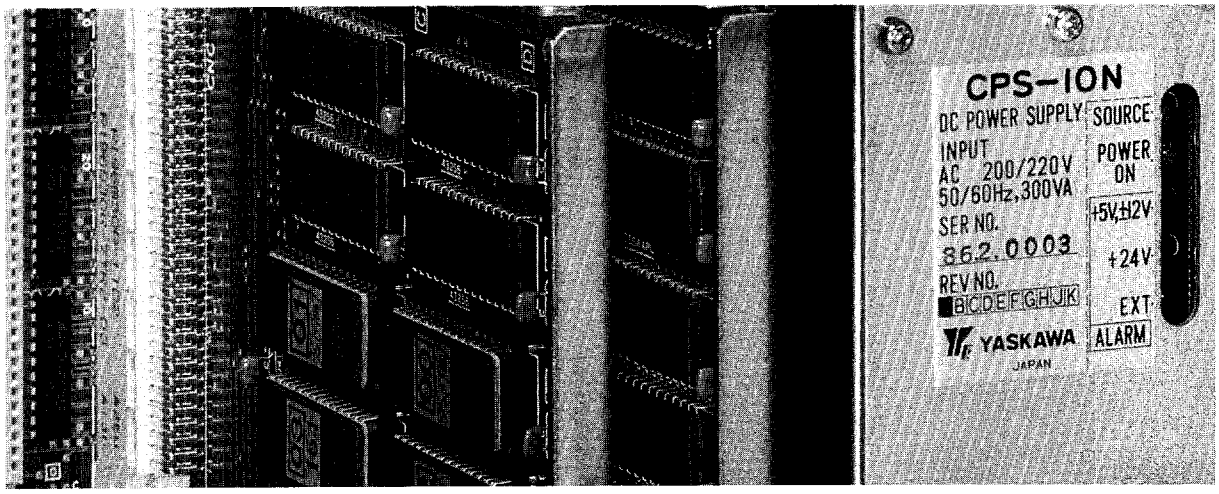
YASNAC LX3

CNC SYSTEM FOR TURNING APPLICATIONS

UPGRANDING FUNCTION



Before initial operation read these instructions thoroughly, and retain for future reference.

YASKAWA

Specification upgrading of several functions is possible witn YASNAC LX3 if it has a "Compact Interactive Function." (optional).

This manual describes these specification upgrading items and operation.

The reader is requested to refer to the YASNAC LX3 Operator's Instruction Manual (TOE-C843-9.20) regarding those items that are not covered here. The data No. of the operator's manual referred to in this manual is TOE-C843-9.20.

CONTENTS

1. STANDARD FUNCTION UPGRADING / *2*

1.1 UPGRADING OF MULTIPLE CANNED CYCLE FUNCTION / *2*

1.2 UPGRADING AND CHANGE IN 9″ SCREEN DISPLAY ITEM / *6*

1.3 UPGRADING 9″ EDIT OPERATION / *8*

2. NEW TOOL SETTER FUNCTION / *9*

2.1 GENERAL / *9*

2.2 WRITING TOOL COORDINATE MEMORY / *9*

2.3 WRITING Z-AXIS WORK COORDINATE SYSTEM SHIFT AMOUNT / *11*

2.4 OFFSET DISPLAY / *13*

3. NEW COORDINATE SYSTEM SETTING FUNCTION / *15*

3.1 GENERAL / *15*

3.2 DESCRIPTION FUNCTIONS / *15*

3.3 PRECAUTIONS IN USING CONVENTIONAL G 50 T**** FUNCTION / *16*

4. FS AUTO EDIT FUNCTION / *17*

4.1 GENERAL / *17*

4.2 INPUT AND OUTPUT / *17*

4.3 DESCRIPTIONS OF FUNCTIONS / *18*

4.4 PRECAUTIONS / *21*

5. AUTO NOSE R FUNCTION / *22*

5.1 GENERAL / *22*

5.2 CONDITIONS TO MAKE AUTO NOSE R COMPENSATION FUNCTION EFFECTIVE / *22*

5.3 COMPENSATION CALCULATIONS OF AUTO NOSE R COMPENSATION APPROACH AND RETRACT / *23*

5.4 CANCEL FUNCTION (G40, G01) / *27*

5.5 PRECAUTIONS / *27*

6. Z-AXIS TOOL CHUCK BARRIER FUNCTION / *27*

6.1 GENERAL / *27*

6.2 OPERATION / *27*

6.3 PRECAUTIONS / *28*

7. OTHER ADDITIONAL FUNCTIONS / *29*

7.1 HIGH-FREQUENCY SKIP SIGNAL APPLIED FOR MACHINE SETTER / *29*

7.2 WORK SHIFT FUNCTION ADDED (G54) / *29*

7.3 ABSOLUTE COORDINATE SWITCHING ADDED WHEN MACHINE LOCK EFFECTIVE IS SWITCHED TO INEFFECTIVE / *29*

7.4 ADDITIONAL PARAMETER SWITCHING TO AUTO NOSE R COMPENSATION AND COMMAND T (INCLUDING G 50 T) FUNCTION / *30*

7.5 HANDLE INTERPOLATION FUNCTION / *30*

8. LIST OF NC PARAMETERS FOR COMPACT INTERACTIVE FUNCTION / *31*

1. STANDARD FUNCTION UPGRADING

1.1 UPGRADING OF MULTIPLE CANNED CYCLE FUNCTION

1.1.1

Nose R compensation can be executed with the G 70 to G 73 cycles even while the finish profile program is being performed.

By this, G41 and G42 are added to the G codes that can be designated by the blocks other than blocks Nsf and Nnf during finish profile program execution.

During the rough finish and finish cycles, nose R compensation will become effective during the finish cycle by the blocks instructed by G41 to G42.

Always instruct with the G00 or G01 moving command when instructing G41/G42 to the block to start the finish profile program.

Commands with single-blocks are prohibited.

- Upgrading (d) and (e) of par. 2.8.25.2, "Rules in programming" in the YASNAC LX3 Operator's Instruction Manual.
- Upgrading in (6) of par. 2.8.25.4, "NOTES" in the YASNAC LX3 Operator's Instruction Manual.
- Upgrading in (2) of par. 2.8.25.5, "NOTES" in the YASNAC LX3 Operator's Instruction Manual.

1.1.2

The T code command can be executed during the G70 to G73 cycles even during the finish profile program.

By this, the codes F, S and T instructed during the finish profile program become effective during the finish cycle (G70) and is disregarded during rough cutting of the external form.

- Upgrading in (c) of par. 2.8.25.2, "Rules in programming" in the YASNAC LX3 Operator's Instruction Manual.
- Upgrading in (1) of par. 2.8.25.4, "NOTES" in the YASNAC LX3 Operator's Instruction Manual.

1.1.3

A command with different start and end points in a finish profile program can be executed during the G70 to G72 cycles.

- Upgrading in (a) of par. 2.8.25.2, "Rules in programming" in the YASNAC LX3 Operator's Instruction Manual (Condition of BA being parallel to Axis Z is lost.)
- Upgrading in par. 2.8.25.3, "Rules and cautions in programming finish shape" in the YASNAC LX3 Operator's Instruction Manual (Condition of BA being parallel to Axis Z is lost.)

1.1.4

Execution using finish allowances U and W as stock removal allowances is possible during the cycles G70 to G73 if both stock removal allowances I and

K are omitted. Upgrading is possible with the related parameter #6009 D1 = 1.

1.1.5

The recessing fixed cycle can be executed by adding the Command A (number of cutting-off stages) and Command B (cutting-off bite blade width) to the functions G74 (end face cutting-off cycle) and G75 (outside diameter cutting-off cycle).

(1) Given the following command, G74 executes the cycle as shown in Fig. 1.

· Upgrading in (1) of par. 2.8.25.6 of the YASNAC LX3 Operator's Manual.

