

CNC System Specifications TOSNUC999

Standard Specifications including pack specifications (Items marked with ☆ are included in the pack specifications.)

A. Controlled axes

☆Number of controlled axes	4 axes (X, Y, Z and W)
Simultaneously controllable axes	3 axes for positioning (G00) and linear interpolation (G01) 2 axes [X-Y, Y-Z (W), and Z (W)-X] for circular interpolation (G02, G03)
☆Synchronized operation	WM and WS axes

B. Programming Methods

Programming resolution	Linear axes: 0.001 mm
Maximum programmable dimension	Linear axes ±99999.999 mm (±9999.9999 in.)
Data code	Automatic recognition of ISO/EIA JIS B6311 ISO 6983/1 EIA RS-358-B EIA RS-244-B
Data format	Variable block with decimal point programming Word address format
Absolute/incremental programming	G90/G91
Decimal point input	Calculator type/programming resolution type

C. Interpolation

Positioning	A positioning operation is performed at a rapid traverse rate by executing G00 command.
Linear interpolation	A linear interpolation is performed by executing G01 command.
Circular interpolation	A circular interpolation is performed by executing G02/G03 command.

D. Feed

Rapid traverse rate	(See the section of machine specifications for numerical values)
Feedrate	F 5-digit direct programming in mm per minute
Dwell G04	Dwell time can be specified in the range of 0 to 999.99 seconds with F (or P) code.

Continuous jog feed
A selected axis is continuously moved at a rapid traverse rate or cutting feedrate by manual operation.

Rapid traverse override
Rapid traverse can be overridden in the range of 0 to 100 % in 10 % increments.

Feedrate override
Specified feedrate can be overridden in the range of 0 to 200 % (stepless).

Automatic acceleration or deceleration
Linear acceleration or deceleration can be effective on rapid traverse rate and jog feedrate.

Automatic acceleration or deceleration for cutting feed
Linear acceleration or deceleration can be effected on feedrate by executing G08/G09 or G50/G51 command.

S-type acceleration or deceleration for rapid traverse
S-type acceleration or deceleration can be effected on rapid traverse.

☆Thread cutting
Thread cutting synchronized with spindle revolution can be performed by executing G33 command.

☆Feed per revolution/feed per minute
Feed per minute or feed per revolution can be selected for the feedrate specified by F code by executing G94/G95 command.

☆Dwell per revolution
Dwell per revolution can be performed by executing G05 command.

☆Handwheel feed with portable type box 0.001, 0.01 or 0.1 mm/division
☆Thread-cutting initiation at optional angle
Thread cutting is initiated at the optional angle of Spindle.

E. Part Program Storage and Edit

☆The part program storage capacity of 600 m [1970 ft/approx. 285 kB] (the numbers of registrable programs: 1512) (The customer's storage capacity is decreased by approximately 100 to 200 m which is used for the selected optional functions by manufacturer.)

Program edit (Background edit)

Various editing operations are possible for programs memorized. Program delete, program copy, program rename, search, jump, cancel, deletion and copy by designating range, replace, program insertion, register in abbreviation, line mark designation, sequence number change, space insertion between words, program input in S. F. auto mode, simultaneous opening of two programs, editing of various data during program editing, undo of deleted program, English comment

Program name

A program name is specified with the alphanumeric characters up to eight characters following address \$ or O.

Program comment of maximum 32 characters can be included.

Sequence number

A sequence number is specified with a 5-digit number following address N.

Sequence number search

A block containing specified sequence number can be searched.

Programs nesting list

A list of program nesting is displayed.

Program offset list

List of the following data is displayed after searching from the head of program.
Fixture offset

Syntax check

The format of a program is checked.

F. Operation and Display

☆Operation panel 10.4 in. color TFT liquid crystal display
Keyboard with membrane switches (80 keys)

Customized key

Registering a series of key-in operations to be performed very often into a key, the operation can be performed quickly, thus improving efficiency of the operation.

