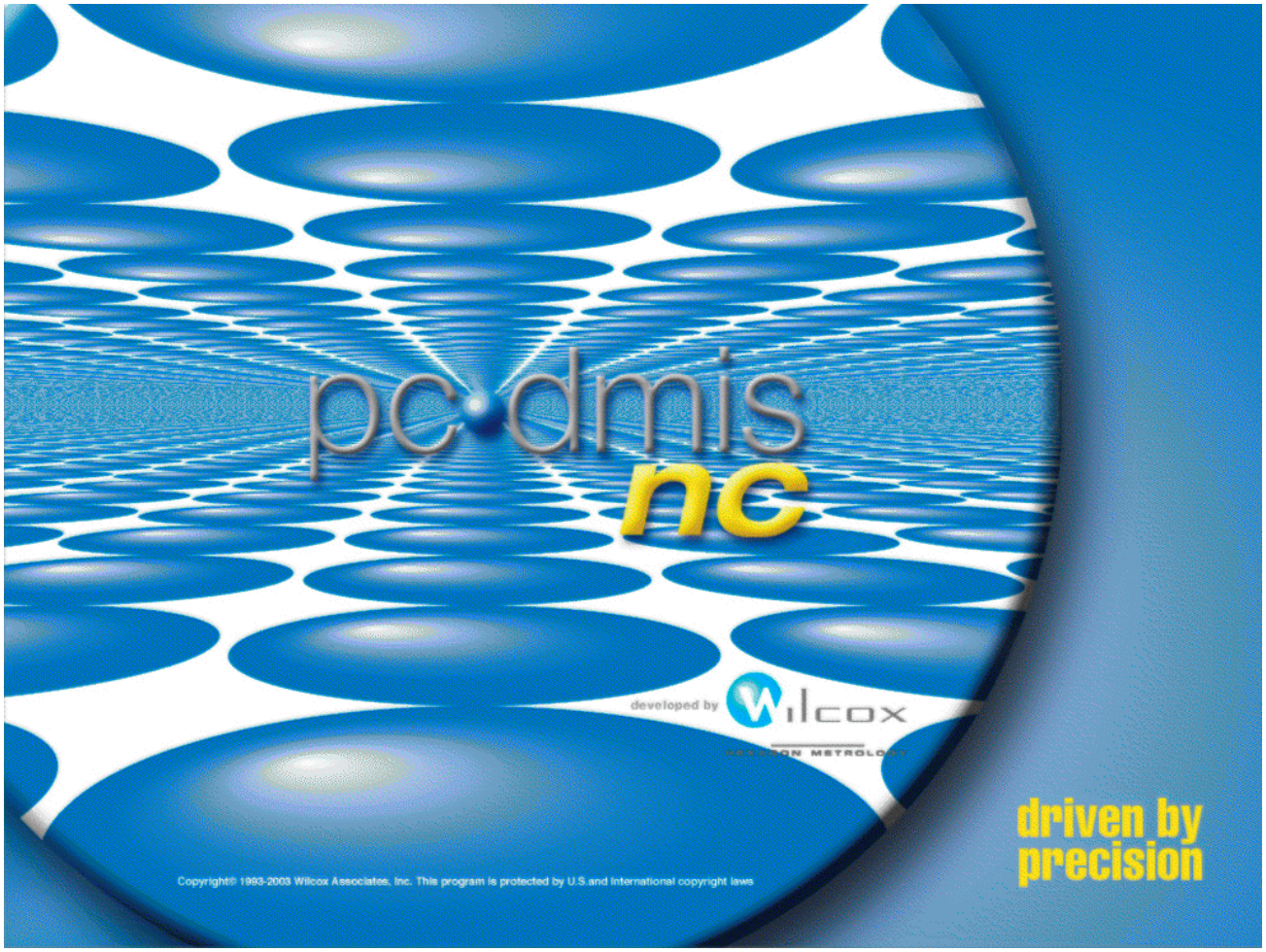
Version 1.0

PC-DMIS/NC Manual



By Wilcox Associates, Inc.

