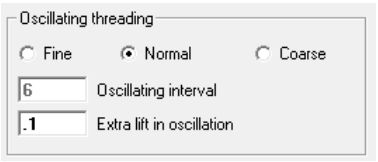
Camtek Oy
Vapaalantie 23 A
01650 VANTAA

19.12.2023

New features of WinCAM version 3.7.0

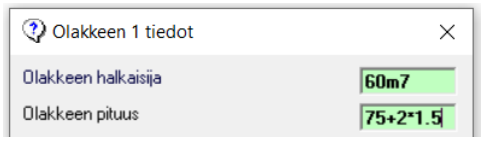
WinCAM version 3.7.0 has been improved both in terms of software usability and cutting methods.



At the subcontracting fair, the new feature of *oscillating threading* created for the WinCAM turning attracted a lot of interest. During thread cutting, the tool makes a wave-like movement by varying the depth of the cut, which causes the chip to break and the initial vibration to decrease. The strength and frequency of the waves can be adjusted. The new feature makes it easier when threading difficult materials with long chips and long slender pieces. Oscillation is available not only for normal threads but also for cylindrical profile threads. You can see how the method works in practice from the attached link.

https://www.youtube.com/watch?v=PHKd2quyp3Q

A filtering option has been added to the tool library so that only tools of a certain type will be visible. If you use the new filtering to search for tools, you should first check the box *Show only these* and then select the desired machining method. Now, also in the turning application, there are new browsing buttons, which make it easier to find another similar tool when programming a tool change.



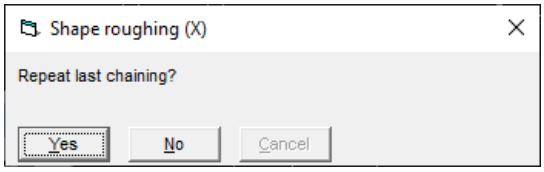
The number of fields that understand calculations has been increased in many queries made by WinCAM. Now, for example, values for macros can be given with ISO tolerance codes, or when importing DXF data, the scaling factor can be given as a fraction, etc. Smart fields are distinguished from others by their own color, which can be adjusted using the parameter file.

The data transfer functions have been added with the option to save all the characters that are sent during the transfer - including normally invisible characters - in log files (LahetysLoki.txt and Vastottoloki.txt) during transmission or reception. For example, a snippet from the transfer log:

%\13\10O0123 (ESIM)\13\10G54\13\10G50 S2000\13\10T101\13\10G97 G99 S636 M3\13\10…

You can now also set the characters to be sent at the beginning and end of the transfer yourself. For now, all these new adjustments only happen via the parameter file. Invisible ASCII characters are marked with the ASCII code after the \ character. For example, \13 or \10 (CR and LF).

Possibility of repeating the previous contour without having to show the lines again has been added to the contour chaining. The function can be accessed by pressing the *Esc* key when asked to display the first line of the contour. If the answer to the query is *No*, the function is interrupted as before. The new feature is available in all functions in programming that chain a shape path - for example roughing and finishing.

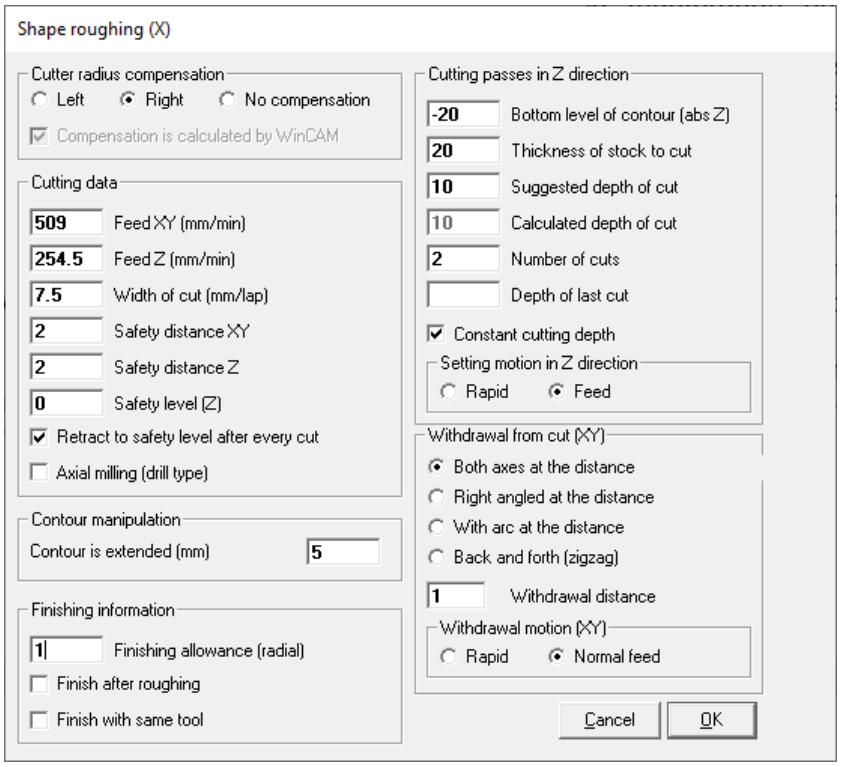


A slider has also been added to the side of the main window of WinCAM to adjust the simulation speed. At least in the beginning, it is in the turning at the bottom and in the machining center application on the right side of the main window, vertically. The speed can still be adjusted using the old methods, i.e., through the settings sheets and with the arrow keys during the simulation.