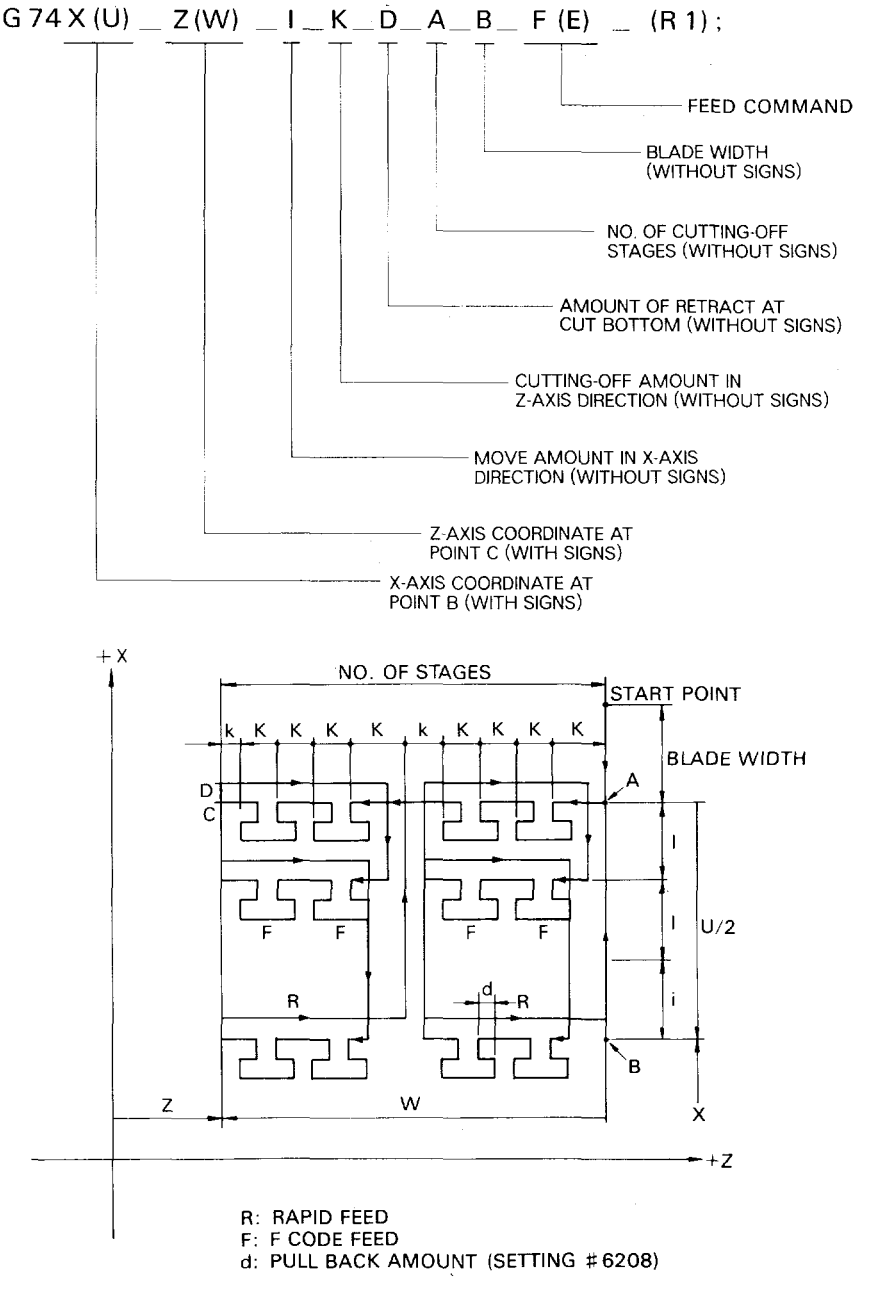


Fig. 1 G74 Cycle Execution

1.1.5 (Cont'd)

(2) Given the following command, G75 executes the cycle as shown in Fig.2.

· Upgrading in (1) of par. 2.8.25.7 in the YASNAC LX3 Operator's Manual.

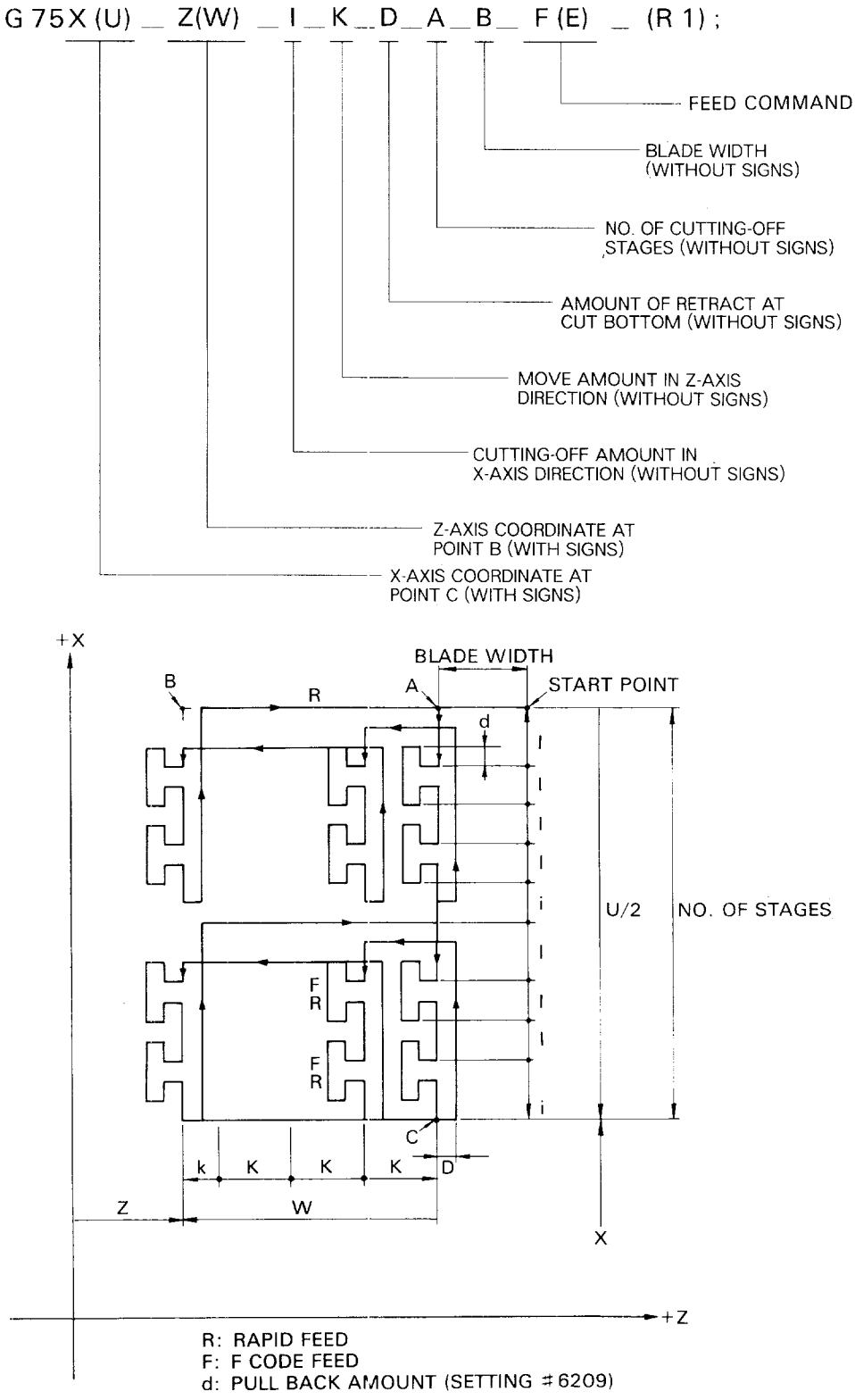


Fig. 2 G75 Cycle Execution

1.1.6 Upgrading of G76 Thread Cutting Cycle Function

(1) The final nth cycle specified by L is executed by adding command L to original G76 command.

G76X(U)_Z(W)_I_K_D_F(E)_A_L ;
L0 : Executes the command of the final cycle.
L1 : Executes the commands in one cycle before the final cycle and the final cycle.
⋮
Ln : Executes the commands from nth cycle before the final cycle to the final cycle.

When n is a larger number than the number (N) of normal cutting cycles ($n \geqq N$), a normal cutting cycle is executed.

(2) Zig zag cutting with constant cutting amount can be performed by adding command P to original G76 command.

G76X(U)_Z(W)_I_K_D_F(E)_A_P_;

The following plunges are obtained by command P :

Without P : Constant normal cutting amount, one-blade cutting
P_1 : Constant normal cutting amount, one-blade cutting
P_2 : Constant normal cutting amount, zig zag cutting
P_3, P_4, P_5 ···· : Constant normal cutting amount, one-blade cutting

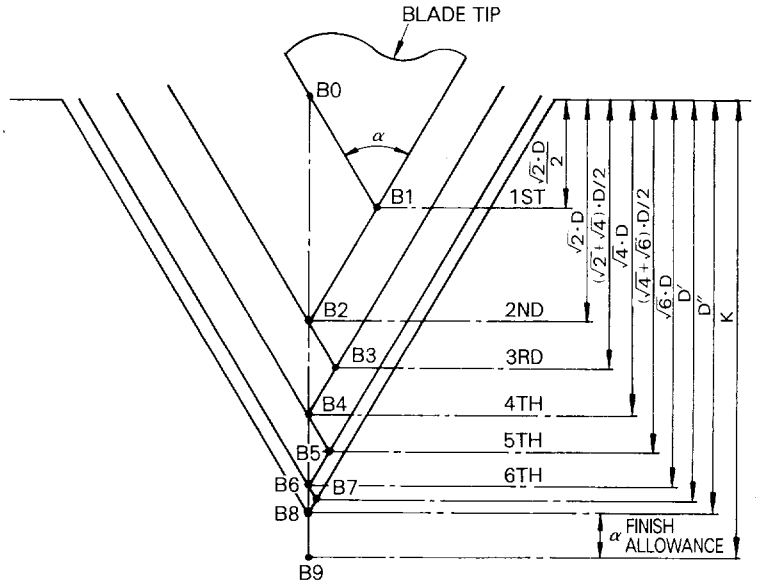


Fig. 3 Constant Cutting Amount, Zig Zag Cutting

1.1.7 Dwell Added to G74 and G75 Cycles

Dwell can be executed on the hole bottom by setting parameter (#6214) to G74 or G75. When command A is provided (with number of stages), dwell is executed on the hole bottom of the last step.

#6214 0 — — 65535 1=1ms

1.1.8 Precautions in Upgrading of Multiple Canned Cycle Function

(1) The following conditions are added to pars. 1.1.1, 1.1.2 and 1.1.3.

(a) Upgrading becomes effective with the related parameter #6011 D1=1.

(b) This upgrading increases the maximum memory capacity of the finish profile programs to 39 blocks.
Change in (2) of par.2.8.25.1 in the YASNAC LX3 Operator's Manual.

(2) Precautions to G74/G75 upgrading in par.1.5.

(a) If neither Command A nor Command B is given, the execution will be the same as in previous G74/G75.

(b) If Command B only is given, blade width shifting is executed at the beginning and end of G74/G75 as mentioned in detail below:

(i) The first motion is shifting by the blade width in the X-axis command direction with G74 and in the Z-axis command direction with G75 from the position of the block immediately before G74/G75.

(ii) The last motion is returning to the position of the block immediately before G74/G75 after being shifted by the blade width amount.

(c) If Command A only is given, only retract is executed without executing blade width shifting.

(d) If Command A is given, the pull back amount with G74 and G75 will be the set amount of #6208 and 6209. Pecking is not performed if this is "0."

- Addition to par. 2.8.25.6, "NOTES" in the YASNAC LX3 Operator's Instruction Manual

(e) Alarm 96 with the groove width < B (blade width) command.