Copyright © 2004 Wilcox Associates Incorporated. All rights reserved.

Contents

Using PC-DMIS/NC 1

Introduction: 1
- Uses for PC-DMIS/NC 1
- Differences between PC-DMIS/NC and PC-DMIS 2
- Required Components for PC-DMIS/NC 6
- Files Used by PC-DMIS/NC 6
- Data Flow Chart 6

Testing PC-DMIS/NC 7
- Step 1: Install the Renishaw Probe 7
- Step 2: Install PC-DMIS/NC and Define a Machine 8
- Step 3: Perform an Offline Test 9
- Step 4: Set up the Serial Link 10

Using PC-DMIS/NC 10
- Step 1: Calibrate the Probe 10
- Step 2: Set the Work Offset 11
- Step 3: Define an NC Machine 11
- Step 4: Create a NC Program File 11
- Step 5: Start the PC-DMIS NC Server Program 12
- Step 6: Load the NC Program File 12
- Step 7: Execute the NC Program File 12
- Step 8: Analyze and Report the Data 12

Working with CNC Edit Window Commands 13
- Set Protection 13
- Use Work Offset 14
- Update Work Offset 14

Working with PC-DMIS/NC in Offline Mode 15

Using the PC-DMIS NC Server Application 16
- The PC-DMIS NC Server Menubar 16
- The PC-DMIS NC Server Command Buttons 17
- The Machine Status Fields 18
- Stopping and Resuming PC-DMIS NC Server 18
- Exiting PC-DMIS NC Server 19
- Moving Programs to the NC Controller 19
- Executing Journal Files 19
- Creating and Modifying Machines 20
- Working with the MonitorView Window 27

Glossary of Terms 33

Index 37

Using PC-DMIS/NC

Introduction:

PC-DMIS/NC (Numerical Control) is a specialized version of the PC-DMIS software that allows you to perform part measurements using NC machine tools. These machine tools are also referred to as CNC (Computer Numerical Control) machines. PCDMIS/NC part programs may be created in offline mode and used to generate an NC program.

In order to measure a part using NC tools, you will need to first fit your NC machine with a probing device. You can then execute the generated part program and return point measurements via a communications interface to the PC-DMIS NC Server program running on a computer. The PC-DMIS NC Server program activates PC-DMIS/NC and tells it to execute a PC-DMIS part program using the supplied data. You can use PC-DMIS's reporting and statistics options for the part program.

This manual discusses the following main topics. For information on PC-DMIS in general, see the main PC-DMIS reference manual.

- Testing PC-DMIS/NC
- Using PC-DMIS/NC
- Using the PC-DMIS NC Server Application
- Working with CNC Edit Window Commands
- Working with PC-DMIS/NC in Offline Mode
- Using the PC-DMIS NC Server Application

Uses for PC-DMIS/NC

Below are some situations where you will want to use PC-DMIS/NC:

Use PC-DMIS/NC for Difficult Setups

Parts which have contoured surfaces, such as turbine blades, castings or mold dies, are challenging to setup on an NC machine using the typical tools available to the machinist for these reasons:

- The datums on the parts may be derived from the surface contours, requiring a best fit to CAD data.

- Castings may need to be positioned in such a way that material is cut from all required areas.
- The shape of the part may vary such that the location of machined features is dependent on the surface.

PC-DMIS/NC alleviates these problems by allowing the part to remain on the NC machine for the measurement process.

Use PC-DMIS/NC for Large Parts

Large machines are typically required to machine large parts. If you can't measure on a NC machine you'll have to deal with the following:

- The part may need to be removed from it’s setup and placed on a CMM.
- An equally large CMM is required to measure the part. This is costly in both shop space and money.
- The part may need to be returned to it’s original setup to continue machining operations, perhaps based on the CMM measurements.

Use PC-DMIS/NC for Process Monitoring

During the machining cycle, the measurement of features and their relationships is an indicator of the stability of the process. PC-DMIS/NC can archive the data in a statistics database for immediate review.

Use PC-DMIS/NC as a Common Metrology Tool

You can use the same part programs on both NC machines and on CMMs. Since both PC-DMIS/NC and PC-DMIS use identical analysis algorithms, the computed results will always be the same.

Differences between PC-DMIS/NC and PC-DMIS

The following differences exist between the PC-DMIS/NC application and PC-DMIS:

- PC-DMIS/NC does not run interactively with the NC machine. PC-DMIS/NC creates a stand-alone NC program file which is executed on the NC controller. Embedded in the NC program file is all the information PC-DMIS NC Server needs to select the associated PC-DMIS program. The PC-DMIS NC Server program file then loads probing data from the NC machine into PC-DMIS/NC and executes the selected program.
- PC-DMIS/NC version 1.0 cannot send data to the NC controller during execution.
- Probing paths are static and cannot be changed. These paths are generated from the part model's nominal geometry prior to execution. Interactive functions that modify the path during program execution have been disabled in the PC-DMIS/NC menus. Since you create the part program in offline mode the path cannot change. For example, you will be unable to modify a path based on a previously measured alignment.

- Alignments computed from features will only be used during the analysis phase.

The following PC-DMIS topics list particular items as either supported or unsupported in PC-DMIS/NC. For information on these specific items, view the main PC-DMIS reference manual:

Automatic Features

Common Items

- Permanent Hits and Spacer – Supported. Runtime sample hits will be taken.
- Initial Hits – Unsupported. These are learn-time sample hits. There is no learning function in V1.0.
- Re-Measure – Unsupported. There is no benefit to re-measuring using the same path.
- Read Position – Unsupported.
- Auto PH9 – Unsupported. 5 axis machine tools with multi-axis wrists may be supported in later versions.
- On Error – Unsupported. PCDMIS will be operating in a Server mode where the Operator will generally not see the screen.
- Relative Measurement – Unsupported. This requires path modification based on measurement.
- NOMINALS – FINDNOMS – MASTER – VECTOR list. NOMINALS only supported.

Circle

- Circular Move – Supported. The path will be generated with straight line segments by PC-DMIS.
- Find Hole – Unsupported. Requires controller based path generation

Cylinder

- Find Hole – Unsupported.
- Circular Move – Supported.
- Use Theoretical – Supported.
- Circular Move – Supported.
- Use Theoretical – Supported.

Cone

- Circular Move – Supported.

Line

- For CMT Features Only – Unsupported.

Corner Point

- Supported since path is not dependent on measurements.

Angle Point

- Supported since the path is not dependent on measurements.

Edge Point

- Unsupported. The points on the edge are determined by the surface point(s) which is a dynamically generated path.

High Point

- Unsupported.

Regular Measured Features

In all cases, Regenerate Hit Targets is Active.

- Circle – Supported.
- Cone – Supported.
- Cylinder – Supported.
- Line – Supported.
- Point – Supported.
- Plane – Supported.
- Sphere – Supported.

Read Point

- Unsupported.

Scans

- Advanced Scans – Unsupported. A CAD model is required to generate scan paths since there is no interactive learning. The model is used to generate a move / measure point set which is downloaded to the Cnc controller for execution.
- BASIC Scans – Unsupported. These are controller dependent for generating the scan data.
- Execute Mode – NORMAL only is supported. Relearn is not supported.
- Nominals – MASTER and FINDNOMS are supported.

Moves

- Exclusion Zone – Unsupported.
- Move Sync – Unsupported.
- Rotary Table – Unsupported.

Parameter Change

Measurement Speed

- Move Speed – Supported.
- Touch Speed – Unsupported.
  This parameter is set within the Renishaw Macros
- Scan Speed – Unsupported.
- Acceleration – Unsupported. This is a machine service parameter not accessed by operators or programmers.

Probe

- Manual / DCC Mode – DCC only is supported.
- Probing Parameters – All are Unsupported. Probing parameters are defined within the Renishaw macros.
- Auto-Relative Measure – Unsupported.

Clamp Value

- Unsupported.

Temperature Compensation

- Manual compensation is unsupported. There is no mechanism in PC-DMIS/NC for the operator to enter parameters in a running CNC program for manual compensation.
- Automatic compensation is unsupported via PC-DMIS/NC. This requires custom software to access these parameters if they are available in the NC control.

NC-100 Video Settings

- Unsupported.

Flow Controls

- Supported if they do not alter the path, measurement sequence, or require measurement results inside a loop.

Calibrate

- Unsupported. All calibration will be performed on the machine as part of the Renishaw macro set. Probe information (tool offset and effective ball diameter) will be requested from the control after every tool change operation.

Basic Scripts

- Supported if they do not affect the measurement sequence or the path which is static.

Operations

- Move-To – Unsupported.
- Column Recovery – Unsupported.
- Master/Slave Mode – Unsupported.
- Calibrate/Edit – Unsupported.
- Rotate Table to Active – Unsupported.

Edit

- Markings – Supported as long as the part program when the Cnc program was generated remains the same when the program is executed by the server.

Breakpoints

- Supported but since breakpoints are a useful debugging tool, once a program is running on an unattended server, all breakpoints should be removed.

Required Components for PC-DMIS/NC

Below are the components you need in order to use PC-DMIS/NC:

NC Machine with Probing – An NC machine with Renishaw probing and probing macros installed.

Server Computer with PC-DMIS/NC – A Computer running the Windows 2000 or Windows XP operating system. This computer must have the communication interfaces to accept data from the NC machine.

Communications Link – A communications link is required from the NC machine to the server computer (the computer running PC-DMIS NC Server). PC-DMIS/NC version 1.0 currently supports a serial interface to communicate between the two.

Port lock – A hardware lock with the NC option programmed into it and licensed with the number of supported NC machines.

Files Used by PC-DMIS/NC

The following files are used by PC-DMIS/NC

PC-DMIS Program Files - The standard PC-DMIS part program (.prg) and CAD (.cad) files stored in the default location.

NC Program Files - These are the NC machine programs generated from the PC-DMIS part program and have a .cnc extension. You can place them in any location, but you should probably create a separate CNC subfolder somewhere on your hard drive.

Journal Files - Journal files (.cnj and .cnp files) contain the transactions received from the NC machine during execution of the NC program file (.cnc file). They are stored in the directory specified in the **Program** tab of the **MachineDefinition** dialog box of the PC-DMIS NC Server program. PC-DMIS NC Server also expects to find the PC-DMIS part programs (.prg files) in this directory during the analysis phase.

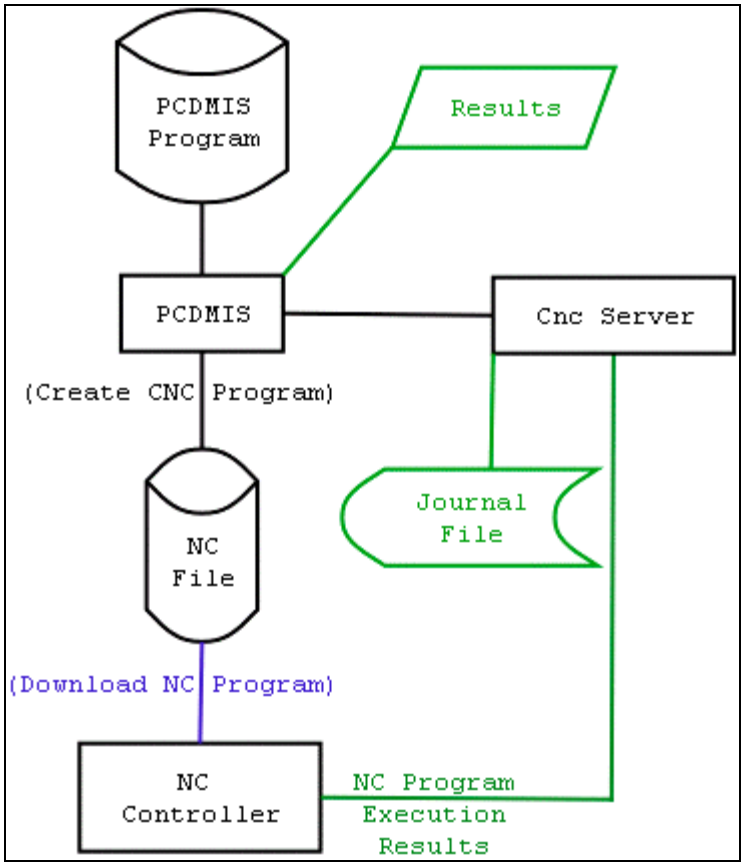
Example: If PC-DMIS/NC was installed to C:\PCDMISW\, journal files for PCDMIS program Block.prg would be stored in C:\PCDMISW\Journal\Block\.

