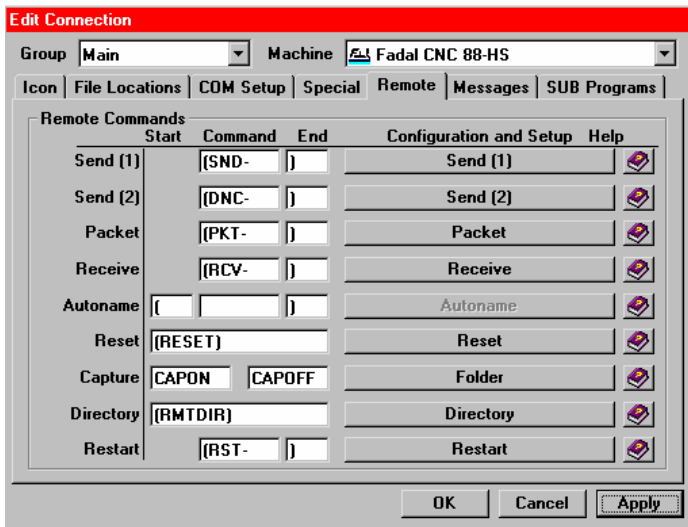




The **Remote Feature** of **NetDNC for Windows™ XP to 95**

Our **Remote** mode is one of the most useful and popular functions of Memex's **NetDNC**. **Remote** allows CNC operators to send, receive, run "drip feed" DNC communications and do other functions directly from their CNC controls without having to walk back to the DNC computer. CNC operators asking for tapes, or to send and receive CNC programs from the CAD/CAM workstation do not have to interrupt your CAM programmer. Both programmers and CNC operators can save a tremendous amount of time as result of this automation. With **NetDNC** there is no need for extra terminals by the CNC control, as the control itself does all the work !

When a Machine Connection is in **Remote** mode, the COM port is set to "**scan**" for a certain sequence or "**string**" of characters associated with the legal file name of a requested file. This particular string of characters triggers **NetDNC** to either find the single requested program and send it to the CNC control **or** to store the program that is being received as it comes in from the CNC control, as well as several other functions. The following dialogs set up all of the **Remote** parameters, which can be different for each Machine Connection in a **NetDNC** system.



There are eight separate Remote functions

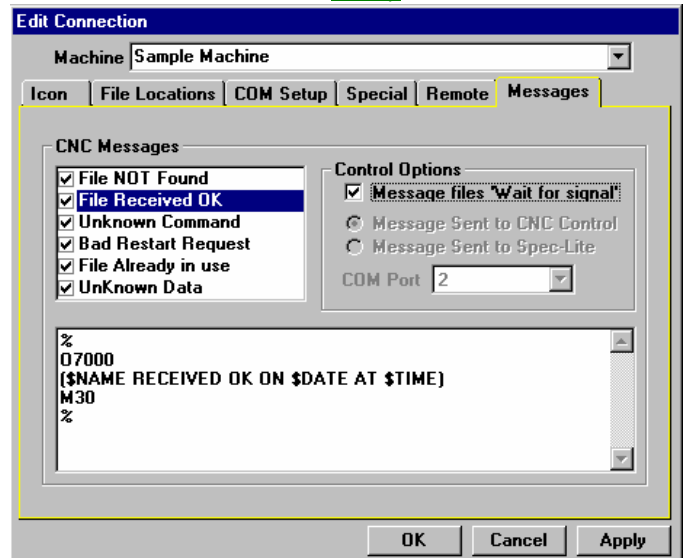
- **Send** is the string of characters that precedes a single file name when you request a file **from** the DNC computer. There are two **Send** Commands that can be used.
- **Packet** is the string of characters that precede the packet name (multiple files) when you request it **from** the DNC computer.
- **Receive** is the string of characters that sets up a file name **before** you send a file **to** the DNC computer for storage.
- **Autaname** is a secondary method to receive files at the DNC computer. The filename is **embedded** in the CNC program itself.
- **Reset** is the string of characters that resets the Remote function directly from the CNC control.
- **Capture** is the string of characters that precedes data output from CNC controls or other devices such as **measuring probes**.
- **Directory** is the string of characters that commands **NetDNC** to send back a current directory **listing** to the CNC control.
- **Restart** sets up the string of characters that precede the command to tell **NetDNC** to "**backup**" and then re-send a file.

Each Remote Function has a Configuration Menu and Help screens

Six possible Messages can be sent back to the CNC

Messages are pre-done blocks of text formatted as CNC programs that are optionally sent back to the CNC control when certain events occur. Messages can be formatted differently for each CNC control with optional Message variables and are edited within this dialog box.

- **File NOT Found Message** is sent back to the CNC control when the requested file does not exist.
- **File Received OK Message** is sent back to the CNC control after a file has been successfully stored and named at the DNC system. Variables can be used to show the file information.
- **Unknown Command Message** is sent back to the CNC control when the syntax of the Remote Command partially matches, but the DNC system cannot resolve the request.
- **Bad Restart Request Message** is sent back to the CNC control when the syntax of the Restart request is incorrect.
- **File Already in Use Message** tells the CNC operator that a requested file is in use, usually because it is open in **The Editor**.
- **Unknown Data Message** is sent back to the CNC control when the DNC system cannot resolve the request because it is totally garbled.



Messages help the CNC operator to do Remote functions

Receiving a File at the CNC control from NetDNC

To receive a file at the CNC control in the **Remote** mode, the CNC operator creates a small program like this:

```
%  
O3333  
(SND-1234)  
M30  
%
```

This program requests a file named "1234" in the Machine Connection's Send folder on the DNC computer. The file extension is pre-set as " *.NC ". The CNC operator "punches" out this program, when it is done, he gets the CNC control ready to "read" and 1234.NC is sent back after the pre-set delay time. This method works on all CNC controls that can type in alphanumeric characters inside comments.

This type of a "Command" program works for CNC controls that can't edit comments, such as the Fanuc 6 Series:

```
%  
O8000  
/F123456  
M30  
%
```

This Command program requests a file named 123456.NC in the DNC computer's Send folder for this Machine Connection. It is stored as a separate program name (O8000) on the CNC control (like a sub-program) and is edited and reused each time for **Remote**. The string of characters " /F " can be configured to any string, but it should be a non "move" code to avoid any problems if this sub-program is accidentally run.

A partial or full path can be specified (including network drives) if the file is in a different place than the pre-set path.

```
%  
O7000  
(SND-C:\DNC\FADAL\PART-4454-321A)  
(PUNCH O7000, WAIT 5 SEC, TYPE TA,1)  
M30  
%
```

This O7000 Command program requests a file named "PART-4454-321A.NC" from the DNC computers C:\DNC\FADAL directory, which is different from the pre-set path. If just the file name is given in a **Remote** Send request, it is automatically sent from the Machine Connection's Send folder. **Remote FULLY** supports Windows XP to 95 "long" file names.

Also note the instructions for the CNC operator. The previous examples shown were referenced to Fanuc and Fadal controls, but **Remote** will work on **ANY** CNC control that can store more than one program. If your CNC control keypad does not have the "\ " and " : " characters, use " / " and " # " and they will automatically be substituted.

Sending a directory listing back to the CNC control

Another feature of Remote is the ability to receive a directory listing at the CNC control of all of the files that are available for downloading. To do this the CNC operator punches a small program like this:

```
%  
O8000  
(RMTDIR)  
M30  
%
```

The "RMTDIR" command sent to the DNC system will return a program to the CNC control that shows all of the programs that the operator can download. This is a listing of all of the programs in the Machine Connection's Send folder with each program listed on a line by itself. Optional variables can also show each files size and last date saved.

```
%  
O1234  
(16-22-40-REV1.NC)  
(BAR.NC)  
(O4569.NC)  
(OPERATION#12.NC)  
and so on for the entire "Send" folder.....
```

Using a "Wildcard" Send Command

This file name listing shows all of the files in the Send folder that match the pre-set file mask of "*.NC". Also shown is a mixed combination of "short" 8.3 format file names and "long" file names that are now possible in 32 bit operating systems. A "template" can be created for each Machine Connection to format the Directory output file correctly for that particular CNC control.

```
%  
O7001  
(SND-105)  
M30  
%
```

