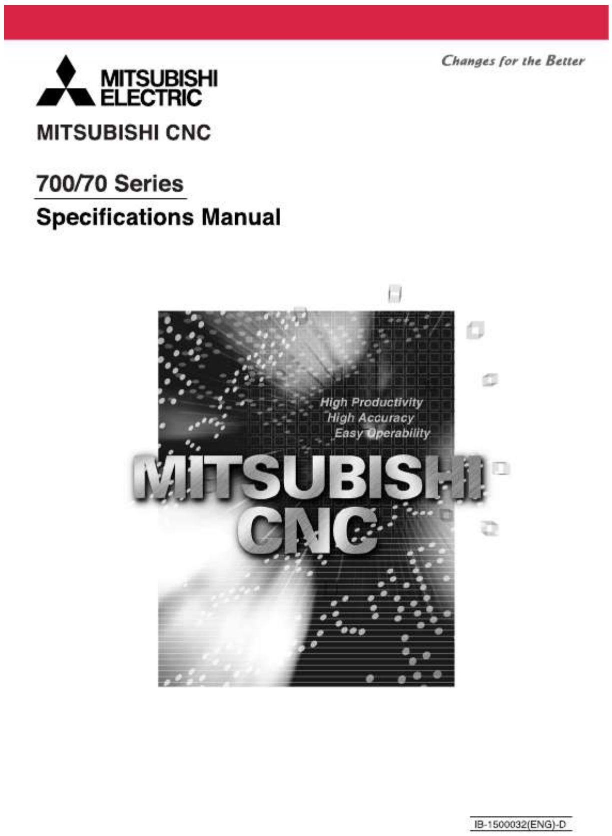
Changes for the Better

MITSUBISHI ELECTRIC

MITSUBISHI CNC

700/70 Series

Specifications Manual



IB-1500032(ENG)-D

*Manual abstract:*

*Details described in this manual At the beginning of each item, a table indicating it's specification according to the model. : Standard : Option : Selection : Special option CAUTION The items that are not described in this manual must be interpreted as "not possible". This manual is written on the assumption that all option functions are added. Some functions may differ or some functions may not be usable depending on the NC system (software) version. General precautions (1) When the contents of this manual is updated, the version (A, B, ...) on the cover will be incremented. Precautions for Safety Always read this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use. Understand all the conditions described in this manual before using the unit.*

*We rank the safety precautions into "DANGER", "WARNING" and "CAUTION" for the manuals issued by Mitsubishi, including this manual. DANGER WARNING CAUTION When there is a great risk that the user could be subject to fatalities or serious injuries if handling is mistaken. When the user could be subject to fatalities or serious injuries if handling is mistaken. When the user could be subject to injuries or when physical damage could occur if handling is mistaken. CAUTION", may lead to major results depending on the situation.*

*Note that even items ranked as " In any case, important information that must always be observed is described. DANGER Not applicable in this manual. WARNING Not applicable in this manual. CAUTION 1. Items related to product and manual The items that are not described in this manual must be interpreted as "not possible".*

*This manual is written on the assumption that all option functions are added. Some functions may differ or some functions may not be usable depending on the NC system (software) version. 2. Items related to start up and maintenance Follow the power specifications (input voltage range, frequency range, momentary power failure time range) described in this manual. Follow the environment conditions (ambient temperature, humidity, vibration, atmosphere) described in this manual. ! Follow the remote type machine contact input/output interface described in this manual. (Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.) If the parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation. The parameter for the temperature rise detection function will be validated forcibly when the NC unit is turned ON.*

*CONTENTS*

  *1. Control Axes 1.1 Control Axes 1.*

*Control Axes The NC axis, spindle, PLC axis and auxiliary axis are generically called the control axis. The NC axis is an axis that can be manually operated, or automatically operated with the machining program. The PLC axis is an axis that can be controlled from the PLC ladder. 1.1 Control Axes 1.*

*1.1 Number of Basic Control Axes (NC Axes) M70 Type B 3 2 M70 Type A 3 2 M720 3 2 M730 3 2 M750 3 2 M system L system 1.1.2 Max. Number of Axes (NC Axes + Spindles + PLC Axes) Max.*

*number of axes (NC axes + spindles + PLC axes) M system L system M70 Type B 9 9 M70 Type A 11 11 M720 12 12 M730 16 16 M750 16 16 A number of axes that are within the maximum number of axes, and that does not exceed the maximum number given for the NC axis, spindle and PLC axis can be used. Connection specifications of NC axis, PLC axis and spindle There are two channels with which the servo and spindle are connected. Maximum 8 axes can be connected with each channel. NC axis, PLC axis, spindle : These can be connected with the optical servo communication channel (OPT). The connection* number of first axis to eighth axis is assigned to each channel. Connect them from the first axis in order. More than one axis must be connected with the channel 1. Max. number of NC axes (in total for all the part systems) M system L system M70 Type B 4 4 M70 Type A 6 7 M720 6 12 M730 16 16 M750 16 16 Max. number of spindles Includes analog spindles.