In the machining center application, one improvement that increases usability is that you don't have to put the screen in XYZ mode first if you want to rotate the image. It is enough to press the Ctrl key and start moving the mouse while pressing the left button, even if the screen is in one of the plane modes XY, XZ, or YZ. Switching to XYZ mode happens automatically.

WinCAM version 3.6.0

In version 3.6.0, the main focus has been on modernizing the roughing functions of milling. Now, all functions related to roughing are defined with a new form in the same way as has been in e.g. contour milling.



A lot of new features have been added to roughing. In principle, roughing in milling is now very similar to turning operations.

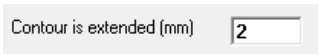
What is new is, e.g., different approaches and exits from the cut, such as approaching and exiting with an arc movement.

Roughing can also be done *axially*, i.e., with a drilling motion.

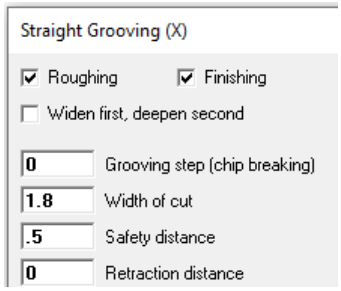
The last line of the defined contour can be extended by the desired distance.

Finishing can optionally take place *with the same contour specifications* after roughing, either with the same or a different tool.

Two new buttons have been added to the tool library of the machining center software to facilitate tool selection in those programming functions where the software suggests suitable tools. Such functions include various methods of machining holes and face milling. With the new buttons, you can, e.g., continue with the tool on the spindle despite the new suggestion or request a new suggestion as a tool from the library.



In the roughing and finishing functions of the lathe software, the extension of the last line of the contour has been changed to be adjustable. Previously, the length of the joint depended on the nose radius.

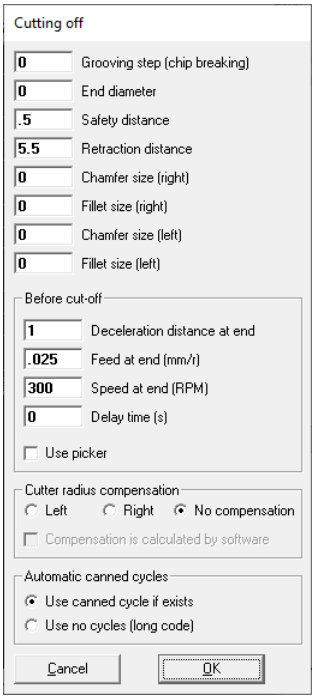


In addition to the safety distance, a separate retraction distance has been added to all the grooving and cutting functions of turning, which enables grooving operations without the risk of collision to the bottom of a cavity - e.g., another groove.

To facilitate various distance measurements, a new function, *Measure,* has been added to the geometry menu of all WinCAM applications. A new button has also been added for the function. The measurement function is very similar to drawing a straight line.

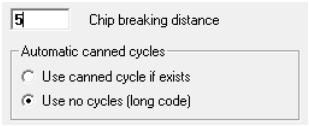
WinCAM version 3.5.0

After the previous version, many good development ideas have been received from users for the turning application. Many of them have been implemented as far as possible.



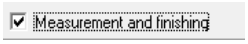
As a completely new function, *Cut off,* has been added to the programming. It also has its own new button. In the past, there has been cutting off in some postprocessors at the end of the program. Now, it can be done with all postprocessors at any time and as many times as necessary. The user can adjust the cut-off event in many ways. If necessary, the cutting blade can also be used to make chamfers or fillets either on the end of the workpiece or the bar or both. The chip can also be broken, and at the end, before cutting off, both the feed and the spindle speed can be slowed down, and a pick-up tool can be used, if there is one.

The straight grooving was also improved in a similar way. Now, it is also possible to make asymmetric shapes, i.e., chamfers and fillets can be defined for both edges separately.

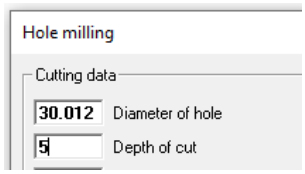


The possibility of breaking the chip with a freely selectable step has been added to the roughing functions in the same way as it was already possible in drilling and grooving. Chip breaking is only possible with most controls *if a roughing cycle is not used*.

The finishing allowance left after roughing can now be saved as default values in the software settings (parameter file).



In threading, in the new version, it is possible to print a program stop for measuring and adjusting the tool offsets, as well as finishing the thread after this. This speeds up the work of threading single occasional work pieces.

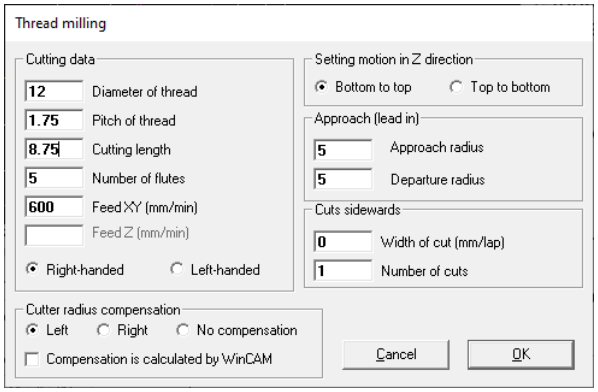


The possibility of changing the diameter of the hole from what is defined in the geometry was added to the hole milling. In this way, fine adjustments can be made, or even several hole milling operations of different diameters can be carried out with the same hole geometry, as long as the status information of the hole is provided from time to time.

In the cutting software an option to adjust the orientation weight factor was added when using directional routing strategies with automatic cutting and pattern marking.

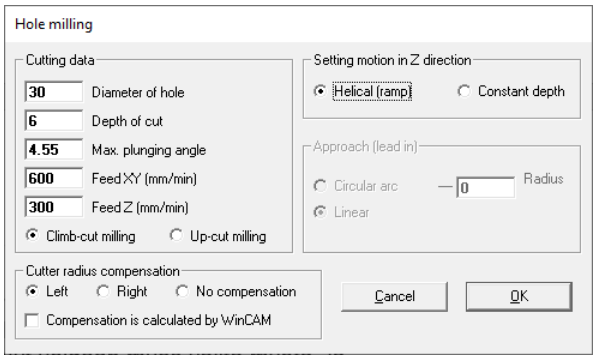
WinCAM version 3.4.0

In version 3.4.0, the biggest changes have been made to the machining center application based on the users' wishes. As far as turning is concerned, there weren't very many suggestions for improvement over the course of the year.



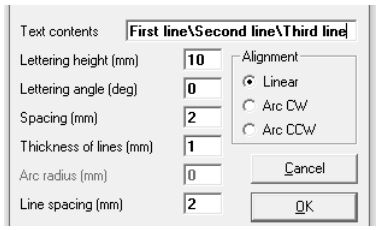
The most requested development by users has been the diversification of thread milling. In the new version, it is now possible to cut either inwards or outwards in Z-direction. Multiple side cuts have also been added. All in all, defining thread milling in the new form is much easier than before; even the diameter and pitch of the thread can still be adjusted during the programming phase - as well as the threading length or number of teeth of the thread mill.

.

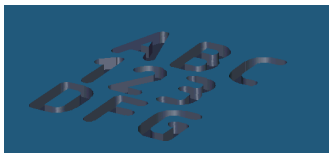


A separate form has also been made for hole milling, where the different options for hole milling can be easily specified. A new adjustment option is to enter the entry angle instead of the cutting depth in ramp machining. Now, you can also choose between climb-cut and up-cut milling.

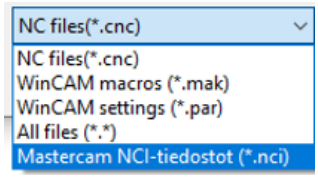
All milling functions have been added with a parameter file adjustable selection of whether to print the compensation command (G41/G42) even when the compensation has already been calculated. This was previously a built-in feature of the postprocessor.



In order to facilitate the alignment of the lines of multi-line texts, the possibility of adding line breaks to the texts has been added to the text milling and to the definition of the texts created in the geometry. The same possibility is available in the creation of text geometries of all WinCAM applications. Either character \ or | can be used as a line break character



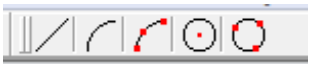
At the request of users, multi-line functionality has also been added to the Find and Replace functions of the Editor of all applications. Using the same line break characters mentioned above, you can now search for single or multi-line text fragments and replace them with other single or multi-line texts. This makes it easier, for example, to edit programs that contain a lot of tool changes.



Starting with version 3.4.0, the WinCAM machining center software can be equipped with a postprocessor, which can also read NCI files of Mastercam and perform postprocessing, the end result of which is an NC program with the same structure as when programming with WinCAM. There are two options for performing postprocessing. The NCI file can be opened via the Open NC program function of WinCAM's File menu, and if the software is equipped with a postprocessor, it automatically turns the read NCI data into an NC code, which is also simulated. Another option is to configure Mastercam so that a specific WinCAM application is selected as its Editor. In this case, when starting Mastercam's postprocessing (G1) with the NCI file editing and saving option, the WinCAM application starts and performs postprocessing and opens the final result in the Editor, which can, of course, be edited and simulated.

