

CNC 501

Programming and Operation of Lathes



York Technical College
452 South Anderson Road
Rock Hill, SC 29730

TABLE OF CONTENTS

I. General Safety and Standard Operating Procedures

II. OSP Control Functions

- Primary and Secondary Modes
- Parameters; Soft Limits vs. Stroke End Limits
- Function of all keys on Machine Operation Panel

III. Manual Machine Control

- Door Interlock restrictions

IV. Coordinate System and Program Zero Point

- Coordinate System
- Absolute Position Encoder Advantages

V. Program Codes

- G Codes
- Additional G Codes
- M Codes

VI. Program Format and Data Word/Address

- Refer to LB25-T min Program at Front
- Begin with Simple Examples; T-Command will be covered in detail later
- Discuss comments inside Parentheses
- Tool Offset relationship to zero set

VII. Angle Command

VIII. Circular Interpolation

- Write Simple Program on Board Part DR202-3
 1. Eliminate speed and gear range
 2. Write program with zero at front and back
- Write Program on Board for DR201-3
 1. Have a student key in the program on simulator as you write
 2. Add graphics commands to program, without explanation to students
- Call Up and Test Program
- Review

IX. Machining Guidelines

- Surface Footage, Feed rate and Depth of Cut
- Cutter Radius Compensation
 1. What CRC is; how it works; and it's advantages
 2. CRC Cancellation

X. Auto Chamfer and Automatic Radius

XI. LAP Cycles

- LAP Cycle Concept
- Types of LAP Cycles
- Write new program for DR201-3 and modify to use LAP cycles

XII. Miscellaneous Cycles

- Drilling Cycles
- Grooving Cycles
- Tapping Cycles

XIII. Threading Fixed Cycles

XIV. Subprograms, Schedule Programs, and Additional File Types

- Schedule Program – Explanation
- Common Variables
- Local Variables; use variable jaw-boring program as an example

XV. Graphic Commands

XVI. Appendix

Programming and Operation of 2 Axis Lathe

Course Objectives - Upon completion, the individual will be proficient in all basic skills necessary to allow the functional/productive operation of the machine tool and associated safety practices.

The course is designed to provide the knowledge and skills required to “translate” the part drawing into a finished product. The individual will be capable of defining the list of required processes, their logical/optimum sequence, create the complete CNC part program, install the appropriate tools correctly, establish the program zero point, and perform corresponding tool offsets.

Course emphasis is a blend of classroom instruction, time spent on the machine tool, and individually displayed skills.

Mastery of the topics is measured by actual demonstration and examples.

**GENERAL SAFETY
AND
STANDARD OPERATING
PROCEDURES**

General Safety and Standard Operating Procedures

SAFETY PRECAUTIONS/STANDARD OPERATING PROCEDURES.....

PRE-POWER UP CHECKLIST.....

CHUCK PRECAUTIONS.....

PRECAUTIONS FOR MACHINE OPERATION.....

SETUP.....

WORK PIECE LOADING AND UNLOADING.....

END-OF-DAY CHECKLIST.....

WHEN A PROBLEM OCCURS.....

OTHER GENERAL PRECAUTIONS.....

SAFETY PRECAUTIONS

Okuma machines are fully equipped with various safety devices to prevent operators and the machine itself from accidents. However, operators are urged to operate the machine with safety in mind. Strict observance of all safety guidelines indicated in the documentation provided with the machine is essential. The following are some points to observe when working with any machine tool.

PRE-POWER UP CHECKS

- 1) Close the doors of the electric control cabinet and the operation panel.
- 2) Never place obstacles around the machine.
- 3) Turn on power to the machine in the following sequence:
 - a) Depress the Emergency stop button
 - b) Turn on the Main power disconnect.
 - c) CONTROL ON button on the operation panel. (POWER ON button on the control cabinet on machining centers.)
- 4) Once the control 'boots up', release E-stop and press the Control On button.

CHUCK PRECAUTIONS

- 1) Before starting the spindle or cutting operations, close the front door.
- 2) Strictly observe the allowable spindle speed for the chuck installed. Never exceed the maximum allowable spindle speed.
- 3) When a chuck or fixture unique to the user's applications is used, check the allowable maximum spindle speed and run the spindle within the allowable range. Pay due attention to work piece gripping force and balance.
- 4) The maximum spindle speed can be limited by inputting the spindle speed with G50. To ensure safety in operation, input this spindle speed limiting command in the program (G50S****).
- 5) If the spindle is operated at a speed close to the allowable maximum speed:

Avoid imbalance in the work piece clamped in the chuck. Apply the maximum allowable pressure to grip the work piece since increased centrifugal force reduces the chuck's gripping force.

The maximum allowable spindle speed and applicable pressure for the chuck are indicated on the name plate attached to the front door as well as on the chuck body. The maximum allowable speed and the applicable pressure ensure a chucking force larger than one-third the original chuck gripping force with the standard soft-top jaw set in line with chuck body's outer periphery.

- 6) When special jaw larger than standard soft-top jaws are used:

Lower the spindle speed because the chuck's gripping force is reduced due to increased centrifugal force and lowered efficiency.

If the jaw-clamping nut (jaw nut) is outside the chuck's outer periphery, only one clamping bolt holds the jaws in place, causing very dangerous conditions. Jaw nuts must always be located within the chuck body's outer periphery

Machine the jaws to the shape of the work piece.

- 7) Tighten the bolts on the chuck body, jaws, and block securely. Clamping force should be greater than 40 to 50 kg.

DAILY CHECKS

- 1) Before starting daily operations, always check the lubrication oil levels.
- 2) Always use the specified brand or grade of lubrication oil.
- 3) For cutting fluid (coolant), use Okuma's recommendation whenever possible.
- 4) Change and replenish lubrication oil for each reservoir at the predetermined schedule as explained in the operation & maintenance manual.
- 5) Clean the filters periodically according to the schedule explained in the operation & maintenance manual.
- 6) Check the pressure gauges of the air and hydraulic lines to make sure they all read the correct values as specified in the operation & maintenance manual.
- 7) For any work required inside the machine door, turn off power and ensure safety before hand. For work done at the back of the machine that requires the operator to enter the machine-operating zone, do not forget to turn off power before attempting any work.

PRECAUTIONS FOR MACHINE OPERATION

- 1) Always follow the instructions given in the operation manuals.
- 2) Never run the machine without protective covers and doors, such as the front door and chuck cover.

- 3) Close the front door first before starting the machine.
- 4) With a new program, never attempt to start actual cutting operations. First run the program without setting a work piece in the machine to check machine operations and interference; after making sure that the program is completely free of bugs, cut a work piece in the single block mode operation. Only after making sure that the work piece can be cut without problems in the single block mode operation should the automatic mode operation be started.
- 5) Before attempting the following, always make sure that untended operation can be accomplished safely:

Spindle rotation
Turret indexing
Axes movements
- 6) While the spindle is revolving, never touch chips or the work piece.
- 7) Never try to stop a revolving object with hands or tools.
- 8) Confirm the jaw installation conditions, hydraulic pressure, and allowable maximum speed for the power chuck.
- 9) Check the installed conditions and arrangement of the tools.
- 10) Confirm the tool and zero offset settings.
- 11) Set the spindle speed and feed rate override dials to 10%.
- 12) Before feeding the turret, confirm the soft-limit settings and the emergency limit switch settings for both X and Z-axes.
- 13) Confirm the position where the turret index or rotation is allowed.
- 14) Confirm the tail stock position.
- 15) Make sure that cutting is conducted within the allowable transmission power and torque ranges.
- 16) Clamp the work piece in the chuck or fixture securely.
- 17) Check the cutting fluid nozzle positions. Set them at positions to supply cutting fluid correctly to the cutting point.

SETUP

- 1) Always make sure that the setup is complete
- 2) After changing the setup, operate the machine step by step to make sure the cutting can be conducted without problems.

- 3) Before replacing the chuck and/or chuck jaws, make sure that the new set is for the job intended.
- 4) When two or more workers work as a group, establish the necessary safety signs, for example, when lifting or setting heavy objects confirm with other workers whether or not it's 'okay' to start the next process.
- 5) When handling heavy objects, use a crane or equivalent tool.
- 6) When attempting unfamiliar setups, check the setup again before going on to the next step.

WORK PIECE LOADING AND UNLOADING

- 1) Load and unload work pieces securely.
- 2) Retract the turret before loading and unloading a work piece to a position where the cutting tools on the turret will not injure the operator's hands.
- 3) Before attempting to load or unload a work piece, make sure the spindle has come to a complete stop.
- 4) Before running a new program, first rotate only the spindle to make sure the work piece is securely clamped in the chuck.
- 5) To machine irregularly shaped work pieces, make sure the work piece is clamped in the chuck securely without imbalance.
- 6) When handling heavy work pieces, use a crane, hoist, or other tool.
- 7) Before setting a work piece in the machine, make sure the work piece has portions that can be used for proper chucking.

AT THE END OF THE DAY

- 1) Clean the machine.
- 2) Locate the turret at the predetermined retraction position.
- 3) Before leaving the machine, turn off all power switches.
- 4) Turn off power to the machine in the following sequence:
CONTROL OFF button on the operation panel. The main power disconnects.

WHEN A PROBLEM OCCURS

- 1) Stop all spindle(s) and axis movement by pushing the closest EMERGENCY STOP switch.
- 2) Contact the maintenance person to determine what action to take.
- 3) Use only the fuses and other replacement parts of the specified rating.
- 4) Be extra careful when handling the following high-voltage units:

- Main Breaker
- Servo Drive unit (BL-11D)
- VAC drive unit
- Power cables

OTHER GENERAL PRECAUTIONS

- 1) Wear suitable safety clothes.
- 2) Keep work areas clean as well as the machine.
- 3) Do not touch controls with wet hands.

OSP CONTROL FUNCTIONS

Primary/Secondary Modes

Parameters - Limits

Function of all keys on Machine Operation Panel

Turning the Power ON and OFF

Turning the Power ON

< Procedure >

- (1) Turn ON the main switch at the control box.
- (2) Press the [CONTROL ON] button on the NC operation panel.
- (3) The NC control software is loaded from the data storage memory to the operation memory and the NC starts running. File names are displayed on the screen, as they are loaded to the operation memory.
- (4) SBP 6-A V*.**-*(???)
MEMORY TEST :0000
BOOTDEV FROA:
LOAD:SYS
OPERATING SYSTEM PROGRAM V*.**-*(???)
OKUMA 1997.**.**

File names of NC application software to be loaded
PBU FILE ON LOADING
FIRMWARE FILE ON LOADING
GRAPHIC DATA ON INITIAL PROCESSOR

Turning the Power OFF

< Procedure >

- (1) Make sure that all machine operating commands are completed.
- (2) If a peripheral (printer, punch, etc.) is connected to the NC, switch off the peripheral.
- (3) Press the [CONTROL OFF] button on the NC operation panel.
- (4) Turn OFF the main switch at the control box.

Notice: NC control data and mode information are backed up to the data storage memory at preset intervals. Therefore, the data may not be backed up if the power is turned OFF immediately after changing the data. Back up the data, press the function key (BACK UP) before turning the power off.

Emergency Stop

(1) Emergency Stop

Press the [EMERGENCY STOP] button to stop the machine in an emergency. The machine stops immediately if the [EMERGENCY STOP] button is pressed.

(2) Recovery from the Emergency Stop State

The [EMERGENCY STOP] button is a push-to-lock type switch and it is locked in the pressed state. To unlock the button, turn the button in the direction indicated by an arrow on the button.

After unlocking the [EMERGENCY STOP] button, press the [CONTROL ON] button on the NC operation panel, so that the NC can recover from the emergency stop state.

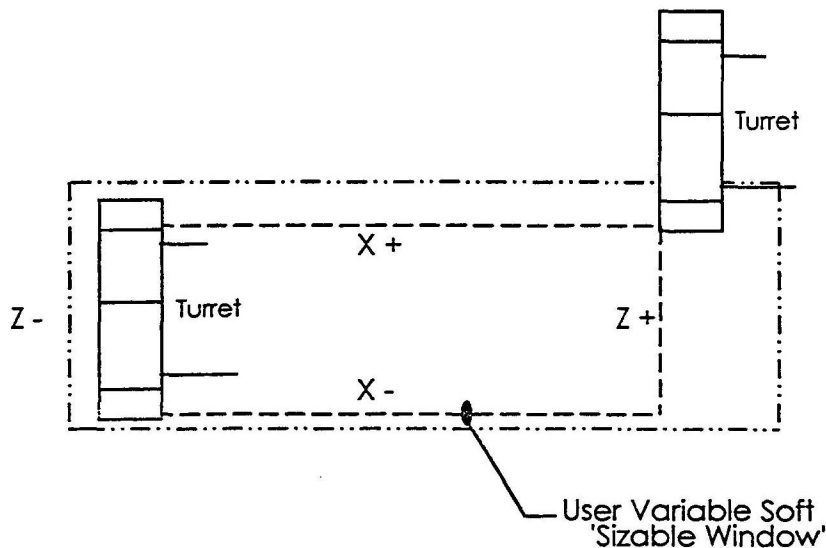
Turret Home Position - this refers to the position to which the turret must move to before the turret can index a commanded tool into the cutting position. The turret home position is also called the turret index position.

For the turret to be able to index, it must be positioned on either the positive X, or Z axis limits .

Whenever the turret slide is at a 'limit', the OSP panel light illuminates, indicating a valid position to index the turret.

For obvious reasons, the position for turret indexing should always allow for the longest tool in the turret to safely clear the chuck, etc. . Given the rapid axis speeds moving an additional 3 or 4 inches from the work piece will improve overall safety and cause little if any increase in net cycle time.

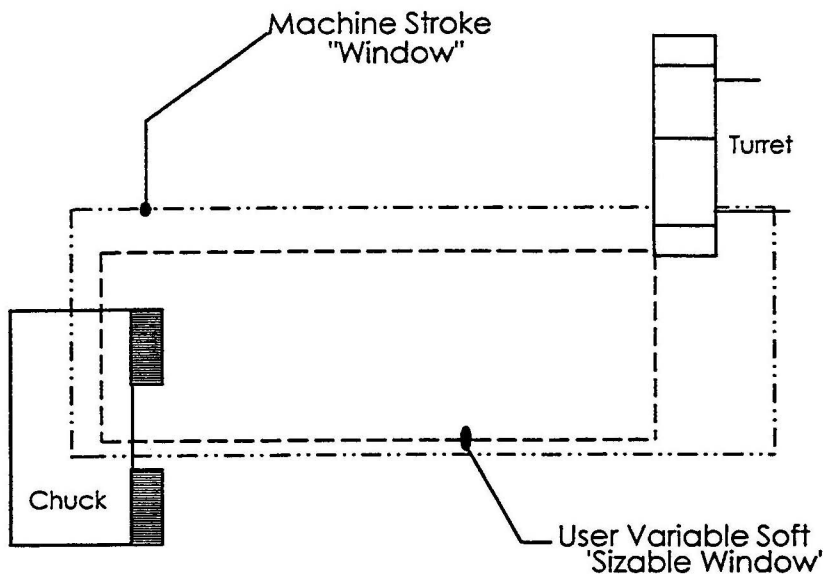
The turret indexing position can be placed anywhere within the machine working range. To index the furthest distance from the chuck, the turret should move to the positive stroke end limits - in X and Z.



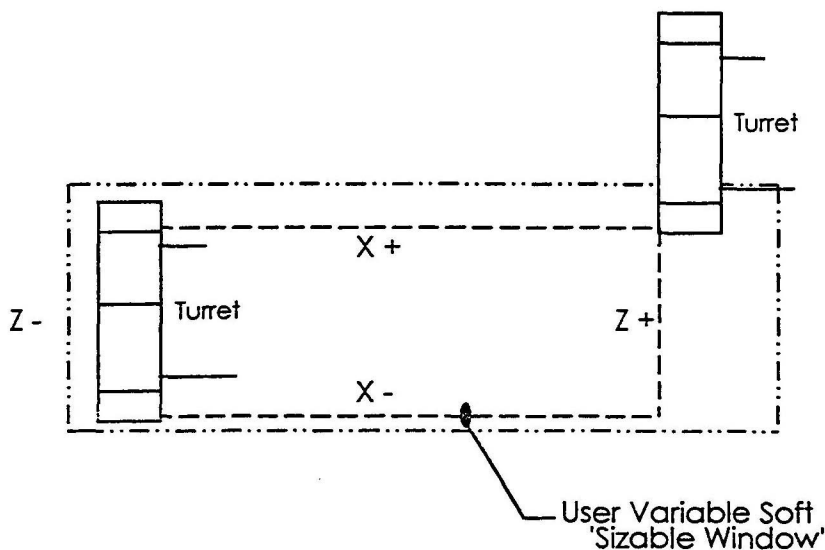
User Soft Limit / Variable Soft Limit - within the machine's stroke end limits, it is possible to establish/define a smaller 'window' used to define a smaller working range. The boundaries of this smaller window are called user soft limits.

The primary reason to establish a 'soft limit' is to reduce the distance the turret has to travel in order to 'be at a limit' and receive the confirmation signal, so that a turret index command may be executed.

The machine's turret will not index until either the 'X' positive, or 'Z' positive *limit* condition is satisfied .



The following illustration depicts the 'working range' within which the turret can travel, based on those 'soft limits' that have been defined in the OSP control.



Note: In the above illustration, the 'chuck' is not shown for reasons of clarity.

To make the turret travel to its [X+, Z+] limit, the user should use the following command:

```
G00 X50 Z50
```

This command will position the turret to the current soft limits, which may or may not be representative of the machine's maximum stroke end travel.

Notes:

1. Positioning the turret to the "limits" cannot be performed by using a G01 command.
2. The user must select a 'X' and 'Z' value that will always exceed the machine's working range. A general rule of thumb is to double the actual axis stroke amounts. Using X50, Z50 will suffice for the majority of instances, except for those machines with considerable bed length.

Problems

- 1) If your part has a program zero at the front of the part, explain why the Zs in the program have a negative sign in front of the numbers?

- 2) What is meant by soft limits?

- 3) Where is the turret home position?

- 4) How do you know if the turret slide is at a slide limit?

- 5) What is meant by the positive stroke end limit?

- 6) Also, what can you do with the turret when it reaches that limit?

- 7) What is the line code that will allow you to index the turret?

OPERATION

Basic Construction of Operation Panels

For operating the machine, the following four kinds of man-machine interfaces are provided:

(1) NC Operation Panel

It is used for file operation and data setting.

(2) Machine Operation Panel

These switches and keys used mainly for manual operation. They are located in the controls operation panel.

(3) Option Panel

An option panel is provided, which contains switches and indicating lamps required for user specified options. Arrangements of the switches and indicating lamps differ depending on the selected optional specification.

(4) Foot Pedal (Foot Switch)

A foot pedal is provided to control chuck open/close and tail stock spindle advance/retract.

Various types of operator panels are included, depending on the shape of the panel and the arrangement of the controls. External views of the operator panels are provided in the Appendix of this manual.

Outline of Controls on Operation Panel

Operation Mode Selection Keys

(1) AUTO Key

Select the automatic mode when operating the machine using a stored part program.



(2) MDI Key

Select the MDI mode for block operation, permitting input of the data necessary for operation by the keyboard in the NC operation panel.



(3) MANUAL Key

Selects the manual mode for operator by the switches on the machine operation panel.



Data Setting Mode Selection Keys

(1) EDIT AUX. Key

Selects the program operation mode for operating program files and data files. In this program operation mode, it is possible to edit input/output, display or delete a program file.



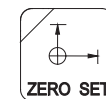
(2) PARAMETER Key

Selects the parameter mode for setting, changing or displaying parameter data necessary for NC control.



(3) ZERO SET Key

Selects the zero set mode for setting, changing, or checking the zero offset data and zero shift data which define a program coordinate system.



(4) TOOL DATA

Selects the tool data mode for setting, changing, or displaying the tool offset data, nose R compensation data, tool shape data, and load monitor data.



(5) MacMan

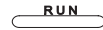
Selects the MacMan (machining management function).



NC Status Indicating lamps

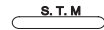
(1) RUN Lamp

The RUN lamp is on when the machine is in operation in the automatic or MDI mode.



(2) S.T.M Lamp

The S.T.M. indicating lamp is on while auxiliary function operation such as spindle gear range change, tool change, and spindle rotation, is executed.



When an axis movement command is designated with an S, T, and /or M command, the axis movement command is executed after the completion of the S, T, and /or M command designated in the same block.

If a spindle gear range command, spindle speed command, or tool number command is changed using the manual intervention function (called out by pressing the [MID AUTO MANUAL] key), the S.T.M. indicating lamp blinks.

(3) SLIDE HOLD Lamp

The SLIDE HOLD indicating lamp is on when the [SLIDE HOLD] button is pressed in the automatic or MDI mode.



For the two-saddle option, it also goes on if the operation of either of the two saddles enters the slide hold state with the other saddle having completed single block mode operation in the automatic mode.

(4) PROGRAM STOP Lamp

The PROGRAM STOP indicating lamp is on during the execution of a program stop (M00) or optional stop (M01) function in the automatic or MDI mode.



The indication lamp blinks during the execution of a dwell (G04) function.

(5) LIMIT Lamp

The LIMIT indicating lamp is on when either X- or Z - axis reaches its limit position.



For the two-saddle option, the indicating lamp is on if any of XA, ZA-, XB-, and ZB - axis reaches the limit.

The indicating lamp blinks if the actual spindle speed reaches the maximum or minimum speed of the selected gear range, or when it reaches the spindle speed specified using the maximum spindle speed specification function.

(6) ALARM Lamp

The ALARM indicating lamp is on when the machine malfunctions or an incorrect program is input. It is also on if the computer fails to function correctly.

ALARM

Status Indicating Lamp	Condition for ON	Condition for Blinking
RUN	- The machine is normally running in the AUTO or MDI mode (except for during the SLIDE HOLD and PROGRAM STOP mode).	-The program selection command is a schedule program is being executed.
S.T.M	<p>-The machine is waiting for the operation completion of an M code command. (Spindle rotation command, gear command, etc.).</p> <p>- A spindle speed command is given (S command).</p> <p>- A turret rotation command is given (T command).</p> <p>Only for the multi-machinig options:</p> <p>-The machine is waiting for the operation completion of a multi-machining M code command. (C-axis joint command , M-tool spindle rotation command, etc.).</p> <p>-An M-tool spindle speed command is given (SB command).</p> <p>-The C-axis brake pressure is switched between high and low.</p> <p>Only for the ATC specification</p> <p>- The machine is waiting for the operation completion of the MG, MT or TN command.</p>	<p>-The following items have been changed during manual intervention.</p> <p>Tool number Spindle rotation/gear command</p> <p>Only for the multi-machining specification</p> <p>C- axis joint state C-axis clamp M-tool spindle rotation/gear command</p>
SLIDE HOLD	<p>-The SLIDE HOLD button has been pressed in the AUTO or MDI mode.</p> <p>-A block of program commands has been executed on one saddle while the other saddle is placed in the slide hold mode with the single block function activated in the AUTO mode.</p>	Does not blink.
PROGRAM STOP	-M00 OR M01 is designated in the AUTO or MDI mode.	-The dwell command (G04) is executed.

Status Indicating Lamp	Condition for ON	Condition for Blinking
LIMIT	-Either X- or Z -axis has reached the variable soft-limit position.	<p>-The spindle speed has reached the limit in the selected gear range.</p> <p>-The spindle speed has reached the limit specified by the maximum spindle speed designation function.</p> <p>-The spindle position is controlled after the completion of spindle orientation (DIFF control).</p> <p>-The M-tool spindle position is controlled after the completion of spindle orientation.</p>
ALARM	-Any erroneous operation is attempted (An alarm of level A, B, C, or D).	Does not blink.

Other Controls on NC Operation Panel

(1) Function Keys: F1 to F8

There are eight function keys on the NC operation panel.

When an operator selects a desired operation mode, the screen displays the T operation functions at the bottom line. Each function corresponds to a function key (F1 through F8). Select the function to execute and press the corresponding function key.

If all functions called out by the selection of a mode cannot all be displayed simultaneously, the ([EXTEND]) message is displayed for function key [F8]. In this case, press [F8] to display the rest of functions.

[F1] [F2] [F3] [F4] [F5] [F6] [F7] [F8]

(2) ? (Help) Key

This key is used to display the description of the alarm which occurred during machine operation and also the alarm history.

It shows actual position data, part program data, block data, zero offset values, tool offset values, parameter data, alarm description, etc.

The basic format of display on the screen is shown below.

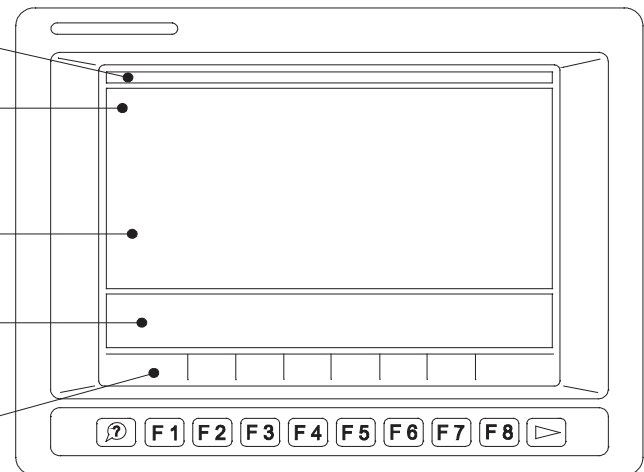
1st line: Mode currently selected
(guide display, title display)

2nd line: Alarm contents

3rd line to 18th line: Data

19th line to 22nd line: Console lines
(display of input data)

23rd line to 24th line: Effective function of
respective function keys



(3) WRITE Key

The [WRITE] key to select an operation and also confirms input data.



(4) BS (Backspace) Key

The [BS] key is used when erroneous data has been input. Each time this key is pressed, the character input last is erased.



For the display of file index and list, this key is used to display the next page.

(5) CAN Key

The [CAN] key is used when erroneous data has been input. Each time this key is pressed, one line of the data is erased.



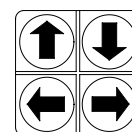
(6) Turret Selection (AB) Keys

These keys are used to select the appropriate turret or saddle (A saddle side, B saddle side) for NC control of the two-saddle or two-turret models.



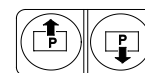
(7) Cursor Keys

Four cursor keys are used to move the cursor displayed on the screen.



(8) Page keys

If the information called out is displayed in more than one page, the page keys are used to change the display page.



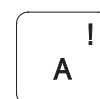
(9) Operator Keys

These keys are used when an operator edits a program or enters data.



(10) Character Keys

Character keys are used for inputting a character for data input, program operation, and file edit operation.



(a) To input a character shown at the upper right corner of a key top, use the [UPPER CASE] key.

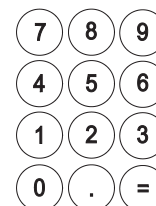


(b) When the [CAPS LOCK] key is pressed (indicating lamp at the upper left corner lit), upper case alphabetic letters A to Z are input. When the [CAPS LOCK] key is not pressed, lower case alphabetic letters a to z are input.



(11) Numerical Key Pad.

These keys are used for inputting numbers for data input, program operation, and file edit operation.



(12) Contrast Adjusting Keys

(only for Operation Panel with Monochrome STN Screen)

These keys are used to adjust the contrast for the display.



Controls on Machine Operation Panel

Flat keys used on the machine operation panel have features as indicated below depending on whether or not they have an indicator lamp.

< Flat keys with an indicating lamp >

The indicating lamp in a key indicates if the function of the key is valid or not.

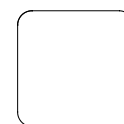
- Indicating lamp lit Key function is valid.

- Indicating lamp unlit Key function is invalid.



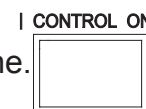
< Flat keys without an indicating lamp >

The function of the key is valid only while the key is held down. If the key is not pressed, the function is invalid.



(1) CONTROL ON Switch

The [CONTROL ON] switch is used to turn on the control power of the NC unit after turning on the main switch of the machine.

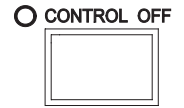


The pilot lamp in this switch lights when the control power is turned on.

If the [EMERGENCY STOP] button is pressed, the pilot lamp in this switch goes off. To leave the emergency stop state, press the [CONTROL ON] switch.

(2) CONTROL OFF Switch

The [CONTROL OFF] switch is used to turn off the control power of the NC unit.



When shutting off the power, turn off the control power first by pressing the [CONTROL OFF] switch before turning off the main switch of the machine.

(3) RESET Key

The NC unit is reset when the [RESET] key is pressed. The key is used to reset an alarm.



(4) NC PANEL Switch

(a) UNLOCK position

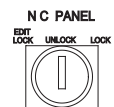
Enables all controls on the NC and machine operation panels.

(b) EDIT LOCK position

Operations in the program operation (EDIT AUX) mode and parameter setting mode are displayed.

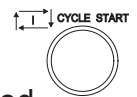
(c) LOCK position

All controls on the NC operation panel are disabled.



(5) CYCLE START Button

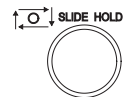
The [CYCLE START] button is used to start the execution of the program.



The CYCLE START signal is output when the button is pressed and released.

(6) SLIDE HOLD Button

Axis movements of X-, Z-, and C-axis stop immediately when the [SLIDE HOLD] button is pressed. To resume axis movements, press the [CYCLE START] button .



If this button is pressed while an axis is not moving, the slide hold becomes valid after the completion of the sequence having been executed at the time the [SLIDE HOLD] button was pressed or when the next axis movement is executed.

(7) EMERGENCY STOP Button

Press the [EMERGENCY STOP] button when an emergency occurs.

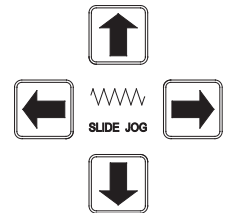


The power supply to the NC is shut off when the [EMERGENCY STOP] button is pressed.

To release the emergency stop state, unlock the [EMERGENCY STOP] button and press the [CONTROL ON] button.

(8) SLIDE JOG Buttons

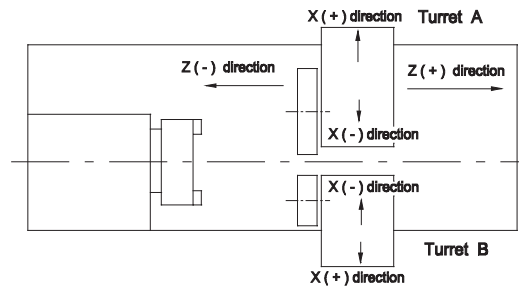
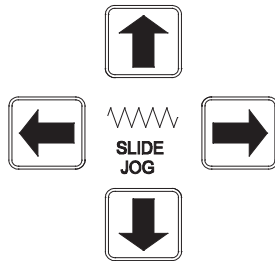
These are used to jog an axis in the manual mode. The button is valid only while it is pressed and the setting of the FEED RATE override dial is appropriate for jog feed operation.



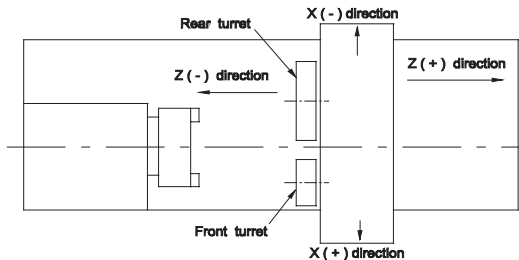
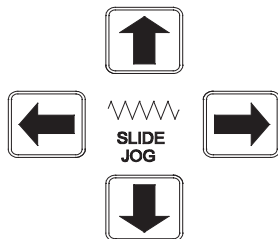
For the two-saddle option, the saddle selected by the turret selection keys moves.

Notice Feed rates differ depending on the machine models and options. With some special order machines, the feed patterns will differ from those illustrated below.

(a) Other than flat-bed machines

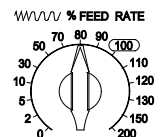


(b) Flat-bed machines



(9) FEED RATE

The FEED RATE override dial overrides the specified feed rate in the range 0 to 200% in 15 steps.

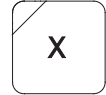


Notice In single block OFF operation in automatic mode, override is not valid for a rapid feed command (G00).

Override is not valid for thread cutting operation.

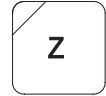
(10) PULSE HANDLE X Key

Select this key to operate the X-axis using the pulse handle.



(11) PULSE HANDLE Z Key

Select this key to operate the Z-axis using the pulse handle.



(12) PULSE HANDLE 1/1 Key

Select this key to move an axis using the pulse handle at the rate of unit distance per pulse.



(13) PULSE HANDLE 10/1 Key

Select this key to extend axis movement per pulse to 10 times the unit distance.



(14) PULSE HANDLE 50/1 Key

Select this key to extend axis movement per pulse to 50 times the unit distance



(15) Pulse Handle

The pulse handle is used to move a selected axis by pulse signals which are generated as the pulse handle is turned.



(16) TOOL INDEX Button

The [TOOL INDEX] button is used to rotate the turret manually. The turret will rotate continuously if the button is held pressed.



(17) SPINDLE STOP Button

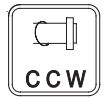
Use the [SPINDLE STOP] button to stop the spindle manually.



For the multiple-machining models, the button is also used to start the M-tool spindle.

(18) SPINDLE CW Button

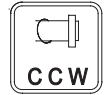
Used to start the spindle in the forward (CW) direction.



For multiple-machining models, the button is also used to start the M-tool spindle in the forward (CW) direction.

(19) SPINDLE CCW Button

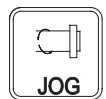
Used to start the spindle in the reverse (CCW) direction.



For multiple-machining models, the button is also used to start the M-tool spindle in the reverse (CCW) direction.

(20) SPINDLE JOG Button

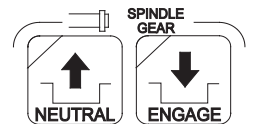
The [SPINDLE JOG] button is used to jog the spindle at the speed set for a parameter.



For multiple-machining models, the button is also used to jog the M-tool spindle.

(21) SPINDLE GEAR ENGAGE Key

For machines with a transmission, press this key to engage the spindle drive gears.



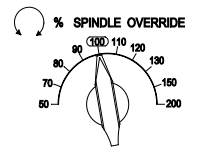
The indicator lamp at the upper left corner of the key lights when the output of the gear range and the input pattern of the gear engage confirmation limit switches agree with each other. It blinks if they do not agree.

(22) SPINDLE GEAR NEUTRAL Key

For equipped with a transmission, press this key to put the spindle drive gears in neutral.

(23) SPINDLE OVERRIDE Dial

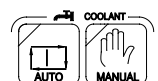
Override the specified spindle speed in 10 steps from 50 to 200% .



If the spindle speed reaches the maximum allowable speed or the spindle speed limit set by G50 when it is overridden, the actual spindle speed is limited to the maximum allowable speed or the spindle speed limit.

(24) COOLANT-MANUAL Key

When the [COOLANT - MANUAL] key is pressed (indicating lamp at the upper left corner lit), coolant can be supplied independent. of the operation mode.



This key cannot be turned ON if the [COOLANT -AUTO] key is ON.

(25) COOLANT-AUTO Key

When the [COOLANT-AUTO] key is pressed (indicating lamp at the upper left corner lit), coolant is supplied according to the coolant command given in the automatic or MDI mode.

This key cannot be turned ON if the [COOLANT -MANUAL] key is ON.

(26) LIGHTING Key

The [LIGHTING] key is used to turn on and off the machine's work light of the machine.



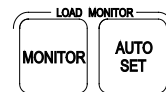
(27) DISPLAY OFF Key

Turn off the display on the screen, make this key valid (indicating lamp at the upper left corner lit) .



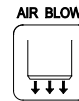
(28) LOAD MONITOR -MONITOR/AUTO SET Keys

Use these keys for machines equipped with load option.



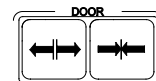
(29) AIR BLOW Key

Air blow is supplied while the key is held pressed for machines equipped with the air blow option.



(30) DOOR -OPEN/CLOSE Keys

Use these keys for machines equipped with the automatic door open/close option.



(31) DOOR INTERLOCK - ON/OFF Switch

The switch is used to select whether or not the door interlock function is operating.



For details of the door interlock function, refer to the Door Interlock Function Manual.

Mode Selection Keys

To operate the machine using a program, a variety of operation modes are provided.

(1) SINGLE BLOCK Key



(a) When the [SINGLE BLOCK] key is on (indicating

lamp at the upper left corner lit), a program is executed in units of blocks. To execute each block, press the [CYCLE START] button.

(b) When the [SINGLE BLOCK] key off (indicating lamp at the upper left corner unlit), program blocks are executed continuously.

(2) BLOCK DELETE Key

(a) When the [BLOCK DELETE] key is on (indicating lamp at the upper left corner lit), commands between a slash (/) code and "ER" code are ignored.



Notice A slash code (/) must be placed at the start of a program block or immediately after the sequence number (or sequence name) of a block.

(b) When the [BLOCK DELETE] key is off (indicating lamp at the upper left corner unlit), commands entered following a slash code (/) are executed.

(3) OPTIONAL Stop Key



(a) When the [OPTIONAL STOP] key is on valid (indicating lamp at the upper left corner lit), operation stops after the execution of an M01 block in a program. Spindle rotation and coolant supply also stop.

When the [CYCLE START] button is pressed, the previous state is recovered and the program is continuously executed .

(b) When the [OPTIONAL STOP] key is off (indicating lamp at the upper left corner unlit), program is continuously executed even after the execution of an M01 block.

(4) DRY RUN Key

(a) When the [DRY RUN] key is on (indicating lamp at the upper left corner lit), axis feed is executed at the feed unit amount set by parameter with exceptions of G00 feed and manual axis feed .



(b) When the [DRY RUN] key is off (indicating lamp at the upper left corner unlit), axis feed is executed at the specified feed rate .

Note: To change the dry run mode on/off state, it is necessary to press the [DRY RUN] key while holding down the [INTERLOCK] key

(5) MACHINE LOCK Key

- (a) When the [MACHINE LOCK] key is on (indicating lamp at the upper left corner lit), all commands in a program are executed without actual machine operation.



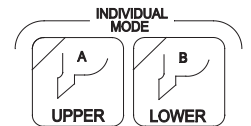
However, the actual position data and block data display are updated as the program is executed. The data display returns to the previous state when the NC is reset.

- (b) When the [MACHINE LOCK] key is off (indicating lamp at the upper left corner unlit), all commands in a part program are executed normally.

Note: To change the machine lock mode on/off state, it is necessary to press the [MACHINE LOCK] key while holding down the [INTERLOCK] key.

(6) INDIVIDUAL MODE -TURRET A/TURRET B Keys

The individual turret operation function is valid only for the two-saddle option.



The two-saddle option allows cutting to be performed by controlling the four axes simultaneously. By using these keys, the following operation modes can be selected.

(a) Independent turret A operation mode

In the automatic or MDI mode of operation, only turret A side operates.

[UPPER A] key: On (indicating lamp at the upper left corner lit)

[LOWER B] key: Off (indicating lamp at the upper left corner unlit)

(b) Independent turret B operation mode

In the automatic or MDI mode of operation, only turret B side operates.

[UPPER A] key: Off (indicating lamp at the upper left corner unlit)

[LOWER B] key: On (indicating lamp at the upper left corner lit)

(c) Independent turret A/turret B operation mode

In the automatic or MDI mode of operation, turret A and turret B operate alternately according to the synchronization codes entered in a part program. **37**

[UPPER A] key: On (indicating lamp at the upper left corner lit)

[LOWER B] key: On (indicating lamp at the upper left corner lit)

(d) Normal operation mode

Simultaneous 4-axis operation is executed according to a part program.

[UPPER A] key: Off (indicating lamp at the upper left corner unlit)

[LOWER B] key: Off (indicating lamp at the upper left corner unlit)

Note: To change the independent turret A/turret B operation mode on/off state, it is necessary to press the [UPPER A] or [LOWER B] key while holding down the [INTERLOCK] key.

(7) SEQUENCE RESTART Key

The [SEQUENCE RESTART] key is used to restart a part program from a specific block.



(8) MID AUTO MANUAL Key

To perform manual operation during automatic or MDI mode operation, press the [MID AUTO MANUAL] key.



(9) INTERLOCK Key

To change the on/off state of the dry run mode, machine lock mode machine lock mode and independent turret A/turret B operation mode, it is necessary to press the [INTERLOCK] key at the same time the corresponding mode key is pressed.



Controls on Additional Panel (C-axis Panel)

An additional panel is provided only for machines equipped with the multiple machining function .

(1) MILLING SPINDLE Key

When the [MILLING SPINDLE] key is turned on (indicating lamp at the upper left corner lit), the [SPINDLE - START], [SPINDLE - CCW], [SPINDLE - CW], and [SPINDLE - JOG] buttons and the [SPINDLE OVERRIDE] dial become valid for the M-tool spindle.



(2) C-AXIS Key

Turn this key on to operate the C-axis manually using the pulse handle. The key functions only in the C-axis control mode.



(3) C-AXIS CLAMP Key

The [C-AXIS CLAMP] key is used to clamp the C-axis manually. The key functions only in the C-axis control mode.



The indicating lamp at the upper left corner is turned on when the C-axis is in the clamped state.

(4) C-AXIS UNCLAMP Key

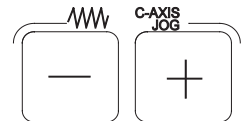
The [C-AXIS UNCLAMP] key is used to unclamp the C-axis manually. The key functions only in the C-axis control mode.



The indicating lamp at the upper left corner is turned on when the C-axis is in the unclamped state.

(5) C-AXIS JOG - +/- Keys

The C-axis jog feed keys used in manual operation. Jog feed occurs when a key is held down. The setting of the FEED RATE override dial on the machine operation panel is valid for jog feed of C-axis. These keys function only in the C-axis control mode.



8. Foot Pedal

In addition to the operation panels, foot pedals are used to operate the machine.

(1) Check Open/Close Foot Pedal

There are two types of foot pedals used for opening and closing a chuck. One type is the standard pedal in which the same pedal is used for opening and closing a chuck, and the other type is the dual-pedal type foot pedal in which the independent pedal is used for opening and closing a chuck.

(2) Tail stock Spindle Advance/Retract Foot Pedal

The foot pedal used for tail stock operation can advance, retract, and inch the tail stock spindle.

MANUAL MACHINE CONTROL

Door Interlock Restriction

Spindle Related Operation

Preparing for Spindle Rotation

- (1) Setting the allowable Chuck Speed

Set the allowable speed of the chuck which is mounted to the machine with a parameter. (machine parameter)

- (2) Inputting the Maximum Spindle Speed

Input the maximum spindle in the MDI operation mode.

```
=IN G50 S_[WRITE] [CYCLE START]
```

- (3) Inputting the Spindle Speed

In the MDI operation mode, input the spindle speed at which the spindle should be operated.

```
=IN S_[WRITE] [CYCLE START]
```

- (4) Inputting the Gear Range

For the machine equipped with a transmission, input the gear range that meets the input spindle speed. Select the MDI mode also for this operation.

```
=IN M_[WRITE] [CYCLE START]
```

Starting the Spindle CW/CCW

<Procedure>

- (1) Select the manual operation mode by pressing the [MANUAL] key on the NC operation panel.
- (2) Press the [SPINDLE - CW] or [SPINDLE - CCW] button on the machine operation panel.

