remoDnc^{5.0}

RemoDNC version 5.0

Windows XP, Vista, Windows 7 and Windows 8 (Pro)

February 2013

Installation and user guide

Document Revision 4
February 12. 2013

remoDnc^{5.0}

RemoDNC version 5.0

www.remodnc.com

support@remodnc.com

Remo DNC version 5.0

for Windows XP, Vista, Windows 7 and Window 8 Desktop.

Remo DNC is an easy to use file transfer program specifically designed for Uploading and Downloading of programs and other data to and from CNC machines.

Remo is small enough to work on laptops and older computers and those serving just one CNC machine. But this version is a modern multi-tasking 'MDI' (Multi Document Interface) application allowing you to run multiple DNC processes at the same time.

Remo DNC 5.0 has been especially designed for use with the very latest versions of Windows such as Windows 8 and Windows 7, but it works just fine on Vista and Windows XP as well.

Remo DNC has larger buttons and menus which you may find interesting if your DNC computer has a touch enabled display. Initial setup may require access to keyboard but once it's configured you'll find that many repetitive day to day operations can be achieved with just one or two mouse clicks **or a tap with a finger on touch enabled displays.**

Remo can also operate in DNC/DripFeed mode where you can pause, stop, restart, restart from any line (for example after a tool break) and even single step through through lines of your program.

REMO Multi-CNC Server. If you have a large machine shop with a number of CNC machines the last thing you want is CNC operators crowding around a computer or running and shouting across the shop. With Remo CNC Server enabled many types of CNC can request, send and receive files without the operator leaving the CNC console.

RemoDNC is the ideal solution for operators who don't require expensive CAD/CAM functionality but do want simple, cost effective, DNC Software.

Introduction

This version of Remo DNC is a Windows "MDI" type application. (Multi-Document Interface) where the main program window (seen below) is a container for a number of separate DNC task windows. MDI technology allows RemoDNC to multi-task to manage, send and receive data to more than one CNC at the same time.

The Remo DNC Management 'container' window.
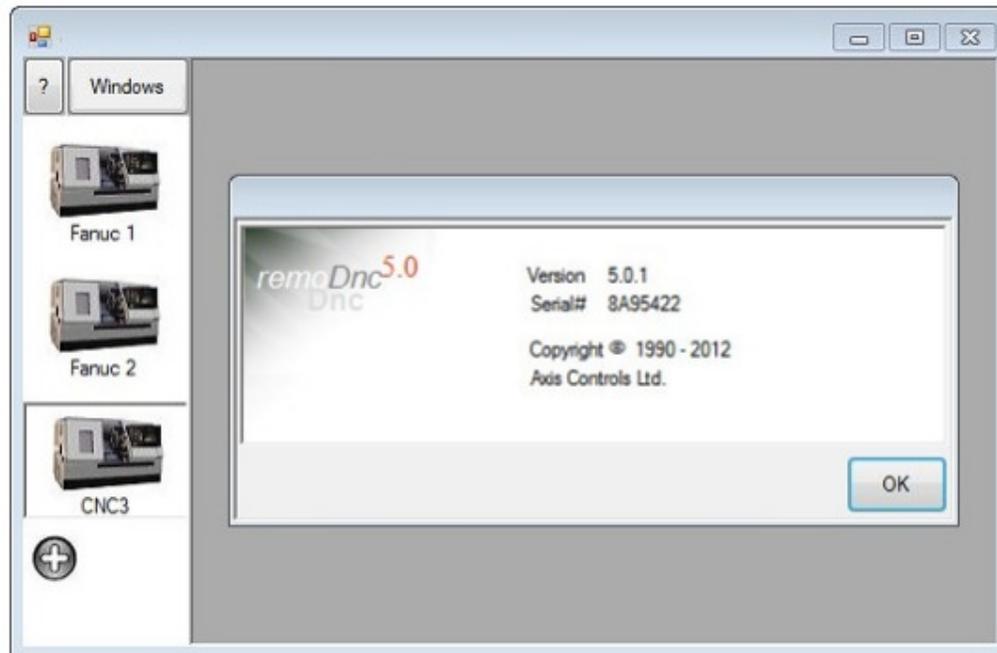
You can run a number of DNC tasks at the same time - drag and size the windows any way you want.
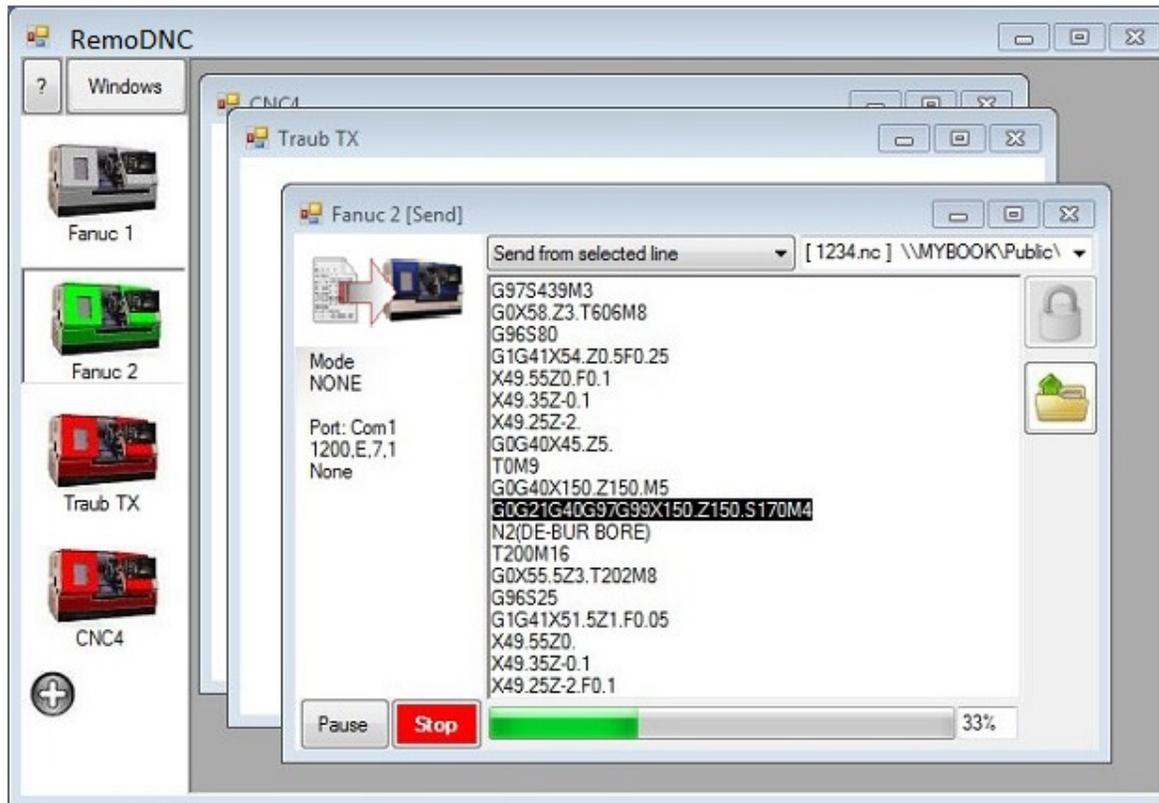
You can run a number of DNC tasks at the same - drag and size the windows any way you want.

