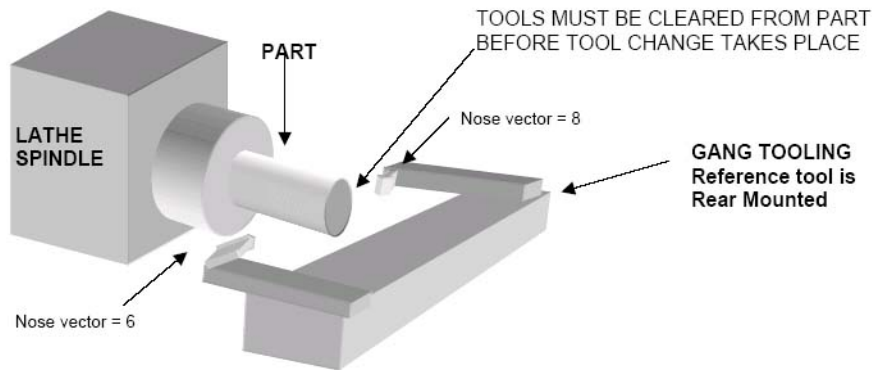


TB128 (Rev1) - Gang Tooling



Overview

Setup for Gang Tooling use. With these settings you will not have to press cycle start after each tool change and the spindle will stay running at speed during the tool change.

Software Version

It is recommended to use the latest version of operating and programming software available so that you can make use of the latest time saving features. V. 8.11 has added specific changes to the Tool library, which improve the ease of use for Gang tooling. We recommend 8.11 or higher.

Procedure

To set the control configuration up for gang tooling...

Update your control software to the latest version. If the control software is not at Version 8.11 or higher, update the system. (Note: to find out what version you have press F1 Setup from the main screen and the version number will be displayed)

Once the control software is at 8.11 or higher, set the parameters below to the following values in the control config parameter screen. (To get to the parameter screen: F1 Setup, F3 config, F3 parameters).

-Set Parameter # 163 = 1 This turns on the gang tooling function in the tool library.

- Parameter #1 Parameter 1 tells the control from which direction the reference tool is approaching the work from and changes the graphical displays to show if the reference tool is front or rear mount. If the reference tool is approaching from the front then Parameter #1 should be set to 1. If your reference tool is approaching from the back then set Parameter #1 to 0. See "Configuration" Chapter 14 of the Lathe Manual for more information about Parameter #1.

-Set Parameter #6 = 1 This makes ATC F7 key available for tool changes and when running a program it will NOT stop and ask the operator to press CYCLE START to continue when a new tool is called out in the program. The Auto Tool Changer unlock code #203 has to be unlocked when parameter #6 = 1.

-Create a file: CNC7.TCH in the c:\CNC7T directory in CNC7 software,
or **CNC10.TCH** in the /cncroot/c/cnc10t directory in CNC10 software,
or **CNCTCH.MAC** in the /cncroot/c/cnct directory in CNC11 software that consist of this one line.

M99

(Use the DOS editor or the built in editor in the control software, or you can create the file on another computer and copy it into c:\cnc7 directory)

When using INTERCON to program parts go to "F10" Setup from the main screen of Intercon and select:

"Use G28 for tool change" = NO.

This tells INTERCON not to use the G28 command when programming a tool change. G28 is a clearance point, used for a position to retract to when using a turret style automatic tool changer. Since with gang tooling a "safe" return point will be different for each tool you do not want to use the G28 return point with a gang tooling setup. It is up to the programmer to program in the clearance moves for each tool before the tool change takes place.

Now you are ready to set up tools:

-Go to the tool library and you will see a column where you can select whether or not a particular tool is coming into the work from the Front or the Back.

Set each tool to its proper approach from "Front" or "Back" setting.

EXAMPLE: Tool Library

Tool Coolant Station Approach Description

T01 Flood 01 Back

T02 Flood 02 Front

T03 Flood 03 Front

T04 Flood 04 Back

T05 Flood 05 Front

T06 Flood 06 Back

T07 Flood 07 Back

T08 Flood 08 Front

T09 Flood 09 Back

T10 Flood 10 Front

-Now set the Nose Vectors for each tool. Look at the nose vector diagram in the tool library and pick the number that matches the direction that the tool will be approaching the work piece. See "Setting the Nose Vector" in Chapter 3 of the Lathe Manual for more information about setting nose vectors.

EXAMPLE #1. Reference tool is Rear Mounted. Parameter #1 = 0. OD straight in with back approach is Vector #8 and OD straight in with front approach is Vector #6. ID straight in with back approach is Vector #6 and ID straight in with front approach is Vector #8.

For more information on selecting nose vectors see "Setting the Nose Vector" in Chapter 3

-Set the tool offsets like the lathe manual describes in Chapter 3.

- While programming parts using gang tooling it is up to the operator to "clear" the current tool before changing to the next tool. For instance after a canned turning cycle, the end point of the cycle is the same as the start point, in other words the tool starts and stops at the same location. Before changing to another tool, the next step in programming would be to add a few lines to clear this (and other) tools away from the work piece. Once the tools are clear and away in a position ready to change tools, then the tool change can be programmed to take place.

-Advice: Study the graph carefully before cutting a part to verify that you have set up the tools properly and have programmed it correctly. You may want to cut a few parts in plastic to verify your offsets and programming.

If using Intercon v7.13 or earlier to program a part, by default Intercon will command a G28 (Return to Reference Point) for each tool change. You will have to edit the g code to remove this G28 if you are using Gang tooling. On version 7.14 and later, use of the G28 command is user selectable in Intercon and can be turned on or off. To turn off G28 go into Intercon select F7 (Setup) scroll down and toggle the G28 command to "off".

Intercon versions prior to V8.10 did NOT support cycles approaching on the negative side of the part. If the program was written manually in G-Code or posted using third party software it will work.

The Auto Tool Changer unlock code #203 has to be unlocked for a gang tooling setup. Call your dealer to purchase the ATC unlock code if you don't have it already.

Document History

Rev1 Created on *0000-00-00* by *#240*