CNC System Specifications TOSNUC PX100

one eyetein opeenieute	
Standard and pack specifications	Program offset list
	Lists of blocks contai
(Items marked 🛱 are included in the pack specifications	.) played after search fro Eixture offect list. T and
Controlled axes 3 axes (X, Y, Z	Program check
Simultaneously controlled axes	Program format check
3 axes for positioning (G00) and linear interpolation (G01)	F. Operation and Displa
2 axes (X-Y, Y-Z or Z-X) for circular interpolation (G02, G03)	☆ Operation panel 15 in
B. Programmable Methods	Keyboard w
Maximum programmable dimension Linear axis: 0.001 mi	<u>T</u> Customizing key When a series of key i
Linear axis: ±99999.999 mr	n registered in a key b
Data code Automatic recognition of ISO/EIA cod	e quickly, thus improving
JIS B6311 ISO 6983/1	Tool file
EIA RS-358-B EIA RS-244-E	A combination of tool I
Data format Variable block with a decimal poin	nt and other tool data car
Absolute/incremental programming G90/G9	1 Automatic operation M
Decimal point input	cal command are poss
Calculator type/Programming resolution typ	e S.F manual setting
C. Interpolation	S and F codes can be
Positioning G00 - Positioning at rapid travers	e S.F auto setting
Circular interpolation G02 (CWI)/G03 (CCW	Spindle motor load display
D. Feed	Load exerted on the spi
Rapid traverse rate	Run hour display
Feedrate F5-digit direct programming in mm/mi	n Calendar timer
Dwell: G04	Management of progra
Seconds following address E (or P)	9 Machining record Machining start time
Continuous jog feed	program executed in th
A selected axis is continuously moved at rapid traverse of	or User's name registration
feedrate by manual operation.	A user's name is displa
Rapid traverse rate override	Jog feedrate switch
to 100 % in 10 % increments	 Display of manual interrupt A log shift amount of
Feedrate override	operation can be disp
Feedrate can be overridden steplessly in the range of 0 t	o current position displa
200 %.	tion display and interru
Automatic acceleration/deceleration	G. I/O Function and Dev
verse rate and log feedrate.	Input and output of N
Automatic acceleration/deceleration for feed	are possible via the El
Linear acceleration or deceleration is effected on fee	e- USB memory
drate by commanding G08/G09 and G50/G51.	A machine could be data ata ara input/aut
S-type acceleration/deceleration for rapid traverse	- connected onto persor
verse rate.	☆ Remote buffer operation
☆ Thread-cutting	Automatic operation is
I nread-cutting synchronized with spinale revolution can b	e gram sent from the
∑ Feed per revolution	Protocol A (Hand-shi
Feed per minute or feed per revolution can be selected for	 Protocol B (DC contr
feedrate as specified by F code, using G94/G95.	H. S, T and M Functions
Dwell per revolution	Spindle speed function (
The set of	following address S
0.001/0.01/0.1 mm (per divisior	n) Spindle speed override
Stroke limit dwell function (only for Z-axis)	Tool function (T-function)
Even if a program resulting in axis overtravel is con	1- Tool number can be s
alarm but the Z-axis moves to the stroke limit at rapid tro	e <u>IOWING address I.</u> Miscollanoous function (I
verse and stavs there until it returns to the stroke range b	Miscellaneous function
another G00 command.	integer following addre
E. Part Program Storage and Edit	I. Tool Offset
Program storage	I ool length offset
(About 50 m is reduced to keep it as the maker's are	a selected plane by mea
according to the optional functions selected.)	Tool offset
Part program edit (Background edit)	Tool offset expansion o
Various editing operations are possible for programs memorized	selected plane by mean
cel deletion and conv for specified range replace program	m Cutter compensation i
insertion, register in abbreviation, line mark designation	n. plane by commanding
sequence number change, space insertion between words, pro	 No. of tool offsets
gram input in S.F auto mode, simultaneous opening of tw	10
programs, editing of various data during program editing, undo	No
Program name	Automatic reference poir
A program name is specified by a maximum of eigl	nt G28: A
alphanumeric characters following address \$ or O. Up to 32	2-
character program comment can be included.	Coordinate
A sequence number is specified by a 5-diait number fo	- A coordinate system A
lowing address N.	that current axis positio
Sequence number search	nate.

- A block containing specified sequence number can be searched.
- Program nesting list

11

A list of program nesting is displayed.