M system L system M70 Type B 2 2 M70 Type A 2 3 M720 4 4 M730 4 4 M750 4 4 Max. number of PLC axes M system L system M70 Type B 4 4 M70 Type A 4 4 M720 2 2 M730 2 2 M750 2 2 1 1. Control Axes 1.2 Control Part System 1.1.3 Max. Number of Auxiliary Axes (MR-J2-CT) M70 Type B 0 0 M70 Type A 0 0 M720 4 4 M730 6 6 M750 6 6 M system L system Auxiliary axis: This can be connected to the channel (SV2) for J2-CT. 1.1.4 Number of Simultaneous Contouring Control Axes Simultaneous control of all axes is possible as a principle in the same part system.
However, for actual use, the machine tool builder specification will apply. M system L system M70 Type B 4 4 M70 Type A 4 4 M720 4 4 M730 4 4 M750 8 8 1.1.5 Max. Number of NC Axes in a Part System M70 Type B 4 4 M70 Type A 6 6 M720 6 6 M730 8 8 M750 8 8 M system L system 1.
2 Control Part System 1.2.1 Standard Number of Part Systems M70 Type B 1 1 M70 Type A 1 1 M720 1 1 M730 1 1 M750 1 1 M system L system 1.2.2 Max. Number of Part Systems M70 Type B 1 1 M70 Type A 1 2 M720 1 2 M730 2 4 M750 2 4 M system L system For actual use, the machine tool builder specification will apply. 2 1. Control Axes 1.3 Control Axes and Operation Modes 1.3 Control Axes and Operation Modes 1.3.1 Tape (RS-232C Input) Mode M70 Type B M system L system M70 Type A M720 M730 M750 In this mode, operation is performed using the machining program data from the RS-232C interface built in the NC unit. A paper tape reader must be provided if machining programs on paper tape are to be run. 1.3.

2 Memory Mode M70 Type B M system L system M70 Type A M720 M730 M750 The machining programs stored in the memory of the NC unit are run. 1.3.3 MDI Mode M70 Type B M system L system M70 Type A M720 M730 M750 The MDI data stored in the memory of the NC unit is executed. Once executed, the MDI data is set to the "setting incomplete" status, and the data will not be executed unless the "setting completed" status is established by screen operations. 1.3.4 High-Speed Program Server Mode (CF Card in Control Unit) M70 Type B M70 Type A M720 M730 M750 M system L system The machining program stored in CF card can be operated by installing a CF card in the control unit CF (compact flash) card interface. Machining programs can be copied to CF card with the front IC card or Ethernet on the input/output screen. When a machining program stored in CF card is searched while "DS" is selected for device during operation search, the machining program in CF card can be operated as a main program.
(The operation mode is "memory mode".) Also, when "M198 Pp;" is commanded in the main program, the machining program in CF card can be called and operated as a sub program. Macros such as WHILE, IF and GOTO can be used during high-speed program server mode, as well. Also, calling the sub program and macro program stored in memory or CF card is possible during high-speed program server mode operation. 3 1.
Control Axes 1.3 Control Axes and Operation Modes 1.3.5 Front IC Card Mode M70 Type B M system L system M70 Type A M720 M730 M750 The machining program stored in PCMCIA card can be operated by installing a PCMCIA card on the front of control unit. When a machining program stored in PCMCIA card is searched while "IC" is selected for device during operation search, the machining program in PCMCIA card can be operated as a main program.
(The operation mode is "memory mode".) Also, when "M98 Pp ,Dd;" ("d" for designating a unit) is commanded in the main program, the machining program in PCMCIA card can be called and operated as a sub program. Macros such as WHILE, IF and GOTO can be used during IC card operation, as well. Also, calling the sub program and macro program stored in memory or PCMCIA card is possible during IC card operation. 1.3.6 Hard Disk Mode M70 Type B M70 Type A M720 M730 M750 M system L system The machining program stored in the hard disk can be operated when using a high-resolution type display (a display with a hard disk mounted). When a machining program stored in hard disk is searched while "HD" is selected for device during operation search, the machining program in the hard disk can be operated as a main program. (The operation mode is "memory mode".) Also, when "M98 Pp ,Dd;" ("d" for designating a unit) is commanded in the main program, the machining program in the hard disk can be called and operated as a sub program.

Macros such as WHILE, IF and GOTO can be used during hard disk operation, as well. Also, calling the sub program and macro program stored in memory or the hard disk is possible during hard disk operation. 4 2. Input Command 2.1 Data Increment 2. Input Command 2.1 Data Increment 2.1.1 Least Command Increment Least command increment: 1 µm (Input setting increment 1µm) M70 Type B M system L system M70 Type A M720 M730 M750 Least command increment: 0.

