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!

McCLAIN
MACHINE TOOL SOLUTIONS
Zeman Technologies

Committed to 100% Customer Satisfaction
Call (314) 432-3220

Did You Know?
"Last Part Program" on Citizen Lathes

Citizen lathes are designed to machine the front and back halves of complex parts simultaneously. **Last Part Program** allows the operator to stop the running of a continuous cycle and finish the back half of the last part, without starting another part on the front spindle. Benefit: Gain an extra part!

An Example of Implementation on the L20:

15.11 Executing the Last Program (G999)

This command must be specified in the last portion (end process) of each axis control group ($1, $2) program that includes the last program.

In general, the last program is executed to perform back machining for workpieces with which front machining is completed. The back machining is performed in the last cycle while the machine is in the stopped state (e.g., 1-cycle stop or product counting by the counter).

Specify the G999 command for each axis control group to automatically enter the axis control groups in the queuing state. The last program between G999 and N999 is executed in the 1-cycle or 1-block operation mode.

Be sure to specify the N999 command at the end of the last program contents of each axis control group. To finish program creation, specify three commands following N999 at the end of the end process. The commands must be specified in the sequence of M02, M99 and then %.

- Command format

G999 Last program execution

- Axis control group

Specify this command for both the axis control groups $1 and $2.

Below, G999 (TOP LEFT) is the start of our Last Part Program , concluding with N999, M2 and M99. Observe the highlighted use of Sub-Programs in the

McCLAIN
MACHINE TOOL SOLUTIONS
Zeman Technologies

Committed to 100% Customer Satisfaction
Call (314) 432-3220

Did You Know?
Standard Control Features on Enshu Horizontals

ENSHU pallet-changing Horizontal Machining Centers are built to last, with 5,000 Hours Mean Time Between Failure. The following control features standard on ENSHU contribute to speed, reliability, and long term accuracy.

Manual Operation of Automatic Pallet Changer

This feature allows an operator to rotate the pallet shuttle by hand to easily clean, maintain, and check fixture clearance safely while pallets exchange.

1. Select Manual Mode
2. Select desired pallet using the APC Drive Switch
3. Press 'APC Drive' button to step through pallet change
4. Release the 'APC Drive' button at any time to stop that operation, and press again to resume operation.

Interested in learning more about Enshu?

Tool Change Arm Recovery

If the tool change arm is interrupted, there are two recovery methods: