

RYM Presents: CNC Monitoring with Macro Commands (DPRNT)

An introduction to DPRNT

Macro commands available in many CNC controllers allow the CNC to output information while it runs. If you have used a touch probe macro to output data to a terminal or serial printer then you used a DPRNT command (pronounced “D print”). Some types of CNC controllers use other command words instead of DPRNT. Fadal controllers use SPRINT and Okuma uses PUT and WRITE commands, for example. Because the majority of CNCs on the market use Fanuc, Haas, Yasnac, Mitsubishi, Mazak, or other controllers which use DPRNT as the command word – we generically refer to this functionality as, “DPRNT Capability.”

How DPRNT works

If your CNC supports DPRNT, it will not automatically output data on its own – part programs or macros must be modified to output serial data. The basic commands are straightforward, and many users modify their CAD/CAM system post-processor to automatically generate the appropriate commands when posting a new program. If you want to make a part program to output a single word, here is what that program might look like:

```
O1111
POPEN
DPRNT [HELLO]
PCLOS
M30
```

Let’s look at this program line by line:

- **O1111** is your program number (name).
- **POPEN** tells the CNC to stand by to send data out the RS232 port.
- **DPRNT[HELLO]** tells the CNC to send whatever is in between the square brackets, in this case the word HELLO.
- **PCLOS** tells the CNC to close the serial port because we are through transferring. Every POPEN should have an accompanying PCLOS.

M30 is your “end of program” command.

The relationship between DPRNT and your DNC System

To take advantage of macro commands for machine monitoring, something must be listening to the RS232 port at all times for any data that may come out. Because RS232 is also used for file transfer, it makes sense that this functionality is usually built into a DNC system.

- Of course, RYM’s DNC systems are designed to communicate directly with FactoryWiz, making the connection from DNC to monitoring system easy and transparent. Also, certain features are available on FactoryWiz Monitoring that can complement your RYM DNC system, such as file transfer functionality available directly from within the FactoryWiz web site

How DPRNT can be used for CNC Monitoring with FactoryWiz

Obviously just sending HELLO out the serial port won't be very useful to you. Instead, what you will want to do is embed commands in your program that tell your monitoring system when a program has begun and when it has finished. For example:

```
O1111
POPEN
DPRNT [START]
DPRNT [PART-AB12331]
PCLOS
.
.
(your part program here)
(could be a few lines)
(or thousands)
.
.
POPEN
DPRNT [FINISH]
PCLOS
M30
```

In this example, the CNC outputs a command to tell the monitoring system that a part cycle has begun, followed by a command to tell the system that it is running program AB12331. At the very end of the program, the CNC indicates it is finished just before going out of cycle. It is up to the monitoring system to keep track of the time that each of these commands was received so that an accurate cycle time can be determined.

Advanced Features

Frequently it is desirable to output some piece of information from the CNC's offset or macro variable tables. This is also straightforward. You can output and save any data you can access with a macro command into one of a number of custom fields set aside in FactoryWiz Monitoring for this purpose. Users have taken advantage of this feature to save touch probe data with the production record for that cycle, or to override FactoryWiz Monitoring's automatic part counter with the value of the CNC's internal part counter (so they will have synchronized numbers if the operator resets it).

Commands used to send to FactoryWiz Monitoring using DPRNT

Part 1: Basic Required Commands for Monitoring

Command *Description*

START	Start of a new cycle
FINISH	End of a cycle
PART-xxxxxxx	Indicates the CNC is running the part called xxxxxxxx

Part 2: Basic Required Command for Capturing Probe Data

Command *Description*

APPEND-filename	Whatever comes after this command will be saved into a text file called "filename" (will append, not overwrite). Handy for probe data.
APPEND-	Turns off the append command to resume normal operation

Limitations

Using DPRNT, there are several limitations compared to other monitoring methods:

- All part programs must be modified with the appropriate commands or the collected data will be inaccurate.
- If the CNC is stopped mid-program, such as by pressing reset, emergency stop, or by an alarm, the CNC will appear to still be in cycle since a FINISH command was never issued. The issue will last until the next data is received from the CNC or a manual override barcode is scanned.
- If the CNC attempts to output data with a DPRNT and the DNC system is not turned on, or is not functioning, the CNC can stop running while it waits for the DNC system to become ready. RYM generally jumpers pins 6-8-20 and 4-5 on the cables of any CNC's running DPRNT monitoring to prevent this from happening.
- No alarm status is recorded, so DPRNT cannot be used to perform an automatic page during unattended operation if the CNC stops.

Does your CNC have DPRNT?

DPRNT is usually associated with an option called Macro B, but not all manufacturers use the same name. Macro B tends to be disabled on simple, less expensive machines unless it has been specifically ordered. Basic lathes and vertical machining centers fall into this category. On the other hand, turning centers with sub-spindles, Swiss machines, and

horizontal machining centers almost always have the feature turned on since macro programs play a large part in programming these machines. Similarly, less expensive brands of CNC such as Excel tend not to have the option turned on, but a large percentage of the more expensive machines such as Robodrill, Toyoda, Matsuura, and Kitamura do have the option turned on. Ultimately, the only way to know for sure is to load a sample program into the CNC to test.

If you have a computer that is able to receive data from your CNC, either standalone or through a DNC system, you can try setting that computer to receive data and run one of the sample programs near the end of this document. If the CNC tries to space the letters apart, such as P O P E N instead of POPEN, or won't let you type the letters consecutively, that is a good indication that it doesn't recognize the command. If the program executes without error – and you see data arrive back at the computer – you should be all set!

Command Examples

Fanuc / Haas / Yasnac / Mitsubishi / Mazak

(described on page 1)

```
O1111
POPEN
DPRNT [HELLO]
PCLOS
M30
```

Also, below is the format for outputting text and macro variable 500's value together. The [40] indicates that the CNC should output 6 digits before the decimal and 0 digits after.

```
DPRNT [PARTCOUNT- #500 [60]]
```

Yasnac / Mitsubishi / Mazak (alt)

```
O1111
POPEN [1]
DPRNT [HELLO]
PCLOS
M30
```

Only difference is in POPEN command. Some machines need port 1 or port 2 specified to tell the CNC which port to use.

Fadal

```
O1111
N2#WAIT
N3#SPRINT"HELLO"
N4M30
```

The #WAIT command tells the CNC to not jump ahead and execute the SPRINT before all previous lines have been run. This is often necessary on Fadals.

Okuma

```
$O1111.MIN%
PUT 'HELLO'
WRITE 0
M30
```

The Okuma looks quite different. There is no equivalent to POPEN or PCLOS. Instead, data is placed in a buffer with the PUT command, and then written to the RS232 port (0 means port CNO:)

Contact Us

Refresh Your Memory, Inc.

6920 Santa Teresa Blvd, Suite 201, San Jose, CA 95119

(408) 224-9167

Or visit us online at www.factorywiz.com