*1 µm (Input setting increment 0.*

*1µm) M70 Type B M system L system M70 Type A M720 M730 M750 Least command increment: 0.01 µm (10nm) (Input setting increment 10nm) M system L system M70 Type B M70 Type A M720 M730 M750 Least command increment: 0.001 µm (1nm) (Input setting increment 1nm) M system L system M70 Type B M70 Type A M720 M730 M750 The data increment handled in the controller includes the input setting increment and command increment. Each type is set with parameters. (1) The input setting increment indicates the increment handled in the internal processing of the controller.*

*The counter and tool compensation data, etc., input from the screen is handled with this increment. This increment is applied per part system (1st to 4th part system, PLC axis). Input setting increment (parameter) 1µm (B) 0.1µm (C) 10nm (D) 1nm (E) Metric unit system Linear axis Rotary axis (Unit = mm) (Unit = °) 0.*

*001 0.001 0.0001 0.0001 0.00001 0.00001 0.000001 0.000001 Inch unit system Linear axis Rotary axis (Unit = inch) (Unit = °) 0.0001 0.001 0.*

*00001 0.0001 0.000001 0.00001 0.0000001 0.000001 (Note) The inch and metric systems cannot be used together. (2) The command increment indicates the command increment of the movement command in the machining program. This can be set per axis. Command increment (parameter) 10 100 1000 10000 Metric unit system Linear axis Rotary axis (Unit = mm) (Unit = °) 0.001 0.*

*001 0.01 0.01 0.1 0.1 1.*

*0 1.0 Inch unit system Linear axis Rotary axis (Unit = inch) (Unit = °) 0.0001 0.001 0.001 0.*

*01 0.01 0.1 0.1 1.0 (Note) The inch and metric systems cannot be used together. 5 2. Input Command 2.1 Data Increment 2.1.2 Least Control Increment The least control increment includes 0.*

*01µm and 0.001µm. These are increments which determine the NC's internal operation accuracy. Least Control Increment 0.01µm (10nm) M70 Type B M system L system M70 Type A M720 M730 M750 Least Control Increment 0.001µm (1nm) M system L system M70 Type B M70 Type A M720 M730 M750 2.1.3 Indexing Increment M70 Type B M70 Type A M720 M730 M750 M system L system This function limits the command value for the rotary axis. This can be used for indexing the rotary table, etc. It is possible to cause a program error with a program command other than an indexing increment (parameter setting value).*

*(Example) When the indexing increment setting value is 2 degrees, only command with the 2-degree increment are possible. G90 G01 C102. 000 ; G90 G01 C101. 000 : G90 G01 C102 ; ..*

*. Moves to the 102 degree angle. ...*

*Program error ... Moves to the 102 degree angle. (Decimal point type II) 6 2. Input Command 2.2 Unit System 2.2 Unit System 2.2.1 Inch/Metric Changeover M70 Type B M system L system M70 Type A M720 M730 M750 The unit systems of the data handled in the controller include the metric unit system and inch unit system.*

*The unit (inch/mm) for the setting and display, as well as for the handle/incremental feed can be switched with either the parameters or machining program (G20/G21 command). An option is required when the unit is switched with the machining program command. Unit system Metric unit system Inch unit system Length data 1.0 1.0 Meaning 1.0 mm 1.0 inch (Note) For the angle data, 1.0 means 1 degree (°) regardless of the unit system. Data Parameter Machining program Screen data (Compensation amount, user parameter, counter, etc.) / Feedrate of handle, etc.*

*Machine parameter / PLC I/F machine position, etc. 0 A 1 B 0 1 G20 G21 G20 G21 Inch unit system Metric unit system Inch unit system Metric unit system Metric unit system Not affected Inch unit system Not affected Metric unit system Inch unit system Not affected (Note 1) The parameter changeover is valid after the power is turned ON again. (Note 2) The unit system for the PLC axis can be switched with a parameter different from the one used with the NC axis. The PLC axis unit system cannot be switched with the machining program (G20/G21 command). (Note 3) When the power is turned ON or resetting is performed, the command increment depends on the parameter setting.*

*7 2. Input Command 2.2 Unit System 2.2.2 Input Command Increment Tenfold M70 Type B M system L system M70 Type A M720 M730 M750 - The program's command increment can be multiplied by an arbitrary scale with the parameter designation.*

*This function is valid when a decimal point is not used for the command increment. For example, when running a machining program already created with a 10µm input command increment with a CNC unit for which the command increment is set to 1µm and this function's parameter value is set to "10", machining similar to before this function is possible. The scale is set with the parameters. (Note 1) This function cannot be used for the dwell function G04_X_(P_);. (Note 2) This function cannot be used for the compensation amount of the tool offset input. (Note 3) This function can be used when decimal point type I is valid, but cannot be used when decimal point type II is valid. 8 2. Input Command 2.3 Program Format 2.3 Program Format 2.*

*3.1 Program Format The G-code of L system is selected by parameter. This specification manual explains the G function with G-code list 3 as standard. 2.3.1.1 Format 1 for Lathe (G-code List 2, 3) M system L system M70 Type B M70 Type A M720 M730 M750 - 2.3.1.2 Format 2 for Lathe (G-code List 4, 5) M system L system M70 Type B M70 Type A M720 M730 M750 - 2.*

*3.1.3 Special Format for Lathe (G-code List 6, 7) M system L system M70 Type B M70 Type A M720 M730 M750 - 2.3.1.*

*4 Format 1 for Machining Center (G-code List 1) M70 Type B M system L system M70 Type A M720 M730 M750 - 2.3.1.5 Format 2 for Machining Center (M2 Format) M70 Type B M system L system M70 Type A M720 M730 M750 - 2.3.*

*1.6 MITSUBISHI CNC Special Format M system L system M70 Type B M70 Type A M720 M730 M750 - The formats of the turning fixed cycles (G77 to G79), multiple repetitive turning fixed cycles (G71 to G76) and drilling fixed cycles (G80 to G89) can be switched to the MITSUBISHI CNC special formats. 9 2. Input Command 2.4 Command Value 2.4 Command Value 2.4.1 Decimal Point Input I, II M70 Type B M system L system M70 Type A M720 M730 M750 There are two types of the decimal point input commands and they can be selected by parameter. (1) Decimal point input type I When axis coordinates and other data are issued in machining program commands, the assignment of the program data can be simplified by using the decimal point input. The minimum digit of a command not using a decimal point is the same as the least command increment.*

*Usable addresses can be applied not only to axis coordinate position but also to speed commands and dwell commands. The decimal point position serves as the millimetre unit in the metric mode, as the inch unit in the inch mode and as the second unit in a time designation of dwell command. (2) Decimal point input type II As opposed to type I, when there is no decimal point, the final digit serves as the millimetre unit in the metric mode, as the inch unit in the inch mode and as the second unit in the time designation.*

*The "." (point) must be added when commands below the decimal point are required. Unit interpretation (for metric system) G00 G1 G2 G4 G4 X100. Y-200.5 X100 F20. Y200 F100 (*1) X1.5 X2 Type I X100mm, Y-200.*

*5mm X100µm, F20mm/min Y200µm, F100mm/min Dwell 1.5 s 2ms Type II X100mm, F20mm/min Y200mm, F100mm/min 2s (*1) The F unit is mm/min for either type (inch system : inch/min). 10 2. Input Command 2.4 Command Value 2.*

*4.2 Absolute/Incremental Command M70 Type B M system L system M70 Type A M720 M730 M750 (1) M system When axis coordinate data is issued in a machining program command, either the incremental command method (G91) that commands a relative distance from the current position or the absolute command method (G90) that moves to a designated position in a predetermined coordinate system can be selected. The absolute and incremental commands can be both used in one block, and are switched with G90 or G91. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations. G90 .*

*.. Absolute command (absolute command) G91 ... Incremental command (incremental command) These G codes can be commanded multiple times in one block. Example G90 X100. Absolute position G91 Y200. Incremental position G90 Z300. Absolute position ; (Note 1) As with the memory command, if there is no G90/G91 designation in the MDI command, the previously executed modal will be followed.*

*(Incremental command) G 91 X 100. Y100. ; End point Y100. Y100. Y100. End point (Absolute command) G 90 X 100. Y100. ; Current position X 100. Current position Program coordinate (0, 0) X 100. (0, 0) X100.*

*11 2. Input Command 2.4 Command Value (2) L system When axis coordinate data is issued in a machining program command, either the incremental command method that commands a relative distance from the current position or the absolute command method that moves to a designated position in a predetermined coordinate system can be selected. When issuing an incremental command, register the axis address to be commanded as the incremental axis name in the parameter. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.*

*Absolute command (absolute command) ... X, Z Incremental command (incremental command) ..*

*. U, W Example G00 X100. Absolute position W200. Incremental position ; (Incremental command) (Absolute command) G 00 U u1 W w1 ; X Current position G 00 X x1 Z z1 ; X End point Current position u1 2 x1 Z End point w1 z1 Z (0,0) The above drawing shows the case for the diameter command. The above drawing shows the case for the diameter command. (Note) Absolute command and incremental command can be switched by the parameter. In addition to the command method using the axis addresses as indicated above, a command method using G code (G90/G91) may be selected. 12 2. Input Command 2.4 Command Value 2.*

*4.3 Diameter/Radius Designation M70 Type B M70 Type A M720 M730 M750 - M system L system For axis command value, the radius designation or diameter designation can be changed over with parameters. When the diameter designation is selected, the scale of the length of the selected axis is doubled. (For instance, an actual length of 1 mm will be treated as 2 mm.) This function is used when programming the workpiece dimensions on a lathe as diameters. Changing over from the diameter designation to the radius designation or vice versa can be set separately for each axis. X-axis radius designation X-axis diameter designation X X u4 x6 u4 x6 Z Coordinate zero point Coordinate zero point Z The difference in the diameter designation and radius designation is shown below. Absolute command Radius designation Diameter designation Actual movement amount = x1 Actual movement amount = 2 x1 Incremental command Radius designation Actual movement amount = u1 Diameter designation Actual movement amount = 2 u1 13 3. Positioning/Interpolation 3.1 Positioning 3.*

