History of MTS software modification

20/Nov./2013

Nano Corporation

The history of MTS software modification is created as the updating version when the software is corrected or updates a function. Please read about the portion which became newer than the version of the software which you are using now. About the version of software, it can check with the Help menu of a manual operation screen.

***Please keep in mind that the item written in the red character is an important portion in connection with operation of a processing machine especially.**

Contents of modification of ver.464 → ver.468

Modified item	Contents
Movement of program execution status display position	The execution state of the program displayed in "execution" tab of the automatic operation window until now was moved into the manual operation window.
Change of the display position of a message box	All the display positions of the message box currently displayed in the center of each screen, such as manual operation window and automatic operation window, were changed so that it might be displayed in the center of a personal computer screen.
The addition of a drawing magnification modification function	The button for changing the display magnification of the tool path drawn in the graphic area of an automatic operation screen was added.
The addition of a macro-variables function	The macro-variables function was added so that macro variables could be used within G code as an option.
Correspondence to the edge number of tool for a lathe system model	About the model of lathe system, although the edge number of tool which can be used until now were only No. 2 and No. 3 implicitly, it was corrected so that it might correspond to the edge-of-a-blade numbers from one to eight.

Contents of modification of ver.463 → ver.464

Modified item	Contents
An improvement of the termination processing of a turning centre	Since the cables with which the turret was equipped by being ended even if the turret has not returned to the origin point are twisted when the "x" button of a manual operation screen is used at the time of the end of a turning centre, it has been improved so that the "x" button cannot be used.
Addition of the tab screen for adjustment of tool offset data	Since a user was only allowed to check for one kind of registered tool offset data on software until now, a screen in which the user can look through the registered offset data of all the tools and can correct the data was added.
Polar correction on the display of tool offset data	About the portion whose tool offset data did not correspond with the coordinates polarity of an axial monitor of a manual operation screen, it was corrected so that axial monitor's polarity and the polarity of tool offset data might be in agreement.(This is correction of only a display only, and even if tool offset data registered before are used, it does not have influence on operation in the case of automatic operation.)

Contents of modification of ver.462 → ver.463

Modified item	Contents
Correction of the of the shank exchange program for turning centers	Since a movement of shank exchange in backup memory recovering in case of turning centers was wrong, it was corrected.

Contents of modification of ver.458 → ver.462

Modified item	Contents
Addition of the screen which can choose the model type used by MTS5R	The screen where a user can choose one of the model types of a milling machine or a lathe as the software start-up in MTS5R was added.
The addition of the format check processing at the time of loading a program in DNC mode	In the case of DNC mode, the format check of the program was performed during program execution, but it has improved so that a format check is performed when a program is loaded, and operation is not performed if there is a format error.

Correction of processing of a program in which G92 was used	Since fault was shown in the conversion process of coordinates, or the display on a screen in processing of the program for which G92 was used by automatic operation, it was corrected.
A fault improvement of processing of a sensor when the locking pin for the spindle is operated	In the model which equipped the spindle with the lock pin, the fault of processing when a reset button and a servo-on button are operated has been improved.
Recovering processing of shank exchange operation for the turning centre at the time of memory evaporation	It has been improved so that it may be restored with checking the "BackUpRecover" menu, when shank exchange movement of a turning center stops operating by evaporation of a backup memory.

Contents of modification of ver.457 → ver.458

Modified item	Contents
Correction about check processing of the zero return at the time of program execution	Whenever "execution" button was pushed until now, the mechanical zero return was demanded each time. Unless the error took place about this problem, once the zero return was performed after starting of software, it corrected so that a program might be run any number of times as it is.

Contents of modification of ver.455 → ver.457

Modified item	Contents
Correction of the processing at the time of pushing the "emergency stop" button of a manual operation screen during program operation	About the fault whose automatic operation stop was not completed even if it pushed the "emergency stop" button of the manual operation screen during program operation, it corrected so that automatic operation might be stopped normally.

Correction which avoids execution of program operation at the time of the servo OFF of spindle	When performing program operation, it corrected to the fault which was able to perform operation even when the servo of the spindle was come by off so that execution evasion of the program operation might be carried out, when the servo of a spindle was OFF.
Correction about feeding speed instructions within a subprogram	When newly not being ordered a feeding speed within a subprogram, the fault in which the feeding speed value ordered just before the block which calls the subprogram was not reflected was corrected.
Restriction of the spindle rotation in case of connecting a centering sensor	Where a spindle is equipped with a centering sensor unit, when the centering sensor unit of the option of a turning centre was connected to a machine, and continuation rotation of the spindle was carried out, the fault that a cable was rolled round was corrected. It was exchanged in the kind of sensor, and restriction on software was carried out so that continuation rotation of the spindle could not be performed, when the sensor unit was connected according to it.
Correction about the stop processing of the subspindle at the time of program operation	About the problem which was not able to stop rotation of the subspindle even if it pushed a "stop" button, an "emergency stop" button, etc., when the subspindle was being operated during program operation, it corrected so that a subspindle could be stopped.
Correction about the processing at the time of G84 instructions	When it was ordered G84, the fault software is forced by the transaction rejection of software depending on the combination of instructions of the address of J and K to terminate was corrected.
The addition of the log file processing for operation record	The processing which records the operating procedure which the user performed as a log file as a tool which supports correction of the fault which occurred during software operation was added.

[For lathe type only] The improvement about a tool offset display in the parameter screen at the time of turning centre use	The fault by which a registered tool offset value may not be reflected in a screen on a parameter screen in a turning centre at the time of use has been improved.
The display of the instruction angle at the time of spindle inching operation	Since the instruction angle was not displayed on the monitor when inching operation of the principal axis was carried out on a manual operation screen, it corrected so that an instruction angle might be displayed.
Automation of download operation of a program	The process was simplified as performing automatically the process which read the processing program with the former "Load File" button, and had transmitted the program to the control box with the "PC -> CtrlBox" button in a control box to transmission of a program within the process of "Load File."

Contents of modification of ver.454 → ver.455

Modified item	Contents
The addition of the check before program execution in case of change of tool offset or feed speed	When tool offset data or a feeding speed were changed and it was going to perform program operation, without reloading a program, the processing which displays the confirmation message of whether to perform as it is was added.
Improvement in the number of program steps which can be performed at the time of DNC operation	The number of steps of the program which can be run in the DNC mode of operation was improved from 100,000 old steps to 1 million steps (the number of characters which can be used for one step is to 128 characters at an ASCII character).
Correction of the fault which happens when Feed speed is changed in DNC mode	When Feed speed was changed on the manual operation screen after choosing the DNC mode of operation, the error which the error message came out at the instruction execution time of the program, saying " Designated data cannot be written", and was not able to run a program was corrected.

Contents of modification of ver.451 → ver.454

Modified item	Contents
The addition of a backup memory restoration function	The function in which restoration processing can be performed on software to management when "Backup Memory Error" comes out by electric discharge of the battery for memory backup on NC control board in a control box was added.
[For lathe type only] Correction of the problem by which a work size parameter is reflected mutually	When using the machine of lathe type, the parameter that is "Work length", or "Work diameter" and so on inputted on the automatic operation screen deleted the processing automatically reflected in the parameter screen which rose again. The treatment of the parameter in connection with work size was corrected so that what was inputted on the parameter screen might be reflected only in one way at the start-up of an automatic operation screen.
[For lathe type only] Correction of simple CAD/CAM software	The fault from which an error takes place by the program of a circle orbit of operation was corrected by the main coordinates of a circle at the time of inputting cutting form shifting the actual condition and a little, and registering them with a mouse from the machine of lathe type, at the time of use of simple CAD/CAM software.
[For lathe type only] Correspondence to a zero coordinates point input	At the time of use of the program G71 or for cycle operation of G72, in a lathe type machine, when ordered the point that the coordinate value on work coordinates (0, 0) etc. serves as zero, the fault whose path generation was not completed normally was corrected.

Contents of modification of ver.448 → ver.451

Modified item	Contents
Modification of main axis rotation direction	We set up turning of main axis clockwise as normal rotation "CW" and turning of main axis counterclockwise as reverse rotation "CCW" when we saw main axis from back side, in line with idea of ordinary machine tool, and reversed handling of main axis rotation direction. (In both manual operation and automatic operation).

