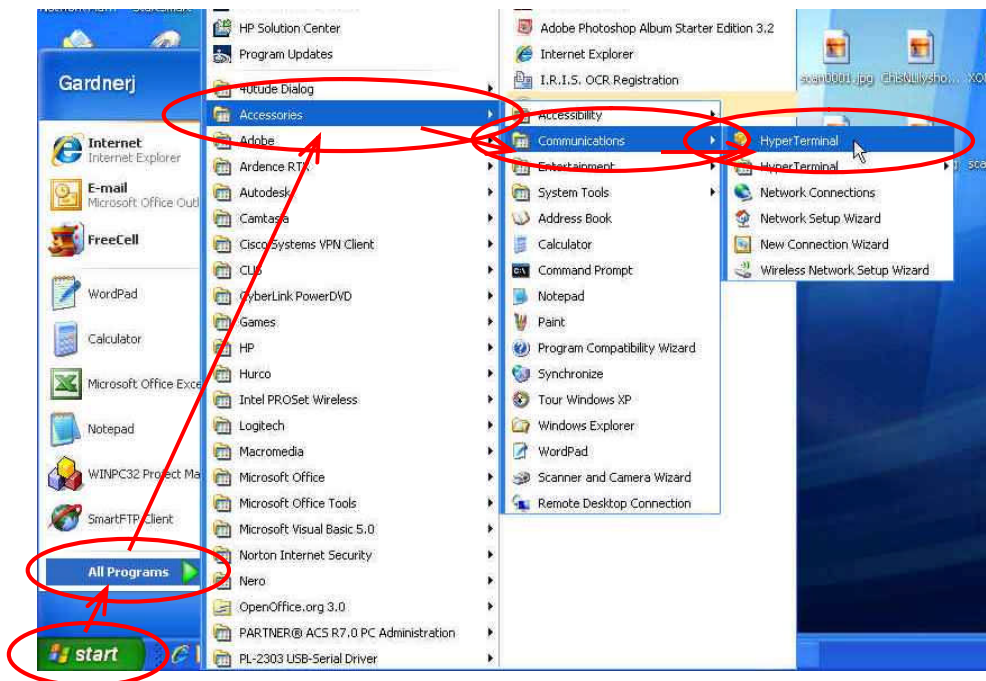


## Communicating with a DynaPath CNC through a Serial Port using HyperTerminal

HyperTerminal is a program included with various Windows® operating systems which facilitates serial communication between two computers. This document explains how to launch and set up HyperTerminal to allow communication between a computer (either desktop or laptop) running Windows (in this case, XP) and a DynaPath (or Autocon) CNC control. There may be slight variations in HyperTerminal through the various Windows operating systems.

**Step 1 – Launch HyperTerminal:** The HyperTerminal program is usually accessible in the Accessories folder. Using your mouse pointer, click on ‘Start,’ then roll your pointer in succession over ‘All Programs,’ then ‘Accessories,’ then ‘Communication,’ and finally clicking on ‘HyperTerminal.’ Refer to the picture below:



Launching HyperTerminal in this manner will start a dialog which allows you to describe and configure your serial port. This configuration can be done later using the Properties dialog box, but is easiest to do when you start the program for the first time.

Step 2 – Completing the Initial Dialog Box: The first dialog box that the program leads you through is shown below:



Enter any name you wish and select any icon. In our example, the connection will be named 'DynaPathExample.' Once you have completed your selections, click the mouse pointer on the 'OK' button. The 'Connect To' dialog box will appear.

Step 3 – Select the Appropriate COM Port:



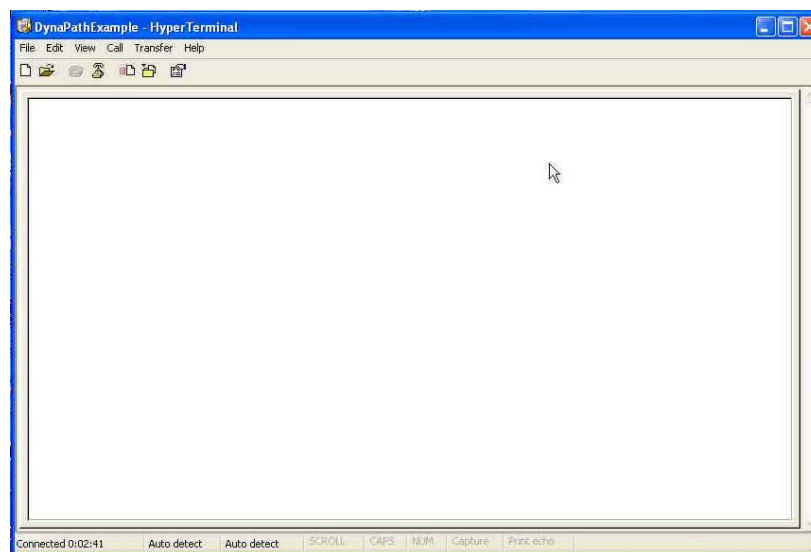
Ignore the first three entries in this dialog box and open the drop box in the entry marked 'Connect using' by clicking on the down arrow at the right side of the entry. Scroll past any modem descriptions and select the correct COM port (which will always be towards the bottom of the list.) In the example, COM3 is selected. Once a COM port is selected, the initial three entries will be "greyed out." Click the mouse pointer on the 'OK' button.

