CNC specifications TOSNUC PX100

Basic and pack specifications
Items with mark "☆" are pack specifications.

A. Controlled axes
Number of controlled axes	4 axes (XM, XS, Y, and Z)
Number of simultaneously controlled axes	
Positioning (G00) and linear interpolation (G01)	4 axes
Circular interpolation (G02 and G03)	2 axes
☆Synchronously controlled axes	XM, XS

B. Input command
Programming resolution	
Linear axis	0.001 mm
Maximum programmable dimensions	
Linear axis	±99999.999 mm
Data code	Automatic recognition of ISO/EIA code
	JIS B6311 ISO 6983/1
	EIA RS-358-B, EIA RS-244-B
Data format	Variable block with decimal point
	word address format
Absolute/incremental programming	G90/G91
Decimal point input	

C. Interpolation functions
Positioning	G00
Linear interpolation	G01
Circular interpolation	G02 (CW), G03 (CCW)

D. Feed functions
Rapid traverse rate	refer to machine specifications
Feedrate	F5 direct programming in mm/min
F1-digit feedrate programming	
Dwell	G04 and
	0 to 999.99 seconds with "F" or "P" code
Jog feed	
The selected axis is moved continuously at a rapid traverse rate or cutting feedrate by manual operation.	
Rapid traverse override	0 ~ 100 % in 10% increments
Feedrate override	0 ~ 200 % in 10% increments
Automatic acceleration/deceleration	
Linear type acceleration/deceleration on rapid and jog feed	
Automatic acceleration/deceleration for cutting feed	
G08/G09, or G50/G51	
Linear acceleration/deceleration on cutting feed	
S-shape acceleration/deceleration for rapid traverse	

☆Threading

G33, In-feed is synchronized with spindle revolution.

☆Feed per minute/ Feed per revolution	G94/G95
☆Dwell per revolution	G05
☆Tapping range	G63
☆Spindle inertia threading	G84

☆Hand wheel feed (portable type)

Linear axis	0.001 mm, 0.01 mm, 0.1 mm/division

☆Random start angle threading

Threading is initiated at the specified spindle angle for multi-start thread.

E. Program memory and editing
☆Part program storage capacity

600 m (1970 ft) or approx. 258 kB, 512 program can be registered.

100 ~ 200m (32.8 to 65.6 ft) is reserved for optional functions by the manufacturer.

Part program edit (in background)

Various editing operations are possible on programs in memory.

Program delete, program copy, program rename, search, jump, deletion and copy, by designating a range, replace, program insertion, etc.

Program name

8 digit characters following address $ or O.

Program comment in ( ) can be 32 characters.

Sequence number	N5

Sequence number search

A block containing specified sequence number is searched in forward or backward.

Program nesting list	List of program nesting is displayed

Program offset list

List of following data is displayed on a screen after searched from the head of program, fixture offsets, tools

Syntax check	program format is checked.

F. Operation and display
☆Operation panel	Keyboard with membrane switches

Customized keys

Series of key-in operation operated very often can be registered into one key for quick and efficient operation.

Parameter editing	Parameters can be edited.

Tool file

List of tool data such as length, diameter and offset are displayed on a screen.

Display capability

Part program, positions, compensation value, etc are displayed on the main screen, window screen or sub screen.

Screen clear

Screen is erased when no operation is performed in a specified time or more.

S, F manual setting

S and F code can be used in the manual mode.

S, F automatic setting

S and F code can be recorded automatically in the manual mode

Spindle motor load indication

Power consumption on spindle drive motor is displayed.

Run hour indication

NC working time is displayed.

Machined workpiece counting

Number of workpieces finished is displayed.

Calendar timer

Date and time are displayed on the run hour screen.

Machining record

Machining history in auto-mode is recorded.

User name registration

A user name is displayed at system start-up.

Memory operation

The machine is controlled by a part program in AUTO mode.

MDI operation

Two or more blocks can be input and executed in MDI mode.

G. Input and output functions and devices
RS-232-C interface port A

NC program, tool offset data, etc. can be input and outputted via this port.

☆User media

NC program, tool offset data, etc. can be input and outputted via USB port and/or CF card slot.