- is executed
- ch color TFT liquid crystal display
- ith membrane switches (80 keys) input operation used very often is
- peforehand, it can be executed efficiency of operation.
- length, cutter diameter, tool offset n be displayed and edited.
- IDI operation and manual numerisible
- e set in the manual mode.
- set automatically in the manual mode.
- ndle drive motor is displayed. The NC working time is displayed.
- am creation date, and time display.
- actual machining time, etc. of a
- he AUTO mode are displayed.
- layed at system startup.
- tive amount and exit from interruption an axis made during automatic
- played on the screen next to the ay, using the automatic interrupption function.
- IC programs and tool offset data A RS232C interface.
- operated and part program, tool tput, from USB memory. nal computer HMI
- executed by means of a NC pro-
- host computer according to the nake system)
 - rol code system)
 - S-function) e specified by a five-digit integer
- 50~200 % in 10 % increments
- specified by a six-digit integer fol-
- M-function) n can be specified by a four-digit ess M.
- be set on an axis perpendicular to ans of G43, G44 or G49.
- or reduction is effected on an axis in ns of G45, G46, G47 or G48.
- sation C is effected on an axis in selected
- G40, G41 or G42. No. of tool length offset: 60 sets
- lo. of cutter compensation: 60 sets nt return
- utomatic return to reference point G29: Return from reference point
- G20: Reference point check can be set by means of G92 so
- ion can be the commanded coordi-
- Fixture offset
- Fixture offset is effected by means of G53 or G57 Fixture offset 2
- Fixture offset is effected by means of G54, G55 or G56.

- 2nd to 4th reference point return Axes are returned to the 2nd to 4th reference point utomatically K. Operation Support Function
- Control in/out
- Information flanked by the control in and control out codes can be neglected Single block
- A program can be executed block by block during automatic or MDI operation.
- Dotional stop Ontional block skir
- A block containing a slash (/) code at the top can be ignored. (One pc.) Drv run
- Axes move at a feedrate specified by parameter in lieu of programmed feedrate.
- Machine lock Output of axis command pulses to the machine side is stopped. Auxiliary function lock
- M, S or T command is not output to the machine side. Z-axis feed cancel
- Output of Z-axis command pulses to the machine side is stopped. Manual absolute ON/OFF
- A jog travel distance can be added to the current coordinate value according to the status of the absolute ON/OFF switch.
- erride cance Override on feedrate and spindle speed can be ignored
- and clamped at 100 %, by commanding M48 or M49. clea The CNC internal memory can be initialized by pressing
- the ALL CLEAR pushbutton switch. A command currently executed can be reset. eed hold
- Axis feed can be stopped temporarily by pressing the FEED HOLD pushbutton switch during automatic or MDI operation
- Cycle stop Axis feed and spindle revolution can be stopped tem-
- porarily by pressing the CYCLE STOP pushbutton switch during automatic or MDI operation. estart
- When machining was interrupted during automatic operation for tool exchange, etc., the operation can be resumed from the specified block after taking necessary steps.
- equence number collation and stop Operation can be stopped after the block preceding a block bearing specified sequence number has been executed.
- Manual numerical command Data can be input and executed in the manual mode Data that can be input: G00/G01, F, M, S, T, axis data
- (incremental) Single block suppression Single block ON or OFF in the single operation mode
- can be selected by means of G990 or G991 Feed hold suppress
- Feed hold ON or OFF can be selected by means of G992 or G993.
- erride suppression Feedrate override ON or OFF can be selected by means of G994 or G995.
- Handwheel feed interruption suppression Handwheel feed interruption ON or OFF can be selected
- by means of G996 or G997 Handwheel feed interruption
- Interruption by means of the MPG handwheel can be permitted during cutting feed.
- Manual tool length and tool diameter measurement An offset value from the master tool is measured and memorized under specified offset number.
- . Programming Support Function Plane selection
- A machined plane can be selected by means of G17, G18 or G19. Circular interpolation by radius programming
- Radius of an arc can be specified directly by R command. Circle cutting G12, G22: Inner circle cutting CW Circle cutting
 - G13, G23: Inner circle cutting CCW G222: Outer circle cutting CW G223: Outer circle cutting CCW
- Machine coordinate system positioning command
- Axes can be moved to the coordinates characteristic of the machine by means of G73. Subprogram ca
- A subprogram stored in the memory can be called and executed by means of G72. The subprogram name should be specified by eight alphanumeric characters following address \$ or O.
- Arbitrary angle chamfering/corner R Arbitrary angle chamfering or corner R can be inserted
- between cutting feed commands of consecutive two blocks.

CNC System Specifications TOSNUC PX100

Involute interpolatior

Synchronous tapping

t program storage

1 200 m equivalent

3.000 m equivalent

5,400 m equivalent

7.800 m equivalent

10,200 m equivalent

s memory (

machine side

DNC interface

protocol

protocol

I. Tool Offset

☆ Wear offset memory

Teaching functior

J. Coordinate System

Binary operation

1-speed LAN linkar

nalish specification

D. Feed

G845

Canned cycle A drilling canned cycle can be executed by commanding G77~G89.

- Automatic corner of Automatic override of inside corner
 - Change of inside arc cutting speed
- Mirror image Mirror image can be set on each axis by means of relevant pushbutton switch.
- ☆ Macro program
- A macro program can be called and executed by commanding G72, G74, G75 or G76 -> Pattern cvc
- Regular hole position pattern or milling pattern can be exe cuted by commanding G109~G119 or G121~G132.
- ☆ Coordinate conversior Parallel move and rotation of a coordinate system can be the snindle
- executed by commanding G10 or G11. M. Mechanical Error Compens

- 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
- single direction by means of G60. ☆ Pitch error gradient compensation
- Pitch error of each axis feed screw of machine system can be compensated by approximating with up to 30 straight lines

Each axis feed can be permitted or banned by means of

Emergency stop can be effected on the machine by

pressing the EMERGENCY STOP pushbutton switch.

Emergency stop is effected by means of external overtravel signal.

Axis feed overrunning a predetermined stroke is prohibited.

Axis entry into a predetermined off-limit area is prohibited

An error in NC program, CNC system, servo and

machine system can be monitored and an alarm is

Setting of axis interference area and axis interference

check ON/OFF can be performed by means of G24 or G25.

When the CNC system door is open, the primary power is turned off.

Select this function when controlling an NC rotary table

through the TOSNUC PX100. Detailed specifications

Programming in the inch system or metric system can be

Helical interpolation can be executed by G02/G03 and a 5

Setting and cancel of hypothetical axis can be com-manded by "G07 α 0/1" (α : axis address). An axis

When G67 is commanded, cylindrical interpolation is possible

axis) for operations such as grooving of a cylindrical cam.

When G140, G141 or G142 is commanded, grooving is performed,

by combining a straight axis with a rotary axis (additional $rac{1}{\sim}\overline{F}$

specified as the hypothetical axis will not move.

using a spring-necked turning tool mounted on the spindle

and the scope of work shall be discussed as necessary.

Absolute encoders (absolute position detection)

ations, which are included in the standard specification

AC servo motors

To reset, the EMG RESET pushbutton switch is used.

N. Machine Control Support Function Axis interloc

external sia P. Safety and Maintenance

Emergency stop

Overtravel check

Interference check II

erference check I

by means of G26 or G27.

Optional specifications

selected by means of G70 or G71

; marked 🕸 are nack A spec

A. Controlled axes

Additional controlled axes

B. Programming Methods

ch/metric selection

elical interpolation

linear axis command

Cylindrical interpolation

Hypothetical axis interpolation

Spindle normal direction control

C. Interpolation

Stroke check

Self-diagnosis

processed.

Q. Servo System

Position detectors

☆ Door interlock

MPEseries

- Involute interpolation is executed with two orthogonal axes by commanding G105 Archimedes interpolation
- Archimedes interpolation (spiral interpolation) is executed with orthogonal two axes or with three axes including a vertical axis by commanding G102/G103.
- Tapping operation is possible by synchronous control of 🖄 the spindle and feed axis by commanding G843, G844 or
- Synchronous tapping speed range: 40~1,150 min⁻¹ Start of thread-cutting at selected angle Thread-cutting can be started from a desired angle of
- Synchronous thread-cutting
- Large-diameter thread-cutting is possible by synchronous control of spindle and feed axis. . Part Program Storage and Edit

punched tape (No. of programs: 1,024)
punched tape (No. of programs: 1,024)
punched tape (No. of programs: 1,024)
punched tape (No. of programs: 1,536)
punched tape (No. of programs: 1,536)
2 GB

F. Operation and Displa

External position display Each axis position display can be equipped on the

G. I/O Function and Device

S232C interface port B Input and output of NC programs, tool offset data, etc. are possible via the EIA RS232C interface

DNC interface pursuant to the EIA SP1292.