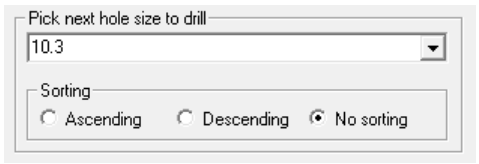
WinCAM version 3.3.0

Many improvements have been added to the new WinCAM version 3.3.0 based on ideas from users. In version 3.3.0, new geometry definition methods *Arc with three points* and *Circle with three points* were added to all WinCAM applications. New buttons were also added to the functions.



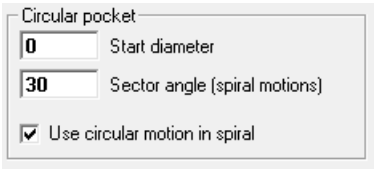
Shortcuts have been added to the *visualization* of all applications for the most common viewing directions, such as top, front, right, etc., and seven new buttons have been added for selecting directions.

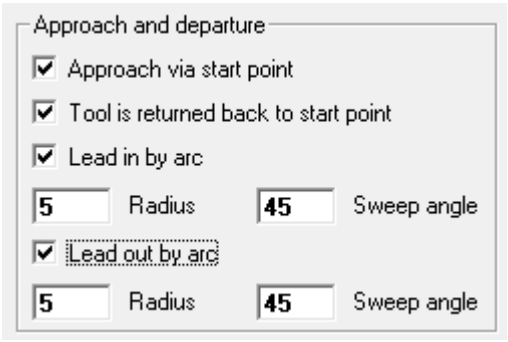
When defining the machining values of the *tool library*, it is now *possible to create a new material by copying* the old material with the machining values as the basis for the new one. This greatly speeds up the determination of machining values for the added material.



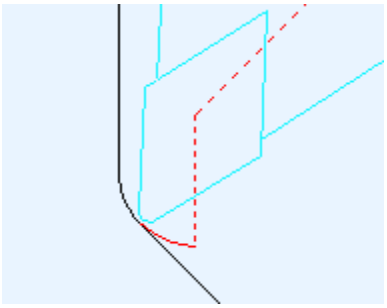
An alternative sorting of holes in either ascending or descending order has been added to the machining center software's drilling and other functions containing hole selections, such as *Select holes*.

Spiral movements of a circular pocket milling can now be performed with circular arc movements. In the past, the spiral was always made with straight movements. The rotation of the spiral is divided into individual movements based on the sector angle given by the user. For example, the attached sector angle 30 produces 12 movement commands in one round.





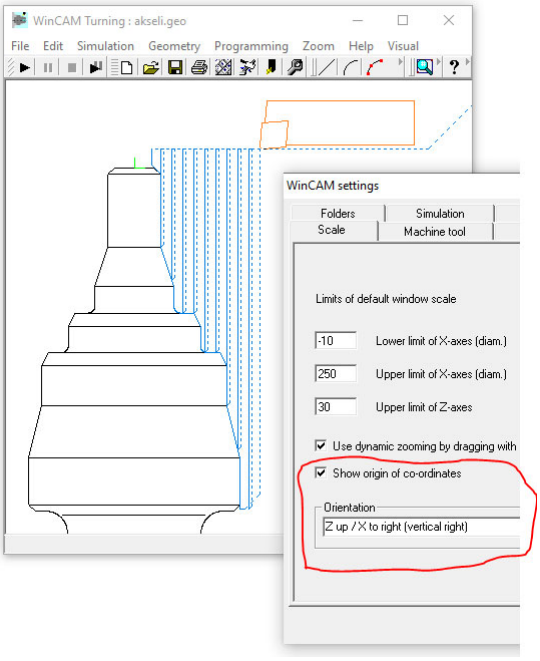
The option to approach and exit with an arc movement has been added to the finishing function of turning. Arc radius and sweep angle can be set.



Some errors found in previous versions have been fixed in the roughing functions of the machining center software.

Programming options for path milling using macros have been developed. It is now easier than ever to program milling for paths calculated with macros, such as gears and sprockets.

WinCAM version 3.2.0



In version 3.2.0, the long-desired reform for vertical lathes was implemented. Now, when simulating turning and defining the geometry, the position of the coordinate system (workpiece) can be chosen so that it looks the same as it does at the lathe. The workpiece can now be upright on the table, and the tool can approach from either the right or the left. With a horizontal lathe, the tool can approach from above or below.

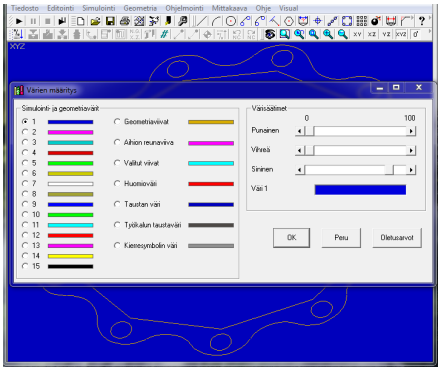
The possibility of displaying the origin (zero point) of the coordinate system in the image, if desired, has been added to all applications. Origo can also be used as a grip when using mouse. This makes drawing and zero-point shifts easier.