Here we're sending to two CNC machines at the same time. We selected the option to "Tile Horizontally" and easyDnc sizes and arranges the windows automatically giving us a clear view of both DNC processes.
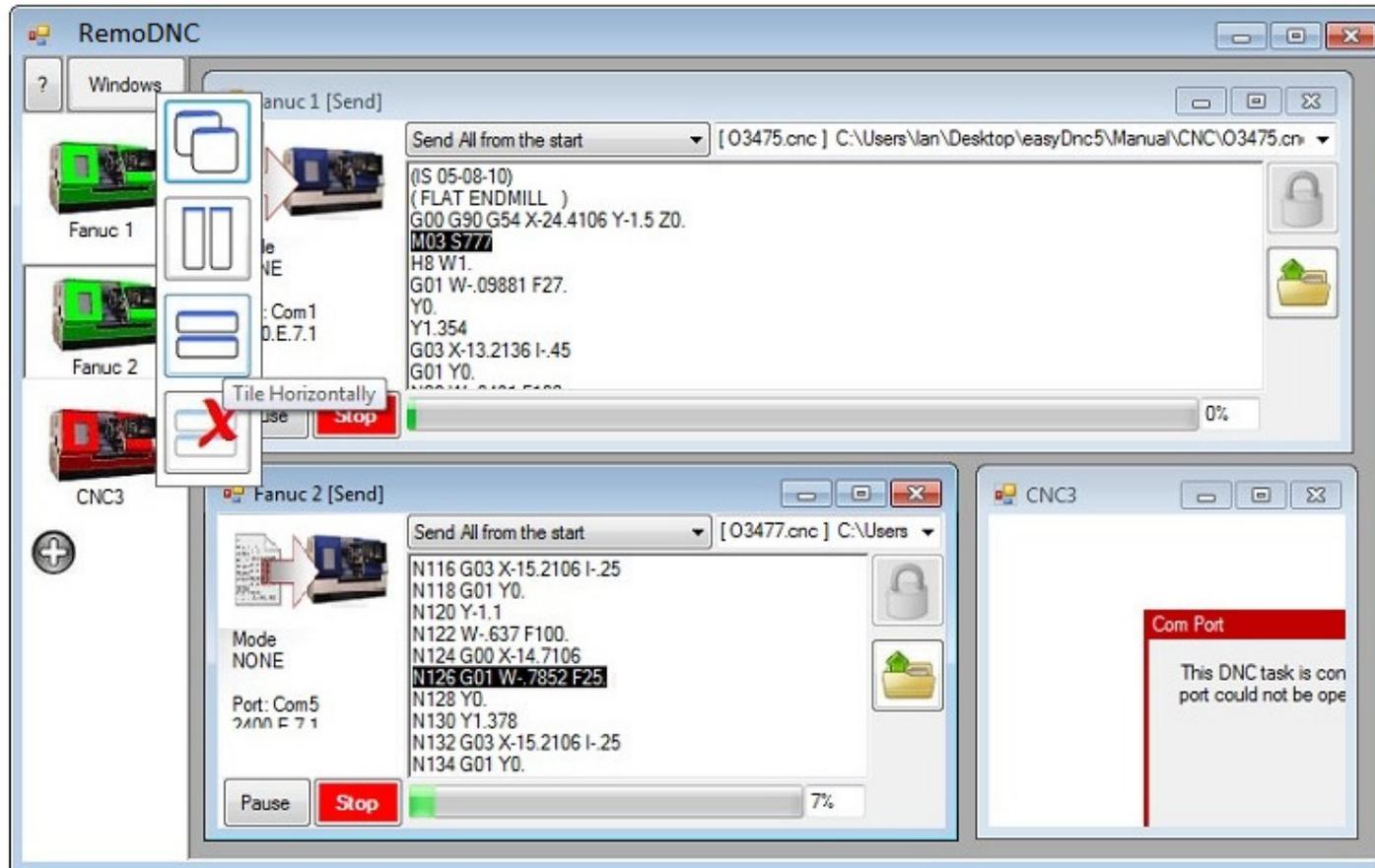
Installation / Configuration

In most case easyDnc will be installed using our downloadable installer package available from our website at www.easydnc.com

When the installation process is complete you'll find easyDnc5 icons in your windows start menu and on your Windows Desktop.

The first time you use easyDnc it will show one CNC icon in the panel. You can add more CNC icons by clicking the (+) button.

Each CNC icon is a machine profile. It contains the DNC configuration for one CNC. To edit a machine profile you click the CNC icon and select "Setup" from the menu.
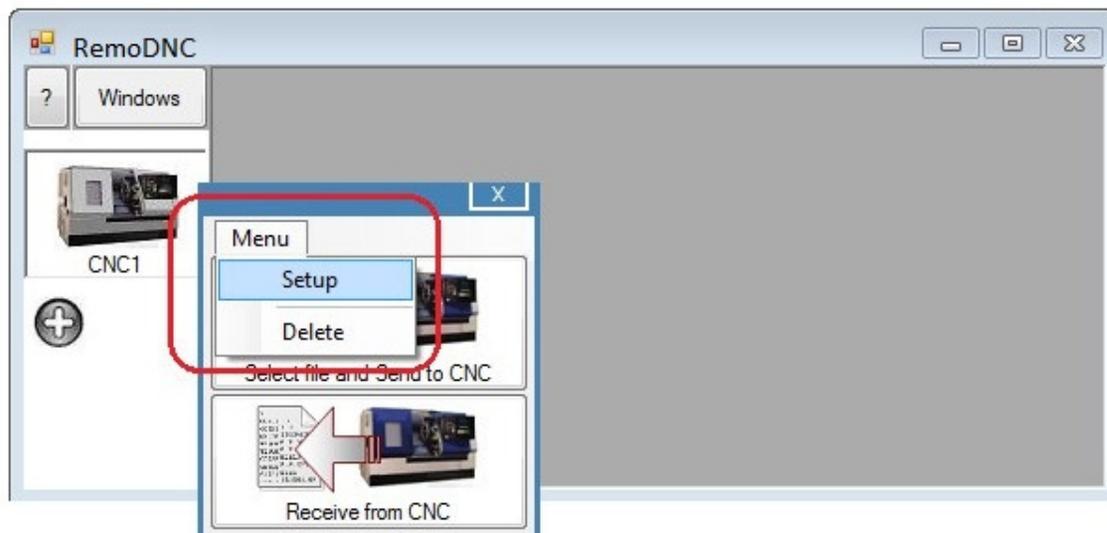
Installation / Configuration