An option for **Remote** is to send a "partial" or "wildcard" Send Command. In the example on the left, if just a single file that started with " 105 " then it would be sent. If more than one file started with " 105 ", then a list of those files would be sent to the CNC control as a formatted program as shown on the right.

```
%  
O8001  
F1(105621.OP1)  
F1(105621.OP2)  
F1(105624.NC)  
And so on.....
```

Using a “Wildcard” Send Command (continued)

```
%  
O8001  
/F1(105621.OP1)  
F1(105621.OP2)  
/F1(105624.NC)  
And so on.....
```

When the CNC operator receives the list of multiple files, he adds a block delete character “ / ” in front of the single or multiple files that he wants to download, and punches the O8001 program back to **NetDNC**. The DNC program then sends back those files to him. **NetDNC** can have two different Remote Send commands, one for getting a single file, and one for this multiple file situation. Two different strings of characters can also be sent before each Send command. This allows the use of embedding either “ TA,1 ” or “ DNC ” strings before sending programs to a **Fadal** control to automatically put it in either Memory or Direct DNC mode.

Sending a file to NetDNC in Remote for storage

The CNC operator just needs to punch out the program to **NetDNC** in the normal way; if the DNC computer is setup in **Remote** for that Machine Connection, the Autaname function will automatically name the received file on the DNC computer and save it to the Machine Connection’s Receive folder. The file will be named according to the first comment like this:

```
%  
O5523  
(4623.TXT)  
N1 G0 G90 G80 G40  
N2 G43 Z 2.0  
Rest of CNC code.....
```

Using Autaname, the file on the left will be stored as “4623-2.TXT” If your CNC control can’t use comments, then using “ O ” for Autaname as on the right will also work and store the program as “1234” This also works if the CNC control output is “ :1234 ” and it also can be set up to be stored as “O1234”, by adding the “ O ” to the file name.

```
%  
O1234  
N1 G0 G90 G80 G40  
N2 G43 Z 2.0  
Rest of CNC code.....
```

Sending other data to NetDNC

Remote can also be used to store other types of data that can be output from CNC controls, such as formatted values output from probing macros or SPC gage output, as long as the data is preceded by the string of characters specified in the **Capture** section of the **Remote** dialog. CNC controls that have Macro capability (DPRINT) can output text messages in ASCII text form from their RS-232 ports and Memex is using this to develop an add-on programs for **NetDNC** that will track the usage of multiple CNC machines in a production environment.

The Remote Restart function

When doing direct (drip feed) **DNC** operations, the CNC operator can stop the flow of data when a tool breaks, and then send a small program back to **NetDNC** that will command it to “backup” and restart the sending of the file. The Remote Restart request can tell **NetDNC** to backup a given number of lines, backup to an absolute line number, or backup to a “bookmark”. A bookmark is a pre-set code in the CNC program, usually at a tool change, where the CNC program could be started over. Your CAM system post processor can be configured to insert this code at each tool change. After **NetDNC** receives the Restart request, it backs up to the requested position, then waits for the CNC operator to hit Cycle Start to begin cutting again.

Restart can also be done from the **NetDNC** interface at the DNC PC, by stopping the “flow” of data at the CNC control, then selecting Operations-Restart from the DNC Main menu. Your current file will appear on the screen with the line highlighted where the communication was stopped. Simply scroll backwards through the file until you find the spot you want to restart, and pick OK. The file position will back up, ready for the CNC operator to hit Cycle Start

Because our **NetDNC** software is built on a powerful 32 bit multi-threaded communications “engine” that enables true multi-tasking, any CNC control that is connected can request a file at any time. No switches to change, no tapes to find, no more interruptions of your programmer to get the next job, no more coffee breaks to get a program, and no more yelling across the shop to get a new program. Production shops that may never use the direct DNC (drip feed) mode of communication need **NetDNC** just for **Remote**. And in fact, over 40% of our customers buy the DNC software just for **Remote**. If you have any questions regarding **Remote** or any other of the features of **NetDNC**, please contact Memex Automation.

Memex Automation Inc.

777 Walker’s Line, Burlington, Ontario Canada L7N 2G1 - Phone: (905) 635-3040 Fax: (905) 631-9640

www.memex.ca