In version 3.2.2, grooving functions were improved when using the "External grooving" type tool with bar angles of 180 and 270. Now, both straight and shaped grooving can be done both as a back stitch and as an inside stitch, also with an outer holder. The directions of the text to be added to the geometry were also corrected in the coordinate systems of the carousel lathes.

In version 3.2.5, errors that appeared randomly when saving to network disks have been fixed. In version 3.2.6, a parameter was added to enable Fanuc's P command in spindle commands.

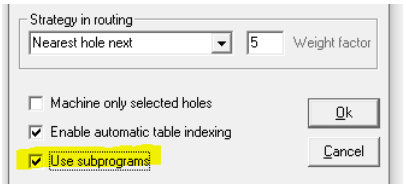
WinCAM version 3.1.0

WinCAM has received significantly more colors in version 3.1.0. Now, there are unlimited background colors, and at the same time, the possibilities of adjusting the colors of different



lines have increased. The function "Recent..." has been added to the most important file menus of the software, which can be used to quickly open the last processed nc programs and geometries. Opening geometry files (*.ge*) in WinCAM directly from Windows by clicking with the mouse is now also possible if Windows has first been told which program to connect to which file extension. Previously, this was only possible for NC programs. The direct programming of input and rapid movement has been simplified so that both can be programmed from the same form without visiting the menu in between them. The feed can also be given at same time.

The possibility of using subprograms in connection with drilling paths has been added to the drilling functions of the machining center software so that the hole pattern is programmed only once, even if it is used with several tools.



WinCAM version 3.0.0

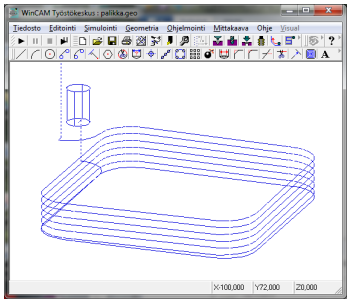
The calculation accuracy of WinCAM's geometry functions has been improved, which makes it easier, especially when calculating tangents and trimmings that require precision, as well as making divisible shapes - such as chain wheels - by multiplying by rotation. Now, you can create the entire shape path with much more certainty. The user has the option to choose whether to save the geometry with this higher accuracy or in the old format. Reading geometries saved with double precision is not possible with older WinCAM versions. On the other hand, compatibility is maintained. If files are saved in the new, more accurate format, geometries are now compatible between different applications (including turning)

The routing strategies familiar from pattern milling and cutting software have been introduced into the drilling functions of the machining center software, which allow the programmer to influence the drilling order of the hole sets.

The option to use canned cycles or print a long code has been added to the grooving functions of turning. You can now also adjust the calculation of the nose radius compensation from the grooving form. If desired, tools can be searched for in the simulation based on the offset number alone (see Settings/Simulation).

WinCAM version 2.9.0

A few new functions have appeared in the WinCAM geometry menu in version 2.9.0. All applications now have a new method for drawing lines, Normal (perpendicular), which makes drawing easier in many situations. At the request of users, a new convenient function, Polygon, has been added to the machining center and cutting applications, which can be used to draw an equilateral polygon consisting of straight lines and possible corner fillets with an arbitrary number of corners. For turning and machining center applications have been added from the cutting application already familiar function of splitting arcs into quarters.



The milling functions of the machining center software have been developed at the request of users with a couple of new features. Now the helical-type milling, which has been used for a long time in hole milling, is also possible in other milling paths, i.e., in contour milling, the tool can be put to advance also feeding the Z axis so that the tool keeps cutting all the time and there is no need to make approach and exit movements except at the beginning and the end. The known retraction distance from the holes can now be added to the data of standard-shaped pockets (rectangular and circular pockets).

In turning, the roughing of free-form blanks (Cast roughing) has been added to the final approach to the new cut with a feed from the safety distance given by the user to increase safety when the dimensions of the blank vary (e.g., castings). Now, even at the beginning of the cutting movement, it is possible to drive through the surface of the blank with a reduced feed (feed factor in penetration).

The option to use a turning U-drill as a tool has been added to the drilling of turning, which is controlled according to the outer edge and not the center (the tool type is "internal turning"). This makes it easier for situations where drilling is done with the same tool and then continued with e.g., internal roughing.

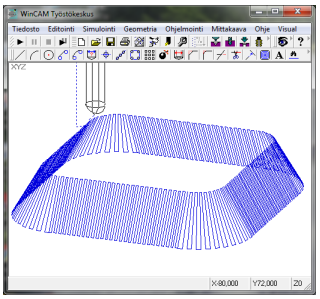
A scaling option has been added to all applications for importing CAD files. In the machining center application, the standard depth of the holes, for example, in plate-like pieces, can now also be entered into the imported geometry. The side of the table can now be given when importing.

In all applications, it has been made possible to use all zooms during paused simulation and continue the simulation without problems afterwards. Through the WinCAM parameter file, it is now possible to define the colors used in geometry selection and queries (previously fixed cyan and red) and also the color of the blank line (previously fixed magenta).