When you select 'Setup' from the menu (Shown on the previous page) the setup window appears.

At first this might look a little complicated but the good news is that you can ignore most of the options – easyDnc will do most things automatically so the defaults will work fine in many cases.

```
CNC1

                    NC Name              Mode              Com Port
                    Fanuc 1              NONE        ▼     Com 1         ▼
                                                           [ Detect Ports ]

    Com Port Settings

       Baud/Data rate      Data Bits and Parity    Stop Bits    Flow Control
       2400         ▼      7 bits, Even Parity ▼   1       ▼    XonXoff          ▼


    [ Send (To CNC) ][ Receive (From CNC) ]

    End of Block           Remo Reqst Tag
    CRLF            ▼       REQST                  ☑ Allow/Send Comments (Text in Parenthis)

    Block Delay            Remo TX Delay           ☑ Smart Pace          Advanced
    0               ▼      5              ▼        ☐ Block Space          Options


    Default Data Folder (Files from the CNC)
    C:\Users\Public\Music\Sample Music                              [ Browse ]


    [ Help ]                                        [ Cancel ]  [ Save/Exit ]
```

Receive Data from the CNC

To receive data from the CNC you put easyDnc into 'receive' mode.

Click the relevant CNC icon to open the menu and select the option 'Receive from CNC'
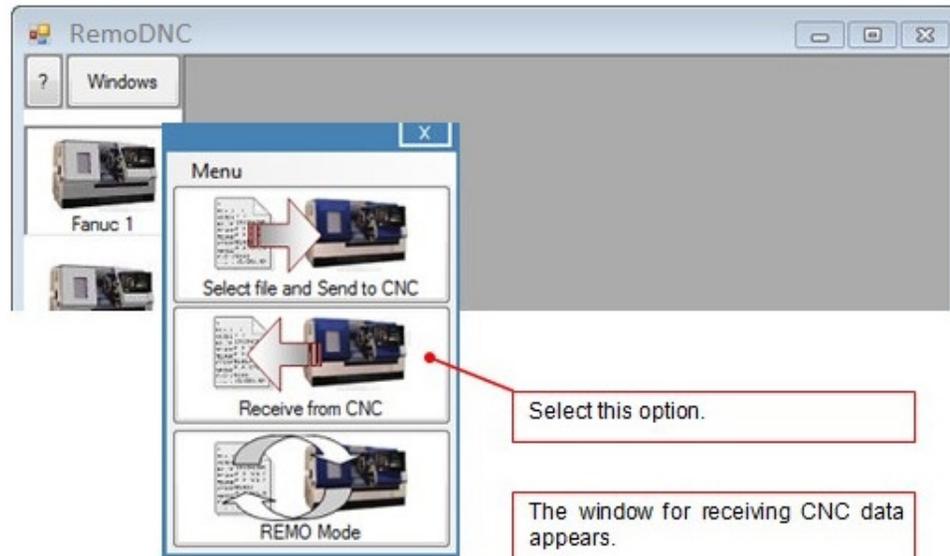
Select this option.

The window for receiving CNC data appears.

Now go to the CNC console and send / punch the program/file from the CNC.
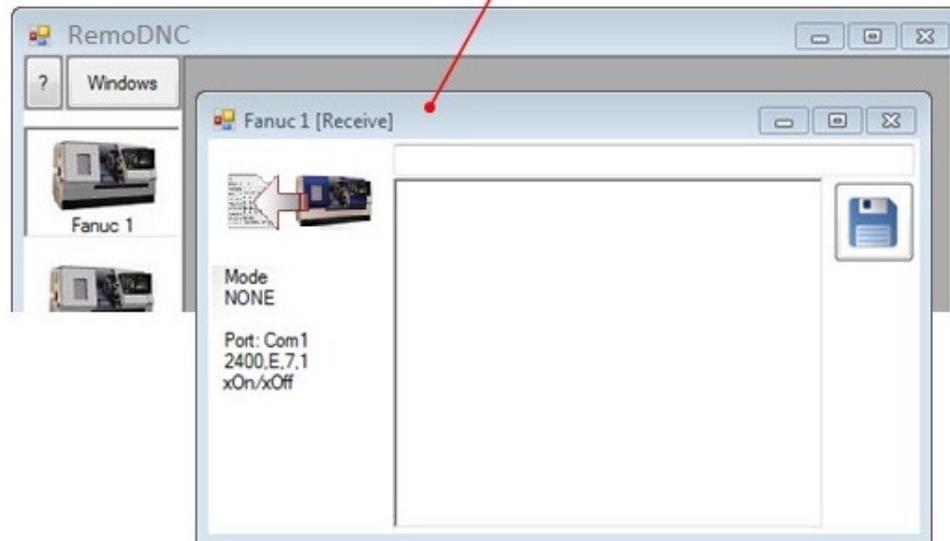
Receive Data from the CNC

As soon as the CNC begins to send the data you should see it appear on easy Dnc's receiving window.

The receive window also functions as an editor. You can edit directly into the window to change the data then save it to disk or even send it back into the CNC.

Use the 'disk/save' button to save.