Modification of turret rotation direction	With reference to turret rotation direction in case of manual operation of turning center, we set up turning of tool upward as “+” and turning of tool downward as “—” in order to conform with movement at the time of program operation, and reversed handling of turret rotation direction.
Modification of movement polarity of Y axis of milling machine	With reference to Y axis movement direction in case of manual operation of milling machine , we set up turning of tool upward as “+” and turning of tool downward as “—” in order to conform with movement at the time of program operation, and reversed handling of Y axis movement direction.
Correction of coordinate polarity at setup of G54 ~ G59 work coordinate system	We corrected trouble which coordinate polarity (offset direction) actually shifted was opposite in setup handling of G54~G59 work coordinate system.
Modification of display polarity of X axis of milling machine	With reference to axis monitor on manual operation window for milling machine, we modified software so that direction which X axis is away from motor is “+” , and its reverse direction is “—” in X axis display polarity of work coordinate system.
Finish of automatic operation window in DNC operation → modification of software crash caused by restart	We modified software so that re-start of automatic operation window after stop of automatic operation may not cause software crash in case of closing of automatic operation window during program operation including DNC.
Modification for program description without semicolon “ ; “	We modified software so that program description without semicolon “ ; “ at the end of each block can be accepted in program description read in at the time of automatic operation. (Program description with semicolon “ ; “ until now can be accepted as before).

Addition of preliminary determination of minus coordinate in movement command	We added the handling to prevent the trouble which movement command is given to absolute coordinate at motor side from machine origin by non-reading of offset data at the time of reading of program on automatic operation window and limit error is caused at the time of automatic operation.
Addition of G77 outer diameter/internal diameter cutting cycle function	We added G77 “outer diameter/internal diameter cutting cycle” function to function which can be handled by automatic operation.
Addition of G78 single screw cutting cycle function	We added G78 “single screw cutting cycle” function to function which can be handled by automatic operation.
Addition of G79 end face cutting cycle function	We added G79 “end face cutting cycle” function to function which can be handled by automatic operation.
Addition of G80 drilling cycle cancellation function	We added G80 “drilling cycle cancellation” function to function which can be handled by automatic operation.
Addition of G81 spot drilling cycle function	We added G81 “spot drilling cycle” function to function which can be handled by automatic operation.
Addition of G82 spot drilling cycle dwell function	We added G82 “spot drilling cycle dwell” function to function which can be handled by automatic operation.
Addition of G83 peck drilling cycle (high speed deep hole) function	We added G83 “peck drilling cycle (high speed deep hole)” function to function which can be handled by automatic operation.
Addition of G84 peck drilling cycle (detail) function	We added G84 “peck drilling cycle (detail)” function to function which can be handled by automatic operation.
Addition of G98 drilling cycle initial point return function	We added G98 “drilling cycle initial point return” function to function which can be handled by automatic operation.
Addition of G99 drilling cycle R point return function	We added G99 “drilling cycle R point return” function to function which can be handled by automatic operation.

Addition of G94, G95 feed function	With reference to feed rate at the time of automatic operation which has corresponded to feed rate only per minute with F value in the past, we modified software so that feed rate can correspond to both feed rate per minute and feed rate per rotation of main axis by the combined use of G94 and G 95.
Modification of main axis inching movement distance	We modified movement distance at the time of main axis movement mode “INC” in manual operation to setup of 1 degree “ ° ” unit which corresponded with distance of inching movement distance setup button.
Modification of main axis indexing rotation command C	We modified command unit of main axis indexing rotation command C in automatic operation to “ ° ”.
Addition of M03/M04/M05 main axis function	With reference to main axis rotation setup which corresponded with command format of S value only until now in automatic operation, we modified software so that main axis rotation setup can correspond with M03 (normal rotation), M04 (reverse rotation) and M05 (stop).
Correspondence with main axis rotation number display	We modified software so that main axis rotation number can be displayed by main axis rotation number monitor on manual operation window in accordance with main axis rotation command in automatic operation.
Addition of cutting tool centering setup unit handling	We added handling for optional “centering setup unit” which can be used for turning center.
Addition of electric sub-spindle unit control function	We added control function for optional “electric sub-spindle unit” which can be used for turning center.
Addition of main axis lock censor handling	We added handling for “main axis lock censor” attached to new type main axis.
Addition of manual zero return function	We added function which can execute zero return by hand when automatic zero return cannot normally executed by some reason.