*Positioning/Interpolation 3.1 Positioning 3.1.1 Positioning M70 Type B M system L system M70 Type A M720 M730 M750 This function carries out high-speed positioning with rapid traverse rate, following the movement command given in a program. G00 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously) Xx1, Yy1, Zz1: Position data The above command positions the tool by rapid traverse.*

*The tool path takes the shortest distance to the end point in the form of a straight line. For details on the rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate". Since the actual rapid traverse feed rate depends on the machine, refer to the specifications of the machine concerned. (1) The rapid traverse feed rate for each axis can be set independently with parameters. (2) The number of axes which can be driven simultaneously depends on the specifications (number of simultaneously controlled axes).*

*The axes can be used in any combination within this range. (3) The feed rate is controlled within the range that it does not exceed the rapid traverse rate of each axis and so that the shortest time is taken. (Linear type) Parameter setting enables movement at the rapid traverse rates of the respective axes independently for each axis. In this case, the tool path does not take the form of a straight line to the end point. (Non-Linear type) (Example) (Example) Linear type (Moves lineary to the end point.) G 00 G 91 X 100. Y 100. ; Y End point Non-linear type (Each axis moves at each parameter speed.) G 00 G 91 X 100. Y 100.*

*; Y End point 100. 100. Current position 100. Current position 100. X X (4) The tool is always accelerated at the start of the program command block and decelerated at the end of the block. 14 3. Positioning/Interpolation 3.1 Positioning 3.1.2 Unidirectional Positioning M70 Type B M system L system M70 Type A M720 M730 M750 - The G60 command always moves the tool to the final position in a direction determined with parameters.*

*The tool can be positioned without backlash. G60 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously) Xx1, Yy1, Zz1: Position data With the above command, the tool is first moved to a position distanced from the end point by an amount equivalent to the creep distance (parameter setting) with rapid traverse and then moved to its final position. For details on the rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate". Since the actual rapid traverse feed rate depends on the machine, refer to the specifications of the machine concerned. Positioning to the final point is shown below (when this positioning is in the "+" direction.*

*) + (Example) G60 G91 X100. Y100. ; Interim point End point Y100. 1. The rapid traverse rate for each axis is the value set with parameters as the G00 speed.*

2. The vector speed to the interim point is the value produced by combining the distance and respective speeds. 3. The creep distance between the interim and end points can be set independently for each axis by parameters. Current position X100. (Note 1) The processing of the above pattern will be followed even for the machine lock and Z-axis command cancel. (Note 2) On the creep distance, the tool is moved with rapid traverse. (Note 3) G60 is valid even for positioning in drilling in the fixed cycle. (Note 4) When the mirror image function is on, the tool will be moved in the reverse direction by mirror image as far as the interim position, but operation over the creep distance with the final advance will not be affected by the mirror image. 15 3.

Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.2 Linear/Circular Interpolation 3.2.1 Linear Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system Linear interpolation is a function that moves a tool linearly by the movement command value supplied in the program at the cutting feed rate designated by the F code. G01 Xx1 Yy1 Zz1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously) Xx1, Yy1, Zz1: Position data Ff1 : Feed rate data Linear interpolation is executed by the above command at the f1 feed rate. The tool path takes the shortest distance to the end point in the form of a straight line. For details on the f1 command values for NC, refer to the section entitled "Cutting Feed Rate". Since the actual cutting feed rate depends on the machine, refer to the specifications of the machine concerned. (Example) G01 G91 X100.
Y100. F120 ; Y End point Feed rate (120mm/min) 1. The cutting feed rate command moves the tool in the vector direction. 2. The component speeds of each axis are determined by the proportion of respective command values.
100. (85mm/min) Current position 100. (85mm/min) X (1) The number of axes which can be driven simultaneously depends on the specifications (number of simultaneously controlled axes). The axes can be used in any combination within this range. (2) The feed rate is controlled so that it does not exceed the cutting feed rate clamp of each axis.
(3) When a rotary axis has been commanded in the same block, it is treated as a linear axis in degree(°) units (1° = 1mm), and linear interpolation is performed. 16 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.2.2 Circular Interpolation (Center/Radius Designation) M70 Type B M70 Type A M720 M730 M750 M system L system (1) Circular interpolation with I, J, K commands This function moves a tool along a circular arc on the plane selected by the plane selection G code with movement command supplied in the program. G02(G03) Xx1 Yy1 Ii1 Jj1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W) G02, G03 Xx1, Yy1 Ii1, Jj1 Ff1 : Arc rotation direction : End point coordinate : Arc center : Feed rate The above commands move the tool along the circular arc at the f1 feed rate. The tool moves along a circular path, whose center is the position from the start point designated by distance "i1" in the X-axis direction and distance "j1" in the Y-axis direction, toward the end point. The direction of the arc rotation is specified by G02 or G03. G02: Clockwise (CW) G03: Counterclockwise (CCW) The plane is selected by G17, G18 or G19.