- [Supplement]
- | | |
|---|---|
| A | The setting of the SPINDLE OVERRIDE dial is valid for the specified spindle speed. |
| B | The [SPINDLE - CW] OR [SPINDLE - CCW] button may be released after pressing it once. The spindle keeps rotating even if the button is released. |
| C | To start the spindle, the chuck must be in the closed state. |

- D An alarm occurs if the actual spindle speed exceeds 120% of the allowable speed of the chuck.
- E For center-work operation, the tailstock spindle must be set in the correct position.
- F The door must be closed.

Stopping the Spindle

<Procedure>

1. Select the manual operation mode by pressing the [MANUAL] key on the NC operation panel.
2. Press the [SPINDLE - STOP] button on the machine operation panel.

Jogging the Spindle

<Procedure>

1. Press the [SPINDLE - JOG] button on the machine operation panel.

[Supplement]

- A Spindle jog speed is set with a parameter (machine parameter).
- B The spindle jogs only while the [SPINDLE - JOG] button is being pressed.
- C The setting of the SPINDLE OVERRIDE dial is not valid for spindle jog operation.

Axis Feed Operaiton (X-, Z-, and C-axis)

An axis (X-, Z-, or C-axis) can be moved in either of the following methods.

- (1) Using SLIDE JOG buttons.
- (2) Using the pulse handle.

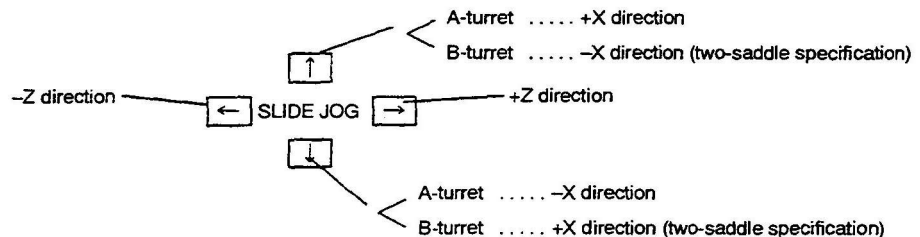
SLIDE JOG Buttons

<Procedure>

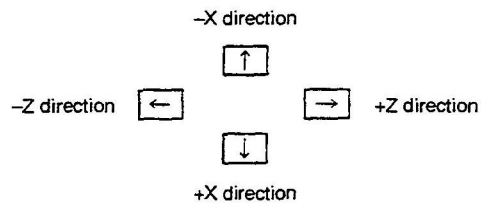
- (1) Select the manual operation mode by pressing the [MANUAL] key on the NC operation panel.
- (2) For the two-saddle specification, select the saddle to be operated by the turret selection keys.
- (3) Press the [SLIDE JOG] button corresponding to the required axis feed direction.

Note that the arrow symbol in a button indicates the direction of saddle movements and it is not related to the positive and negative directions in the program coordinate system. The [C-AXIS JOG] keys do not have the arrow symbol in them, but they have the “+” and “-” signs which indicate the positive and negative directions in the program coordinate system.

- [Supplement]
- A An axis moves only while a [SLIDE JOG] button is held down.
 - B The saddle can move within the range defined by the variable limits which are set with parameters.
 - C Jog feedrate can be changed by the setting of the FEED RATE override dial on the machine operation panel.
 - D The illustration below shows the saddle movement directions.



-Note that the direction of axis movement differs for the following models:
LH35, LH55, LS30



Pulse Handle

<Procedure>

- (1) Select the manual operation mode by pressing the [MANUAL] key on the NC operation panel.
- (2) For the two-saddle specification, select the saddle to be operated by the turret selection keys.
- (3) Select the axis to move by pressing the [PULSE - X] or [PULSE - Z] key on the machine operation panel.
- (4) Select the distance to move an axis per pulse by selecting the [PULSE - 1/1], [PULSE - 10/1], or [PULSE - 50/1] key on the machine operation panel.

	Linear axis		Rotary axis
	mm Unit System	inch Unit System	degree Unit System
1/1	.001 mm/pulse	0.0001 inch/pulse	0.001 deg/pulse
10/1	.010 mm/pulse	0.0010 inch/pulse	0.010 deg/pulse
50/1	.050 mm/pulse	0.0050 inch/pulse	0.050 deg/pulse

- (5) Turn the pulse handle.

The axis moves in the positive direction in the program coordinate system if the pulse handle is turned clockwise. It moves in the negative direction if the pulse handle is turned counterclockwise.

- [Supplement]
- A When operating an axis by the pulse handle, it is not possible to select more than one axis at a time.
 - B To operate the C-axis by the pulse handle, set the C-axis in the engaged state beforehand in the MDI operation mode.
 - C =N M110 [WRITE] [CYCLE START]

Turret Rotation

<Procedure>

- (1) Move the saddle to the position where turret rotation is possible. The turret rotation enabled position differs depending on the machine model.
- (2) For the two-saddle specification and the two-turret specification, select the turret to be operated by the turret selection keys.
- (3) Press the [TURRET INDEX] button on the machine operation panel.

- [Supplement]
- A The turret keeps rotating if the [TURRET INDEX] button is held down.
- B The turret rotation direction should be set in the MDI operation mode before indexing the turret.

If the turret is rotated without setting the rotation detection, the turret rotates in the direction the station numbers increase. For the flat-bed construction machines (LH series), the turret can be rotated only in the direction the station numbers increase.

- C To rotate the turret, the following conditions must be satisfied.

LBII series	LT15/25	Either the X- or Z-axis is positioned at the variable limit in the positive direction. (The [LIMIT] status indicating lamp must be lit.)
LU series	LC series	
LCC series	LCS series	
V-turret specification of LH series/LS30N		
H4 specification of LH series/LS30N LT10		The X-axis is positioned at the variable limit in the positive direction.
H6 specification of LH series/LS30N		The Z-axis is positioned at the variable limit in the positive direction.
H8 specification of LH series/LS30N		
Two-turret specification of LH series/LS30N		The saddle may be at any position.

Chuck Open/Close Operation

A foot pedal is provided to open and close the chuck.

- (1) Procedure (Standard Foot Pedal)
 - (1) Step on the foot pedel to open or close the chuck. Each time the foot pedal is stepped on, the chuck opens and closes alternately.
- (2) Procedure (Dual-Pedal Foot Pedal)
 - (1) To close the chuck, step on the chuck close foot pedal.
 - (2) To open the chuck, step on the chuck open foot pedal.

Tailstock Body Positioning

With the simplified tow-along tailstock specifcaiton, it is possible to move the tailstock body using the saddle movement in the Z-axis direction after joining the tailstock body to the saddle (A saddle for the two-saddle specification). When the tailstock body is joined to the saddle by the joint pin, the tailstock body is unclamped from the rail. When the saddle in not joined to the tailstock, the tailstock body is clamped against the rail.

<Procedure>

- (1) Select the manual operation mode by pressing the [MANUAL] key on the NC operation panel.
- (2) Move the saddle in the Z-axis direction to the position where the match mark (↑) (↓) in the saddle is aligned with the match mark in the tailstock body.
- (3) Manually push the joint pin into the saddle and tailstock body joint holes to join the saddle and the tailstock body.
- (4) Move the saddle in the Z-axis direction using the Z-axis jog button or the pulse handle to tow the tailstock body.

[Supplement] A Tailstock body movable range

The tailstock body is repositioned by moving the saddle in the Z-axis direction after joining it to the saddle. When tailstock body is joined to the saddle, the saddle has a Z-axis movement range that is different from the saddle Z-axis movement range when disconnected from the tail stock body. Therefore, the movable range of the saddle in the Z-axis direction with the tailstock body joined to the saddle is set for W axis stroke end limit of the system

parameter to restrict the saddle movable range when the tailstock body is joined to the saddle.

If an axis movement command is specified that causes the Z-axis to move beyond the set range, the command is disregarded.

B Feedrate

Feedrate of the Z-axis operated by pressing a SLIDE JOG button is determined by the setting for system parameters W axis JOG FEED and W axis ACCEL JOG FEED.

C Torque monitor

When moving the Z-axis in the state the tailstock body is joined to the saddle, torque monitor is executed based on the value set for the system parameter W axis TORQUE VALUE

If the torque exceeds the value determined by the setting and the torque rating, an alarm occurs.

Tailstock Spindle Advance/Retract Operation

A foot pedal is provided to operate the tailstock spindle (advance (inching)/advance/retract).

The foot pedal used for tailstock spindle operation is of a dual-pedal type and one pedal is used for the tailstock spindle retract operation. The other pedal has the two-position switch and it is used for the tailstock spindle advance (inching)/advance operation.

(1) Procedure (Tailstock Spindle Advance (Inching))

- (1) Step on the tailstock spindle advance side foot pedal to the first-step switch position

The tailstock spindle advances only while the first-step switch is kept pressed.

- (2) Release the foot pedal and the tailstock spindle stops advancing.

(2) Procedure (Tailstock Spindle Advance)

- (1) Step on the tailstock spindle advance side foot pedal until the second-step switch is actuated. Once the second-step switch is actuated, the tailstock spindle keeps advancing even if the foot pedal is released.

(3) Procedure (Tailstock Spindle Retract)

- (1) Step on the tailstock spindle retract side foot pedal. Once the switch in the pedal is actuated, the tailstock spindle keeps retracting even if the foot pedal is released.

Coolant

<Procedure>

- (1) Press the [COOLANT - MANUAL] key on the machine operation panel to turn on the indicating lamp in it.
- (2) Coolant stops when the [COOLANT - MANUAL] key is pressed again to turn off the indicating lamp on it.

[Supplement]

It is not possible to select the [COOLANT - MANUAL] key while the [COOLANT - AUTO] key is selected (indicating lamp at the upper left corner lit). If the manual coolant operation is selected, automatic coolant operation is made invalid. Similarly, if the automatic coolant operation is selected while the manual coolant operation is valid, the manual coolant operation is made invalid.

MDI Operation

1. Press MDI key in Mode Select Area.
2. Press function key (F3) (Program)
3. *Select "A" or "B" turret.
4. Press Page until MDI display appears with current/buffer.
5. *The Individual/Simultaneous switches should be in individual or simultaneous settings
6. Press function key (F1) (Data Input)
7. Enter the MDI data (G00X20,Z20,S800T101010)
8. Press Write Key - at this point the MDI data is transferred to the buffer side of the MDI screen.
9. Before pressing cycle start, decrease the feed rate switch to minimize RAPID movements.
10. Press Cycle Start - at this point MDI data is transferred from "Buffer" to "Current" side of MDI screen and the machine executes your command.

Starting Spindle Rotation

1. Make sure Chuck is clamped.
2. Press MDI key in Mode Select Area.
3. Press function key (F3) (Program)
4. Page until you find MDI/Current-Buffer
5. *Select "A" or "B" turret.
6. *The Individual/Simultaneous switches should be in either individual or simultaneous setting.
7. Press function key (F1) (Input)
8. Key in M 41 S 500 M03
9. Press Write Key, at this point the MDI data is transferred to the Buffer side of the screen.
10. Press Cycle Start button

11. The machine will now execute your command.

*For 2 turret model machine only.

Setting Z Zero Offsets

There are two methods of offsetting on the Okuma control. This first method is used for Z-axis.

1. *Select "A" turret.
2. Select a tool for the Zero Set Tool. (Only one tool per turret will be used for Zero setting.)
3. In manual, face the part, then withdraw the tool in the X-axis.
4. Stop the spindle and measure overall length. Write this number down as it will be used later.
5. Select Zero Set in the Mode Select Area.
6. With the cursor shift keys, locate the cursor to the Z-axis position.
7. Press function key (F3) (CAL), then key in dimension value.
8. Press Write Key.
9. The CRT displays the machine coordinate position.
10. Check the actual position display by pressing manual mode, then function key (F2) (ACTUAL POSITION).
11. *Select "B" turret.
12. Select a tool for the Zero set tool.
13. *Move Zero Set Tool to touch the face of the part.
14. *Repeat steps 5 thru 8.

Setting X Zero Offsets

This method is used for zero setting for a drill from the management data card (X-axis only).

1. Select Zero Set Mode
2. *Select "A" turret
3. With the cursor shift keys, locate the cursor to the X-axis zero offset.
4. Press function key (F1) (SET), then key in the value from management data card. (89.1256) (EXAMPLE)
5. Press Write Key.
6. The CRT will display this zero offset value.
7. *Select "B" turret.
8. Repeat steps 3 thru 5.

Adjusting Zero Offsets

On the control, there is also a method to adjust the zero set value. After properly zero setting and tool offsetting, this method allows us to shift the zero set value further away from the part, thus allowing us to run through a machine cycle without cutting any material.

1. Select Zero Set Mode.
2. Select "A" turret.
3. With the cursor shift keys, locate the cursor to the Z zero offset position.
4. Press function key (F2) (ADD) then key in amount of adjustment.
5. Press the Write Key.
6. The CRT will display the modified zero set value.
7. *Select "B" turret.
8. *Repeat steps 3 thru 5.

Setting Tool Offsets

There are two methods of tool offsetting. The first is used when the tool offsets are unknown. This will be the most commonly used method.

1. Select Manual Mode.
2. Select Tool Data Mode.
3. *Select "A" turret.
4. Select a tool for offsetting.
5. Manually touch off a faced part in Z axis.
6. With the cursor shift key, locate the cursor to this tool's offset number then select Z axis.
7. Press function key (F3) (CAL) Key in Dimension Value.
8. Press Write Key.
9. The CRT will display the offset.
10. Next, touch off a turned diameter for X-axis offset. If you do not have a diameter to work from manually turn a diameter and measure this diameter. Write this number down.
11. With the cursor shift key select X-axis.
12. Press function key (F3) (CAL) then key in the dimension value.
13. Press Write Key.
14. The CRT will display the new offset value.
15. Repeat steps 4 thru 13 until all "A" turret tools are offset.
16. *Select "B" turret.
17. Repeat steps 4 thru 13 until all "B" turrets tools are offset.

Setting Tool Offsets

This method of setting tool offsets will be used when preset tools are placed in the turret station, that is, the tooling has been premeasured for tool length and tool width prior to being placed in the turret stations. This method can also be used to erase offsets.

EXAMPLE: Drill X offset = 0.000

1. Press the Tool Data key in this Mode Select area.
2. *Select "A" turret.
3. With the cursor shift key locate the cursor to the tool offset number then select X or Z axis.
4. Press function key (F8) (EXTEND)
5. Press function key (F1) (SET) key in premeasured value.
6. Press Write key.
7. Repeat steps 3 thru 6 as needed for "A" turret tools.
8. *Select "B" turret.
9. Repeat steps 3 thru 6 as needed for "B" turret tools

Adjusting Tool Offsets

This method is used to compensate for insert wear by incrementally adjusting the offset value for that particular tool. The rule of thumb for adjusting the tool offset is: the part dimension is oversized you must subtract (-) the oversize amount from the original offset value. If the part dimension is undersized you must add (+) the undersized amount to the original offset value.

1. Press the Tool Data Key.
2. *Select the proper turret "A" or "B".
3. With the cursor shift keys locate the cursor to the proper tool offset position.
4. Press function key (F2) (ADD) and key in the adjustment value (oversize = -value, undersize = +value).
5. Press Write Key.
6. The CRT will display the new offset calculation.

7. Repeat steps 3 thru 5 as needed for additional tool offset adjustment.

Notes:

1. A constant for adjustment can be set in Optional Parameter Long Word 32.
2. To prevent accidental over adjustment of an offset, a limit can be set in Optional Parameter Long Word 33.
3. A limit can be set in Optional Parameter Long Word 33 to prevent accidental adjustment of offsets.

Setting Soft Stroke End Limits

Method 1

Soft wired stroke end limit positions are used as a home position for indexing the turret. There are two methods of setting these limits. The first method is called the PLAYBACK METHOD. This method allows us to physically move the turret to a safe index position and set the soft limits.

1. Manually move "A" turret to the desired "SAFE" position in X and Z axis.
2. Press the Parameter key in the Mode Select area.
3. Press function key (F6) or (F7) until the "User Parameter" display appears.
4. *Select "A" turret.
5. *With cursor shift keys locate the cursor to the X-axis data position.
6. *Press function key (F3) (CAL) (Value is not necessary).
7. *Press Write Key.

The CRT will display the new soft limits (manually check these limits).

8. Locate the cursor to the Z axis data position.
9. Repeat steps 6 and 7.
10. *Select "B" turret.
11. Manually move the "B" turret to the desired "SAFE" position in X and Z axis.
12. *Repeat steps 5 thru 9.

Method 2

This method for setting the stroke end limits is used when it is necessary to calculate the stroke end limits from the program zero point without having to move the turret(s) to that position.

1. Press the Parameter Key in the Mode Select area.
2. Press function key (F6) or (F7) until the "User Parameter" display appears.
3. *Select "A" turret.
4. With cursor shift keys located the cursor to the X-axis data position.
5. Press function key (F1) (SET) and then key in soft limit value.
6. Press Write Key.
7. With cursor shift keys locate the cursor to the Z-axis data position.
8. Repeat steps 5 and 6.
9. *Select "B" turret.
10. *Repeat steps 4 thru 8.
11. "The CRT will display the new soft limit positions.

CAUTION - The amount of adjustment cannot exceed the maximum stroke end limits.

Adjusting the Soft Stroke End Limits

This method is used to incrementally adjust the stroke end limits.

1. Press the Parameter Key in the Mode Select area.
2. Press function key (F6) or (F7) until the "User Parameter" display appears.
3. *Select "A" turret.
4. With the cursor shift keys locate the cursor to the Z-axis data position.
5. Press function key (F2) (ADD) and key in the amount of adjust (+ or -1).
6. Press Write Key.
7. *Select "B" turret.
8. *Repeat steps 4 thru 8.

9. The CRT will display the new soft limit position.

CAUTION - The amount of adjustment cannot exceed the maximum stroke end limits.

Procedure to Store Programs into Memory from Tape

1. Set the N.C. part program tape in the tape reader.
2. Press the Edit Aux. key in the Mode Select area.
3. Press function key (F3) (PIP).
4. Press function key (F1) (READ).
5. Press Write Key.
6. Press function (F7) (QUIT) when tape reader is finished reading the N.C. tape.

Program Selection

Method 1

1. Press Auto in the Mode Select area.
2. Press function key (F1) (Program Select) then the * on the key pad.
3. Press Write Key.
4. Move cursor to the desired program.
5. Press Write Key.

Method 2

1. Press Auto in the Mode Select area.
2. Press function key (F1) (Program Select).
3. Key in the program name.
4. Press the Write Key

*For 2 turret mode machines only.

Block to Block Operation

This method allows the operator to “single step” through the N.C. program one block at a time.

*Steps 1 thru 5 need only be done when you are selecting a new program.

1. Press the Auto Key in the Mode Select area.
2. Press “Single Block” switch on.
3. Place the individual/simultaneous switches to the appropriate settings.
4. Turn feed rate override to 10% to reduce rapid movements.
5. Press Cycle Start button.

NOTE: With the Single Block switch on you must depress cycle start after every block. (For automatic continuous operation, at Step 4, press Single Block switch off) and feed rate override to 100%.)

Sequence Re-Start

This method is used to restart the program to a specific section in the N.C. program by designating that sequence number.

CAUTION - After pressing the Sequence Restart button the machine advances to one block before the commanded restart block

1. Press the Auto key in the Mode Select Area.
2. Press function key (F3) (Part Program)
3. Press page until program is displayed. “For 4-axis machines page until both “A” and “B” programs are shown in a split screen.
4. “Select the “A” turret.
5. Press function key (F8) (EXTEND) to display F2/restart button.
6. Press “Single Block” switch “ON”
7. Press function key (F2) (restart).
8. Key in sequence number for upper turret program (N-).
9. Press Write Key.

NOTE: For 2-axis machines jump to procedure #14.

10. *Select "B" turret.
11. *Press function key (F2) (RESTART)
12. *Key in sequence number for lower turret.
13. *Press Write Key
14. Wait for cursors to stop on the program.
15. Press the Sequences Restart button.
16. Press Single Block switch off, press cycle start to run.

CAUTION - Before pressing Sequence Restart button decrease the feed rate override switch to minimize rapid movements.

Sequence Restart - Block Counter Method

This method of Sequence Restart entails using the block counter, which is located at the extreme top right corner of the CRT next to the "N" sequence number.

CAUTION - After pressing the Sequence Restart button the machines advances to one block before the commanded restart block.

1. Press the Auto key in the Mode Select area.
2. Press function key (F3) (Part Program)
3. Press Page until program is displayed. "For 4-axis machines page until both "A" and "B" programs are shown in split screen.
4. *Select "A" turret.
5. Press function key (F8) (EXTEND) to display F2 restart key.
6. Press function key (F2) (RESTART)
7. Key in block counter number for upper turret program (for example 15).
8. Press Write Key.

NOTE: For 2-axis machines jump to procedure #13.

9. *Select "B" turret.
10. *Press function key (F2) (RESTART).
11. Key in block counter number for lower turret program (for example 75)

12. *Press Write Key.
13. Wait for cursor to stop in the program.
14. Press Single Block key.
15. Decrease the feed rate override switch to minimize rapid movements
16. Press the Sequence Restart button.

*For 2 turret mode machines only.

MID-Auto Manual Mode and Restart Function

This method allows the operator to interrupt the machining cycle, manually jog away from the work piece, stop the spindle, change inserts, and then return back to the position where the machining cycle was interrupted.

1. Press the "Slide Hold" button.
2. Press the Mid-Auto Manual Button - (NOT THE MANUAL BUTTON)
3. Manually move the turret away from the work piece. At this time, the operator can stop the spindle and index the turret to change inserts.
4. Return turret to original station and restart spindle to get the STM light to stop blinking.
5. Manually move the turret close to the point where the slide hold button was originally pressed.
6. Press the "Single Block" switch on and decrease the feed rate override switch.
7. Press "Sequence Restart" button.

CAUTION - After pressing the Sequence Restart button, the machine moves in rapid toward the work piece.

8. Return feed rate override switch to 100%
9. Press the "Cycle Start" button.

The functions that are operative under this function are:

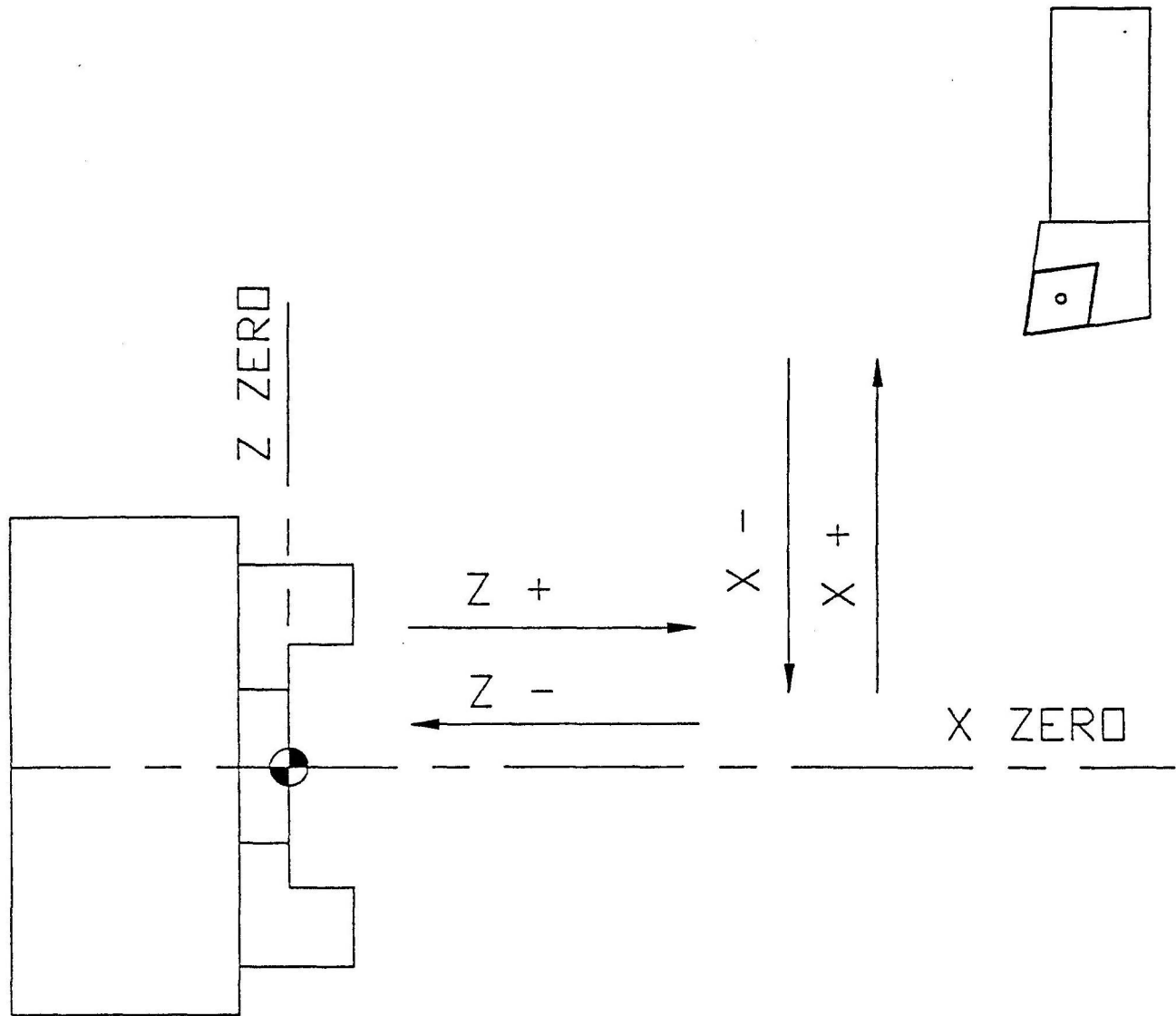
- A. Manual axis feed.
- B. Axis feed using the pulse handle.
- C. Spindle rotation (CCW, CW, STOP, JOG)
- D. Spindle gear range selection.
- E. Turret Indexing

COORDINATE SYSTEM AND PROGRAM ZERO POINT

Coordinate System

Absolute Position Encoder Advantages

COORDINATE SYSTEM AND ZERO POINT

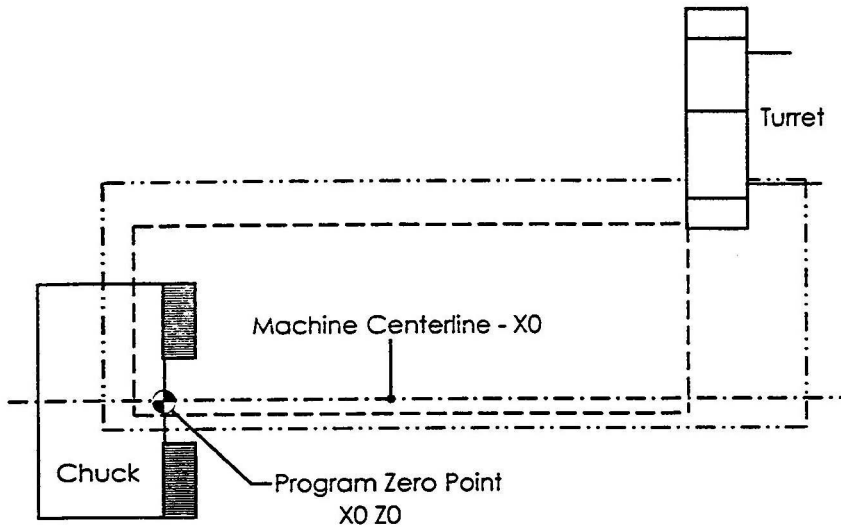


CNC machine tool motions must be identified carefully. Motions formerly done manually must now be explained numerically to the machine. Cutting tool and metal must come together at the right place and cutting conditions must be established. Points in space are specified to the machine by numerical coordinates along specified axes. This manual explains programming for Okuma Lathes with two perpendicular axes (common to most lathes).

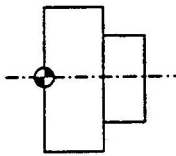
Positions on the work surface are identified by X and/or Z commands. All points are determined by their distance from a Zero point in each axis. Positions are positive or negative, depending on their position relative to Zero.

Program Zero Point - the program zero point defines a 'relative' X0 Z0 position with regards to the machine's center line. It defines the point along the machine's center line, which in turn, correlates to the part program's origin.

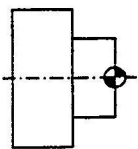
The illustration below depicts a case which the user has defined the program zero to be at the face of the chuck body. This 'zero' is often referred to as the part's 'Z' zero. This point corresponds to the programmer's X0 Z0 point, or 'origin' in the programming process.



Note:

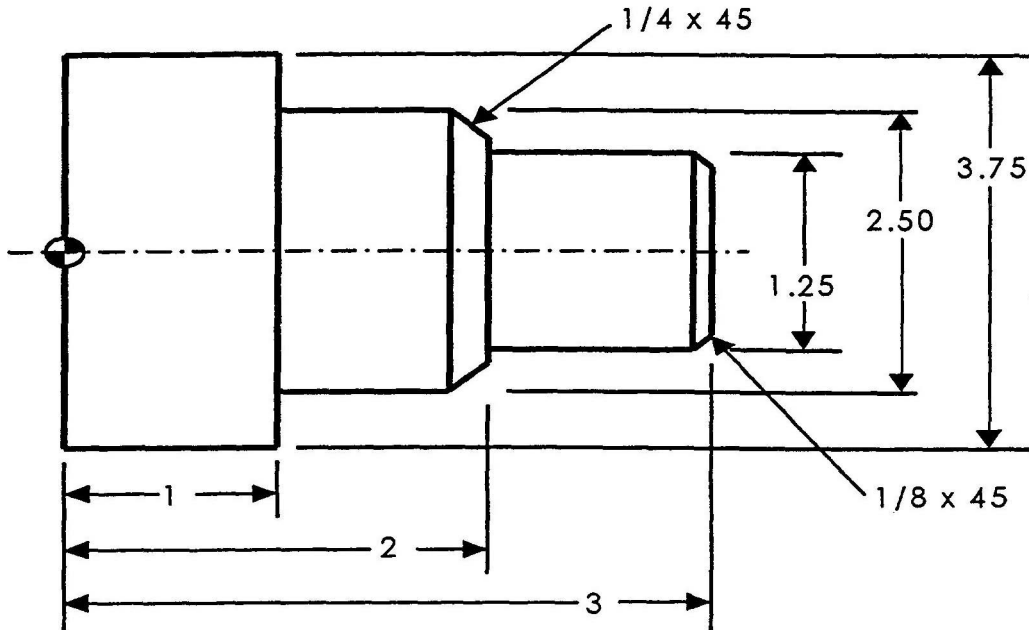


Fundamentally, there are two (2) methods of deciding the location of the Program Zero. In the Okuma method, the program zero point is typically placed at the left end of the work piece. The main advantage of this 'style' is that ALL programmed dimensions will - by default - be positive. Additionally, there is the benefit that the 'zero' point is effectively represented by a physical surface of the work holder.

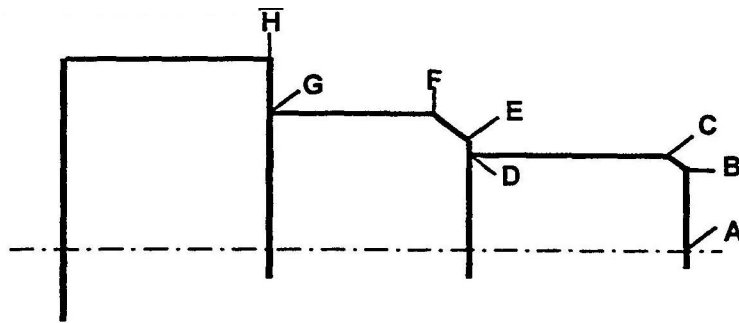


The other method places the program zero point at what is effectively the part's right end. Therefore, part programs that use this 'style', have negative 'Z' program dimensions. The decision of which method to use is mainly a matter of personal preference.

Coordinate Positions / Program Points



Often a cylindrical part is redrawn to show only one half of the basic part profile, and the program points in a logical sequence. The following illustration reflects this basic technique for the part, shown above.

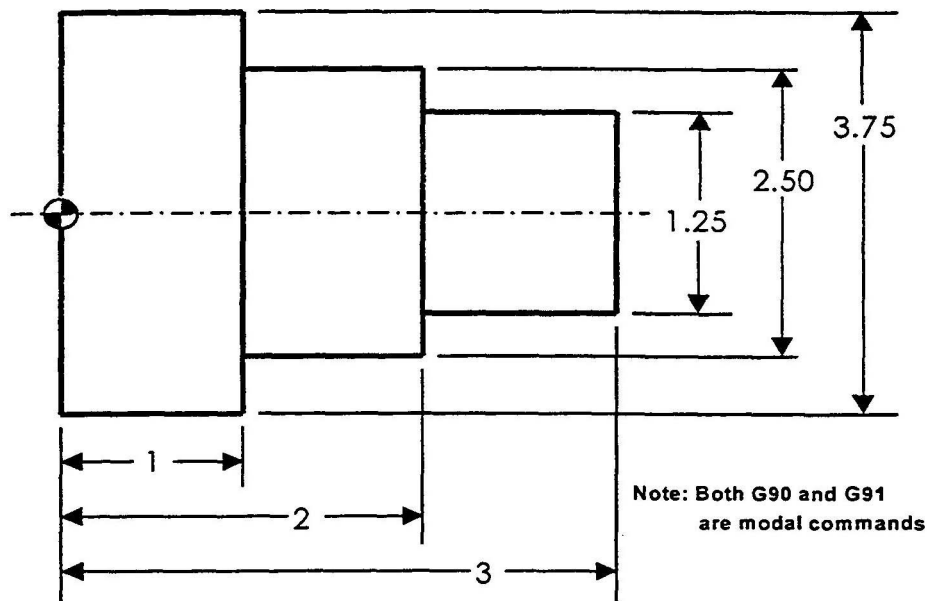


Identify the X and Z coordinates of the following points/positions:

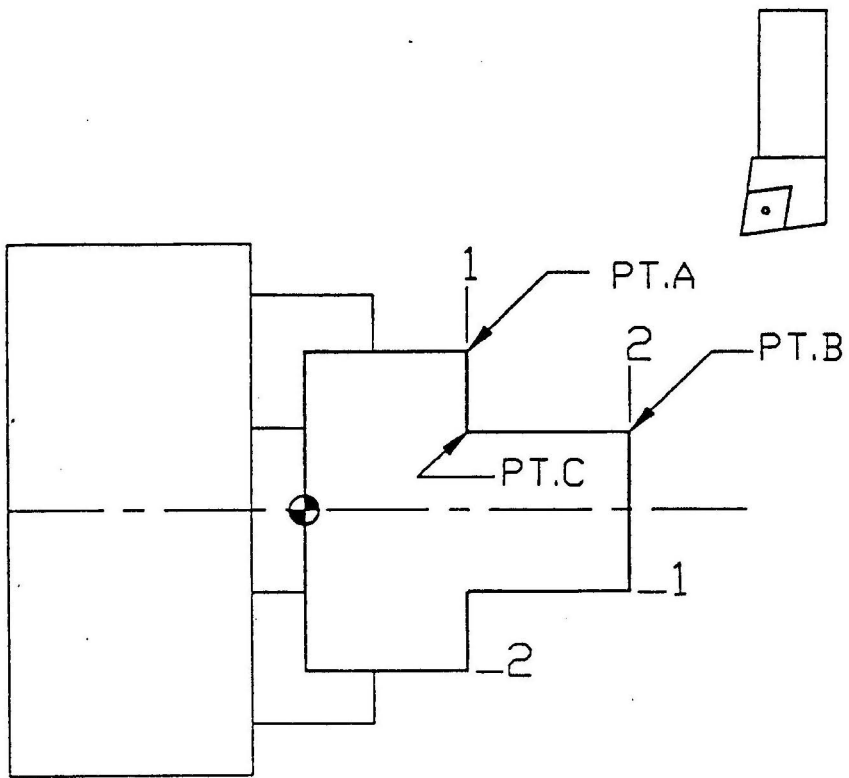
Point 'A'	[X]	[Z]
Point 'B'	[X]	[Z]
Point 'C'	[X]	[Z]
Point 'D'	[X]	[Z]
Point 'E'	[X]	[Z]
Point 'F'	[X]	[Z]
Point 'G'	[X]	[Z]
Point 'H'	[X]	[Z]

G90 Absolute Positioning - this is most common method of programming. The absolute method defines all the dimensions of the part based on parts origin point [X0 Z0]. It is the preferred method because it simplifies relating the part drawing dimensions to that of the actual part program X, Z dimensions. The majority of engineering drawings today are based on the 'datum dimensioning' technique, which, corresponds directly to absolute programming.

G91 Incremental Positioning - this method is only used when necessary. It describes the program path based on the tool's current coordinate in X and Z, and 'incrementally' defines the next 'target position' relative to the actual current position. It is similar in concept to chain dimensioning .

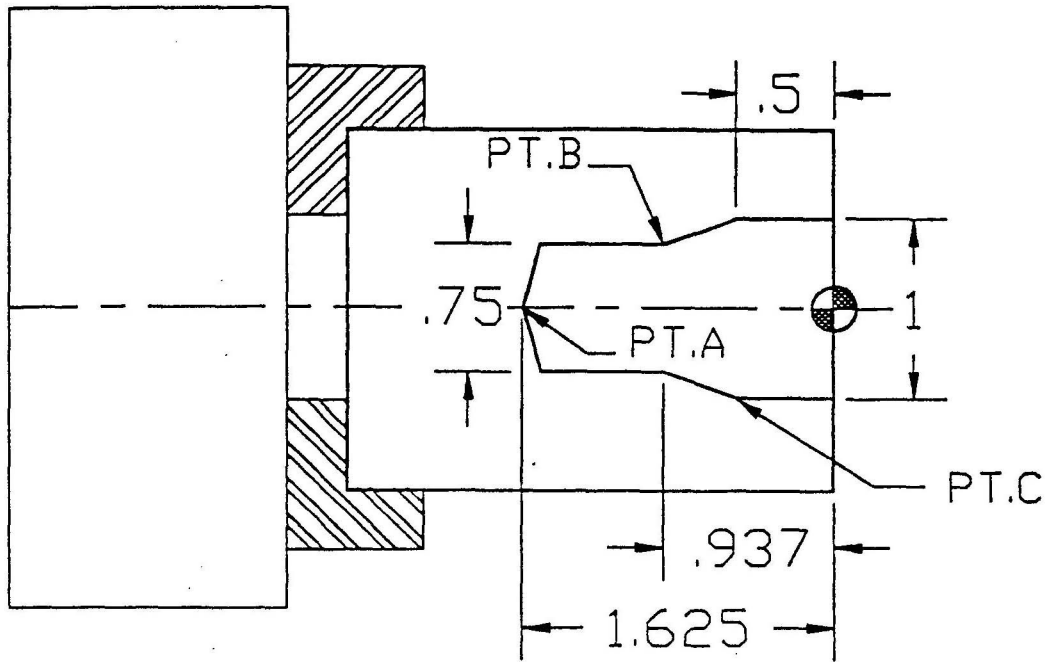


ABSOLUTE Method	INCREMENTAL Method
G50 S2000	G50 S2000
G00 X50 Z50	G00 X50 Z50
X1.25 Z3 T0101 G97S900 M03	X1.25 Z3 T0101 G97S900 M03
G01 Z2 F.015	G91 G01 Z-1 F.015
X2.5	X1.25
Z1	Z-1
X3.75	X1.25
G00 X50 Z50	G90 G00 X50 Z50
M02	M02



IDENTIFY
COORDINATES:

	X	Z
PT.A		
PT.B		
PT.C		



IDENTIFY
COORDINATES:

	X	Z
PT.A		
PT.B		
PT.C		

Program Codes

G Codes

Additional G Codes

M Codes

G Codes

G Code	Description	Notes
G00	Rapid Travel Positioning	
G01	Linear Interpolation	
G02	Circular Interpolation (Clockwise)	
G03	Circular Interpolation (Counter clockwise)	
G04	Dwell, expressed as seconds (used with the "F" format word)	
G13	Designates 'A' Turret (upper)	
G14	Designates 'B' Turret (lower)	
G32	Fixed Threading Cycle (face)	
G33	Fixed Threading Cycle (longitudinal)	
G34	Increasing Lead Threading Cycle	
G35	Decreasing Lead Threading Cycle	
G40	Cancellation of G41 or G42 Tool Nose Radius Compensation	
G41	Tool Nose Radius Compensation (left of material)	
G42	Tool Nose Radius Compensation (right of material)	
G50	Used with 'S' Format Word, for Maximum spindle Speed Designation	
G64	Droop Control 'OFF'	
G65	Droop Control 'ON'	
G71	Longitudinal Threading Compound Cycle	
G72	Transverse (end face) Thread Cutting Cycle	
G73	Grooving Cycle, Feeding on the 'X' Axis	
G74	Face Grooving/Drilling Cycle, Feeding on the 'Z' Axis	
G75	Automatic Chamfering (45 degree corners and fillets only)	
G76	Automatic Radius (corners and fillets only)	
G77	Right Hand Tapping Cycle	
G78	Left Hand Tapping Cycle	
G80	End of Contour Definition for LAP	
G81	Start of Longitudinal Contour Definition for LAP	
G82	Start of Transverse Contour Definition for LAP	
G83	LAP 4-High Speed Copy Turning	
G84	Change of Rough Cutting Conditions in the LAP 'G85' Cycle	
G85	Rough Cutting Cycle for LAP	
G86	Copy Turning Cycle for LAP	
G87	Finish Cutting Cycle for LAP	
G88	Continuous Thread Cutting Program	

G Codes	Description	Notes
G90	Absolute Coordinate Programming	
G91	Incremental Coordinate Programming	
G94	Feed rate Expressed as INCHES PER MINUTE	
G95	Feed rate Expressed as INCHES PER REVOLUTION	
G96	Constant Surface Footage	
G97	Direct Spindle RPM Command	
G110	Constant Surface Footage (cutting) - 'A' Turret Priority	
G111	Constant Surface Footage (cutting) - 'B' Turret Priority	
G152	Call for Programmable Tail stock	

Additional G Codes

FREQUENTLY USED CODES

G50 Spindle Speed Limit

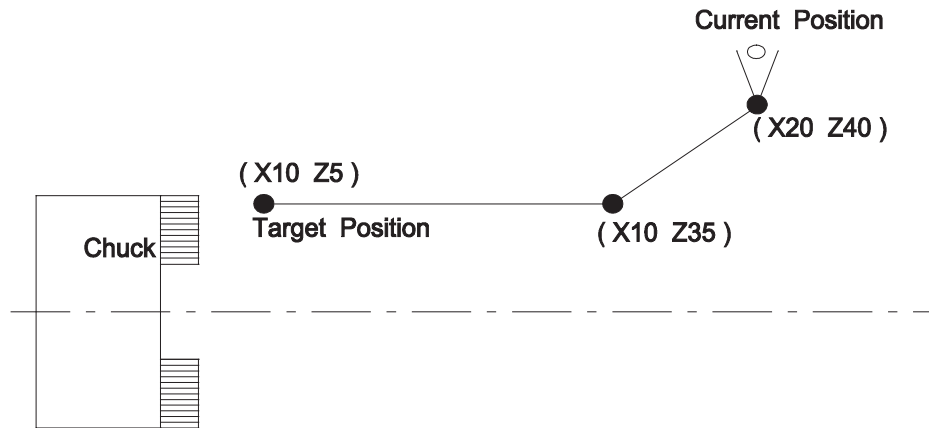
Power chucks are used on CNC lathes have a maximum safe RPM. This is usually stamped on the chuck. This maximum RPM must be input on the first line of every part program.

FORMAT:

G50 S _____

G0 Rapid Travel

Used to move the slides at full speed. Note the tool path in the G0. At rapid travel both axes move at full speed and no linear interpolation takes place. Though the axes move simultaneously they move individually until they reach the specified coordinate.



G0 X50 Z50: Rapid to home position for Turret Index

Set Soft Limits so that the longest tool can safely index. Also allow room for the operator to be able to move parts from the chuck without interference with the tools in the turret. The OSP will not allow the turret to index unless the machine is in this position

G94 and G95 - IPR and IPM Feed

In a cutting mode, the rate by which the turret or the X-axis and Z-axis is moved, is controlled by the “F” or feed rate word.

The mode of the “F” word is dictated by which “G” command code is chosen by the programmer.

The feedrate choices are as follows:

G94: Inches per minute

G95: Inches per revolution

The control powers up in the G95 mode, when the machine is turned on.

FEEDRATE: Fxxxx.xxx Inch per Minute

Fxxxx.xxx Inch per Revolution

Feedrate is used in conjunction with the G-codes G94 and G95 to control the rate of feed in a linear or circular contouring mode.

Most of the programmed feedrates will be in inches per revolution; however, if inch per minute programming is needed use this formula to calculate the value:

$$\text{IPM} = (\text{IPR} * \text{RPM})$$

ADDITIONAL USE FOR “F” WORD:

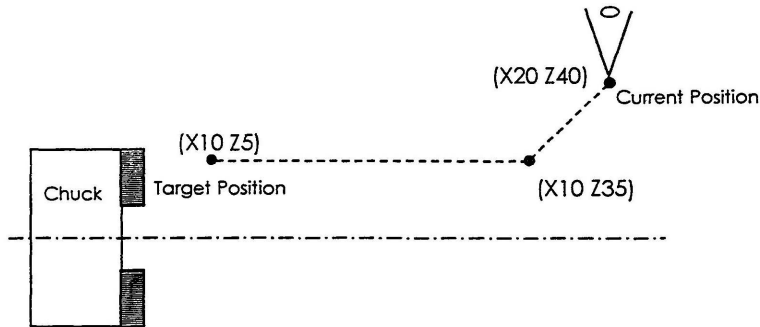
When using the G04 preparatory function the “F” word is the method used to describe dwell in seconds

G00 Rapid Travel Positioning - this command causes all the machines axis slides to move at 'full speed' to the target position.

Note that a G00 command does not always result in a 'straight-line' path of motion between the current position and the target position.

Effectively, both axes move simultaneously - (45 degree) until such time where either axis has met its target point, and any continued axis motion is done to satisfy the remaining axis.

FORMAT: G00.....Z.....



N1 G00 X50 Z50
N2 X10 Z5

Notes: example performed under MDI, assumes 'inch' mode.

Line 'N1' positions the turret at either of the machine's positive stroke end limits, or the current 'soft limit' positions. Main spindle rotation is not required during a 'G00' movement. The G00 command becomes modal following its initial occurrence. As no tool number is 'active' the positioning does not include any tool offset values. Positioning is relative to the current 'Z' zero offset position, and the machine's X and Z offset origins.

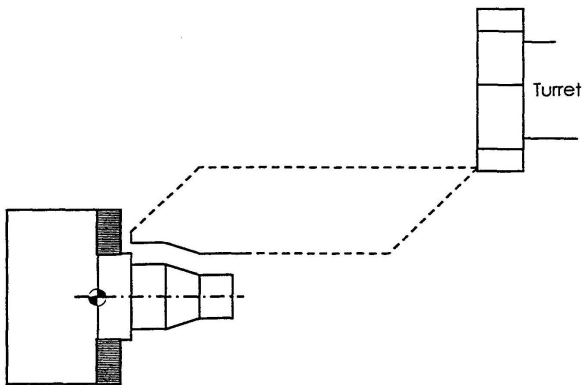
Line 'N2' positions the axes at X10 Z5. This line can be changed to: X10 Z5 T0101 to include the offset value for tool number 1. Note the difference in the final position, when including the tool offset value.

G01 Linear Interpolation - a fundamental program command that is used whenever 'cutting' takes place. A 'G01' command *requires* that an associated feed rate be either contained on the same 'line', or has been established prior to the G01 line. In addition, it is required that the main spindle be *rotating*, that condition being established either prior to the G01 command, or included on the same program line.

Unlike the G00 command, a G01 move will always be a 'linear' move. The machine's OSP control will always calculate the 'X' and 'Z' axis *coordination*, such that the tool tip will travel from its current position, to the (G01) 'commanded' position in a straight line.

The length of time it takes to complete the 'feed' move is a function of the actual spindle RPM, and the programmed feed rate ['F' Word]. The illustrations that follow depict a simple example.

FORMAT: G01 X.....Z.....F.....



When depicting the actual tool path motion, it is customary for the rapid paths to be identified as a dashed line, and the cutting paths to be shown as a solid line. This helps in the visualization of the machine's motions.

Feed Rate 'F' - this 'word' is used to define the desired feed per revolution during the cutting process. It can be thought of as the amount the tool will move axially for every revolution of the work piece.

Generally, rough turning tool feed rates range from 0.01 to 0 .02 per revolution, and finish tool turning typically ranges from 0.004 to 0 .01 per revolution.

At the time of initial program creation, the user only needs to select an approximate feed rate based on past experience, or cutting data tables. The optimum feed rate is ultimately tied to the complete cutting condition, material, and chosen insert.

Examples: ↗

G01 X Z F.010	G01 X Z F.020	G01 X Z F.004	G01 X Z F.0075
---------------	---------------	---------------	----------------

NOTE - the 'F' value is related to G95 or G94 selection.

◆◆◆

G04 'Dwell' and 'F' - occasionally there will be a need to program a dwell – or *wait*- directly into the part program. Perhaps the operation of grooving might need to allow for a dwell after the tool reaches its target position, allowing the cutting forces to relax, providing a better surface at the minor diameter.

While the dwell command uses the 'F' word, it has NO association or effect on G01 commands containing its own 'F' word.

FORMAT: G04 F.....

Examples: ↗

G04 F5	G04 F0.5	G4F8	G4F8.50
--------	----------	------	---------

G94 - Inch Per Minute - for the most part, this mode works with the 'F' word when the machine axes need to be moved at a controlled rate **-AND THE MAIN SPINDLE IS NOT REVOLVING.**

An example of this would be the use of a 'work pusher' or the use of a bar puller.

Example:

```
G50 S2000
G00 X50 Z50
G00X. . . .Z. . . .T. . . .
G94 G01 X. . . .Z. . . . F95
G04 F1
M84
G01 Z. . . . F40
M83
G04 F1
G01 X. . . .Z. . . . F95
G00 X50 Z50
G00 X. . . .Z. . . . S. . . .T. . . .M. . . .G95
```

The user must assure that the part program 'returns' the control to the G95 mode prior to returning the routine cutting mode.

G95 - Inch Per Revolution - during cutting, the 'F' word determines the rate at which the axis is moved. For turning equipment, the 'F' word is based on the *unit of inch per revolution*, or IPR. Traditionally, insert manufacturers state cutting data in this IPR method.

- The OSP control powers up with G95 mode as its default, and at every occurrence of control reset.

While it does not have to be placed into the part program, it is good practice to include a 'G95' at the 'top' of the program for ultimate authority in programming so that the program's initial mode is always G95.

Example:

```
G50 S2000
G00 X50 Z50
G00 X. . . . .Z. . . . .S . . . . .T. . . . . M . . . . .
G95 G01 X. . . . .Z. . . . . F. . . . . . . . . . .
```

Common M-Codes

M Codes	Description	Notes
M00	Program Stop	
M01	Optional Program Stop	
M02	End of Program	
M03	Spindle 'ON' Forward (clockwise)	
M04	Spindle 'ON' Reverse (counter clockwise)	
M05	Spindle 'OFF'	
M08	Coolant 'ON'	
M09	Coolant 'OFF'	
M20	Tail stock Barrier 'OFF'	
M21	Tail stock Barrier 'ON'	
M22	Thread Chamfering 'OFF'	
M23	Thread Chamfering 'ON'	
M24	Chuck Barrier 'OFF'	
M25	Chuck Barrier 'ON'	
M26	Thread Lead Along Z-Axis	
M27	Thread Lead Along X-Axis	
M28	Tool Interference 'OFF'	
M29	Tool Interference 'ON'	
M30	End of Program	
M32	Straight In Feed: Cutting on Right Hand Side of Threading Tool	
M33	Zig-Zag in Feed: Cutting path is Calculated as Side to Side	
M34	Angled in Feed: Cutting on Left Hand Side of Threading Tool	
M40	Spindle Gear Range - Neutral	
M41	Spindle Gear Range- Selection Depends on Machine Model	
M42	Spindle Gear Range- Selection Depends on Machine Model	
M43	Spindle Gear Range - Selection Depends on Machine Model	
M44	Spindle Gear Range - Selection Depends on Machine Model	
M55	Tail stock Quill Retract	
M56	Tail stock Quill Advance	
M60	Spindle Speed Answer Neglect 'OFF'	
M61	Spindle Speed Answer Neglect 'ON' G96 Mode	
M73	In Feed Pattern 1 (one) for Thread Cutting Depths	
M74	In Feed Pattern 2 (two) for Thread Cutting Depths	

M Codes	Descriptions	Notes
M75	In Feed Pattern 3 (three) for Thread Cutting Depths	
M83	Chuck Clamp	
M84	Chuck Unclamp	
M86	Turret Indexing Direction: Clockwise (Reverse)	
M87	Cancel M86 (Forward)	
M88	Air Blower 'ON'	
M89	Air Blower 'OFF'	
M90	Door (cover) Close	
M91	Door (cover) Open	

Program Format and Data Word/Address (letter/number combinations)

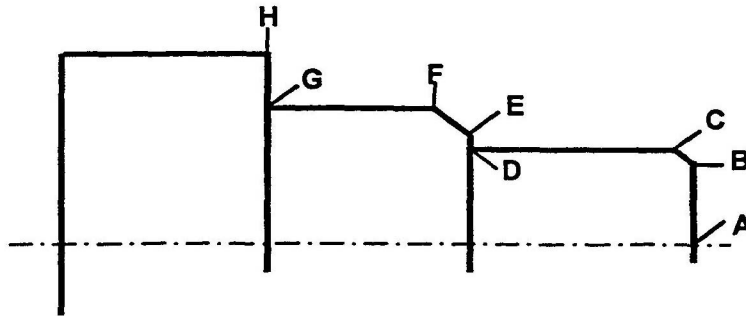
Refer to LB25-T min Program at Front

**Begin with Simple Examples; T-Command Will
be Covered in Detail Later**

Discuss comments inside Parentheses

Basic Program

The following program *example* uses the program points from the preceding part example.



G50 S2000	← Max. Spindle RPM
G00 X50 Z50	← Rapid to Home/ Index Position
X0 Z3.1 T0101 G97S975 M03 M08	← Approach Move, select tool speed
G01 Z3 F.01	← Feed to "A"
X1	← Feed to "B"
X1.25 Z2.875	← Feed to "C"
Z2	← Feed to "D"
X2	← Feed to "E"
X2.5 Z1.75	← Feed to "F"
Z1	← Feed to "G"
X3.75 +.1	← Feed to "H" and off part
G00 X50 Z50	← Rapid to home
M02	← End of Program

N001 G50 S2000	<p>Line Numbering – for ease in identifying a specific line within a part program, the program can accept a "N" word and up to 4 trailing alphanumeric characters.</p> <p>This illustration depicts applying "line numbering" to the above program example</p> <p>In addition to this the sequence line numbers are used when a "program restart" is performed.</p>
N005 G00 X50 Z50	
N010 X0 Z3.1 T0101 G97S975 M03 M08	
N020 G01 Z3 F.01	
N025 X1	
N030 X1.25 Z2.875	
N035 Z2	
N040 X2	
N045 X2.5 Z1.75	
N050 Z1	
N055 X3.75 +.1	
N060 G00 X50 Z50	
N065 M02	

S,T,M Execution - all S, T, M words which are part of any 'program block line' will always be executed on a 'first priority' level of hierarchy.

In the preceding example, the M08, M03, S650, and T0101 will ALL be 'established' **before** the machine slides are allowed to move to the commanded position.

The order in which the S, T, or M words are placed on a program line are not critical, but it must be possible for the OSP control to execute them in the specific order.

•

Depending on the programmers '**Style**', the following examples are given to show the many possibilities for a simple program.

G50 S2000 G00 X50 Z50 G00X3 Z8.2 T0101 G97 S650 M3 M8 G01 Z6 F.012 G01X4 Z4 G01Z2 G01X5.1 G00 X50 Z50 M02	G50 S2000 G00 X50 Z50 X3 Z8.2 T0101 G97 S650 M3 M8 G01 Z6 F.012 X4 Z4 Z2 X5.1 G00 X50 Z50 M02
G50 S2000 G00 X50 Z50 T0101 G97S650 M3M8 X3 Z8.2 G01 Z6 F.012 X4 Z4 Z2 X5.1 G00 X50 Z50 M02	G50 S2000 G00 X50 Z50 X3 Z8.2 T0101 G97 S650 M3 M8 G01 Z6 F.012 X4 Z4 Z2 X5.1 G00 X50 Z50 M02

The style of programming is virtually limitless. As long as the program 'code' is presented in an executable order, the control will accept any programming style.

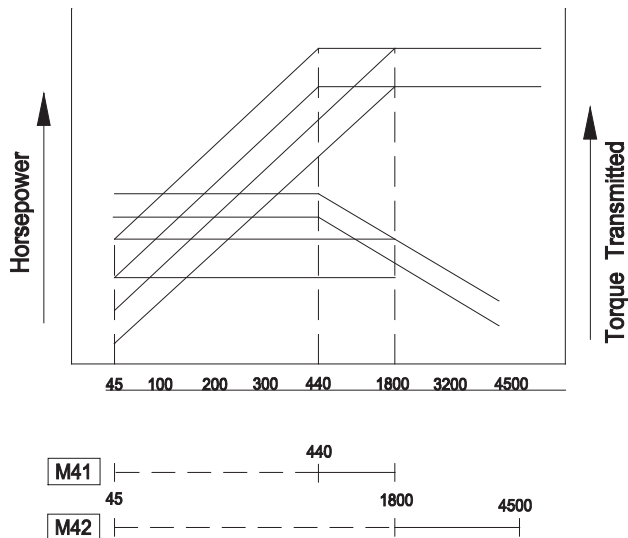
Spindle Ranges

Depending on your machine type, the headstock unit may have up to four (4) ranges. There are different methods that allow the headstock to change from one range to another. Please research your machine type to determine if your programs need to contain all the fundamental 'M' codes relating to spindle ranges.

Ranges: M41, M42, M43, M44

The smaller to moderate machines have the ability to change ranges 'on the fly'. Therefore, the spindle does not come to a stop, and higher productivity is the end result.

As a programmer, this requires knowledge of the torque / horsepower available for a given (spindle) range. The following charts are representative information that SHOULD BE REFERENCED PRIOR to setting a cutting condition or process.



For heavy-duty cutting, select a spindle speed in the — area so that cutting is performed within a constant output range.

Low speed/high speed range is changed by VAC motor coil connection (takes approx... 1 sec.)

Tool Selection Command - this command allows part program to select the desired tool to be indexed into the cutting position.

For reasons of clarity, a tool number is synonymous with a turret station number. The tool is identified by using a 'T' word, and trailing digits.

Formats

T__
T____
T_____

T010101



Ultimately, the command can consist of up to 3 pairs (6 digits) of data for a complete tool description.

Depending on the tool type being utilized, not all 3 pairs of data be present.

The following examples depict various 'T' word scenarios.

```
G50 S1000  
G00 X50 Z50  
X10 Z10 T0100
```



This program containing '**T0100**' allows for tool #1 to be placed at the cutting position, BUT, does **not** 'read' the number #1 tool's X and Z offset values when positioning to 'X10 Z10'. Therefore positioning will be based on current machine **origin** offsets. The turret body is placed at the commanded position, and **not** the cutting tool itself.

```
G50 S1000
G00 X50 Z50
X10 Z10 T010101
```

This program, containing 'T010101' allows tool #1 to be placed at the cutting position, reads the number #1 tools X and Z offset values, and reads, the corresponding X,Z nose radius values. Programs that utilize the tool nose radius function, require a corresponding **fully** described tool (**all** data elements). Failure to define the tool's nose radius register #, will disable **TNR** function (correction) cutting.

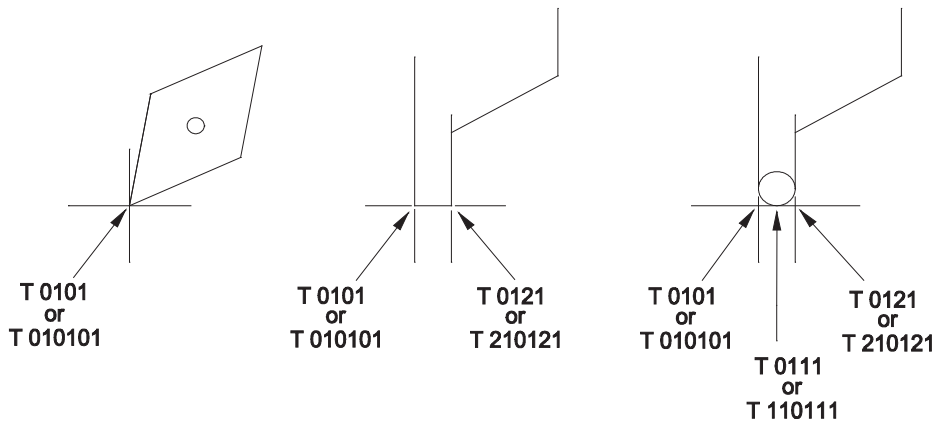
```
G50 S1000
G00 X50 Z50
X10 Z10 T0101
```

This program, containing T0101 allows for tool #1 to be placed at the cutting position, and in addition, reads the number #1 tool's X and Z offset values when positioning to X10 Z10. Therefore positioning will be based on the machine **zero** offsets, and the X, Z tool offsets.

Tool position is correct.



The following illustrates the fundamental positions of an 'offset', as well as depicts some common tools that typically require more than one offset to simplify program creation. Independent offset control is a simple means of controlling a part features final size.

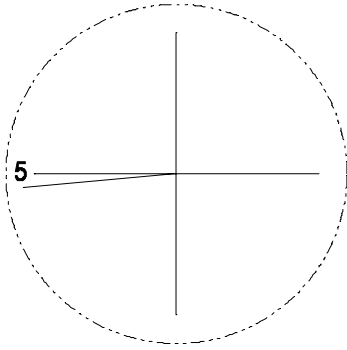


Problems

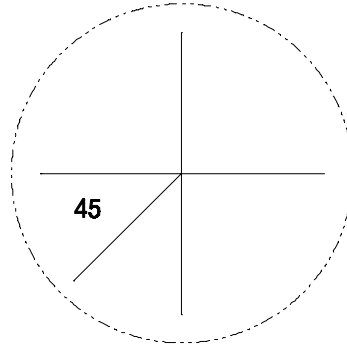
1. What is the difference a G90 and a G91 code?
2. What are the two primary axis of a lathe?
3. G00 causes the axis slide to move, how?
4. What are the two uses of the "F" word?
5. What is the difference between the G94 and the G95 codes and how do they affect the "F" word?
6. What word do you use for line numbering?
7. What is the importance of the "S", "M", "T" words?

ANGLE COMMAND

1. What are the angles for each circle?



A = _____, or _____, or _____



A = _____, or _____, or _____

2. When a tool is designated as a 4-place number, what are the meaning of the numbers?

3. Will the tool nose compensation be picked up by the control with a 4-place designation? Explain.

CIRCULAR INTERPOLATION

Write Simple Program on Board Part DR202-3

**Eliminate Speed and gear range
Write program with zero at front and back**

Write Program on Board for DR201-3

**Have a student key in the program on simulator as you write
Add graphics commands to program, without explanation to students**

Call Up and Test Program

Review

Circular Interpolation

Circular interpolation uses either a “G02” or “G03” command to allow the machining of arcs or radii either external or internal.

Fundamentally, an arc has a direction based on the advance of the cutting tool. An arcs direction is said to be either clockwise or counterclockwise in determining its program format.

FORMAT: G02 or G03 X.....Z...I +/-.....K+/-.....((L) option).....

While using a G02 or G03 command is not always mandatory to generate arcs, radii, or spherical features, this command is routinely used in manual programming, and virtually universal in its format to all CNC programming.

While the OSP control has other simplified programming methods to meet the needs of many circular features, the G02 and G03 commands can always handle the most complex of profiles.

The following illustrations depict various arc's and their corresponding direction. Understanding the arc's direction is a fundamental root level of knowledge for the programmer.

After the direction of the arc is defined, it is now necessary to define both the start point, or origin of the arc, and the end point of the arc.

Please note that the actual G02/G03 program command uses the X and Z words for the end point of the arc. HOWEVER, the above start points must be known such that the program line immediately prior to the G02/G03 command positions the cutting tool correctly.

The following illustration is provided to depict an example of both start and end points as related to an arc or radius.

TOOL DATA SET		SEIGYO-1.MIN		N	O	
		97/06/03		17:46:10		
PAGE	1	A - turret		UNIT 1mm		
		* TOOL OFFSET *		* NOSE-R COMP *		
NO.	T	XA	ZA	XA	ZA	P
1	A	0.000	0.000	0.000	0.000	O
2	A			0.000	0.000	O
3	A			0.000	0.000	O
4	A			0.000	0.000	O
5	A			0.000	0.000	O
6	A			0.000	0.000	O
7	A			0.000	0.000	O
8	A			0.000	0.000	O
9	A			0.000	0.000	O
10	A			0.000	0.000	O
11	A			0.000	0.000	O
12	A			0.000	0.000	O

LAST DATA * = 0.000 X = 500.000 Z = 500.000 TOOL T 7

=

SET	SEARCH	CAL	ONETOU			
			-CH CAL	ITEM ↑	ITEM ↓	[EXTEND]

At this time, the programmer needs to decide whether to define the arc's center point by "I" and "K" words, or use the simpler format and "L" word.

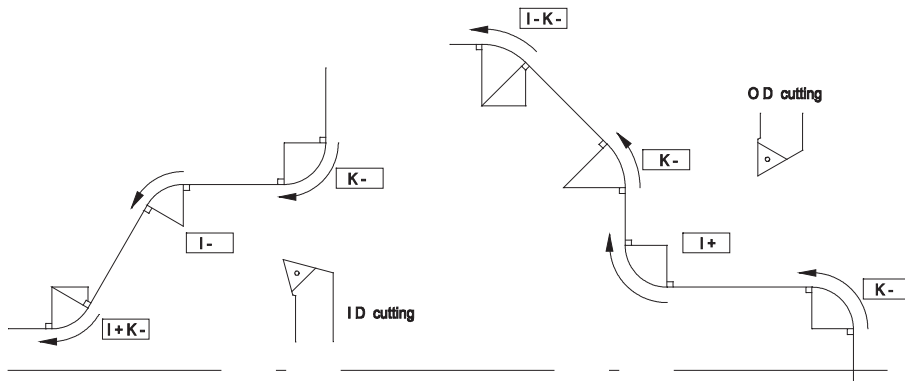
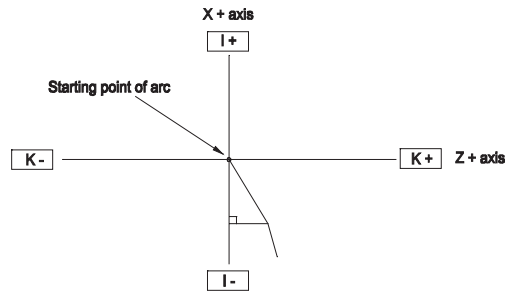
Ultimately it becomes a decision based on what style the programmer is most comfortable with and ultimately what information is given on the part drawing.

Either method is acceptable and equally accurate.

Admittedly the "L" word is oftentimes much easier to use as the "L" word equals the size of the desired radius.

For those cases where the programmer chooses to use the "I" and "K" words, the following chart is provided to assist the programmer in determining both the identification of "I" and "K" words and most importantly the respective "sign" (+/-)

Fundamentally, the "I" and "K" words describe the distance from the arcs start point to the center of the arc. These distances need to be defined **incrementally**.



When the cutting tool is positioned at the start point of the arc, the following 2 questions will aid the programmer. Imagine the tool tip being at the start point of the arc then ask the following.

1. Is the center of the arc to the left/right of the tool ? This determines the sign (+/-) of the "K" word.
2. Is the center of the arc equal to, further away, or below the current "X" position? This determines the sign (+/-) of the "I" word.

What is the actual distance from the arcs start point to the arc center.

These distances become the incremental values for "I" and "K" respectively. Please note that for many cases either the "I" or "K" word value could/can be zero.

In summary, four things must be specified to machine an arc on any CNC machine: They are as follows:

1. Origin of the arc.
2. Center of the arc.
3. End point of the arc.
4. The direction of travel.

Programming Steps:

1. Position the tool to the origin (start point) of the arc.
2. Define the direction of travel:

G02 Clockwise
G03 Counterclockwise

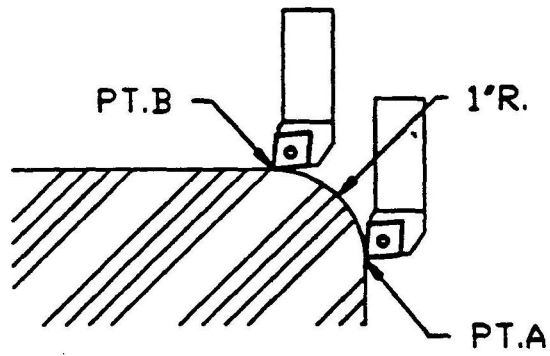
3. Define the end point in X and Z.
4. Define the center of the arc by the two following words:

“I” incremental distance from the origin to the center in X
“K” incremental distance from the origin to the center in Z

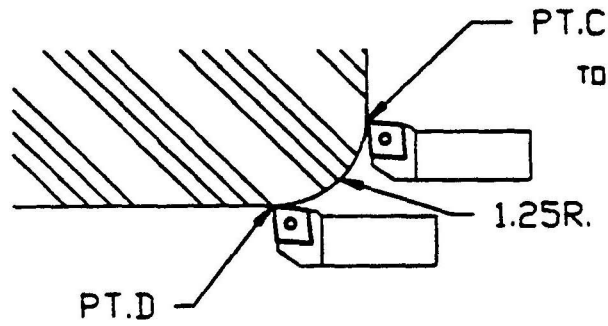
OR

“L” radii of the arc.

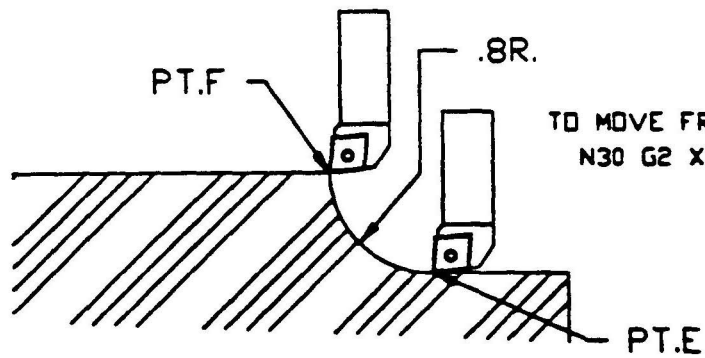
More detailed explanation of L follows. “L” is much easier to use than the traditional use of I and K.



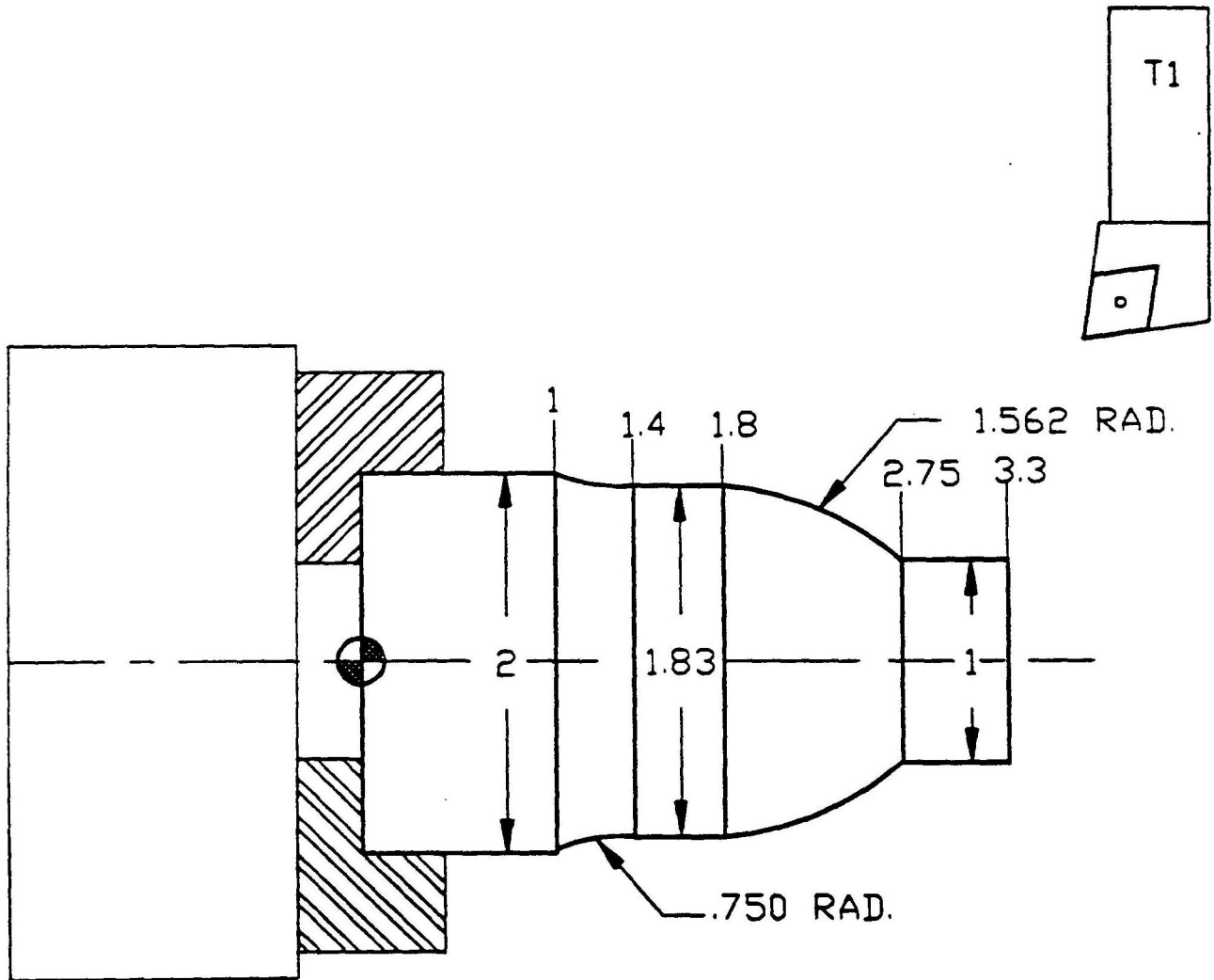
TO MOVE FROM PT.A TO PT.B:
 N20 G3 X4.2 Z3.12 L1



TO MOVE FROM PT.C TO PT.D:
 N25 G2 X1.6 Z-1.25 L1.25



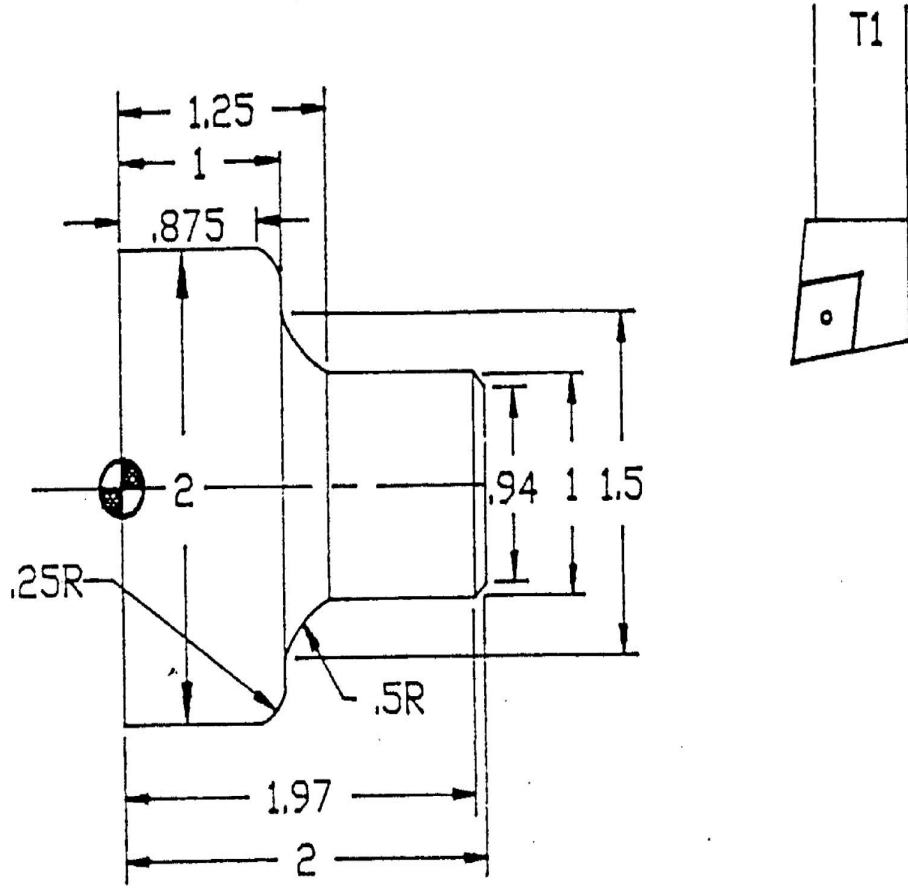
TO MOVE FROM PT.E TO PT.F:
 N30 G2 X5.625 Z2 L.8



WRITE SIMPLE PROGRAM ON BOARD PART DR202-3

Eliminate speed and gear range

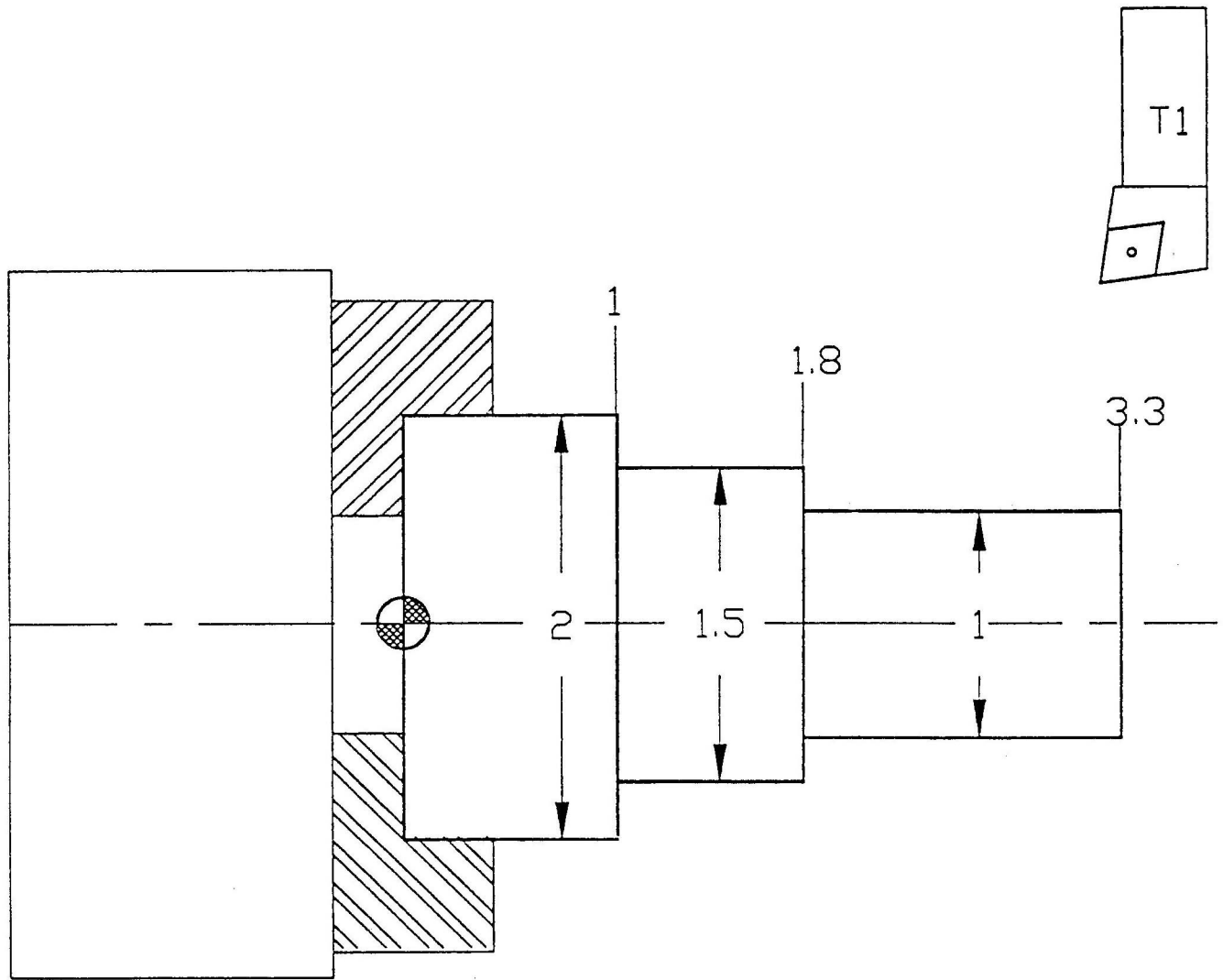
Write program with zero at front and back



WRITE PROGRAM ON BOARD FOR DR201-3

**Have a student key in the program
on simulator as you write**

**Add graphics commands to program,
without explanation to students**



CALL UP AND TEST PROGRAM

REVIEW

MACHINING GUIDELINES

Surface Footage, Feedrate and Depth of Cut

Cutter Radius Compensation

Discuss what CRC is, how it works, and it's advantages

RPM and SFPM FORMULAS

1) Revolutions per minute (RPM) is a machine setting that must be given to the lathe operator. RPM describes how fast the workpiece is turning and is simply the number of revolutions the workpiece makes during one minute. RPM is independent of the workpiece diameter and is one of several machining parameters that are used to estimate machine capability.

Formula:

$$\text{RPM} = 12 \times \text{SFPM} / 3.1416 \times \text{DIA. OF WORKPIECE}$$

2) Surface Feet Per Minute (SFPM) is a measure of cutting speed universally used by tooling manufacturers. SFPM is the distance that a point on the outer diameter of a rotating workpiece travels during one minute.

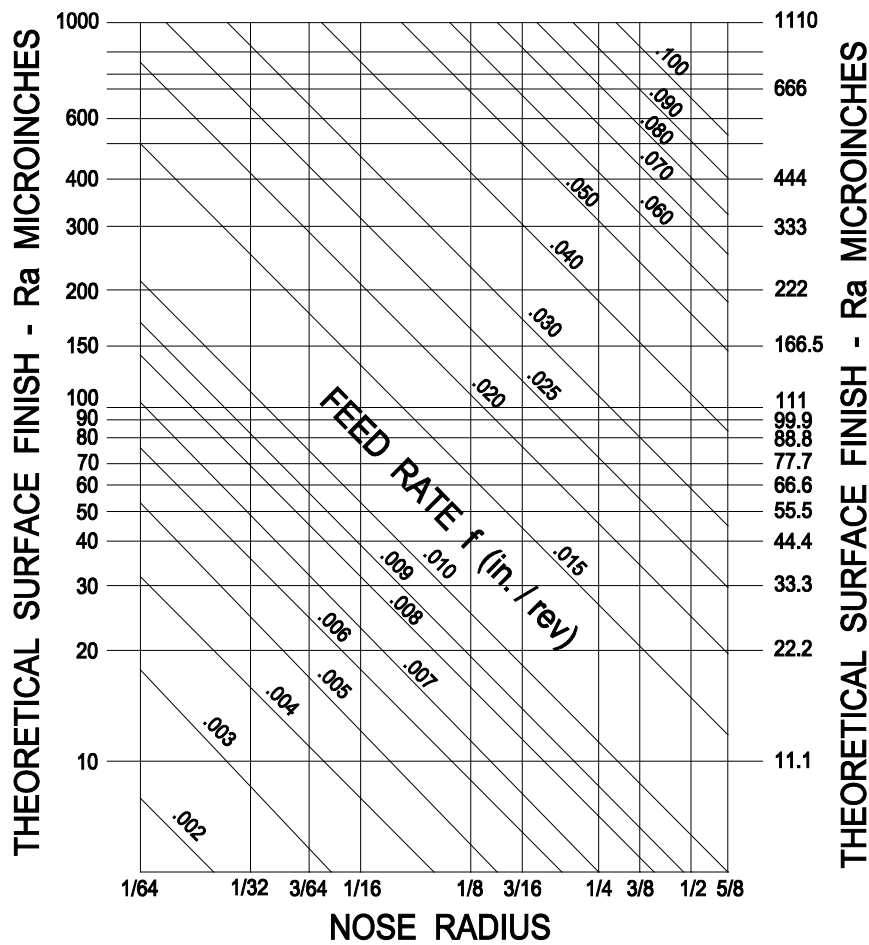
Formula:

$$\text{SFPM} = 3.1416 \times \text{DIA. OF WORKPIECE} \times \text{RPM} / 12$$

Note:

When machining with carbide inserts, SFPM is a critical operating parameter specified by tooling manufacturers. Whereas RPM is simply a machine setting, SFPM is the speed of the workpiece relative to the tool, directly impacting tool life and productivity.

Nose Radius Selection and Surface Finish



Nose radius and feed rate have the greatest impact on surface finish. To determine the nose radius required for a theoretical surface finish, use the following procedure and the chart above.

1. Locate the required surface finish (RMS and R_a) on the vertical axis.
2. Follow the horizontal line corresponding to the desired theoretical finish to where it intersects the diagonal line corresponding to the intended feed rate.
3. Project a line downward to the nose radius scale and read the required nose radius.
4. If this line falls between two values, choose the larger value.
5. If no available nose radius will produce the required finish, feed rate must be reduced.
6. Reverse the procedure to obtain the surface finish from a given nose radius.

