



HEIDENHAIN



MANUALplus 620

The Control for CNC and
Cycle Lathes



MANUALplus 620
with 15.6-inch touchscreen and
machine operating panel



Windows 7, Windows 8, and Windows 10 are trademarks of Microsoft Corporation

Options are functions that are integrated into the control and that allow you to tailor the feature range of the MANUALplus 620 to your actual requirements, including retroactively if necessary. Some options need to be adapted by the machine tool builder.

The features and specifications described here apply to the following control and NC software versions:

MANUALplus 620 with NC software version
548431-06 (no export license required)

This brochure supersedes all previous editions, which thereby become invalid. **Subject to change without notice.**

Contents

The MANUALplus 620...

Where can it be used?	Compact and versatile <ul style="list-style-type: none">– MANUALplus 620, the control for CNC and cycle lathes	4
What does it look like?	Intuitive and user-friendly <ul style="list-style-type: none">– The MANUALplus 620 in dialog with the user	6
What can it do?	Universally applicable <ul style="list-style-type: none">– The right programming mode for every task	8
	Easy machining with cycles (option) <ul style="list-style-type: none">– Preprogrammed machining steps– From single parts to series	10
	Well thought out, simple, and flexible <ul style="list-style-type: none">– Simple programming with smart.Turn (option)– NC program at the stroke of a key with TURN PLUS (option)	14
	Describing and importing contours <ul style="list-style-type: none">– Interactive Contour Programming (ICP) (option)	18
	Realistic testing before machining <ul style="list-style-type: none">– Graphic simulation	20
	Expandable for complex tasks <ul style="list-style-type: none">– Full-surface machining including the C axis and Y axis (option)– Turning, drilling, and milling in one setup (option)– Working in a tilted plane with the B axis (option)	22
	Rapid availability of tool data and cutting data <ul style="list-style-type: none">– The tool database and technology database of the MANUALplus 620	28
	Reliable machining <ul style="list-style-type: none">– Intelligent supervision with Load Monitoring (option)	30
	Open for communication <ul style="list-style-type: none">– Rapid availability of all information– Connected Machining– The DataPilot MP 620 programming station	31
	What accessories are there?	Workpiece measurement <ul style="list-style-type: none">– Setup, presetting, and measuring with touch trigger probes
Tool measurement <ul style="list-style-type: none">– Measuring the length, radius, and wear directly in the machine		36
... At a glance	Overview <ul style="list-style-type: none">– User functions– Options– Accessories– Specifications	37

Compact and versatile

– MANUALplus 620, the control for CNC and cycle lathes

The MANUALplus 620 is a compact and versatile contouring control that is particularly well suited for cycle-controlled lathes. The MANUALplus 620 optimally combines the ease of use of conventional lathes with the advantages of CNC-controlled machines.

Regardless of whether you are manufacturing single parts or batches or whether your workpieces are simple or complex, the control adapts to the needs of your company. The MANUALplus 620 is characterized by its simple operation and programming. It is easy to learn and requires minimum training time.

MANUALplus 620 for cycle lathes

On the MANUALplus 620, rework or simple tasks can be done in the same way as on a conventional lathe. You move the axes in the familiar way by turning the handwheels. For difficult cuts such as tapers, undercuts, or threads, you can utilize the cycles of the MANUALplus 620. This makes getting started with automated production very easy even for conventional lathe operators without previous knowledge of NC technology.

For small and medium-size production runs, you have the benefit of cycle programming. When machining the first workpiece, you can record the machining cycles and thereby save valuable time starting immediately with the second workpiece.

For increased requirements and complex machining tasks, you will profit from the programming mode smart. Turn.



MANUALplus 620 for cycle lathes

MANUALplus 620 for CNC lathes

Regardless of whether you are turning simple parts or complex workpieces, the MANUALplus 620 provides you with the benefits of graphical contour input and convenient programming with smart.Turn.

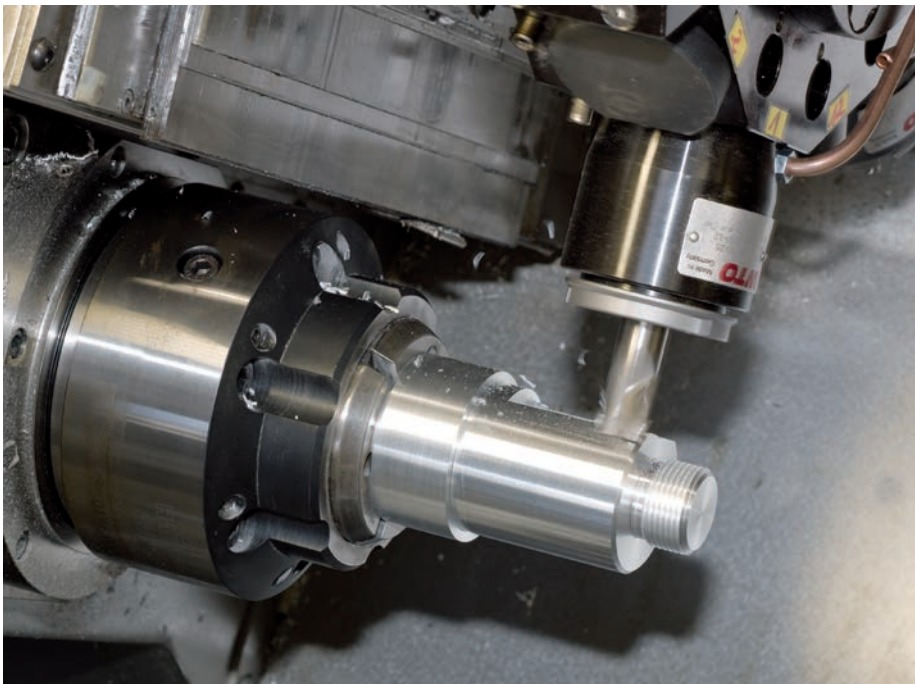
Programming with variables, controlling special machine components, or using externally created programs, etc. is no problem: simply switch to DIN PLUS. In this programming mode, you will find the solution to your specialized tasks.

The MANUALplus 620 is designed for both CNC and cycle lathes and is suitable for horizontal and vertical machines. The MANUALplus 620 supports lathes with simple tool holders and lathes with tool turrets. The tool carrier of horizontal lathes can be located in front of or behind the workpiece.

The MANUALplus 620 supports lathes with main and counter spindle, one slide (X and Z axis), C axis or positionable spindle, and driven tools as well as machines with Y and B axes.

Compatibility

You can also run existing NC programs from old HEIDENHAIN lathe controls on the MANUALplus 620. With a convenient import filter, you can easily transfer older programs (e.g., from the CNC PILOT 4290) to the new control for continued use.



Milling operation with the Y axis on a CNC lathe

Intuitive and user-friendly

– State-of-the-art multitouch operation and clear-cut display

The screen

The MANUALplus 620 is equipped with a practical 15.6-inch touchscreen designed for harsh shop conditions. It is splash-proof, scratch-resistant, and features an IP54 protection rating. The screen can be operated with gestures like those commonly used on your smartphone or tablet.

The MANUALplus 620 gives you a clear view of all the information you need in order to program, operate, and check the control and machine, including NC blocks, informational texts, error messages, etc.

While you are inputting the program, the required parameters are explained through help graphics. In the simulation, all movements of the tool are displayed in realistic detail. During program run, the MANUALplus 620 shows you complete information on the tool position, speed, and load of the drives, as well as the current machine status.

The keyboard

The MANUALplus 620 needs very few keys, and their functions are clearly indicated by easily understood symbols.

The numeric keypad is used both for inputting data and for selecting functions. The menu window graphically displays the available functions. You can use soft keys to modify the selected functions, confirm position and technology values, and control the data input.

For easy entry of program names, tool texts, comments, etc., an ASCII keyboard can be displayed.

Ergonomic and sturdy design

The optimized stainless-steel operating panel of the MANUALplus 620 features a special protective coating and is thus particularly resistant to contamination and wear. The labeling of the ergonomically formed keys can withstand extreme shop conditions. With the handy control knobs, you can make fine adjustments to the feed rate and spindle speed.

Intuitive touchscreen operation

The conventional means of operating the MANUALplus 620 has proven itself over many years. Users around the world operate their MANUALplus 620 controls using its dialog keys, navigation keys, and soft keys.

In a touchscreen version, the MANUALplus 620 now supports you with a particularly innovative and user-friendly operating interface. It combines the proven advantages of the HEIDENHAIN controls with a new type of operation by tapping, swiping, and dragging.

Practical touchscreen

The touchscreen is ideally suited to a shop environment. It can still be operated even when exposed to cooling lubricant, oil, etc.. If you wish to clean your screen, you can simply select the "Touchscreen Cleaning" mode to lock it. The touchscreen can also be operated with different types of work gloves.

PLC soft keys for machine functions

Operating modes

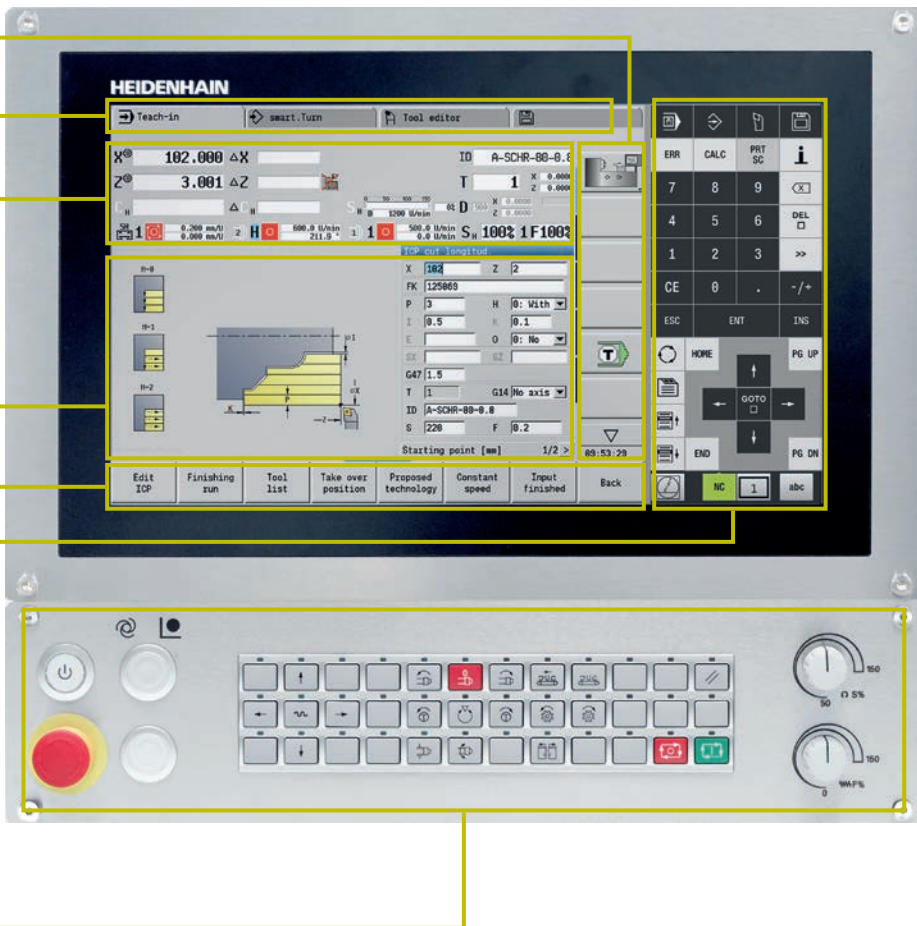
Display of the machine status. The display is configurable. You can select the suitable display field for each of the 16 segments and save different display fields for the automatic and manual modes.

Straightforward input forms for cycle programming, smart.Turn programming, or DIN PLUS programming. Help graphics illustrate the inputs during NC programming

Self-explanatory soft keys for the current operating mode

Operating panel with the option of switching between a numerical keypad and a small ASCII keyboard, function keys for switching the dashboard, HEROS system menu, calculator, help keys, and much more besides

Machine operating panel with override potentiometer



Gestures for multitouch operation

The screen of the MANULplus 620 can be operated with gestures like those commonly used on your smartphone or tablet. For example, you can zoom in and out with two fingers, while swiping allows you to quickly navigate through soft-key rows, programs, and menus.

Symbol	Gesture
	Tap
	Double tap
	Long press
	Swipe
	Drag
	Two-finger drag
	Spread
	Pinch

Universally applicable

– The right programming mode for every task

With its various programming operating modes—**cycle programming**, **smart.Turn**, and **DIN PLUS**—the MANUALplus 620 always provides the right support for your tasks.

Regardless of whether you want to perform rework or machine a single part on a cycle lathe, the cycles of the MANUALplus 620 will simplify your tasks. For manufacturing batches, you can create a cycle program by saving your cycles while machining the first

workpiece. And for more demanding tasks, the effective smart.Turn programming mode is there to help you create NC programs with speed and ease.

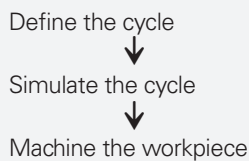
Manual

- Simple, non-repetitive machining tasks
- Rework
- Thread repair

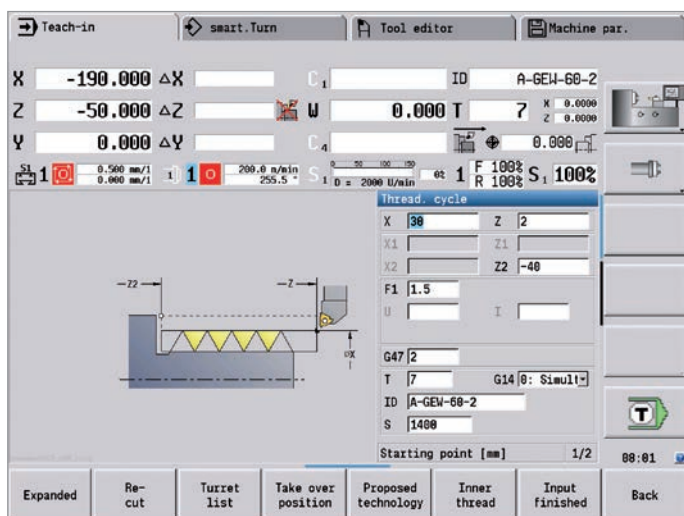
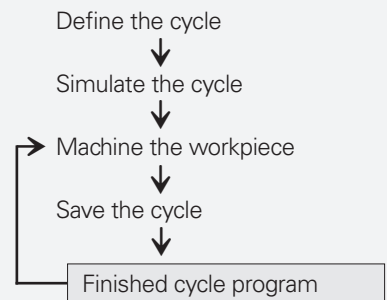
Teach-in

- Manual-oriented machining of small and medium-sized production runs
- Graphic description of complex contours

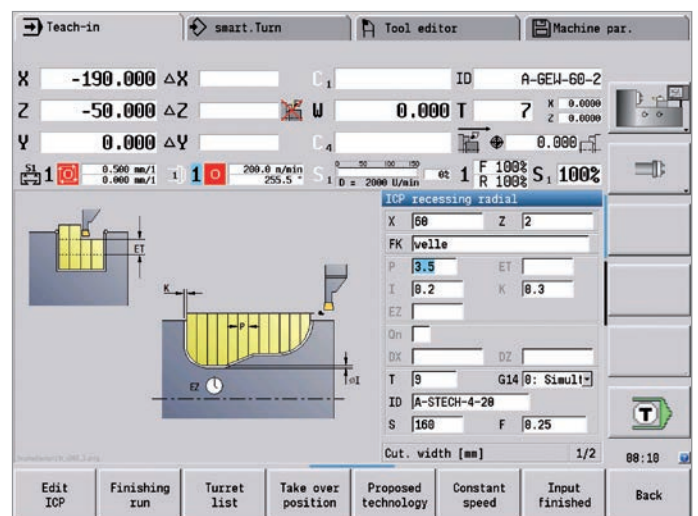
Machining with cycles



Machining with cycles



Manual cycle



Teach-in cycle

For work on CNC lathes, creating programs with smart.Turn is recommended. This programming mode from HEIDENHAIN is based on input forms and allows you to

create structured and easy-to-read NC programs. In these programs, you can even store all of the setup information needed for machining the workpiece.

And if you program with variables, need to handle special requirements, or want to use externally created NC programs, then DIN PLUS will give you the right support.