All applications have new buttons, and therefore, the old button layouts are no longer valid except for the Editor and visualization. New layouts are saved after saving the software settings under the name WinCAM29Eng.LYT, WinMill29Eng.LYT and WinBurn29Eng.LYT.

WinCAM version 2.8.0

In version 2.8.0, the WinCAM macro system has also been extended to the geometry side. Calling all the most important geometry creation commands from macros is now possible. This enables, for example, parameterized drawing of part families. It is also possible to call the most important programming commands directly from the macro. Thanks to it, macros can take advantage of the features of the postprocessor, and the construction of macros that are not dependent on control is possible. Examples of such macros can be found in the folder "Macros (280)" on the USB.

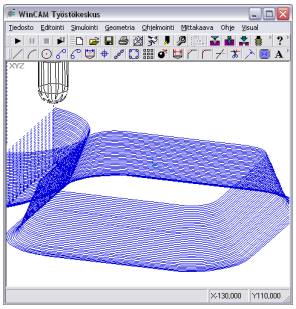


As a new machining method, *the axial milling* option of path milling has been added with the possibility of machining in a zigzag manner so that the lateral movement takes place alternately at the top and bottom, i.e., the tool is cutting all the time. This machining method is often not successful in roughing, but in finishing, it speeds up machining and improves the surface quality. Zigzag-type axial milling can be used both for vertical and inclined profiles.

In the machining center and cutting software, two fields have been added to the geometry form. In the drawing of arcs, instead of the radius, the diameter can now be entered as an alternative. In the rotation function, instead of the angle of rotation, the number of teeth can be given, which often improves accuracy and reduces the need for calculations when making divisible shapes such as chain wheels or circular saw blades.

Other new features in the 2.8.0 version include the removal of rest material in profile threading, thanks to which the opening and finishing of the thread can be done with a differently shaped tool. Saving temporary files (TMP) in their own folder and blocking the saving has been made possible. The scanning of serial ports can now be skipped when opening the data transfer, which has caused a slight delay in some situations if all the devices in the system are not on.

WinCAM version 2.7.3



In version 2.7.3, the presentation of the tools, especially the copy mills, has been improved in the XYZ image. You can now adjust the drawing resolution of the tool yourself in the settings. The possibility of divide the cuts evenly along the profile has been added to the

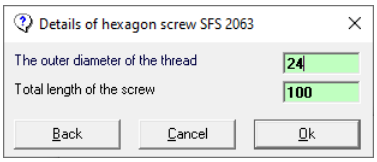
shape profile milling, which improves the surface quality on curved surfaces.

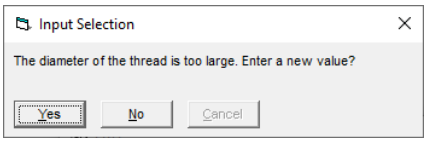
As a new machining method, axial or drilling milling has also been added to contour and copy milling operations. Thread sizes have been added to the specification of the threaded hole, and now the drilling diameter is also automatically retrieved from the standard.

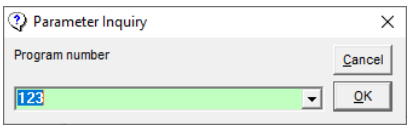
From *version 2.7.2* onwards, the use of WinCAM macro features for automating one's own functions is even more convenient and pleasant. The features of the INPUT commands that make queries have been added with the possibility of giving hint texts related to the queried data without long variable names that are difficult to come up with. Here are a couple of examples:

```
Query:!
INPUT LIST Details of hexagon screw SFS 2063
        #D_ulko "The outer diameter of the thread"
        #L_Pultti "Total length of the screw”
END LIST
----
Wrong_size:!
INPUT SELECTION The diameter of the thread is too
large. Enter a new value?
IF (#Yes=1) GOTO Query
GOTO Exit
----

INPUT #OhjNo "Program number"
```



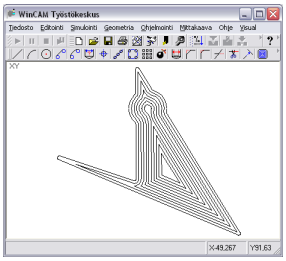




New operators AND and OR have been added to the logical conditional statement (IF), which enable the simultaneous comparison of several conditions in one IF statement, for example:

IF (#Bolt_diameter<16 OR #Bolt_diameter>36) GOTO Wrong_Size

In the USB folder Macros\Turning\Fanuc you can find an example macro Kuusioruuvi (ver 2_7_2).mak from new features.



With *version 2.7.1,* an even smarter offset function became available in WinCAM, which makes it easy to build non-hooking paths as a basis for pattern and path milling as well as roughing and finishing turning. Clearing multi-shaped areas with offset path set and pattern milling is also now easier.

In version 2.7.0, the possibility of using an envelope curve with an adjustable distance was added to the cutting software when nesting workpieces. This significantly facilitates the quick and efficient nesting of workpieces. New features also include text and pattern marking.

In the turning simulator (visualization), the effect of the radius compensation of the milling cutter can now also be seen in the face milling, at least with Fanuc, Siemens, Okuma, and Traub controls.

