

# Indexer Box Setup Procedures

## **Power up**

- Only connect/disconnect rotary table motor cables when the power is off.
- Use a 110-120V outlet at the wall.
- Do not use an under rated extension cord.

## **Manual operation setup**

- The rotary system will be ready to operate directly out of the box.
- Each line of code in the indexer memory will run with a single push of the START button.

## **M function signal setup**

- Your CNC must have M code finish functionality.
- Wire the provided cable from the indexer to the CNC control following the wiring diagram in the back of the manual.
- The state of the indexer “finishing relay” setting is “normally open” for most CNCs.
- Each line of code in the indexer memory will run with an M code signal from the CNC.

## **Air brake** (if available)

- 90 psi recommended
- Set “brake on/off delay” and the brake will automatically operate after every move.

## **Advanced settings (Needed for USB to CNC and DNC)**

### **RS-232 cable testing**

- You must use the supplied “Straight Thru” RS-232 serial cable.
- Measure the pin voltages using the RS-232 wiring worksheet.

### **RS-232 communication setup** (see pages 6-8)

- If you are currently using a PC to communicate to the CNC, use these same settings for the indexer.
- If your CNC control settings are unknown, we can make recommendations.
- Set indexer “baud rate”, “data/parity/stop”, and “flow/control” to match CNC settings.

### **“DNC from CNC” indexer setup** (see pages 7, 17)

- The RS-232 cable is used for data communication.
- The M function cable is used to signal cycle start and finish.
- CNC data must end with a carriage return, “end of block character” for DNC data.
- See “DNC from CNC” operation procedures in the manual.

### **“DNC to CNC” indexer setup** (see pages 7, 18)

- Your CNC must have the ability to receive DNC data.
- The RS-232 cable is used for data communication.
- The M function cable is used to signal cycle start and finish.
- Special M function signal setup: Save the indexer .INI file to a USB disk. Edit the .INI file with a PC text editor to enter the M code value that your CNC uses for signaling. This M code value will be sent to the CNC instead of the AB move.
- If your program already has % at start and end, then set setting “start Tx” and “end Tx” to “none”. Otherwise, you may need to verify that start and end text to your CNC needs a % or not.
- See “DNC to CNC” operation procedures in the manual.

**Important:** Save your .INI settings to a USB for each unique CNC application. (pages 9-10)