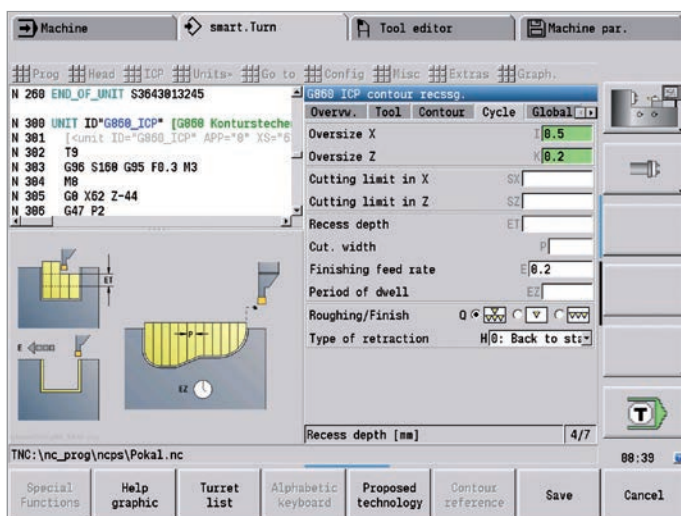
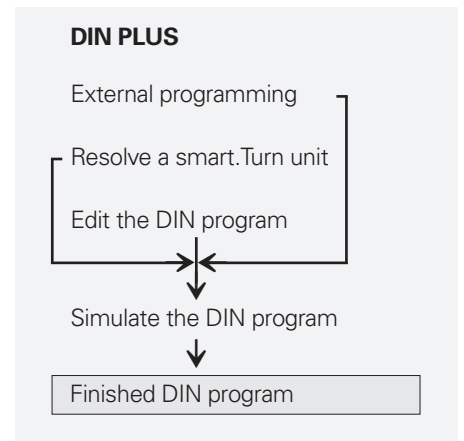
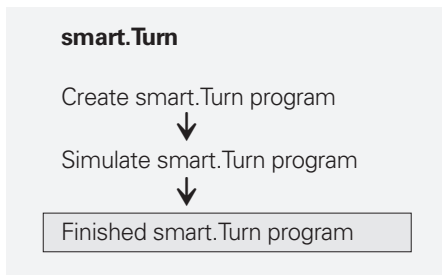
smart.Turn

- Convenient smart.Turn programming
- Graphic contour description
- Numerous machining units
- Compatible with cycle programs

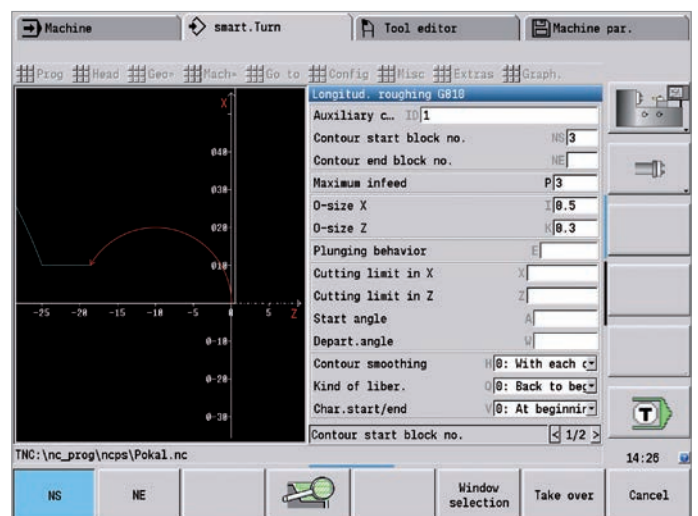
- Clearly structured and easy-to-read programs
- All required data contained in the respective working block

DIN PLUS

- Familiar DIN programming
- Graphic contour description
- Numerous fixed cycles
- Variable programming and subprograms
- Resolving smart.Turn units into DIN commands
- Compatible with externally created DIN programs



smart.Turn cycle



DIN PLUS cycle

Easy machining with cycles (option)

– Preprogrammed machining steps

With the MANUALplus 620, you can perform simple operations such as turning or facing by means of a handwheel, just as with a conventional lathe. All standard machining operations, such as area clearance, slot milling, recess turning, undercutting, parting, thread cutting, boring, drilling, and milling are stored in the MANUALplus 620 as cycles. You simply enter the positions, dimensions, and specifications, and the control will automatically run the machining program.

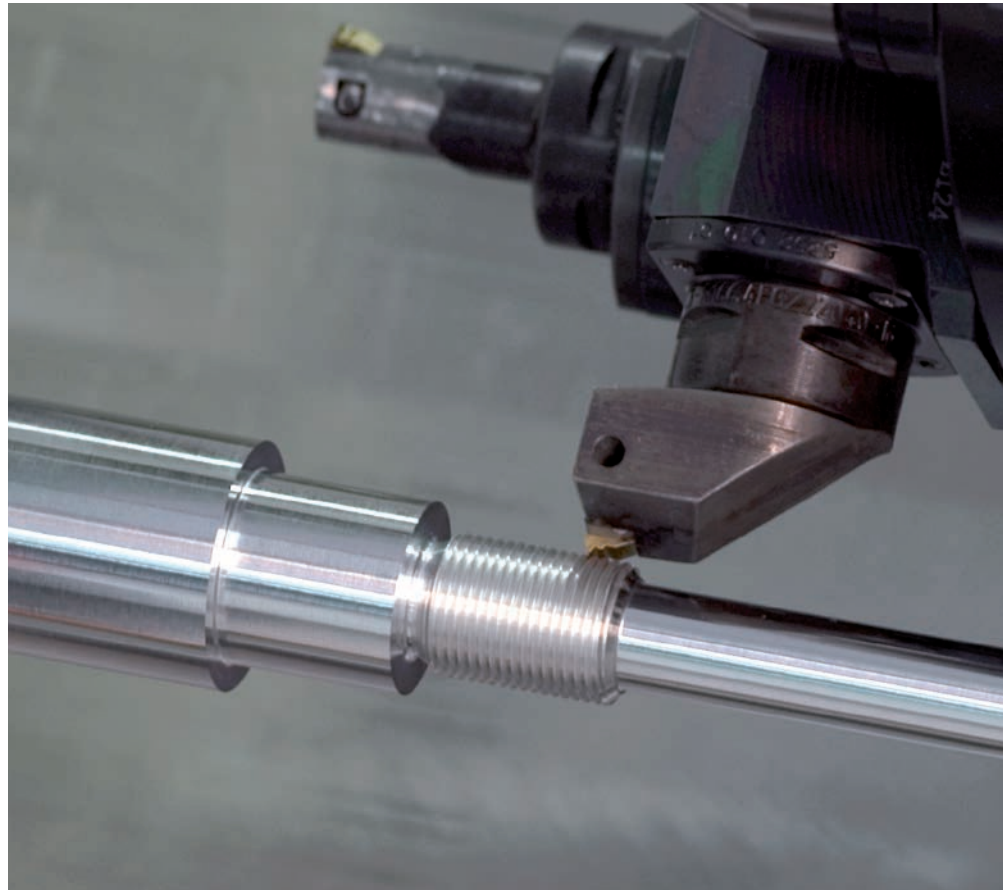
Manual workpiece machining

The MANUALplus 620 simplifies manual turning with numerous functions, without requiring you to learn complicated procedures. This enables you, for example, to adjust the feed rate and spindle speed steplessly during machining, to machine with preset tools, or to part with constant cutting speed.

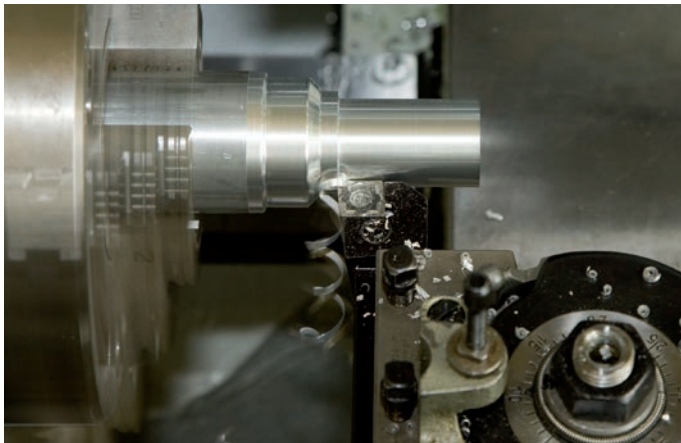
Workpiece machining with cycles

For simple, non-recurring tasks, rework, thread repair, or small production runs, the cycles of the MANUALplus 620 simplify your work.

Help graphics and dialogs illustrate the few entries that are needed for the cycles. Before cutting, use the simulation to make sure that the machining operation will run as planned.

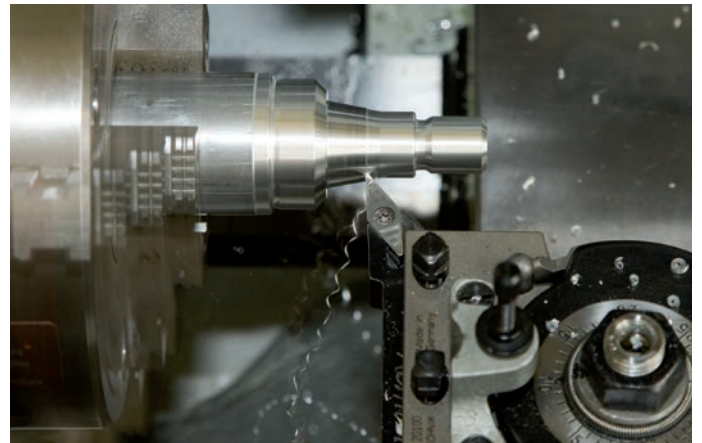


Thread cutting—three times faster with a cycle



Turning cylinders manually:

You work as usual with the handwheels and the position display on the screen.



Turning tapers automatically:

You enter the dimensions, and the MANUALplus 620 will move the saddle and cross-slide automatically.



Fewer calculations

The MANUALplus 620 automatically calculates the number of cuts for roughing, recessing, recess turning, and thread cutting. For pecking, it determines the required number of infeeds. When turning a taper, you can enter either the starting point and end point or the starting point and the taper angle—whichever is shown on your drawing.

Continuous availability of tool data

The MANUALplus 620 uses a tool database. Tool data such as cutting radius, tool angle, and point angle must only be entered once, and you can determine the setting dimensions via touch-off, for example. The MANUALplus 620 saves the data. The next time you use the tool, you simply call the tool number; the MANUALplus 620 automatically adjusts for the correct tool size, and you can immediately work to dimension.

When turning a contour, the MANUALplus 620 automatically compensates for the deviations from the cutting-edge radius. This increases the precision of your workpiece.

Technology data as suggested values

The MANUALplus 620 saves the cutting data according to the criteria of workpiece material, tool material, and machining mode. Because you have already entered the cutting material in the tool definition, you need only enter the material of your workpiece. This provides the cycle with all of the data required for suggesting the cutting data.

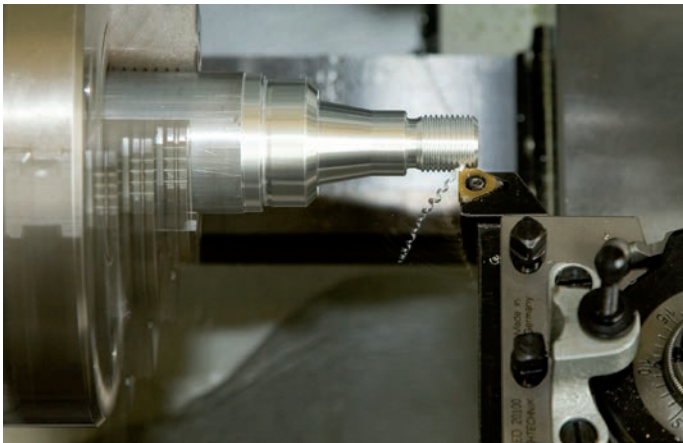
Presets

You can define the **workpiece datum** by touching the workpiece with the tool or by entering the datum coordinates.

Approach the **tool change point** once and store this position. Afterwards, a simple cycle call suffices to return to the tool change point.

Protective zone for the spindle

For every tool movement in the negative Z direction, the MANUALplus 620 checks whether the programmed protective zone will be violated. If so, it stops the movement and responds with an error message.



Running a cycle to cut a thread automatically:

Call the appropriate fixed cycle and enter the dimensions.

The MANUALplus 620 will perform all of the work steps automatically.

Easy machining with cycles (option)

– From single parts to series

The first workpiece

Machine the workpiece cycle for cycle in the usual way, and save the machining steps. When you have finished machining, you then save the created cycle program. By this means, you can create the working plan for the workpiece. The MANUALplus 620 displays the individual machining steps in the proper sequence on the screen.

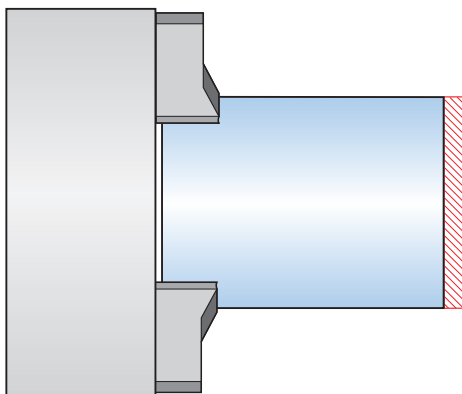
Repeating individual cycles

After you have saved the cycles for a part, you can always go back later to edit or delete the steps used to machine it, or to insert new steps.

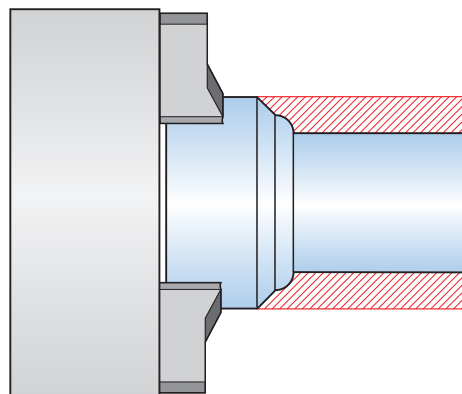
You can save a great deal of time with the MANUALplus 620 when using this feature to produce a family of parts, for example.

Saving your work for the next part

You can save all of the cycles and rerun them automatically. For each additional machined part, you save time and cost.



1. Transverse turning

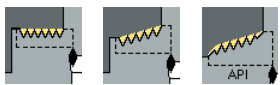


2. Longitudinal turning

Cycles for turning

You will always find the appropriate cycle in the cycle menus of the MANUALplus 620. Help graphics and dialogs explain the work step, the required dimensions, and any additional entries. After entering these values, you can graphically inspect the cutting process and let the operation run automatically.

Threads and undercuts



Single- or multi-start longitudinal, taper, or API threads



Undercuts as per DIN 76, DIN 509 E, or DIN 509 F

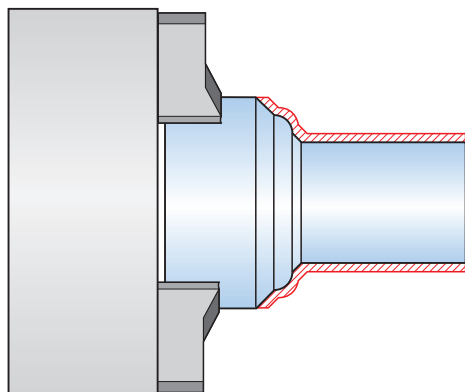
Undercuts and parting



Undercut form H, form K, or form U



Parting



3. Finishing

Turning—cutting and finishing



Longitudinal/transverse cutting for simple contours



Longitudinal/transverse cutting with plunging



Longitudinal/transverse ICP cutting for any contours



Longitudinal/transverse ICP contour-parallel cutting

Recessing and recess turning—cutting and finishing



Radial/axial recessing for simple contours



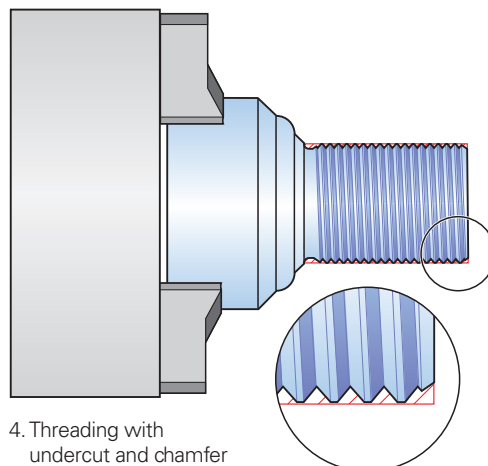
Longitudinal/transverse recess turning for simple contours



Radial/axial ICP recessing for any contours



Longitudinal/transverse ICP recess turning for any contours



4. Threading with undercut and chamfer

Well thought out, simple, and flexible

– Easy programming with smart.Turn (option)

Has the safety clearance been correctly entered, is the speed limit taken into account, and how are the oversizes defined? All of these items need to be considered not only by the beginner but also by the experienced NC programmer when creating conventional DIN programs.