To send the data back to the machine click the ➜ arrow (top right corner of the window). The text is copied into the easy DNC 'Transmit' window ready to be sent to the CNC.

```
Fanuc 1 [Receive]                              [_] [□] [✕]

                                          ➜        [💾]

              X9.668 Z-12.4
              G02 X9.4 Z-12.9 I0.866 K-0.5
              G01 Z-13.0
              G02 X11.4 Z-14.0 I01.0 K0.0
Mode          G01 X13.0
NONE          G00 G40 X20.0 Z5.0
              G40 Z150.0 M05
Port: Com1    M01
2400,E,7,1    T11 D11 M8
xOn/xOff      G95 S200 M3
              G00 X14.0 Z2.0
              G00 X14.0 Z5.0
              G40 X150.0 M05 M09
              Z150.0
              M30
              %
```
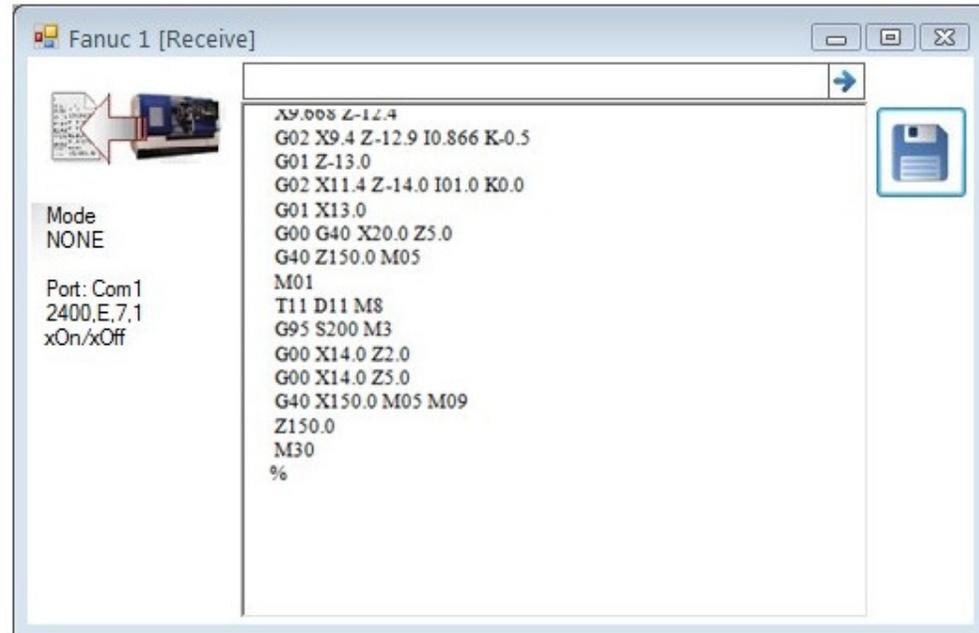
Sending Data to the CNC

So that the file can be sent to the CNC it needs to be loaded into the eas yDnc transmit buffer/window.

Click the relevant CNC icon and select the option to Send to CNC.
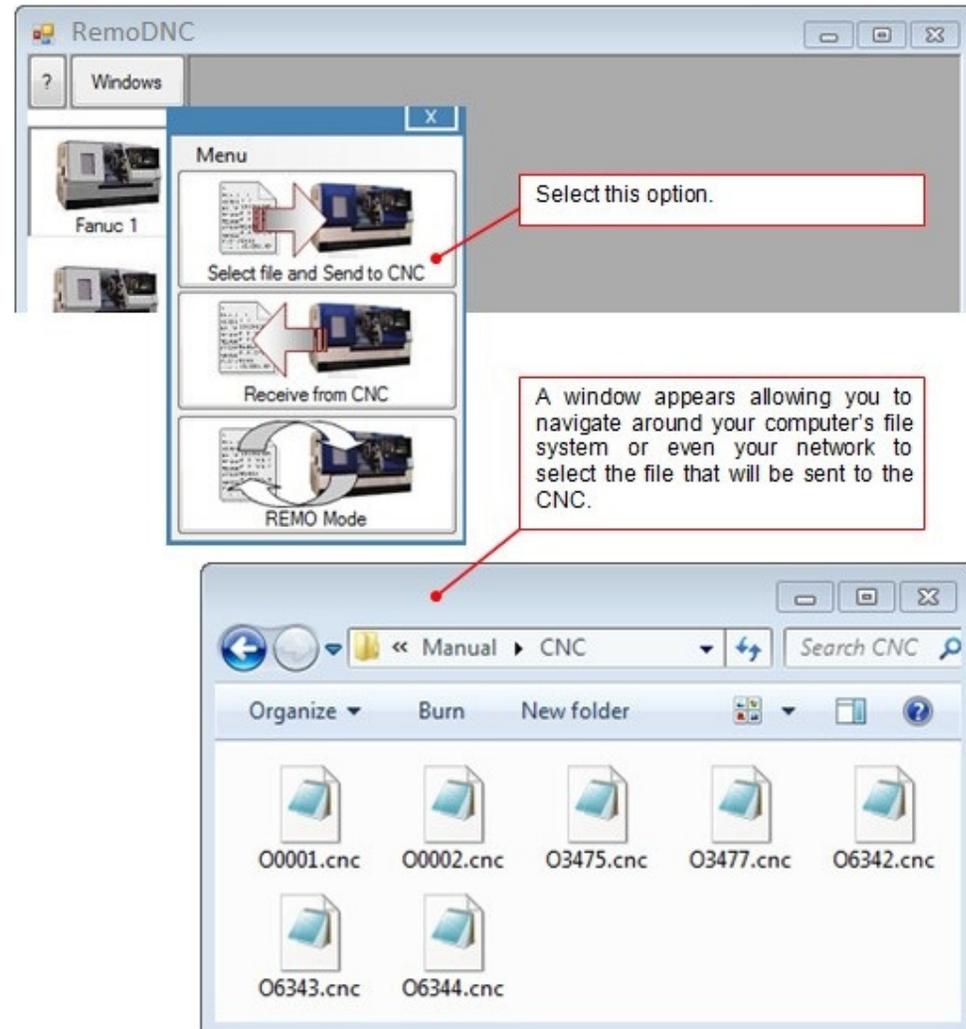
In our example we'll select the file O3475 and show how to send it to the CNC on the following page.

Note:
It doesn't matter if you want to load the file into CNC memory or DNC Drip-Feed to the CNC. The procedure is the same.

Select this option.

A window appears allowing you to navigate around your computer's file system or even your network to select the file that will be sent to the CNC.

