

B-01: Program CNC Lathe

SAFETY FIRST

- Follow all Caterpillar facility safety standards when performing this task.

EQUIPMENT

- Mazak QuickTurn 28N

RESOURCES

- Mazatrol T32-2 Operating Manual
- Mazatrol T32-2 Programming Manual

Program CNC Lathe

1. Power up the CNC lathe.

- Turn ON the main power switch, below the control panel.
- Press the power button to turn ON the control panel power.

2. Set the machine zero.

- Press the <Home> key.



Red Light Will Illuminate
When Button is Pressed

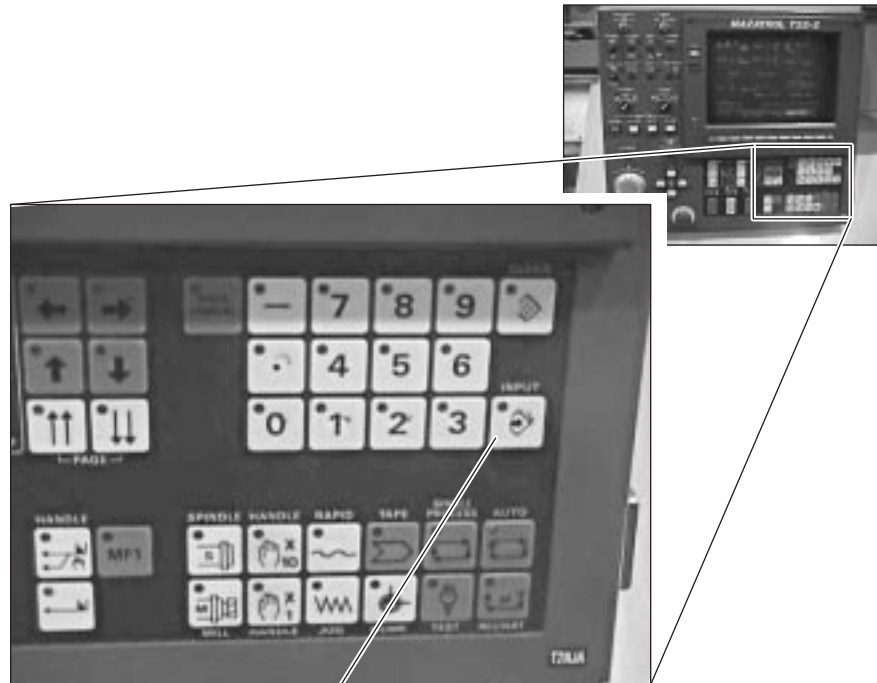


- Home the X axis first to avoid damaging the machine.
 - Press the manual control (Ÿ) positive X axis feed until the X axis home value reads zero on the control panel.
 - Press the manual control (Ɔ) positive Z axis feed until the Z axis home value reads zero on the control panel.
3. **Press the left green softkey to view the menu options.**
 4. **Select an unused program number to start a new CNC program.**
 - Press the <Program File> softkey to view the existing program numbers (Wno.).

Enter General Piece Parameters

1. **Press the left green softkey to begin entering the first process parameters.**
2. **Press the <Program> softkey to begin programming.**
3. **Press the <Work No.> softkey to select a work number.**