CUTTER RADIUS COMPENSATION

Discuss what CRC is, how it works, and its advantages

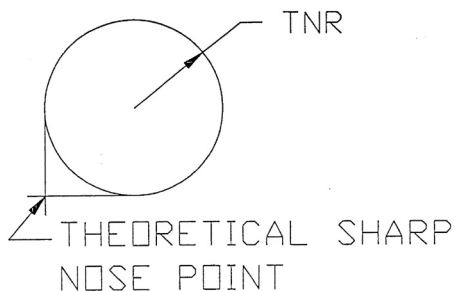
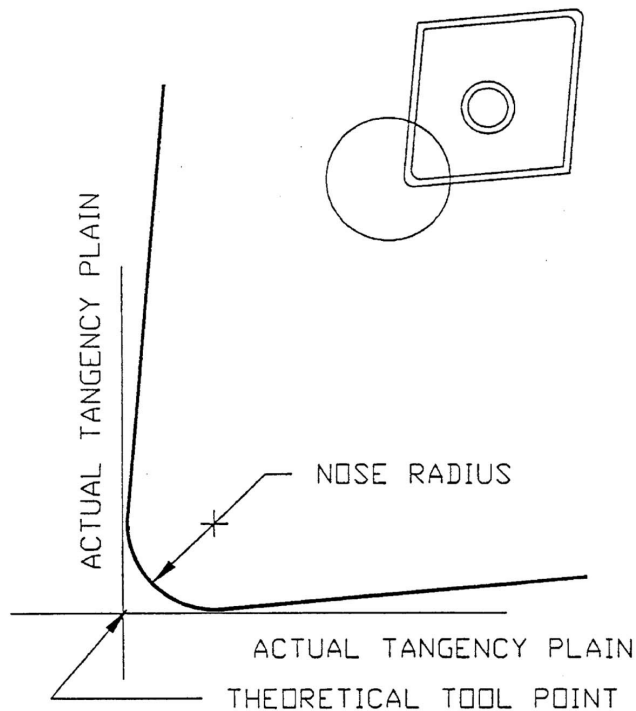
Tool Nose Radius Compensation

FORMAT:

- G40 Cancel of TNR
- G41 TNR Compensation (Left)
- G42 TNR Compensation (Right)

For straight turning and facing cuts, the THEORETICAL sharp point tool nose and the actual tangent point of the tool radius doing the cutting, are the same. When turning an angle or radius, there is a difference in the location of the THEORETICAL sharp point and the actual tangent points of the radius that must be compensated. This is the function of TOOL NOSE RADIUS COMPENSATION.

Radius Compensation is also helpful in programming because part print dimensions can be programmed, instead of having to do tool center off-set calculations for each motion.



When using TOOL NOSE RADIUS COMPENSATION all programming is done to the THEORETICAL SHARP TOOL NOSE POINT.

This means that the part program is a reflection of your engineering drawing. It also eliminates the costly and time consuming calculations needed to compute tool center line offsets for angles or arcs.

Consideration must now be given to the relationship between the THEORETICAL SHARP TOOL NOSE POINT and the surface being cut. This is determined from the direction from which the surface is approached and which direction the tool travels along the surface.

Two “G” codes are provided for this purpose:

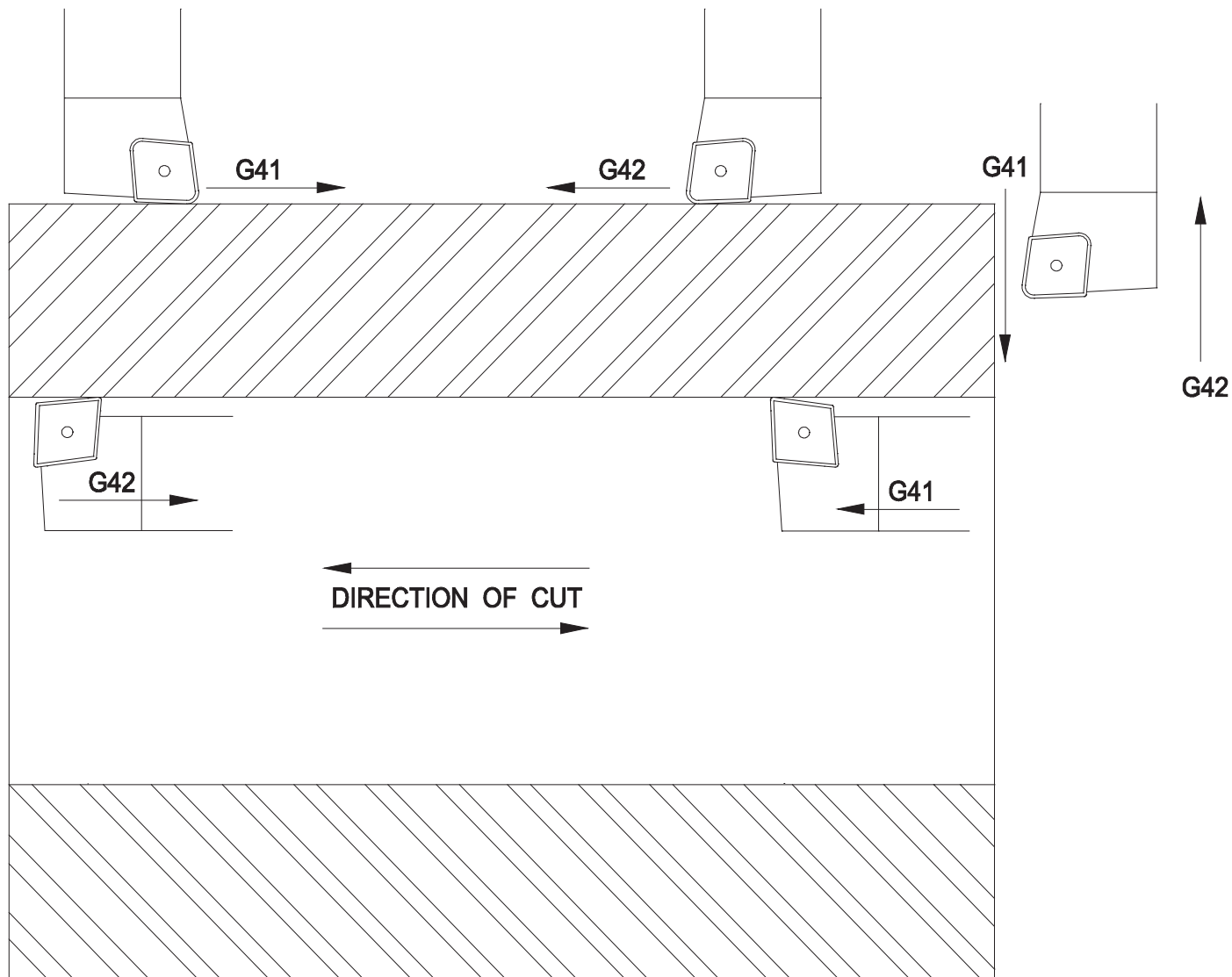
G41: Tool nose radius or cutter compensation – LEFT

A code for tool nose radius compensation when the tool center is on the LEFT side of the work surface to be machined, looking the direction of the tool motion.

G42: tool nose radius or cutting compensation – Right

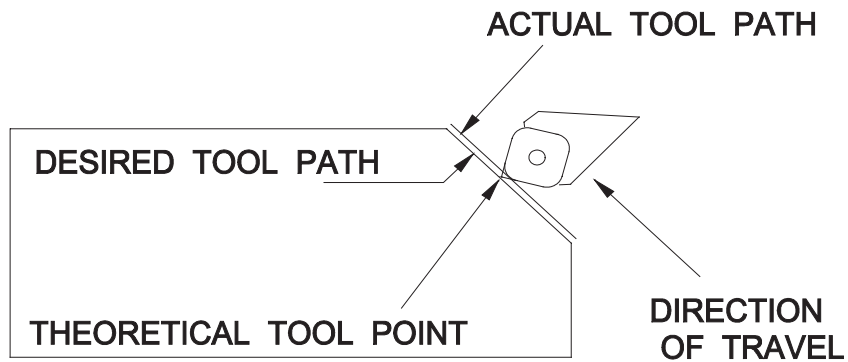
A code for tool nose radius compensation when the tool center is on the RIGHT side of the work surface to be machined, looking in the direction of the tool motion.

TNR3 TOOL NOSE COMP. DIRECTION

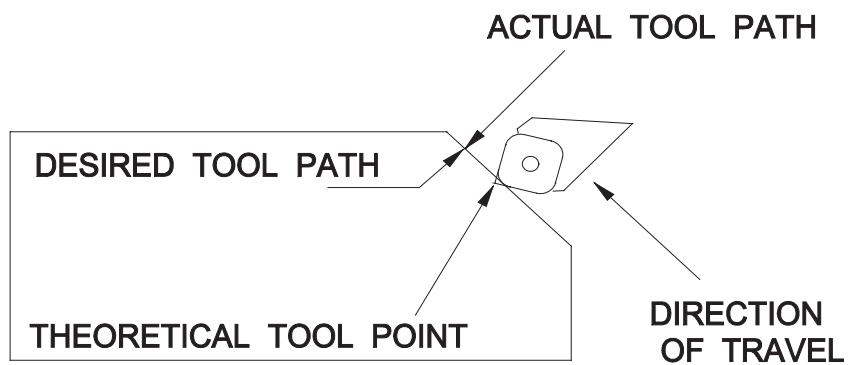


Shown below is the improper tool path caused by not using TOOL NOSE RADIUS COMPENSATION and the proper tool path resulting from TOOL NOSE RADIUS COMPENSATION.

Note location of the THEORETICAL SHARP TOOL NOSE POINT in both the examples



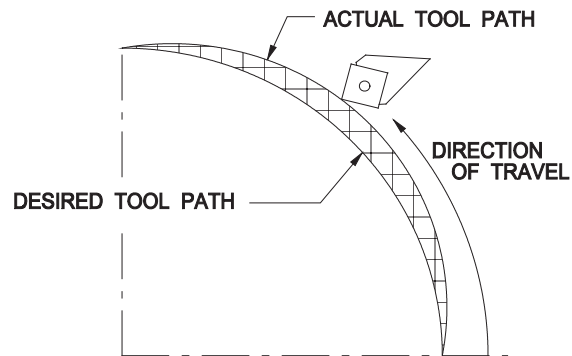
MOTION PROGRAMMED TO THEORETICAL TOOL POINT WITHOUT TOOL NOSE RADIUS COMP... RESULTS IN WRONG CUTTER PATH.



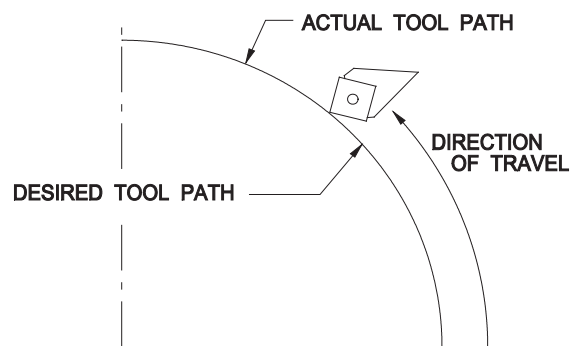
MOTION PROGRAMMED TO THEORETICAL TOOL POINT WITH TOOL NOSE RADIUS COMP.. RESULTS IN CORRECT CUTTER PATH.

Shown below is the improper tool path caused by not using TOOL NOSE RADIUS COMPENSATION and the proper tool path resulting from TOOL NOSE RADIUS COMPENSATION.

Note location of the THEORETICAL SHARP NOSE POINT in both of the examples.

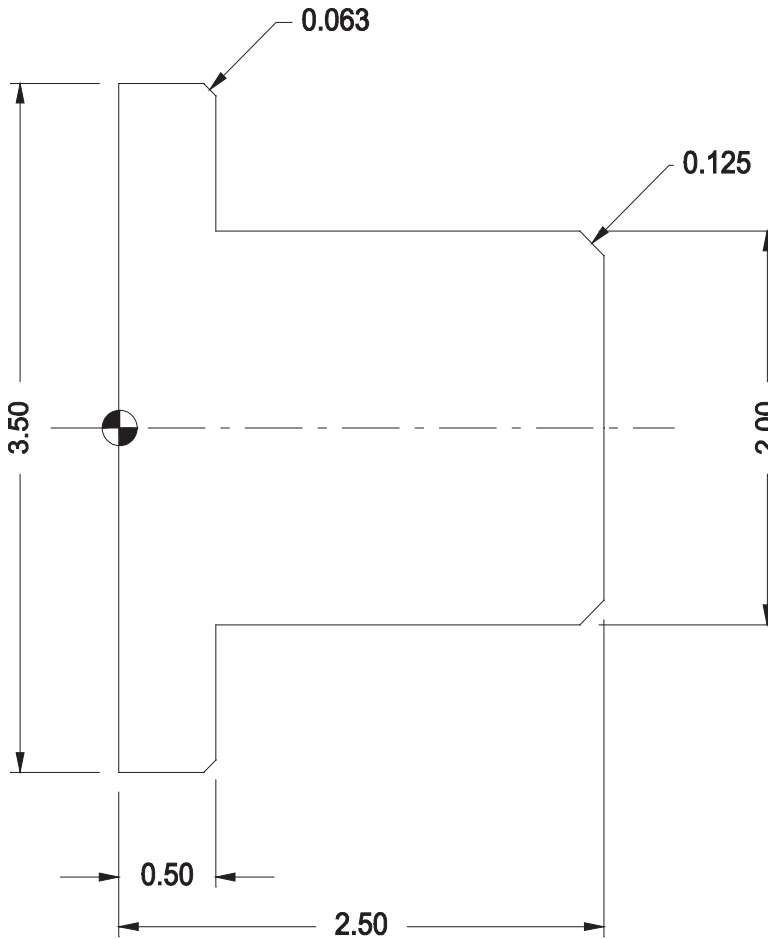


MOTION PROGRAMMED TO THEORETICAL TOOL POINT WITHOUT TOOL NOSE RADIUS COMP. RESULTS IN WRONG CUTTER PATH.



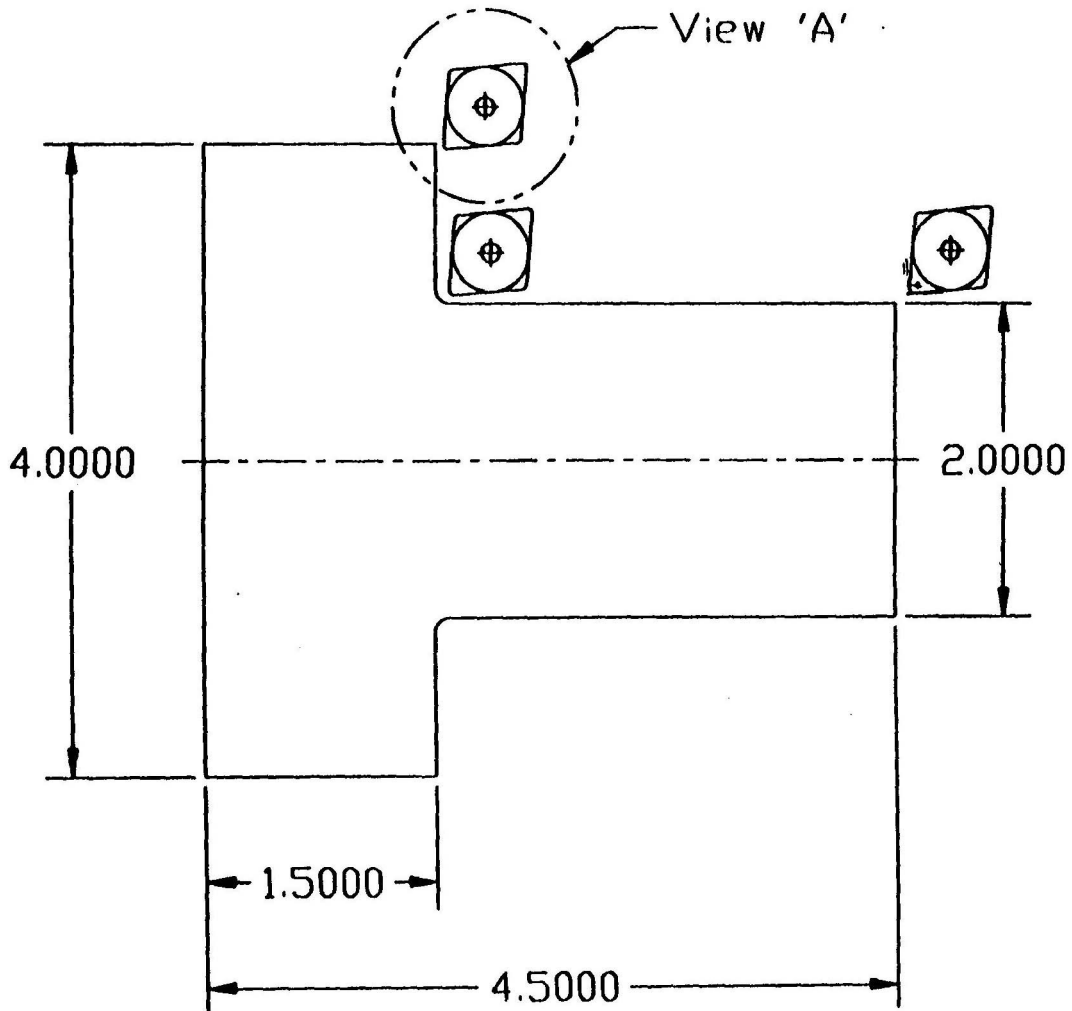
MOTION PROGRAMMED TO THEORETICAL TOOL POINT WITH TOOL NOSE RADIUS COMP. RESULTS IN CORRECT CUTTER PATH.

PROGRAMMING EXAMPLE:
TOOL NOSE RADIUS COMPENSATION G42



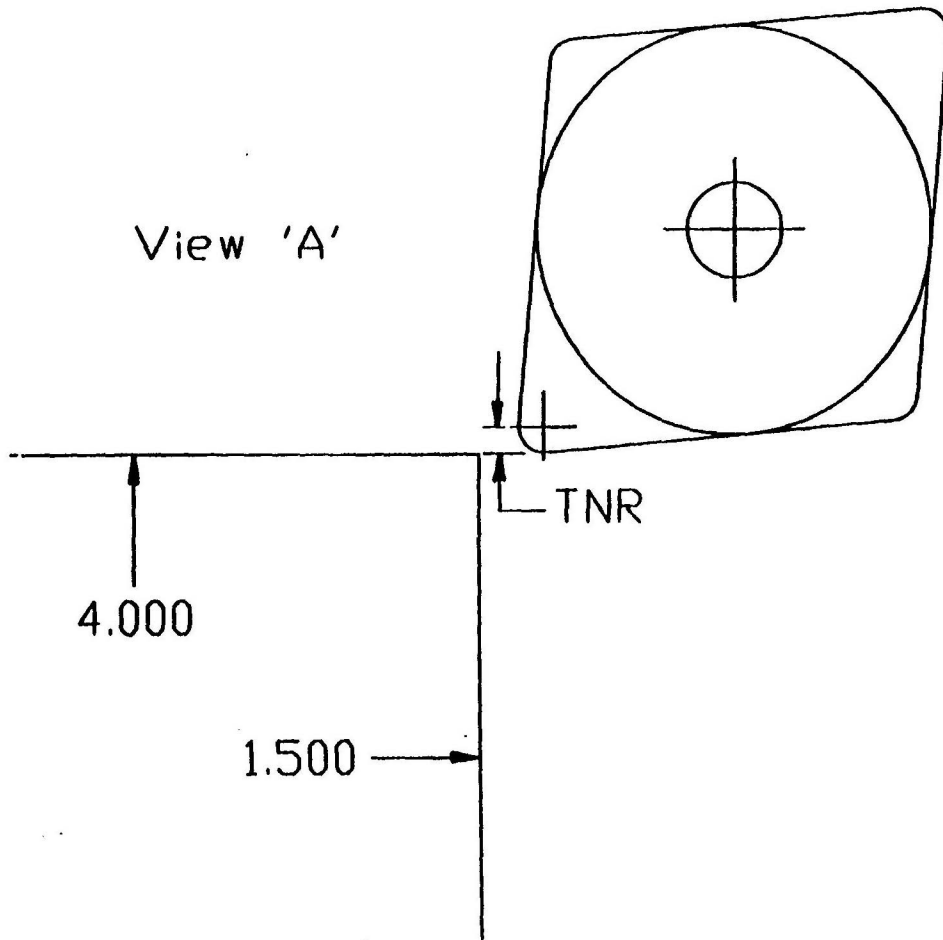
```
G13  
G50 S3200  
G0 X25 Z25  
N1 X1.75 Z2.6 S1500 T20202 M3 M42 M8  
N2 G42 G1 Z2.5 F.012  
N3 X2 Z2.375  
N4 Z.5  
N5 X3.375  
N6 X3.5 Z.4375  
N7 G40 X3.6 K-1  
G0 X25 Z25  
M2
```

TNR CANCEL METHOD



```
G50 S2000  
G00 X50 Z50  
G42 X2 Z4.6 T010101 G97 S1100 M03  
G01 Z1.5 F.015  
X4  
G40  
G00 X50 Z50  
M02
```

G40 with "k" word

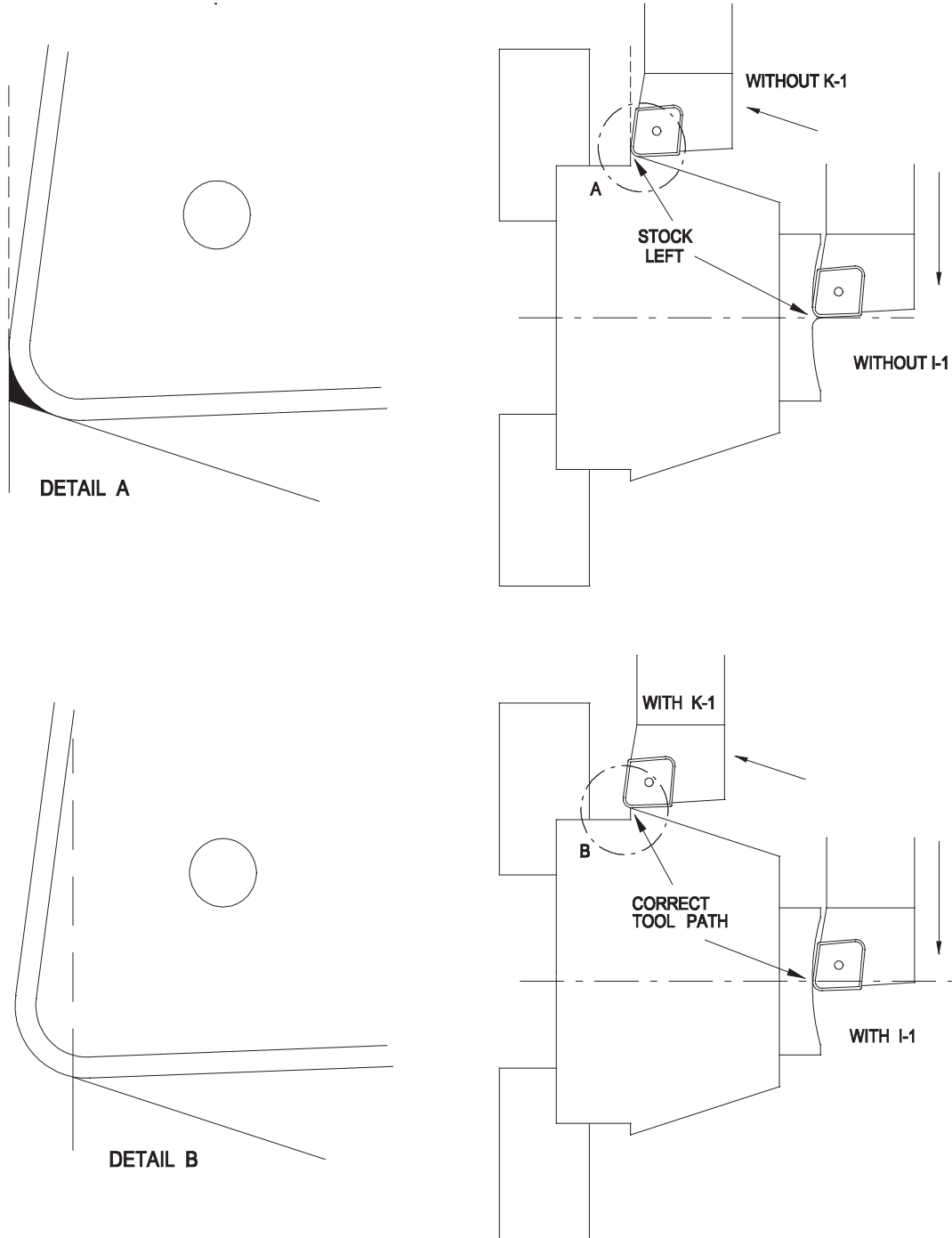


```
G50 S2000
G00 X50 Z50
G42 X2 Z4.6 T010101 G97 S1100 M03
G01 Z1.5 F.015
G40 X4 K-1 (positions as shown)
G00 X50 Z50
M02
```

Use of I and K when cancelling tool Nose Comp.

TNR6

Without K-1



TOOL DATA SET N 8

Page 1 A-turret UNIT 1in

NO.	T	* TOOL OFFSET *		* NOSE-R COMP *		P
		XA	ZA	XA	ZA	
1	A	0.4867	-0.0000	0.0310	0.0000	3
2	A	-0.9537	0.0000			3
3	A	123.0000	0.0000			0
4	A	456.0000	0.0000			0
5	A	789.0000	0.0000			0
6	A	0.0000	0.0000			0
7	A	0.0000	0.0000			0
8	A	0.0000	0.0000			0
9	A	0.0000	0.0000			0
10	A	0.0000	0.0000			0
11	A	0.0000	0.0000			0
12	A	0.0000	0.0000	0.0000	0.0000	0

LAST DATA == 0.0000 X= 13.2674 Z= 0.9853 TOOL T 6

DIRC

BACK UP ADD CONST. ADD CONST. SUB GUIDE ON/OFF ITEM# ITEM# [EXTEND]

F1 F2 F3 F4 F5 F6 F7 F8 ▶

Setting Compensation Value

Set the tool compensation value in the NOSE R COMP columns on the TOOL DATA SET screen. The compensation amounts can be set in the same manner as setting tool offsets.

Orientation of nose R center in reference to the imaginary tool tip may be set either by a positive or negative sign preceding the compensation amount or P number. If P numbers are used, set the number in the P column.

Tool Offset and Nose R Compensation Data Setting Screen



<Procedure>

1. Select the tool data setting mode.
2. For the two-saddle or two-turret machines, select the screen for turret A by pressing the turret selection [A] key on the NC operation panel.
3. Move the cursor to X (XA) column of NOSE – R COMP using the cursor keys.

4. Press function key [F1] {SET) and input the compensation data.

Input the compensation data for Z (ZA) also.

The allowable maximum setting value of nose-R compensation value is ± 999.999 mm.

5. Set the direction of imaginary tool tip by coded number. Move the Cursor to P column and set the coded number of the direction of imaginary tool tip. If "O" is set for P, the direction of imaginary tool tip is determined by the sign of the nose-R compensation data.

It is possible to set the tool form on the screen up to tool No. 12 on standard machines. When tool form data is set for tool No. 13 and above, graphic display of the tool form is not available. And the sign must be input carefully when setting the data.

In such cases, simplified graphic display is provided for tool form. By setting the parameter bit, it is possible to display the graphic information needed to confirm the direction of nose R compensation.

Using this feature it is possible to check the direction of nose R compensation, which is determined by the X-axis data, Z-axis data and the P code number, while inputting the data on the tool data setting screen.

Parameter bit position (OPTIONAL PARAMETER (OTHER FUNCTION 2))

Nose-R setting graphic (1: special/0: standard)

- 0: Standard graphic display
- 1: Simplified graphic display



AUTO CHAMFER & AUTOMATIC RADIUS

G50 S2000	← Max. Spindle RPM
G00 X50 Z50	← Rapid to Home/ Index Position
X0 Z3.1 T0101 G97 S1200 M03 M08	← Rapid approach position, tool select
G01 Z3 F.011	← Feed to face
G75 X1.25 L.125	← Feed to face and .125 chamfer
Z2	← Feed to shoulder
G75 X2.5 L.25	← Feed up face and .25 chamfer
Z1	← Feed to shoulder
G75 X3.75 L.375	← Feed up face, and .375 chamfer
Z.1	← Feed off
G00 X50 Z50	← Feed to Home Position
M02	← End of Program

Notes:

G75 is effective only with G01 mode.

G75 is non-modal and active only in the commanded program block

The axis movement must be greater than the absolute value of the “L” word.

The program block can contain only one dimension word either X or Z.

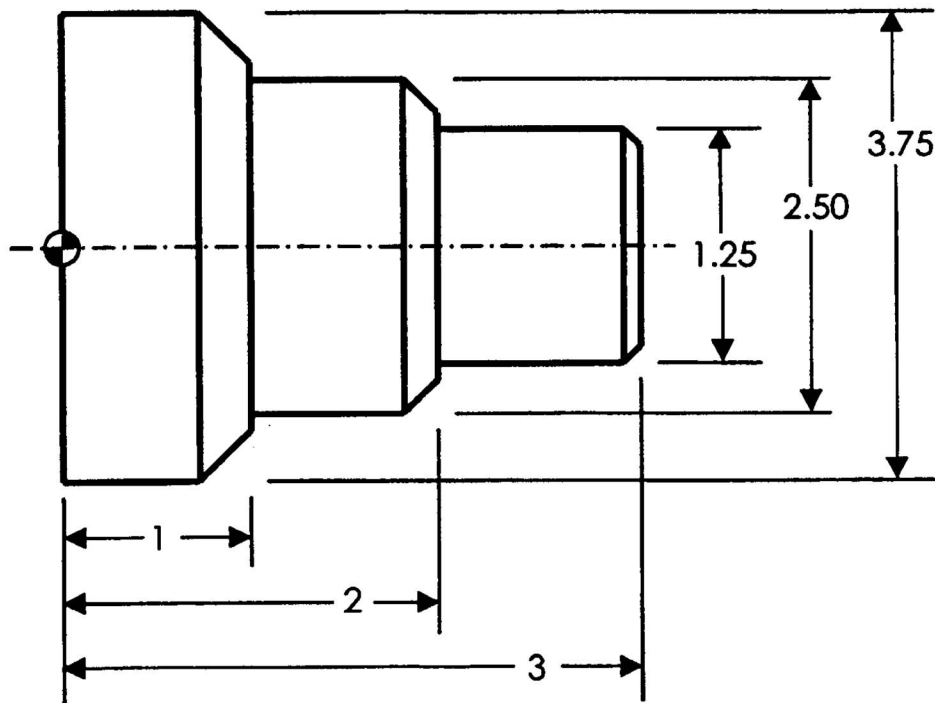
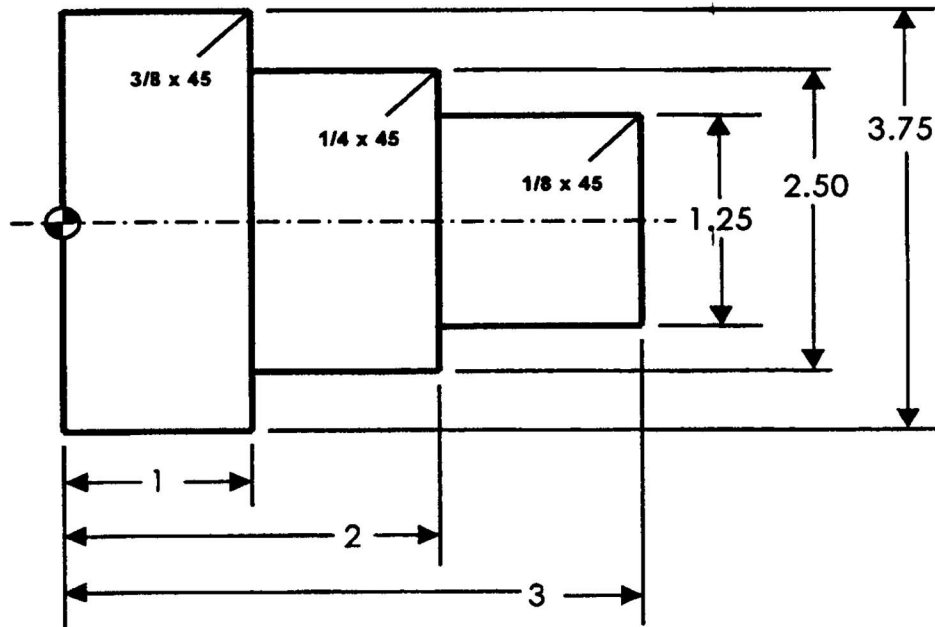
G75 is effective for LAP cycles.

G75 is effective under TNR function.

The following program is equally valid, yet contains fewer characters.

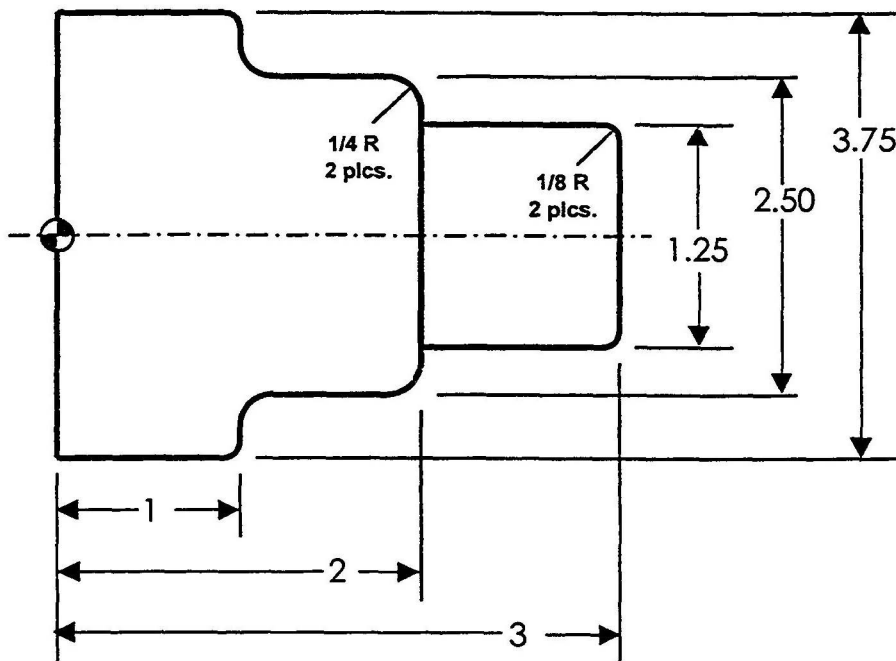
```
G50 S2000
G00 X50 Z50
X0Z3.1 T010101 G97 S1200 M03 M08
G01 Z3 F.011 G42
G75 X1.25 L.125
Z2
G75 X2.5 L.25
Z1
G75 X3.75 L.375
Z.1
G00 X50 Z50
M02
```

G75 Auto Chamfering – 45 Degree only – the following illustration shows how the OSP control can utilize a G75 command to allow simplified programming of either external or internal 45 degree chamfers.



G76 Radius Chamfering – 90 Degree – the following illustration shows how the OSP control can utilize a G76 command to allow simplified programming of either external or internal 90 degree radius chamfers.

FORMAT: G76 G01 X.. or..ZL.....F.....



G50 S200

G00 X50 Z50

X0Z3.1 T0101 G97 S120 M03 M08

G01 Z3 F.011

G76 X1.25 L.125

G01 Z2

G76 X2.5 L.25

G76 Z1 L.25

G76 X3.75 L.125

G01 Z.1

G00 X50 Z50

M02

← Max. Spindle RPM

← Rapid to Home/Index Position

← Rapid approach position, tool select

← Feed to face

← Feed up face, and .125 radius

← Feed to shoulder

← Feed up face, and .25 radius

← Feed left, .25 'fillet' radius, end on face

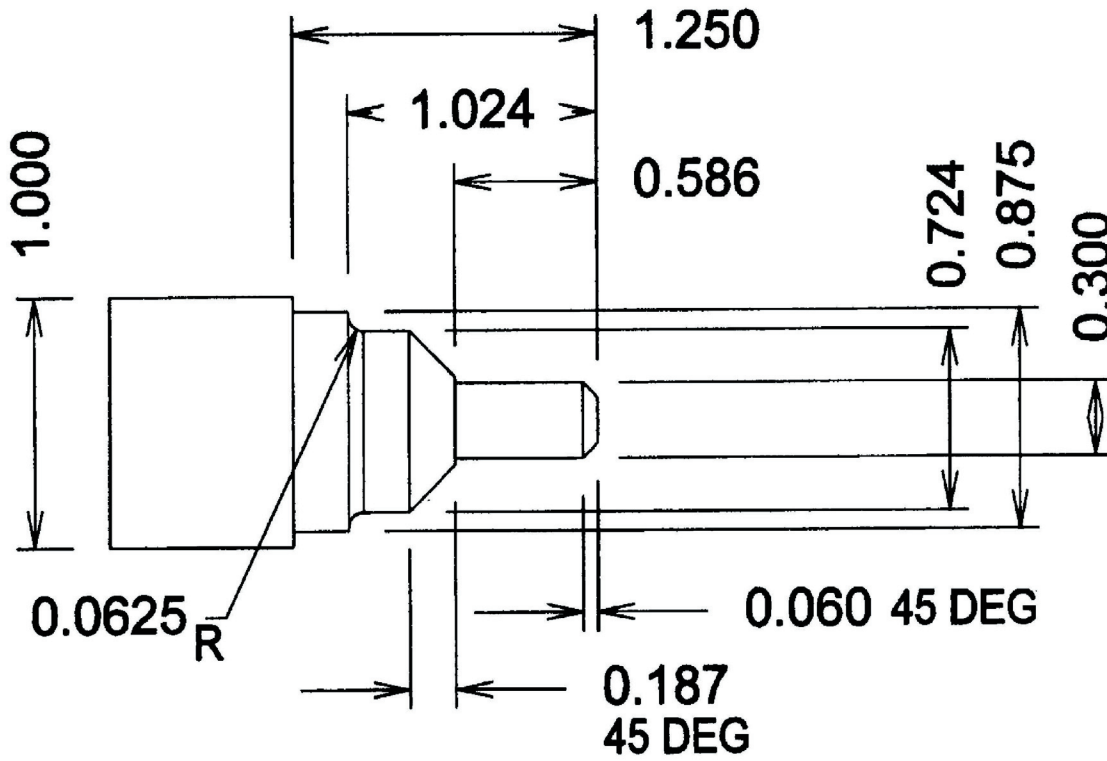
← Feed up face, and .125 radius

← Feed off

← Rapid to Home Position

← Program End

Program this part using G75 and G76



LAP CYCLES

LAP Cycle concept

Types of LAP Cycles

Write new program for DR201-3 and modify to use LAP cycles

LATHE AUTO-PROGRAMMING **FUNCTION (LAP3/LAP4)**

General Description:

LAP (Lathe Auto-Programming) makes full use of the control's high-speed processing capability. With this function, the control automatically generates a tool path to produce the required part contour.

In this function, dimension data of the final contour to be finished, including rough cut conditions, is prepared as the Contour Definition Program; when it is called out with the cutting conditions specified, the control automatically generates the tool path for the rough cut cycles and then finishes the work piece to the programmed dimensions.

This feature permits the programmer to complete the part program simply by picking up the dimensions specified in an engineering drawing, simplifying programming as well as reducing programming time.

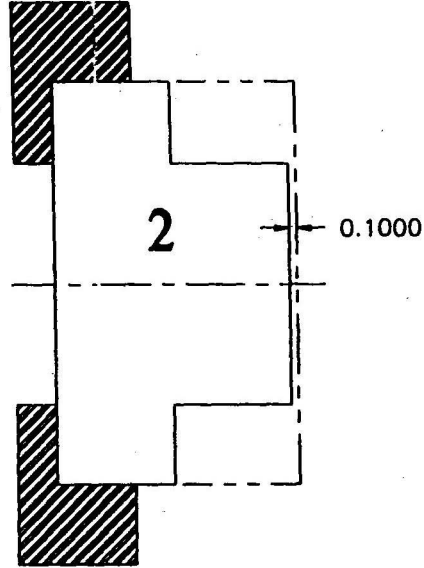
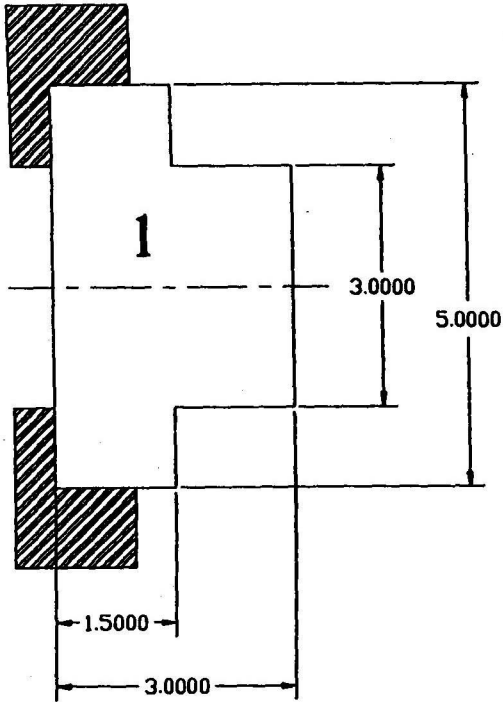
The cutting modes available with the LAP can cope with any type of cutting.

Features of LAP are:

1. No special programming language is needed. Normal programming techniques are used for LAP.
2. Programming preparation time can be greatly reduced.
3. Programming for the rough cut cycle can be eliminated, and this simplifies manual calculations required for programming.
4. Change of cutting conditions such as depth of cut and feed rate is possible during the rough cut cycle.
5. By entering the blank work piece shape, unnecessary air-cutting tool paths can be eliminated to improve cutting efficiency (LAP4).

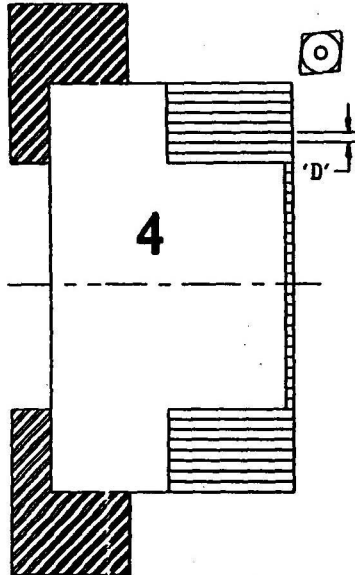
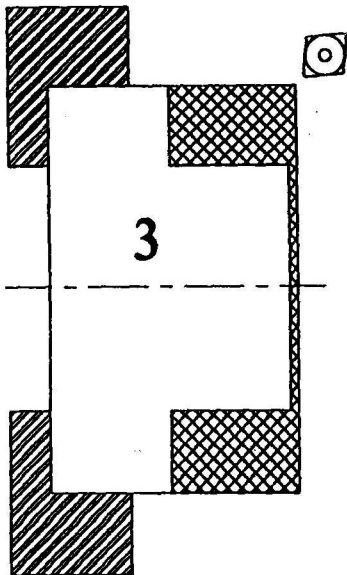
CLASSIFICATION OF LAP CYCLES

	<i>G81</i>	<i>G82</i>	<i>G80</i> END OF SHAPE
	LONGITUDINAL CUTTING	TRANSVERSE CUTTING	
<i>G85</i>			BAR STOCK
<i>G86</i>			COPY TURNING FOR FORGINGS AND CASTINGS
<i>G73</i>			THREADING
<i>G83</i>			HIGH SPEED COPY TURNING FOR FORGINGS AND CASTINGS
<i>G87</i>			FINISH TURNING



G85 Cycle

- 1. Finished Part
- 2. Extra Stock 'length'
- 3. Material to Remove
- 4. Cutting Tool Path



```

G50 S2000
G00 X50 Z50
X5.2 Z3.25 T0101 G97 S500
M03 M08
G96 S675
G85 NLAP D.4 F.016 U.05
W.005
NLAP G81
G00 X-.062 Z3.15
G01 Z3 F.008
X3
Z1.5
X5.1
G80
G97 S500
G00 X50 Z50
M01
X5.2 Z3.25 T0202
G96 S650
G87 NLAP
G00 X50 Z50 M09
M02
  
```

Parameters for LAP Cycles

Parameter	Description
D	Depth of cut in rough cut cycle 1) Alarm 2) $D > 0$
DA	Depth of cut after rough cut conditions change point A 1) $DA = D$ 2) $DA > 0$
DB	Depth of cut after rough cut conditions change point B 1) $DB = DA$ 2) $DB > 0$
FA	Feedrate after rough cut conditions change point A 1) $FA = F$ 2) $FA > 0$
FB	Feedrate after rough cut condition change point B 1) $FB = FA$ 2) $FB > 0$
E	Feedrate in rough cut cycle along finish contour 1) F active at entry of LAP mode 2) $E > 0$
XA	X coordinate of rough cut condition change point A 1) No change of cutting conditions 2) $ XA < 99999.999$
XB	X coordinate of rough cut condition change point B 1) No change of cutting condition change B 2) $ XB < 99999.999$
ZA	Z coordinate of rough cut condition change point A 1) No change of cutting conditions 2) $ ZA < 99999.999$
ZB	Z coordinate of rough cut condition change point B 1) No change of cutting condition at point B 2) $ ZB < 99999.999$
U	Stock removal amount in X-axis direction for finish cut cycle 1) $U = 0$ 2) $U > 0$
W	Stock removal amount in Z-axis direction for finish cut cycle 1) $W = 0$ 2) $W > 0$
H	Thread height in G88 thread cutting cycle 1) Alarm 2) $H > 0$
B	Tip point angle of thread cutting tool in G88 1) $B = 0$ 2) $0 < B < 180^\circ$

Note 1: The following words should be specified in incremental values.