A DNC window appears containing the selected program text.
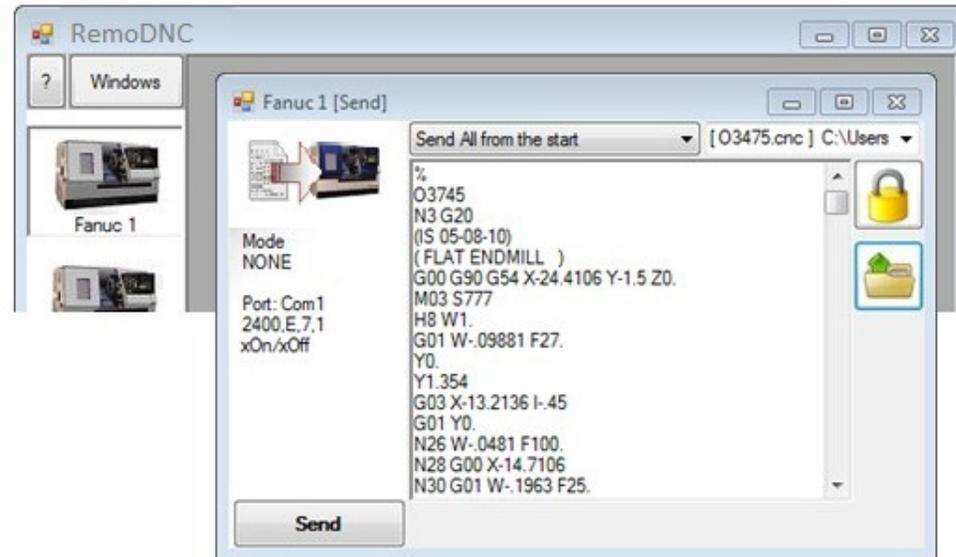
When you hit the [Send] button the data starts feeding into the CNC and you can see the progress as data scrolls up the through the buffer window and the progress bar moves left to right.

Also notice that the Send button has been replaced by other buttons allowing you to Stop or temporarily pause the transmission.
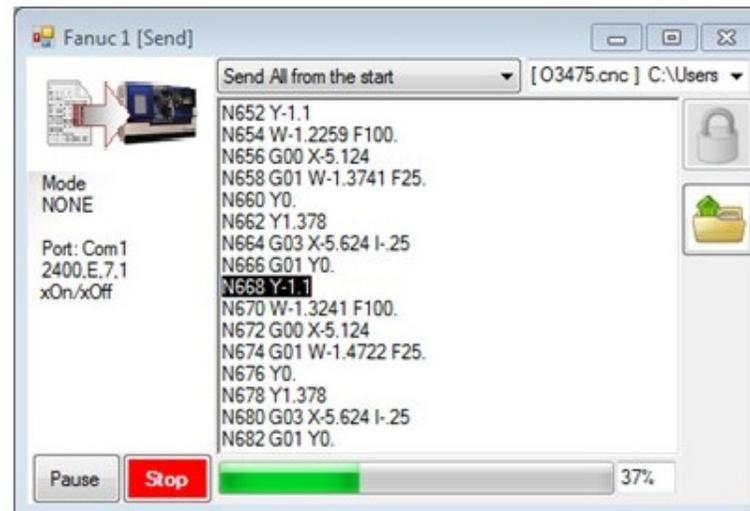
Drip Feeding (DNC) to CNC

Remo DNC has no special drip feeding mode. It's automatic. But your CNC probably has a different switch setting. For example a typical Fanuc would be switched to 'Tape Mode' for drip feeding and then press Cycle Start on the CNC. At this point nothing will happen – press the'Send' button on the DNC Software to begin the machining process. (See your machine documentation for setting/options related to your specific type of CNC

Although Remo DNC doesn't need any special DNC drip feed mode it does offer a couple of interesting options which you might find useful when DNC drip feeding.
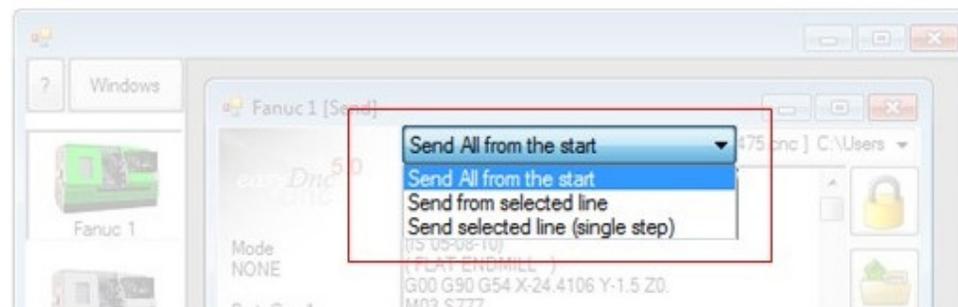
Send All From the Start

No matter what part of the program you're viewing, when you select this option and click the Send button RemoDNC will rewind right back to the top and start sending from the very beginning of the file.

Send All From Selected Line

When you click the Send button RemoDNC will start sending from the currently selected line.

Send Selected Line (Single Step)

When you click the Send button RemoDNC will send the currently selected line and immediately apply the 'Pause' button. You can repeatedly click (deselect) the Pause button to Single Step through your program.

When drip feeding long programs you might experience a tool break. When you're ready to restart DNC you wouldn't necessarily want to go right back to the start. You'd probably want to select a specific line and run DNC from that point. If you weren't sure about a specific area of your program you might want to single step through it.

During drip feeding at full speed you can hit pause and switch the mode to single step DNC - for example if you wanted to single step through an area you wanted to observe closely - and at any time switch the DNC back to full speed machining.

Remo. (MultiCNC Server)

In many cases the CNC machine may be located a distance away from the DNC computer or maybe you've got a number of machines connected to a central computer. In either case the last thing you want is machine operators running or shouting across the shop or crowding around the DNC computer.

If your CNC machines were based on a modern operating system then they might be connected to your company's ethernet networks and be able to browse through a remote server's files. In most cases though the NC control just doesn't have any networking capability so can't do anything like that.

One thing most CNCs can do though is send and receive files via RS232. They can't request files **from** computers but they can send files **to** computers. So, if we created a small file on the CNC and wrote our request into that file and sent that file to the computer. As long as the DNC computer, in this case 'RemoDnc', was expecting such a file it would be able to open the file, read the request, and complete whatever action was contained within. The good news is this version of RemoDnc has been programmed to expect such simple file requests from pretty much any CNC as long as the CNC is capable of punching a CNC program..