G17: XY plane G18: ZX plane G19: YZ plane (Example) See below for examples of circular commands. Y Start point F G03 Z Y G17 G02 G03 X G19 G02 X G18 G02 G03 Z I, J Center End point Y X (a) (b) (c) (d) The axes that can be commanded simultaneously are the two axes for the selected plane. The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle. Circular interpolation can be commanded within a range extending from 0° to 360°. The max. value of the radius can be set up to six digits above the decimal point. (Note 1) The arc plane is always based on the G17, G18 or G19 command. If a command is issued with two addresses which do not match the plane, an alarm will occur. (Note 2) The axes configuring a plane can be designated by parameters. Refer to the section entitled "Plane Selection".
17 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation (2) R-specified circular interpolation Besides the designation of the arc center coordinates using the above-mentioned I, J and K commands, arc commands can also be issued by designating the arc radius directly. G02(G03) Xx1 Yy1 Rr1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W ) G02, G03 Xx1, Yy1 Rr1 Ff1 : Arc rotation direction : End point coordinate : Arc radius : Feed rate G02 or G03 is used to designate the direction of the arc rotation. The arc plane is designated by G17, G18 or G19.
The arc center is on the bisector which orthogonally intersects the segment connecting the start and end points, and the point of intersection with the circle, whose radius has been designated with the start point serving as the center, is the center coordinate of the arc command. When the sign of the value of R in the command program is positive, the command will be for an arc of 180° or less; when it is negative, it will be for an arc exceeding 180°. (Example) G02 G91 X100. Y100. R100.
F120 ; Y Feed rate: 120mm/min Arc end point (X, Y) R100. Current position (arc start point) X (a) The axes that can be commanded simultaneously are the two axes for the selected plane. (b) The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle. (Note 1) The arc plane is always based on the G17, G18 or G19 command. If a command is issued with two addresses which do not match the plane, an alarm will occur. 18 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.2.3 Helical Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system With this function, any two of three axes intersecting orthogonally are made to perform circular interpolation while the third axis performs linear interpolation in synchronization with the arc rotation.

This simultaneous 3-axis control can be exercised to machine large-diameter screws or 3-dimensional cams. G17 G02(G03) Xx1 Yy1 G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Pp1 Ff1 ; (Specify arc center) Zz1 Rr1 Ff1 ; (Specify arc radius "R") G17 G02, G03 Xx1, Yy1 Zz1 Ii1, Jj1 Pp1 Ff1 Rr1 : Arc plane : Arc rotation direction : End point coordinate values for arc : End point coordinate value of linear axis : Arc center coordinate values : Number of pitches : Feed rate : Arc radius (1) The arc plane is designated by G17, G18 or G19. (2) G02 or G03 is used to designate the direction of the arc rotation. (3) Absolute or incremental values can be assigned for the arc end point coordinates and the end point coordinates of the linear axis, but incremental values must be assigned for the arc center coordinates.

(4) The linear interpolation axis is the other axis which is not included in the plane selection. (5) Command the speed in the component direction that represents all the axes combined for the feed rate. Pitch l1 is obtained by the formula below. l1 = z1/((2 · p1 + )/2) = e  s = arctan (ye/xe)  arctan (ys/xs) (0 < 2) Where xs, ys are the start point coordinates xe, ye are the end point coordinates The combination of the axes which can be commanded simultaneously depends on the specifications. The axes can be used in any combination under the specifications. The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

19 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation (Example) G91 G17 G02 X0. Y200. Z100.

I100. J100. Z Command program path Y End point End point X W J100 I-100 Start point Y Start point X XY plane projection path in command program (Note 1) Helical shapes are machined by assigning linear commands for one axis which is not a circular interpolation axis using an orthogonal coordinate system. It is also possible to assign these commands to two or more axes which are not circular interpolation axes. Z When a simultaneous 4-axis command is used with the V axis as the axis parallel to the Y axis, helical interpolation will be carried out for a cylinder which is inclined as shown in the figure on the right. In other words, linear interpolation of the Z and V axes is carried out in synchronization with the circular interpolation on the XY plane. V ·End point X · point Start Y 20 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.2.4 Spiral/Conical Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system - - - - This function interpolates arcs where the start point and endpoint are not on the circumference of the same circle into spiral shapes. There are two types of command formats which can be changed with the parameters. (1) For command format type 1 (a) Spiral interpolation G17 G02.1(G03.1) Xx1 Yy1 Ii1 Jj1 Pp1 Ff1 ; G17 G02.