D, DA, DB, U, W and H

Note 2: D, DA, DB, XA, XB, U and H words should be specified in diameter.

Note 3: In a thread cutting cycle using the M73 pattern, "H-U" must be greater than or
Equal to D: $H-U > D$

=

In the M74 and M75 patterns, it must be positive:

$H - U > 0$

Note 4: When more than one alphabetic character is used in succession, the control interprets the expression as a variable. Therefore, it is necessary to use a comma as a delimiter for extended address characters: DA=, DB=, FA=, FB=, XA=, XB=, ZA=, and ZB=

G85 – Bar Turning Cycle

[N0103] [G85] [NLAP1] [] [D] [F] [U] [W] [G84]

N0103	Sequence number
G85	G code calling out bar turning cycle. To be provided right after sequence number (name)
NLAP1	Sequence name in the first block of contour defining blocks
Blank	Enter either tab or space code
D	Depth of cut in rough cut cycle
F	Feedrate in rough cut cycle
U	Stock removal in finish cut cycle, X component
W	Stock removal in finish cut cycle, Z component
G84	Change of rough cut conditions

With the commands above, the control starts searching for the contour definition program beginning with the sequence name NLAP1. After assigning parameter data of D, F, U, W and G84 for NLAP1, the controls starts bar turning cycle.

Note 1: No S, T, or M code may be provided in the G85 block.

Note 2: D word is used to specify depth of cut in rough cut cycle. When the command indicating change of cutting conditions is provided, the D word is effective up to such point, XA and ZA.

D word which has a positive value must be provided in the G85 block if the D word is not positive or if it is omitted, an alarm results.

Note 3: F word is used to specify the feedrate in rough cut cycle. When the command indicating change of cutting conditions is provided, the F word is effective up to such point, XA and ZA.

Note 4: When U and/or W word is not provided, U and/or W is assumed "0". U and W words must be positive or zero.

Note 5: Finish feedrate must be specified in the finish contour definition.

G87 – Finish Cut Cycle

Format:

[N0203] [G87] [NLAP1] [] [U] [W]

N0203	Sequence number
G87	G code calling out finish cut cycle. To be provided right after the sequence Number (name)
NLAP1	Sequence name of the first block of contour defining blocks
Blank	
U	Stock removal in finish cut cycle, X component – Optional
W	Stock removal in finish cut cycle, Z component - Optional

With this command, the control starts searching for the contour definition program beginning with the sequence name NLAP1. After assigning parameter data U and W of NLAP1, the control starts the finish cut cycle.

Note 1: No S, T or M code may be provided in the G87 block.

Note 2: No feed rate command is used because the finishing feed rate is defined in the contour definition

If no F word is provided in the contour definition program the feed rate defaults to that of the previous block.

Note 3: When U and /or W word is not provided, U and /or W is assumed to be “0”. U and W words must be positive or zero. If not, an alarm results.

LAP Programming Precautions:

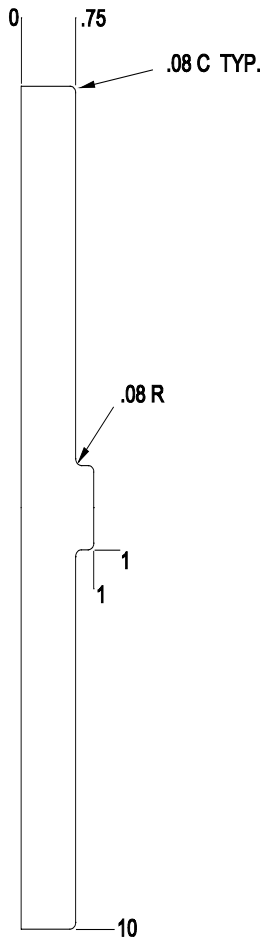
1. Be sure to include the contour defining sequence name or number right after the “G” code calling for execution of the LAP program: G85, G86, G87, G88.
2. The “G” codes (G81 and G82) are used to indicate the start of the LAP CONTOUR DEFINITION and must be programmed with a sequence name or number.
3. The selected mode, G90 absolute or G91 incremental that is active prior to the LAP CONTOUR DEFINITION stays in effect for the LAP mode. The mode can be changed within the LAP CONTOUR DEFINITION by programming G90 or G91.
4. If the “G” codes, G64, G65, G94, G95, G96, OR G97, are executed prior to using G85, G86, G87, or G88 they will stay active during the LAP cycle. Once active, however, they cannot be changed within the LAP CONTOUR DEFINITION.
5. If the “G” codes, G00, G01, G02, G03, G32, G33, G34, G35, G64, G65, G90, G91, G95, G96, OR G97, are executed prior to using G85, G86, G88, they become active after the completion of the LAP mode.
6. Nesting or branching from one LAP CONTOUR DEFINITION to another LAP CONTOUR DEFINITION is not allowed and will cause an alarm.
7. If a “G” code is programmed, calling for a LAP CONTOUR DEFINITION (G85, G86, G87, and G88), while tool nose radius compensation mode is active an alarm will occur.
8. Tool nose radius compensation can be programmed within the LAP CONTOUR DEFINITION. It must, however, be cancelled prior to the G80 block, which indicates the end of the LAP CONTOUR DEFINITION.

EXAMPLE:

```
NOD1 G82  
  
G0 G41 X_____ Z _____  
  
G1 Z_____  
  |  
  | (LAP contour definition)  
  
G40 X _____ K1  
  
G80
```

Questions

1. Why is part contour definition so important?
2. What “G” word is at the end of your part contour definition on a line by itself?
3. The first letter of the lap name must always be a _____.
4. What is the difference between G81 and G82 for establishing the lap cycle?
5. Could tool nose compensation be used within the lap contour definition?
6. Letter U and W are used to signify what?

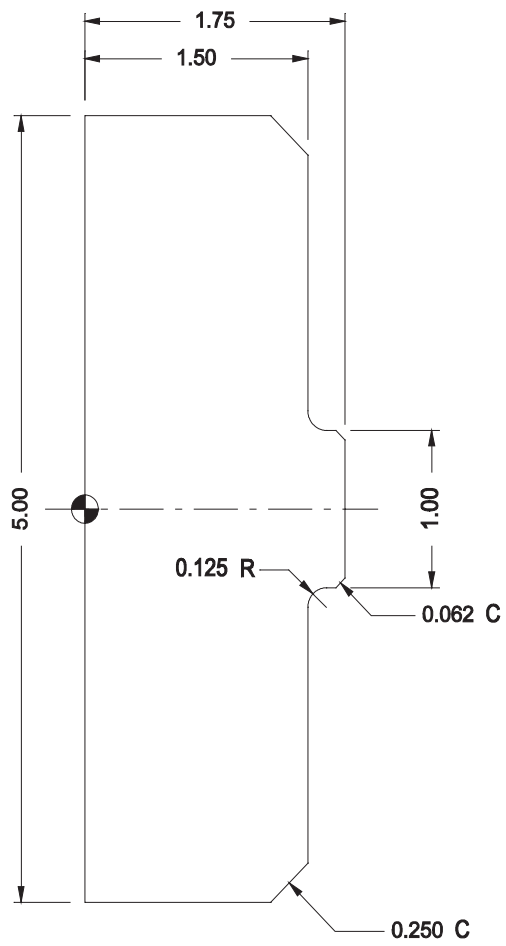


```

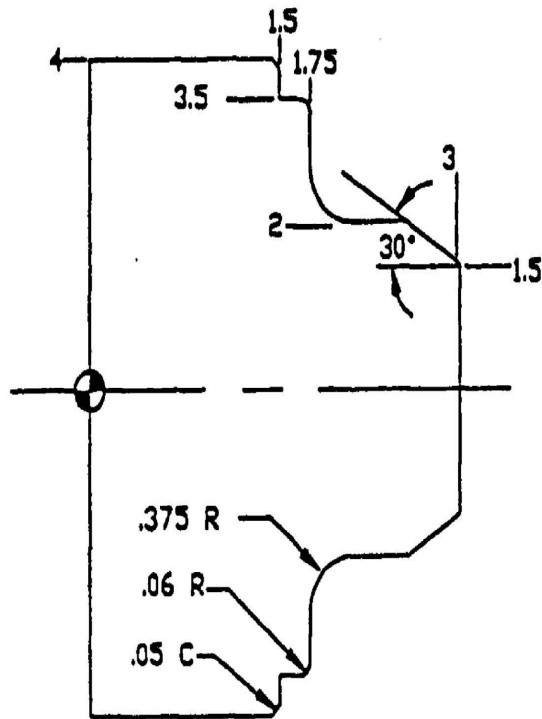
N1 G50 S3500
N2 G0 X25 Z25
N3 G96 X10.1 Z1.25 S500 T10101 M3 M41
N4 G85 NFCE D.12 U.010 W.005 F.01
N5 G0 X 25 Z25
N6 S1000 T20202 M42
N7 G87 NFCE
N8 G0 X25 Z25
NFCE G82
N10 G0 G41 X10.1 Z.5
N11 G1 X10 F.008
N12 G75 Z.75 L.08
N13 G76 X1 L.08
N14 G75 Z1 L.08
N15 X-.03
N16 G40 Z1.2 I1
N17 G80
N18 M2

```

Practice Programming



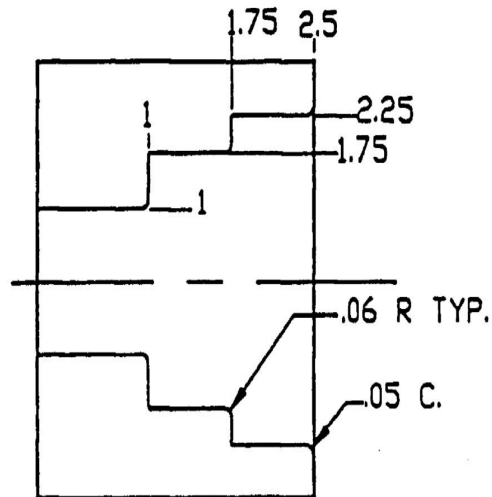
PROGRAMMING EXAMPLE:
BAR TURNING WITH LAP CYCLE



```

G50 S4000
G0 X25 Z25
N11 X4 Z3.1 F.018 S500 T30303 M3 M41 M8
N12 G96 S1200
N13 G85 NOD1 D.3 U.03 W.005
N14 G97 S500
N15 G0 X25 Z25
N16 T50505 S1000 M3
N17 X4 Z3.1
N18 G96 S1500
N19 G87 NOD1
N20 G0 X25 Z25 M5 M9
NOD1 G81
N1 G0 G42 X1 Z3.1 F.01
N2 G1 Z3
N3 X1.5
N4 X2 A-30
N5 G76 Z1.75 L.375
N6 G76 X3.5 L-.06
N7 Z1.5
N8 G75 X4 L-.05
    Z1.4
N9 G40 X4.2 K-1
N10 G80
N21 M2
    
```


PROGRAMMING EXAMPLE
BAR TURNING WITH LAP CYCLES



```

DEF WORK
PS LC,[0,0],[25,30]
PS LC,[0,0],[25,71],0
END
DRAW
N1 G50 S3500
N2 G0 X25 Z25
N3 X.75 Z2.6 F.015 S134 T20202 M3 M41
N4 G85 NID1 D.2 U.03 W.005
N5 G0 X25 Z25
N6 S1890 T50505
N7 G87 NID1
N8 G0 Z2.75
N9 X25 Z25
→ N10 NID1 G81
N10 G0 G41 X2.5 Z2.6
N11 G1 Z2.5 F.008
N12 G75 X2.25 L.05
N13 Z1.75 E.01
N14 G76 X1.75 L.06 E.01
N15 Z1
N16 G76 X1 L.06
N17 Z-.1
N18 G40 X.9
→ N19 G80
N20 M2

```

G84 - Change of Cutting Conditions in Bar Turning Cycle

FORMAT: N..... G85 N.....

\$	G84	XA=(ZA=)	DA=	FA=
\$		XB=(ZB=)	DB=	FB=
(1)		(2)	(3)	(4)

1. Indicates that the commands is a continuation from a previous command line. (must be specified at the beginning of the block.)
2. Specifies the point where the cutting conditions are changed.
3. Depth of cut after cutting condition point.
4. Feed rate after cutting condition change point.

The commands should be programmed in the block containing the G85 code that calls out the bar turning cycle. Since the number of characters in one line will be very large if these commands are specified in the same line with the G85 command, the G84 command can be continue on the next line. The continuation line must begin with "\$" which indicates that the commands in these lines belong on the same block.

With these commands, cutting conditions can be changed at desired point(s) during rough cut cycle. If a change of cutting conditions is not necessary, omit them.

Note 1: G84 and following commands must be provided "N...G85 N...".

Note 2: For OD turning, coordinate values of the "LAP starting point", "rough cut condition change point "A" must be larger than "rough cut condition change point B" must be provided so that they become smaller in this order. For ID turning, point "B" should be larger point "A".

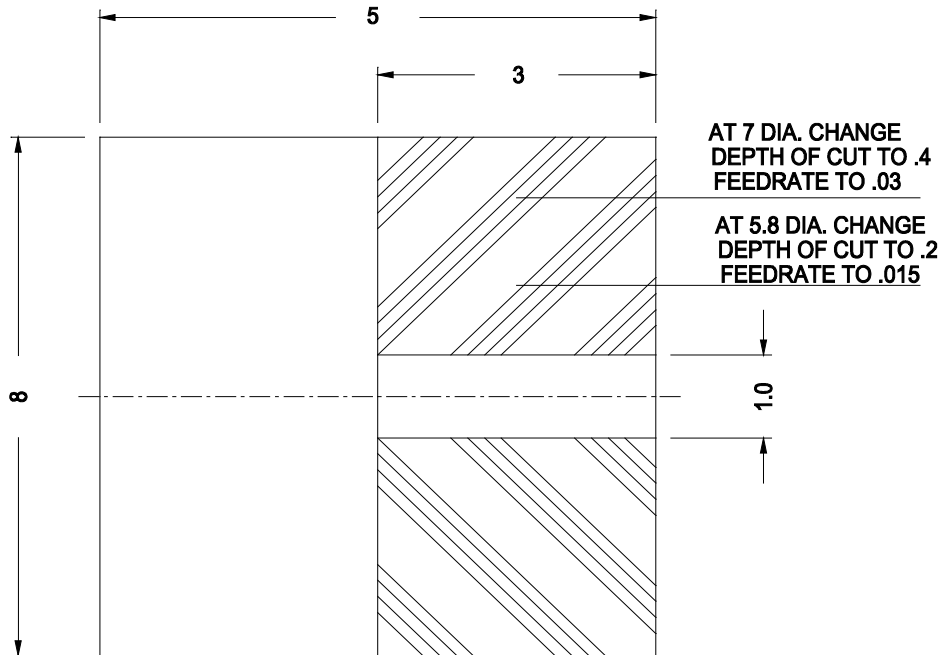
Note 3: If rough cut condition change points A and B both exist within a range where the cut is made in the current position, the cycle is performed with D provided the current position is outside XA, and with DA when the current position is on XA.

Note 4: If tool path exceeds XA when cutting cycle is performed with the depth of cut D from the current position, the cycle is performed with D provided the current position is outside XA and with DA when the current position is on XA.

Note 5: In longitudinal cutting, ZA = and ZB - commands may not be provided. In transverse cutting, XA = and XB = commands may not be provided.

PROGRAMMING EXAMPLE: G84 – 1

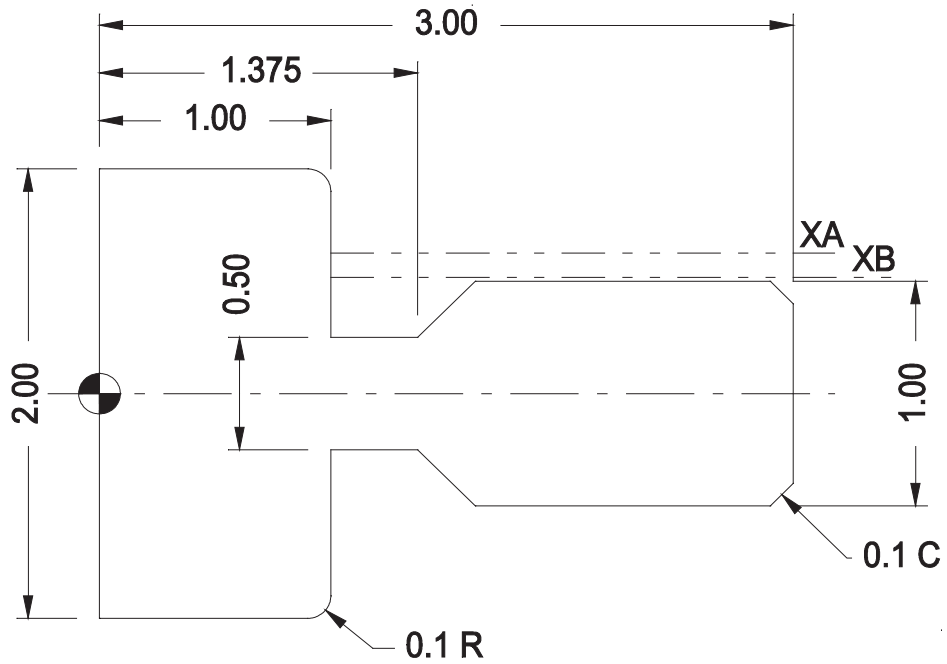
USING G85, G81 AND G84



The LAP program, to remove the material indicated in the cross-hatches areas, would be as follows:

```
N1 G50 S3000
N2 G00 X20. Z20.
N3 X8. Z.Z1 T010101 G96 S600 M03 M08 M42
N4 G85 NTRY1 D.2 U.010 W.005F.015
$ G84 XA = 7. DA = .4 FA = .030
$   XB=5.8 DB=.2 FB=.015
NTRY1 G81
G00 X1. Z.1
G01 Z-3. F.008
X8.1
G80
G87 NTRY1
N5 G00 X20. Z20.
N6 M02
```

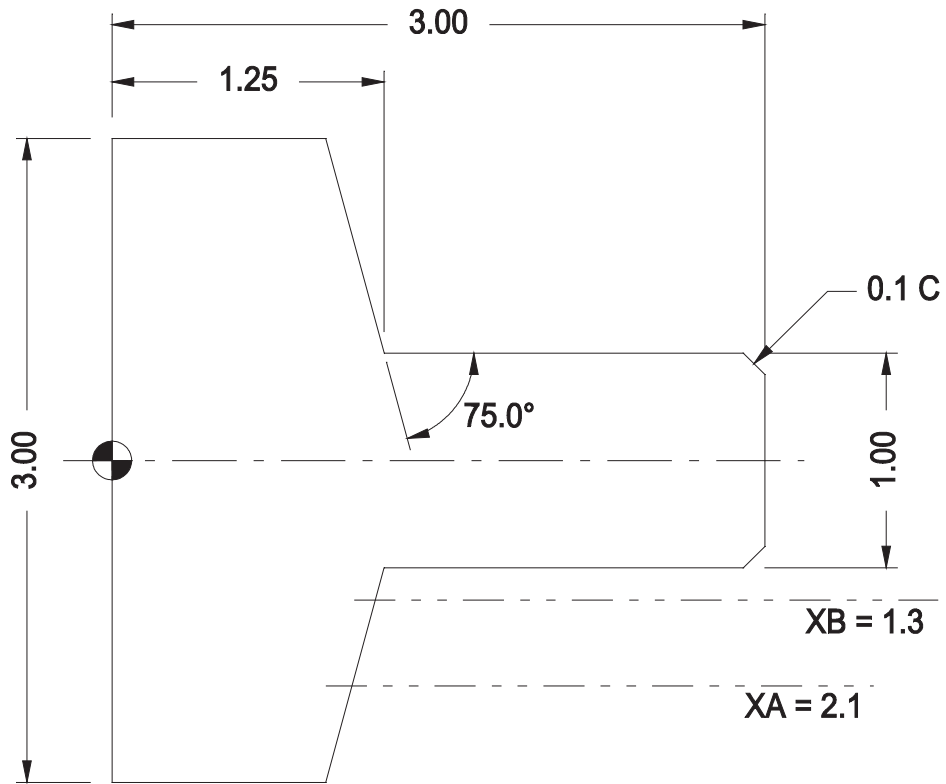
PROGRAMMING EXAMPLE
LAP CYCLE WITH CHANGE OF CUTTING CONDITIONS



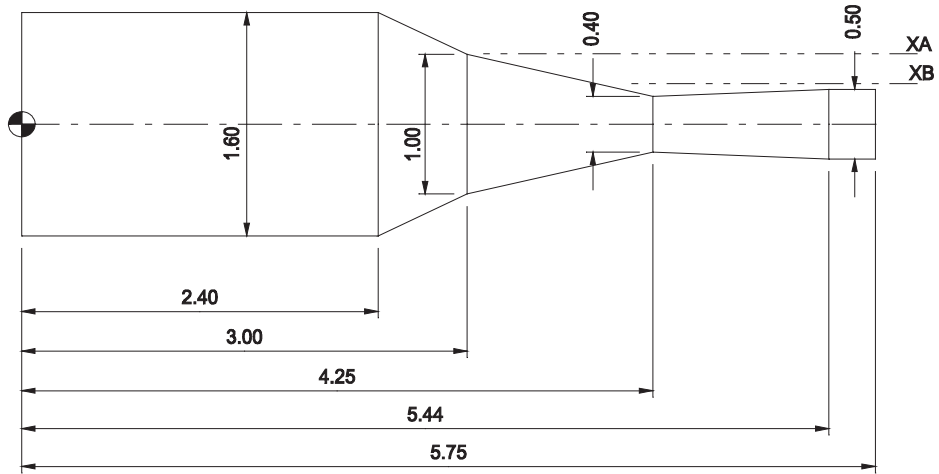
```

N1 G50 S4200
N2 G0 X25 Z25
N3 G96 X2 Z3.1 F. 015 S500 T90909 M3 M42
→ N4 G85 NOD2 D.2 U.0
→ $ G84 XA = 1.25 DA=.15 FA=.012
→ $ XB = 1.01 DB=.1 FB=.008
N5 G0 X25 Z25
N6 S1000 T111111 M42
N7 G87 NOD2
N8 G0 X25 Z25
NOD2 G81
N9 G0 G42 X.7 Z3.1
N10 G1 Z3 F.007
N11 G75 X1 L.1
N12 G75 Z1.375 L .25
N13 Z1
N14 G76 X2 L .1
N15 G40 X2.1 K 1
N16 G80
N17 M2
    
```

PROGRAMMING PRACTICE
PROGRAM USING G84



PROGRAMMING PRACTICE



G86 – Copy Turning

FORMAT

[N0123] [G86] [NLAP2] [] [D] [F] [U] [W]

N0123 Sequence number

G86 G code which calls out copy turning cycle. It is located to be right after the sequence number (name)

NLAP2 Sequence name in the first block of contour defining blocks

Blank

D Depth of cut

F Feed rate

U Stock removal in finish cut cycle, X component

W Stock removal in finish cut cycle, Z component

With this command, the control starts searching for the contour definition program beginning with the sequence name NLAP2. After assigning parameter data of D, F, U, and W of NLAP2, the control starts copy turning cycle.

Note 1: No S, T or M code can be used in the G86 block.

Note 2: The D word is used to specify depth of cut in each cycle and must be provided in the G86.

The D word value must be positive. If not, an alarm results.

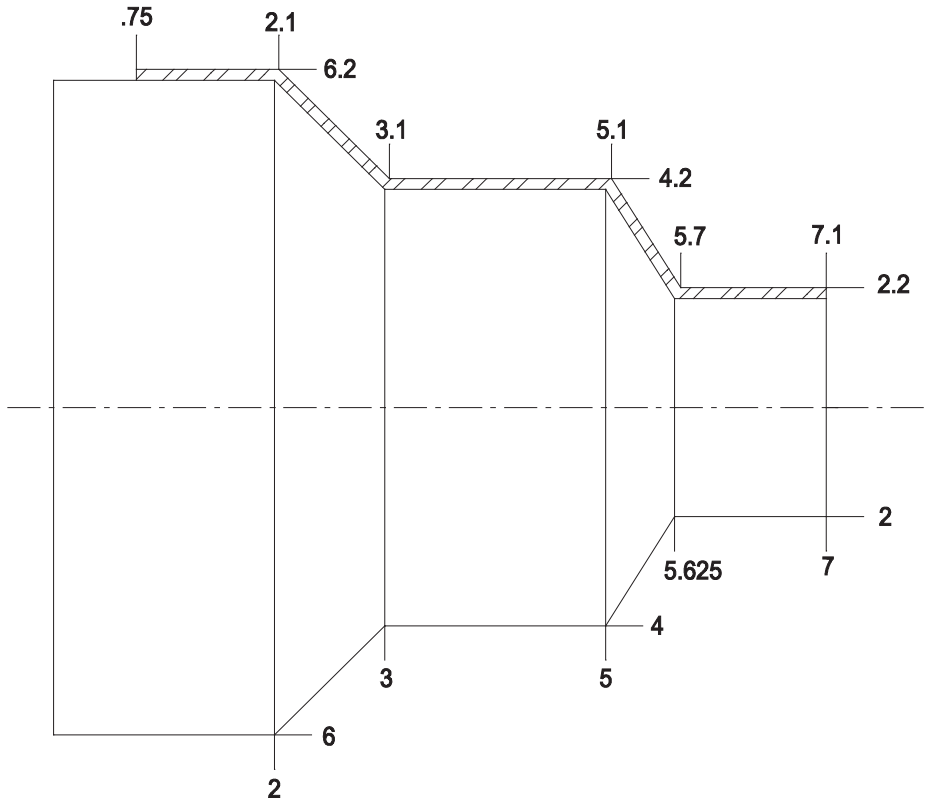
Note 3: The F word specifies the feed rate for the blocks until an E word is provided in the contour definition program.

If no F word is provided in the G86 block, the feed rate effective before the execution of the G86 block is used.

The F word must be positive. If not, an alarm results.

Note 4: When U and /or W words are not provided, U and /or W is assumed to be “0”. The U and W words must be positive or zero. If not, an alarm results.

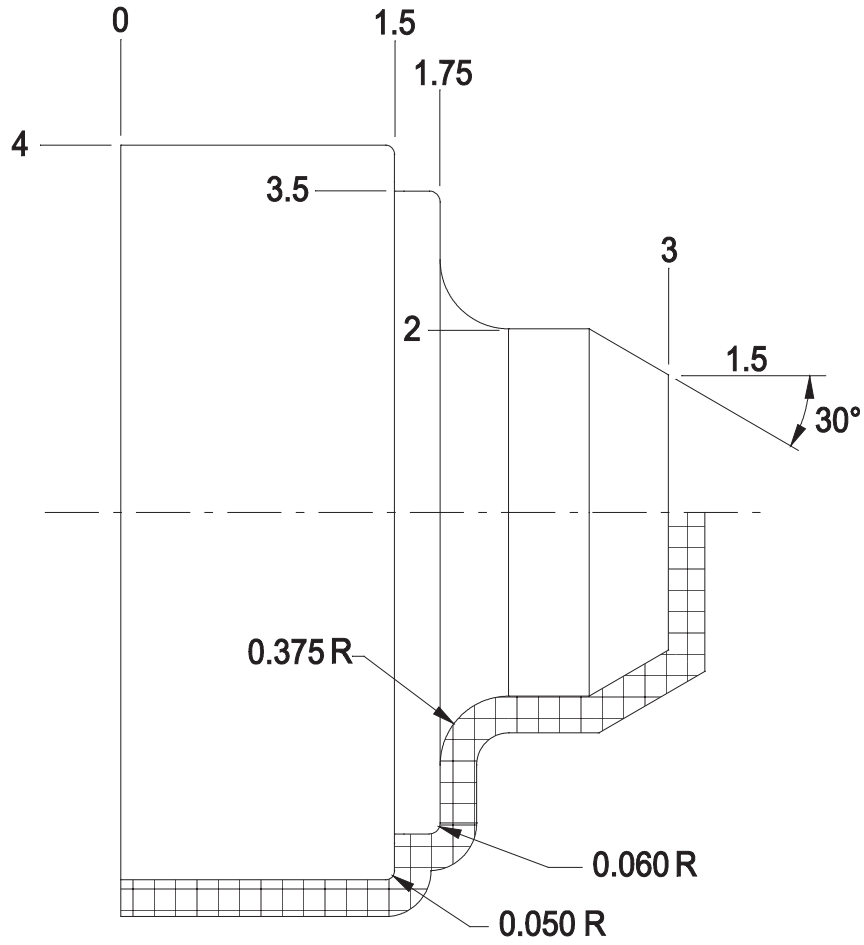
LAP 4 PROGRAMMING EXAMPLE



```

N10 G85 NG83 D.1 F.015 U.01 W.005
NG83 G83
G1 X2.2 Z7.1
Z5.7
X4.2 Z5.1
Z3.1
X6.2 Z2.1
Z.6
NG81 G81
G42 X2 Z7.1
Z5.625
X6 Z2
Z.6
G40 X6.2 K-1
G80
    
```


PROGRAMMING PRACTICE
PROGRAM USING G85 FOR FACE
G86 FOR OD



REVIEW

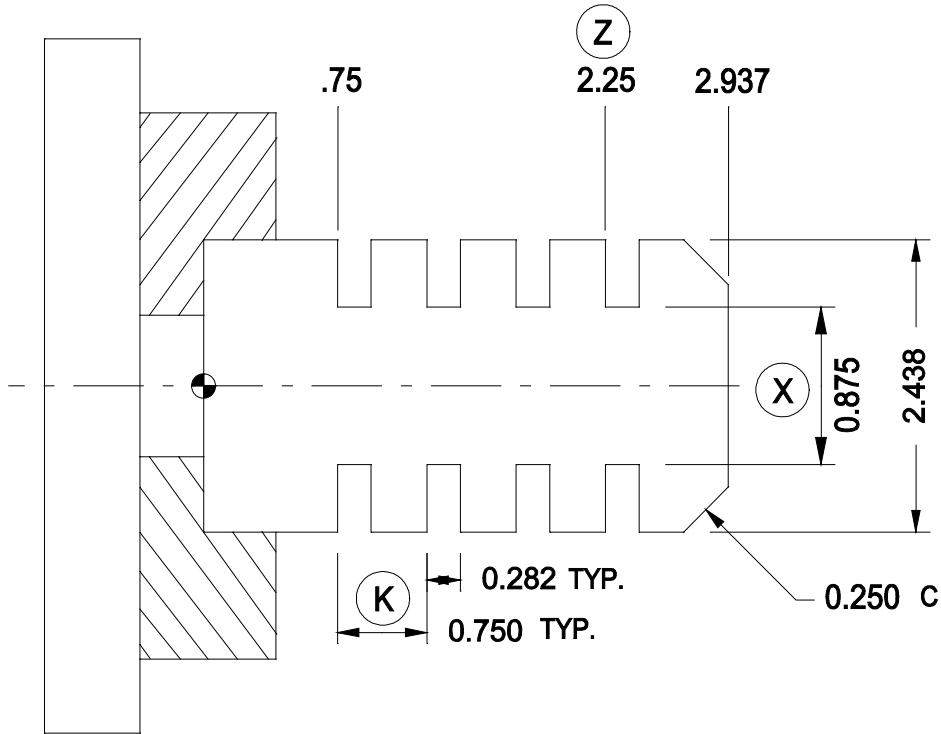
MISCELLANEOUS CYCLES

**Drilling Cycles
Grooving Cycles
Tapping Cycles**

Programming Practice: DR607-2 G73 using "K" for multiple grooves

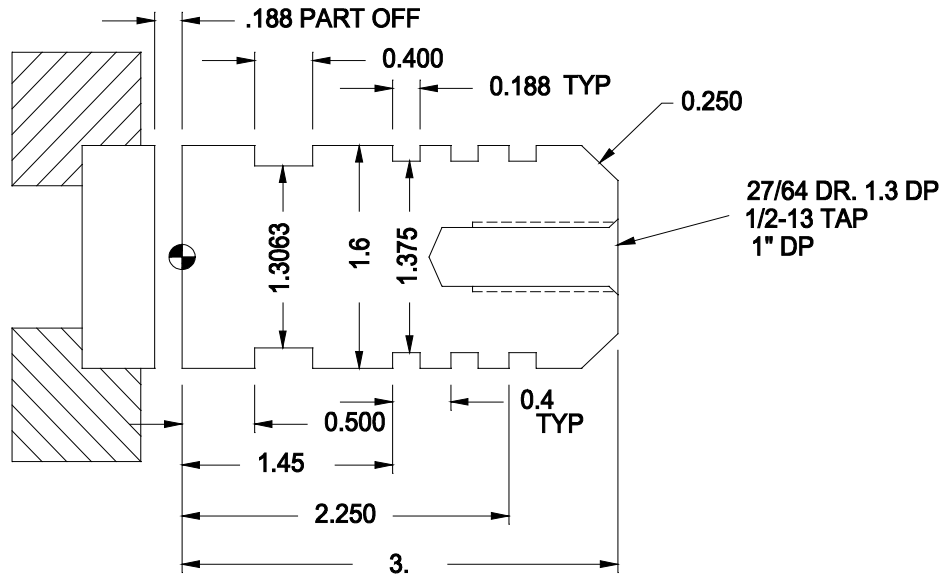
When programming multiple grooves of equal widths
to the tool being used:

- K = pitch of the grooves
- Z = chuck side of the last groove
- T word is not used



Note: All grooves are equally spaced and the groove width is equal to the tool width.

Programming Practice



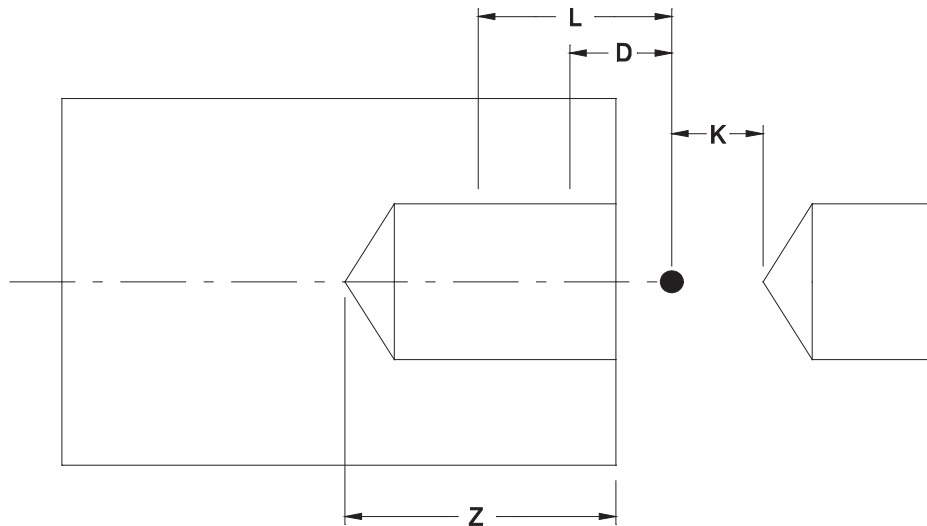
Operations:

1. Face to Length
2. Turn 1.6 OD
3. Drill & Tap
4. Groove
5. Part Off

G74 DRILLING FIXED CYCLE

[G74] [X0] [Z] [D] [K] [L] [F] [E]

*OPTIONAL



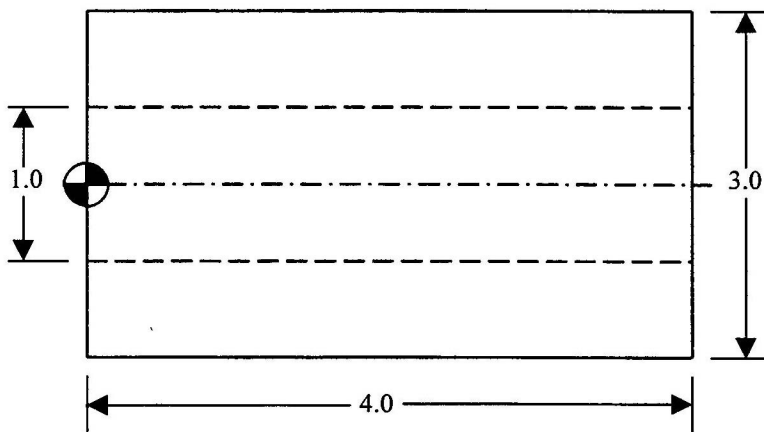
This example is to show the drill program, to position, drill to depth, and rapid retract back out past the start position point.

Definitions:

For G74 line of programming

- X X-axis diameter coordinate must always be zero. It must be programmed, even if the line above it specifies the "X" coordinate as zero.
- Z Z-axis coordinate to final drill depth from program zero
- D is used to "peck" feed to the "Z" target depth. Amount of "peck" is set to the input value of "D". The "peck" will start from the approach "Z" value or from the "K" value. After each "peck" tool will withdraw the value of the "PARAMETER SETTING" then feed back the distance of the PARAMETER SETTING + one "peck") until the "Z" target depth is reached.
- K The "K" word is used to advance the tool from the Z-axis rapid position dimension to any safe distance from the work piece at a rapid feed rate. This is used to reduce "air cutting" time at the start of drilling. This value is not doubled but must be incremental.
- L Used to trigger full retract of tool past the start position point for chip removal. This function is controlled by the "D" word, in that, if "L" is larger then "D" several "Ds" will occur until "L" is reached at which point the tool will retract. If "L" is smaller than "D" the tool will retract fully every time.
- E Dwell at hole button

PROGRAMMED EXAMPLE
DRILLING G74

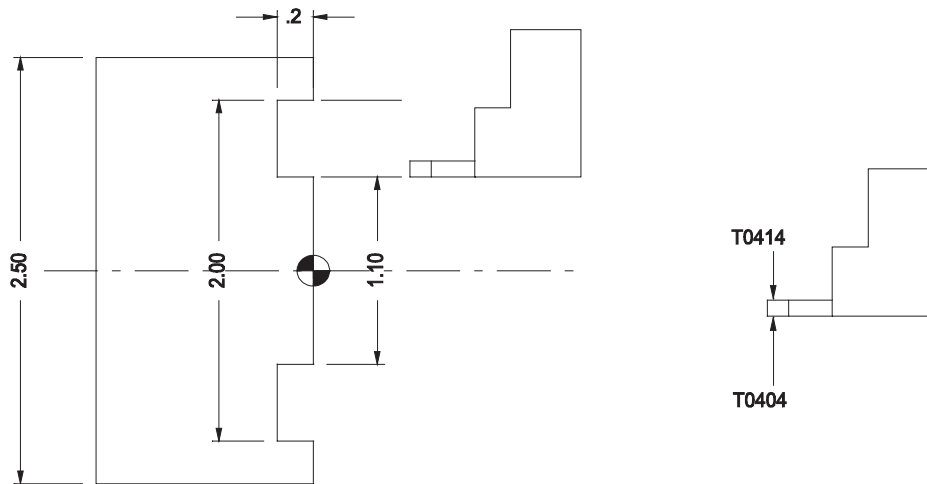


```
G50 S3500  
G0 X25 Z25  
Z4.1 S760 T505 M3 M41  
X0  
G74 X0 Z-.4 D.6 L1 F.008  
G0 X25  
Z25  
M2
```

G74 - Face Grooving Cycle

G74 X__ Z__ K__ I__ D__ L__ F__ E__ T__

- X X-axis diameter coordinate to end-point of groove (from program zero)
- Z Z-axis coordinate to bottom of groove (from program zero)
- K Rapid advance to the work piece from the Z-axis start position.
- I Cutter shift amount (plus) on X-axis must be entered as diametrical value.
- D Depth of cut for "peck" feed into work piece
- L Full tool retract
- F Feed rate expressed as either IPM or IPR
- E Dwell expressed in seconds at stroke end (bottom of groove).
- T New tool offset for end-point of groove tool.



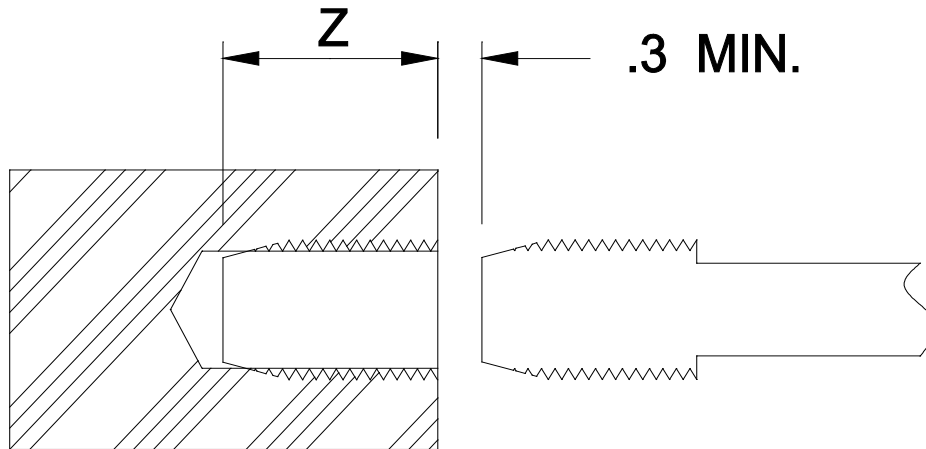
```
G50 S3000
G00 X20. Z20
X1.1 Z.1 G97 S400 T0404 M03 M08 M41
G74 X2. Z-0.2 K.09 I.15 D.05 L1.F.015 E2 T0414
G00 X20 Z20
M02
```

Notes:

1. The amount to shift the tool over for each successive plunge is based on experience. The general rule of thumb is 75% of width of tool.
2. The cycle can proceed from outside to inside by initially positioning with T0414 and using T0404 within the G74 line.

G77/G78 TAPPING

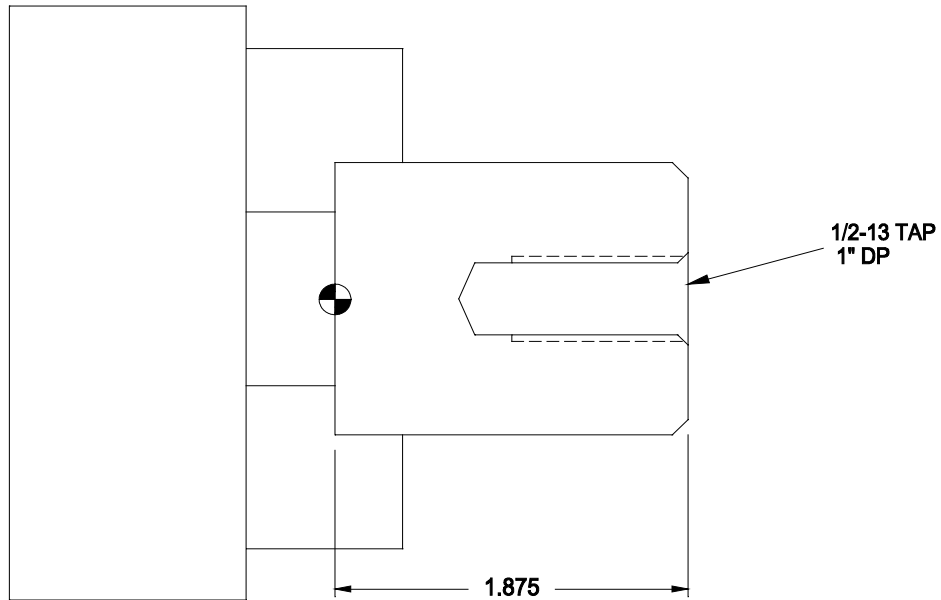
G77 X0 Z___ F___ (K___)



Definitions:

- X Allows 0, must always be on G77 line
 - Z End point coordinate of tap depth.
 - K Air cut reduction distance from start point, incremental value. Do not start too close to the hole, distance must be kept for slide acceleration.
 - F Feed rate
- F = 1/Threads Per Inch

PROGRAMMING PRACTICE
TAPPING G77 & G78



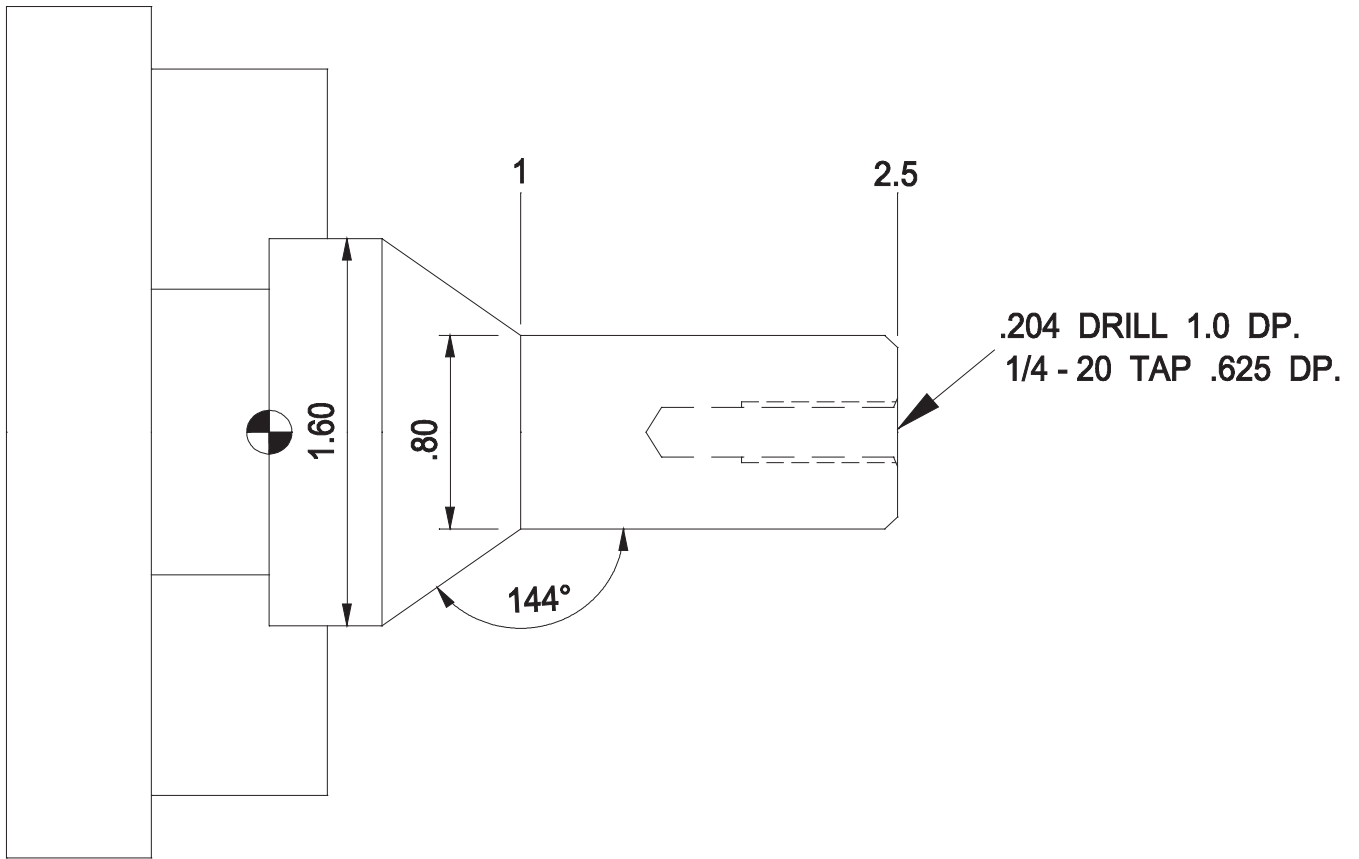
G77 RH TAP

```
G50 S350  
N1 G0 X25 Z25  
N2 X- Z2.3 S500 T404 M3 M41  
N3 G77 X0 Z.875 F1.13 K.1  
N4 G0 X25 Z25  
N5 M2
```

G78 LH TAP

```
G50 S350  
N1 G0 X25 Z25  
N2 X0 Z2.3 S500 T404 M4 M41  
N3 G78 X0 Z.875 F1.13 K.1  
N4 G0 X25 Z25
```

PROGRAMMING PRACTICE



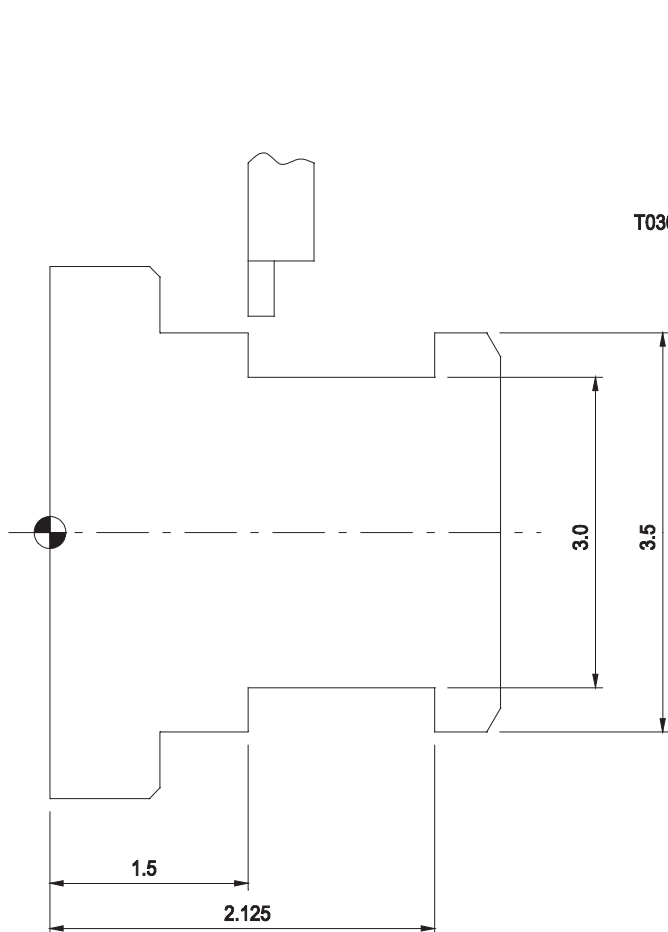
INSTRUCTIONS:

1. FACE
2. TURN OD
3. DRILL
4. TAP

G73 LONGITUDINAL GROOVING - FIXED CYCLE

G73 X__ Z__ I__ K__ D__ L__ F__ E__ T__

X	X-axis coordinate (groove bottom)
Z	Z-axis coordinate on right wall of groove
I	X axis cutter shift advance to the workpiece - rapid feed
K	Z axis "stepover" amount
D	Depth of infeed
L	Depth for Full retraction
F	Feedrate
E	Dwell - at bottom
T	Grooving tool second offset



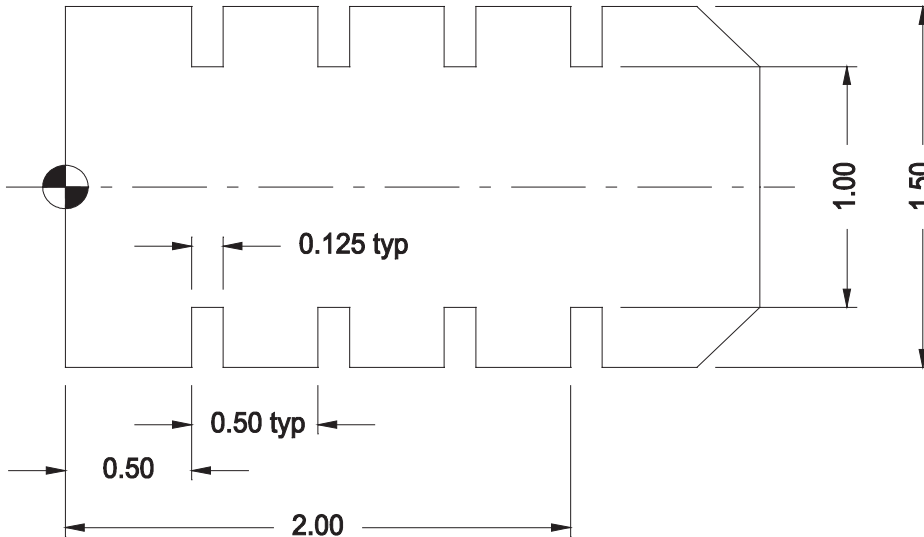
```
G50 S3500
G00 X25 Z25
X3.6 Z1.6 S475 T0303 M03 M41
G73 X3 Z2.125 K.2 D.15 L.3 F.006 E.8
T13
G00 X25 Z25
M02
```

Notes:

1. The amount to shift the tool over for each successive plunge is based on users experience. The general rule of thumb is 75% of width of tool.
2. Alternately, the cycle can proceed from right to left by initial positioning with T0313 and using T03 within the G73 line.

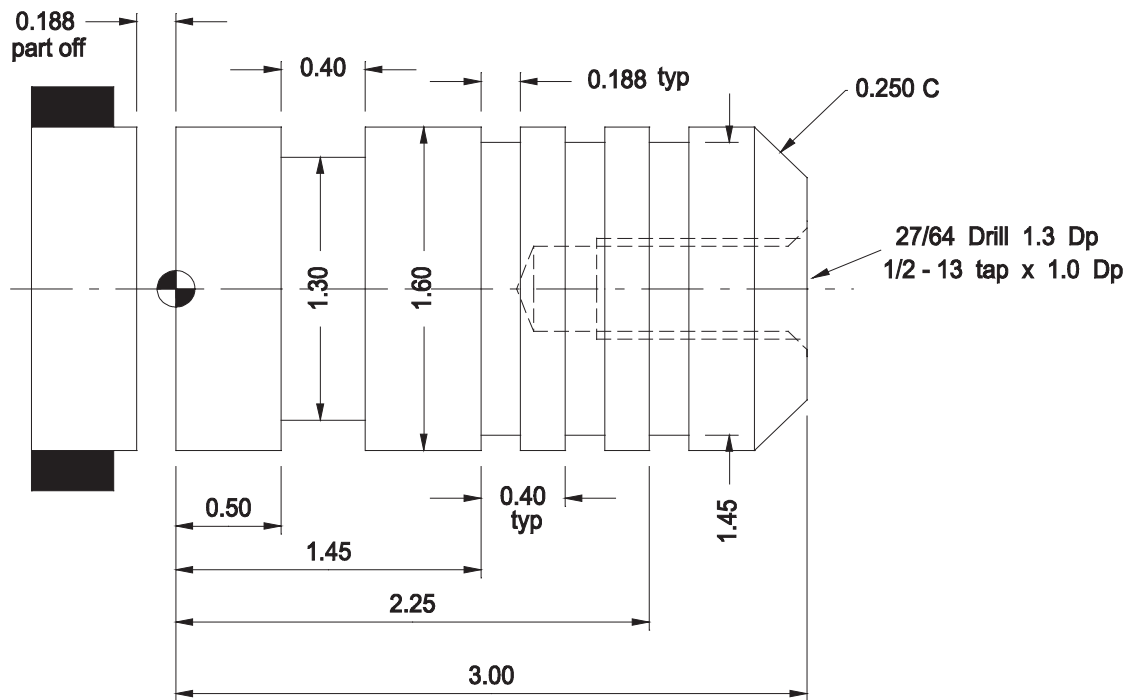
Using G73 for Multiple Grooves - Fixed pitch

The following example shows how to use the G73 cycle to produce grooves along a diameter at a uniform spacing. Please be aware that the groove width will be the width of the insert and therefore insert tolerance could be a problem if trying to hold a groove width. The secondary offset corner is not used - therefore it is critical that the Z dimension be correct. Essentially, the cycle uses the "shift amount" (K word) as equaling the desired pitch of the grooves.



```
G50 S4500  
G00 X50 Z50  
X1.6 Z.5 T0303 G97 S1000 M42 M03 M08  
G73 X1 Z2 K.5 D.12 L.24 F.004 E1  
G00 X50 Z50  
M02
```

PROGRAMMING PRACTICE



OPERATIONS:

1. FACE TO LENGTH
2. TURN 1.6 OD
3. DRILL & TAP
4. GROOVE
5. PART OFF

THREADING FIXED CYCLED

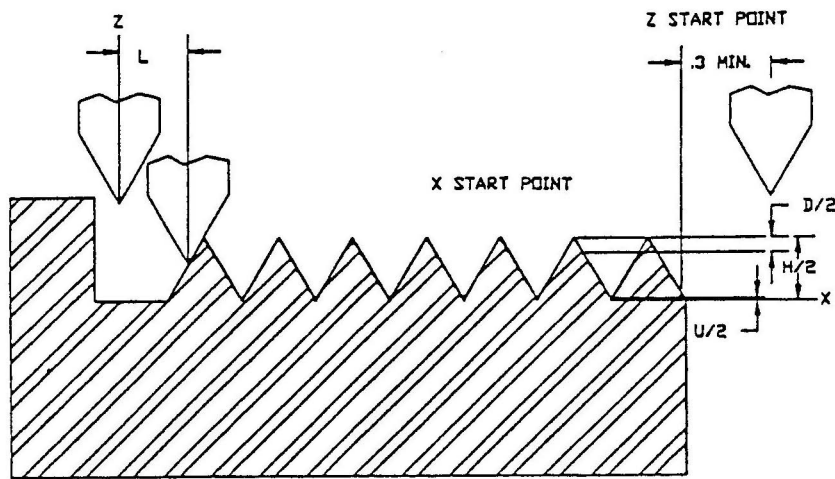
GENERAL DESCRIPTION

Thread cutting requires several passes, ordinarily requiring many blocks of programs. By using the threading fixed cycle we can take many passes with only one line of program.

Two modes of thread cutting cycles are available, G71 longitudinal thread cutting and G72 transverse (end face) thread cutting cycle. In addition, combination of "M" codes, one designating cutting mode and another selecting the proper infeed pattern, permits the programmers to select the most desirable mode of thread cutting from NINE available types of thread cutting cycles.

G71 - Longitudinal Thread Cutting Cycle (fixed cycle)

```
[G71] [X___] [Z___] [B60] [D___] [F1] [J___] [H___] [(U___)] [A___] [(I___)] [(E1___)] [(Q___0]
[(L___)]
```



X Coordinate value (dia.) for final diameter of thread.

Minor diameter for O/D threads
Major diameter for I/D threads

Z Coordinate value of thread end-point in Z-axis direction.

A Tapered thread angle, measured from the positive Z-axis zero or three o'clock position. A counter-clockwise value is plus and a clockwise value is negative.

Note: Do not use if using "I" command.

I Incremental radial difference between start point and end point of thread, the taper thread cutting.

"+" Increasing taper
"-" Decreasing taper

Note: Do not use if using the "A" command

B Cutter tip point angle control will assume zero if not specified

D Depth of cut for the first thread cutting cycle. This value must be expressed as a diametrical value, this means that if a .010 depth of cut (one side) is desired, the value must be entered as twice that amount or .020 depth of cut.

U Depth of cut for an extra threading pass. This value must be expressed as a diametrical value, this means that the threading pass prior to the last or finish pass will be controlled by this value and if a .001 depth of cut is desired, the value must be entered as twice that

amount or .002 depth of cut. If no "U" value is provided, the control will not perform an extra pass.

- H Thread height, expressed as the difference between the MAJOR and MINOR thread diameters. Must be expressed as a diametrical value.
- L Chamfering or pull-out distance at end of thread. When no "L" word is provided the control will assume that "L" is equal to one thread lead, specified at start of thread by the "F" lead word. The "L" condition can only be active when M23 is added to the G71 line of programming.
- E Lead variation per lead for cutting a variable lead thread,
- F Thread lead, or, (1/number of threads per inch). If "J" word is used "F" value should be one.
- J Is equal to number of threads per inch.

Example:

1-20 Thread would be	F1.J20
3/4-16 Thread would be	F1.J16
1/2-13 Thread would be	F1.J13

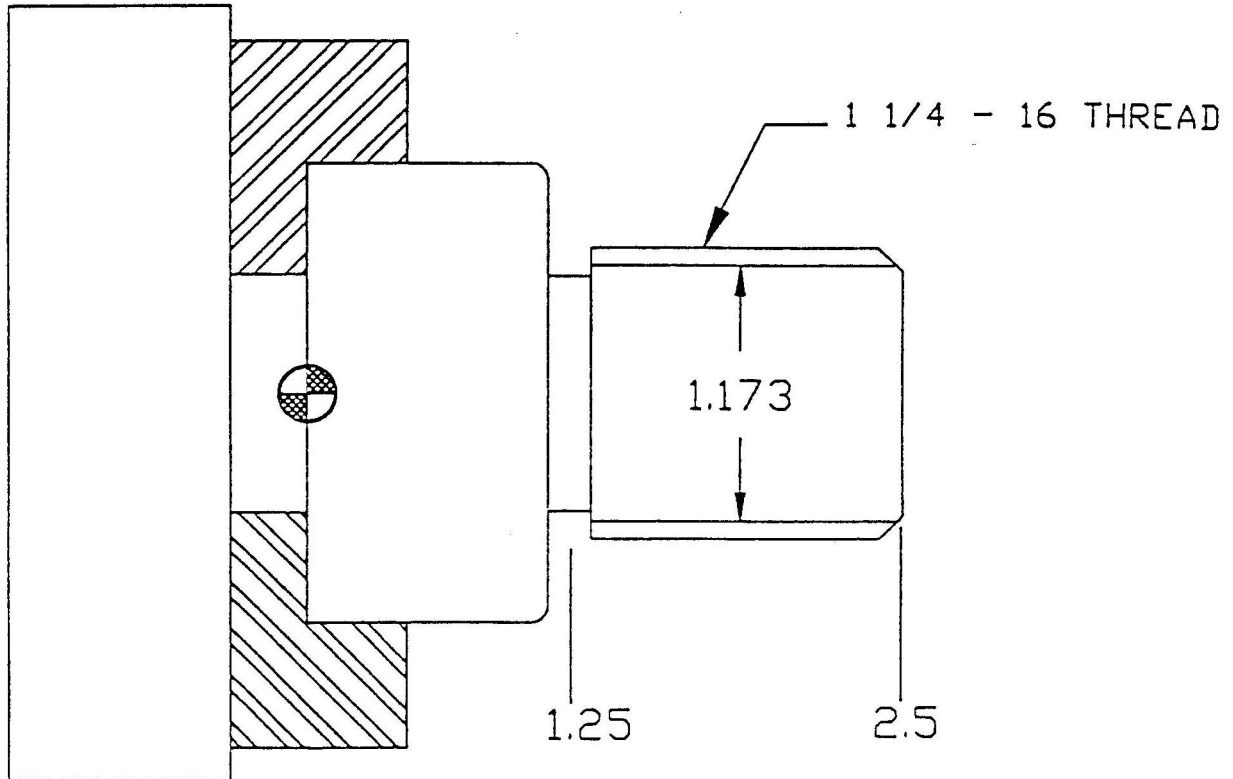
- M Used to select mode of cutting infeed (M32, M33, M34) or used to select thread cutting depths of cut (M73, M74, M75)
- Q Used to program multiple start threading

Notes: Cannot exceed 276 IPM when in threading cycle. The formula to calculate IPM is

$$\text{IPM} = ((1/\text{NUMBER OF THREADS PER INCH}) * \text{RPM})$$

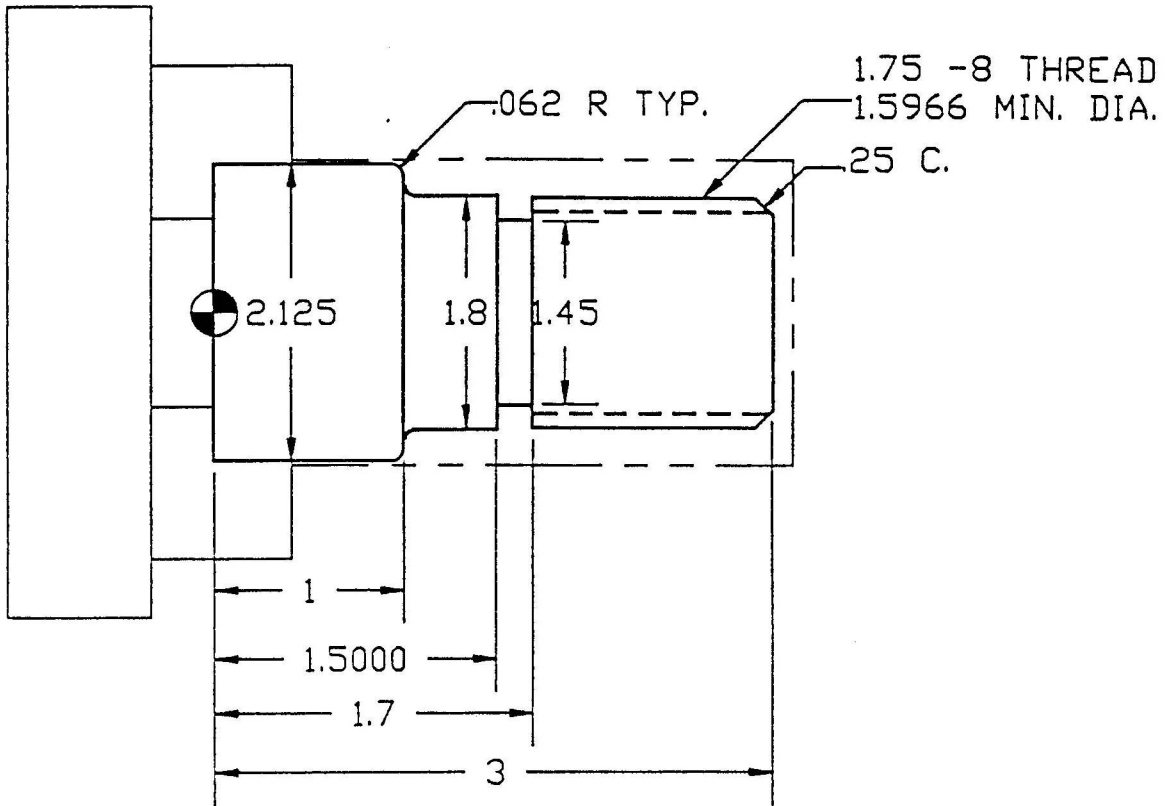
If no "M" codes are selected at cycle definition, the control will default to M32 and M73.

PROGRAMMING EXAMPLE



```
G50 S3500  
N1 G0 X25 Z25  
N2 X1.4 Z2.9 S1200 T606 M3 M41  
N3 G71 X1.173 Z1.25 B60 D.01  
$ U.002 H.077 F1 J16 (M32 M73)  
N4 G0 X25 Z25  
N5 M2
```


Programming Practice



ROUGH STOCK IS 2.25 DIA.
3.125 LONG

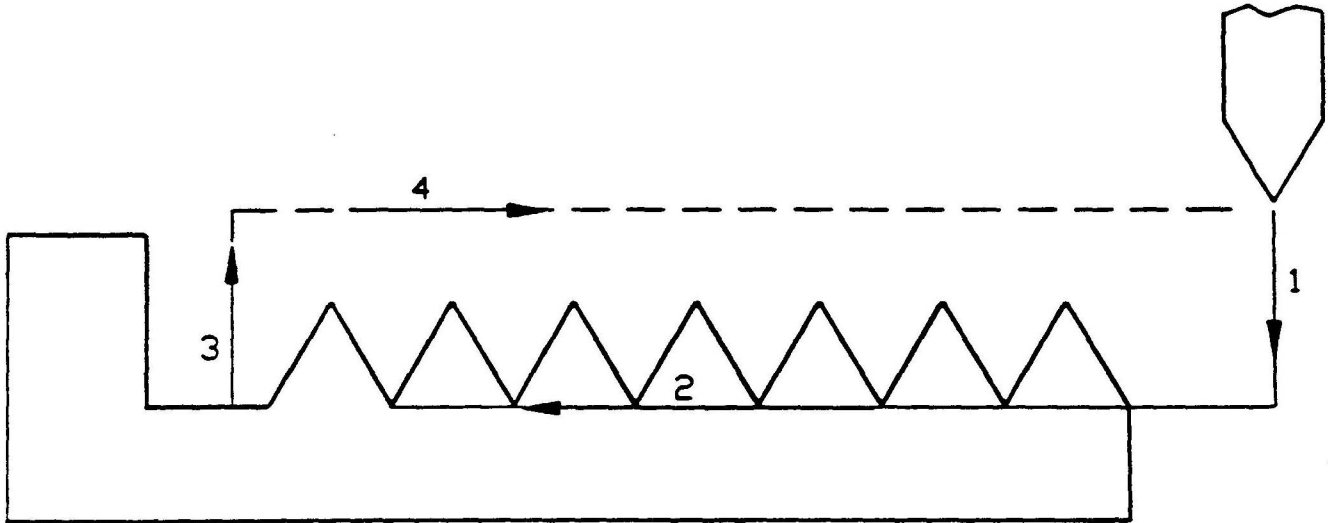
OPERATIONS

1. FACE
2. ROUGH TURN
3. FINISH TURN
4. GROOVE
5. THREAD

Motion of Thread Cutting Tool

When using the thread cutting cycle called for by G71 or G72, tool paths (1) and (4) are executed at a rapid traverse rate, (2) at the feed rate specified by the "F" word, and (3) at the feed rate determined by a parameter setting.

Parameter Long Word No.8



"M" Codes Specifying Cutting Mode and Infeed Pattern

In the block calling for "G71" Thread Cutting Fixed Cycle, "M" codes specifying mode and infeed pattern should be programmed.

1. "M" Codes for Infeed Pattern:

- M32 Infeed Pattern to Cut on Front Face of Thread - Default
- M33 Infeed Pattern to Cut Zig-Zag
- M34 Infeed Pattern to Cut Rear Face of Thread

If none of these "M" codes are programmed the control will automatically select M32 Infeed Pattern.

2. "M" Codes for Cutting Mode (depth of cuts)

M73 Cutting Mode the control will calculate the needed depth and number of passes, based on the "X" dia., "H" thread height, "U" an extra pass and "D" depth of cut.

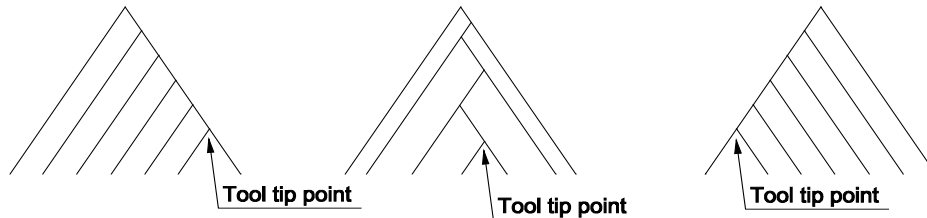
The control will calculate the passes in the following manner for an O/D thread:

- First Pass: (Major Dia. - D)
- Second Pass: (First Pass - D)
- Third Pass: (Second Pass - D)
- ETC

Cutting Modes

- M32 Straight infeed along thread face (on the left face)
- M33 Zigzag infeed
- M34 Straight infeed along thread face (on the right face)

Designate a B command (tool tip point angle) at the same time.



Infeed Patterns

1) M73 Infeed pattern 1

Infeed is made by D (in diameter) in each thread cutting cycle until a point which is away from the “H - U (W)” level by D . After this point has been reached, cutting is performed by changing the infeed amount as $D/2$, $D/4$, $D/8$ and $D/16$, while leaving finishing stock U (W). The finish cutting cycle is not performed unless U (W) is specified.

2) M74 Infeed pattern 2

Infeed is made by D (in diameter) in each thread cutting cycle until “H - U (W)” level is reached. After that, the finish cutting cycle is performed with infeed amount U (W). The finish cutting cycle is not performed unless U (W) is specified.

3) M75 Infeed patterns 3 and 4

The depth of cut is controlled in each thread cutting cycle so that the metal removal rate is optimal. Whether pattern 3 or pattern 4 is selected is determined by the setting at optional parameter (bit) No. 3, bit 5

Optional parameter (Bit) No. 3, bit 6:

- 0.... Infeed pattern 3
- 1.... Infeed pattern 4

The control continues in this manner until it reaches a calculated diameter that is six passes from the end. The six passes are calculated in this manner.

- This pass is equal to ("X" + "U" + "D")
- This pass is equal to ("X" + "U" + "D"/2)
- This pass is equal to ("X" + "U" + "D"/4)
- This pass is equal to ("X" + "U" + "D"/8)

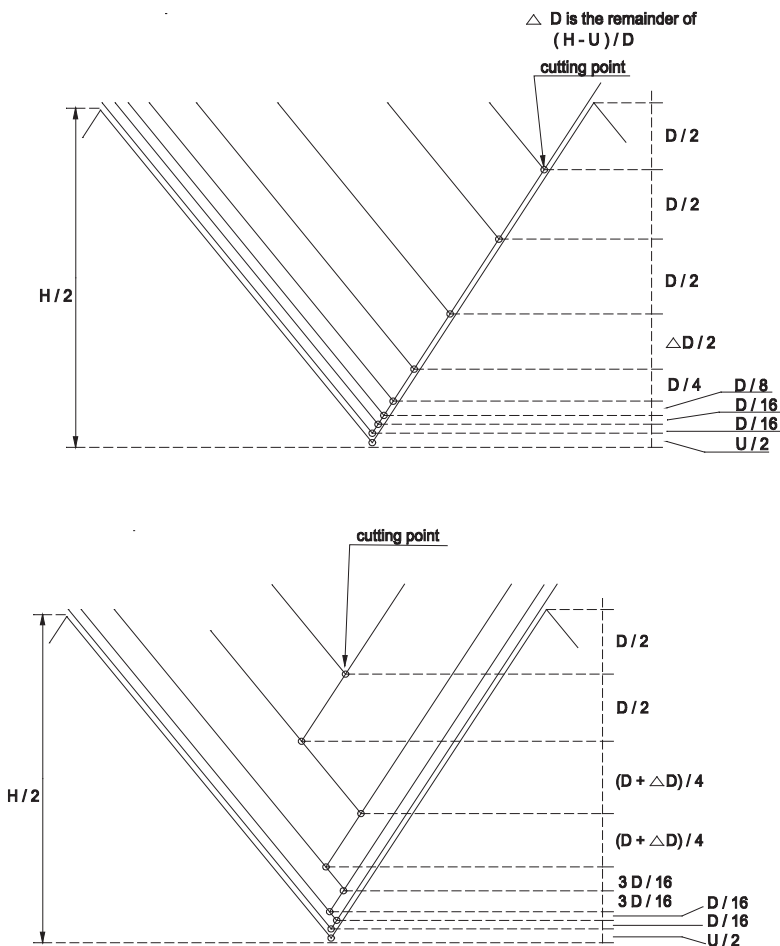
An extra pass is equal to ("X" + "U")

Last pass is equal to "X" diameter (minor dia. on O/D, major dia. on I/D)

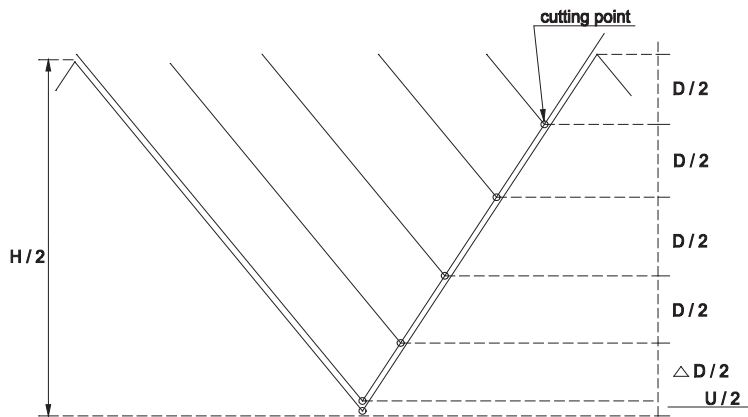
Of course, it will be rare that the thread pass before the sixth pass from the end will work out to be in even increments. When the control reaches a diameter that it cannot subtract "D" value from, without violating the calculated SIXTH pass from the end, it will transfer to the SIXTH pass value taking a pass less than the "D" value.

With combining the M codes specifying cutting mode and infeed pattern, six types of thread cutting are available for longitudinal thread cutting cycle and transverse thread cutting cycle, respectively.

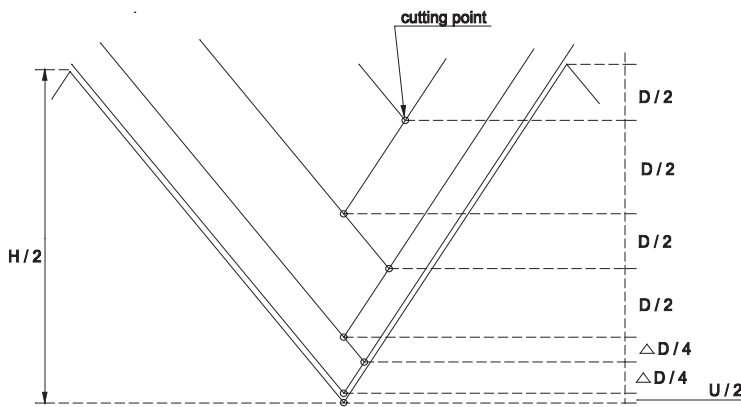
1. Longitudinal Thread Cutting Compound Fixed Cycle



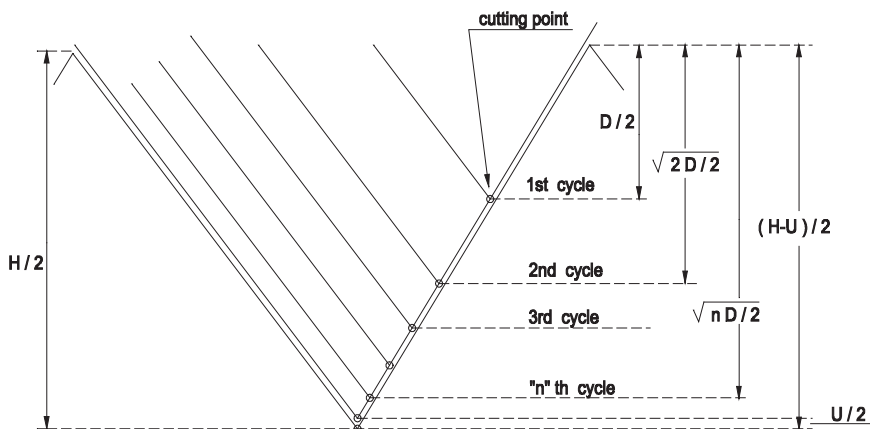
C. M32 + M74 Mode



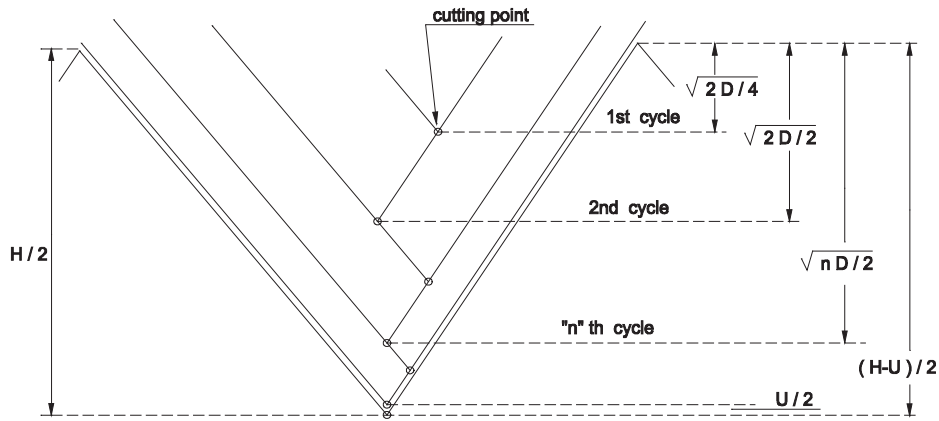
D. M33 + M74 Mode



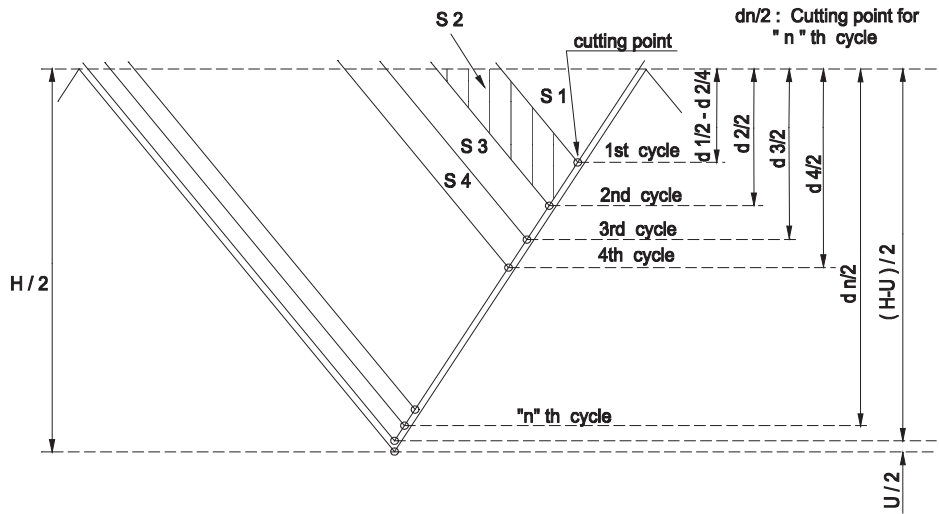
E. M32 + M75 Mode (Infeed Pattern 3, $D_2 > (H^2 - (H - U(W))^2)$)



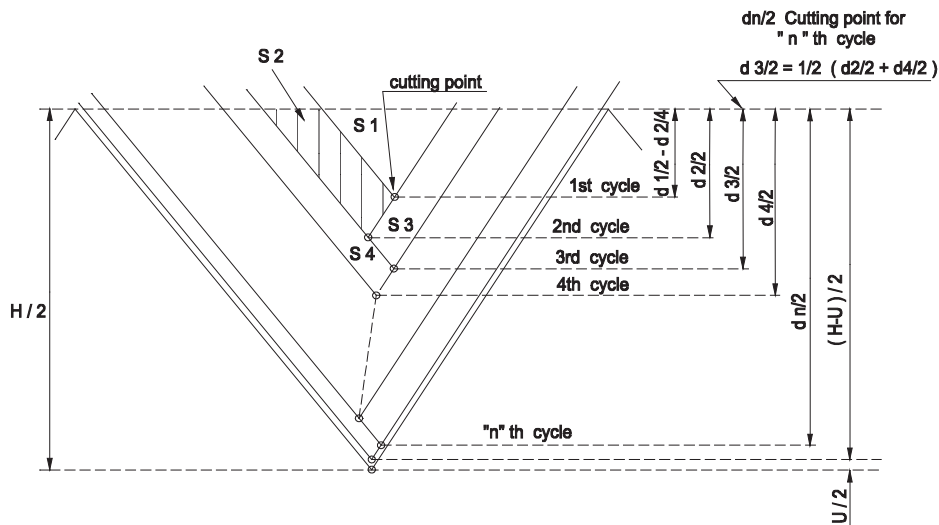
F. M33 + M75 Mode (Infeed Pattern 3, $D^2 \geq (H^2 - H - U(W))^2$)



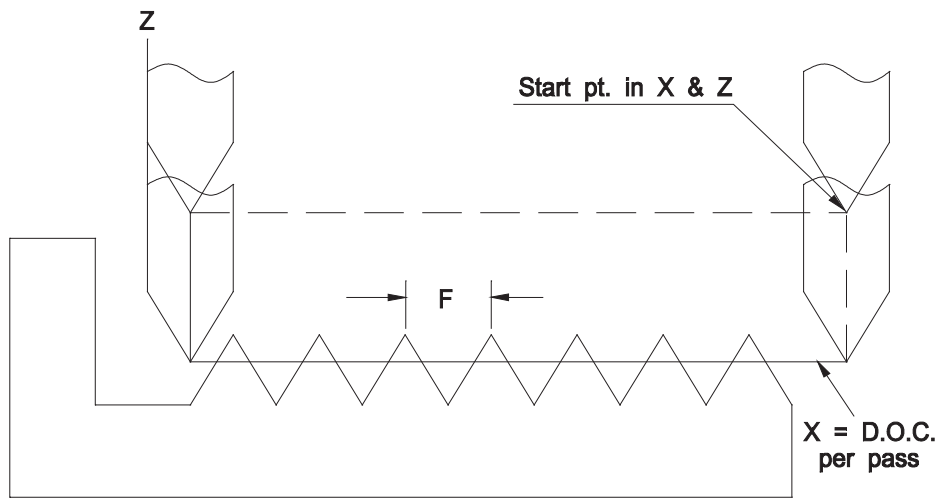
G. M32 + M75 Mode - (Infeed Pattern 3, $D^2 \geq (H^2 - H - U(W))^2$) Infeed Pattern 4



H. M33 + M75 Mode - (Infeed Pattern 3, $D^2 \geq (H^2 - H - U(W))^2$) or Infeed Pattern 4



G33 - Fixed Threading Cycle (longitudinal) - Each pass individually programmed.



Definitions:

- X Coordinate value (dia.) of "each" thread cutting pass.
- Z Coordinate value of thread end-point in Z-axis direction.
- F Thread lead, or, (1/number of threads per inch). If "J" word is used "F" value should be one for even or odd number of threads and two for decimal threads, such as 11.5 threads per inch.
- K Incremental value used to shift the thread cutting start point. Value must be positive and can not be used when either "I" or "A" are active.

Note: When no "K" word is programmed, the control assumes K=0

- A Tapered thread angle measured from the positive Z-axis zero or three o'clock position. A counterclockwise value is plus and a clockwise value is negative.

Note: Do not use "A" if using the "I" command.

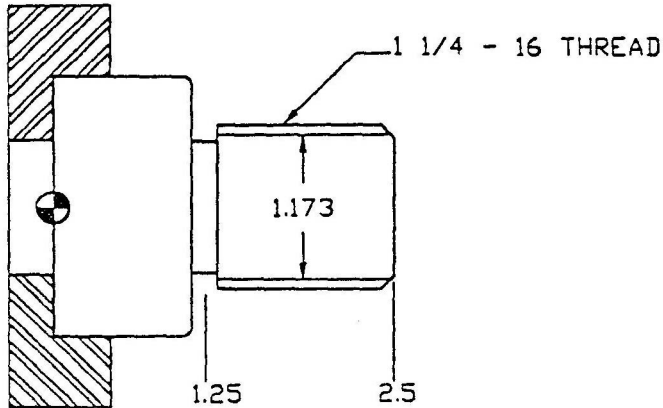
- I Incremental radial difference between start point and end point of thread for taper thread cutting.

"+"	Increasing taper
"-"	Decreasing taper

Note: Do not use "I" if using the "A" command.

- L Chamfering or pull-out distance at end of thread. When no "L" word is provided the control will assume that "L" is equal to one thread specified at start of thread by the "F" (lead) word. The "L" condition can only be active when M23 is added to the G33 line of programming.
- J Is equal to number of threads per inch. Must always be entered as a whole number when used with "F" thread leads.

Programming Example: G33-2



```
G50 S3500
N1 G0 X25 Z25
N2 X1.3 Z2.9 S1200 T606 M3 M41
N3 G33 X1.239 Z1.125 F1 J16
N4 X1.228
N5 X1.217
N6 X1.206
N7 X1.195
N8 X1.184
N9 X1.177
N10 X1.173
N11 G0 X25 Z25
M2
```

1 1/4 -16 - O.D. thread with "Z" zero at left face of part. Program sample is written for a steel part under 300 BHN., number of passes is eleven.

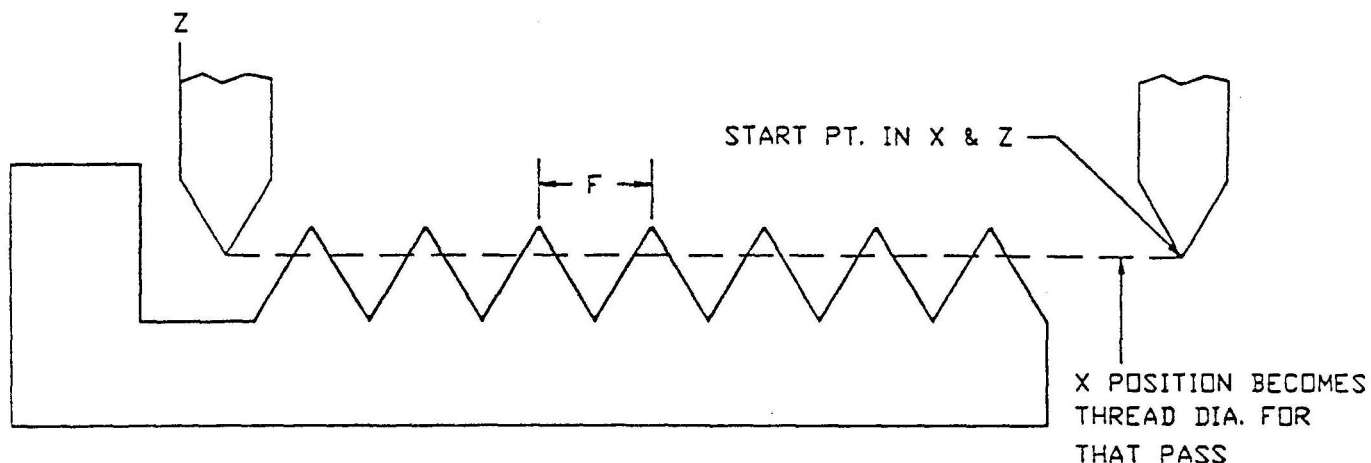
Length of thread = 1/10 or .1 inch

Pitch or "F" = 1/10 or .1 inch

When the thread cutting cycle is activated by using the "G33" the tool will rapid down to the thread dia. called for, feed across the dia. at the feed rate established by the RPM and "F" word, retract at a feed determined by the parameter setting and then rapid back to the thread cutting start point.

G34 - Variable Lead Threading, Increasing (non-fixed cycle)

[G34] [X___] [Z___] [F1] [J___] [(E___)]



Definitions:

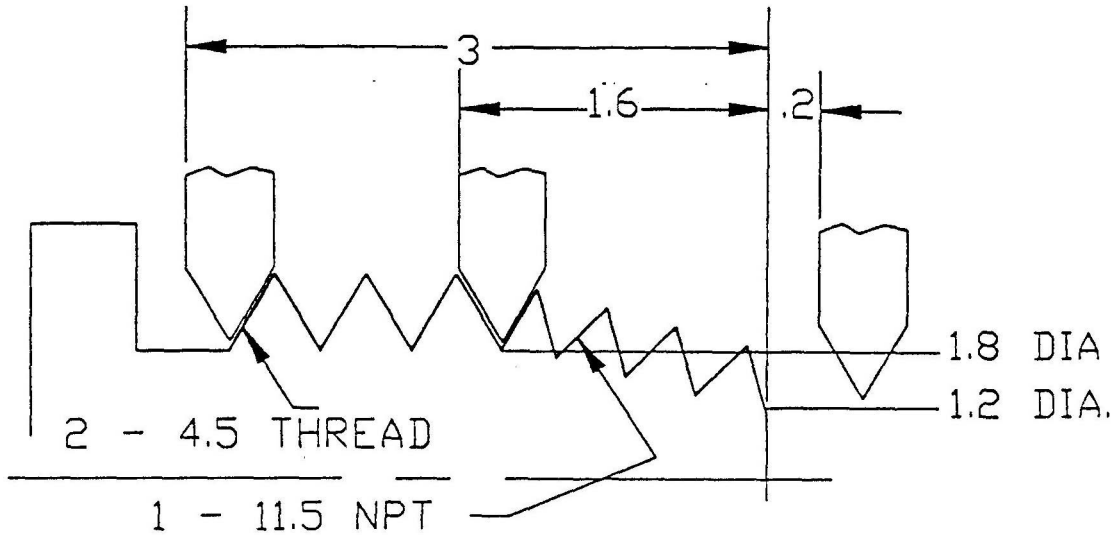
When using the “G34” cycle one must remember that it is a non-fixed cycle. The tool **MUST** be positioned in the X-axis and Z-axis to the desired threading tool pass prior to using the cycle. Once positioned, any tool path can be programmed: vertical, horizontal or slope. Upon completion of the tool path, the programmer must retract that tool and reposition for another pass if needed.

- X Coordinate value (dia.) of “each” thread cutting pass.
- Z Coordinate value of thread end-point for “each” programmed thread cutting pass, in Z-axis direction.
- E Lead variation per lead in cutting variable lead threads increasing (plus) values only.
- F Thread lead or 1/number of threads per inch. If “J” word is used “F” value should be **one** for even or odd number of threads and **two** for decimal threads such as 11.5 threads per inch.
- J Is equal to number of threads per inch. Must always be entered as a whole number when used with “F” thread lead.

Example:

1 - 20 thread would be	F1.J20
3/4 - 11 thread would be	F1.J11
1/2 - 11.5 thread would be	F2.J11.5

PROGRAMMING EXAMPLE: G34-1



The following program is written to show one pass on the threaded part

```
N1 G50 S3000  
N2 G00 X20.Z20  
N3 X1.2 Z.2 G97 S200 T0101 M08 M03 M41  
N4 G34 X1.8 Z-1.6 F2 J11.5  
N5 Z-3 F2. J9.  
N6 G00 X2.4  
N7 X20. Z20. M09  
N8 M02
```

This programming is conveniently used for special thread cutting: combined threads, straight threads, tapered threads, variable lead threads and straight constant lead threads.

To achieve full depth of thread, several passes would have to be taken.

SUBPROGRAMS, SCHEDULE PROGRAMS AND ADDITIONAL FILES

**Divide an existing program into subprograms
and discuss System Subprogram concept**

Have students divide another existing program

Explain Schedule Program operation

Review the concept of Variable Programming

Common Variables

Local Variables

Program Files

There are four types of files as described latter. They correspond to filed documents, or ledgers; to manage NC data, each file is stored with the name of a part such as gear, shaft or flange assigned to it. File names consist of main file names and extensions.

Main file name.....Consists of a character-string of 16 characters beginning with an alphabetic character

Extension.....Consists of a character-string of 3 characters beginning with an alphabetic character.

Note that in assigning usable characters file names are: alphabetic characters, numeral characters and hyphen.

A file name is assigned in the order of main file name first then an extension.

A period must be entered as a delimiter between the main file name and the extension.

Types of files:

	<u>Main file Name</u>	<u>Extension</u>
- Main program file.....	Character string (16 characters) beginning with an alphabetic character	.MIN
- Subprogram file.....	Character string (16 characters) beginning with an alphabetic character	.SUB
- System subprogram file.....	Character string (16 characters) beginning with an alphabetic character	.SSB
- Schedule program file.....	Character string (16 characters) beginning with an alphabetic character	.SDF

1. Main Program File

A file comprising one or more main programs with the main file name followed by the extension "MIN".

To make a main program file, there are two methods:

- Classified depending on types of work pieces: gear, shaft, and flange.
- Program(s) for each work piece to be assigned with one file name

2 Subprogram File

Patterns often repeated in cutting parts such as Vee-groove and parting off cycle are filed in the subprogram file when preparing part programs. The subprogram file comprises one or more subprograms; the main file name is followed by the extension "SUB".

When the called main program contains the command that calls the subprograms, it is necessary to call the subprograms file of these subprograms also when selecting the program(s).

3 System Subprogram File

The system subprograms file is the same as the subprogram file explained above; in the system subprogram file, repetitive patterns often used by respective users are filed.

4 Schedule Program File

This file is used when controlling several part programs prepared to meet the production schedule. A schedule program file consists of one schedule program, and its file name is suffixed by "SDF"

5 Program Name or Number

A program name or number is made up from:

- Program name.....Address character "O" and four alphanumeric characters beginning with an alphabetic character.
- Program number...Address character "O" and four numeral characters.

There are four types of program names and program numbers that correspond to be the type of files explained in

(1) Main Program

A main program begins with program number and ends with M02 or M30.

(2) Subprogram

A subprogram begins with program number and ends with RTS

(3) System Subprogram

A system subprogram begins with program number and ends with RTS as a subprogram indicated above.

(4) Schedule Program

Schedule Operation

To machine different kinds of work pieces continuously, the scheduled operation feature is very effective. In scheduled operation, part programs used to machine work pieces and those for controlling work piece loading/unloading operation are prepared and the order the number of execution of these programs are specified in the schedule program. Continuous operation is performed according to the schedule program.

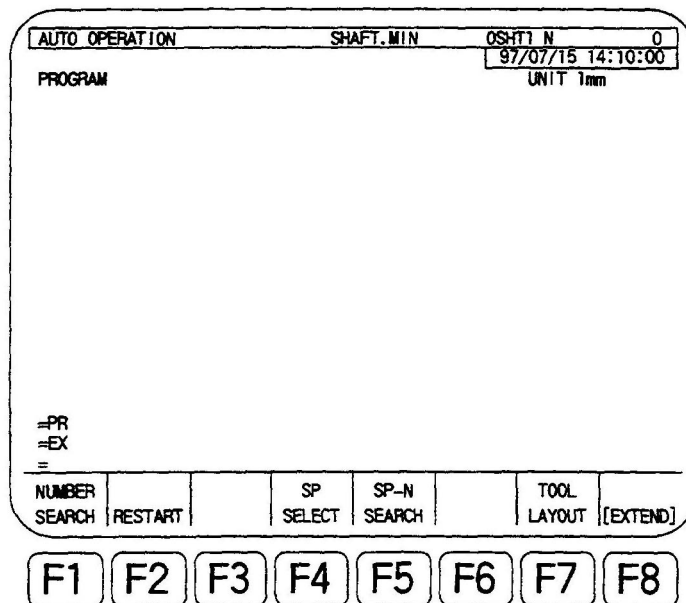
For programming of a schedule program refer to PROGRAMMING MANUAL.

1 Schedule Program Selection

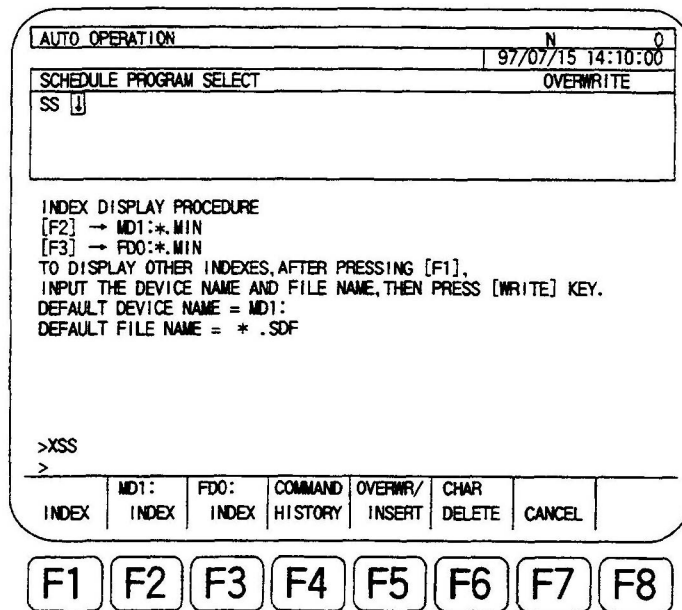
(1) Selection from the Directory Display Screen

<Procedure>

- (a) Press the [AUTO] mode key.
- (b) Press function key [F3] (PART PROGRAM) and press the page keys to display the schedule program screen.
- (c) Press function key [F8] (EXTEND) to display the schedule program selection function.
- (d) Press function key [F4] (SP SELECT).



The screen changes to the index-selection-based file operation screen where “SCHEDULE PROGRAM SELECT” and prompt “SS” are displayed



- (5) Press function key [F2] (MD1: INDEX).
- (6) Move the cursor to the desired file name and press the [WRITE] key.
- (7) Press the [WRITE] again. This selects the program at the cursor position and the screen returns to the previous screen. Make sure that the specified schedule program has been selected.
- (8) Press the [CYCLE START] button.

This starts the continuous operation according to the selection schedule program.

(2) Direct Designation of Schedule Program

Selection of a schedule program is also possible by directly inputting a file name and program name. In this operation, schedule program to be selected must be stored in memory.

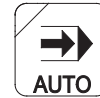
<Example of stored programs>

File Name	Contents
A.SDF	Schedule program 1

The explanation below is given based on this example program.

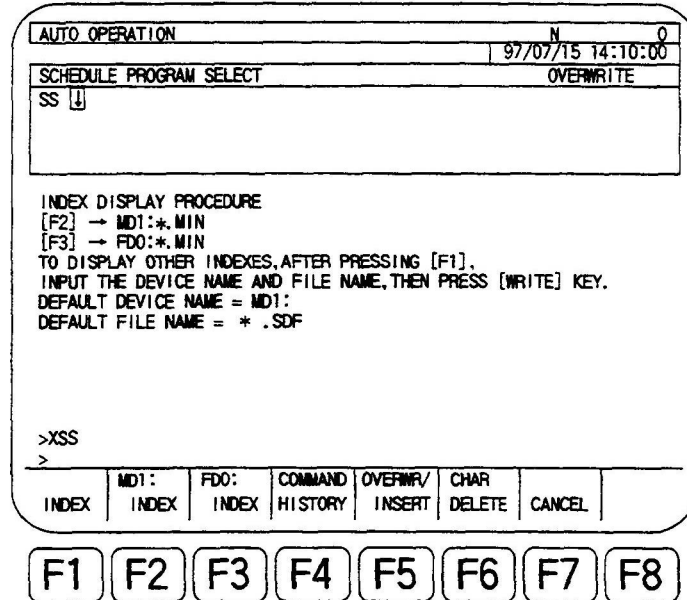
<Procedure>

(1) Press the [AUTO] selection key.



(2) Press function key [F8] (EXTEND) to display the schedule program selection function.

(3) Press function key [F4] (SP SELECT)



(4) Input the selection program file name.

=SS A.SDF

(5) Press the [WRITE] key.

This selects the schedule program to be executed. On the program display screen make sure that the specified schedule program has been selected.

(6) Press the [CYCLE START] button.

The schedule program file A.SDF is executed.

[Supplement]

- A Wild card (? . *) can be used in the selection of a schedule program file.
- B Selection of a schedule program should be made only after resetting the NC. If schedule program selection is made while the NC is running an error occurs.
- C To carry out normal operation (automatic operation by selecting a program) after selecting a schedule program select the desired program again.

[Supplement Cont.]

- D When the scheduled operation is started with the single block function ON, a main program is selected by the schedule program first and the NC waits for cycle start operation. Pressing the [CYCLE START] button causes the NC to run in the normal single block mode.
- E When the [RESET] key is pressed during continuous operation in the scheduled operation mode, and if the interrupted program is resumed, the program selected at the time the [RESET] key was pressed is executed from the beginning.

F Cycle stop

If the machine has a cycle stop switch and operation is executed with that switch set ON, the NC enters the cycle start waiting state after executing a main program.

This feature is used for schedule operation of the machine equipped with a robot or a bar feeder; the machine is placed in the cycle start waiting state at the completion of a program (execution of M02 or M30) by inputting an external cycle stop signal to the NC.

Variables

There are three types of variables used with User Task 2:

- Common variables
- Local variables
- System variables

These three types of variables differ in their use and characteristics.

(1) Common Variable

Common variables are common to main programs and subprograms. When the same variable is used in two or more programs, the variable number used in those programs must be identical. Therefore, the common variable, the result of calculation in one program, can be referenced to in other programs.

a) Program format

<V> <one or two digits>=numerical is used as:

N101 V5=10, or N101 V5+1

Note 1: Common variables are used in common for A-turret and B-turret

Note 2: Common variables are effective both in main and subprogram.

Note 3: Common variables are not affected by control reset or turning power off. The data are retained unless they are re-set or control software is installed.

Note 4: Common variables can be set or changed by setting a parameter or by changing them in a program. For detailed information of parameter setting, refer to 3.

“OPERATION IN PARAMETER MODE” of the Operation Manual for OSP70001.

Common Variables

There are 200 common variables, V1 through V200.

In the parameter setting operation, these variables can be set or changed. They can be used freely, independent of the system.

<Function selection>

SET: When directly inputting the value to be set.

ADD: When a value is already set and the value to be added to or subtracted from the existing value should be set.

CAL: Cannot be used.

SEARCH:

Selection of a common variable for which the data should be set can be made by either using the cursor keys or the search function.

After pressing function key [F4] (SEARCH), input the common variable where the cursor should be moved, and the cursor moves directly to the specified position.

PARAMETER SET SEIGYO-1.MIN N 0
97/06/03 17:46:10

Page 1 A-turret

* COMMON VARIABLE *

NO.	NO.	NO.	NO.	NO.			
1	0	11	0	21	0	31	0
2	0	12	0	22	0	32	0
3	0	13	0	23	0	33	0
4	0	14	0	24	0	34	0
5	0	15	0	25	0	35	0
6	0	16	0	26	0	36	0
7	0	17	0	27	0	37	0
8	0	18	0	28	0	38	0
9	0	19	0	29	0	39	0
10	0	20	0	30	0	40	0

SET ADD CAL SEARCH BACK UP ITEM ↑ ITEM ↓

F1 F2 F3 F4 F5 F6 F7 F8

NOTICE: Since common variables are stored as floating-point data, the data is processed by rounding-off and as the result of this processing numbers displayed to the right of a decimal point may differ from the set numbers.

Local Variables

Local variables are the variables that a user can set as desired with a unique name defined by the user. Up to 127 local variables can be used for A-turret and B-turret, respectively.

a) Program format

<Letter> <Letter> <two alphanumerics>= Numerical data or expression

“O”, “N” and “V” cannot be used.

Example: “DIA1” ITH5”

*Note: The same name as used for <function name> *1 <comparison operator> *1, <boolean operator>*1, and <extended address character> *2 cannot be used.*

*1: Explained in 3-3. “Arithmetic Operation Function”

*2: Extended address characters are provided to realize the LAP, pattern processing, user-specific fixed cycles. The following extended characters are used presently. <AA><AB><DA><DB><FA> <IA> <IB> <KA><KB><LA><LB><RA><RB><SA><SB><TA> <TB><UA><UB><WA><WB><XA><XB><ZA><ZB> <BC>

b) Characteristics of Local Variables

- 1) Local variables can be used independently on A-turret and B-turret, and are cleared when the control is reset.
- 2) When a new local variable is set in a main program, that is, when data is assigned to a new local variable name, that local variable name and corresponding data are registered in the memory.

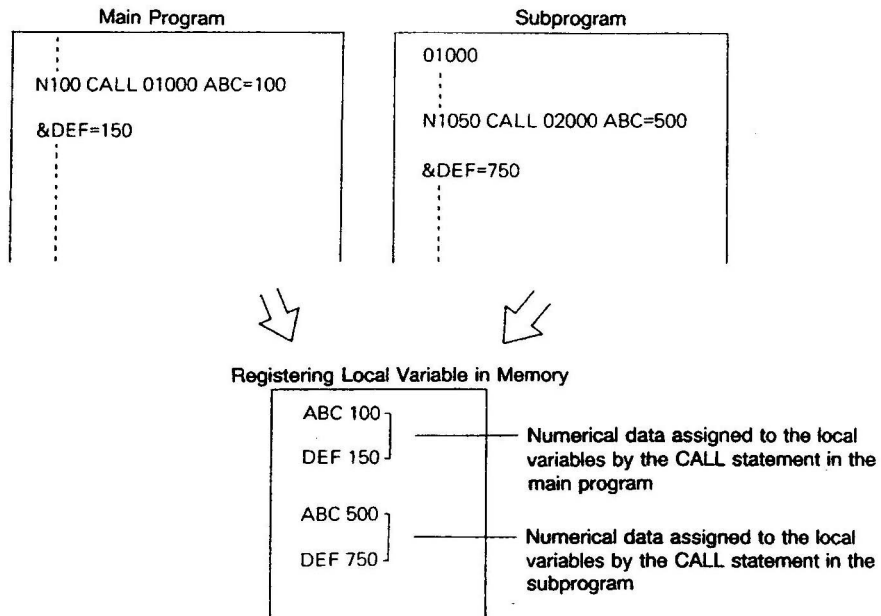
Note: If a local variable name is used without assigning the data, an alarm results.

- 3) When new data is assigned to the local variable already registered with another data, that old data is replaced.

Main program

: In N0010, numerical data “160” is :
assigned to local variable name
N0010 DIA1= 160 “DIA1”, which remains effective
: up to sequence N0049. In
N0049 N0050, new numerical value “200
new is assigned to the same local
N0050 DIA1= 200 variable name “DIA1”. This
clears the value data “160” and substitutes with the
value “200”

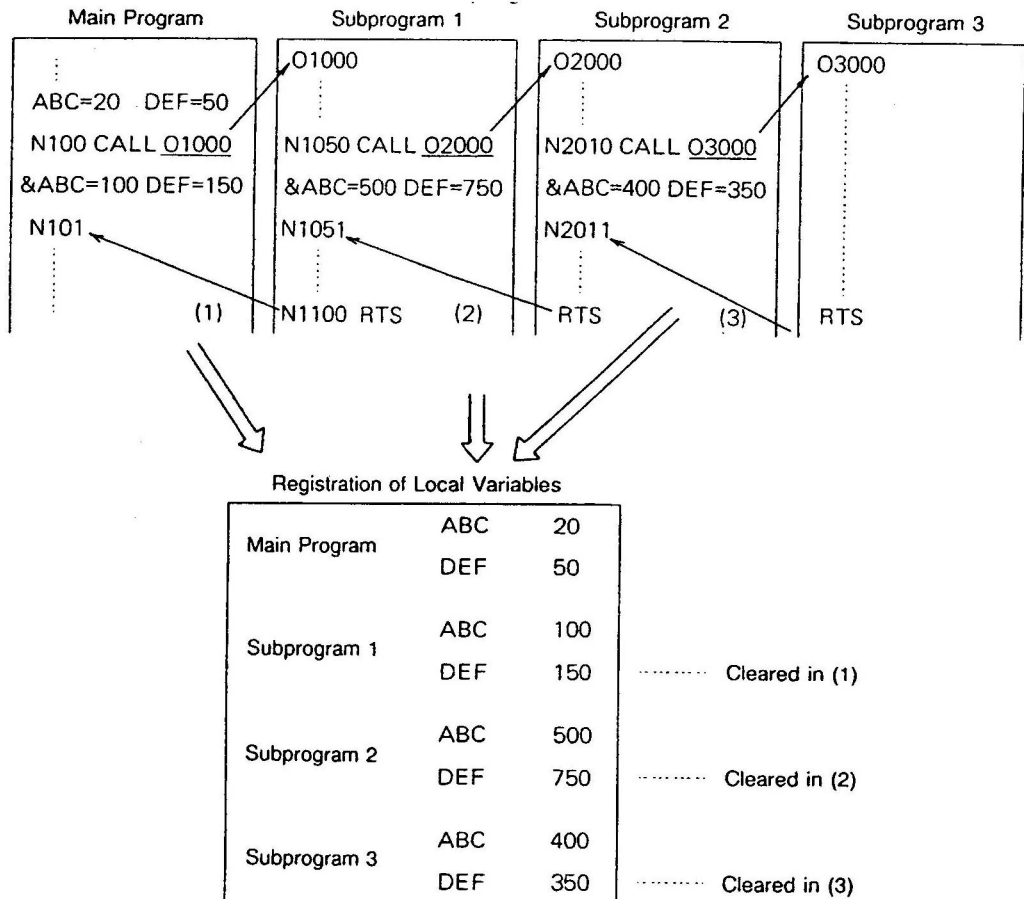
- 4) Up to 127 local variables can be used on A-turret and B-turret, respectively.
- 5) Local variable values are not passed between the main program and subprograms. A subprogram may have a local variable of the same name as one in the main program, but their values can be different.



As shown above, the variables having the same name as those already defined are newly defined different variables.

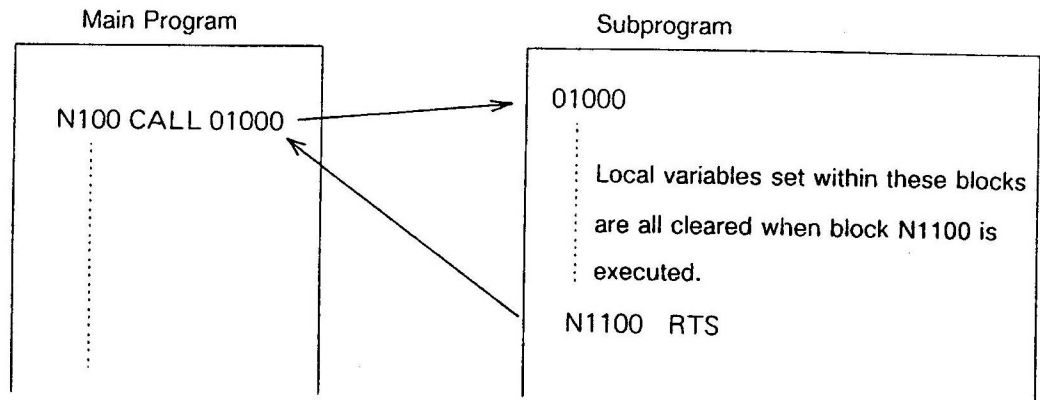
- 6) When using local variables in a subprogram, the numerical data assigned last to the local variable is used when there are several local variables assigned with the same name registered in the memory.

The local variables set in the block containing a CALL statement are cleared when the RTS statement in the called subprogram is executed.

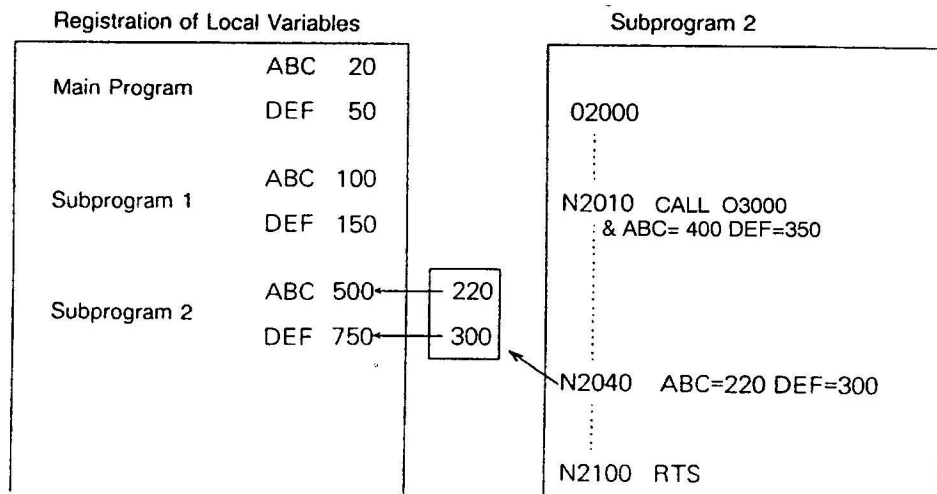


In the example above, execution of N2010 in subprogram 2 defines 4 kinds of local variables: ABC and DEF. The values stored last, i.e., ABC = 400 and DEF = 350 are passed to the subprogram. At the end of the subprogram 03000 the RTS statement in 03000 is executed and the local variables registered by the subprogram CALL 03000, ABC = 400 and DEF = 350 are cleared from the memory.

- 7) When a local variable is newly set in a subprogram, its name and numerical value are stored in the memory. They are effective only in the subprogram in which they are set, and are cleared when the RTS statement in the subprogram is executed.



- 8) When a new value is assigned to a local variable name, while a subprogram is executed, the numerical data is renewed. If several local variables having the same variable name are stored in the memory, the value of the local variable stored last is restored.



While the subprogram 2 is executed, local variables ABC = 400 and DEF = 350 are stored in the memory but they are cleared by executing program 3 RTS. Therefore, the variables defined when program 2 is called in N 1050 are effective up to block N2039. When block N2040 is executed, the numerical values of local variables ABC and DEF defined in subprogram 2 are changed to 220 and 300, respectively.

The subprogram defining the contour, prepared using local and common variables, can be programmed as below according to Table 2-1.

\$ SHAFT-ABC.SUB
%
0100
NLAP1 G81
N1001 G00 X=DX1 Z=LZ1+2
N1002 G01 Z=LZ2 F0.2
N1003 X=DX2
N1004 Z=LZ3
N1005 X=DX3
N1006 Z=0
N1007 G80
N1010 G00 X=800 Z=400
N1011 G96 X=XS Z=ZS S=V3 T=V1 M03 M08
N1012 G85 NLAP1 D4 F0.35 U=UDX1 W=WLZ1
N1013 G97 S=1000/3.14*V3/XS
N1020 G00 X=800 Z=400
N1021 G96 X=XS Z=XS S=V4 T=V2
N1022 G87 NLAP1
N1023 G00 G97 X=XS Z=ZS S=1000/3.14*V4/XS
N1024 RTS

The main program for the work piece is programmed as shown below.

Work piece A
\$ SHAFT-A.MINa)
%b)
0100c)
N101 G00 X800 Z400
N102 CALL 01000 V1=0101 V2=0202 V3=100 V4=120d)
LZ1=200 LZ2=150 LZ3=80
\$ DX1=30 DX2=50 DX3=80 WLZ1=0.1 UDX1=0.2 XS=100 ZS=210
N103 M02

a) File name of the main program.....program "\$" must precede the file name

b) Key in the main program name as follows:

\$, program name, MIN, feed holes, %, CR, LF.

c) Program name0100 in this example.

d) With the commands in N102, the subprogram 01000 is called and executed. Numerical data for the variables used in the subprogram are set. When commands in this block cannot be written on line. They are separated on to several lines by placing a "\$" code at the beginning of the each continuation line. The control recognizes those commands as programmed in one block.

Work piece B \$ SHAFT-B.MIN
%
0101
N101 G00 X800 Z400
N102 CALL 01000 V1=0303 V2=0404 V3=110 V4=130 LZ1=250 LZ2=170 LZ3=100 \$ DX1=40 DX2=70 DX3=120 WLZ1=D.15 UDX1=0.25 XS=140 ZS=260
N103 M02

Work piece C \$ SHAFT-C.MIN
%
0102
N101 G00 X800 Z400
N102 CALL 01000 V1=0505 V2=0606 V3=90 V4=150 LZ1=300 LZ2=200 LZ3=120 \$ DX1=50 DX2=90 DX3=150 WLZ1=0.2 UDX1=0.3 XS=170 ZS=310
N103 M02

Note: To call the machining program follow the steps below:

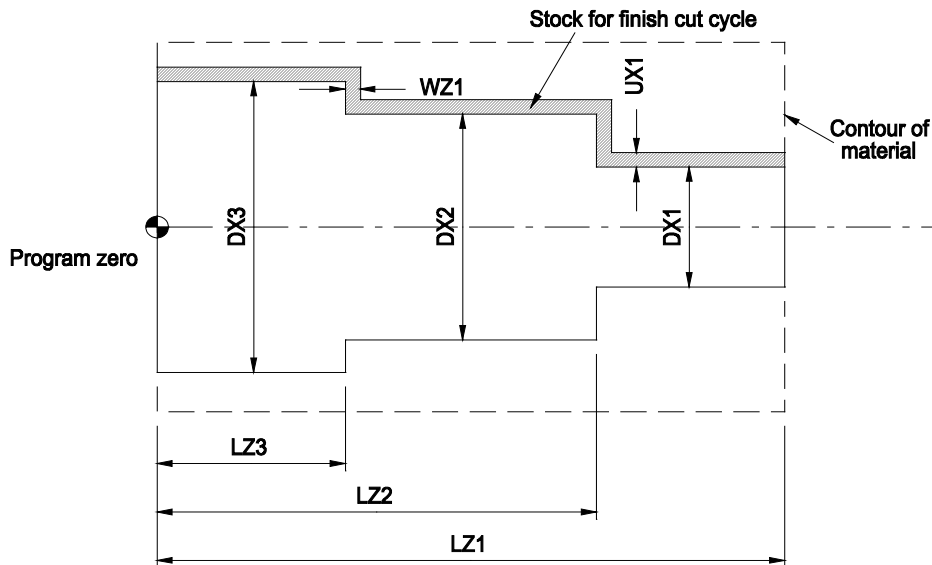
- (1) Press the AUTO key to select the automatic operation mode
- (2) Press the function key [F1] (PROGRAM SELECT)
- (3) Key in the main program name, subprogram name and select [WRITE].

PROGRAM EXAMPLES

Three typical program examples are provided on the following pages. They are prepared only for purpose of illustration, and they do not cover all the functions available with User Task 2.

Please refer to the examples so that you can make the most of the User Task function for preparing programs.

<Program Example 1>.....[Shaft work with similar contour]



Assuming three different work pieces having similar contours as shown above are to be cut, programs are prepared using task function.

- (1) Assign respective work pieces with file names:

SHAFT-A

SHAFT-B

SHAFT-C

- (2) Since these work pieces have similar contours, their contours are defined by a subprogram. File name of the subprogram is "SHAFT-ABC.SUB".
- (3) The elements (dimensions) used to define the contour and the tool numbers and the cutting speeds are expressed using local variables and common variables, respectively.

V1 = Rough tool

DX1 = Diameter DX1

V2 = Finishing tool

DX2 = Diameter DX2

V3 = Cutting speed in roughing cycle

DX3 = Diameter DX3

V4 = Cutting speed in finishing cycle

WLZ1 = Finish allowance in longitudinal
Direction WZI

LZ1 = Longitudinal dimension LZ1

UDX1 = Finish allowance on diameter UX1

LZ2 = Longitudinal dimension LZ2

XS = X coordinate of LAP Starting point

LZ3 = Longitudinal dimension LZ3

ZS = Z coordinate of LAP Starting point

(4) For cutting the work piece, LAP mode is used.

(5) Table 2-2 , below, is a summary of previously listed variables.

		Work A	Work B	Work C
		SHAFT-A	SHAFT-B	SHAFT-C
Roughing tool	V1	0101	0303	0505
Finishing tool	V2	0202	0404	0606
Cutting speed in rough cycle	V3	100	110	90
Cutting speed in rough cycle	V4	120	130	150
LZ1	LZ1	200	250	300
LZ2	LZ2	150	170	200
LZ3	LZ3	80	100	120
DX1	DX1	30	40	50
DX2	DX2	50	70	90
DX3	DX3	80	120	150
WZI	WLZ1	0.1	0.15	0.2
UX1	UDX1	0.2	0.25	0.3
XS	XS	100	140	170
ZS	ZS	210	260	300

Variable Programming SHAFT EXAMPLE

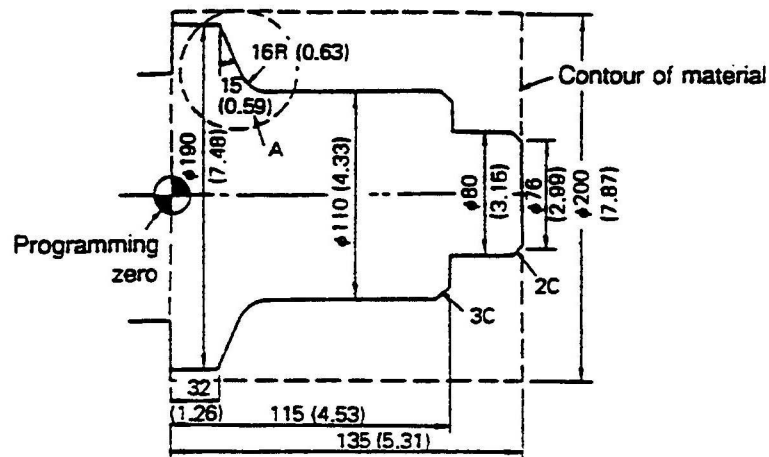
```
G13
(SHAFT-A.MIN)
G50S4200
N101 GOX800Z800
N102 CALL O1000 V1=1010 V2=1111 V3=160 V4=200
$ LZ1=200 LZ2=150 LZ3=80 DX1=30 DX2=50 DX3=80 WLZ1=.1 UDX1=2
$ XS=100 ZS=210
N103 G0X800Z800
N104 M2
```

```
G13
(SHAFT-B.MIN)
G50S4200
N101 GOX800Z800
N102 CALL O1000 V1=1010 V2=1111 V3=160 V4=200
$ LZ1=250 LZ2=170 LZ3=100 DX1=40 DX2=70 DX3=120 WLZ1=.1
$ UDX1=2 XS=140 ZS=260
N103 G0X800Z800
N104 M2
```

```
G13
(SHAFT-C.MIN)
G50S4200
N101 GOX800Z800
N102 CALL O1000 V1=1010 V2=1111 V3=160 V4=200
$ LZ1=300 LZ2=200 LZ3=120 DX1=50 DX2=90 DX3=150 WLZ1=.1
$ UDX1=2 XS=170 ZS=310
N103 G0X800Z800
N104 M2
```

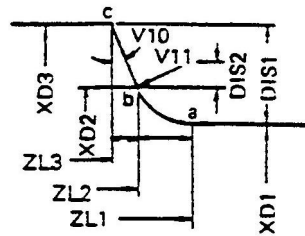
```
O1000
(SHAFTABC.SSB)
NLAP1 G81
N1001 G00 X=DX1 Z=LZ1+2
N1002 G01 Z=22 F0.35
N1003 X=DX2
N1004 Z=LZ3
N1005 X=DX3
N1006 Z=0
N1007 G80
N1010 G00 X=8—Z=800
N1011 G96 X=XS Z=ZS S=V3 T=V1 M03 M08
N1012 G85 NLAP1 D8 F0.45 U=UDX1 W=WLZ1
N1013 G97 S=1000/3.14*V3/XS
N1020 G00 X=800 Z=800
N1021 X=XS Z=ZS T=V2
N1022 G96 S=V4
N1022 G87 NLAP1
N1024 G00 G97 X=XS Z=ZS S=1000/3.14*V4/XS
N1025 RTS
```

<Program Example 2>.....(Cutting contour requiring calculation for defining 1)



When cutting a contour containing a circular arc and a taper and when the point(s) of intersection is not indicated on the part drawing, the user task function featuring operation function should be used to prepare the program. .

- (1) With enlarging section A, have the control calculate the points of intersection using the variables and the operation function of the user task.



The points that must be calculated are Z-coordinate of point "a" and X- and Z-coordinate of point "b". To obtain them, variables are set as follows.

Set using common variables

$$V10 = \text{Taper angle } (15^\circ)$$

$$V11 = \text{Arc radius } (16 \text{ mm})$$

Set using local variables

XD1	=	Diameter of point “a” (110 mm)
XD2	=	Diameter of point “b”
XD3	=	Diameter of point “c” (190 mm)
ZL1	=	Z-coordinate of point “a”
ZL2	=	Z-coordinate of point “b”
ZL3	=	Z-coordinate of point “c” (32 mm)
DIS1	=	Distance: DX3 – DX1
DIS2	=	Distance between point arc and point “b” (along Z-axis)
DIS3	=	Distance between point “a” and point “b” (along Z-axis)
DIS4	=	Distance between point “b” and “c” (along Z-axis)

The point of intersection can be calculated from the following equations

DIS1	=	$[XD3 - XD1]/2$
DIS2	=	$V11 * \sin [V10]$
DIS3	=	$V11 * \cos [V10]$
DIS4	=	$[DIS1 + DIS2 - V11] * \tan[V10]$
XD2	=	$ZD1 + 2 * [V11 - DIS2]$
ZL2	=	$ZL3 + DIS4$
ZL1	=	$ZL2 + DIS3$

(2) Since the pattern appearing in Section A can be used in common with other work pieces, it is advisable to program such contour as a subprogram. Name subprogram “RADIUSTAPER.SUB”. Setting of variables <XD2>, <ZL1>, and <ZL2> is made within this subprogram; other variables are set in the main program.

Subprogram RADIUS-TAPER.SUB
%
ORT01
N1000 XD2=XD1 + 2*[V11 - DIS2] 2L2=ZL3 + DIS4 \$ ZL = AL2 + DIS3
N1001 G01
N1002 G02 X=XD2 Z=ZL2 L=V11
N1003 G03 X=XD3 Z=ZL3
N1004 RTS

- Variables are set in block N1000.
- Z coordinate of point “a” is specified in block N1001.
- X and Z coordinates of “b” and arc radius are specified in block N1002.
- X and Z coordinates of “c” are specified in block N1003.
- RTS in block N1004 indicates the end of the subprogram.

(3) Prepare the cutting program as a main program.

File name of the main program is “FLANGE-1.MIN”. LAP and nose radius compensation are used in this program.

The main program is shown below.

Main Program
\$ FLANGE-1.MIN
%
O100
N101 V10 = 15 V11 = 16 XD1 = 110 XD3 = 90 ZL3 = 32
N102 G00 X800 Z300
NLAP1 G81
N103 G00 X76 Z137
N104 G42 G01 Z135 F0.2
N105 G75 X80 L2
N106 G01 Z115
N107 G75 X=XD1 - 6
N108 X=XD1 Z112
N109 CALL ORT01 DIS1 = [XD3 - XD1]/2 D1S2 = V11*SIN [V10] \$ DIS3 = V11*COS [V10] DIS4=[DIS1 + DIS2 - V11]*TAN [V11]
N110 Z-2
N111 G40 G00 X200 110
N112 G80
N120 G00 X800 Z300
N121 G96 X202 Z137 S110 TO10101 M03 M08
N122 G85 NLAP1 D6 F0.35 U0.2 W0.1
N123 G00 X800 Z300
N124 S130 T020202
N125 G87 NLAP1
N126 G00 X800 Z300
M05 M09
N128 M02

Note: The subprogram ORT01 is called by the command in block N109. ORT01 defines the contour consisting of circular arc and a taper. Variables used for defining this contour are set in this block.