Before you can use the server mode within RemoDnc you need to enable it and set it up. We'll explain that in a moment. First we'll give an example of what you can do with it. In our examples we'll describe usage with a machine like a Fanuc because there's lots of other makes and models of CNC which adhere to the same ISO data standards. But on following pages we'll also show a couple of alternatives so you can adjust this method to suit other machines.

Once RemoDNC is running (listening) in Remo mode you can leave the computer and do all of your work at the CNC console.

In the CNC control create a small program which isn't used by the machine. We call this program 'The Runner' (because we send it, just like a runner, back to the computer with our message) Lets use program number 9999 as our runner (O9999). You can of course use any program number you want. In our example we want Remo to send program 1234 into the machine. On a Fanuc we'd create a small 'runner' program like this:-

```
%
O9999
(1234)
%
```

That's it. Just those 4 lines. When you're ready, send (Punch) the Runner to the computer and quickly switch the CNC into 'Input/Read' mode. When Remo receives the runner the DNC Software opens the file, sees the text (1234) and knows that the CNC is waiting for program 1234. The DNC Software pulls the file from the machine's default folder, loads it up and sends it to the CNC.

Store the runner program on your CNC so you can use it again. Next time you use it just edit the text between the braces to request any other program.

This page gave a quick example of what can be acheived with Remo. On the next pages we'll explain how to enable and set it up.

Remo setup

Open RemoDNC setup and select the 'Remo Setup' tab and then check the box to enable Remo.

Remo Reqst Tag

When Remo is enabled it's always 'listening' for data from the CNC. Such data would usually be an incoming file which will be saved to the CNC's default program folder. But when the CNC sends the Runner you need RemoDNC to recognise it and read the instructions it contains.

The Reqst Tag tells RemoDNC how to recognise the runner. This can be the program name. In the picture to the right you'll see we've entered O9999 as the Reqst Tag. We're telling Remo that we're going to use program O9999 as the runner. A typical Runner file looks like this:-

```
%
O9999
(1234)
%
```

Save this file on the CNC and whenever you want Remo to send a specific program just edit the text between the braces and punch/output the file to Remo. This example tells Remo to load program 1234 into the CNC.

Note:- See more Runner examples on the following page.

Remo TX Delay (Time period in seconds)

The length of time Remo will wait before sending the file to the CNC.

Remo RX Timeout (Time period is seconds)

When receiving a file that will be stored to disk this is the length of time Remo waits at the end of file before closing the file and saving to disk.

Incoming Program Names

When a file is saved to disk Remo DNC needs to scan the file to extract the f name. Setting this option to 'Numeric' tells Remo DNC to ignore letters and o symbols and expect a numeric program name. For example a Fanuc name is usually O5555 with a preceding letter 'O'. Numeric mode tells Remo DNC to ignore the 'O' and save the file as '5555' . Setting this option to Alphanumeric would tell Remo DNC to include letters and therefore save the file as O5555

Extension is appended to the filename when saved. eg 5555.CNC It's also appended to file requests. if the server requests file 5555 then Remo will look the default folder for a file called 5555.CNC

More detailed information on the following page

On the previous page we showed a simple example of a Runner program.

```
%
O9999              O9999 is the program name and Reqst Tag.
(1234)             1234 is the program requested by the CNC.
%
```

If you enter 'O9999' into Remo setup as the Reqst Tag then you will always use that program as the runner.

But, instead of the program name, you can use any text within the runner. For example if you setup the word **SEND** as the Remo Reqst Tag then the following syntax is allowed:-

```
%
O2315 (USE ANY PROGRAM NUMBER AS THE RUNNER)
( SEND 1234 )
%
```

In the above example the actual program name and any comments are ignored by Remo. It's simply looking for the word SEND.

If your CNC is something like an Okuma then your filenames will probably be full alphnumeric. (See the Alphanumeric setting in Remo setup) In your Okuma your runner could look like this:-

```
$SOME_FILE_NAME.MIN%
( ANY COMMENT )
( SEND 1234ABC.MIN )
```

In this case Remo would ignore everything up to the word SEND and then take the rest of that line as the requested file name and send the program 1234ABC.MIN to the CNC.

In the preceding example Remo is only seeing a requested filename and the file must exist in the CNC's default folder.

The CNC's default folder is selected in Remo setup 'transmit options'.

But it is also possible to be quite specific within the Runner telling Remo exactly where to go to find the file. For example a specific drive and folder rather than the default folder. See this runner example:-

```
%
O9999
( C:\SOME_FOLDER\1234.TXT )
%
```

Giving a drive letter and folder this tells Remo to go to that specific drive and folder for that file. The drive can be a local drive or remote network server mapped as a logical drive letter. In this case Remo's default folder and file extension are ignored.

The following is also possible:-

```
%
O9999
( PROJECT_X\1234.TXT )
%
```

Similar to the example at the top but here no drive letter is given. The path does not begin with a drive letter so Remo assumes that this is a sub folder within the CNC's default folder. (Think default folder for all programs but within the default folder there are sub folders for specific programs?)

Note the use of the backslash in the above examples. Some types of CNC do not have backslash character. You can therefore use either back \ or forward / slash:-

C:**/**SOME_FOLDER**/**1234.TXT

It's not standard MS Windows syntax but Remo accepts it.

Starting and using Remo Server

At the Computer.

A Remo process can be started to serve one CNC machine. You can start a number of Remo server processes so that you can serve a number CNC machines at the same time.

To start Remo click the relevant CNC icon and select 'Remo Mode' from the menu. (If you don't see this option then you didn't enable it in Setup)

A Remo window appears and you see the status in the lower left corner of the Remo window. "Remo Listening". You can leave this window as it is if you want to observe it. Or you can size, position or even minimize it so you can do other things while Remo is running in the background. While Remo is listening you can do everything else at the CNC console.

At the CNC Console.

To save a CNC program to the computer simply punch/output the file. Remo is already listening so you don't need to do anything at the computer. Remo receives the file, saves it to disk and resets back to Listening mode.