Step 4 – Set the COM Port Parameters: The next dialog box to appear is the COMx Properties box as seen below:



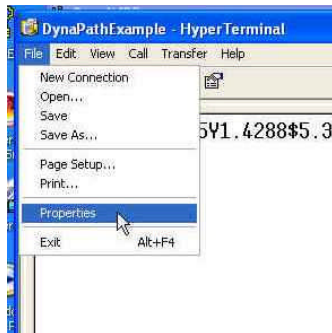
Set 'Bits per second:' to match the baud rate selected on the CNC control. Typically, this is set to 9600, but you may want to increase or decrease this setting. 'Data bits:' and 'Parity:' will be set one of two ways. Either set 'Data bits:' to 8 and 'Parity:' to None or set 'Data bits:' to 7 and 'Parity:' to Even. Leave 'Stop bits:' at 1 and 'Flow control:' on Hardware.

The initial set up of HyperTerminal is now complete. The display should appear similar to the one below:

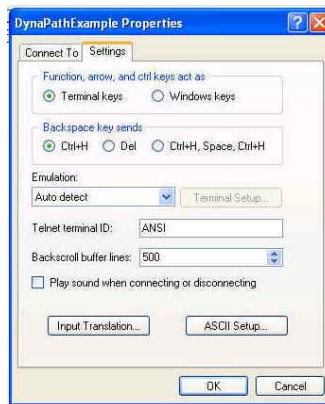


Note that it is normal for the word 'Connected' to appear in the lower left corner, whether the connection between the computer and the CNC is made or not.

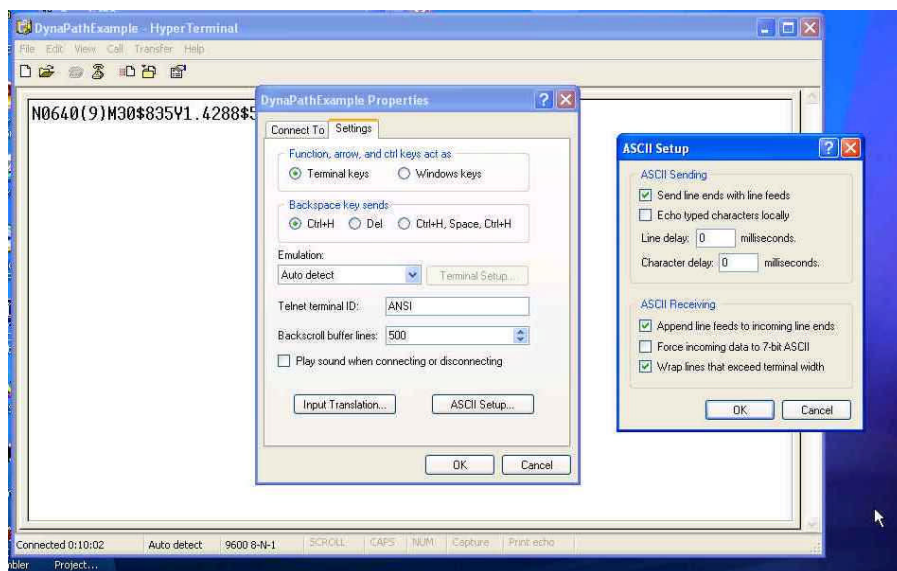
**Step 5 – Complete the ASCII Setup:** There are two more parameters that must be set for the program to function properly with the DynaPath CNC control. Open the 'Properties' dialog box by clicking on 'File' on the Menu bar, then 'Properties':



The Properties dialog box is now displayed. Click on the tab labeled 'Settings':

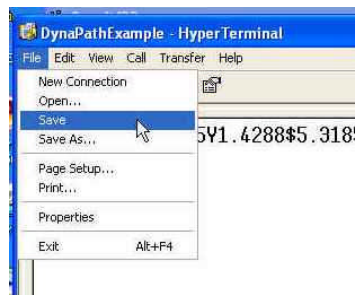


Click on 'ASCII Setup...' to open the next dialog box:



Check the box marked 'Send line ends with line feed' in the upper area of the dialog box marked 'ASCII Sending.' Also check the box marked 'Append line feeds to incoming line ends' in the lower area marked 'ASCII Receiving.' Once done, press 'OK' in each of the two open dialog boxes to close them.