Tool file

Tool data such as tool length, cutter diameter and tool offset can be collectively displayed in a unified manner and edited.

Display capabilities

Commanded values, current positions, compensation values or the like are displayed on the main screen, window screen and sub screen respectively.

Screen erasing function

A screen is erased when a key-in operation is not performed in specified time or more.

Operation Automatic operation, MDI operation, and manual numerical command are possible.

SF manual setting S and F codes can be set in the manual mode.

SF automatic setting S and F codes are set automatically in the manual mode.

Spindle motor load display The power consumption of spindle drive motor is displayed.

Run hours display The NC working time is displayed.

Machined workpiece counting The number of machined workpiece is counted when the M code specified in the parameter is executed.

Calendar timer

A calendar and time are displayed on the run hour display screen. Program creation dates and time are displayed in the program list.

Machining record

A history such as machining start time, actual machining time or the like which is recorded after operating a machine in the automatic mode by executing a program are displayed.

User name registration

A user name is displayed at the time of a system startup.

Memory operation

A program in the NC memory is executed in an automatic mode.

MDI operation

In MDI mode, a program constituted of a plurality of blocks can be inputted and executed.

G. I/O Function and Device

RS232C interface port A

NC programs, tool offset data or the like can be inputted and outputted via the EIA RS232C interface.

☆User media

Various data such as NC programs or tool offset data can be input and output via a USB memory or a CF card slot. Memory slots are arranged on the right side of the pendant operation box.

H. S, T and M functions

Spindle-speed function (S function)

Spindle speed can be specified with a five (5)-digits integer following address S.
Spindle speed override 50 to 200 % in 10 % increments.

Tool function (T function)

Tool number can be specified with a six(6)-digits integer following address T.
Miscellaneous function (M function)
Miscellaneous function can be specified with a four (4)-digits integer following address M.

I. Tool Offset

Tool length offset

Tool length offset can be set to an axis perpendicular to the selected plane by executing G43/G44/(G49) command.

Tool offset

Tool extension or contraction is effected on an axis in selected plane by executing G45/G46/G47/G48 command.

Cutter compensation type C

Cutter compensation is effected on an axis in selected plane by executing G40/G41/G42 command.

☆Number of tool offsets The number of tool length offsets: 899 sets
The number of cutter compensations: 899 sets

J. Coordinate System

Automatic reference point return G28: Automatic return to a reference point
G29: Return from a reference point
G20: Reference point return check

Coordinate system setting

A coordinate system can be set so that the current positions of axes constitute commanded coordinate values by executing G92 command.

☆Fixture offset 99 sets (H901-H999)
Fixture offset becomes effective by executing G53/G57 command.

Fixture offset 2

Fixture offset becomes effective by executing G54/G55/G56 command.

2nd to 4th reference point return

Axes are returned to the 2nd to 4th reference points automatically by executing G21 command.

K. Operation Support Function

Control in/out

Information inserted in the section between control out and control in codes can be neglected.

Single block

A program can be executed block by block during automatic or MDI operation.

Optional stop

A program is stopped at a block containing M01 code in a state that M01 push button switch is ON.

☆Optional block skip

A block containing a slash (/) code at the top there of can be skipped.
3 pcs

Dry run

Axes moves at the feedrate set to the parameter in place of the programmed feedrate.

Machine lock

Output of axis command pulse to the machine side is suspended.

Miscellaneous function lock

M, S, and T commands are not outputted to the machine side.

Z-axis feed cancel

Output of Z-axis command pulse to the machine side is suspended.

Manual absolute ON/OFF

Whether a travel of an axis moved by manual operation is added to the current coordinate value is selected according to the status of the absolute ON/OFF switch.

Override cancel

An override for a feedrate and spindle speed is ignored to clamp the feedrate and spindle speed at 100 % by executing M48 or M49 command.

