

CNC MACROS

Exceptional ROI

Thank you for allowing us to create your Custom Macro Solution. Please read through this documentation. **This is an updated document for the updated programs. Please disregard any previous documents.**

What's included with purchase and program descriptions.

Program O8888


This program is strictly to test that the behaviors of the machine act correctly. This program to be deleted after testing is complete. Detailed instruction on how to use this program are found on page 6 of this document.

Program O7901

The command "M98P7901" will need to be added to all of your programs. This command will need to be added after all M03 and M04 calls in your programs. Using your Demo Program as an example.


Before

```
N1
M139
(FACE PART TO 1.95 )
(2 INCH SANDVIK HF)
G00G91G28Z0.
G17G20G49G90
T06M06
M01
T52
M11(B AXIS UNCLAMP)
G00G90G54.1P1
G68.2P1Q123X0.Y0.Z0.I0.J0.K0.
G53.1
M10(B AXIS CLAMP)
G90X.625Y-6.3002S3180M03
G43H#999Z1.M09
Z.15
G01Z.025F450.
Y-5.1002
Y0.
G00Z.15
```



After

```
N1
M139
(FACE PART TO 1.95 )
(2 INCH SANDVIK HF)
G00G91G28Z0.
G17G20G49G90
T06M06
M01
T52
M11(B AXIS UNCLAMP)
G00G90G54.1P1
G68.2P1Q123X0.Y0.Z0.I0.J0.K0.
G53.1
M10(B AXIS CLAMP)
G90X.625Y-6.3002S3180M03
M98P7901 ←
G43H#999Z1.M09
Z.15
G01Z.025F450.
Y-5.1002
Y0.
G00Z.15
X.625Y-6.3002
Z.125
G01Z0.
Y-5.1002
Y0.
G00Z.15
```



This program will set the internal "Hours" counter to zero. #3002 to 0.

CNC MACROS

Exceptional ROI

Program O7902

The command “M98P7902” will need to be added to all of your programs. This command will need to be added after all M05 calls in your programs. Using your Demo Program as an example.

Before	After
G54.1P20X.625Y-6.3002	G54.1P20X.625Y-6.3002
Z.15	Z.15
G01Z.025	G01Z.025
Y-5.1002	Y-5.1002
Y0.	Y0.
G00Z.15	G00Z.15
X.625Y-6.3002	X.625Y-6.3002
Z.125	Z.125
G01Z0.	G01Z0.
Y-5.1002	Y-5.1002
Y0.	Y0.
G00Z.15	G00Z.15
Z1.M09	Z1.M09
G69	G69
G00G91G28Z0.M05	G00G91G28Z0.M05
M01	M98P7202 ←
	M01
N2 (2" SANDVIK CORNCOB)	N2 (2" SANDVIK CORNCOB)
G00G91G28Z0.	G00G91G28Z0.

This program will take the spindle on time in the “Hours” counter #3002. And transfer the time to macro number 600 plus the tool number. Example – Tool 21’s accumulated time will be in macro #621.

Heavy tools are done slightly different. They are store by tool number minus 70 plus 600. Example: Tool 131’s time will be store in Macro #661. ($131-70=61$ $61+600=\#661$.)

There are 6 allotted macro numbers for heavy tools.

- Tool 131 = #661
- Tool 132 = #662
- Tool 133 = #663
- Tool 134 = #664
- Tool 135 = #665
- Tool 136 = #666



If program encounters Tool 0 or a Tool greater than 136 it will skip this program.

CNC MACROS

Exceptional ROI

Program O7900

The command "M98P7900" will need to be added to all of your programs. This command will need to be added at the beginning of all your programs. I would add this after the preparatory blocks. Using your Demo Program as an example.

Before	After
O1043(132497 OP1 - 2" CORN COB) (10-14-2020) (USE MITEE BITE FIXTURES)	O1043(132497 OP1 - 2" CORN COB) (10-14-2020) (USE MITEE BITE FIXTURES)
(5.65 PER PART FOR OP1 AND OP2)	(5.65 PER PART FOR OP1 AND OP2)
(NUM DESCRIPTION) (--- -----) (T06 2" INCH SANDVIK HF) (T52 2" CORN COB) (T02 NINE9 SUMO SPOT) (T15 .433 SUMO DRILL) (T132 5" X .197 SAW) (T131 6" X .422 SAW) (T04 9NINE BACK CHAMFER) (T12 0.4375 CARBIDE TIPPED REAMER) (T60 PROBE) (T22 CHIP FAN)	(NUM DESCRIPTION) (--- -----) (T06 2" INCH SANDVIK HF) (T52 2" CORN COB) (T02 NINE9 SUMO SPOT) (T15 .433 SUMO DRILL) (T132 5" X .197 SAW) (T131 6" X .422 SAW) (T04 9NINE BACK CHAMFER) (T12 0.4375 CARBIDE TIPPED REAMER) (T60 PROBE) (T22 CHIP FAN)
G17G40G49G80G90G20G69 G54.2P0(CANCEL ROTARY TABLE DYNAMIC FIXTURE OFFSET) G00G91G28Z0.(Z AXIS RETRACT TO HOME) 	G17G40G49G80G90G20G69 G54.2P0(CANCEL ROTARY TABLE DYNAMIC FIXTURE OFFSET) G00G91G28Z0.(Z AXIS RETRACT TO HOME) M98P7900 
(ENTER 1 TO SET WORK OFFSETS AND PROBE THE PART) (ENTER 0 TO SKIP) #100=0 IF[#100EQ0]GOTO1 M00(MACHINE IS GOING TO SET WORK OFFSETS AND PROBE)	(ENTER 1 TO SET WORK OFFSETS AND PROBE THE PART) (ENTER 0 TO SKIP) #100=0 IF[#100EQ0]GOTO1 M00(MACHINE IS GOING TO SET WORK OFFSETS AND PROBE)

