



Zeman Technologies

The Finest Machine Tools

IL - (630) 321-1500

WI - (262) 789-1600



Zeman Technologies

## Did You Know? Macro Programs Using G & M Codes

On Fanuc & Mitsubishi controlled machines, users can create custom macro programs that are called with simple G (Geometric) or M (Machine) codes that the user defines in parameters. Any redundant action - returning axes home, drilling cycles, custom boring cycle, etc. - can be stored as a macro program that is called with a G or M code...

### Example Macro Program: Move to Home Position

1. Set Parameter 6050 to 25 (See Fanuc/Mitsubishi Manual)
2. This Causes G25 to Call Program 9010
3. Write Program 9010 Below:

```
O9010  
G00G53 Z0  
G53X0Y0  
M99
```

Now G25 and Cycle Start saves 3+ button pushes and a mode selection.

[Watch our video tutorial on Special G Codes here](#)

## Explanation

By setting a G code number from -9999 to 9999 used to call a custom macro program (O9010 to O9019) in the corresponding parameters Nos.6050 to 6059, the macro program can be called in the same way as with G65. To call custom macro program O9040 to O9049 using a G code with a decimal point, set bit 0 (DPG) of parameter No. 6007 to 1 and set the G code number in the corresponding parameters Nos. 6060 to 6069.

The number of decimal places of a G code is 1. Set the value obtained by multiplying a desired G code number by 10 in the corresponding parameter.

[Example] When parameter No. 6060 is set to 234, custom macro program O9040 is called using G23.4.

When a negative G code is set, a modal call is performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.

For example, when a parameter is set so that macro program O9010 can be called with G81, a user-specific cycle created using a custom macro can be called without modifying the machining program.

**Why Macro Programs?** Users often utilize macro programming to save memory by looping a repetitive task, rather than duplicating lines of code. This is especially helpful with families of parts, where users can change values of variables rather than write multiple part programs.

[Want to learn from our expert engineers? Click Here](#)

## Explanation

By setting an M code number from 3 to 99999999 used to call custom macro program O9020 to O9029 in the corresponding parameters Nos. 6080 to 6089, the macro program can be called in the same way as with G65.

### - Correspondence between parameter numbers and program numbers

Parameter number	Corresponding program number
6080	O9020
6081	O9021
6082	O9022
6083	O9023
6084	O9024
6085	O9025
6086	O9026
6087	O9027
6088	O9028
6089	O9029

Example) When parameter No. 6080 is set to 990, O9020 is called using M990.

**Miss our Last Tech Tips? Click Below!**



The Finest Machine Tools

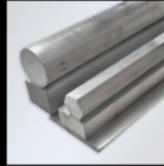
IL - (630) 321-1500  
WI - (262) 789-1600



Zeman Technologies

### Did You Know? Skip the Milling, Run Non-Conformed Stock!

Sometimes it makes sense to run hex or other stock rather than milling to shape. To set up the machine, use a round bar loader collet to fit over points of the stock. In Spindle 1, use a chucking sleeve with a pin to hold the collet in place as a new bar is loaded. To teach the machine rotational relationships, you will now perform phase adjustment.



You can adjust phase between the following on your Citizen:

- Main spindle and guide bushing
- Main spindle and back spindle
- Main spindle, guide bushing, and back spindle

In the phase adjustment menu, there are three (3) selections located on the right. Enter the desired number in the Operation Select field, close all doors, and press the menu key [Phase Adj] and Start Key simultaneously. A phase adjustment confirmation message will appear.



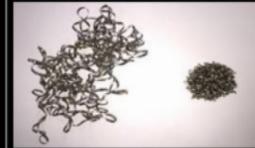
The Finest Machine Tools

IL - (630) 321-1500  
WI - (262) 789-1600



Zeman Technologies

### Did You Know? Cincom Low Frequency Vibration-Cutting



**L.F.V.-Produced Chips:** Two 14.3 gram samples of 304 Stainless, which would you prefer?

#### What is L.F.V?

Chip Control: Servo-controlled axes oscillate in the axial direction, and machining is carried out while synchronizing this vibration with spindle rotation. The air cutting time created during machining leads to intermittent chip removal.

[View our Webinar on Low Frequency Vibration Here](#)



NYLON

[Website](#)

[Products](#)

[News](#)

[Support](#)

[Contact Us](#)