In all applications, it is now possible to import CAD geometries so that it is added as a new geometry group (cf. Insert geometry). Automatically splitting arcs of a circle into quarters is also a new option in CAD import. A new button has been added to the software settings in the machining center software for moving the zero point in programming. On the display page of the software settings, it is now possible to adjust the drawing resolution of the arcs on the screen. In the Editor, thanks to user feedback, it is possible to automatically add the time and date to the text either from the menu or by pressing F5.

Managing tools in an Excel spreadsheet

In the “Työkalut Exceliin” folder on the USB Flash drive, there is an Excel application, ”Työkalujen hallinta.xls” that can be used to open any WinCAM machining center software tool library in

Excel, modify it there, and save it back to WinCAM. In addition to normal Excel functions, the application has many automatic functions that can be selected with buttons for maintaining and editing the library.

Latest features of previous versions

In version 2.6, the biggest changes have been made to the machining center software. At the request of the users, the possibility of milling contours of freely shaped cross-sections has been added to the contour and copy milling, in addition to the previous vertical and chamfered ones. Now, the side profile of the contour, which is formed freely from the segments and curves, can be drawn anywhere and used in the calculation of the path at different depths. The only condition is that the profile remains constant throughout the path.

Due to the users' request, proper taper thread programming has also been added to the lathe software for special profile threading. It is no longer necessary to fiddle with the pull-out movement, but the taper angle and the pull-out angle can be given separately, and the acceleration movement is also in the direction of the taper. Based on user feedback, many usability improvements have been made to all applications, and the shortcomings of the previous major update 2.5 have been corrected.

WinCAM version 2.5 was made in a new development environment with more efficient programming tools, that's why the version number also has a clear leap in honor of the recently ignored 25th anniversary. Thanks to the improved efficiency, WinCAM's graphics window, where geometry and simulation trajectories are presented, has been made more dynamic. Instead of the former bitmap graphics, which required a lot of redrawing, vector graphics are now used. It enables lightning-fast zooming and panning with the mouse at any time without re-simulating. In the machining center software, the image can also be rotated; that is, paths and geometry can be viewed from different directions. It has also been possible to radically reduce the need for re-simulation when canceling programming. These improvements significantly speed up the creation and testing of large NC programs in particular. Many smaller improvements have also been made, e.g., chamfering and rounding now have a predictive corner display, making it easier to indicate corners.

The installation package for version 2.5 and later is not compatible with older versions, but when updating an old version, changing only the application file (exe) is usually sufficient. Old and new versions can be used on the same computer. Support for old Windows environments has been maintained, so even old Windows 95 machines can still be used.

For the use of the 2.5 version, see separate additional instructions.

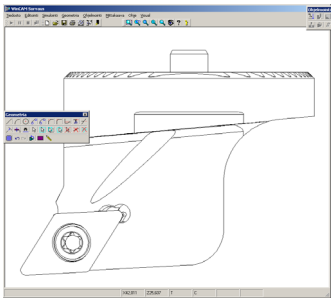
In version 2.3.24, an option was added to change the precision with which continuous contour chains are searched for in path selection, machining, and offset path creation. This reduces the need to correct imprecisely drawn paths. The accuracy can be changed in the Decimals tab of the software settings (*Chaining Tolerance*). Paths can now also be patched more easily by making a broken path into a complete offset path.

All grooving functions of turning have been added with the grooving method recommended by tool manufacturers, which can be used for wide and deep grooving. In it, a value greater than the width of the blade and no more than two times the width of the blade is given as the lateral transfer. The first stitches are made across the entire width of the blade using the given step. After that, the remaining ridges (in the middle) are removed. In this method, the blade is loaded and wears evenly, and the leaf of the soft blades intended for deep stitching does not bend. In connection with form grooves, this method is currently only used for opening the rectangular area of the form groove, and the form edges are inserted in the same order as before but using as wide a step as possible.

The long-awaited complete overhaul of pocket milling in the machining center software was done in version 2.3.23 for those controls that do not have proper pocket milling cycles (e.g., Fanuc). There are several options for penetrating into the pocket (drilling, helical, and pendulum). A circular pocket (still a special case of a rectangular pocket) has its own spiral-like machining method and can also be defined with a starting diameter (pre-hole). The rectangular pocket can be at any angle and also rotates in the hole circle along with the dividing angle. Sequential holes were also included in both hole milling and U-drilling. The handling of macro variables has been changed so that variable names containing only numbers (e.g., #123) are interpreted as variables

of Fanuc user macros so that they can be printed from WinCAM macros. Printing the tool list at the beginning of the NC program is now also possible with those controls that can have comment lines in the program. Small cosmetic changes have also been made to the bottom bar of the WinCAM window so that it would also be better visible in XP, Vista, and Windows 7 styles.

In version 2.3.22, the capabilities of the tool library and simulation were improved to show the actual shape of the tool.