☆ Remote buffer operation Automatic operation is executed as per an NC program sent from the host computer according to the transmission

> Protocol A (Hand-shake system) Protocol B (DC control code system)

Automatic operation is executed as per the binary data sent from the host computer according to the transmission

age	
Host	FTP serve
Protocol	TCP/I
NC input cable	100 base
Maximum canacity: 2	GB (includin

30 MB for OS Note) The following items shall be provided by customer.

1) Network construction and setup) 10 base-T cable for connection with the machine

Additional number of tool offset

Tool length offset: 899 sets including the standard ones Cutter compensation: 899 sets including the standard ones

A wear offset memory can be added to the tool offset memory. Three-dimensional tool compensation

Three-dimensional compensation of a tool path is possible by means of G30 or G31.

lditional number of fixture offset

Fixture offset: 99 sets including the standard ones (H901 \sim H999) K. Operation Support Function

dition of optional block skip A total of three including the standard one Foreground plotting function

A tool path of an active NC program can be plotted on the display. Manual alignment (or centering) function

An offset of the coordinate system is automatically calculated by macro program after workpiece measurement. L. Programming Support Function

Virogrammable mirror image Mirror image can be set on each axis by means of G62 or G66.

A program can be created automatically based on a block executed in the MDI mode, manual axis feed, etc. rogrammable data input

The contents of the tool offset memory or fixture offset memory can be updated by means of G58 or G59. set data input

When G158 is commanded, programmed each axis value can be added, taking the already registered offset value as the reference.

Programmable parameter input

Reading and writing of setting parameter and system parameter values are possible by means of G58 or G59.

A profile specified in an NC program can be reduced or expanded by means of G64 or G65. igure copy function

Entire subprogram can be executed by effecting coordinate rotation or coordinate shift by means of G721 or G722. Circle cutting compensation

erruptive macro

A macro program can be called and executed by means of external signal. Three-dimensional coordinate conversion

Three-dimensional parallel move and rotation of a coordinate system are possible by means of G14.

Plane conversion

A program created based on the G17 plane can be converted into the one supposing other plane and exe cuted by means of G35~G39. Machining time estimate & NC plotting function

Program check, machining time estimate and tool path plotting for a non-active program can be executed in the background.

Interactive type automatic programming function

- This function helps create NC programs for face milling, side face milling, pocket milling, contouring and drilling operations Zone machining function
- In die and mold machining, this function allows the following processing in a zone set beforehand in the CNC internal memory.
- 1) Change in depth of cut in Z-axis direction
- 2) Change of cutting conditions
- 3) Composite processing of 1) and 2) above

M. Machine Control Support Funct Straightness compensation

Straightness of the machine system can be compensated by approximating with up to nine straight lines per each axis. O. Automation Support Function

Skip function

When G61 is commanded, axis feed currently executed can be stopped by means of skip signal input from an external device

Fool breakage and tool wear detection

Cutting load condition can be monitored to detect a tool breakage or tool wear. Counting of tool working time

The tool working time is totaled. When it has reached a specified tool life, an alarm will generate.

eedrate regulation

Feedrate can be controlled so that cutting load (i.e. load imposed on the spindle drive motor) can be a predetermined value.

Spare tool selection

When a tool that cannot be used any further due to end of life. breakage or wear, a spare tool set beforehand is selected. ool wear coefficient function

When totaling the tool life and tool working time, they can be counted by multiplying a predetermined tool wear coefficient. (M code output only) Two external M codes

M192, M193 (M code output only)

R. Servo System ape recognition preview control function (including preview control) When machining consecutive blocks of short line of tool path programmed at high speed, impact at a corner and error due to delayed response of servo system can be prevented to assure high-speed and high-precision machining CNC DASH

It is possible to setup acceleration and slowdown time separately for each axis X, Y, Z. S.F function

When cutting a sculptured surface with a ball end mill, spindle speed can be controlled so that cutting speed can be made constant at ever-changing contact points. Also, feed per revolution can be controlled according to the spindle speed controlled. S. Others

JRBS interpolatior

Three-dimensional NURBS (Non-Uniform Rational B-Spline) interpolation is possible with defined data of NURBS curve

Note) When selecting the functions marked **. consult with us beforehand.

Note) Marked with *, selectable between two options