1, G03.1 Xx1, Yy1 Ii1, Jj1 Pp1 Ff1 : Arc plane : Arc rotation direction : End point coordinate : Arc center : Number of pitches : Feed rate The circular interpolation operation is performed at the feed rate f1 by the commands listed above. The tool draws a spiral arc path whose center is at the position from the start point which is designated by distance i1 for the X-axis direction and distance j1 for the Y-axis direction as the tool moves toward the end point. The arc plane is designated by G17, G18 or G19. G17: XY plane G18: ZX plane G19: YZ plane The direction of the arc rotation is designated by G02.1 or G03.1. G02.1: Clockwise (CW) G03.1: Counterclockwise (CCW) The number of pitches (number of rotations) is designated by p1.

By assigning zero to p1, the pitch designation can be omitted in this case, the interpolation is obtained as a spiral rotation of less than one full turn. Assigning 1 to p1 yields a spiral rotation of more than one full turn but less than two full turns. Y Example: G91 G17 G01 X60. F500 ; Y140. ; G2.

1 X60. Y0 I100. P1 F300 ; G01 X-120 ; G90 G17 G01 X60. F500 ; Y140.; G2.

1 X120. Y140. I100. P1 F300 ; G01 X0 ; 140. End point Start point Center X60. I100. W 60. 160. X · The combination of the axes which can be commanded simultaneously depends on the specifications. Any combination can be used within the specified range.

· The feed rate is a constant tangential rate. (Note 1) This function cannot be used in combination with a tool radius compensation command (G41, G42). (Note 2) The arc plane is always based on the G17, G18 or G19 command. Arc control is performed on a plane by the G17, G18 or G19 command even when two addresses which are not on the selected plane are designated. 21 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation (b) Conical interpolation When an axis other than the ones for the spiral interpolation plane has been designated at the same time, the other axis will also be interpolated in synchronization with the spiral interpolation. G17 G91 G02.1 X100. Z150.

I150. P3 F500 ; In the example given above, truncated cone interpolation is performed. Y XY plane X W Z Z XZ plane X X W (2) For command format type 2 (a) Spiral interpolation G17 G02(G03) Xx1 Yy1 Ii1 Jj1 Qq1/Ll1 Ff1 ; G17 G02, G03 Xx1, Yy1 Ii1, Jj1 Qq1 Ll1 Ff1 : Arc plane : Arc rotation direction : End point coordinate : Arc center : Incremental/decremental amount of radius per spiral rotation : Number of pitches : Feed rate · Relation between Q and L L = (arc end point radius  arc start point radius) / Q · Q takes precedence if both Q and L have been designated at the same time. (b) Conical interpolation G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Kk1 /Qq /Ll1 Ff1 ; G17 : Arc plane G02, G03 : Arc rotation direction Zz1 : End point coordinate in height direction Ii1, Jj1 : Arc center Kk1 : Amount by which height is incremented or decremented per spiral rotation Qq1 : Amount by which radius is incremented or decremented per spiral rotation Ll1 : Number of pitches Ff1 : Feed rate · Relation between L and (I, J) K L = | Height | / | Amount by which height is incremented or decremented (I, J, K) | · Q takes precedence over K which in turn takes precedence over L if Q, K and L have been designated at the same time. · The tolerable error range (absolute position) for when the commanded end point position is deviated from the end point position obtained from the number of pitches and increment/decrement amount is set with the parameters.

22 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.2.5 Cylindrical Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system This function transfers the shape that is on the cylinder's side surface (shape yielded by the cylindrical coordinate system) onto a plane, and when the transferred shape is designated in the program in the form of plane coordinates, the shape is converted into a movement along the linear and rotary axes of the original cylinder coordinates, and the contours are controlled by means of the CNC unit during machining.

Since the programming can be performed for the shapes produced by transferring the side surfaces of the cylinders, this function is useful when it comes to machining cylindrical cams and other such parts. This function can be used only with the G code list 6 or 7. Program coordinate plane Z axis C axis X axis C axis Cylindrical interpolation machining Z axis Cylinder radius value (1) Cylindrical interpolation mode start (G07.1 name of rotary axis cylinder radius value;) Cylindrical interpolation is performed between the rotary axis designated in the G07.1 block and any other linear axis. (a) Linear interpolation or circular interpolation can be designated in the cylindrical interpolation mode.

*However, assign the G19 command (plane selection command) immediately before the G07.1 block. (b) The coordinates can be designated with either absolute command or incremental command. (c) Tool radius compensation can be applied to the program commands.*

*Cylindrical interpolation is performed for the path after tool radius compensation. (d) For the feed rate, designate a tangential rate over the cylinder transfer surface using the F command. The F rate is in either mm/min or inch/mm units. (2) Cylindrical interpolation mode cancel (G07.1 name of rotary axis 0;) If "C" is the name of the rotary axis, the cylindrical interpolation cancel mode is established with the command below. G07.1 C0 ; 23 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.2.*

*6 Polar Coordinate Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system This function converts the commands programmed by the orthogonal coordinate axes into linear axis movements (tool movements) and rotary axis movements (workpiece rotation) to control the contours. It is useful for cutting linear cutouts on the outside diameter of the workpiece, grinding cam shafts, etc. This function can be used only with the G code list 6 or 7. X axis C axis Z axis Hypothetical axis Polar coordinate interpolation plane (G17 plane) (1) Polar coordinate interpolation mode (G12.1) The polar coordinate interpolation mode is established by designating the G12.*