All clear

The NC internal memory can be initialized by pressing the ALL CLEAR push-button switch.

Reset A command currently executed can be reset.

Feed hold

Axis travel can be suspended by pressing the FEED HOLD push-button switch in automatic and MDI operation.

Cycle stop

Axis travel and spindle rotation can be suspended by pressing CYCLE STOP push-button switch in automatic and MDI operation.

Restart

When machining is interrupted in automatic operation, the operation can be resumed from the specified block after taking required steps such as tool changing.

Sequence number collation and stop

Operation can be stopped after executing the block preceding a block with the specified sequence number.

Manual numerical command

Data can be inputted and executed in manual-operation mode.
Data that can be inputted : G00/01, F, M, S, T, axis data (incremental)

Single block cancel

Single block ON or OFF in the single operating mode can be selected by executing G990/G991 command.

Feed hold cancel

Feed hold ON or OFF can be selected by executing G992/G993 command.

Override cancel

Feedrate override ON or OFF can be selected by executing G994/G995 command.

Handwheel feed interruption cancel

Handwheel feed interruption ON or OFF can be selected by executing G996/G997 command.

Manual interruption and manual return

A function for interposing manual operations in automatic operation. After each axis is moved in a manual interruption mode, the axis is automatically returned to the position before interrupting by pressing manual return push-button switch.

☆Handwheel feed interruption

Interruption by means of a MPG handwheel is permitted in cutting feed operation.

☆Manual tool length and tool diameter measurement

An offset value of a tool to be measured from the calibration tool is measured and memorized as the data of specified offset number.

L. Programming Support Function

Plane selection

A machined plane can be selected by executing G17/G18/G19 command.

Circular interpolation by radius programming

Radius of an arc can be specified directly by R command.

Circle cutting G12, G22: Inner circle cutting CW
G13, G23: Inner circle cutting CCW
G222: Outer circle cutting CW
G223: Outer circle cutting CCW

Machine coordinate system position command

Axes can be moved to a position on the coordinates unique to the machine by executing G73 command.

Subprogram call
A subprogram stored in the memory can be called and executed by commanding G72. The subprogram name should be specified with eight (8) alphanumeric characters following address \$ or O.

Arbitrary angle chamfering/corner R
Arbitrary angle chamfering or corner R can be inserted between two (2) consecutive blocks including cutting feed commands.

Canned cycle
A drilling canned cycle can be executed by commanding each of G77 to G89.

Automatic corner override
Automatic override for inside corner/Change of inside arc cutting speed

☆**Programmable mirror image**
Mirror image can be set for each axis by executing G62 or G66 command.

☆**Plane conversion**
A NC program based on G17 plane can be executed on the other planes by executing each of G35 to G39 commands.

☆**Macro program**
A macro program can be called and runs by executing a command such as G72, G74, G75 or G76.

☆**Pattern cycle**
Regularly arranged hole position pattern cycle and milling pattern cycle can be executed by commanding each of G109 to G119 or G121 to G132.

☆**Coordinate conversion**
A parallel shift and rotation of a coordinate system can be made by executing G10/G11 command.

☆**Three-dimensional coordinate conversion**
A parallel shift and rotation of a coordinate system can be made in three dimensions by executing G14 command.

☆**Spindle C-Axis control**
Rotational position control of Spindle can be performed using a hypothetical rotational-axis command.

M. Mechanical error compensation

Backlash compensation Backlash of the machine is compensated.
Pitch error compensation Pitch error of the machine is compensated.

Unidirectional positioning
Final positioning of axes can always be performed from one direction by executing G60 command.

☆**Pitch-error gradient compensation**
Pitch error of each axis feed screw of mechanical system can be compensated linearly by approximating with up to thirty (30) straight lines per axis.

☆**Straightness compensation**
Straightness of the mechanical system can be compensated linearly by approximating with up to nine (9) straight lines per axis.

N. Machine Control Support Function

Axis interlock
Feed of each axis can be prohibited or permitted in response to the external signal.

