MACH3 LATHE TOOL CHANGE
HOW THE TOOL CHANGE WORKS

So the question now is how to save different tools and then call it from MDI?

In the MDI line type:
- **M6 T0101** – the M6 will be run and tool # 1 becomes the current too with offset 1.
  Note that M6 use is affected Lathe Configuration (info below)
- **T0202** - will change the current tool to tool# 2 with offsets of tool #2

PREFACE

There are many ways to do a tool change for the lathe and this write up addresses the topic. How a tool change is done will vary depending on the actual lathe in use, level of system automation, and how the USER needs / wants the tool change to work.

Mach3 provides for tool changes and configuration of the software is required. The lathe manual covers many topics related to a tool change. I assume that you have a level of understanding of the following topics, and if not, READ THE MANUAL. Mach3 is software, software is stupid, and needs to know information in order control a machine.

- Tool Table
- Macro understanding
- G Code definition and commands
- Software Configuration

Information from Mach3Turn manual:

8.1.1.3 G28 positions

The machine coordinates in setup units to which the machine should co when a G28 is performed are defined for axes X to C on this dialog. For more details see Chapter 10.

10.7.8 Return to Home - G28 and G30

A home position is defined by Config>Softlimits & Homing. The values are in terms of the absolute coordinate system, but are in setup length units.
To return to home position by way of the programmed position, program G28 X~ Z~ (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

10.7.9 Reference axes - G28.1

Program G28.1 X~ Z~ to reference the given axes. The axes will move at the current feed rate towards the home switch(es), as defined by the Configuration. When the absolute machine coordinate reaches the value given by an axis word then the feed rate is set to that defined by Config>Homing/Limits. Provided the current absolute position is approximately correct, then this will give a soft stop onto the reference switch(es).

10.1.13 Tool Change

Mach3 allows you to implement a procedure for implementing automatic tool changes using macros or to change the tools by hand when required.

10.7.4 Dwell – G04

For a dwell, program G04 P~ . This will keep the axes unmoving for the period of time in seconds or milliseconds specified by the P number. The time unit to be used is set up on the Configure>Logic dialog. For example, with units set to Seconds, G4 P0.5 will dwell for half a second. It is an error if:

· the P number is negative

10.8.3 Tool change – M06

M06 is not needed as tool changing is actually performed when the T word is used.

You will need a custom M6 start macro for the lathe you are using.

Typicaly code content for M6Start could be:
-turn spindle off
-turn cooling off
-go to a save place for toolchange
-anounce the new tool to System for all the the new offsets

Typicaly code content for M6End could be:
-turn spindle on
-turn cooling on
-go back to a place where it it save to restar

10.10.3 Select Tool – T

To select a tool, program T~ where the T number is entry number in the tool table for the tool to be used and the entry number for the offsets to be applied.

For example T0202 (or equivalently T202) selects tool 2 with its own offsets and T0207 selects tool 2 wth the offsets for tool 7. T02 is treated as equivalent to T0202 and similarly for any other value less than or equal to 99.

Provided tool change requests are not to be ignored (as defined in Configure>Logic), Mach3 will call a macro M6Start when the command is encountered. It will then, optionally, wait for Cycle Start to be pressed, execute the macro M6End and continue running the part program. If an automatic tool changer is in use then the Cycle Start is not needed and M6End will not be called.

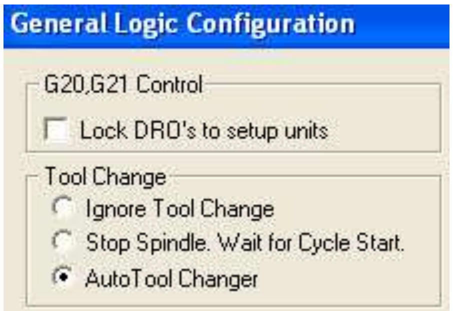
You can provide Visual Basic code in the macros to operate your own mechanical tool changer and to move the axes to a convenient location to tool changing if you wish. Full details can be found in *Customizing Mach3 wiki.*

If tool change requests are set to be ignored (in Configure>Logic) then the macros are not called..

It is OK to program T0; no tool will be selected. This is useful if you want the spindle to be empty after a tool change. It is an error if:

· a negative T number is used, or a T number larger than 9999 is used.

<u>Lathe Configuration</u>



Under the General Logic Configuration you have a choice of how a YOU want the call for a tool change to be handled. Once you selected which of the options you want to use you can then you place code in the macro to do whatever you want to happen.

- Ignore Tool Change - Mach will ignore the M6 command in the g-code T404 M6.
  One could think of this selection as "switch" which turns the tool change off.

- Stop Spindle. Waite for Cycle Start - Mach will use both the M6 start and M6 end macros.

  Typicaly code content for M6Start could be:
  -turn spindle off
  -turn cooling off
  -go to a save place for toolchange
  -anounce the new tool to System for all the the new offsets

  Typicaly code content for M6End could be:
  -turn spindle on
  -turn cooling on
  -go back to a place where it it save to restart

- Auto Tool Changer - only the M6 start macro will be used