

Acorn Axis Driven Turret Instructions

This is a “universal” ACORN lathe tool turret PLC program and supporting files that facilitates the operation of a lathe tool turret. This document covers the axis driven turrets that uses the 4th axis as the turret.

V4.60+ Allows the use of the wizard for those who wish to use the beta ATC PLC.

Installation:

1. Install ACORN CNC12 V4.60+ on PC System.
2. (Optional) If upgrading restore report from previous version to retain Parameter settings.
3. Extract all files from centroid_acorn_v460_lathe_universal file into the [c:\cnc1](#) directory and overwrite all files.
4. Start CNC12 and open the Acorn Wizard. Assign Drive Type and Required I/O for your system. The Axis Driven Turret has no specific I/O required for its operation.
5. The turret axis is default on axis 4. Set the wizard axis settings for the 4th axis to Letter “A”. Ensure that the turns/rev is correctly set so that 1 = 1 turret position.
6. Set the following CNC12 parameters. From the main screen press F1(Setup)→F3(Config)→137(Or User specified password)→F3(Params).
 - A. Parameter #6 = 1
 - B. Parameter #160 = 1 (Sets ATC to non-random type meaning it puts tools back into same positions)
 - C. Parameter #161 = Maximum number of tool positions
 - D. Parameter #820 = 1 (Sets Machine type to Lathe)
 - E. Parameter #830 = 6 ****NEW**** (Sets what type of ATC you have for PLC and macros, values are subject to change in future so ensure to check documentation on every update)
7. The Turret must be initialized to work properly on every bootup with the use of the m18 command. Read the “Initialize the turret” section below for more information.

Initialize the turret:

Since the turret is axis driven, it will need to be homed or “initialized” just like any other axis for the control to know where it’s at. Provided is an M18 macro that will home the turret, by default the operator will be asked the current tool location of the turret and the turret will be homed to that tool. The M18 command can be edited into the cnc.hom file to home the turret during machine home. An M18 can be called anytime in MDI in the situation that the turret position was lost.

Indexing the turret:

The included M21 macro indexes the turret. The M21 macro determines the current tool location that it’s at and calls the tool change macro to go to the next tool location. If it’s at the maximum bin location then it goes to tool location 1. One can assign a free AUX button on the VCP for convenience to index the turret ‘manually’. M21 is simply “mapped” to a free AUX button. Mapping an M code to a free AUX button is covered in [Tech Bulletin #300](#). For example, if one is using the AUX8 button then parameter 195 will be set to 2111. That means that the custom macro, M21, will be “mapped” to that AUX button and now when the AUX8 button is pressed, CNC12 will run the custom M21 macro.

Since the Introduction of VCP 2.0, it is possible to reassign the turret index button to a “aux key” which will allow it to call the macro. Please read the documentation on [VCP 2.0](https://www.centroidcnc.com/centroid_diy/downloads/centroid_vcp_users_manual.pdf) if interested in doing so.
(https://www.centroidcnc.com/centroid_diy/downloads/centroid_vcp_users_manual.pdf)