4. Type the new work number (Wno.), and press the <Input> key.



Press Input to
Enter Values

Press the <Input> Key After Typing a Value

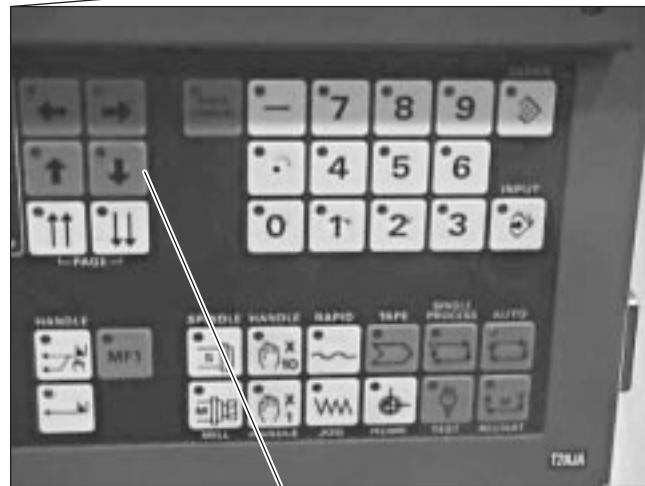
PNo. 0 - Enter Part Parameters

1. Press the <Mazatrol Program> softkey to display the beginning program screen.
2. Press the corresponding material (MAT) softkey.
 - Use the process sheet or print to determine the material type, if necessary.
3. Type the value for the largest outside diameter (OD-Max), and press <Input>.
4. Type the value for the smallest inside diameter (ID-Min), and press <Input>.
5. Type the length of the piece, and press <Input>.
6. Type the maximum spindle RPM limit (rpm), and press <Input>.
7. Type the amount of finished stock desired on the X axis (Fin-X), and press <Input>.
 - Typically the finish stock value is .01 to .002 inches.
8. Type the finish allowance - Z axis (Fin-Z), and press <Input>.
 - The finish allowance is typically about .003 inches.
9. Type the stock removal of work face, and press <Input>.

Enter the First Operation (PNo1) Parameters (Turn on the Coolant)

1. Press the next to last button, on the far right.
2. Press the <M Code> softkey.
3. Type 8 and press <Input> to turn on the coolant.

4. Press the down (↓) arrow key to begin a new process (PNo.).



Press to Start a New Process

PNo2: Select the First Machining Process (Edge Face)

1. Press the <EDG> (Edge Face) softkey.



Press this Key
to Cut an Edge

2. Press the <Auto Set> softkey to enter the mode data automatically.
 - Edit the roughing (maximum) depth of cut if necessary (it is often too heavy).
3. Type the value for the desired rough tool number, and press <Input> twice.
4. Type the value for the desired finish tool number, and press <Input> twice.

Entering the Data for Sequence 1 in PNo2

1. Type the starting point for the X axis, and press <Input>.
 - On rough stock, the starting point should allow .100" clearance to avoid hitting the piece.

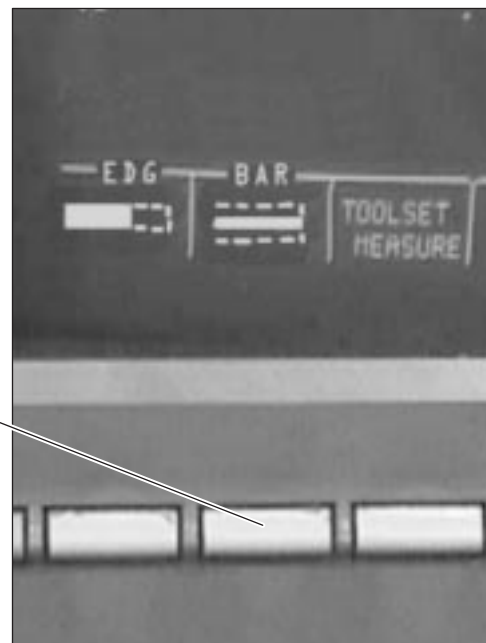


2. **Type the starting point for the Z axis, and press <Input>.**
 - The Z axis starting point is usually the work face value.
3. **Type the final point for the X axis, and press <Input>.**
 - The final point will be zero (0).
4. **Type the final point for the Z axis, and press <Input>.**
 - The final point for the Z axis should be zero.
5. **Enter the surface finish feed rate.**
 - Press the <roughness> softkey.
 - Press the <5> softkey.