Now, the shape of the tool can consist of a maximum of 5,000 lines, which enables the use of dxf images available on tool manufacturers' websites. Attached is an example of a tool retrieved from Sandvik's website. The use of external image files in the tool library is described in, e.g., Turning instructions and examples in chapter 6 on page 12 of the material. Pictures can be found, for example, at http://www.sandvik.coromant.com/

In version 2.3.21, capabilities for multilingual versions were added. In the multilingual version, the operating language is always selected at startup. At the moment, you can choose Finnish and English, as well as Swedish for turning.

Undo options have been added in both geometry and programming functions starting from version 2.3.20. Completely new is undoing undo, i.e., redoing (Undo/Redo). Up to 20 recent actions can be undone and redone. In the machining center application, there is an added option for threading to separate the threads from each other based not only on the diameter information but also on the basis of the pitch. New buttons have been added to both geometry and programming, e.g., *Select holes.*

In version 2.3.19, the functions *Select holes* and *Group properties* were added to the geometry menu. The possibility of selecting holes based on diameter makes it easier to examine and supplement the hole data of geometry imported from CAD, for example. The group information in the function *Group properties* can be freely examined, and geometry group numbering and zero points can be changed, for e.g., multi-piece programming.

Installation on a hard drive from a USB Flash Memory (update)

The update can be done either as an initial installation (see below) or by copying the new exe files from the machine-specific folder on the USB memory to the computer's WinCAM folder. If error messages appear when starting WinCAM after the update, it is a sign that one of the Windows libraries also needs to be updated. In that case, the initial installation must be performed for at least one application according to the next *paragraph*.

Installation on a hard drive from a USB Flash Memory (initial installation)

Start Windows (log in with administrator rights if possible) and insert the USB memory into one of the USB ports. *Open* the folder of the machine tool or NC control from the USB memory (e.g., Heidenhain) and click *Setup.exe* to start the program from there. If there are several applications to be installed, repeat this for all of them.

The installation can be performed as a system administrator even if you are logged in as a normal user (however, passwords are required). When the USB window has opened, open the machine tool or NC control folder and right-click on *the Setup.exe* icon from there. In the menu that opens, select *Run as administrator.* If there are several programs to be installed, repeat this for all of them.

The installer copies all the necessary files to the desired folder and creates the necessary startup short cuts for Windows. When the software is started for the first time, you should check all folder paths (File/Folders...) to see if the software was not installed in the location suggested by the installer. If the folders are changed, you should save the software settings (*File / WinCAM settings / Save settings*) afterwards.

If during installation you receive a notification about a file to be installed: *Setup is about to replace a pre-existing file(s)... Cancel setup?* Answer the question by pressing the button No. After that, you may receive a notification: *Installing over an existing installation... Are you sure you want to continue?* Answer the question by pressing the YES button.

After installation, you should start WinCAM at least once with system administrator rights and then copy the startup icons to the necessary user profiles or to the profile "all users."

If there are problems starting the software, see *ReadMe.txt* and *ReadMe (Windows 7).txt* from the folder Instructions and examples\Installation on the USB Flash memory.

License terms

The purchased software license always includes the right to use the software on all computers of one of the company's offices. Wider use and transfer of the software to a third party is strictly prohibited. If the machine tool for which the software has been purchased is sold, the transfer of the right to use to the buyer of the machine tool must always be negotiated with the software supplier.

WinCAM software instructions

The USB memory has a support package in the folder "*Instructions and examples*" with more than 300 pages of illustrated step-by-step overview of NC programming with WinCAM containing many practical examples. There are also chapters on building, editing, and visualizing the tool library. The text is available in both pdf and doc format. The drawings, geometries, macro programs, and tool libraries for the examples are also located in their own folders.

The operating instructions have been implemented using the Windows Help system to be browsed on the screen. With the help of the F1 key, you can get quick help related to the topic on the screen almost at any time. The operating instructions work in some Windows versions only with the help of an operating system update that can be downloaded from Microsoft (instructions can be found on the USB Memory in the file Lueminut.txt). Windows 10 and 11 no longer support the traditional Help system, but help texts can be found in the Help folder of the USB media in several different formats.

Help texts on the installation media.

Help texts for all WinCAM applications in Word format are in the *Help* folder of the installation media. These are freely editable if you want to make your own material.

Shortcuts and software settings

The installer creates a shortcut to launch the software. The icon can be found by opening *the Start* menu as a window (with the right mouse button). Open also the *Programs* window inside it. From there, if necessary, the icon can be copied to the desktop with the mouse.

If you create the shortcuts yourself (or let Windows do it), it is important to check in the target field of the shortcut properties that the name of the program to be started is followed by the name of the parameter file (=software settings), e.g.

Target: C:\wincam\Fanuc0M.EXE Fanuc0M.PAR

There can be several shortcuts and parameter files (for different machines). When WinCAM is started, it always reads the data of the above machine. For example

Target: C:\wincam\Fanuc0M.EXE Kafo.PAR or
C:\wincam\Fanuc0M.EXE Femco.PAR

Additional Information

If there are problems with installation or starting, you can ask for instructions or give any kind of feedback at the e-mail address: info@camtek.fi