1.2 UPGRADING AND CHANGE IN 9" SCREEN DISPLAY ITEM

Basically, this function is upgrading of and changes in the 9" character CRT. The same functions are obtained with the compact interactive function also.

1.2.1 Upgrading

(1) The program directory menu was displayed in the alarm function.
By upgrading, the menu is displayed in the program function.
Therefore, the alarm function does not show a program directory menu.

By depressing PROG in the program function regardless of the mode, the program directory and program menu are shown cyclically.

The program directory menu is displayed when the operation returns to the program function after changing from the program directory menu to another function.

The page key in the program directory menu is operated by the same method as before.

(2) The external present value display can be changed to "0" by the operation in the all-position menu [X], [Z] "[ORG]" . The external present value display can be changed to "0" by the following key operation when the all-position menu is displayed.

(a) [X] "[ORG]" : Change the value of the external current value display X-axis to "0."

(b) [Z] "[ORG]" : Change the value of the external current value display Z-axis to "0."

(c) [X] [Z] "[ORG]" : Change the values of the external current value display X and Z axes to "0."

The values other than the external current values do not change.
Similar operation is possible also with the external current value menu.

1.2.2 Changes

(1) "R" was displayed after No.51 of the offset display.
However, this has been deleted.

(2) The first display when power was turned on was an alarm display. However, now the first display is the all-position display. The data displayed in the all-position display has not changed.

(3) There were independent position increment displays. However, these displays are now eliminated.

The displays change cyclically in the following sequence if the page key is depressed in the position function mode:

1. All-position display
2. Program restart information display
3. Display for remaining numbers of pulses of stored stroke limit
4. Spindle PG pulse number display
5. Servo position deviation display
6. Command pulse integration register display
7. External current value display
8. Current value display

(4) By depressing [OPS] function key the following four menus are displayed cyclically:

1. All-position display
2. Spindle PG pulse number display
3. External current value display
4. Current value display

1.3 UPGRADING 9″ EDIT OPERATION

Basically, this function upgrades and changes the 9″ character CRT. The compact interactive function has the same function.

1. Adds the [N] **** " [ERASE] "function to the edit function.

The section between the word in with the cursor on and the sequence No. designated by "N****" is erased.

Erasing is performed as follows:

(1) Key in Address N and subsequent number in the program menu. ([N] [1] [2] [3] [4])

(2) Depress the [ERASE] key to erase up to word N****. The cursor then moves to the top position of the word next to the erased word.
"NOT FOUND" is displayed and the display remains as before if N**** cannot be found.

2. Add the No. O automatic renaming function during program input by tape. No. O input by keying is registered as the program No. if No. O input by keying in [O] **** [IN] operation differs from No. O in the taped program. Input No. O in the taped program by keying in and inputting the NC machine after rewriting it to program No.
Rewriting is not executed if tape without No. O is input.

3. The function with/without feed during tape output (O to 9999 including "OUT" operation) is added by the NC machine. This function becomes effective by establishing #6008 D4 = 1.

2. NEW TOOL SETTER FUNCTION

2.1 GENERAL

Offset amounts and work coordinate system shift amounts can be written automatically by manual operation even if the machine is not equipped with a touch sensor.

This function simplifies setup during tool change.

This function differs from the previous tool setter function with a touch sensor in tool coordinate memory value and in work coordinate shift amount.

The previous tool coordinate memory data was the differential distance value between the touch sensor and the tip of the selected tool. This has been changed to the differential distance value between the workpiece end face and the tip of the tool. The work coordinate system shift amount has been changed the shift amount from the projected home position to that from the workpiece end face.

However, the approach to the coordinate system does not change. This function is called the new tool setter function.

2.2 WRITING TOOL COORDINATE MEMORY

Regardless of whether or not a measuring tool for a setter in the machine exists, the tool coordinate memory amount can be written by manual operation as follows.

The tool coordinate memory amounts written in this case will be the differential distance value between the workpiece end face and the tip position of the applicable tool.

2.2.1 Writing Procedures

(1) Home position return is executed by manual operation.

(2) Mount the workpiece.

(3) Set the tool coordinate memory value writing mode setting (6001 D6) to "1." ("PST 1" key ON in compact interaction). The write mode is set by this setting.

The display automatically changes to the

function OFFSET (TOOL-SHIFT) display as shown in Fig. 3 and the

lamp blinks.

(4) Select the machining tool.
The tool coordinate memory matching the tool post No. (tool post No. input in #1317) selected at this time is automatically selected.

This function is effective only if parameter #6020 D3=1. If the tool coordinate memory No. is automatically selected, memory No. selection by the cursor or by the page key is prohibited.

If the tool coordinate memory No. is not selected automatically, the cursor is moved to the tool coordinate memory No. by operating the cursor key or page key.

(5) Make one cut of the workpiece skin.

2.2.1 Writing Procedures (Cont'd)

(6) Have the Z-axis only retract and stop the spindle, keeping the X-axis as it is.

(7) Measure the workpiece diameter.

(8) If the above setting is not turned on, select the tool coordinate memory picture [T] ** of the [OFS] function manually.

(9) By inputiing the [MX] workpiece diameter and [WR] , the tool coordinate memory "X" is automatically written by the following calculation:
(Tool coordinate memory "X") = (workpiece diameter)
− (Position machine "X" value)

(10) Make one cut of the workpiece end face.

(11) Have the X-axis only retract and stop the spindle, keeping the Z-axis as it is.

(12) Input [MZ] , [O] and [WR] to automatically write in the tool coordinate memory "Z" by the following calculation.
(Tool coordinate memory "Z") = − (Position machine "Z" value)

(13) Operate (4) to (12) for all tools, both for X and Z axes.

(14) Set the tool coordinate memory value write mode setting (#6001 D6) to "0" . ("METER 1" key OFF in compact interaction.)

The write mode is cancelled and the [OFS] lamp stops blinking.

Precaution :

Tool coordinate memory value write mode setting is not effective if home position return is not performed even once after turning on power.

2.3 WRITING Z-AXIS WORK COORDINATE SYSTEM SHIFT AMOUNT

2.3.1 Writing Procedures

(1) Home position return is executed by manual operation.

(2) Mount the workpiece.

(3) Set work coordinate system shift value writing mode setting (#6001 D5) to "1." ("PST 2" : key ON in compact interaction.) After this setting, the write mode is set up.

At this time, OFFSET (WORK-SHIFT) display of the OFS function automatically changes to T00 and the OFS function key lamp blinks.

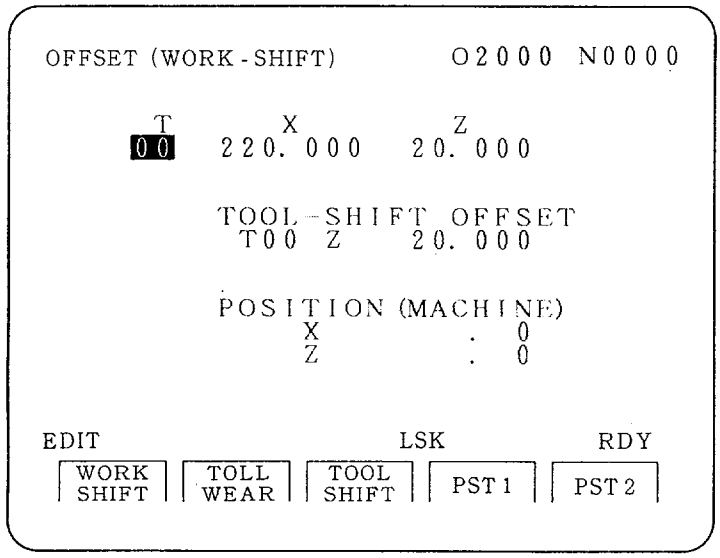


Fig. 4 Work Coordinate System Shift Display (Compact Interactive Function)

(4) Make one cut of the workpiece end face.

(5) Have the X-axis only retract, keeping the axis Z as it is.

(6) Measure the workpiece cutting allowance.

(7) Input [P] [Z], cutting allowance and [WR]. The result of the following calculation is automatically written in the work coordinate system shift memory "Z." The tool offset memory matches the tool post No. input in #1317.

(Work coordinate system shift memory "Z")
= (Cutting allowance) − (Tool coordinate memory "Z" value)
− (Position machine "Z" value)

(8) Changing Z-axis work coordinate system shift amount.

(a) If further shifting of the program is desired (e.g. a cutting allowance from the workpiece end face) , the work coordinate system shift memory can be rewritten for both the X and Z axes by MDI operation.

Calculate the shift amount, depress the [U] or [W] key and input the value to add the value keyed in to the value presently displayed.
The code for addition is by #6018 D6.

#6018 D6=0 : Workpiece shift amount + cutting allowance amount (with W code)

#6018 D6=1 : Workpiece shift amount − cutting allowance amount (with W code)

The procedure for rewriting the work coordinate system shift memory "X" value by [U] is the same.

#6018 D6=0 : Workpiece shift amount + cutting allowance amount (with code U)

#6018 D6=1 : Workpiece shift amount − cutting allowance amount (with code U)

(b) Normally, the [WR] and numeral keys are effective in this case. Minute changes such as cutting allowance values are input by adjusting by the [U] or [W] key or by inputting [P] [Z], cutting allowance value and [WR] again.

(9) Set the writing mode setting (#6001 D5) of the work coordinate system shift memory value to "0." ("PST 2" and key OFF in compact interaction.) The writing mode is cancelled and the [OFS] lamp stops blinking.

(10) By manual home position return after changing the work coordinate system shift memory, the work coordinate system is automatically set.
If parameter #6011 D0 = 1 (if the tool coordinate memory No. follows the last two digits of T ** SS), refer to the Coordinate Setting Operator's Instruction Manual. Work coordinate system setting is not performed in manual home position return.

Precaution :

Mode setting to write work coordinate system shift memory values is not effective if home position return is not performed even once after turning power on.

