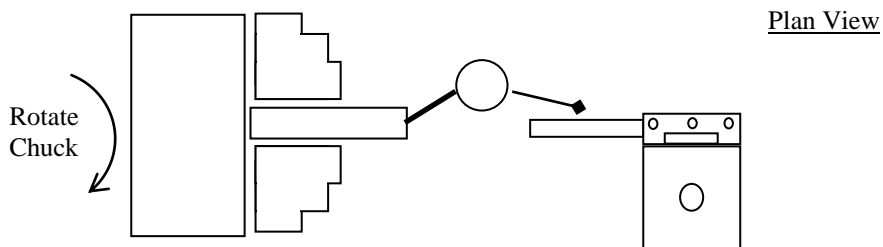




ALPHA PROCEDURE TO REFERENCE X AXIS

**PLEASE MAKE SURE YOU ARE IN CNC MODE
(PLEASE NOTE THAT THIS PROCEDURE IS ONLY TO BE USED WHEN ALARM 300 APC
X AXIS NEEDS ZRN IS DISPLAYED)**

1. Set any Workshift values to ZERO.
2. Put machine into MDI Mode
3. Press the Offset/Setting Key.
4. Press the Setting Softkey displayed on the screen and cursor to the PWE option.
5. Change the "0" setting to "1".
6. Press the "Input" key (This should display a P/S 100 alarm condition).
7. Press the System key.
8. Type in 1240, then press the Number Search Soft key, and make a note of the X axis value.
(Note 1)
9. Clock the X axis and manually move the machine to X zero position (spindle centreline).



10. Move the clock away from the bar in the Z axis direction only.
11. Press the POS key and then the REL Softkey. Press the U key and then the ORIGIN softkey, the screen should now read U 0.000.
12. Using the X axis handwheel wind the X axis slide until the U value displayed equals the figure recorded in parameter 1240.
13. Put machine into MDI Mode
14. Press the SYSTEM key.

15. Type in 1815, then press the Number Search Softkey.
16. Use the arrow keys to illuminate the X axis APZ column.
17. Change the value shown to 1.
18. Press the emergency stop button.
19. Turn the CNC off and back on
20. Press the Offset Setting Key.
21. Press the Setting Softkey.
22. Change PWE back to zero.
23. Type in 0 and then press the Input Key.
24. Release the Emergency Stop.
25. Press the green power "on" button.
26. Press the Reset button.

Note 1 : On later models this value may be set at 0.000, in this instance the reference position will be set on the centre line of the machine. Therefore ignore points 11,12 & 13