H. S, T, M function
Spindle speed (S) function	5 digits integer following address "S"
Spindle speed override	50 ~ 200% in 10 % increment
Tool (T) function	6 digits integer following address "T"
Miscellaneous (M) function	4 digits integer following address "M"

I. Tool offset
Tool length offset	G43, G44, G49
Tool offset	G45, G46, G47, G48
Cutter compensation C	G40, G41, G42
☆Tool offset capacity	899 sets for Tool length offset
	899 sets for Cutter compensation

J. Coordinate system
Automatic reference point return	G28, G29, G20
Coordinate system setting	G92
Fixture offset	G53, G57
	9 sets
☆Additional fixture offset capacity	90 sets
Fixture offset 2	G54, G55, G56
2nd, to 4th reference point return	G21

K. Operation support function
Control IN/OUT

Data and comments in "( )" are ignored.

Single block

Program is executed block by block.

Optional stop	M01

☆Optional block skip

Blocks containing " / " at head of them are ignored.

End of program	M02, M30

Dry run

A machine moves at feedrate set in a parameter in place of programmed feedrate.

Machine lock	All travel commands are neglected.
Auxiliary function lock	M, S and T commands are neglected.
Z axis feed cancel	Z axis travel commands are neglected.

Manual absolute ON/OFF

Jog travel distance is neglected or added on the current position data on the screen.

Override cancel	M48, M49

Mirror image

All clear	The push button initializes NC memories.
Reset	The push button resets the current command.
Cycle start	The push button starts a program.
Feed hold	The push button holds travel in AUTO mode.

Cycle stop

The push button holds travel and spindle rotation in AUTO mode.

Re-start

Restart after interrupted require several measures for safety.

Sequence number collation and stop

Machine stop after executed operation in the block with the specified sequence number.

Manual numerical command

Data input via keyboard is available in manual mode.

Single block suppression	G990, G991
Feed hold suppression	G992, G993
Override suppression	G994, G995
Hand wheel interruption suppression	G996, G997

Manual interruption and manual return

Machine can return to the interrupted position after some manual interrupting operation.

☆Additional optional block skip

Blocks containing " // " and " /// " at head of them are ignored.

☆Hand wheel interruption	G996 or G997

☆Manual tool length and diameter measurement

Offset data are checked based on reference tool and stored on memories specified in manual.

L. Programming support function
Plane selection	G17, G18, G19
Radius programming in circular interpolation	
	R data is radius on circle.
Circle cutting	G12, G13, G22, G23 and G222, G223
Positioning on machine coordinate system	G73
Sub program call	G72
Arbitrary angle chamfering / corner R	
Canned cycle	G77 to G89, and G98, G99
Automatic corner override	G08, G09 and G50, G51
☆Programmable mirror image	G62, G66
☆Plane conversion	G35 to G39
☆Macro programming	G72, G74, G75, G76
☆Pattern cycle	G109 to G119, and G121, to G132
☆Coordinate conversion	G10, G11
☆Three dimensional coordinate conversion	G14
☆Spindle C-axis control	
	Spindle rotational position is controlled as a rotary axis.

M. Mechanical error compensation
Backlash compensation type A

At time of reversing travel direction.

Backlash compensation type B

Referring position detector on feed motor.

Backlash compensation type C

For hybrid control axis.

Backlash compensation for MPG
　For hand wheel travel
Backlash compensation for each axis feed
　Compensation data is different for rapid and cutting feed mode.
Fluent backlash compensation
　Compensation based on travel distance from return point.
Pitch error compensation
　Compensation points are mot more than 160.per axis.
Uni-directional positioning
　Positioning approach is always single direction specified.
☆Pitch error gradient compensation
　Approximation with up-to 30 line
☆Straightness compensation
　Approximation with up-to 9 straight line.

N. Machine control support function

| Feed interlock | External signal stop machine travel. |
| ☆External deceleration | External signal slow down federate. |

O. Safety and maintenance

Emergency stop
　Push button stop machine in emergency.
Overtravel check
　External over travel signal stop machine travel.
Stored stroke check
　Allowable stroke are stored for each axis.

| Axis interference check Ⅱ | G26, G27 |
| Self-diagnosis | Errors are monitored in controller. |

Software configuration display
Alarm screen and alarm record
Recording past operation, alarm, and machine conditions
Screen copy

| ☆Interference check Ⅰ | G24, G25 |

☆Door interlock
　Power will be turned off when door on controller is open.