☆**External feedrate-reduction function**
Feedrate is slowed down in response to the external signal.

O. Safety and maintenance

Emergency stop
The machine comes to an emergency stop by pressing the EMERGENCY STOP push-button switch. Emergency stop can be canceled by pressing the EMG RESET push-button switch.

Overtravel check
The machine comes to an emergency stop in response to the external overtravel signal.

Stroke check
Axis travel exceeding the predetermined stroke is prohibited.

Interference check II
Axis travel into the predetermined interference zone is prohibited by executing G26/G27 command.

Self-diagnosis
A malfunction of NC program, NC system, servo and mechanical system can be monitored, and relevant alarm is processed.

☆**Interference check I**
Axis travel into the predetermined interference zone is prohibited by executing G26/G27 command.

☆**Door interlock**
An interlock function which shuts off the primary power supply when the control-cabinet door is opened.

P. Installation requirements

Power source AC 200/220 V +10 % ~ -15 %
50/60 Hz±1 Hz, Three (3) phases

Environmental conditions Temperature: 0 to 45 °C
Relative humidity: 75 % or less (no condensation)

Q. Servo system

Servo motor AC servo motor
Position detector X, Y, Z and W axis: Absolute encoders (Absolute position detection)

Optional specifications

A. Controlled axes

Additional controlled axes
Select this function when using a NC rotary table to be controlled by TOSNUC999. Detailed specifications and scope of work shall be discussed as required.

B. Programming Methods

Inch/metric selection
Programming in inch or metric system can be selected by executing G70/G71 command.

C. Interpolation

Helical interpolation
The helical interpolation becomes effective by executing G02/G03 command and a linear interpolation command simultaneously.

Hypothetical axis interpolation
Setting and cancel of a hypothetical axis can be commanded by "G07 α 0/1" (α : axis address). An axis specified as the hypothetical axis does not move.

Cylindrical interpolation
When G67 command is executed, the cylindrical interpolation becomes effective by combining a linear axis with a rotation axis (additional axis) for operations such as grooving of a cylindrical cam.

Involute interpolation
The involute interpolation by means of orthogonal 2 axes becomes effective by executing G105 command.

Archimedes interpolation
The Archimedes interpolation (spiral interpolation) by means of orthogonal 2 axes or three (3) axes including orthogonal 2 axes and a normal axis becomes effective by executing G102/103 command.

Spindle normal direction control
Spindle normal direction control becomes effective for operation such as a grooving with the spring-necked turning tool by executing G140/G141/G142 command.

D. Feed

Synchronous tapping
Tapping can be performed by controlling a feed axis in synchronization with Spindle rotation.
M843/M844/M845 Spindle speed for synchronous tapping : 20 - 750 min⁻¹

Synchronous thread cutting
Large-diameter thread cutting can be performed by controlling a feed axis in synchronization with Spindle rotation.

E. Part program storage and edit

Part program storage capacity
Part program storage capacity of 1,200 m [3,900 ft] (approx. 538 KB) (the number of registrable programs: 1,024)

Part program storage capacity of 3,000 m [9,800 ft] (approx. 1.3 MB) (the number of registrable programs: 1,024)

Part program storage capacity of 5,400 m [17,700 ft] (approx. 2.2 MB) (the number of registrable programs: 1,024)

Part program storage capacity of 7,800 m [25,500 ft] (approx. 3.3 MB) (the number of registrable programs: 1,536)

Part program storage capacity of 10,200 m [33,400 ft] (approx. 4.2 MB) (the number of registrable programs: 1,536)

(Part program storage capacity includes a capacity of 600 m as the pack specification.)