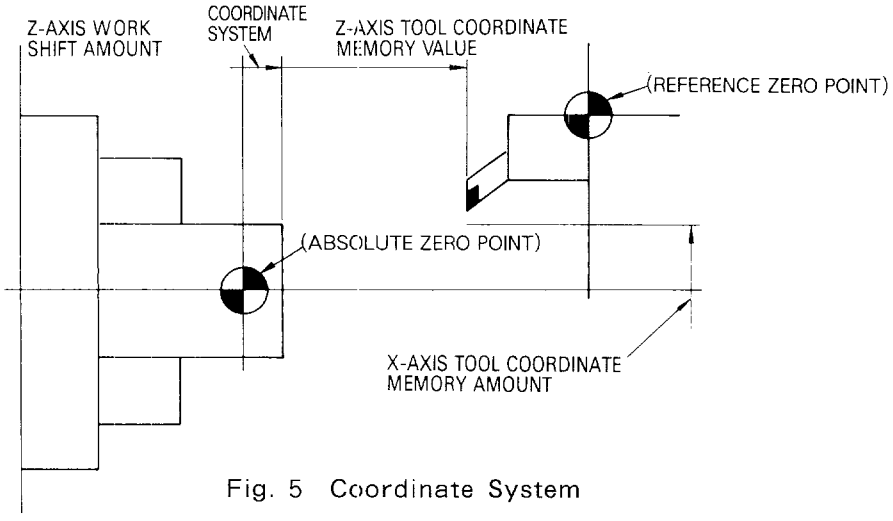


Fig. 5 Coordinate System

2.4 OFFSET DISPLAY

The following offset display will be shown if the new tool setter function is added.

The following displays can be changed cyclically by depressing the OFS key.

The OFS key is disregarded during the PST 1 or 2 mode.

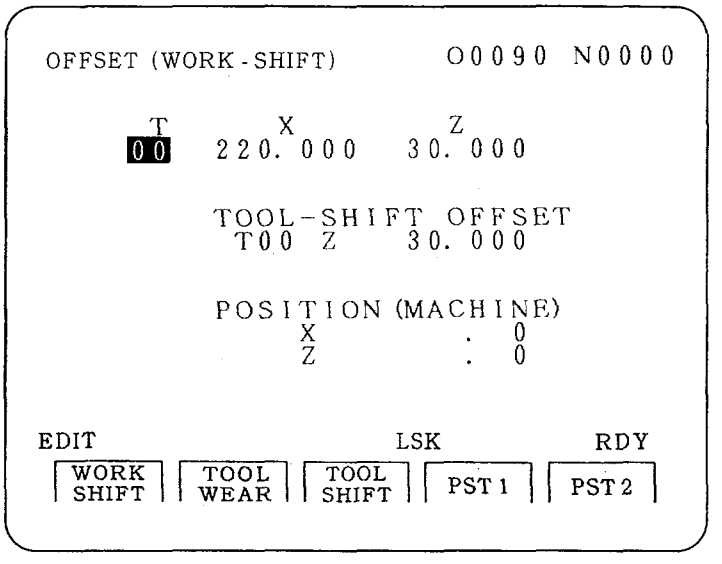


Fig. 6 Display No.1
Work Coordinate System Shift Display
(Compact Interaction)

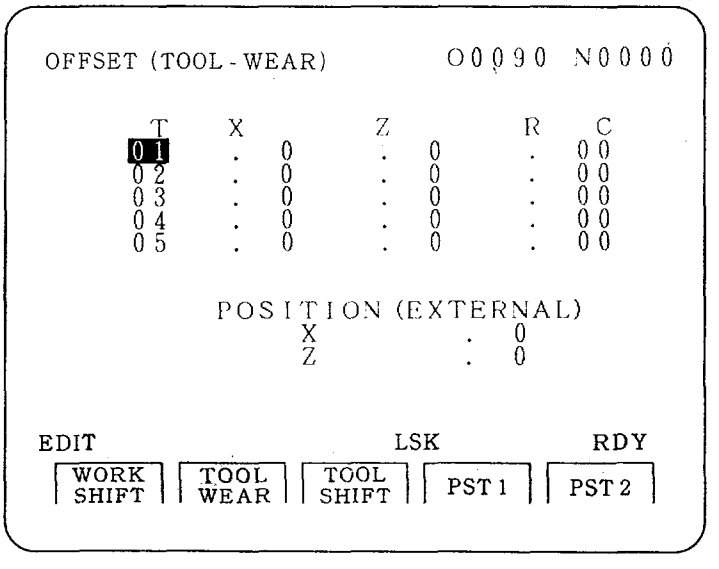


Fig. 7 Display No.2
Tool Offset Memory Display
(Compact Interaction)

2.4 OFFSET DISPLAY (Cont'd)

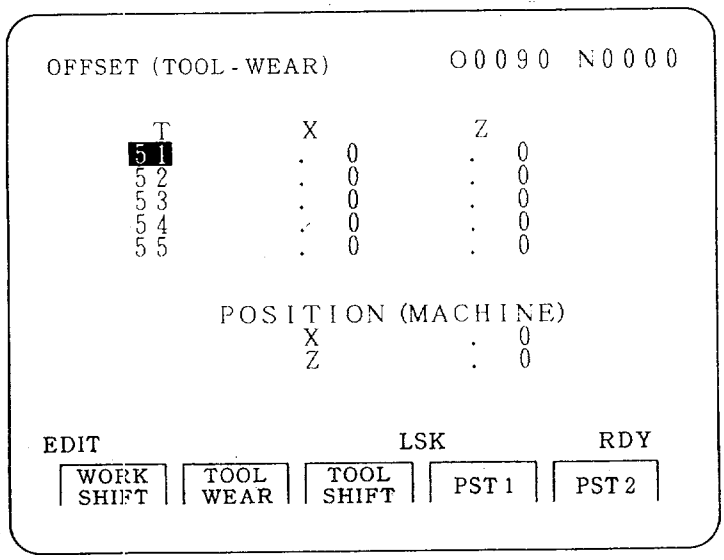


Fig.8 Display No.3
Tool Coordinate Memory Display
(Compact Interaction)

3. NEW COORDINATE SYSTEM SETTING FUNCTION

3.1 GENERAL

This function has a new approach to coordinate setting which is entirely different from previous ones. The features are shown below :

(a) The approach to coordinates is machine coordinate system.

(b) The blade tip point can always be displayed as the current value (absolute).

(c) Coordinate system setting is not required in programs.

3.2 DESCRIPTION OF FUNCTIONS

How and on what timing are coordinate system set?

3.2.1 Operational Expressions for Coordinates Setting

Various types of timing are used to set coordinates.
However, the following operational expressions are always used in setting coordinate system :

(X-axis coordinate value)
= (Machine position) + (Tool coordinate system memory value)
+ (Work coordinate system shift amount)

(Z-axis coordinate value)
= (Machine position) + (Tool coordinate system memory value)
+ (Work coordinate system shift amount)

(1) The machine position is the position machine.

(2) Tool coordinate memory value Nos. are divided into two systems depending on whether the timing of next coordinate system setting is manual or automatic.

(3) The work coordinate system shift amount is data of offset T 00, X, and Z.

3.2.2 Coordinate System Setting Timing in Manual Mode

In the manual mode, coordinate system is set on timing (a) to mentioned below (c). The tool coordinate memory Nos. in this case are produced based on tool No. binary values set from I/O inputs #13174 (TP1) to #13178 (TP8) and are used for arithmetic operations.

Set up is performed in two modes, namely, setting up inside the NC machine and coordinate system setting by sequencer requests. On a sequencer request, coordinate system setting is executed and Output #121974 (coordinate system setting end output) is turned on if Input #13127 (coordinate system setting request input) turns on.

(a) At the time of the home position return.

(b) When the blade tip contacts the sensor with the tool setter during measurement.

(c) When the turret is indexed manually.

Precaution :

Coordinate system setting in the manual mode is not performed if Parameter #6011 D0 = 1. If the tool coordinate memory No. follows the last two digits of T**$$, refer to par. 2.3.

3.2.3 Coordinate System Setting Timing in Auto Mode

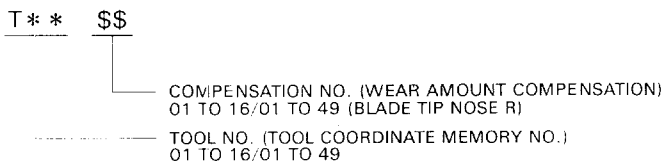
Coordinate setting is performed in the auto mode if the turret is called by the T code. In this case, the tool coordinate memory No. uses the turret No. or offset No. instructed to the command display for arithmetic operations.

Unlike the T code command by the conventional compensation method, the T code command in the coordinate system setting specification instructs as follows :

The tool coordinate memory No. can be selected to be set in the first or last two digits of T4 digits by setting parameter #6011 D0.

Depending on parameter setting, the execution of coordinate system setting differs as follows :

(1) Setting first two digits (#6011 D0 = 0)



Precaution :

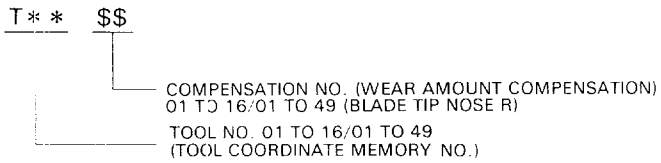
The tool coordinate memory No. will be (tool No. + 50).
For this reason, select data of (tool No. + 50) as the tool coordinate memory value.

(a) By executing the T**$$ command, coordinate system corresponding to the selected tool coordinate memory No. is set after the tool post moves by the offset amount to the compensation No. simultaneously with turret calling regardless of the tool post location.

(b) T**00 command executes turret calling and offset value cancel shift simultaneously regardless of the tool post location. Then coordinate system is set corresponding to the selected coordinate memory No.

(c) By this coordinate setting, the coordinate system at the tool blade tip are always set.

(2) Setting last two digits (#6011 D0 = 1)



The tool coordinate memory No. will be compensation No. + 50. For this reason, the tool coordinate memory value will select compensation No. + 50.