The smart.Turn principle

The working block—called a unit—plays a pivotal role in smart.Turn programs. A unit completely describes and unambiguously defines a single working step. The unit includes the tool call, the technology data, the cycle call, the approach and departure strategies, and global data such as the safety clearance, etc. All of these parameters are summarized in a clearly structured dialog box.

The smart.Turn principle gives you the reassurance that the working block is defined correctly and completely. In the NC program, smart.Turn lists the unit's DIN PLUS commands. This gives you an overview of all working-block details at any time.

Programming made easy

With smart.Turn, you program with the aid of simple and intuitive fillable forms. The overview form provides you with an overview of the selected unit, while subforms provide information on the details of a working block. Clearly arranged help graphics illustrate all of the required inputs. If alternative inputs are available, then smart.Turn displays them in a list, from which you can then make a selection.

Global program parameters, such as oversizes, safety clearances, coolants, etc., are defined once in the start unit. Then smart.Turn transfers these parameters to the other units.

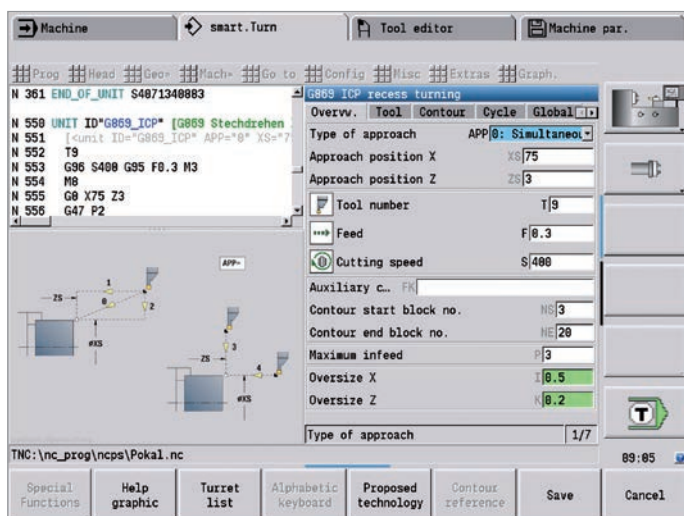
The smart.Turn programming mode supports units for roughing, finishing, recessing, recess turning, thread cutting, boring, drilling, tapping, and milling, as well as special units for program start, program end, moving the C axis in/out, subprograms, and program-section repeats.

By the way, you do not need to stop the manufacturing process to program with smart.Turn. You can create and test the smart.Turn program while the program is actually running.

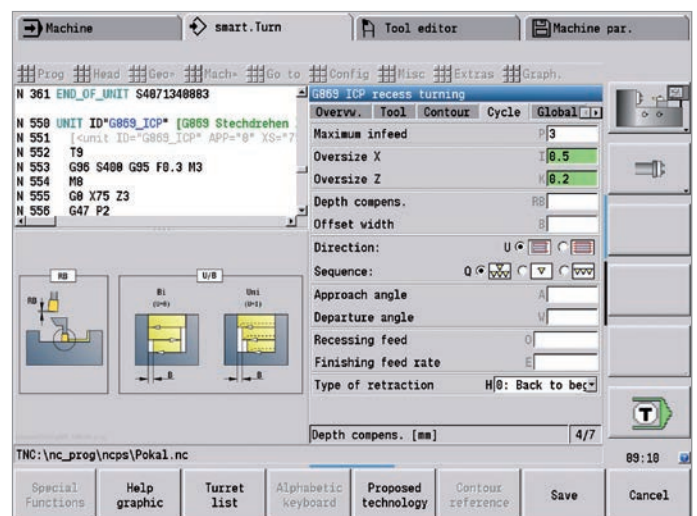
Structured and easy-to-read

Programs created with smart.Turn are characterized by their clear structure and easy understandability. The smart.Turn tool uses section codes that clearly distinguish between the program head with the setup information, the turret assignment, the workpiece description, and the actual machining operation.

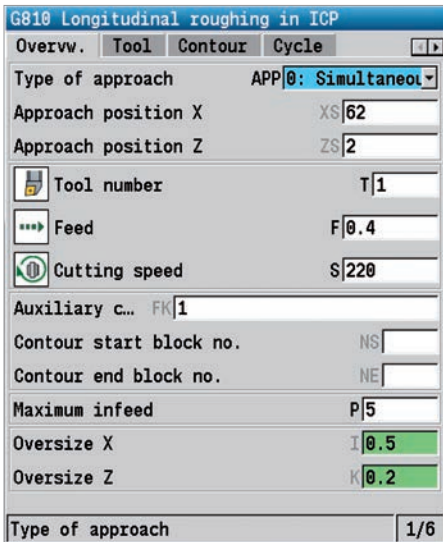
The smart.Turn method not only ensures that the program is easy to read; it also makes it possible to save all of the information required for producing the workpiece in the NC program.



Overview form in smart.Turn



Subform in smart.Turn



```

N 150 UNIT ID*G869_ICP* [G869 Stechdrehen ICP]
N 151 [

```

Programming contours

Simple contours can be defined with just a few entries in the cycle. Complex contours are described with ICP graphical interactive programming. Workpiece descriptions that are available in DXF format can be easily imported.

Contour follow-up

When you define the workpiece blank, smart.Turn works with contour follow-up. This means that the MANUALplus 620 calculates the newly created workpiece blank after every cut. The machining cycles are thereby always conformed to the current state of the workpiece blank. The contour follow-up helps you to avoid air cuts and to optimize approach paths, even through space where workpiece material has been removed.

Technology data as suggested values

The MANUALplus 620 saves the cutting data according to the criteria of workpiece material, tool material, and machining mode. Because you specify the cutting material in the tool definition, you need only still enter the material of your workpiece. This provides smart.Turn with all of the data it needs in order to set default values for the cutting data.

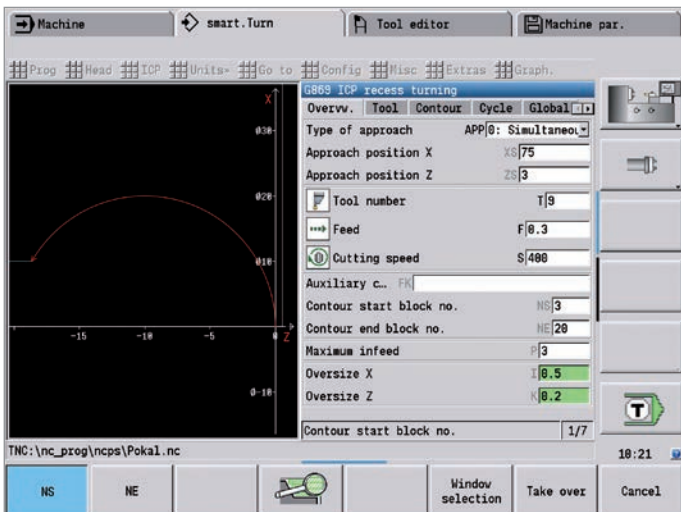
Programming in DIN PLUS

The smart.Turn tool offers units for all machining tasks, as well as units for special functions.

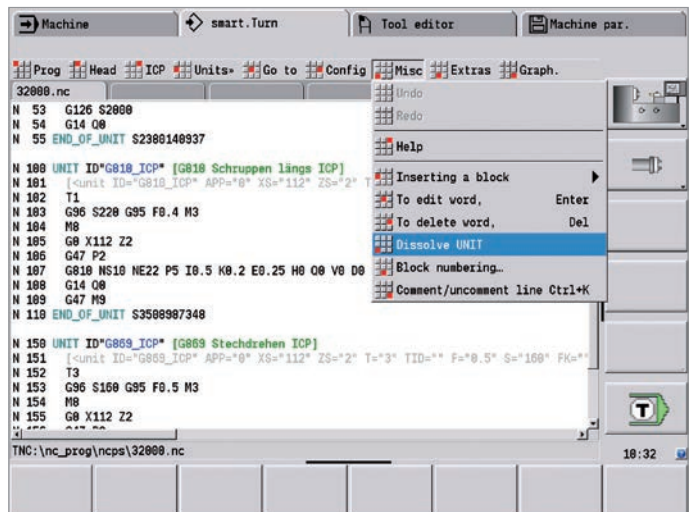
However, if you wish to control special machine components or use the variable programming function or other complex functions that are not provided by smart.Turn, then use DIN PLUS. It provides powerful machining cycles, program branches, and programming with variables. Within a program, you can switch back and forth between the smart.Turn and DIN PLUS programming modes.

Because the units are based on DIN PLUS, you can "resolve" a unit into blocks at any time in order to modify and optimize the resulting DIN PLUS program section.

Of course, the MANUALplus 620 also allows you to create a DIN program, or to import and use externally created programs.



Contour selection in ICP



Resolving a DIN PLUS unit

Well thought out, simple, and flexible

– NC program at the stroke of a key with TURN PLUS (option)

With TURN PLUS, you can create NC programs in very little time. After you have described the contour of the workpiece blank and finished part, you need only select the material and clamping devices. TURN PLUS takes care of the rest automatically: it generates the working plan, selects the machining strategy, selects the tools and cutting data, and generates the NC blocks.

The result is a comprehensively commented smart.Turn program with working blocks (units). This gives you leeway for optimization and safety when you first use the NC program.

TURNPLUS can do all this for boring, drilling, and milling operations with the C or Y axis on the front face and lateral surfaces as well as on rear-face surfaces in machines with counter spindles.

The NC program at a keystroke

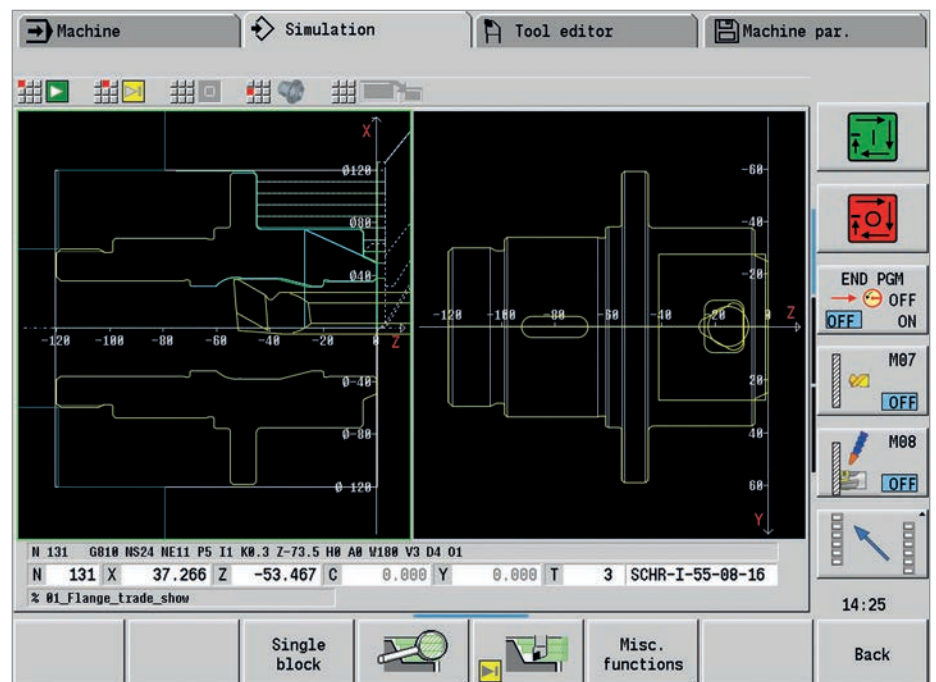
If short programming times are important, you can generate all of the working steps with only a single keystroke. Based on the entered contour and the information from the technology database, TURN PLUS creates the working plan and selects suitable machining strategies, tools, and cutting data, all on its own. This only takes a few seconds. You can monitor each individual step in the control graphic. TURN PLUS uses a reasonable sequence of possible operations, such as “first roughing transverse, then roughing longitudinal” or “finishing outside, then finishing inside.” However, you can also adapt this sequence on your own to suit various tasks. In this way, the MANUALplus 620 can also harness your company’s know-how during automatic working plan generation.

Automatic program generation for the complete contour

The MANUALplus 620 can also automatically generate the NC program for complex workpieces that need to be machined on the front face, rear face, and lateral surfaces. After defining the geometry, you can benefit from time savings of approximately 90 %—time otherwise needed for the programming of the machining operations.

Automatic program generation for the second setup

TURN PLUS knows the contour of the clamping devices and takes it into account when it generates the working plan. It thus automatically limits the tool path to a safe distance from the clamping device. Once the program for the first setup is completed, you can “rechuck” using interactive graphics. The control then automatically generates the program for the second setup without the workpiece geometry needing to be re-entered.



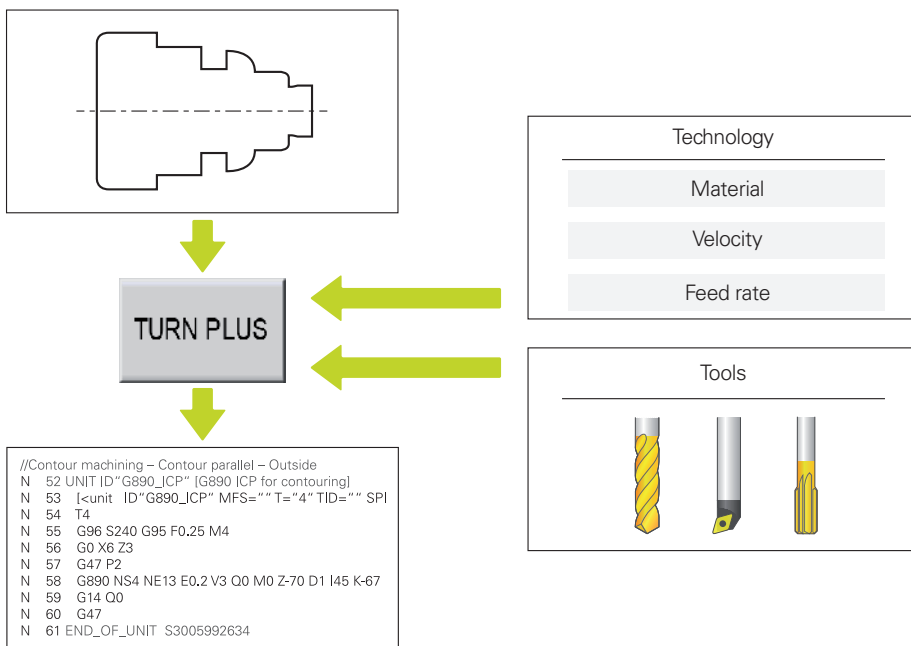
Inclined contours are no problem

The control faces unique demands when it comes to automatically generating the NC program for inclined contours. Often, the angle at which the contour falls off is steeper than the tool tip angle. In this case, the control automatically selects a different tool and performs the machining operation in the opposite direction or as a recess. In any case, the result is an executable NC program.

Defining the machining sequence

In the TURN PLUS dialog, you can define a standard machining sequence. You can store various machining sequences, such as for chuck parts, shaft machining, etc.

From the global main types of machining, such as "roughing," "finishing," or "drilling," to details such as defining a tool for a specific operation, the automatic working plan generation (AWG) feature can be adapted to the user's requirements.



Go from programming to the first cut in the shortest possible time with TURN PLUS

Describing and importing contours

– Interactive Contour Programming (ICP) (option)

For jobs that cannot be machined with the standard cycles due to the complexity of the workpiece or the lack of certain dimensions in the workpiece drawing, you need ICP. You describe the contour elements directly as they appear in the workpiece drawing. Or—if the drawing is available in DXF format—you simply import the contour.

Contour programming with ICP

You define an ICP contour by entering the contour elements one after the other in the graphic editor. When selecting the contour elements, you specify the direction of the line or the direction of rotation of the circular arc. This way, the MANUALplus 620 requires very little information about the contour element.

When entering the data, you decide whether the coordinates are to be absolute or incremental, and whether lines are defined by their end point or length and circular arcs by their center point or radius. You also specify whether the path to the next contour element should be tangential or non-tangential.

The MANUALplus 620 calculates missing coordinates, intersections, center points, etc., provided that they are mathematically defined. If the entered data permit multiple mathematically possible solutions, then you can view the alternative proposals and select the one you prefer. You can also add to or modify existing contours.

Superimposing form elements

The ICP editor recognizes the chamfer, rounding, and undercut form elements (DIN 76, DIN 509 E, DIN 509 F, etc.). You can enter these form elements during the course of the sequential contour definition. However, it is often easier to first define the “rough” contour and then superimpose the form elements. This is done by selecting the corner on which the form element is to be placed and then inserting the element.

ICP contours for cycle programs

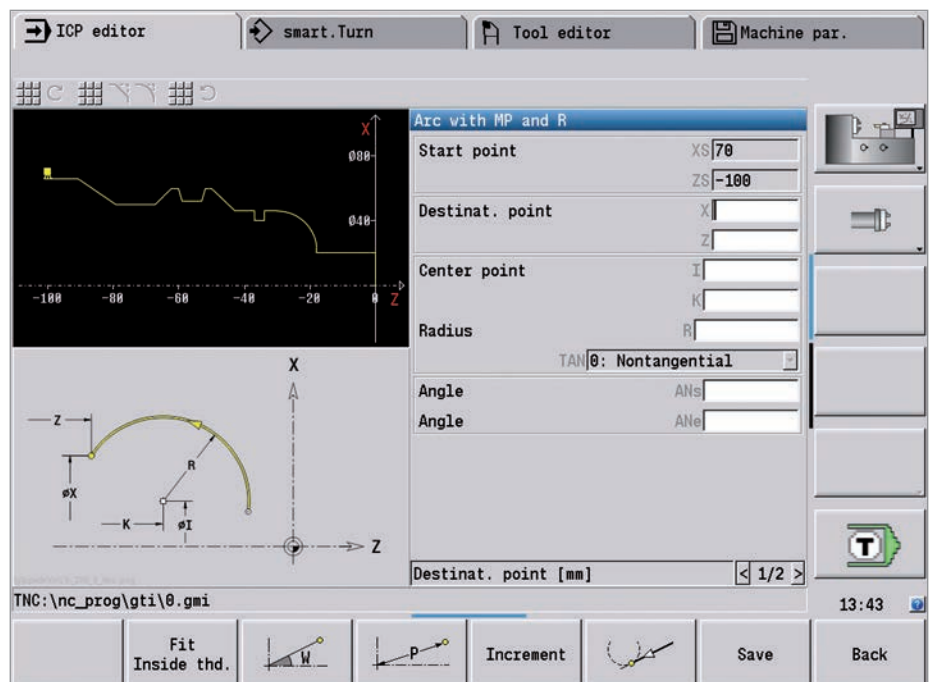
In turning or milling operations, standard contours are defined in the cycle. Complex contours are described with ICP, and this ICP contour is then called in an ICP cycle for roughing, recessing, recess turning, or milling.

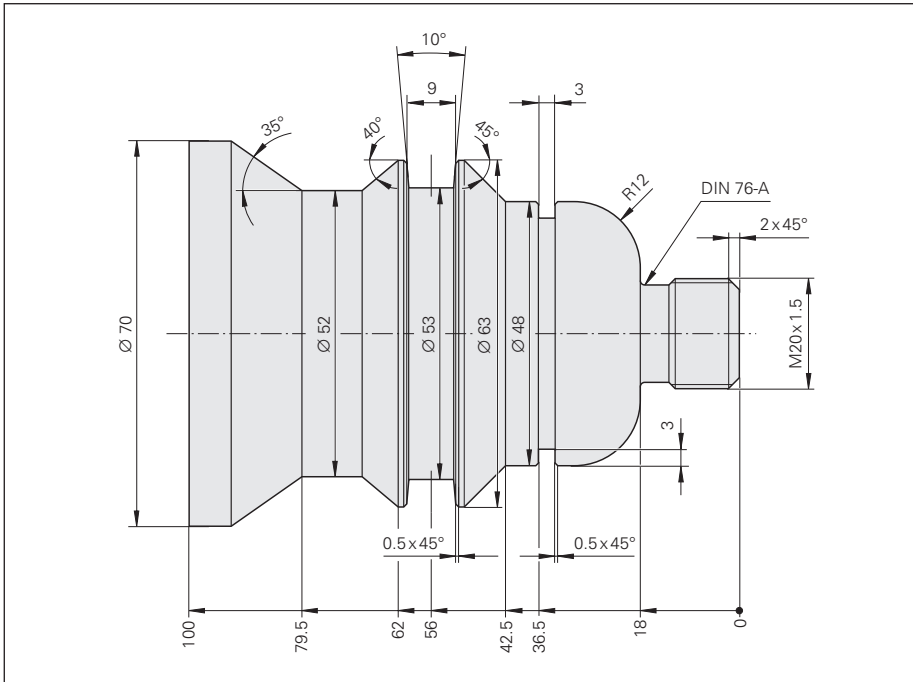
The ICP editor can be called directly during cycle programming.

ICP contours for smart.Turn and DIN PLUS

In smart.Turn, you have various possibilities for describing the contour to be machined. You can describe simple contours right within the unit, or you can use ICP for complex turning or milling contours as well as linear or circular drilling and milling patterns. The contour defined with ICP is incorporated into the smart.Turn program. Within the unit, you enter a reference to the contour section to be machined.

If you are working in DIN PLUS mode, you can likewise use ICP to describe the turning and milling contours, as well as the linear and circular patterns. In the contour-related cycles, you enter a reference to the contour section to be machined.

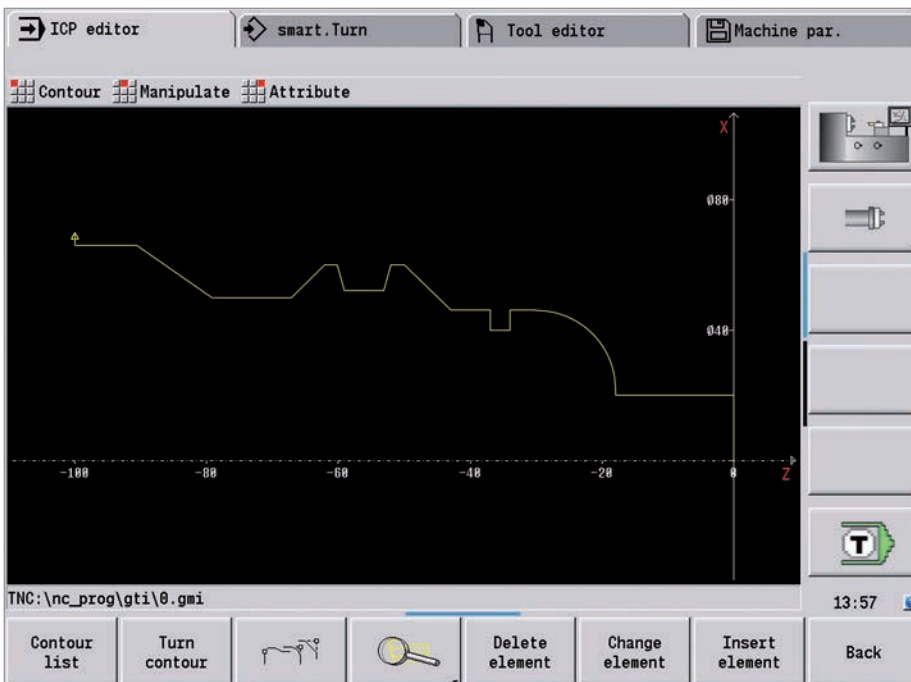




DXF import of contours (option)

Why should you painstakingly enter contour elements if the data already exists in the CAD system? ICP makes it possible to import contours in DXF format directly into the MANUALplus 620. Not only do you save time that would otherwise be spent on programming and testing, but you also ensure that the finished contour corresponds exactly to the design engineer's specification. DXF contours can describe workpiece blanks, finished parts, contour trains, and milling contours. They must be present as two-dimensional elements in a separate layer (i.e., without dimension lines, wrap-around edges, etc.).

First, you download the DXF file to the MANUALplus 620 over the network or through a USB flash drive. Since the DXF format is fundamentally different from the ICP format, the contour is converted from DXF to the ICP format during the import. This contour is then treated as a normal ICP contour, and is available for smart.Turn, DIN PLUS, or cycle programming.



Realistic testing before machining

– Graphic simulation

Timely detection of errors is very important for the production or repair of single parts. With its graphic simulation feature, the MANUALplus 620 supports you in checking the program for errors—with exactness and with the real dimensions of the contour and cutting edge.

Graphic simulation

Before actual machining, you can use the graphic simulation to inspect

- the machining sequence,
- the proportioning of cuts,
- the finished contour,
- the approach and departure behavior.

In the graphic simulation, you can display the cutting edge. You can view the cutting-edge radius, width, and position rendered true to scale. This helps you recognize machining details or collision risks ahead of time.

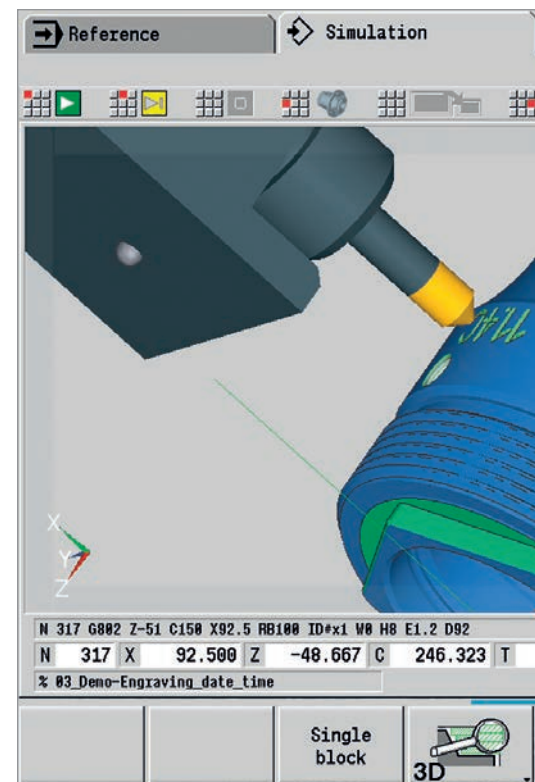
Wire-frame or cutting-path graphics, machining simulation

The MANUALplus 620 supports various views of the tool paths and machining process. You can choose the type of verification best suited to the tool or machining process used.

The **wire-frame graphics** is particularly convenient if you only need a quick overview of the proportioning of cuts. The wire-frame graphics illustrate the path of the theoretical cutting point.

More accurate contour verification is provided by the **cutting-path graphics**, which account for the exact geometry of the tool tip. You immediately see whether material was left behind, the contour is damaged, or the overlaps are too large. The cutting-path graphics are particularly helpful for recessing, boring, drilling, or milling operations in which the tool shape has a decisive influence on the final result.

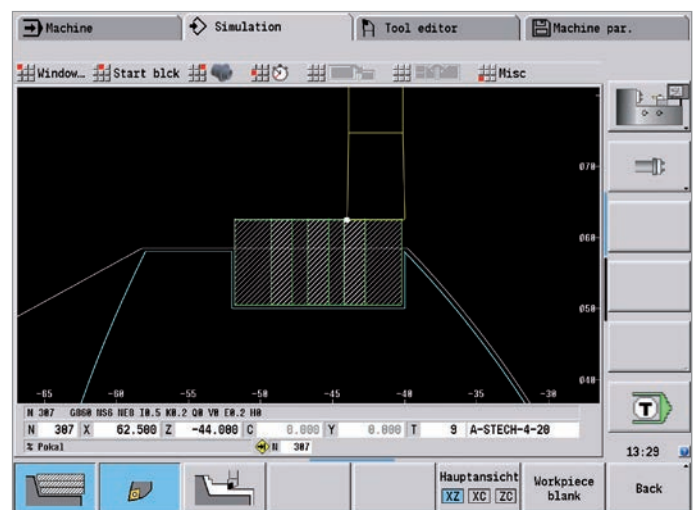
The **machining simulation** (material removal graphic) realistically demonstrates the cutting process. The workpiece blank is shown as a filled-in surface. The MANUALplus 620 simulates every tool movement at its programmed cutting speed and removes the “machined” material.



Finished part in a 3-D graphic



Wire-frame graphics



Cutting-path graphics



Configuring the views

If your lathe is equipped with driven tools and a positionable spindle, a C axis, or a Y axis, then the MANUALplus 620 also simulates machining operations on the front face and lateral surface, or on the XY and YZ planes. You can select the combination of windows best suited for the job. This gives you everything you need for a precise inspection of your boring, drilling, and milling operations.

The MANUALplus 620 depicts C-axis machining on the lateral surface on an "unrolled" plane surface.

3-D simulation

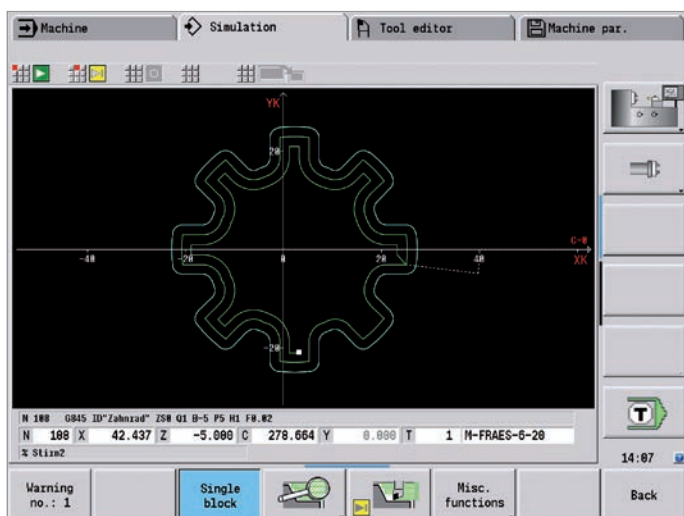
With the high-resolution and intricately realistic 3-D simulation, you can precisely evaluate the production results from turning, drilling, or milling processes prior to actual machining. Milling and turning operations are shown in different colors.

The view, which is freely rotatable about all axes, permits visual inspection of the workpiece blank and finished part from all angles. Thanks to intuitive mouse and keyboard operation, you can navigate and zoom onto every programmed detail—including, of course, with C-axis contours on the cylindrical surface or front face, and with Y-axis contours in the tilted plane. In this way, the 3-D simulation makes it possible for you to detect even the smallest errors prior to machining.

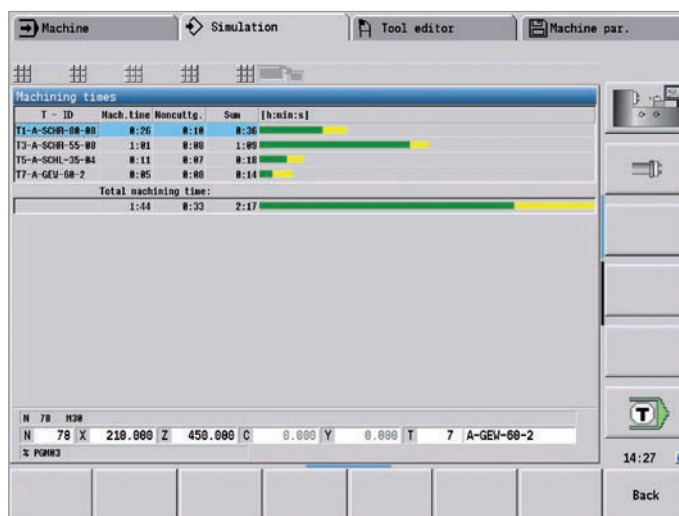
Calculating the machining time

If your customer needs an offer in a hurry, and you require exact information within very little time, the MANUALplus 620 is a valuable aid with its machining time calculator. During simulation of the smart.Turn or DIN PLUS program, the MANUALplus 620 calculates the time required per piece for the programmed machining work.

Along with the total time, the table displays the machining time and idle time of each cycle or each tool used. And this can help with more than just cost calculations, since you can also tell at a glance whether there are ways to further optimize the machining process.



Machining the front face



Calculation of machining time

Expandable for complex tasks

– Full-surface machining including the C axis and Y axis (option)

The MANUALplus 620 provides a solution for any machining task and any machine configuration: it performs complex machining tasks with a C or Y axis. It also controls full-surface cutting on dual-spindle machines.

Likewise, for C-axis, Y-axis, and full-surface machining, you can select from the DIN PLUS, smart.Turn or Teach-in programming modes.

C axis or positionable spindle*

For more complex tasks, the MANUALplus 620 can be expanded to additionally support a C axis or positionable spindle and a driven tool. The driven tool makes it possible to drill off-center and to tap holes while the spindle is at rest. The C axis or positionable spindle permits boring, drilling, and milling operations on the front face and lateral surface of the workpiece. For the programming and inspection of these machining sections, the workpiece is shown in the side and front face views. The Y axis is supported in the smart.Turn and DIN programming modes.

Y axis*

With the Y-axis option of the MANUALplus 620, you can machine slots or pockets with plane floors and perpendicular slot walls. By defining the spindle angle, you can determine the position of the milling contours on the workpiece. For programming and verification of these machining sections, the workpiece is shown in the side and front face views. The Y axis is supported in the smart.Turn and DIN programming modes.

Dual-spindle option

For full-surface lathes, the MANUALplus 620 provides the following control features:

- Counter spindle with second C axis
- Movable tailstock (W axis)

These features are complemented by additional functions such as coordinate transformation, spindle synchronization, and traversing to a stop surface.

Coordinate transformation

Contours of workpiece blanks and finished parts can be mirrored about the X axis or shifted in relation to the workpiece datum.

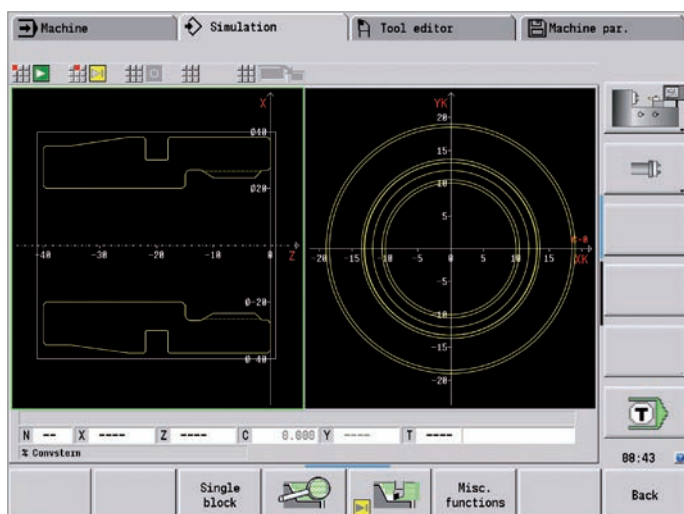
Spindle synchronization

Counter spindles are electronically coupled and rotate synchronously. This makes it possible to transfer the workpiece from one spindle to the other while they are rotating, thereby saving time otherwise spent braking and starting the spindles. The MANUALplus 620 detects any angular offset and compensates for it during subsequent milling on the counter spindle.

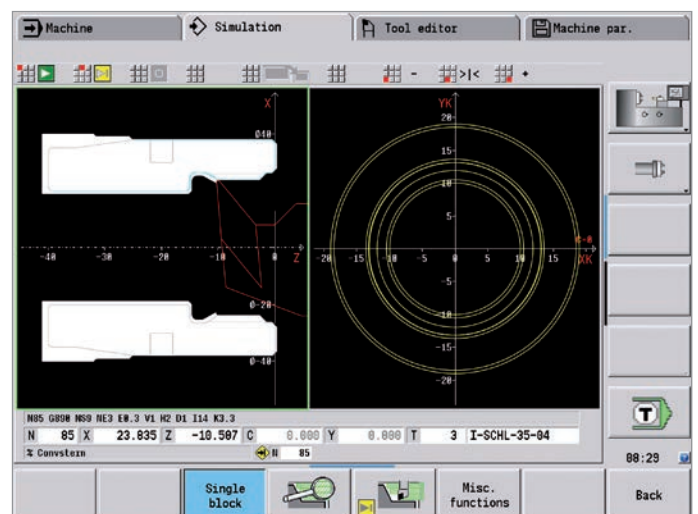
Traversing to a fixed stop

To ensure that the workpiece is firmly pressed onto the opposite spindle surface, the control monitors the nominal and actual positions while the longitudinal axis is in motion and thereby detects the fixed stop. The MANUALplus 620 monitors the motor torque and uses it to achieve the programmed contact force.

* The machine and MANUALplus 620 must be adapted to this function by the machine tool builder.



Graphic contour programming for C-axis machining (boring, drilling, and milling)



First clamping

Eccentric turning and non-circular turning*

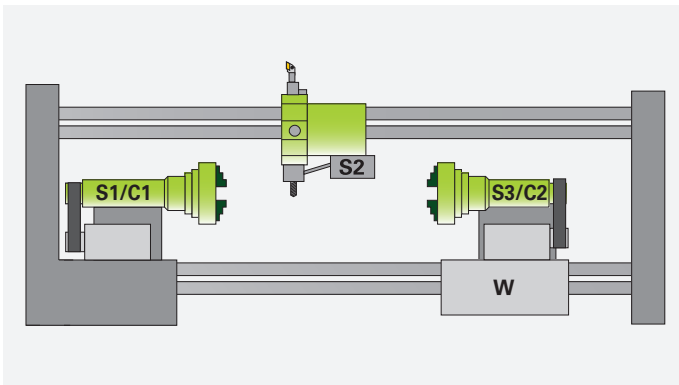
The MANUALplus 620 offers convenient cycles for eccentric turning and for the manufacture of oval and polygonal parts. In this case—besides the actual contour machining—additional traversing movements of the X and Y axes are superimposed in synchronism with the rotational movement of the spindle. The manufacture of cams and non-circular parts is possible without additional machine elements.

* The machine and the control must be adapted to this function by the machine tool builder.

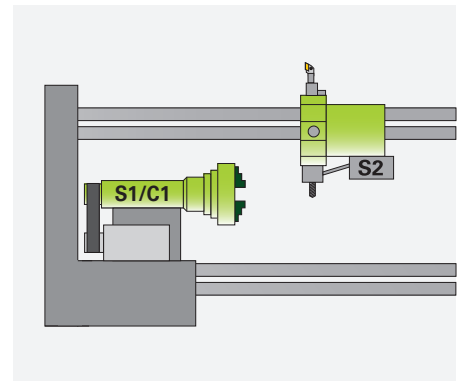
Hobbing of external gears

Cycle G808 HOBGING allows you to machine external cylindrical gears or helical gears in any angle. During hobbing, the rotation of the tool and the spindle are synchronized. In addition, the gear hob moves along the workpiece in axial direction. Cycle G808

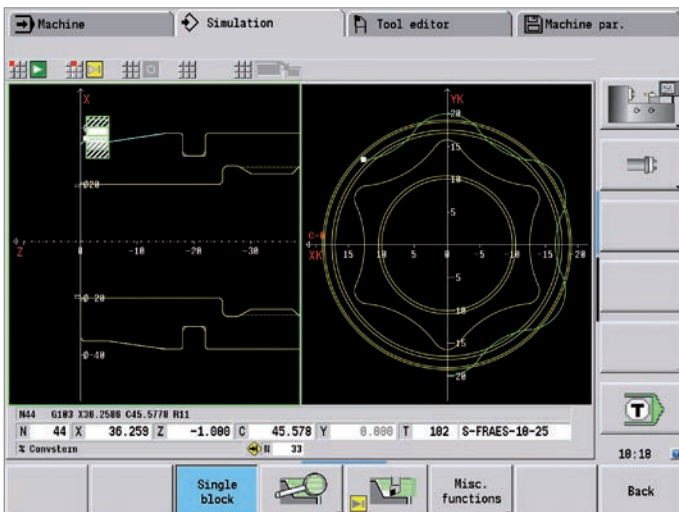
automatically controls these complex movements and allows you to easily and practically enter all of the relevant values. You can use gear parameters directly from your drawing, and the cycle will then use them to calculate the sequence of motion.



Full-surface machining: counter spindle (S3) with C axis (C2) on secondary axis (W) and driven tool (S2)



Spindle 1 (S1) with C axis (C1) and driven tool (S2)



Rear-face machining on the counter spindle after automatic workpiece transfer



Expandable for complex tasks

– Turning, drilling, and milling in one setup (option)

You can use the MANUALplus 620* to turn, drill, and mill your workpiece on the front face or lateral surface in a single setup. In addition, the control offers numerous functions and practical cycles.

* Optional, the machine and MANUALplus 620 must be adapted to this function by the machine tool builder.

Drilling, deep-hole drilling, tapping

The MANUALplus 620 drills, pecks, and taps individual holes with the C or Y axis. Using parameters, you can easily program infeed reductions for spot drilling and for the drilling of through-holes.

Drilling and milling patterns

If holes, slots, or ICP milling cycles are located at regular intervals along a straight line or circular arc, the MANUALplus 620 greatly simplifies your work: you can create these patterns on the front face or lateral surface with just a few key strokes.

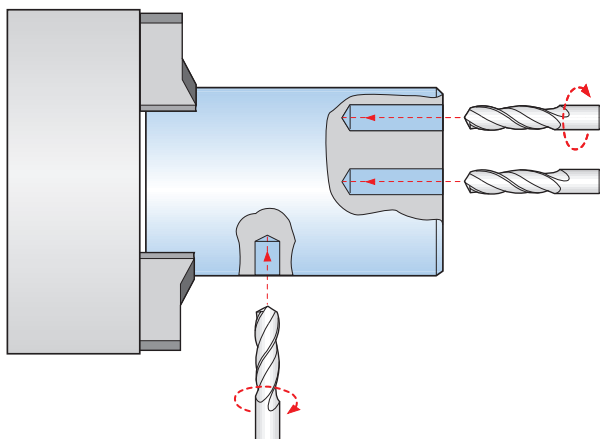
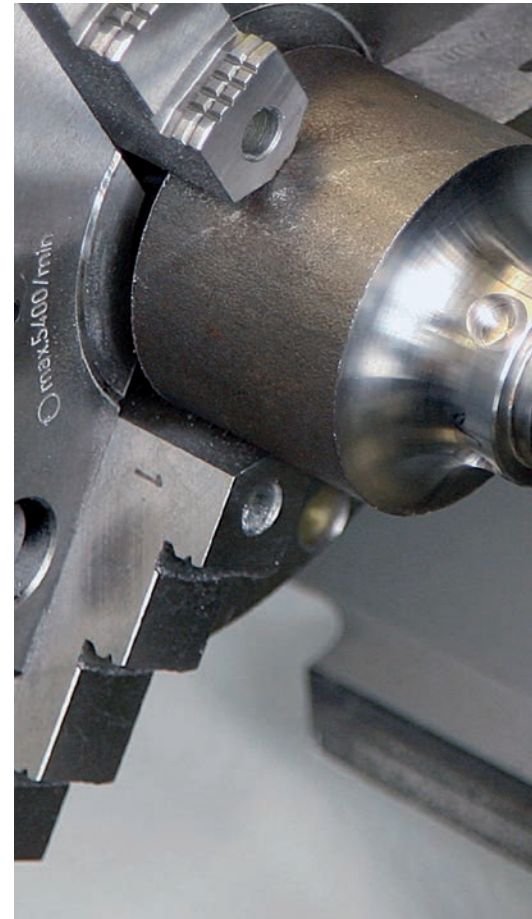
Thread milling

On lathes equipped with a C or Y axis, you can take advantage of thread milling because the MANUALplus 620 supports special thread-milling tools.

Milling slots and simple figures

Slot milling is very simple with the MANUALplus 620. You define the position and depth of the slot as well as the cutting values, and the milling cycles will take care of the rest automatically.

For simple contours such as circles, rectangles, and equilateral polygons, defining the figure and position likewise requires only a few keystrokes.



Drilling or tapping



Drilling



Deep-hole drilling

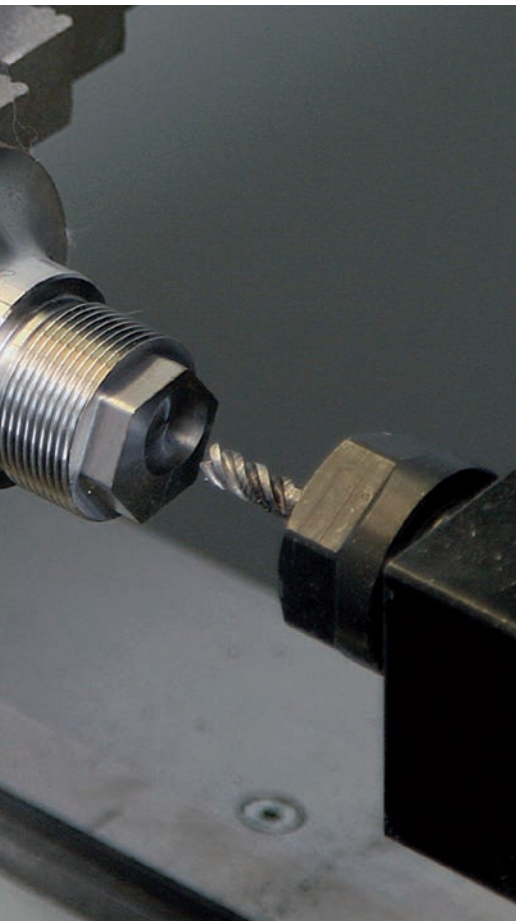


Tapping



Thread milling

Cycles for drilling



Contour and pocket milling

The milling cycles of the MANUALplus 620 support both contour and pocket milling. You determine all of the essential details, such as the machining direction, milling direction, approach and departure behavior, infeeds, etc. The MANUALplus 620 automatically compensates for the tool radius. You can mill the pocket in two stages—first roughing, and then finishing. The result is high accuracy and good surface quality.

In smart.Turn and DIN programming, the MANUALplus 620 supports various infeed strategies. You can choose between direct, reciprocating, or helical infeed, or infeed at the predrilling position.

Face milling

The face milling cycle machines individual surfaces, equilateral polygons, or a circle—even off-center.

Helical slot milling

The helical slot milling cycle is useful for machining lubrication grooves. You specify all of the important parameters such as pitch and milling at multiple infeeds, etc.

Engraving cycles

Do you want to inscribe your workpieces? That's no problem with the MANUALplus 620. The smart.Turn engraving units require only a few parameters in order to engrave characters of any size on the front face or lateral surface, or on the XY or YZ plane.

On the front face of the workpiece, you can arrange the characters along a line or an arc. On the lateral surface, and when engraving with the Y axis, you can define the angle at which the characters are to be arranged. With the time function, you can engrave the current time and date on your workpiece.

Of course, the engraving cycles are also available as DIN PLUS cycles.

Deburring

The MANUALplus 620 supports special units or DIN PLUS cycles for deburring. You enjoy the benefit of being able to program this operation with only a few parameters.

Trochoidal milling

The trochoidal milling method allows you to efficiently machine pockets in different shapes (circle, rectangle, and polygon), as well as any desired slot. With this technique, roughing is performed with circular movements that are superimposed onto a linear forward motion. Trochoidal milling enables high removal rates and reduces chatter.



Slot milling



Figure milling (circles, rectangles, equilateral polygons)



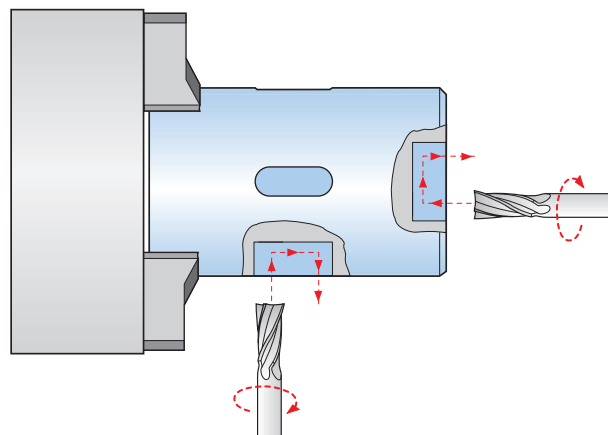
ICP contour milling



Face milling (single surfaces, flat surface, polygon)



Helical slot milling



Front face and lateral-surface milling

Expandable for complex tasks

– Working in a tilted plane with the B axis (option)

The B axis makes it possible to perform boring, drilling, and milling operations in tilted planes. At first glance, the programming work for such operations seems very complex and compute-intensive. But with the MANUALplus 620, you simply tilt the coordinate system to the required position and program the machining operation as usual in the main plane. The machine will then execute the machining operation in the tilted working plane.

The B axis also provides benefits in turning operations. By tilting the B axis and rotating the tool, you can bring it into positions that enable you to use a single tool to perform machining in the longitudinal and transverse directions on the main and counter spindles. You can thereby reduce the number of tools needed as well as forgo certain tool changes.

Programming

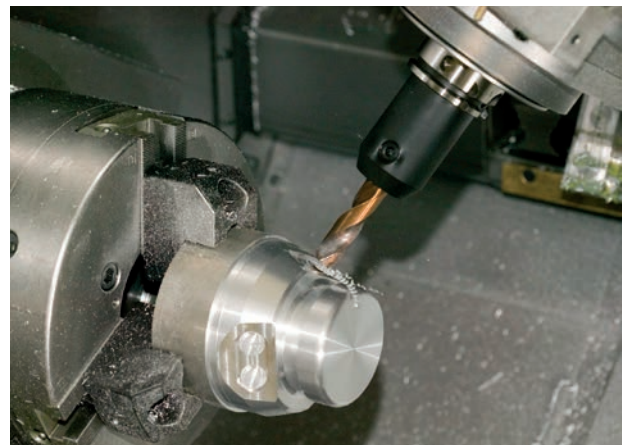
The usual separation of the contour description from the machining operation on the MANUALplus 620 also applies to boring, drilling, and milling operations in a tilted plane.

First, you rotate and shift the coordinate system so that it lies in the tilted plane. Then you describe the hole pattern or the milling contour as you would in the YZ plane. The hole pattern and figure definitions of the MANUALplus 620 are available. This means that, for linear or circular patterns, as well as for simple figures (circles, rectangles, equilateral polygons, etc.), only a few more entries are needed in order to describe the pattern or figure in the tilted plane.

Simulation

In the Side View window, the simulation shows the hole pattern and milling contour perpendicularly to the tilted plane, and therefore without distortion. This ensures the easy verification of programmed hole patterns and milling contours. You can also verify the tool movements in the Side View window. If you want to check the machining operation in the tilted plane with respect to the turning contour or the front face, then add the Lathe Window or Front Window. In the position display (below the simulation window), the MANUALplus 620 displays the angle of the tilted plane and the tilt angle of the B axis. Do you want to view the active coordinate system? No problem—with a simple keystroke, the MANUALplus 620 shows the current datum and the direction of the active coordinate system.

* The machine and MANUALplus 620 must be adapted to this function



Machining in the tilted plane

Flexible use of tools*

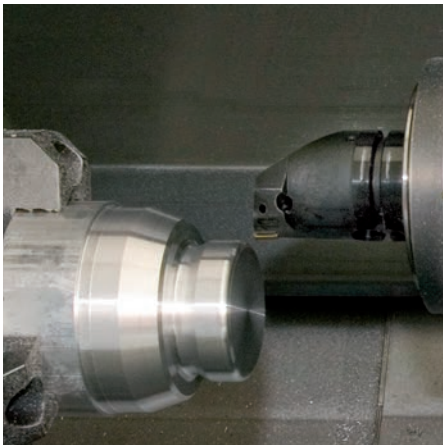
If your machine is equipped with a B axis, then you can use your tools much more efficiently than before. Whereas on conventional lathes you need four different tools for longitudinal and transverse turning on main and counter spindles, you can perform these tasks with just a single tool when you employ a B axis.

You simply tilt the B axis and rotate the tool to the normal position or to the position for machining from behind the workpiece—whichever is required for longitudinal or transverse turning on the main or counter spindle. All you need is a single call: the MANUALplus 620 calculates the tool lengths, the tool angle, and the other tool data for you.

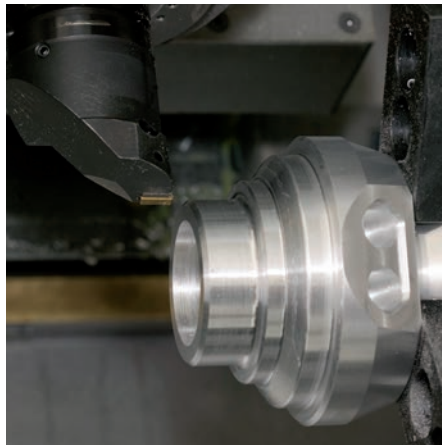
Tool-use flexibility is increased significantly when multiple tools are mounted in one holder. For example, with a roughing, finishing, and recessing tool you can perform essential portions of turning and recessing operations on a main spindle and counter spindle—without changing the tool. And programming is very easy. You simply indicate which tooth of the tool to use and then define the tilting angle and the tool position. This is sufficient because the MANUALplus 620 already contains the rest position and the data of each tool tooth in its database.

This type of flexibility reduces the number of your tools, and you save on machining time by reducing the number of tool changes.

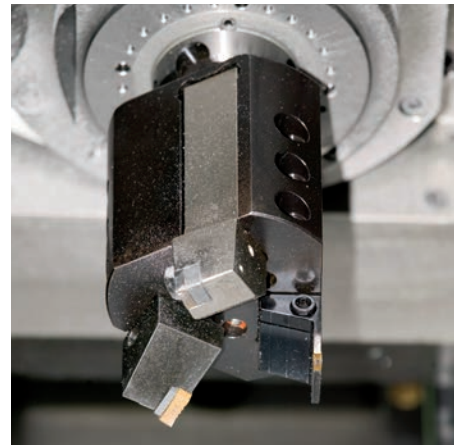
*The machine and MANUALplus 620 must be adapted to this function



Facing and ...



... longitudinal turning with the same tool ...



... and with multiple tools on one holder.

Fast availability of tool data and cutting data

– The tool database and technology database of the MANUALplus 620

Tool database

The standard tool database of the MANUALplus 620 stores 250 tools. The tool database can be expanded to contain 999 tools (option).

The MANUALplus 620 differentiates between various types of turning, drilling, and milling tools. The data that are input vary depending on the tool type. In this way, you can be sure that all of the important parameters are specified despite reduced data input.

The tool data are entered through dialog prompts in which you enter parameters such as the cutting-edge radius, tool angle, point angle, cutting material, and tool description. The data to be input is explained by context-sensitive help graphics.

Tool list

The MANUALplus 620 shows all of the tools in a straightforward tool list. Various sorting criteria help you to quickly find the desired tool.

This list not only gives you a good overview of your tools—it is also the basis for transferring tool data during manual machining and during the creation of NC programs.

Wear compensation

The MANUALplus 620 offers a simple means of compensating for tool wear in both the X and the Z axes. You enter the compensation values during or after workpiece machining.

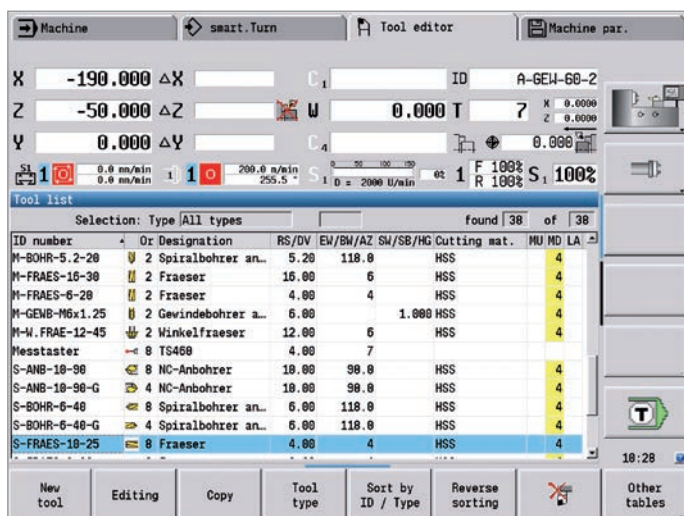
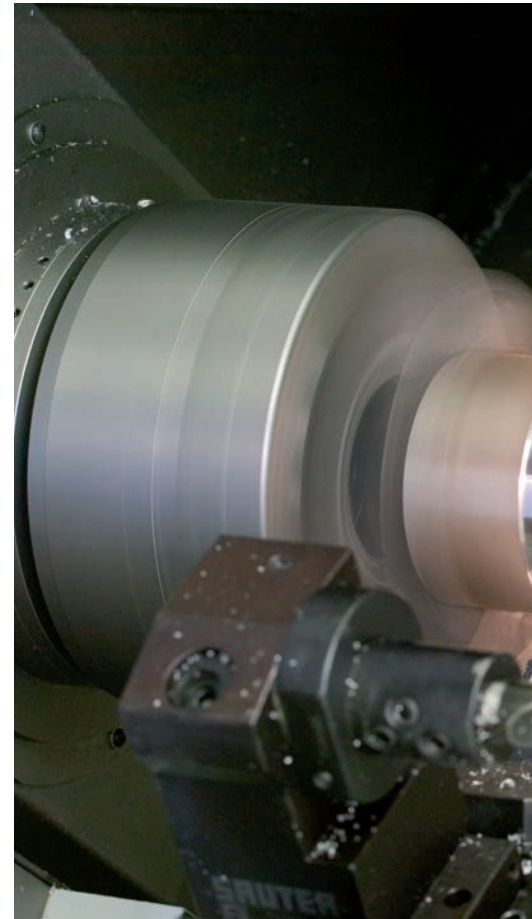
Tool measurement

The MANUALplus 620 offers various possibilities for the measurement of tools directly on the machine:

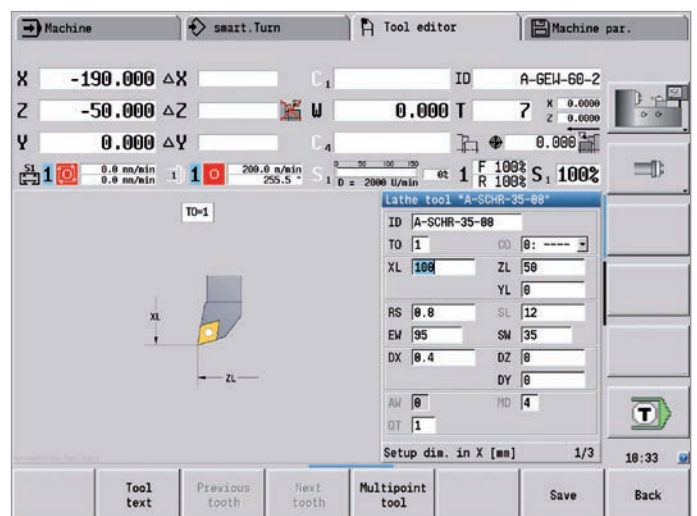
- By touching the workpiece
- With the aid of an optical gauge* (option): the tool is manually traversed to the crosshairs of the optical measuring device, and the value is saved with a keystroke
- With a tool touch probe* (option): the tool moves in the direction of measurement. The tool setting dimension is ascertained and adopted when the tool touch probe releases a trigger signal, e.g. the TT 160 touch trigger probe with cuboid probe contact.

You can determine the tool data with great ease, reliability, and precision during tool measurement with an optical gauge or tool touch probe.

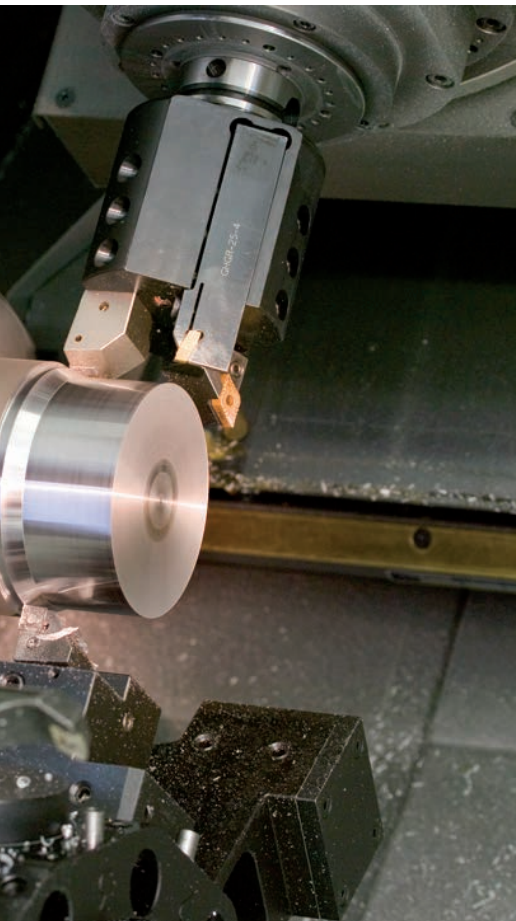
* The machine and MANUALplus 620 must be adapted to this function by the machine tool builder.



Tool management in the tool list



Tool editor



Turret assignment

If your lathe is equipped with a tool turret, you can view the programmed turret assignment at any time. The MANUALplus 620 displays all of the important tool parameters.

If you want to change the tool assignment or the tools in the turret, you can additionally display the tool database entries in the lower window. Now you need only select the desired turret pocket and choose the correct tool from the database. At the stroke of a key, you can apply the tool data to the turret assignment entry.

Tool life monitoring (option)

With smart.Turn and DIN PLUS programs, in addition to the simple tool life monitoring feature you can also use the "Tool Life Monitoring with Sister Tool" option. The MANUALplus 620 then automatically inserts a replacement tool as soon as the active tool is worn out. The MANUALplus 620 does not stop the program run until the last tool of the tool change sequence is worn out.

Technology data (option)

With the MANUALplus 620, you need to enter the cutting data only once. The control saves the data in accordance with the criteria of the workpiece material, cutting material, and machining mode. Thanks to this three-dimensional table, the control always knows the correct feed rate and the correct cutting speed.

The MANUALplus 620 determines the machining mode based on the cycle or the unit. The cutting material is entered during the tool description. Now you need only define the workpiece material at the beginning of the cycle program or the smart.Turn program, and the MANUALplus 620 will provide the correct suggested values for your machining operation. You can use the suggested cutting parameters or adjust them if required.

In its standard version, you can store the cutting data for nine workpiece-material / tool-material combinations in the technology database of the MANUALplus 620. It can be expanded to a total of 62 combinations (option). Each workpiece-material / tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes.

T #	ID number	Or Designation	RS/DV	EM/BM/AZ	SW/SB/HG	Cutting mat.	MU MD LA
1	A-SCHR-88-08	1 Schruppen Aussen	0.80	95.0	80.0	Hartmetall	
2	A-SCHR-55-08	1 Schruppen Aussen	0.80	95.0	55.0	Hartmetall	
3							
4	A-SCHL-35-04	1 Schlichten Auss...	0.40	95.0	35.0	Hartmetall	
5	S-FRAES-10-25	8 Fraezer	4.00		4	HSS	
6	A-GEW-60-2	1 Gewinde Aussen	0.10	60.0	60.0	Hartmetall	
7	S-FRAES-8-20	8 Fraezer	8.00		4	HSS	
8	A-STECH-4-20	1 Stechen Aussen	1.00		4.00	Hartmetall	
9	M-FRAES-16-30	2 Fraezer	16.00		6	HSS	
10							
11							

Tool selection for turret assignment

TASK	CUTMAT	CSP	FDR	AFDR	DEP	OCOL
Predrilling	Hartmetall	98	0.25	0	0	0
Roughing	Hartmetall	200	0.35	0.25	5	0
Finishing	Hartmetall	220	0.15	0.1	0	0
Thread cutting	Hartmetall	120	0	0	0	0
Contour recessg.	Hartmetall	160	0.25	0.2	0	0
Parting	Hartmetall	140	0.25	0.18	0	0
Centering	Hartmetall	0	0	0	0	0
Drilling	Hartmetall	80	0.28	0	0	0
Countersinking	Hartmetall	0	0	0	0	0
Reaming	Hartmetall	0	0	0	0	0
Tapping	Hartmetall	60	0	0	0	0
Milling	Hartmetall	64	0.05	0.02	5	0
Finish milling	Hartmetall	74	0.03	0.01	5	0
Deburring	Hartmetall	0	0	0	0	0
Engraving	Hartmetall	0	0	0	0	0
Recess turning	Hartmetall	160	0.5	0.3	5	0

Input of cutting values in the technology editor

Reliable machining

– Intelligent supervision with Load Monitoring (option)

Load Monitoring – detecting tool wear and breakage during machining*

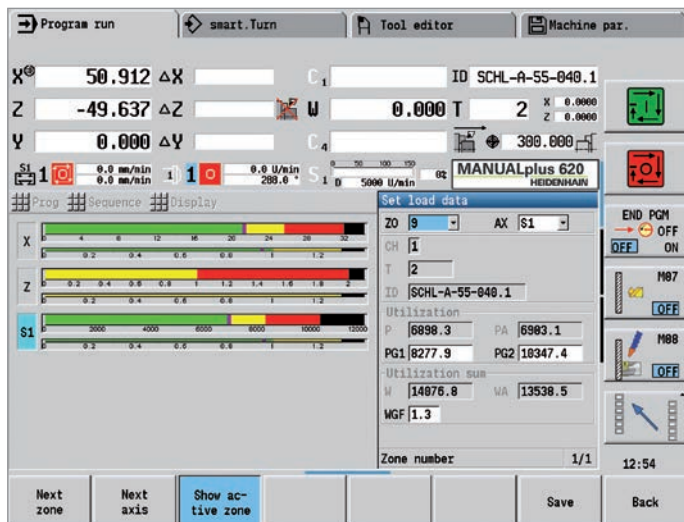
Load Monitoring tracks the machine's spindle and motor loads while comparing them with the load values of a reference operation. The MANUALplus 620 can graphically display the load values in a separate window.

You can set two limit values that trigger different error reactions. After the first limit value is exceeded, the current tool is flagged as worn out, and the control automatically exchanges it on the next tool call with a previously defined replacement tool. After the second limit value is exceeded, the MANUALplus 620 assumes that there is an impermissible load (e.g., tool breakage) and stops the machining process. This improves process reliability during machining, particularly during unmanned shifts.

Batch mode

The batch mode function allows you to run multiple main programs automatically. You can also define the part programs and the respective number of parts in a program list. The MANUALplus 620 can then execute this list on its own and without you having to select and start programs in between. Batch mode is particularly suitable for different operations with the same raw material supplied by bar loaders or robots.

* The machine and MANUALplus 620 must be adapted to this function by the machine tool builder.



Graphic display of the load values



Open for communication

– Rapid availability of all information

Do you have questions about a programming step, but your User's Manual is not at hand? No problem: the MANUALplus 620 numerical control and DataPilot MP 620 programming station feature TURNguide, a convenient help system that can display the user documentation in a separate window.

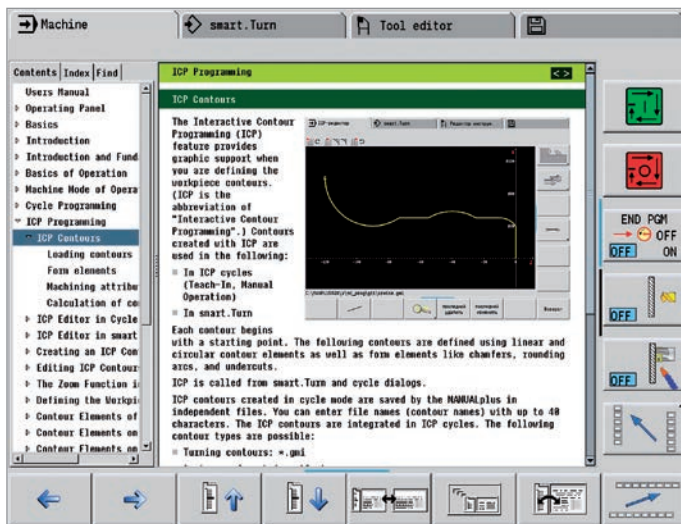
You can activate TURNguide by simply pressing the Info soft key.

Typically, TURNguide directly displays the information in the correct context (context-sensitive help). This means that you immediately receive the information that you need. This function is particularly helpful for the programming of cycles. The respective operating method is explained as soon as you press the Info soft key in the opened dialog.

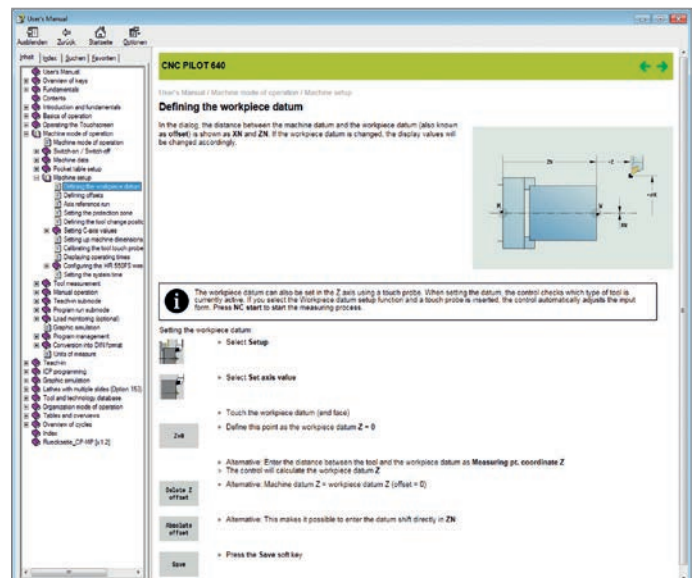
You can download the documentation in the desired language from the HEIDENHAIN homepage to the corresponding language directory on your control.

The following manuals are available in the help system:

- MANUALplus 620 User's Manual
- smart.Turn and DIN Programming User's Manual
- DataPilot MP 620 User's Manual (included only in the programming station)



TURNguide integrated into the control, e.g., on the MANUALplus 620 ...



... or at the programming station.

Open for communication

– Uniformly digital job management with Connected Machining



The networked MANUALplus 620

You can integrate the MANUALplus 620 with the functions of **Connected Machining** into your corporate network and, via the control, connect the shop to PCs, programming stations, and other data storage devices in the following departments:

- Design
- Programming
- Simulation
- Production planning
- Production

Even in its standard version, the MANUALplus 620 features a Gigabit Ethernet interface of the latest generation. The MANUALplus 620 communicates with NFS servers and Windows networks in TCP/IP protocol without requiring additional software. The fast data transfer at rates of up to 1000 Mbps ensures very short transfer times. The MANUALplus 620 therefore meets the ideal technological requirements for **Connected Machining**, the networking of the control in the workshop with all production-related areas in your company.

Standard range of functions

For you to be able to use the data that you transfer to the MANUALplus 620 over a standard network connection, the MANUALplus 620 offers attractive applications, even as part of its standard range of functions. A CAD viewer, PDF viewer, or the web browser Mozilla Firefox make the simplest form of **Connected Machining** possible: access to manufacturing process data right at the control.

The operation of web-based documentation software or ERP systems is just as much possible in this case as is the access to your e-mail inbox. The following further file formats can also be opened directly on the MANUALplus 620:

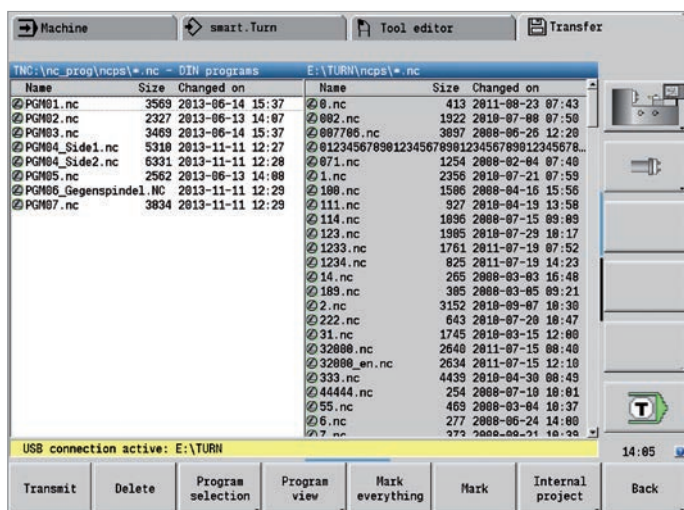
- Text files ending with .txt or .ini
- Graphic files ending with .gif, .bmp, .jpg, or .png
- Table files ending with .xls or .csv
- HTML files

Data transfer with Connected Machining

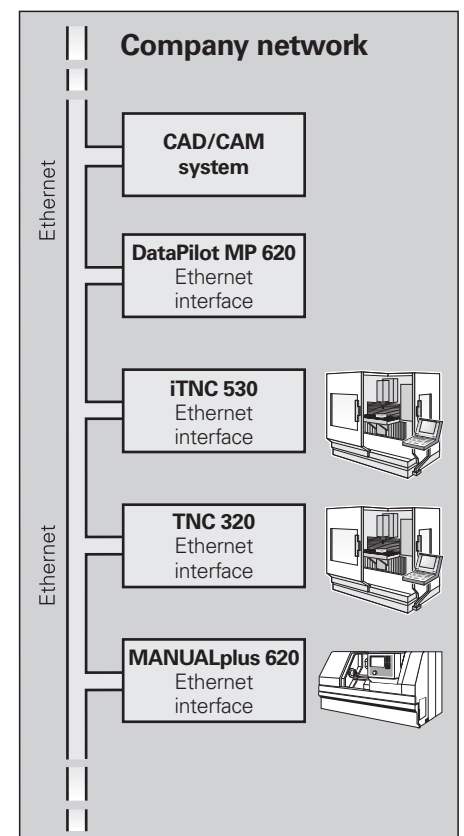
An additional solution for uniformly digital job management as part of **Connected Machining** is the free **TNCremo** software for PCs. With it, you can

- transfer remotely stored part programs and tool tables in both directions, even via Ethernet, and
- make backups.

With the powerful **TNCremoPlus** PC software, you can also transfer the screen contents of the control to your PC using the live-screen function.



Data transfer on the control



USB interface

Of course, for data transfer you can also use one of the USB ports. By using USB removable memory media (such as flash drives), you can quickly and easily exchange DXF contours, ICP contour descriptions, NC programs, tool parameters, etc., between systems that are not linked to each other.

All programs at a glance

After you have entered the path of the partner terminal, your own MANUALplus 620 programs will appear on the left side of the screen, with your partner's programs appearing on the right. Now select the programs that you want to transfer and press the send or receive button. The data is transferred very quickly and reliably.

Transferring programs

Data transfer is easiest and most convenient when you integrate the systems into your company network.

When transferring NC programs, the MANUALplus 620 even takes into account the files related to the cycle program, smart.Turn program, or DIN PLUS program, such as contour descriptions, DIN macros, or subprograms.

Exchanging tool data

You can also transfer already acquired tool data. This is important not only for data backup but is also an advantage when you use the DataPilot PC programming station. The benefits are that there is no redundant data acquisition and that your files are always up to date.

Detailed data for the optimal organization of the production process

HEIDENHAIN DNC¹⁾ enables, among other things, the connection of MANUALplus 620 controls to inventory management systems and production activity control systems. Automatic feedback messages about active production processes, for example, can be set up over this interface.

¹⁾ The machine must be adapted to this function by the machine tool builder.

One aspect of Connected Machining is the **StateMonitor²⁾** PC software, which gives you access to the status of the current machining operations. You can use the software with any device that has a web browser. This allows you to use StateMonitor not only on your control or PC but also on your smartphone or tablet. In a well-designed display, you can quickly gain an overview of the current machine status or see whether any machine messages are pending. This allows you to react immediately and take appropriate action. You can also easily configure StateMonitor to send you an e-mail for specific events, such as at the end of a program, for a machine stop, or for a servicing message.

²⁾ Option 18 required



Open for communication

–The DataPilot MP 620 programming station

DataPilot MP 620 is the PC programming station for the MANUALplus 620 and the organizational system for the shop and design office.

That is why DataPilot MP 620 is the ideal supplement to the MANUALplus 620 for program creation and archiving, and for basic and advanced training.

Creating programs

Programming, testing, and optimization of smart.Turn or DIN PLUS programs with the programming station significantly reduce machine idle time. This does not even require an adjustment in thinking, because, with DataPilot, programming and testing are performed in the same manner as on the lathe. DataPilot is based on the same software that is in the control. This also

gives you the assurance that a program created with DataPilot can be run on the machine immediately.

Archiving programs

Even though the MANUALplus 620 features a large memory capacity, you should also back up your programs on an external system. The MANUALplus features a USB port and an Ethernet interface. This gives you everything you need to integrate the MANUALplus 620 into your existing network or to connect the DataPilot PC directly to the control.

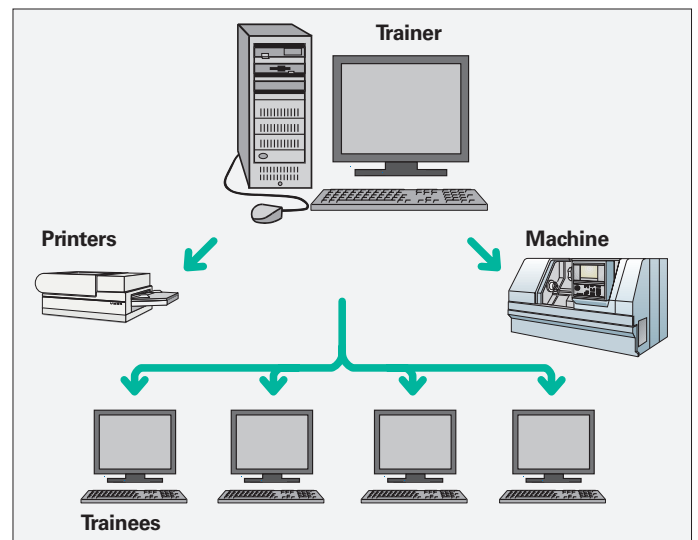
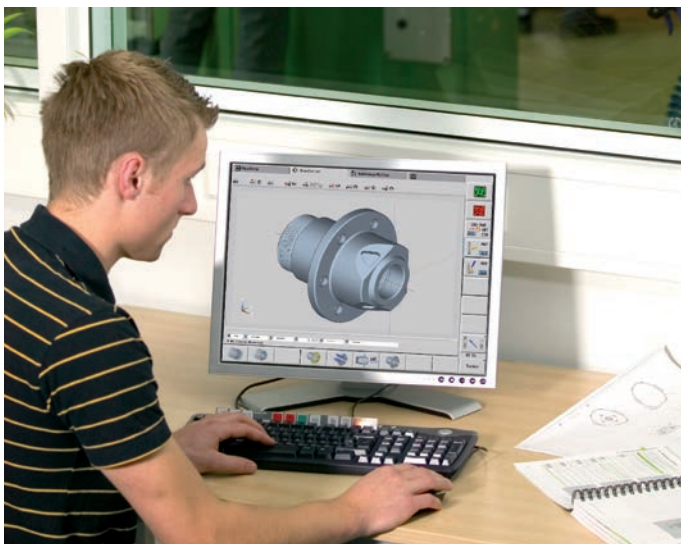
Convenient program transfer functions support both programming and archiving on the DataPilot PC.

Training with DataPilot MP 620

Because DataPilot MP 620 is based on the same software as the MANUALplus 620, it is also ideally suited for training purposes. Programming and program testing on the DataPilot PC are performed in exactly the same way as on the machine. DataPilot even simulates setup functions such as defining the workpiece datum, tool measurement, or running individual cycles or smart.Turn or DINPLUS programs. This gives trainees the confidence they will need for working on the actual machine.

System requirements

The DataPilot software can run on PCs with the Windows 7, 8, 8.1, or 10 (64-bit) operating systems.



Workpiece measurement

- Setup, presetting, and measuring with touch trigger probes

Inspecting workpieces for proper machining and dimensional accuracy

The MANUALplus 620 features measuring cycles for checking the geometry of the machined workpieces. For the measuring cycles, you simply insert a 3-D touch probe from HEIDENHAIN into the turret instead of a tool:

- Check whether all of the machining operations have been properly performed
- Determine infeeds for finishing
- Detect and compensate for tool wear
- Inspect workpiece geometry and sort parts
- Log measured data
- Ascertain the machining error trend

Workpiece touch probes from HEIDENHAIN help you to reduce costs in the shop and in series production: together with the MANUALplus 620's probing cycles, touch probes can automatically perform setup, measuring, and inspection functions.

The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. At that moment the TS generates a trigger signal that, depending on the model, is transmitted to the control either by cable, radio transmission, or an infrared beam.

HEIDENHAIN touch probes* for workpiece measurement are available in various versions. The ball tips, which are made of ruby, are available in different diameters with various stylus lengths.

* The touch probes must be interfaced to the MANUALplus 620 by the machine tool builder.

Touch probes with **cable connection for signal transmission** for machines with manual tool change, as well as for grinding machines and lathes:

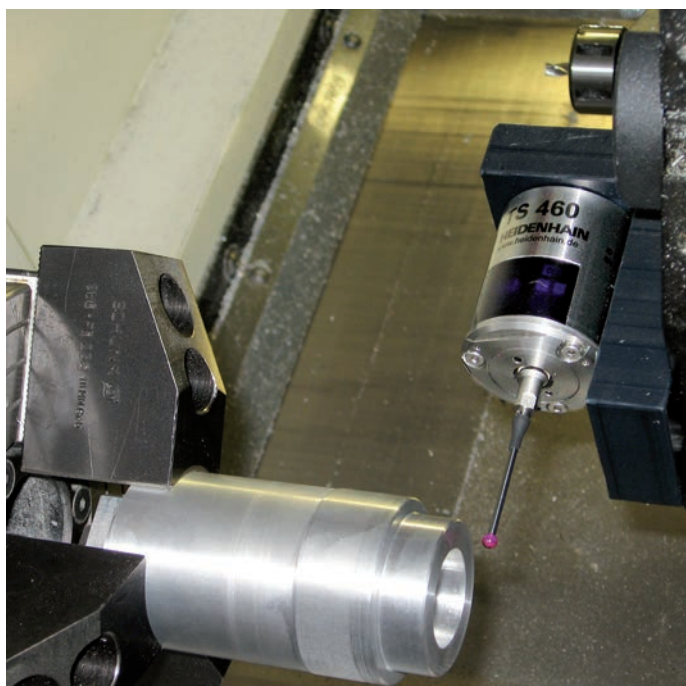
TS 260 – new generation, axial or radial cable connection

TS 248 – like the TS 260, but with reduced deflection forces

Touch probes with **radio or infrared signal transmission** for machines with an automatic tool changer:

TS 460 – new generation standard touch probe for radio and infrared transmission, with compact dimensions

TS 740 – high probing accuracy and repeatability, low probing force, with infrared transmission



Further information:

Detailed descriptions about touch probes are available on the Internet at www.heidenhain.de or in the brochure *Touch Probes for Machine Tools*.

Tool measurement

– Measuring the length, radius, and wear directly in the machine

Exact measurement of the tool dimensions is a decisive factor in ensuring a consistently high level of production quality. The TT tool touch probes from HEIDENHAIN serve this purpose.

TT 160 – new generation, signal transmission to the NC over a cable

TT 460 – new generation, signal transmission over radio and infrared beam to the transceiver

The SE 660 is a common transceiver for tool and workpiece touch probes with radio or infrared transmission.

The TT 160 and TT 460 are 3-D touch trigger probes for the measurement and inspection of tools. The disk-shaped probe contact of the TT is deflected during the physical probing of a tool. During probing, the TT generates a trigger signal that is transmitted to the control for further processing. The trigger signal is generated by means of a wear-free and highly reliable optical sensor.

With their rugged design and high protection rating, these tool touch probes can be installed directly within the machine tool's work envelope, thereby enabling tool measurement directly within the machine. You can thus determine the tool dimensions quickly, easily and, above all, very precisely. The TT tool touch probes from HEIDENHAIN are the ideal addition for improving the efficiency and quality of your production system.



SE 660



TT 160

Further information:

Detailed descriptions about touch probes are available on the Internet at www.heidenhain.de or in the brochure *Touch Probes for Machine Tools*.

Overview

– User functions

User functions	Standard	Option	
Configuration	•	0-6 55+0-6 70+0-6 54+0-6 94+0-6 132+0-6	Basic version: X and Z axis, main spindle Driven tool and auxiliary axes (U, V, W) C axis and driven tool Y axis B axis Parallel axes U, V, W (display function and compensation) Counter spindle
	•		Digital current and speed control
Operating modes			
Manual operation	•		Manual slide movement through axis-direction keys, an intermediate switch, or electronic handwheels
	•	11	Graphical support for entering and running cycles without storage of the machining steps in direct alternation with manual machine operation Thread reworking (thread repair) in a second workpiece setup
Teach-in mode		8	Sequential arrangement of fixed cycles, with each cycle being run immediately after input or graphically simulated and subsequently saved
Program run	•	9 8	Each in Single-Block mode and Full-Sequence mode DIN PLUS programs smart.Turn programs Cycle programs
Setup functions	•	17 17 17	Workpiece datum setting Definition of tool-change position Definition of protection zone Definition of machine dimensions Manual programs Tool measurement by touching the workpiece Tool measurement with a TT tool touch probe Tool measurement with an optical gauge Workpiece measurement with a TS workpiece touch probe
Programming			
Cycle programming		8 8 8 8 8 8 8 8 8 8 8+55 8+55 8+55 8+55 8+55 8 8 8 8+9	Turning cycles for simple or complex contours, and contours described with ICP Contour-parallel turning cycles Recessing cycles for simple or complex contours, and contours described with ICP Repetitions with recessing cycles Recess turning cycles for simple or complex contours, and contours described with ICP Undercut and parting cycles Engraving cycles Threading cycles for single or multi-start longitudinal, taper, or API threads, and threads with variable pitch Cycles for axial and radial drilling, deep drilling, and tapping operations with the C axis Thread milling with the C axis Axial and radial milling cycles for slots, figures, single surfaces, and centric polygon surfaces, and for complex contours described with ICP for machining with the C axis Helical slot milling (multi-start) with the C axis Deburring of ICP contours Linear and circular patterns for boring, drilling, and milling operations with the C axis Context-sensitive help graphics Transfer of cutting values from the technology database Use of DIN macros in cycle programs Conversion of cycle programs to smart.Turn programs

Overview

– User functions

User functions	Standard	Option	
Interactive Contour Programming (ICP)		8/9 8/9 8/9 8/9 8/9 8/9 8/9 8/9 8/9+55 9+70 8/9+55+70+132 8/9+42	Contour definition with linear and circular contour elements Immediate display of entered contour elements Calculation of missing coordinates, intersections, etc. Graphic display of all solutions for selection by the user if more than one solution is possible Chamfers, rounding arcs, and undercuts available as form elements Input of form elements immediately during contour creation or through later superimposition Changes to existing contours can be programmed Machining attributes available for individual contour elements C-axis machining on the front face and lateral surface: <ul style="list-style-type: none"> • Description of individual holes and hole patterns (only in smart.Turn) • Description of figures and figure patterns for milling (only in smart.Turn) • Creation of freely definable milling contours Y-axis machining on the XY and ZY planes (only in smart.Turn): <ul style="list-style-type: none"> • Description of individual holes and hole patterns • Description of figures and figure patterns for milling operations • Creation of freely definable milling contours Programming of the rear face for full-surface machining with the C and Y axes DXF import: importing of contours for turning and milling operations
smart.Turn programming		9 9 9 9 9 9 9 9 9+55/70 9+55/70 9+55+135 9+55 9+55/70 9 9 9	The basis is the "unit," which is the complete description of a machining block (geometry, technology, and cycle data) Dialog boxes divided into overview and detail forms Fast navigation between the forms and the input groups via the "smart" keys Context-sensitive help graphics Start unit with global settings Transfer of global values from the start unit Transfer of cutting values from the technology database Units for all turning and recessing operations for simple contours and ICP contours Units for boring, drilling, and milling operations with the C or Y axis for holes, milling contours, and drilling and milling patterns that are simple or that have been programmed with ICP Units for trochoidal milling operations for pockets and slots Hobbing unit for the machining of gear teeth Special units for activating/deactivating the C axis, as well as for subprograms and section repeats Verification graphics for the workpiece blank and finished part, as well as for C- and Y-axis contours Turret assignment and other setup information in the smart.Turn program Parallel programming Parallel simulation
TURN PLUS		63	Automatic smart.Turn program generation with <ul style="list-style-type: none"> • Automatic tool selection • Automatic turret assignment • Automatic calculation of cutting data • Automatic generation of the machining sequence in all working planes, also for C-axis machining (with option 55) and Y-axis machining (with option 70) • Automatic cutting limit through chucking equipment • Automatic generation of rechucking work blocks for turning with opposing spindles • Automatic generation of work blocks for rear-face machining (with option 132)

User functions	Standard	Option	
Programming DIN PLUS programming	<ul style="list-style-type: none"> • • • • • • • • • • • • • • • • • • • 	55 70 131/132 132 8/9 9	Programming in DIN 66025 format Expanded command format (IF ... THEN ... ELSE ...) Simplified geometry programming (calculation of missing data) Powerful fixed cycles for area clearance, recessing, recess turning, and thread machining Powerful fixed cycles for boring, drilling, and milling with the C axis Powerful fixed cycles for boring, drilling, and milling with the Y axis Subprograms Technology functions for full-surface machining: <ul style="list-style-type: none"> • Traversing to a fixed stop • Controlled parting • Spindle synchronization • Mirroring and converting • Mechatronic tailstock Programming with variables Contour description with ICP Program verification graphics for workpiece blank and finished part Turret assignment and other setup information in the DIN PLUS program Conversion of smart.Turn units into DIN PLUS command sequences Parallel programming Parallel simulation
Simulation	<ul style="list-style-type: none"> • • • • • • • • • • • • • • • • 	55 54 132	Graphic simulation of the cycle process, or of the cycle program, smart.Turn program, or DIN PLUS program Depiction of the tool paths as pencil-trace or cutting-path graphics; special identification of the rapid traverse paths Machining simulation (2-D material-removal graphic) Side or front face view, or 2-D view of lateral surface for verification of C-axis machining Display of programmed contours View of the tilted plane (B-axis machining) View of front face and YZ plane for verification of Y-axis machining Three-dimensional display of the workpiece blank and finished part Simulation of mirrored contours for rear-face machining Shifting and magnifying functions Block scan in the simulation
B axis machining	<ul style="list-style-type: none"> • 	54 54	Machining with the B axis Tilting the working plane Rotating the machining position of the tool
Eccentric machining		135 135	Cycles for eccentric turning and for the manufacture of oval and polygonal contours Superimpositioning of traversing movements of the X and Y axes synchronously to the rotational motion of the spindle
Machining time analysis	<ul style="list-style-type: none"> • • • 		Calculation of machining times and idle times Consideration of the switching commands triggered by the CNC Representation of time per individual cycle or per tool change
Monitoring function		151	Load Monitoring—detecting tool wear and breakage during machining

Overview

– User functions

User functions	Standard	Option	
Tool database	<ul style="list-style-type: none"> • • • • • • • • • • 	<p>10</p> <p>10</p>	<p>For 250 tools For 999 tools</p> <p>Tool description can be entered for every tool</p> <p>Automatic inspection of the tool-tip position relative to the contour</p> <p>Compensation of the tool-tip position in the X/Y/Z plane</p> <p>Fine compensation of tool via handwheel with application of compensation values in the tool table</p> <p>Automatic tool-tip and cutter radius compensation</p> <p>Tool monitoring for the service life of the indexable insert or of the number of produced workpieces</p> <p>Tool monitoring with an automatic tool change after expiration of tool life</p> <p>Management of multipoint tools (multiple inserts or reference points)</p> <p>Support of quick-change tool systems</p>
Technology database		<p>8/9</p> <p>8/9</p> <p>8/9</p> <p>8/9</p> <p>10</p>	<p>Access to cutting data after definition of workpiece material, cutting material, and machining mode. The MANUALplus 620 distinguishes between 16 machining modes. Each workpiece-material / tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes.</p> <p>Automatic determination of the machining modes from the cycle or the machining unit</p> <p>The cutting data are entered in the cycle or in the unit as suggested values</p> <p>Nine workpiece-material / tool-material combinations (144 entries)</p> <p>62 workpiece-material / tool-material combinations (992 entries)</p>
User administration	<ul style="list-style-type: none"> • 		<p>Configurable linkage of permissions to user roles</p> <ul style="list-style-type: none"> • Logging on to the control with a user account • User-specific HOME folder for simplified data management • Role-based access to the control and network data
Conversational languages	<ul style="list-style-type: none"> • 		<p>English, German, Czech, French, Italian, Spanish, Portuguese, Dutch, Swedish, Danish, Finnish, Norwegian, Slovenian, Slovak, Polish, Hungarian, Russian (Cyrillic), Romanian, Turkish, Chinese (traditional and simplified), Korean</p>

Overview

– Options

Option number	Option	As of NC software 54843x-	ID	Comment
0 1 2 3 4 5 6	Additional Axis	01 01 01 01 01 01	354540-01 353904-01 353905-01 367867-01 367868-01 370291-01 307292-01	Additional control loops 1 to 7
8	Teach-In	01	632226-01	Cycle programming <ul style="list-style-type: none"> • Contour description with ICP • Cycle programming • Technology database with nine workpiece-material / tool-material combinations
9	smart.Turn	01	632227-01	smart.Turn <ul style="list-style-type: none"> • Contour description with ICP • Programming with smart.Turn • Technology database with nine workpiece-material / tool-material combinations
10	Tools and Technology	01	632228-01	Tools and technology <ul style="list-style-type: none"> • Tool database expanded to 999 entries • Technology database expanded to 62 workpiece-material / tool-material combinations • Tool life monitoring with exchange tools
11	Thread Recutting	01	632229-01	Threads <ul style="list-style-type: none"> • Thread recutting • Handwheel superimpositioning during thread cutting
17	Touch Probe Functions	01	632230-01	Tool measurement and workpiece measurement <ul style="list-style-type: none"> • Determining tool-setting dimensions with a tool touch probe • Determining tool-setting dimensions with an optical gauge • Automatic workpiece measurement with a workpiece touch probe
18	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component
24	Gantry Axes	01	634621-01	Gantry axes via master-slave torque control
42	DXF Import	01	632231-01	DXF import: importing of DXF contours
46	Python OEM Process	01	579650-01	Python application on the MANUALplus 620
49	Double Speed Axes	01	632223-01	Short control-loop cycle times for direct drives
54	B-Axis Machining	01	825742-01	B axis: tilting the working plane, rotating the machining position of the tool
55	C-Axis Machining	01	633944-01	C-axis machining
63	TURN PLUS	01	825743-01	TURN PLUS: automatic generation of smart.Turn programs

Overview

- Options
- Accessories

Option number	Option	As of NC software 54843x-	ID	Comment
70	YAxis Machining	01	661881-01	Y-axis machining
77	4 Additional Axes	03	634613-01	4 additional control loops
94	Parallel Axes	01	679676-01	Support of parallel axes (U,V,W) Combined display of principal axes and secondary axes
131	Spindle Synchronism	01	806270-01	Synchronization (of two or more spindles)
132	Counter Spindle	01	806275-01	Counter spindle (spindle synchronism, rear-face machining)
135	Synchronizing Functions	03	1085731-01	Advanced synchronization of axes and spindles
137	State Reporting	06	1232242-01	State Reporting Interface (SRI): provision of operating conditions
143	Load Adapt. Control	01	800545-01	LAC: load-dependent adaptation of the control parameters
151	Load Monitoring	03	1111843-01	Monitoring of the tool load

Accessories	
Electronic handwheels	<ul style="list-style-type: none"> • HR 180 panel-mounted handwheels with connection at position inputs, plus • One HR 130 serial, panel-mounted handwheel or an HR 510, HR 520, or HR 550 portable serial handwheel
Tool measurement	3-D touch trigger probes: <ul style="list-style-type: none"> • TT 160 with a cuboid probe contact and cable connection • TT 460 with a cuboid probe contact and radio or infrared transmission
Workpiece measurement	<ul style="list-style-type: none"> • TS 260 touch trigger probe with cable connection, or • TS 460 touch trigger probe with radio or infrared transmission, or • TS 740 touch trigger probe with infrared transmission
DataPilot CP 640, MP 620	Programming station for PC for programming, archiving, and training for the MANUALplus 620 <ul style="list-style-type: none"> • Full version with license for single station or multiple stations • Demo version (free of charge)
Software for PCs	<ul style="list-style-type: none"> • TeleService software for remote diagnostics, monitoring, and operation • TNCremo software for data transfer—free of charge • TNCremoPlus software for data transfer with live-screen function

Overview

– Specifications

Specifications	
Components	MC main computer with integrated 15.6-inch screen (1366 × 768 pixels) for multi-touch operation; operating panel can be shown CC or UEC controller unit Machine operating panel with 36 exchangeable snap-on keys as well as spindle and feed rate override potentiometers
Operating system	HEROS 5 real-time operating system for machine control
NC program memory	1.8 GB (on CFR compact flash memory card)
Input resolution and display step	X axis: 0.5 µm, diameter: 1 µm U, V, W, Y, Z axes: 1 µm B, C1/C2 axes: 0.001°
Interpolation	Straight line: in two principal axes (max. ±100 m), optional in three principal axes Circle: in two axes (radius max. 999 m), optional additional linear interpolation in the third axis C1/C2 axis: interpolation of X and Z linear axes with the C1/C2 axis (option) B axis: interpolation of the X, Z, Y, B, C axes (max. four axes)
Feed rate	mm/min or mm/revolution Constant surface speed Max. feed rate (60 000/pole pair number × spindle pitch) with $f_{PWM} = 5000$ Hz
Main spindle	Maximum of 60 000 rpm (with option 49: max. 120 000 rpm) for motors with one pole pair
Axis feedback control	Integrated digital drive control for synchronous and asynchronous motors Position loop resolution: signal period of the position encoder/1024 Cycle time of position controller: 0.2 ms Cycle time of speed controller: 0.2 ms Cycle time of current controller: minimum 0.05 ms
Error compensation	Linear and nonlinear axis error, backlash, reversal spikes during circular movements Static friction
Data interfaces	1000BASE-T Gigabit Ethernet interface MC 7410T: 5 x USB (1 x front USB 2.0; 4 x back panel USB 3.0) MC 8420T: 4 x back panel USB 3.0
Diagnostics	Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	Operation: in electrical cabinet: 5 °C to 40 °C in operating panel: 0 °C to 50 °C Storage: –20 °C to 60 °C

HEIDENHAIN

Mastering nanometer accuracy



HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

☎ +49 8669 32-5061

✉ info@heidenhain.de

www.heidenhain.com



HEIDENHAIN
worldwide