The first few times that you do this we'd suggest going to the computer and looking at the file that was saved to make sure that the file name has been correctly identified and that the file is stored in the folder where you expected it to be. (Of course we're duty bound to suggest you always check files are on the computer before deleting them from CNC memory)

To bring a file into the CNC you would edit your Runner program so that it contains the name of the file you want and Punch/Output the runner program then quickly switch the CNC to Input/Receive mode. Remo will receive and recognize the runner, open it, see the requested program name, load that program and send it to the CNC. When the transfer is complete Remo resets back to listening mode. (See setup pages for information about setting up your Runner program)

Select this option

A Remo window appears
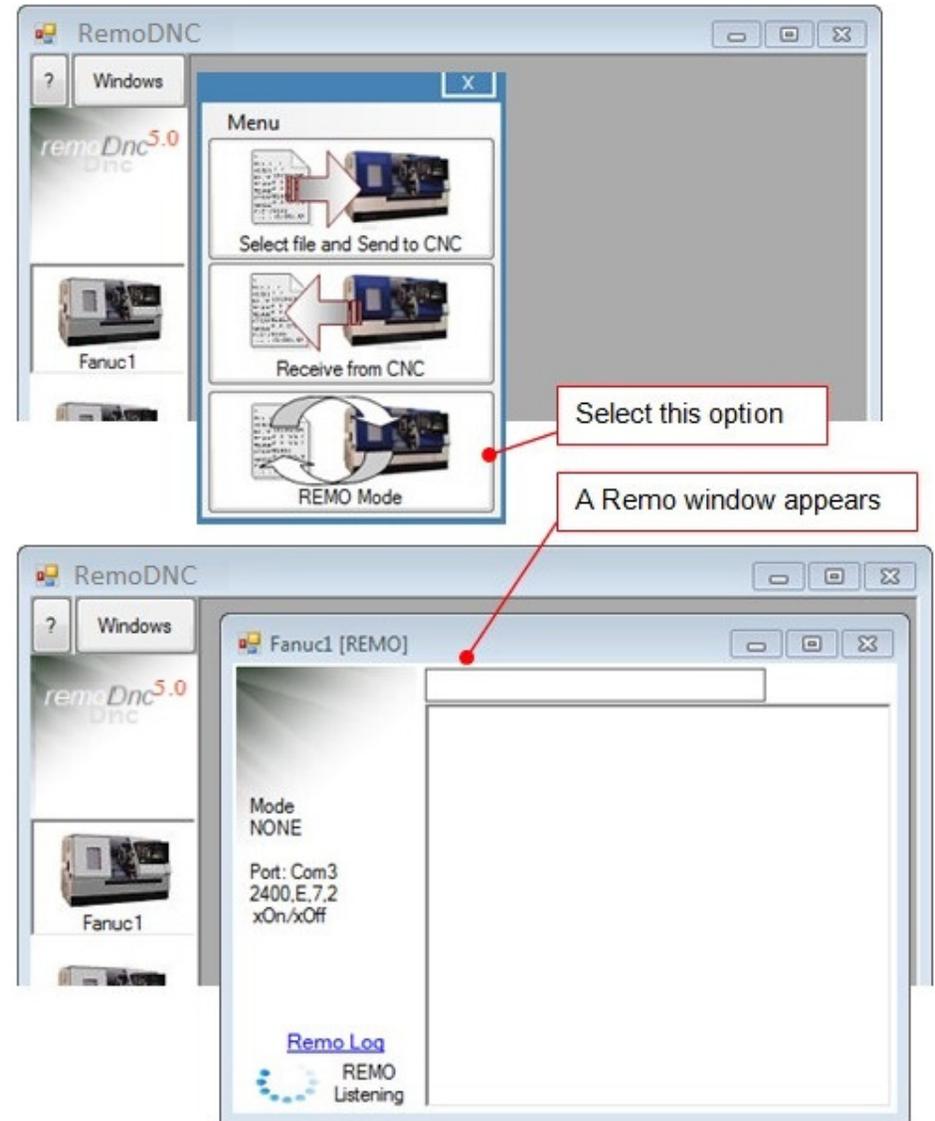
Detect DNC settings using 'Auto Detect'.

Auto Detect can usually detect the DNC settings automatically.

Select the relevant CNC icon and enter setup.

Then from the setup menu select "Auto Detect".

When you select the "Auto Detect" option RemoDNC will open a receive window and display additional instructions specific to your version of RemoDNC..

The procedure is similar for all versions - After placing a RemoDNC task into Auto-Detect mode you go to the CNC and send/punch a sizable program file which RemoDNC will analyse.

It's quite normal to see strange characters and symbols as RemoDNC scans the incoming data. After a few seconds clear CNC code should appear. When Analysis is complete RemoDNC will return to setup mode where you can review and save the detected settings.

If the DNC settings could not be detected (clear CNC code does not appear) try sending the data from the CNC again. If settings can not be detected then you will need to configure RemoDNC manually using the setup window. You can get settings for many types of CNC at the website:-

www.dncsetup.com
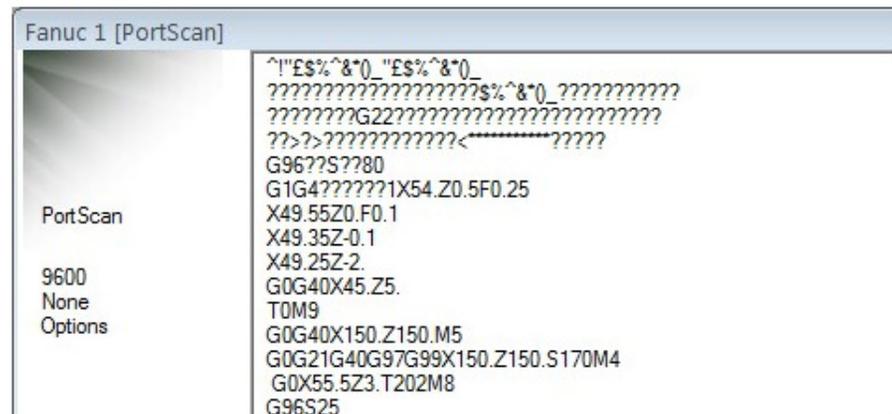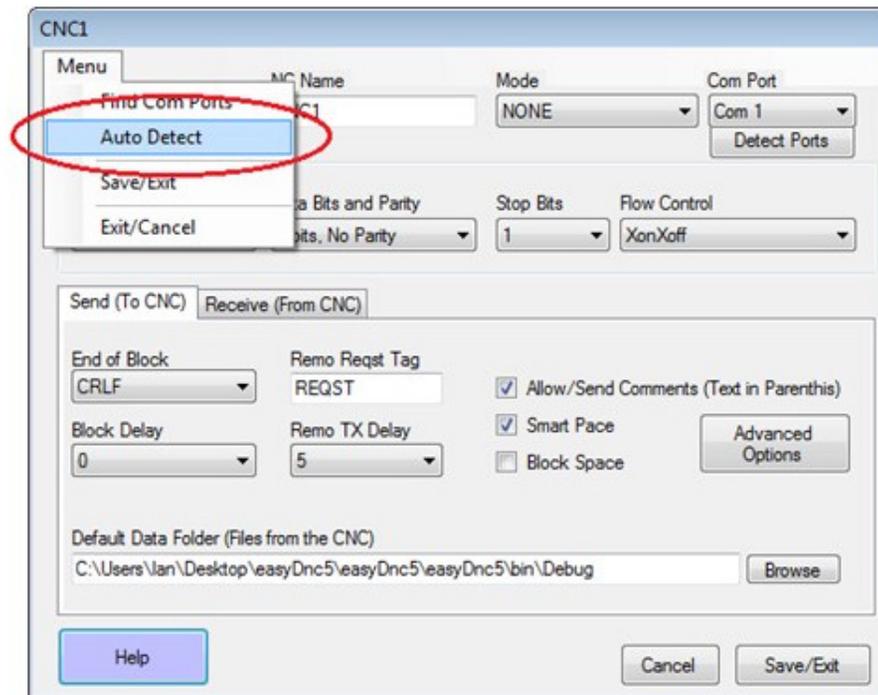
Editor.