(a) By executing the T**$$ command coordinate system corresponding to the selected tool coordinate memory No. is set after moving by the offset amount to the compensation No. simultaneously with turret calling regardless of the tool post location.

(b) By executing the T**00 command moving of cancel operation of the offset amount simultaneously with turret calling is executed regardless of the tool post location. Coordinate system setting is not executed at this time.

(c) By this set up, coordinate system at tool blade tip are always set.

3.3 PRECAUTIONS IN USING CONVENTIONAL G 50 T****FUNCTION

Do not issue the G 50 T**** command when using in the coordinate system setting specification. An error results if the command is issued.

4. FS AUTO EDIT FUNCTION

4.1 GENERAL

This function creates optimum programs by storing effective values of feed speed and spindle revolution speed during auto operation and feeds them back to the NC part program. In the following descripitions, feed speed (F code) and spindle revolution speed (S code) are called "FS."

4.2 INPUT AND OUTPUT

The I/O signals needed in FS editing are as follows :

4.2.1 Input

(1) FSCM (#13134) :

FS edit mode. Prepares for FS memory and for FS memory change during FS editing when the signal is on.

(2) FSMEM (#13135) :

FS memory. Call up feed speed and spindle revolution speed in the internal memory during auto operation when the signal is on.

(3) FSCH (#13136) :

FS memory change. Reflects data called up in automatic operation in the NC PART program while auto operation pauses when the signal is on.

(4) FSCLR (#13126) :

FS data clear. Clears the FS data stored when the signal is on.

4.2.2 Output

(1) FSMD (#12183) :

During FS edit mode. The signal shows that FSCM is on and that FS storage and FS memory edit is ready. Feed speed and spindle revolution speed are cancelled when the status of this signal changes from on to off.

(2) FSCE (#12184) :

FS memory change end. The status of this signal becomes on when the status of FSCH becomes on and data is reflected on the NC PART program. The status of this signal changes to off if the status of FSCH becomes off again.

(3) FSCLRE (#12233) :

FS data clear end. The status of this signal becomes on if the status of FSCLR becomes on and if the stored FS data is cleared. The status of this signal changes to off if the status of FSCLR becomes off again.

4.3 DESCRIPTIONS OF FUNCTIONS

Rough calculations of FS data only are programmed by the NC PART program and the program is optimized by feed and spindle override after cutting.

The optimum data and positions of actual cutting in the program are stored in the memory contained in the NC machine. By turning on FS memory change input, the data and positions are reflected in the part program.

4.3.1 Operation Procedures

(1) Set up the system ready to operate.

(2) Search the operation program in the memory mode.

(3) Trun on FSCM.

(4) Check that FSMD has been output and start memory operation.

(5) Change the feed or spindle revolution speed for optimum value.

(6) Make FSMEM when optimum condition is achieved.

(7) Execute steps (5) and (6) repetitively.

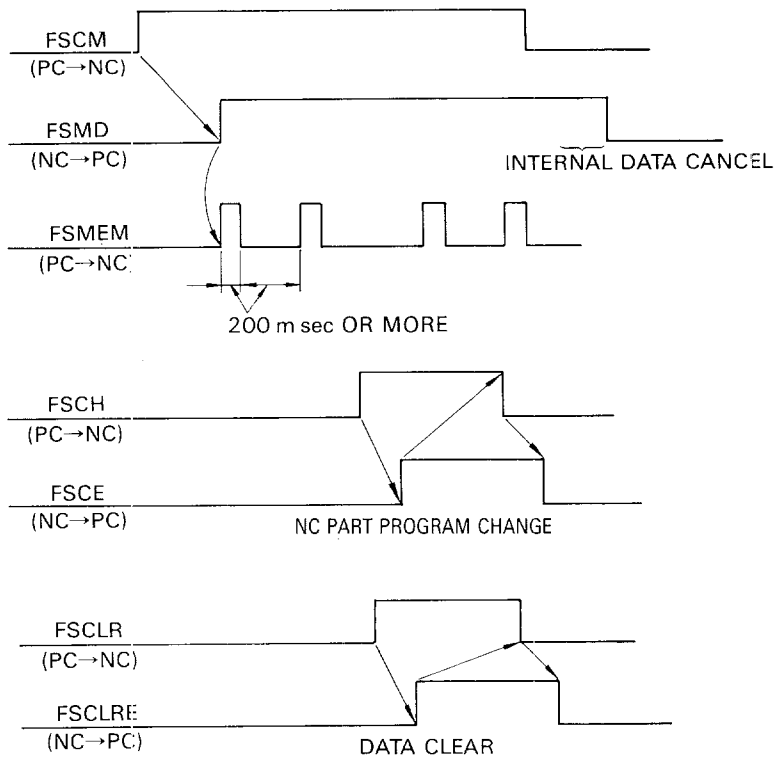
(8) Set up the operation pause status (label skip status).

(9) Turn on FSCH. Put the execution values stored in FSMEM in the PART program.

(10) Turn off FSFSCH.'SCE breaks.

(11) Turn off FSCM. The data stored in step (5) and (6) are cancelled.

4.3.2 Timing Chart



4.3.3 Examples of FS Editing

The following two edit patterns are available by selecting parameter #6008 D6.

(1) F/S is edited every time (#6008 D6 = 0)

(a) (Before editing)

```
O 1000 ;
N 1 G28 U0 W0 ;
N 2 G50 TS100 ;
N 3 T0103 ;
N 4 G40 G97 G99 S1500 M08 ;
N 5 G00 X43.236 Z10. M03 ;
N 6 Z1. ;
N 7 G01 X39.216 Z-6.5 F.15 ; - F override 80% S override 110% FSMEM is
    on.
N 8 X38.864 ;
N 9 G03 X18. Z-20.142 R19.85 ; - F override 60% S override 100% FSMEM is
    on.
N 10 G01 Z-20.5 ;
N 11 X12. Z-22.232 ; - F override 50% S override 80% FSMEM is on.
N 12 G40 X10. F1. ;
.
.
.
.
.
.
.
.
```

(b) (After editing)

```
O 1000 ;
N 1 G28 U0 W0 ;
N 2 G50 TS100 ;
N 3 T0103 ;
N 4 G40 G97 G99 S1500 M08 ;
N 5 G00 X43.236 Z10. M03 ;
N 6 Z1. ;
N 7 G01 X39.216 Z-6.5 F.12 S1650 ; - F override 80% S override 110%
N 8 X38.864 ;
N 9 G03 X18. Z-20.142 R19.85 F.09 S1500 ; - F override 60% S override 100%
N 10 G01 Z-20.5 ;
N 11 X12. Z-22.232 F.08 S1200 ; - F override 50% S override 80%
N 12 G40 X10. F1. ;
.
.
.
.
.
.
.
.
```

4.3.3 Examples of FS Editing (Cont'd)

(2) F/S is edited retroactively (#6008 D6 = 1)

In this case F/S is edited retroactive to the point where FSMEM was last closed.

(a) (Before editing)

```
O 1000 ;
N 1 G28 U0 W0 ;
N 2 G50 TS100 ;
N 3 T0103 ;
N 4 G40 G97 G99 S1500 M08 ;
N 5 G00 X43.236 Z10. M03 ;
N 6 Z1. ;
N 7 G01 X39.216 Z-6.5 F.15 ; - F override 80% S override 110% FSMEM
    is on.
N 8 X38.864 ;
N 9 G03 X18. Z-20.142 R19.85 ; - F override 60% S override 100% FSMEM
    is on.
N 10 G01 Z-20.5 ;
N 11 X12. Z-22.232 ; - F override 50% S override 80% FSMEM is on.
N 12 G40 X10. F1. ;
.
.
.
.
.
.
.
.
.
```

(b) (After editing)

```
O 1000 ;
N 1 G28 U0 W0 ;
N 2 G50 TS100 ;
N 3 T0103 ;
N 4 G40 G97 G99 S1200 M08 ; - S override 80% is reflected.
N 5 G00 X43.236 Z10. M03 ;
N 6 Z1. ;
N 7 G01 X39.216 Z-6.5 F.08 ; - F override 50% is reflected.
N 8 X38.864 ;
N 9 G03 X18. Z-20.142 R19.85 ;
N 10 G01 Z-20.5 ;
N 11 X12. Z-22.232 ;
N 12 G40 X10. F1. ;
.
.
.
.
.
.
.
.
.
```

4.3.4 Specification Conditions for FS Editing

(1) Conditions to store FS data in internal memory

(a) During FSMD output.

(b) During automatic operation (STL is on).

(c) Actual feed and spindle revolution speed values are stored by the rise of FSMEM.

(d) All the data stored in (3) are cancelled by the fall of FSCM.

(e) The maximum memory cycles (the cycles to turn on FSMEM) are 128 cycles.

(2) Conditions to reflect FS data in part program

(a) During FSMD output.

(b) During automatic operation pause. (Label skip status)

By changing the status of FSCH from off to on under this condition, F/S data are inserted in each block in accordance with the stored data.

4.4 PRECAUTIONS

(1) The newest feed and spindle revolution speeds become effective if the status of FSMEM is on more than twice in one block. The FSMEM count is not increased.

(2) FS data is inserted in the sub-program itself if the status of FSMEM is on during a sub-program. Caution must be exercised as other programs are also called up.

(3) FS editing during a compound fixed cycle allows only F/S of O.D. rough cutting cycles.

(4) Blocks in on status are changed with F if FSMEM is on in constant peripheral speed control. Blocks are not changed with S.

(5) The CYCLE START switch does not operate while FSCH is on even if it is depressed.

(6) F/S data is stored if FSMEM is on during cutting.

(7) S data is stored if FSMEM is on during rapid feed.

(8) A warning is issued if FSMEM is on more than 128 times, and data are not stored even if FSMED is on further.

(9) FSMD is not on even if the status of FSCM is on during automatic operation.

(10) The CYCLE START switch does not operate while FSMEM is on.