*1 command. Polar coordinate interpolation plane consists of a linear axis and a hypothetical axis, which are at right angles to each other. Polar coordinate interpolation is performed on this plane. (a) Linear interpolation and circular interpolation can be designated in the polar coordinate interpolation mode. (b) Either absolute command or incremental command can be issued.*

*(c) Tool radius compensation can be applied to the program commands. Polar coordinate interpolation is performed for the path after tool radius compensation. (d) For the feed rate, designate a tangential rate on the polar coordinate interpolation plane (orthogonal coordinate system) using the F command. The F rate is in either mm/min or inch/mm units. (2) Polar coordinate interpolation cancel mode (G13.1) The polar coordinate interpolation cancel mode is established by designating the G13.1 command. 24 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.*

*2.7 Milling Interpolation M70 Type B M70 Type A M720 M730 M750 - M system L system When a lathe with linear axes (X, Z axes) and rotary axis (C axis) serving as the control axes is to perform milling at a workpiece end face or in the longitudinal direction of the workpiece, this function uses the hypothetical axis Y which is at right angles to both the X and Z axes to enables the milling shape to be programmed as the X, Y and Z orthogonal coordinate system commands. With this function, the workpiece can be treated as a cylinder with radius X, and commands can be designated on the plane formed by transferring the cylinder side surface instead. With milling interpolation, the commands programmed by the orthogonal coordinate system are converted into linear axis and rotary axis movements (workpiece rotation) to control the contours. X Z C Y (Hypothetical axis) G12.1 ; Milling mode ON G13.1 ; Milling mode OFF (Turning mode) G16 (Y-Z cylindrical plane) G17 (X-Y plane) G19 (Y-Z plane) X X X Z Y Z Z Y Plane on which radius X cylinder is developed. Select this to machine the cylindrical plane of a workpiece. X-Y plane in XYZ orthogonal coordinate system. Select this to machine the workpiece end face.*

*Y Y-Z plane in XYZ orthogonal coordinate system. Select this to machine a plane of a cylinder cut in the longitudinal direction. 25 3. Positioning/Interpolation 3.2 Linear/Circular Interpolation 3.*

*2.8 Hypothetical Axis Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system - - - Take one of the axes of the helical interpolation or spiral interpolation, including a linear axis, as a hypothetical axis (axis with no actual movement) and perform pulse distribution. With this procedure, an interpolation equivalent to the helical interpolation or spiral interpolation looked from the side (hypothetical axis), or SIN or COS interpolation, will be possible. The setting of this hypothetical axis is commanded with G07. G07 G07 Y0 Y1 ; ; X axis command cancel ON X axis command cancel OFF G07 Y : Hypothetical axis interpolation command : Designate the axis for which hypothetical axis interpolation is performed Designation of the axis for which axis command cancellation is performed applies for all the NC axes.*

*(0: Cancel (normal), 1: Handle as hypothetical axis) (1) Interpolation functions that are used for hypothetical interpolation are helical interpolation and spiral interpolation. (2) During G070; to G071;, axis will be the hypothetical axis. Thus, when axis is commanded independently during this time, dwell mode will be held until finishing the pulse distribution to the hypothetical axis. Y Y (Example) G07 Z0 ; G18 G02 X50.Z0.Y100. K30. P3 ; X X (Note) In order to perform hypothetical axis interpolation, helical interpolation must be added. 26 3. Positioning/Interpolation 3.*

*3 Curve Interpolation 3.3 Curve Interpolation 3.3.2 Exponential Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system With this function, the rotary axis movement is changed into exponential functions vis-a-vis the linear axis movements. When exponential function interpolation is performed, linear interpolation is performed between the other axes and the linear axis. This makes it possible to machine tapered grooves (regular helix machining of tapered shapes) whose helix angle is always constant. The function can be used for slotting and grinding end mills and other tools. [Regular helix machining of tapered shapes] Z axis (G00) (G01) (G01) (G02.3/G03.3) A axis (rotary axis) J1 J2 J3 X axis (linear axis) Helix angle : J1=J2=J3 [Relationship between linear and rotary axes] X axis (linear axis) X=B (e -1) {B, C = constants} CA A axis (rotary axis) 27 3.*

*Positioning/Interpolation 3.3 Curve Interpolation 3.3.3 Spline Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system - - - This function automatically generates spline curves that smoothly pass through rows of dots designated by a fine-segment machining program, and performs interpolation for the paths along the curves. This enables high-speed and high-accuracy machining to be achieved.*

*To use this function, the high-accuracy control function 1 (G08P1) is required. 3.3.4 NURBS Interpolation M70 Type B M70 Type A M720 M730 M750 M system L system - - This function realizes NURBS curve machining by commanding NURBS curve parameters (number of stages, weight, knot, control point). The path does not need to be replaced with fine segments.*