P. Enclosures and room condition

Power specifications	AC 3 phase 200/220 V + 10% ~ -15%
	50/60 Hz +/- 1 Hz
Room conditions	Temperature 0 to 45 degree centigrade
	Humidity 75 % or less (No condensation)

Q. Servo system

| Servo motors | AC servo motors |
| Position detectors | Absolute position detectors on each axis |

Optional specifications

A. Controlled axes

Additional controlled axis
　When the machine has CNC controlled additional options.
Hybrid control
　When the machine has optional scales on a axis.

B. Input command

| Inch/metric selection | G70/G71 |

C. Interpolation functions

| Helical interpolation | G02/G03 |
| Hypothetical axis interpolation | G07 α0/1 |

　α is an axis address. The axis specified for α will never move in this program.

Cylindrical interpolation	G67 for machining cylindrical cam
Involute interpolation	G105
Archimedean interpolation	G102/G103
Spindle normal direction control	G140/G141/G142

D. Feed functions

| Synchronous tapping | G843/G844/G845 |
| | Available spindle speed is 20 ~ 750 min^{-1}. |

Synchronous threading
　For threading on large diameter by boring tool

E. Program memory and editing

Part program storage capacity	1200 M (approx. 3940 ft.), 538 kB
	Number of program is 1024
	3000 M (approx. 9840 ft.), 1.3 MB
	Number of program is 1024
	5400 M (approx. 17720 ft.), 2.2 MB
	Number of program is 1024
	7800 M (approx. 25590 ft.), 3.3 MB
	Number of program is 1536
	10200 M (approx. 33460 ft.), 4.2 MB
	Number of program is 1536
Mass memory (1) (CF)	Compact flash memory with 256 MB

F. Operation and display

| Selection of display language | English |
| | Chinese |

G. Input and output functions and devices

DNC connection	EIA SP1292 level 3 protocol
Remote buffer operation	Protocol A (handshake type)
	Protocol B (DC control code type)
Binary operation	Binary data

External data output
High-speed LAN linkage (1)

Host	FTP server
Protocol	TCP/IP
Connecting cable	10 base-T
Capacity	2 GB

　NOTE
　Followings are customer's responsibility.
　　1, Installation and set-up of LAN network
　　2, Connecting cable and connection to the LAN network
　Select one of the High-speed LAN linkage or Compact flash memory (CF).

I. Tool compensation

Wear offset memory

| Three dimensional tool offset | G30/G31 |

K. Operation support function

Foreground plotting
　Tool path of active program on vertical spindle will be shown on the screen, but except program for the horizontal spindle and such machine sequence as ATC and likes.
Help
　Help message for alarms are indicated on the screen.
S&F analogue override

Manual alignment
　Check a workpiece and set a coordinate for machining automatically with help of special macro program.
Manual tracing back
　Up to 30 points are memorized for tracing back in manual.

L. Programming support function

Teaching
　A program is regenerated based on operation in MDI and manual mode automatically.

Programmable data input	G58/G59
Programmable parameter input	G58/G59
Fixture offset data input	G158
Scaling	G64/G65
Figure copy	G721/G722

Compensated circle cutting
　Radius is adjusted in circle cutting.
Estimation of machining time and NC plotting
　Executed in background
Pattern cycle conversion to normal extended program
　A short condensed program of pattern cycle is converted to general program which consisted of many blocks

M. Compensation function for mechanical accuracy

| Z axis thermal expansion compensation | In Z axis direction |

O. Automation support function

| Skip function | G61 for several kinds of measurement |
| Tool breakage/wear detection | Load on the tool is monitored. |

Counting tool working time
　Working time is counted, and alarm is the result when life has expired.
Feedrate regulation
Spare tool selection
　A spare tool will be selected automatically when a tool had such trouble as life, breakage, wear

Tool wear coefficient function	For Figuring tool working time
Automatic measuring	Measuring on workpiece
Scheduled operation	NEXUS schedule

Program check and listing slated tool
　Format on the next program will be checked and prepare a tool list for it while a current program is working.
Interruptive macro
　A macro program is activated by a external signal and interrupt machine movement.

| Two additional M codes output | M192, M193 |

Q. Servo system

Shape recognition preview positioning control
(CNC shape Ⅱ)
　Target are accurate shape and smooth surface in contouring surface machining.

| NRBS interpolation | Three dimensional NURBS |
| Spindle load factor setting | M400, M499 |