# Automatic Tool Changer Operations Manual

For machines equipped with a WinCNC controller

8/1/2015 Laguna Tools

# **Table of Contents**

Tool Changer Overview	3
Manually Changing Tools	3
Executing Tool Call Commands	3
Quick Start Guide for Generating Programs with Tool Calls	4
Storing Tool Locations	7
What happens during a tool change?	7

The automatic tool changer (ATC) adds another layer of automation to the CNC machine. The advantages of an automatic tool changer are as follows:

- ATC increases the machine's productivity.
- Helps to minimize total machining time.
- Increases the flexibility of the CNC machine.
- Improves tool carrying capacity of the CNC machine.

#### MANUALLY CHANGING TOOLS

Each machine equipped with an ATC will have a means to chuck or release a tool in the spindle manually. This is useful when the project only requires one tool and does not use the ATC.

Laguna Tools offers a variety of spindle options, but each spindle compatible with an ATC will have a button that releases a chucked tool. The button will be located on the spindle or on the CNC machine near the spindle.

In the picture on the right, the release button is on the front of the spindle.



Figure 1. Tool gripper release button.

#### **EXECUTING TOOL CALL COMMANDS**

Tool changes can also be made by executing tool call commands in the command line of the controller interface. The number of tools each ATC can support will vary between machines. The most common is an eight slot tool rack.

If the machine has eight tools, then each tool can be called by executing the command "T#" where the # represents a numerical digit from 0-8.

Changing tools only requires executing a tool call command.

- Prior to executing a tool change make sure any blades have been removed from the TCM-3 and EOT-2 modules. Long blades may collide with tool rack during tool changes.
- Enter the command "T#" where the # symbol represent a number 1-8. For example, the command "T8" will cause the spindle to grab tool 8.
- If there is a tool in the spindle when executing the "T8" command, the tool presently in the spindle will be parked first.

Tool Commands	Description
то	Unloads Tool.
T1-T8	Changes tool, activates last stored tool measure.

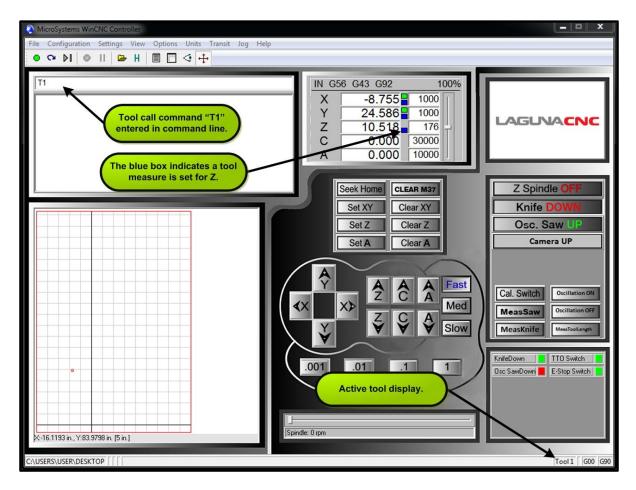


Figure 2. Screen capture demonstrating ATC fundamentals.

For a simple demonstration three concentric circles were drawn. Each circle has been assigned a different tool to cut along the perimeter of each circle. In figure 3 below is a screen shot of the Aspire software after creating three separate toolpaths, each using a different tool.

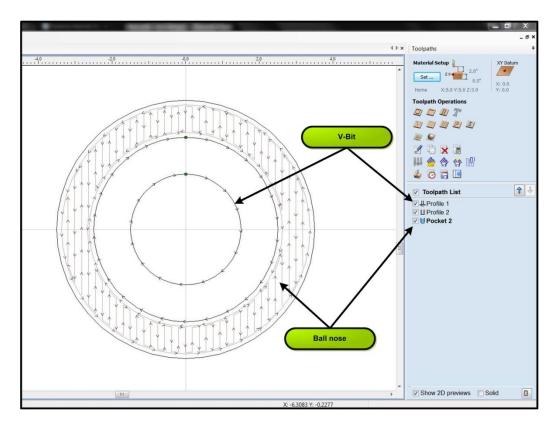


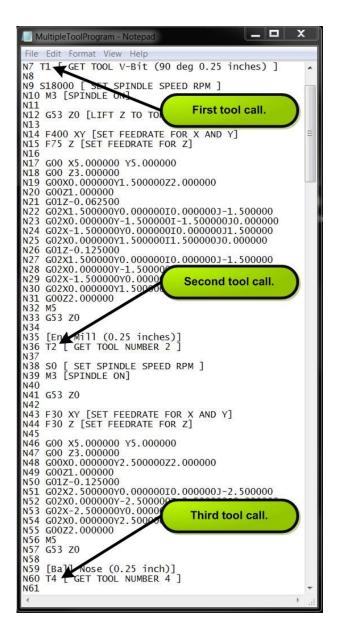
Figure 3. Demonstrating multiple toolpaths using different tooling.

	Tool Info		Tool	Info			Tool Info				
Notes Cometry Diameter (D) 0.25 inches Notes Cutting Parameters Pass Depth 0.052 inches Notes Cutting Parameters Pass Depth 0.052 inches Notes Cutting Parameter Pass Depth 0.050 inches Stepover 0.1 inches 40.0 • \$% Cutting Parameter Pass Depth 0.050 inches Stepover 0.1 inches 40.0 • \$% Cutting Parameter Pass Depth 0.050 inches Stepover 0.1 inches 40.0 • \$% Cutting Parameter Pass Depth 0.050 inches Stepover 0.1 inches 40.0 • \$% Cutting Parameter Pass Depth 0.050 inches Stepover 0.1875 inches Stepover 0	Name V-Bit (90 d	deg 0.25 inches)	Name	End M	ill (0.25 inches)		Name	Ball Nose (	0.25 inch)		
indued Angle 90.0 ↓ degrees indued Angle 90.0 ↓ degrees indued Angle 90.0 ↓ degrees indued Angle 90.0 ↓ degrees indued Angle 90.0 ↓ degrees indues 40.0 ↓ 0.	Tool Type V-Bit	•	Tool	Type End Mi	ill 👻		Tool Type	Ball Nose		•	
Geometry   Diameter (D)   0.25 inches   Inches <th< td=""><td>Notes</td><td></td><td>Note</td><td></td><td></td><td></td><td>Notes</td><td></td><td></td><td></td><td></td></th<>	Notes		Note				Notes				
Tool Numbers     Cutting Parameters   Cutting Parameters     Pass Depth   0.062   inches     Final Pass Stepover   0.1   inches   40.0   \$6     Clearance Pass   0.1   inches   40.0   \$6     Spindle Speed   18000   r.p.m   Feeds and Speeds   5     Feed Rate   40.0   inches/min   1   inches/min   1     Plunge Rate   75.0   1   1   inches/min   1		0.25 inches 🔻	Geor		0.25 inches V	Fo⊣		)	0.25	inches 🔻	<b>⊢</b> ∎-1
Pass Depth   0.062   inches   Pass Depth   0.09   inche   Pass Depth   0.25   inches     Final Pass Stepover   0.1   inches   40.0   %   %   Stepover   0.1   inches   %   Stepover   0.1875   inches   %   %   %   Stepover   0.1875   inches   %   %   %   Stepover   Stepover   0.1875   inches   %   %   %   Stepover   Stepover   Stepover   Stepover   Stepover   Stepover   Stepover   Stepover <td></td> <td>90.0 🔹 degrees</td> <td>-</td> <td>(</td> <td>Tool Numbers</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>V.</td>		90.0 🔹 degrees	-	(	Tool Numbers						V.
Final Pass Stepover 0.1 inches 40.0 %   Final Pass Stepover 0.1 inches 40.0 %   Clearance Pass 0.1 inches 40.0 %   Spindle Speeds 18000 r.p.m Feeds and Speeds   Spindle Speed 18000 r.p.m Spindle Speed   Plunge Rate 75.0 Plunge Rate 30.0		0.062 inches			0.99 inch		-	meters	0.25	inches	
Feeds and Speeds   Spindle Speed 18000 r.p.m Feeds and Speeds   Spindle Speed 0 r.p.m   Feed Rate 400.0   Plunge Rate 75.0     Feed Rate 30.0   Plunge Rate 30.0	Final Pass Stepover	0.1 inches 40.0	% Ste		0.1 inches 40.0	%					<u>^</u> %
Spindle Speed 18000 r.p.m Spindle Speed 0 r.p.m   Feed Rate 400.0 inches/min Feed Rate 30.0 inches/min Feed Rate 100.0   Plunge Rate 75.0 Plunge Rate 30.0 inches/min 90.0 Inches/min		indico inte i		and Speeds			Feeds and S	peeds			
Plunge Rate 75.0 Plunge Rate 300 Plunge Rate 30.0 Plunge Rate 30.0		18000 r.p.m		0.540.000 <b>.</b> Market of					15000	r.p.m	
Plunge Rate 75.0 Plunge Rate 300 Plunge Rate 30.0	Feed Rate		Fee	d Rate		_	Feed Pate		100.0		
hol Number 1 Apply Tool Number 2 Apply Tool Number 4 Apply Apply	Plunge Rate	75.0	Plu	ige Rate	300	-	Plunge Rate		30.0	inches/min	•
	ool Number	1	Apply Tool	Number	2	Apply	Tool Number		4		Apply

Figure 4. Snapshot of the tool info menu, highlighting the tool number parameter.

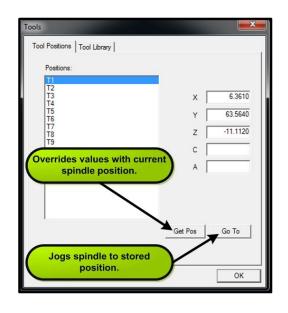
 With all toolpaths selected, save the toolpath list after selecting the WinCNC ATC(\*.tap) post processor.





### STORING TOOL LOCATIONS

It is possible that a location will need to be reset. To view and edit a tool location, follow the path Settings  $\rightarrow$  Tool Positions. This opens the tool positions dialog box.



## WHAT HAPPENS DURING A TOOL CHANGE?

The flow diagram below captures the order of events that occur when a tool command is executed.