This program will check Tools for expired tool lives. This is where you will have to edit the program to place the appropriate tool life desired. See figure #2 below to see where to edit program.

CNC MACROS

Exceptional ROI

```
IF[#601GE9200.0000]GOTO1(Tool #1 Max Life)
IF[#602GE9200.0000]GOTO2(Tool #2 Max Life)
IF[#603GE9200.0000]GOTO3(Tool #3 Max Life)
IF[#604GE9200.0000]GOTO4(Tool #4 Max Life)
IF[#605GE9200.0000]GOTO5(Tool #5 Max Life)
IF[#606GE9200.0000]GOTO6(Tool #6 Max Life)
IF[#607GE9200.0000]GOTO7(Tool #7 Max Life)
IF[#608GE9200.0000]GOTO8(Tool #8 Max Life)
IF[#609GE9200.0000]GOTO9(Tool #9 Max Life)
```

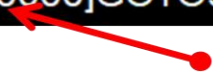


Figure 2

Currently the time value of 9200.0000 is set for all tools, as max cut time. Change this to fit your needs. This can be set to 1.25 for a time of 1 and a quarter hours.

Once an expired tool is encountered, the machine will do an auto tool change to the expired tool. Move the tool to the door. And then throw a machine alarm. On the screen it will show a message that the tool has expired and what tool number.

The alarm number generated will be #3000 plus tool number. So if you look in the alarm history and you see alarm #3047 that will mean tool 47 came up with expired tool life.

Heavy tools are treated different. The machine will only pause with a message to change a tool. Pressing cycle start will continue to run the program. Or the operator can press reset and tend to the expired tool.

Tool life automatically sets back to zero for all tools.

If no tools are found to have expired tool life, the main program will run as normal.

Program O9988

In order to use your program O9988 to move the tool to the door at the expiration of tool life, one modification will need to be made to this program. Since it appears that this program is called by using an MDI call. We have to add the following lines to the end of the program just before the **M30** command to return control back to program O9019. The added lines are:

G31

IF[#668EQ9019]THENM99 (Add This Line to program)

I have included this program in the attachments for you to reload, or you can add the above lines yourself to the existing program.

CNC MACROS

Exceptional ROI

Also included with purchase:

1ea. Microsoft Excel File called- "Master Macro Map REV2". This is used to track what macro numbers are being used per information provided by you the customer and what numbers are being used by our custom tool life macro programs. If you hover over the cell you will see a comment telling you what the macro number does.

1ea. This document (#Lei01REV2) detailing how each custom tool life macro program works. Also included in this document, detailed instructions on how to use program O8888 to test the behaviors of the custom macro programs prior to using them with your main production programs. And where to add certain M98 commands.

Disclaimer - Under no circumstance shall we (CNCMACROS.COM) have any liability to you for any loss or damage of any kind incurred as a result of the use of the purchased macro programs or documentation. Your use of any provided programs and your reliance on any information provided is solely at your own risk.

Program Loading & Proofing Procedures.

1. Load downloaded programs into your normal program directory.
 - a. Included Programs – O7900, O7901, O7902, O9988, and O8888.
 - b. Program #O8888 is strictly a testing program to ensure everything works correctly. And can be deleted once program is proof out.
2. Set Macro #601 value to **9300.00**. This will cause an expired tool condition for tool #1.
3. Set program #O8888 as the main program.
4. Place machine control in “**OPT Stop Mode**”.
5. Press **Cycle Start**.
6. When the control reads the line with the “M98P7900” call in it. This will cause the machine to do a tool change to Tool #1. Move the tool to the door, and cause machine alarm “3001”. “Tool life expired”.
7. Press “**Reset**”.
8. While still in “**OPT Stop Mode**”, press cycle start. Machine will do a tool change, to tool #2. And stop at the **M01**.
9. Press **Cycle Start** again. This will cause the spindle to start (M03) and be at 1000rpm.
10. Machine will dwell for 10 seconds with the spindle at 1000 rpm and then turn off the spindle (M05).
 - a. Once the spindle stops, and the control stops at the M01, look at macro #602, there should be a value larger than 0. Estimated value 0.002777.
11. Press **Cycle Start** again. Program will reset Tool’s #1 & #2 tool life (Macro’s #601 & #602) back to zero and end the program.

Program has now been proof and is ready to run with the production programs. You can delete program #8888.

Disclaimer - Under no circumstance shall we (CNCMACROS.COM) have any liability to you for any loss or damage of any kind incurred as a result of the use of the purchased macro programs or documentation. Your use of any provided programs and your reliance on any information provided is solely at your own risk.