The compact download of easyDnc 5 (PreRelease version) might not include any separate CNC Text Editor. You can download our "Free" Editor from the website dedicated to "cncCoder" the easyDnc editor.

www.cncCoder.com

If you don't want to use cncCoder then we would suggest that you use Windows NotePad or some other plain/ascii editor specifically for CNC text. Do not use MS Word or Windows WordPad. If used incorrectly both of those tools can insert windows specific control codes, fonts and other formatting information that is not compatible with standard ISO/ASCII CNC text.

cncCoder

The next couple of pages describe our editor cncCoder.

Use cncCoder just like Windows NotePad.

Or enable syntax highlighting to make things a little clearer.

When you enable syntax highlighting cncCoder switches the display color instantly – this highlighting is only visual – it only affects the computer display. It does not change the actual file text in any way.

Syntax highlighting is fully configurable. Use any colors you want and highlight as much, or as little, as you want. (See the color picker on the next page)

Note: cncCoder can be installed on as many computers as you wish at no extra cost. You can therefore use cncCoder on any number of computers in the office without having to purchase additional copies of easyDnc.

cncCoder.

Each individual part of your CNC syntax can be configured separately.

As an example maybe you're editing something where you want to pay extra attention to the X values - Simply open the color picker, select 'X' followed by a color and 'Save' and every X value throughout the display area is instantly shown in the new color.

As mentioned previously this only affects the display within cncCoder - the colors or other formatting are not applied to the underlying CNC text so does not alter the format of your CNC files or affect the machining process.

cncCoder also includes the usual features such as 'Find' and 'Find & Replace'.

Remember, you can install the "Free" cncCoder on as many computers as you wish. No need to buy more DNC software when all you want to do is edit CNC text while you're away from your DNC workstation.
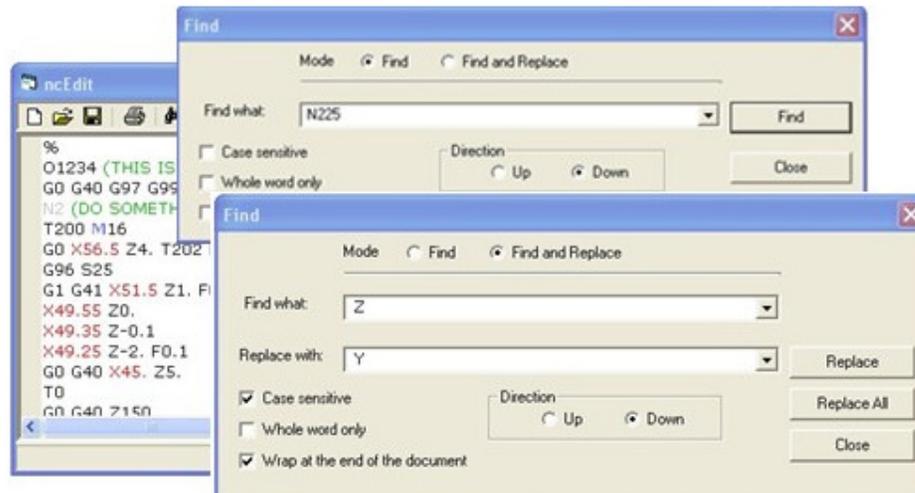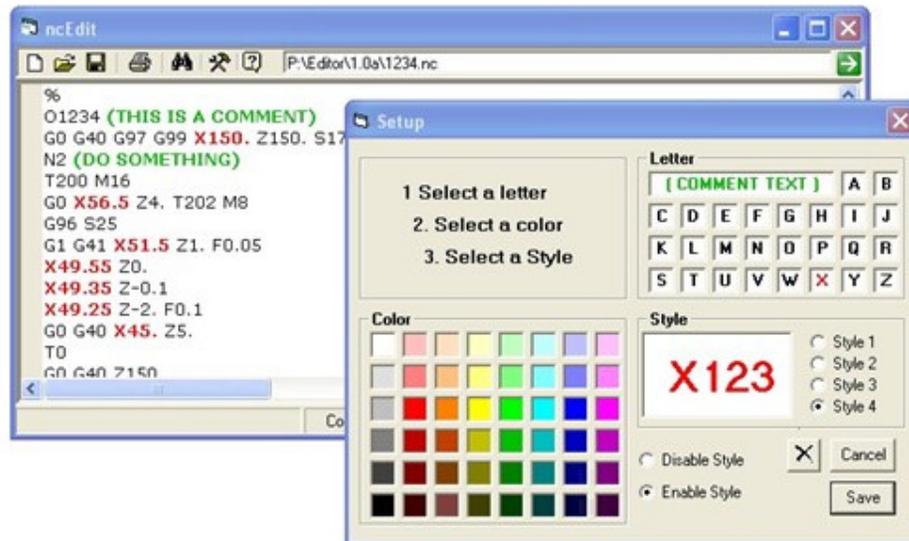
Cable Diagrams.

This 'quick' user guide does not contain any CNC machine specific settings, parameters or cable diagrams.

Please visit the website below to see settings and cable diagrams for many different types of CNC.

www.dncsetup.com

Notes.