**Step 6 – Save the Connection Properties:** The configuration of the HyperTerminal program should be saved at this point so that the setup does not have to be completed each time the program is used. Refer to the picture below:

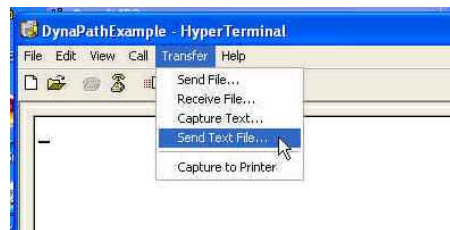


Once the configuration of the program has been completed, part programs can be sent to or received from the CNC control. Part programs on a DynaPath or Autocon control are text files and do not require any special protocol. In transferring these files, always prepare and start the receiving device first, then start the transmitting device.

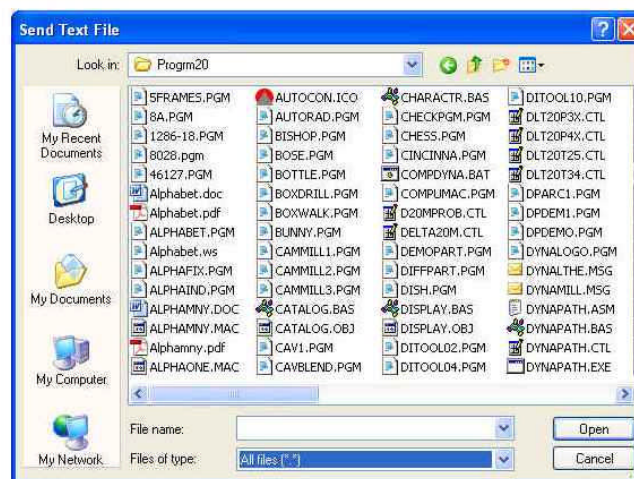
## Step 7 – Sending a Part Program from the Computer to the CNC:

Prepare the CNC control to receive the part program (the word 'Load' was used on older controls.) Refer to your Customer Information Manual for details on how to prepare the control to receive part programs.

Start transmitting from HyperTerminal by clicking on 'Transfer' on the Menu bar, then 'Send Text File...' as shown below:



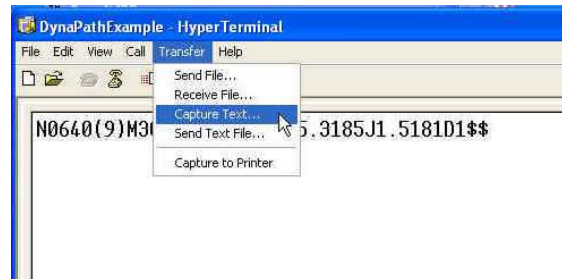
A dialog box will open, allowing you to select the file you wish to transmit:



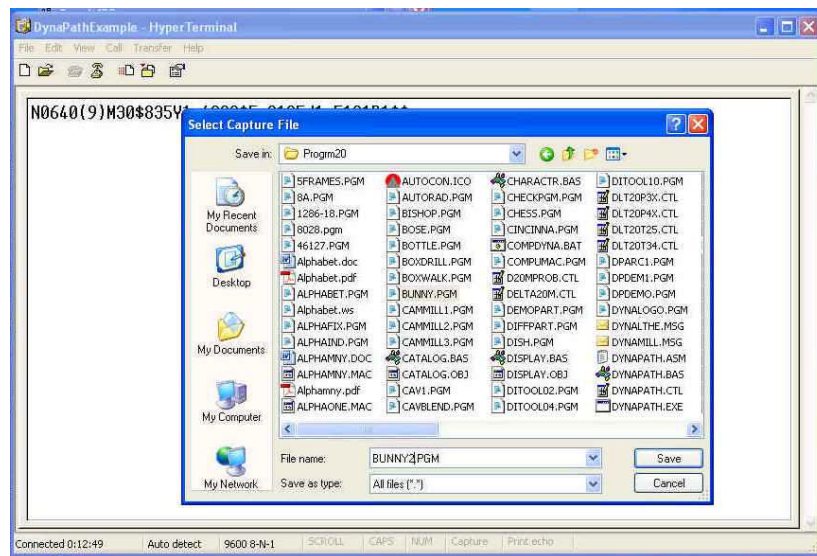
The transmission of the part program will begin as soon as you select a file, so make sure that the CNC control is waiting to receive it. You should see data scrolling down the HyperTerminal screen, as long as the part program file was formatted properly.

## Step 8 – Sending a Part Program from the CNC to the Computer:

Prepare HyperTerminal to receive a part program file by clicking on 'Transfer' on the Menu bar, then 'Capture Text...' as shown below:



The 'Select Capture File' dialog box will be opened which allows specifying the folder and the name for the file. You can either overwrite an existing file or create a new file. Refer to the following:



Once the file has been selected or the new file created, HyperTerminal will be ready to receive the part program file from the CNC control.

Start transmitting the part program from the CNC control (the word 'Record' was used on older controls.) Refer to your Customer Information Manual for details on how to prepare the control to transmit part programs.

As the information is transmitted from the CNC control to the computer, the text being received should scroll down the HyperTerminal screen.

Once transfer is complete, HyperTerminal must be told to close the receiving file. Again click on 'Transfer' on the Menu bar, and then click on 'Capture Text.' Now click on 'Stop' to close the file:

