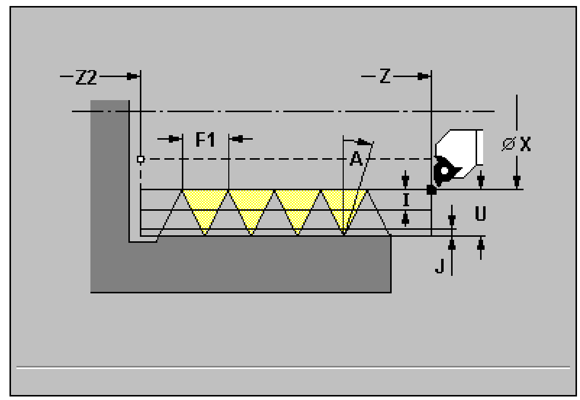
Expanded

Press the ***Expanded*** soft key.

Inner thread

Inner thread soft key
On: Internal thread
Off: External thread





This cycle cuts a single or multi-start external or internal thread. The thread starts at starting point X and ends at end point Z2 (without a thread run-in or run-out).

Cycle parameters

- **X, Z starting point** of thread (without run-in)
- **Z2 end point of thread** (without run-out)
- **F1 thread pitch** (= feed rate)
- **U thread depth**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **I 1st cutting depth**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth down to J.
  - I=U: One cut
  - No input: Calculation from U and F1
- **A feed angle (default: 30°):**
  Range: –60° < A < 60°
  - A<0: Infeed on left thread flank
  - A>0: Infeed on right thread flank
- **J remaining cutting depth**—default: 1/100 mm
- **D number of thread starts**—default: 1 single-start thread
- **E incremental gradient**—default: 0
  - E=0: Constant pitch
  - E>0: Increase the pitch per revolution by E
  - E<0: Decrease the pitch per revolution by E
- **T tool number**
- **S spindle speed / cutting speed**

Cycle run

1 Calculate the proportioning of cuts.

2 Start first pass for first thread groove at Z.

3 Move to end point Z2 at programmed feed rate.

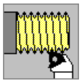
4 Return on paraxial path and approach for next thread groove.

5 Repeat 3 and 4 for all thread grooves.

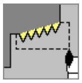
6 Approach for next pass, taking the reduced cutting depth and the "feed angle A" into account.

7 Repeat 3 to 6 until "no. threads D" **and** "depth U" are reached.

Tapered thread



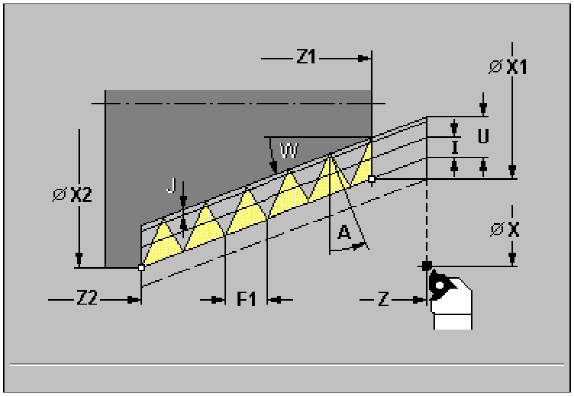
Call the thread-cutting menu.

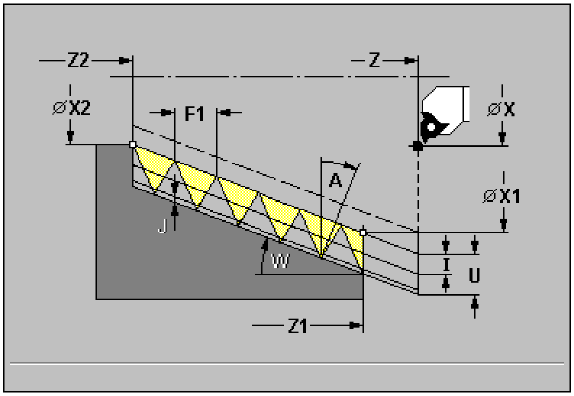


Select "Tapered thread."

Inner thread

Inner thread soft key
On: Internal thread
Off: External thread





This cycle cuts a single or multi-start tapered external or internal thread.

Cycle parameters

- **X, Z starting point**
- **X1, Z1 starting point** of thread (without run-in)
- **X2, Z2 end point of thread** (without run-out)
- **F1 thread pitch** (= feed rate)
- **U thread depth**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **I 1st cutting depth**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth down to J.
  - I=U: One cut
  - No input: Calculation from U and F1
- **A feed angle** (default: 30°): Range: –60° < A < 60°
  - A<0: Infeed on left thread flank
  - A>0: Infeed on right thread flank
- **W taper angle:** Range: –60° < A < 60°
- **J remaining cutting depth**—default: 1/100 mm
- **T tool number**
- **S spindle speed / cutting speed**
- **D number of thread starts**—default: 1 single-start thread
- **E incremental gradient**—default: 0
  - E=0: Constant pitch
  - E>0: Increase the pitch per revolution by E
  - E<0: Decrease the pitch per revolution by E

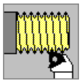
Parameter combinations for taper angle:

- X1/Z1, X2/Z2
- X1/Z1, Z2, W
- Z1, X2/Z2, W

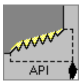
Cycle run

1. Calculate the proportioning of cuts.
2. Move to starting point X1, Z1.
3. Move to end point Z2 at programmed feed rate.
4. Return on paraxial path and approach for next thread groove.
5. Repeat 3 and 4 for all thread grooves.
6. Approach for next pass, taking the reduced cutting depth and the "feed angle A" into account.
7. Repeat 3 to 6 until "no. threads D" **and** "depth U" are reached.

API thread



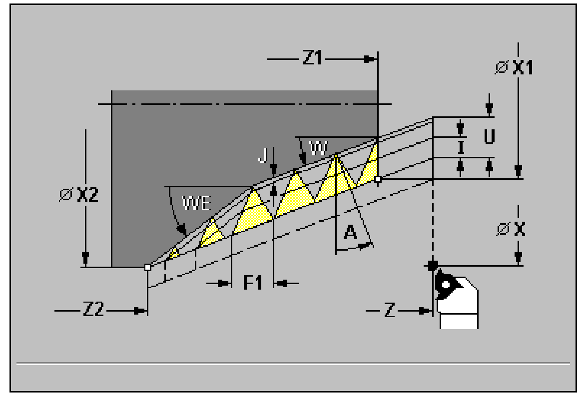
Call the thread-cutting menu.

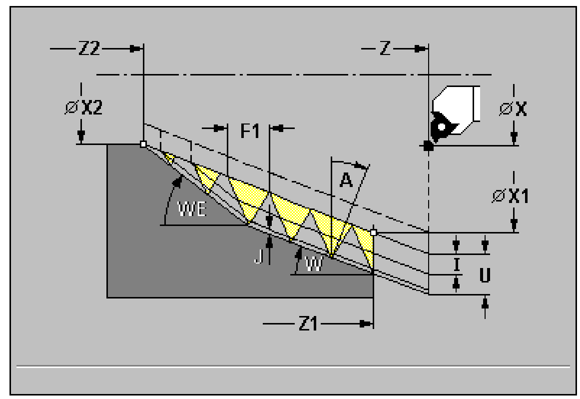


Select "API thread."

Inner thread

Inner thread soft key
On: Internal thread
Off: External thread





This cycle cuts a single or multi-start API external or internal thread. The depth of thread decreases at the overrun at the end of thread.

Cycle parameters

- **X, Z starting point**
- **X1, Z1 starting point** of thread (without run-in)
- **X2, Z2 end point of thread** (without run-out)
- **F1 thread pitch** (= feed rate)
- **U thread depth**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **I 1st cutting depth**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth down to J.
  - I=U: One cut
  - No input: Calculation from U and F1
- **A feed angle** (default: 30°):
  Range: –60° < A < 60°
  - A<0: Infeed on left thread flank
  - A>0: Infeed on right thread flank
- **W taper angle:** Range: –45° < W < 45°
- **WE run-out angle:** Range: 0° < WE < 90°
- **J remaining cutting depth**—default: 1/100 mm
- **T tool number**
- **S spindle speed / cutting speed**
- **D number of thread starts**—default: 1 single-start thread

Parameter combinations for taper angle:

- X1/Z1, X2/Z2
- X1/Z1, Z2, W
- Z1, X2/Z2, W

Cycle run

1 Calculate the proportioning of cuts.

2 Move to thread starting point X1, Z1.

3 Move to end point Z2 at programmed feed rate, taking the "run-out angle WE" into account.

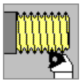
4 Return on paraxial path and approach for next thread groove.

5 Repeat 3 and 4 for all thread grooves.

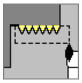
6 Approach for next pass, taking the reduced cutting depth and the "feed angle A" into account.

7 Repeat 3 to 6 until "no. threads D" **and** "depth U" are reached.

Recut (longitudinal) thread



Call the thread-cutting menu.



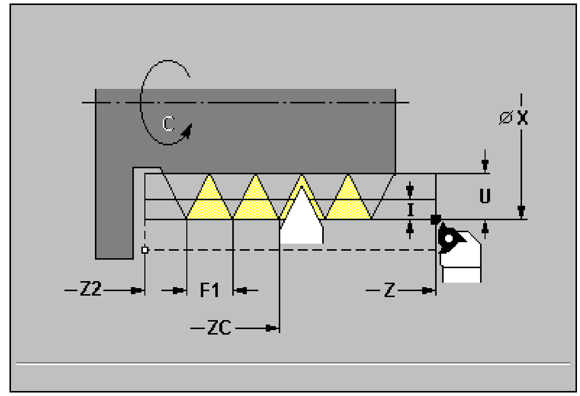
Select the "Thread cycle."

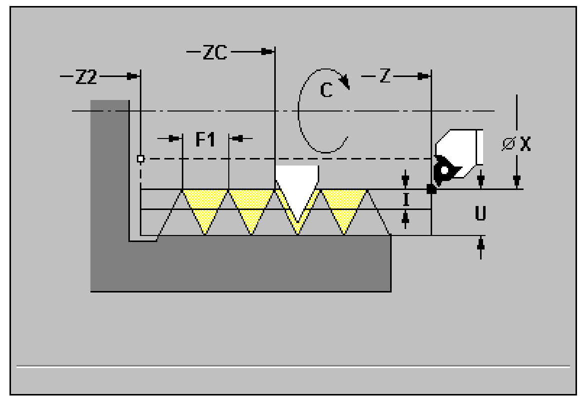
Re-cut

Press the ***Recut*** soft key.

Inner thread

Inner thread soft key
On: Internal thread
Off: External thread





The cycle reworks a single-start thread. Since you have already unclamped the workpiece, MANUALplus needs to know the exact position of the thread. For this, place the cutting edge of the tap drill in the center of a groove and transfer the positions to the parameters C and ZC by pressing the ***Take over position*** soft key. From these values the cycle then calculates the angle of the spindle at the starting point Z.

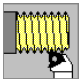
Cycle parameters

- **Z2 end point of thread**
- **C measured angle**
- **ZC measured position**
- **F1 thread pitch** (= feed rate)
- **U thread depth**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **I 1st cutting depth**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth
  - I=U: One cut
  - No input: Calculation from U and F1

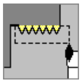
Cycle run

1. Pre-position threading tool to center of thread groove.
2. Transfer the tool position ZC and the spindle angle C with ***Take over position.***
3. Move the tool manually out of the thread groove.
4. Position the tool to starting point X, Z.
5. Start cycle with "Input finished", then press "Cycle START."

Recut (longitudinal) thread—Expanded



Call the thread-cutting menu.



Select the "Thread cycle."

Expanded

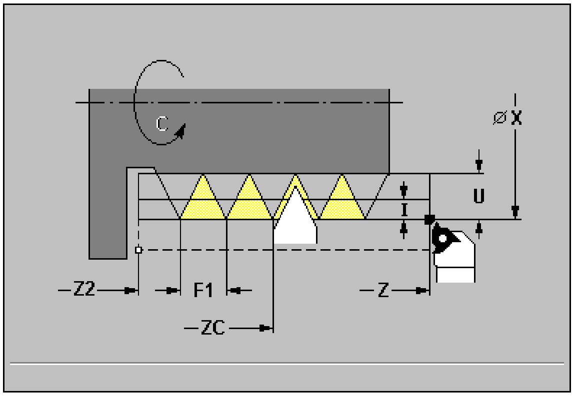
Press the ***Expanded*** soft key.

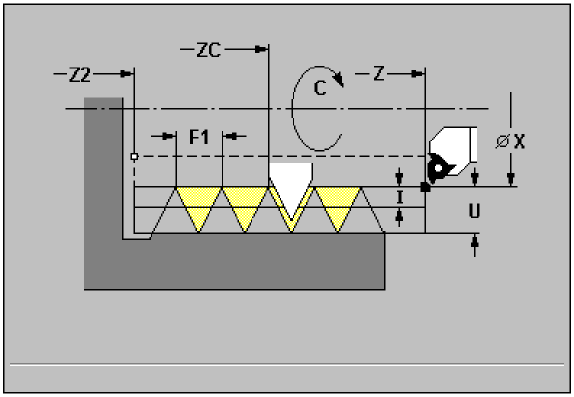
Re-cut

Press the ***Recut*** soft key.

Inner thread

Inner thread soft key
On: Internal thread
Off: External thread





This cycle recuts a single or multi-start external or internal thread. Since you have already unclamped the workpiece, MANUALplus needs to know the exact position of the thread. For this, place the cutting edge of the tap drill in the center of a groove and transfer the positions to the parameters C and ZC by pressing the ***Take over position*** soft key. From these values the cycle then calculates the angle of the spindle at the starting point Z.

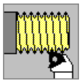
Cycle parameters

- **Z2 end point of thread** (without run-out)
- **C measured angle**
- **ZC measured position**
- **F1 thread pitch** (= feed rate)
- **U thread depth**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **I 1st cutting depth**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth down to J.
  - I=U: One cut
  - No input: Calculation from U and F1
- **A feed angle (default: 30°):** Range: –60° < A < 60°
  - A<0: Infeed on left thread flank
  - A>0: Infeed on right thread flank
- **J remaining cutting depth**—default: 1/100 mm
- **D number of thread starts**—default: 1 single-start thread

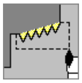
Cycle run

1. Pre-position threading tool to center of thread groove.
2. Transfer the tool position ZC and the spindle angle C with ***Take over position.***
3. Move the tool manually out of the thread groove.
4. Position the tool to starting point X, Z.
5. Start cycle with "Input finished", then press "Cycle START."

Recut tapered thread



Call the thread-cutting menu.



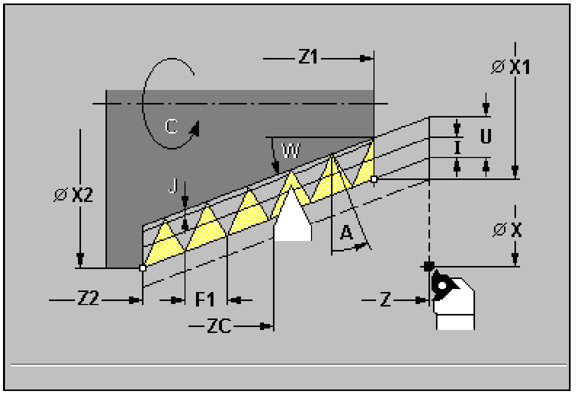
Select "Tapered thread."

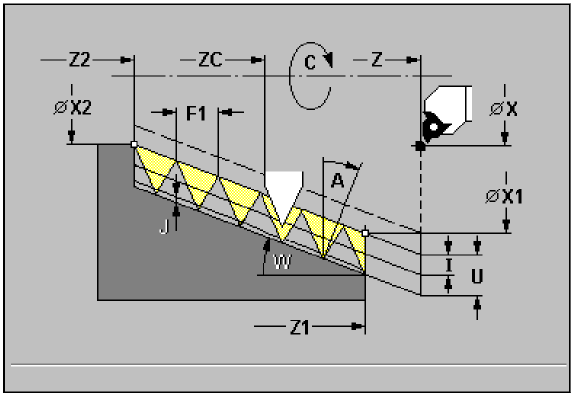
Re-
cut

Press the ***Recut*** soft key.

Inner
thread

Inner thread soft key
On: Internal thread
Off: External thread





The cycle recuts a single or multi-start tapered external or internal thread. Since you have already unclamped the workpiece, MANUALplus needs to know the exact position of the thread. For this, place the cutting edge of the tap drill in the center of a groove and transfer the positions to the parameters C and ZC by pressing the ***Take over position*** soft key. From these values the cycle then calculates the angle of the spindle at the starting point Z.

Cycle parameters

- **`X1, Z1 starting point`** of thread (without run-in)
- **`X2, Z2 end point of thread`** (without run-out)
- **`C measured angle`**
- **`ZC measured position`**
- **`F1 thread pitch`** (= feed rate)
- **`U thread depth`**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **`I 1st cutting depth`**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth down to J.
  - I=U: One cut
  - No input: Calculation from U and F1
- **`A feed angle`** (default: 30°):
  Range: –60° < A < 60°
  - A<0: Infeed on left thread flank
  - A>0: Infeed on right thread flank
- **`W taper angle:`** Range: –60° < A < 60°
- **`J remaining cutting depth`**—default: 1/100 mm

Cycle run

1 Pre-position threading tool to center of thread groove.

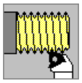
2 Transfer the tool position ZC and the spindle angle C with ***Take over position.***

3 Move the tool manually out of the thread groove.

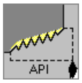
4 Position the tool **in front of** the workpiece.

5 Start cycle with "Input finished", then press "Cycle START."

Recut API thread



Call the thread-cutting menu.



Select "API thread."

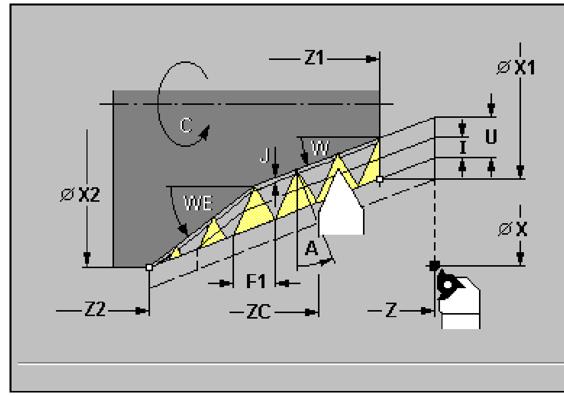
Re-cut

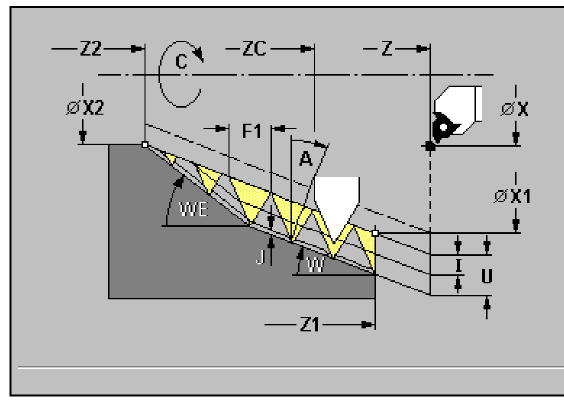
Press the ***Recut*** soft key.

Inner thread

Inner thread soft key
On: Internal thread
Off: External thread

The cycle recuts a single or multi-start API external or internal thread. Since you have already unclamped the workpiece, MANUALplus needs to know the exact position of the thread. For this, place the cutting edge of the tap drill in the center of a groove and transfer the positions to the parameters C and ZC by pressing the ***Take over position*** soft key. From these values the cycle then calculates the angle of the spindle at the starting point Z.





Cycle parameters

- **X1, Z1 starting point** of thread (without run-in)
- **X2, Z2 end point of thread** (without run-out)
- **C measured angle**
- **ZC measured position**
- **F1 thread pitch** (= feed rate)
- **U thread depth**
  - No input: Depth is calculated
    External thread: U=0.6134*F1
    Internal thread: U=–0.5413*F1
- **I 1st cutting depth**
  - I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth down to J.
  - I=U: One cut
  - No input: Calculation from U and F1
- **A feed angle** (default: 30°):
  Range: –60° < A < 60°
  - A<0: Infeed on left thread flank
  - A>0: Infeed on right thread flank
- **W taper angle:** Range: –45° < W < 45°
- **WE run-out angle:** Range: 0° < WE < 90°
- **J remaining cutting depth**—default: 1/100 mm

Cycle run

1 Pre-position threading tool to center of thread groove.

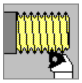
2 Transfer the tool position ZC and the spindle angle C with ***Take over position.***

3 Move the tool manually out of the thread groove.

4 Position the tool **in front of** the workpiece.

5 Start cycle with "Input finished", then press "Cycle START."

Undercut DIN 76

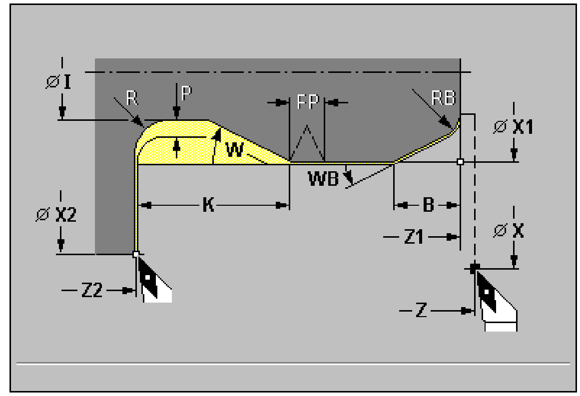


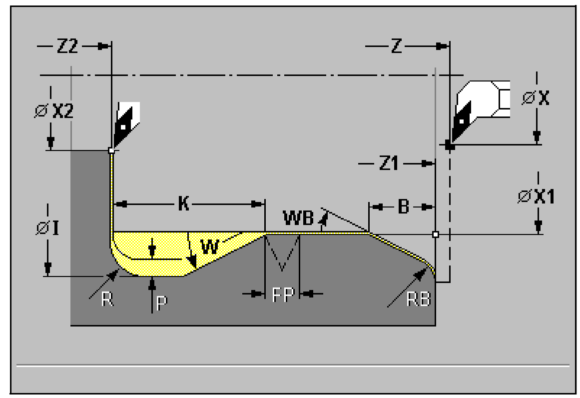
Call the thread-cutting menu.

Select the "Undercut DIN 76" cycle.

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position (see figures at right).
On: Tool returns to the starting point (see figures on next page).





The cycle machines a thread undercut according to DIN76, a thread chamfer, then the cylinder, and finishes with the plane surface. The thread chamfer is executed when you enter at least one of the parameters "B" or "RB."

Cycle parameters

- **X, Z starting point**
- **X1, Z1 starting point of cylinder**
- **X2, Z2 end point on plane surface**
- **FP thread pitch**—default: Value from standard table
- **E reduced feed rate** for the plunge cut and the thread chamfer—default: Feed rate F
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**
- **I undercut diameter**—default: Value from standard table
- **K undercut length**—default: Value from standard table
- **W undercut angle**—default: Value from standard table
- **R undercut radius** (on both sides of the undercut)—default: Value from standard table
- **P undercut oversize**
  - P>0: Division into pre-turning and finish-turning—"P" is the longitudinal oversize; the transverse oversize is always 0.1 mm
  - No input: Machining in one cut
- **B cylinder start chamfer**—default: No start chamfer
- **WB 1st cut angle**—default: 45°
- **RB chamfer radius**—default: No chamfer radius

All parameters that you enter will be accounted for—even if the standard table prescribes other values. Undercut parameters that are not defined are automatically calculated from the standard table (see "DIN 76—undercut parameters" on page 525"):

- "Thread pitch FP" is calculated from the diameter X1.
- The parameters I, K, W, and R are calculated from FP.

Cycle run

1 Approach workpiece from X, Z
- to starting point X1, or
- for the thread chamfer.

2 Machine thread chamfer, if defined.

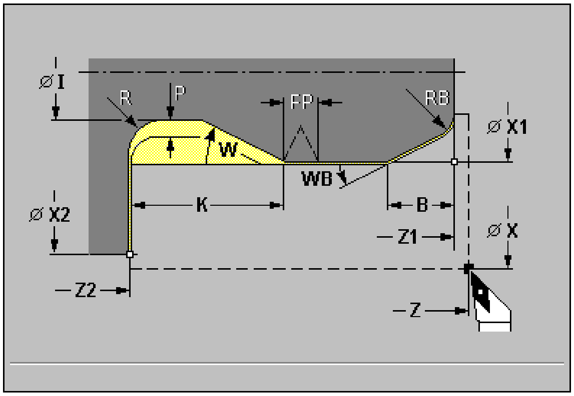
3 Finish cylinder up to beginning of undercut.

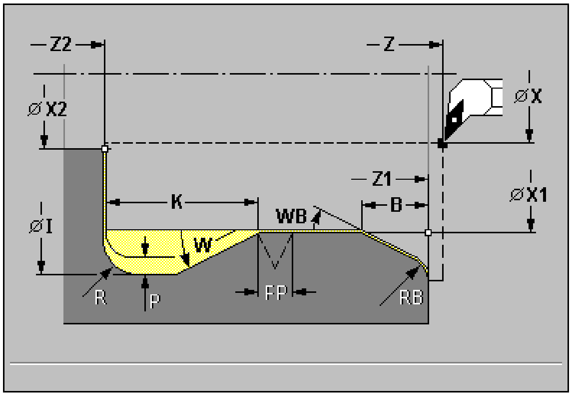
4 Pre-machine undercut, if defined.

5 Machine undercut.

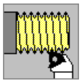
6 Finish to end point X2.

7
- Without return: Tool remains at the end point X2.
- With return: Return to starting point on diagonal path.





Undercut DIN 509 E

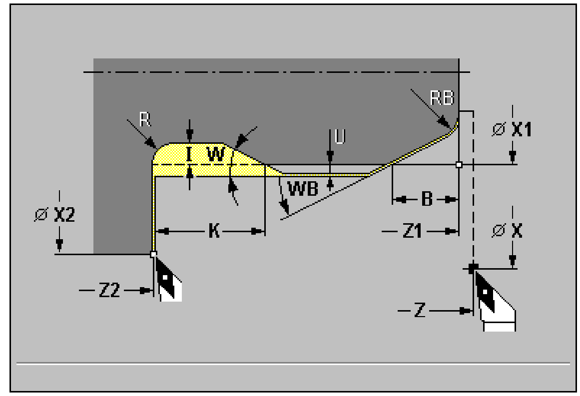


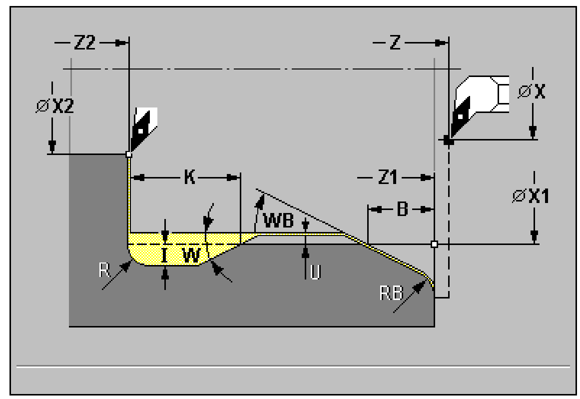
Call the thread-cutting menu.

Select the "Undercut DIN 509 E" cycle.

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position (see figures at right).
On: Tool returns to the starting point (see figures on next page).





The cycle machines a thread undercut according to DIN 509 type E, a cylinder start chamfer, then the adjoining cylinder, and finishes with the plane surface. You can define a finishing oversize for the area of the cylinder. The cylinder start chamfer is executed when you enter at least one of the parameters "B" or "RB."

Cycle parameters

- **X, Z starting point**
- **X1, Z1 starting point of cylinder**
- **X2, Z2 end point on plane surface**
- **E reduced feed rate** for the plunge cut and the thread chamfer—default: Feed rate F
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**
- **I undercut depth**—default: Value from standard table
- **K undercut length**—default: Value from standard table
- **W undercut angle**—default: Value from standard table
- **R undercut radius** (on both sides of the undercut)—default: Value from standard table
- **B cylinder start chamfer**—default: No start chamfer
- **WB 1st cut angle**—default: 45°
- **RB chamfer radius**—default: No chamfer radius
- **U finishing oversize** for the area of the cylinder—default: 0

All parameters that you enter will be accounted for—even if the standard table prescribes other values. If the parameters I, K, W and R are not defined, MANUALplus determines these values from the cylinder diameter in the standard table (see “DIN 509 E, DIN 509 F—undercut parameters” on page 527).

Cycle run

1 Approach workpiece from X, Z
- to cylinder starting point X1, or
- for the thread chamfer.

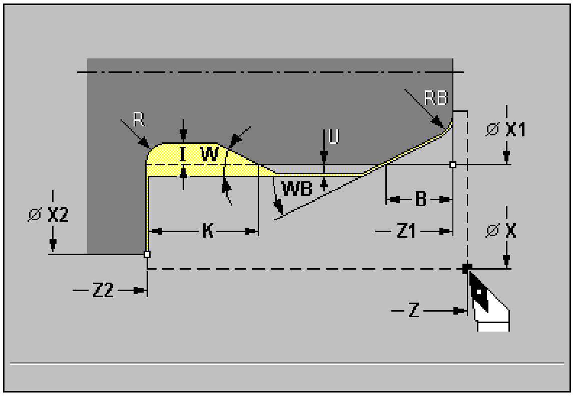
2 Machine thread chamfer, if defined.

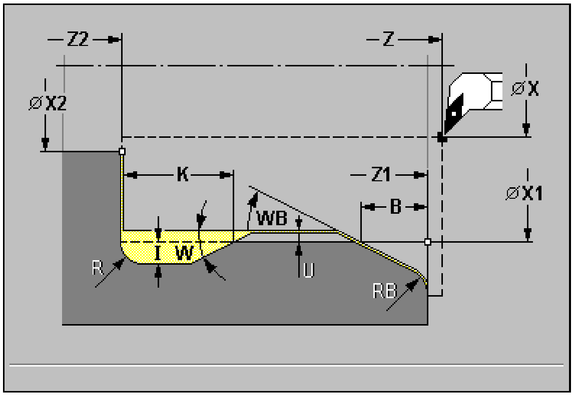
3 Finish cylinder up to beginning of undercut.

4 Machine undercut.

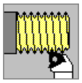
5 Finish to end point X2 on plane surface.

6
- Without return: Tool remains at the end point X2.
- With return: Return to starting point on diagonal path.





Undercut DIN 509 F

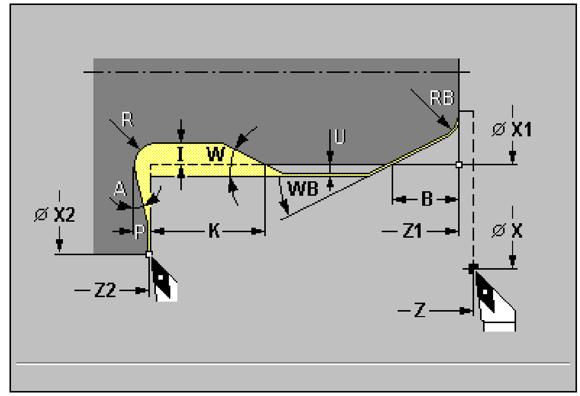


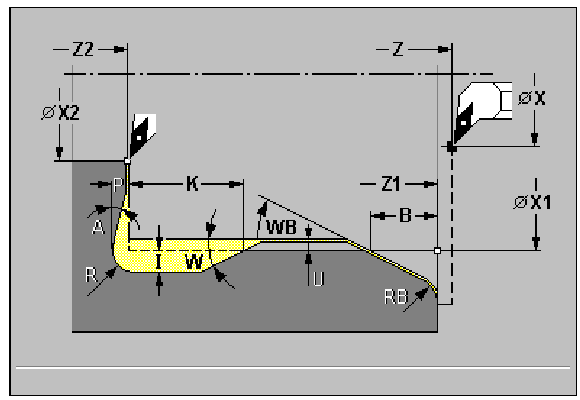
Call the thread-cutting menu.

Select the "Undercut DIN 509 F" cycle.

With return

With return soft key:
Off: When the cycle is completed, the tool remains at the cycle end position (see figures at right).
On: Tool returns to the starting point (see figures on next page).





The cycle machines a thread undercut according to DIN 509 type F, a cylinder start chamfer, then the adjoining cylinder, and finishes with the plane surface. You can define a finishing oversize for the area of the cylinder. The cylinder start chamfer is executed when you enter at least one of the parameters "B" or "RB."

Cycle parameters

- **`X, Z starting point`**
- **`X1, Z1 starting point of cylinder`**
- **`X2, Z2 end point on plane surface`**
- **`E reduced feed rate`** for the plunge cut and the thread chamfer—default: Feed rate F
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**
- **`I undercut depth`**—default: Value from standard table
- **`K undercut length`**—default: Value from standard table
- **`W undercut angle`**—default: Value from standard table
- **`R undercut radius`** (on both sides of the undercut)—default: Value from standard table
- **`P transverse depth`**—default: Value from standard table
- **`A transverse angle`**—default: Value from standard table
- **`B cylinder start chamfer`**—default: No start chamfer
- **`WB 1st cut angle`**—default: 45°
- **`RB chamfer radius`**—default: No chamfer radius
- **`U finishing oversize`**—default: 0

All parameters that you enter will be accounted for—even if the standard table prescribes other values. If the parameters I, K, W, R, P and A are not defined, MANUALplus determines these values from the cylinder diameter in the standard table (see "DIN 509 E, DIN 509 F—undercut parameters" on page 527).

Cycle run

1 Approach workpiece from X, Z
- to cylinder starting point X1, or
- for the thread chamfer.

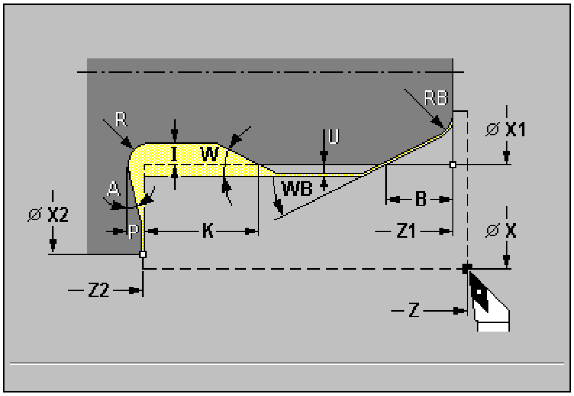
2 Machine thread chamfer, if defined.

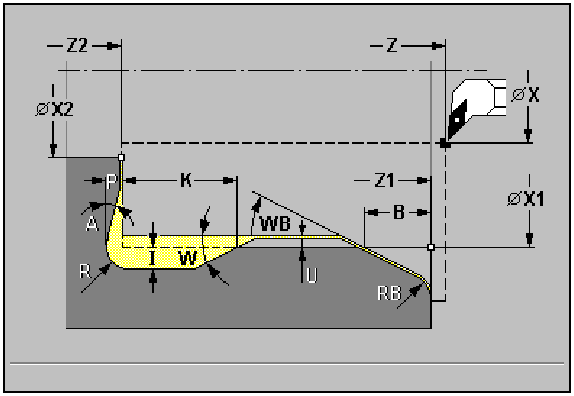
3 Finish cylinder up to beginning of undercut.

4 Machine undercut.

5 Finish to end point X2 on plane surface.

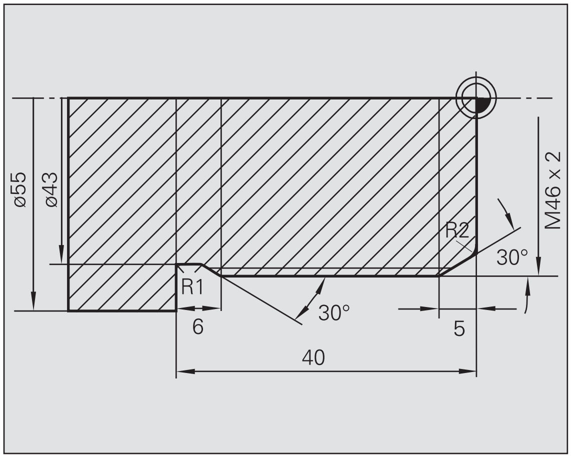
6
- Without return: Tool remains at the end point X2.
- With return: Return to starting point on diagonal path.

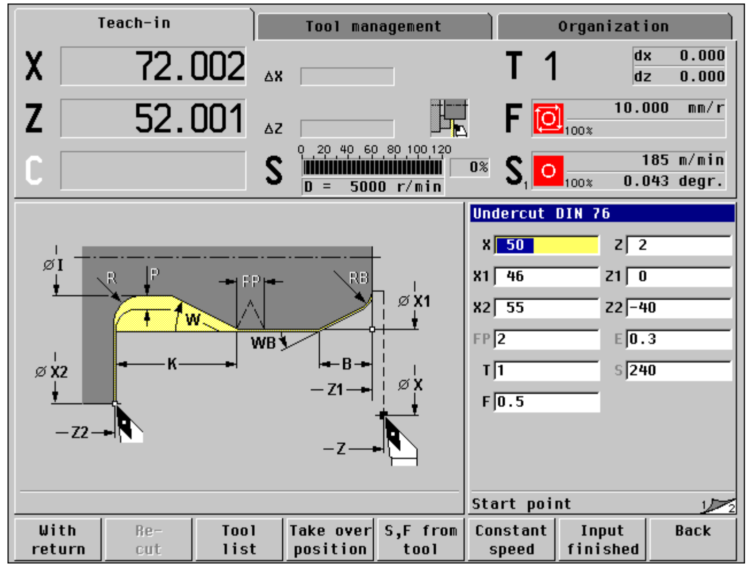




Examples of thread and undercut cycles

External thread and thread undercut

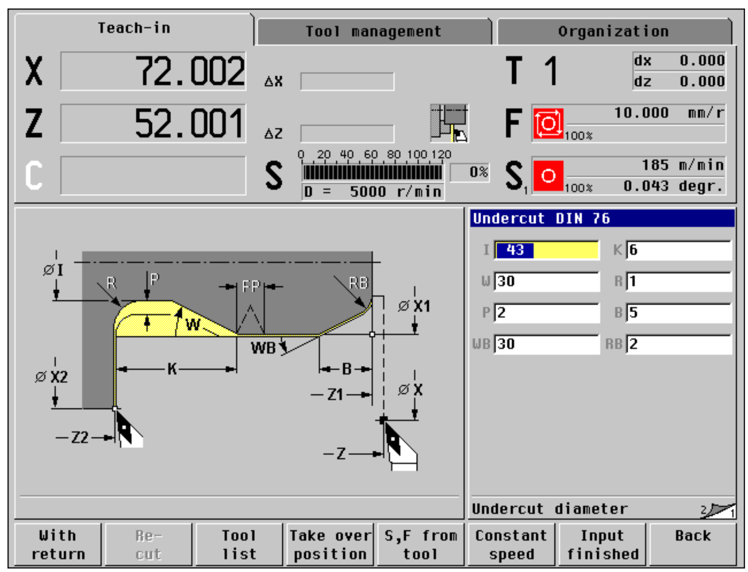




The machining operation is to be performed in two steps. Thread undercut DIN 76 produces the undercut and thread chamfer. In the second step, the thread cycle cuts the thread.

First step

The parameters for the undercut and thread chamfer are programmed in two superimposed input windows (see figure at right).



Tool data

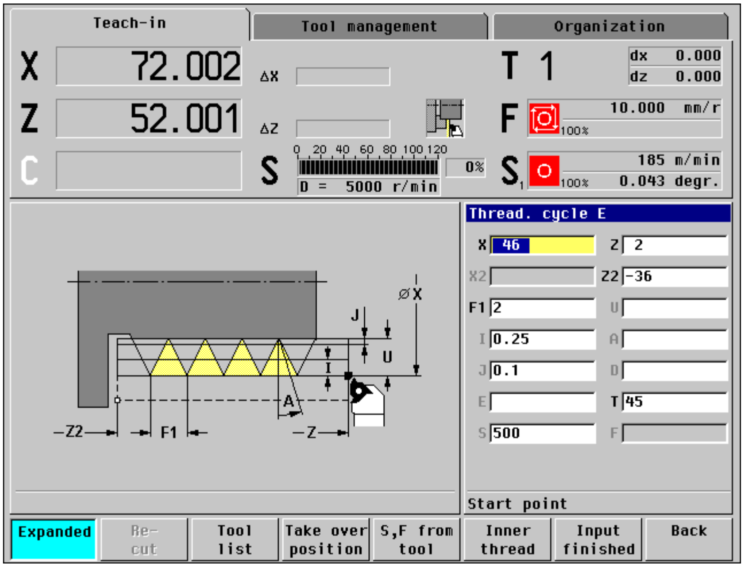
- Lathe tool (for external machining)
- WO = 1—Tool orientation
- A = 93°—Tool angle
- B = 55°—Point angle

Second step

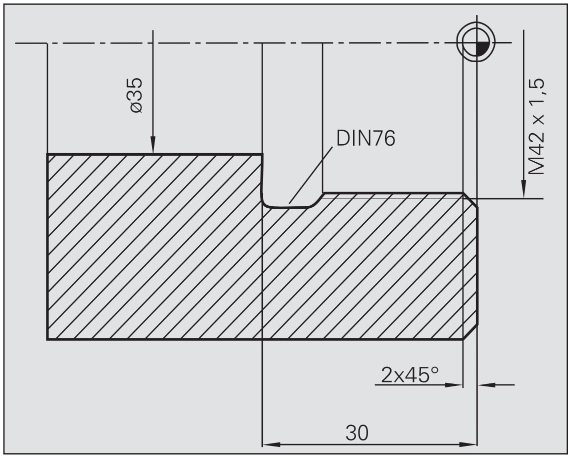
The "Thread cycle (longitudinal)—Expanded" cuts the thread. The cycle parameters define the thread depth and the proportioning of cuts (see figure at top right).

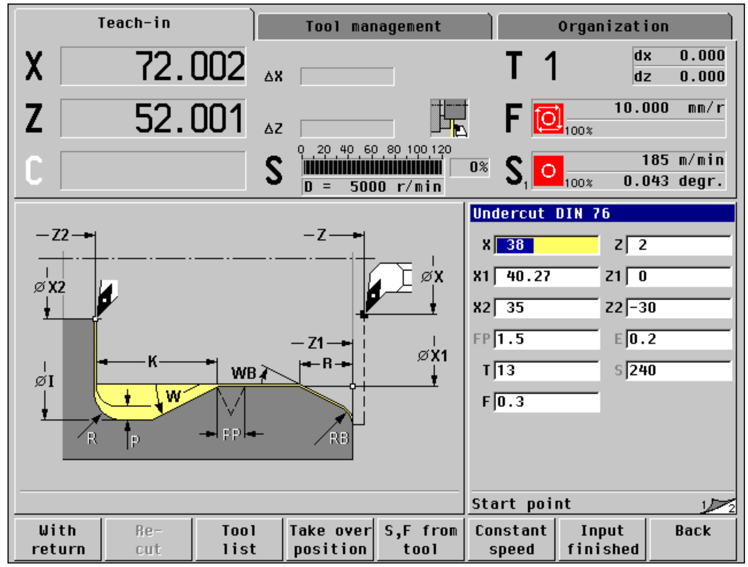
Tool data

- Threading tool (for external machining)
- WO = 1—Tool orientation



Internal thread and thread undercut





The machining operation is to be performed in two steps. Thread undercut DIN 76 produces the undercut and thread chamfer. In the second step, the thread cycle cuts the thread.

First step

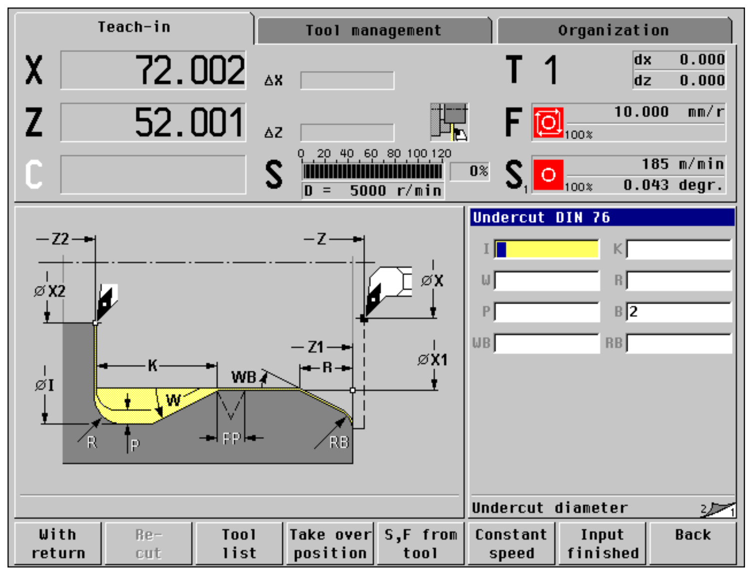
The parameters for the undercut and thread chamfer are programmed in two superimposed input windows (see figure at bottom right and figure on next page at top right).

MANUALplus determines the undercut parameters from the standard table.

For the thread chamfer, you only need to enter the chamfer width. The angle of 45° is the default value for the "1st cut angle WB."

Tool data

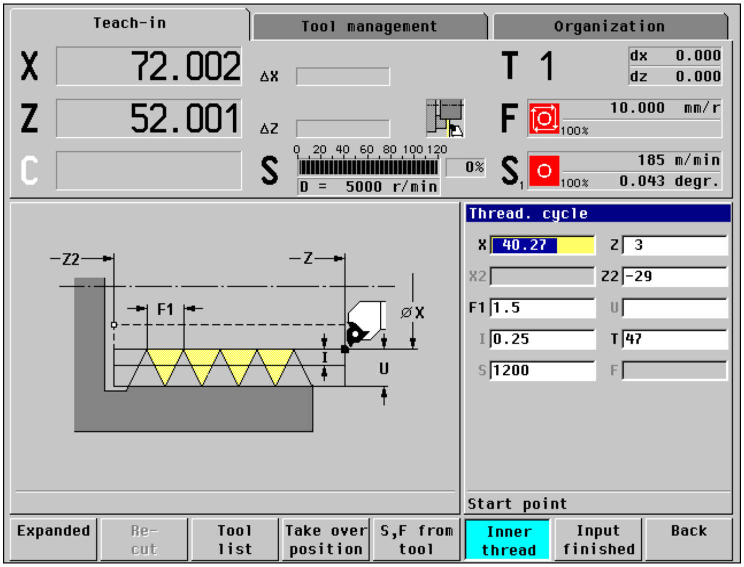
- Lathe tool (for internal machining)
- WO = 7—Tool orientation
- A = 93°—Tool angle
- B = 55°—Point angle



Second step

The "Thread cycle (longitudinal)" cuts the thread. The thread pitch is defined. MANUALplus automatically determines all other values from the standard table (see figure at right).

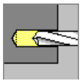
You must pay attention to the setting of the ***Inner thread*** soft key.



Tool data

- Threading tool (for internal machining)
- WO = 7—Tool orientation

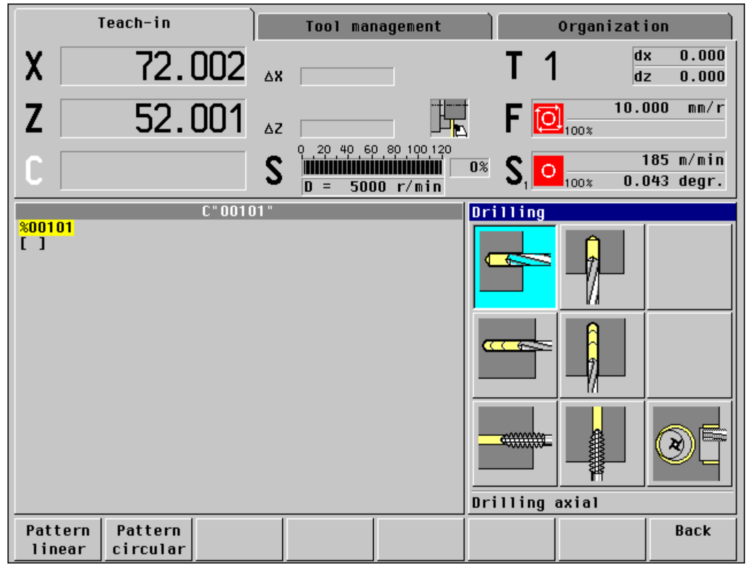
4.7 Drilling Cycles

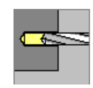
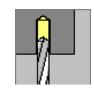
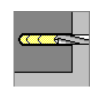
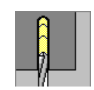
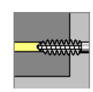
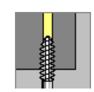
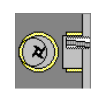


The drilling cycles allow you to machine axial and radial holes.

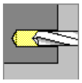
For pattern machining, see "Drilling/ Milling Patterns" on page 227.

> The "constant cutting speed" may only be programmed with driven tools on machines with spindle control.

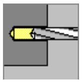


Drilling cycles	Symbol	
Axial/Radial drilling cycle For drilling single holes and patterns		
Axial/Radial deep-drilling cycle For drilling single holes and patterns		
Axial/Radial tapping cycle For drilling single holes and patterns		
Thread milling For milling threads in existing holes		

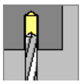
Drilling, axial/radial



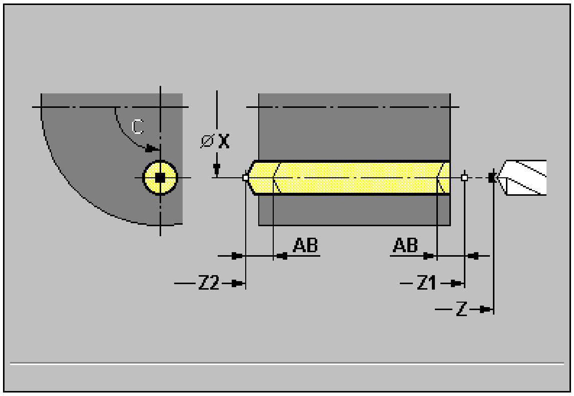
Call the drilling menu.

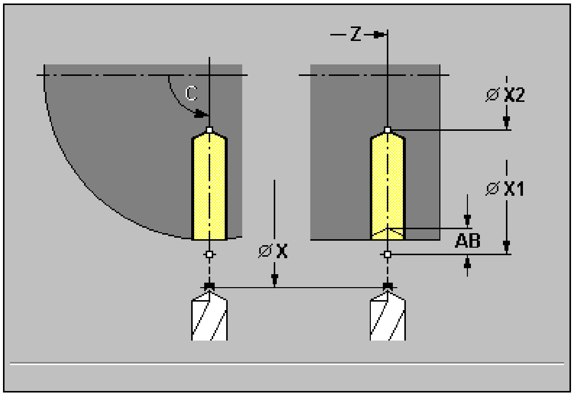


Select the "Drilling, axial" cycle.



Select the "Drilling, radial" cycle.





This cycle drills a hole on the face / lateral surface of the workpiece.

Cycle parameters

- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **E dwell time** for chip breaking at end of hole—default: 0
- **AB drilling lengths**—default: 0
- **V drilling variants**—default: 0
  - 0: No feed rate reduction
  - 1: Feed reduction for through-boring
  - 2: Feed reduction for pre-drilling
  - 3: Feed reduction for pre-drilling and through-boring
- **D retreat**—default: 0
  - 0: Rapid traverse
  - 1: Feed rate
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Drilling axial:

- **Z1 starting point of hole**—default: Drilling starts from position Z
- **Z2 end point of hole**

Drilling radial:

- **X1 starting point of hole**—default: Drilling starts from position X
- **X2 end point of hole**

- If "AB" **and** "V" are programmed, the feed rate is reduced by 50% during both pre-drilling and through-boring.
- MANUALplus uses the tool parameter "driven tool" to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run

1 Position spindle to "spindle angle C" (in Manual mode, machining starts from the current spindle angle).

2 If defined, move at rapid traverse to
- Z1 (axial).
- X1 (radial).

3 Start drilling at reduced feed rate, if defined.

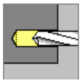
4 Depending on "V":
- Drill at programmed feed rate to
  End point Z2 (axial).
  End point X2 (radial).
- Remain at end of hole for dwell time "E," if defined.

  or
- Drill at programmed feed rate to position
  Z2 – AB (axial).
  X2 – AB (radial).
- Drill at reduced feed rate to
  End point Z2 (axial).
  End point X2 (radial).

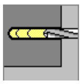
5 If X1/Z1 has been defined, retract to
- Starting point of hole Z1 (axial).
- Starting point of hole X1 (radial).

  If X1/Z1 has **not** not been defined, retract to
- Starting point Z (axial).
- Starting point X (radial).

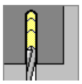
Deep-hole drilling, axial/radial



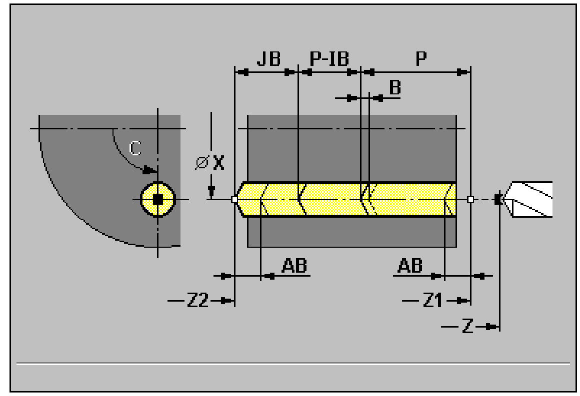
Call the drilling menu.

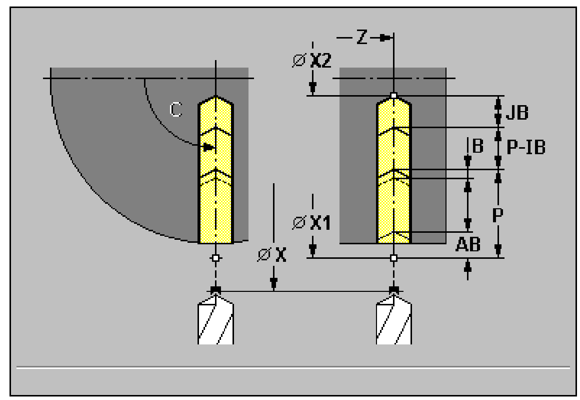


Select the "Deep-hole drilling, axial" cycle.



Select the "Deep-hole drilling, radial" cycle.





The cycle produces a bore hole on the face / lateral surface in several passes. After each pass, the drill retracts and, after a dwell time, advances again to the first pecking depth, minus the safety clearance. You define the first pass with "1st hole depth P." MANUALplus then automatically reduces the drilling depth with each subsequent pass by the "reducing value IB," however, without falling below the "minimum drilling depth JB."

Cycle parameters

- **`X, Z starting point`**
- **`C spindle angle`** (C-axis position)—default: Current spindle angle
- **`P 1st hole depth`**—default: Hole will be drilled in one pass
- **`IB hole depth reduction value`**—default: 0
- **`JB minimum hole depth`**—default: 1/10 of P
- **`B return length`**—default: Retract to starting point of hole
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**
- **`E dwell time`** for chip breaking at end of hole—default: 0
- **`AB drilling lengths`**—default: 0
- **`V drilling variants`**—default: 0
  - 0: No feed rate reduction
  - 1: Feed reduction for through-boring
  - 2: Feed reduction for pre-drilling
  - 3: Feed reduction for pre-drilling and through-boring
- **`D retreat`** retraction speed and infeed within the hole—default: 0
  - 0: Rapid traverse
  - 1: Feed rate

Drilling axial:

- **`Z1 starting point of hole`**—default: Drilling starts from position Z
- **`Z2 end point of hole`**

Drilling radial:

- **`X1 starting point of hole`**—default: Drilling starts from position X
- **`X2 end point of hole`**

> - If "AB" **and** "V" are programmed, the feed rate is reduced by 50% during both pre-drilling and through-boring.
> - MANUALplus uses the tool parameter "driven tool" to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run

1 Position spindle to "spindle angle C" (in Manual mode, machining starts from the current spindle angle).

2 If defined, move at rapid traverse to
- Z1 (axial).
- X1 (radial).

3 First pass (pecking depth: P)—Drill with reduced feed rate, if defined.

4 Retract by "B"—or to starting point of hole and advance again to last pecking depth minus safety clearance.

5 Next pass (pecking depth: "last depth – IB" or JB).

6 Repeat 4 to 5 until end point Z2/X2 is reached.

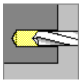
7 Last pass—depending on "V":
- Drill at programmed feed rate to
  End point Z2 (axial).
  End point X2 (radial).
- Remain at end of hole for dwell time "E," if defined.

or
- Drill at programmed feed rate to position
  Z2 – AB (axial).
  X2 – AB (radial).
- Drill at reduced feed rate to
  End point Z2 (axial).
  End point X2 (radial).

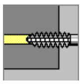
8 If X1/Z1 has been defined, retract to
- Starting point of hole Z1 (axial).
- Starting point of hole X1 (radial).

If X1/Z1 has **not** not been defined, retract to
- Starting point Z (axial).
- Starting point X (radial).

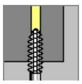
Tapping, axial/radial



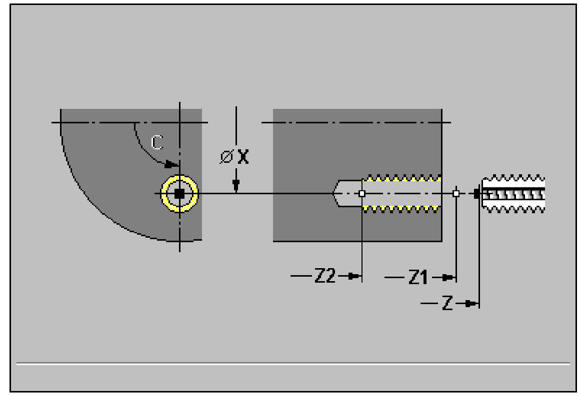
Call the drilling menu.

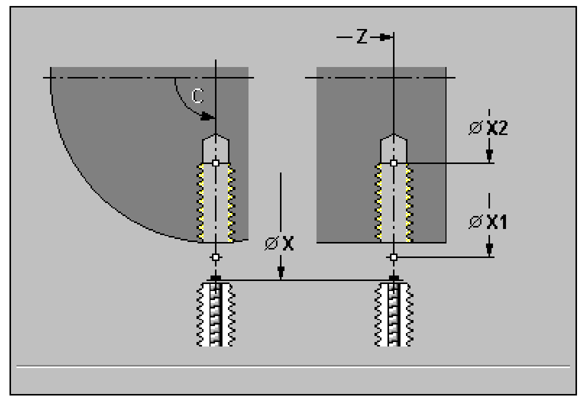


Select the "Tapping, axial" cycle.



Select the "Tapping, radial" cycle.





With this cycle, you can tap a thread on the face / lateral surface of a workpiece.

Meaning of "retraction length L": Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the "retract length." The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the drill is pulled away from the chuck by the "retraction length." With this method you can achieve higher service life from the taps.

Cycle parameters

- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **F1 thread pitch** (= feed rate)—default: Feed rate from tool definition
- **B run-in length** (default: 2 * thread pitch F1) to reach the programmed spindle speed and feed rate
- **SR return speed** (default: Same spindle speed as for tapping) for enabling the tap to retract rapidly
- **Retraction length L** (default: 0) when using floating tap holders
- **T tool number**
- **S spindle speed / cutting speed**

Drilling axial:

- **Z1 starting point of hole**—default: Drilling starts from position Z
- **Z2 end point of hole**

Drilling radial:

- **X1 starting point of hole**—default: Drilling starts from position X
- **X2 end point of hole**

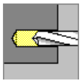
MANUALplus uses the tool parameter "driven tool" to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run

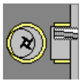
1. Position spindle to "spindle angle C" (in Manual mode, machining starts from the current spindle angle).
2. If defined, move at rapid traverse to
   - Z1 (axial).
   - X1 (radial).
3. Tap thread to
   - End point Z2 (axial).
   - End point X2 (radial).
4. If X1/Z1 has been defined, retract at return speed SR to
   - Starting point of hole Z1 (axial).
   - Starting point of hole X1 (radial).

   If X1/Z1 has **not** not been defined, retract to
   - Starting point Z (axial).
   - Starting point X (radial).

Thread milling, axial

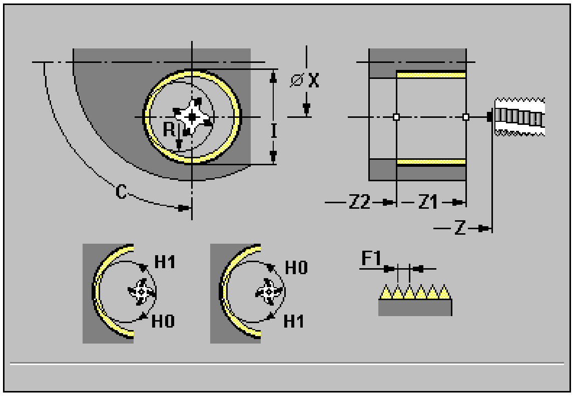


Call the drilling menu.



Select the "Thread milling, axial" cycle.

The cycle mills a thread in existing holes.



Use threading tools for this cycle.

Danger of collision!

Be sure to consider the hole diameter and the diameter of the milling cutter when programming "approaching radius R."

Cycle parameters

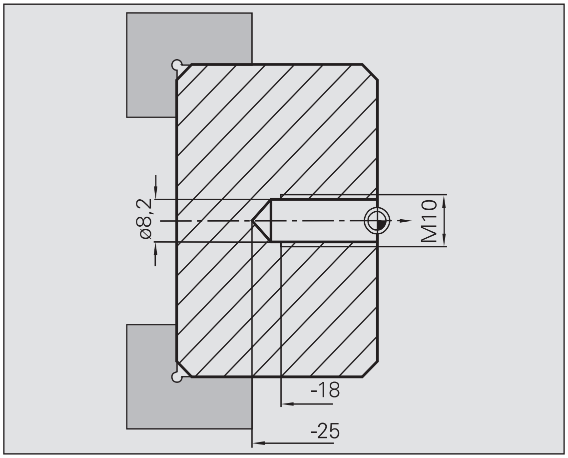
- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **Z1 starting point of thread**—default: Starting point Z
- **Z2 end point of thread**
- **I diameter of thread**
- **R approaching radius**—default: (I—milling diameter)/2
- **F1 thread pitch**
- **J direction of thread**—default: 0
  - J=0: Right-hand
  - J=1: Left-hand
- **H cutting direction**—default: 0
  - H=0: Up-cut milling
  - H=1: Climb milling
- **SR return speed** for enabling the tap to retract rapidly—default: Same spindle speed as for tapping
- **T tool number**
- **S spindle/cutting speed** for the driven tool

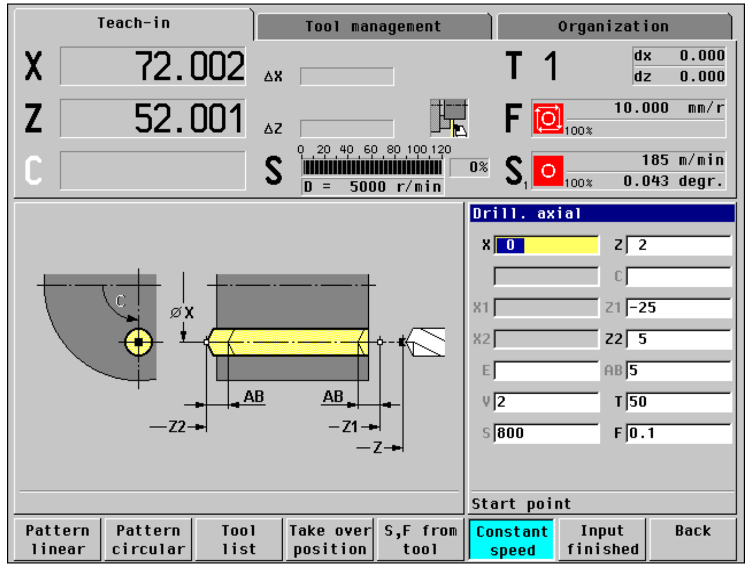
Cycle run

1. Position spindle to "spindle angle C" (in Manual mode, machining starts from the current spindle angle).
2. Position the tool to "milling floor Z2" inside the hole.
3. Approach on "approach arc R."
4. Mill the thread in a rotation of 360°, while advancing by "thread pitch F1."
5. Retract the tool and return it to the starting point.

Examples of drilling cycles

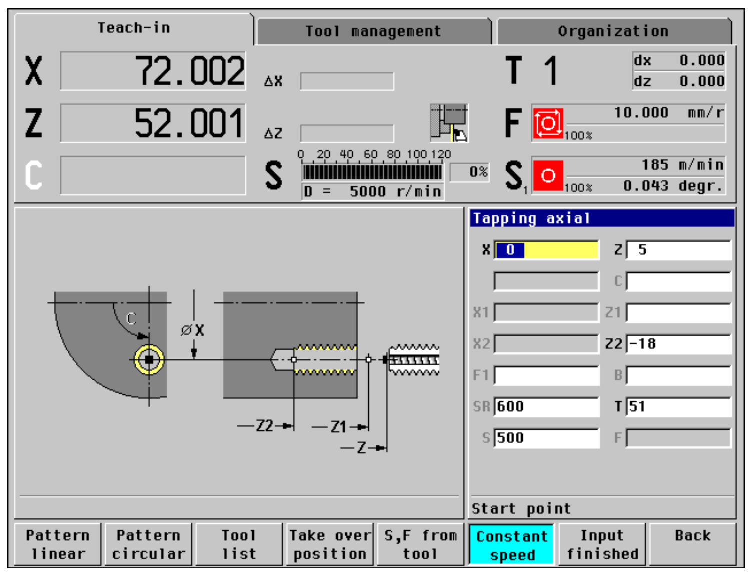
Centric drilling and tapping





The machining operation is to be performed in two steps. In the first step, the "Drilling, axial" cycle drills the hole. In the second, the "Tapping, axial" cycle taps the thread.

The drill is positioned at the safety clearance to the workpiece surface (starting point X, Z). The hole starting point Z1 is therefore not programmed. In the parameters "AB" and "V," you program a feed reduction (see figure at upper right).



The thread pitch is not programmed. MANUALplus uses the thread pitch of the tool. The "return speed SR" ensures that the tool is retracted quickly (see figure at lower right).

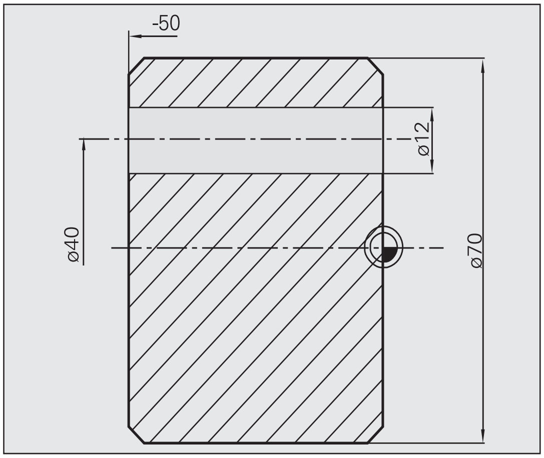
Tool data (drill)

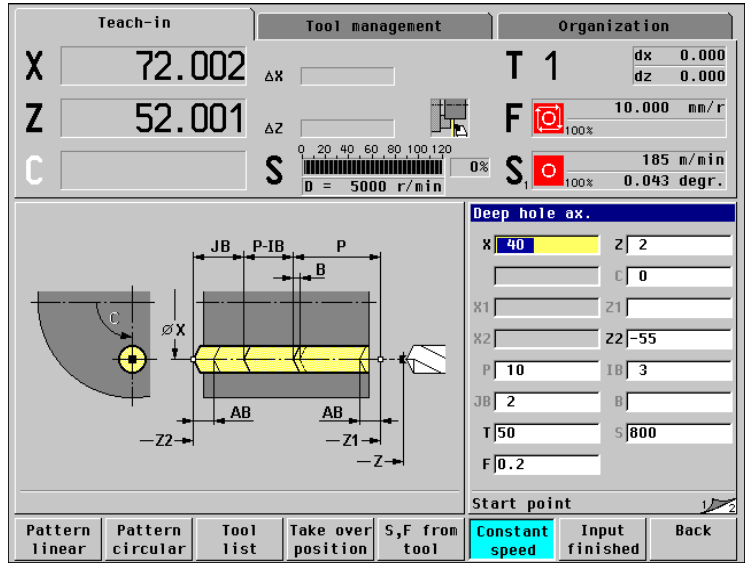
- WO = 8—Tool orientation
- I = 8.2—Drilling diameter
- B = 118—Point angle
- H = 0—The tool is not a driven tool

Tool data (tap)

- WO = 8—Tool orientation
- I = 10—Thread diameter M10
- F = 1.5—Thread pitch
- H = 0—The tool is not a driven tool

Deep-hole drilling





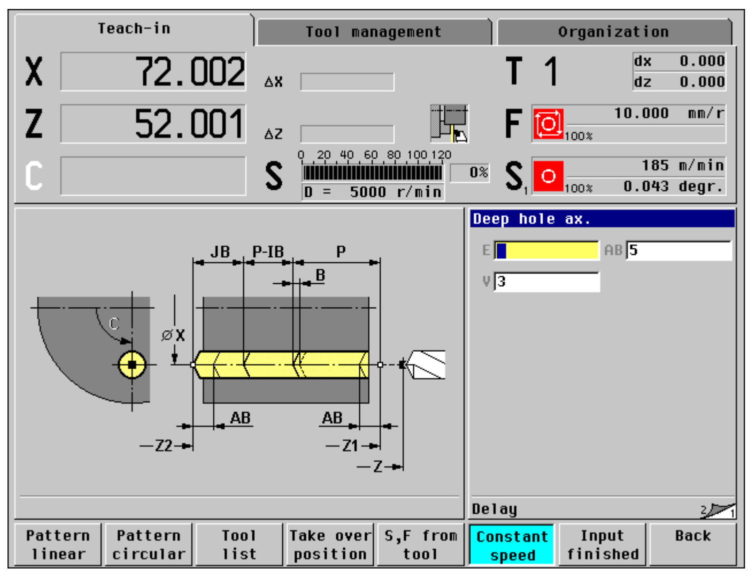
A hole is to be bored through the workpiece outside the turning center with the cycle "Deep-hole drilling, axial." This machining operation requires a traversable spindle and driven tools.

The parameters "1st hole depth P" and "hole depth reduction value IB" define the individual passes and the "minimum hole depth JB" limits the hole reduction value.

As the "return length B" is not defined, the drill therefore retracts to the starting point after each pass, remains there for the programmed dwell time, and then advances again to the safety clearance for the next pass.

Since this example is to illustrate how you drill a through hole, the hole end point Z2 is programmed such that the tool has to drill all the way through the workpiece before it reaches the end point.

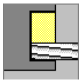
The parameters "AB" and "V" define a feed reduction for both pre-drilling and through-boring.



Tool data

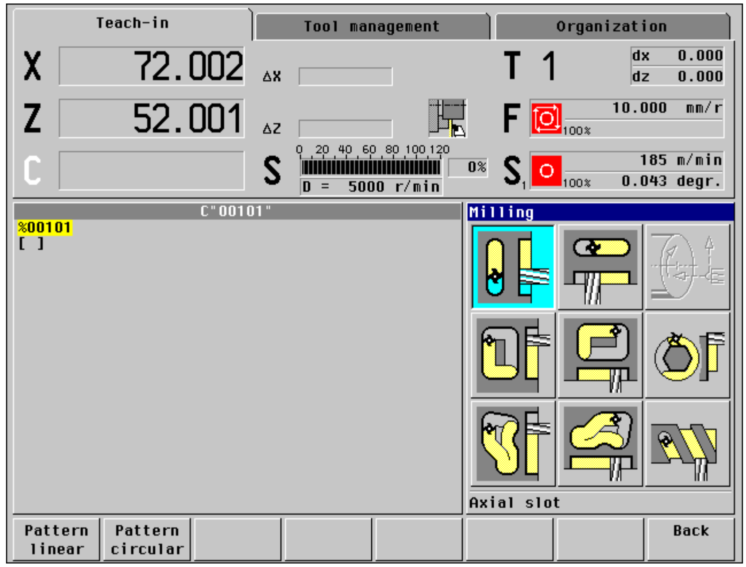
- WO = 8—Tool orientation
- I = 12—Drilling diameter
- B = 118—Point angle
- H = 1—The tool is a driven tool

4.8 Milling Cycles



Milling cycles for axial and radial slots, contours, pockets, surfaces and polygons.

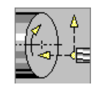
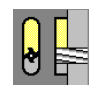
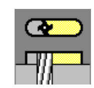
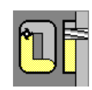
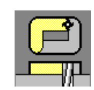
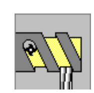
For pattern machining, see "Drilling/ Milling Patterns" on page 227.



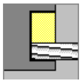
In Teach-in mode these cycles include the activation/ deactivation of the C axis and the positioning of the spindle.

In Manual mode you can activate the C axis with "Rapid traverse positioning" and position the spindle **before** the actual milling cycle. The milling cycles then automatically deactivate the C axis.

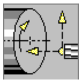
> The "constant cutting speed" may only be programmed with driven tools on machines with spindle control.

Milling cycles	Symbol
Rapid traverse positioning Activate C axis; position tool and spindle	
Slot axial/radial For milling single slots or slot patterns	 
Figure axial/radial For milling a single figure	 
Radial/axial ICP contour For milling single ICP contours or contour patterns	 
Face milling For milling surfaces or polygons	
Helical-slot milling, radial For milling a helical slot	

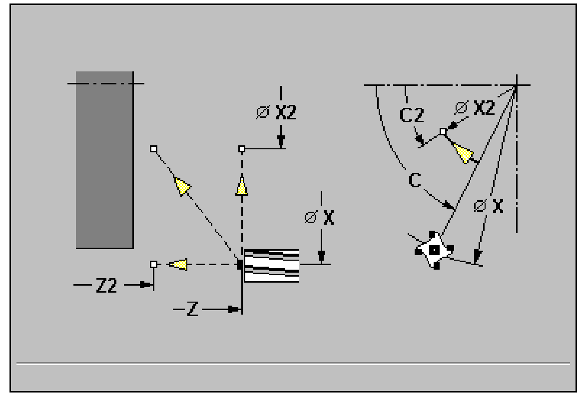
Rapid traverse positioning



Call the milling menu.



Select the "Rapid traverse positioning" cycle.



The cycle activates the C axis and positions the spindle (C axis) and the tool.

- "Rapid traverse positioning" is only required in Manual mode.
- The C axis is deactivated by a subsequent manual milling cycle.

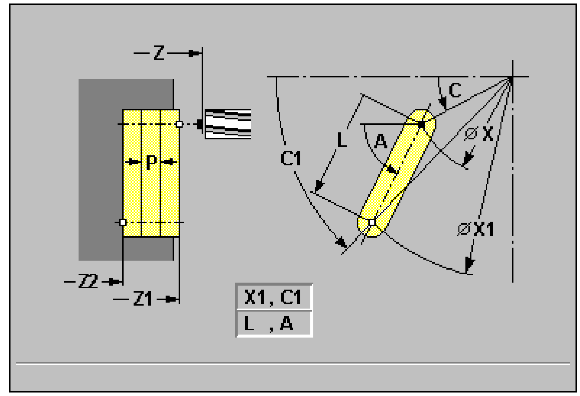
Cycle parameters

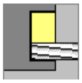
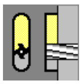
- **`X2, Z2 target point`**
- **`C2 end angle`** (C-axis position)—default: Current spindle angle

Cycle run

1. Activate C axis.
2. Position to end angle C2 at rapid traverse.
3. Position the tool to target point X2, Z2 at rapid traverse.

Slot, axial



	Call the milling menu.
	Select the "Slot, axial" cycle.

This cycle mills a slot on the face of the workpiece. The slot width equals the diameter of the milling cutter.

Cycle parameters

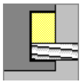
- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **C1 angle of slot target point**—default: Spindle angle C
- **X1 slot target point in X** (diameter value)
- **Z1 milling top edge**—default: Starting point Z
- **Z2 milling floor**
- **L slot length**
- **A angle to X axis**—default: 0
- **P infeed depth**—default: Total depth in one infeed
- **FZ infeed rate**—default: Active feed rate
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Parameter combinations for the position and orientation of the slot: See graphic support window

Cycle run

1. Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).
2. Calculate the proportioning of cuts.
3. Approach to safety clearance.
4. Approach at infeed rate FZ.
5. Machine to end point of slot.
6. Approach at infeed rate FZ.
7. Machine to starting point of slot.
8. Repeat 4 to 7 until the milling depth is reached.
9. Position to starting point Z and deactivate C axis.

Figure, axial



Call the milling menu.

Select the "Figure, axial" cycle.

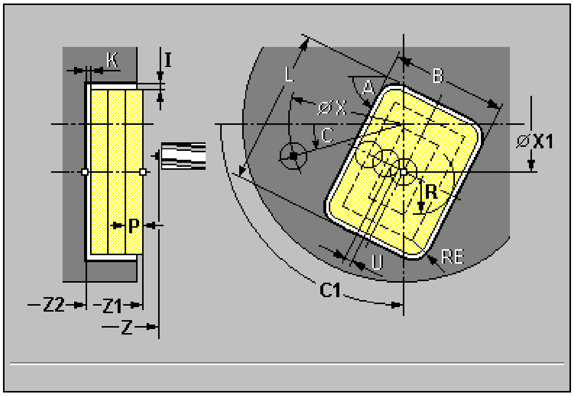
Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the face:

- Rectangle (Q=4, L<>B)
- Square (Q=4, L=B)
- Circle (Q=0, RE>0, L and B: No input)
- Triangle or polygon (Q=3 or Q>4, L>0)

Notes on parameters/functions:

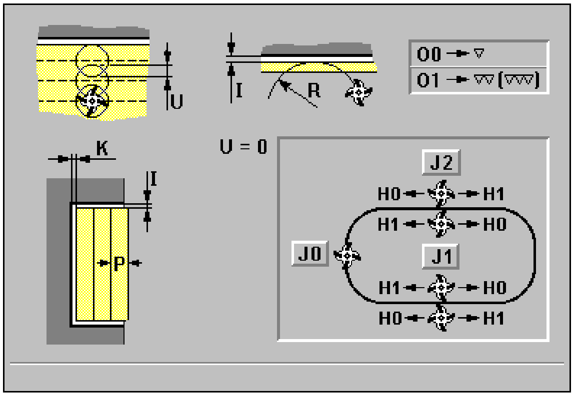
- **Machining of contour or pocket:** defined in "U."
- **Milling direction:** depends on definition in "H" and the direction of tool rotation (see "Cutting direction for contour milling and pocket milling" on page 224).
- **Milling cutter compensation:** effective (except for contour milling with J=0).
- **Approach and departure:** The point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element (for rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to "approaching radius R."
- **Contour milling:** "J" defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour.
- **Pocket milling—roughing (O=0):** "Contour milling J" defines whether a pocket is machined from the inside towards the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. "J" defines whether a pocket floor is finished from the inside towards the outside, or vice versa.

Cycle parameters (first input window)



- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **C1 angle of figure center**—default: Spindle angle C
- **X1 diameter of figure center**
- **Z1 milling top edge**—default: Starting point Z
- **Z2 milling floor**
- **L rectangle length**
  - Rectangle: Length of rectangle
  - Square, polygon: Edge length
  - Circle: No input
- **B rectangle width**
  - Rectangle: Width of rectangle
  - Square: L=B
  - Polygon, circle: No input
- **RE rounding radius**—default: 0
  - Rectangle, square, polygon: Rounding radius
  - Circle: Radius of circle
- **A angle to X axis**—default: 0
  - Rectangle, square, polygon: Position of figure
  - Circle: No input
- **Q number of edges**—default: 0
  - Q=0: Circle
  - Q=4: Rectangle, square
  - Q=3: Triangle
  - Q>4: Polygon
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Cycle parameters (second input window)



- **P infeed depth**—default: Total depth in one infeed
- **U overlap factor**
  - No input: Contour milling
  - U>0: Pocket milling—(minimum) overlap of milling paths = U*milling diameter
- **I contour-parallel oversize**
- **K oversize in infeed direction**
- **FZ infeed rate**—default: Active feed rate
- **E reduced feed rate** for circular elements—default: Active feed rate
- **H cutting direction**—default: 0
  - H=0: Up-cut milling
  - H=1: Climb milling
- **J contour milling** (default: 0) depending on "U," the following applies:
  - Pocket milling and J=0: On the contour
  - Pocket milling and J=1: Inside
  - Pocket milling and J=2: Outside
  - Contour milling and J=0: From the inside towards the outside
  - Contour milling and J=1: From the outside towards the inside
- **O roughing/finishing:** Milling sequence (only for pocket milling)—default: 0
  - O=0: Roughing
  - O=1: Finishing
- **R approaching radius:** Radius of approaching/departing arc—default: 0
  - R=0: Contour element is approached directly; feed to starting point above the milling plane—then vertical plunge.
  - R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially.

Cycle run

1 Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).

2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes).

Contour milling:

3 Depending on "R," approach the workpiece and plunge to the first milling plane.

4 Mill the first plane.

5 Plunge to the next milling plane.

6 Repeat 5 to 6 until the milling depth is reached.

7 Position to starting point Z and deactivate C axis.

Pocket milling—Roughing:

3 Move to the safety clearance and plunge to the first milling plane.

4 Depending on "J," machine the milling plane either from the inside towards the outside, or vice versa.

5 Plunge to the next milling plane.

6 Repeat 4 to 5 until the milling depth is reached.

7 Position to starting point Z and deactivate C axis.

Pocket milling—Finishing:

3 Depending on "R," approach the workpiece and plunge to the first milling plane.

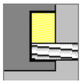
4 Finish-machine the edge of the pocket—one working plane after the other.

5 Depending on "J," finish-machine the milling floor either from the inside towards the outside, or vice versa.

6 Finish-machine the pocket at the programmed feed rate.

7 Position to starting point Z and deactivate C axis.

ICP contour, axial



Call the milling menu.

Select "ICP contour, axial."

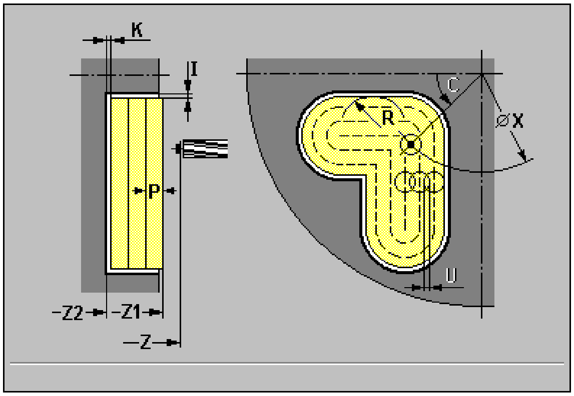
Depending on the parameters, the cycle mills a contour or roughs/ finishes a pocket on the face.

Notes on parameters/functions:

- **Machining of contour or pocket:** defined in "U."
- **Milling direction:** depends on definition in "H" and the direction of tool rotation (see “Cutting direction for contour milling and pocket milling” on page 224).
- **Milling cutter compensation:** effective (except for contour milling with J=0).
- **Approach and departure:** For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element (for rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to "approaching radius R."
- **Contour milling:** "J" defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour.
- **Pocket milling—roughing (O=0):** "Contour milling J" defines whether a pocket is machined from the inside towards the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. "J" defines whether a pocket floor is finished from the inside towards the outside, or vice versa.

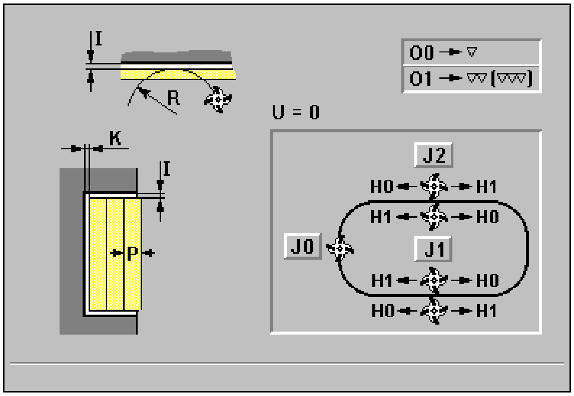
Cycle parameters (first input window)

- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **C1 angle of figure center**—default: Spindle angle C
- **Z1 milling top edge**—default: Starting point Z
- **Z2 milling floor**
- **P infeed depth**—default: Total depth in one infeed
- **U overlap factor**
  - No input: Contour milling
  - U>0: Pocket milling—(minimum) overlap of milling paths = U*milling diameter
- **I contour-parallel oversize**
- **K oversize in infeed direction**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**



Cycle parameters (second input window)

- **N ICP contour number**
- **FZ infeed rate**—default: Active feed rate
- **E reduced feed rate** for circular elements—default: Active feed rate
- **H cutting direction**—default: 0
  - H=0: Up-cut milling
  - H=1: Climb milling
- **J contour milling** (default: 0) depending on "U," the following applies:
  - Pocket milling and J=0: On the contour
  - Pocket milling and J=1: Inside
  - Pocket milling and J=2: Outside
  - Contour milling and J=0: From the inside towards the outside
  - Contour milling and J=1: From the outside towards the inside
- **O roughing/finishing:** Milling sequence (only for pocket milling)—default: 0
  - O=0: Roughing
  - O=1: Finishing
- **R approaching radius:** Radius of approaching/departing arc—default: 0
  - R=0: Contour element is approached directly; feed to starting point above the milling plane—then vertical plunge.
  - R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially.



Cycle run

1. Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes).

Contour milling:

3. Depending on "R," approach the workpiece and plunge to the first milling plane.
4. Mill the first plane.
5. Plunge to the next milling plane.
6. Repeat 5 to 6 until the milling depth is reached.
7. Position to starting point Z and deactivate C axis.

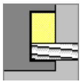
Pocket milling—Roughing:

3. Move to the safety clearance and plunge to the first milling plane.
4. Depending on "J," machine the milling plane either from the inside towards the outside, or vice versa.
5. Plunge to the next milling plane.
6. Repeat 4 to 5 until the milling depth is reached.
7. Position to starting point Z and deactivate C axis.

Pocket milling—Finishing:

3. Depending on "R," approach the workpiece and plunge to the first milling plane.
4. Finish-machine the edge of the pocket—one working plane after the other.
5. Depending on "J," finish-machine the milling floor either from the inside towards the outside, or vice versa.
6. Finish-machine the pocket at the programmed feed rate.
7. Position to starting point Z and deactivate C axis.

Face milling



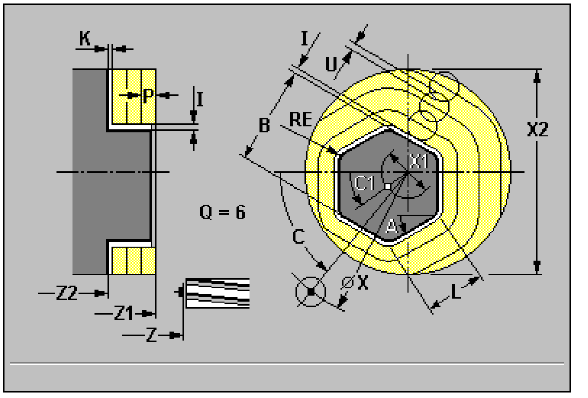
Call the milling menu.

Select the "Face milling" cycle.

Depending on the parameters, the cycle mills the following contours on the face.

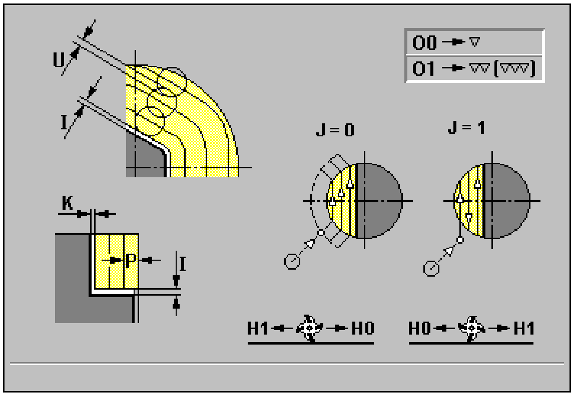
- One or two surfaces (Q=1 or Q=2, B>0)
- Rectangle (Q=4, L<>B)
- Square (Q=4, L=B)
- Triangle or polygon (Q=3 or Q>4, L>0)
- Circle (Q=0, RE>0, L and B: No input)

Cycle parameters (first input window)



- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **C1 angle of figure center**—default: Spindle angle C
- **X1 diameter of figure center**
- **Z1 milling top edge**—default: Starting point Z
- **X2 limiting diameter**
- **Z2 milling floor**
- **L edge length**
  - Rectangle: Length of rectangle
  - Square, polygon: Edge length
  - Circle: No input
- **B width across flats:**
  - For Q=1, Q=2: Remaining thickness (remaining material)
  - Rectangle: Width of rectangle
  - Square, polygon (Q>=4): Width across flats (use only for even number of surfaces; program "B" as an alternative to "L")
  - Circle: No input
- **RE rounding radius**—default: 0
  - Polygon (Q>2): Rounding radius
  - Circle (Q=0): Radius of circle
- **A angle to X axis**—default: 0
  - Polygon (Q>2): Position of figure
  - Circle: No input
- **Q number of edges**—default: 0
  - Q=0: Circle
  - Q=1: One surface
  - Q=2: Two surfaces offset by 180°
  - Q=3: Triangle
  - Q=4: Rectangle, square
  - Q>4: Polygon
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Cycle parameters (second input window)



- **P infeed depth**—default: Total depth in one infeed
- **U overlap factor**
  - No input: Contour milling
  - U>0: Pocket milling—(minimum) overlap of milling paths = U*milling diameter
- **I contour-parallel oversize**
- **K oversize in infeed direction**
- **FZ infeed rate**—default: Active feed rate
- **E reduced feed rate** for circular elements—default: Active feed rate
- **H cutting direction**—default: 0
  - H=0: Up-cut milling
  - H=1: Climb milling
- **J milling direction:** For surfaces or polygons (with "RE = 0"), "J" defines whether a unidirectional or bidirectional milling operation is to be executed.
  - J=0: Unidirectional
  - J=1: Bidirectional
- **O roughing/finishing:** Milling sequence (only for pocket milling)—default: 0
  - O=0: Roughing
  - O=1: Finishing

Cycle run

1. Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes).
3. Move to the safety clearance and plunge to the first milling plane.

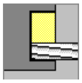
Roughing

4. Machine the milling plane, taking "J" (unidirectional or bidirectional) into account.
5. Plunge to the next milling plane.
6. Repeat 4 to 5 until the milling depth is reached.
7. Position to starting point Z and deactivate C axis.

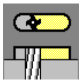
Finishing:

4. Finish-machine the edge of the island—one working plane after the other.
5. Finish-machine the floor from the outside towards the inside.
6. Position to starting point Z and deactivate C axis.

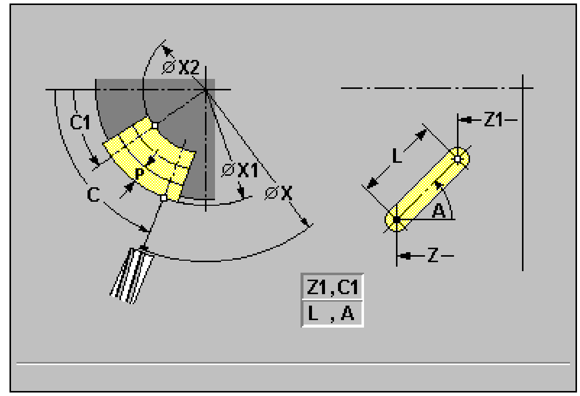
Slot, radial



Call the milling menu.



Select the "Slot, radial" cycle.



This cycle mills a slot on the lateral surface. The slot width equals the diameter of the milling cutter.

Cycle parameters

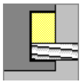
- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **C1 angle of slot target point**—default: Spindle angle C
- **X1 milling top edge** (diameter)—default: Starting point X
- **Z1 target point of slot**
- **X2 milling floor**
- **L slot length**
- **A angle to Z axis**—default: 0
- **P infeed depth**—default: Total depth in one infeed
- **FZ infeed rate**—default: Active feed rate
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

Parameter combinations for the position and orientation of the slot: See graphic support window

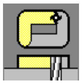
Cycle run

1. Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).
2. Move at rapid traverse to slot starting point X, Z if defined.
3. Approach at infeed rate FZ.
4. Mill to slot end point at programmed feed rate.
5. Retract to starting point X.

Figure, radial



Call the milling menu.



Select the "Figure, radial" cycle.

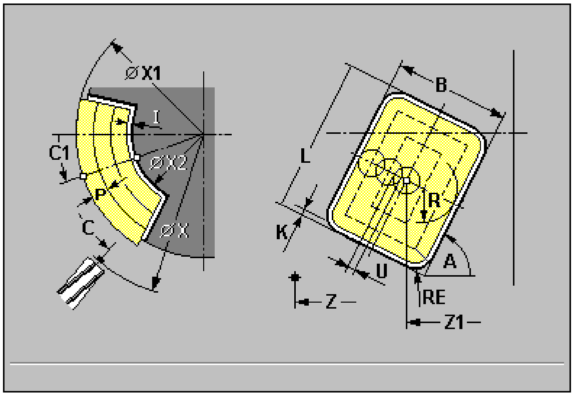
Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the lateral surface:

- Rectangle (Q=4, L<>B)
- Square (Q=4, L=B)
- Circle (Q=0, RE>0, L and B: No input)
- Triangle or polygon (Q=3 or Q>4, L>0)

Notes on parameters/functions:

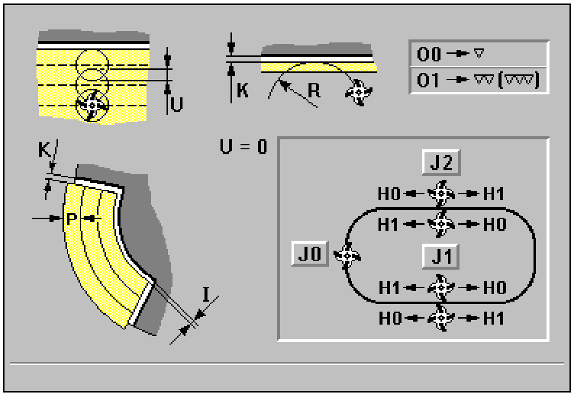
- **Machining of contour or pocket:** defined in "U."
- **Milling direction:** depends on definition in "H" and the direction of tool rotation (see “Cutting direction for contour milling and pocket milling” on page 224).
- **Milling cutter compensation:** effective (except for contour milling with J=0).
- **Approach and departure:** The point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element (for rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to "approaching radius R."
- **Contour milling:** "J" defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour.
- **Pocket milling—roughing (O=0):** "Contour milling J" defines whether a pocket is machined from the inside towards the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. "J" defines whether a pocket floor is finished from the inside towards the outside, or vice versa.

Cycle parameters (first input window)



- **`X, Z starting point`**
- **`C spindle angle`** (C-axis position)—default: Current spindle angle
- **`C1 angle of figure center`**—default: Spindle angle C
- **`X1 milling top edge`**—default: Starting point X
- **`Z1 figure center`**
- **`X2 milling floor`**
- **`L rectangle length`**
  - Rectangle: Length of rectangle
  - Square, polygon: Edge length
  - Circle: No input
- **`B rectangle width`**
  - Rectangle: Width of rectangle
  - Square: L=B
  - Polygon, circle: No input
- **`RE rounding radius`**—default: 0
  - Rectangle, square, polygon: Rounding radius
  - Circle: Radius of circle
- **`A angle to Z axis`**—default: 0
  - Rectangle, square, polygon: Position of figure
  - Circle: No input
- **`Q number of edges`**—default: 0
  - Q=0: Circle
  - Q=4: Rectangle, square
  - Q=3: Triangle
  - Q>4: Polygon
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

Cycle parameters (second input window)



- **P infeed depth**—default: Total depth in one infeed
- **U overlap factor**
  - No input: Contour milling
  - U>0: Pocket milling—(minimum) overlap of milling paths = U*milling diameter
- **I oversize in infeed direction**
- **K contour-parallel oversize**
- **FZ infeed rate**—default: Active feed rate
- **E reduced feed rate** for circular elements—default: Active feed rate
- **H cutting direction**—default: 0
  - H=0: Up-cut milling
  - H=1: Climb milling
- **J contour milling** (default: 0) depending on "U," the following applies:
  - Pocket milling and J=0: On the contour
  - Pocket milling and J=1: Inside
  - Pocket milling and J=2: Outside
  - Contour milling and J=0: From the inside towards the outside
  - Contour milling and J=1: From the outside towards the inside
- **O roughing/finishing:** Milling sequence (only for pocket milling)—default: 0
  - O=0: Roughing
  - O=1: Finishing
- **R approaching radius:** Radius of approaching/departing arc—default: 0
  - R=0: Contour element is approached directly; feed to starting point above the milling plane—then vertical plunge.
  - R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially.

Cycle run

1 Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).
2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes).

Contour milling:

3 Depending on "Approaching radius R," approach the workpiece and plunge to the first milling plane.
4 Mill the first plane.
5 Plunge to the next milling plane.
6 Repeat 5 to 6 until the milling depth is reached.
7 Position to starting point Z and deactivate C axis.

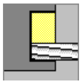
Pocket milling—Roughing:

3 Move to the safety clearance and plunge to the first milling plane.
4 Depending on "J," machine the milling plane either from the inside towards the outside, or vice versa.
5 Plunge to the next milling plane.
6 Repeat 4 to 5 until the milling depth is reached.
7 Position to starting point Z and deactivate C axis.

Pocket milling—Finishing:

3 Depending on "Approaching radius R," approach the workpiece and plunge to the first milling plane.
4 Finish-machine the edge of the pocket—one working plane after the other.
5 Depending on "J," finish-machine the milling floor either from the inside towards the outside, or vice versa.
6 Finish-machine the pocket at the programmed feed rate.
7 Position to starting point Z and deactivate C axis.

ICP contour, radial



Call the milling menu.

Select "ICP contour, radial."

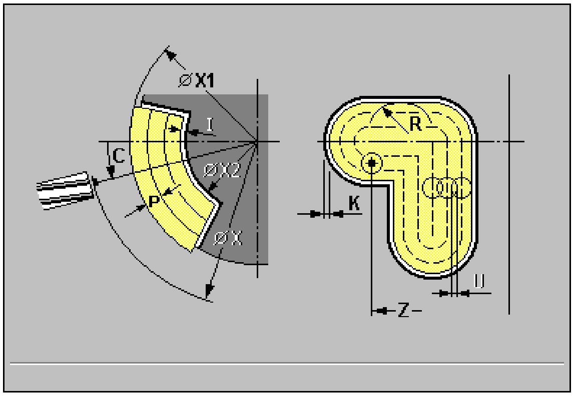
Depending on the parameters, the cycle mills a contour or roughs/finishes a pocket on the lateral surface.

Notes on parameters/functions:

- **Machining of contour or pocket:** defined in "U."
- **Milling direction:** depends on definition in "H" and the direction of tool rotation (see “Cutting direction for contour milling and pocket milling” on page 224).
- **Milling cutter compensation:** effective (except for contour milling with J=0).
- **Approach and departure:** For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element (for rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to "approaching radius R."
- **Contour milling:** "J" defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour.
- **Pocket milling—roughing (O=0):** "Contour milling J" defines whether a pocket is machined from the inside towards the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. "J" defines whether a pocket floor is finished from the inside towards the outside, or vice versa.

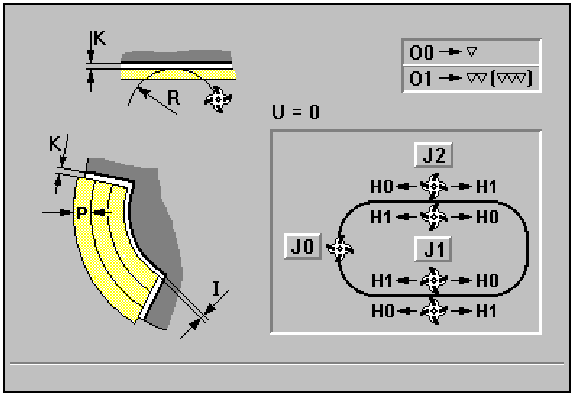
Cycle parameters (first input window)

- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **X1 milling top edge**—default: Starting point X
- **X2 milling floor**
- **P infeed depth**—default: Total depth in one infeed
- **U overlap factor**
  - No input: Contour milling
  - U>0: Pocket milling—(minimum) overlap of milling paths = U*milling diameter
- **I oversize in infeed direction**
- **K contour-parallel oversize**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**



Cycle parameters (second input window)

- **N ICP contour number**
- **FZ infeed rate**—default: Active feed rate
- **E reduced feed rate** for circular elements—default: Active feed rate
- **H cutting direction**—default: 0
  - H=0: Up-cut milling
  - H=1: Climb milling
- **J contour milling** (default: 0) depending on "U," the following applies:
  - Pocket milling and J=0: On the contour
  - Pocket milling and J=1: Inside
  - Pocket milling and J=2: Outside
  - Contour milling and J=0: From the inside towards the outside
  - Contour milling and J=1: From the outside towards the inside
- **O roughing/finishing:** Milling sequence (only for pocket milling)—default: 0
  - O=0: Roughing
  - O=1: Finishing
- **R approaching radius:** Radius of approaching/departing arc—default: 0
  - R=0: Contour element is approached directly; feed to starting point above the milling plane—then vertical plunge.
  - R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially.



Cycle run

1 Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).

2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes).

Contour milling:

3 Depending on "R," approach the workpiece and plunge to the first milling plane.

4 Mill the first plane.

5 Plunge to the next milling plane.

6 Repeat 5 to 6 until the milling depth is reached.

7 Position to starting point Z and deactivate C axis.

Pocket milling—Roughing:

3 Move to the safety clearance and plunge to the first milling plane.

4 Depending on "J," machine the milling plane either from the inside towards the outside, or vice versa.

5 Plunge to the next milling plane.

6 Repeat 4 to 5 until the milling depth is reached.

7 Position to starting point Z and deactivate C axis.

Pocket milling—Finishing:

3 Depending on "R," approach the workpiece and plunge to the first milling plane.

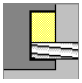
4 Finish-machine the edge of the pocket—one working plane after the other.

5 Depending on "J," finish-machine the milling floor either from the inside towards the outside, or vice versa.

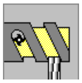
6 Finish-machine the pocket at the programmed feed rate.

7 Position to starting point Z and deactivate C axis.

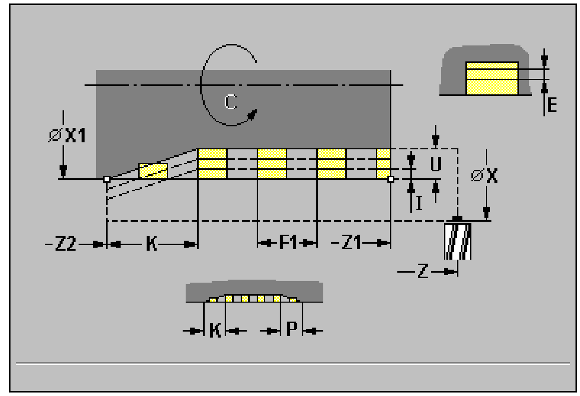
Helical-slot milling, radial



Call the milling menu.



Select the "Helical-slot milling, radial" cycle.



The cycle mills a helical slot from Z1 to Z2. Starting angle C1 defines the starting position for the slot. The slot width equals the diameter of the milling cutter.

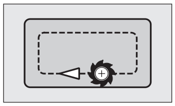
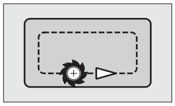
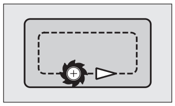
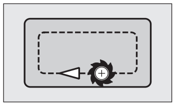
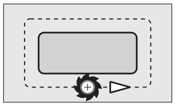
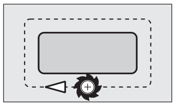
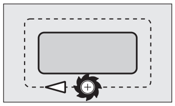
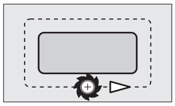
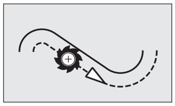
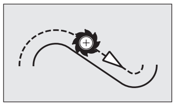
Cycle parameters

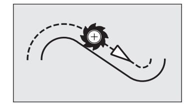
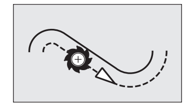
- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **C1 starting angle**
- **X1 diameter of thread**
- **Z1 starting point of thread**
- **F1 thread pitch**
  - F1 positive: Right-hand thread
  - F1 negative: Left-hand thread
- **Z2 end point of thread**
- **P run-in length:** Ramp at the beginning of the slot
- **K run-out length:** Ramp at the end of the slot
- **U thread depth**
- **I maximum infeed:** The infeed movements are reduced down to >= 0.5 mm according to the following calculation: Following that, each infeed movement will amount to 0.5 mm.
  - Infeed 1: "I"
  - Infeed n: I * (1 – (n–1) * E)
- **E cutting depth reduction**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

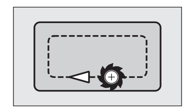
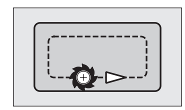
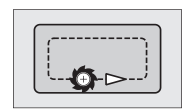
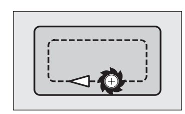
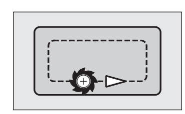
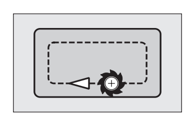
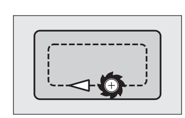
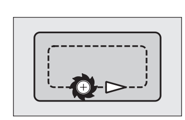
Cycle run

1. Activate the C axis and position to spindle angle C at rapid traverse (only in Teach-in mode).
2. Calculate current infeed.
3. Position the tool for the first pass.
4. Machine up to end point Z2 at the programmed feed rate, taking the ramps at the beginning and end of the slot into account.
5. Return on paraxial path and approach for next pass.
6. Repeat 4 to 5 until the slot depth is reached.

Cutting direction for contour milling and pocket milling

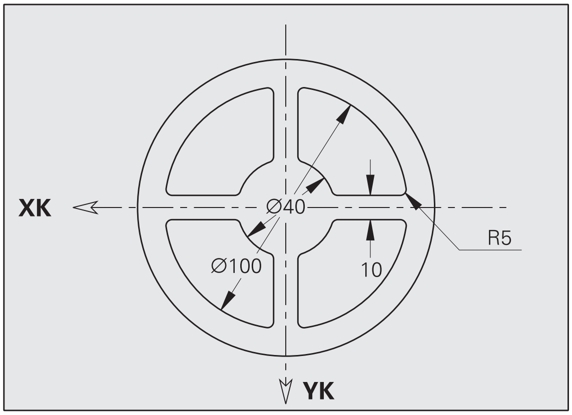
Milling direction for contour milling				
Cycle type	**Cutting direction**	**Direction of tool rotation**	**TRC**	**Execution**
Inside (J=1)	Up-cut milling (H=0)	Mx03	Right	
Inside	Up-cut milling (H=0)	Mx04	Left	
Inside	Climb milling (H=1)	Mx03	Left	
Inside	Climb milling (H=1)	Mx04	Right	
Outside (J=2)	Up-cut milling (H=0)	Mx03	Right	
Outside	Up-cut milling (H=0)	Mx04	Left	
Outside	Climb milling (H=1)	Mx03	Left	
Outside	Climb milling (H=1)	Mx04	Right	
Right (J=3)	Up-cut milling (H=0)	Mx03	Right	
Left (J=3)	Up-cut milling (H=0)	Mx04	Left	

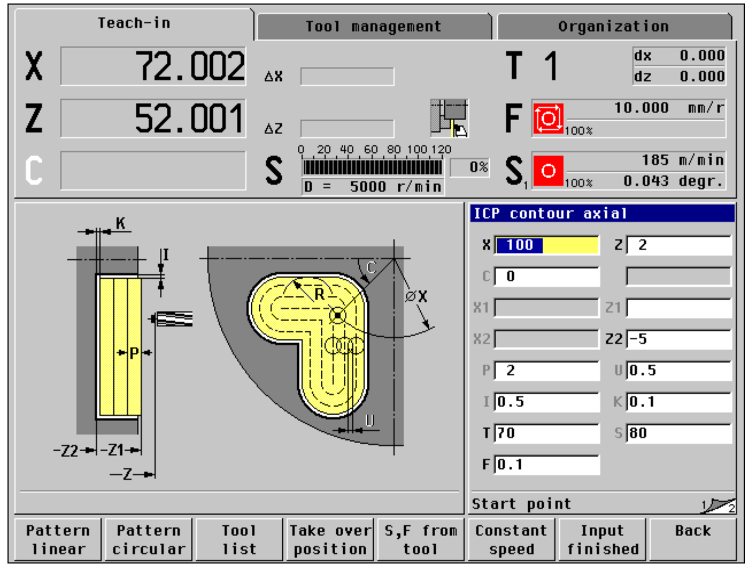
Milling direction for contour milling				
Cycle type	**Cutting direction**	**Direction of tool rotation**	**TRC**	**Execution**
Left (J=3)	Climb milling (H=1)	Mx03	Left	
Right (J=3)	Climb milling (H=1)	Mx04	Right	

Milling direction for pocket milling				
Machining operation	**Cutting direction**	**Machining direction**	**Direction of tool rotation**	**Execution**
Roughing	Up-cut milling (H=0)	From inside towards outside (J=0)	Mx03	
Finishing	Up-cut milling (H=0)	—	Mx03	
Roughing	Up-cut milling (H=0)	From inside towards outside (J=0)	Mx04	
Finishing	Up-cut milling (H=0)	—	Mx04	
Roughing	Climb milling (H=0)	From outside towards inside (J=1)	Mx03	
Roughing	Up-cut milling (H=0)	From outside towards inside (J=1)	Mx04	
Roughing	Climb milling (H=1)	From inside towards outside (J=0)	Mx03	
Finishing	Climb milling (H=1)	—	Mx03	
Roughing	Climb milling (H=1)	From inside towards outside (J=0)	Mx04	
Finishing	Climb milling (H=1)	—	Mx04	
Roughing	Climb milling (H=1)	From outside towards inside (J=1)	Mx03	
Roughing	Up-cut milling (H=1)	From outside towards inside (J=1)	Mx04	

Examples of milling cycles

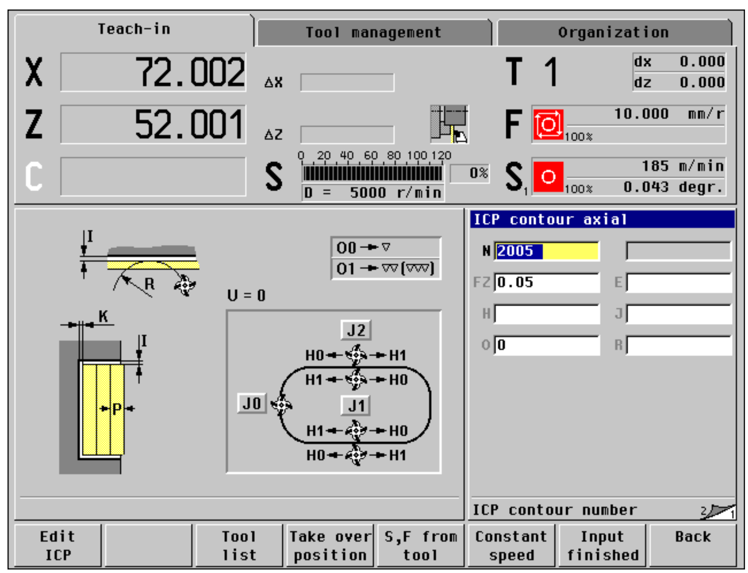
Milling on the face





In this example, a pocket is milled. The milling example in "9.8 ICP Example, Milling Cycle" illustrates the complete machining process on the face, including contour definition.

The machining process is performed with the cycle "ICP contour, axial." To describe a contour, define the basic contour first. Then superimpose the rounding arcs.



Tool data (milling cutter)

- WO = 8—Tool orientation
- I = 8—Milling diameter
- K = 4—Number of teeth
- TF = 0.025—Feed per tooth

4.9 Drilling/Milling Patterns

Note on using drilling/milling patterns:

- **Drilling patterns:** MANUALplus generates the machine commands M12, M13 (apply/release block brake) under the following conditions: the drill/tap must be entered as driven tool (parameter "Driven tool H") and the "direction of rotation MD" must be defined.
- **ICP milling contours:** If the contour starting point is outside the coordinate datum, the distance between contour starting point and coordinate datum is added to the pattern position (see “ICP Example "Milling Cycle"” on page 507).

Drilling/milling pattern linear, axial

Drilling pattern linear, axial

Call the drilling menu.

Select "Drilling axial " (see figure at upper right).

Select "Deep-hole drilling, axial " (see figure at center right).

Select "Tapping, axial " (see figure at lower right).

Pattern linear

Press the ***Pattern linear*** soft key.

Milling pattern linear, axial

Call the milling menu.

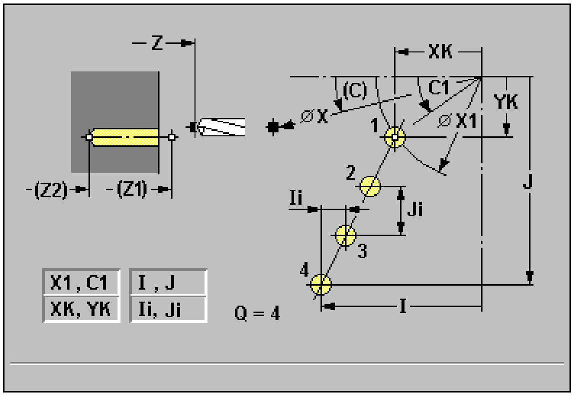
Select "Slot axial " (see figure at top of next page).

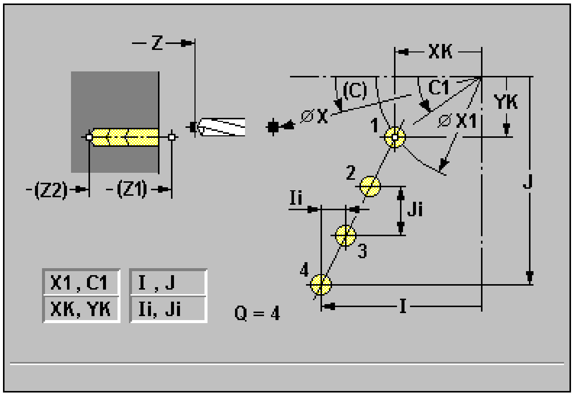
Select "ICP contour, axial" (see middle figure on next page).

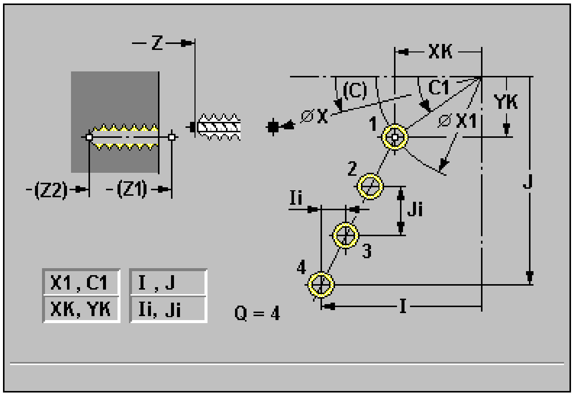
Pattern linear

Press the ***Pattern linear*** soft key.

Press "Pattern linear" to machine hole patterns or figure patterns in which the individual features are arranged at a regular spacing in a straight line on the face.







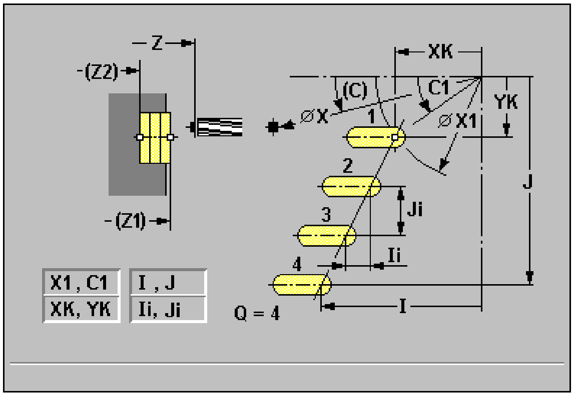
Cycle parameters

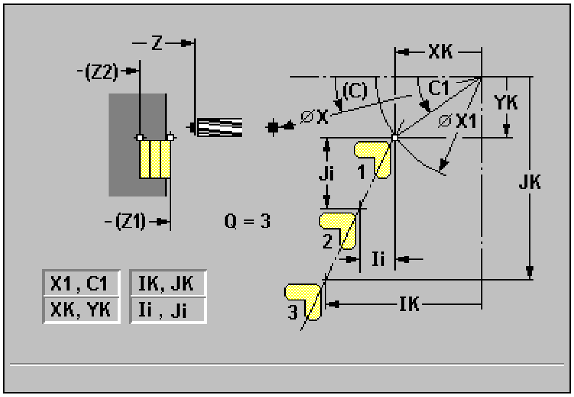
- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **X1, C1 starting point of pattern:** Position, starting angle (polar coordinates)
- **XK, YK starting point of pattern:** (Cartesian coordinates)
- **I, J end point of pattern** (Cartesian coordinates)
- **Ii, Ji pattern spacing (incremental)**
- **Q number of holes/slots**—default: 1
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

MANUALplus also interrogates the parameters that are required for machining the respective elements.

Use the following parameter combinations to define the:

- Starting point of pattern
  - X1, C1 or
  - XK, YK
- Pattern positions:
  - Ii, Ji and Q
  - I, J and Q



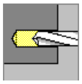


Cycle run

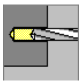
1. Positioning (depending on the machine configuration):
   - Without C axis: Position to "spindle angle C."
   - With C axis: Activate C axis and position to "spindle angle C" at rapid traverse.
   - Manual mode: Machining starts from current spindle angle.
2. Calculate the pattern positions.
3. Position to starting point of pattern.
4. Execute drilling/milling operation.
5. Position for next machining operation.
6. Repeat steps 4 and 5 until all machining operations have been completed.
7. Return to starting point X, Z.

Drilling/milling pattern circular, axial

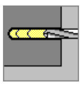
Drilling pattern circular, axial



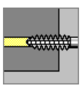
Call the drilling menu.



Select "Drilling axial " (see figure at upper right).



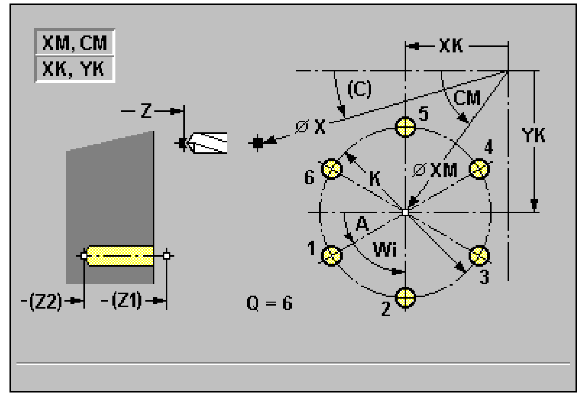
Select "Deep-hole drilling, axial " (see figure at center right).

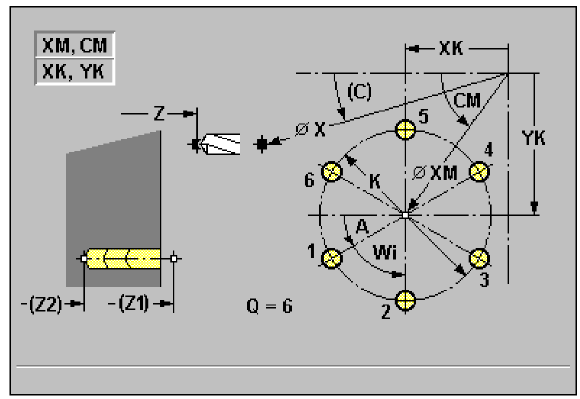


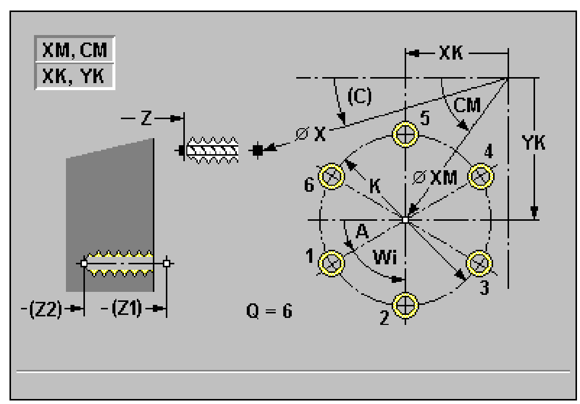
Select "Tapping, axial " (see figure at lower right).

Pattern circular

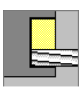
Press the ***Pattern circular*** soft key.



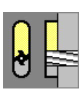




Milling pattern circular, axial



Call the milling menu.



Select "Slot axial " (see figure at top of next page).

Select "ICP contour, axial" (see middle figure on next page).

Pattern circular

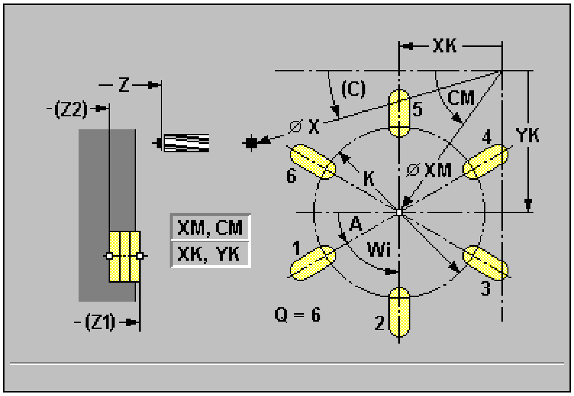
Press the ***Pattern circular*** soft key.

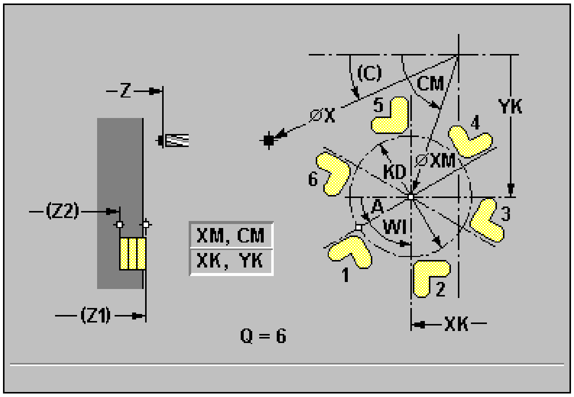
Press "Pattern circular" to machine hole patterns or figure patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the face.

Cycle parameters

- **`X, Z starting point`**
- **`C spindle angle`** (C-axis position)—default: Current spindle angle
- **`XM, CM pattern center:`** Position, angle (polar coordinates)
- **`XK, YK pattern center`** (Cartesian coordinates)
- **`K/KD pattern diameter`**—default: Starting point X is the pattern diameter
- **`A angle of 1st hole/slot`**—default: 0°
- **`Wi angle increment`** (pattern spacing)—default: Holes, slots, etc., arranged at a regular spacing in a circle
- **`Q number of holes/slots`**—default: 1
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

MANUALplus also interrogates the parameters that are required for machining the respective elements.

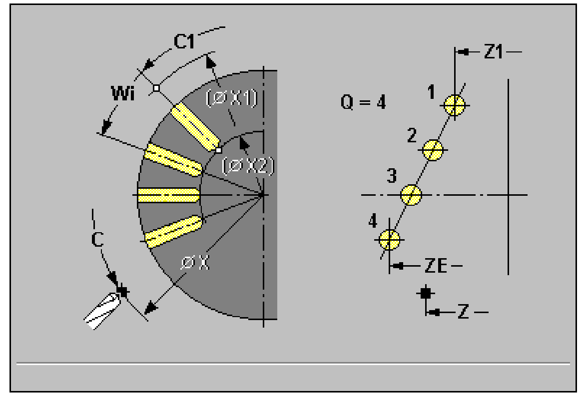


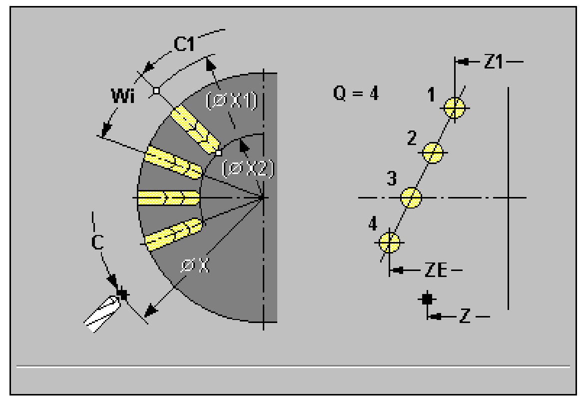


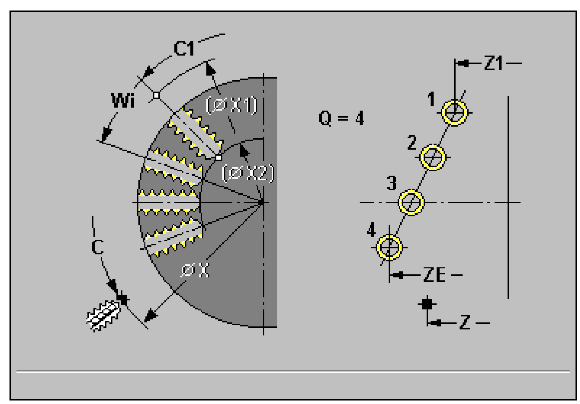
Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to "spindle angle C."
   - With C axis: Activate C axis and position to "spindle angle C" at rapid traverse.
   - Manual mode: Machining starts from current spindle angle.
2. Calculate the pattern positions.
3. Position to starting point of pattern.
4. Execute drilling/milling operation.
5. Position for next machining operation.
6. Repeat steps 4 and 5 until all machining operations have been completed.
7. Return to starting point X, Z.

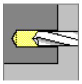
Drilling/milling pattern linear, radial



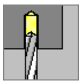




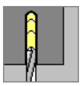
Drilling pattern linear, radial



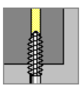
Call the drilling menu.



Select "Drilling, radial " (see figure at upper right).



Select "Deep-hole drilling, radial " (see figure at center right).

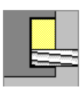


Select "Tapping, radial " (see figure at lower right).

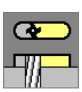
Pattern linear

Press the ***Pattern linear*** soft key.

Milling pattern linear, radial



Call the milling menu.



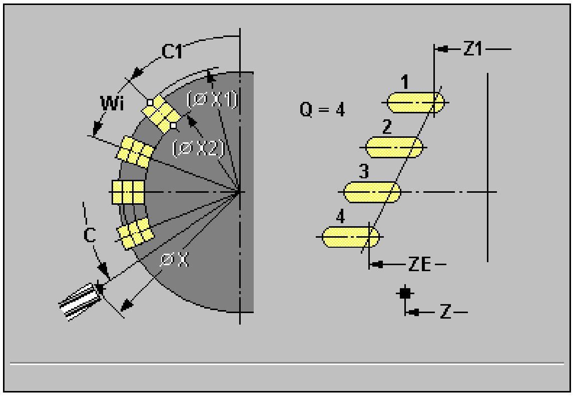
Select "Slot radial" (see figure at top of next page).

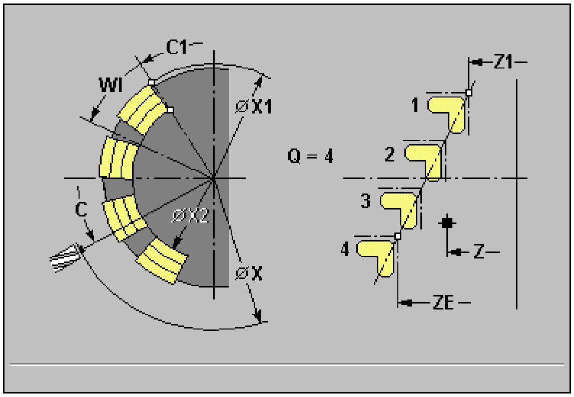
Select "ICP contour, radial" (see middle figure on next page).

Pattern linear

Press the ***Pattern linear*** soft key.

Press "Pattern linear" to machine hole patterns or figure patterns in which the individual features are arranged at a regular spacing in a straight line on the lateral surface.





Cycle parameters

- **`X, Z starting point`**
- **`C spindle angle`** (C-axis position)—default: Current spindle angle
- **`Z1 starting point of pattern:`** Position of 1st hole/slot (polar coordinates)
- **`C1 angle of 1st hole/slot:`** Starting angle (polar coordinates)
- **`ZE end point of pattern`**—default: Z1
- **`Wi angle increment (pattern spacing)`**—default: Holes, slots, etc. are arranged at a regular spacing on the circumference.
- **`Q number of holes/slots`**—default: 1
- **`T tool number`**
- **`S spindle speed / cutting speed`**
- **`F feed per revolution`**

The pattern positions are defined with "ZE – Wi" or "Wi – Q."

MANUALplus also interrogates the parameters that are required for machining the respective elements.

Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to "spindle angle C."
   - With C axis: Activate C axis and position to "spindle angle C" at rapid traverse.
   - Manual mode: Machining starts from current spindle angle.
2. Calculate the pattern positions.
3. Position to starting point of pattern.
4. Execute drilling/milling operation.
5. Position for next machining operation.
6. Repeat steps 4 and 5 until all machining operations have been completed.
7. Position to starting point Z and deactivate C axis.

Drilling/milling pattern circular, radial

Drilling pattern circular, radial

Call the drilling menu.

Select "Drilling, radial " (see figure at upper right).

Select "Deep-hole drilling, radial " (see figure at center right).

Select "Tapping, radial " (see figure at lower right).

Pattern circular

Press the ***Pattern circular*** soft key.

Milling pattern circular, radial

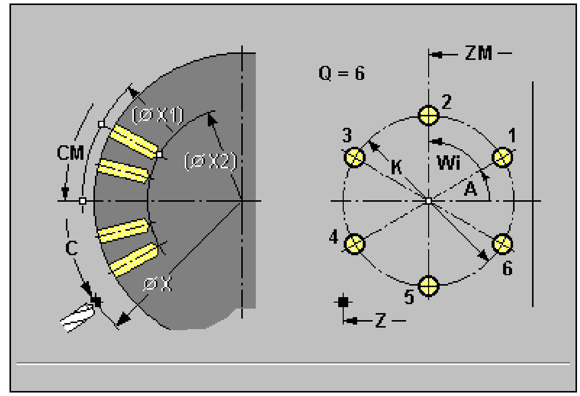
Call the milling menu.

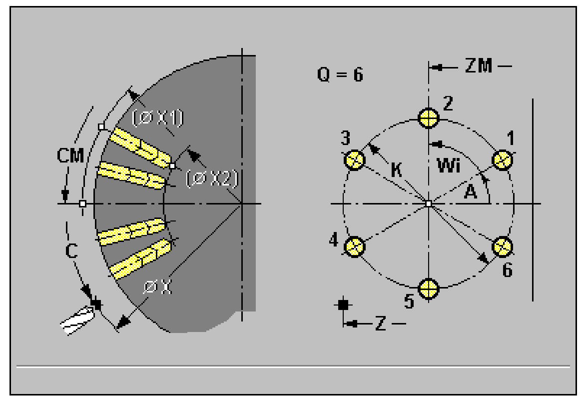
Select "Slot radial" (see figure at top of next page).

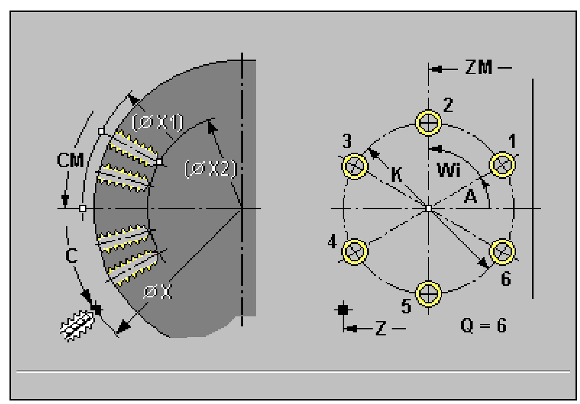
Select "ICP contour, radial" (see middle figure on next page).

Pattern circular

Press the ***Pattern circular*** soft key.





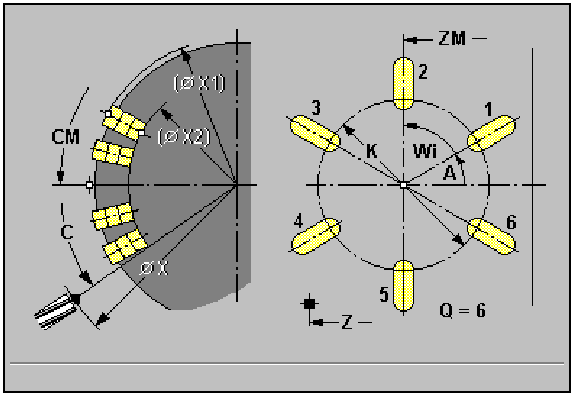


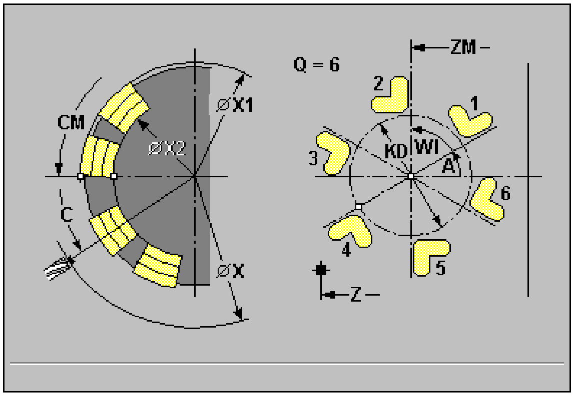
Press "Pattern circular" to machine hole patterns or figure patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the lateral surface.

Cycle parameters

- **X, Z starting point**
- **C spindle angle** (C-axis position)—default: Current spindle angle
- **ZM, CM pattern center:** Position, angle (polar coordinates)
- **K/KD pattern diameter**—default: Starting point X is the pattern diameter
- **A angle of 1st hole/slot**—default: 0°
- **Wi angle increment** (pattern spacing)—default: Holes, slots, etc., arranged at a regular spacing in a circle
- **Q number of holes/slots**—default: 1
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

MANUALplus also interrogates the parameters that are required for machining the respective elements (see corresponding cycle descriptions).



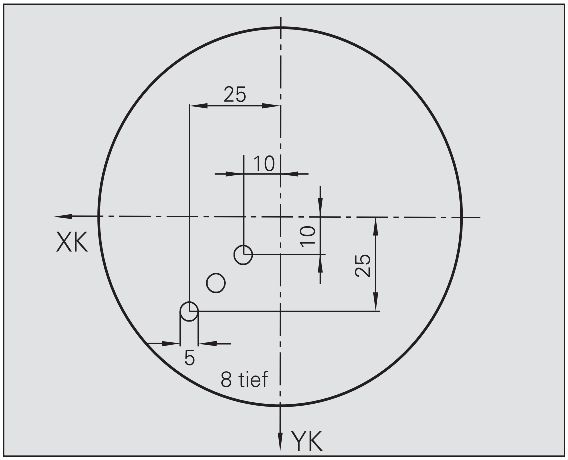


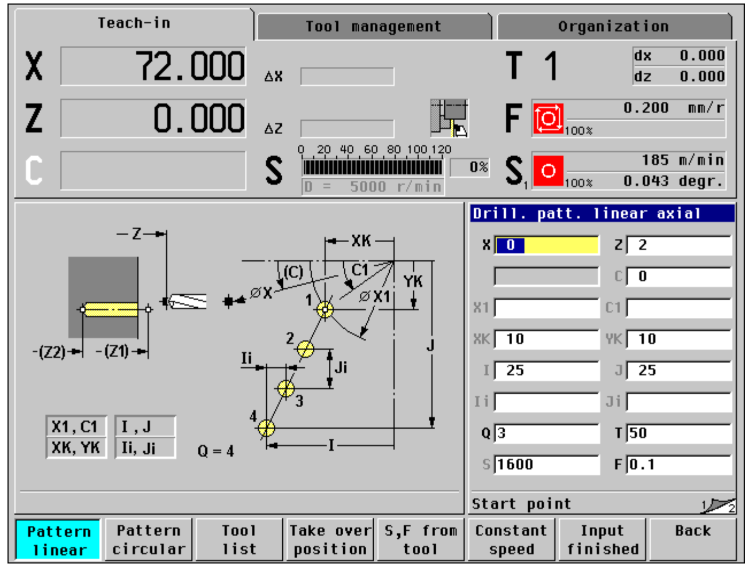
Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to "spindle angle C."
   - With C axis: Activate C axis and position to "spindle angle C" at rapid traverse.
   - Manual mode: Machining starts from current spindle angle.
2. Calculate the pattern positions.
3. Position to starting point of pattern.
4. Execute drilling/milling operation.
5. Position for next machining operation.
6. Repeat steps 4 and 5 until all machining operations have been completed.
7. Position to starting point Z and deactivate C axis.

Examples of pattern machining

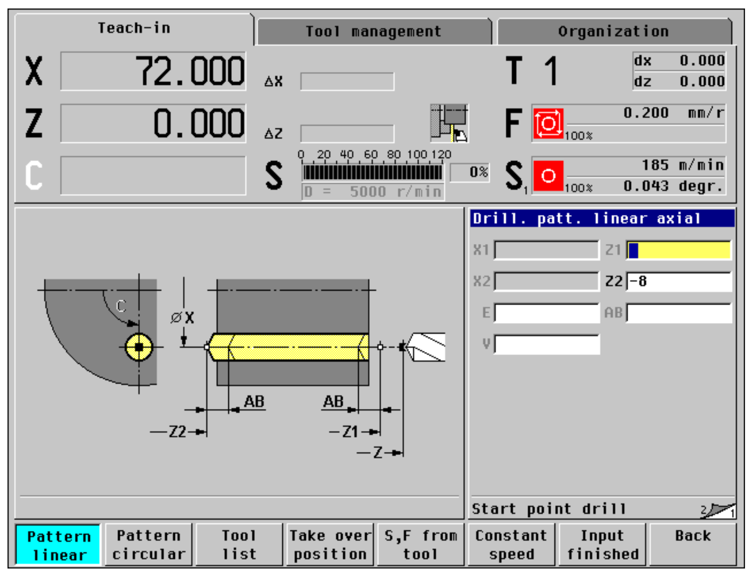
Linear hole pattern on face





A linear hole pattern is to be machined on the face of the workpiece with the "Drilling, axial" cycle. This machining operation requires a traversable spindle and driven tools.

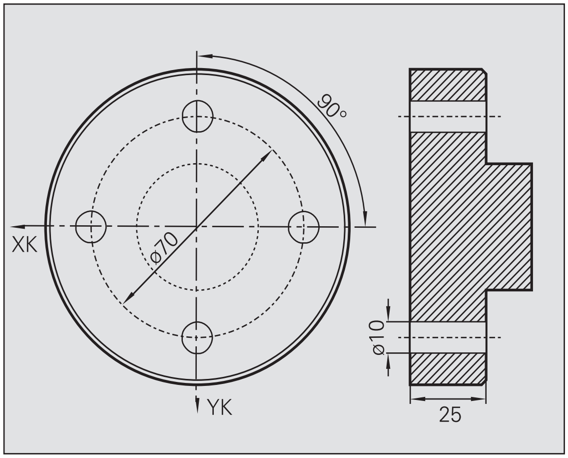
The pattern is programmed by entering the coordinates of the first and last hole, and the number of holes (see figure at upper right). Only the depth is indicated for the drilling cycle (see figure at lower right).

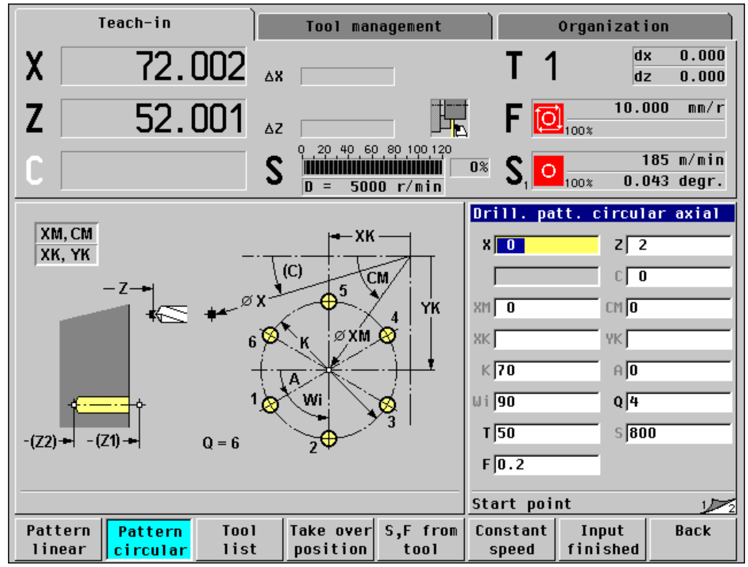


Tool data

- WO = 8—Tool orientation
- I = 5—Drilling diameter
- B = 90—Point angle
- H = 1—The tool is a driven tool

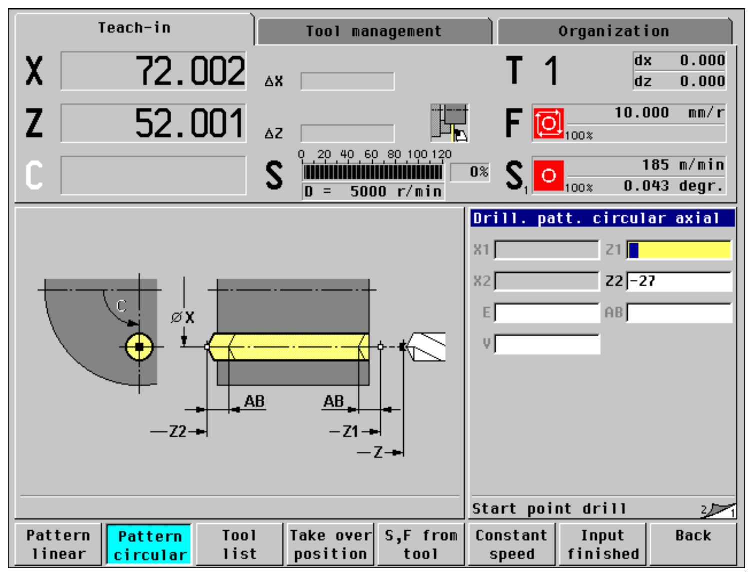
Circular hole pattern on face





A circular hole pattern is to be machined on the face of the workpiece with the "Drilling, axial" cycle. This machining operation requires a traversable spindle and driven tools.

The pattern center point is entered in Cartesian coordinates (see figure at top right).

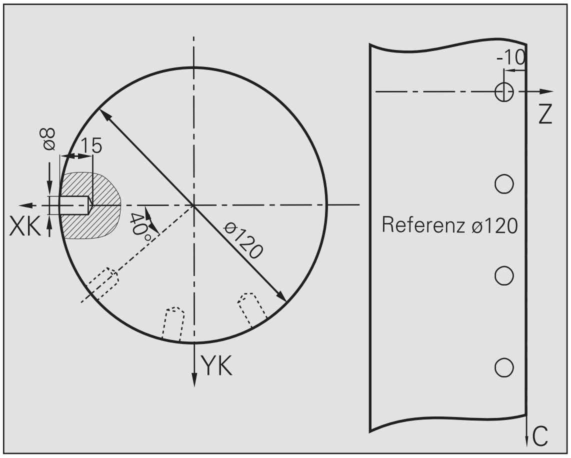


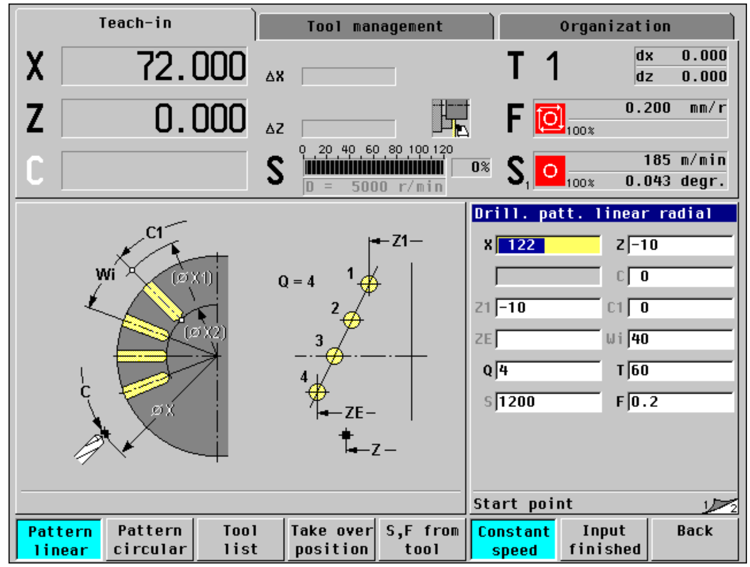
Since this example is to illustrate how you drill a through hole, the hole end point Z2 is programmed such that the tool has to drill all the way through the workpiece before it reaches the end point. The parameters "AB" and "V" define a feed reduction for both pre-drilling and through-boring (see figure at lower right).

Tool data

- WO = 8—Tool orientation
- I = 10—Drilling diameter
- B = 90—Point angle
- H = 1—The tool is a driven tool

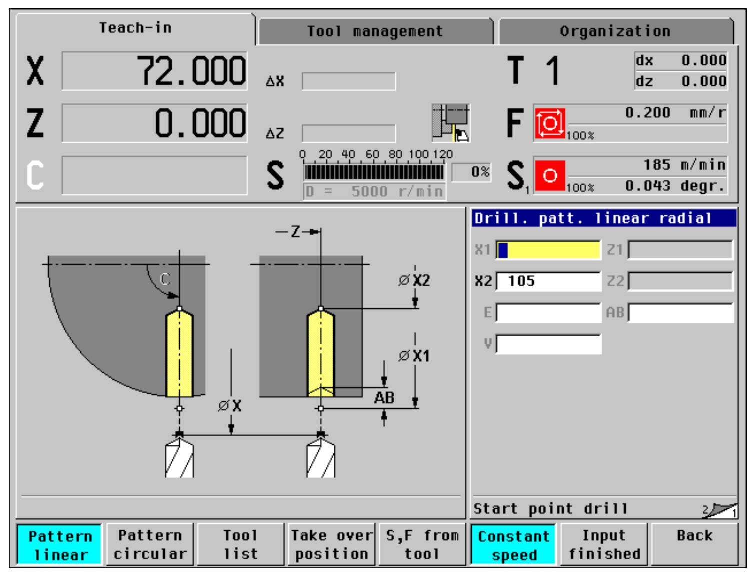
Linear hole pattern on lateral surface





A linear hole pattern is to be machined on the lateral surface of the workpiece with the "Drilling, axial" cycle. This machining operation requires a traversable spindle and driven tools.

The drilling pattern is defined by the coordinates of the first hole, the number of holes, and the spacing between the holes (see figure to the upper right). Only the depth is indicated for the drilling cycle (see figure at lower right).



Tool data

- WO = 2—Tool orientation
- I = 8—Drilling diameter
- B = 90—Point angle
- H = 1—The tool is a driven tool

4.10 DIN Cycles

„Select "DIN cycle."

This function allows you to select a DIN cycle (DIN macro) and integrate it in a MANUALplus cycle program.

The machine data that are programmed in the DIN cycle (in Manual mode, the currently active machine data) become effective as soon as you start the DIN macro. You can change the machine data (T, S, F) at any time by editing the DIN macro.

Danger of collision!

- **Cycle programming:** With DIN macros, the zero point shift is reset at the end of the cycle. Therefore, do not use any DIN macros with zero point shifts in cycle programming.
- In this cycle, no starting point is defined. Please keep in mind that the tool moves on a diagonal path from the current position to the first position that is programmed in the DIN macro.

Cycle parameters

- **N DIN macro number**
- **T tool number**
- **S spindle speed / cutting speed**
- **F feed per revolution**

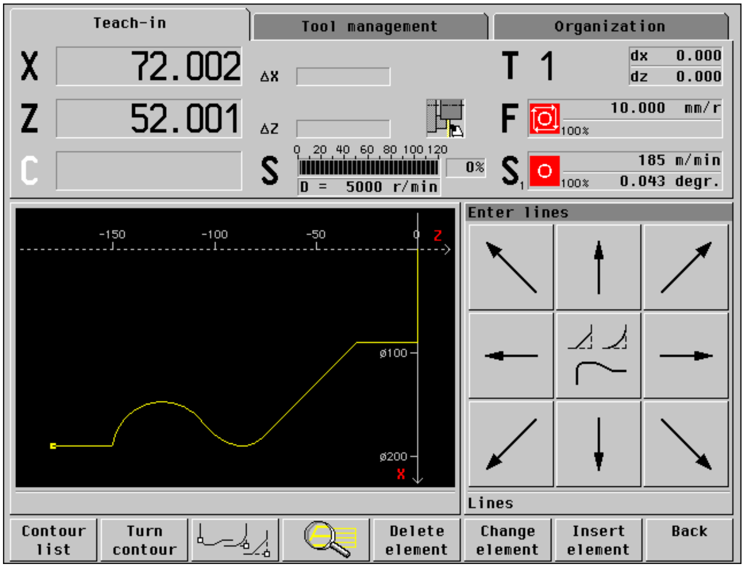
5

ICP Programming

5.1 ICP Contours

The Interactive Contour Programming (ICP) feature provides graphic support when you are defining the workpiece contours for ICP cycles. (ICP is the abbreviation of "**I**nteractive **C**ontour **P**rogramming".)

The contours are defined using linear and circular contour elements as well as form elements like chamfers, roundings, and undercuts.



> For **lathes** used in the machining of ICP contours, you need to define the tool angle and point angle.

Calculation of contour geometry

MANUALplus automatically calculates all missing coordinates, points of intersection, center points, etc. that can be derived mathematically.

If the entered data permit several mathematically possible solutions, you can inspect the individual solutions and select the proposal that matches the drawing.

Each **unsolved contour element** is represented by a small symbol below the graphic window. MANUALplus displays all contour elements that can be drawn, even if they are not yet fully defined.

Form elements

You can insert chamfers and roundings at each corner of the contour. Undercuts according to DIN 76, DIN 509 E, DIN 509 F are only possible at paraxial, orthogonal contour corners.

You have the following alternatives for entering form elements:

- Enter all contour elements, including the form elements, in the sequence in which they are given in the workpiece drawing.
- First define the "rough contour" without the form elements. Then you superimpose the form elements (see also "Programming Changes to ICP Contours" on page 254).

Special feed

You can assign a **special feed** to contour elements, which will be used for finish-machining these elements.

5.2 Editing ICP Contours

An ICP contour consists of definitions of the individual contour elements it is made up of. Each ICP contour is clearly identified by its number and a short description. ICP contours are integrated in ICP cycles.

You program an ICP contour by entering the individual contour elements one after the other in the correct sequence. The **starting point** is defined when you describe the first contour element. The **end point** is determined by the target point of the last contour element.

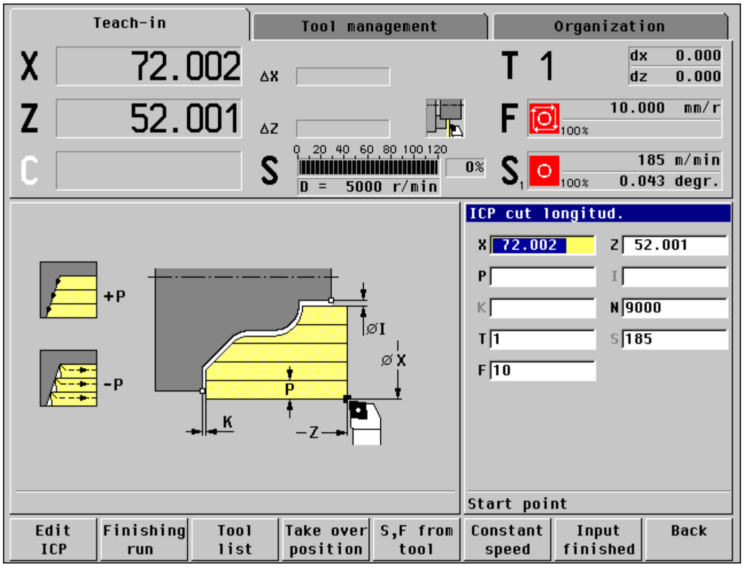
Existing contours can be **edited** or **added to.**

The contour elements / subcontours are displayed as soon as they are programmed. With the zoom and panning functions, you can adjust the graphics as required.

You can call ICP contours either with "Select ICP contours," or by defining an ICP contour number ("ICP contour input box). To switch between the list of ICP contours and the "ICP contour" input box, press the vertical arrow keys or ENTER.

You can successively program/edit several ICP contours. After exiting the ICP editor, the last active "ICP contour number" is transferred to the cycle.

For copying or deleting ICP contours, see “Program Management” on page 75.



Calling the ICP editor

Edit ICP

Press ***Edit ICP.***

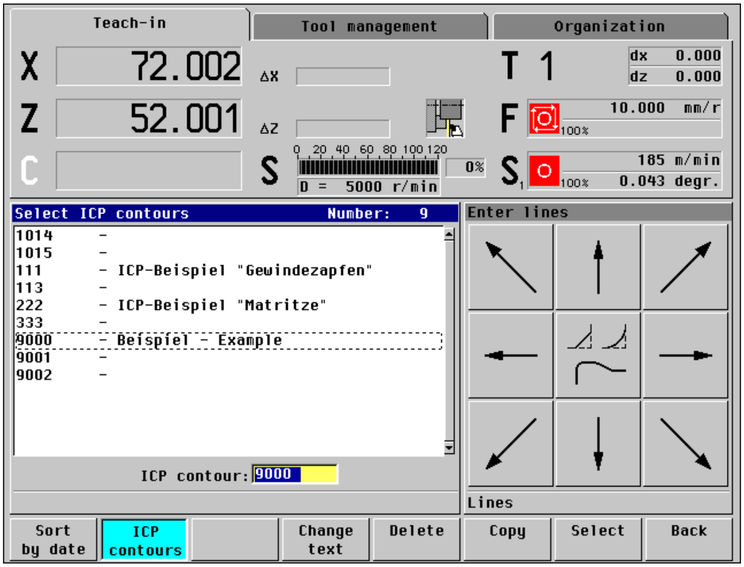
- Define a new ICP contour number, or
- select an existing ICP contour number.

Select

Press ***Select.***

You can now use the ICP editor to

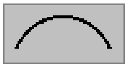
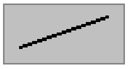
- enter a new contour, or
- display the selected existing contour for subsequent editing.



Programming and adding to ICP contours

After selecting a contour element, you enter the known parameters. MANUALplus automatically calculates parameters that have not been defined from the adjoining contour elements. You can usually program the contour elements with the dimensions given in the production drawing.

You can toggle between the lines and arcs menus by soft key. Form elements (chamfers, roundings, and undercuts) are selected with the menu key.

Soft keys	
	Call the arcs menu.
	Call the lines menu.

ICP contour programming.

Insert element	Press ***Insert element.***

Select the element type:

- Direction of line ("Enter lines" menu)
- Direction of rotation and type of dimensioning of circular arcs ("Enter arcs" menu)
- Type of form element

Define the parameters.

You can **add to** an ICP contour by entering additional contour elements that are "appended" to the existing contour. A newly entered contour element is always linked to the "last contour element." MANUALplus identifies the last contour element by a small square at the end of the contour, when the contour is being displayed, but not machined.

Absolute or incremental dimensions

The setting of the ***Increment*** soft key determines which type of coordinate is active. Incremental parameters will have the appendix "i" (Xi, Zi, etc.).

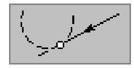
Transitions between contour elements

A transition between two contour elements is called "tangential" when one contour element makes a smooth and continuous transition to the next. There is no visible kink or corner at the intersection. With geometrically complex contours, tangential transitions are useful for reducing the input of dimensional data to a minimum and eliminating the possibility of mathematically contradictory entries.

To be able to calculate unsolved contour elements, MANUALplus must know the type of transition that connects the contour elements.

The transition to the next contour element is determined by soft key.

> Error messages that occur during definition of the ICP contour are often caused by "forgotten" tangential transitions.

Soft keys for tangential transitions	
	**Tangential transition** from linear to circular element
 	**Tangential transition** from circular element to circular or linear element (for direction of rotation, see symbol)

Contour graphics

As soon as you have entered a contour element, MANUALplus checks whether the element is "solved" or "unsolved." A "solved" element is a contour element that is fully and unambiguously defined. It is drawn immediately.

"Unsolved" contour element:

- has not yet been fully defined.
- MANUALplus shows a symbol below the graphics window, which reflects the element type and the line direction / direction of rotation.
- An unsolved linear element is represented if the starting point and direction are known.
- An unsolved circular element is represented as a full circle if the circle center and the radius are known (see figure at upper right).

As soon as an unsolved contour element can be calculated, it is drawn by MANUALplus. The symbol is then deleted (see figure at lower right).

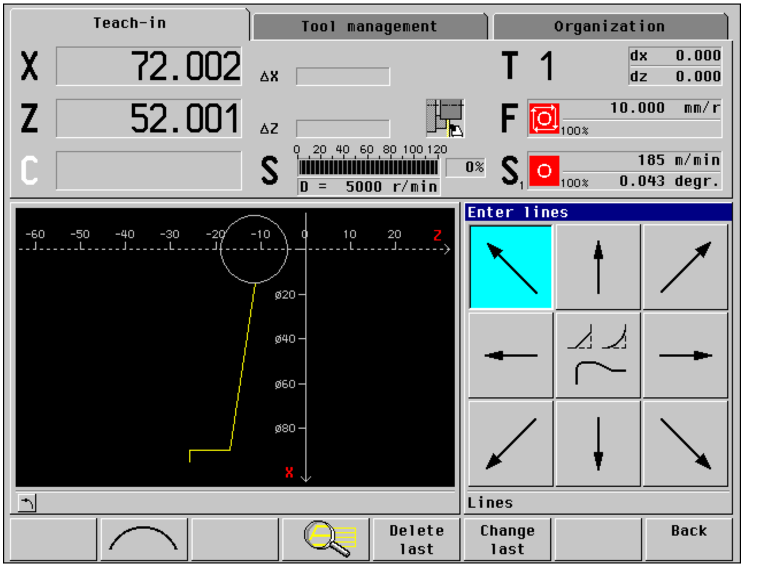
An incorrect contour element is displayed if possible. In addition, an error message is issued.

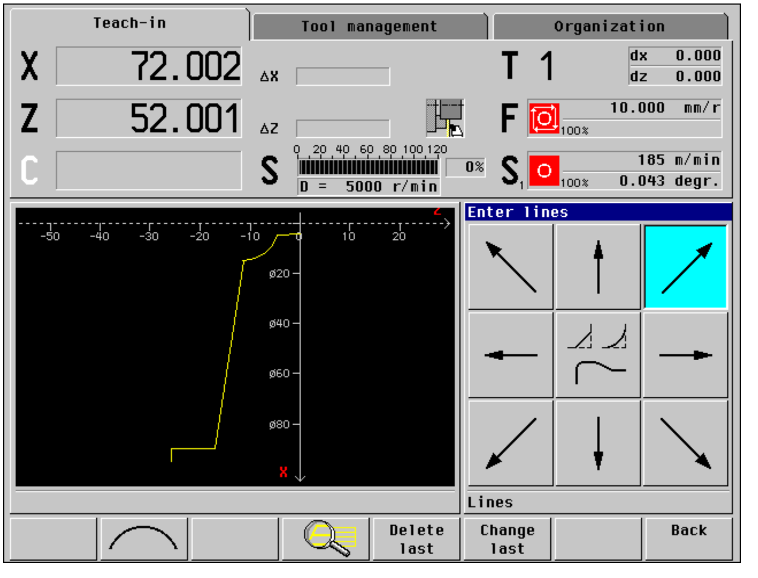
Colors in contour graphics

Solved and unsolved contour elements as well as selected contour elements, selected contour corners, and remaining contours are depicted in different colors. (The selection of contour elements / contour corners and remaining contours is important when you are editing ICP contours.)

Colors:

- Yellow: For solved elements
- Gray: For unsolved or incorrect elements
- Red: Selected solution, selected element
- Blue: Remaining contour



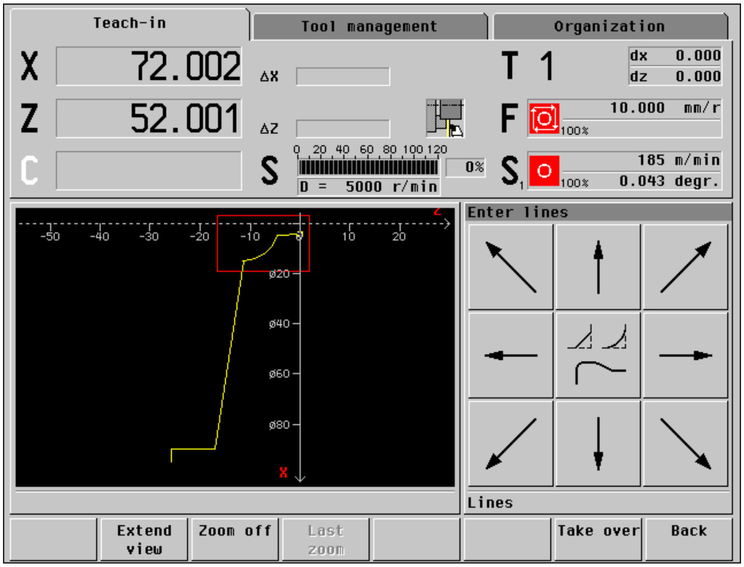


Changing the ICP contour graphics

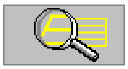
MANUALplus selects the area to be represented such that all entered contour elements are displayed.

You can magnify/reduce the displayed graphics with the PgUp/PgDn keys, and pan the detail with the arrow keys. These functions are available when the contour is displayed but is not being edited.

To isolate and display a precise detail, proceed as follows:



Calling and setting the "Zoom" function



When you press ***Zoom,*** a "red frame" appears on the screen with which you can select the detail you wish to isolate.

"Red frame"

– To move the red frame, use the arrow keys.

– To enlarge the red frame, use the PgDn key.

– To reduce the red frame, use the PgUp key.

Take over: The adjusted "red frame" will be displayed as a picture.

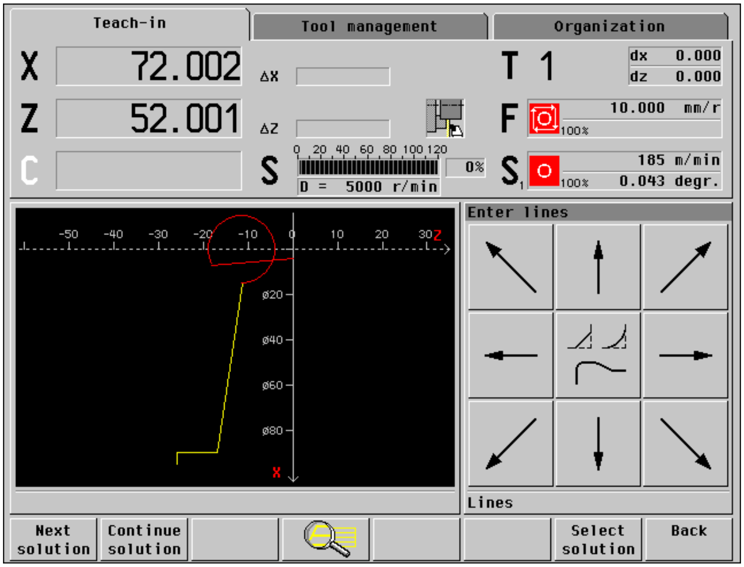
Extend view: By reducing the contour, a greater area can be displayed (if, for example, an area of the workpiece is not displayed in the graphics window).

Zoom off: All previously defined contour elements are shown as large as possible.

Last zoom: Return to the last setting of Zoom.

Selection of solutions

If the entered data permit several possible solutions, you can inspect the mathematically possible solutions with ***Next solution*** / ***Continue solution*** and confirm the correct solution with ***Select solution*** (see figures at right).



> If the contour still contains unsolved contour elements when you exit the editing mode, MANUALplus will ask you whether to cancel these elements.

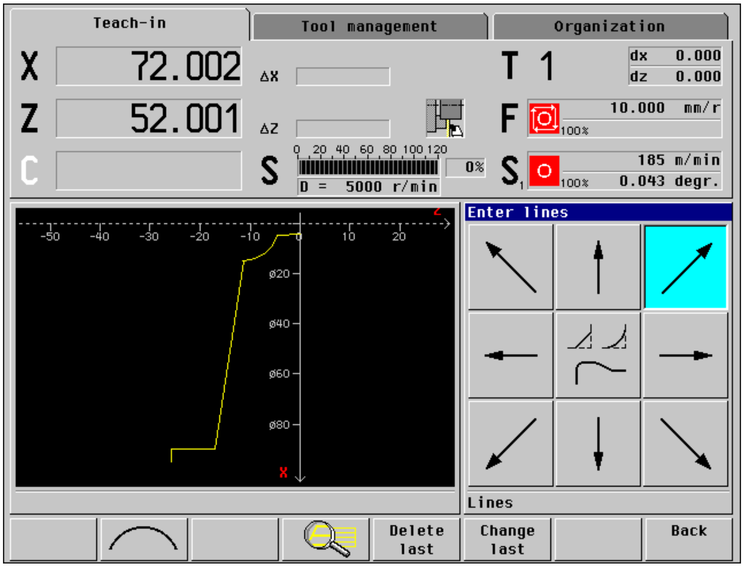
Editing the last contour element

By pressing ***Change last,*** you can call the data of the last entered contour element and edit it as desired.

Depending on the adjoining contour elements, corrections of linear or circular elements are either transferred immediately or the corrected contour is displayed for inspection. Contour elements that have been edited are highlighted in color. If the change permits several possible solutions, you can check all mathematically possible solutions with ***Next solution*** / ***Continue solution.***

The changes will not become effective until you press ***Select solution.*** If you press ***Back,*** the change will not be transferred and the previous description of the last contour element becomes effective again.

The type of contour element (linear or circular element), the direction of a linear element, or the direction of rotation of a circular element cannot be changed with this function. Should this be necessary, you must delete the last contour element and add a new contour element.



Deleting the last contour element

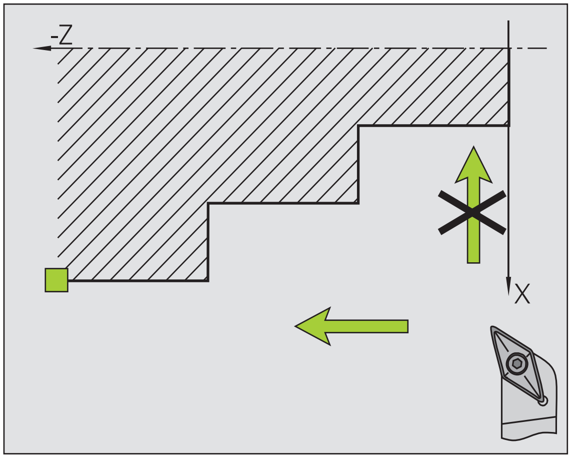
If you press ***Delete last,*** the data of the last entered contour element are canceled. You can use this function repeatedly to delete several successive contour elements.

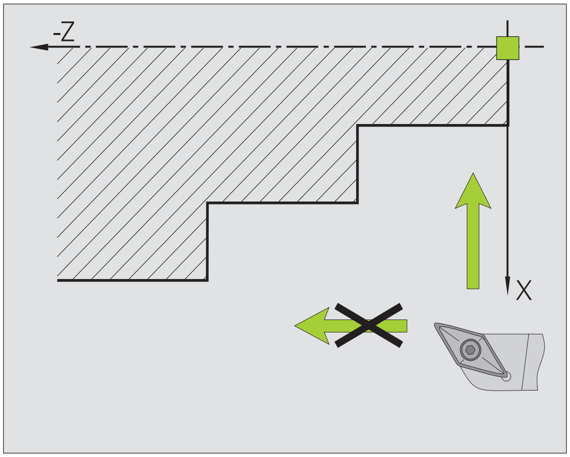
Contour direction

The cutting direction depends on the direction of the contour. If the contour is described in the negative Z-axis direction, the control uses a longitudinal cycle (see figure to the upper right). If the contour is described in the negative X-axis direction, the control uses a transverse cycle (see figure at center right).

- **ICP cut, longitudinal/transverse (roughing)**
  MANUALplus machines the workpiece in the contour direction.
- **ICP finishing, longitudinal/transverse**
  MANUALplus finishes the workpiece in the contour direction.

> An ICP contour which was defined for a roughing operation with the cycle "ICP cut longitudinal" cannot be used for machining with the cycle "ICP cut transverse." You can reverse the contour direction with the ***Turn contour*** soft key.





5.3 Importing of DXF Contours

Fundamentals

Contours available in DXF format can be imported into the ICP editor.

DXF contours describe contour trains for ICP cycles (recessing, cutting and milling cycles).

For contour trains for recessing and cutting cycles, DXF layers should contain only one contour. For contours for milling cycles multiple DXF contours can be contained and imported.

DXF import is available as of software versions 507 807-11 and 526 488-03.

Requirements of a DXF contour or DXF file:

- Only two-dimensional elements
- The contour must be in a separate layer (without dimension lines, without wraparound edges, etc.)
- Depending on the setup of the lathe, contours for recessing or cutting cycles must be either before or behind the workpiece.
- No complete circles, no splines, no DXF blocks (macros), etc.
- The imported contours may consist of no more than 4000 elements (lines, arcs). In addition, up to 10 000 polyline points are permitted.

Preparation of contours during DXF import

The contour is converted from DXF format to ICP format during importation. The following changes are made to the contour representation, since there are basic differences between the DXF and ICP formats:

- Possible gaps between contour elements are closed
- Polylines are transformed into linear elements

In addition, the following features, which are necessary for a ICP contour, are specified:

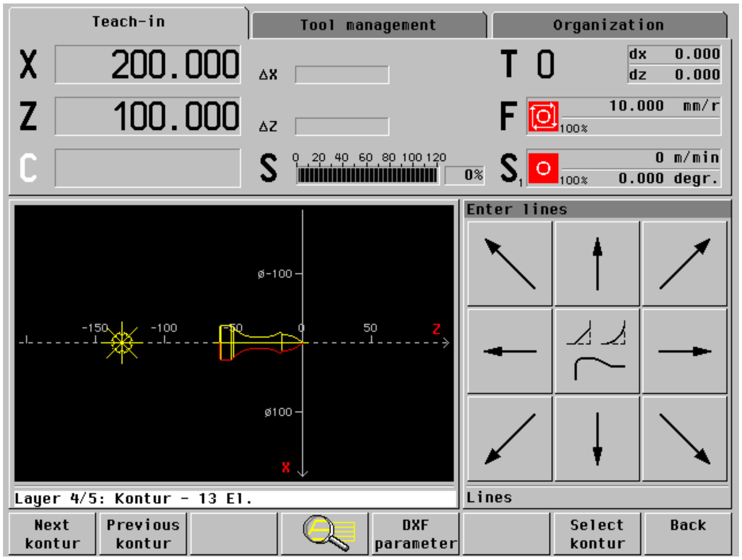
- The starting point of the contour
- The direction of rotation of the contour

Sequence of a DXF importation:

- Selection of the DXF file
- Selection of the layer, which contains only the contour(s)
- Import of the contour(s)

DXF import

The ICP editor offers the DXF import during the contour entry phase.



DXF import

Edit ICP — Press ***Edit ICP.***

Insert element — Press ***Insert element.***

DXF import — Press ***DXF Import.***

Select the file with the DXF contour(s).

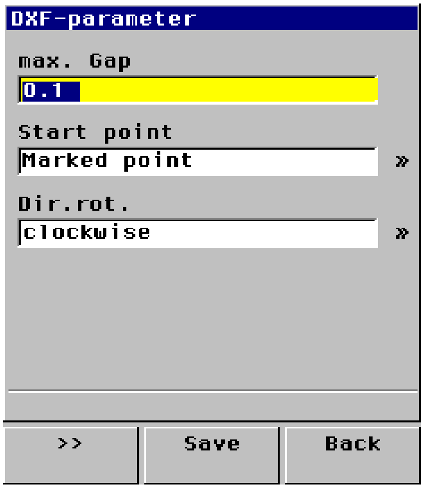
Next kontur / Previous kontur — Press ***Next contour*** or ***Previous contour*** to select the DXF layer.

Select kontur — Press ***Assume contour.*** The ICP editor loads the selected DXF contour and converts it to ICP format.

> Contour elements that have already been entered or loaded are overwritten immediately when you press the ***Assume contour*** soft key.

Configuring the DXF import

After you have selected a file with DXF contours, you can adapt the parameters for configuring the DXF import.



Adapting the DXF parameters

DXF parameter

Press ***DXF parameters.*** The MANUALplus opens the "DXF parameters" dialog box.

Enter the DXF parameters (see the meanings below).

Save

Press ***Save.*** The MANUALplus assumes the parameters.

Setting the DXF parameters to standard values

Call the **DXF parameters** dialog box.

reset

Press ***Reset.*** The MANUALplus enters the standard parameters.

Save

Press ***Save.*** The MANUALplus assumes the parameters.

Meaning of the DXF parameters:

- **Maximum gap:** There might be small gaps between contour elements in the DXF drawing. With this parameter you specify how large the distance between two contour elements may be.
  - If the **maximum gap** is not exceeded, then the following element is seen as being part of the "current" contour.
  - If the **maximum gap** is exceeded, then the following element is considered an element of the "new" contour.
- **Starting point:** The DXF import analyzes the contour and determines the starting point. The possible settings have the following meaning:
  - **right, left, top, bottom:** The starting point is set to be the contour point that is the furthest to the right (or left, or ...). If more than one contour point satisfies this requirement, then one of these points is selected automatically.

- **Maximum distance:** The DXF import sets the starting point to one of the two contour points farthest apart from each other. The program automatically determines which of these points is the starting point. It is not possible to influence this decision.
- **Marked point:** If one of the contour points in the DXF drawing is marked with a complete circle, then this point is specified as the starting point. The contour point must be at the center of the complete circle.

- **Direction of rotation:** Indicate whether the contour is aligned in clockwise or counterclockwise direction.

5.4 Programming Changes to ICP Contours

You can edit existing contours by:

- Editing contour elements
- Deleting contour elements
- Extending the contour (adding to the contour)
- Editing individual contour sections
- Superimposing form elements (refining the contour)

Editing a contour element

Editing a contour element

Change element	Press ***Change element***—a contour element is marked "selected" (highlighted in color).
Next element Previous element	Select the contour element to be edited.
Change	Display the contour element for editing.

Make the changes.

Over-write	Transfer the changes.

The contour or, if applicable, the possible solutions are displayed for inspection. With form elements and unsolved elements the changes are transferred immediately.

Select solution	Transfer the desired solution.

Editing an unsolved contour element

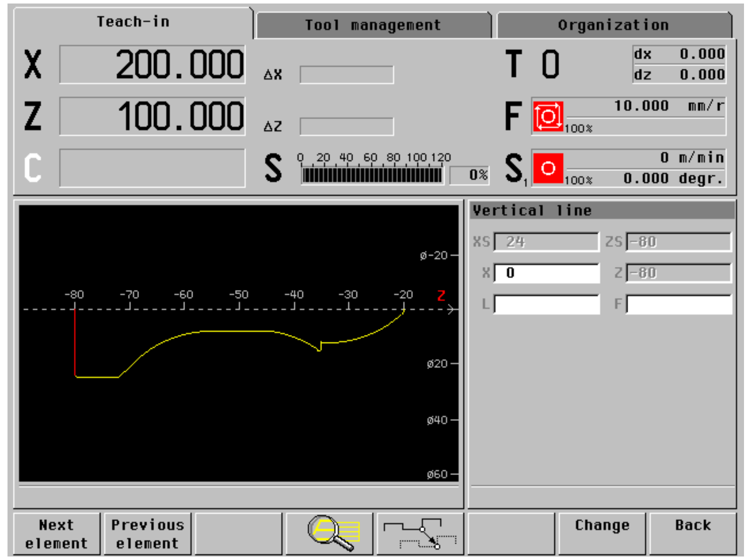
If a contour contains "unsolved" contour elements, the "solved" elements cannot be changed. You can, however, set or delete the "tangential transition" for the contour element located directly before the unsolved contour area.

If the element to be edited is an unsolved element, the associated symbol is marked "selected."

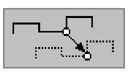
The element type and the direction of rotation of a circular arc cannot be changed. Should it be necessary to change the element type / direction of rotation, you must delete the contour element and then add it again.

Shifting a contour

You can shift a contour if it is not at the desired position. Select a suitable contour element (reference element). For shifting you enter the new position of the **starting point of the reference element.** The entire contour is shifted when function is completed.

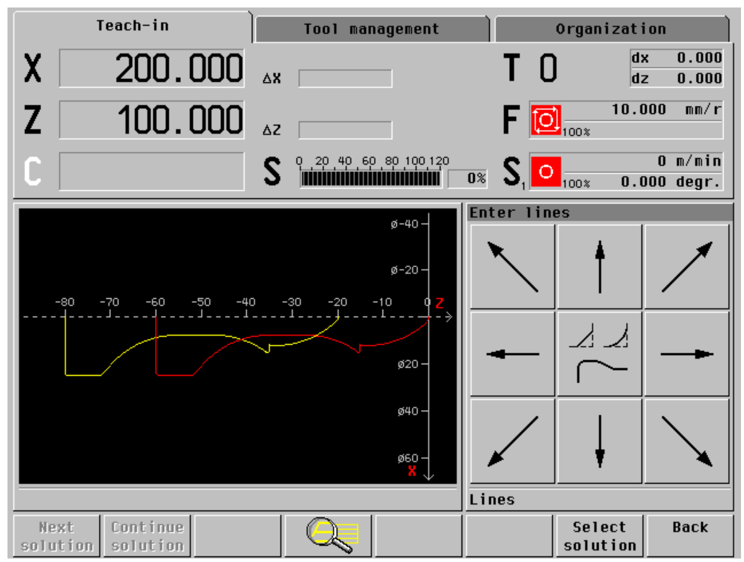


Shifting a contour

Change element	Press ***Change element***—a contour element is marked "selected" (highlighted in color).
Next element Previous element	Select the reference element
	Press ***Shift contour.***

Enter the new starting point of the reference element.

Over-write	Assume the new starting point (= new position). The MANUALplus shows the shifted contour.
Select solution	Assume the new position for the contour.



Adding a contour element

Adding a contour element

Insert element	Press ***Insert element.***

"Append" additional contour elements to the existing contour.

Deleting a contour element

Deleting a contour element

Delete element	Press ***Delete element***—a contour element is marked "selected" (highlighted in color).
Next element Previous element	Select the contour element to be deleted.
Delete	Delete the contour element.

You can delete several successive contour elements.

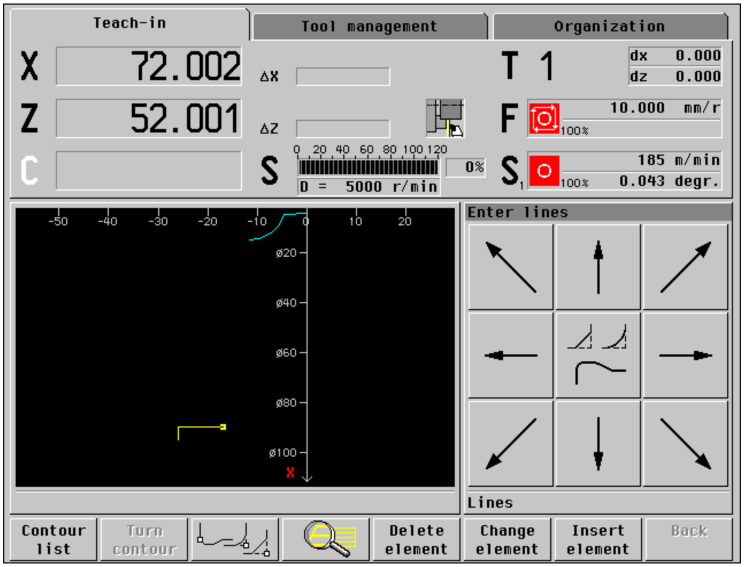
If the element to be deleted is an unsolved element, the associated symbol is marked "selected."

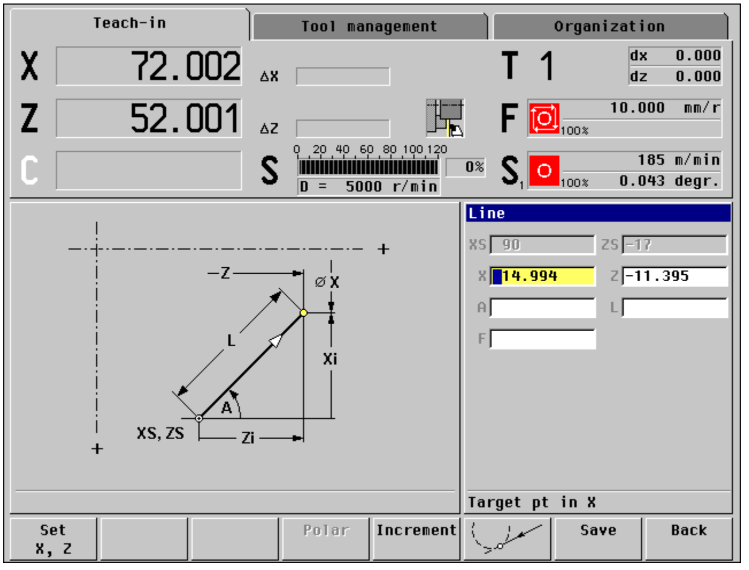
"Splitting" a contour

If you delete a contour element which is located between preceding **and** subsequent elements, the contour is split into a basic contour and a remaining contour (see figure at upper right).

The remaining contour cannot be edited—you can, however, change the basic contour and "link" it to the remaining contour. This is usually done by inserting new contour elements. "Linking" the "last contour element" to the "remaining contour" is also permitted if such a link is possible.

MANUALplus supports this possibility by taking over the starting point coordinates of the remaining contour. You simply press ***Set X, Z*** (see figure at right).

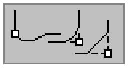




Superimposing form elements

When superimposing form elements, select the corner to be superimposed and then insert the desired form element.

Superimposing form elements



Press ***Superimpose form elements.***

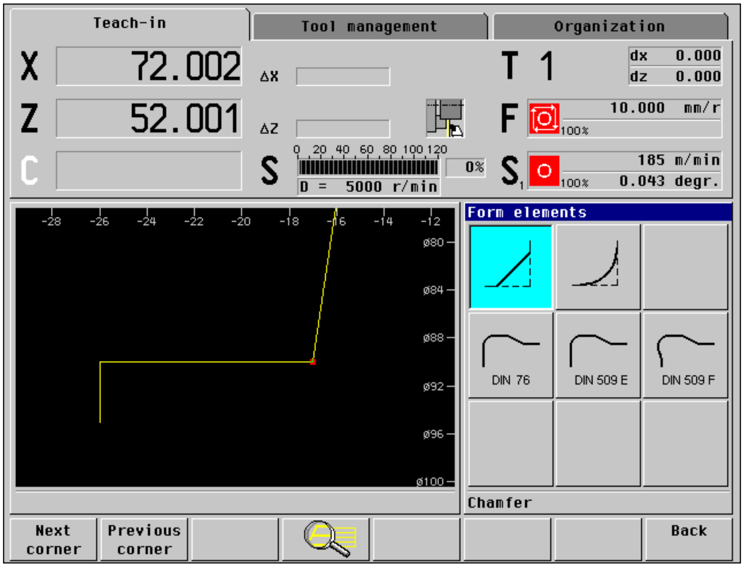
Next corner

Previous corner

Select the corner to be changed.

Select the desired **form element.**

Define the form element.



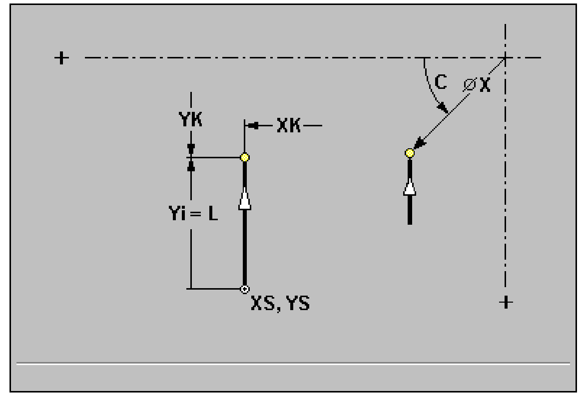
Superimposing on contours with "unsolved" contour areas

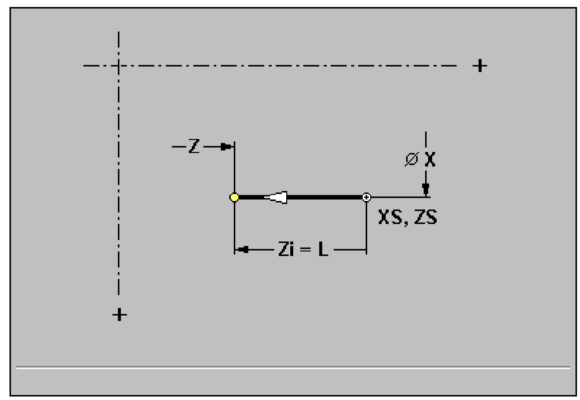
You can superimpose form elements even if the contour still contains unsolved contour areas.

Exit the input mode with ***Back*** and call ***Superimpose form elements.***

5.5 ICP Contour Elements, Turning Contour

Entering lines, turning contour

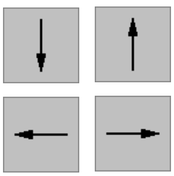




Use the menu symbol to select the direction of the contour element and assign it a dimension.

When defining horizontal and vertical linear elements, it is not necessary to enter the X and Z coordinates, respectively. MANUALplus inhibits the corresponding input field if no unsolved elements exist.

Vertical/horizontal lines



Select the line direction.

You enter the dimensions of the line and then define the transition to the next contour element.

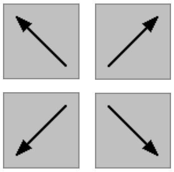
Parameters for vertical line

- **XS, ZS starting point in X/Z** (= end point of last element)
- **X target point in X**
- **Xi target point in X** (incremental): Distance from starting point to target point
- **L length of line**
- **F special feed**

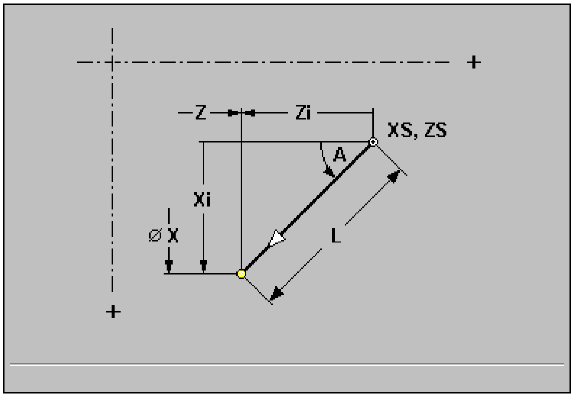
Parameters for horizontal line

- **XS, ZS starting point in X/Z** (= end point of last element)
- **Z target point in Z**
- **Zi target point in Z** (incremental): Distance from starting point to target point
- **L length of line**
- **F special feed**

Line at angle



Select the line direction.



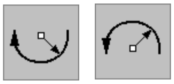
You enter absolute or polar dimensions for the line and then define the transition to the next contour element. The direction of the angle is shown in the graphic support window.

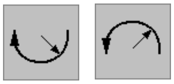
Parameters

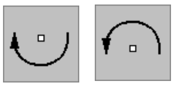
- **XS, ZS starting point in X/Z** (= end point of last element)
- **X, Z target point in X/Z**
- **Xi, Zi target point in X/Z** (incremental): Distance from starting point to target point
- **L length of line**
- **A angle to Z axis**
- **F special feed**

Entering circular arcs, turning contour

Select direction of rotation and type of dimensioning


Arc with center and radius

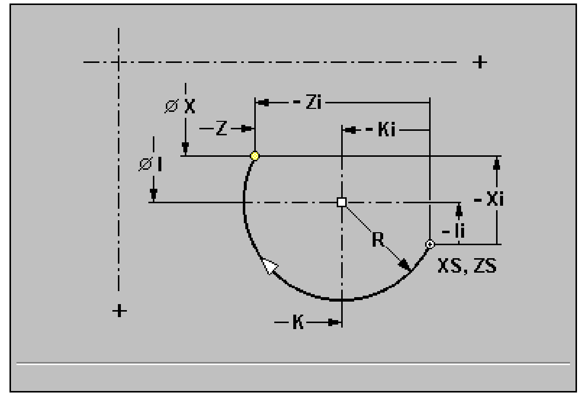

Arc with radius

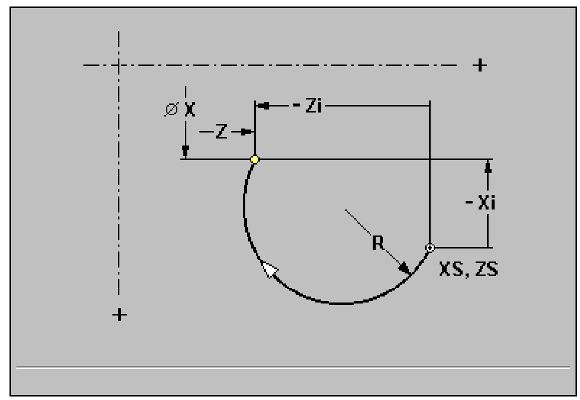

Arc with center

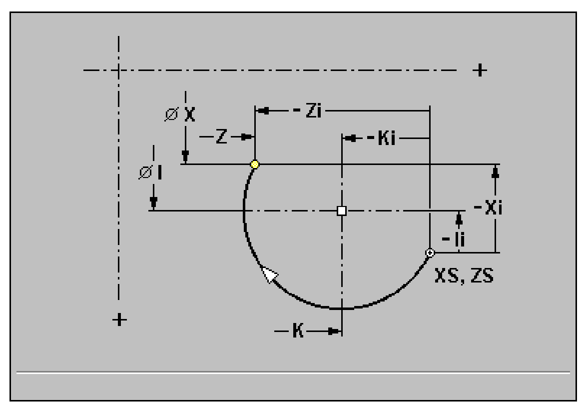
You enter the dimensions of the arc and then define the transition to the next contour element.

Parameters (for "Arc with radius," the center is not requested. For "Arc with center point," the radius is not requested.)

- **`XS, ZS starting point in X/Z`** (= end point of last element)
- **`X, Z target point in X/Z:`** End point of circular arc
- **`Xi, Zi target point in X/Z`** (incremental): Distance from starting point to target point
- **`I, K center point in X/Z:`** Center of circular arc
- **`Ii, Ki center point in X/Z`** (incremental): Distance from starting point to center point
- **`R radius`**
- **`F special feed`**





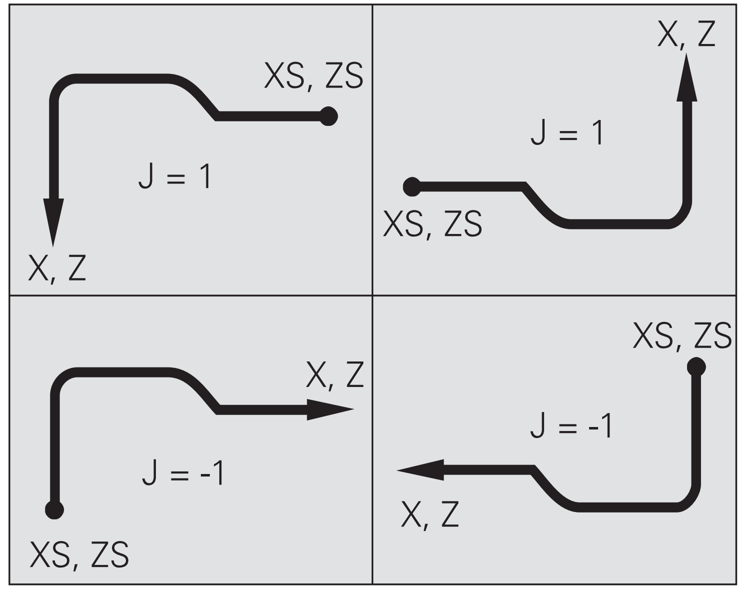


Entering form elements

Chamfer/rounding

Chamfers/roundings are defined on contour corners. A "contour corner" is the point of intersection between the approaching and departing contour elements. MANUALplus cannot calculate a chamfer or rounding until the departing contour element is known. During the parameter input for the chamfer/rounding, the coordinates of the corner are shown in "starting point XS, ZS."

If an ICP contour starts with a chamfer/rounding, the "approaching contour element" is missing. You can then use the parameter "element position J" to clearly define the position of the chamfer/rounding. MANUALplus converts a chamfer or rounding at the start of the contour to a linear or circular element.



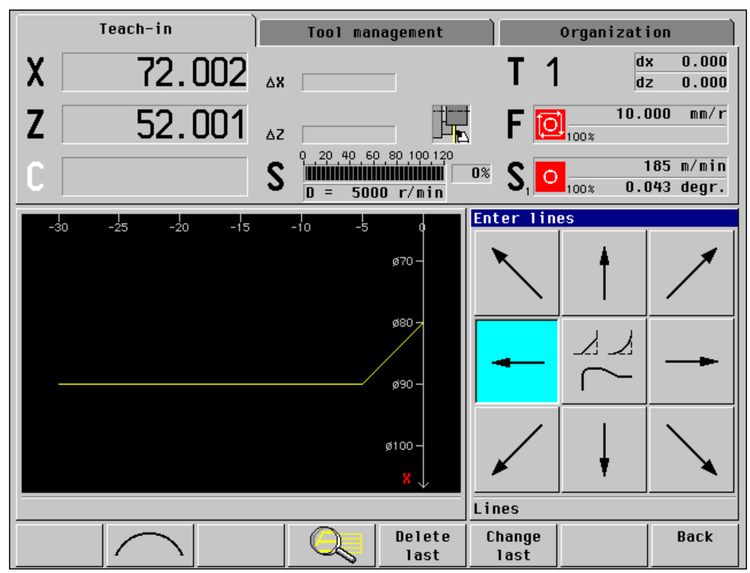
Undercut

The form element "undercut" consists of a longitudinal element, the actual undercut and a transverse element. An undercut can start with a longitudinal or transverse element.

If the undercut corner is not known, the parameter "Element position J" defines whether the undercut starts with the longitudinal or transverse element (see figure at upper right).

Example: External chamfer at contour start

If you program "element position J=1," the imaginary approaching reference element is a transverse element in the positive X-axis direction (see figure to the bottom right).



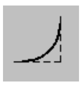
Chamfer/rounding, turning contour

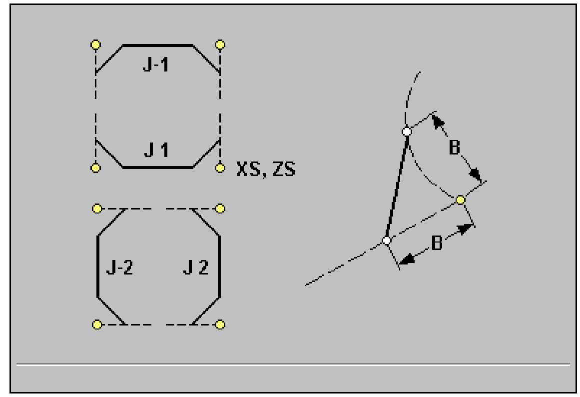
Chamfer

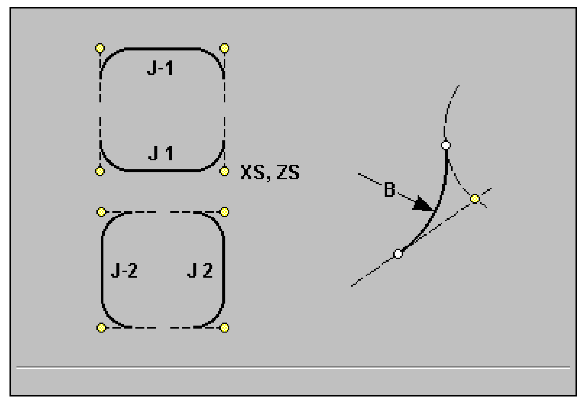


Select form elements or chamfer/rounding.


Choose chamfer.


Choose rounding.



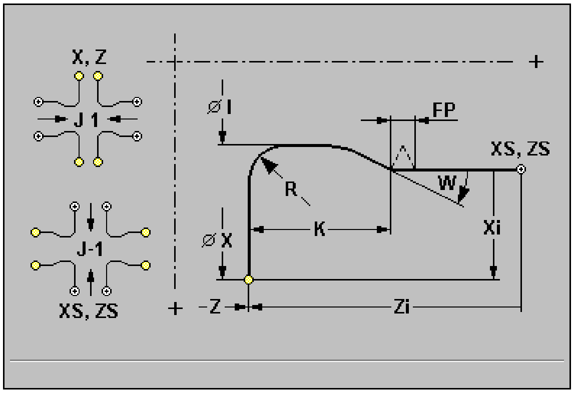


The corner is predefined by the starting point. You need only enter the "chamfer width B" or "rounding radius B."

Parameters

- **B chamfer width**—or
- **B rounding radius**
- **J element position:** "imaginary approaching reference element"
  - J=1: Transverse element in the positive X-axis direction
  - J=–1: Transverse element in the negative X-axis direction
  - J=2: Longitudinal element in the positive Z-axis direction
  - J=–2: Longitudinal element in the negative Z-axis direction
- **F special feed**

Undercuts, turning contour



Thread undercut DIN 76

Select form elements.

Select thread undercut DIN 76.

For thread undercut DIN 76, the diameter of the longitudinal element represents the thread diameter (or, with internal threads, the core diameter).

Parameters

- **`XS, ZS starting point in X/Z:`** Starting point of undercut
- **`X, Z target point in X/Z:`** End point of undercut
- **`FP thread pitch`**—default: Value from standard table
- **`I undercut diameter`**—default: Value from standard table
- **`K undercut length`**—default: Value from standard table
- **`W undercut angle`**—default: Value from standard table
- **`R undercut radius`**—default: Value from standard table
- **`J element position`**—default: 1
  - J=1: Undercut starts with the longitudinal element and ends with the transverse element.
  - J=–1: Undercut starts with the transverse element and ends with the longitudinal element.
- **`F special feed`**

Parameters that are not defined are automatically calculated from the standard table (see “DIN 76—undercut parameters” on page 525):

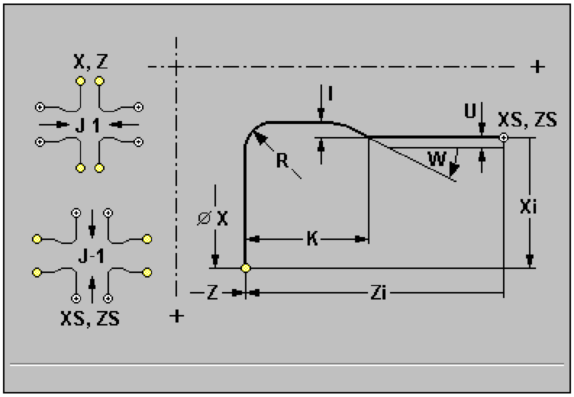
- The "thread pitch FP" is determined from the diameter ("starting point XS").
- The parameters I, K, W, and R are calculated from the thread pitch FP.

> - If you are programming an internal thread, it is advisable to preset the "thread pitch FP," since the diameter of the longitudinal element is not the thread diameter. If you have MANUALplus calculate the thread pitch automatically, slight deviations may occur.
> - The "element position J" cannot be entered when superimposing the undercut and cannot be changed when programming changes to ICP contours. (The contour corner has already been clearly defined.)

Undercut DIN 509 E

Select form elements.

Select undercut DIN 509 E.

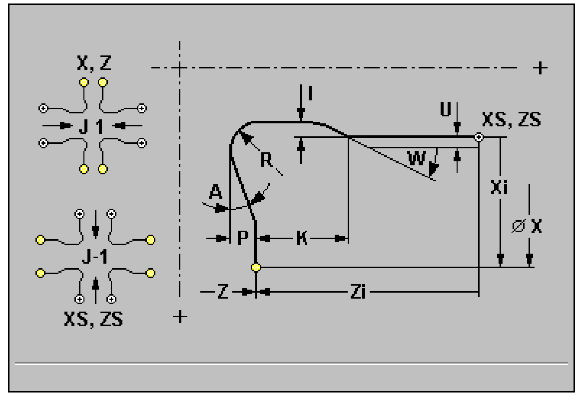


Parameters that are not entered are automatically calculated from the standard table (see “DIN 509 E, DIN 509 F—undercut parameters” on page 527"):

Parameters

- **`XS, ZS starting point in X/Z:`** Starting point of undercut
- **`X, Z target point in X/Z:`** End point of undercut
- **`I undercut diameter`**—default: Value from standard table
- **`K undercut length`**—default: Value from standard table
- **`W undercut angle`**—default: Value from standard table
- **`R undercut radius`**—default: Value from standard table
- **`Finishing oversize`**—default: No finishing oversize
- **`J element position`**—default: 1
  - J=1: Undercut starts with the longitudinal element and ends with the transverse element.
  - J=–1: Undercut starts with the transverse element and ends with the longitudinal element.
- **`F special feed`**

> The "element position J" cannot be entered when superimposing the undercut and cannot be changed when programming changes to ICP contours. (The contour corner has already been clearly defined.)



Undercut DIN 509 F

Select form elements.

Select undercut DIN 509 F.

Parameters that are not entered are automatically calculated from the standard table (see "DIN 509 E, DIN 509 F—undercut parameters" on page 527"):

Parameters

- **`XS, ZS starting point in X/Z:`** Starting point of undercut
- **`X, Z target point in X/Z:`** End point of undercut
- **`I undercut diameter`**—default: Value from standard table
- **`K undercut length`**—default: Value from standard table
- **`W undercut angle`**—default: Value from standard table
- **`R undercut radius`**—default: Value from standard table
- **`P transverse depth`**—default: Value from standard table
- **`A transverse angle`**—default: Value from standard table
- **`Finishing oversize`**—default: No finishing oversize
- **`J element position`**—default: 1
  - J=1: Undercut starts with the longitudinal element and ends with the transverse element.
  - J=–1: Undercut starts with the transverse element and ends with the longitudinal element.
- **`F special feed`**

The "element position J" cannot be entered when superimposing the undercut and cannot be changed when programming changes to ICP contours. (The contour corner has already been clearly defined.)

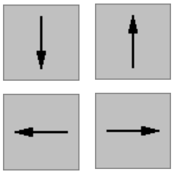
5.6 ICP Contour Elements on the Face

Enter the dimensions of the contour elements on face and lateral surface in Cartesian or polar values. You must pay attention to the setting of the ***Polar*** soft key. MANUALplus distinguishes Cartesian coordinates from polar coordinates by different address letters.

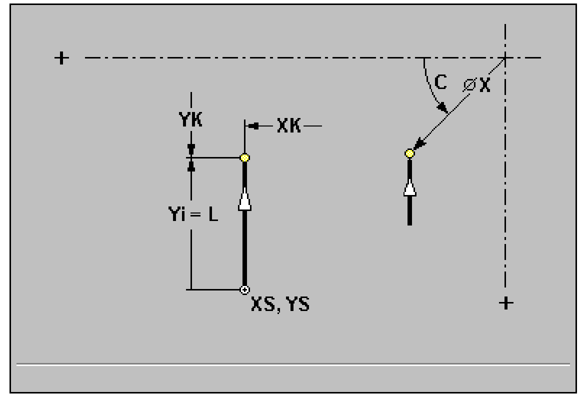
- **Polar coordinates**:
  - XD, X: Diameter
  - CS, C: Angle to positive XK axis
- Depending on the cycle, you can have the MANUALplus display ICP contours for either the face or the lateral surface by calling the "selection of ICP contours" with the ***Contour list*** soft key.

Entering lines on the face

Vertical/horizontal lines



Select the line direction.

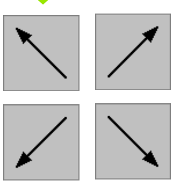


You enter the dimensions of the line and then define the transition to the next contour element.

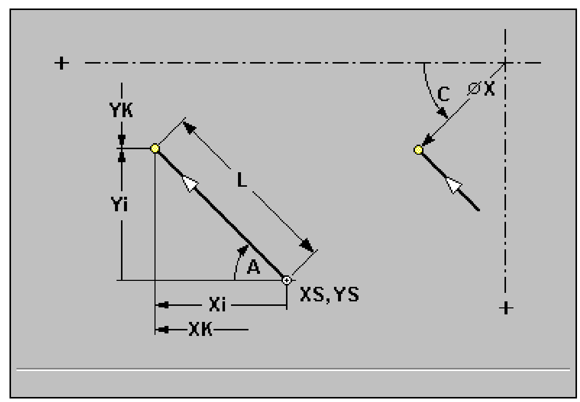
Parameters

- **`XS, YS starting point`** (Cartesian coordinates)
- **`XD, CS starting point`** (polar coordinates)
- **`XK, YK target point`** (Cartesian coordinates)
- **`X, C target point`** (polar coordinates)
- **`L length of line`**
- **`F special feed`**

Line at angle



Select the line direction.



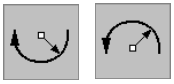
You enter the dimensions of the line and then define the transition to the next contour element.

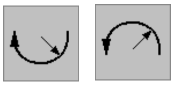
Parameters

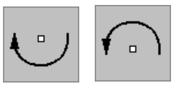
- **`XS, YS starting point`** (Cartesian coordinates)
- **`XD, CS starting point`** (polar coordinates)
- **`XK, YK target point`** in Cartesian coordinates
- **`X, C target point`** (polar coordinates)
- **`A angle to XK axis`**—for direction of angle, see graphic support window
- **`L length of line`**
- **`F special feed`**

Entering circular arcs on the face

Entering an arc


Arc with center and radius

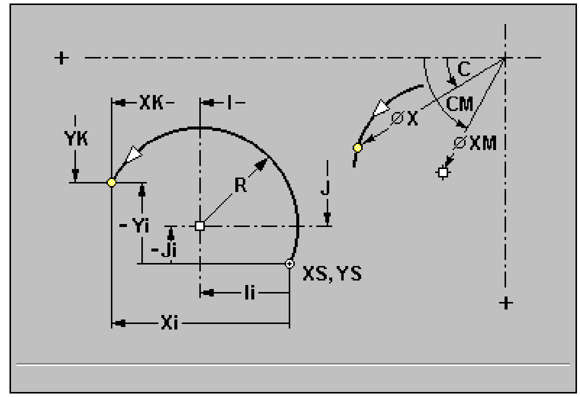

Arc with radius

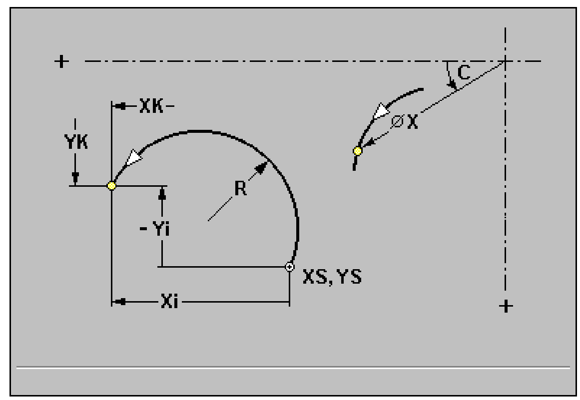

Arc with center

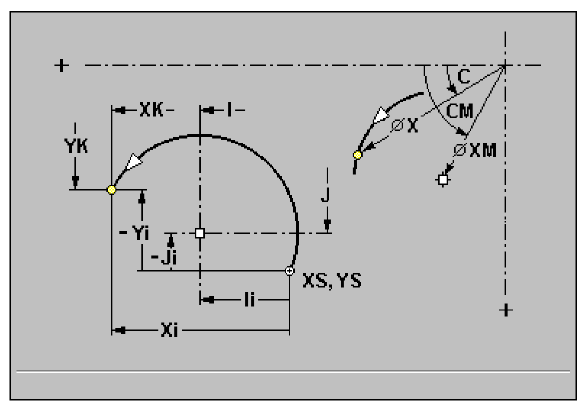
You enter the dimensions of the arc and then define the transition to the next contour element.

Parameters (for "Arc with radius," the center is not requested. For "Arc with center point," the radius is not requested.)

- **`XS, YS starting point`** (Cartesian coordinates)
- **`XD, CS starting point`** (polar coordinates)
- **`XK, YK target point`** (Cartesian coordinates)
- **`X, C target point`** (polar coordinates)
- **`I, J center point`** (Cartesian coordinates)
- **`XM, CM center point`** (polar coordinates)
- **`R radius`**
- **`F special feed`**





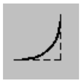


Entering chamfers/roundings on the face

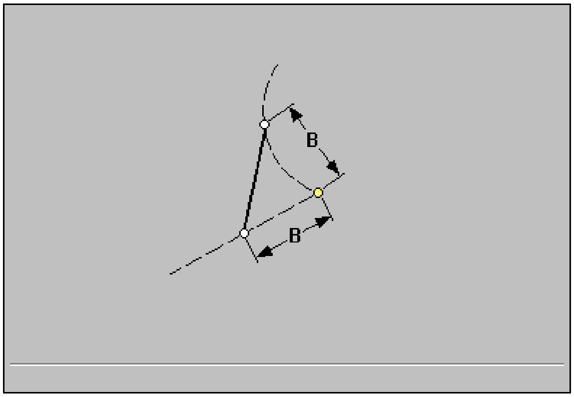
Entering a chamfer/rounding

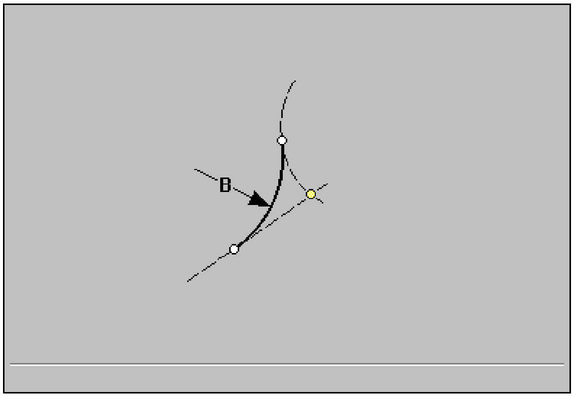
Select chamfer/rounding.

Choose chamfer.



Choose rounding.





The corner is predefined by the starting point. You need only enter the "chamfer width B" or "rounding radius B."

Parameters

- `B chamfer width`—or
- `B rounding radius`
- `F special feed`

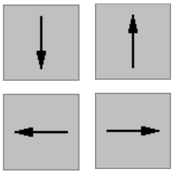
5.7 ICP Contour Elements on the Lateral Surface

You can use the linear dimension as an alternative to the angular dimension. The setting of the ***Polar*** soft key determines which type of dimensioning is active. MANUALplus distinguishes angular dimensions from linear dimensions by different address letters.

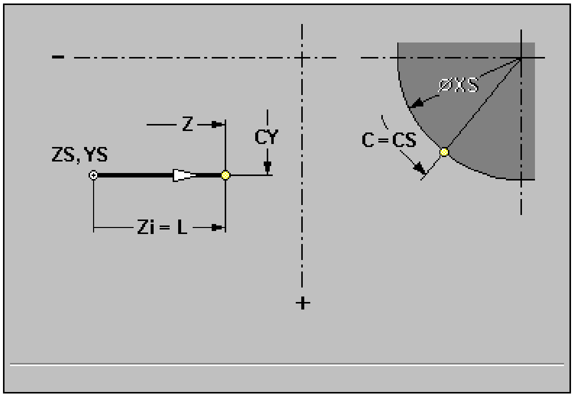
- Depending on the cycle, you can have the MANUALplus display ICP contours for the face or the lateral surface by calling the "selection of ICP contours" with the ***Contour list*** soft key.
- When defining the first element for lateral surface contours, specify **unrolled diameter XS.** The **linear dimensions** of all subsequent contour elements are referenced to this diameter.

Entering lines on the lateral surface

Vertical/horizontal lines



Select the line direction.

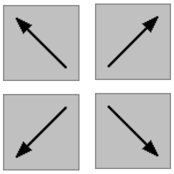


You enter the dimensions of the line and then define the transition to the next contour element.

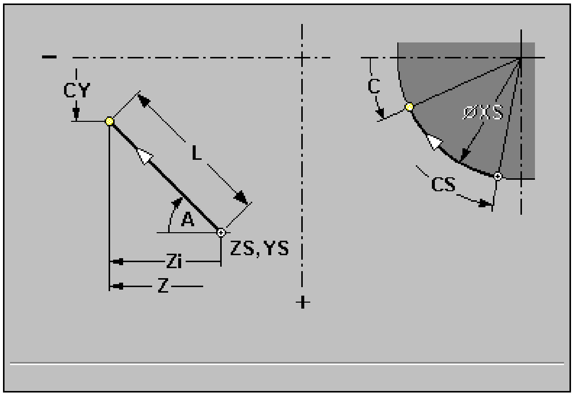
Parameters

- **ZS, YS starting point** (YS as linear dimension—reference: diameter XS)
- **CS starting point** as angular dimension
- **XS** unrolled diameter
- **Z target point**
- **CY target point** as linear dimension (reference: diameter XS)
- **C target point** as angular dimension
- **L length of line**
- **F special feed**

Line at angle



Select the line direction.



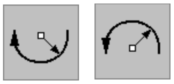
You enter the dimensions of the line and then define the transition to the next contour element.

Parameters

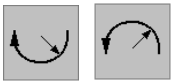
- **ZS, YS starting point** (YS as linear dimension—reference: diameter XS)
- **CS starting point** as angular dimension
- **XS** unrolled diameter
- **Z target point**
- **CY target point** as linear dimension (reference: diameter XS)
- **C target point** as angular dimension
- **A angle to Z axis**—for direction of angle, see graphic support window
- **L length of line**
- **F special feed**

Entering circular arcs on the lateral surface

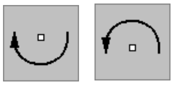
Entering an arc



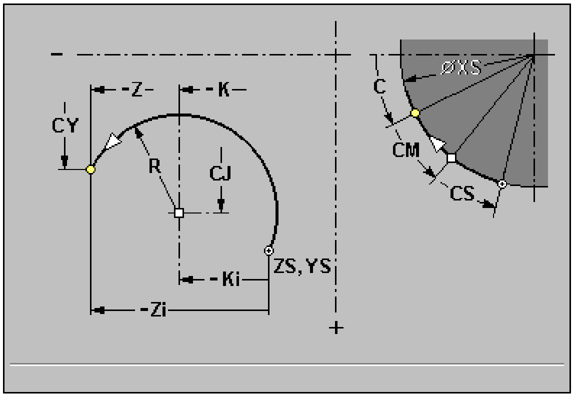
Arc with center and radius



Arc with radius



Arc with center



You enter the dimensions of the arc and then define the transition to the next contour element.

Parameters (for "Arc with radius," the center is not requested. For "Arc with center point," the radius is not requested.)

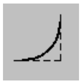
- **`ZS, YS starting point`** (YS as linear dimension—reference: diameter XS)
- **`CS starting point`** as angular dimension
- **`XS`** unrolled diameter
- **`Z target point`**
- **`CY target point`** as linear dimension (reference: diameter XS)
- **`C target point`** as angular dimension
- **`K, CJ center point`** (CJ as linear dimension—reference: diameter XS)
- **`CM center point`** as angular dimension
- **`R radius`**
- **`F special feed`**

Entering chamfers/roundings on the lateral surface

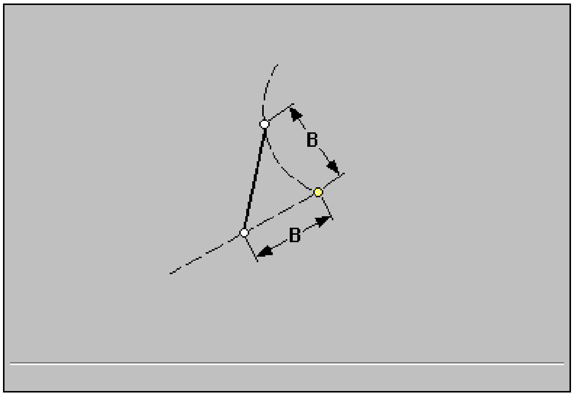
Entering chamfers/roundings

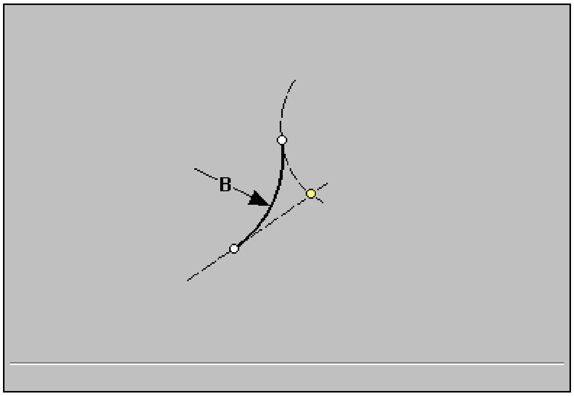
Select chamfer/rounding.

Choose chamfer.



Choose rounding.





The corner is predefined by the starting point. You need only enter the "chamfer width B" or "rounding radius B."

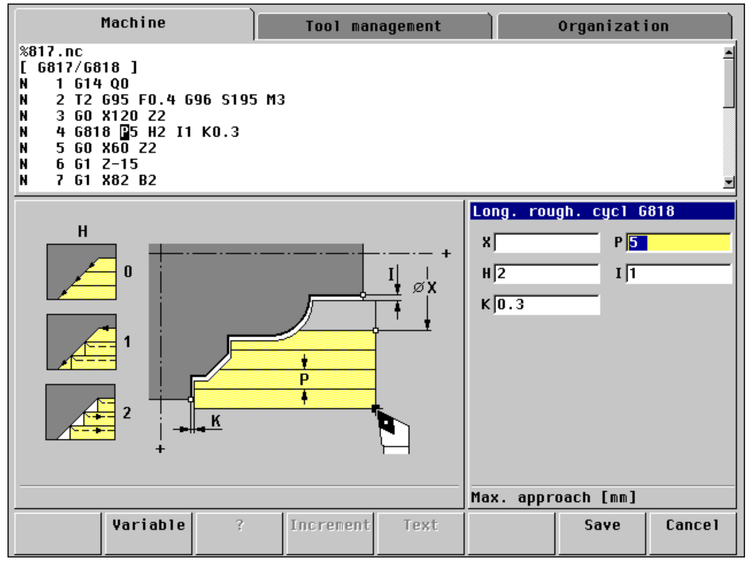
Parameters

- `B chamfer width`—or
- `B rounding radius`
- `F special feed`

6

DIN Programming

6.1 DIN Programming



The structure of programs and program blocks follows the standard DIN 66025 (ISO 6983) and is therefore called "DIN programming." MANUALplus supports DIN programs and DIN macros.

DIN programs are independent NC programs. In other words, they contain all traversing and switching commands that are necessary for producing the desired workpiece.

DIN macros are integrated into cycle programs. They are "not independent" but only define a specific machining operation within a cycle program. How you apply DIN macros depends entirely on the job at hand. DIN macros use the complete range of commands that are available for DIN programs. A distinction between programs and macros is not necessary in this chapter. They are therefore simply referred to as "DIN programs" or "NC programs."

Test-running DIN programs and DIN macros

You can test DIN programs and DIN macros with the graphic simulation function. With DIN macros, this feature is available in the cycle programming mode. With DIN programs, you switch to the "Program run" mode and call the simulation function.

Help graphics

The functionalities and parameters of the traverse commands and cycles are illustrated in the graphic support window. These graphics usually show an external machining operation. The Circle key allows you to switch to the help graphics for internal machining, and

▶ to switch between the help graphics for internal and external machining.

Notes on the elements used in the graphic support window:

- Broken line: Rapid traverse path.
- Continuous line: Feed path.
- Dimension line with arrow head on one side: "Directional dimension"—the algebraic sign defines the direction.
- Dimension line with arrow head on both sides: "Absolute dimension"—the algebraic sign is of no importance.

Program and block structure

Program structure

- Program number, starting with the character "%" followed by up to eight characters and the extension "nc" for main programs or "ncs" for subprograms.
- Program designation (definition in the second program line).
- NC blocks or comment blocks.
- The term "END" with main programs or "RETURN" with macros and subprograms.

The first and last lines of an NC program cannot be edited. The program designation can be edited, but not deleted.

The program designation can be displayed in the "program list" and edited with the "program selection" functions.

NC blocks begin with an "N" followed by a block number (with up to four digits). The block numbers do not affect the sequence in which the program blocks are executed. They are intended for identifying the individual blocks.

An NC block comprises **NC commands.** These are traversing, switching or organizational commands. Traversing and switching commands start with a letter followed by a number (such as G1, G2, G81, M3, M30) and the parameters. Organizational commands consist of "key words" (WHILE, RETURN, etc.), or of a combination of letters/numbers. The parameters are preceded by **address letters** (such as X100, Z–2, etc.).

You can program several NC commands in one block unless they have the same address letters or opposing functionalities.

Examples
Permissible combination:

N10 G1 X100 Z2 M8

Non-permissible combination:

N10 G1 X100 Z2 G2 X100 Z2 R30 (same address letters are used more than once)

or

N10 M3 M4—opposing functionality

You can also program NC blocks containing only variable calculations.

Comments are enclosed in brackets "[...]." They are located at the end of an NC block or in a separate NC block. An NC block consisting only of a comment does not have a block number.

Example: Program and block structure

```
%12345678.nc
[Beispiel – Example]
N1 G21 X80 Z110 B10 J1
N2 G1 Z-15 B-1
N3 G1 X102 B2
N4 G1 Z-22
N5 G1 X90 Zi-12 B1
N6 G1 Zi-6
N7 G1 X100 A80 B-1
N8 G1 Z-47
N9 G1 X110
N10 G80
N11 G14 Q0
N12 T3 G95 F0.25 G96 S200 M3
N13 G0 X0 Z2
N.. . . .
END
```

Overview of DIN commands

- **Traversing commands**
  For moving the slide on a linear or circular path.
- **Cycles**
  For roughing, recessing, finishing, threading, and drilling.
- **Switching commands**
  For machine components.
- **Zero point shifts**
  For adjusting the dimensional system.
- **Commands for program organization**
  Program branches, program repeats and subprograms.
- **Comments**
  For explaining the program.
- **Variable functions**
  Instead of fixed address parameters, you can use variables which are entered or calculated before execution of the program.
- **Mathematical operations**
  For calculating address parameters or variables.
- **Simplified programming**
  Values (coordinates) which have not been dimensioned in the workpiece drawing are calculated by MANUALplus, if mathematically possible.
- **Operator communication**
  Via input of variable values and output of texts and variables.

Not all of these functions are performed by NC commands (G commands, M commands, etc). Some of these functions such as "simplified programming," "variable functions" or "mathematical operations" can be used for programming with variables, working with simplified geometry, etc. Instead of a fixed value, you then enter a variable, mathematical expression or question mark "?" for the address parameter.

6.2 Editing DIN Programs

Loading the desired DIN program

Edit DIN	Call the DIN editor.
Program list	Call the program list.
DIN programs	Select DIN programs.
DIN macros	Select DIN macros.

Select a DIN program / DIN macro or define a new program number.

Select	Call the DIN program / DIN macro.

Block functions

With the arrow keys and paging keys, you move the cursor within the DIN program to the position you wish to delete, change or add to. Place the cursor at the beginning of a block, NC word, or parameter.

The soft keys or menu keys enable you to select the desired function. MANUALplus then asks you to enter additional parameters. When you are programming cycles or traversing commands (G functions), the function and the parameters are illustrated in the graphic support window.

Place the cursor at the beginning of the block to select **block functions.**

Soft keys for block functions	
Insert block	Insert a new NC block with the "next" block number below the highlighted block.
Extend block	The "Function selection" menu can be used to add further NC commands.
Delete block	The block at which the cursor is located is deleted.
Change block no.	The number of the block at which the cursor is located can now be edited.

Changing block numbers

Position the cursor on the NC block.

Change block no.

Press ***Change block no.***

Enter the new block number.

Save

Transfer the new block number.

Renumbering blocks

Position the cursor on any NC block.

Change block no.

Press ***Change block no.***

Renumber

Press ***Renumber.***

Define the block number increment.

Renumber

Press ***Renumber*** again.

The block number increment you defined is also effective for automatic block numbering.

Word functions

The functions ("Delete word," "Change word," etc.) refer to the "word" at which the cursor is located. What is actually deleted or changed depends on the meaning of the "word." Examples:

- The cursor is located on a **G command.**
  - ***Change word:*** First the command and then the associated parameters can be edited.
  - ***Delete word:*** The command and the associated parameters are deleted.
- The cursor is located on an **address letter** of a parameter.
  - ***Change word:*** All parameters of the function can be edited.
  - ***Delete word:*** Only the parameter concerned is deleted.

An NC command can be changed within "a group." You can, for instance, change the number of a G command, but you cannot change the G command into an M command. The editor deletes the parameters that are no longer required after a change.

Soft keys for word functions	
Delete word	The NC command or parameter (word) at which the cursor is located is deleted.
Change word	The NC command or parameter (word) at which the cursor is located can now be edited.
Extend block	The "Function selection" menu can be used to add further NC commands.

Address parameters

Enter the address parameters as follows:

- **Absolute dimension:** The dimension is referenced to the workpiece zero point.
- **Incremental dimension:** The dimension is referenced to the last active coordinate.
- **Variable:** The value of the variable or the result of the mathematical expression represents the value of the address parameter.
- **Simplified geometry programming:** This coordinate is calculated by MANUALplus—if mathematically possible.

The input data that are supported by the individual address parameters depend on the meaning of this parameter. MANUALplus inhibits all functions that are not permissible.

"Absolute dimensions" is the default setting for the input fields. If you wish to define "incremental dimensions," simply press ***Increment.*** The address letter receives the attribute "i" (for example, "Zi"). The attribute is transferred to the DIN program. The selected absolute or incremental setting is only effective for the **currently active** input field.

To enter variables, press the ***Variable*** soft key (see “Variables as Address Parameters” on page 403).

With the ***?*** soft key you inform MANUALplus that this coordinate is to be calculated. The character ***?*** is transferred to the DIN program.

Comments

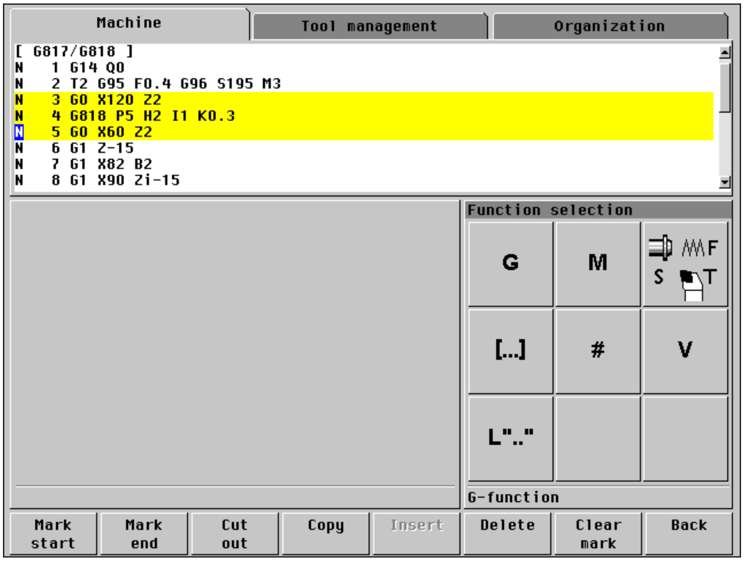
If you enter a comment in an empty block, the block number is deleted and only the comment is stored in this block. (An "empty block" is a block that consists of the block number only.) If the NC block already contains NC commands, the comment is appended to the end of the block.

To change comments, place the cursor at the beginning of the comment and press ***Change word.*** MANUALplus then displays the "alphanumeric keyboard" and the current text of the comment. You can now edit the comment as desired by changing, adding to, etc.

If you wish to delete comments, place the cursor at the beginning of the comment and press ***Delete block*** or ***Delete word,*** as applicable. The comment is deleted.

Block functions

Mark several successive NC blocks (block sequence) to be able to cut, copy or delete them. If you cut or copy the block sequence, it is taken into the clipboard. You can then insert this block sequence at a different position in the program, or call a different DIN program and insert the block sequence there. The block sequence remains stored in the clipboard until it is overwritten or MANUALplus is switched off.



Copying and pasting blocks

Position the cursor on the beginning of a block.

Block function Call the block functions.

Mark start Mark the beginning of the block.

Position the cursor on the end of a block.

Mark end Mark the end of the block.

Copy Copy the block and transfer it to the clipboard.

Cut out Cut the block out and transfer it to the clipboard.

Load a new DIN program (if required).

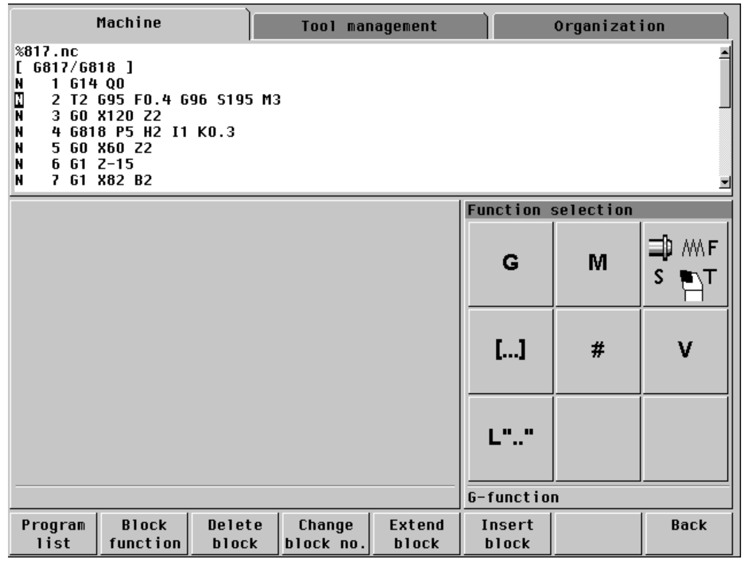
Place the cursor on the position where the block is to be inserted.

Insert Transfer the block sequence from the clipboard (the NC blocks are inserted below the cursor).

Menu structure

Select the function group by menu key.

- G and M functions: The function number and further parameters that vary depending on the function are entered subsequently.
- Comment, subprogram and T, S, F: The required parameters are entered subsequently.
- Variable functions: MANUALplus switches to other menus for entering further data.



DIN functions	Menu key
G function Traversing commands, cycles, and other G commands.	G
M function Switching functions for machine components and program control functions (see “M Functions” on page 408).	M
Machine data Entry of F, S, T (see “Set T, S, F” on page 392).	F S T
Comment Entry of comments (see “Editing DIN Programs” on page 281).	[...]
Program variable functions Switch to the "Program variable menu" (see “Programming Variables” on page 396).	#
Machine variable functions Switch to the "Machine variable menu" (intended for special cases and of no importance to the DIN programmer).	V
Subprogram call Program a subprogram call (see “Subprograms” on page 406).	L".."

Programming G functions

Direct programming of a G function

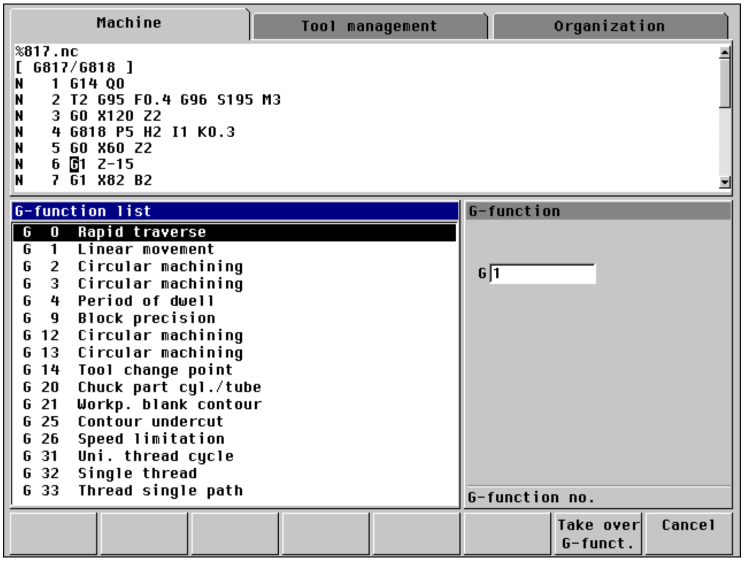
G — Select "G function."

Enter the G number.

Select — Call the G function.

Enter the parameters.

Save — Transfer the G function.



If you do not know the number of the G function, you can select it from the list of G functions.

Selecting a G function

G — Select "G function."

G-funct. list — Call the "G function list."

Select the G function.

Take over G-funct. — Transfer the G function.

Select — Call the G functions.

Enter the parameters.

Save — Transfer the G function.

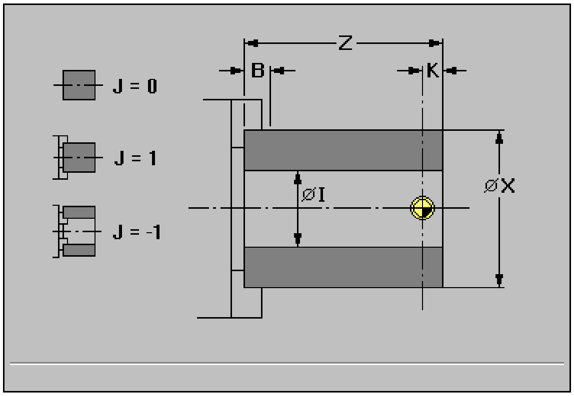
6.3 Definition of Workpiece Blank

Chuck part, cylinder/tube G20

G20 describes the workpiece blank and the setup used. This information is evaluated during the simulation.

Parameters

- **X diameter**
- **Z length** (including transverse allowance and clamping range)
- **K right edge** (transverse allowance)
- **I inside diameter** for workpiece blank "tube"
- **B clamping range**
- **J type of clamping**
  - 0: Not clamped
  - 1: Externally clamped
  - 2: Internally clamped



Example: G20

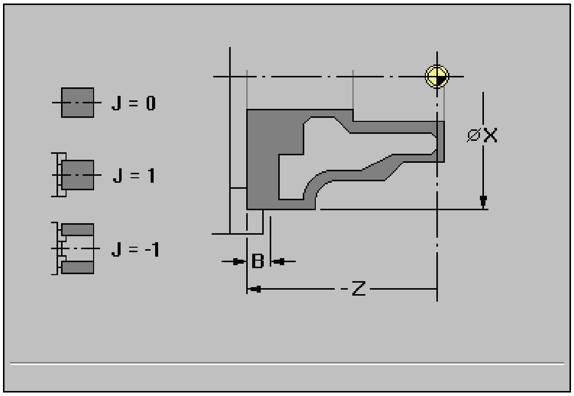
```
%20.nc
[G20]
N1 G20 X80 Z100 K2 B10 J1
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
N.. . . .
END
```

Workpiece blank contour G21

G21 describes the setup used. The workpiece blank is described with G1, G2, G3, G12 and G13 commands that follow immediately after G21. G80 concludes the contour description. This information is evaluated during the simulation.

Parameters

- **X diameter**
- **Z length**
- **B clamping range**
- **J type of clamping**
  - 0: Not clamped
  - 1: Externally clamped
  - 2: Internally clamped



Example: G21

```
%21.nc
[G21]
N1 G21 X80 Z110 B10 J1
N2 G1 Z-15 B-1
N3 G1 X102 B2
N4 G1 Z-22
N5 G1 X90 Zi-12 B1
N6 G1 Zi-6
N7 G1 X100 A80 B-1
N8 G1 Z-47
N9 G1 X110
N10 G80
N11 T3 G95 F0.25 G96 S200 M3
N12 G0 X0 Z2
N.. . . .
END
```

6.4 Tool Positioning without Machining

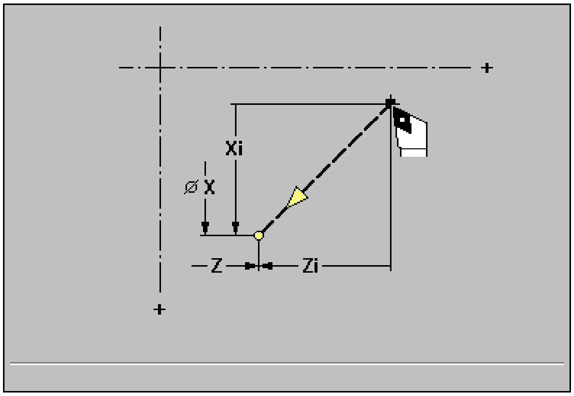
Rapid traverse G0

Geometry command: G0 defines the starting point of contour definition.

Machining command: The tool moves at rapid traverse along the shortest path to the target point X, Z. Rapid traverse paths can be executed when the spindle is stationary.

Parameters

- **X target point** (diameter value)
- **Z target point**



Example: G0

```
%0.nc
[G0]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G819 P5 I1 K0.3
N4 G0 X80 Z2
N5 G1 Z-15 B-1
N6 G1 X102 B2
N7  G1 Z-22
N8 G1 X90 Zi-12 B1
N9 G1 Zi-6
N10 G1 X100 A80 B-1
N11 G1 Z-47
N12 G1 X120
N13 G80
END
```

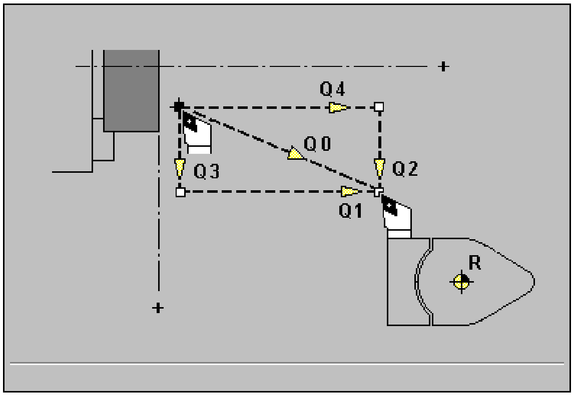
Tool change point G14

The slide moves at rapid traverse to the tool change position. In setup mode, define permanent coordinates for the tool change point (see “Defining the tool change position” on page 52).

Parameters

- **Q sequence** (default: 0): Determines the sequence of traverse.
  - Q=0: Diagonal path of traverse
  - Q=1: First X, then Z direction
  - Q=2: First Z, then X direction
  - Q=3: Only X direction, Z remains unchanged
  - Q=4: Only Z direction, X remains unchanged

G14 is converted to the basic commands "Rapid traverse to machine coordinates G701." With G701, "target point X, Z" is referenced to the machine zero point. The slide is referenced to the slide reference point.



Example: G14

```
%14.nc
[G14]
N1 G14 Q0
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
N.. . . .
END
```

6.5 Simple Linear and Circular Movements

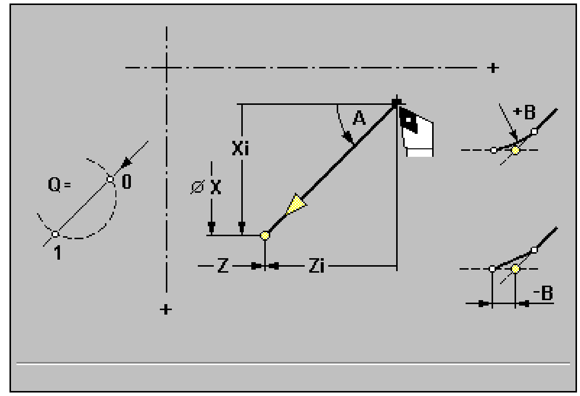
Linear path G1

Geometry command: G1 defines a linear segment in a contour.

Machining command: The tool moves on a linear path at feed rate to the end point X, Z.

Parameters

- **X end point** (diameter value)
- **Z end point**
- **A angle**—for angle direction, see graphic support window.
- **B chamfer/rounding:** At the end of the linear path you can program a chamfer/rounding or a tangential transition to the next contour element.
  - No entry: Tangential transition
  - B=0: No tangential transition
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **E special feed rate** for chamfer/rounding—default: Active feed rate
- **Q point of intersection** (default: Q=0): Specifies the end point if two solutions are possible (see graphic support window).



Example: G1

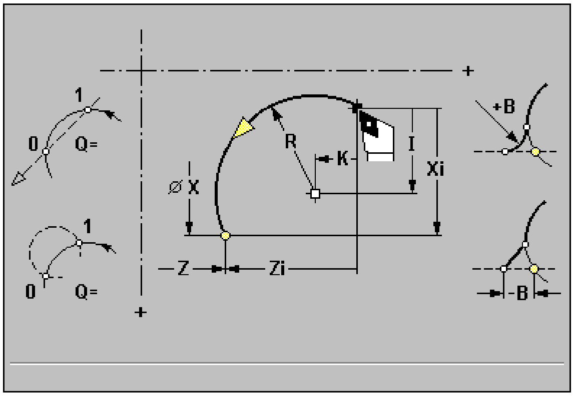
```
%1.nc
[G1]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X20 B-0.5
N6  G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G40
END
```

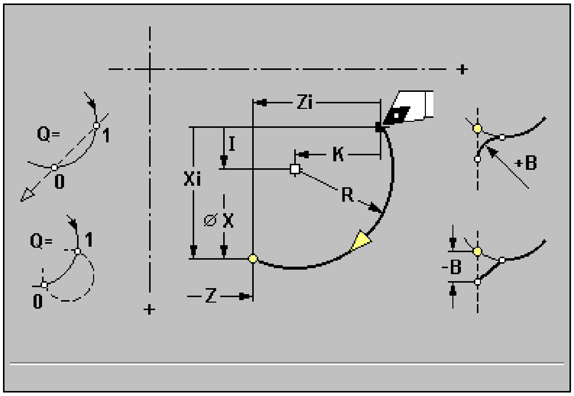
Circular path G2, G3—incremental center coordinates

Geometry command: G2/G3 defines a circular arc in a contour.

Machining command: The tool moves on a circular arc at feed rate to the end point.

The direction of rotation is shown in the graphic support window.





Parameters G2, G3

- **X end point** (diameter value)
- **Z end point**
- **R radius**
- **I center point** incremental—(distance from starting point to center point; diameter value)
- **K center point** incremental—(distance from starting point to center point)
- **Q point of intersection** (default: Q=0): Specifies the end point if two solutions are possible (see graphic support window).
- **B chamfer/rounding:** At the end of the circular arc you can program a chamfer/rounding or a tangential transition to the next contour element.
  - No entry: Tangential transition
  - B=0: No tangential transition
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **E special feed rate** for chamfer/rounding—default: Active feed rate

- If you do not program the center, MANUALplus automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.
- The direction of rotation of G2/G3 is shown in the graphic support window.

Example: G2, G3

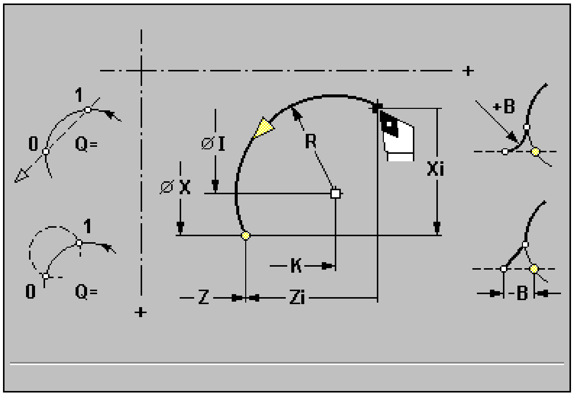
```
%2.nc
[G2, G3]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X15 B-0.5 E0.05
N6 G1 Z-25 B0
N7 G2 X45 Z-32 R36 B2
N8 G1 A0
N9 G2 X80 Z-80 R20 B5
N10 G1 Z-95 B0
N11 G3 X80 Z-135 R40 B0
N12 G1 Z-140
N13 G1 X82 G40
END
```

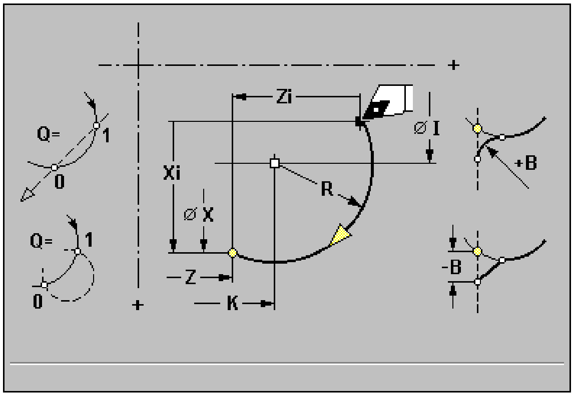
Circular path G12, G13—absolute center coordinates

Geometry command: G12/G13 defines a circular arc in a contour.

Machining command: The tool moves on a circular arc at feed rate to the end point.

The direction of rotation is shown in the graphic support window.





Parameters G12, G13

- **X end point** (diameter value)
- **Z end point**
- **R radius**
- **I center point** absolute—(diameter value)
- **K center point** absolute
- **Q point of intersection** (default: Q=0): Specifies the end point if two solutions are possible (see graphic support window).
- **B chamfer/rounding:** At the end of the circular arc you can program a chamfer/rounding or a tangential transition to the next contour element.
  - No entry: Tangential transition
  - B=0: No tangential transition
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **E special feed rate** for chamfer/rounding—default: Active feed rate

- If you do not program the center, MANUALplus automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.
- The direction of rotation of G12/G13 is shown in the graphic support window.

Example: G12, G13

```
%12.nc
[G12, G13]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X20
N6 G1 A0 B0 Q1
N7 G12 X50 R22 I60 K-5 B2
N8 G1 A0 B2
N9 G13 X60 R17.5 I30 K-55 B2
N10 G1 A0 B2
N11 G13 X80 R48 I-10 K-107 Q1 B0
N12 G1 Z-107
N13 G1 X82 G40
END
```

6.6 Feed Rate and Spindle Speed

Speed limitation G26/G126

G26: Speed limitation for spindle

G126: Speed limitation for spindle 1 (driven tool)

The speed limit remains in effect until a new value is programmed for G26/G126.

Parameters

- **S speed:** Maximum speed

- The speed limitation remains in effect even after concluding the DIN program and exiting "Program run" mode. You can define a new speed limit in the "F, S, T menu" or via parameters.
- If the speed programmed with G26/G126 is greater than the speed set in the machine parameter "General parameters for spindle—Absolute max. speed," then the speed limit of this parameter takes effect.

Example: G26, G126

```
%26.nc
[G26, G126]
N1 G14 Q0
N1 G26 S2000
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
N4 . . .
END
```

Interrupted feed G64

G64 interrupts the programmed feed for a short period of time. This function is used to ensure continuous chip breaking.

G64 without parameters deactivates the interrupted (intermittent) feed rate.

Parameters

- **E pause duration:** Range: 0.01s < E < 999s
- **F feed duration:** The slide accelerates to the programmed feed rate and decelerates again to "zero feed" at the end of the period. Range: 0.01s < E < 999s

Example: G64

```
%64.nc
[G64]
N1 T3 G95 F0.25 G96 S200 M3
N2 G64 E0.1 F1
N3 G0 X0 Z2
N4 G42
N5 G1 Z0
N6 G1 X20 B-0.5
N7  G1 Z-12
N8 G1 Z-24 A20
N9 G1 X48 B6
N10 G1 Z-52 B8
N11 G1 X80 B4 E0.08
N12 G1 Z-60
N13 G1 X82 G40
N14 G64
END
```

Feed per tooth G193

G193 defines the feed rate with respect to the number of teeth of the cutter.

Parameters

- **F feed per tooth** in mm/tooth or inch/tooth

The actual value display shows the feed rate in mm/rev.

Example: G193

```
%193.nc
[G193]
N1 M5
N2 T71 G197 S1010 G193 F0.08 M104
N3 M14
N4 G152 C30
N5  G110 C0
N6 G0 X122 Z-50
N7 G744 X122 Z-50 ZE-50 C0 Wi90 Q4
N8 G792 K30 A0 X100 J11 P5 F0.15
N9 M15
END
```

Constant feed G94 (feed per minute)

G94 defines the feed rate independent of drive.

Parameters

- **F feed per minute** in mm/min or inch/min

Example: G94

```
%94.nc
[G94]
N1 G14 Q0
N2 T3 G94 F2000 G97 S1000 M3
N3 G0 X100 Z2
N4 G1 Z-50
N5 . . .
END
```

Feed per revolution G95/G195

G95/G195 defines the feed rate as a function of drive.

G95: Reference—No. of revolutions of spindle

G195: Reference—No. of revolutions of spindle 1 (driven tool)

Parameters

- **F feed per revolution** in mm/rev or inch/rev

Example: G95, G195

```
%95.nc
[G95, G195]
N1 G14 Q0
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
N7 . . .
END
```

Constant cutting speed G96/G196

G96/G196 defines a constant cutting speed.

G96: The speed of the spindle depends on the X position of the tool tip.

G196: The spindle speed depends on the diameter of the tool.

Parameters

- **S cutting speed** in m/min or ft/min

Example: G96, G196

```
%96.nc
[G96, G196]
N1 T3 G195 F0.25 G196 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X20 B-0.5
N6  G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G40
END
```

Spindle speed G97/G197

G97/G197 defines a constant spindle speed.

G97: For the **spindle**

G197: For **spindle 1** (driven tool)

Parameters

- **S speed** in revolutions per minute

Example: G97, G197

```
%97.nc
[G97, G197]
N1 G14 Q0
N2 T3 G95 F0.25 G97 S1000 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
N7 . . .
END
```

6.7 Tool-Tip / Milling-Cutter Radius Compensation

Fundamentals

Tool-tip radius compensation (TRC)
If TRC is not used, the theoretical tool tip is the reference point for the paths of traverse. This might lead to inaccuracies when the tool moves along non-paraxial paths of traverse. The TRC function corrects programmed paths of traverse (see “Tool-tip radius compensation (TRC)” on page 28).

With Q=0, the TRC reduces the feed rate at arcs (G2, G3, G12, G13) and rounding arcs if the "shifted radius < original radius." The "special feed rate" is corrected when a rounding as transition to the next contour element is machined.

Reduced feed rate = feed rate * (offset radius / original radius)

Milling cutter radius compensation (MCRC)
When the MCRC function is not active, the system defines the center of the cutter as the zero point for the paths of traverse. With the MCRC function, MANUALplus accounts for the outside cutting radius when moving along the programmed paths of traverse (see “Milling cutter radius compensation (MCRC)” on page 29).

Recessing, roughing and milling cycles already include TRC/MCRC calls. You must therefore ensure that TRC/MCRC is disabled before you call these cycles. There are a few exceptions to this rule that will be described where concerned.

- If "tool radii > contour radii," the TRC/MCRC might cause abrasive cuts. **Recommendation:** Use the finishing cycle G89 / milling cycles G793/G794.
- Never select MCRC during a perpendicular approach to the machining plane.
- Note on calling subroutines: Switch the TRC/MCRC off
  - in the subprogram in which it was switched on.
  - in the main program if it was switched on there.

Function of the TRC/MCRC

N.. . . .
N.. G0 X10 Z10
N.. G41 G0 Z20 [Path of traverse: from X10/Z10 to X10+TRC/ X20+TRC]
N.. G1 X20 [The path of traverse is "shifted" by the TRC]
N.. G40 G0 X30 Z30 [Path of traverse: from X20+TRC/ Z20+TRC to X30/X30]
N.. . . .

G40: Switch off TRC/MCRC

- The TRC/MCRC remains in effect until a block with G40 is reached.
- The block containing G40, or the block after G40 only permits a linear path of traverse (G14 is not permissible).

G41/G42: Switch on TRC/MCRC

- A straight line segment (G0/G1) must be programmed in the block containing G41/G42 or after the block containing G41/G42.
- The TRC/MCRC is taken into account from the next path of traverse.

G41: Internal machining (with traverse in negative Z direction)—compensation of the tool-tip / cutter radius to the left of the contour in traverse direction.

G42: External machining (with traverse in negative Z direction)—compensation of the tool-tip / cutter radius to the right of the contour in traverse direction.

Parameters

- **Q plane** (default: 0)
  - Q=0: TRC on the turning plane (XZ plane)
  - Q=1: MCRC on the face (XC plane)
  - Q=2: MCRC on the lateral surface (ZC plane)
- **H output** (default: 0)
  - H=0: Intersecting areas which are programmed in directly successive contour elements are not machined.
  - H=1: The complete contour is machined—even if certain areas are intersecting.
- **O feed rate reduction** (default: 0)
  - O=0: Feed rate reduction active
  - O=1: No feed rate reduction

Example: G40, G41, G42

```
%40.nc
[G40, G41, G42]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X20 B-0.5
N6  G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G40
END
```

6.8 Compensation Values

(Changing the) cutter compensation G148

MANUALplus manages three wear compensation values for recessing tools (DX, DZ, and DS). The parameter "O" allows you to define which wear compensation values are to be taken into account.

DX, DZ become effective after program start and after a T command (G148 O0). The compensation values defined with G148 remain in effect until the next T command or the end of the program.

Before you can use G148, you must assign the compensation values DX, DZ and DS to the cutting edges of the recessing tool tip with the "tool orientation" function (see “Recessing and recess-turning tools” on page 421).

Parameters

- **O selection** (default: 0)
  - O=0: DX, DZ active—DS inactive
  - O=1: DS, DZ active—DX inactive
  - O=2: DX, DS active—DZ inactive

> The recessing cycles G861 to G868 automatically take the "correct" wear compensation into account.

Example: G148

```
%148.nc
[G148]
N1 T31 G95 F0.25 G96 S160 M3
N2 G0 X62 Z2
N3 G0 Z-29.8
N4 G1 X50.4
N5 G0 X62
N6 G150
N7  G1 Z-20.2
N8 G1 X50.4
N9 G0 X62
N10 G151 [Recessing finishing]
N11 G148 O0
N12 G0 X62 Z-30
N13 G1 X50
N14 G0 X62
N15 G150
N16 G148 O2
N17 G1 Z-20
N18 G1 X50
N19 G0 X62
END
```

Additive compensation G149

MANUALplus manages 16 tool-independent compensation values, which are assigned the designations D901 to D916. These compensation values are added to the active wear compensation values of the tools.

Additive compensations are effective from the block in which they are programmed with G149 and remain in effect up to the

- Next "G149 D900"
- Next tool change
- Program end

Parameters

- **D additive compensation** (default: D900):
  - D900: deactivates the additive compensation
  - D901 to D916: activates the additive compensation

Example: G149

```
%149.nc
[G149]
N1 T3 G96 S200 G95 F0.4 M4
N2 G0 X62 Z2
N3 G89
N4 G42
N5 G0 X27 Z0
N6 G1 X30 Z-1.5
N7  G1 Z-25
N8 G149 D901
N9 G1 X40 B-1
N10 G1 Z-50
N11 G149 D902
N12 G1 X50 B-1
N13 G1 Z-75
N14 G149 D900
N15 G1 X60 B-1
N16 G1 Z-80
N17 G1 X62
N18 G80
END
```

Compensation of right-hand tool nose G150
Compensation of left-hand tool nose G151

With recessing tools, the "tool orientation" function defines whether the tool reference point is set at the left or the right side of the tool tip (see “Recessing and recess-turning tools” on page 421). G150/G151 switches the reference point.

G150: Reference point on right tip

G151: Reference point on left tip

G150/G151 is effective from the block in which it is programmed and remains in effect up to the

- Next tool change
- Program end

Example: G150, G151

```
%148.nc
[G148]
N1 T31 G95 F0.25 G96 S160 M3
N2 G0 X62 Z2
N3 G0 Z-29.8
N4 G1 X50.4
N5 G0 X62
N6 G150
N7  G1 Z-20.2
N8 G1 X50.4
N9 G0 X62
N10 G151 [Recessing finishing]
N11 G148 O0
N12 G0 X62 Z-30
N13 G1 X50
N14 G0 X62
N15 G150
N16 G148 O2
N17 G1 Z-20
N18 G1 X50
N19 G0 X62
END
```

6.9 Zero Point Shifts

Zero point shift G51

G51 shifts the workpiece zero point by "Z" (or "X"). The shift is referenced to the workpiece zero point defined in setup mode (see “Defining the workpiece zero point” on page 50).

Even if you shift the zero point several times with G51, it is still always referenced to the workpiece zero point defined in setup mode.

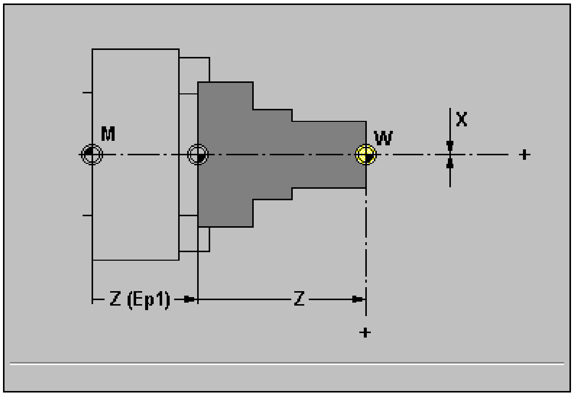
The workpiece zero point defined with G51 remains in effect up to the end of the program, or until it is canceled by another zero point shift.

Parameters

- `X shift` (diameter value)
- `Z shift`

Danger of collision!

Cycle programming: With DIN macros, the zero point shift is reset at the end of the cycle. Therefore, do not use any DIN macros with zero point shifts in cycle programming.



Example: G51

```
%51.nc
[G51]
N1 T30 G95 F0.25 G96 S200 M3
N2 G0 X62 Z-15
N3 G862 Q0
N4 G0 X60 Z-19.2327
N5 G3 X58.5176 Z-20.1986 R1 I-1 K0
N6 G1 X48 Z-21.6077 B1
N7 G1 Z-28.3923 B1
N8 G1 X58.5176 Z-29.8014
N9 G3 X60 Z-30.7673 R1 I-0.2588 K-0.9659
N10 G80
N11 G51 Z-28
N12 G0 X62 Z-15
N13 G862 Q0
N14 G0 X60 Z-19.2327
N.. . . .
N.. G80
N.. G51 Z-56
N.. . . .
END
```

Additive zero point shift G56

G56 shifts the workpiece zero point by "Z" (or "X"). The shift is referenced to the currently active workpiece zero point.

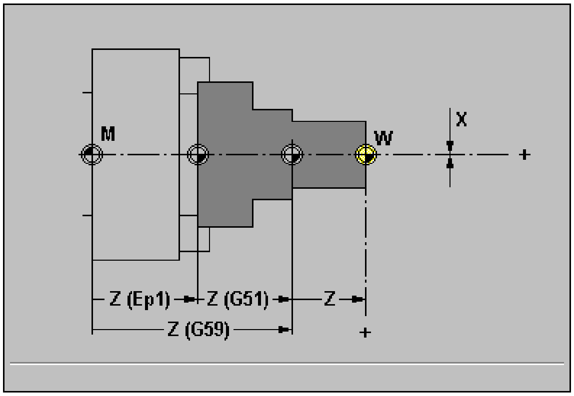
If you shift the workpiece zero point more than once with G56, the shift is always added to the currently active zero point.

Parameters

- **X shift** (diameter value)
- **Z shift**

> G51 or G59 cancel additive zero point shifts.

> **Danger of collision!**
>
> **Cycle programming:** With DIN macros, the zero point shift is reset at the end of the cycle. Therefore, do not use any DIN macros with zero point shifts in cycle programming.



Example: G56

```
%56.nc
[G56]
N1 T30 G95 F0.25 G96 S200 M3
N2 G0 X62 Z-15
N3 G862 Q0
N4 G0 X60 Z-19.2327
N5 G3 X58.5176 Z-20.1986 R1 I-1 K0
N6 G1 X48 Z-21.6077 B1
N7 G1 Z-28.3923 B1
N8 G1 X58.5176 Z-29.8014
N9 G3 X60 Z-30.7673 R1 I-0.2588 K-0.9659
N10 G80
N11 G56 Z-28
N12 G0 X62 Z-15
N13 G862 Q0
N14 G0 X60 Z-19.2327
N.. . . .
N.. G80
N.. G56 Z-28
N.. . . .
END
```

Absolute zero point shift G59

G59 sets the workpiece zero point to the position "X, Z." The new zero point remains in effect to the end of the program.

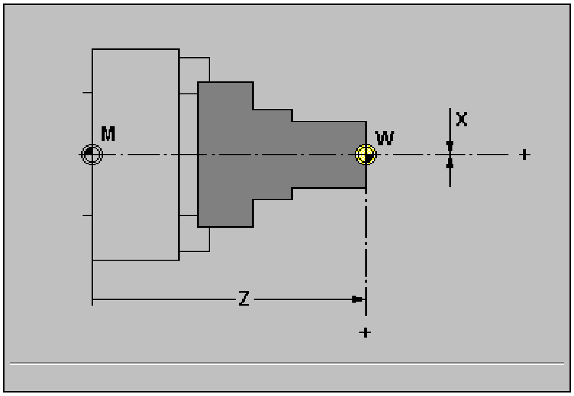
Parameters

- **`X zero point shift`** (diameter value)
- **`Z zero point shift`**

G59 cancels all previous zero point shifts (with G51, G56 or G59).

Danger of collision!

Cycle programming: With DIN macros, the zero point shift is reset at the end of the cycle. Therefore, do not use any DIN macros with zero point shifts in cycle programming.



Example: G59

```
%59.nc
[G59]
N1 G59 Z256
N2 G14 Q0
N3 T3 G95 F0.25 G96 S200 M3
N4 G0 X62 Z2
N5 . . .
END
```

6.10 Oversizes

Axis-parallel oversize G57

G57 defines different oversizes for X and Z. G57 is programmed before the recessing or roughing cycle.

Parameters

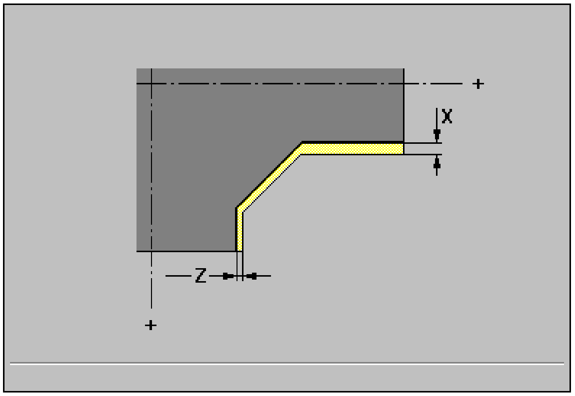
- `X oversize X` (diameter value)
- `Z oversize Z`

The following **cycles** take the oversizes into account:

- Roughing cycles: G81, G817, G818, G819, G82, G827, G828, G829, G83
- Recessing cycles: G86x
- Recess turning cycles: G81x, G82x

The cycles G81, G82 and G83 do **not** cancel the oversizes after execution of the cycle.

> If the oversizes are programmed with G57 **and** in the cycle itself, the cycle oversizes apply.



Example: G57

```
%57.nc
[G57]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G57 X0.2 Z0.5
N4 G819 P5
N5 G0 X80 Z2
N6 G1 Z-15 B-1
N7 G1 X102 B2
N8 G1 Z-22
N9 G1 X90 Zi-12 B1
N10 G1 Zi-6
N11 G1 X100 A80 B-1
N12 G1 Z-47
N13 G1 X120
N14 G80
END
```

Contour-parallel oversize (equidistant) G58

G58 defines a contour-parallel oversize. G58 is programmed before recessing or roughing cycles.

Parameters

- **P oversize**

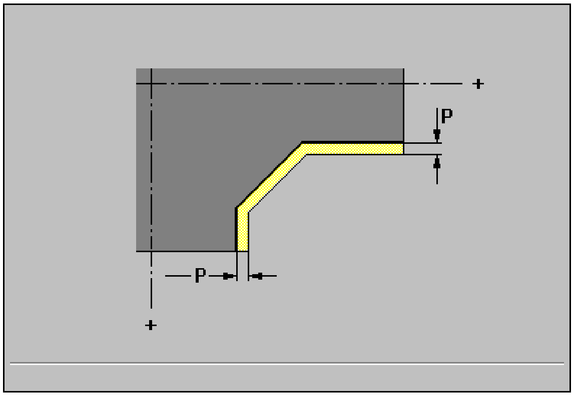
A negative oversize is permitted with the cycle G89.

The following **cycles** take the oversizes into account:

- Roughing cycles: G817, G818, G819, G827, G828, G829, G83
- Recessing cycles: G86x
- Recess turning cycles: G81x, G82x

The cycle G83 does **not** cancel the oversizes after execution of the cycle.

> If an oversize is programmed with G58 **and** in the cycle, the oversize from the cycle is used.



Example: G58

```
%58.nc
[G58]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G58 P2
N4 G819 P5
N5 G0 X80 Z2
N6 G1 Z-15 B-1
N7 G1 X102 B2
N8 G1 Z-22
N9 G1 X90 Zi-12 B1
N10 G1 Zi-6
N11 G1 X100 A80 B-1
N12 G1 Z-47
N13 G1 X120
N14 G80
END
```

6.11 Contour-Based Turning Cycles

Contour definition

For contour-based cycles (turning / recessing / recess turning cycles), the cycle call is followed by the contour definition:

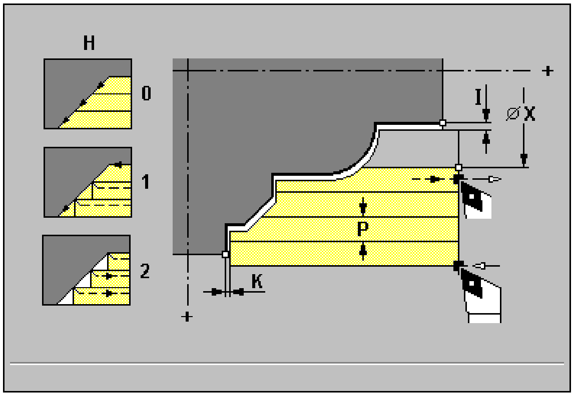
- G0 defines the starting point of the contour section.
- The contour section is described with G1, G2, G3, G12 and G13 commands.
- G80 concludes the contour definition.

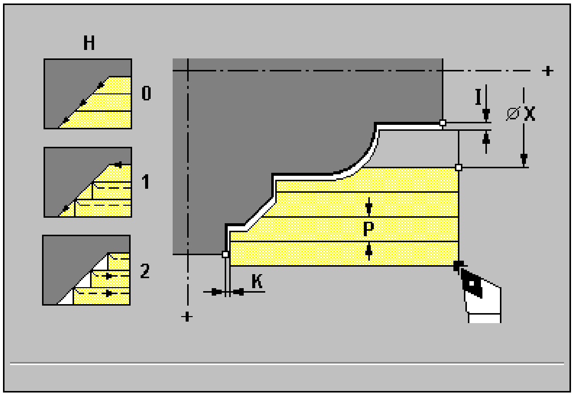
End of cycle G80

G80 concludes the contour definition after roughing, recessing and undercut cycles. A block with G80 must not contain any other commands.

Longitudinal contour roughing G817/G818

The cycles machine the contour area described by the current tool position and the data defined in the subsequent blocks in longitudinal direction **without** recessing (see "Contour definition" on page 310).





Parameters G817, G818

- **X cutting limit** (diameter value): The control machines up to the cutting limit.
- **P maximum infeed:** The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the infeed distance is <= P.
- **H type of departure** (default: 1):
  - 0: Machine contour outline after each pass
  - 1: Retract at 45°; machine contour outline after last pass
  - 2: Retract at 45°; do not machine contour outline
- **I oversize X** (diameter value)—(default: 0)
- **K oversize Z** (default: 0)

Note on the execution of the cycle:

- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point / end point of the contour area.
- Tool position at the end of the cycle:
  - G817: Cycle starting point Z; last retraction diameter X
  - G818: Cycle starting point

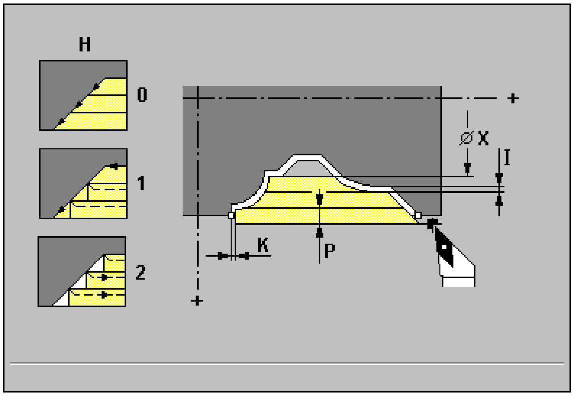
- Descending contour elements are not machined.
- The tool must be located outside the defined contour area.
- **Cutting radius compensation:** Active.
- G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.
- **Safety clearance** after each step: Parameter "Current parameters—Machining—Safety distances."

Example: G817, G818

```
%817.nc
[G817, G818]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G818 P5 H2 I1 K0.3
N4 G0 X60 Z2
N5 G1 Z-15
N6 G1 X82 B2
N7 G1 X90 Zi-15
N8 G80
N9 G0 X120 Z-28
N10 G817 X90 P4 H0 I1 K0.3
N11 G0 X90 Z-28
N12 G1 Z-45 B-3
N13 G1 X102 B2
N14 G1 X120 A30
N15 G80
END
```

Longitudinal contour roughing with recessing G819

The cycle machines the contour area described by the current tool position and the data defined in the subsequent blocks in a longitudinal direction **with** recessing (see "Contour definition" on page 310).



Parameters

- **X cutting limit** (diameter value): The control machines up to the cutting limit.
- **P maximum infeed:** The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the infeed distance is <= P.
- **E plunge feed:** The tool enters the material at the plunge feed E.
  - E=0: Descending contours are not machined.
  - No entry: The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%).
- **H type of departure** (default: 1):
  - H=0: Machine contour outline after each pass.
  - H=1: Retract at 45°; machine contour outline after last pass.
  - H=2: Retract at 45°; do not machine contour outline.
- **I oversize X** (diameter value)—(default: 0)
- **K oversize Z** (default: 0)

Note on the execution of the cycle:

- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point / end point of the contour area.
- Tool position at the end of the cycle: Cycle starting point

> - The tool must be located outside the defined contour area.
> - **Cutting radius compensation:** Active.
> - G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.
> - **Safety clearance** after each step: Parameter "Current parameters—Machining—Safety distances."

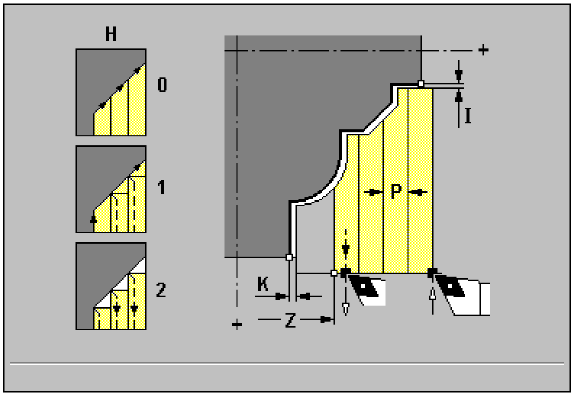
> **Danger of collision!**
>
> If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.

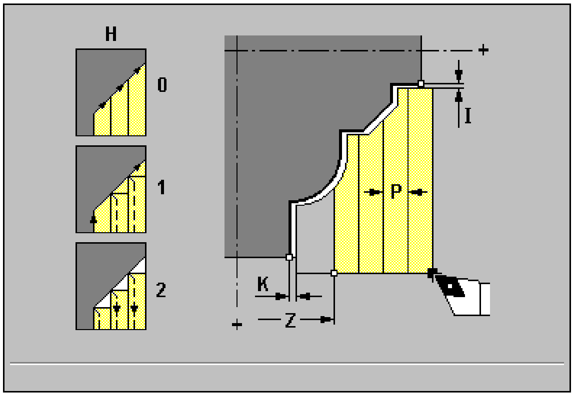
Example: G819

```
%819.nc
[G819]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G819 P5 I1 K0.3
N4 G0 X80 Z2
N5 G1 Z-15 B-1
N6 G1 X102 B2
N7  G1 Z-22
N8 G1 X90 Zi-12 B1
N9 G1 Zi-6
N10 G1 X100 A80 B-1
N11 G1 Z-47
N12 G1 X120
N13 G80
END
```

Transverse contour roughing G827/G828

The cycle machines the contour area described by the current tool position and the data defined in the subsequent blocks in transverse direction **without** recessing (see “Contour definition” on page 310).





Parameters

- **Z cutting limit:** The control machines up to the cutting limit.
- **P maximum infeed:** The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the infeed distance is <= P.
- **H type of departure** (default: 1):
  - H=0: Machine contour outline after each pass.
  - H=1: Retract at 45°; machine contour outline after last pass.
  - H=2: Retract at 45°; do not machine contour outline.
- **I oversize X** (diameter value)—(default: 0)
- **K oversize Z** (default: 0)

Note on the execution of the cycle:

- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point / end point of the contour area.

Tool position at the end of the cycle:

- G827: Cycle starting point X; last retraction diameter in Z
- G828: Cycle starting point

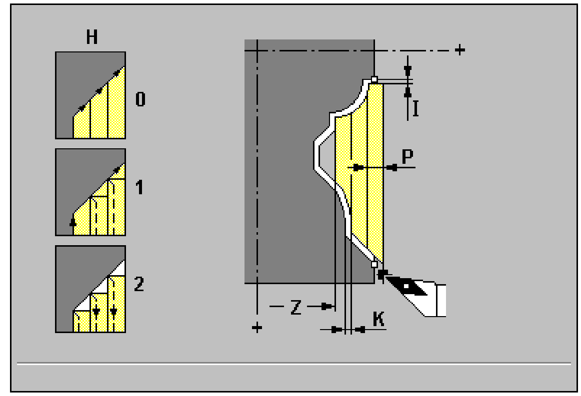
- Descending contour elements are not machined.
- The tool must be located outside the defined contour area.
- **Cutting radius compensation:** Active.
- G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.
- **Safety clearance** after each step: Parameter "Current parameters—Machining—Safety distances."

Example: G827, 828

```
%827.nc
[G827, G828]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G827 Z-15 P5 H0 I1 K0.3
N4 G0 X120 Z-15
N5 G1 X62 B3
N6 G1 Z-8 B2
N7 G1 X40 B-2
N8 G1 Z0 B-2
N9 G1 X30
N10 G80
N11 G0 X120 Z-15
N12 G828 Z-38 P4 H1 I1 K0.3
N13 G0 X120 Z-38
N14 G1 X103 B-3
N15 G1 Z-25
N16 G1 Z-15 A195 B2
N17 G1 X80
N18 G80
END
```

Transverse contour roughing with recessing G829

The cycle machines the contour area described by the current tool position and the data defined in the subsequent blocks in transverse direction **with** recessing (see "Contour definition" on page 310).



Parameters

- `Z cutting limit:` The control machines up to the cutting limit.
- `P maximum infeed:` The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the infeed distance is <= P.
- `E plunge feed:` The tool enters the material at the plunge feed E.
  - E=0: Descending contours are not machined.
  - No entry: The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%).
- `H type of departure` (default: 1):
  - H=0: Machine contour outline after each pass.
  - H=1: Retract at 45°; machine contour outline after last pass.
  - H=2: Retract at 45°; do not machine contour outline.
- `I oversize X` (diameter value)—(default: 0)
- `K oversize Z` (default: 0)

Note on the execution of the cycle:

- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point / end point of the contour area.
- Tool position at the end of the cycle: Cycle starting point

> - The tool must be located outside the defined contour area.
> - **Cutting radius compensation:** Active.
> - G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.
> - **Safety clearance** after each step: Parameter "Current parameters—Machining—Safety distances."

> **Danger of collision!**
>
> If the tool angle and the tool point angle have **not** been defined, the tool plunge-cuts at the plunging angle. If the tool and point angles have been defined, the tool plunge-cuts at the maximum possible plunging angle. In this case, the resulting contour will not be completely finished and may need to be reworked.

Example: G829

```
%829.nc
[G829]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X125 Z0
N3 G829 P5 H1 I1 K0.3
N4 G0 X120 Z-12
N5 G1 Z-3 A195 B3
N6 G1 X90 B2
N7 G1 Z-9 A-65 B-2
N8 G1 X50
N9 G1 Z-11 A-60
N10 G1 X32 B1
N11 G1 X24 Z0
N12 G80
END
```

Contour-parallel roughing G836

G836 machines the workpiece sections parallel to the contour. The starting point of the contour is defined either in the cycle with X,Z or in the G0 block after the cycle call. The blocks following G836 describe the contour area. G80 concludes the contour description.

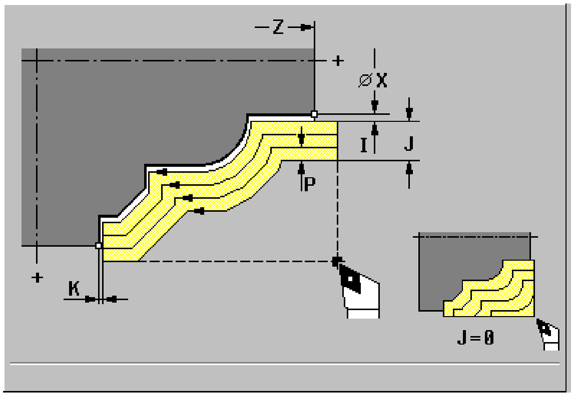
Parameters

- **`X starting point`** (diameter value)
- **`Z starting point`**
- **`P maximum infeed:`** The infeed depth is determined taking J into account. The proportioning of cuts is calculated so that an "abrasive cut" is avoided.
  - J=0: P is the maximum infeed depth. The cycle reduces the infeed depth if the programmed infeed is not possible in the transverse or longitudinal direction due to the cutting geometry.
  - J>0: P is the infeed depth. This infeed is used in the longitudinal and transverse directions.
- **`I oversize X`** (diameter value)—(default: 0)
- **`K oversize Z`** (default: 0)
- **`J workpiece blank oversize`**—the cycle machines:
  - J=0: From the current tool position.
  - J>0: The area defined by the workpiece blank oversize.
- **`Q transverse roughing`** (default: 0): Longitudinal or transverse machining
  - Q=0: Longitudinal machining
  - Q=1: Transverse machining

Note on the execution of the cycle:

- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point / end point of the contour area.
- Tool position at the end of the cycle: Cycle starting point

- At the start of the cycle, the tool must be located outside the defined contour area.
- **Cutting radius compensation:** Active.
- G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.
- **Safety clearance** after each step: Parameter "Current parameters—Machining—Safety distances."
- For **workpiece blank oversize J>0**: Set the "infeed depth P" to the smaller infeed, if the maximum infeed differs for the longitudinal and transverse directions due to the cutting geometry.
- The cycle parameter **`workpiece blank oversize J`** is available as of NC software versions 507 807-16 and 526 488-08. With earlier software versions, the cycle starts the machining operation from the current tool position.



Example: G836

```
%836.nc
[G836]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G836 P4 I1 K0.3
N4 G0 X80 Z0
N5 G1 Z-15 B-1
N6 G1 X102 B2
N7  G1 Z-22
N8 G1 X90 Zi-12 B1
N9 G1 Zi-6
N10 G1 X100 A80 B-1
N11 G1 Z-47
N12 G1 X110
N13 G0 Z2
N14 G80
END
```

Contour finishing G89

G89 finishes the contour area defined in the subsequent blocks (see "Contour definition" on page 310).

In the NC block after G89, the tool-tip radius compensation (TRC) is called with G41/G42 (without parameters) and allows you to define the position of the tool (reference: contour direction):

- G41: Tool moves to the right of the contour.
- G42: Tool moves to the left of the contour.

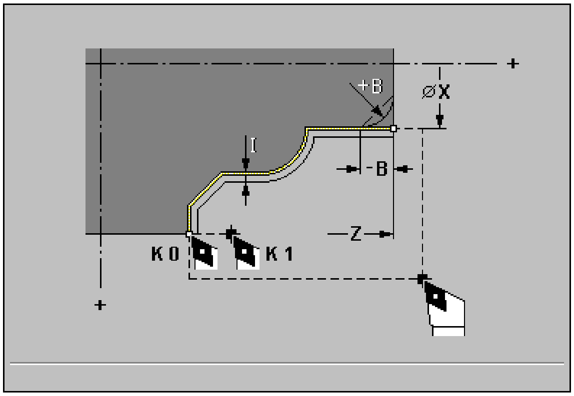
MANUALplus switches off the TRC at the end of the cycle. If you do not define G41/G42, the TRC function does not become effective.

Parameters

- **`B chamfer/rounding`** at start of contour
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **`I oversize:`** Equidistant oversize—a negative oversize is permitted.
- **`K retraction mode at the end of cycle`**—defines the tool position at the end of the cycle:
  - No input: Return to starting point of cycle
  - K=0: Tool remains at cycle end position
  - K>0: Tool retracts by K
- **`J element position:`** When the contour section begins with a chamfer/rounding, J defines the position of the "imaginary reference element" (default: 1).

  Reference element:
  - J=1: Transverse element in the positive X-axis direction
  - J=–1: Transverse element in the negative X-axis direction
  - J=2: Longitudinal element in the positive Z-axis direction
  - J=–2: Longitudinal element in the negative Z-axis direction

Oversizes: An oversize programmed with G58 is taken into account if I is not defined in the cycle. After the cycle has been executed, the oversize is canceled.



Example: G89

```
%89.nc
[G89]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X70 Z2
N3 G89 B-2 I2 K1 J1
N4 G42
N5 G0 X40 Z0
N6 G1 Z-20 B3
N7 G1 X60 B-2
N8 G1 Z-32
N9 G25 H5 W30
N10 G1 X70
N11 G80
END
```

6.12 Simple Turning Cycles

Roughing longitudinal G81

G81 machines the contour area defined by the current tool position and "X, Z" in longitudinal direction.

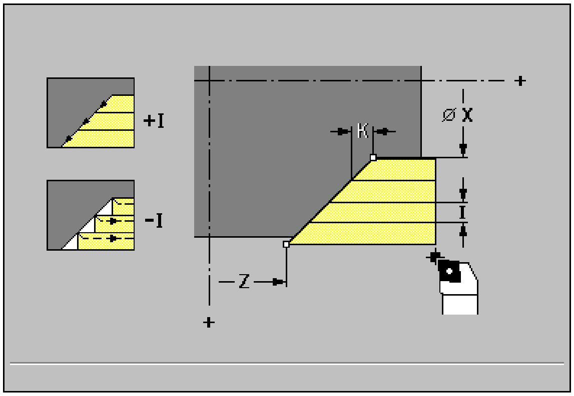
Parameters

- **X starting point** of contour section (diameter value)
- **Z end point** of contour section
- **I maximum infeed** in X: The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the calculated infeed distance is <= I.
  - I>0: With machining contour outline
  - I<0: Without machining contour outline
- **K offset:** Infeed in Z (default: 0)
- **Q G function infeed:** Infeed is executed through G function
  - Q=0: Infeed with G0
  - Q=1: Infeed with G1
- **V type of retraction** (default: 0)
  - V=0: Return to cycle starting point in Z and last retraction diameter in X
  - V=1: Return to starting point of cycle

Note on the execution of the cycle:

- If you wish to machine an oblique cut, you can define the angle with I and K.
- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point / end point of the contour area.

- **Cutter radius compensation:** is not carried out.
- **Oversizes:** Oversizes programmed with G57 are taken into account. The oversizes remain **in effect** after execution of the cycle.
- **Oversizes for inside contours:** Program negative oversizes with G57 (possible only with "Free entry").
- **Safety clearance** after a pass is 1 mm.



Example: G81

```
%81.nc
[G81]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G81 X100 Z-70 I4 K4 V0
N4 G0 X100 Z2
N5 G81 X80 Z-60 I-4 K2 V1
N6 G0 X80 Z2
N7 G81 X50 Z-45 I4 Q1
END
```

Roughing transverse G82

G82 machines the contour area defined by the current tool position and "Z, X" in transverse direction.

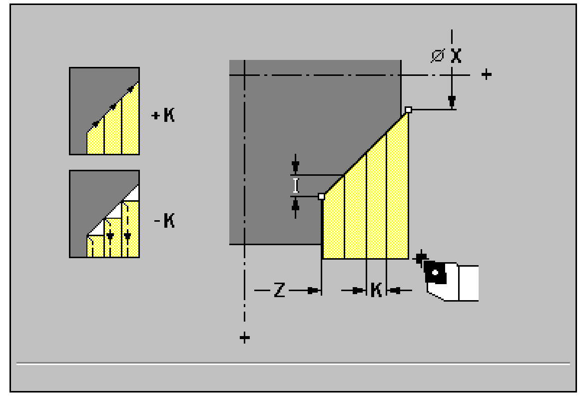
Parameters

- **`X end point`** of contour section (diameter value)
- **`Z starting point`** of contour section
- **`I offset:`** Infeed in Z (default: 0)
- **`K maximum infeed`** in X: The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the calculated infeed distance is <= K.
  - K>0: With machining contour outline
  - K<0: Without machining contour outline
- **`Q G function infeed:`** Infeed is executed through G function
  - Q=0: Infeed with G0 (rapid traverse)
  - Q=1: Infeed with G1 (feed rate)
- **`V type of retraction`** (default: 0)
  - V=0: Return to cycle starting point in Z and last retraction diameter in X
  - V=1: Return to starting point of cycle

Note on the execution of the cycle:

- If you wish to machine an oblique cut, you can define the angle with I and K.
- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point / end point of the contour area.

> - **Cutter radius compensation:** is not carried out.
> - **Oversizes:** Oversizes programmed with G57 are taken into account. The oversizes remain **in effect** after execution of the cycle.
> - **Safety clearance** after a pass is 1 mm.



Example: G82

```
%82.nc
[G82]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G82 X20 Z-15 I4 K4 V0
N4 G0 X120 Z-15
N5 G82 X50 Z-26 I2 K-4 V1
N6 G0 X120 Z-26
N7 G82 X80 Z-45 K4 Q1
END
```

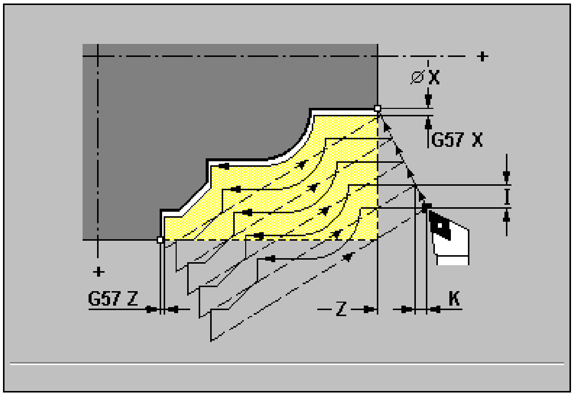
Simple contour repeat cycle G83

G83 repeatedly executes the machining cycle programmed in the subsequent blocks. The machining cycle may contain simple traverse paths or cycles (without contour definition). G80 ends the machining cycle.

"X, Z" define the starting point of the contour. G83 starts the cycle execution from the current tool position. Before each pass, the tool advances by the infeed distance defined in "I, K." The control then executes the machining operation which is programmed in the blocks after G83, taking the distance from the tool position to the contour starting point as an "oversize." G83 repeats this operation until the "starting point" is reached.

G83 is used for:

- Machining contour-parallel workpiece sections (roughing of pre-shaped workpiece blanks).
- Repeating machining operations (for example, for slot-cutting).



Parameters

- **X starting point** (diameter value)
- **Z starting point**
- **I maximum infeed** in X direction (I is entered without the algebraic sign)
- **K maximum infeed** in Z direction (K is entered without the algebraic sign)

Note on the execution of the cycle:

- If the number of infeeds differs for the X and Z axes, the tool first advances in both axes with the programmed values. As soon as the target dimension is reached in one axis, the tool no longer advances in this axis.
- MANUALplus automatically determines the cutting and infeed directions from the current tool position relative to the starting point of the contour area.
- Tool position at the end of the cycle: Starting point of contour

- G83 must not be nested, not even by calling subprograms.
- At the start of the cycle, the tool must be located outside the defined contour area.
- **Cutter radius compensation:** is not carried out—You can program TRC separately.
- **Oversizes:** Oversizes programmed with G57 are taken into account. An oversize programmed with G58 is accounted for, provided that the TRC function is active. The oversizes remain in effect after execution of the cycle.

Danger of collision!
After each pass, the tool returns on a diagonal path before it advances for the next pass. If there is danger of collision, you must program an additional path of rapid traverse to avoid a collision.

Example: G83

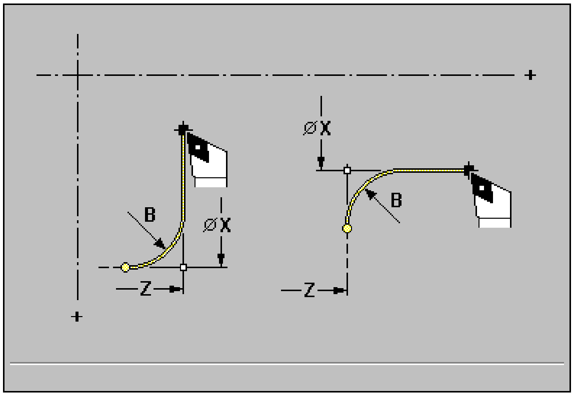
```
%83.nc
[G83]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G83 X80 Z0 I4 K0.3
N4 G0 X80 Z0
N5 G1 Z-15 B-1
N6 G1 X102 B2
N7  G1 Z-22
N8 G1 X90 Zi-12 B1
N9 G1 Zi-6
N10 G1 X100 A80 B-1
N11 G1 Z-47
N12 G1 X110
N13 G0 Z2
N14 G80
END
```

Line with radius G87

G87 machines transition radii at orthogonal, paraxial inside and outside corners. A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed. The radii are machined in one pass.

MANUALplus determines the direction of the radius from the "tool orientation" (see “Lathe tools” on page 419).

Tool position at the end of the cycle: End point of radius



Parameters

- **X corner point** (diameter value)
- **Z corner point**
- **B radius**
- **E reduced feed rate:** default: Active feed rate

- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.

Example: G87

```
%87.nc
[G87]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X70 Z2
N3 G1 Z0
N4 G87 X84 Z0 B2
N6 . . .
END
```

Line with chamfer G88

G88 machines chamfers at orthogonal, paraxial outside corners. A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed. The chamfers are machined in one pass.

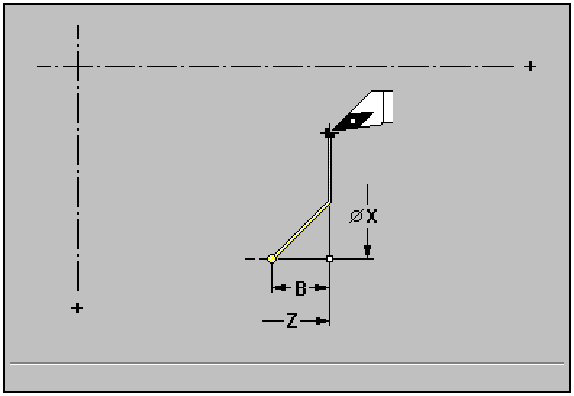
MANUALplus determines the direction of the chamfer from the "tool orientation" (see “Lathe tools” on page 419).

Tool position at the end of the cycle: End point of chamfer

Parameters

- **X corner point** (diameter value)
- **Z corner point**
- **B chamfer width**
- **E reduced feed rate:** default: Active feed rate

- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.



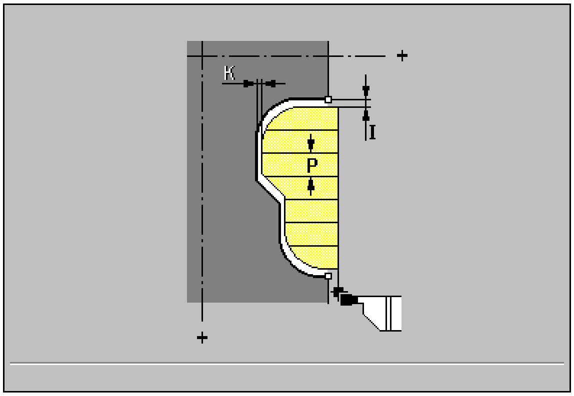
Example: G88

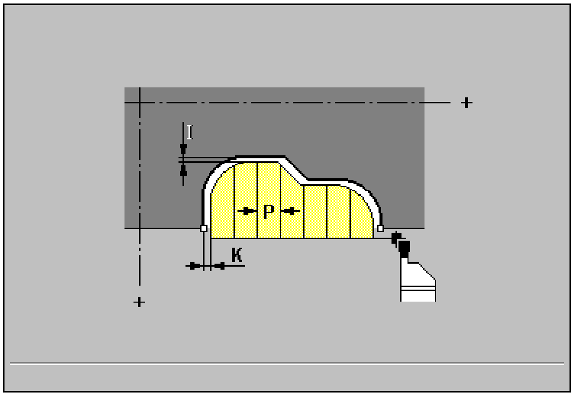
```
%88.nc
[G88]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X70 Z2
N3 G1 Z0
N4 G88 X84 Z0 B2
N5 . . .
END
```

6.13 Recessing Cycles

Contour recessing axial G861 / radial G862

The cycles machine an axial/radial recess in the contour area described by the current tool position and the data defined in the subsequent blocks (see “Contour definition” on page 310).





Parameters

- **`P recessing width`**
  - P is not defined: Infeeds <= 0.8 * cutting width of tool
  - P is defined: Infeeds <= P
- **`I oversize X`** (default: 0)
- **`K oversize Z`** (default: 0)
- **`Q roughing/finishing`**
  - Q=0: Only roughing
  - Q=1: The recess is first rough-machined with consideration of the oversizes and then finish-machined at finishing feed E.
- **`E finishing feed`** (default: Active feed rate)

Note on the execution of the cycle:

- MANUALplus determines the cutting direction from the current tool position relative to the starting point / end point of the contour area.
- Tool position at the end of the cycle: Cycle starting point

- **Cutting radius compensation:** Active.
- G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.

Example: G861

```
%861.nc
[G861]
N1 T38 G95 F0.15 G96 S200 M3
N2 G0 X110 Z2
N3 G861 I1 K0.2 Q1
N4 G0 X100 Z2
N5 G1 Z-6 B3
N6 G1 X88 B2
N7 G1 Z-13 A-20 B2
N8 G1 X60 B3
N9 G1 Z0 B-1
N10 G1 X55
N11 G80
END
```

Example: G862

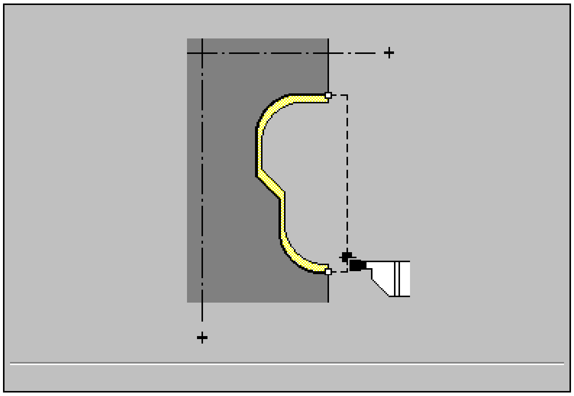
```
%862.nc
[G862]
N1 T30 G95 F0.15 G96 S200 M3
N2 G0 X87 Z-35
N3 G862 I0.5 Q1 E0.11
N4 G0 X85 Z-29.5
N5 G1 X84 Z-30
N6 G1 X75 A-75 B2
N7 G1 Z-42 B-1
N8 G1 X70
N9 G1 Z-58.5 B3
N10 G1 X85 Z-63 B-2
N11 G1 Z-66
N12 G80
END
```

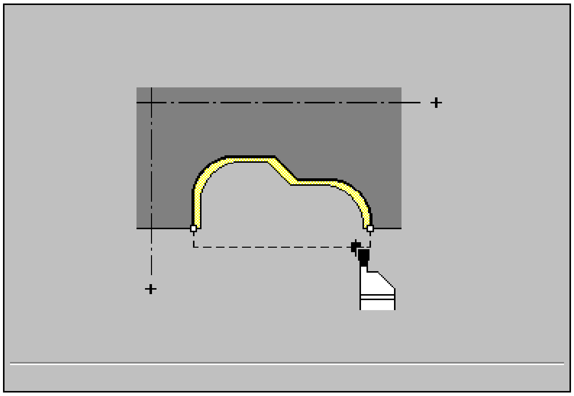
Contour recessing cycle, finishing, axial G863 / radial G864

The cycles axially/radially finish the contour area defined in the subsequent blocks (see "Contour definition" on page 310).

Parameters

- `E finishing feed rate`





Note on the execution of the cycle:

- Tool position at the end of the cycle: Cycle starting point

Cutting radius compensation: Active.

Example: G863

```
%863.nc
[G863]
N1 T38 G95 F0.15 G96 S200 M3
N2 G0 X110 Z2
N3 G863 E0.08
N4 G0 X100 Z2
N5 G1 Z-6 B3
N6 G1 X88 B2
N7 G1 Z-13 A-20 B2
N8 G1 X60 B3
N9 G1 Z0 B-1
N10 G1 X55
N11 G80
END
```

Example: G864

```
%864.nc
[G864]
N1 T30 G95 F0.15 G96 S200 M3
N2 G0 X87 Z-35
N3 G864 E0.11
N4 G0 X85 Z-29.5
N5 G1 X84 Z-30
N6 G1 X75 A-75 B2
N7 G1 Z-42 B-1
N8 G1 X70
N9 G1 Z-58.5 B3
N10 G1 X85 Z-63 B-2
N11 G1 Z-66
N12 G80
END
```

Simple recessing cycle, axial G865 / radial G866

The cycles axially/radially machine the rectangle described by the tool position and "X, Z."

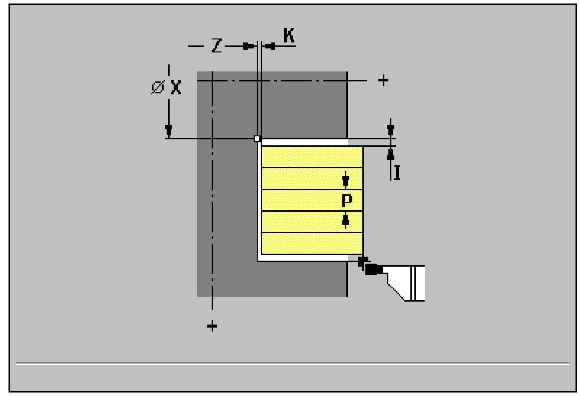
Parameters

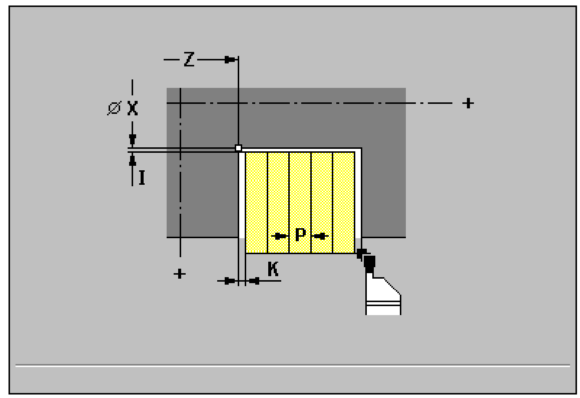
- **X base corner X** (diameter value)
- **Z base corner Z**
- **P recessing width**
  - P is not defined: Infeeds <= 0.8 * cutting width of tool
  - P is defined: Infeeds <= P
- **I oversize X** (default: 0)
- **K oversize Z** (default: 0)
- **Q roughing/finishing**
  - Q=0: Only roughing
  - Q=1: The recess is first rough-machined with consideration of the oversizes and then finish-machined at finishing feed E.
- **E finishing feed rate** or dwell time
  - For Q=0: Dwell time (for chip breaking)—default: Time of two revolutions
  - For Q=1: Finishing feed—default: Active feed rate

Note on the execution of the cycle:

- MANUALplus determines the cutting direction from the current tool position relative to the starting point / end point of the contour area.
- Tool position at the end of the cycle: Cycle starting point

> - **Cutting radius compensation:** Active.
> - G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.





Example: G865

```
%865.nc
[G865]
N1 T38 G95 F0.15 G96 S200 M3
N2 G0 X120 Z1
N3 G865 X102 Z-4 I0.5 K0.2 Q1 E0.11
END
```

Example: G866

```
%866.nc
[G866]
N1 T30 G95 F0.15 G96 S200 M3
N2 G0 X62 Z-18
N3 G866 X54 Z-30 I0.2 K1 Q1 E0.12
END
```

Recessing finishing, axial G867 / radial G868

The cycles axially/radially finish the contour area described by the tool position and "X, Z."

Tool position at the end of the cycle: Cycle starting point

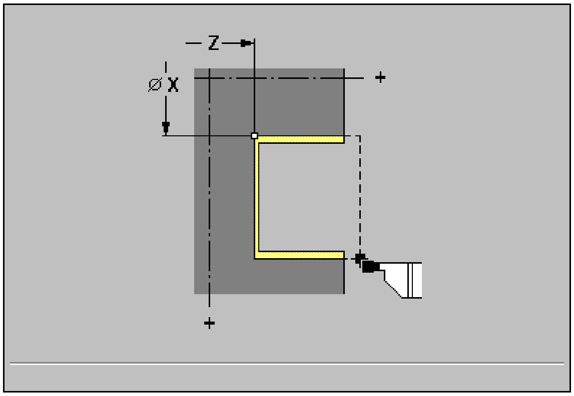
Parameters

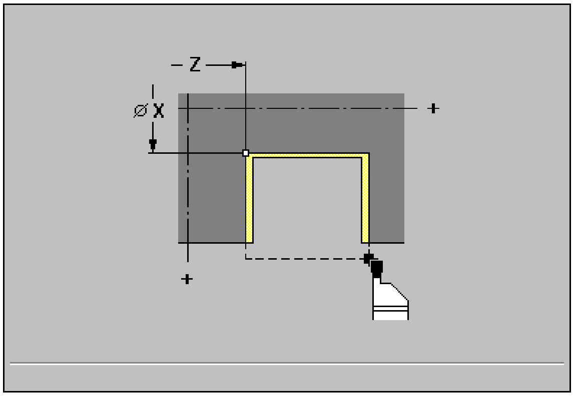
- `X base corner X` (diameter value)
- `Z base corner Z`
- `E finishing feed` (default: Active feed rate)

Note on the execution of the cycle:

- Tool position at the end of the cycle: Cycle starting point

Cutting radius compensation: Active.





Example: G867

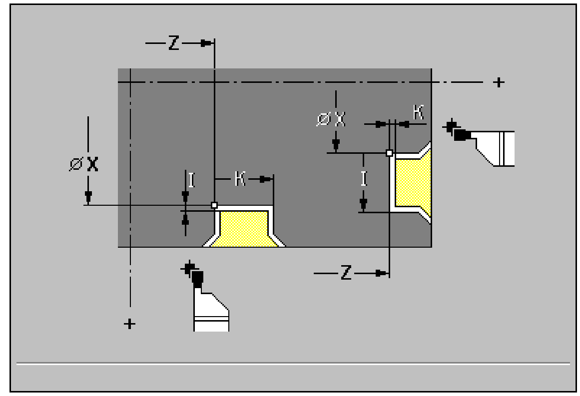
```
%867.nc
[G867]
N1 T38 G95 F0.15 G96 S200 M3
N2 G0 X120 Z1
N3 G867 X102 Z-4 E0.11
END
```

Example: G868

```
%868.nc
[G868]
N1 T30 G95 F0.15 G96 S200 M3
N2 G0 X62 Z-18
N3 G868 X54 Z-30 E0.12
END
```

Simple recessing cycle G86

G86 machines simple radial/axial inside and outside recesses with chamfers. From the "tool orientation," the control determines the type of recess (radial/axial; inside/outside, see “Lathe tools” on page 419).



Parameters

- **X base corner X** (diameter value)
- **Z base corner Z**
- **I oversize**
  - Radial recess: Oversize for precutting
  - Axial recess: Recess width—no input: A **single** cut is machined (recess width = tool width).
- **K width**
  - Radial recess: Recess width—no input: A **single** cut is machined (recess width = tool width).
  - Axial recess: Oversize for precutting
- **E dwell time** (for finishing)—default: length of time for one revolution

Note on the execution of the cycle:

- If you program an oversize, the control always rough-machines the recess first. In the second step, the recess is then finish-machined.
- If you do not wish to cut the chamfers, you must position the tool at a sufficient distance from the workpiece. Calculation for radial recess:
  XS = XK + 2 * (1.3 – b)
  XS: Starting position (diameter value)
  XK: Contour diameter
  b: Chamfer width
  The starting position is calculated accordingly for an axial recess.
- At the end of the cycle, the tool is located at:
  - For radial recess:
  - X: Starting position
  - Z: Last recess position
  - For axial recess:
  - X: Last recess position
  - Z: Starting position

- **Cutter radius compensation:** is not carried out.
- **Oversizes:** are not taken into account.

Example: G86

```
%86.nc
[G86]
N1 T30 G95 F0.15 G96 S200 M3
N2 G0 X62 Z2
N3 G86 X54 Z-30 I0.2 K7 E2 [radial]
N4 G14 Q0
N5 T38 G95 F0.15 G96 S200 M3
N6 G0 X120 Z1
N7 G86 X102 Z-4 I7 K0.2 E1 [axial]
END
```

6.14 Recess-Turning Cycles

Function of recess turning cycles

The defined contour area is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements.

The contour to be machined may contain various valleys. If required, the area to be machined is divided into several sections.

To influence recess-turning operations, use the following parameters:

- **Recessing feed O:** Feed rate for recessing - If "O" is not defined, the "active feed rate" for turning and recessing operations is effective.
- **Turning operation, unidirectional/bidirectional U:** You can perform a unidirectional or bidirectional turning operation. With **radial** recess-turning cycles, unidirectional turning operations are always performed in the direction of the spindle. With **axial** recess-turning cycles, the machining direction corresponds to the direction of contour definition.
- **Offset width B:** After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by "B." Each time the system switches on this side, the path is reduced by "B"—in addition to the previous offset. The total offset is limited to 80% of the effective cutting width (effective cutting width = cutting width –2*cutting radius). If required, the MANUALplus reduces the programmed offset width. After precutting, the remaining material is removed with a single cut.
- **Depth compensation R** (only with G815/G825): Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with "R" during finish-machining. The depth compensation factor is usually determined empirically.
- **Roughing/finishing Q:** Define whether the contour area is to be rough-machined and/or finish-machined. "Q" can be programmed such that the workpiece is rough-machined in the first cycle; then insert another tool and finish-machine the workpiece, using a further cycle.

These cycles require the use of **recess-turning tools.**

Simple recess-turning cycle, longitudinal G811 / transverse G821

The cycles machine the rectangle described by the tool position and "X, Z."

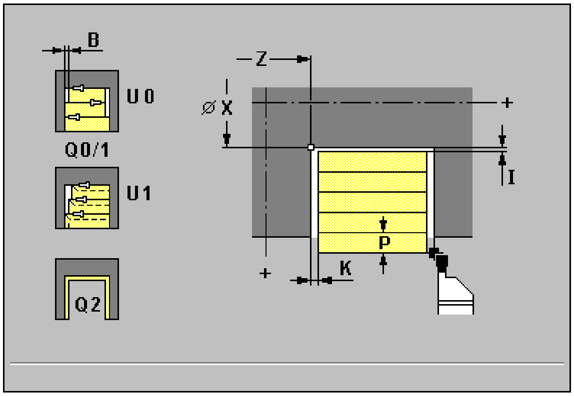
Parameters

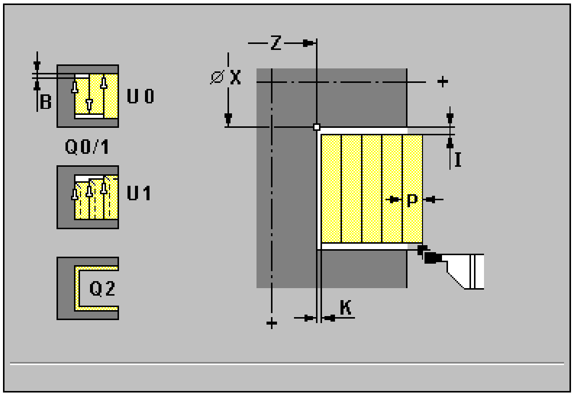
- **X base corner X** (diameter value)
- **Z base corner Z**
- **P maximum infeed:** The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the infeed distance is <= P.
- **I oversize X** (default: 0)
- **K oversize Z** (default: 0)
- **Q roughing/finishing** (default: 0)
  - Q=0: The recess is first rough-machined with consideration of the oversizes and then finish-machined at finishing feed E.
  - Q=1: Only roughing
  - Q=2: Finishing only—"I, K" defines the material to be machined.
- **U unidirectional turning** (default: 0):
  - U=0: bidirectional
  - U=1: Unidirectional
    – G811: In direction of spindle
    – G821: In direction of "base corner X"
- **B offset width** (default: 0)
- **O recessing feed rate** (default: Active feed rate)
- **E finishing feed** (default: Active feed rate)

Note on the execution of the cycle:

- Tool position at the end of the cycle: Cycle starting point

- It is absolutely necessary to define the **oversizes I, K** for recess turning—finishing (Q=2), since they define the material to be machined during the finishing cycle.
- **Cutting radius compensation:** Active.
- G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.





Example: G811

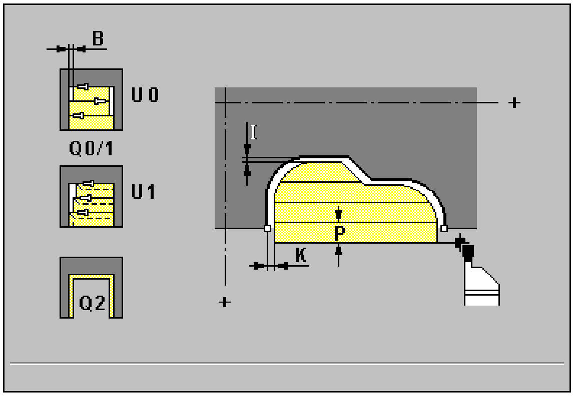
```
%811.nc
[G811]
N1 T38 G95 F0.4 G96 S140 M3
N2 G0 X122 Z-30
N3 G811 X80 Z-60 P2 Q1 B0.1 O0.2
END
```

Example: G821

```
%821.nc
[G821]
N1 T30 G95 F0.4 G96 S140 M3
N2 G0 X100 Z5
N3 G821 X60 Z-15 P2 Q1 B0.1 O0.25
END
```

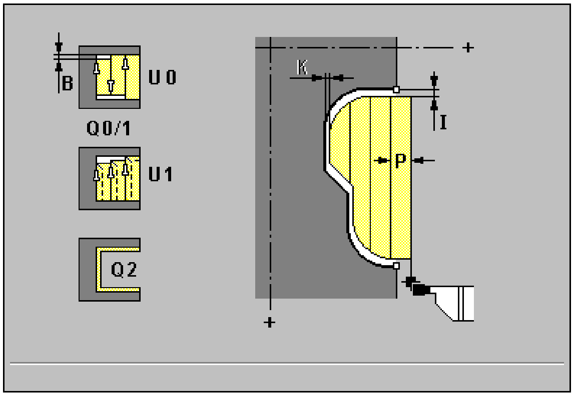
Recess-turning cycle, longitudinal G815 / transverse G825

The cycles machine the contour area described by current tool position and the data defined in the subsequent blocks (see "Contour definition" on page 310).



Parameters

- **X cutting limit** (diameter value)
- **Z cutting limit**
- **P maximum infeed:** The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the infeed distance is <= P.
- **I oversize X** (default: 0)
- **K oversize Z** (default: 0)
- **Q roughing/finishing** (default: 0)
  - Q=0: The recess is first rough-machined with consideration of the oversizes and then finish-machined at finishing feed E.
  - Q=1: Only roughing
  - Q=2: Finishing only—"I, K" defines the material to be machined.
- **U unidirectional turning** (default: 0):
  - U=0: bidirectional
  - U=1: Unidirectional
    – G815: In direction of spindle
    – G825: In direction of contour definition
- **B offset width** (default: 0)
- **R turning depth compensation** (default: 0)
- **O recessing feed rate** (default: Active feed rate)
- **E finishing feed** (default: Active feed rate)



Note on the execution of the cycle:

- Tool position at the end of the cycle: Cycle starting point

- It is absolutely necessary to define the **oversizes I, K** for recess turning—finishing (Q=2), since they define the material to be machined during the finishing cycle.
- **Cutting radius compensation:** Active.
- G57/G58 **oversizes** are taken into account if I/K is not programmed. After the cycle has been executed, the oversizes are canceled.

Example: G815

```
%815.nc
[G815]
N1 T38 G95 F0.4 G96 S140 M3
N2 G0 X62 Z-5
N3 G815 P3 I2 K1 B0.1 O0.32 E0.28
N4 G0 X60 Z-5
N5 G3 X54.2229 Z-9.5323 R5 I-5 K0 B1.5
N6 G1 X49.5 Z-32 B1.5
N7 G1 X35 Z-34 B1.5
N8 G1 Z-45 B1.5
N9 G1 X60 Z-49 B1.5
N10 G1 Z-51
N11 G80
END
```

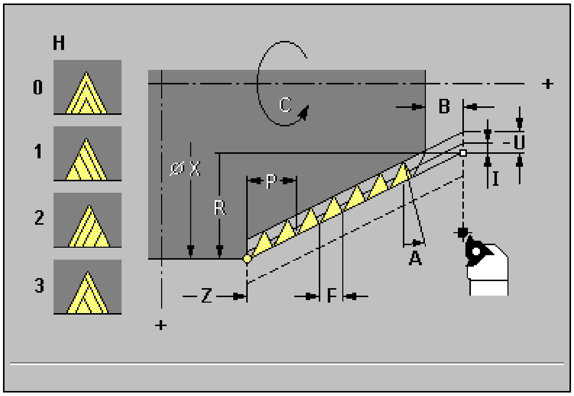
Example: G825

```
%825.nc
[G825]
N1 T30 G95 F0.4 G96 S140 M3
N2 G0 X82 Z2
N3 G825 P3 I2 K1 Q1 B0.1 O0.3
N4 G0 X81 Z0
N5 G1 X79.9477 Z-0.4993
N6 G1 X79.8355 Z-1.5698
N7 G1 X70 Z-2
N8 G1 X60 Z-7 B1
N9 G1 X46.2248 B1
N10 G1 X45 Z0 B-1
N11 G1 X42
N12 G80
END
```

6.15 Thread Cycles

Universal thread cycle G31

G31 cuts threads in any desired direction and position (longitudinal, tapered or transverse threads; internal or external threads). You can also machine successions of threads.



Parameters

- **`X end point`** of thread (diameter value)
- **`Z end point`** of thread
- **`F thread pitch`**
- **`U thread depth`**
  - U>0: Internal thread
  - U<=0: External thread (lateral surface or front face)
  - U= +999 or –999: Thread depth is calculated
- **`I maximum infeed`**
- **`R difference in radii`** (default: 0): Difference between the diameters at the start of thread (XA) and end of thread (X). With descending contours, R must be programmed as a negative value. R=(X–XA)/2
- **`B run-in length:`** Distance required to accelerate to the programmed feed rate.
  No input: Internal calculation (see "Thread run-in / thread run-out" on page 163)
- **`P run-out length:`** Distance required to decelerate the slide.
  No input: Internal calculation (see "Thread run-in / thread run-out" on page 163)
- **`A feed angle:`** Range: 0° < A < 60°
  No input: A=arctan (0.5*F/U)
- **`V type of approach`** (default: 0)
  - V=0: Constant cross section for all cuts
  - V=1: Constant feed
  - V=2: With distribution of remaining cut
  - V=3: Without distribution of remaining cut
- **`H type of tool offset`** (default: 0)
  - H=0: Without offset
  - H=1: Offset from the left toward the thread base
  - H=2: Offset from the right toward the thread base
  - H=3: Tool is offset alternately from the right and left (zigzag)
- **`Q number of air cuts`** after the last cut (default: 0)
- **`C starting angle:`** Position of the spindle at the thread start (default: 0°)

- **G31 without contour definition:** "X, Z" is programmed. The thread starts at the current tool position and ends at the end point X, Z.

Example: G31

```
%31.nc
[G31]
N1 T45 G97 S800 M3
N2 G0 X20 Z5
N3 G31 Z-50 F1.5 I0.2
END
```

- **G31 with contour definition:** "X, Z" is not programmed. G31 is followed by NC blocks defining up to 6 contour elements on which the thread is to be machined. Contour definition is completed with G80.

Transverse threads or successive threads are programmed "with contour definition."

Internal or external threads: See algebraic sign of "U."

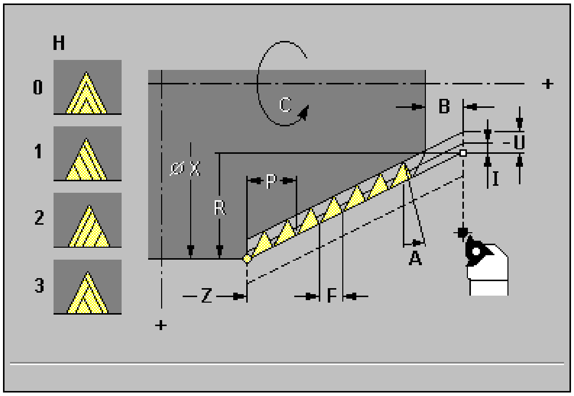
The **infeeds** are calculated on the basis of "V:"

- V=0: Constant cross section for all cuts. "I" defines the first (maximum) infeed. All further infeeds are executed in such a way that the same cross section as for the first cut is used.
- V=1: The thread is machined with constant infeeds <=I.
- V=2: If the division U/I provides a remainder, the first feed is reduced. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8
- V=3: If the division U/I provides a remainder, the first feed is reduced.

- Transverse threads:
  - Are machined with recessing tools.
  - The difference in radii must be programmed.
- "Cycle STOP" becomes effective at the end of a thread cut.
- Feed rate override is not effective during cycle execution.
- **Feed forward control** is switched on.

Single thread G32

G32 cuts a simple thread in any desired direction and position (longitudinal, tapered or transverse thread; internal or external thread). The thread starts at the current tool position and ends at the "end point X, Z."



Parameters

- **X end point** of thread (diameter value)
- **Z end point** of thread
- **F thread pitch**
- **U thread depth**
  - U>0: Internal thread
  - U<=0: External thread (lateral surface or front face)
  - U= +999 or –999: Thread depth is calculated
- **I maximum infeed**
- **B remainder cuts** (default: 0)
  - B=0: The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8.
  - B=1: Without distribution of remaining cut
- **Q number of air cuts** after the last cut (default: 0)
- **K run-out length** at end point of thread (default: 0)
- **W taper angle** (default: 0): Position of the tapered thread with reference to longitudinal or transverse axis. For cutting a descending tapered thread, W must be programmed with a negative algebraic sign.
  Range: –45° < W < 45°
- **C starting angle:** Position of the spindle at the thread start (default: 0°)
- **H type of tool offset** (default: 0)
  - H=0: Without offset
  - H=1: Offset from the left toward the thread base
  - H=2: Offset from the right toward the thread base
  - H=3: Tool is offset alternately from the right and left (zigzag)

Internal or external threads: See algebraic sign of "U."

Infeeds: If the division U/I provides a remainder, the first feed is reduced. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8

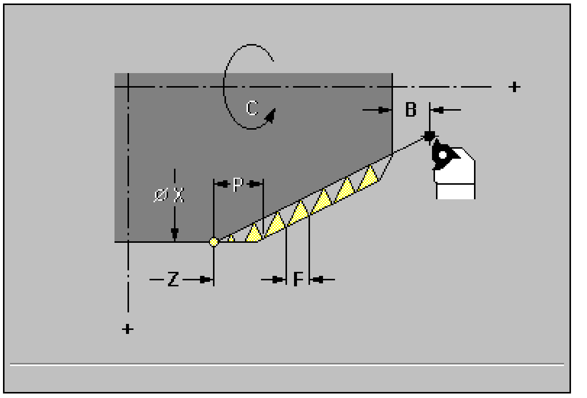
- Transverse threads are machined with recessing tools.
- "Cycle STOP" becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle execution.
- **Feedforward control** is switched off.

Example: G32

```
%32.nc
[G32]
N1 T45 G97 S800 M3
N2 G0 X16 Z4
N3 G32 X16 Z-29 F1.5 U-0.9 I0.2
END
```

Thread single path G33

G33 cuts threads in any desired direction and position with variable pitch (longitudinal, tapered or transverse threads; internal or external threads). The thread starts at the current tool position and ends at the "end point X, Z."



Parameters

- **X end point** of thread (diameter value)
- **Z end point** of thread
- **F thread pitch**
- **B run-in length** (default: 0): Distance required to accelerate to the programmed feed rate
- **P run-out length** (default: 0): Distance required to decelerate the slide
- **C starting angle:** Position of the spindle at the thread start (default: 0°)
- **Q number of spindle** (default: 0=master spindle)
- **H reference direction** for thread pitch (default: 3)
  - H=0: Feed rate on the Z axis (for longitudinal and taper threads up to a max. angle of +45°/–45° to the Z axis)
  - H=1: Feed rate on the X axis (for transverse and taper threads up to a max. angle of +45°/–45° to the X axis)
  - H=3: Contouring feed rate
- **E variable pitch** (default: 0)
  - E>0: Increase the pitch per revolution by E
  - E<0: Decrease the pitch per revolution by E

> - "Cycle STOP" becomes effective at the end of a thread cut.
> - Feed rate override is not effective during cycle execution.
> - **Feed forward control** is switched on.

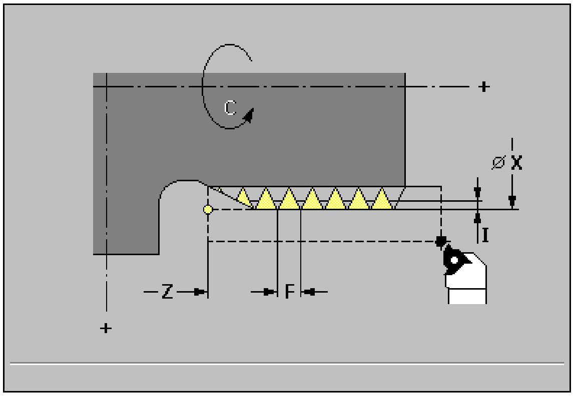
Example: G33

```
%33.nc
[G33]
N1 T45 G97 S1100 G95 F0.5 M3
N2 G0 X101.84 Z5
N3 G83 X100 Z5 I0.15
N4 G33 X120 Z-80 F1.5
N5 G33 X140 Z-122.5 F1.5
N6 G0 X150 Z5
N7 G80
END
```

Metric ISO thread G35

G35 cuts a longitudinal thread (internal or external thread). The thread starts at the current tool position and ends at the "end point X, Z."

From the tool position relative to the end point of the thread, MANUALplus automatically determines whether an internal or external thread is to be cut.



Parameters

- **`X end point`** of thread (diameter value)
- **`Z end point`** of thread
- **`F thread pitch`**—default: F is determined from the diameter in the standard table (see "Thread Pitch" on page 524).
- **`I maximum infeed`**—no input: I is calculated from the thread pitch and the thread depth.
- **`Q number of air cuts`** (default: 0): after the last cut
- **`B remainder cuts`** (default: 0)
  - B=0: The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8.
  - B=1: Without distribution of remaining cut

Infeeds: If the division U/I provides a remainder, the first feed is reduced. The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8

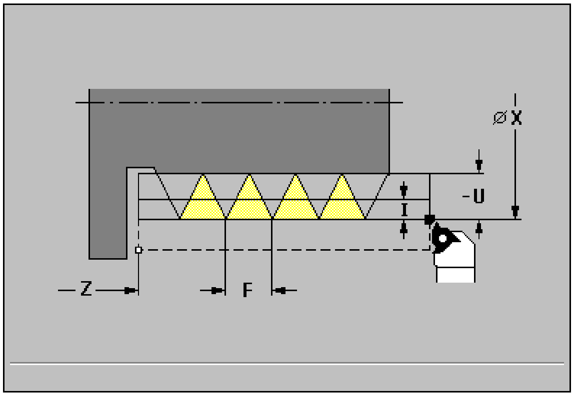
- "Cycle STOP" becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle execution.
- If you are programming an internal thread, it is advisable to preset the "thread pitch F," since the diameter of the longitudinal element is not the thread diameter. If you have MANUALplus calculate the thread pitch automatically, slight deviations may occur.
- **Feed forward control** is switched on.

Example: G35

```
%35.nc
[G35]
N1 T45 G97 S1500 M3
N2 G0 X16 Z4
N3 G35 X16 Z-29 F1.5
END
```

Simple longitudinal single-start thread G350

G350 cuts a longitudinal thread (internal or external thread). The thread starts at the current tool position and ends at the "end point X, Z."



Parameters

- **Z end point** of thread
- **F thread pitch**
- **U thread depth**
  - U>0: Internal thread
  - U<=0: External thread (lateral surface or front face)
  - U= +999 or –999: Thread depth is calculated
- **I maximum infeed**—no input: I is calculated from the thread pitch and the thread depth.

Internal or external threads: See algebraic sign of "U."

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- **X direction:** Depends on the current cutting depth; the starting and end points of the thread are not exceeded.
- **Z direction:** No more than one turn; the starting and end points of the thread are not exceeded.

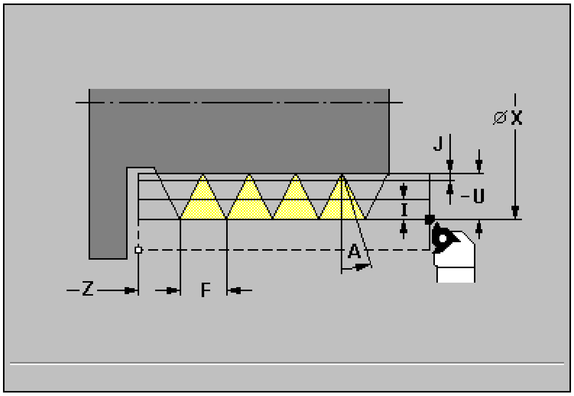
- "Cycle STOP" becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle execution.
- Handwheel superposition is activated with a switch located on the machine operating panel.
- **Feedforward control** is switched off.

Example: G350

```
%350.nc
[G350]
N1 T45 G97 S1500 G95 F1.5 M3
N2 G0 X16 Z4
N3 G350 Z-29 F1.5 U-999
END
```

Extended longitudinal multi-start thread G351

G351 machines a single or multi-start longitudinal thread (internal or external thread) with variable pitch. The thread starts at the current tool position and ends at the "end point X, Z."



Parameters

- **Z end point** of thread
- **F thread pitch**
- **U thread depth**
  - U>0: Internal thread
  - U<=0: External thread (lateral surface or front face)
  - U= +999 or –999: Thread depth is calculated
- **I maximum infeed**—no input: I is calculated from the thread pitch and the thread depth.
- **A feed angle** (default: 30°): Range: –60° < A < 60°
  - A>0: Infeed on right thread flank
  - A<0: Infeed on left thread flank
- **D number of thread starts** (default: 1)
- **J remaining cutting depth** (default: 1/100 mm)
- **E variable pitch** (default: 0)
  - E>0: Increase the pitch per revolution by E
  - E<0: Decrease the pitch per revolution by E

Internal or external threads: See algebraic sign of "U."

Proportioning of cuts: The first cut is performed at the cutting depth defined for "I" and is reduced with each cut until the tool reaches the "remaining cutting depth J."

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- **X direction:** Depends on the current cutting depth; the starting and end points of the thread are not exceeded.
- **Z direction:** No more than one turn; the starting and end points of the thread are not exceeded.

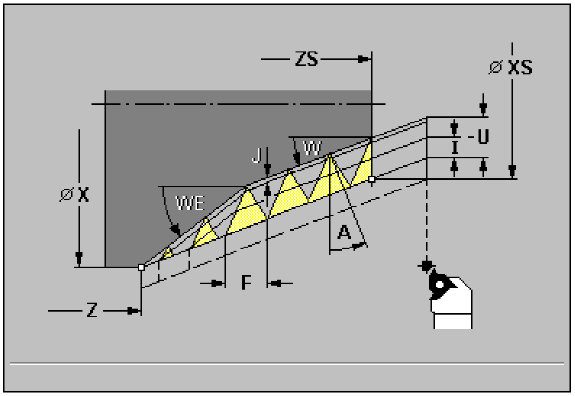
- "Cycle STOP" becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle execution.
- Handwheel superposition is activated with a switch located on the machine operating panel.
- **Feedforward control** is switched off.

Example: G351

```
%351.nc
[G351]
N1 T45 G97 S1500 M3
N2 G0 X16 Z4
N3 G351 Z-29 F1.5 U-0.9 I0.2
END
```

Tapered API thread G352

This cycle cuts a tapered single or multi-start API thread. The depth of thread decreases at the overrun at the end of thread.



Parameters

- **X end point** of thread (diameter value)
- **Z end point** of thread
- **XS starting point** of thread (diameter value)
- **ZS starting point** of thread
- **F thread pitch**
- **U thread depth**
  - U>0: Internal thread
  - U<=0: External thread (lateral surface or front face)
  - U= +999 or –999: Thread depth is calculated
- **I maximum infeed**—default: I is calculated from the thread pitch and the thread depth.
- **A feed angle** (default: 30°): Range: –60° < A < 60°
  - A>0: Infeed on right thread flank
  - A<0: Infeed on left thread flank
- **D number of thread starts** (default: 1)
- **W taper angle** (default: 0°): Range: –45° < W < 45°
- **WE run-out angle** (default: 12°): Range: 0° < WE < 90°
- **J remaining cutting depth** (default: 1/100 mm)

Internal or external threads: See algebraic sign of "U."

Proportioning of cuts: The first cut is performed at the cutting depth defined for "I" and is reduced with each cut until the tool reaches the "remaining cutting depth J."

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- **X direction:** Depends on the current cutting depth; the starting and end points of the thread are not exceeded.
- **Z direction:** No more than one turn; the starting and end points of the thread are not exceeded.

Definition of **taper angle**: XS/ZS, X/Z, or XS/ZS, Z, W, or ZS, X/Z, W

- "Cycle STOP" becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle execution.
- Handwheel superposition is activated with a switch located on the machine operating panel.
- **Feedforward control** is switched off.

Example: G352

```
%352.nc
[G352]
N1 T45 G97 S1500 M3
N2 G0 X13 Z4
N3 G352 X16 Z-28 XS13 ZS0 F1.5 U-999
WE12
END
```

Tapered thread G353

G353 cuts a tapered single or multi-start thread with variable pitch.

Parameters

- **X end point** of thread (diameter value)
- **Z end point** of thread
- **XS starting point** of thread (diameter value)
- **ZS starting point** of thread
- **F thread pitch**
- **U thread depth**
  - U>0: Internal thread
  - U<=0: External thread (lateral surface or front face)
  - U= +999 or –999: Thread depth is calculated
- **I maximum infeed**—default: I is calculated from the thread pitch and the thread depth.
- **A feed angle** (default: 30°):
  Range: –60° < A < 60°
  - A>0: Infeed on right thread flank
  - A<0: Infeed on left thread flank
- **D number of thread starts**—default: 1
- **W taper angle** (default: 0°): Range: –45° < W < 45°
- **J remaining cutting depth** (default: 1/100 mm)
- **E variable pitch** (default: 0)
  - E>0: Increase the pitch per revolution by E
  - E<0: Decrease the pitch per revolution by E

Internal or external threads: See algebraic sign of "U."

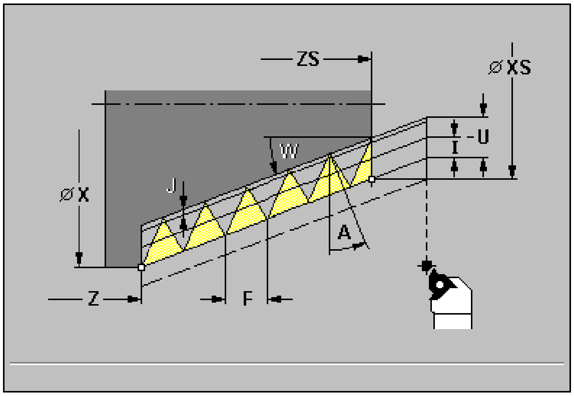
Proportioning of cuts: The first cut is performed at the cutting depth defined for "I" and is reduced with each cut until the tool reaches the "remaining cutting depth J."

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- **X direction:** Depends on the current cutting depth; the starting and end points of the thread are not exceeded.
- **Z direction:** No more than one turn; the starting and end points of the thread are not exceeded.

Definition of **taper angle**: XS/ZS, X/Z, or XS/ZS, Z, W, or ZS, X/Z, W

- "Cycle STOP" becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle execution.
- Handwheel superposition is activated with a switch located on the machine operating panel.
- **Feedforward control** is switched off.



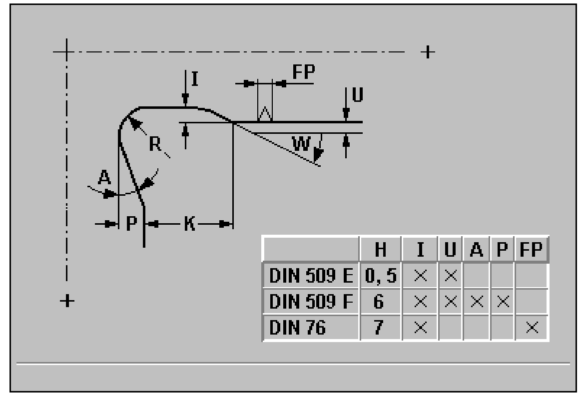
Example: G353

```
%353.nc
[G353]
N1 T45 G97 S1500 M3
N2 G0 X13 Z4
N3 G353 X16 Z-28 XS13 ZS0 F1.5 U-999
END
```

6.16 Undercut Cycles

Undercut contour G25

G25 generates an undercut form element (DIN 509 E, DIN 509 F, DIN 76) that can then be integrated in roughing or finishing cycles. The table in the graphic support window describes the parameters for undercuts.



Parameters

- **`H type of undercut`** (default: 0)
  - H=0, 5: DIN 509 E
  - H=6: DIN 509 F
  - H=7: DIN 76
- **`I undercut depth`** (default: Value from standard table)
- **`K undercut width`** (default: Value from standard table)
- **`R undercut radius`** (default: Value from standard table)
- **`P transverse depth`** (default: Value from standard table)
- **`W undercut angle`** (default: Value from standard table)
- **`A transverse angle`** (default: Value from standard table)
- **`FP thread pitch`**—no input: FP is calculated from the thread diameter
- **`U finishing oversize`** (default: 0)
- **`E reduced feed rate`** for machining the undercut (default: Active feed rate)

Note:
If the parameters are not defined, MANUALplus determines the following values from the diameter or the thread pitch (undercut DIN 76) in the standard table (see “Thread Pitch” on page 524):

- DIN 509 E: I, K, W, R
- DIN 509 F: I, K, W, R, P, A
- DIN 76: I, K, W, R and FP (determined from the diameter)

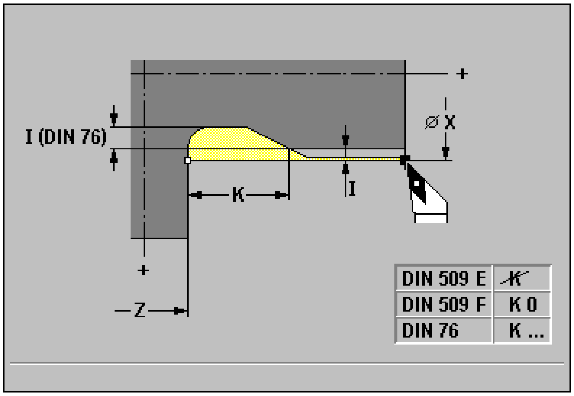
> - All parameters that you enter will be accounted for—even if the standard table prescribes other values.
> - If you are programming an internal thread, it is advisable to preset the "thread pitch FP," since the diameter of the longitudinal element is not the thread diameter. If you have MANUALplus calculate the thread pitch automatically, slight deviations may occur.

Example: G25

```
%25.nc
[G25]
N1 T1 G95 F0.4 G96 S150 M3
N2 G0 X62 Z2
N3 G819 P4 H0 I0.3 K0.1
N4 G0 X13 Z0
N5 G1 X16 Z-1.5
N6  G1 Z-30
N7 G25 H7 I1.15 K5.2 R0.8 W30 FP1.5
N8 G1 X20
N9 G1 X40 Z-35
N10 G1 Z-55 B4
N11 G1 X55 B-2
N12 G1 Z-70
N13 G1 X60
N14 G80
END
```

Undercut cycle G85

With the function G85, you can machine undercuts according to DIN 509 E, DIN 509 F and DIN 76 (thread undercut). The adjoining cylinder is machined if you position the tool at the cylinder diameter "in front of" the cylinder. If the tool is not positioned at the cylinder diameter, it approaches the workpiece on a diagonal path to machine the undercut.



Parameters

- **`X target point`** (diameter value)
- **`Z target point`**
- **`I finishing oversize/depth`**
  - DIN 509 E, F: Finishing oversize (default: 0)
  - DIN 76: Undercut depth
- **`K undercut length`** (and undercut type)
  - No input: Undercut DIN 509 E
  - K=0: Undercut DIN 509 F
  - K>0: Undercut length for DIN 76
- **`E reduced feed rate:`** for machining the undercut (default: Active feed rate)

The control determines the undercut parameters from the cylinder diameter (see tables).

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutter radius compensation:** is not carried out—You can program TRC separately with G41/G42 and switch it off again with G40.
- **Oversizes:** are not taken into account.

Example: G85

```
%85.nc
[G85]
N1 T21 G95 F0.23 G96 S248 M3
N2 G0 X62 Z2
N3 G85 X60 Z-30 I0.3
N4 G1 X80
N5 G85 X80 Z-40 K0
N6 G1 X100
N7 G85 X100 Z-60 I1.2 K6 E0.11
N8 G1 X110
END
```

Undercut parameters DIN 509 E (dimensions in mm)			
Diameter	**Undercut depth I**	**Undercut length K**	**Undercut radius R**
< 18	0,25	2	0,6
> 18 - 80	0,35	2,5	0,6
> 80	0,45	4	1

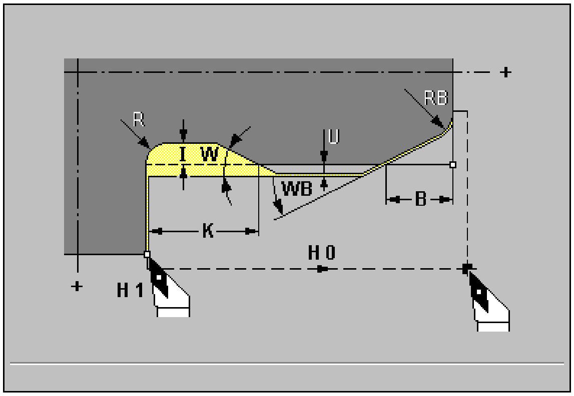
Undercut parameters DIN 509 F (dimensions in mm)				
Diameter	**Undercut depth I**	**Undercut length K**	**Undercut radius R**	**Transverse depth P**
< 18	0,25	2	0,6	0,1
> 18 - 80	0,35	2,5	0,6	0,2
> 80	0,45	4	1	0,3

Undercut angle (for undercuts according to DIN 509 E and F): 15°

Transverse angle (for undercuts according to DIN 509 F): 8°

Undercut according to DIN 509 E with cylinder machining G851

The cycle machines the adjoining cylinder, the undercut, and finishes with the plane surface. It also machines a cylinder start chamfer when you enter at least one of the parameters "B" or "RB."



Parameters

- **I undercut depth** (default: Value from standard table)
- **K undercut length** (default: Value from standard table)
- **W undercut angle** (default: Value from standard table)
- **B cylinder 1st cut length**—no input: No chamfer machined at start of cylinder
- **RB 1st cut radius**—no input: No chamfer radius is machined
- **WB 1st cut angle** (default: 45°)
- **E reduced feed rate** (default: Active feed rate): For the plunge cut and the cylinder start chamfer
- **H type of departure** (default: 0):
  - H=0: Tool returns to the starting point
  - H=1: Tool remains at the end of the plane surface
- **U finishing oversize** for the area of the cylinder (default: 0)

Note:
Parameters that are not programmed are automatically calculated from the diameter of the cylinder in the standard table (see "DIN 509 E, DIN 509 F—undercut parameters" on page 527).

Blocks following the cycle call

```
N.. G851 I.. K.. W..  /Cycle call
N.. G0 X.. Z..  /Corner point of cylinder start chamfer
N.. G1 Z..  /Undercut corner
N.. G1 X..  /End point of plane surface
N.. G80  /End of contour definition
```

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.

Example: G851

```
%851.nc
[G851]
N1 T21 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G851 I3 K15 W30 R2 B5 RB2 WB30 E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

Undercut according to DIN 509 F with cylinder machining G852

The cycle machines the adjoining cylinder, the undercut, and finishes with the plane surface. It also machines a cylinder start chamfer when you enter at least one of the parameters "B" or "RB."

Parameters

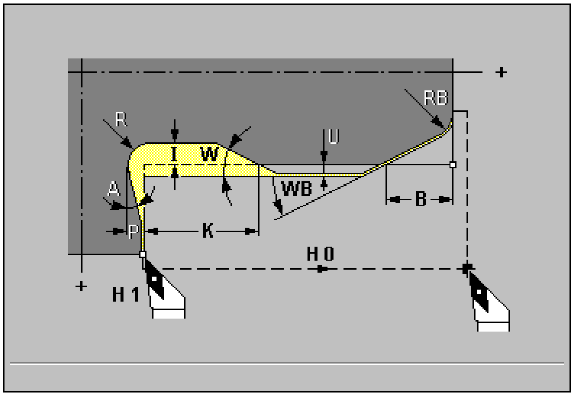
- **I undercut depth** (default: Value from standard table)
- **K undercut length** (default: Value from standard table)
- **W undercut angle** (default: Value from standard table)
- **R undercut radius** (default: Value from standard table)
- **P transverse depth** (default: Value from standard table)
- **A transverse angle** (default: Value from standard table)
- **B cylinder 1st cut length**—no input: No chamfer machined at start of cylinder
- **RB 1st cut radius**—no input: No chamfer radius is machined
- **WB 1st cut angle** (default: 45°)
- **E reduced feed rate** (default: Active feed rate): For the plunge cut and the thread chamfer
- **H type of departure** (default: 0):
  - H=0: Tool returns to the starting point
  - H=1: Tool remains at the end of the plane surface
- **U finishing oversize** for the area of the cylinder (default: 0)

Note:
Parameters that are not programmed are automatically calculated from the diameter in the standard table (see “DIN 509 E, DIN 509 F—undercut parameters” on page 527).

Blocks following the cycle call

```
N.. G852 I.. K.. W..  /Cycle call
N.. G0 X.. Z..  /Corner point of cylinder start chamfer
N.. G1 Z..  /Undercut corner
N.. G1 X..  /End point of plane surface
N.. G80  /End of contour definition
```

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.

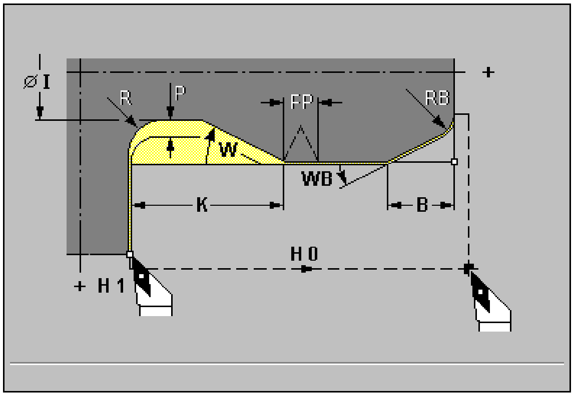


Example: G852

```
%852.nc
[G852]
N1 T21 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G852 I3 K15 W30 R2 P0.2 A8 B5 RB2 WB30
E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

Undercut according to DIN 76 with cylinder machining G853

The cycle machines the adjoining cylinder, the undercut, and finishes with the plane surface. It also machines a cylinder start chamfer when you enter at least one of the parameters "B" or "RB."



Parameters

- **FP thread pitch**
- **I undercut diameter** (diameter value) (default: Value from standard table)
- **K undercut length** (default: Value from standard table)
- **W undercut angle** (default: Value from standard table)
- **R undercut radius** (default: Value from standard table)
- **P oversize**
  - P is not defined: The undercut is machined in one pass
  - P is defined: Division into pre-turning and finish-turning
    – P = longitudinal oversize
    – The transverse oversize is preset to 0.1 mm
- **B cylinder 1st cut length**—no input: No chamfer machined at start of cylinder
- **RB 1st cut radius**—no input: No chamfer radius is machined
- **WB 1st cut angle** (default: 45°)
- **E reduced feed rate** (default: Active feed rate): For the plunge cut and the thread chamfer
- **H type of departure** (default: 0):
  - H=0: Tool returns to the starting point
  - H=1: Tool remains at the end of the plane surface

Note:
Parameters that are not programmed are automatically calculated from the standard table (see "DIN 76—undercut parameters" on page 525):

- FP from the diameter
- I, K, W, and R from FP (thread pitch)

Blocks following the cycle call

```
N.. G853 FP.. I.. K.. W..  /Cycle call
N.. G0 X.. Z..  /Corner point of cylinder start chamfer
N.. G1 Z..  /Undercut corner
N.. G1 X..  /End point of plane surface
N.. G80  /End of contour definition
```

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.

Example: G853

```
%853.nc
[G853]
N1 T21 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G853 FP1.5 I47 K15 W30 R2 P1 B5 RB2
WB30 E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

Undercut type U G856

Cycle G856 machines an undercut and finishes the adjoining plane surface. A chamfer or rounding (optional) can be machined.

Parameters

- **I undercut diameter** (diameter value)
- **K undercut length**
- **B chamfer/rounding**
  - B>0: Radius of rounding
  - B<0: Width of chamfer

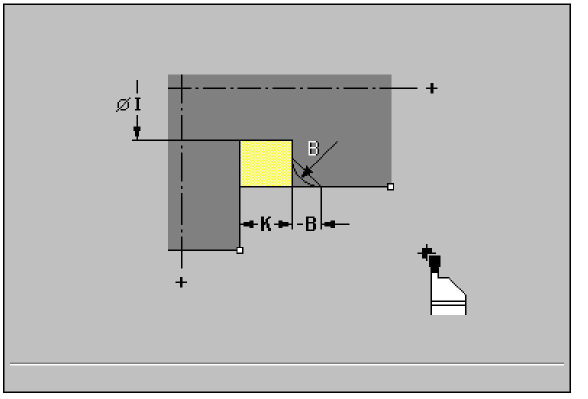
Note on the execution of the cycle:

- At the end of cycle, the tool returns to the starting point.
- If the cutting width of the tool is not defined, the control assumes that the tool's cutting width equals "K."

Blocks following the cycle call

```
N.. G856 I.. K.. /Cycle call
N.. G0 X.. Z..  /Undercut corner
N.. G1 X..  /End point of plane surface
N.. G80  /End of contour definition
```

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.



Example: G856

```
%856.nc
[G856]
N1 T30 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G856 I47 K7 B1
N4 G0 X50 Z-30
N5 G1 X60
N6 G80
END
```

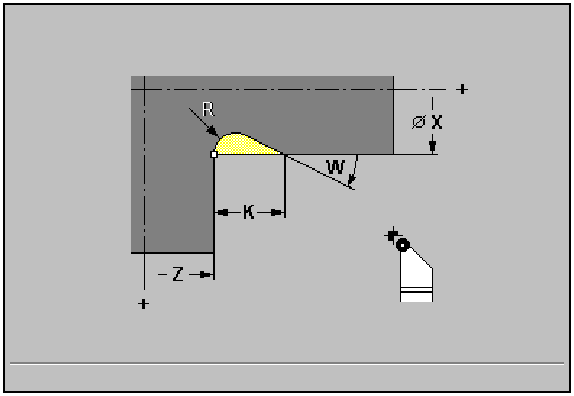
Undercut type H G857

The cycle G857 machines an undercut. The end point is determined from the plunging angle in accordance with "Undercut type H." At the end of cycle, the tool returns to the starting point.

Parameters

- **X contour corner** (diameter value)
- **Z contour corner**
- **K undercut length**
- **R radius**—no input: No circular element (tool radius = undercut radius)
- **W plunge angle**—no input: W is determined from "K" and "R."

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.



Example: G857

```
%857.nc
[G857]
N1 T21 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G857 X50 Z-30 K7 R2 W30
END
```

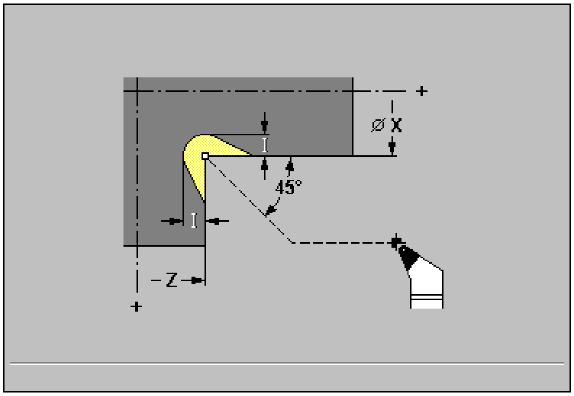
Undercut type K G858

The cycle G858 machines an undercut. This cycle performs only one cut at an angle of 45°. The resulting contour geometry therefore depends on the tool that is used. At the end of cycle, the tool returns to the starting point.

Parameters

- **X contour corner** (diameter value)
- **Z contour corner**
- **I undercut depth**

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.
- **Cutting radius compensation:** Active.
- **Oversizes:** are not taken into account.



Example: G858

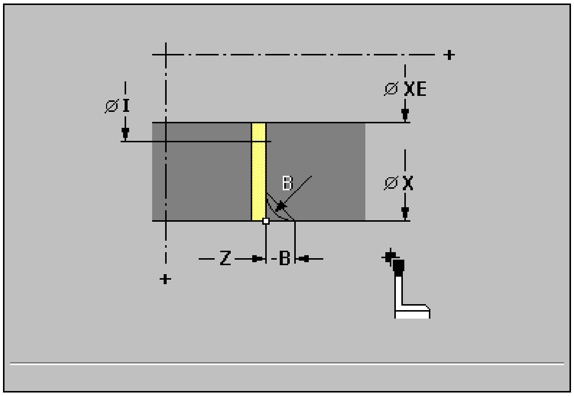
```
%858.nc
[G858]
N1 T9 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G858 X50 Z-30 I0.5
END
```

6.17 Parting Cycle

Parting cycle G859

The cycle G859 parts the workpiece. If programmed, a chamfer or rounding arc is machined on the outside diameter. At the end of cycle, the tool retracts and returns to the starting point.

You can define a feed rate reduction after position "I."



Parameters

- **`X parting diameter`**
- **`Z parting position`**
- **`I diameter for feed reduction`**
  - I is defined: The control switches to feed rate "E" after this position.
  - I is not defined: No feed rate reduction
- **`XE inside diameter`** (tube)
- **`E reduced feed rate`**—default: Active feed rate
- **`B chamfer/rounding`**
  - B>0: Radius of rounding
  - B<0: Width of chamfer

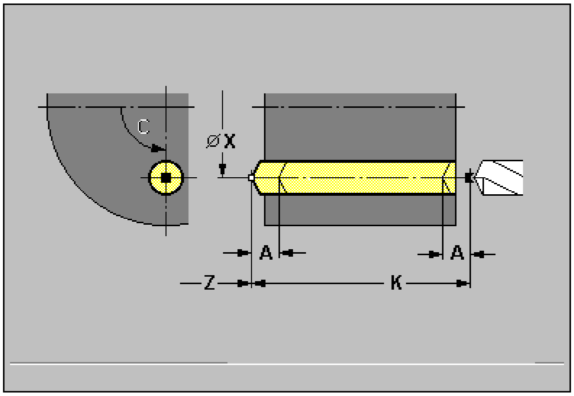
Example: G859

```
%859.nc
[G859]
N1 T30 G95 F0.23 G96 S248 M3
N2 G0 X60 Z-28
N3 G859 X50 Z-30 I10 XE8 E0.11 B1
END
```

6.18 Drilling Cycles

Drilling cycle G71

You can use cycle G71 with stationary tools for drilling axial holes in the turning center and with driven tools for drilling axial and radial holes.



Parameters

- **X end point** of axial hole (diameter value)
- **Z end point** of radial hole
- **A drilling lengths** (default: 0)
- **E dwell time** for chip breaking at end of hole (default: 0)
- **V drilling variants**—Feed rate reduced by 50% during both pre-drilling and through-boring
  - 0: No feed rate reduction
  - 1: Feed reduction for through-boring
  - 2: Feed reduction for pre-drilling
  - 3: Feed reduction for pre-drilling and through-boring
- **K drilling depth** (radial holes: radius)
  - K is defined: The starting point of the hole is calculated from the hole end point and "K."
  - K is not defined: "K" is calculated from the hole end point and the current tool position.
- **D retreat**—default: 0
  - 0: Rapid traverse
  - 1: Feed rate

Notes:

- The control starts execution of the cycle at the current tool and spindle position. The starting point is approached at rapid traverse.
- **Axial hole:**
  - Do not program "X."
  - Define "Z."
- **Radial hole:**
  - Define "X."
  - Do not program "Z."
- X **and** Z are programmed: The control uses the "tool orientation" to decide whether a radial or an axial hole is machined (see “Drilling tools” on page 423).

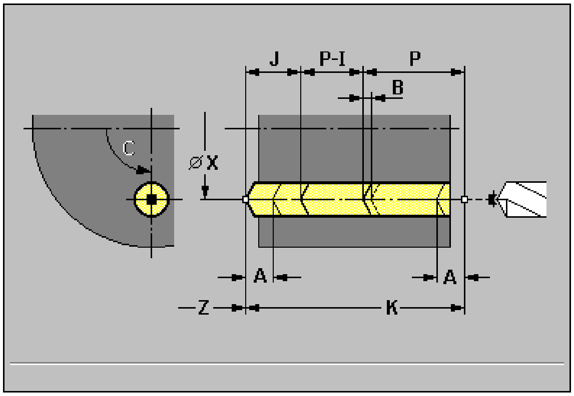
Example: G71

```
%71.nc
[G71]
N1 T50 G97 S1000 G95 F0.2 M3
N2 G0 X0 Z5
N3 G71 Z-25 A5 V2
END
```

Deep-hole drilling cycle G74

You can use cycle G74 with stationary tools for drilling axial holes in the turning center and with driven tools for drilling axial and radial holes.

The hole is drilled in several passes. After each pass, the drill retracts and advances again to the first drilling depth, minus the "safety clearance." The drilling depth is reduced with each subsequent pass.



Parameters

- **X end point** of axial hole (diameter value)
- **Z end point** of radial hole
- **R safety clearance** no input: Value from "Current parameters—Machining—Safety distances"
- **P 1st hole depth**—no input: Hole will be drilled in one pass
- **I reduction value** (default: 0): With each subsequent pass, the drilling depth is reduced by "I," however, without falling below "J."
- **B return distance** (default: Retract to starting point of hole)
- **J minimum hole depth** (default: 1/10 of P)
- **A drilling lengths** (default: 0)
- **E dwell time** for chip breaking at end of hole (default: 0)
- **V drilling variants**—Feed rate reduced by 50% during both pre-drilling and through-boring
  - 0: No feed rate reduction
  - 1: Feed reduction for through-boring
  - 2: Feed reduction for pre-drilling
  - 3: Feed reduction for pre-drilling and through-boring
- **K drilling depth** (radial holes: radius)
  - K is defined: The starting point of the hole is calculated from the hole end point and "K."
  - K is not defined: "K" is calculated from the hole end point and the current tool position.
- **D retreat** retraction speed and infeed within the hole—default: 0
  - 0: Rapid traverse
  - 1: Feed rate

Example: G74

```
%74.nc
[G74]
N1 M5
N2 T49 G197 S1000 G195 F0.2 M103
N3 M14
N4 G110 C0
N5 G0 X80 Z2
N6 G745 XK0 YK0 Z2 K80 Wi90 Q4 V2
N7 G74 Z-40 R2 P12 I2 B0 J8
N8 M15
END
```

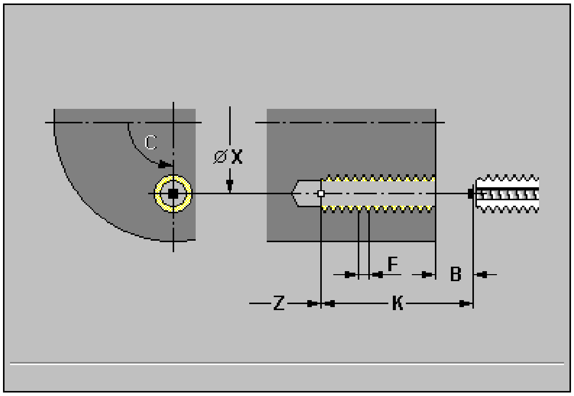
Notes:

- The control starts execution of the cycle at the current tool and spindle position. The starting point is approached at rapid traverse.
- **Axial hole:**
  - Do not program "X."
  - Define "Z."
- **Radial hole:**
  - Define "X."
  - Do not program "Z."
- X **and** Z are programmed: The control uses the "tool orientation" to decide whether a radial or an axial hole is machined (see “Drilling tools” on page 423).

Tapping G36

You can use cycle G36 with stationary tools for cutting axial threads in the turning center and with driven tools for cutting axial and radial threads.

Meaning of "retraction length J": Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the "retract length." The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the drill is pulled away from the chuck by the "retraction length." With this method you can achieve higher service life from the taps.



Parameters

- **X end point** of thread for axial machining (diameter value)
- **Z end point** of thread for radial machining
- **F feed per revolution:** Thread pitch
- **B run-in length** (default: 2 * thread pitch F1): Distance for reaching the programmed spindle speed and feed rate
- **Q number of spindle**
  - Q=0: Master spindle (stationary tool)
  - Q=1: Driven tool
- **H reference direction** for thread pitch (default: 0)
  - H=0: Feed rate on the Z axis
  - H=1: Feed rate on the X axis
- **S retraction speed** (default: Same spindle speed as for tapping)
- **K drilling depth** (radial holes: radius)
  - K is defined: The starting point of the hole is calculated from the hole end point and "K."
  - K is not defined: "K" is calculated from the hole end point and the current tool position.
- **Retraction length J** (default: 0) when using floating tap holders

Notes:

- The control starts execution of the cycle at the current tool and spindle position. The starting point is approached at rapid traverse.
- **Axial hole:**
  - Do not program "X."
  - Define "Z."
- **Radial hole:**
  - Define "X."
  - Do not program "Z."
- X **and** Z are programmed: The control uses the "tool orientation" to decide whether a radial or an axial hole is machined (see “Tapping tools” on page 424).

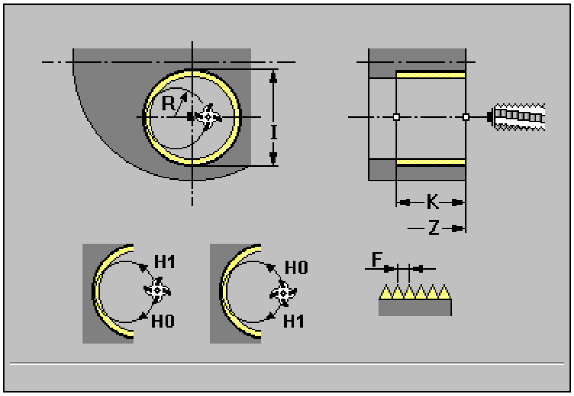
Example: G36

```
%36.nc
[G36]
N1 T50 G97 S1000 G95 F0.2 M3
N2 G0 X0 Z5
N3 G71 Z-30
N4 G14 Q0
N5 T51 G97 S600 M3
N6 G0 X0 Z8
N7 G36 Z-25 F1.5 B3 Q0
END
```

Thread milling, axial G799

The cycle mills a thread in existing holes.

Place the tool on the center of the hole before calling G799. The cycle positions the tool on the end point of the thread within the hole. The tool then approaches on "approaching radius R," mills the thread in a rotation of 360°, while advancing by "F." Following that, the cycle retracts the tool and returns it to the starting point.



Parameters

- **I inside diameter of thread**
- **Z starting point** of thread
- **K thread depth**
- **R approaching radius**—default: R = (I – milling diameter) / 2
- **F thread pitch**
- **J left-hand, right-hand** (default: 0): Direction of thread
  - J=0: Right-hand
  - J=1: Left-hand
- **H cutting direction** (default: 0)
  - H=0: Up-cut milling
  - H=1: Climb milling

Use thread-milling tools for cycle G799.

Danger of collision!

Be sure to consider the hole diameter and the diameter of the milling cutter when programming "approaching radius R."

Example: G799

```
%799.nc
[G799]
N1 T70 G195 F0.2 G197 S800
N2 G0 X100 Z2
N3 M14
N4 G110 Z2 C45 X100
N5 G799 I12 Z0 K-20 F2 J0 H0
N6 M15
END
```

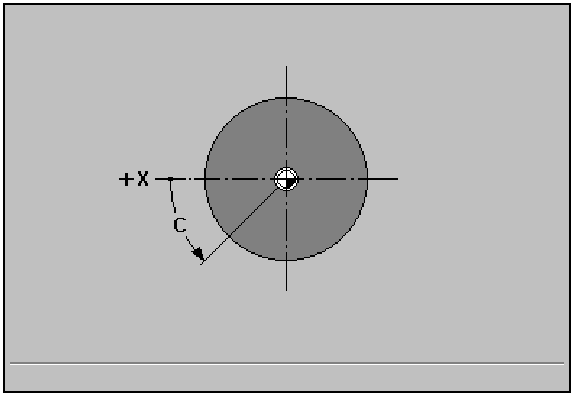
6.19 C-Axis Commands

Zero point shift, C axis G152

G152 defines an absolute zero point for the C axis (reference: machine parameter 1005, "Reference point, C axis"). The zero point is valid until the end of the program.

Parameters

- **C angle:** Spindle position of "new" C-axis zero point



Example: G152

```
%152.nc
[G152]
N1 M5
N2 T71 G197 S1010 G193 F0.08 M104
N3 M14
N4 G152 C30
N5  G110 C0
N6 G0 X122 Z-50
N7 G744 X122 Z-50 ZE-50 C0 Wi90 Q4
N8 G792 K30 A0 X100 J11 P5 F0.15
N9 M15
END
```

Standardize C axis G153

G153 resets a traverse angle >360° or <0° to the corresponding angle modulo 360°—without moving the C axis.

> G153 is only used for lateral-surface machining. An automatic modulo 360° function is carried out on the face.

6.20 Face Machining

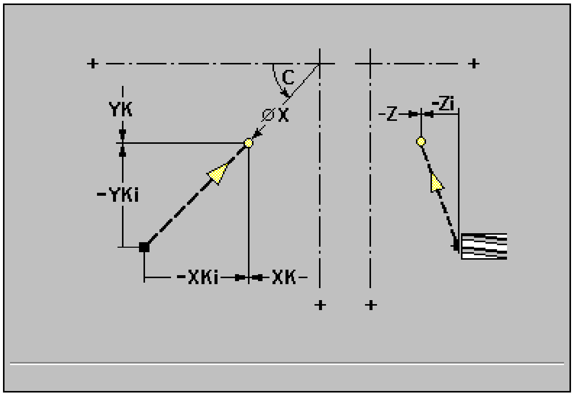
Starting point of contour / rapid traverse G100

Geometry command: G100 defines the starting point of a contour on the face.

Machining command: The tool moves at rapid traverse along the shortest path to the end point.

Parameters

- **X end point** (diameter value)
- **C end angle**—for angle direction, see graphic support window
- **XK end point** (Cartesian coordinates)
- **YK end point** (Cartesian coordinates)
- **Z end point**



Danger of collision!
During G100 the tool moves on a linear path—even if you program "C" only. To position the workpiece to a defined angle, use G110.

- Define the contour starting point or end point either in polar or Cartesian coordinates.
- Permitted as machining command only for G100: Parameter Z

Example: G100

```
%100.nc
[G100, G101, G102, G103]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G793 Z2 ZE-5 P2 U0.5 R0 I0.5 F0.15 H0 Q0
N6 G100 XK20 YK5
N7 G101 XK50 B5
N8 G103 XK5 YK50 R50 Q1 B5
N9 G101 XK5 YK20 B5
N10 G102 XK20 YK5 R20 B5
N11 G80
N12 M15
END
```

Linear segment, face G101

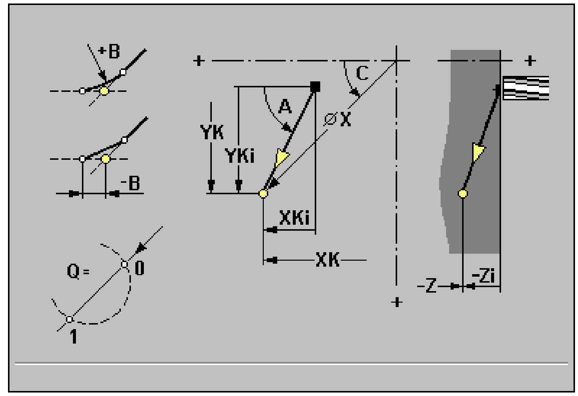
Geometry command: G101 defines a linear segment in a contour on the face.

Machining command: The tool moves on a linear path at feed rate to the end point.

Parameters

- **X end point** (diameter value)
- **C end angle**—for angle direction, see graphic support window
- **XK end point** (Cartesian coordinates)
- **YK end point** (Cartesian coordinates)
- **Z end point**
- **A angle** to positive XK axis
- **Q point of intersection** (default: Q=0): If entered data permit two possible solutions for the end point, "Q" defines the end point.
- **B chamfer/rounding arc:** Transition to the next contour element When entering a chamfer/rounding, program the theoretical end point of the contour element.
  - B no input: Tangential transition
  - B=0: No tangential transition
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **Z end point**

> - Define the end point either in polar or Cartesian coordinates.
> - Permitted as geometry command only for G101: Parameters Q, B
> - Permitted as machining command only for G101: Parameter Z



Example: G101

```
%101.nc
[G101, G102, G103]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z2
N5 G100 XK50 YK0
N6  G1 Z-5
N7  G42 Q1
N8 G101 XK40
N9 G101 YK30
N10 G103 XK30 YK40 R10
N11 G101 XK-30
N12 G103 XK-40 YK30 R10
N13 G101 YK-30
N14 G103 XK-30 YK-40 R10
N15 G101 XK30
N16 G103 XK40 YK-30 R10
N17 G101 YK0
N18 G100 XK110 G40
N19 G0 X120 Z50
N20 M15
END
```

Circular arc, face G102/G103

Geometry command: G102/G103 defines a circular arc in a contour on the face.

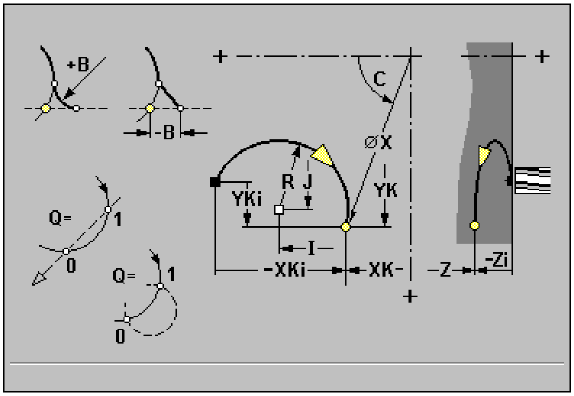
Machining command: The tool moves on a circular arc at feed rate to the end point.

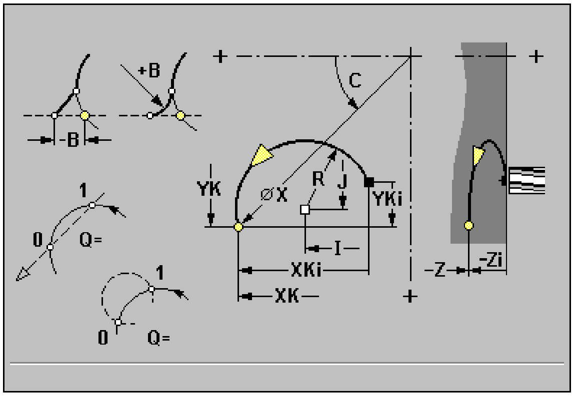
The direction of rotation is shown in the graphic support window.

Parameters

- `X end point` (diameter value)
- `C end angle`—for angle direction, see graphic support window
- `XK end point` (Cartesian coordinates)
- `YK end point` (Cartesian coordinates)
- `R radius`
- `I center point` (Cartesian coordinates)
- `K center point` (Cartesian coordinates)
- `Q point of intersection` (default: Q=0): If entered data permit two possible solutions for the end point, "Q" defines the end point.
- `B chamfer/rounding arc:` Transition to the next contour element When entering a chamfer/rounding, program the theoretical end point of the contour element.
  - B no input: Tangential transition
  - B=0: No tangential transition
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- `Z end point`

> - Define the end point either in polar or Cartesian coordinates.
> - End point in the coordinate origin: Program XK=0, YK=0.
> - Program either center or radius.
> - If you do not program the center, MANUALplus automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.
> - Permitted as geometry command only for G102/G103: Parameters Q, B
> - Permitted as machining command only for G102/G103: Parameter Z



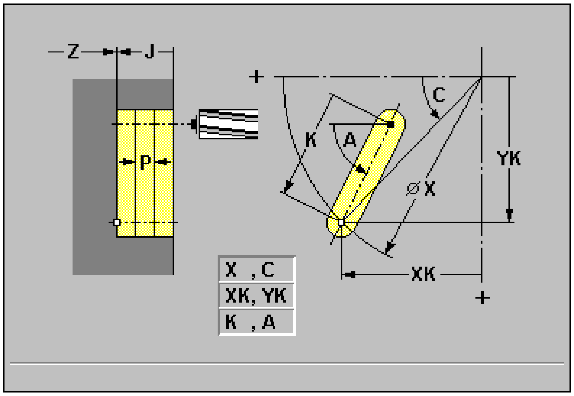


Example: G102, G103

```
%100.nc
[G100, G101, G102, G103, G793]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G793 Z2 ZE-5 P2 U0.5 R0 I0.5 F0.15 H0 Q0
N6 G100 XK20 YK5
N7 G101 XK50 B5
N8 G103 XK5 YK50 R50 Q1 B5
N9 G101 XK5 YK20 B5
N10 G102 XK20 YK5 R20 B5
N11 G80
N12 M15
END
```

Linear slot, face G791

G791 mills a slot from the current tool position to the end point. The slot width equals the diameter of the milling cutter. Oversizes are not taken into account.



Parameters

- **X diameter**—end point of slot
- **C end angle**—end point of slot—for angle direction, see graphic support window
- **XK end point** of slot (Cartesian coordinates)
- **YK end point** of slot (Cartesian coordinates)
- **K length** of slot—referenced to the cutter center
- **A angle** of slot—reference: see graphic support window
- **Z milling floor**
- **J milling depth**
  - J is defined: The tool approaches to safety clearance and then mills the slot.
  - J is not defined: The milling cycle starts from the tool position.
- **P maximum infeed** (default: Total depth in one infeed)
- **F feed rate** for infeed (default: Active feed rate)

Possible **parameter combinations** for definition of the end point:

- Diameter X, end angle C
- End point XK, YK
- Slot length K, angle A

Notes:

- Rotate the spindle to the desired angle position **before** calling G791.
- If you use a spindle positioning device (no C axis), an axial slot is machined centrically to the rotary axis.

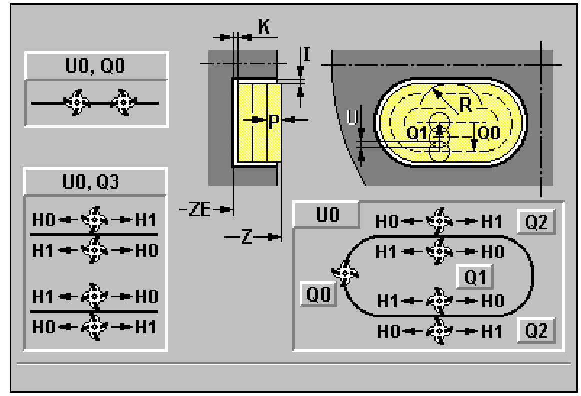
Example: G791

```
%791.nc
[G791]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G100 XK20 YK5
N6 G791 XK30 YK5 Z-5 J5 P2
N7 M15
END
```

Contour and figure milling cycle, face G793

G793 mills figures or (open or closed) "independent" contours on the face. G793 is followed by:

- **The figure to be milled** with:
  - Circle (G304), rectangle (G305) or polygon (G307)
  - Conclusion of milling contour (G80)
- **The independent contour** with:
  - Starting point of milling contour (G100)
  - Milling contour (G101, G102, G103)
  - Conclusion of milling contour (G80)



Parameters

- **`Z milling top edge`**
- **`ZE milling floor`**
- **`P maximum infeed`** (default: Total depth in one infeed)
- **`U overlap factor:`** Machining of contour or pocket (default: 0)
  - U=0: Contour milling
  - U>0: Pocket milling—minimum overlap of milling paths = U*milling diameter
- **`R approaching radius:`** (Radius of approaching/departing arc)—(default: 0)
  - R=0: Contour element is approached directly; feed to starting point above the milling plane—then vertical plunge.
  - R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially.
- **`I oversize`** contour-parallel
- **`K oversize Z`** (in infeed direction)
- **`F feed rate`** for infeed (default: Active feed rate)
- **`E reduced feed rate`** for circular elements (default: Active feed rate)
- **`H cutting direction`** (default: 0): The **cutting direction** (see graphic support window) can be changed with H and the direction of tool rotation.
  - H=0: Up-cut milling
  - H=1: Climb milling

Example: G793

```
%100.nc
[G100, G101, G102, G103, G793]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G793 Z2 ZE-5 P2 U0.5 R0 I0.5 F0.15 H0 Q0
N6 G100 XK20 YK5
N7 G101 XK50 B5
N8 G103 XK5 YK50 R50 Q1 B5
N9 G101 XK5 YK20 B5
N10 G102 XK20 YK5 R20 B5
N11 G80
N12 M15
END
```

- **`Q cycle type`** (default: 0): Depending on "U," the following applies:
  - **Contour milling (U=0):**
    – Q=0: Milling center on the contour
    – Q=1—closed contour: Inside milling
    – Q=1—open contour: Left in machining direction
    – Q=2—closed contour: Outside milling
    – Q=2—open contour: Right in machining direction
    – Q=3—open contour: Milling location depends on "H" and the direction of tool rotation—see graphic support window
  - **Pocket milling (U>0):**
    – Q=0: From the inside toward the outside
    – Q=1: From the outside toward the inside
- O roughing/finishing—default: 0
  - O=0: Roughing
  - O=1: Finishing—first, the edge of the pocket is machined; then the pocket floor is machined.

Notes:

Milling depth: The cycle calculates the depth from "Z" and "ZE," taking the oversizes into account.

Milling cutter radius compensation: effective (except for contour milling with Q=0).

Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For contour milling and finishing (pocket milling), define with "R" whether the tool is to approach directly or in an arc.

Oversizes are taken into account if I, K are not programmed:

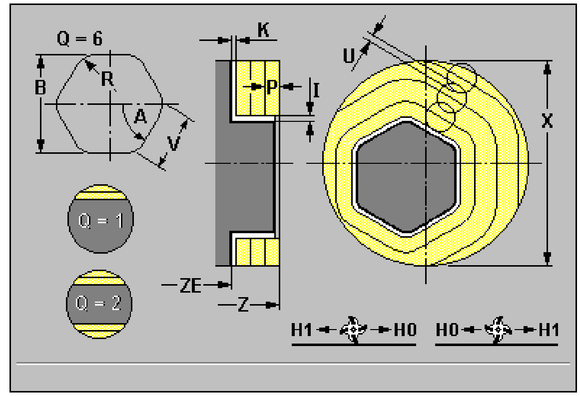
- G57: Oversize in X, Z direction
- G58: The oversize "shifts" the milling contour as follows:
  - With inside milling and closed contour: The contour is contracted
  - With outside milling and closed contour: The contour is expanded
  - With open contour and Q=1: Left in machining direction
  - With open contour and Q=2: Right in machining direction

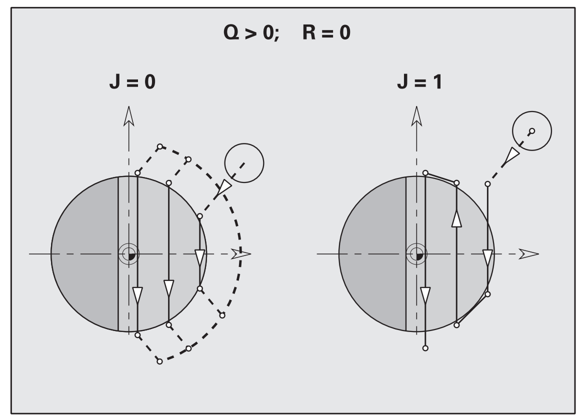
Area milling, face G797

Depending on "Q," G797 mills surfaces, polygons or the figure defined in the command following G797.

Parameters

- **X limiting diameter**
- **Z milling top edge**
- **ZE milling floor**
- **B width across flats** (omit for Q=0): B defines the remaining material. For an even number of surfaces, you can program "B" as an alternative to "V."
  - Q=1: Remaining thickness
  - Q>=2: Width across flats
- **V edge length**—omit for Q=0
- **R chamfer/rounding arc**—omit for Q=0
  - R<0: Chamfer length
  - R>0: Rounding arc
- **A slope angle** (reference: see graphic support window)—omit for Q=0
- **Q number of surfaces** (default: 0): Range: 0 <= Q <= 127
  - Q=0: G797 is followed by a figure definition
  - Q=1: One surface
  - Q=2: Two surfaces offset by 180°
  - Q=3: Triangle
  - Q=4: Rectangle, square
  - Q>4: Polygon
- **P maximum infeed** (default: Total depth in one infeed)
- **U overlap factor** (default: 0.5): Minimum overlap of milling paths = U*milling diameter
- **I oversize** contour-parallel
- **K oversize Z** (in infeed direction)
- **F feed rate** for infeed (default: Active feed rate)
- **E reduced feed rate** for circular elements (default: Active feed rate)
- **H cutting direction** (default: 0): The **cutting direction** (see graphic support window) can be changed with H and the direction of tool rotation.
  - H=0: Up-cut milling
  - H=1: Climb milling





Example: G797

```
%797.nc
[G797]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G797 X100 Z0 ZE-5 B50 R2 A0 Q4 P2 U0.5
N6 G100 Z2
N7 M15
END
```

- **O roughing/finishing** (default: 0)
  - O=0: Roughing
  - O=1: Finishing
- **J milling direction:** For polygons without chamfers/roundings, J defines whether a unidirectional or bidirectional milling operation is to be executed.
  - J=0: Unidirectional
  - J=1: Bidirectional

Notes:

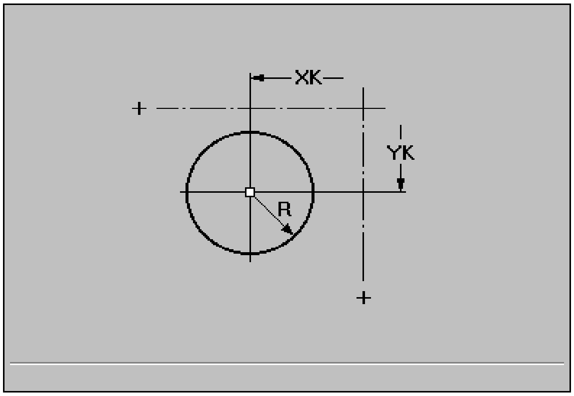
- With "Q=0," one of the following figures is programmed in the subsequent command. A G80 is programmed after the command.
  - G304—circle
  - G305—rectangle
  - G307—polygon
- A polygon that has been defined with G797 (Q>0) is in the center. A figure defined in the subsequent command can be **outside the center.**
- The cycle calculates the milling depth from "Z" and "ZE," taking the oversizes into account.

Figure definition: Full circle, face G304

G304 defines a full circle on the face. Program this figure in conjunction with G793 or G797.

Parameters

- **XK** center
- **YK** center
- **R** radius of circle



Example: G304

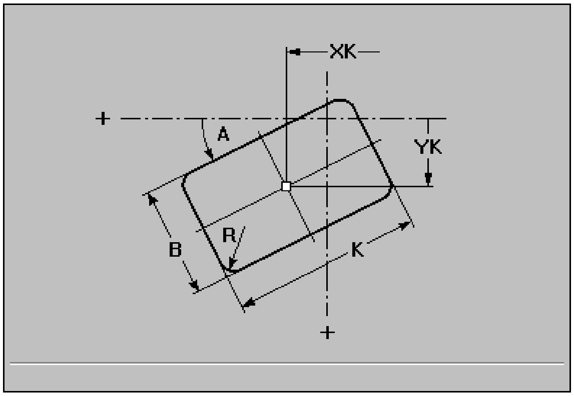
```
%304.nc
[G304]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G793 Z2 ZE-5 P2 U0.5 R0 I0.5 F0.15 H0 Q0
N6 G304 XK20 YK5 R20
N7 G80
N8 M15
END
```

Figure definition: Rectangle, face G305

G305 defines a rectangle on the face. Program this figure in conjunction with G793 or G797.

Parameters

- **XK center**
- **YK center**
- **A angle**—reference: see graphic support window
- **K length** of rectangle
- **B height** of rectangle
- **R chamfer/rounding**
  - R<0: Chamfer length
  - R>0: Rounding arc

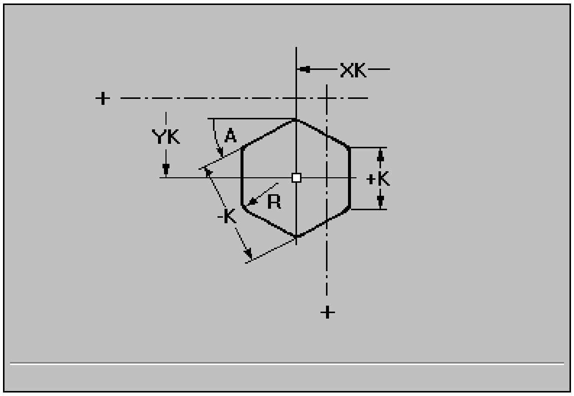


Example: G305

```
%305.nc
[G305]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G793 Z2 ZE-5 P2 U0.5 R0 I0.5 F0.15 H0 Q0
N6 G305 XK20 YK5 A0 K15 B10 R2
N7 G80
N8 M15
END
```

Figure definition: Eccentric polygon, face G307

G307 defines a polygon on the face. Program this figure in conjunction with G793 or G797.



Parameters

- **XK center**
- **YK center**
- **Q number of edges:** Range: 3 <= Q <= 127
- **A angle**—reference: see graphic support window
- **K width across flats (SW) / length**
  - K<0: Width across flats (inside diameter)
  - K>0: Edge length
- **R chamfer/rounding**
  - R<0: Chamfer length
  - R>0: Rounding arc

Example: G307

```
%307.nc
[G307]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G793 Z2 ZE-5 P2 U0.5 R0 I0.5 F0.15 H0 Q0
N6 G307 XK20 YK5 Q4 A0 K-10 R2
N7 G80
N8 M15
END
```

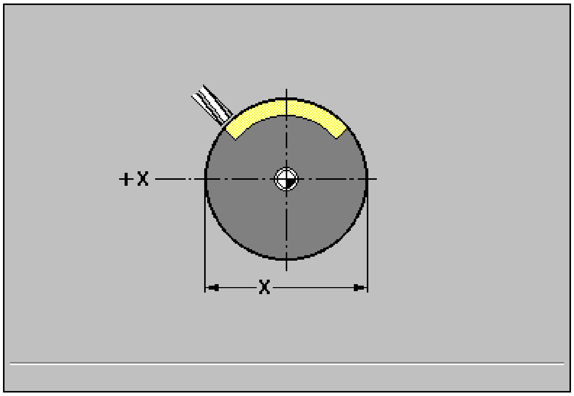
6.21 Lateral Surface Machining

Reference diameter G120

G120 determines the reference diameter of the unrolled lateral surface. Program G120 if you use "CY" for G110 to G113. G120 is a modal function.

Parameters

- `X diameter`



Example: G120

```
%111.nc
[G111, G120]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N6 G41 Q2 H0
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635
N12 G40
N13 G110 X105
N14 M15
END
```

Starting point of contour / rapid traverse G110

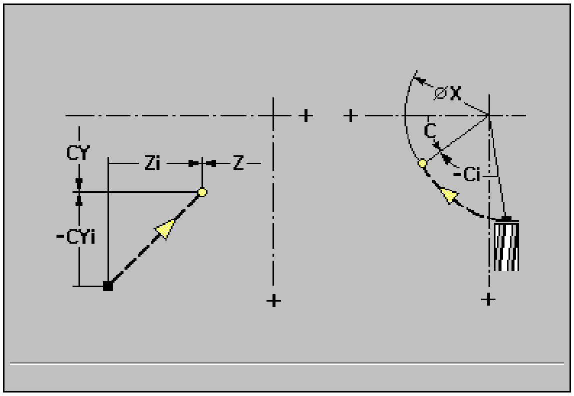
Geometry command: G110 defines the starting point of contour definition on the lateral surface.

Machining command: The tool moves at rapid traverse along the shortest path to the end point.

Parameters

- `Z end point`
- `C end angle`
- `CY end point` as linear value (reference: G120 reference diameter)
- `X end point` (diameter value)—(default: Current X position)

- Define the contour starting point or end point either with "C" or "CY."
- Use **G110** to position the C axis to a defined angle (programming: N.. G110 C...).
- Permitted as machining command only for G110: Parameter X



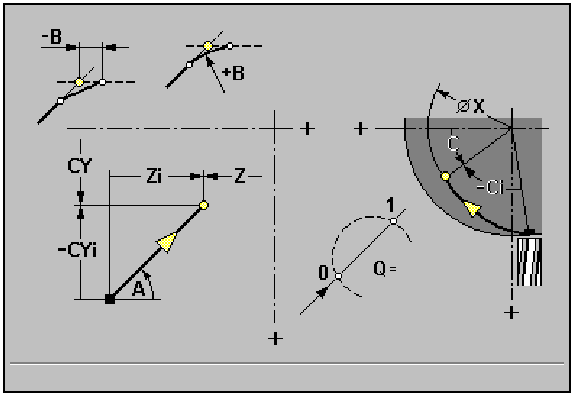
Example: G110

```
%110.nc
[G110, G111, G113, G794]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N6 G794 X100 XE97 P2 U0.5 R0 K0.5 F0.15 H0 Q0
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635 B0
N12 G80
N13 M15
END
```

Linear segment, lateral surface G111

Geometry command: G111 defines a linear segment in a contour on the lateral surface.

Machining command: The tool moves on a linear path at feed rate to the end point.



Parameters

- **Z end point**—default: Current Z position
- **C end angle**—for angle direction, see graphic support window
- **CY end point** as linear value (reference: G120 reference diameter)
- **A angle**—reference: see graphic support window
- **Q point of intersection** (default: Q=0): If entered data permit two possible solutions for the end point, "Q" defines the end point.
- **B chamfer/rounding arc:** Transition to the next contour element When entering a chamfer/rounding, program the theoretical end point of the contour element.
  - B no input: Tangential transition
  - B=0: No tangential transition
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **X end point:** infeed (diameter value)—(default: Current X position)

- Define the end point either with "C" or "CY."
- Permitted as geometry command only for G111: Parameters Q, B
- Permitted as machining command only for G111: Parameter X

Example: G111

```
%111.nc
[G111, G120]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N6 G41 Q2 H0
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635
N12 G40
N13 G110 X105
N14 M15
END
```

Circular arc, lateral surface G112/G113

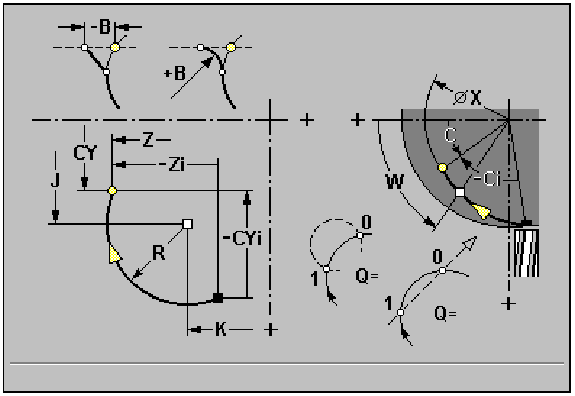
Geometry command: G112/G113 defines a circular arc in a contour on the lateral surface.

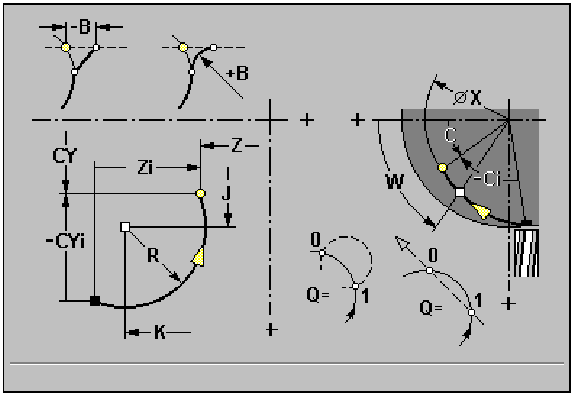
Machining command: The tool moves on a circular arc at feed rate to the end point.

The direction of rotation is shown in the graphic support window.

Parameters

- **Z end point** (default: Current Z position)
- **C end angle**—for angle direction, see graphic support window
- **CY end point** as linear value (reference: G120 reference diameter)
- **R radius**
- **K center point**
- **J center point** as a linear value (reference: unrolled G120 reference diameter)
- **W center point:** Angle to center—for angle direction, see graphic support window
- **Q point of intersection** (default: Q=0): If entered data permit two possible solutions for the end point, "Q" defines the end point.
- **B chamfer/rounding arc:** Transition to the next contour element When entering a chamfer/rounding, program the theoretical end point of the contour element.
  - B no input: Tangential transition
  - B=0: No tangential transition
  - B>0: Radius of rounding
  - B<0: Width of chamfer
- **X end point:** infeed (diameter value)—(default: Current X position)





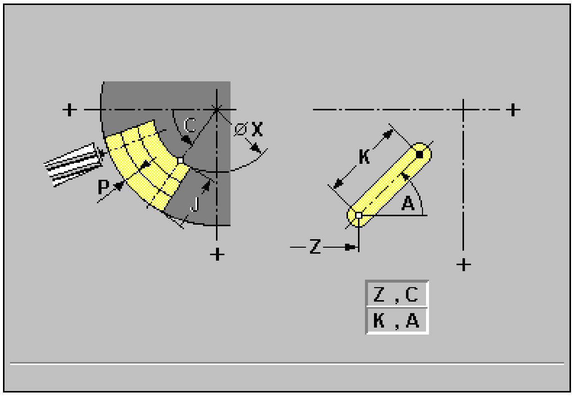
- Define the center or end point either with "C/W" or "CY/J."
- Program either center or radius.
- If you do not program the center, MANUALplus automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.
- Permitted as geometry command only for G112/G113: Parameters Q, B
- Permitted as machining command only for G112/G113: Parameter X

Example: G112, G113

```
%110.nc
[G110, G111, G113, G794]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N6 G794 X100 XE97 P2 U0.5 R0 K0.5 F0.15 H0
Q0
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635 B0
N12 G80
N13 M15
END
```

Linear slot, lateral surface G792

G792 mills a slot from the current tool position to the end point. The slot width equals the diameter of the milling cutter. Oversizes are not taken into account.



Parameters

- `Z end point`
- `C end angle`
- `K length` of slot—referenced to the cutter center
- `A angle` of slot—reference: see graphic support window
- `X milling floor` (diameter value)
- `J milling depth`
  - J is defined: The tool approaches to safety clearance and then mills the slot.
  - J is not defined: The milling cycle starts from the tool position.
- `P maximum infeed` (default: Total depth in one infeed)
- `F feed rate` for infeed (default: Active feed rate)

Possible **parameter combinations** for definition of the end point:

- End point Z, end angle C
- Slot length K, angle A

Notes:

- Rotate the spindle to the desired angle position **before** calling G792.
- If you use a spindle positioning device (no C axis), a radial slot is machined parallel to the Z axis.

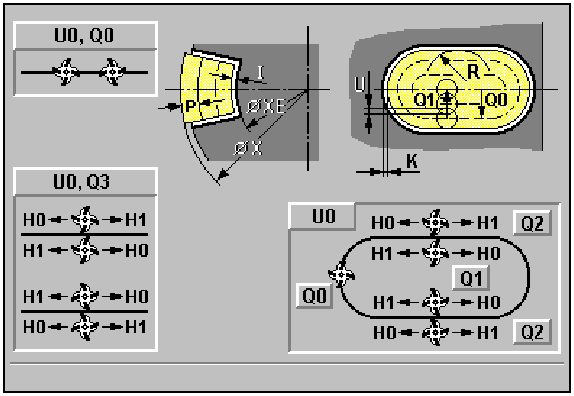
Example: G792

```
%792.nc
[G792]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z5
N5 G0 X102 Z-30
N6 G792 K25 A45 X97 J3 P2 F0.15
N7 M15
END
```

Contour and figure milling cycle, lateral surface G794

G794 mills figures or (open or closed) "independent" contours on the lateral surface. G794 is followed by:

- **The figure to be milled** with:
  - Circle (G314), rectangle (G315) or polygon (G317)
  - Conclusion of contour definition (G80)
- **The independent contour** with:
  - Starting point (G110)
  - Contour definition (G111, G112, G113)
  - Conclusion of contour definition (G80)



Parameters

- **X milling top edge**
- **XE milling floor**
- **P maximum infeed** (default: Total depth in one infeed)
- **U overlap factor:** Machining of contour or pocket (default: 0)
  - U=0: Contour milling
  - U>0: Pocket milling—minimum overlap of milling paths = U*milling diameter
- **R approaching radius:** (Radius of approaching/departing arc)—(default: 0)
  - R=0: Contour element is approached directly; feed to starting point above the milling plane—then vertical plunge.
  - R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element.
  - R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially.
- **K oversize** contour-parallel
- **I oversize X** (in infeed direction)
- **F feed rate** for infeed (default: Active feed rate)
- **E reduced feed rate** for circular elements (default: Active feed rate)
- **H cutting direction** (default: 0): The **cutting direction** (see graphic support window) can be changed with H and the direction of tool rotation.
  - H=0: Up-cut milling
  - H=1: Climb milling

Example: G794

```
%110.nc
[G110, G111, G113, G794]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G120 X100
N4 G110 C0
N5 G0 X110 Z5
N6 G794 X100 XE97 P2 U0.5 R0 K0.5 F0.15 H0 Q0
N7 G110 Z-20 CY0
N8 G111 Z-40
N9 G113 CY39.2699 K-40 J19.635
N10 G111 Z-20
N11 G113 CY0 K-20 J19.635 B0
N12 G80
N13 M15
END
```

- **Q cycle type** (default: 0): Depending on "U," the following applies:
  - **Contour milling (U=0):**
    – Q=0: Milling center on the contour
    – Q=1—closed contour: Inside milling
    – Q=1—open contour: Left in machining direction
    – Q=2—closed contour: Outside milling
    – Q=2—open contour: Right in machining direction
    – Q=3—open contour: Milling location depends on "H" and the direction of tool rotation—see graphic support window
  - **Pocket milling (U>0):**
    – Q=0: From the inside toward the outside
    – Q=1: From the outside toward the inside
- **O roughing/finishing**—default: 0
  - O=0: Roughing
  - O=1: Finishing—first, the edge of the pocket is machined; then the pocket floor is machined.

Notes:

Milling depth: The cycle calculates the depth from "Z" and "ZE," taking the oversizes into account.

Milling cutter radius compensation: effective (except for contour milling with Q=0).

Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For contour milling and finishing (pocket milling), define with "R" whether the tool is to approach directly or in an arc.

Oversizes are taken into account if I, K are not programmed:

- G57: Oversize in X, Z direction
- G58: The oversize expands or contracts the contour to be milled, depending on your definition of the "cycle type." The "outside milling" cycle type expands the contour. For open contours, the contour is shifted to the left or right, depending on the cycle type.
  - With inside milling and closed contour: The contour is contracted
  - With outside milling and closed contour: The contour is expanded
  - With open contour and Q=1: Left in machining direction
  - With open contour and Q=2: Right in machining direction

Helical-slot milling G798

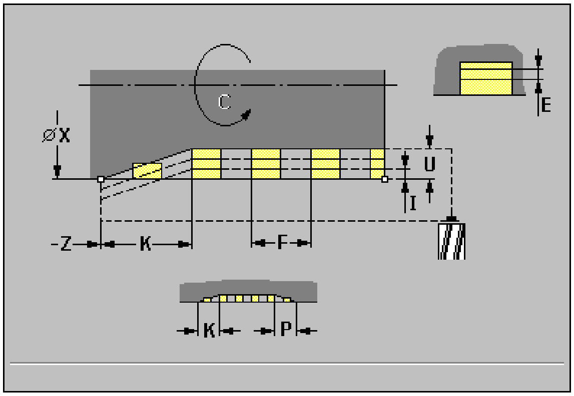
G798 mills a helical slot from the current tool position to end point X, Z. The slot width equals the diameter of the milling cutter.

For the first infeed, "I" is effective—MANUALplus then calculates all further infeed movements as follows:

Current infeed = I * (1 – (n–1) * E)

n: nth infeed

The infeed movement is reduced down to >= 0.5 mm. Following that, each infeed movement will amount to 0.5 mm.



Parameters

- **X end point** (diameter value)—(default: Current X position)
- **Z end point** of slot
- **C starting angle:** Starting position of the slot (default: 0)
- **F pitch**
  - F positive: Right-hand thread
  - F negative: Left-hand thread
- **F pitch**
- **P run-in length:** Ramp at the beginning of the slot (default: 0)
- **K run-out length:** Ramp at the end of the slot (default: 0)
- **U thread depth**
- **I maximum infeed** (default: Total depth in one infeed)
- **E reduction value** for infeed reduction (default: 1)

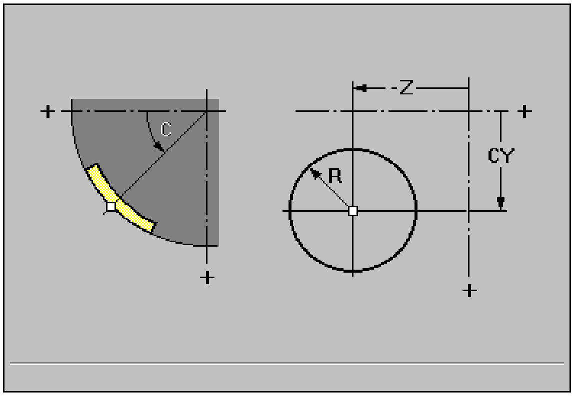
> You can mill a helical slot only on the outside.

Example: G798

```
%798.nc
[G798]
N1 T71 G197 S800 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X80 Z15
N5 G798 X80 Z-120 C0 F20 K20 U5 I1
N6 M15
END
```

Figure definition: Full circle, lateral surface G314

G314 defines a full circle on the lateral surface. Program this figure in conjunction with G794.



Parameters

- **`Z center point`**
- **`CY center point`** as linear value (reference: G120 reference diameter)
- **`C center point`** Angle to center—for angle direction, see graphic support window
- **`R radius`** of circle

Example: G314

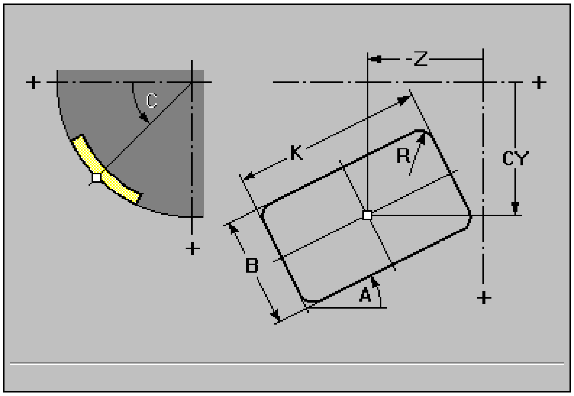
```
%314.nc
[G314]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z5
N5 G794 X100 XE97 P2 U0.5 R0 K0.5 F0.15 H0 Q0
N6 G314 Z-35 C0 R20
N7 G80
N8 M15
END
```

Figure definition: Rectangle, lateral surface G315

G315 defines a rectangle on the lateral surface. Program this figure in conjunction with G794.

Parameters

- **Z center point**
- **CY center point** as linear value (reference: G120 reference diameter)
- **C center point** Angle to center—for angle direction, see graphic support window
- **A angle**—reference: see graphic support window
- **K length** of rectangle
- **B height** (width) of rectangle
- **R chamfer/rounding**
  - R<0: Chamfer length
  - R>0: Rounding arc



Example: G315

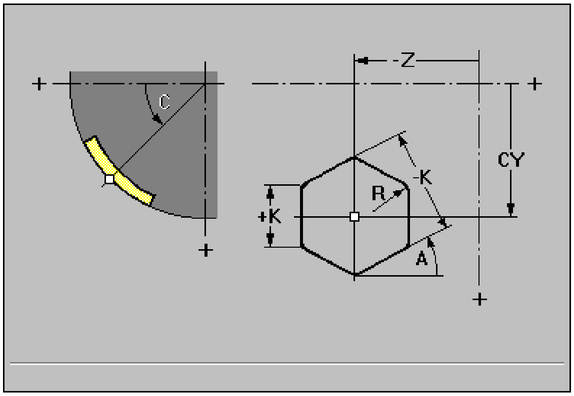
```
%315.nc
[G315]
N1 T71 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z5
N5 G794 X100 XE97 P2 U0.5 R0 K0.5 F0.15 H0
Q0
N6 G315 Z-35 C0 A5 K30 B15 R3
N7 G80
N8 M15
END
```

Figure definition: Eccentric polygon, lateral surface G317

G317 defines a polygon on the lateral surface. Program this figure in conjunction with G794.

Parameters

- **Z center point**
- **CY center point** as linear value (reference: G120 reference diameter)
- **C center point** Angle to center—for angle direction, see graphic support window
- **Q number of edges:** Range: 3 <= Q <= 127
- **A angle**—reference: see graphic support window
- **K width across flats (SW) / length**
  - K<0: Width across flats (inside diameter)
  - K>0: Edge length
- **R chamfer/rounding**
  - R<0: Chamfer length
  - R>0: Rounding arc



Example: G317

```
%317.nc
[G317]
N2 T71G197 S1200 G195 F0.2 M104
N3 M14
N4 G110 C0
N5 G0 X110 Z5
N6 G794 X100 XE97 P2 U0.5 R0 K0.5 F0.15 H0 Q0
N7 G317 Z-35 C0 Q6 A5 K-25 R3
N8 G80
N9 M15
END
```

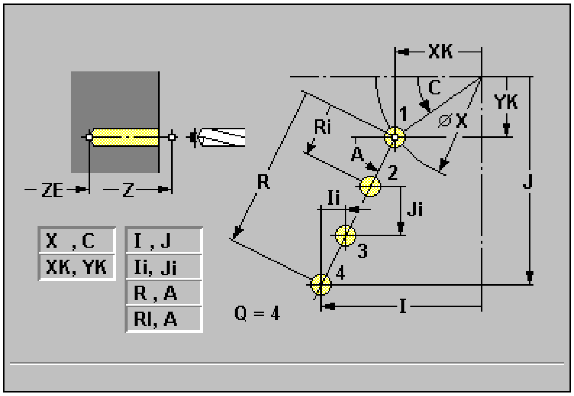
6.22 Pattern Machining

Linear pattern, face G743

With cycle G743, you can machine linear hole patterns or figure patterns in which the individual features are arranged at a regular spacing on the face.

If "ZE" has not been defined, the drilling/milling cycle of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with

- Drilling cycles (G71, G74, G36)
- The milling cycle for a linear slot (G791)
- The contour milling cycle with "independent contour" (G793)



Parameters

- **XK starting point** of pattern (Cartesian coordinates)
- **YK starting point** of pattern (Cartesian coordinates)
- **Z starting point** of drilling/milling operation
- **ZE end point** of drilling/milling operation
- **X diameter** (polar coordinates)
- **C starting angle** (polar coordinates)
- **A pattern angle**
- **I end point** of pattern (Cartesian coordinates)
- **J end point** of pattern (Cartesian coordinates)
- **Ii end point:** Pattern distance (Cartesian coordinates)
- **Ji end point:** Pattern distance (Cartesian coordinates)
- **R length:** Distance between first and last position
- **Ri length:** Distance to next position
- **Q number** of holes/figures (default: 1)

Parameter combinations for defining the starting point and the pattern positions:

- Starting point of pattern:
  - XK, YK
  - X, C
- Pattern positions:
  - I, J and Q
  - Ii, Ji and Q
  - R, A and Q
  - Ri, Ai and Q

Example: G743

```
%743.nc
[G743]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G743 XK20 YK5 A45 Ri30 Q2
N6 G791 X50 C0 Z-5 P2 F0.15
N7 M15
END
```

Examples for command sequences:

```
[ Simple drilling pattern ]
N.. G743 XK.. YK.. Z.. ZE.. I.. J.. Q..
. . .
```

```
[ Drilling pattern with deep-hole drilling ]
N.. G743 XK.. YK.. Z.. I.. J.. Q..
N.. G74 Z.. P.. I..
. . .
```

```
[ Milling pattern with linear slot ]
N.. G743 XK.. YK.. Z.. I.. J.. Q..
N.. G791 K.. A.. Z..
. . .
```

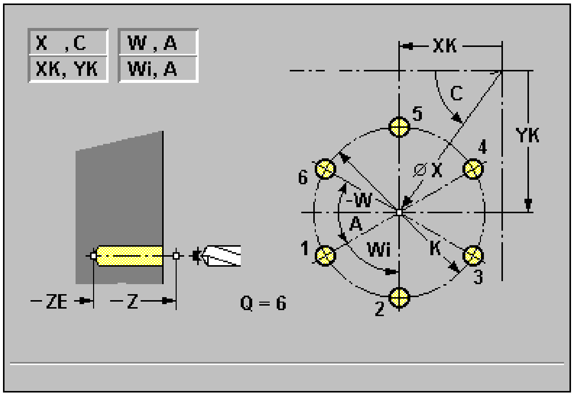
```
[ Milling pattern with "independent contour" ]
N.. G743 XK.. YK.. Z.. I.. J.. Q..
N.. G793 ZE.. U.. Q..
N.. G100 XK.. YK..
N.. . . .
N.. G80
. . .
```

Circular pattern, face G745

With cycle G745, you can machine hole patterns or figure patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the face.

If "ZE" has not been defined, the drilling/milling cycle of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with

- Drilling cycles (G71, G74, G36)
- The milling cycle for a linear slot (G791)
- The contour milling cycle with "independent contour" (G793)



Parameters

- **XK center** of pattern (Cartesian coordinates)
- **YK center** of pattern (Cartesian coordinates)
- **Z starting point** of drilling/milling operation
- **ZE end point** of drilling/milling operation
- **X diameter** (polar coordinates)
- **C angle** (polar coordinates)
- **K diameter:** Pattern diameter—default: The current X position is transferred.
- **A starting angle**—position of the first hole/figure
- **W end angle**—position of the last hole/figure
- **Wi end angle**—distance to the next position
- **Q number** of holes/figures (default: 1)
- **V direction of rotation** (default: 0): Position of holes/figures (required if W is defined):
  - V=0: Holes are placed on the longer arc.
  - V=1: Holes are arranged clockwise, starting at A.
  - V=2: Holes are arranged counterclockwise, starting at A.

Parameter combinations for defining the center of the pattern and the pattern positions:

- Center of pattern:
  - X, C
  - XK, YK
- Pattern positions:
  - A, W and Q
  - A, Wi and Q

Example: G745

```
%745.nc
[G745]
N1 T70 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G745 XK0 YK0 K50 A0 Q3
N6 G791 K30 A0 Z-5 P2 F0.15
N7 M15
END
```

Examples for command sequences:

[Simple drilling pattern]
N.. G745 XK.. YK.. Z.. ZE.. A.. W.. Q..
. . .

[Drilling pattern with deep-hole drilling]
N.. G745 XK.. YK.. Z.. ZE.. A.. W.. Q..
N.. G74 Z.. P.. I..
. . .

[Milling pattern with linear slot]
N.. G745 XK.. YK.. Z.. ZE.. A.. W.. Q..
N.. G791 K.. A.. Z..
. . .

[Milling pattern with "independent contour"]
N.. G745 XK.. YK.. Z.. ZE.. A.. W.. Q.
N.. G793 ZE.. U.. Q..
N.. G100 XK.. YK..
N.. . . .
N.. G80
. . .

Linear pattern, lateral surface G744

With cycle G744, you can machine linear hole patterns or figure patterns in which the individual features are arranged at a regular spacing on the lateral surface.

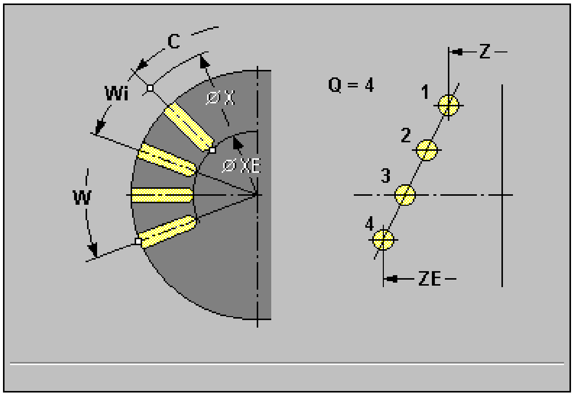
Parameter combinations for defining the starting point and the pattern positions:

- Starting point of pattern: Z and C
- Pattern positions:
  - W and Q
  - Wi and Q

If XE has not been defined, the drilling/milling cycle or the figure definition of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with drilling cycles (G71, G74, G36) or milling cycles (figure definitions with G314, G315, G317).

Parameters

- **Z starting point** (polar coordinates)
- **C starting angle** (polar coordinates)
- **X starting point** of drilling/milling operation (diameter value)
- **XE end point** of drilling/milling operation (diameter value)
- **ZE end point** of pattern (default: Z)
- **W end angle** of pattern—no input: Holes/figures are arranged on the lateral surface at regular spacing
- **Wi end angle:** Angle increment—distance to next position
- **Q number** of holes/figures (default: 1)



Example: G744

```
%744.nc
[G744]
N1 T60 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z2
N5 G744 X102 Z-10 ZE-35 C0 W270 Q5
N6 G71 X102 K7
N7 M15
END
```

Examples for command sequences:

```
[ Simple drilling pattern ]
N.. G744 Z.. C.. X.. XE.. ZE.. W.. Q..
. . .
```

```
[ Drilling pattern with deep-hole drilling ]
N.. G744 Z.. C.. X.. XE.. ZE.. W.. Q..
N.. G74 Z.. P.. I..
. . .
```

```
[ Milling pattern with linear slot ]
N.. G744 Z.. C.. X.. XE.. ZE.. W.. Q..
N.. G792 K.. A.. X..
. . .
```

```
[ Milling pattern with "independent contour" ]
N.. G744 Z.. C.. X.. XE.. ZE.. W.. Q..
N.. G794 XE.. U.. Q..
N.. G110 Z.. C..
N.. . . .
N.. G80
. . .
```

Circular pattern, lateral surface G746

With cycle G746, you can machine hole patterns or figure patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the lateral surface.

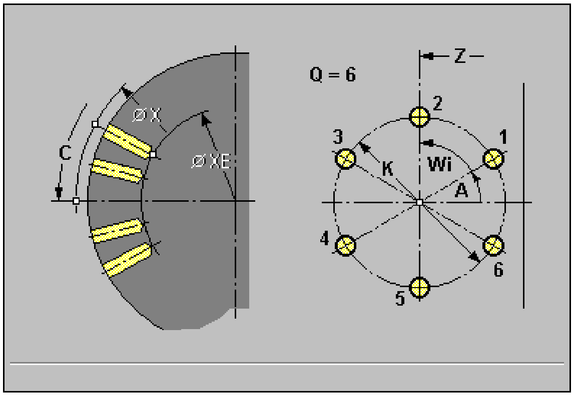
Parameter combinations for defining the center of the pattern and the pattern positions:

- Center of pattern: Z and C
- Pattern positions:
  - W and Q
  - Wi and Q

If XE has not been defined, the drilling/milling cycle or the figure definition of the next NC block is used as a reference. Using this principle, you can combine pattern definitions with drilling cycles (G71, G74, G36) or milling cycles (figure definitions with G314, G315, G317).

Parameters

- **Z center** of pattern (polar)
- **C angle:** Center of pattern (polar)
- **X starting point** of drilling/milling operation (diameter value)
- **XE end point** of drilling/milling operation (diameter value)
- **K diameter:** Pattern diameter
- **A starting angle**—position of the first hole/figure
- **K diameter:** Pattern diameter (default: Current X position)
- **W end angle**—position of the last hole/figure
- **Wi end angle**—distance to the next position
- **Q number** of holes/figures (default: 1)
- **V direction of rotation** (default: 0): Position of holes/figures (required if W is defined):
  - V=0: Holes are placed on the longer arc.
  - V=1: Holes are arranged clockwise, starting at A.
  - V=2: Holes are arranged counterclockwise, starting at A.



Example: G746

```
%746.nc
[G746]
N1 T60 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X110 Z2
N5 G746 Z-40 C0 K40 Q8
N6 G71 X102 K7
N7 M15
END
```

Examples for command sequences:

```
[ Simple drilling pattern ]
N.. G746 Z.. C.. X.. XE.. K.. A.. W.. Q..
. . .
```

```
[ Drilling pattern with deep-hole drilling ]
N.. G746 Z.. C.. X.. XE.. K.. A.. W.. Q..
N.. G74 Z.. P.. I..
. . .
```

```
[ Milling pattern with linear slot ]
N.. G746 Z.. C.. X.. XE.. K.. A.. W.. Q..
N.. G792 K.. A.. X..
. . .
```

```
[ Milling pattern with "independent contour" ]
N.. G746 Z.. C.. X.. XE.. K.. A.. W.. Q..
N.. G794 XE.. U.. Q..
N.. G110 Z.. C..
N.. . . .
N.. G80
. . .
```

6.23 Other G Functions

Period of dwell G4

The system interrupts the program run for the programmed length of time before executing the next command.

If G4 is programmed together with a path of traverse in the same block, the dwell time only becomes effective after the path of traverse has been executed.

Parameters

- **F dwell time:** Range: 0 sec < F < 999 sec

Precision stop G9

If you program G9 in conjunction with a traverse command (G1, G2, G3, G12 or G13) in a block, the feed rate is reduced to zero at the end of the path of traverse. The tool tip stops exactly at the programmed position before executing the next movement. This results in a square-edged corner.

Deactivate protection zone G60

G60 is used to cancel protective zone monitoring. G60 is programmed **before** the traversing command to be monitored or not monitored. G60 is a modal function.

Application example:
With G60, you can temporarily deactivate a programmed monitoring of the protective zone in order to machine a centric through hole.

Parameters

- Activate/deactivate **Q.**
  - Q=0: Activate protective zone monitoring.
  - Q=1: Deactivate protective zone monitoring.

Example: G60

```
%60.nc
[G60]
N1 T49 G97 S1000 G95 F0.3 M3
N2 G0 X0 Z5
N3 G60 Q1
N4 G71 Z-60 K65
N5  G60 Q0
END
```

Wait for moment G204

The execution of a DIN program is interrupted up to the defined point in time. (You can use this function, for example, for a warm-up program.)

The parameter "day D" refers to the next possible date. If D is not defined, the parameters "hour H, minute Q" refer to the next possible point in time.

Parameters

- **D day [1-31]**
- **H hour [0-23]**
- **Q minute [0-59]**

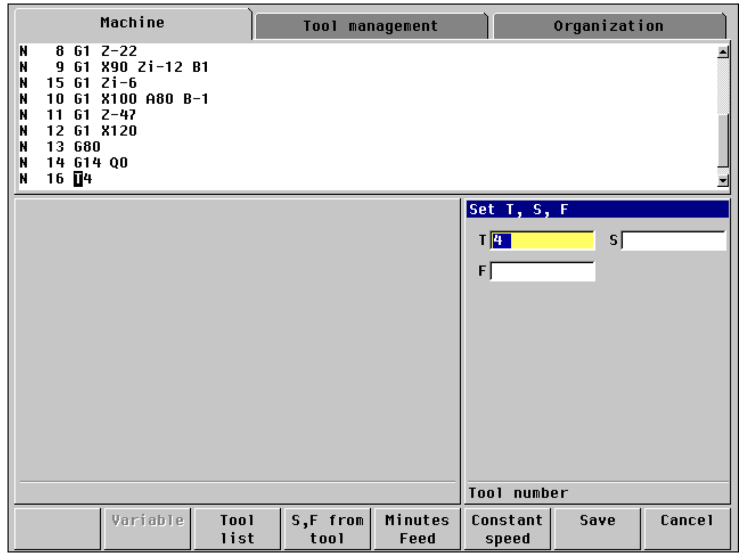
6.24 Set T, S, F

Tool number, spindle speed /cutting speed and feed rate

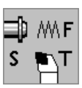
The values for feed rate and spindle speed that are programmed with "Set T, S, F" always refer to the spindle.

MANUALplus then transfers the parameters to the DIN program together with the identification letters or G functions.

- T: "T.."
- S: G96/G97 S..
- F: G94/G95 F..



Entering T, S, F



Select "Set T, S, F."

Select the soft keys, enter the parameters.

- **T tool number**
- **S cutting speed or spindle speed** (selected by soft key)
- **F feed per revolution or feed rate** (selected by soft key)

Conclude data input with **Save.**

Example: T, S, F

```
%819.nc
[G819]
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G819 P5 I1 K0.3
N4 G0 X80 Z2
N5 . . .
END
```

6.25 Data Input and Data Output

INPUT

INPUT (assigning values to variables)

#

Press "Program variable function."

INPUT

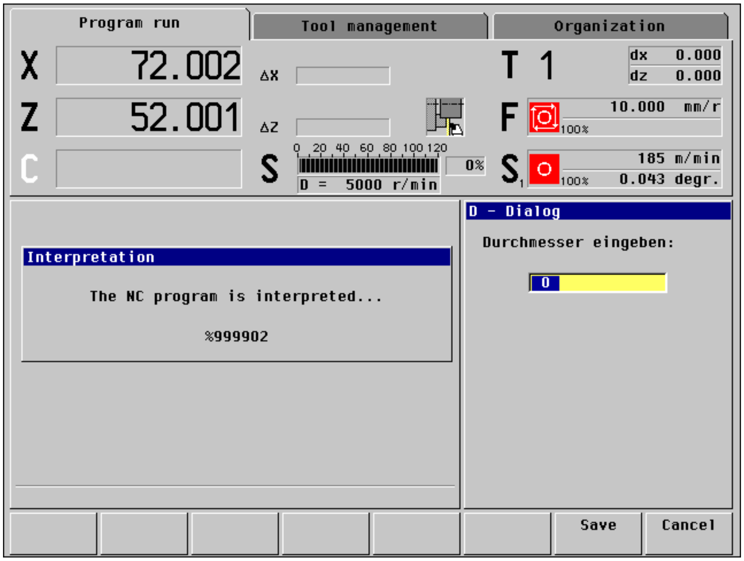
Select "Input function" (see top right figure on next page).

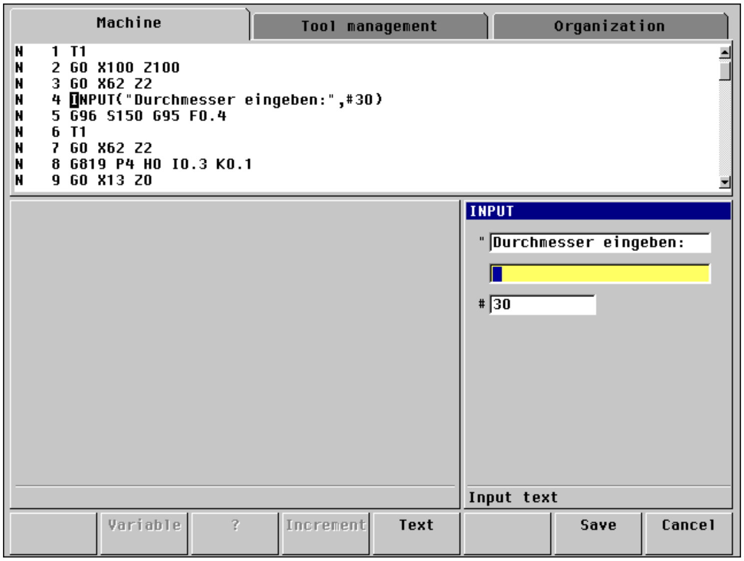
- Define the "input text."
- Enter the "Variable number for request" (see figure at top right).

When programming the "INPUT command," you define the "input text" and number of the "variable for request." The "input text" explains the input.

During the interpretation of this command, MANUALplus displays a screen window with the "input text" and the input field for the variable (see figure to the bottom right). Program interpretation continues after you have entered the data.

The value entered is assigned to the "variable for request."





WINDOW

WINDOW (defining the output window)

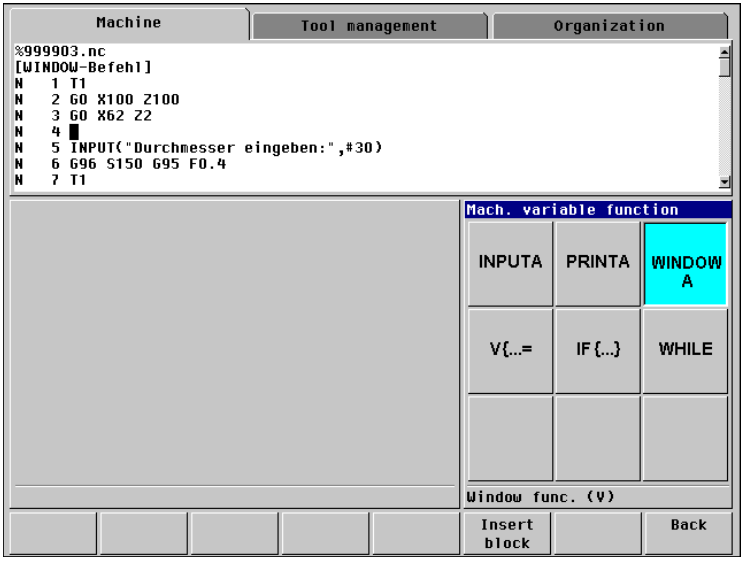
#

Press "Program variable function."

WINDOW

Select "WINDOW" (see figure to the top right).

- Select the size of the output window with "lines for output."
- Close the output window with "lines for output = 0."



With the "WINDOW" command, you can define a specific size for the "output" window that is to be used for output of information to the machinist. If you do not program "WINDOW," the control uses an output window with three lines for the output of information.

The output window is displayed in the lower part of the "list and program window." It appears on the screen during the first output of information and is displayed until you close it or until the interpretation of the DIN program is concluded.

You can close the output window with a "WINDOW call" and the parameter value "lines for output = 0."

PRINT

PRINT (output of information)

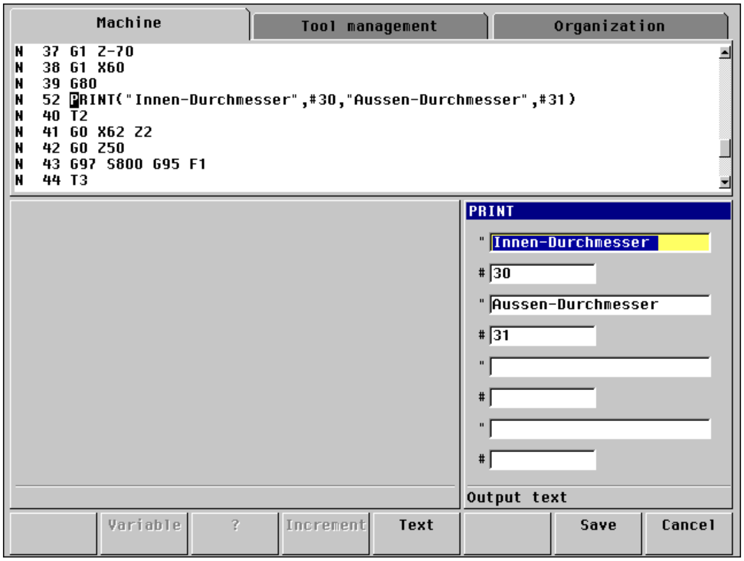
#

Press "Program variable function."

PRINT

Select "PRINT" (see figure to the top right).

- Define the "output texts" and
- the "variable numbers" (see figure to the right).



During execution of this command, MANUALplus displays the "output text" and the value (contents) of the defined "program variable for output" in the "output window."

The PRINT command can be used for defining more than one text and variable.

- The "output window" is cleared after program run (interpretation) and before execution of the DIN program.
- The window is not cleared, however, during simulation. It is not cleared until the simulation is restarted.

6.26 Programming Variables

Fundamentals

The MANUALplus interprets NC programs before the program run. The system therefore differentiates between two types of variables:

- # variables are evaluated during **NC program interpretation.**
- V variables (or events) are evaluated during **NC program run.**

The following rules apply:

- Multiplication/division before addition/subtraction
- Up to 6 bracket levels
- **Integer variables** (only for V variables): Integer values between –32767 and +32768
- **Real variables** (with # and V variables): Floating point numbers with max. 10 integers and 7 decimal places
- The V variables are retained even if the control has been switched off in the meantime.

Syntax	Mathematical functions
+	Addition
–	Subtraction
*	Multiplication
/	Division
SQRT(...)	Square root
ABS(...)	Absolute amount
TAN(...)	Tangent (in degrees)
ATAN(...)	Arc tangent (in degrees)
SIN(...)	Sine (in degrees)
ASIN(...)	Arc sine (in degrees)
COS(...)	Cosine (in degrees)
ACOS(...)	Arc cosine (in degrees)
ROUND(...)	Round
LOGN(...)	Natural logarithm
EXP(...)	Exponential function ex
INT(...)	Cut decimal places
Only with **# variables:**	
SQRTA(.., ..)	Square root of (a^2+b^2)
SQRTS(.., ..)	Square root of ($a^2–b^2$)

variables

MANUALplus uses value ranges to define the **scope of variables:**

- **#0 .. #45 global variables**
  Global variables are retained after the program has been completed and can be processed by the following NC program.
- **#46 .. #50 variables only for expert programs**
  Do not use these variables in your NC program.
- **#256 .. #285 local variables**
  These variables are effective only within a subprogram.

Reading-in parameter values

Syntax:**#1 = PARA(x,y,z)**

x = Parameter group

- 1: Machine parameters
- 2: Control parameters
- 3: Setup parameters
- 4: Machining parameters
- 5: PLC parameters

y = Parameter number

z = Sub-parameter number

Information contained in variables

The following variable information on tool data and your NC program can be read out (see the tables to the right and on the next page).

Positions and dimensions are always indicated in metric form. This also applies when an NC program is run in inches.

Example: "# variables"

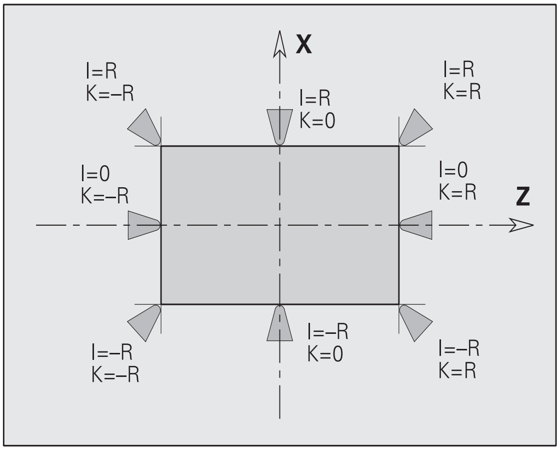
```
. . .
N.. #1=PARA(1,7,2) [reads "machine
dimension 1 Z" in variable #1 ]
N.. . . .
N.. #1=#1+1
N.. G1 X#1
N.. G1 X(SQRT(3*(SIN(30)))
N.. #1=(ABS(#2+0.5))
. . .
```

# variable	NC information
#768, #770	Last programmed position X (radius value), Z
#771	Last programmed position C [°]
#774	TRC/MCRC status 40: G40 active; 41: G41 active; 42: G42 active
#776	Active wear compensation (G148) 0: DX, DZ; 1: DS, DZ; 2: DX, DS
#778	Unit of measure: 0=metric; 1=inch
#785, #786	Distance between tool tip and slide zero point Z, X
#787	Reference diameter for lateral surface machining (G120)
#791..#792	G57 oversizes X, Z
#793	G58 oversize P
#794..#795	Cutting width in X, Z by which the tool reference point is shifted with G150/ G151
#796	Number of spindle for which the last feed rate was programmed
#797	Number of spindle for which the last speed was programmed

Precondition for tool information: A tool call must be programmed for the variables to become effective.

The assignment of variables #519..#521 varies depending on the type of tool.

# variable	Tool information
#514	Tool type: ■ 1: Turning tools ■ 2: Recessing tools ■ 3: Threading tools ■ 4: Drilling tools ■ 5: Taps ■ 6: Milling tools
#515	Tool orientation:
#519	For tool type 6: Number of teeth (K)
#520	For tool type: ■ 1, 2: Cutting radius (R) ■ 6: Milling diameter (I)
#521	For tool type 2: Cutting width (K)
#522	Tool orientation (reference: machining direction of tool): ■ 0: On the contour ■ 1: To the right of the contour ■ – 1: To the left of the contour
#523..#524	Set-up dimensions (Z, X)
#526..#527	Position of tool tip center I, K (see figure below)



V variables

The MANUALplus uses value ranges to define the following **scope of variables:**

- Real: V1 .. V199
- Integer: V200 .. V299
- Reserved: V300 .. V900

Requests and assignments:

- Read/write **machine dimensions** (machine parameter 7)

  Syntax: **V{Mx[y]}**

  x = dimension 1..9
  y = coordinate: X, Y, Z, U, V, W, A, B or C

- Interrogate **external events**
  Is the **bit** value 0 or 1? The significance of the external event is determined by the machine manufacturer.

  Syntax: **V{Ex[y]}**

  x = slide 1
  y = bit: 1..16

- Interrogate **sequential events**
  The tool life monitoring function and the function for searching the start block trigger sequential events.

  Syntax:**V{Ex[1]}**

  x = event: 20, 90

  - 20: Life of this tool has expired (global information)
  - 90: Define start block (0=not active; 1=active)

- Read/write **tool compensation**

Syntax: **V{Dx[y]}**

x = T number
y = length compensation: X, Y, or Z

Example: "V variable"

```
. . .
N.. V{M1[Z]=300}  [ sets "machine dimension 1 Z" to "300" ]
. . .
N.. G0 Z{M1[Z]}  [ moves to "machine dimension 1 Z" ]
. . .
N.. IF{E1[1]==0}  [ Interrogates "external event 1 – bit 1“ ]
. . .
N.. V{D5[X]=1.3}  [ set "compensation X for tool 5" ]
. . .
N.. V{V12=17.4}
N.. V{V12=V12+1}
N.. G1 X{V12}
. . .
```

Information contained in variables

- V901, V902 and V919 are used for the G functions G901, G902 and G903 (see table).

- X values are saved as radius values.
- Note: Functions G901, G902 and G903 overwrite the variable! This also applies to variables that have not yet been evaluated.

Variable assignment		
Slide 1 (X, Z)	V901	V902
C axis	V919	

Note on interpreter stop (G909)

The MANUALplus pre-interprets approx. 15 to 20 NC blocks. If variables are assigned shortly before the evaluation, "old values" would be processed. An **interpreter stop** ensures that the variables contain the new value.

G909 stops the pre-interpretation. The NC blocks are processed up to G909—after G909 the subsequent NC blocks are processed.

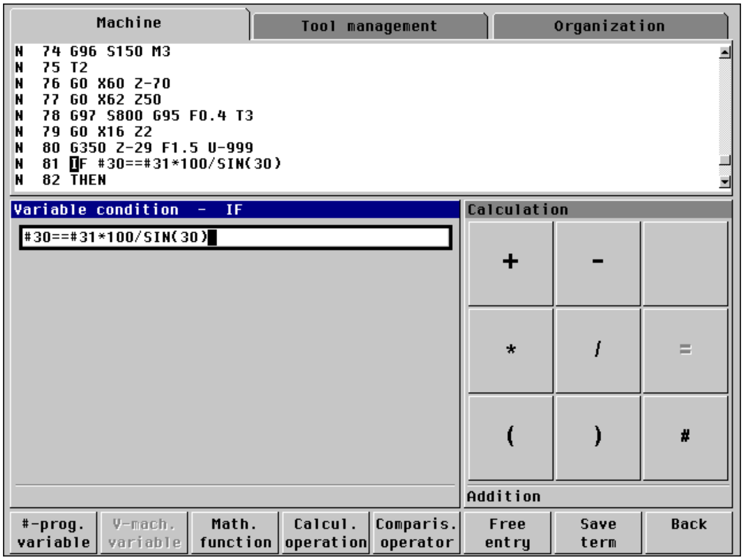
- Program an **interpreter stop** if variables or external events are modified shortly before the block is run.
- Each interpreter stop lengthens the run time of the NC program.
- Several G functions include the interpreter stop.

6.27 Program Branches, Program Repeats

IF (...) (conditional program branch)

#	Press "Program variable function."
IF (...)	Select "Conditional program branch."

Enter the "variable condition" (see figure to the top right).



You can use both "Mathematical functions" and "Calculating operations" in the same mathematical expression. The mathematical functions are arranged on two menu levels. To switch to the next menu level, press ">>."

The "condition" includes a variable or mathematical expression on either side of the relational operator (see figure to the top right).

A "conditional branch" consists of the elements:

- "IF"—followed by a condition (comparison).
- "THEN"—if the condition is fulfilled, the THEN branch is executed.
- "ELSE"—if the condition is not fulfilled, the ELSE branch is executed.
- "ENDIF"—concludes the conditional program branch.

After entering the "conditional program branch," program the NC blocks to be executed.

The "ELSE branch" can be omitted.

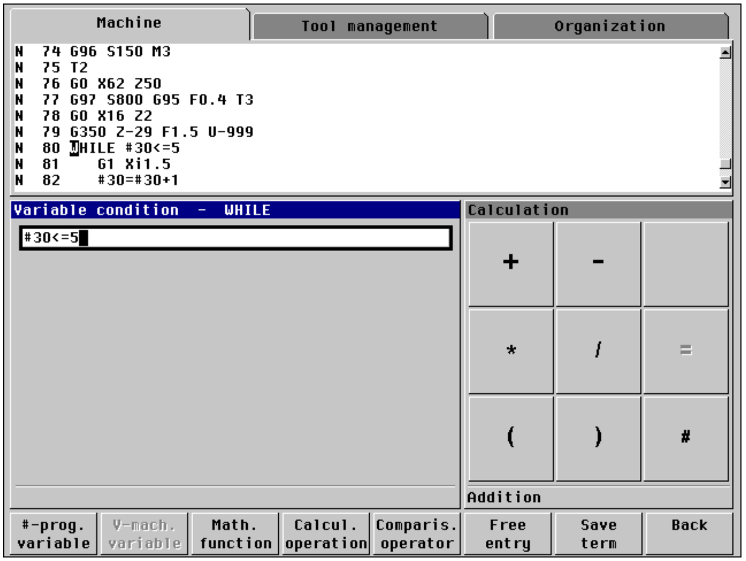
Relational operators	
<	Less than
<=	Less or equal
<>	Not equal
>	Greater than
>=	Greater or equal
==	Equal
AND	Logical AND operation
OR	Logical OR operation

WHILE (program repeat)

Press "Program variable function."

WHILE Select "Program repeat."

Enter the "variable condition" (see figure to the top right).



A "program repeat" consists of the elements:

- "WHILE"—followed by a condition (comparison).
- "ENDWHILE"—concludes the conditional program branch.

The NC blocks that are programmed between WHILE and ENDWHILE are executed repeatedly for as long as the "condition" is fulfilled. If the condition is not fulfilled, MANUALplus continues execution of the program with the block programmed after "ENDWHILE."

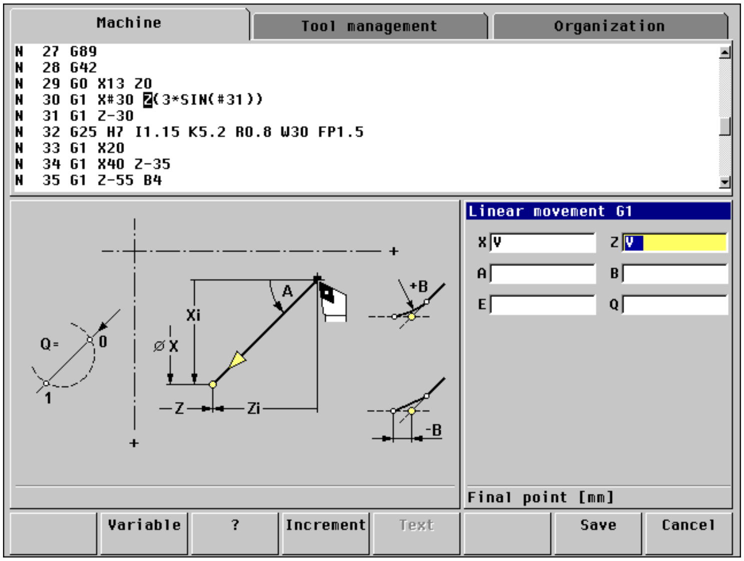
The "condition" includes a variable or mathematical expression on either side of the relational operator (see figure to the top right).

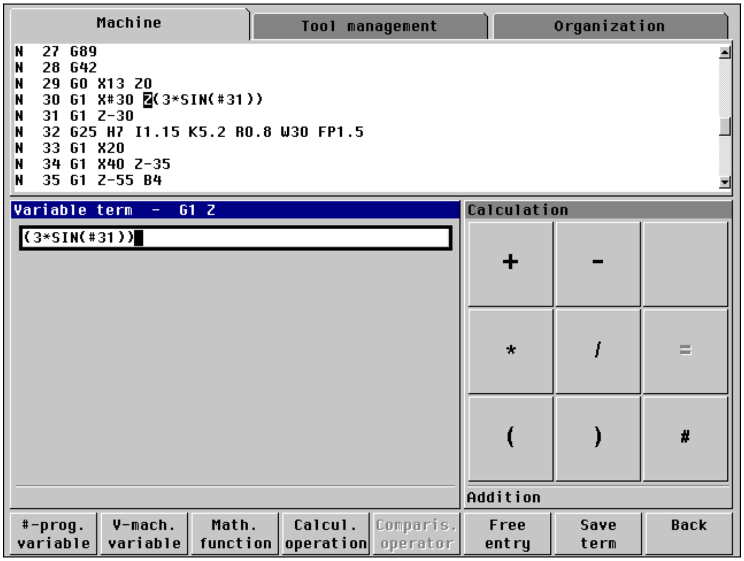
After entering the "program repeat," program the NC blocks to be executed.

> If the condition you program in the WHILE command is always true, the program remains in an "endless loop." This is one of the most frequent causes of error when working with program repeats.

Relational operators	
<	Less than
<=	Less or equal
<>	Not equal
>	Greater than
>=	Greater or equal
==	Equal
AND	Logical AND operation
OR	Logical OR operation

6.28 Variables as Address Parameters





Variables as address parameters

Select the input parameters (see figure to the top right)

Variable

Press ***Variable.***

#-prog. variable

Press ***# program variable.***

#

Select "Program variable."

Enter the variable number.

Select

Transfer the variable number.

Enter a mathematical expression, if required:

Math. function

Mathematical function
or

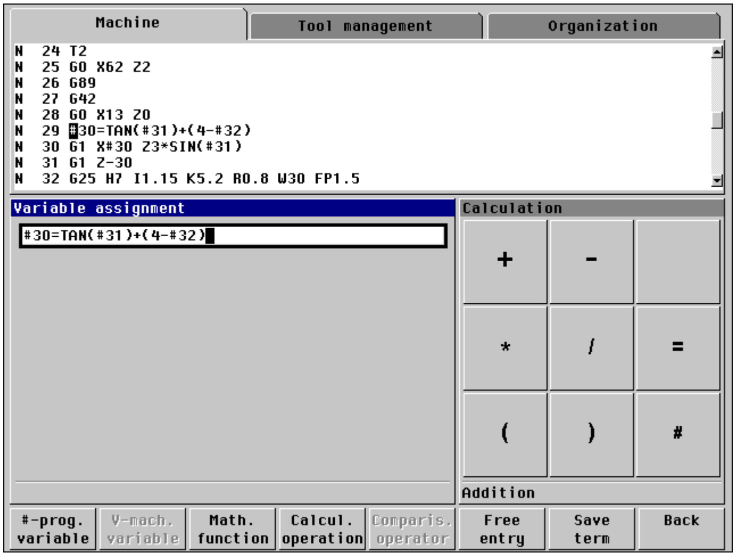
Calcul. operation

Select the ***Calculating operation*** (see figure to the lower right).

Save term

Transfer the variable/variable calculation as address parameter.

- In the input field, the letter "V" appears. The control, however, transfers the complete variable designation or mathematical expression to the DIN program (see figures at right).
- A mathematical expression should be enclosed in brackets (example: G1 X(3*SIN(#30)) Z#31).



Overview of mathematical functions	
SIN	Sine (degrees)
COS	Cosine (degrees)
TAN	Tangent (degrees)
ATAN	Arc tangent (degrees)
ABS	Absolute amount
ROUND	Round
INT	Round off (truncate)
SQRT	Square root
SQRTA	Square root ($a^2 + b^2$)
SQRTS	Square root ($a^2 – b^2$)
LOGN	Natural logarithm
EXP	Exponential function
+	Addition
–	Subtraction
*	Multiplication
/	Division
=	Assign
(	Opening bracket
)	Closing bracket

You can program NC blocks that contain only variable calculations (see figure at right).

Calculating variables

— Press "Program variable function."

#...= — Select "Assignment (#)."

Enter the variable number.

Select — Transfer the variable number.

Enter the mathematical expression:

Math. function — ***Mathematical function*** or

Calcul. operation — Select the ***Calculating operation*** (see figure to the lower right).

Save term — Transfer the variable/variable calculation as address parameter.

The mathematical expression is calculated during the translation (interpretation). The result is assigned to the variable.

You can use both "Mathematical functions" and "Calculating operations" in the same mathematical expression. The mathematical functions are arranged on two menu levels. To switch to the next menu level, press ">>."

The following rules apply:

- Multiplication/division before addition/subtraction
- You can form up to six levels, using brackets.

6.29 Subprograms

Calling a subprogram

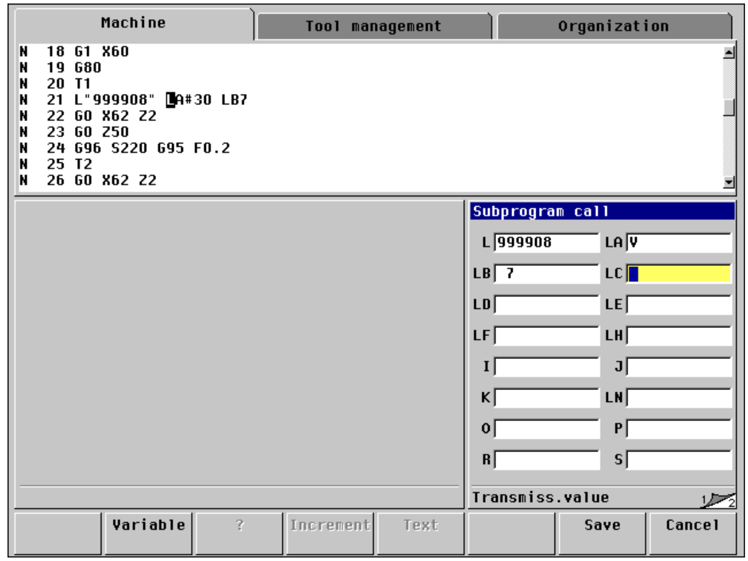
`L".."` Select the "Subprogram call."

`DIN-makro list` Select the ***DIN macro list.***

Call the subprogram.

`Take over DIN-makro` Select ***Assume DIN macro.***

Enter transfer parameters

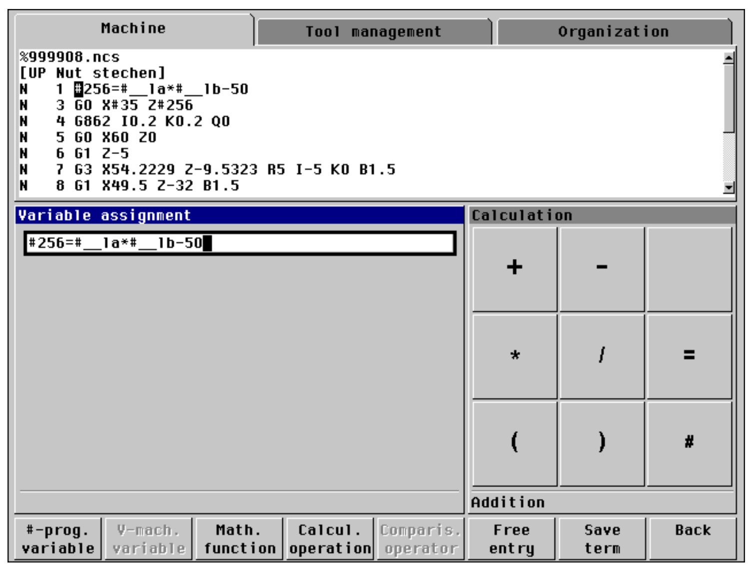


Entering subprogram names directly

`L".."` Select the "Subprogram call."

Enter the program name (see figure at top right).

Enter transfer parameters



General information on subprograms:

- Subprograms are defined in a separate file. They can be called from any main program or other subprogram. (DIN macros are subprograms.)
- Subroutines can be "nested" up to 6 times. Nesting means that another subprogram is called from within a subprogram.
- Recursion should be avoided.

- You can add up to 20 "transfer values" to a subprogram. These are: LA to LF, LH, I, J, K, O, P, R, S, U, W, X, Y, Z.
  The transfer values are available as variables within the subprogram. The identification code is: "#__..", followed by the parameter designation in lowercase letters (for example: #__la).
  You can use these transfer values when programming with variables within the subprogram (see figure to the bottom right).
  The transfer parameter **LN** is reserved for transferring integer values from 0 to 9999.
- The variables #256 – #285 are available in any subprogram for internal calculations (local variables).
- If a subprogram is to be executed repeatedly, enter the number of times the subprogram is to be repeated in the parameter "number repeats Q."

Dialog texts

You can define the parameter descriptions that precede/follow the input fields in an external subprogram.

MANUALplus automatically sets the unit of measure for parameter values to the metric system or inches.

A maximum of 19 descriptions can be entered. The parameter descriptions can be positioned within the subprogram as desired.

Parameter descriptions:

[//]—beginning

[pn=n; s=parameter text (up to 16 characters)]

[//]—end

pn: Parameter designations (la, lb, ...)

n: Conversion number for units of measurement

- 0: Non-dimensional
- 1: "mm" or "inches"
- 2: "mm/rev" or "inch/rev"
- 3: "mm/min" or "inch/min"
- 4: "m/min" or "feet/min"
- 5: Rev/min
- 6: Degrees (°)
- 7: "µm" or "µinch"

Example:

```
. . .
[//]
[la=1; s=bar diameter]
[lb=1; s=starting point in Z]
[lc=1; s=chamfer/rounding (-/+)]
. . .
[//]
. . .
```

6.30 M Functions

With M functions, you can control the program run and program switching functions for the machine (machine commands).

Entering M functions

M — Select "M function."

Enter the number of the M function. Define the parameters, if applicable.

M commands for program-run control

- **M00 Program stop** interrupts execution of a DIN program. Program run is continued after Cycle START has been pressed.
- **M01 Optional stop:** In "Program run" mode, you can use ***Continuous run*** to determine whether cycle programs or DIN programs are to be interrupted at an M01 command. If this function is disabled, MANUALplus interrupts execution of the program when M01 is reached and continues program run after Cycle START has been pressed.
- **M30 End of program** indicates the end of a program or subprogram. (M30 does not need to be programmed.) If you press "Cycle START" after M30, program execution is repeated from the start of the program.
- **M99 End of program with return jump** to start of program or to the defined block number and restart. MANUALplus restarts program execution from:
  - The start of program if no "next block NS" is defined, or
  - From the block number NS if a "next block NS" is defined.
- **M417** deactivates protection zone monitoring.
- **M418** activates protection zone monitoring.

Note on using M99: All modal functions (feed rate, spindle speed, tool number, etc.) which are effective at the end of program remain in effect when the program is restarted. You should therefore reprogram the modal functions at the start of program or at the startup block.

M commands for program-run control	
M00	Program STOP
M01	Optional STOP
M30	End of program
M99	NS.. End of program with return jump to start of program or to block number "NS.." and restart
M417	Deactivate protection zone monitoring
M418	Activate protection zone monitoring

Machine commands

The effect of machine commands depends on the configuration of your machine. The table lists the M commands used on most machines.

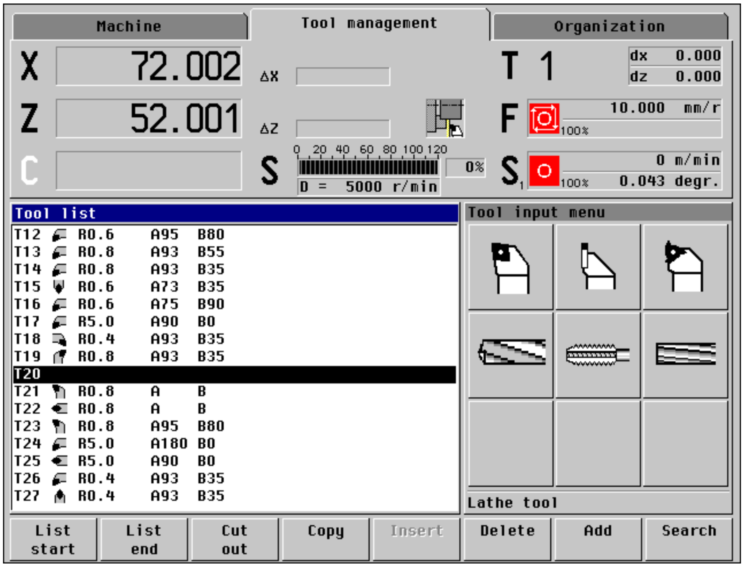
Please refer to your machine manual for detailed information on which of the M commands listed are supported by your machine and which additional M commands are available.

M commands as machine commands	
M03	Spindle ON (CW)
M04	Spindle ON (CCW)
M05	Spindle STOP
M12	Lock spindle brake
M13	Release spindle brake
M14	C axis ON
M15	C axis OFF
M19	C.. STOP spindle at position "C"
M40	Shift gear to range 0 (neutral)
M41	Shift gear to range 1
M42	Shift gear to range 2
M43	Shift gear to range 3
M44	Shift gear to range 4
M103	Spindle 1 (auxiliary spindle for driven tool) ON (CW)
M104	Spindle 1 (auxiliary spindle for driven tool) ON (CCW)
M105	Spindle 1 (driven tool) STOP

7

Tool Management Mode

7.1 Tool Management Mode of Operation



You usually program the coordinates for the contour by taking the dimensions from the drawing. To enable MANUALplus to calculate the slide path, compensate the cutting radius and determine the proportioning of cuts, you need to enter the tool length, cutting radius, tool angle, etc.

MANUALplus can save tool data for up to 99 tools, whereby each tool is identified with a number (1...99). For each tool, you can enter an additional tool description which makes it easier to find the tool data again when needed.

The Machine mode has functions for determining the tool length dimensions (see “Setting up Tools” on page 54).

Wear compensation is managed separately. This allows you to enter new compensation values at any time, even during program run.

You can assign specific **cutting data** to the tools (spindle speed, feed rate) which is then transferred simply at the touch of a key as cycle parameters or machine data. This saves you a lot of time since you only need to determine and enter the cutting data once.

Tool types

Tools for drilling, recessing, finishing, etc., have very different shapes. Therefore, the reference points for determining the tool length and other tool data also vary.

MANUALplus differentiates:

- **Lathe tools**—this group comprises:
  - Roughing tools
  - Finishing tools
  - Fine-finishing tools
  - Copying tools
  - Button tools
- **Recessing tools**—this group comprises:
  - Recessing tools
  - Undercut tools
  - Parting tools
  - Recess-turning tools
- **Thread-cutting tools:** All kinds of threading tools except tapping tools

- **Drilling tools**—this group comprises:
  - Centering tools
  - NC center drills
  - Twist drills
  - Indexable-insert drills
  - Countersinks/counterbores
  - Reamers
- **Taps:** All kind of tapping tools
- **Milling tools**—this group comprises:
  - Twist drill cutter
  - End milling cutter
  - Thread cutter

You will certainly use more than these tool types. Special care has been taken to clearly structure the tool types available on the MANUALplus.

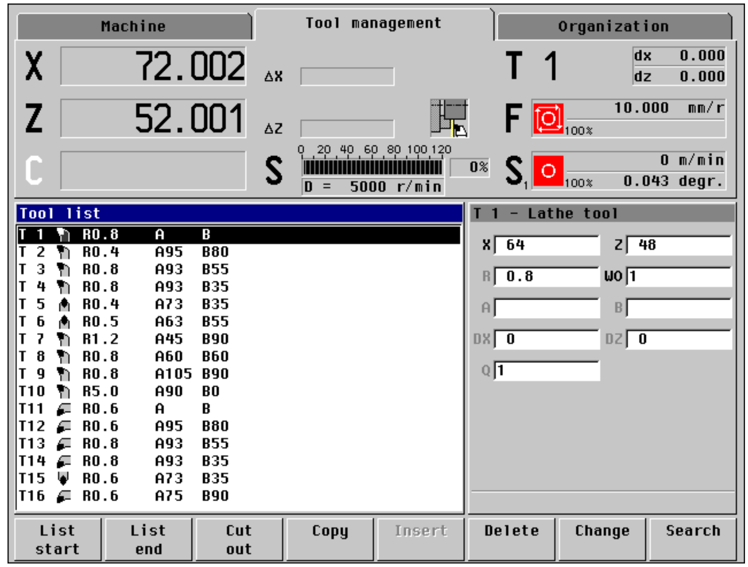
Tool life management

In Tool Organization you can set the life for the cutting edge or define the number of workpieces that will be machined with each cutting edge. The system monitors use of the tool and displays a message as soon as the programmed tool life expires or the programmed number of parts is reached (see “Tool life monitoring” on page 59 and “Tool Data— Supplementary Parameters” on page 426).

7.2 Tool Organization

The entries in the tool list are designated **T1...T99.** The tool tip in the graphic display shows the tool type and the tool orientation. In the tool list, the MANUALplus displays important parameters and the tool description. The input window shows additional data on the tool that is highlighted in the tool list.

You can navigate within the tool list with the arrow keys and "PgUp/PgDn" to check the entries. You can also "search" entries of a **particular tool type.**



Enter data for new tool

- Position the cursor on a free space.
- Press ***Add.***
- Select tool type—MANUALplus opens the input window and illustrates the individual parameters in the graphic support window.

Edit entry

- Place the cursor on the desired entry.
- Press ***Change.***
- The tool parameters are displayed in a dialog box and can be edited (the tool type cannot be edited).

Copy entry

- Place the cursor on the desired entry.
- Press ***Copy.***
- Position the cursor on a free space.
- Retrieve the copied tool data by pressing ***Insert.***

Relocate entry

- Place the cursor on the desired entry.
- Press ***Cut out*** (the tool data are deleted).
- Position the cursor on a free space.
- Retrieve the tool data by pressing ***Insert.***

Delete entry

- Place the cursor on the entry to be deleted.
- Press ***Delete.***

Find entries

- Press ***Search.***
- Select the tool type with the menu key.
- MANUALplus scrolls through the list and stops at the next entry of this type.
- Press the tool type menu key again: MANUALplus scrolls through the list and stops at the next entry of this type. When MANUALplus has reached the last entry in the list for the selected tool type, it displays the first entry for this type again.
- Available functions:
  – ***Change:*** MANUALplus opens the input window.
  – ***Back:*** Return to tool list

> The internal buffer can only store one entry at a time! If you successively transfer several entries to the buffer with ***Cut out*** or ***Copy*** without inserting them again at another position with ***Insert,*** all entries transferred to the buffer, except the last one, will be lost.

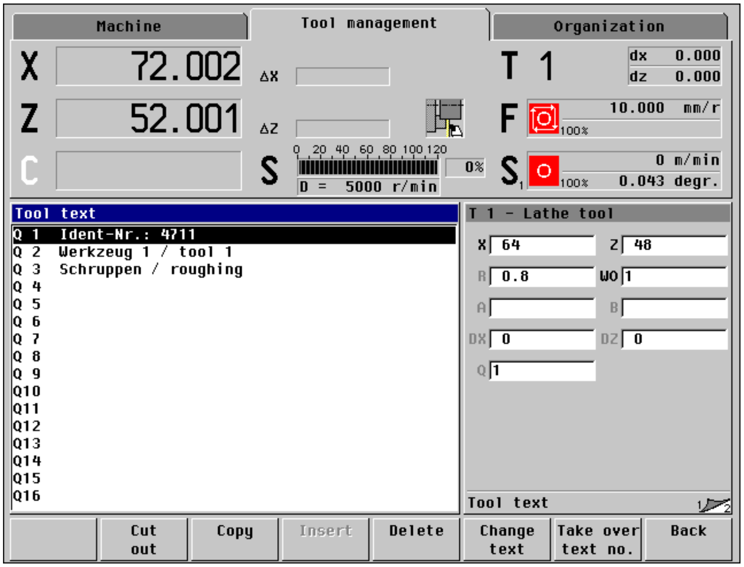
7.3 Tool Texts

A description or designation makes it easier to find a specific tool whenever you need it again. You can describe each tool by an identification number or a general designation, depending on your method of organization.

Connections:

- The descriptions are managed in the **tool text** list. Each entry is preceded by a "Q number."
- The parameter "Tool text Q" contains the reference number for the "tool text" list. The text is then displayed in the tool list.

You can navigate within the tool list with the arrow keys and "PgUp/PgDn" to check the entries.



Create entry

- Position the cursor on a free space.
- Press ***Change text.***
- MANUALplus displays the alphanumeric keyboard for entering the text.

Edit entry

- Position the cursor on the text entry.
- Press ***Change text.***
- MANUALplus displays the alphanumeric keyboard for editing the text.

Copy entry

- Position the cursor on the text entry.
- Press ***Copy.***
- Position the cursor on a free space.
- Retrieve the copied text by pressing ***Insert.***

Relocate entry

- Position the cursor on the text entry.
- Press ***Cut out.***
- Position the cursor on a free space.
- Retrieve the copied text by pressing ***Insert.***

Delete entry

- Place the cursor on the text entry to be deleted.
- Press ***Delete.***

Transfer text number

- Position the cursor on the text entry.
- Press ***Take over text no.***
- MANUALplus transfers the "Q number" of the text entry as "tool text Q" and switches back to the tool data editing mode.

- If you switch back to the tool data editing mode with ***Back,*** the parameter "tool text Q" remains unchanged.
- The internal buffer can only store one entry at a time! If you successively transfer several entries to the buffer with ***Cut out*** or ***Copy*** without inserting them again at another position with ***Insert,*** all entries transferred to the buffer, except the last one, will be lost.

7.4 Tool Data

Tool orientation

From the tool orientation, MANUALplus determines the position of the tool tip and, depending on the selected tool type, additional information such as the setting-angle direction, reference-point position, etc. This information is necessary, for example, for calculating the cutting radius compensation, plunging angle, etc.

Reference point

The "setting dimensions X, Z" refer to the tool reference point. The position of the reference point depends on the tool type (see graphic support window).

Editing tool data

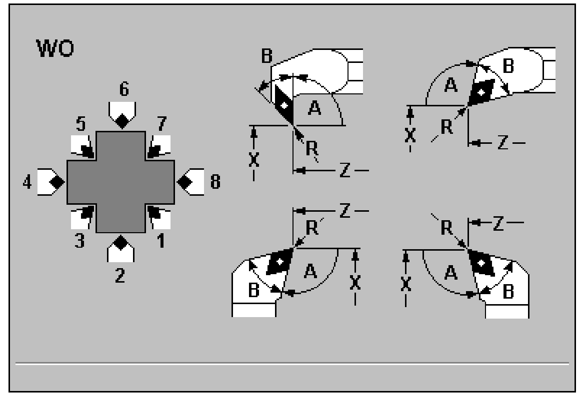
When you press one of the menu keys, MANUALplus displays the associated parameter input window and a graphic support window in which all parameters for this tool type are illustrated.

The tool data are edited in two input windows. The first input window contains the parameters specific to this tool type, the second contains the cutting data as well as information on the spindle rotation direction and tool life monitoring. You only need to switch to the second input window if you want to use this data.

- Tool parameters whose identification letters are shown in gray can be entered optionally. MANUALplus uses these parameters when specific cycle parameters are not entered, when plunging angles or feed rates need to be calculated, etc.
- You will find more information on the use of these parameters in the description of the tool data and the cycles.

Lathe tools

Select lathe tools.



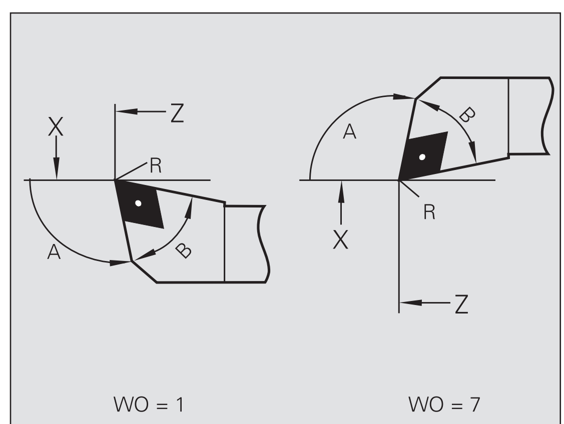
The graphic support window illustrates how goose-necked roughing and finishing tools for longitudinal machining are dimensioned (WO 1, 3, 5 and 7). On the next page you will find information on the dimensions of facing tools, neutral tools and button tools.

Tool parameters

- **X setup dimension in X**
- **Z setup dimension in Z**
- **R cutting radius**
- **WO tool orientation:** For code number, see graphic support window
- **A tool angle:** Range: 0° <= A <= 180°
- **B point angle:** Range: 0° <= B <= 180°
- **DX wear compensation in X:** Range: –100mm < DX < 100mm
- **DZ wear compensation in Z:** Range: –100mm < DZ < 100mm
- **Q tool text:** Reference to tool text
- **MD direction of rotation**—default: Not defined
  - 3: M3
  - 4: M4
- **TS cutting/spindle speed:** default: Not defined
- **TF feed rate**—default: Not defined
- **PT tool life**—default: Not defined
- **RT:** Display field for remaining tool life
- **PZ quantity**—default: Not defined
- **RZ:** Display field for remaining quantity

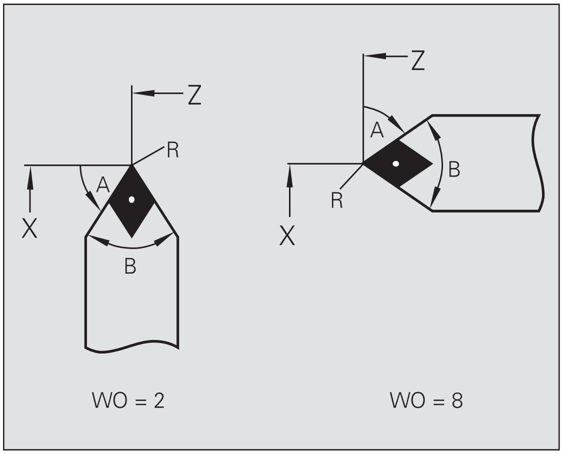
Facing tools

The figure at right explains how to dimension these tools, taking facing tools with the tool orientations WO=1 and WO=7 as an example.



Neutral tools

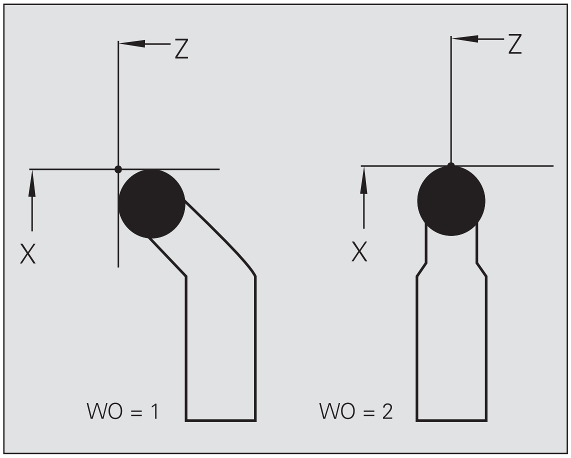
The tool orientation values WO=2, 4, 6, 8 are used for "neutral" tools. Neutral means the cutting edge is perpendicular to the X or Z axis (see figure at right).



Button tools

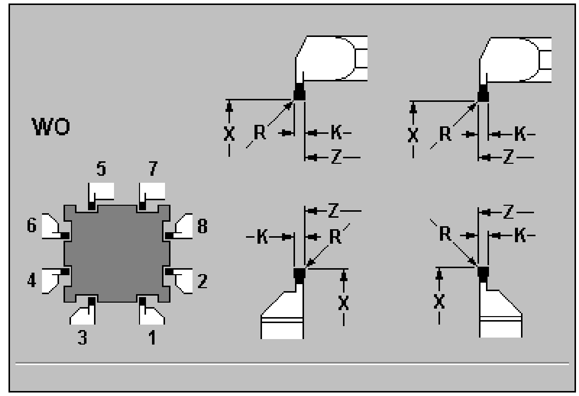
The following aspects are important when dimensioning button tools:

- **Nose angle B=0:** identifies the tool as button tool.
- **Tool angle:** is used for plunge cycles to check or calculate the plunging angle. MANUALplus needs the tool angle during simulation for calculating the tool position.
- **Reference point:** depends on the tool orientation—see figure at right for dimensioning with WO=1 and WO=2 (neutral button tool).



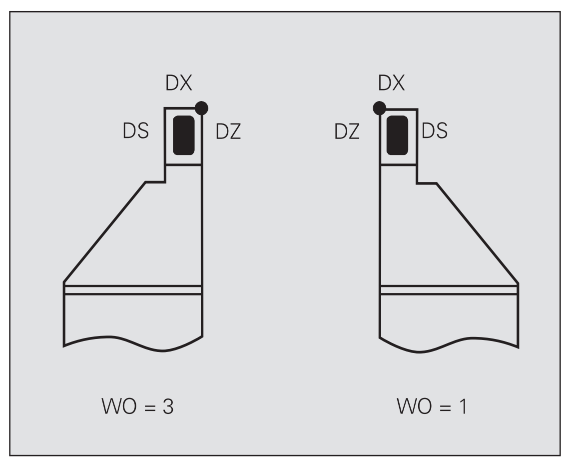
Recessing and recess-turning tools

Select recessing tools.



Tool parameters

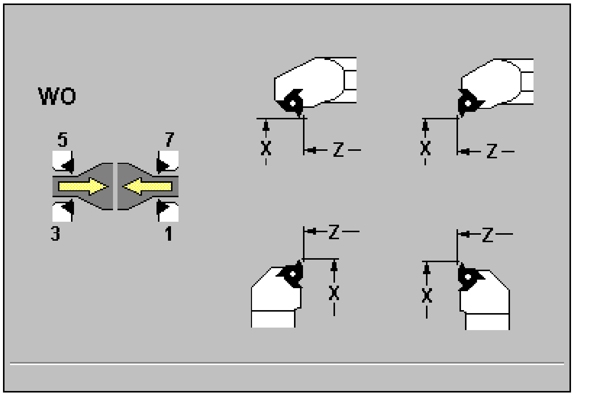
- **`X setup dimension in X`**
- **`Z setup dimension in Z`**
- **`R cutting radius`**
- **`WO tool orientation:`** For code number, see graphic support window
- **`K cutting width`**
- **`DX wear compensation in X:`** Range: –100mm < DX < 100mm
- **`DZ wear compensation in Z:`** Range: –100mm < DZ < 100mm
- **`DS special compensation:`** Range: –100mm < DS < 100mm
- **`Q tool text:`** Reference to tool text
- **`MD direction of rotation`**—default: Not defined
  - 3: M3
  - 4: M4
- **`TS cutting/spindle speed:`** default: Not defined
- **`TF feed rate`**—default: Not defined
- **`PT tool life`**—default: Not defined
- **`RT:`** Display field for remaining tool life
- **`PZ quantity`**—default: Not defined
- **`RZ:`** Display field for remaining quantity



- With recessing tools, you define the position of the reference point with "tool orientation WO."
- "DX, DZ" compensate for wear on the two sides of the tool tip that lie next to the reference point. "DS" compensates for wear on the third side of the tool tip (see figure at right).
- The "cutting width K" is evaluated if the corresponding parameter is not defined in the recessing cycle.

Thread-cutting tools

Select thread-cutting tools.

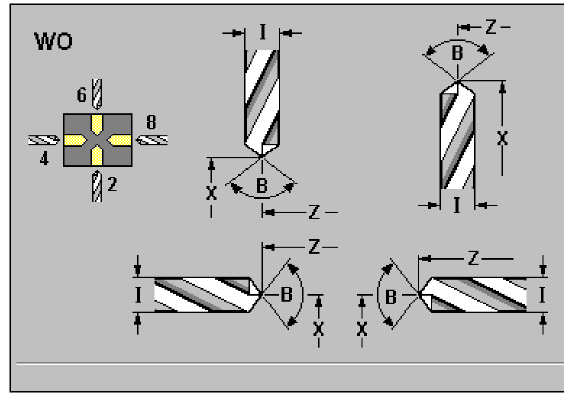


Tool parameters

- **`X setup dimension in X`**
- **`Z setup dimension in Z`**
- **`WO tool orientation:`** For code number, see graphic support window
- **`DX wear compensation in X:`** Range: –100mm < DX < 100mm
- **`DZ wear compensation in Z:`** Range: –100mm < DZ < 100mm
- **`Q tool text:`** Reference to tool text
- **`MD direction of rotation`**—default: Not defined
  - 3: M3
  - 4: M4
- **`TS cutting/spindle speed:`** default: Not defined
- **`TF feed rate`**—default: Not defined
- **`PT tool life`**—default: Not defined
- **`RT:`** Display field for remaining tool life
- **`PZ quantity`**—default: Not defined
- **`RZ:`** Display field for remaining quantity

Drilling tools

Select drilling tools.

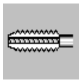


Tool parameters

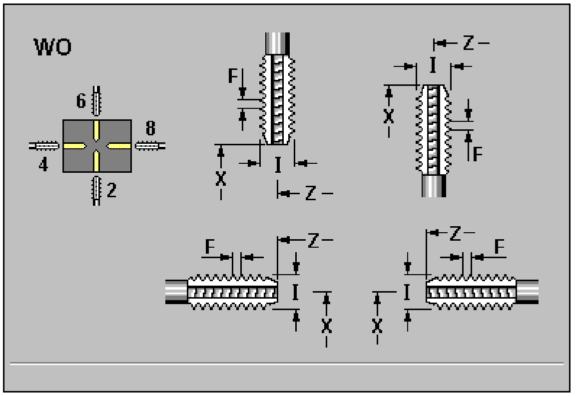
- **X setup dimension in X**
- **Z setup dimension in Z**
- **I hole diameter**
- **WO tool orientation:** For code number, see graphic support window
- **B point angle**—Range: 0° < B <= 180°
- **DX wear compensation in X:** Range: –100mm < DX < 100mm
- **DZ wear compensation in Z:** Range: –100mm < DZ < 100mm
- **Q tool text:** Reference to tool text
- **H tool driven**—default: 0
  - 0: Not driven
  - 1: Driven
- **MD direction of rotation**—default: Not defined
  - 3: M3
  - 4: M4
- **TS cutting/spindle speed:** default: Not defined
- **TF feed rate**—default: Not defined
- **PT tool life**—default: Not defined
- **RT:** Display field for remaining tool life
- **PZ quantity**—default: Not defined
- **RZ:** Display field for remaining quantity

- For drilling operations with "constant cutting speed," "drilling diameter I" is used to calculate the spindle speed.
- The parameters I and B are used to depict the cutting edge during simulation.
- Using the parameter "driven tool H," MANUALplus determines whether a driven tool is being used.

Tapping tools



Select tapping tools.



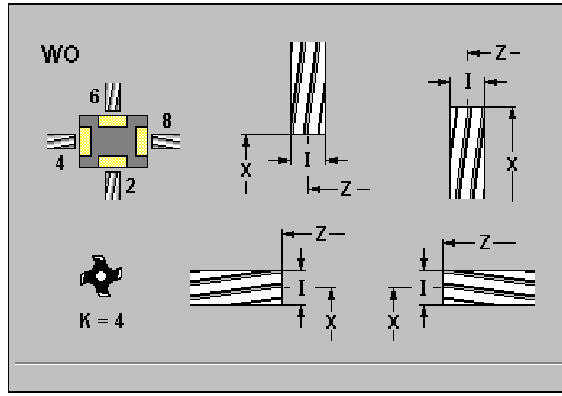
Tool parameters

- **`X setup dimension in X`**
- **`Z setup dimension in Z`**
- **`I diameter of thread`**
- **`WO tool orientation:`** For code number, see graphic support window
- **`F thread pitch`**
- **`DX wear compensation in X:`** Range: –100mm < DX < 100mm
- **`DZ wear compensation in Z:`** Range: –100mm < DZ < 100mm
- **`Q tool text:`** Reference to tool text
- **`H tool driven`**—default: 0
  - 0: Not driven
  - 1: Driven
- **`MD direction of rotation`**—default: Not defined
  - 3: M3
  - 4: M4
- **`TS cutting/spindle speed:`** default: Not defined
- **`TF feed rate`**—default: Not defined
- **`PT tool life`**—default: Not defined
- **`RT:`** Display field for remaining tool life
- **`PZ quantity`**—default: Not defined
- **`RZ:`** Display field for remaining quantity

- The "thread pitch K" is evaluated if the corresponding parameter is not defined in the tapping cycle.
- Using the parameter "driven tool H," MANUALplus determines whether a driven tool is being used.

Milling tools

Select milling tools.



Tool parameters

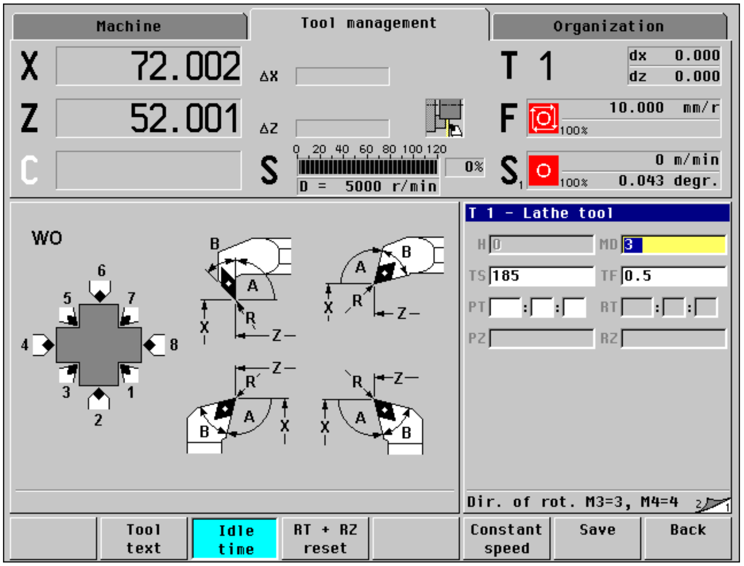
- **X setup dimension in X**
- **Z setup dimension in Z**
- **I cutter diameter**
- **WO tool orientation:** For code number, see graphic support window
- **K number of teeth**
- **DX wear compensation in X:** Range: –100mm < DX < 100mm
- **DZ wear compensation in Z:** Range: –100mm < DZ < 100mm
- **Q tool text:** Reference to tool text
- **MD direction of rotation**—default: Not defined
  - 3: M3
  - 4: M4
- **TS cutting/spindle speed:** default: Not defined
- **TF feed rate**—default: Not defined
- **PT tool life**—default: Not defined
- **RT:** Display field for remaining tool life
- **PZ quantity**—default: Not defined
- **RZ:** Display field for remaining quantity

- For milling operations with "constant cutting speed," "cutter diameter I" is used to calculate the spindle speed.
- The "number of teeth K" is evaluated for "G913 Feed per tooth" (see “Feed Rate and Spindle Speed” on page 297).
- The parameter I is used to depict the tool during simulation.

Danger of collision!

Be sure to specify the direction of tool rotation.

7.5 Tool Data — Supplementary Parameters



The second input window contains information on direction of rotation, cutting data, data on tool life monitoring, etc.

You can switch between the input windows using PgUp/PgDn.

Driven tool

The "Tool driven" parameter allows you to define for drilling and tapping tools whether switching commands are generated for the spindle or the driven tool. Milling tools are always considered "driven tools."

Direction of rotation

If you define a direction of rotation, a switching command (M3 or M4) is automatically generated for the spindle or, with driven tools, for the auxiliary spindle in all cycles that use this tool.

> It depends on the PLC software of your machine whether the generated switching commands are evaluated. If the PLC does not execute the switching commands, they should not be defined. Your machine manual provides more detailed information on switching commands.

Cutting data

Cutting data
You can transfer the parameters "cutting speed TS" and "feed TF" as cycle parameters or machine data by pressing the ***S, F from tool*** soft key.

When presetting the spindle speed, you can choose between "constant speed" and "constant cutting speed." The setting that you define in the tool parameters is then taken over whenever you press ***S, F from tool.***

With driven tools, the cutting data always refer to the auxiliary spindle.

Tool life management

MANUALplus can "count" either the machining time of a tool (i.e. the time a tool is traversed at the programmed feed rate) or the number of parts that were produced with that tool. These two options are used for tool life management.

As soon as the tool life expires or the programmed quantity is reached, the system interrupts machining and asks you to replace the tool or cutting edge. The machining operation, however, is not interrupted until the workpiece that is currently being produced is finished.

Depending on the status of the ***Tool life*** soft key, MANUALplus enables either the **Tool life** input field or the **No. pieces** input field. The fields "Rem. dwell RT / Rem. pieces RZ" show either the remaining life of your tool, or the remaining number of pieces that can be produced.

When you insert a new cutting edge, you must reset the tool life / quantity parameter with ***Reset RT + RZ.*** The parameter is set back to the initially programmed tool life or number of pieces.

- Tool life monitoring is activated/deactivated in "Current parameters—Tool monitoring."
- The quantity is counted when the end of the program has been reached.
- Tool life / quantity monitoring is also continued after a change of program.

8

Organization Mode of Operation

8.1 Organization Mode of Operation

This mode of operation offers various functions for communication with other systems, data security, setting of parameters, and diagnosis.

The following functions are available:

- **Parameter settings**
  Parameters enable you to adapt MANUALplus to your specific requirements. The "Parameter" menu provides functions to display and edit parameters.
- **Transfer**
  Input and output of programs, parameters, and tool data. The transfer functions are used either for exchange of data with other systems or for data security.
- **System Service**
  Some parameter settings and functions may only be accessed by qualified personnel. User registration and administration is done in the System service mode.
  You can also set the date and time and select the screen language in this mode.
- **Diagnosis**
  The Diagnosis menu provides functions for checking the system and for locating errors.

Menu logic
Each menu item in the Organization mode is preceded by a number. By pressing the corresponding key on the numerical keypad, you can activate the function, open a display or input window, or call the next menu level.

Some functions in the service and diagnosis menu can only be accessed by authorized commissioning and service personnel.

Basics of operation
Data is input in the usual way on the MANUALplus (see “Data input” on page 34). The data is transferred to the control when you press ***OK*** or place the cursor on the OK field and press "Enter." If you leave the input window with ***Cancel,*** then entries or changes you made will be lost.

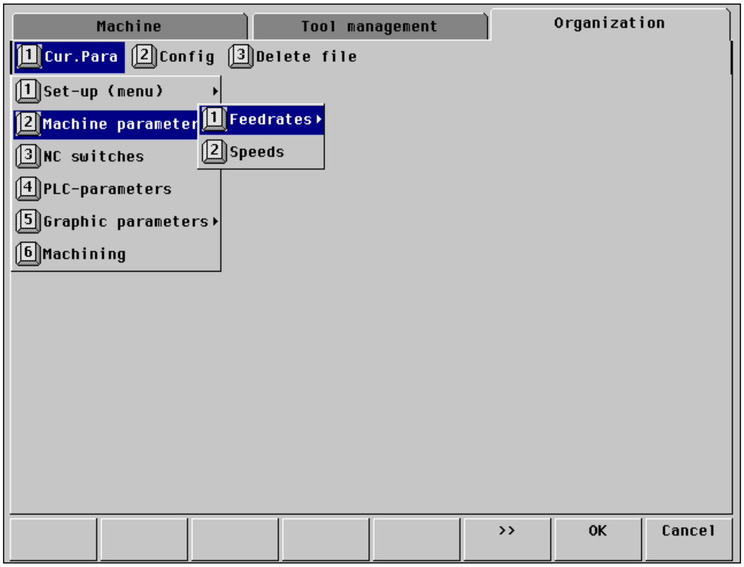
Switching to another function
The "Menu" key takes you back to the main level where you can then select another function of the Organization mode of operation.

8.2 Parameters

Parameters that are preset for the usual "daily operations" are grouped under the menu item **Cur.(rent) para(meters) [1].**

Selecting this menu item calls the following:

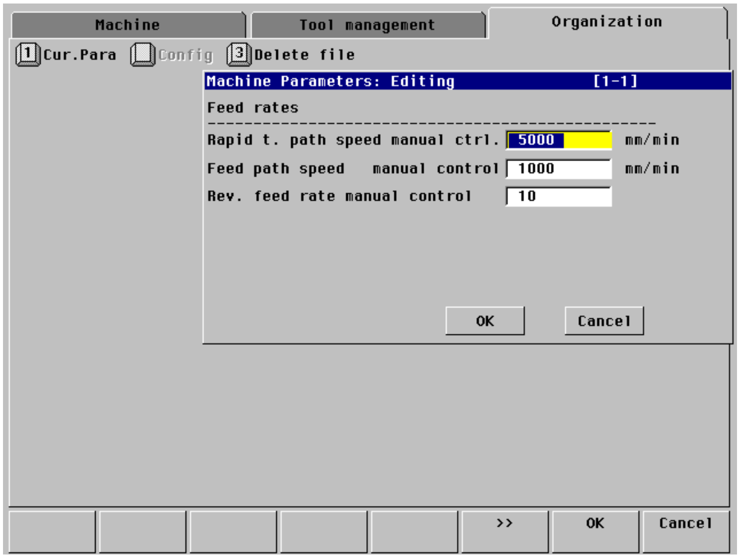
- Setup (menu) [1]
- Machine parameters [2]
- NC switches [3]
- PLC parameters [4]
- Graphics parameters [5]
- Machining [6]



A small arrow to the right of the menu line indicates that a menu item has a submenu (see figure to the top right). After you have selected a parameter, the input window is opened.

Alternately, some parameters can be set in the Machine mode of operation.

You can call the **Config(uration parameters) [2]** menu item only with "system manager" authorization (see "Access authorization" on page 453).



Displaying and editing parameters

A parameter consists of a number of parameter values. The parameter values are displayed and edited in one or several input windows.

The title bar of the input window indicates the parameter designation and the number of windows. Each parameter value is explained by the text next to the input field.

For "current parameters" which refer to a slide or a spindle, the number of the slide or spindle is displayed to the bottom left. With configuration parameters, the parameter number is displayed to the bottom left.

The parameter editor automatically considers the metric or inch units of measure.

> Some parameters are reserved for authorized service and commissioning personnel.

Current parameters

Current parameters	
"Setup (menu) [1]" menu item	
Workpiece zero point [1]—Main spindle [1]	Distance between "Machine zero point and workpiece zero point" (usually determined with "Set axis values"). ■ Zero point coordinate X [mm] ■ Zero point coordinate Z [mm]
Tool change point [2]	Distance between "Machine zero point and tool change point" (usually determined with "Set tool change point"). ■ Tool change point X coordinate [mm] ■ Tool change point Z coordinate [mm]
Zero point shift, C axis [3]	■ Zero point shift, C axis [°]
Tool monitoring [4]	Activate/deactivate tool life monitoring ■ Tool life switch ■ 0: Off ■ 1: On
Additive compensation [5]	16 compensation value sets—you can also define this parameter with "Set additive compensation." ■ Compensation 901 in X ■ Compensation 901 in Z ■ Compensation 902 in X ■ Compensation 902 in Z ■ . . .
"Machine parameters [2]" menu item	
Feed rates [1]—Manual control [1]	■ Rapid traverse contouring speed for manual control ■ Feed rate contouring speed for manual control (usually set with "Set T, S, F") ■ Feed per revolution for manual control
Feed rates [1]—Automatic mode [2]	■ Feed rate for rapid traverse in X ■ Feed-rate for rapid traverse in Z

Current parameters	
Speeds [2]	For spindle 1 (main spindle) and spindle 2 (driven tool): ■ Zero point shift (M19) [°] The parameter determines the offset in position between the spindle reference point and the reference point of the angle encoder (rotary encoder). After receiving the reference pulse from the rotary encoder, the current actual position is overwritten by the parameter value. ■ Number of revolutions for chip breaking Number of additional spindle revolutions for disengaging the tool during spindle stop. ■ M5/M19 Angle (usually set with "Set T, S, F"). ■ Speed value VConstant (G96) (usually set with "Set T, S, F"). ■ Speed value NConstant (G97) (usually set with "Set T, S, F"). ■ Speed limit (G26) (usually set with "Set T, S, F").
"NC switches [3]" menu item	
Display type [1]	The data is displayed in the "Actual value display" fields (machine window). ■ Actual display type—the numbers have the following meaning: ■ 0: Actual value ■ 1: Following error ■ 2: Distance of traverse ■ 3: Tool tip—referenced to machine zero point ■ 4: Slide position ■ 5: Distance between reference cams / reference pulse ■ 6: Nominal position ■ 7: Difference between tool tip / slide position ■ 8: Nominal IPO position
Tool measurement type [2]	This parameter defines how to determine the tool set-up dimensions in setup mode. ■ Type (of tool measurement): ■ 0: Touch-off ■ 1: Touch probe ■ 2: Optical gauge ■ Measuring feed: Feed rate for approaching the touch probe ■ Measuring range: ■ Measuring range: The tool stops when it has traversed to the maximum measuring range without reaching the touch probe.

Current parameters	

Settings [3]	Set the system to "metric mode" or "inch mode" and define the behavior for searching the start block.
	
	Changes do not take effect until the control is restarted.
	■ Output to printer—non-functional
	■ Metric/Inch
	  ■ 0: Metric
	  ■ 1: Inch
	■ Start block search
	  ■ 0: Off
	  ■ 1: On (Note: The system must be prepared for the start block function.)

"PLC parameters [4]" menu item	The PLC parameters are described in your machine manual.
"Graphics parameters [5]" menu item	
Standard window size [1]	These parameters define the display area for the graphic simulation of DIN programs. MANUALplus accounts for the height and width of the screen and may even enlarge the window size in the vertical or horizontal direction. ■ Minimum X coordinate—smallest X coordinate displayed ■ Minimum Z coordinate—smallest Z coordinate displayed ■ Delta X—vertical expansion ■ Delta Z—horizontal expansion
Standard workpiece blank [2]	These parameters define the standard workpiece blank and are used for calculating the "unrolled lateral surface." ■ Outside diameter: The "unrolled lateral surface" is calculated from the diameter. ■ Length of blank: Horizontal dimension of "unrolled lateral surface." ■ Right edge of blank part: Position of the unrolled lateral surface relative to the coordinate origin. If you enter a positive value, the "right blank edge" is located to the right of the coordinate origin. ■ Inside diameter—non-functional
"Machining [6]" menu item	
Safety clearances [1]	The following safety clearances are used in several cycles and during execution of specific DIN cycles (see cycle definitions): ■ External safety clearance [SAR] ■ Internal safety clearance [SIR] ■ External on machined part [SAT] ■ Internal on machined part [SIT]

Configuration parameters

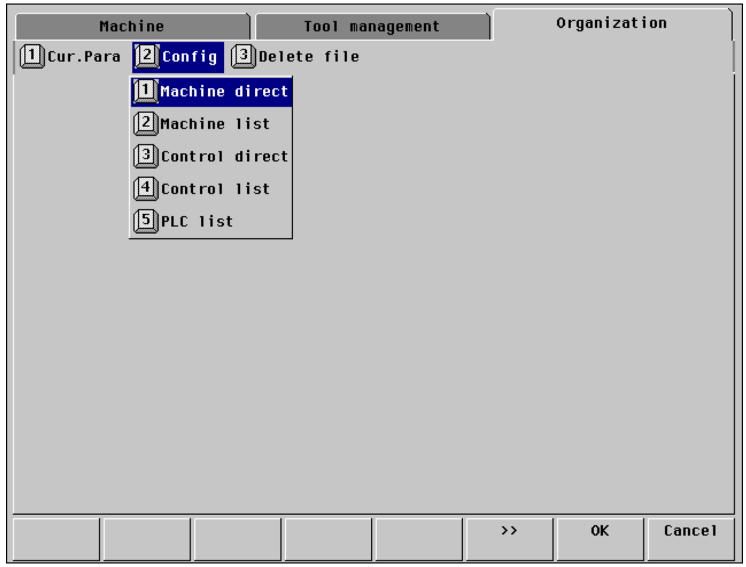
You can call the **Config(uration parameters) [2]** menu item only with "system manager" authorization (see "Access authorization" on page 453).

The configuration parameters are divided into three groups:

- Machine parameters
- Control parameters
- PLC parameters (see machine manual)

The parameters are identified with numbers. You can either call a parameter directly if you already know its number, or display the parameter list. In the parameter list, you then simply highlight the desired parameter and confirm your selection with ENTER.

> Some configuration parameters are also included in the "Current parameters" menu item.



Machine parameters (MP)	

Tool measurement [MP 6]	This parameter defines how to determine the tool set-up dimensions in setup mode.
	■ Type (of tool measurement):
	  ■ 0: Touch-off
	  ■ 1: Touch probe
	  ■ 2: Optical gauge
	■ Measuring feed: Feed rate for approaching the touch probe
	■ Measuring range:
	■ Measuring range: The tool stops when it has traversed to the maximum measuring range without reaching the touch probe.

Machine dimensions [MP 7]	Within the framework of variable programming, machine dimensions can be used in NC programs. ■ Dimension 1 in X [mm] ■ Dimension 1 in Z [mm] ■ . . .

Machine parameters (MP)	
Display setting [MP 17]	The data is displayed in the "Actual value display" fields (machine window). ■ Actual display type—the numbers have the following meaning: ■ 0: Actual value ■ 1: Following error ■ 2: Distance of traverse ■ 3: Tool tip—referenced to machine zero point ■ 4: Slide position ■ 5: Distance between reference cams / reference pulse ■ 6: Nominal position ■ 7: Difference between tool tip / slide position ■ 8: Nominal IPO position
Control configuration [MP 18]	■ PLC is to perform counting of workpieces: 0 = not active 1 = active ■ M0/M1 for all NC channels—configuration parameter ■ Interpreter stop during tool change 0 = not active 1 = active—look-ahead block interpretation is interrupted and only resumed after execution of the G14 function. ■ Option code 1—configuration parameter
Feed rates [MP 204]	■ Rapid traverse contouring speed for manual control ■ Feed rate contouring speed for manual control (usually set with "Set T, S, F") ■ Feed per revolution for manual control
Thread cutting [MP 208]	The coupling/decoupling path is used if the corresponding parameters are not programmed in the DIN program. ■ Coupling path [mm] Acceleration path at the beginning of a threading cut in order to synchronize the feed axis with the rotary axis. ■ Decoupling path [mm] Deceleration path at the end of the threading cut.
Position of the touch probe or optical gauge [MP 211]	To define the **position of the probe**, enter its external coordinates. To define the position of the **optical measuring system**, enter the position of the cross hairs. Reference: Machine zero point ■ Position of touch probe/optical gauge in +X ■ Position of touch probe in –X ■ Position of touch probe/optical gauge in +Z ■ Position of touch probe in –Z

Machine parameters (MP)	

Tool mount n [MP 601, ..]	If you use tool holders in different quadrants, the additional tool holder is defined as "mirrored" (see "Tools in different quadrants" on page 48). The distance between the additional tool holder and the principal tool holder is usually defined in "compensation X, Z."
	■ Compensation in X [mm]
	■ Compensation in Z [mm]
	■ Type of tool holder
	  ■ 0: Standard
	  ■ 1: Mirrored

General parameters for spindle [MP 805] / driven tool [MP 855]	■ Zero point shift (M19) [°] The parameter determines the offset in position between the spindle reference point and the reference point of the angle encoder (rotary encoder). After receiving the reference pulse from the rotary encoder, the current actual position is overwritten by the parameter value. ■ Number of revolutions for chip breaking Number of additional spindle revolutions for disengaging the tool during spindle stop. ■ M5/M19 Angle (usually set with "Set T, S, F"). ■ Speed value VConstant (G96) (usually set with "Set T, S, F"). ■ Speed value NConstant (G97) (usually set with "Set T, S, F"). ■ Speed limit (G26) (usually set with "Set T, S, F").
Tolerance values for spindle [MP 806] / driven tool [MP 856]	Configuration parameters

Backlash compensation in linear axis (X) [MP 1107] / linear axis (Z) [MP 1157]	The backlash compensation takes into account the "value of backlash compensation" for every change of direction. This enables you to compensate for backlash between the speed encoder and the table when the drive and encoder are connected directly.
	■ Type of backlash compensation
	  ■ 0: No compensation
	  ■ 1: Value of backlash compensation is added
	■ Value of backlash compensation

Limit switches, protection zone, feed rates for linear axis (X) [MP 1116] / linear axis (Z) [MP 1166]	For linear axis Z (parameter 1166), the value defined in "Set protect. zone" is transferred. ■ Protection zone dimension negative [mm] ■ Protection zone dimension positive [mm] ■ Feed rate for rapid traverse [mm/min] ■ Reference dimension [mm]
Misalignment compensation in linear axis (X) [MP 1120] /linear axis (Z) [MP 1170]	Configuration parameters

Control parameters (SP)	

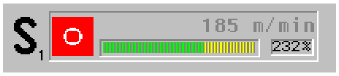
Settings [SP 1]	Set the system to "metric mode" or "inch mode" and define the behavior for searching the start block.
	■ Output to printer—non-functional
	■ Metric/Inch
	  ■ 0: Metric
	  ■ 1: Inch
	■ Start block search
	  ■ 0: Off
	  ■ 1: On (Note: The system must be prepared for the start block function.)

Time calculation for simulation, general [SP 20]	The times set in this parameter are taken into account for calculating the idle machine times. ■ Tool change time [sec] ■ Gear shifting time [sec] ■ Time allowance for M functions [sec]
Time calculation for simulation: M function [SP 21]	For M functions, the control calculates the "time allowance for M functions" that was defined in parameter 20. In parameter 21, you can enter up to 10 M functions for which an additional time allowance is to be calculated. ■ 1st M function ■ Time allowance [sec] ■ 2nd M function ■ Time allowance [sec] ■ . . .
Standard window size [SP 22]	These parameters define the display area for the graphic simulation of DIN programs. MANUALplus accounts for the height and width of the screen and may even enlarge the window size in the vertical or horizontal direction. ■ Minimum X coordinate—smallest X coordinate displayed ■ Minimum Z coordinate—smallest Z coordinate displayed ■ Delta X—vertical expansion ■ Delta Z—horizontal expansion
Standard workpiece blank [SP 23]	These parameters define the standard workpiece blank and are used for calculating the "unrolled lateral surface." ■ Outside diameter: The "unrolled lateral surface" is calculated from the diameter. ■ Length of blank: Horizontal dimension of "unrolled lateral surface." ■ Right edge of blank part: Position of the unrolled lateral surface relative to the coordinate origin. If you enter a positive value, the "right blank edge" is located to the right of the coordinate origin. ■ Inside diameter—non-functional

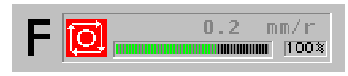
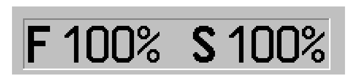
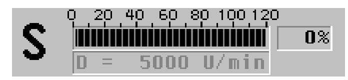
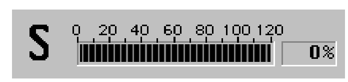
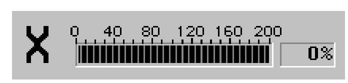
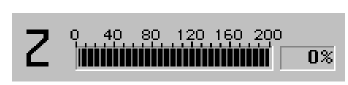
Control parameters (SP)	
Simulation: Settings [SP 27]	The machining simulation is delayed by the "path delay" time after the path has been simulated graphically. The simulation speed can thus be influenced. ■ Path delay
Allocation to interfaces [SP 40] **Interface 1 [SP 41]** **Interface 2 [SP 42]**	MANUALplus stores the "settings" of the serial interface in these parameters. The parameters are usually defined in "Transfer—Settings" (see "Settings in the "Serial" and "Printer" modes" on page 445).
Transfer directory [SP 48]	The parameters are usually defined in "Transfer—Settings" (see "Settings in "Network" mode" on page 444). ■ PCDIRECT directory: Path of directory offered and indicated for data transfer with PCDIRECT. ■ NETWORK directory: Path of directory offered and indicated for data transfer with NETWORK.
Display type 1, manual control [SP 301]	With these parameters, you can configure your machine display. (The arrangement of the machine display fields and code numbers of the display symbols are described in the following tables.) ■ Symbol for field 1: Enter the code number of the symbol ■ Slide/spindle: Enter "0". ■ Component group: Enter "0". ■ Symbol for field 2 ■ . . .

Arrangement of the machine display fields		
Field 1	Field 4	Field 7
Field 2	Field 5	Field 8
Field 3	Field 6	Field 9

Code numbers of the machine display symbols		
0	Special code, no display	
1	Actual X-position display	

Code numbers of the machine display symbols		
60	Actual/nominal value of spindle	
61	Spindle and speed information	

Code numbers of the machine display symbols		
2	Actual Z-position display	
3	Actual C-position display	
5	Distance to go in X	
6	Distance to go in Z	
7	Distance to go in C	
9	Distance to go in Z and protection zone status display	
10	Actual value displays for X, Z and C	
21	Tool display with compensation values (DX, DZ)	
23	Additive compensation	

Code numbers of the machine display symbols		
69	Slide and feed rate information	
70	Actual/nominal value of slide	
81	Overview of enabled elements	
82	Feed rate and speed override	
87	Utilization display for spindle and display of maximum speed	
91	Utilization display for spindle	
92	Utilization display for X axis	
93	Utilization display for Z axis	
99	Empty field	

8.3 Transfer

The Transfer mode is used for **data backup** and **data exchange** with PCs. When we speak of "files" in the following, we mean programs, parameters and tool data. The following file types can be transferred:

- Programs (cycle programs, DIN programs, DIN macros, and ICP contour descriptions)
- Parameters
- Tool data

Data backup

HEIDENHAIN recommends backing up the tool data and programs created on MANUALplus on a PC at regular intervals.

You should also back up the parameters. Since the parameters are not changed very often, however, you only need to back up the parameters from time to time, as required.

- For security reasons, parameters are only transferred after logon as "system manager" (see "Access authorization" on page 453).
- Once you have logged on as "system manager," you can also transfer and print "diagnostic files." The creation and evaluation of "diagnostic files" is intended for service purposes only.

Data exchange with DataPilot 4110

HEIDENHAIN offers the PC program package DataPilot 4110 to complement the MANUALplus control. The DataPilot provides the same programming and test functions as MANUALplus. This offers you the advantage of defining cycle programs, DIN programs or ICP contours with DataPilot, running a graphic simulation of the created programs and contours, and then transferring them to the control.

DataPilot is suited for data backup. Of course, you can also use alternative Windows operating system functions or other PC programs available on the market for data backup.

Printer

MANUALplus supports the output of DIN programs and DIN macros to a printer over the serial interface. (Cycle programs and ICP contour descriptions cannot be printed.)

The data are processed for printout in size A4 format.

Interfaces

Data transfer is carried out over the **Ethernet** or the **serial interface.** We recommend using transfer modes via Ethernet interface, since the transmission rate and transmission security are higher than with serial interfaces.

- **WINDOWS networks** (via Ethernet):
  With a WINDOWS network you can integrate your lathe in a LAN network. MANUALplus supports the networks provided by WINDOWS. MANUALplus allows you to send/receive files. Other computer systems integrated in the network have access to read from and write to "shared" directories, independent of MANUALplus's activities.

Danger of collision!

Other computer systems in the network may overwrite MANUALplus programs. When organizing the network and granting access rights, ensure that only authorized persons have access to MANUALplus.

- **Serial data transfer:**
  It is important to ensure that the interface parameters (baud rate, word length, etc.) comply with those of the remote station.
- **Printer:**
  The interface parameters (baud rate, word length, etc.) must also comply with those of the printer.

Basics of data transfer

MANUALplus (and DataPilot) manage DIN programs, DIN macros, cycle programs and ICP contours in different directories. When you select "Program group," MANUALplus and DataPilot automatically switch to the applicable directories.

Parameters and tool data are stored in the remote station under the file name entered for "Backup name." The "Backup name" is displayed for your information. It can be changed by service personnel only.

Automatic logon

If "Auto-logon—Yes" has been selected, you are automatically logged on to the network by MANUALplus. Enter your "user name" and "password" required for logon into the "Settings" dialog box. If you do not use the automatic logon function, you must enter your user name and password when starting the system. The disadvantage of working without the automatic logon function is that these entries are required each time you start the system and that you can only use numbers for your user name and password.

Access control for networks

Passwords for read/write access to directories can be assigned by the remote system (WINDOWS: "Access control to shared levels"). In this case, the "Enter network password" dialog box appears when you try to access directories of the remote system.

In Diagnosis mode, the MANUALplus files can be assigned passwords for read access or write access (see “Diagnosis” on page 455).

"Enter network password" dialog box: The dialog box is displayed by the WINDOWS operating system. The following rules apply:

- **>>** soft key: moves the cursor to the next input field or to the next button.
- "Store key": edits the entry in the "Save this password in your password list" input field.
- ENTER key: confirms and concludes the dialog box.

Please note: Only use numbers for your password.

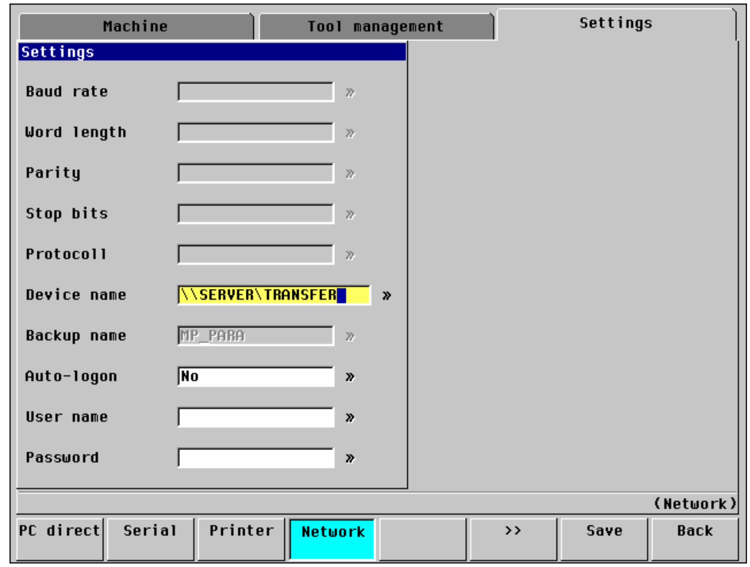
If only **one password** is used, it can be saved. This dialog box will then only appear once (or for changing the password). The password saved is checked each time you try to access further directories. If the password for read permission differs from the one for write permission, the "Enter network password" dialog box appears each time you try to access a directory after having restarted the MANUALplus.

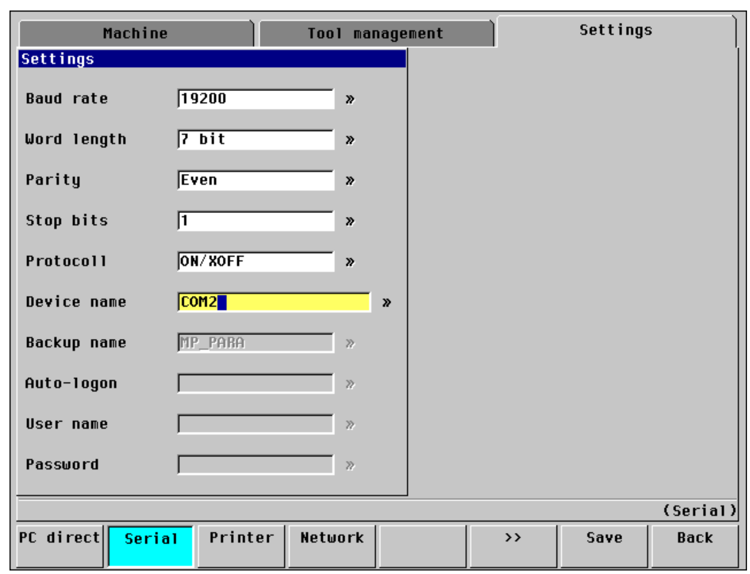
HEIDENHAIN recommends:

- Configuration of the Windows networks by the authorized personnel of your machine manufacturer.
- Use the automatic logon function.

Configuring for data transfer

Settings	Press ***Settings.***
Network	Press ***Network*** and define the directory of the remote station ("Device name" input field)—see figure to the upper right.
Serial	Press ***Serial*** and define the interface parameters—see figure to the lower right.
Printer	Press ***Printer*** and define the interface parameters.
Save	Transfer the settings.
Back	Press ***Back.***





Access authorization as “system manager” is necessary to access the settings (see “Access authorization” on page 453). The active mode is displayed to the right above the soft-key row.

Settings in "Network" mode

- **Device name:** Enter the name and directory of the server as follows:
  //Computer name/path
  (The "/" character corresponds to the "\" character on the PC; The "computer name" and the "share name" are set on the PC of the remote station.)
- **Auto-logon**
  - Yes: Automatic logon is active.
  - No: Automatic logon is not active.
- **User name:** For automatic logon
- **Password:** For automatic logon

Settings in the "Serial" and "Printer" modes

- **Baud rate:** In bits per second
- **Word length:** 7 or 8 bits per character
- **Parity:** Select even/odd parity or "no parity." The setting "word length = 8 bits" is required whenever you want to use "even/odd parity."
- **Stop bits:** 1, 1 1/2 and 2 stop bits
- **Protocol**
  - **Hardware** (hardware handshake) The receiver informs the sender through "RTS/CTS signals" that it is temporarily not able to receive data. A hardware handshake presupposes that the RTS/CTS signals are hardwired in the data transfer cable.
  - **XON/XOFF** (software handshake) The receiving terminal transmits "XOFF" if it is temporarily unable to receive data. As soon as the receiver can receive further data, is signalizes "XON" again. The software handshake does not require the transmission of RTS/CTS signals over the data transfer cable.
  - **ON/XOFF** (software handshake) The receiver transmits "XON" at the beginning of data transmission to indicate that it is ready to receive. When the receiver is temporarily not able to receive data, it sends "XOFF." As soon as the receiver can receive further data, is signalizes "XON" again. The software handshake does not require the transmission of RTS/CTS signals over the data transfer cable.
- **Device name:** Designation of the interface used, in most cases, "COM2."
- **Backup name:** Parameters and tool data are stored in the remote station under the file name entered for "Backup name." The "Backup name" is displayed for your information. It can be changed by service personnel only.

Transferring programs (files)

When selecting a program, place the highlight on the desired program and press ***Mark.*** You can also select all programs with ***Mark all.***

„A marked program is indicated with a diamond. To unmark a program, simply press ***Mark*** once again.

If you want to transfer a single program, place the highlight on the program and press ***Transmit file*** or ***Receive file.***

Below the window, MANUALplus displays the file size of the highlighted program and the time it was last changed.

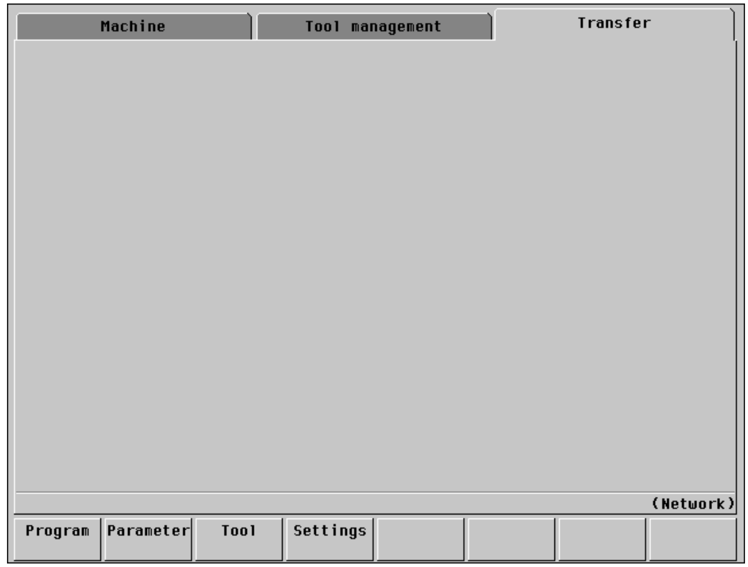
With DIN programs / DIN macros, you can also view the NC program with ***Program view.***

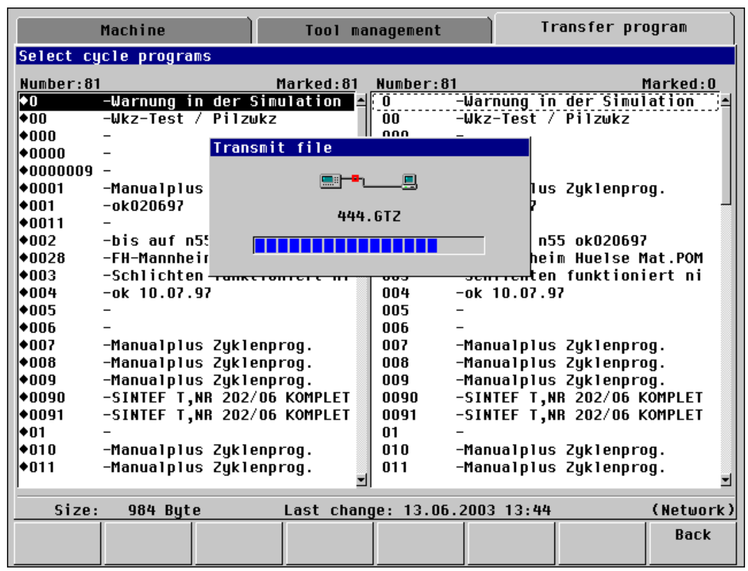
Parameters and tool data are sent/received as "blocks."

During transfer, MANUALplus displays the following information in a "transfer window" (see figure at bottom right):

- The transfer status in the form of a small red square, which moves between the control and the PC during transfer—if transfer is interrupted, the red square remains stationary.
- The name of the program which is currently being transferred.
- Amount of data already transferred is shown in the progress window.

- When receiving parameters and tool data, the previous data are **overwritten.**
- When you start "Network," MANUALplus reads the program names and program descriptions of the remote station. This process can take several minutes and is indicated in the "progress display" above the soft-key row.





Selecting the program group

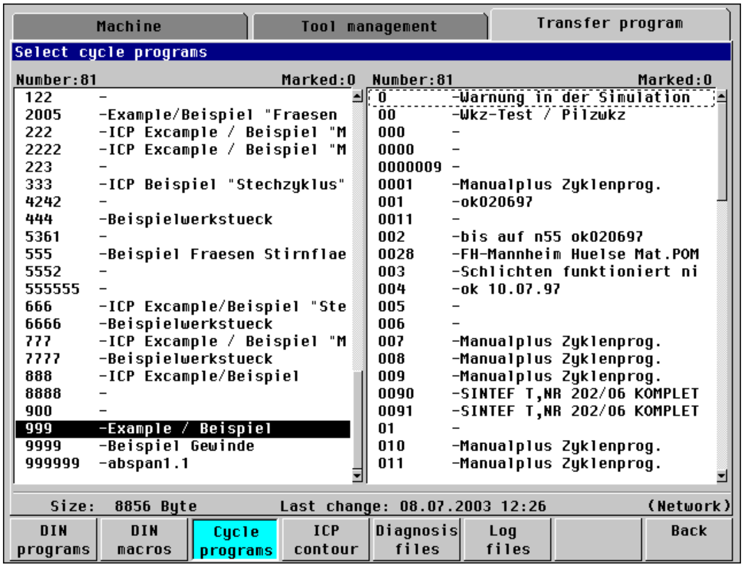
Program	Press ***Program.***
Program selection	Press ***Program selection.***
DIN programs	***DIN programs*** or
DIN macros	***DIN macros,*** or
Cycle programs	***Cycle programs,*** or
ICP contour	***ICP contours,*** or
DXF files	***DXF Import.***
Back	Press ***Back.***

When DIN programs, or cycle programs, or ICP contours have been selected, only the file name is displayed. MANUALplus uses the extensions (see table at right) to differentiate between the individual program groups.

If you log on as a "system manager," you can additionally select the following files:

- Diagnostic files: These files are important for commissioning and service.
- Log files: These files serve to display the stored error log file (file name: "error") and further files for commissioning and service.

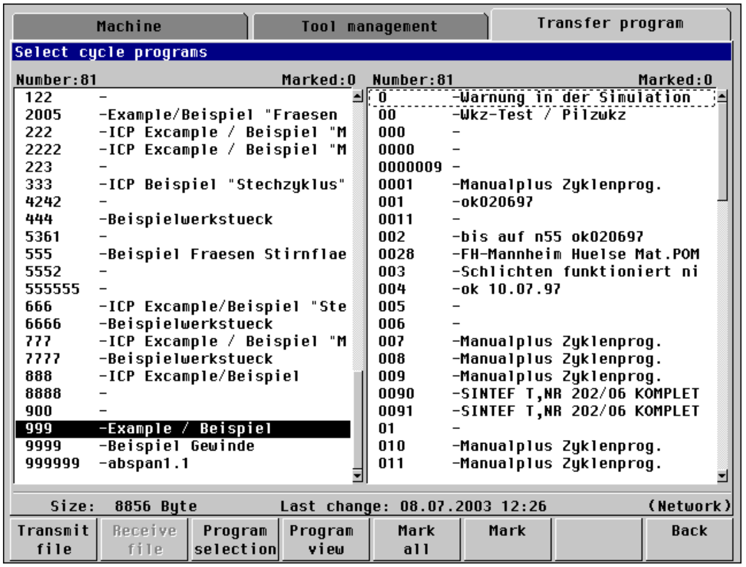
You can use the same program names for DIN programs, DIN macros, ICP turning contours, and ICP contours on face and lateral surface. It is therefore advisable to use the "program description" for explaining the program contents.



Program group extensions	
NC	DIN programs
NCS	DIN subprograms (DIN macros)
GTZ	Cycle programs
GTI	ICP turning contours
GTS	ICP face contours
GTM	ICP lateral surface contours
DXF	DXF contours

Program transfer (Network mode)

"Network" shows its own directory in the left window and the directory of the remote station in the right window (see figure to the right). To switch back and forth between the two windows, press the horizontal arrow keys (or ENTER).



Transmitting files

Program — Press ***Program.***

Place the highlight in the **left window.**

Highlight the program, or

Mark — Select and ***Mark*** programs, or

Mark all — Press ***Mark all.***

Transmit file — Press ***Transmit file.***

Receiving files

Program — Press ***Program.***

Place the highlight in the **right window.**

Highlight the program, or

Mark — Select and ***Mark*** programs, or

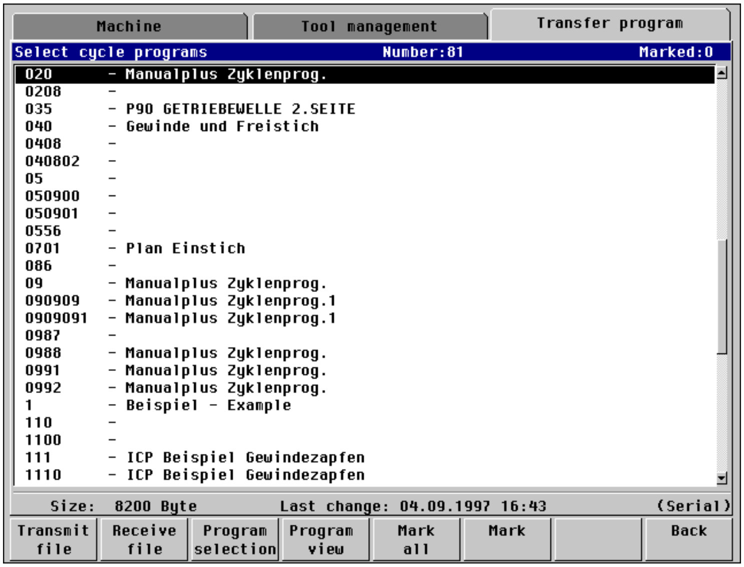
Mark all — Press ***Mark all.***

Receive file — Press ***Receive file.***

The information displayed during transfer is described in "Transferring programs (files)" on page 446.

Program transfer (Serial mode)

MANUALplus displays its own directory (see figure at right).



Transmitting files

Highlight the program, or

Mark — Select and ***Mark*** programs, or

Mark all — Press ***Mark all.***

Transmit file — Press ***Transmit file.***

Receiving files

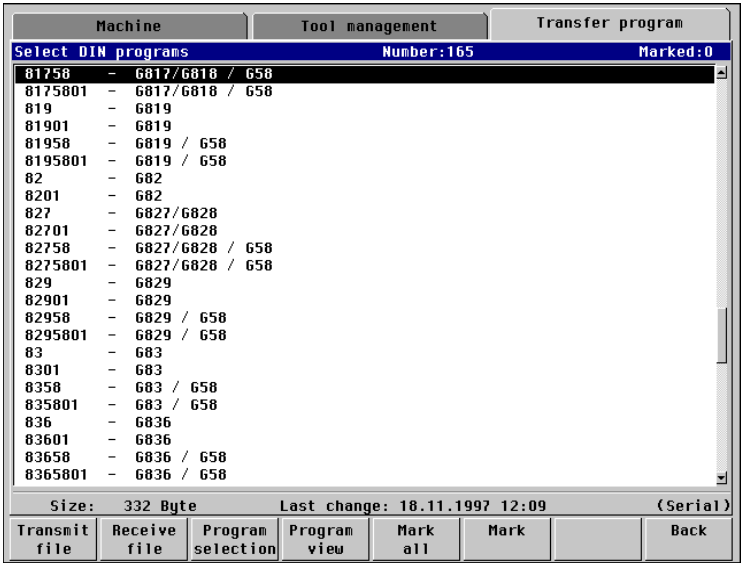
Receive file — Press ***Receive file.***

If you have selected "Receive file," MANUALplus waits for the transmission of data from the serial interface. The "progress display" shows that data transfer is active. If you want to cancel the receiving status, press ***Back.***

When **receiving programs,** MANUALplus accepts all program types (DIN programs, DIN macros, cycle programs, and ICP contours).

The information displayed during transfer is described in “Transferring programs (files)” on page 446.

Printing DIN programs/macros



`Program`	Press ***Program.***
`Program selection`	Press ***Program selection.***
`DIN programs`	Press ***DIN programs*** or
`DIN macros`	***DIN macros.***
`Back`	Press ***Back.***

Highlight the program, or

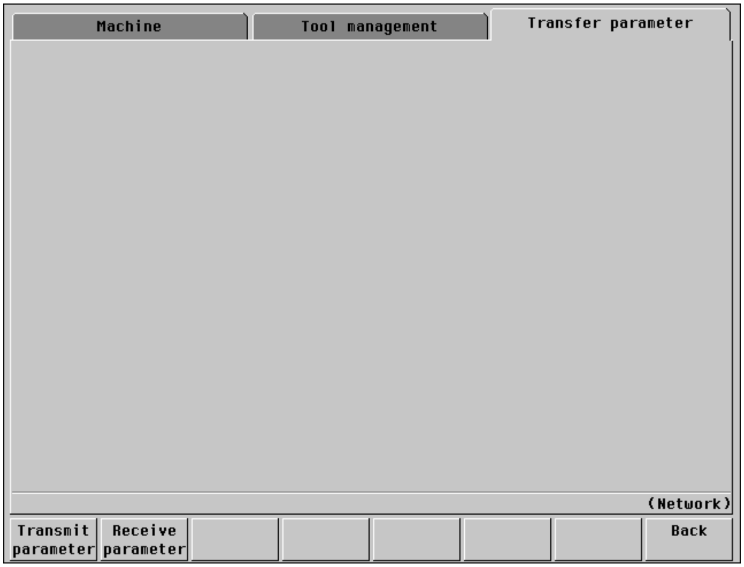
`Mark`	Select and ***Mark*** programs, or
`Mark all`	Press ***Mark all.***
`Transmit file`	Press ***Transmit file.***

The information displayed during transfer is described in "Transferring programs (files)" on page 446.

> You can only print out DIN programs and DIN macros.

Transferring parameters

Parameter	Press ***Parameter.***
Transmit parameter	Press ***Transmit parameter.***
Receive parameter	Press ***Receive parameter.***



The transmitted parameter files receive the file name that was entered for "Backup name" in the "Settings" menu. MANUALplus appends the following extension to the file name:

- *.BEA (machining parameters)
- *.MAS (machine parameters)
- *.PRO (production parameters)
- *.PLC (PLC parameters)
- *.STD (control data)

The information displayed during transfer is described in “Transferring programs (files)” on page 446.

- Receipt of parameter files:
  - "Network" mode: The file name is checked. It must match the file name in "Settings—Backup name."
  - "Serial" mode: The file name is not checked.

Important!

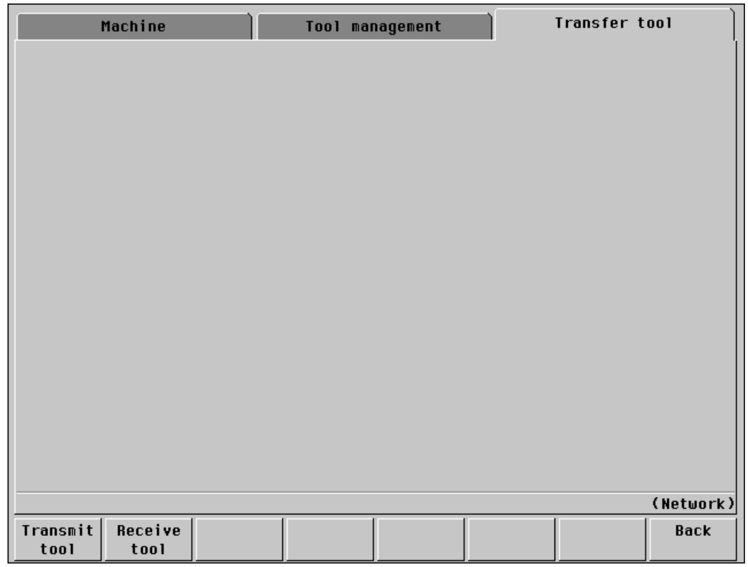
After receiving the parameter files, it is essential to restart MANUALplus.

Transferring tool data

Tool — Press ***Tool.***

Transmit tool — Press ***Transmit tool.***

Receive tool — Press ***Receive tool.***



The transmitted tool files receive the file name that was entered for "Backup name" in the "Settings" menu. MANUALplus appends the following extension to the file name:

- *.TXT (tool texts)
- *.WKZ (tool parameters)

The information displayed during transfer is described in "Transferring programs (files)" on page 446.

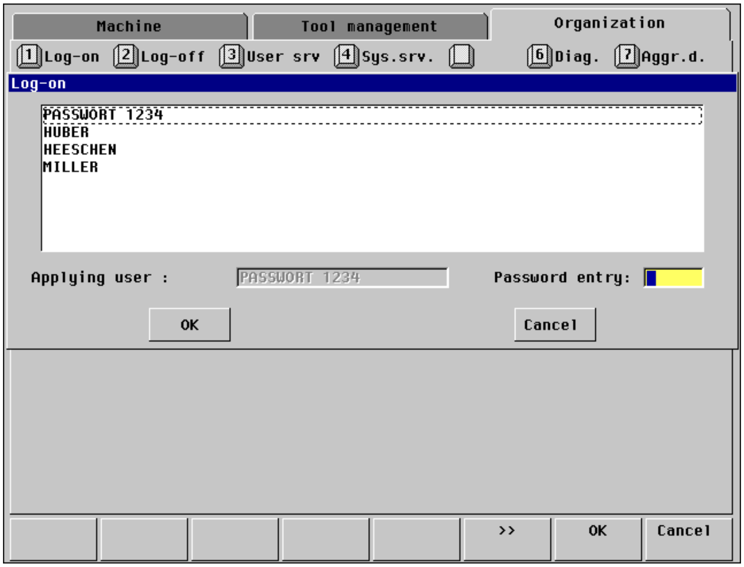
- Receipt of tool files:
  - "Network" mode: The file name is checked. It must match the file name in "Settings—Backup name."
  - "Serial" mode: The file name is not checked.

8.4 Service and Diagnosis

When you select **Service [3],** MANUALplus offers the following functions or function groups:

- Logon [1]
- Logoff [2]
- Usr. Srv. [3] (user service)
- Sys.Srv. [4] (system service)
- Diag.(nosis) [6]

Some service and diagnostic functions are not accessible (reserved for service and commissioning personnel).



Access authorization

The functions logon, logoff, and user service are provided for managing access authorization.

Certain parameter changes and specific functions from service/diagnosis may only be performed by authorized personnel. The control permits access when the correct **password** is entered. This access authorization is canceled again during "logoff." In the "user service" functions, you can enter and delete users, and assign and change passwords.

Each user is assigned a "password" consisting of a four-digit number that has to be memorized. The password is entered "masked" (not visible).

MANUALplus differentiates between the following user groups:

- Without protection class
- NC programmers
- System managers
- Service personnel (of the machine tool builder)

> MANUALplus is delivered with a preset authorization for the user "password 1234". The password is 1234. After you have logged on as the user "Password 1234", you can program the users that operate the machine (with system manager authorization). You should then delete the user "Password 1234".

Logon [1]

When "Logon" is selected, a list of the entered users is displayed. Select your name, press "Enter" and then enter your password. You are now logged on as an NC programmer or a system manager.

This logon remains effective until the "logoff" function is used or another user logs on with his password.

Logoff [2]

The present user is logged off and authorization is reset to "no protection class."

User service [3]

User service functions are only available after log-on as a system manager. The following functions are available:

- **Enter user [1]**
  Enter the name of the new user. For typing in the names, activate the alphanumeric keyboard with the ***>>*** key. Then assign a password for each name. The entered users are subsequently displayed in the "user list."
- **Cancel user [2]**
  Select the user to be deleted and confirm with ***OK.***
- **Change password [3]**
  Every user can change his or her password. To safeguard against misuse, the user must first enter the "old" password before assigning a new one.

System service

"System service" provides the following functions:

- **Date/Time [1]**
  Enter the date and/or time. Error messages are recorded together with the date and time they occurred. You should therefore always ensure that the date and time are correctly set.
- **Language switchover [3]**
  After calling this function, you can select the desired language with the ***>>*** soft key. Then confirm your selection with ***OK.***
  The selected language becomes effective as soon as you restart the control.

Diagnosis

The "Diagnosis" submenu provides information, test and control functions.

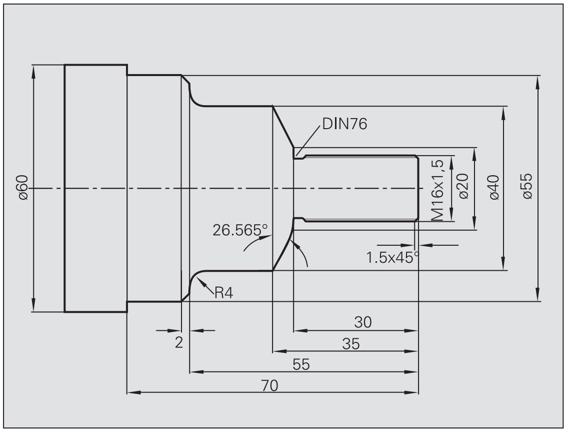
- **Info [1]:** provides information on the software version of your control.
- **Log files [3]—Display error log [1]:** displays the most recent error message. To view further entries, press the PgUp/PgDn keys.
- **Log file [3]—Save error log file [2]:** This function makes a copy of the error log file and stores the file with the name "error.log" in the "Para_Usr" directory. If the directory already contains an "error.log" file, it is overwritten.
  Application example: You save the error messages that have occurred in order to make them available to the service engineer.

Further diagnostic functions are available for commissioning and service.

9

Examples

9.1 Working with MANUALplus



The following example illustrates how to set up the machine and how to machine a workpiece using the cycle programming feature. The machining operation is to be performed in Teach-in mode. This has the advantage that, once you have machined the first workpiece, you have a cycle program that can be repeated any time.

Required tools

- **Roughing tool:**
  - Position T1
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.8 Tool radius
- **Finishing tool:**
  - Position T2
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.8 Tool radius
- **Threading tool:**
  - Position T3
  - WO = 1 Tool orientation

Sequence of working steps

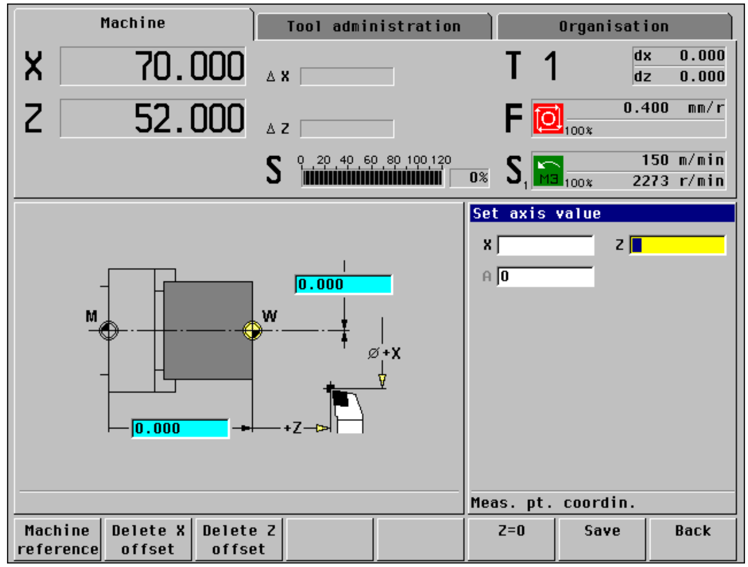
- Clamp a workpiece blank (diameter 60 mm, length 100 mm).
- Machine setup
  – Define the workpiece zero point
  – Measure the tool dimensions
- Switch to "Teach-in" mode.
- Machine the workpiece cycle by cycle.

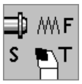
Setting up the machine

Prerequisite: Tools T1, T2, and T3 are entered.

Setting up the machine

Clamp the workpiece blank



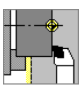


Insert the reference tool and specify the machine data in "Set T, S, F."

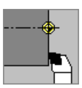
Prepare for setting the workpiece zero point and measuring the tools (in Manual mode with handwheels / jog controls):

- Machine an **end face.**
- Prepare the **diameter.**

Set the workpiece zero point



Select "Setup."

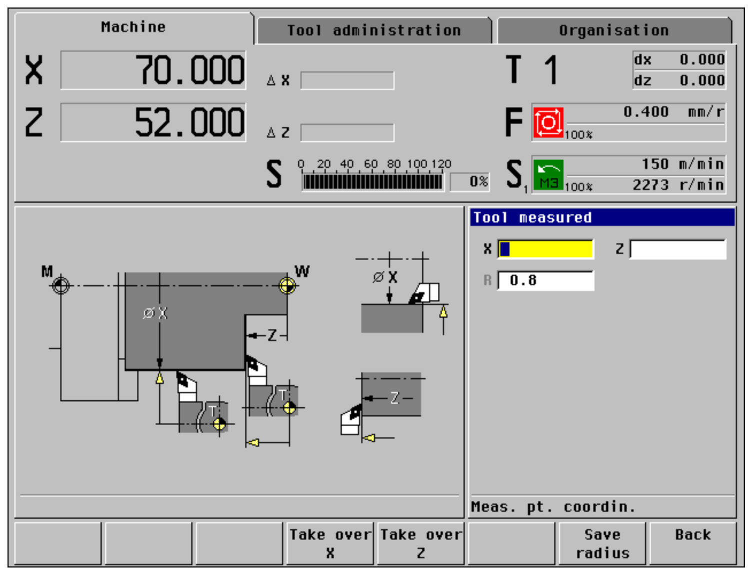


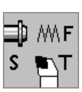
Press "Set axis values."

Z=0

Touch the end face and define this coordinate as the workpiece zero point.

Measure tool (for all tools):





Insert the tool and define the T number in "Set T, S, F."

Measure tool

Press ***Measure tool.***

Touch the diameter and enter the value as "Measuring point coordinate X."

Touch the end face and enter "0" as "Measuring point coordinate Z." (The end face has now been defined as the workpiece zero point.)

Selecting a cycle program

A new cycle program with the number "999" is created.

Creating a cycle program

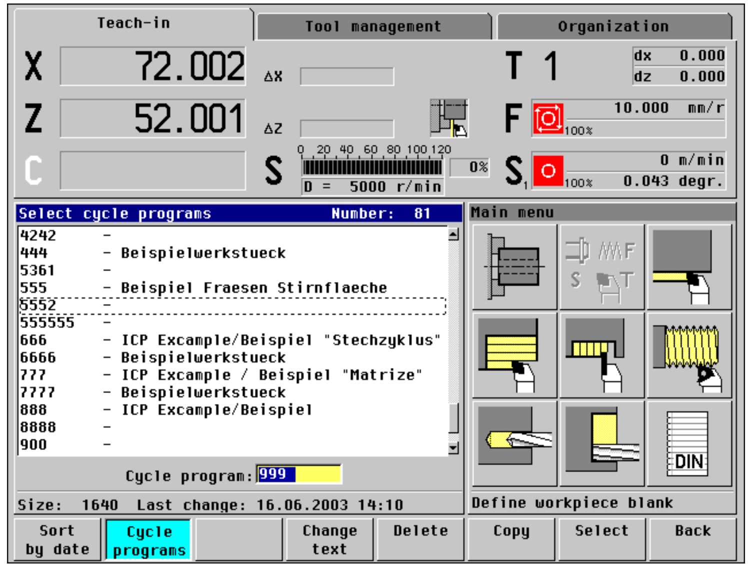
Teach-in	Switch to Teach-in mode.
Program list	Press ***Program list.***

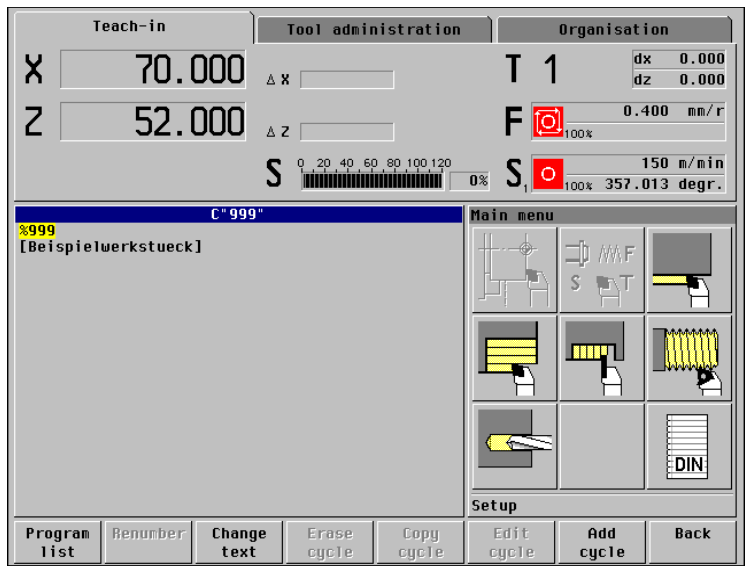
Enter "999" as program number.

Select	Activate program "999".
Change text	Press ***Change text.***

Enter the program designation (here: "Example workpiece").

Save	Transfer the program designation.
Add cycle	Start programming the cycle.





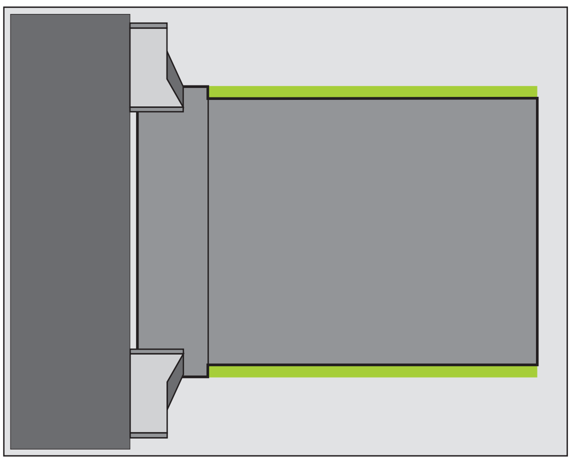
Creating a cycle program

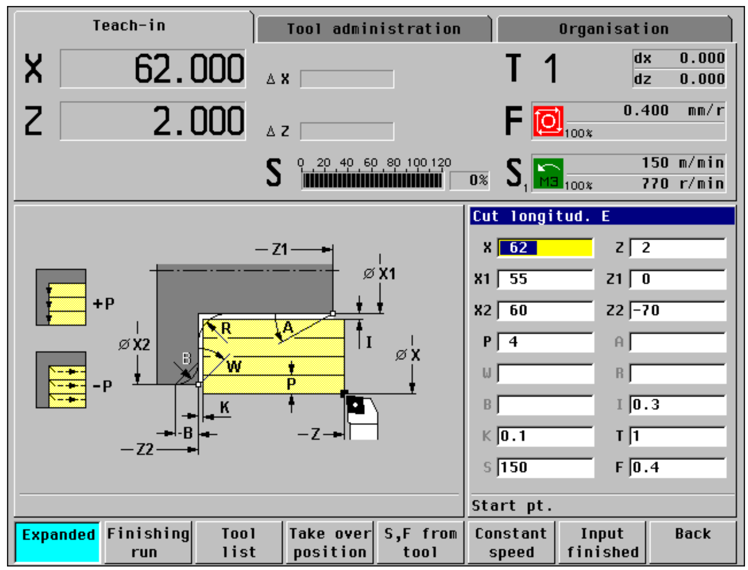
The individual cycles for machining workpieces are described below. The current working step is displayed in the workpiece graphic; the cycles and the cycle parameters are displayed in the graphics to the right. The machine data display indicates the status **after** execution of the cycle.

Sequence of working steps for each cycle:

- Select the cycle.
- Program the cycle.
- Check the cycle by running a graphical simulation.
- Run the cycle.
- Save the cycle.

Roughing cycle 1





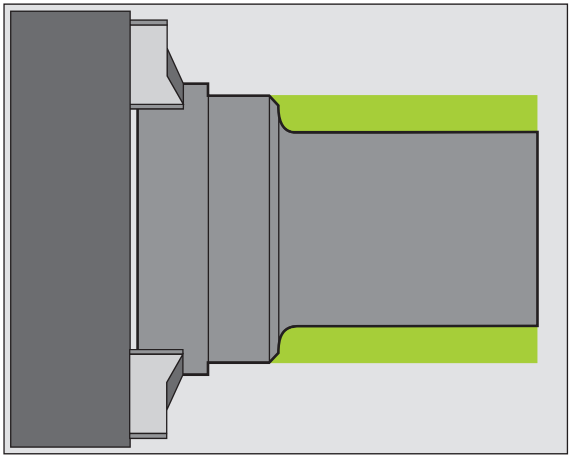
First insert the roughing tool.

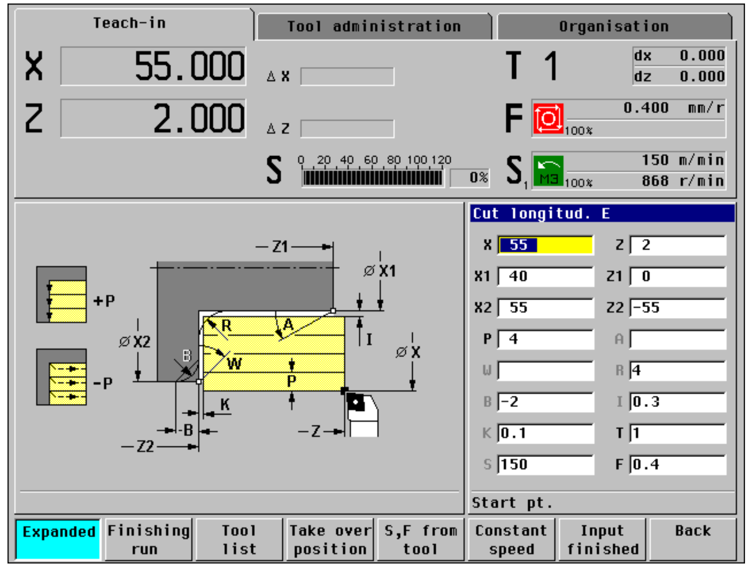
The cycle machines the area marked in the drawing. Expanded mode is selected for defining the allowances.

The "starting point X, Z" is defined such that it is located shortly before the area to be machined. It is approached at rapid traverse.

With roughing cycles, the tool returns to the starting point at the end of the cycle.

Roughing cycle 2

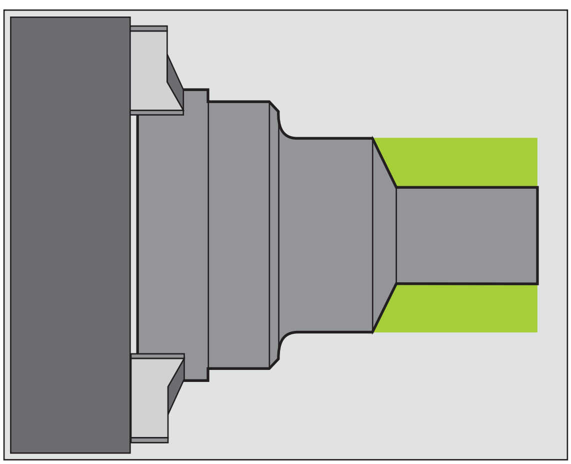


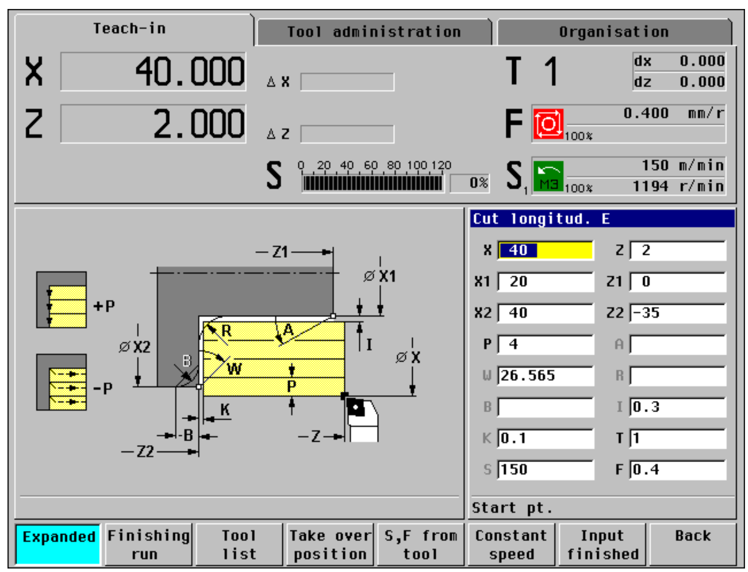


The "starting point X, Z" is defined such that it is located shortly before the area to be machined. It is approached at rapid traverse.

The cycle machines the area marked in the drawing. The expanded mode is selected for programming the allowances, the rounding and the chamfer.

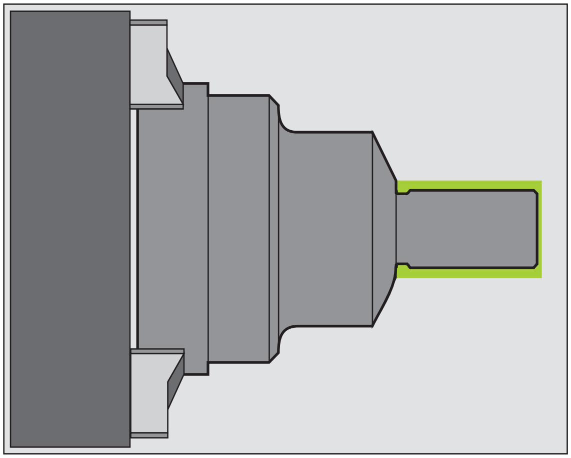
Roughing cycle 3



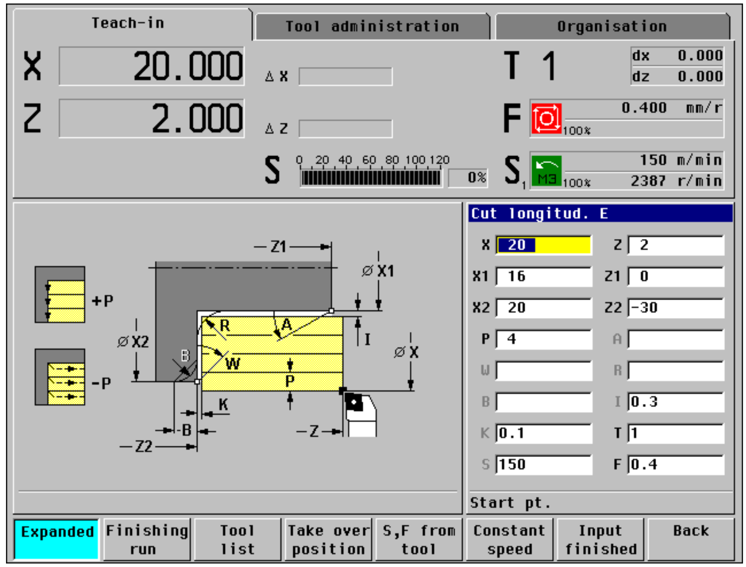


The cycle machines the area marked in the drawing. The expanded mode is selected for programming the allowances and the oblique cut.

Roughing cycle 4

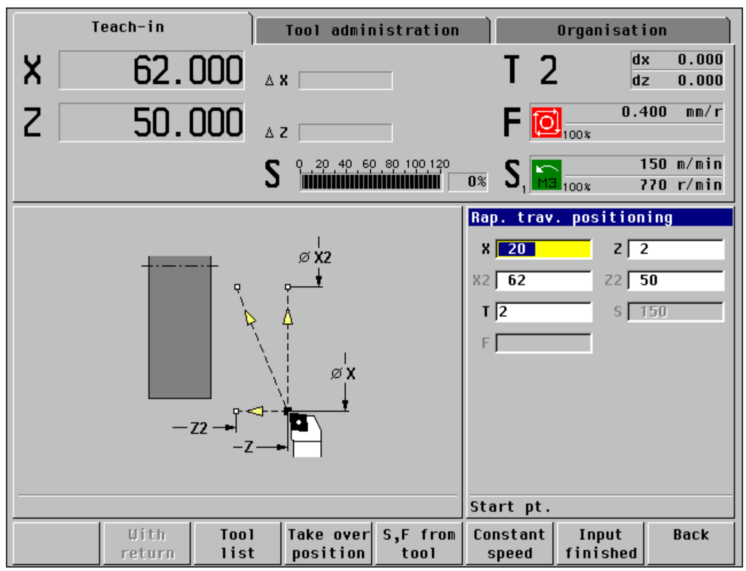


The cycle machines the area marked in the drawing. The expanded mode is selected for programming the allowances.

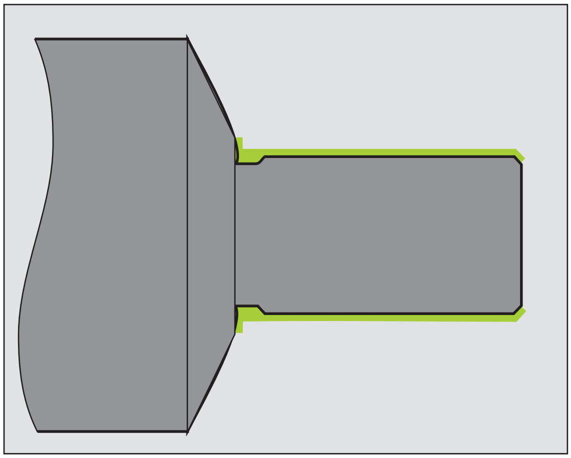


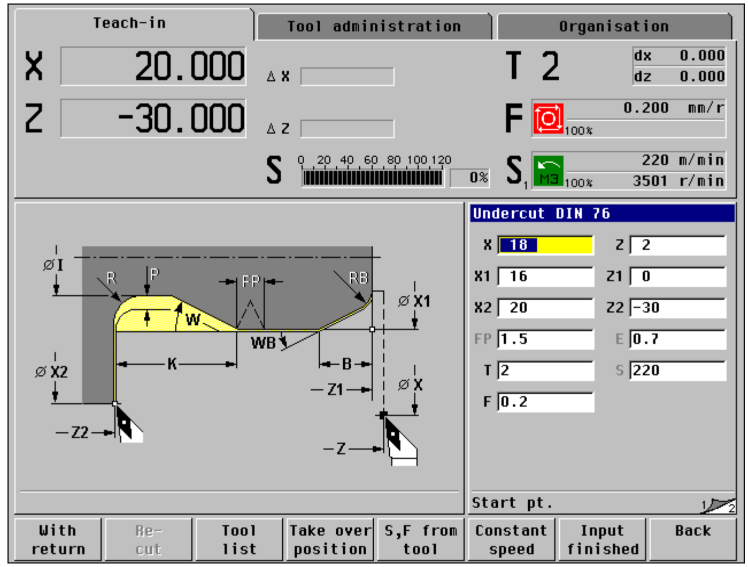
Positioning the tool for tool change

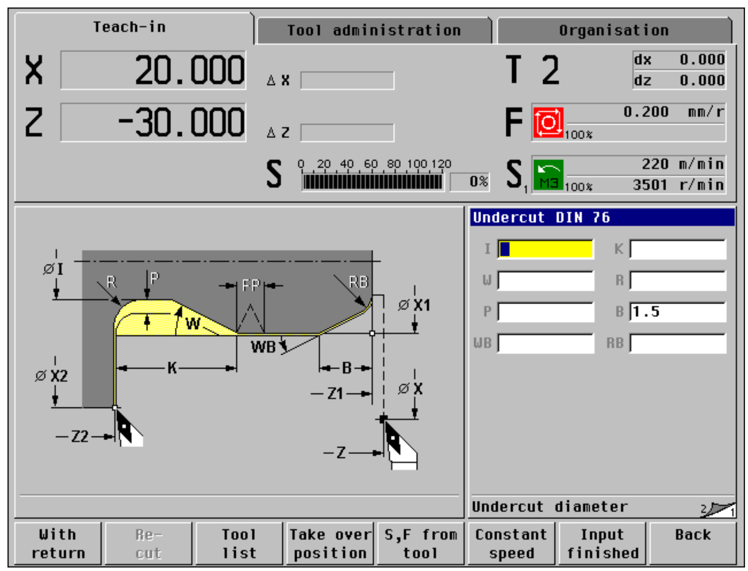
Before you can replace the roughing tool by the finishing tool, you must move it to a "safe position."



Machining a thread chamfer and undercut







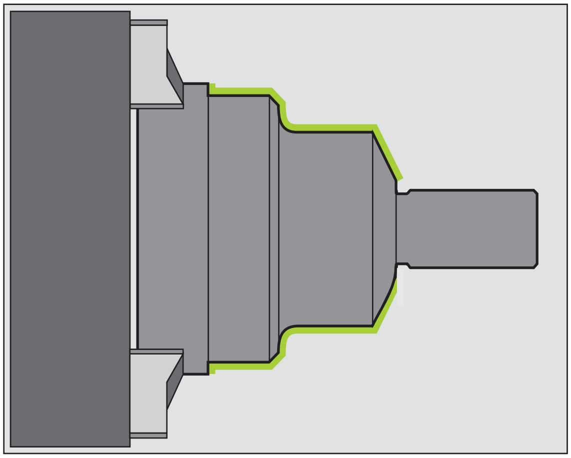
The thread chamfer / undercut and the following finishing cycles are programmed in such a way that the contour area is machined in a single uninterrupted cut.

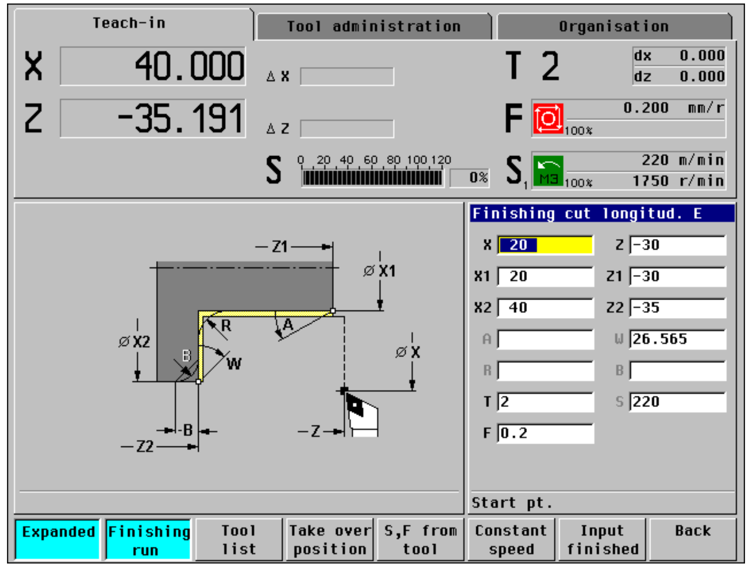
MANUALplus approaches the "starting point X, Z" in rapid traverse. No further positioning movements therefore need to be programmed.

The thread chamfer and undercut are machined with the "Undercut DIN 76" cycle.

"With return" is switched off. This enables you to finish the contour area in a single uninterrupted cut.

Finishing cycle 1

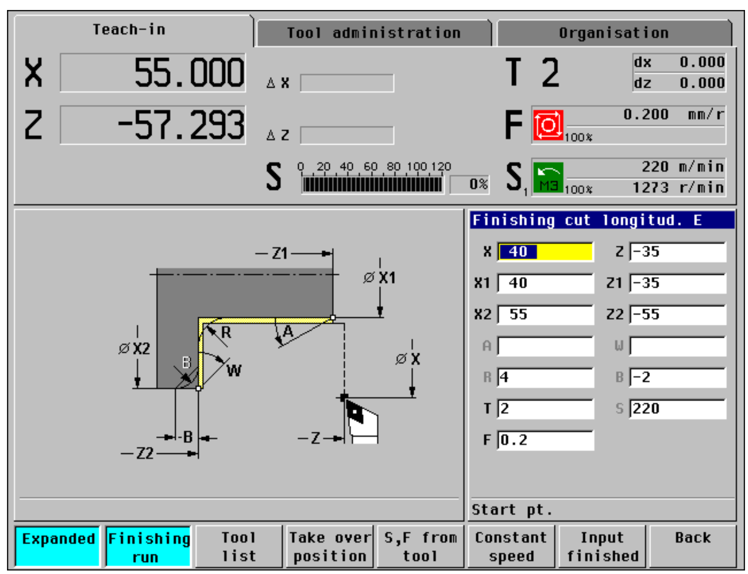




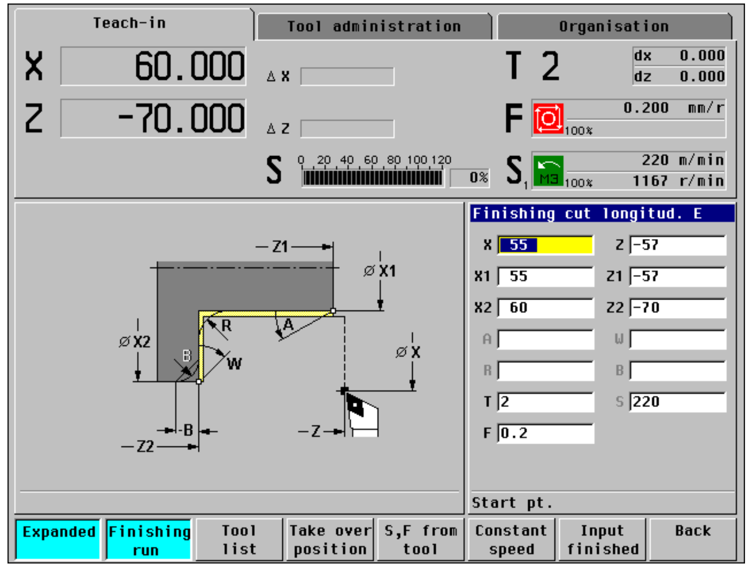
The three following finishing cycles finish the contour area shown in the graphic.

The expanded mode is used for all finishing cycles so that contour elements such as oblique cuts, roundings, or chamfers can be machined. In expanded mode, the tool stops at the end of the cycle. This is necessary to be able to finish the contour area "in a single cut."

Finishing cycle 2

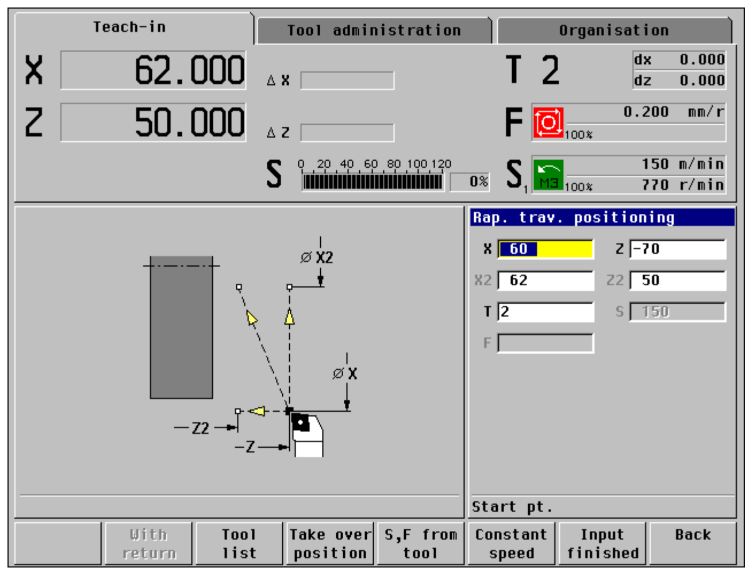


Finishing cycle 3

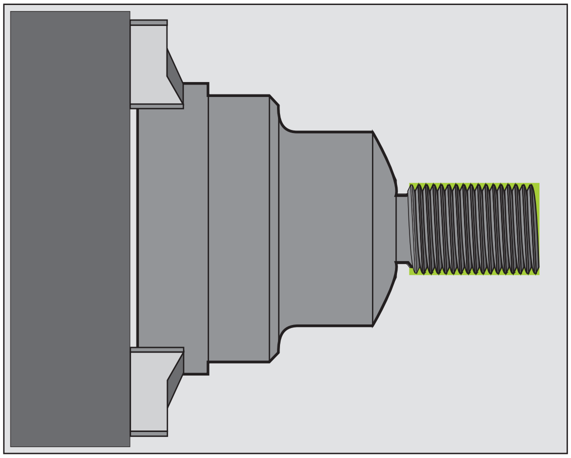


Positioning the tool for tool change

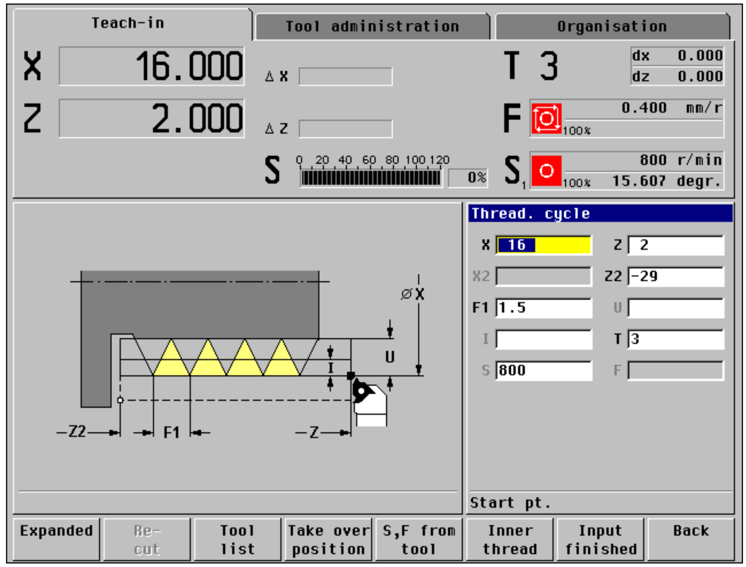
Before you can replace the finishing tool by the threading tool, you must move it to a "safe position."



Thread cycle

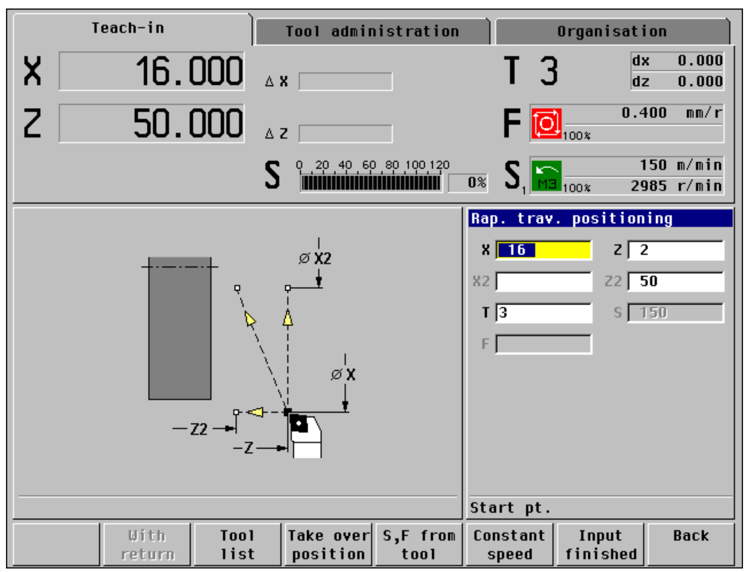


This cycle produces a single-start thread with a thread pitch of 1.5 mm. The depth of thread and the proportioning of cuts is calculated automatically by MANUALplus.



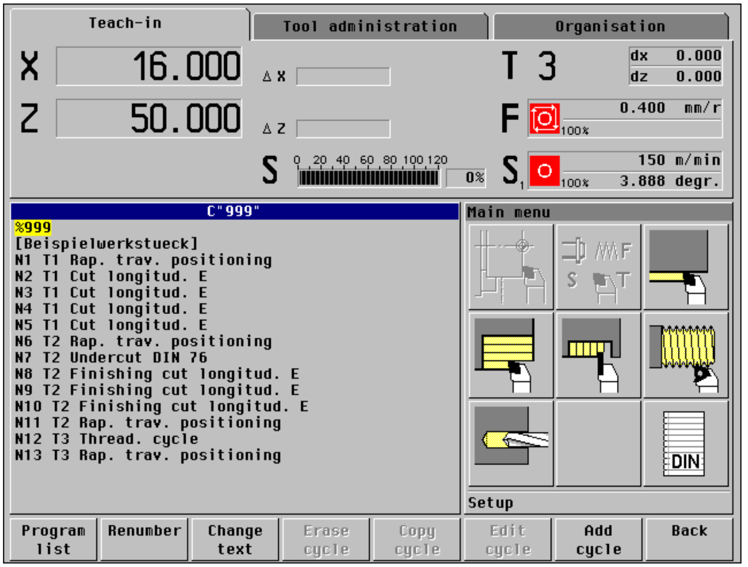
Tool positioning

The workpiece is completely machined. To remove the finished workpiece, you must move the tool to a "safe position."



Program list

The figure to the right shows the resulting cycle program.

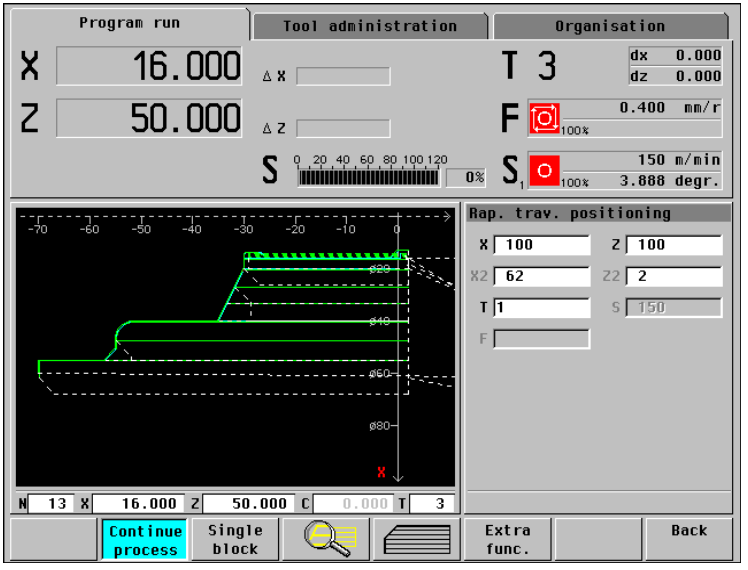


Simulation in program run

The program is simulated in the "Program run" mode.

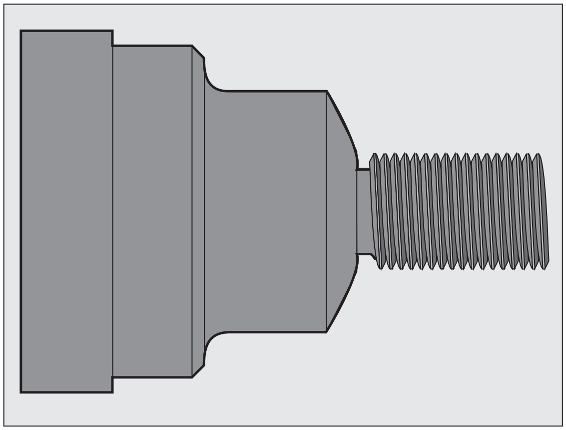
Press the "Menu" key to return to the main menu and select ***Program run.*** MANUALplus loads the program that was last machined. In this case, the cycle program "999" is loaded.

In the figure to the right, the complete machining operation for producing the workpiece was simulated in the Program run mode. In this example, the ***Continuous run*** function is active. MANUALplus therefore runs the simulation without interruption.

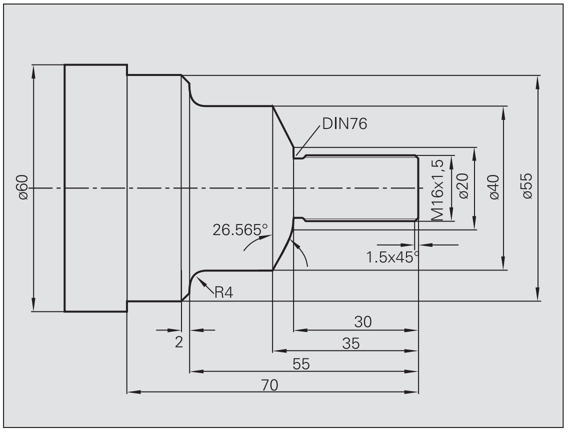


Finished workpiece

The figure to the right shows the resulting workpiece.



9.2 ICP Example "Threaded Stud"



This example illustrates how to machine a threaded stud using the ICP programming feature. The individual working steps for machining the ICP contour and integrating the contour into ICP cycles are based on the workpiece drawing.

The machining operation is performed with the "ICP cutting longitudinal" cycle. In the process described below, you create an ICP contour description and a cycle program for parts production.

Required tools

- **Roughing tool:**
  - Position T1
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.8 Tool radius
- **Finishing tool:**
  - Position T2
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.5 Tool radius
- **Threading tool:**
  - Position T3
  - WO = 1 Tool orientation

Sequence of working steps

- Clamp a workpiece blank (diameter 60 mm, length 100 mm).
- Machine setup
  – Define the workpiece zero point.
  – Measure the tool dimensions.
- Switch to "Teach-in" mode.
- Enter the positioning cycles for tool change.
- ICP contour programming.
- Integrate the ICP contour in the roughing and finishing cycles.
- Machine the thread.

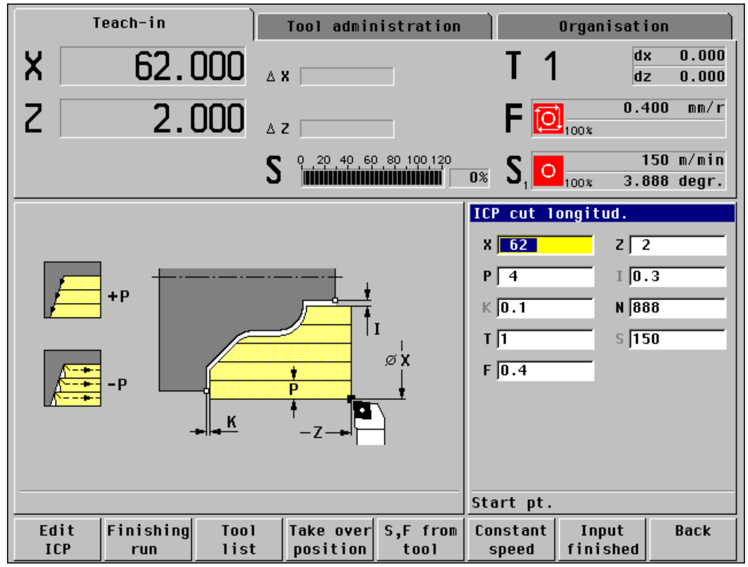
ICP cutting longitudinal

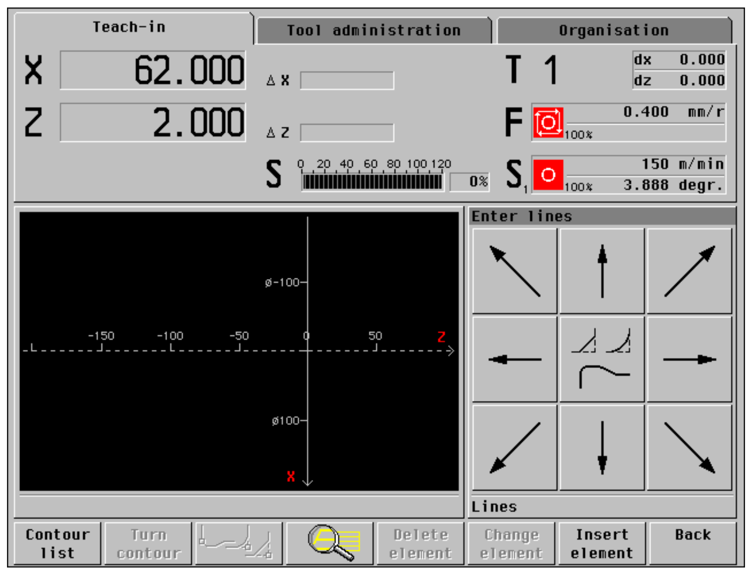
The procedure presupposes that the machine has been set up and the control is in "Teach-in" mode.

The infeed depth and the allowances for roughing are programmed in the ICP cutting cycle. In this example, the number ("888") of the ICP contour is entered **before** calling the ICP editor (see figure to the top right).

You then switch to the ICP editor and press ***Insert element*** to enter the contour elements.

> Since MANUALplus determines the cutting direction from the contour direction, the ICP contour must be described in the negative Z direction.





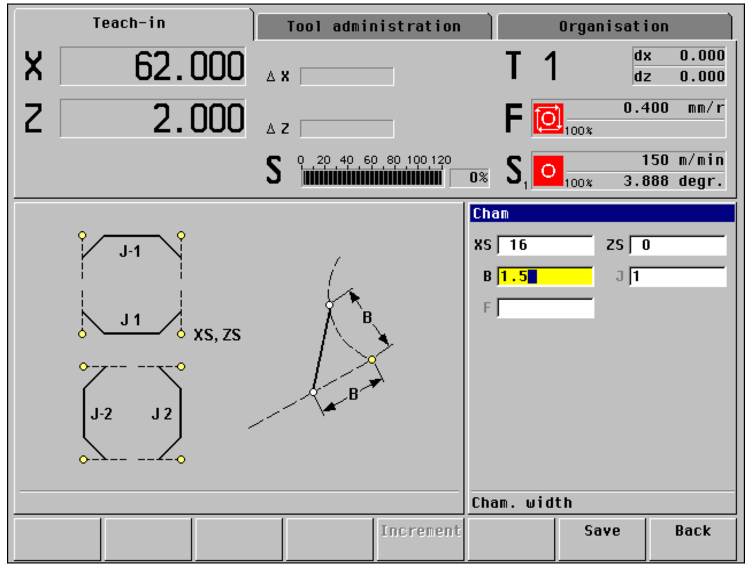
Contour element 1

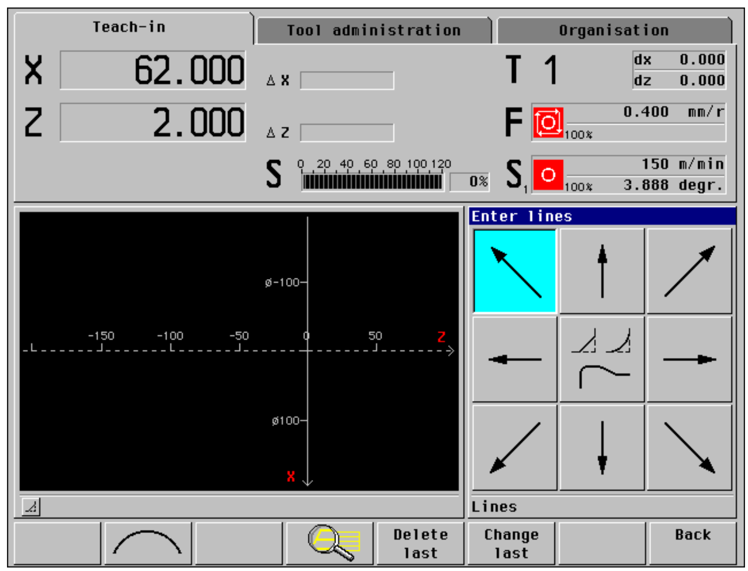
The contour starts with a chamfer (thread chamfer).

The starting point of the contour is defined in "XS, ZS" when programming the first contour element. The starting point, in this case, is the corner that is cut off by the chamfer.

If you program a chamfer as the first contour element, you must specify the position of the chamfer with the parameter "element position J"—here: "J=1" (see figure to the upper right).

The control does not know yet the next connecting contour element. The chamfer is therefore regarded as an "unsolved element." MANUALplus displays the appropriate symbol below the graphics window (see figure to the bottom right).



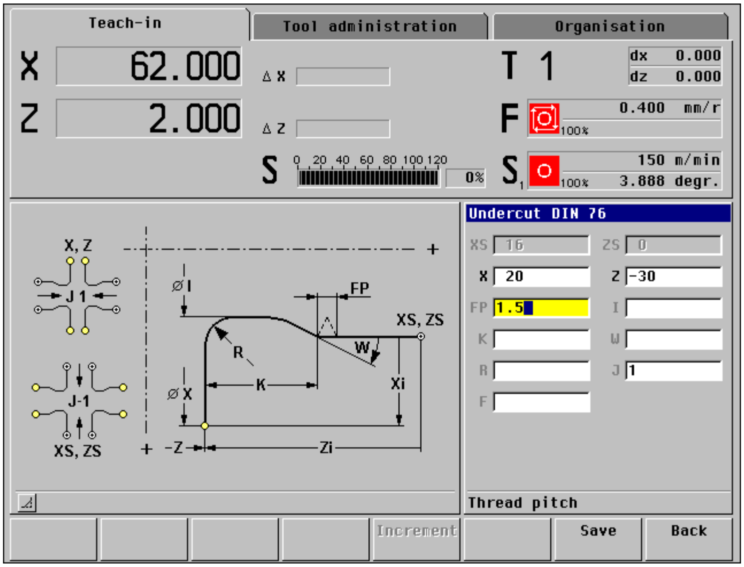


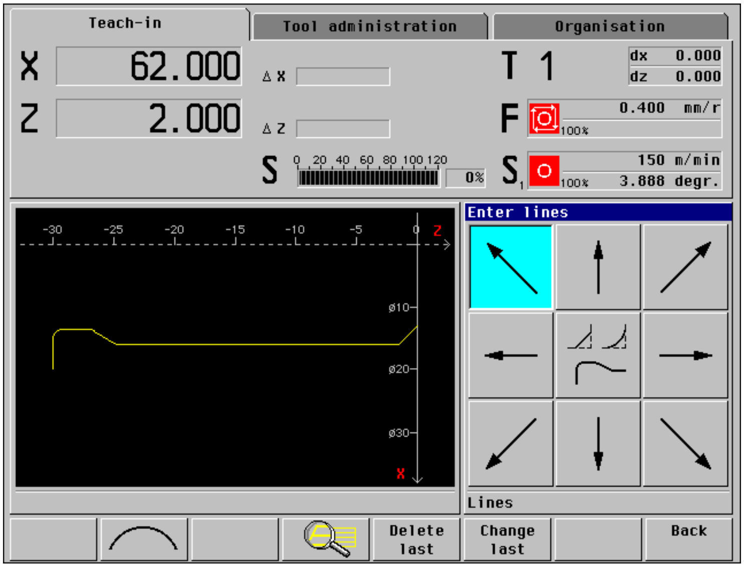
Contour element 2

The next connecting contour element is an undercut. The form element "undercut" describes the preceding cylinder, the actual undercut and the subsequent plane surface.

The part of the contour you have entered up to now is unambiguously defined. MANUALplus draws the contour elements and clears the symbol for the "unsolved chamfer element."

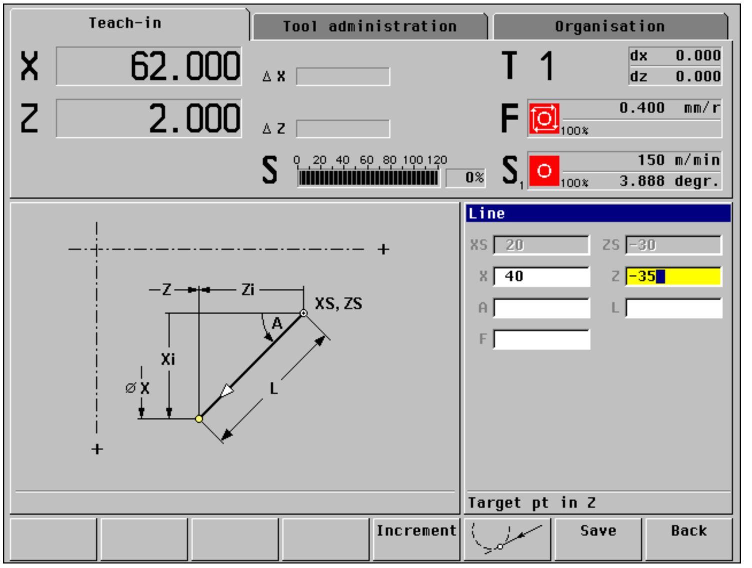
To define the undercut, the thread pitch is programmed in addition to the "target point." MANUALplus automatically determines all other undercut parameters from the standard tables.

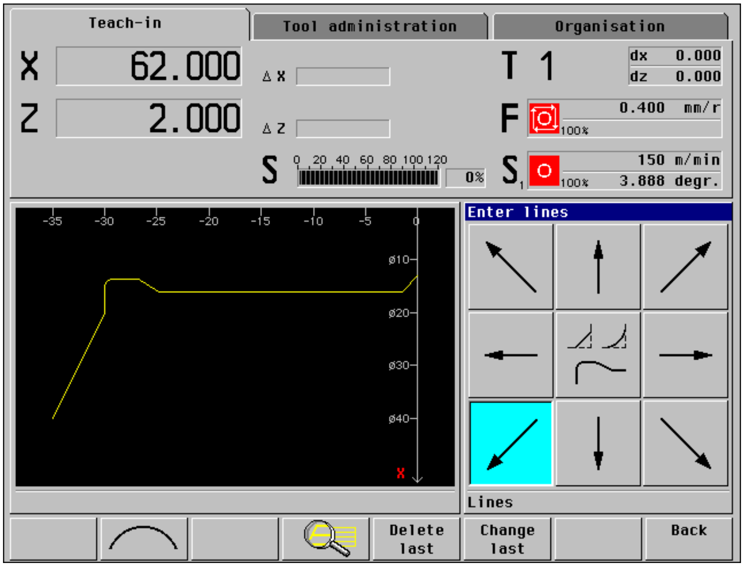




Contour element 3

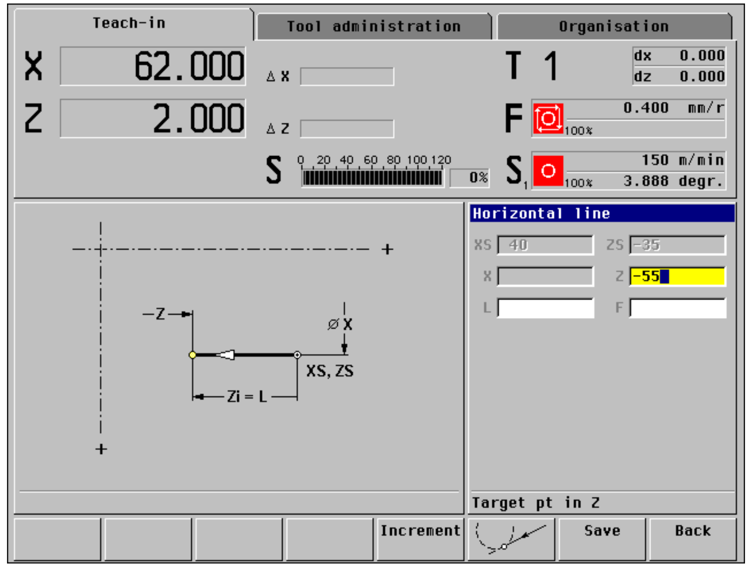
The next connecting contour element is an oblique cut. After you have entered the "target point X, Z," the line is unambiguously defined. MANUALplus draws the contour elements in the graphics window.

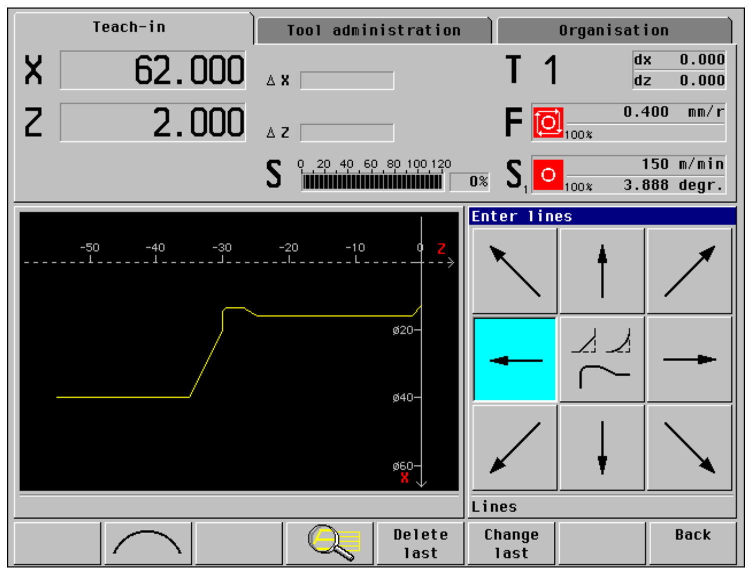




Contour element 4

The next connecting contour element is a horizontal line. After you have entered the "target point Z," the line is unambiguously defined. MANUALplus draws the contour elements in the graphics window.

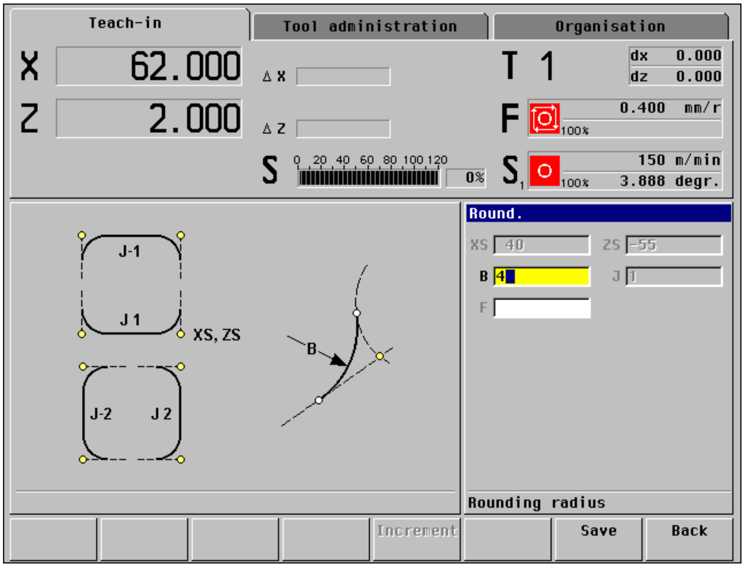


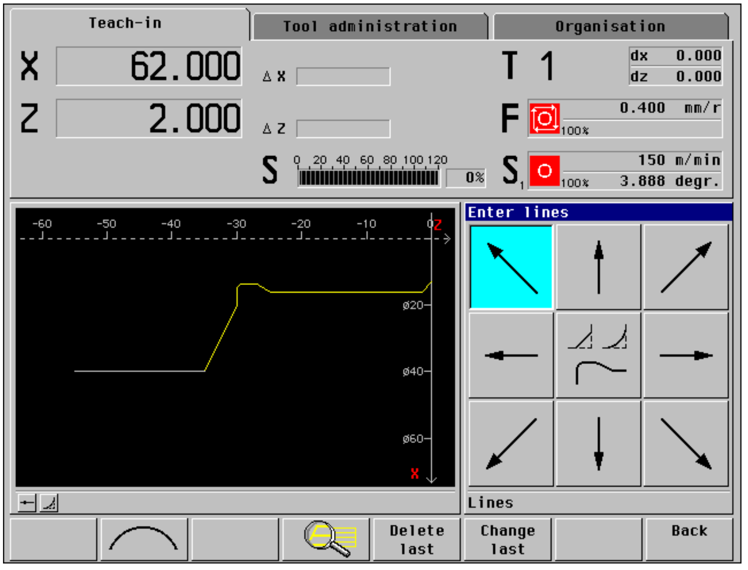


Contour element 5

The next connecting contour element is a rounding. You only need to enter "rounding radius B."

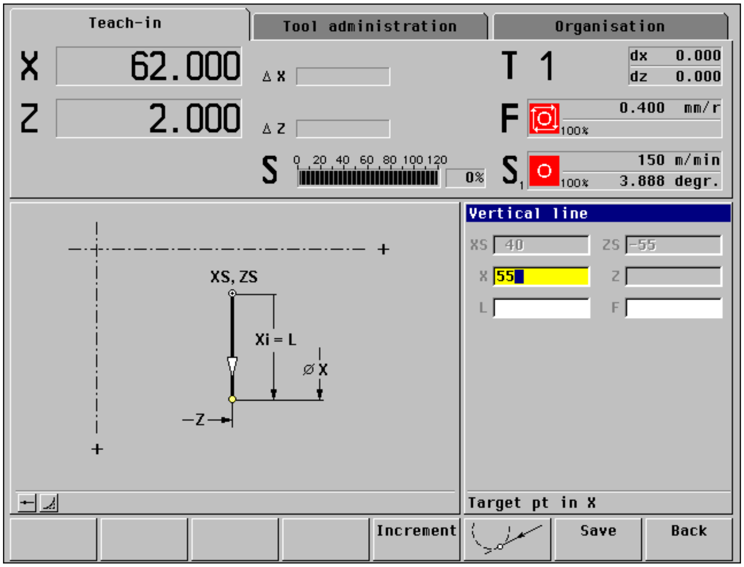
When the rounding is programmed, the control does not yet know the next connecting contour element. The rounding and the preceding linear element are therefore considered "unsolved elements." MANUALplus displays the symbols for these elements below the graphics window and depicts the preceding horizontal line in gray.

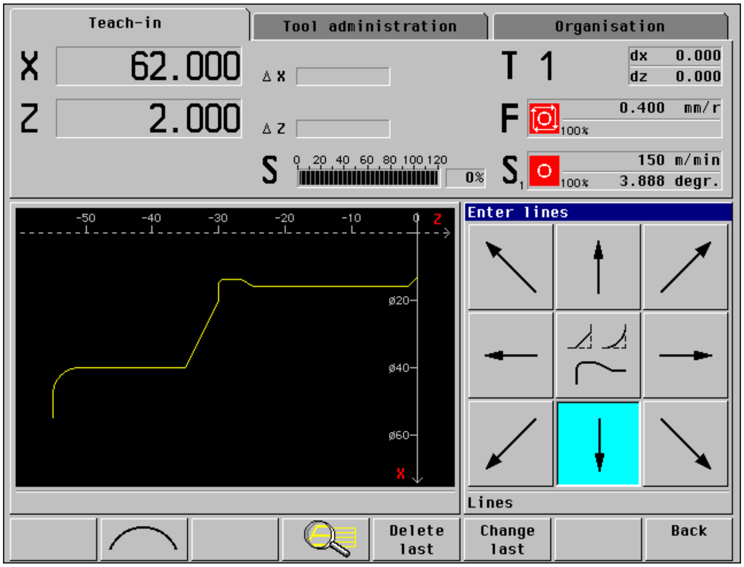




Contour element 6

The next connecting contour element is a vertical line. After you have entered the "target point X," the line and the preceding rounding are unambiguously defined. MANUALplus draws the contour elements and clears the symbols for the "unsolved elements."

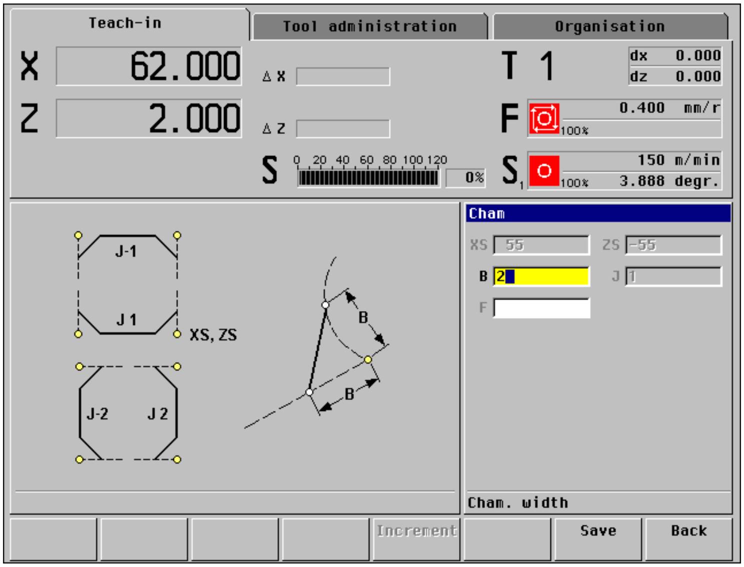


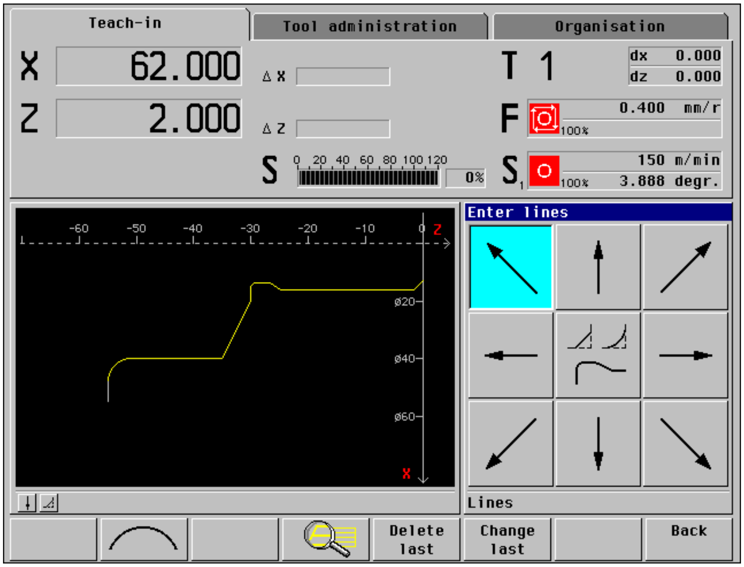


Contour element 7

The next connecting contour element is a chamfer. The only parameter that needs to be defined is the "chamfer width B."

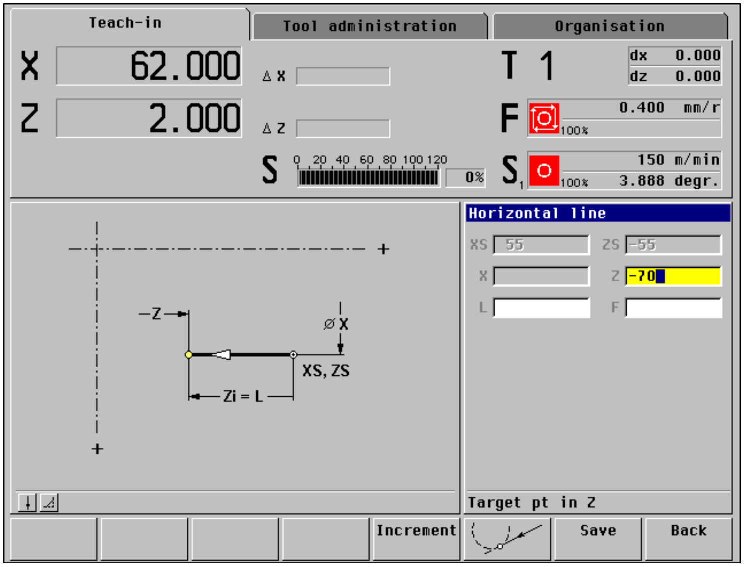
When you program the chamfer, the control does not yet know the subsequent contour element which connects to the chamfer. The chamfer and the preceding linear element are therefore considered "unsolved elements." MANUALplus displays the symbols for these elements below the graphics window and depicts the preceding horizontal line in gray (color for unsolved elements).

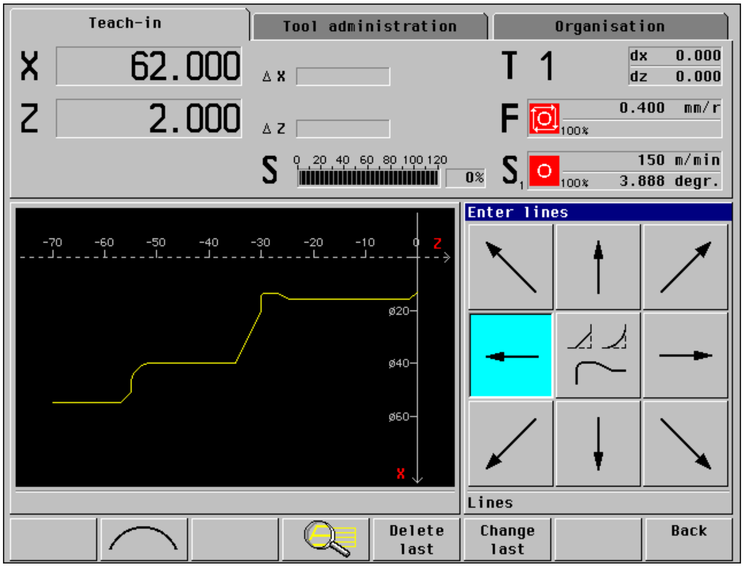




Contour element 8

The next connecting contour element is a horizontal line. After you have entered the "target point Z," the line and the preceding chamfer are unambiguously defined. MANUALplus draws the contour elements and clears the symbols for the "unsolved elements."

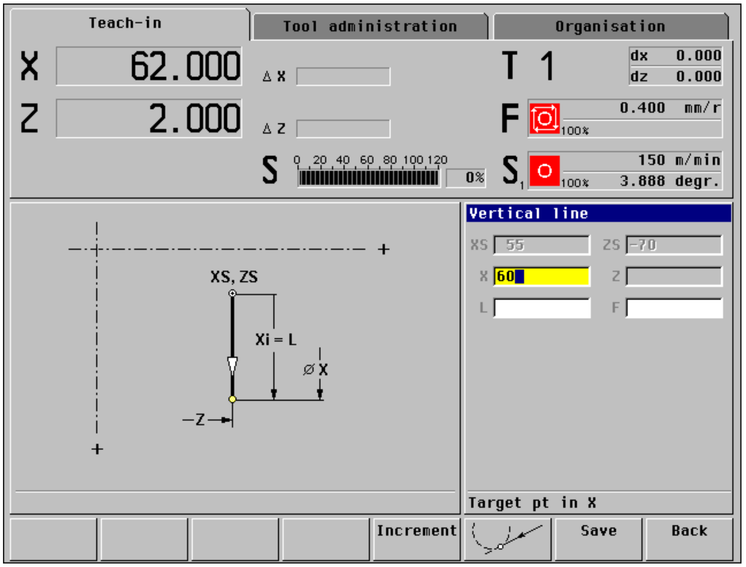


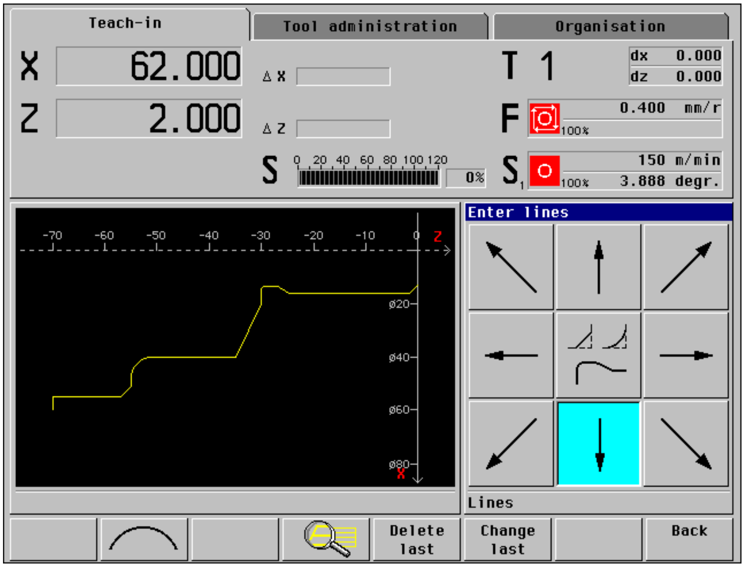


Contour element 9

The next connecting contour element is a vertical line. After you have entered the "target point X," the line is unambiguously defined. MANUALplus draws the contour elements in the graphics window.

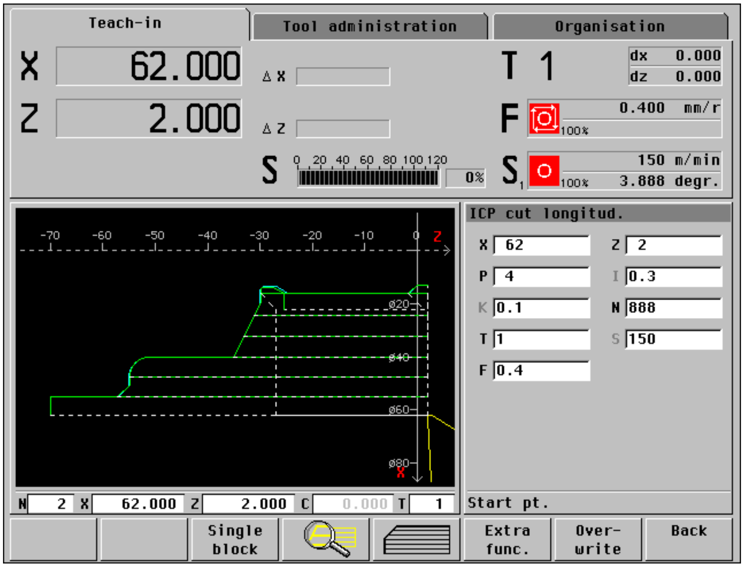
The ICP contour has been completely defined. ***Back*** concludes ICP programming and ***Input finished*** concludes the ICP cycle.





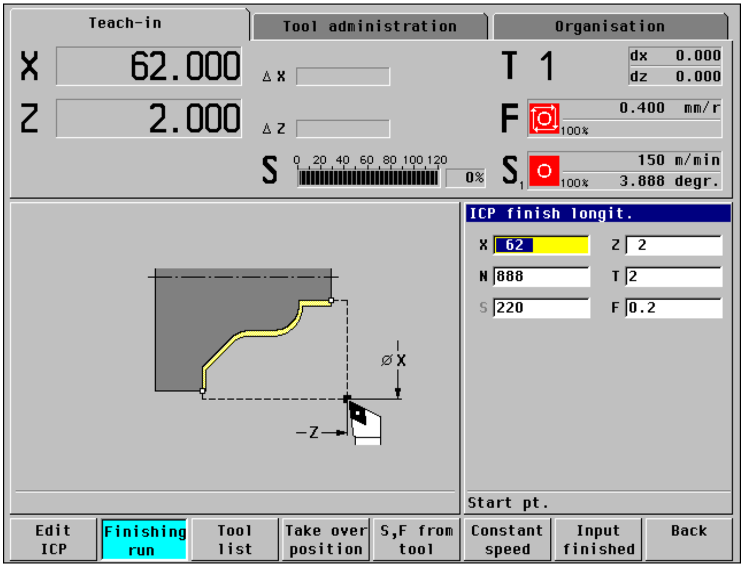
Checking the ICP cutting cycle

With the graphic simulation function, you can check the execution of the cycle (***Graphics*** soft key). You can then transfer the cycle to the cycle program with the ***Save*** or ***Overwrite.***



ICP finishing

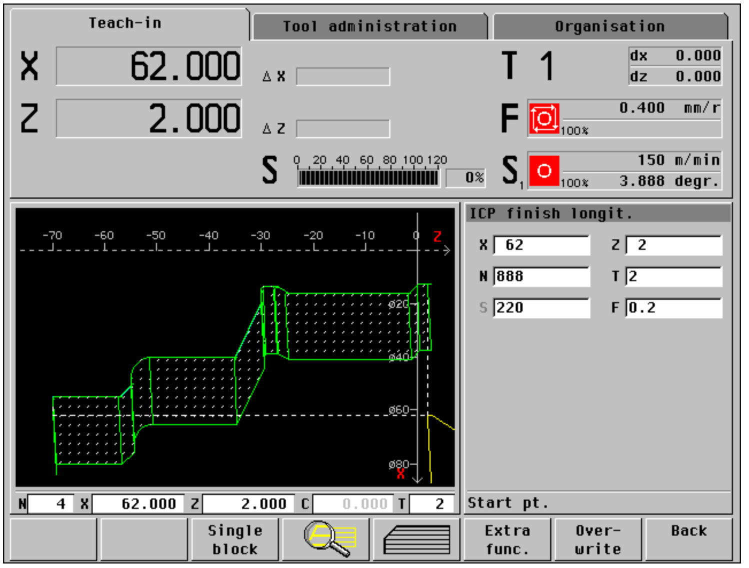
The ICP contour "888" (threaded stud) is also used for the finishing cycle.



Checking the ICP finishing cycle

With the graphic simulation function, you can check the execution of the ICP finishing cycle (***Graphics*** soft key). You can then transfer the cycle to the cycle program with the ***Save*** or ***Overwrite.***

MANUALplus finishes the contour in the defined "contour direction" (see figure to the top right).

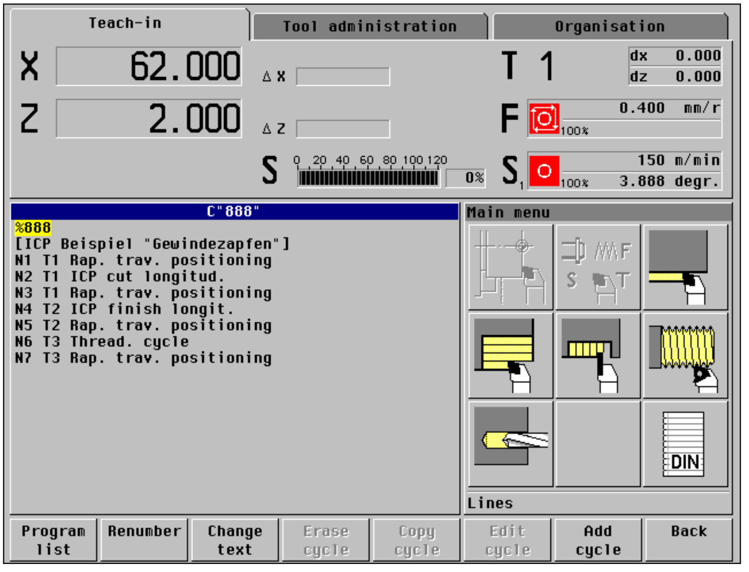


Cycle program "ICP example workpiece"

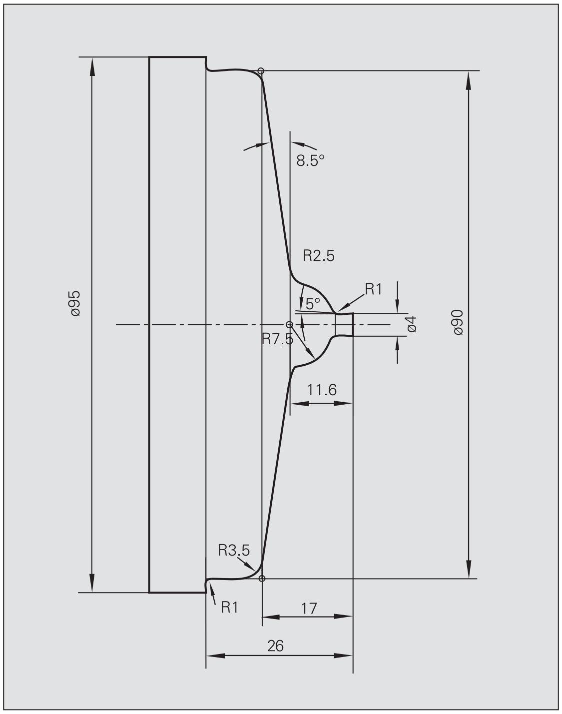
Besides the ICP cycles, the created cycle program also includes the positioning cycles for tool change and the thread cycle (see figure to the lower right).

Functions of the cycles:

- N1: Remove the material (roughing).
- N2: Position the tool for tool change.
- N3: Finish-machine the workpiece.
- N4: Position the tool for tool change.
- N5: Machine the thread.
- N6: Position the tool for removing the workpiece.



9.3 ICP Example "Matrix"



This example illustrates how to machine a matrix using the ICP programming feature. The individual working steps for machining the ICP contour and integrating the contour into ICP cycles are based on the workpiece drawing.