*Large capacity memory (CF card) 2GB

F. Operation and Display

Display language selection (standard: Japanese) English or Chinese

G. I/O Function and Devices

**DNC I/F
DNC interface function compatible with EIA SP1292

Remote operation
Automatic operation is performed based on a NC program sent from the host computer in accordance with the transmission protocol.
Protocol A (handshaking system)
Protocol B (DC control-code system)

Binary operation
Automatic operation is performed based on the binary data sent from the host computer in accordance with the transmission protocol.

*High-speed LAN-LINKAGE
Host FTP Server
Protocol TCP/IP
LAN cable 10 base-T

Maximum capacity of internal memory
2GB (including a capacity of 30MB for storing the operation system)
Note) The following facilities shall be provided by the customer.
1) Construction and set-up for network
2) 10 base-T cables for connecting the LAN system with a machine

I. Tool Offset

Wear offset memory
The Wear offset memory can be added to the tool offset memory.

Three-dimensional tool compensation
A tool path can be offset in three dimensions by executing G30/G31 command.

K. Operation Support Function

Foreground plotting function
A tool path obtained from the active NC program can be plotted on the display screen. This function is not applicable to the program created for machining by using the horizontal spindle and AAI function.

Manual centering function
An offset of the coordinate system is automatically calculated by macro program after measuring a workpiece position.

L. Programming Support Function

Teaching
A program is automatically created based on a block executed by MDI operation, axis travel operated manually or the like.

Programmable data input
The data in the tool offset memory or the fixture offset memory can be updated by executing G58/G59 command.

Fixture offset data input
The programmed offset value for each axis can be added to the registered offset value by executing G158 command.

Programmable parameter input
Retrieving and rewriting of the data of setting parameter and system parameter can be performed by executing G58/G59 command. (To use this function, "programmable data input" is also required.)

Scaling
Reducing or enlarging of a profile based on a tool locus specified by a NC program can be performed by executing G64/G65 command.

Figure copy function
A subprogram can be entirely executed on the coordinate system which is rotated or shifted by executing G721/G722 command.

Circle cutting compensation
In executing the circle cutting command, the circle cutting is performed while compensating the roundness of machine motion by changing the radius of the circle in the longitudinal direction.

Machining time estimation and NC plotting function
Background processing such as program syntax check, machining time estimation, and plotting of a tool path can be performed in a state that the program is not executed.

Development of pattern cycle into NC statement
A hole-position pattern cycle command is developed into an NC statement showing each hole position.

M. Accuracy Compensation for Mechanical System

Z-axis thermal displacement compensation
The error of the mechanical system caused by thermal displacement of Spindle in the Z-axis direction is compensated.

O. Automation Support Function

Skip function
The processing of axis travel in execution is stopped and skipped to the following block in response to a skip signal inputted from the outside. (This function is automatically included in the specifications when the "automatic measuring system" and "automatic tool length measuring system" of machine optional accessories are selected.)

Tool breakage and tool wear detection
Cutting load condition can be monitored to detect tool breakage and tool wear.

Counting of tool working time
The tool working time is counted and an alarm is generated when the time has reached a specified tool life. (tool-life management)

Feedrate regulation
A feedrate is controlled so that a cutting load (load on the spindle motor) assumes a predetermined value.

Spare tool selection
The specified spare tool is selected in place of a tool that cannot be used due to tool life, breakage, or wear based on the result of tool breakage and tool wear detection.

Tool wear coefficient function
The tool-life time and the tool working time can be counted by multiplying the predetermined tool wear coefficient. (M-code output only)

Program check and tool list creation
In the program in execution, the syntax check of a program to be executed next is performed, or the list of tools to be used is created.

Interruptive macro
A macro program can be called and executed in response to an external signal.

Two (2) external M codes M192, M193 (M-code output only)

Q. Servo system

Shape recognition preview control (including preview control)
When machining along the tool path programmed including the successive short line segment block at high speed, the impact at a corner or an error such as delay in the servo system can be prevented, thus performing highly accurate machining at high speed.

Note) Consult with us regarding the DNC interface marked with **.
Note) Marked with *, selectable between two options.