PN03: Select the Next Machining Process (Bar –Turn Piece)

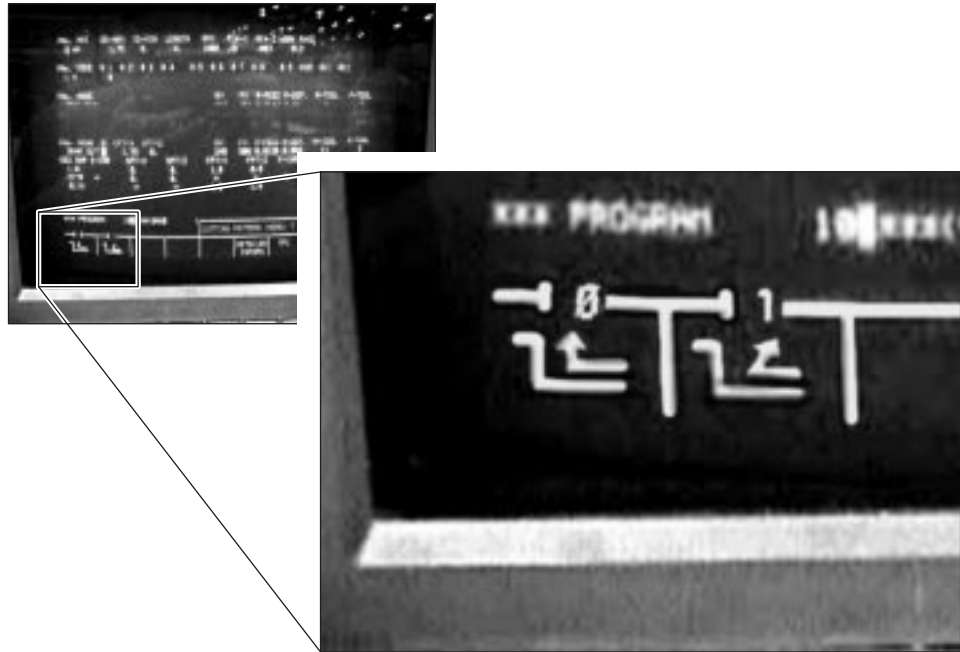
1. **Press the <Bar> softkey.**

Press this Key to
Select the Bar Option




2. **Press the In or Out softkey as required.**

3. Type 0 to select the cutting pattern.



Cutting Pattern Selection

4. Type the starting point for x (.100" from the OD-Max), and press <Input>.
5. Type the cutting point for Z (zero), and press <Input>.
6. Press the <Auto Set> key to enter the cutting parameters.
 - Change the R-Dep (Roughness Depth), if needed.
7. Enter the Roughing Tool number, and press <Input> twice.
8. Enter the Finish Tool number, and press <Input> twice.
9. Press the <Line> softkey. [Seq 1 for PNo3].
10. Specify a particular radius, if a chamfer is not required, and press <Input>.
11. Type the final point (finish diameter) – X, and press <Input>.

- 
12. Type the final point (finish length) - Z, and press <Input>.
 13. Type the final CNR-C (Corner or Chamfer) value, and press <Input>.
 14. Enter the surface finish feed rate.
 - Press the <roughness> softkey.
 - Press the <5> softkey.
 15. Continue entering sequences to the process until all sequences are complete.
 - Begin a new process anytime a tool change is required.
 16. Enter any other processes, as needed.
 17. Press the <Shape End> softkey to end the program.

Check the Program for Errors

1. Press the left green key.
2. Press the Program key.
3. Press the <Toolpath> softkey.
4. Press the <Check Continue> softkey to view the cut of the tool.
5. Correct any errors shown on the screen.
6. Visually observe for any setup problems in the program.
 - Verify that the tool clears the piece.
7. Press the left green softkey.
8. Press the <Set Up> softkey.
9. Press the <Simulation> softkey.

10. Press the <All Erase> softkey to erase the previous toolpath simulation.
11. Press the <Shape> softkey to draw the piece.
12. Press the right green softkey.
13. Press the Auto key at the control panel.
14. Press the <Simulation Start> softkey.

Verify the Tool Layout

1. Press the left green softkey.
2. Press the <Program> softkey.
3. Press the <Layout> softkey.
4. Verify that the tool operations are in logical order.
 - Check that rough cuts are all performed before the finish cut.