(11) A warning is issued if editing is performed while FSMD is on after finishing FS editing.

5. AUTO NOSE R FUNCTION

5.1 GENERAL

Nose R compensation is performed by the nose R control point (0 to 9) command and by G 41 and G 42. Selection of compensation side moving from an end face to outside diameter becomes simple. By executing the T command, a tool, as well as the nose R and control point corresponding to the tool, are selected.

At this time, coordinates of the tool based on this machine coordinate system are set. After issuing the T command, nose R compensation calculations are made by the control point and by G 41 and G 42.

5.2 CONDITONS TO MAKE AUTO NOSE R COMPENSATION FUNCTION EFFECTIVE

Auto nose R compensation becomes effective by four conditions, namely, designation of projected blade tip points set by compensation No., by nose R radius, by setting of the compensation side by the G 41 and G 42 commands, and by the T code command. They are described below.

5.2.1 Designation of Projected Tool Tip Points

(1) Projected edge point direction

Auto nose R is determined by designating the projected tool tip point position by using a numeral between 0 and 9 to regulate the shift direction. The directions are shown in Fig.9 (X plus specification).

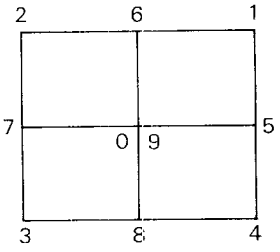


Fig. 9 Projected Tool Tip Point Direction

(2) Method to set projected tool tip point.
Projected tool tip points can be set using the offset screen. (See Fig.10)

(a) Select the [OFS] function.

(b) Select the conpensation memory of the matching tool No. from T 01 to 49.

(c) Input [C], [3] and [WR] by MDI if there are three projected tool tip points.

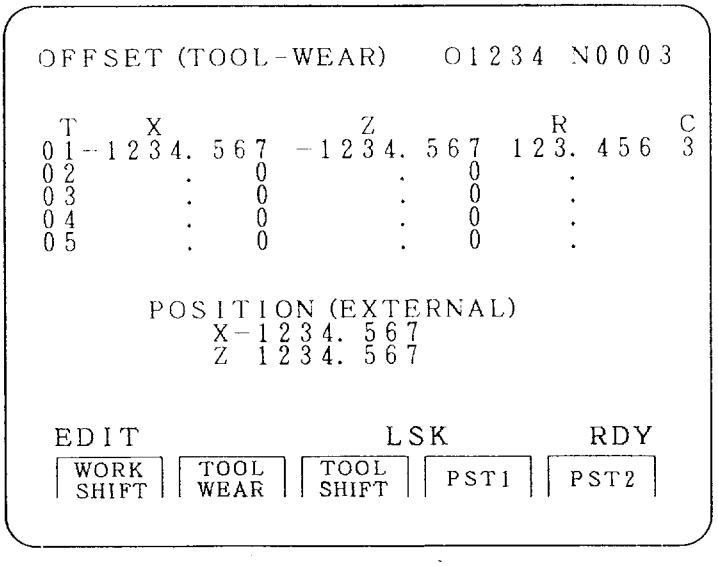


Fig. 10 Offset Display (Compact Interactive)

5.2.2 Setting Tool Tip R Data Memory

(1) Setting nose R data memory
The nose R data memory can be set on the offset display as shown in Fig.10.

(a) Select the [OFS] function.

(b) Select the compensation memory of the corresponding to tool No. from T 01 to T 49.

(c) Input [R] , [8] and [WR] by MDI if the nose R data is 0.8 mm.

5.2.3 G41 and G42 Commands

The compensation direction can be instructed during cutting by the G 41 and G 42 commands. (X plus specification)

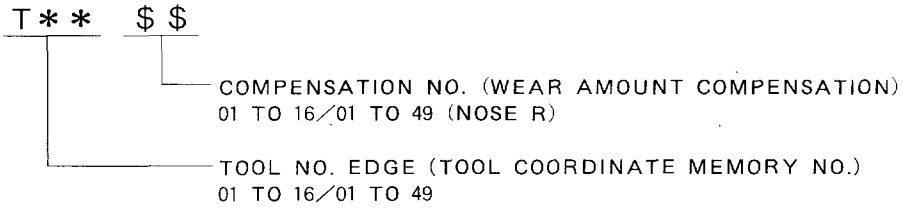
G 40 : Nose R compensation OFF

G 41 : Left side of moving direction is nose R center.

G 42 : Right side of moving direction is nose R center.

5.2.4 T Code Commands

The T code No. (4 digit) commands select coordinates system, wear compensation and nose Rs.
Refer to Sect.3, "NEW COORDINATE SYSTEM SETTING FUNCTION," for details.



5.3 COMPENSATION CALCULATIONS OF AUTO NOSE R COMPENSATION APPROACH AND RETRACT

In auto nose R compensation, the following functions have been upgraded from the conventional YASNAC LX3 nose R compensation function.

5.3.1

Block compensation G 00 → G 00 is not performed.
Positioning only is performed.

5.3.2

In approach G 00 → G 01 (G 02, G 03), a vector (wall) is theoretically set on the tool to make compensation to prevent overcutting and overshock.

In recess G 01 (G 02, G 03) → G 00, a vector (wall) is theoretically set to make compensations as follows to prevent omission in cutting.

(1) Approach G 00 → G 01 or recess G 01 → G 00

(a) If the direction of the cutting command moving axis is larger than 45° relative to Z-axis (| X/2 | > | Z |), a vector (wall) parallel to Z-axis is theoretically erected to make compensation calculations with the cutting moving axis. The tool moves toward this calculated position. See Fig.11.

(b) If the direction of the cutting command moving axis is smaller than 45°, or is equal, relative to Z-axis (| X/2 | ≦ | Z |), a vector (wall) parallel to X-axis is theoretically erected to make compensation calculations with the cutting moving axis. The tool moves toward this calculated position. See Fig.12.

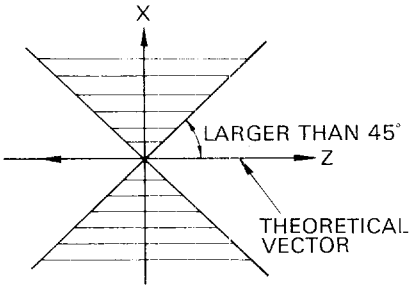


Fig. 11

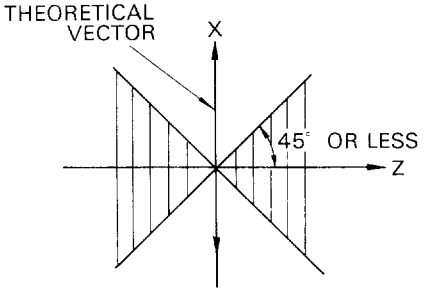


Fig. 12

Example : 3 tool tip points

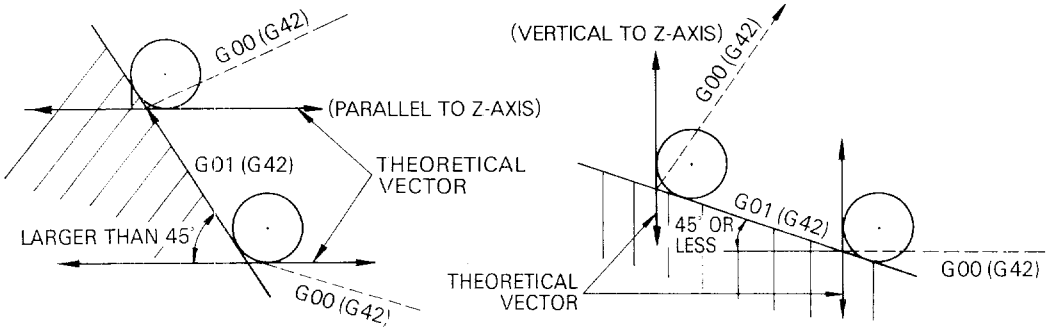


Fig. 13 Example of 3 Tool Tip Points

(2) Approach G00 → G02 (G03) or recess G02 (G03) → G00

(a) If the vector direction of the circular arc cutting command moving axis is larger than 45° relative to Z-axis, a vector (wall) parallel to Z-axis is theoretically erected to make compensation calculations with the cutting moving axis. The tool moves toward this calculated position. See Fig.14.

(b) If the vector direction of the circular arc cutting command moving axis is smaller than 45°, or is equal, relative to Z-axis, a vector (wall) parallel to Z-axis is theoretically erected to make compensation calculations with the cutting moving axis. The tool moves toward this calculated position. See Fig.15.