Variable Programming Flange Example

```
G13
(FLANGE-1.MIN)
G50 S4000
N100 V10=15 V11=16 XD1=110 XD3=190 ZL3=32
N102 G0 X800 Z800
NLAP1 G81
N104 G0 X74 Z137
N106 G42 G1 Z135 F.2
N108 G75 X80 L2
N110 G1 Z115
N112 G75 X=XD1 L2
N114 Z112
N116 CALL ORT01 DIS1=[XD3-XD1]/2 DIS2=V11*SIN[V10]
$ DIS3=V11*COS [V10] DIS4=[DIS1+DIS2-V11]*TAN[V10]
N118 Z-2
N120 G40 G0 X200 110
N122 G80
N124 G0 X800 Z300
N126 G96 X202 Z137 S220 T010101 M3 M8
N128 G85 NLAP1 D10 F.45 U2. W.1
N130 G0 X800 Z800
N1332 S130 TO20202
N134 G87 NLAP1
N136 G0 X800 Z800
M138 M5 M9
N140 M2
```

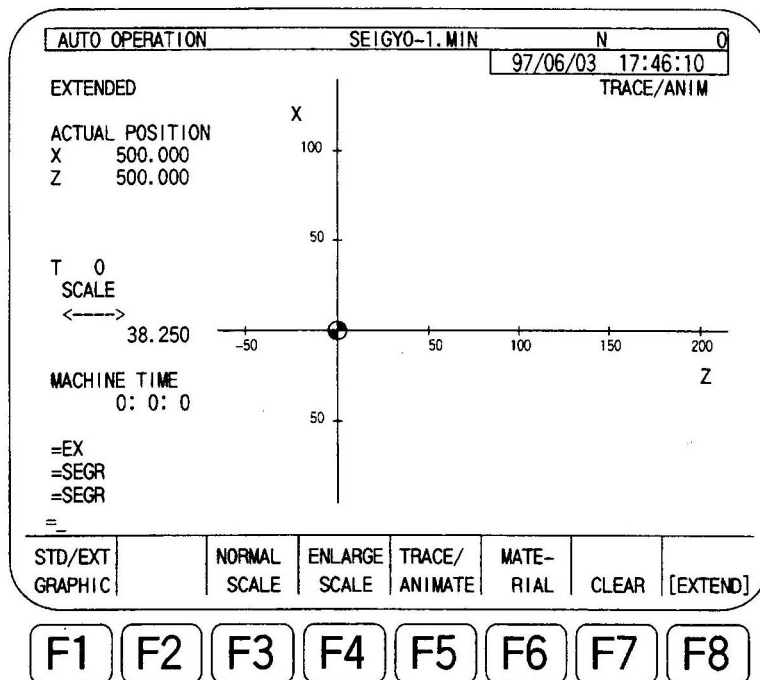
```
ORT01
N100 XD2=XD1+2*[V11-DIS2] ZL2+ZL3+DIS4 ZL1=ZL2+DIS3
N102 G1 Z=ZL1
N104 G2 X=XD2 Z=ZL2 L=V11
N106 G1 X=XD3 Z=ZL3
N108 RTS
```


GRAPHIC COMMANDS

ANIMATION MODE DISPLAY

Function Keys Used for Graphic Display Operation

The graphic display is possible in the auto, MDI and manual operation modes.

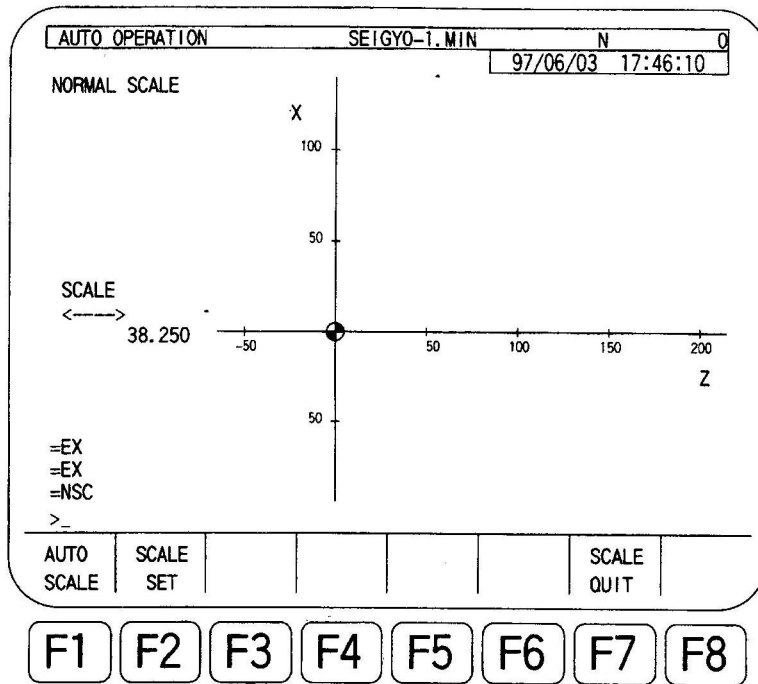


- (1) F1 (STD/EXT GRAPHIC)

This toggles the graphic display mode between NORMAL SCALE and ENLARGE SCALE.

NOTICE: During the execution of a program, it is not possible to change the graphic display mode screens.

F3 (NORMAL SCALE)



This function selects the unit length of an axis on the graphic display. On the graphic display, a dotted line with arrow marks at both ends is displayed with SCALE indication and scale value. This represents the scale length.

The normal scale is set by two different methods:

- (a) Automatic determination
- (b) Arbitrary setting

The procedure for setting the normal scale is explained below. The normal scale graphic display screen should be selected for setting the normal scale.

- (c) Automatic determination

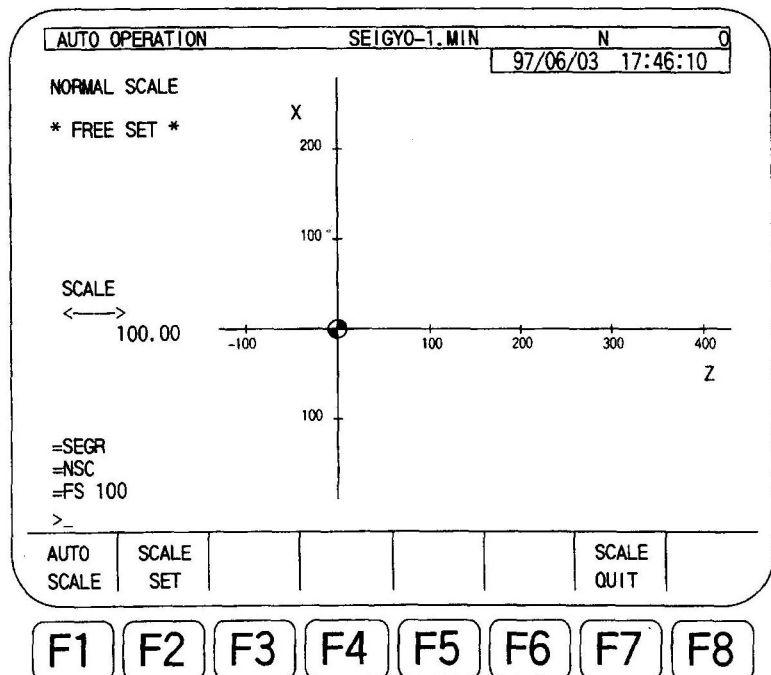
The graphic scale is automatically determined by pressing function key [F1] (AUTO SCALE). When this key is pressed, the program is read up to the end of the program (M02) and the scale is determined automatically so that the machining area can be displayed on the CRT.

- [Supplement]A Before executing auto scaled, select the desired program.
- B If the following commands are used in a part program, they are executed when the program is read by the pressing of function key [F1] (AUTO SCALE)
- READ, WRITE, GET, PUT, DELETE, SAVE and DEF
- C If output variables, system variables, and common variables are used as the left member of an expression in a part program, they are rewritten by the auto scale operation.
- D If the scheduled operation mode is selected, automatic determination of the scale value is possible only for the program selected by the first PSELECT command.
- E If an alarm occurs during the reading of a program for automatic determination, or if the program execution order designing commands and automatic determination of the scale is not completed, reset the NC.
- F The drawing area is automatically determined from calculation from the following formula.

$$(\text{Operating area in the cutting G code}) \times (100 + n)\%$$

Here, "n" is a factor to allow margin in the tool path drawing operation and set by OPTIONAL PARAMETER (ANIMATION) – Margin for automatic determination. ($0 \leq n \leq 100$)

(d) Arbitrary setting

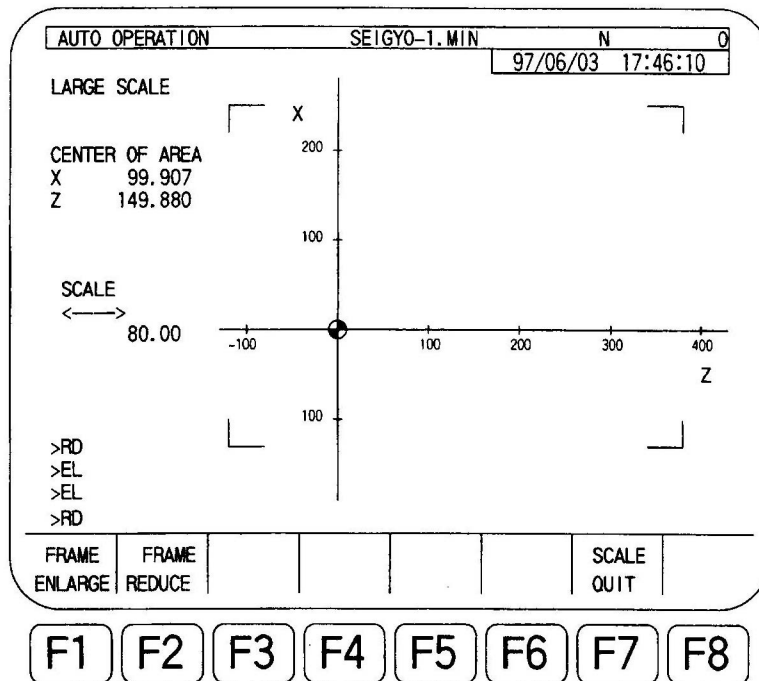


The scale value can be directly entered through the keyboard after pressing function key [F2] (SCALE SET). In this scale setting, setting range is from 12.5 mm (0.49 in.) to 1250 mm (49.21 in.).

The position of the coordinate axes can be set at a required position by using the cursor after setting the scale value.

The normal scale setting operation ends by pressing function key [F7] (SCALE QUIT)

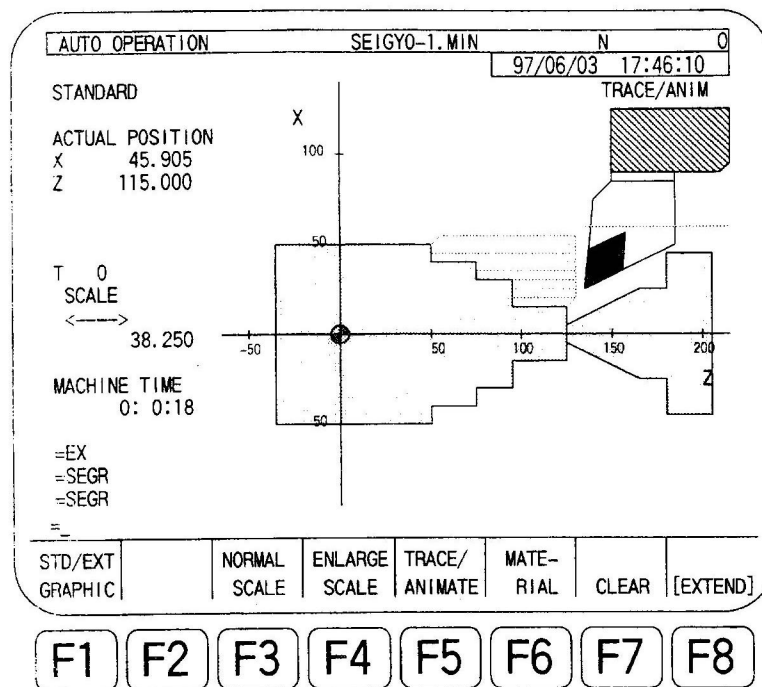
F4 (ENLARGE SCALE)



The display area to be enlarged can be set as needed.

Pressing function key [F1] (FRAME ENLARGE) reduces the size the display area frame. Pressing function key [F2] (FRAME REDUCE) enlarges the display area frame. Setting the position of the display area frame is done using the cursor key. The enlarge scale setting function terminates when function key [F7] (SCALE QUIT) is pressed.

F5 (TRACE/ANIMATE)



The graphic display mode is selectable from the three modes indicated below by pressing function key [F5] (TRACE/ANIMATE). Note that selection of the graphic display mode must be made before starting the operation. The selected mode cannot be changed during the operation.

(a) TRACE/ANIMATE mode

In this mode, tool shape, chuck shape, blank shape, and tail stock spindle shape is displayed in addition to programmed tool paths. The blank shape is modify in accordance with the progress of the part program.

(b) TRACE mode

In this mode, only the programmed tool paths are drawn.

(c) ANIMATE mode

In this mode, tool shape, chuck shape, blank shape and tail stock spindle shape are displayed and blank shape is modified in accordance with the progress of the part program.

Display of the tool paths is not available

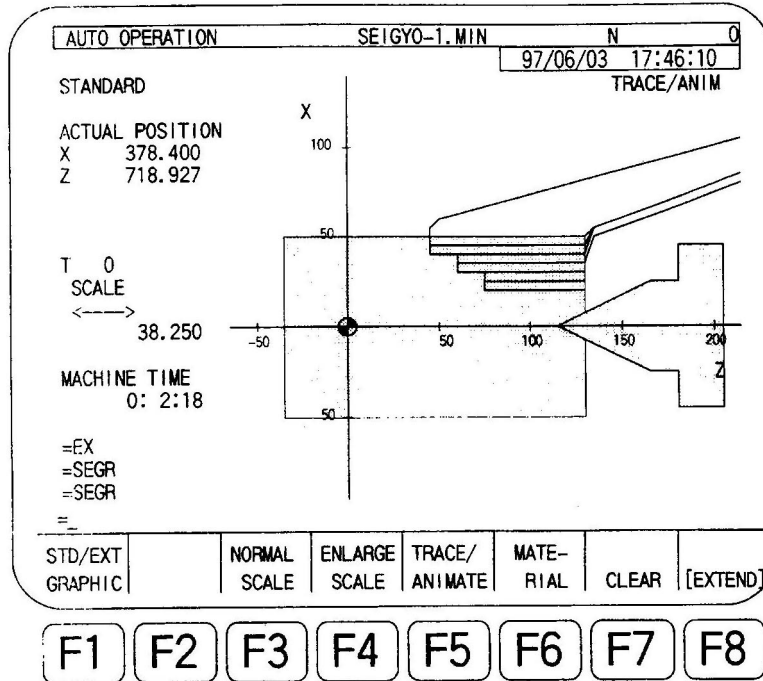
F6 (MATERIAL)

The blank shape, chuck shape and tail stock spindle shape are displayed.

F7 (CLEAR)

Tool paths, blank shape, chuck shape and tail stock spindle shape displayed on the CRT are cleared.

Display Page



Note 1: Data in item "XB" and "ZB" is available for two-saddle models

Note 2: Data in item "C" is available for the multi-machining option.

Machine Time Display

Actual program execution time in the automatic and MDI mode operation is counted and displayed. Time spent in the slide hold mode or the program stop mode is not included in the execution time.

In the high-speed drawing mode (machine lock and dry run on), the machine time display is not available. If machine time calculation is required through the execution of the high speed drawing operation, an optional function is required.

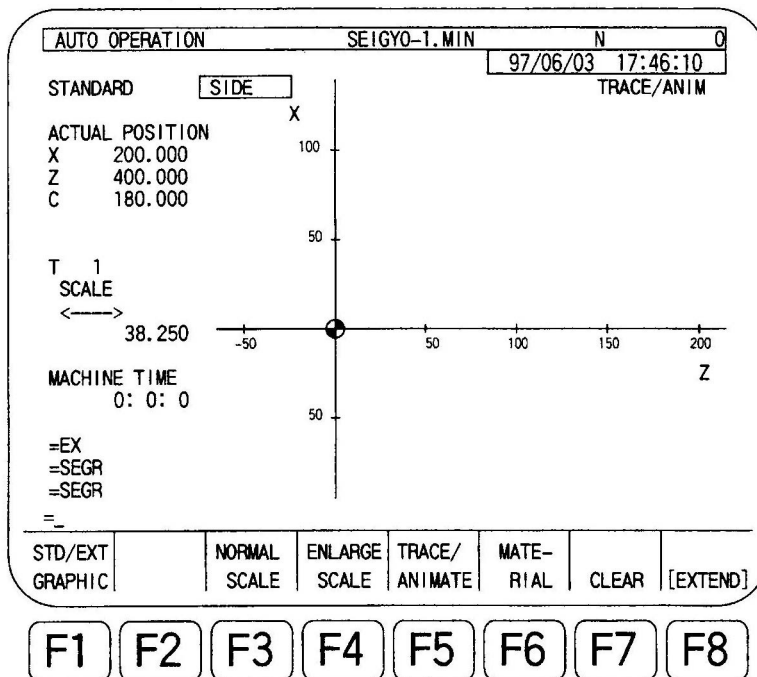
Additional Functions for Multi-machining Models

The available function keys are basically the same as used for the standard models.

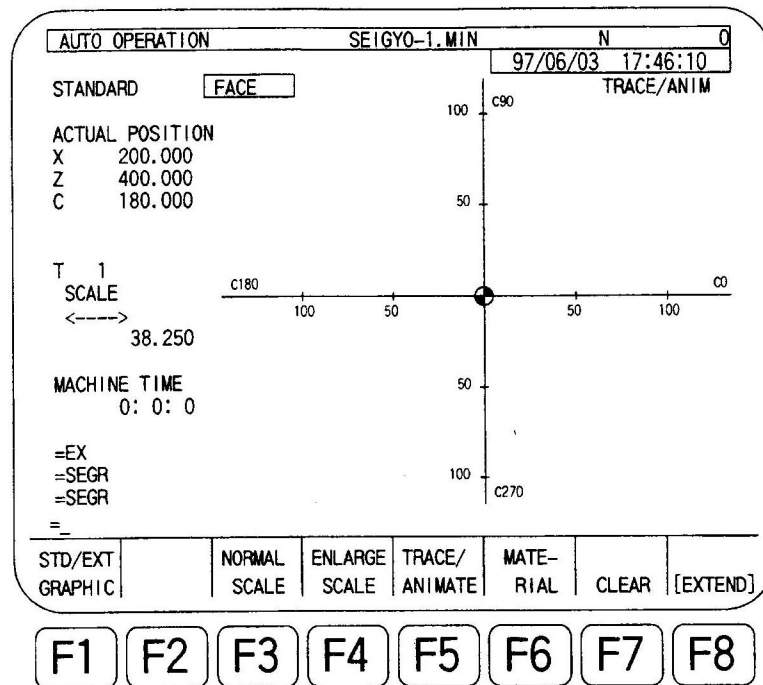
(1) Standard/Enlarged Graphic Display

(a) Standard graphic display mode.

For the blank display, the page key is used for switching the view angle from the side to the front and from the front to the side. This display change is possible even during the progress of cutting operations. Note that the page key is not effective for other than the multi-machining models.



Side View



Front View

The front view is displayed in a coordinate system in which the C-axis angle is fixed as shown above. The indications on the scale represent X-axis values (in radius)

(b) Enlarged graphic display mode

In the enlarged graphic display mode, switching the view angle between the front and side views is possible.

(2) Normal Scale Setting

Independent scale setting on the side and front views is possible. For setting the scale for these two views, first display the required view. When the same scale setting is used for both of the displays in common, then the setting is necessary only from either of the displays.

Setting procedure is the same as used with standard models.

(3) Enlarged Scale Setting

In the enlarged graphic display mode, independent scale setting for the two different displays – front and side is possible. As in the scale setting from the standard display mode, display the required view and setting should be done from that display.

(4) Switching between Trace and Animation Displays

As with the standard models, function key [F5] (TRACE/ANIMATE) is used for selecting the three different display modes.

(a) Trace/Animate

Side View:

Tool shape, chuck shape, blank shape and tail stock spindle shape are displayed. Tool paths are also displayed in cyan and the blank is modified as the cutting progresses.

While the C-Axis is connected (only for A-turret tools in two-saddle model), the tool paths are displayed in magenta and modification of the blank will not occur.

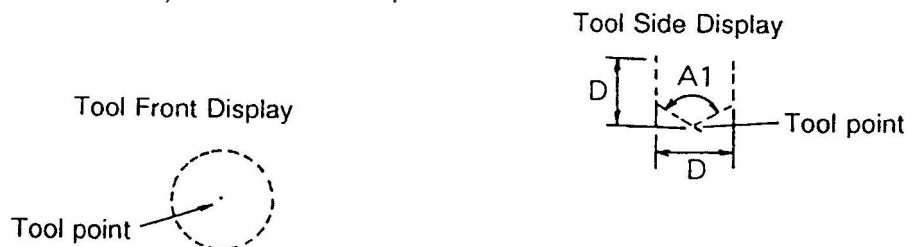
Front View:

The front view of the blank shape is displayed and the tool tip point is indicated by a red marker. The tool path is also displayed in magenta. During cutting, the tool outline is displayed *1. Modification of the blank does not occur.

The marker and tool outline are displayed only while the C-axis is connected.

Display of blank shape, tool path and tool outline is dependent on the setting of certain parameters (see 2-3, "Parameters").

*1 The tool outline is drawn in dots in magenta in accordance with the tool mounting direction, tool diameter (D) and cutting angle (A1) when the revolving tool (M2) is selected (tool classification code: 27-38) as the tool shape



The interval of the tool outline drawing is the distance the tool point has moved from the previous tool outline display position as indicated below:

“n” represents the factor to designate the tool outline drawing interval and is set OPTIONAL through PARAMETER (ANIMATION) – Display distance of tool contour within a range of 10 to 100.

The tool outline is also drawn on the display when the feed rate is changed from the rapid traverse to cutting feed rate or the desired point is reached (with C-axis connected).

(b) Trace

Side View:

The display shows only tool paths, chuck shape and tail-stock spindle shape. As in case a) above, tool path is drawn in magenta while the C-axis is connected (only for A-turret for two saddle model) and in cyan in other cases.

Front View:

The tool point position is identified by a red marker and the tool outline is drawn in magenta.

(c) Animate

Side View:

Tool shape, chuck shape, blank shape and tail stock spindle shape are displayed. The tool path display is not available. Modification of blank occurs as the cutting progresses. Note that when the C-axis is connected (only for A-turret for two-saddle model), modification of the blank does not occur.

Front View:

Same as in case (a) above, but the tool path is not displayed.

(5) Blank Drawing Function Key

Side View:

The blank shape is displayed.

This function key is effective only on the page currently displayed. For

example: pressing this key while the side view is being displayed displays only the side view of the blank.

(6) Delete Function Key

Side View:

Tool paths, blank shape, chuck shape, the tail stock spindle shape and tool shape displayed on the side view page are all deleted.

Front View:

The tool outline drawing and the blank shape displayed on the front view page are deleted

This function key is also effective only on the page currently displayed.

Additional Functions for 2-spindle Models

The available function keys are basically the same as used for standard models.

(1) [F1] (STD/EXT GRAPHIC)

This key toggles the display mode between normal and enlarged. Note however, the display mode cannot be switched while machine operation is in progress.

(2) [F2] (MODE SELECT)

The display pattern for the standard/enlarge graphic display in each operation mode can be selected by pressing this function key. Select the display pattern before starting machine operation. The display mode cannot be switched while machine operation is in progress.

(a) Graphic display pattern

Display Patten	Display Screen	
1	Side view of the 1 st spindle	Front view of the 1 st spindle
2	Side view of the 2 nd spindle	Front view of the 2 nd spindle
3	Side view of the 1 st spindle	Side view of the 2 nd spindle
4	Front view of the 1 st spindle	Front view of the 2 nd spindle

In display patterns 1 and 2, switching between the side view and the front view is possible using page keys even during machine operation.

In display patterns 3 and 4, switching between the 1st spindle and the 2nd spindle is not possible using page keys during machine operation. It is switched automatically by G140/G141 during operation.

- (b) When the multi-machining specification is not selected, patterns 1 through 3 can be selected by pressing function key [F2] (MODE SELECT). The front views for patterns 1 and 2 cannot be selected using the page keys.
- (c) Patterns 1 through 3 can be selected by pressing function key [F2] (MODE SELECT). The front view for pattern 2 cannot be selected using the page keys.
- (d) Patterns 1 through 4 can be selected by pressing function key [F2] (MODE SELECT).

(3) Normal Scale Setting

Independent scale setting on the side and front views is possible. To set the scale for these two views, first display the required view.

Independent scale setting for the 1st and 2nd spindles is not possible.

When the same scale setting is used for both of the displays in common, then setting is necessary only from either of the displays. In this case, specific scale value is set both for the front and side views.

The setting procedure is the same as used with standard models.

(4) Enlarged Scale Setting

In the enlarged graphic display mode, independent scale setting is possible for the two different displays – front and side. As in the scale setting from the standard display mode, display the required view and setting should be done from that display.

(5) Switching between Trace and Animation Displays

As with the standard models, function key [F5] (TRACE/ANIMATE) is used for selecting the three different display modes.

- (a) Trace/Animate
- (b) Trace
- (c) Animate

For details, refer to 2-1, and (4) in 2-7.

(6) Blank Drawing Function Key

Side View:

Blank shape, chuck shape, and tail stock spindle shape are displayed.

Front View:

Blank shape is displayed

This function key is effective only for the page currently displayed. For example, pressing this key while the side view is being displayed displays only the side view of the blank.

(7) Delete Function Key

Side View:

Tool paths, blank shape, chuck shape, tail stock spindle shape and tool shape displayed on the side view page are all deleted.

Front View:

The tool outline drawing and the blank shape displayed on the front view are deleted.

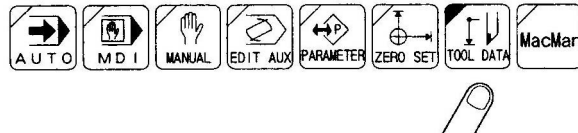
This function key is effective only for the page currently displayed.



TOOL FORM SELECTION

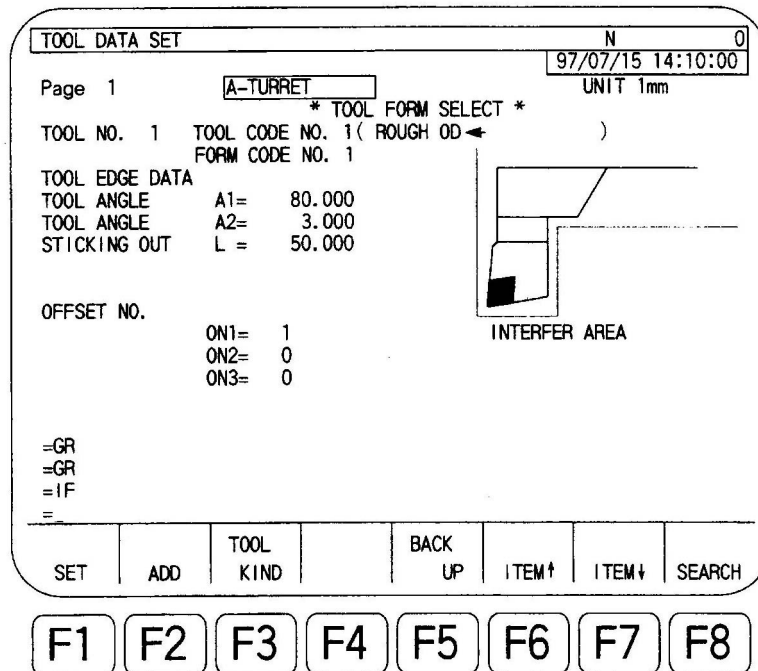
The procedure to set the tool form used in animation display is explained below.

Tool form data must be set in advance for all tools that are used in the program. However, when a tool animation data command (commanded using system variables) is specified in a program such as the program made using the IGF function, the tool form data is automatically set when the program is executed. Thus the setting of tool form data is not necessary in this step.

- (1) Select to TOOL DATA SET mode.



- (2) Press function key [F7] (ITEM↓) to display the TOOL FORM SELECT page.
- (3) Using the PAGE keys ;  , display the screen for the tool number to be set.



NOTICE: In the initial state, an OD turning tool is set as the tool form.

- (4) Press function key [F3] (TOOL KIND) to display the page from which the tool code number can be set.

TOOL DATA SET N 0
97/07/15 14:10:00

Page 1 A-TURRET UNIT 1mm

* TOOL FORM SELECT *

TOOL NO. 1 TOOL CODE NO. 1 (ROUGH OD ←)
 FORM CODE NO. 1

- TOOL CODE TABLE -

NO.	NO.	NO.
1 ROUGH OD ←	10 FINISH OD →	19 GROOVE OD ↓
2 ROUGH ID ←	11 FINISH ID →	20 GROOVE ID ↑
3 ROUGH FACE ↓	12 FINISH FACE ↑	21 GROOVE FACE ←
4 ROUGH OD →	13 THREAD OD ←	22 DRILL, HSS
5 ROUGH ID →	14 THREAD ID ←	23 DRILL, CARBIDE
6 ROUGH FACE ↑	15 THREAD FACE ↓	24 DRILL, CENTER
7 FINISH OD ←	16 THREAD OD →	25 RECESS OD ↖
8 FINISH ID ←	17 THREAD ID →	26 RECESS ID ↗
9 FINISH FACE ↓	18 THREAD FACE ↑	

=GR
 =IF
 =TK
 tool item code NO. 13

SET	ADD	TOOL KIND		BACK UP	ITEM ↑	ITEM ↓	SEARCH
-----	-----	--------------	--	------------	--------	--------	--------

F1

F2

F3

F4

F5

F6

F7

F8

- (5) Input TOOL CODE NO.

If keys [3] and WRITE are pressed, the display indicated below is displayed.

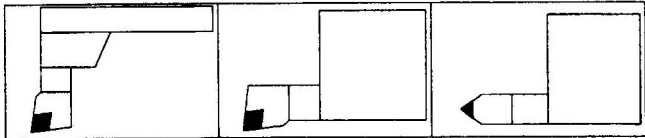
TOOL DATA SET N 0
97/07/15 14:10:00

Page 1 A-TURRET UNIT 1mm

* TOOL FORM SELECT *

TOOL NO. 1 TOOL CODE NO. 1 (ROUGH OD ←)
 FORM CODE NO. 1

- FORM PATTERN FIGURE -



=IF
 =TK
 tool item code NO. 13
 tool form code NO. 13

SET	ADD	TOOL KIND		BACK UP	ITEM ↑	ITEM ↓	SEARCH
-----	-----	--------------	--	------------	--------	--------	--------

F1

F2

F3

F4

F5

F6

F7

F8

(6) Input the FORM CODE NO.

If the form code "3" is specified, the display will change as shown below.

The screenshot shows a CNC control interface titled "TOOL DATA SET". At the top right, it displays "N 0" and a timestamp "97/07/15 14:10:00". Below this, it says "Page 1" and "UNIT 1mm". The main display area contains the following text:

A-TURRET
* TOOL FORM SELECT *
TOOL NO. 1 TOOL CODE NO. 3 (ROUGH OD ▾)
FORM CODE NO. 3
TOOL EDGE DATA
TOOL ANGLE A1= 80.000
TOOL ANGLE A2= 3.000
STICKING OUT L = 50.000
OFFSET NO.
ON1= 1
ON2= 0
ON3= 0
=TK
tool item code NO. !3
tool form code NO. !3
=

To the right of the text is a diagram of a tool tip with a rectangular block representing the workpiece. The area where the tool tip overlaps the workpiece is shaded and labeled "INTERFER AREA".

At the bottom of the screen, there is a row of function keys: SET, ADD, TOOL KIND, BACK UP, ITEM↑, ITEM↓, and SEARCH. Below this row are eight F-key buttons labeled F1 through F8.

(7) From this page, set the TOOL EDGE DATA by locating the cursor at the required data position.

TOOL ANGLE A1
EDGE ANGLE A2
STICKING OUT L

Supplement: The tool interference area is automatically created according to the selected tool form.

For the automatic interference area creation, values specified by parameters are used. If required, change these values.

(8) Input the offset number of the tool by locating the cursor at the required OFFSET NO (ON1-ON3) data position.

For a system having the tool life management specification, ON1 – ON3 corresponds to the offset group 1-3, respectively. On the tool life management table, tool offset numbers identical to those set in the tool life management table are automatically set.

MATERIAL BLANK

It is important to test your program graphically before cutting the first piece

FORMAT:

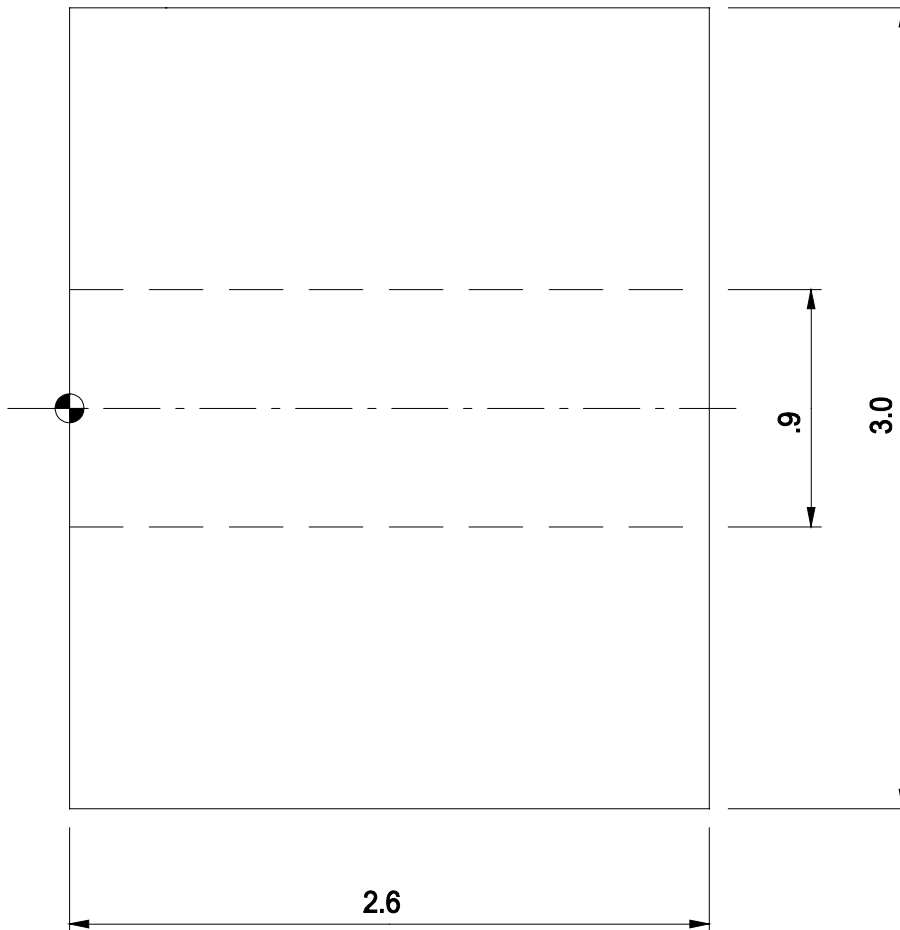
```
CLEAR
DEF_WORK
PS_(REF.ENDFACE), [Z(VALUE),X(VALUE)], [Z(VALUE), X(VALUE)]
END
DRAW
N 1 G50 S3000
```

DEFINITIONS:

CLEAR	Enter this as the first 'graphic' line in your program. The command will erase the graphic material from the last part cycle. Every time CYCLE START is pressed the CRT screen will be cleared.
DEF WORK	DEFine WORK serves to tell the control that material blank data follows. Note there must be a space between the words DEF and WORK
PS_	Abbreviation for PAINT SCREEN. The accompanying data sets up the drawing position, area, and shape. Note that the PS must be followed by a space
(REF ENDFACE)	There are two reference points – LC (left center) for a program with its zero point at the left endface; or, RC (right center) for a program with its zero point at the right endface
[Z(VALUE), X(VALUE)]	The “Z” and “X” distance from the program zero to the ref. Endface that was selected. Typically [0,0]
[Z(VALUE), X(VALUE)]	The absolute “Z” and “X” dimensions for the overall length and diameter of the part. These values must be expressed in the correct unit amount.

END	End of MATERIAL BLANK statements. Must be on a line by itself
DRAW	The defined MATERIAL BLANK is displayed on the CRT screen.

You may need to draw a MATERIAL BLANK that already has a hole in it. This requires two (2) PS statements. For this example, the first PS statement is for the overall blank size, and the second is to define the size of the hole through the center.



The PS statement will be:

```

N1 CLEAR
N2 DEF WORK
N3 PS LC, [0,0], [26,30]
N4 PS LC, [0,0], [26,9],0
N5 END
N6 DRAW

```


APPENDIX

Machining Processes

1. Roughing - a roughing operation is that machine process that is intended to remove a majority of stock material in an efficient manner.

The roughing operation is in most cases followed by either a semi finishing process, or a finishing process. The roughing process typically requires the most horsepower for work piece processing, due to the significant depth of cuts involved, and the more aggressive feed rate.

While the process of roughing is to quickly remove the excess stock, there are some basic guidelines that should be followed, such that the following processes can be completed without “prior process influence”.

For the most part, all roughing processes should be completed before any subsequent finish operation is performed.

Generally, roughing speeds and feeds can be increased until such time as the work piece begins to display some deficient “trait” that is indicative of the rough machining itself. Typical limitations in roughing can be the work holding ability to grip/contain the part under the heavy cutting conditions.

If the part even displays a minor ability to move in the chuck during roughing, either the work holding or the program feed/speed needs attention. The user is cautioned that for those work pieces that are not completely located on the end face surface against the chucks jaws or stop blocks are fundamentally prone to moving if aggressive part processing is attempted. Merely increasing the chuck clamping pressure will do little, other than deform the shape of the work piece. With this in mind, the user needs to understand that occasionally the resultant part geometry produced by roughing process can effect the following finish operation.

In short, the “seasoned’ user is respectful of the fact that roughing does have its limitations, and indirectly can clearly impact the ultimate quality and successful completion of any following process.

The roughing process needs its own level of process stability.

2. Semi Finishing - semi finishing is that operation which can be required to allow better ‘sizing’ control for the final finishing tool, and its corresponding part features. Basically, using a semi finish operation is required for those part features that contain very tight tolerance part features.

In essence, it affords the user the ability to ‘pre qualify’ a specific part feature, such that a consistent cutting condition is presented to the finish tool, and the net (finish) result in final size can be controlled more in terms of size, surface consistency, and longer tool life.

In other cases, a semi finish process is necessitated to establish the complete profile, such that a single subsequent finishing tool can size the complete feature without any tool/part interferences.

Generally, semi finishing operations are only employed in those processes that cannot successfully be attained by conventional rough/finish part processing.

- 3 In general, finishing processes involve light depths of cut, significant increases in RPM, and a marked decrease in the tools feed per revolution, proportionate to the parts requirement of surface finish.

Oftentimes, the chuck clamping pressure is significantly reduced to eliminate the tendency for the part feature to reflect the work holding contact points.

4. Drilling - a drilling process is a very straight forward machine operation. Generally, drilling is performed to produce either a through hole or a 'blind' hole.

In some applications, a drilling process is done simply to allow access of another tool. There are many different styles of drills available, and for the most part, there are pros/cons for each, relative to the process at hand.

Typically, a drilling process is preceded by a centering process, which aids the conventional pointed drill to maintain its position, and greatly reduce a drills tendency to 'walk'.

Similar to the case in roughing, a given drilling process must be respective of any follow-up operations, as the surface condition, etc., of the drilled hole could have an effect on the following tools overall capabilities.

5. Boring - boring is that process which creates those internal, cylindrical shaped part features. Typically, boring is preceded by a drilling process which establishes the basic internal feature, and indirectly allows access for the boring tool itself.

Generally, boring is performed to achieve geometrical roundness, and establish an internal part features actual size, and desired profiles.

Many boring operations are unique in that these operations are always subject to some level of cutting chips either remaining in the bore, and or the presence of chips interfering with the smooth creation of the internal bore feature profile. This is especially true of those boring operations that are performed in blind holes.

To overcome this, oftentimes the user can utilize an internal coolant fed boring bar, which allows for the coolant to 'exit' at the tools tip. This inherently aids in effectively 'back flushing' the chips out of the hole.

Additionally, the use of a higher coolant pressure level can complement the internal coolant fed boring bars ability to evacuate those chips prone to remaining in the parts bore due to other adjacent cavities, voids, and the impact of centrifugal force during part rotation.

6. Threading - threading is that process which creates either an external or internal helical profile that is commonly referred to as a thread. In most cases, the final profile of the thread form is built into the tool or insert itself.

For those threads that have a considerable tooth depth, it is wiser to employ another common turning tool holder to effectively rough out the basic thread prior to using the threading tool itself. This can greatly increase the threading tools useful life.

Based on the actual profile of the desired recess feature, it oftentimes can be created with an existing tool in the basic process, thereby negating the need to add another tool to the overall process.

Recessing is oftentimes synonymous with relieving.

Threading in general is somewhat prone to having an associated chip control problem. This can be offset by determining what effects a change in infeed pattern may have on the chips size, length, and natural direction of the chips exiting path/route tendency.

In recent years, insert manufacturers have made excellent advancements in the threading arena. Many tools now exist that can provide effective chip control, and evacuation especially in those tight internal applications.

In general, threading is performed at approximately two thirds the surface speed of general turning. Excessive speed must be limited, as the basic profile of most threads includes a rather sharp (root) point angel, which can effectively burn up if too high a surface footage is employed.

7. Recessing - recessing is that process, either external/internal, that generally is performed to allow for either reasons of clearance to a mating component, or, is done to provide an area such that a subsequent operation or tool can access the adjacent part feature(s).

Based on the actual profile of the desired recess feature, it oftentimes can be created with an existing tool in the basic process, thereby negating the need to add another tool to the overall process.

Recessing is oftentimes synonymous with relieving.

8. Grooving - grooving is that process, either external/internal, that generally is performed to allow for the placement of a component such as an 'O' ring, or to allow a mating part to occupy the space when assembled.

To a lesser extent, grooving is a form of recessing. There is a tremendous amount of groove, or groove type requirements in a variety of work piece configurations. Therefore, there exists a barrage of dedicated products that comprehensively address virtually any desired groove, its profile, and its associated tooling/insert needs.

There even exist grooving tools that can perform some limited general turning work, thereby offering the user higher flexibility for tool layout, process, and net cycle time.

9. Centering - this process allows the creation of a 'spotted' center drilled feature in the work piece, such that a following drilling (point) will have some aid in maintaining the desired position.

