

F-01: Program/Edit CNC Mill

SAFETY FIRST

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

EQUIPMENT

- Hurco CNC Mill

RESOURCES

- Hurco CNC manufacturer's manual



Program/Edit CNC Mill

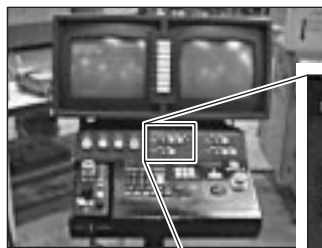
1. Power up the CNC mill.

- Turn on the main power on the back of the mill.
- Allow the processor to boot up.
- Press the <F8> key to exit.



Function Keys
Correspond to
Options on Screen

- Press the Machine Operations <Manual> key.

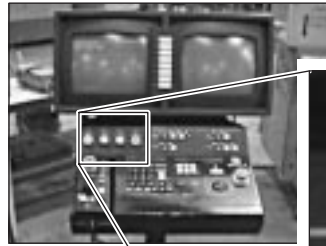


Press for Manual Control



- Press the <Power On> key.

- Press the <F7> key to reset servos and spindles.
- Press the <Start Cycle> button.



Press to Control Work Head



Note: You should now be able to control the CNC workhead manually.

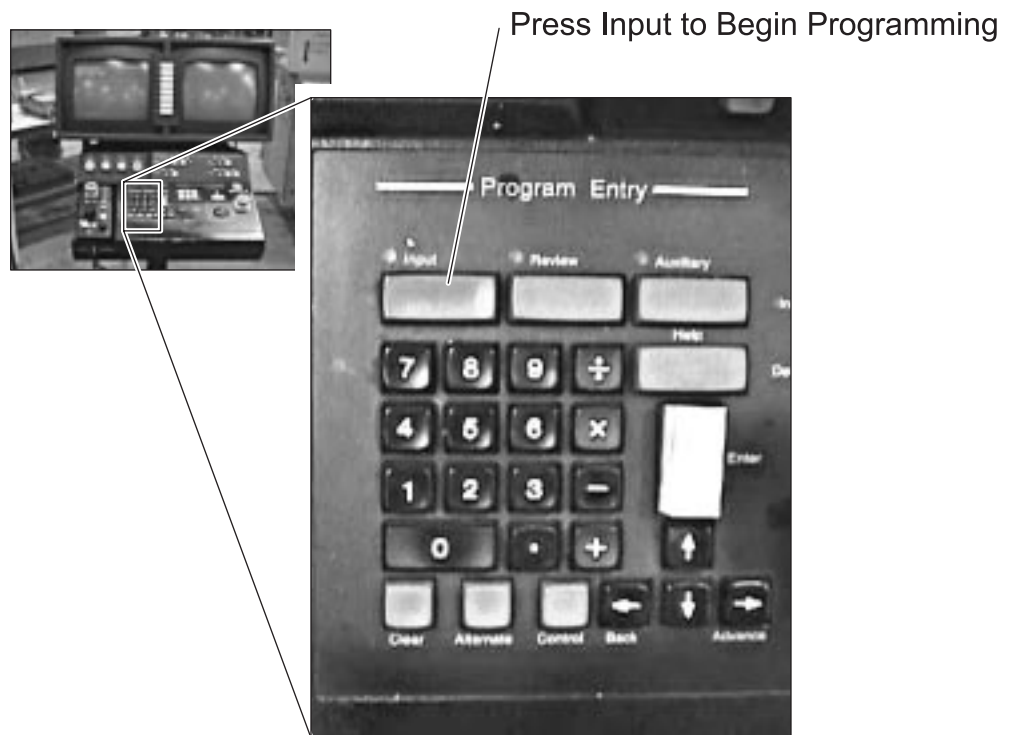
2. Calibrate the machine.

- Press the <F8> key to calibrate the machine.
- Press the <Start Cycle> key to begin calibration.
- Press the <Start Cycle> key again to calibrate the tool changer.

3. Specify the tool in the spindle.

- Press the <F1> key.
- Type the tool number from the Input screen.
- Press the <Tool Setup> key to view the tool parameters.

4. Press the Program Entry <Input> key to begin part programming.



5. Press the <F3> key to display the existing part program.
6. Delete the existing backed up program.
 - Press the <F1> key to delete a block.
 - Continue deleting blocks until "End of Program" is displayed.
7. Press the <F2> key to insert a block.
8. Specify the machining operation.

Note: The following steps are examples of the options used to program the CNC mill. The Holes selection is used to demonstrate the types of options.

- Press the corresponding machining operation key depending upon the operation required from the process sheet and print.



- Options include:
 - <F1> Position
 - <F2> Holes
 - <F3> Milling
 - <F4> Patterns
 - <F5> Miscellaneous
 - <F7> Rotary
- Press <F2> to select Holes.
- The subroutines for Holes include:
 - <F1> Drill Operations
 - <F2> Tap
 - <F3> Bore and Ream

9. Select <F1> Drill Operations.

- The subroutines for Drill Operations include:
 - <F1> Drill
 - <F2> Center Drill
 - <F3> Counter Bore
 - <F4> Spot Face
 - <F5> Counter Sink

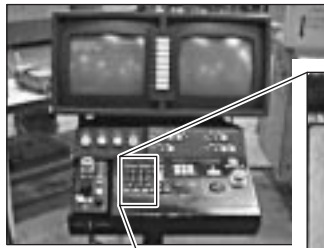
10. Select <F1> Drill.

11. Type the tool number and press <Enter>.

12. Enter the tool parameters, as required.

- Enter the X, Y, and Z Start and Bottom parameters.
- Enter the Tool Speed and Feed, if necessary.
- Enter the peck depth and type.

13. Press Advance (right arrow key) to proceed to the next menu.



Press Once to Proceed to the Next Menu. Press Twice to Start a New Block.

14. Press the <F6> key to select Locations.
15. Enter the X and Y cutting location parameters.
- Press the Graphics Screen button to view a drawing of the current cutting settings.
16. Press Advance (right arrow key) twice to start a new block for a new machining operation.
17. Repeat steps 7 through 16 until all operations are programmed.
18. Press the Machine Mode <Auto> key.
19. Press the <F4> key to "Check for Errors" in the program.
20. Press the <F8> key to "Run Program" when ready to begin machining a piece.

Note: To download the program, follow the procedures listed in the manufacturer's manual.