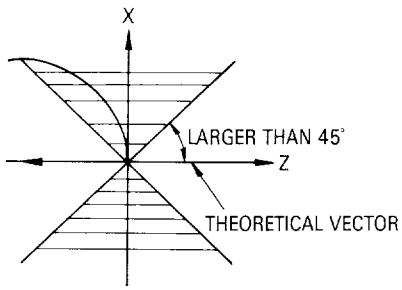


Fig. 14

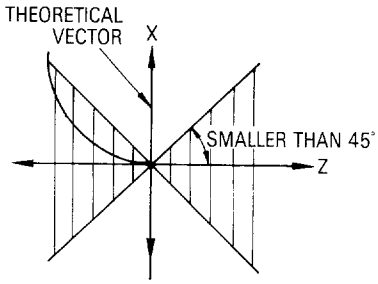


Fig. 15

Example : 3 tool tip points

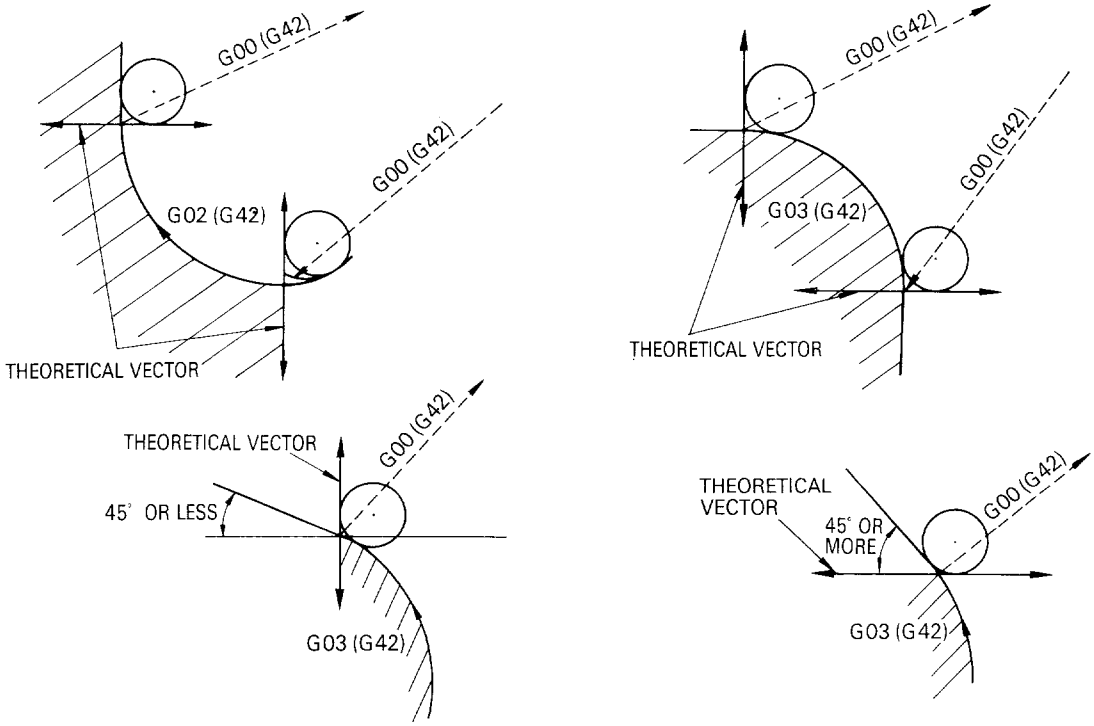
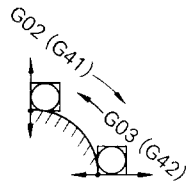
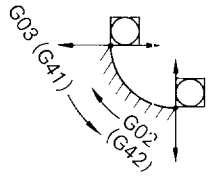
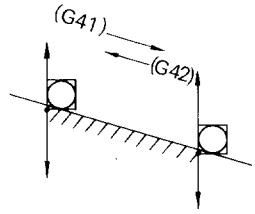
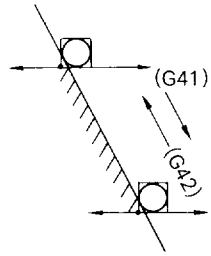
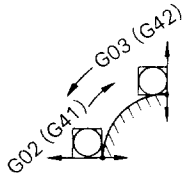
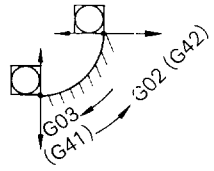
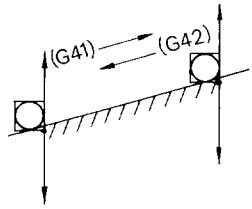
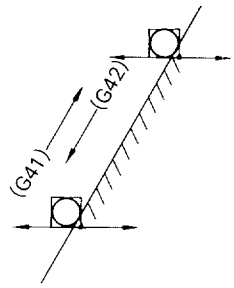
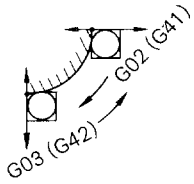
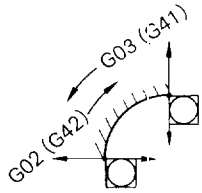
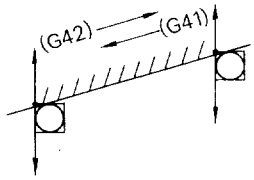
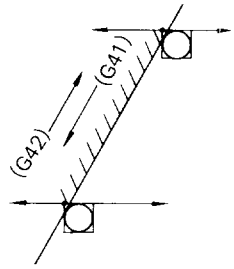
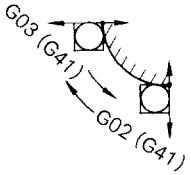
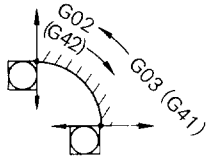
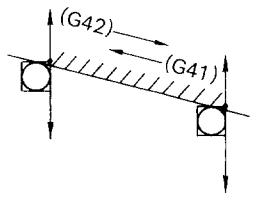
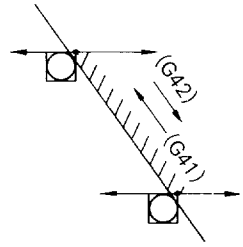


Fig. 16 Example of 3 Tool Tip Points

5.3.3 Patterns of Auto Nose R Approach and Retract

	Circular		Linear	
			45° or less	45° or more
① C=3				
② C=4				
③ C=2				
④ C=1				

5.4 CANCEL FUNCTION (G40, G01)

Auto nose R function can cancel G40 G01 X_Z_I_K_; command. In addition to cancel function of G40 G00 X_Z_I_K_;, G01 has the same function.

This function makes a theoretical wall by command I or K in the block before G40 G01 command and executes nose R calculation.

5.5 PRECAUTIONS

(1) In G40 G00 X_Z_; or G40 independent block, the center of nose R is provided on the end point normal.

(2) In G51 independent block or G28 block, nose R complete cancel is executed in the block immediately before G51 or G28.

(3) In G40 G00 X_Z_I_K_;, node calculation is executed with vector indicated as "I" or "K" in the block immediately before G40 G00.

(4) When G01 and G41 (G42) are commanded simultaneously, the center of nose R is provided on the normal of the starting point of the next block.

In canned cycle, the first block can be started up by G01 G42, however, it cannnot be started up by the same command afterward.

(5) G40 G01 X_K_; and G40 G01 Z_I_; have the same command format as that of beveling command, however, cancel function has priority only when G40 command is provided.

6. Z-AXIS TOOL CHUCK BARRIER FUNCTION

Barrier for each tool in Z-axis direction is automatically set by setting the length between absolute zero point and the barrier by extending normal tool stored stroke limit function.

6.1 GENERAL

When #13181 (tool range change) signal is input in the timing for changing the tool, chuck barrier value corresponding to tool number is set in Z-axis direction by reading the tool numbers set in #13174 to #13178 (TP1 to TP8).

This function is effective when parameter #6011 D4 = 1.

6.2 OPERATION

(1) Each tool coordinate memory value and work coordinate system shift value is set in PST mode.

(2) A value of the length between the absolute zero point to a desired chuck barrier in Z-axis direction is set in setting parameter (#6573).

(3) Stored stroke limit 3 is effective. (#6001 D2 = 1)

6.2 OPERATION (Cont'd)

(4) Stored stroke limit 3 setting parameter (#6507) in Z-axis direction is set in the tool changing timing in manual or automatic mode, by the following operation :

#6507 = #6573 − (Tool coordinate memory value + Work coordinate system shift value)

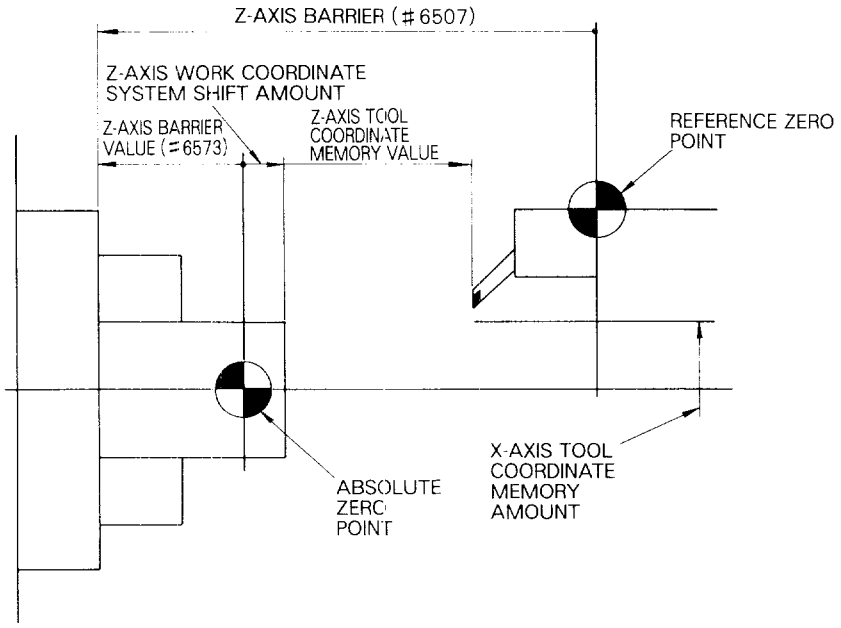


Fig. 17 Coordinate System

6.3 PRECAUTIONS

The following conditions are required for effective functioning

(1) Stored storoke limit Z is available as an option.

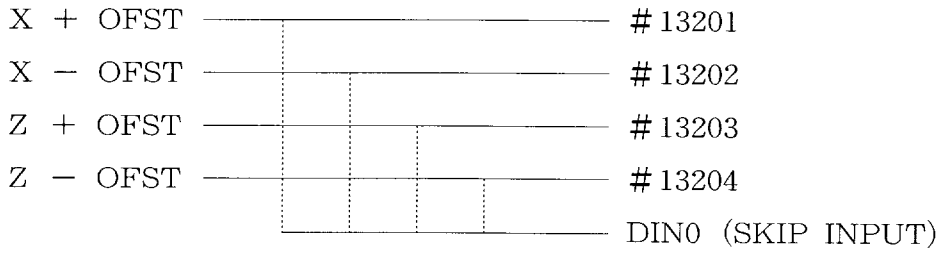
(2) When tool stored stroke limit is available, each tool stored stroke limit value of Z-axis only can be reset.

7. OTHER ADDITIONAL FUNCTIONS

7.1 HIGH-FREQUENCY SKIP SIGNAL APPLIED FOR MACHINE SETTER

When high-frequency skip signal is used together with sensor input signal {X + OFST (#13201), X−OFST (#13202), Z + OFST (#13203), Z−OFST (#13204)}, it will improve absolute accuracy of measured value.

High-frequency skip signal is switched by #6011 D3.



#6011 D3 1 : Effective

0 : Ineffective

7.2 WORK SHIFT FUNCTION ADDED (G54)

The current coordinate system can be shifted for setting amount of #6571(X) and #6572(Z) by G54 independent command.

Shift amount can be cancelled by G50 independent command.

- X-axis coordinate value = Machine position + Tool coordinate value + Work coordinate system shift amount + #6571
- Z-axis coordinate value = Machine position + Tool coordinate value + Work coordinate system shift amount + #6572

G54 ; Coordinate system shift effective

⋮

G50 ; Coordinate system shift ineffective

7.3 ABSOLUTE COORDINATE SWITCHING ADDED WHEN MACHINE LOCK EFFECTIVE IS SWITCHED TO INEFFECTIVE

To prevent an error in coordinate systems when machine lock is ineffective after drawing display at machine lock effective, absolute coordinate system is reset by the tool number currently provided when machine lock is switched from effective to ineffective.

7.4 ADDITIONAL PARAMETER SWITCHING TO AUTO NOSE R COMPENSATION AND COMMAND T (INCLUDING G50T) FUNCTION

A parameter which switches auto nose R compensation and coordinate system setup function by command T to normal nose R compensation and command T (including G50T) function, is added.

#6011 D2 0 : Auto nose R function and coordinate system setup function by command T effective
1 : Normal nose R function and command T (including G50T) function effective

7.5 HANDLE INTERPOLATION FUNCTION

7.5.1 General

When handle interpolation mode is commanded during program execution in memory, tape or MDI mode, cutting speed commanded by G01, G02 or G03 can be controlled by turning the handle in the plus direction.

7.5.2 Operation

(1) Select handle interpolation mode to set by panel key or I/O input in compact interactive function ; by setting parameter or I/O input in 9″ monochrome display.

Setting #6001 D4 1 : Effective
0 : Ineffective

I/O Signal #13066 1 : Effective
0 : Ineffective

(2) Depress cycle start button.

(3) "F" blinks and cutting stops before executing G01 (or G02, G03) program.

(4) The manipulator moves while turning the handle in the plus direction and starts cutting.
The moving speed is in proportion to handle rotating speed and command F is overridden.

EXAMPLE : When 1 rotation/sec = 100%, the relation between rotation speed and override is as follows :

#6349 parameter = 100 (pulse/sec)

1 rotation/sec or more	100%
0.5 rotation/sec	50%
0.01 rotation/sec	1%
0.01 rotation/sec or less	Stop

(5) The manipulator stops moving when the handle is stopped or turned in the minus direction.

(6) Turn off handle interpolation mode after cutting is completed.

7.5.3 Precautions

(1) Turning the handle at 0.2 rotation/sec or more makes a constant movement. If at 0.2 rotation/sec or less, the movement is intermittent.

(2) When turning off the mode during handle interpolation, the remaining movement is executed by command F.

(3) In case of handle 2 axis specification, only the first handle is effective.

(4) Handle interpolation is ineffective during thread cutting.

(5) Handle axis selection is not required for interpolation.

(6) Handle interpolation mode is disregarded during automatic mode handle offset.

(7) Normally, the command is overridden, disregarding setting of feed override selection switch.

(8) In dry run, override varies depending on dry run speed. Therefore, override varies depending on feed override selection switch setting and rapid traverse speed setting.

8. LIST OF NC PARAMETERS FOR COMPACT INTERACTIVE FUNCTION

These parameters are needed to realize the compact interactive function.

Parameters (Fixed)

#6009 D2 The compound compensation function
The T $$ ** command enables simultaneous setting of position offsets and of coordinate system

1 : Effective
0 : Not effective

#6015 D0 The automatic X-axis coordinate system setting effective axis

1 : Effective
0 : Not effective

#6015 D1 The automatic Z-axis coordinate system setting effective axis

1 : Effective
0 : Not effective

#6016 D4 The automatic nose R function

1 : Effective
0 : Not effective

#6016 D5 The simple measured value direct input function

1 : Effective
0 : Not effective

8. LIST OF NC PARAMETERS FOR COMPACT INTERACTIVE FUNCTION (Cont'd)

#6016 D6 The tool coordinate system setting function. The coordinate system setting function by the T4-digit command

1 : Effective
0 : Not effective

Parameters (Selective)

#6008 D4 With or without feed during tape output by NC

1 : Without feed
0 : With feed

#6008 D6 During automatic FS editing

1 : F/S edited retroactively
0 : F/S edited every time

#6009 D1 U and W are used instead if I and K are not available with G 71 to G 73.

1 : Effective
0 : Not effective

#6011 D0 The T**$$ command to make tool coordinate memory Nos.

1 : Lower 2 digits
0 : Upper 2 digits
(Related fixed parameter : #6016 D6)

#6011 D1 Enables the T code, G 41 and G 42 commands during compound fixed cycles

1 : Effective
0 : Not effective

#6011 D2 Interlock of auto nose R function and coordinate setup function by command T

1 : Effective normal nose R function and command T (including G 50 T) function
0 : Effective auto nose R function and coordinte system setup function

#6011 D3 Setter high-frequency skip signal in machine

1 : Effective
0 : Ineffective

#6011 D4 Check barrier function for each tool

1 : Effective
0 : Ineffective

Setting Parameters

#6001 D4 Handle compensation function

1 : Effective
0 : ineffective

#6001 D5 PST2 workpiece shift amount write request

1 : Effective
0 : Not effective

#6001 D6 PST1 tool coordinate system memory amount write request

1 : Effective
0 : Not effective

#6208 G 74 recessing canned cycle pull back amount

Unit : Input unit
Set range : 0 to 65535

#6209 G 75 recessing canned cycle pull back amount

Unit : Input unit
Set range : 0 to 65535

#6210 G 74 recessing canned cycle clearance amount

Unit : Input unit
Set range : 0 to 65535

#6211 G 75 recessing canned cycle clearance amount

Unit : Input unit
Set range : 0 to 65535

#6212 G 74/G 75 recessing canned cycle approach speed override
Unit : 1 = 100%
Set range : 0 to 127

Enables override to F in approach speed after second recessing cycle.

#6213 Reduces cutting amount of each cutting in units of 1% to G 71/G 72 cutting reduction override #6004 (D3 to D7) or to cutting override. D* (#6004 or input override) * (100 - (B 1) * #6013)/100 = cutting amount of each cutting

D : Cutting amount
N : Number of cutting cycles
Unit : 1 = 1%
Set range : 0 to 99

#6214 G 74, G 75 cycle hole bottom dwell time

Unit : 1 = 1ms
Set range : 0 to 65535

#6571 G 54 work shift function X coordinate shift amount

Unit : 1 = 0.0001 mm or 0.0001 inch
Set range : −99999999 to 99999999

NOTE : The NC machine automatically turns #6001 D5 and D6 on and off when the new tool setter function is used.
Therefore, setting them is not possible.

YASNAC LX3

CNC SYSTEM FOR TURNING APPLICATIONS

UPGRANDING FUNCTION

TOKYO OFFICE Ohtemachi Bldg, 1-6-1 Ohtemachi, Chiyoda-ku, Tokyo, 100 Japan
Phone (03) 3284-911' Telex YASKAWA J33530 Fax (03) 3284-9034
SEOUL OFFICE 8th Floor Seoul Center Bldg. 91-1. Sogong-Dong, Chung-ku, Seoul, Korea 100-070
Phone (02) 776-7844 Fax (02) 753-2639
TAIPEI OFFICE Shen Hsiang Tang Sung Chiang Build ng 10F 146 Sung Chiang Road, Taipei, Taiwan
Phone (02) 563-0010, -7732 Fax (02) 567-4677
YASKAWA ELECTRIC AMERICA, INC.
Chicago-Corporate Headquarters 2942 MacArthur Blvd. Northbrook, IL 60062-2028, U.S.A.
Phone (708) 291-2340 Fax (708) 498-2430
Chicago-Technical Center 3160 MacArthu' Blvd. Northbrook, IL 60062-1917, U.S.A.
Phone (708) 291-0411 Fax (708) 291-1018
MOTOMAN INC.
805 Liberty Lane West Carrollton, OH 45449, U.S.A.
Phone (513) 847-6200 Fax (513) 847-6277
YASKAWA ELECTRIC EUROPE GmbH
Niederhöchstädter Straße 73, 61476 Kronberg-Oberhöchstadt, Germany
Phone (06173) 9380 Telex 415660 YASE D Fax (06173) 68421
YASKAWA ELÉTRICO DO BRASIL COMÉRCIO LTDA.
Rua Conde Do Pinhal 8-5°, Andar Sala 51 CEP 01501-São Paulo-SP, Brasil
Phone (011) 35-1911 Fax (011) 37-7375
YASKAWA ELECTRIC (SINGAPORE) PTE. LTD.
Head Office CPF Bldg, 79 Robinson Road # 13-05, Singapore 0106, SINGAPORE
Phone 2217-530 Telex (87) 24890 YASKAWA RS Fax 224-5854
Service Center 221 Henderson Road, # 07-20 Henderson Building Singapore 0315, SINGAPORE
Phone 276-7407 Fax 276-7406
YATEC ENGINEERING CORPORATION
Shen Hsiang Tang Sung Chiang Building 10F 146 Sung Chiang Road, Taipei, Taiwan
Phone (02) 563-0010 Fax (02)567-4677

YASKAWA ELECTRIC CORPORATION

YASKAWA

TOE-C843-9.23D

© Printed in Japan October 19