In the process described below, you create an ICP contour description and a cycle program for parts production.

The machining operation is performed with the "ICP cutting transverse" cycle.

Required tools

- **Roughing tool:**
  - Position T1
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.8 Tool radius
- **Finishing tool:**
  - Position T2
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.5 Tool radius

Sequence of working steps

- Clamp a workpiece blank (diameter 95 mm, length 100 mm).
- Machine setup
  – Define the workpiece zero point.
  – Measure the tool dimensions.
- Switch to "Teach-in" mode.
- Enter the positioning cycles for tool change.
- ICP contour programming.
- Integrate the ICP contour in the roughing and finishing cycles.

ICP cutting transverse

The procedure presupposes that the machine has been set up and the control is in "Teach-in" mode.

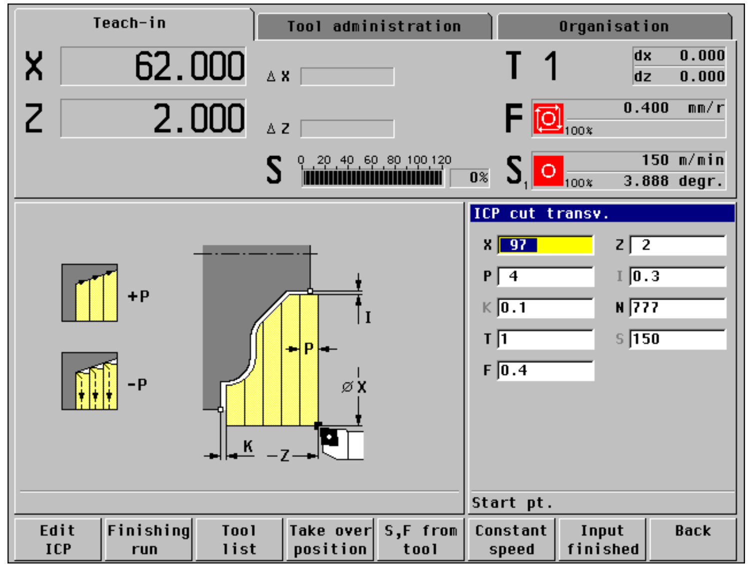
The infeed depth and the allowances for roughing are programmed in the ICP cutting cycle. The number of the ICP contour is entered **before** calling the ICP editor (see figure to the top right).

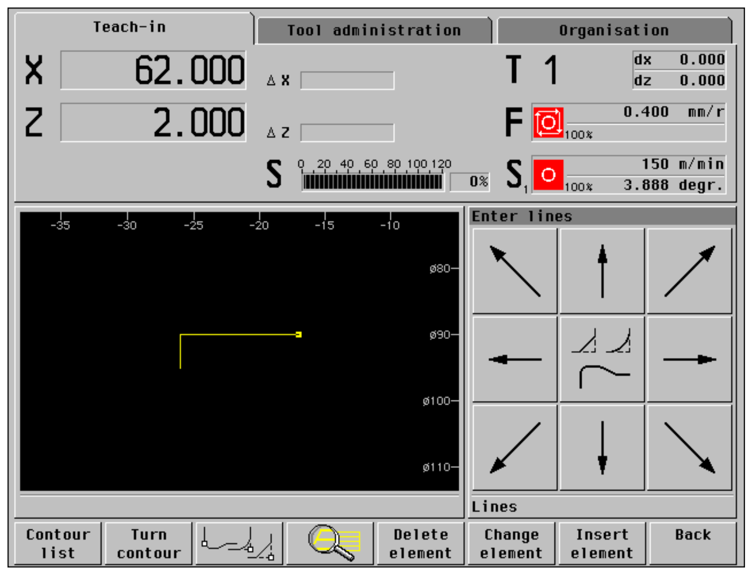
Switch to the ICP programming function with ***Edit ICP.*** In this example, the first two contour elements of the contour "ICP example Matrix" (ICP contour number 777) have already been entered.

First you enter the "rough contour." Then you use the "superimposition" function to define the roundings.

You can now switch to the input mode of the ICP editor by pressing ***Insert element*** (see figure at lower right).

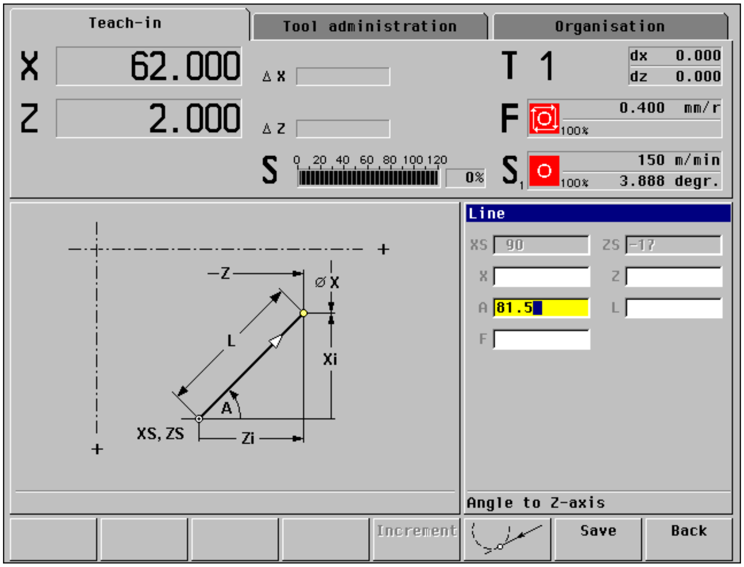
Since MANUALplus determines the cutting direction from the contour direction, the ICP contour must be described in the negative Z direction.

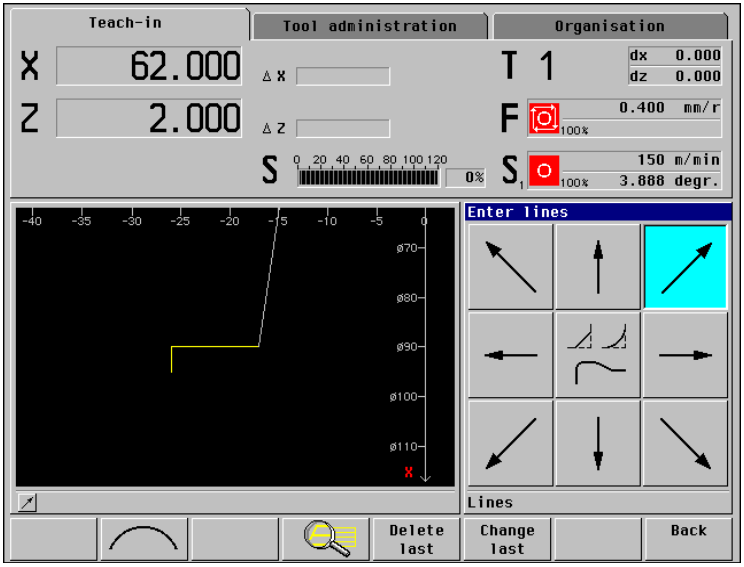




Contour element 3

The next connecting contour element is an oblique cut. Only the angle of the linear element is known. MANUALplus displays the symbol for an "unsolved element" below the graphics window and depicts the unsolved line in gray (color for unsolved elements).

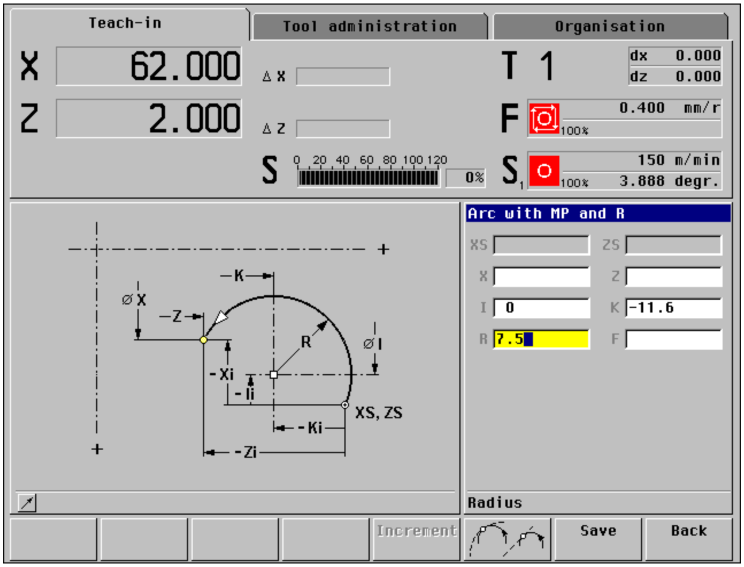


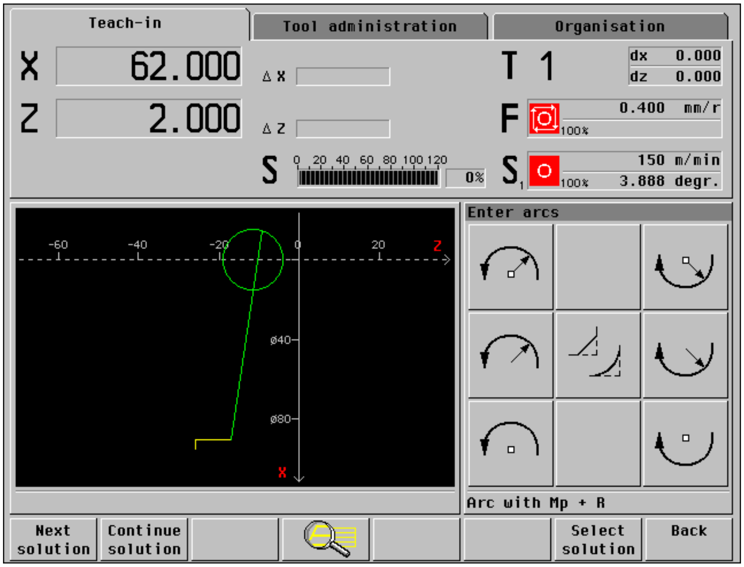


Contour element 4

The next connecting contour element is a circular arc whose center and radius are known.

MANUALplus displays the possible solutions for selection (see figure to the bottom right and, on the next page, to the top right).

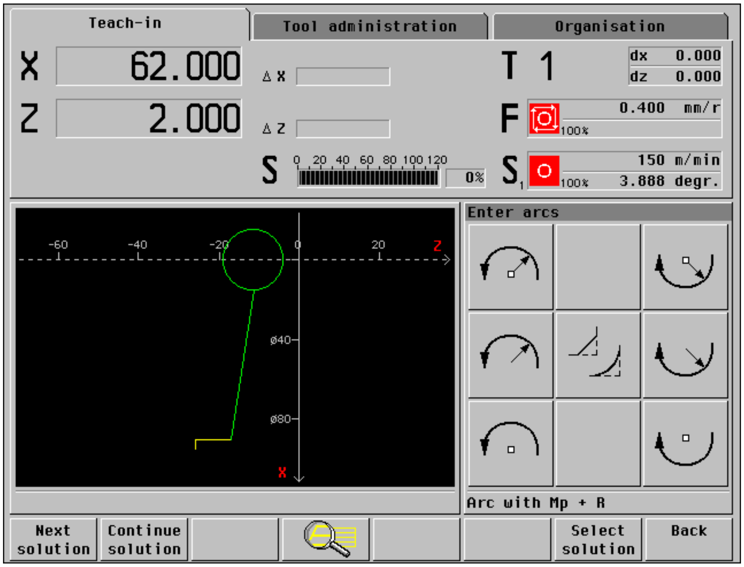


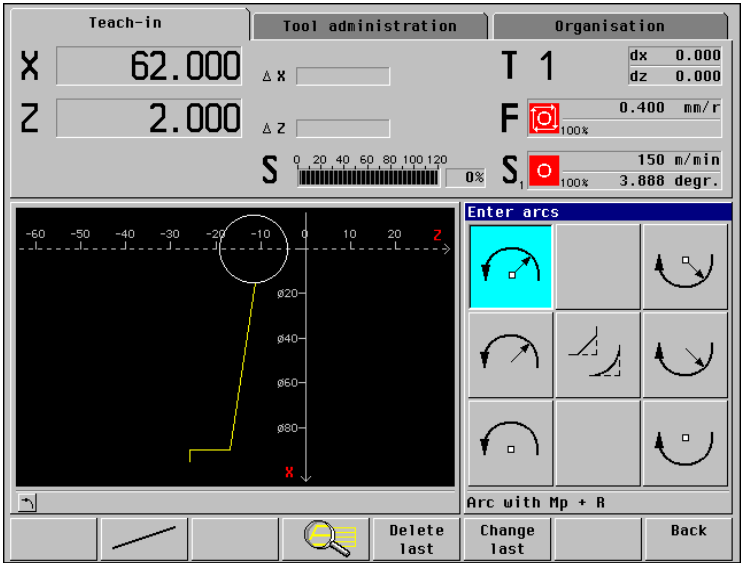


To transfer the correct solution, press ***Select solution.***

The preceding oblique cut is now unambiguously defined. The circular arc still permits several solutions.

MANUALplus displays the symbol for an "unsolved element" below the graphics window and depicts the unsolved line in gray (color for unsolved elements).

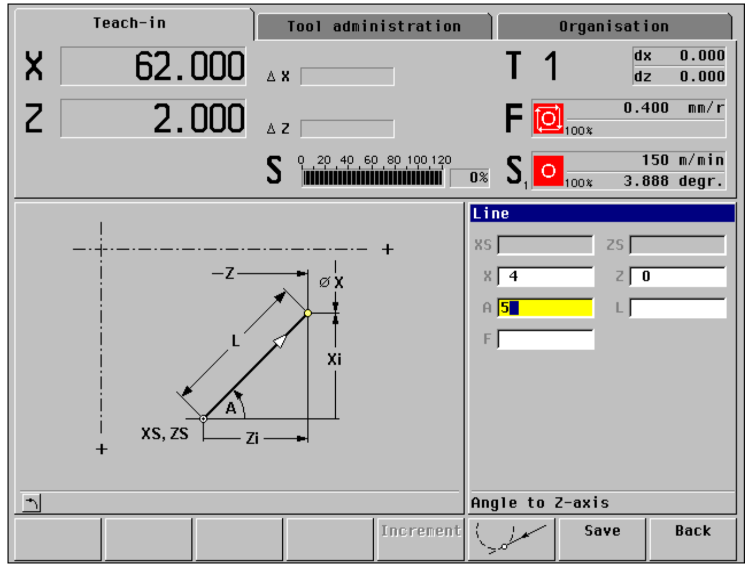


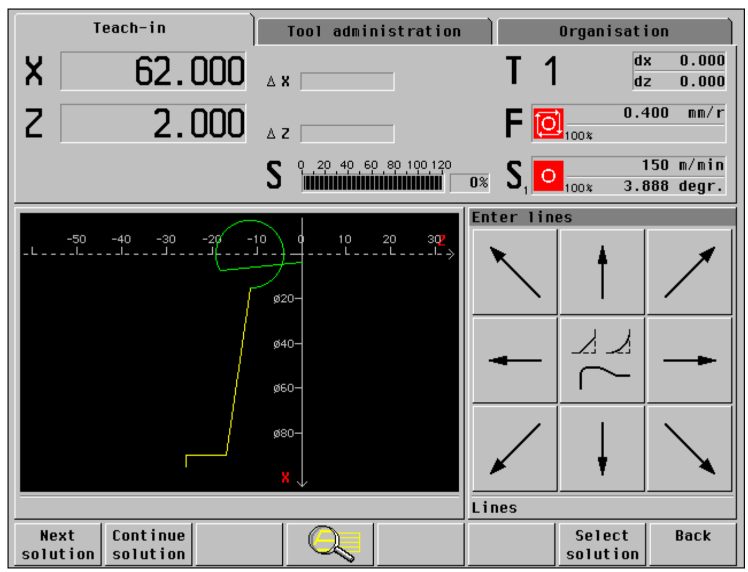


Contour element 5

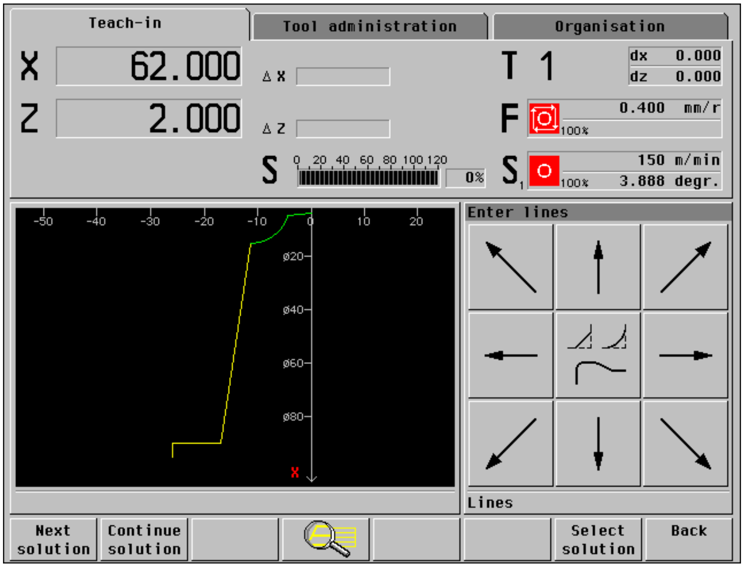
The next connecting contour element is an oblique cut. After you have entered the "target point X, Z" and the "angle A," the line is unambiguously defined.

MANUALplus displays the possible solutions for selection (see figure to the bottom right and, on the next page, to the top right).



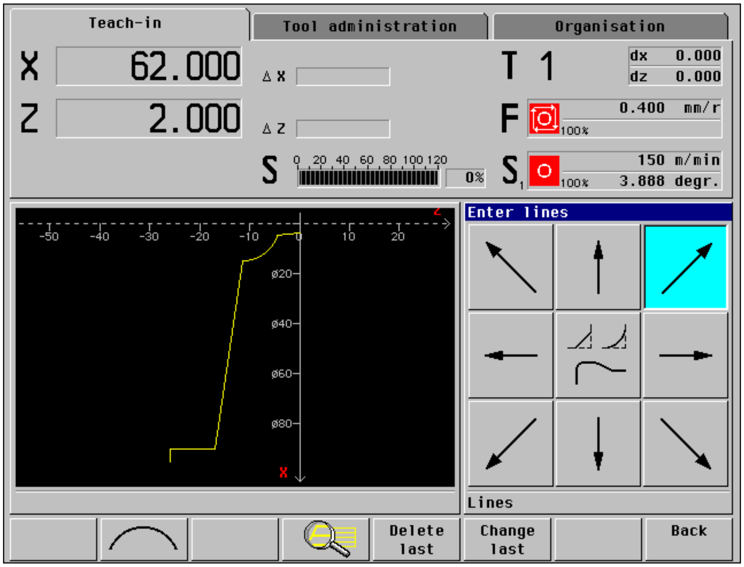


To transfer the correct solution, press ***Select solution.***



The preceding circular arc and the oblique cut are now unambiguously defined. MANUALplus draws the contour elements and clears the symbols for the "unsolved elements."

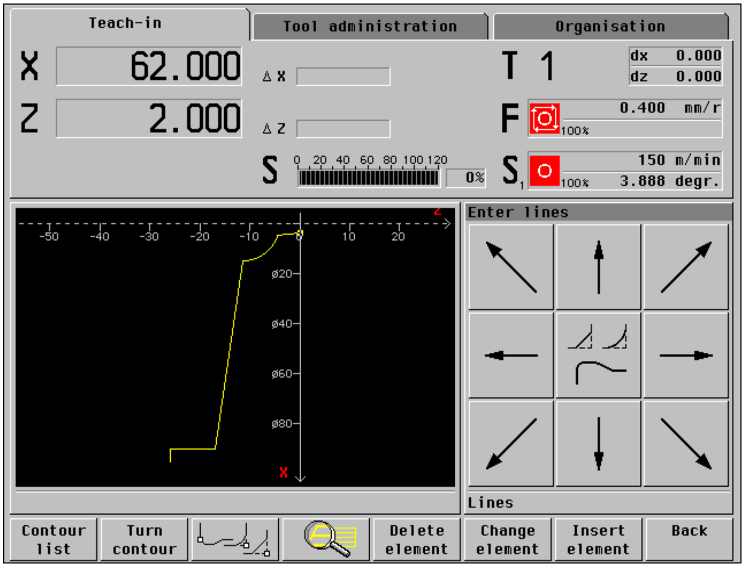
The "rough contour" has been completely defined. You can now exit the input mode with ***Back.***

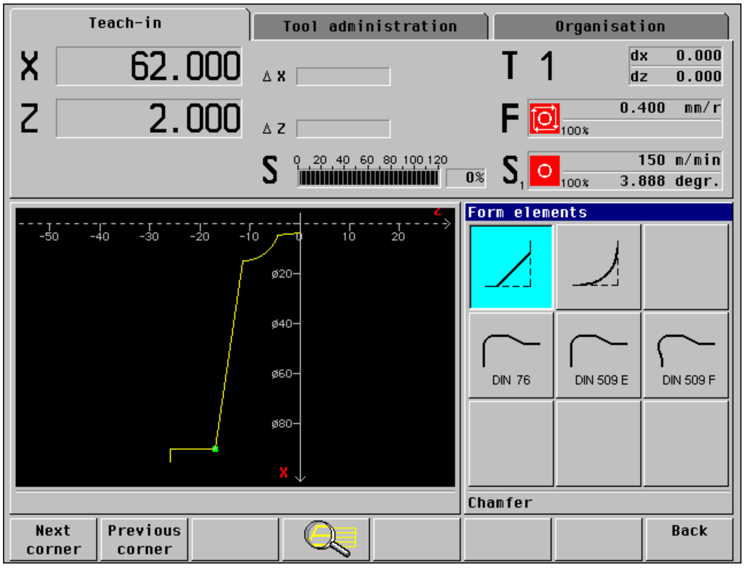


Rounding the corners

The rounding arcs are "superimposed" on the existing contour. This is done by selecting the individual contour corners and defining the corresponding rounding radii.

You call the function for superimposing elements with the ***superposition*** soft key (represented by a symbol—see figure at upper right). You can then select the position of the rounding with ***Previous corner*** / ***Next corner*** (see figure at lower right).

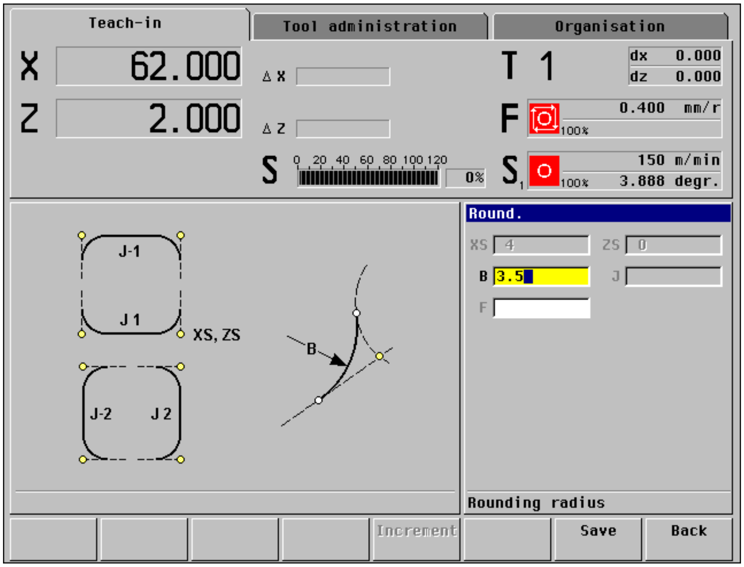


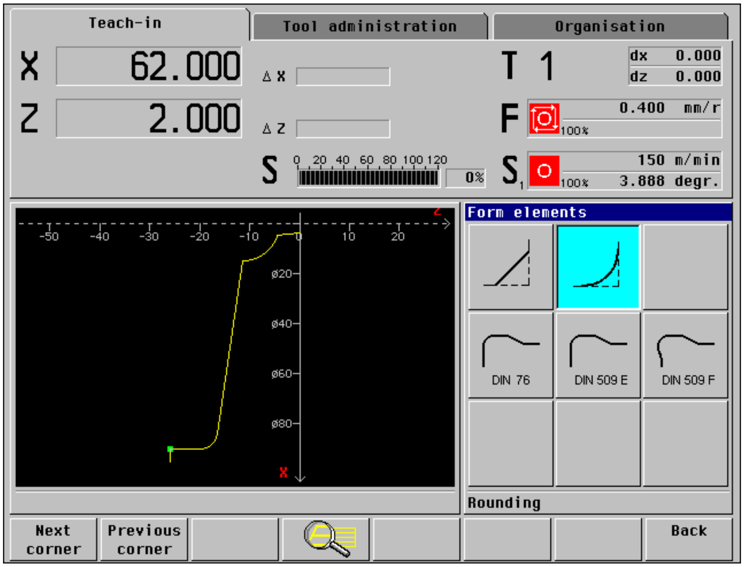


Defining a rounding

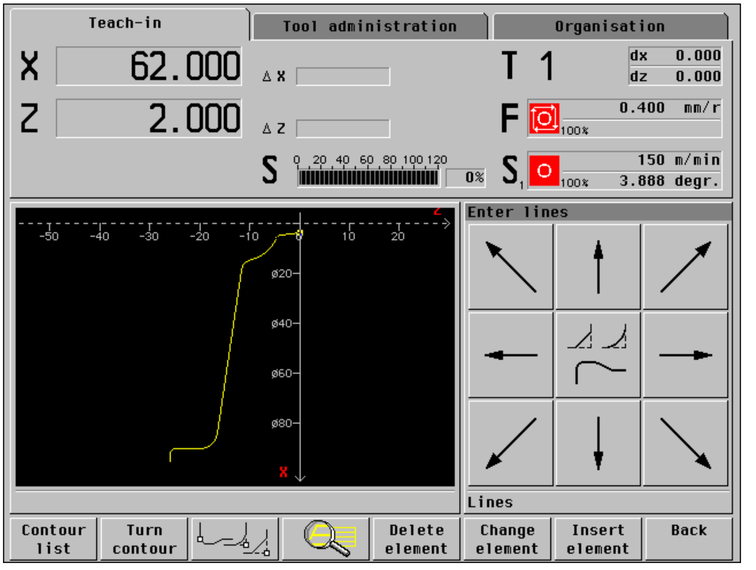
To define the rounding, enter "Rounding radius B." MANUALplus inserts the rounding in the existing ICP contour and draws the "perfected" contour.

If the contour contains further corners, MANUALplus offers the next contour corner for selection (see figure to the bottom right).





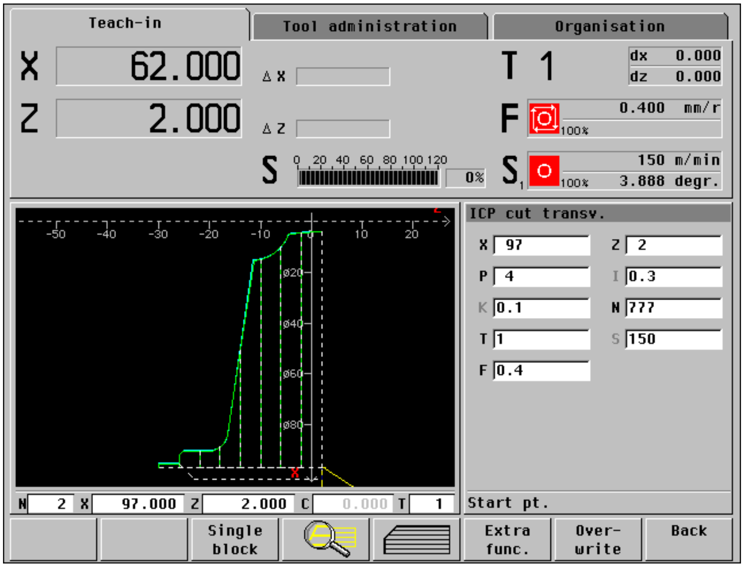
The ICP contour has been completely defined. ***Back*** concludes ICP programming and ***Input finished*** concludes the ICP cycle.



Checking the ICP cutting cycle

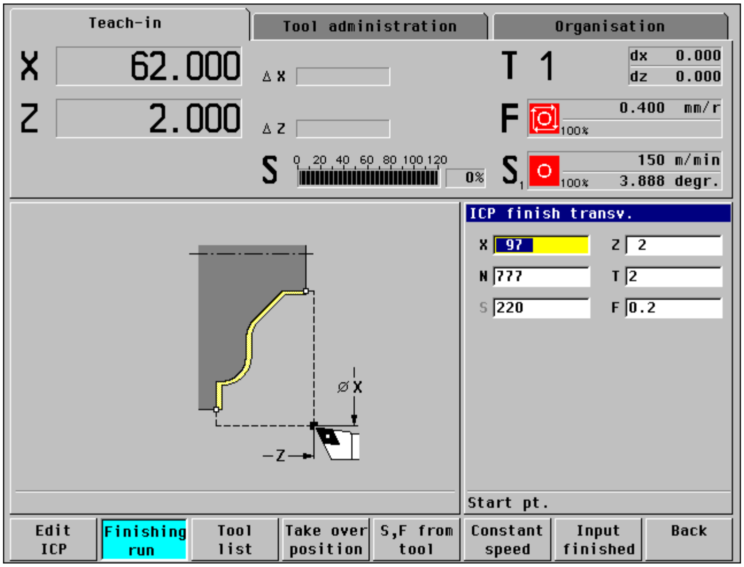
After the cutting operation has been completed, it is checked with the Simulation function. The simulation function is called with the ***Graphics*** soft key.

You can then transfer the cycle to the cycle program with the ***Save*** or ***Overwrite.***



ICP finishing

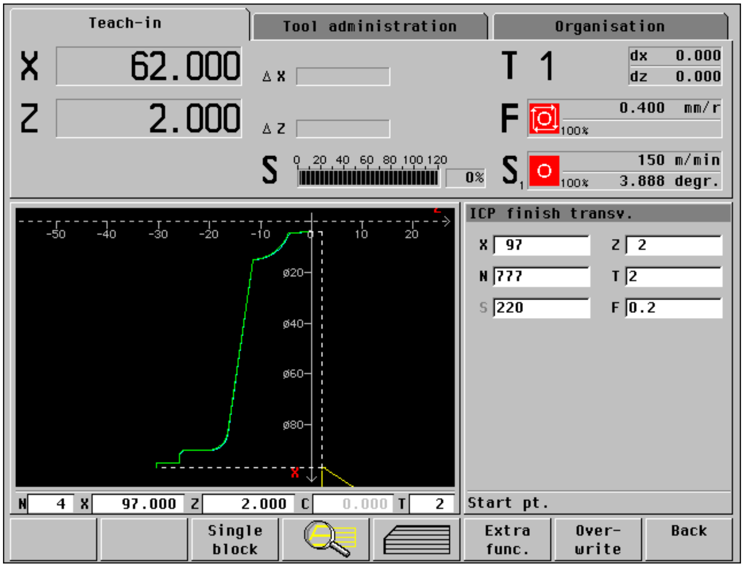
The ICP contour "777" ("Matrix") is also used for the finishing cycle.



Checking the ICP finishing cycle

With the graphic simulation function, you can check the execution of the ICP finishing cycle (***Graphics*** soft key). You can then transfer the cycle to the cycle program with the ***Save*** or ***Overwrite.***

MANUALplus finishes the contour in the defined "contour direction" (see figure to the bottom right).

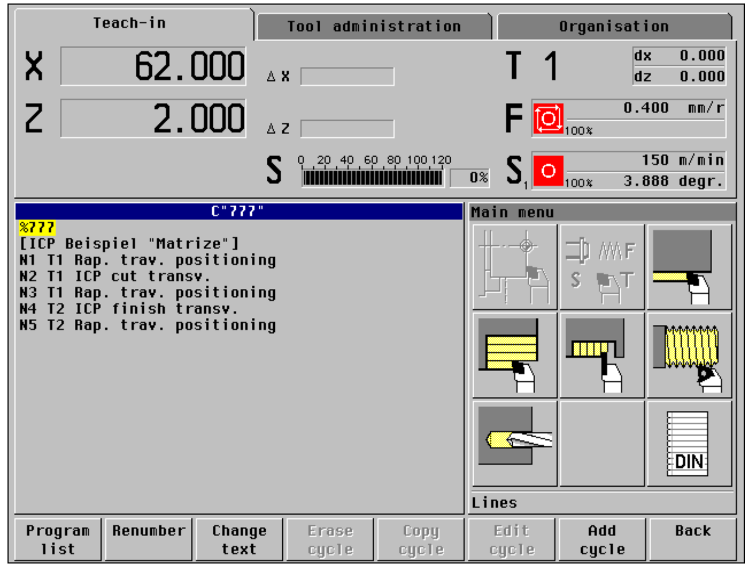


Cycle program "ICP example Matrix"

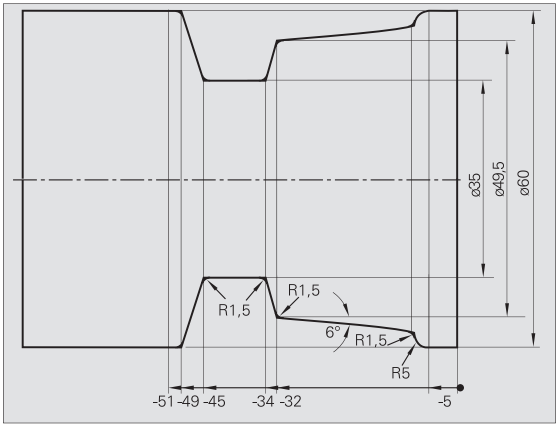
Besides the ICP cycles, the created cycle program also includes the positioning cycles for tool change (see figure to the right).

Functions of the cycles:

- N1: Remove the material (roughing)
- N2: Position the tool for tool change
- N3: Finish-machine the workpiece
- N4: Position the tool for removing the workpiece



9.4 ICP Example "Recessing Cycle"



This example illustrates the use of an ICP recessing cycle. The individual working steps for machining the ICP contour and integrating the contour into ICP cycles are based on the workpiece drawing.

In the process described below, you create an ICP contour description and a cycle program for parts production.

The machining operation is performed with the "ICP recessing radial" cycle.

Required tool

- **Recessing tool**:
  - Position T4
  - WO = 1 Tool orientation
  - R = 0.2 Tool radius
  - K = 5 Cutting width

Sequence of working steps

- Clamp a workpiece blank (diameter 60 mm, length 65 mm).
- Machine setup
  – Define the workpiece zero point
  – Measure the tool dimensions
- Switch to "Teach-in" mode.
- Enter the positioning cycles for tool change.
- Call the "ICP cut radial" cycle.
- ICP contour programming.
- Integrate the ICP contour in the recessing, finishing cycle.

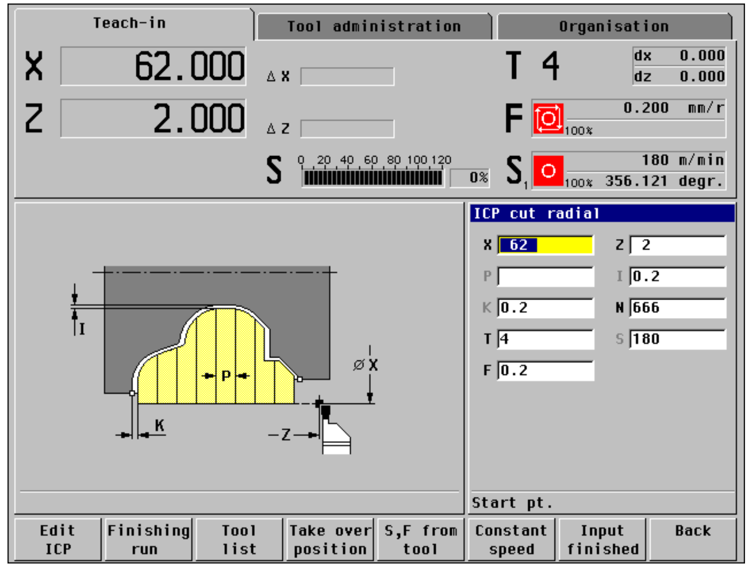
ICP recessing radial

The procedure presupposes that the machine has been set up and the control is in "Teach-in" mode.

The allowances for pre-cutting are programmed in the ICP recessing cycle. The cutting width is not entered. MANUALplus automatically calculates the proportioning of cuts such that the infeed per pass is < 80% of the cutting width defined in the tool data (see figure to the top right).

After defining the cycle parameters, press ***Edit ICP*** to call the ICP programming function. You can now switch to the input mode by pressing ***Add element.***

First you enter the "rough contour." Then you use the "superimposition" function to define the roundings.

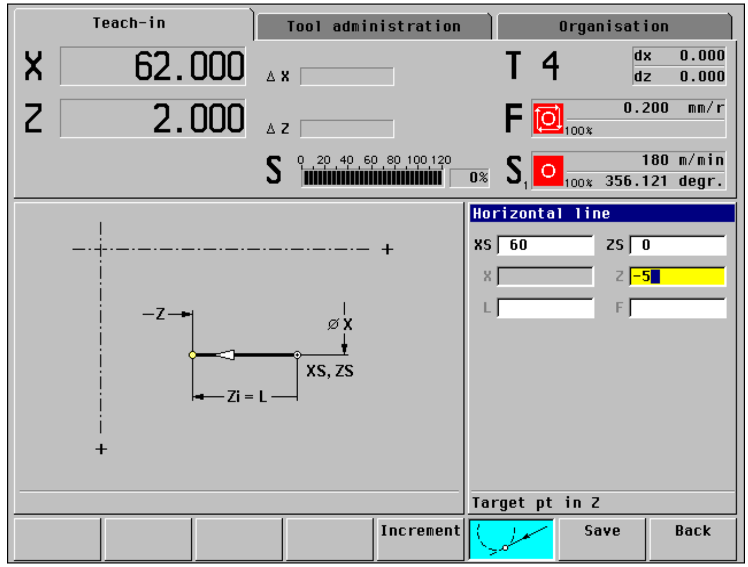


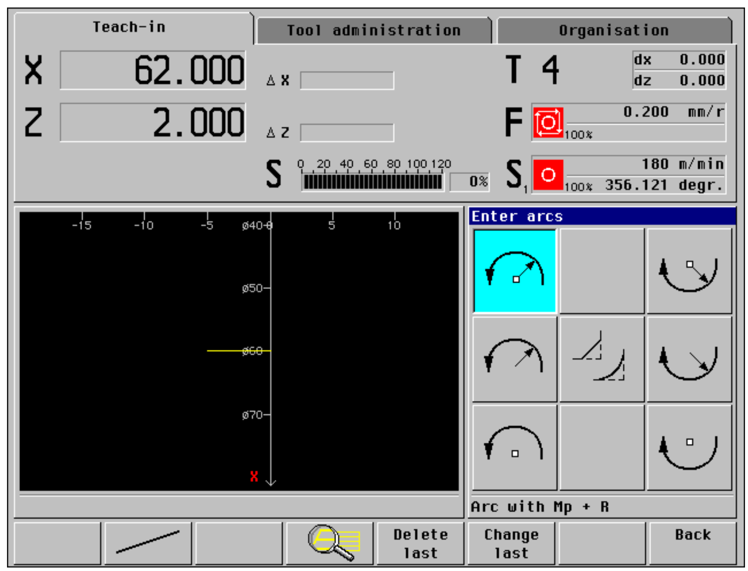
Contour element 1

The contour starts with a horizontal line which is connected "tangentially" to the subsequent circular arc.

The starting point of the ICP contour is defined in "XS, ZS" when programming the first contour element.

After you have entered the "target point Z," the line is unambiguously defined. MANUALplus draws the contour element in the graphics window.

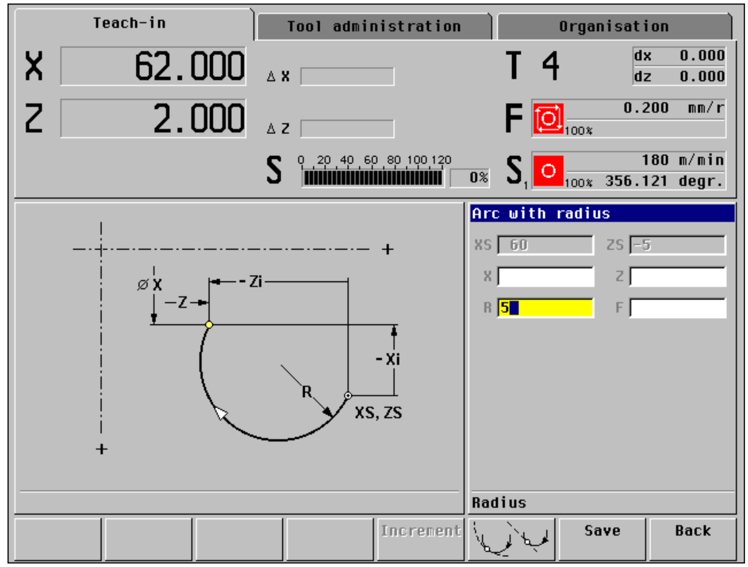


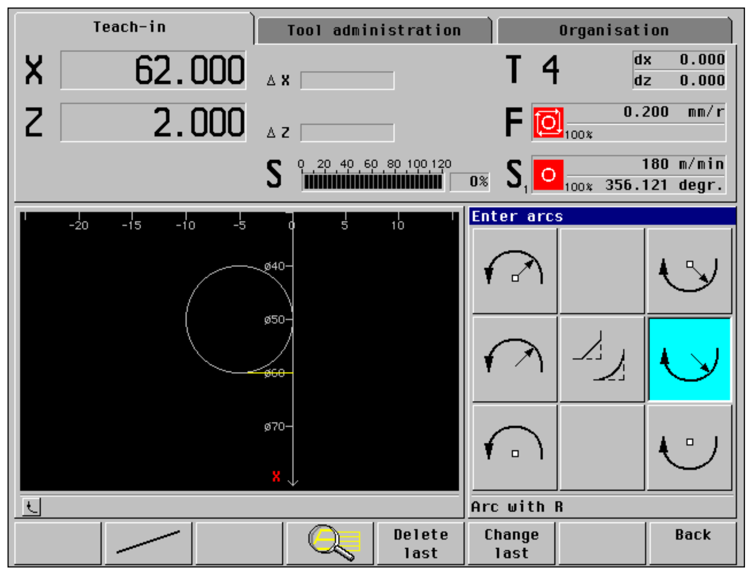


Contour element 2

The next connecting contour element is a circular arc. Only its radius is known. The circular arc still permits several solutions.

MANUALplus displays the corresponding symbol below the graphics window and depicts the arc in gray, which is the color used for identifying unsolved elements.



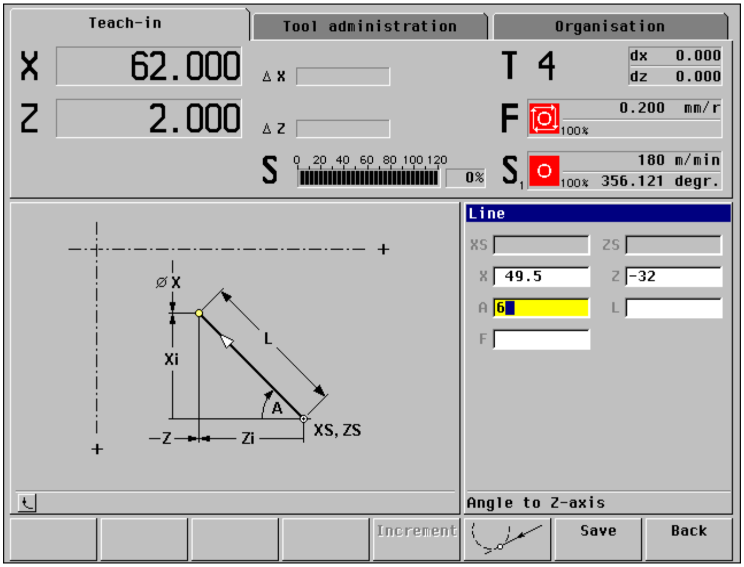


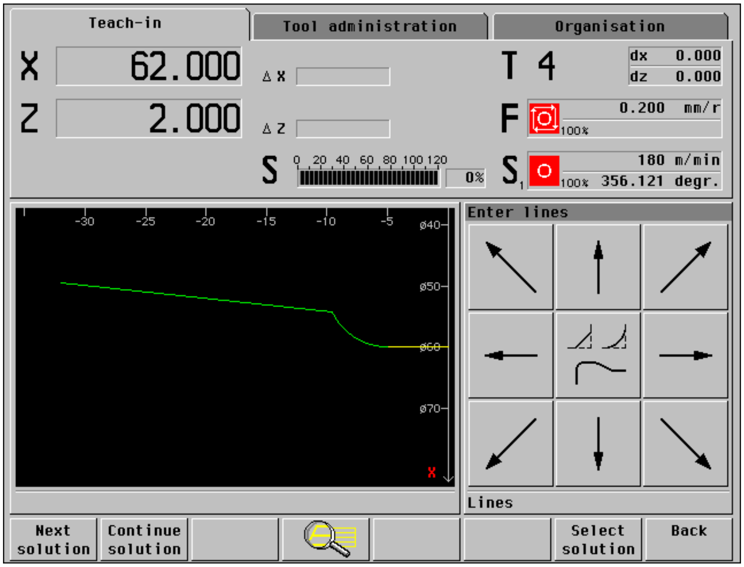
Contour element 3

The next connecting contour element is an oblique cut whose target point and angle are known.

MANUALplus displays the "selection of possible solutions." To transfer the correct solution, press ***Select solution*** (see figure to the lower right).

The preceding circular arc and the oblique cut are now unambiguously defined.

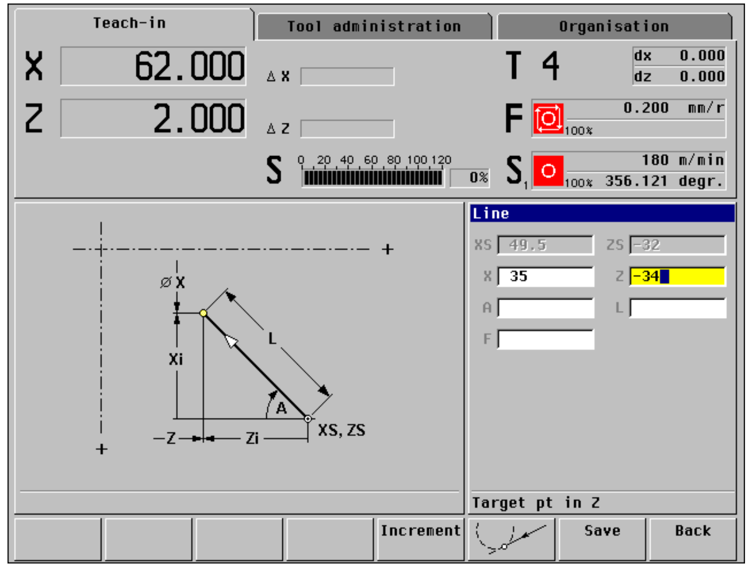


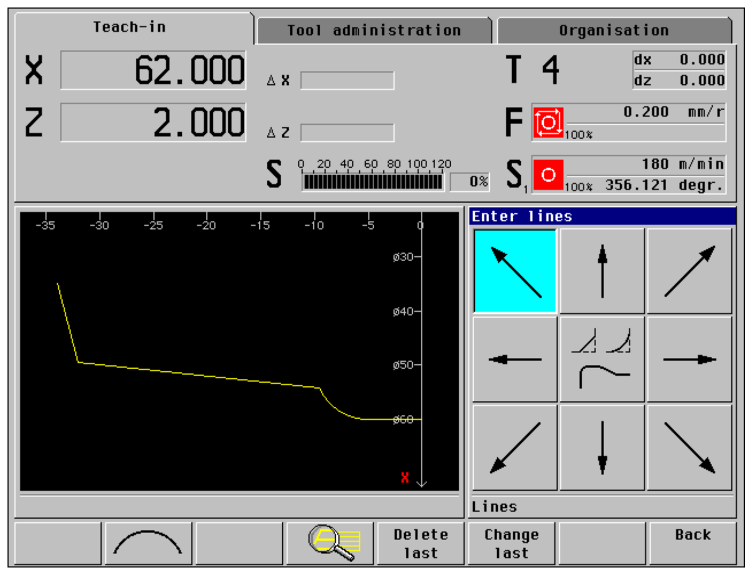


Contour element 4

The next connecting contour element is an oblique cut whose target point is known.

After you have entered the "target point X, Z," the oblique cut is unambiguously defined. MANUALplus draws the contour elements in the graphics window.

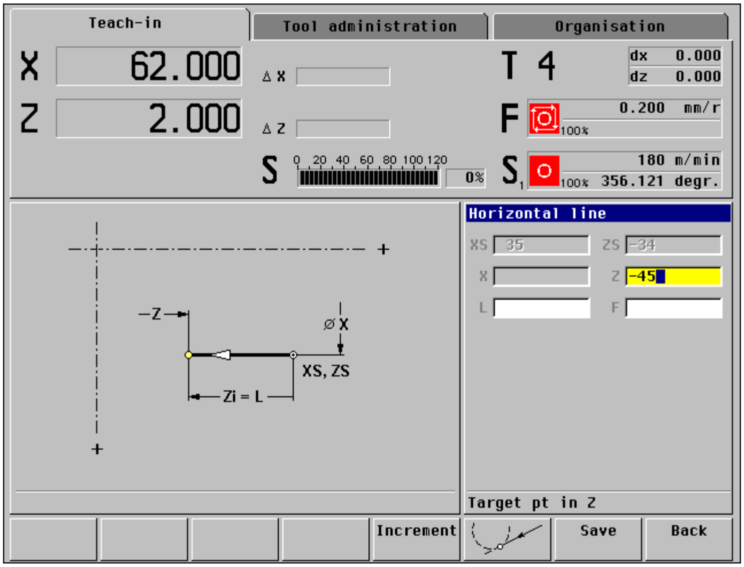


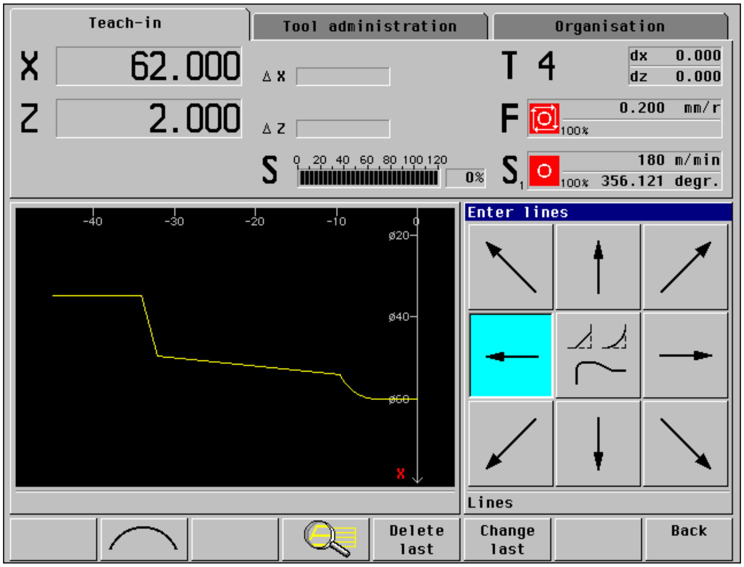


Contour element 5

The next connecting contour element is a horizontal line.

After you have entered the "target point Z," the line is unambiguously defined. MANUALplus draws the contour elements in the graphics window.

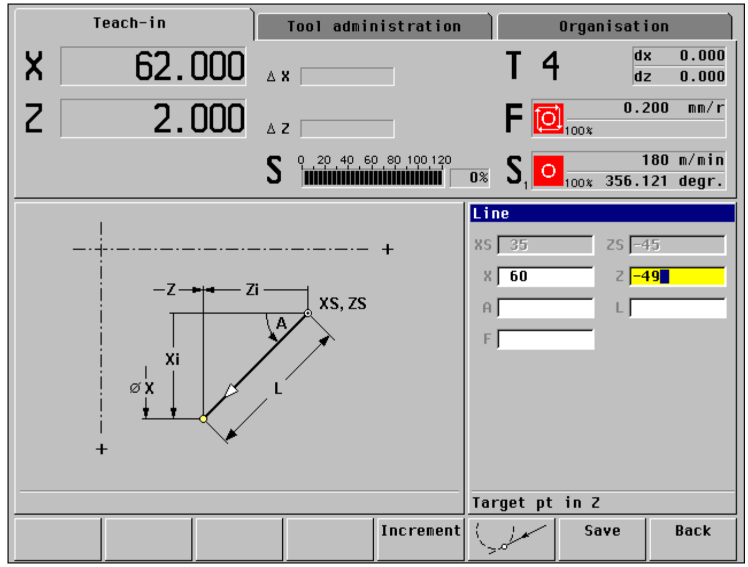


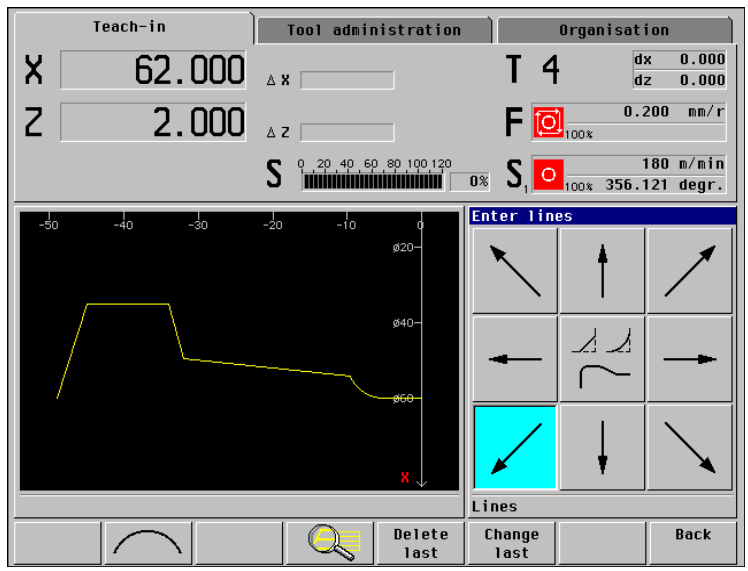


Contour element 6

The next connecting contour element is an oblique cut whose target point is known.

After you have entered the "target point X, Z," the oblique cut is unambiguously defined. MANUALplus draws the contour elements in the graphics window.



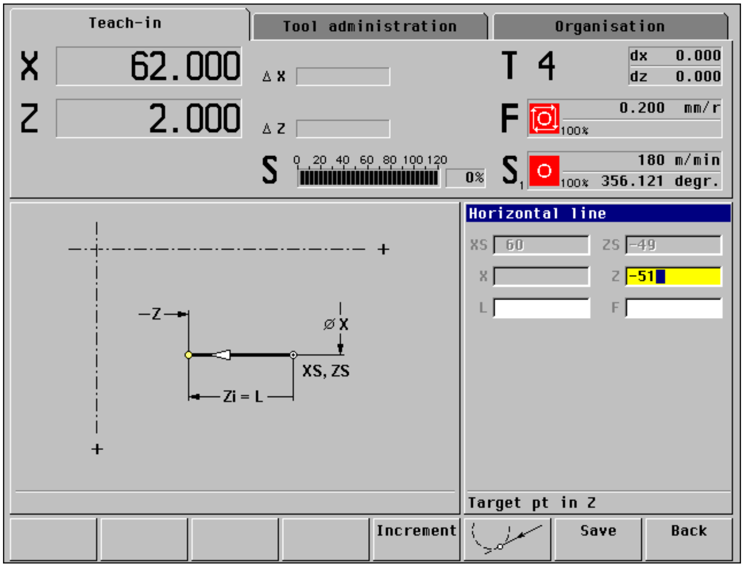


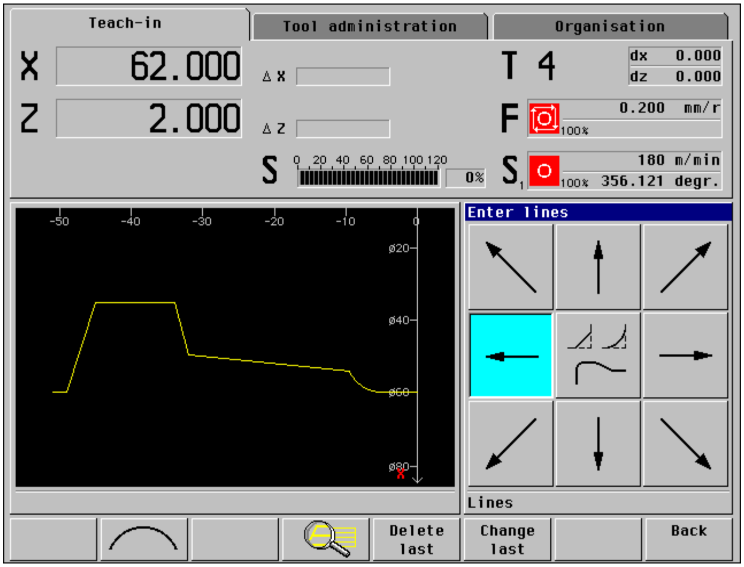
Contour element 7

The next connecting contour element is a horizontal line.

After you have entered the "target point Z," the line is unambiguously defined. MANUALplus draws the contour elements in the graphics window.

The "rough contour" has been completely defined. You can now exit the input mode with ***Back.***



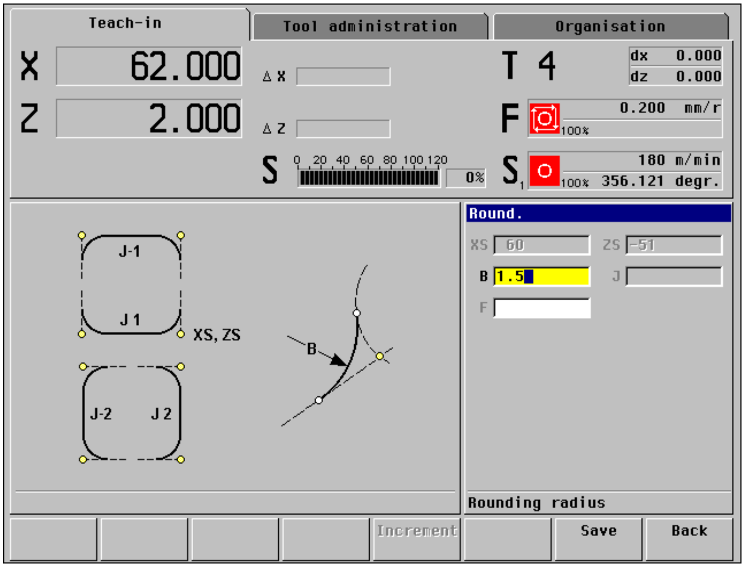


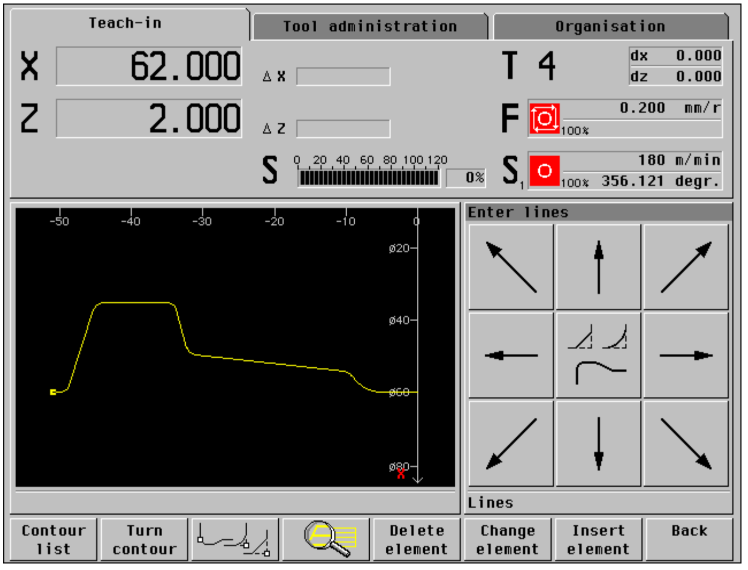
Rounding the corners

The rounding arcs are "superimposed" on the existing contour. Then select the corner ***(Next corner/ Previous corner).*** Following that, define the "rounding radius B." MANUALplus inserts the rounding in the existing ICP contour and draws the "perfected" contour.

MANUALplus offers the next contour corner for selection. In this example, all existing corners need to be rounded.

The ICP contour is completely programmed (see figure at lower right). ***Back*** concludes ICP programming and ***Input finished*** concludes the ICP cycle.

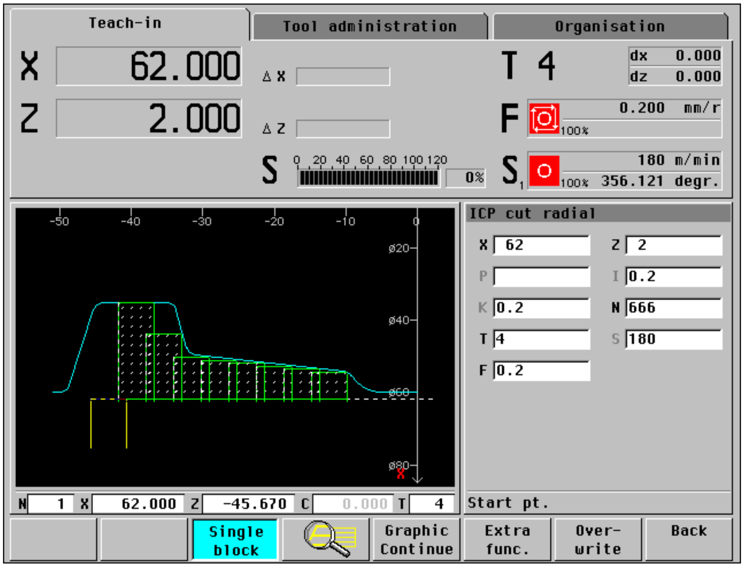




Checking the "ICP recessing radial" cycle

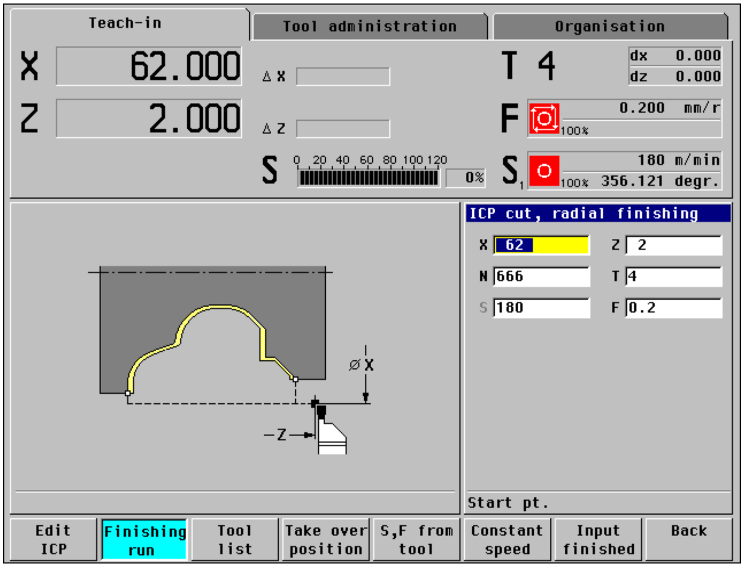
With the graphic simulation function, you can check the execution of the ICP recessing cycle (***Graphics*** soft key). Activating the ***Single block*** function allows you to check the paths of traverse more carefully one block at a time. In the figure to the upper right, the recessing operation has not been completed yet.

You can then transfer the cycle to the cycle program with ***Save*** or ***Overwrite.***



ICP recessing (finishing)

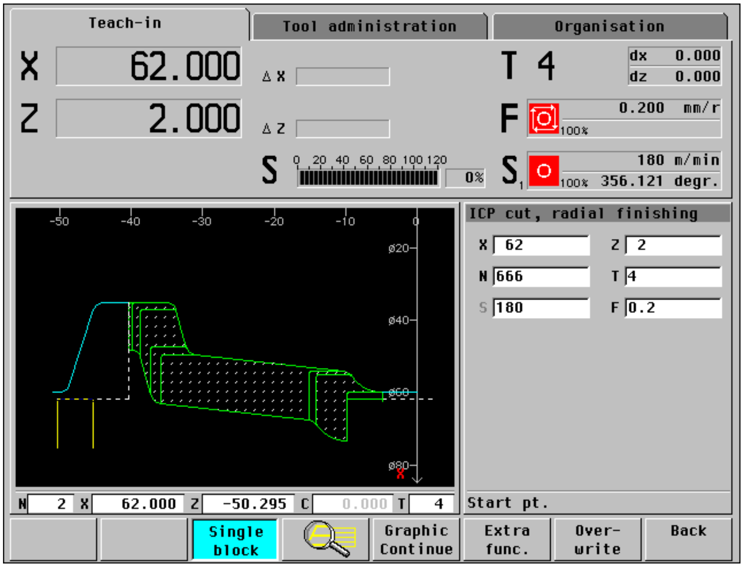
ICP contour "666" (recess) which was defined in the above steps is also used for the finishing cycle.



Checking the "ICP recessing radial, finishing" cycle

With the graphic simulation function, you can check the execution of the ICP recessing, finishing cycle (***Graphics*** soft key). In the figure to the upper right, the finishing operation has not been completed yet.

You can then transfer the cycle to the cycle program with ***Save*** or ***Overwrite.***

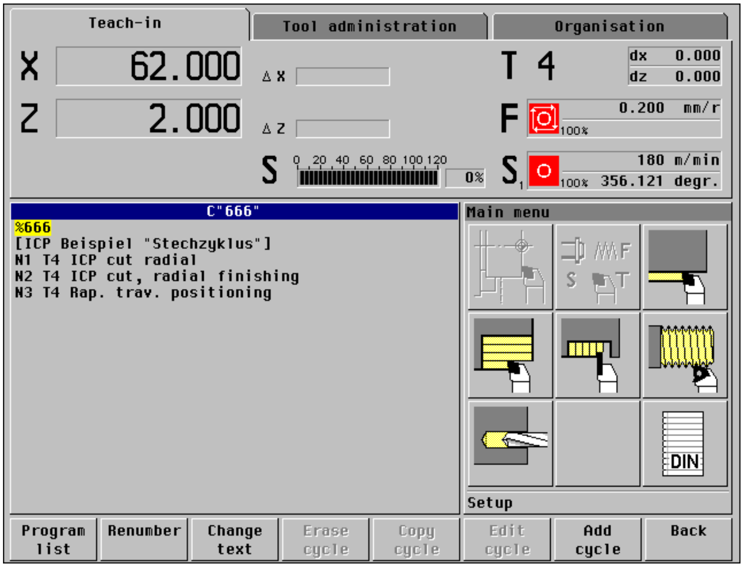


Cycle program "ICP recessing example"

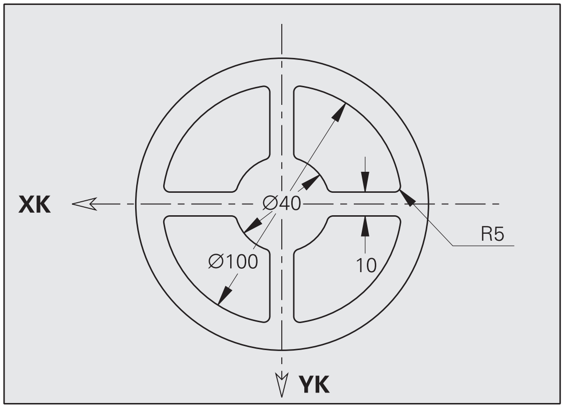
Besides the ICP cycles, the created cycle program also includes the positioning cycles for tool change (see figure to the right).

Functions of the cycles:

- N1: Contour recessing.
- N2: Contour finishing.
- N3: Position the tool for removing the workpiece.



9.5 ICP Example "Milling Cycle"



The milling example illustrates the use of an ICP contour for machining a pattern. The individual working steps for machining the ICP contour and integrating the contour into ICP cycles are based on the workpiece drawing.

In the process described below, you create an ICP contour description and a cycle program for parts production.

The cycle used is "ICP contour, pattern circular, axial."

Required tool

- **Milling tool:**
  - Position T40
  - WO = 8 Tool orientation
  - I = 8 Cutter diameter
  - K = 4 Number of teeth
  - TF = 0.025 Feed per tooth

Sequence of working steps

- Preconditions:
  – The turning operation is completed.
  – The tool dimensions have been determined.
- Switch to "Teach-in" mode.
- Enter the positioning cycles for tool change.
- Call "ICP contour, axial."
- Activate "Pattern, circular" in addition.
- ICP contour programming.
- Integrate the ICP contour in the milling cycle (roughing).
- Generate a milling cycle (finishing).
- Integrate the ICP contour in the milling cycle (finishing).

Note on defining ICP contours in patterns

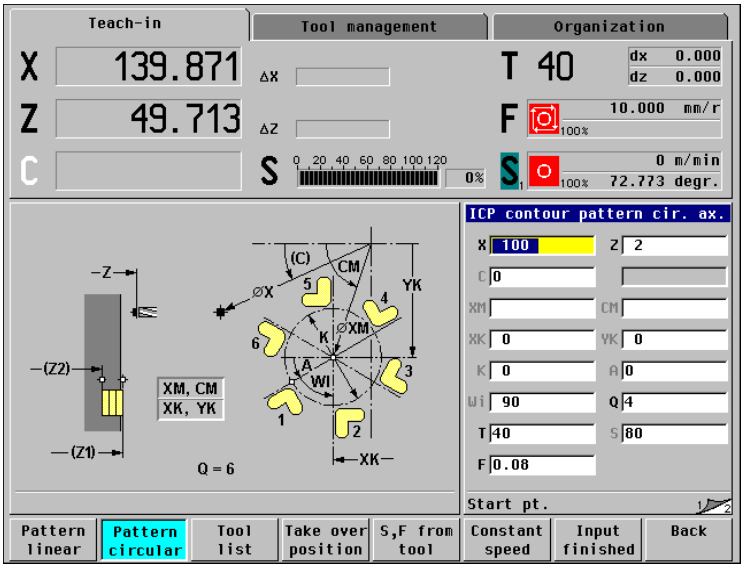
In this example, the first milling contour is programmed as indicated in the workpiece drawing. The coordinate datum therefore is the reference point for the definition of the pattern positions.

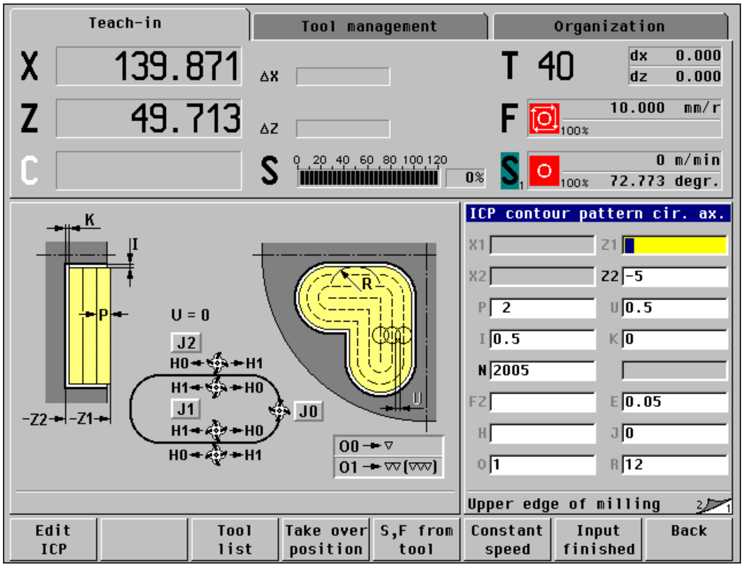
Alternately, you can dimension the first milling contour "in the coordinate datum" and define the position of the milling contours in the pattern positions.

Milling cycle—roughing

The roughing cycle used is "ICP contour, pattern circular, axial." After defining the cycle parameters, press ***Edit ICP*** to call the ICP programming function.

> The pattern diameter is "K=0", since the exact position of the "first milling contour" is defined and the ICP contours are symmetrically arranged around the face center.





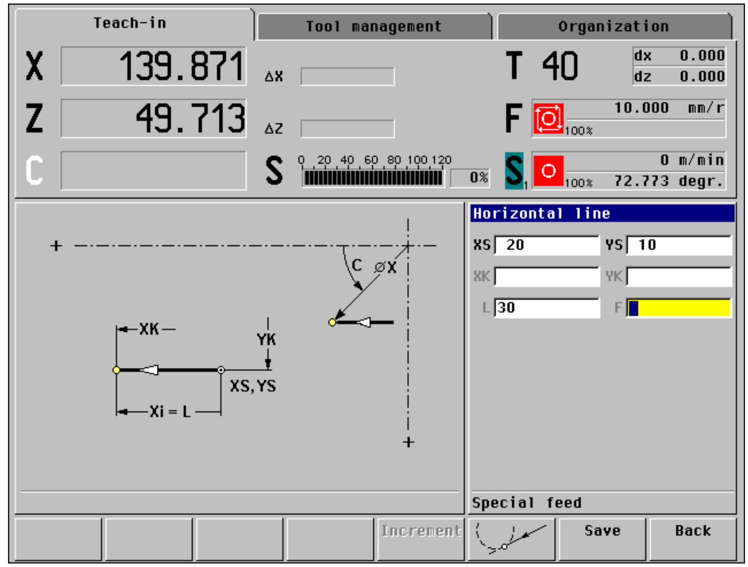
Contour element 1

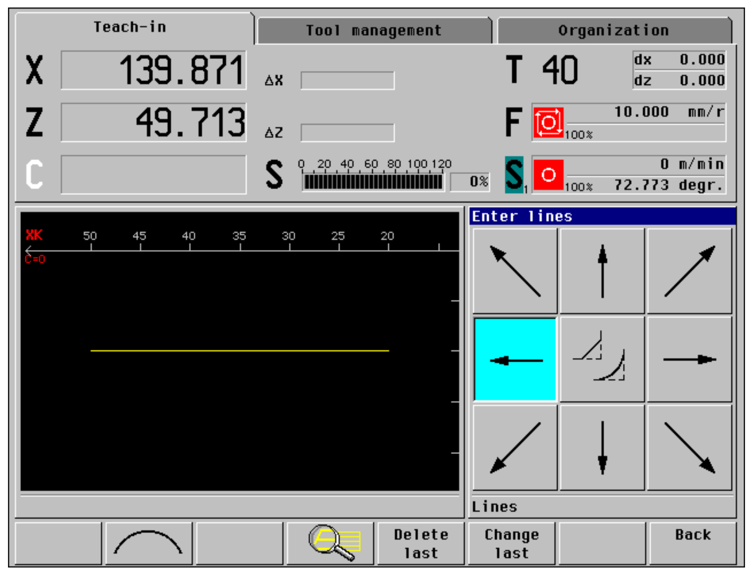
First you enter the "rough contour." Then you use the "superimposition" function to define the roundings.

The contour starts with a horizontal line.

The starting point of the ICP contour is defined in "XS, YS" when programming the first contour element.

The element is defined unambiguously after the "length of line" has been entered. MANUALplus draws the contour element in the graphics window.

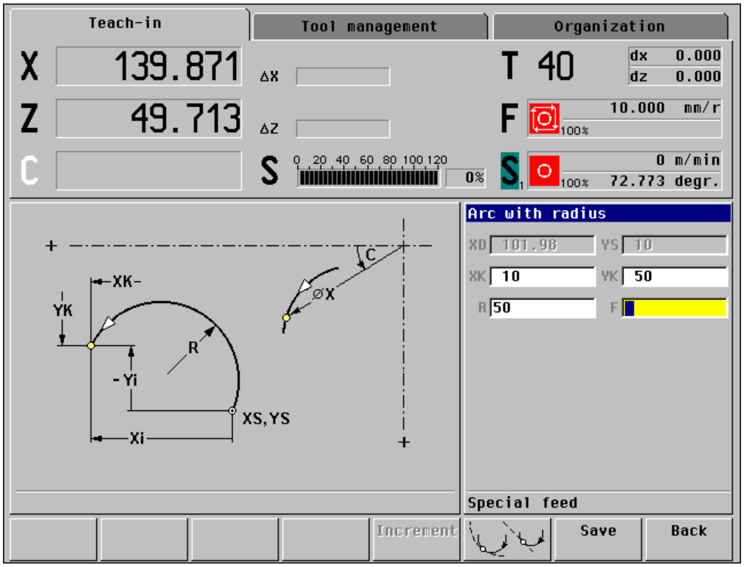


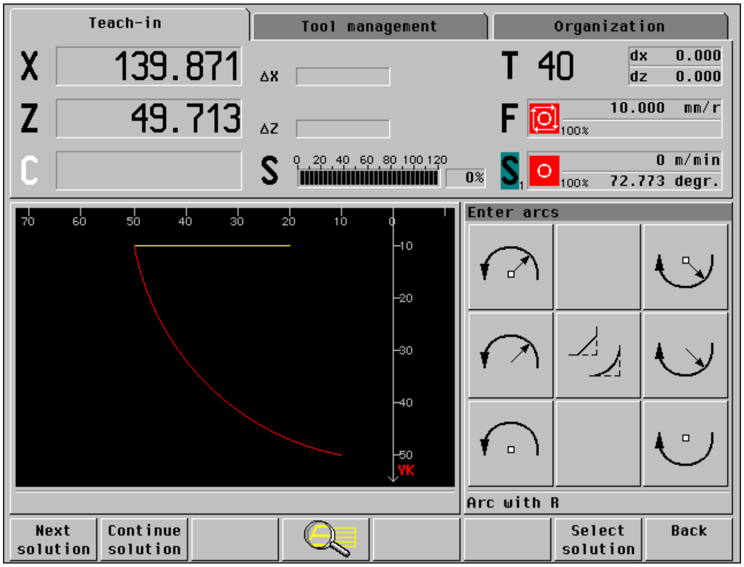


Contour element 2

The next connecting contour element is a circular arc. The target point and the radius must be defined.

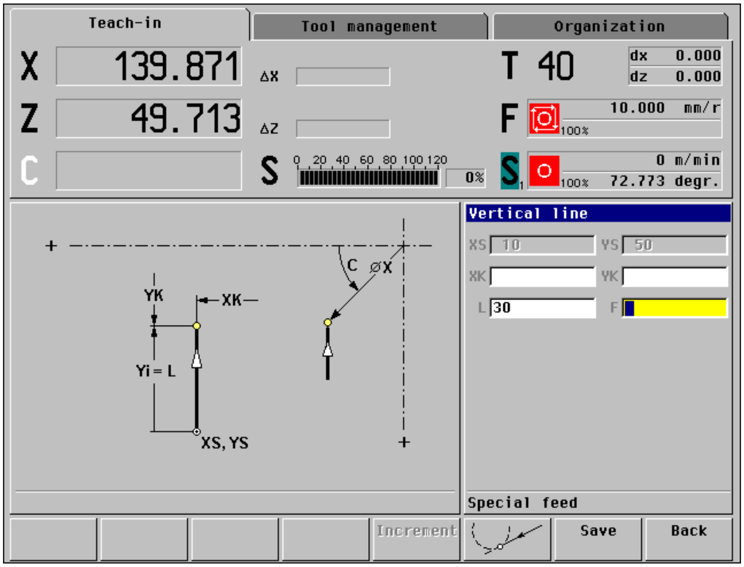
Since there are two solutions, MANUALplus asks which solution is to be used.

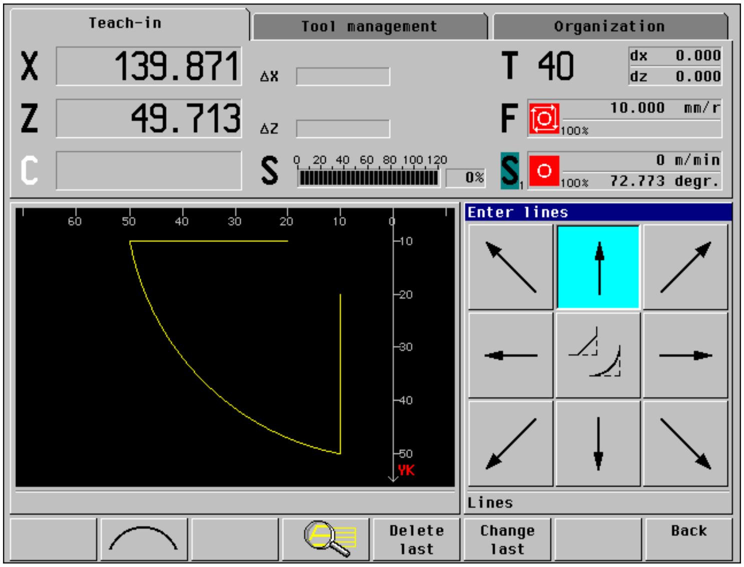




Contour element 3

A vertical line follows. The element is defined unambiguously after the "length of line" has been entered.

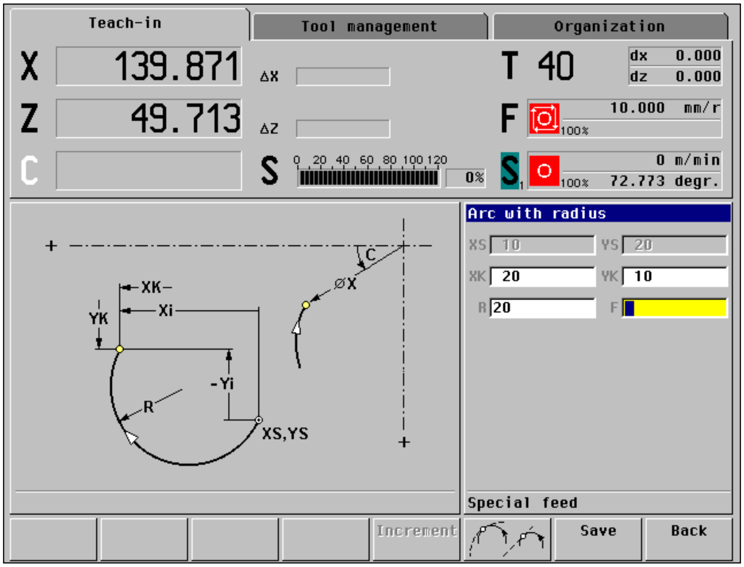


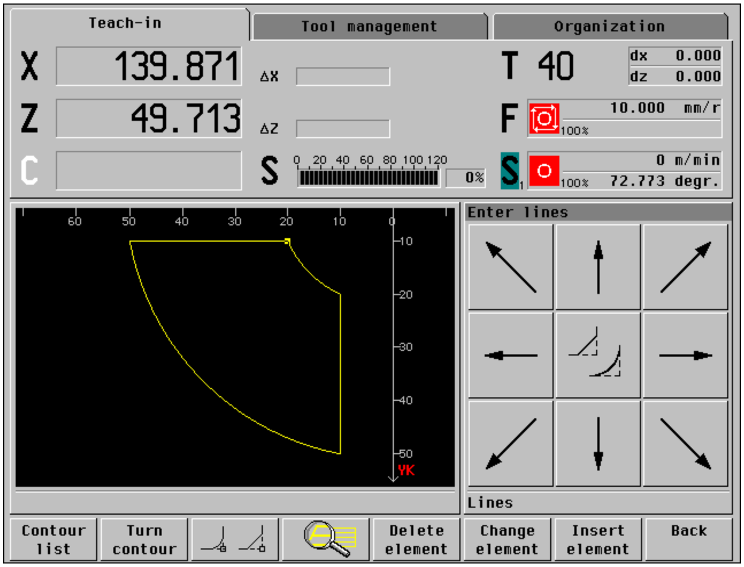


Contour element 4

A circular arc follows. The target point and the radius must be defined. Now the milling contour is closed. This is the precondition for milling pockets.

Since there are two solutions, MANUALplus asks which solution is to be used.



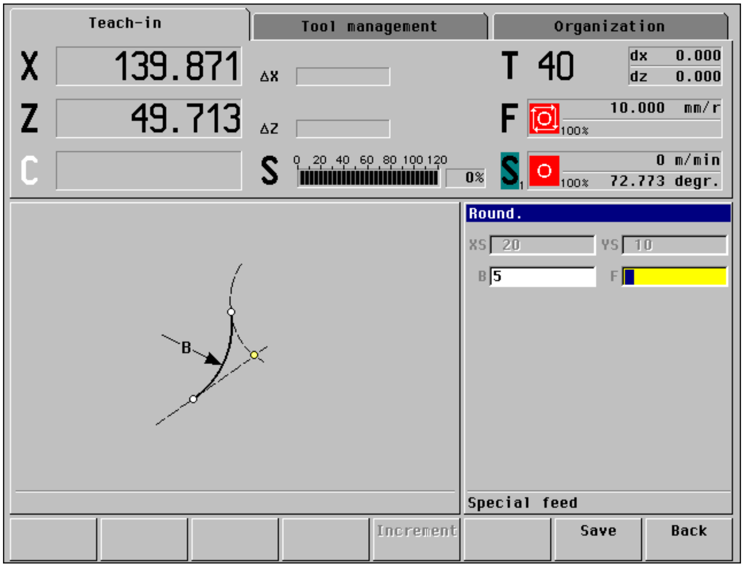


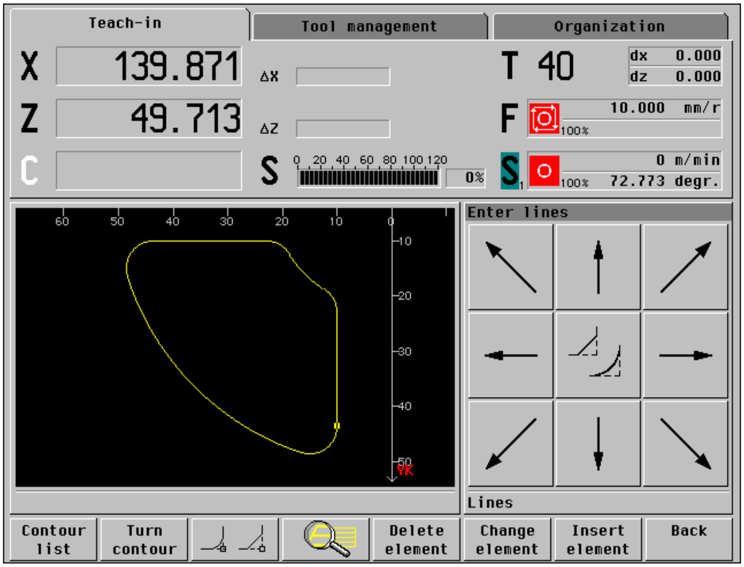
Rounding the corners

The rounding arcs are "superimposed" on the existing contour. Then select the corner ***(Next corner/ Previous corner)***. Following that, define the "rounding radius B." MANUALplus inserts the rounding in the existing ICP contour and draws the "perfected" contour.

MANUALplus offers the next contour corner for selection. In this example, all existing corners need to be rounded.

The ICP contour is completely programmed (see figure at lower right). ***Back*** concludes ICP programming and ***Input finished*** concludes the ICP cycle.



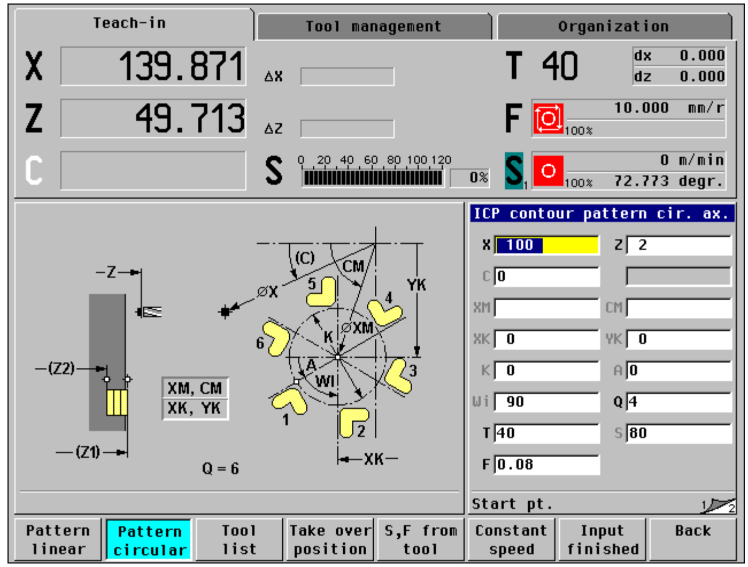


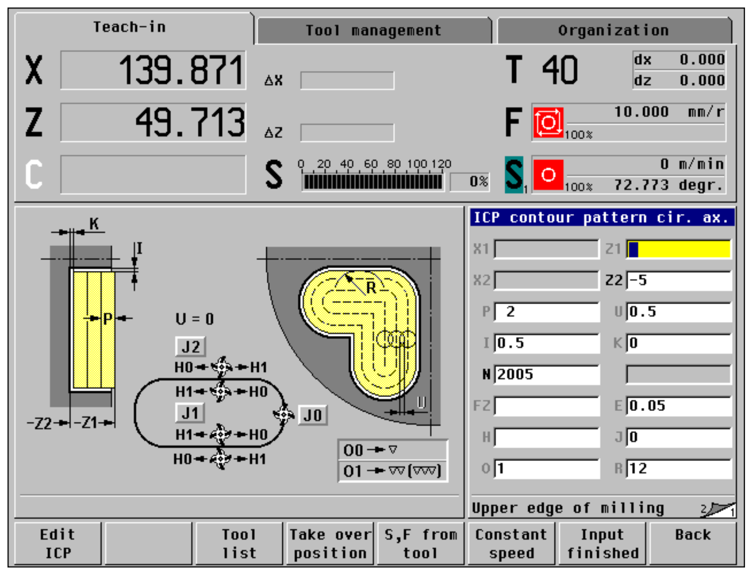
Milling cycle—finishing

The workpiece is machined with "ICP contour, pattern circular, axial" and the generated ICP contour.

"O=1" defines the finishing cycle—with "J=0," the pocket floor is finished from the inside towards the outside.

The same cutter that was used for the roughing cycle is used for the finishing cycle.

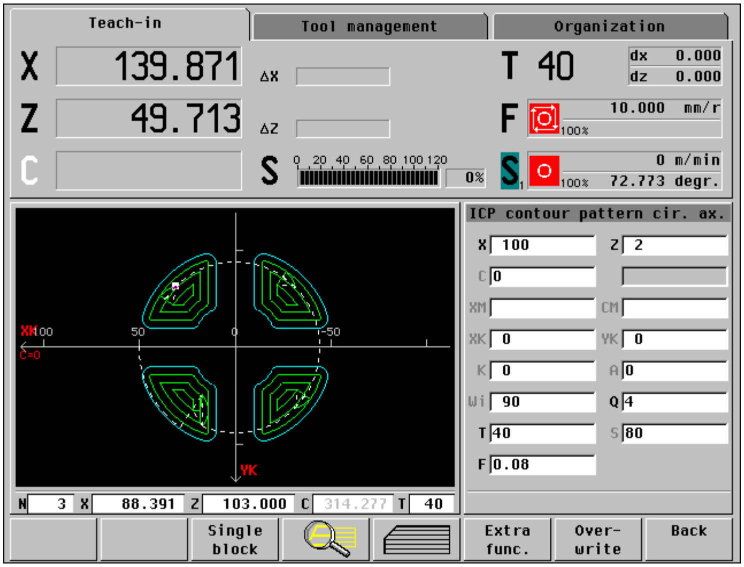




Checking the ICP milling cycle (finishing)

With the graphic simulation function, you can check the execution of the ICP milling (finishing) cycle (***Graphics*** soft key).

You can then transfer the cycle to the cycle program with ***Save*** or ***Overwrite.***

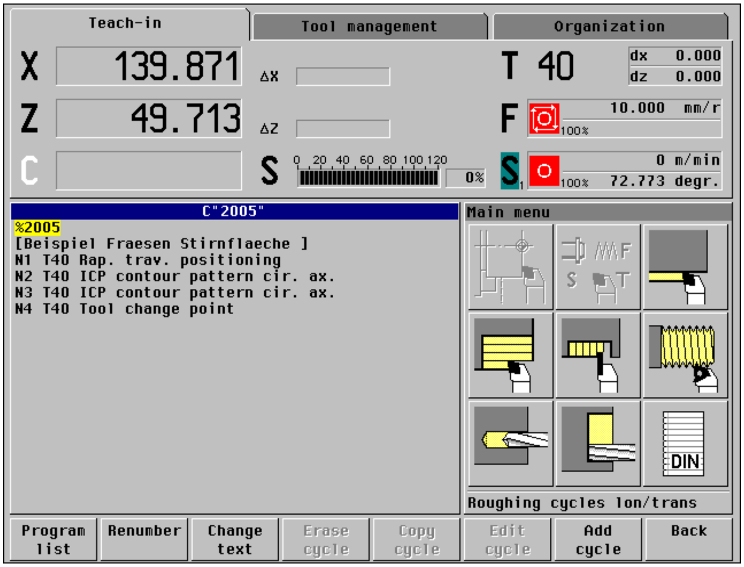


Cycle program "ICP milling example"

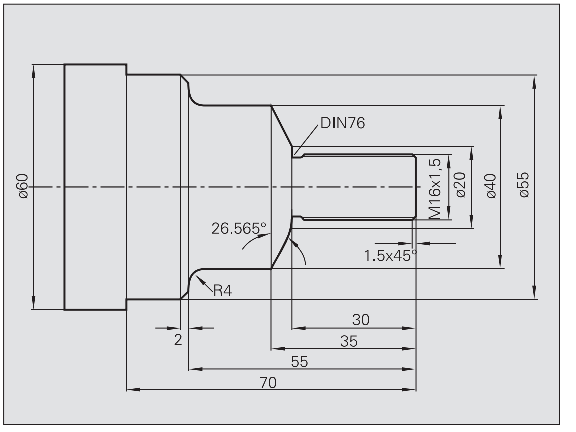
Besides the ICP cycles, the created cycle program also includes the positioning cycles for tool change (see figure to the right).

Functions of the cycles:

- N2: Pocket milling—Roughing.
- N3: Pocket milling—Finishing.
- N4: Position the tool for removing the workpiece.



9.6 DIN Programming Example "Threaded Stud"



This example illustrates how to machine a threaded stud using the DIN programming feature. The individual working steps that are defined in the DIN program are based on the workpiece drawing.

Required tools

- **Roughing tool:**
  - Position T1
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.8 Tool radius
- **Finishing tool:**
  - Position T2
  - WO = 1 Tool orientation
  - A = 93° Setting angle
  - B = 55° Nose angle
  - R = 0.5 Tool radius
- **Threading tool:**
  - Position T3
  - WO = 1 Tool orientation

Sequence of working steps

- Clamp a workpiece blank (diameter 60 mm, length 100 mm).
- Machine setup
  – Define the workpiece zero point.
  – Measure the tool dimensions.
  – Enter the tool change position.
- Switch to the DIN editor.
- Write the DIN program "Threaded stud."
- Simulate the DIN program "Threaded stud."

DIN program "threaded stud"

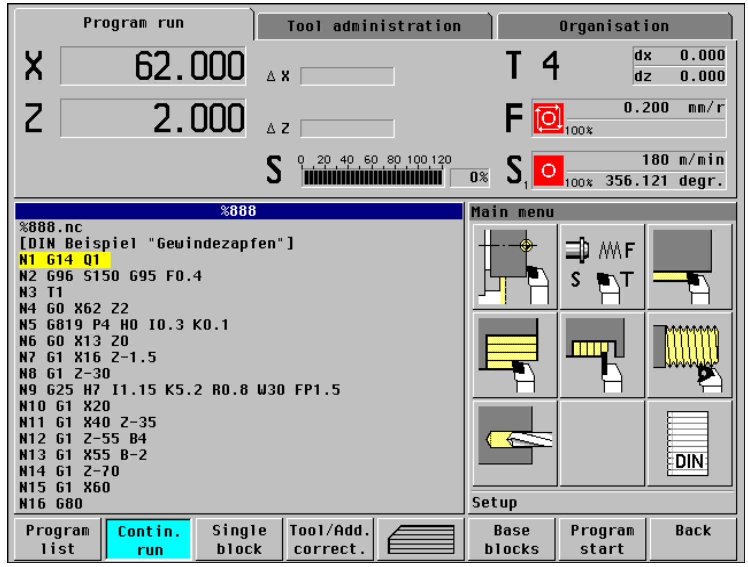
%888.nc	Program number of the DIN program
DIN example "threaded stud"	Program description
N1 G14 Q1	Approach the tool change position, insert the roughing tool
N2 G96 S150 G95 F0.4 T1	Call roughing tool, program spindle speed and feed rate
N3 G0 X62 Z2	Approach the workpiece
N4 G819 P4 H0 I0.3 K0.1	"Longitudinal contour roughing with recessing" cycle
N5 G0 X13 Z0	Start point of contour description (for roughing cycle G819)
N6 G1 X16 Z-1.5	Contour definition
N7 G1 Z-30	
N8 G25 H7 I1.15 K5.2 R0.8 W30 FP1.5	Undercut contour (an element of the contour description)
N9 G1 X20	
N10 G1 X40 Z-35	
N11 G1 Z-55 B4	
N12 G1 X55 B-2	
N13 G1 Z-70	
N14 G1 X60	
N15 G80	End of contour description (for roughing cycle G819)
N16 G14 Q1	Move to tool change point, insert finishing tool
N17 G96 S220 G95 F0.2 T2	Call finishing tool, program spindle speed and feed rate
N18 G0 X62 Z2	Approach the workpiece
N19 G89	Contour finishing cycle
N20 G42	Tool is to the left of the contour
N21 G0 X13 Z0	Starting point of contour description (for finishing cycle G89)
N22 G1 X16 Z-1.5	Contour definition
N23 G1 Z-30	
N24 G25 H7 I1.15 K5.2 R0.8 W30 FP1.5	Undercut contour (an element of the contour description)
N25 G1 X20	
N26 G1 X40 Z-35	
N27 G1 Z-55 B4	
N28 G1 X55 B-2	
N29 G1 Z-70	
N30 G1 X60	
N31 G80	End of contour description (for finishing cycle G89)
N32 G14 Q1	Move to tool change point, insert threading tool
N33 G97 S800 T3	Call threading tool, program (constant) spindle speed
N34 G0 X16 Z2	Move to thread starting point
N35 G350 Z-29 F1.5 U-999	"Simple longitudinal single-start thread" cycle

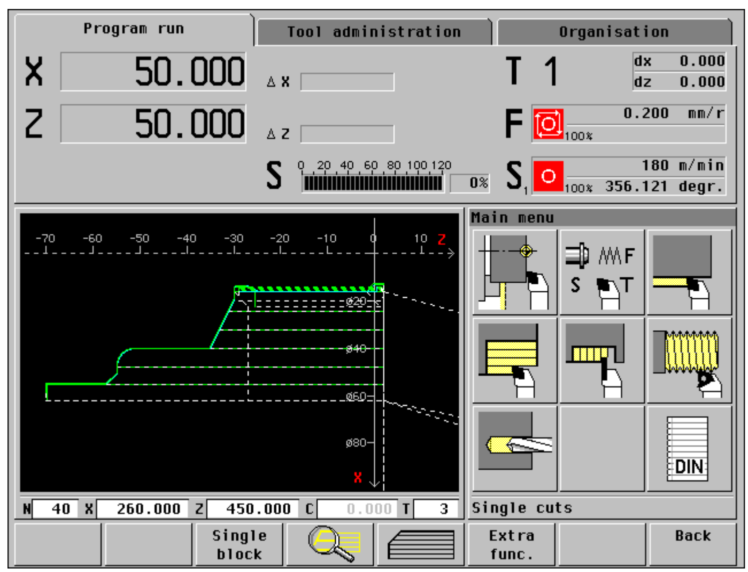
N36 G14 Q1	Retract the tool (approach the tool change position)
N37 M30	End of program
END	

Checking the DIN program

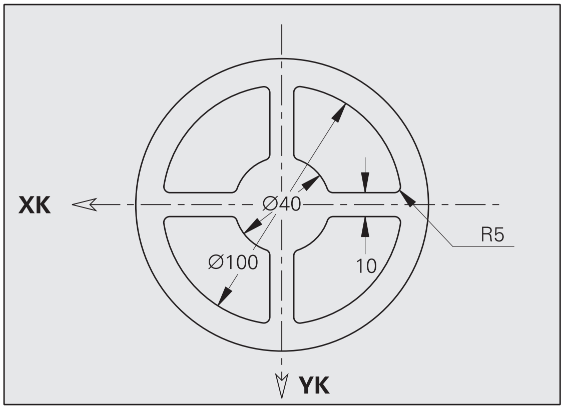
After you have written the DIN program "Threaded stud," switch to the "Program run" mode to test the program (see figure at upper right).

The simulation shows the contour of the threaded stud and each individual tool movement (see figure at lower right).





9.7 DIN Programming Example "Milling Cycle"



This example illustrates how to machine the face using the DIN programming feature.

Required tool

- **Milling tool (roughing and finishing):**
  - Position T40
  - WO = 8 Tool orientation
  - I = 8 Cutter diameter
  - K = 4 Number of teeth
  - TF = 0.025 Feed per tooth

Preconditions:

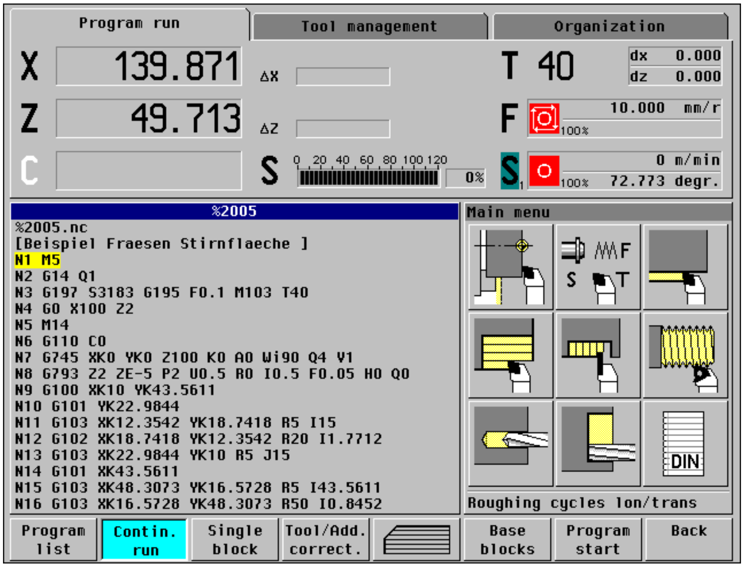
- The turning operation is completed.
- The tool dimensions have been determined.

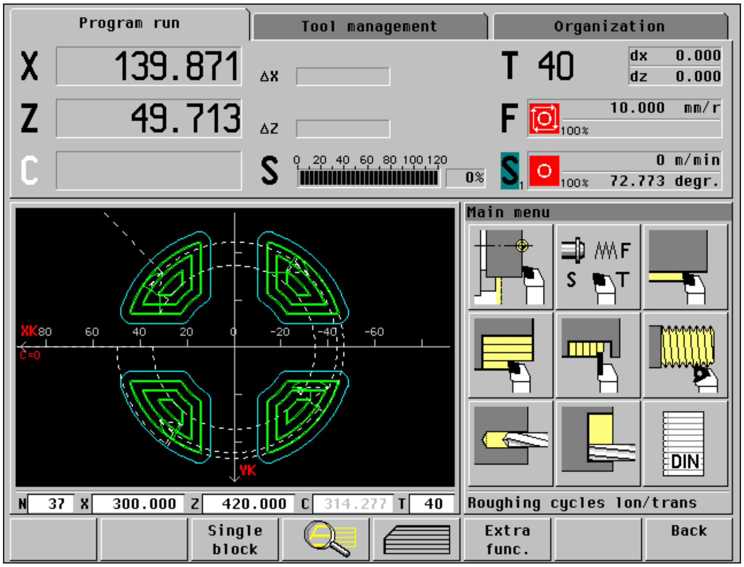
DIN program "face milling"

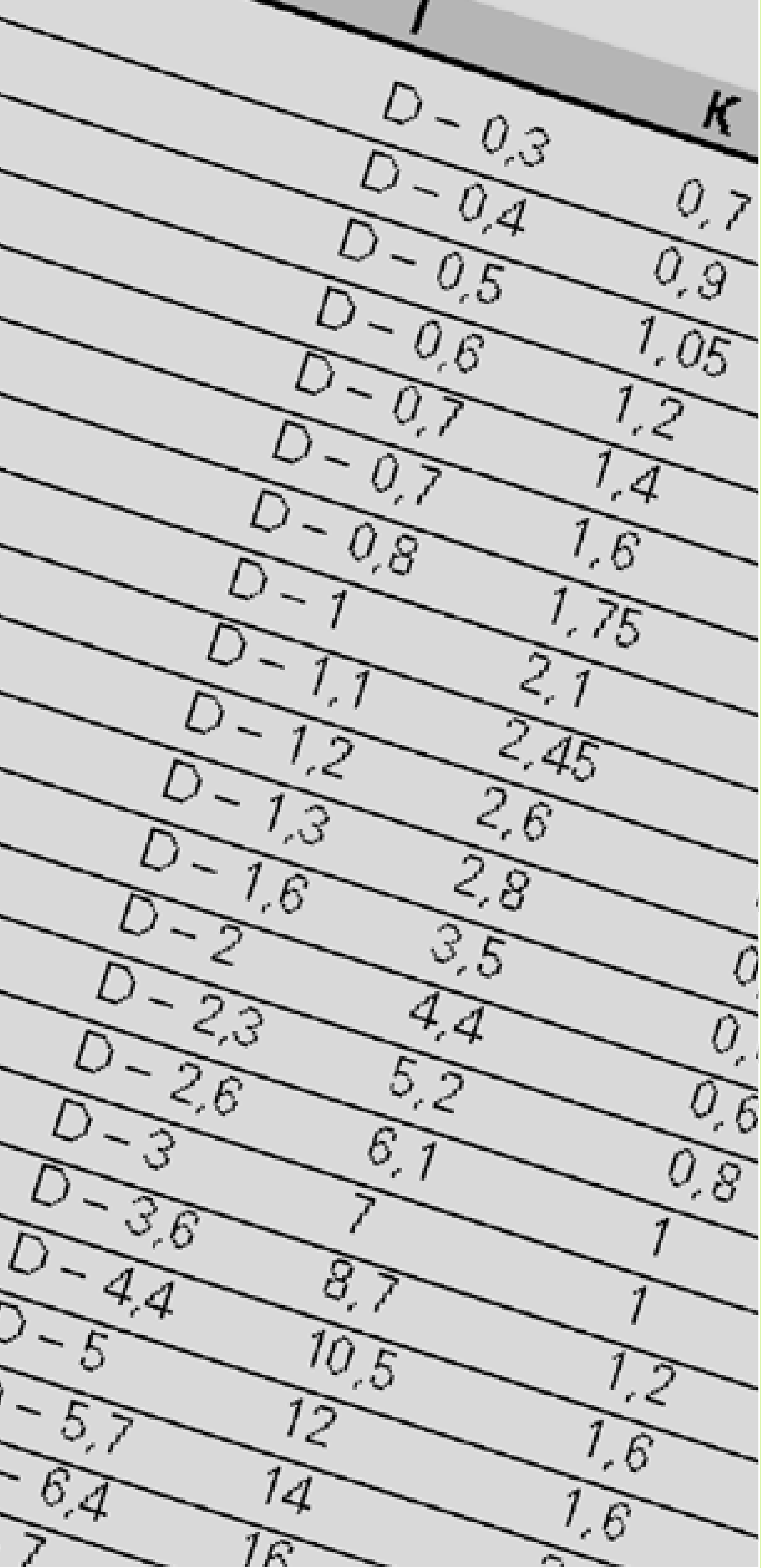
%2005.nc	Program number of the DIN program
[Example of face milling]	Program description
N1 M5	Spindle STOP
N2 G197 S3183 G195 F0.12 M103	Program the speed, feed rate
N3 T40	Call the milling tool
N4 M14	Activate the C axis
N5 G110 C0	Position the C axis
N6 G0 X80 Z2	Approach the workpiece
N7 G793 Z0 ZE-6 P3 U0.5 I1 K0.15 F0.1 E0.08 H0 Q0	"Contour milling, face" (roughing cycle)
N8 G100 XK20 YK5	Starting point of contour definition for cycle G793
N9 G101 XK50 B5	Contour definition
N10 G103 XK5 YK50 R50 Q1 B5	
N11 G101 XK5 YK20 B5	
N12 G102 XK20 YK5 R20 B5	
N13 G80	End of contour definition
N14 M15	Deactivate the C axis
N15 G14 Q0	Retract the tool (approach the tool change position)
N16 M30	End of program
END	

Checking the DIN program

After you have written the DIN program "face milling," switch to the "Program run" mode to test the program (see figure at upper right). To check the contours and each individual tool movement, switch the simulation to "Face view" (see figure to the bottom right).







10

Tables and Overviews

10.1 Thread Pitch

If the thread pitch has not been defined, it is calculated from the diameter according to the following table.

Diameter	Thread pitch
1	0.25
1.1	0.25
1.2	0.25
1.4	0.3
1.6	0.35
1.8	0.35
2	0.4
2.2	0.45
2.5	0.45
3	0.5
3.5	0.6
4	0.7
4.5	0.75
5	0.8
6	1
7	1
8	1.25
9	1.25
10	1.5
11	1.5

Diameter	Thread pitch
12	1.75
14	2
16	2
18	2.5
20	2.5
22	2.5
24	3
27	3
30	3.5
33	3.5
36	4
39	4
42	4.5
45	4.5
48	5
52	5
56	5.5
60	5.5
64	6
68	6

10.2 Undercut Parameters

DIN 76—undercut parameters

MANUALplus determines the parameters from the thread pitch according to the following table.

Designations:

- I = undercut diameter
- K = undercut length
- R = undercut radius
- W= undercut angle

Thread undercut DIN 76—external thread				
Thread pitch	**I**	**K**	**R**	**W**
0.2	D –0.3	0.7	0.1	30°
0.25	D –0.4	0.9	0.12	30°
0.3	D –0.5	1.05	0.16	30°
0.35	D –0.6	1.2	0.16	30°
0.4	D –0.7	1.4	0.2	30°
0.45	D –0.7	1.6	0.2	30°
0.5	D –0.8	1.75	0.2	30°
0.6	D –1	2.1	0.4	30°
0.7	D –1.1	2.45	0.4	30°
0.75	D –1.2	2.6	0.4	30°
0.8	D –1.3	2.8	0.4	30°
1	D –1.6	3.5	0.6	30°
1.25	D –2	4.4	0.6	30°
1.5	D –2.3	5.2	0.8	30°
1.75	D –2.6	6.1	1	30°
2	D –3	7	1	30°
2.5	D –3.6	8.7	1.2	30°

Thread undercut DIN 76—internal thread				
Thread pitch	**I**	**K**	**R**	**W**
0.2	D +0.1	1.2	0.1	30°
0.25	D +0.1	1.4	0.12	30°
0.3	D +0.1	1.6	0.16	30°
0.35	D +0.2	1.9	0.16	30°
0.4	D +0.2	2.2	0.2	30°
0.45	D +0.3	2.4	0.2	30°
0.5	D +0.3	2.7	0.2	30°
0.6	D +0.3	3.3	0.4	30°
0.7	D +0.3	3.8	0.4	30°
0.75	D +0.3	4.0	0.4	30°
0.8	D +0.3	4.2	0.4	30°
1	D +0.5	5.2	0.6	30°
1.25	D +0.5	6.7	0.6	30°
1.5	D +0.5	7.8	0.8	30°
1.75	D +0.5	9.1	1	30°
2	D +0.5	10.3	1	30°
2.5	D +0.5	13	1.2	30°

Thread undercut DIN 76—external thread				
Thread pitch	**I**	**K**	**R**	**W**
3	D –4.4	10.5	1.6	30°
3.5	D –5	12	1.6	30°
4	D –5.7	14	2	30°
4.5	D –6.4	16	2	30°
5	D –7	17.5	2.5	30°
5.5	D –7.7	19	3.2	30°
6	D –8.3	21	3.2	30°

Thread undercut DIN 76—internal thread				
Thread pitch	**I**	**K**	**R**	**W**
3	D +0.5	15.2	1.6	30°
3.5	D +0.5	17.7	1.6	30°
4	D +0.5	20	2	30°
4.5	D +0.5	23	2	30°
5	D +0.5	26	2.5	30°
5.5	D +0.5	28	3.2	30°
6	D +0.5	30	3.2	30°

DIN 509 E, DIN 509 F—undercut parameters

MANUALplus determines the parameters from the diameter according to the following table.

Designations:

- I = undercut depth
- K = undercut length
- R = undercut radius
- W= undercut angle
- undercut depth
- A= transverse angle

Undercut 509 E Diameter	I	K	R	W
<= 1.6	0.1	0.5	0.1	15°
> 1.6 – 3	0.1	1	0.2	15°
> 3 – 10	0.2	2	0.2	15°
> 10 – 18	0.2	2	0.6	15°
> 18 – 80	0.3	2.5	0.6	15°
> 80	0.4	4	1	15°

Undercut 509 F Diameter	I	K	R	W	P	A
<= 1.6	0.1	0.5	0.1	15°	0.1	8°
> 1.6 – 3	0.1	1	0.2	15°	0.1	8°
> 3 – 10	0.2	2	0.4	15°	0.1	8°
> 10 – 18	0.2	2	0.6	15°	0.1	8°
> 18 – 80	0.3	2.5	0.6	15°	0.2	8°
> 80	0.4	4	1	15°	0.3	8°

10.3 Technical Information

Specifications	
Control design	■ Contouring control with integrated motor control ■ 2 controlled axes X/Z, controlled spindle and 1 driven tool
Display	■ Integrated 10.4-inch TFT color flat-panel display ■ Highlighted actual-value and status displays ■ Load display for spindle ■ Error messages in plain language
Program memory	■ Hard disk > 4.5 GB
Input resolution and display step	■ X axis: 0.5 µm, diameter: 1 µm ■ Z axis: 1 µm ■ C axis: 0.001°
Interpolation	■ Straight line in 3 principal axes (max. ±10 m) ■ Circle in 2 axes (max. ±100 m)
Feed rate	■ Max. 9.999 m/min or max. 9.999 mm/rev. ■ Constant cutting speed ■ Maximum threading feed rate up to 99.999 m/rev. ■ Feed rate with chip breaking ■ Maximum rate of rapid traverse 99.999 m/min
Spindle	■ 0 to 9 999 rpm
Axis control	■ Integrated digital drive control for synchronous and asynchronous motors ■ Position control clock pulse: < 3 ms ■ Speed control: < 0.6 ms ■ Current control: < 0.1 ms
Spindle speed	■ Speed: 0 to 9999 rpm
Error compensation	■ Backlash / reversal error ■ Screw pitch error ■ Slope angle of an oblique axis ■ Temperature
Integral PLC	■ 512 KB program memory ■ 124 KB data memory
Data interface	■ RS 232-C, max. 38.4 kilobaud ■ RS 422-C, max. 38.4 kilobaud ■ Ethernet 10 Mb
Operating temperature	■ 0 °C to 45 °C

User functions	
Manual operation	■ Manually controlled slide movement via intermediate switch or electronic handwheels ■ Graphically supported entry and execution of cycles in conjunction with manual machine operation ■ Thread repair function for reworking threads with unclamped and re-clamped workpieces
Teach-in mode	■ Sequential linking of machining cycles ■ Graphic simulation of each machining cycle after completion of data input ■ Immediate execution after input of cycle ■ Storage of machining cycles with automatic program creation
Program run mode	■ Cycle or DIN programs in single-block mode or continuous-run mode
Programming—machining cycles	■ Graphically supported cycle programming in plain language ■ Linear and circular paths, chamfers and roundings ■ Roughing cycles for longitudinal and transverse turning operations, for simple or complex contours, or contours defined with ICP ■ Recessing cycles for simple or complex contours, or contours defined with ICP ■ Recess turning cycles for simple or complex contours, or contours defined with ICP ■ Undercuts as per DIN76, DIN509E, DIN509F ■ Parting cycle ■ Drilling, deep-hole drilling and tapping cycles ■ Linear and circular hole patterns on face and lateral surface ■ Cycles for single or multiple paraxial and tapered threads ■ Axial and radial milling cycles for slots, figures, single surfaces and polygons as well as for complex contours defined with ICP ■ Thread milling ■ Use of DIN macros in cycle programs ■ Hole patterns on face and lateral surface ■ Conversion of cycle programs into DIN programs
Interactive contour programming (ICP)	■ Straight line in 3 principal axes (max. ±10 m) ■ Circle in 2 axes (max. ±100 m)
DIN programming	■ NC programming as per DIN 66025 (ISO 6983) ■ Creation of DIN programs or DIN macros ■ Programming with roughing, recessing, recess-turning, drilling and milling cycles ■ Simple geometry programming (calculation of missing data) ■ Programming variables ■ Subprograms

User functions	
Position data	■ Nominal positions in Cartesian or polar coordinates ■ Absolute or incremental dimensions ■ Input and display in the metric or inch system ■ Distance-to-go display
Tool compensation	■ Compensation of tool-tip position in the X/Z plane ■ Automatic recognition of tool-tip position ■ Precision compensation via handwheel with transfer of compensation values to the tool table ■ Tool-tip / milling-cutter radius compensation
Tool table	■ Tool table for 99 tools with tool descriptions ■ Graphically supported tool entry ■ Tool life monitoring or monitoring of number of parts produced
Test run graphics	■ Graphic simulation of individual cycles, cycles programmed in teach-in mode, or DIN programs ■ Two-dimensional wire-frame or cutting-path graphics ■ Turning view, face view, or depiction of unrolled lateral surface ■ Zoom function for magnifying or reducing isolated details
Machining time analysis	■ Calculation of machining times and idle times ■ Consideration of switching commands triggered by CNC ■ Representation of single times per cycle or per tool change with DIN programs

Accessories	
Electronic handwheels	■ For moving the axes as on a manual lathe; a maximum of two electronic handwheels can be connected. ■ In addition, the portable handwheel HR410 can be connected.
DataPilot	Control software on PCs for: ■ Programming and program test ■ Program management ■ Management of operating-resource data ■ Data backup ■ Training

10.4 Peripheral Interface

Connector: 9-pin, D-sub pins

Pin	Signal	RS-232
1	Do not assign	
2	RxD	Receive Data
3	TxD	Transmit Data
4	DTR	Data Terminal Ready
5	GND	Signal Ground
6	DSR	Data Set Ready
7	RTS	Request to Send
8	CTS	Clear to Send
9	Do not assign	

The interface is linked to the external PC by direct electrical connection. This may lead to interference in the interface, resulting from different power-supply reference levels.

Precautions:

- If possible, use the service jack on the machine for the PC.
- Engage/disengage the connection only when the machine and PC are switched off.
- The cable length must not exceed 20 m (66 ft). Use even shorter cables if there is strong electromagnetic interference.
- Recommendation: Use an adapter with electrical isolation.

Index

D

E

I

J

K

L

M

N

O

P

Q

R

Overview of G functions

Definition of workpiece blank		Page
G20	Standard blank (bar, tube)	288
G21	Contour of workpiece blank	289

Tool positioning without machining		Page
G0	Positioning in rapid traverse	290
G14	Approach the tool change position	291

Simple linear and circular movements		Page
G1	Linear path	292
G2	Circular arc with incr. center dimensioning	293
G3	Circular arc with incr. center dimensioning	293
G12	Circular arc with abs. center dimensioning	295
G13	Circular arc with abs. center dimensioning	295

Feed rate and spindle speed		Page
G26	Speed limitation for spindle	297
G126	Speed limitation, driven tool	297
G64	Interrupted (intermittent) feed	297
G193	Feed per tooth	298
G94	Constant feed	298
G95	Feed per revolution	298
G195	Feed per revolution for driven tool	298
G96	Constant cutting speed	299
G196	Constant cutting speed for driven tool	299
G97	Spindle speed (in rev/min)	299
G197	Spindle speed (in rev/min) for driven tool	299

Tool-tip/cutter radius compensation (TRC/MCRC)		Page
G40	Deactivate TRC	301
G41	Activate TRC	301
G42	Activate TRC	301

Tool compensation		Page
G148	Changing the cutter compensation	302
G149	Additive compensation	303
G150	Compensate right tool tip	304
G151	Compensate left tool tip	304

Zero point shifts		Page
G51	Zero point shift	305
G56	Additive zero point shift	306
G59	Absolute zero point shift	307

Oversizes		Page
G57	Paraxial oversize	308
G58	Contour-parallel oversize	309

Roughing cycles		Page
G80	End of cycle	310
G81	Longitudinal roughing	319
G817	Longitudinal contour roughing	311
G818	Longitudinal contour roughing	311
G819	Longitudinal contour roughing with recessing	313
G82	Transverse roughing	320
G827	Transverse contour roughing	314
G828	Transverse contour roughing	314
G829	Transverse contour roughing with recessing	316
G83	Simple contour repeat cycle	321
G836	Contour-parallel roughing	317
G87	Line with radius	322
G88	Line with chamfer	323
G89	Contour finishing cycle	318

Recessing cycles		Page
G86	Simple recessing cycle	330
G861	Axial contour recessing	324
G862	Radial contour recessing	324
G863	Axial contour recessing, finishing	326
G864	Radial contour recessing, finishing	326
G865	Simple axial recessing cycle	328
G866	Simple radial recessing cycle	328
G867	Axial recessing, finishing	329
G868	Radial recessing, finishing	329

Recess-turning cycles		Page
G811	Simple radial recess-turning cycle	332
G815	Radial recess-turning cycle	333
G821	Simple axial recess-turning cycle	332
G825	Axial recess-turning cycle	333

Thread cycles		Page
G31	Universal thread cycle	335
G32	Single thread cycle	337
G33	Thread single path	338
G35	Metric ISO thread	339
G350	Simple longitudinal single-start thread	340
G351	Extended longitudinal multi-start thread	341
G352	Tapered API thread	342
G353	Tapered thread	343
G36	Tapping cycle	357
G799	Thread milling	358

Undercut cycles, parting cycles		Page
G25	Undercut contour (DIN509 E, DIN509 F, DIN76)	344
G85	Undercut cycle (DIN509 E, DIN509 F, DIN76)	345
G851	Undercut with cylinder machining DIN 509 E	347
G852	Undercut with cylinder machining DIN 509 F	348
G853	Undercut with cylinder machining DIN 76	349
G856	Undercut type U	350
G857	Undercut type H	351
G858	Undercut type K	352
G859	Parting cycle	353

Drilling cycles		Page
G71	Drilling cycle	354
G74	Deep-hole drilling cycle	355
G36	Tapping cycle	357
G743	Linear pattern, face	383
G744	Linear pattern, lateral surface	387
G745	Circular pattern, face	385
G746	Circular pattern, lateral surface	389
G799	Thread milling	358

C axis		Page
G120	Reference diameter, lateral-surface machining	371
G126	Speed limitation, driven tool	297
G152	Zero point shift, C axis	359
G153	Standardize C axis	359
G193	Feed per tooth	298
G195	Feed per revolution for driven tool	298
G196	Constant cutting speed for driven tool	299
G197	Spindle speed (in rev/min) for driven tool	299

Overview of Cycles

Drilling cycles	Page
Overview	190
Axial drilling cycle For drilling single holes and patterns	191
Radial drilling cycle For drilling single holes and patterns	191
Axial deep-hole drilling cycle For drilling single holes and patterns	193
Radial deep-drilling cycle For drilling single holes and patterns	193
Axial tapping cycle For drilling single holes and patterns	195
Radial tapping cycle For drilling single holes and patterns	195
Thread milling For milling threads in existing holes	197
Undercut DIN 509 E Undercut and cylinder chamfer	182
Undercut DIN 509 F Undercut and cylinder chamfer	184

Milling cycles	Page
Overview	201
Rapid traverse positioning Activate C axis; position tool and spindle	202
Slot, axial For milling single slots or slot patterns	203
Figure, axial For milling a single figure	204
Axial ICP contour For milling single ICP contours or contour patterns	208

Milling cycles	Page
Face milling For milling surfaces or polygons	211
Slot, radial For milling single slots or slot patterns	215
Figure, radial For milling a single figure	216
Radial ICP contour For milling single ICP contours or contour patterns	220
Helical-slot milling, radial For milling a helical slot	223
Thread milling For milling threads in existing holes	197

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 32-1000
Measuring systems ☏ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (8669) 31-3105
E-Mail: service.lathe-support@heidenhain.de

www.heidenhain.de

Ve 04
354 267-24 · 1 · 9/2007 · F&W · Printed in Germany · Subject to change without notice