Addition of movement prevention function into shank exchange area	We added handling which movement can be automatically stopped, even if axis unexpectedly moves into shank exchange area (area at motor side from Z axis origin) at the time of manual operation and automatic operation in case of turning center.
Prohibition of double starting of operation window	We modified software so that double window cannot be started, when such operation window as parameter window and automatic operation window etc. is already started.

Contents of modification of ver. 438 → ver.448

Modified item	Contents
Exclusion of connection with program execution button and return key	We corrected trouble which program operation starts when return key on key board is pressed in the condition of starting of automatic operation.
Prohibition of reload of program file in DNC operation	We modified software so that reload of program file can not be performed during DNC operation, as reload of program file during DNC operation is impossible in handling of DNC operation.
Correction of software crash caused by error during DNC operation	We corrected trouble which software is crashed when some errors are caused during DNC operation

Contents of modification of ver.436 → ver.438

Modified item	Contents
Correction of display of program step number during DNC operation	We corrected trouble which display of program step number on automatic operation window is different from actual number, when program over 65535 step is executed during DNC operation.

Contents of modification of ver.429 → ver.436

Modified item	Contents
Correction of G71 ～ G73 cycle operation handling	We corrected trouble in reload of G71～G73 cycle operation program into software.

Correction of feed rate in automatic operation	We corrected setup of feed rate, as setup of feed rate in automatic operation of the machine of which minimum movement unit of feed axis is 0.1 μm is setup to be 1/10.
Elimination of automatic over ride at the time of quick feed in automatic operation	We eliminated automatic setup of up to 200% of rate over ride in software, in case of quick feed command in automatic operation

Contents of modification of ver.411 → ver.429

Modified item	Contents
Correspondence to DNC operation	We added function, in order to be able to correspond to DNC operation as option.
Elimination of selection window for kinds of machine	We modified software so that selection window for kinds of machine by user at the time of software starting is eliminated and kinds of machine and specification can be automatically set up in software by identification of serial ID for each machine.
Elimination of tool drawing	We modified software so that tool drawing in graphic area of automatic operation window at the time of automatic operation is eliminated, and instead, actual tool path can be displayed in red line.
Rotational restriction of turret	We modified software so that tool for turret can not be changed in "TOL" mode, when stage for either X or Z is more than 5 mm away from origin for prevention of collision of working axis by turret rotation for turning center.
Exclusion of influence of single block for shank exchange movement	We modified software so that shank exchange movement is not stopped halfway, even in the condition of selection of single block operation on automatic operation window in shank exchange movement for turning center.
Correction of display monitor for program execution block	We corrected trouble which handling display of first several blocks is delayed on display monitor for program execution block on automatic operation window.

Contents of modification of ver.402 → ver.411

Modified item	Contents
G71 ～ G73 Cycle operation path	We modified feed rate at every cutting upward for constant depth from quick feed to cutting feed in creation of path at the time of cycle operation in G71 ~ G73.
Registration method of cutting tool centering	We modified software so that cutting tool centering can be easily registered by input of work diameter to be used for setup work in cutting tool centering setup for turret for turning center
Handling of English notation	We modified software so that notation on software can be shown in English.
Display of diameter value	We modified software so that diameter value is sure to be displayed by positive value in case of kinds of machine of turning machine line.
Registration method of work coordinate X Y origin	We modified software so that coordinate to be registered as processing center at the time of software starting can be automatically registered on X Y origin of work coordinate system in milling machine.
Display of tool offset	We modified software so that updating of tool number based on switching of turret tool is executed on manual operation window for turning center.
Edit tab for simplified CAM data	We modified partially processing in data edit tab in use of simplified CAM in turning center.

Contents of modification of ver.401 → ver.402

Modified item	Contents
Display of tool offset	We modified software so that T axis tool offset value on parameter window can be automatically switched by switching of turret tool for turning center.