Data Flow Chart

In the chart below, the following process is described:

1. Create a PC-DMIS Part Program
2. Open the Part Program in PC-DMIS/NC.
3. Select the **Operation | Create CNC Program** menu option. PC-DMIS/NC creates the NC program file.
4. Start the PC-DMIS NC Server program.

5. Click **Analyze Hold**.
6. Select **Download NC Program** button. The NC program file gets sent to the NC controller.
7. Click **Analyze Run**.
   - The NC machine with probing capability measures the features by taking probing hits.
   - The NC program execution results get sent to PC-DMIS NC Server.
   - The PC-DMIS NC Server generates a journal file for later analysis and review.
   - PC-DMIS/NC runs the analysis phase to generate measurement results.



Testing PC-DMIS/NC

The following topics describe the steps you should take in order to install and then test PC-DMIS/NC with a simulated NC machine:

Step 1: Install the Renishaw Probe

Using your Renishaw documentation, do the following:

1. Install the Renishaw probing system. It must allow 3 axis Vector Measuring Cycles (#9821).
2. Install the Renishaw probing macros.
3. Verify installation using the vector calibration macro, O9804.

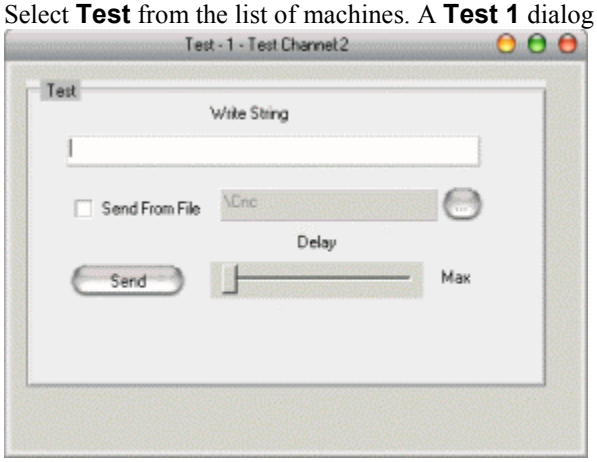
Note: Updates to the Renishaw macros have been made for optimum ball-center probing. If, prior to a probing cycle, your probe drops in the Z axis by the probe's diameter, contact Renishaw for an update.

Step 2: Install PC-DMIS/NC and Define a Machine

1. Attach the port lock to your computer. Your port lock should already be programmed to enable the PC-DMIS/NC functionality.
2. Install PC-DMIS. The PC-DMIS NC Server program should get installed in the background.
3. Start PC-DMIS NC Server by double-clicking on the PC-DMIS NC Server shortcut.
4. From within PC-DMIS NC Server, define a machine:
   - Click **Analyze Hold** to cause the server to stop running.
   - Click **Define Machines**. The **Define Machines** dialog box appears.
   - Click **New**. The **MachineDefinition** dialog box appears.
   - Fill out this dialog box. See "Creating and Modifying Machines" on page 20 for more information.
   - From the **Program** tab, name the machine "Test1".
   - Type a value in the **Machine ID** box.
   - The PC-DMIS NC Server automatically determines where the NC program files (.cnc files) are stored by inserting a pathway to the PC-DMIS directory into the **Program Directory** box.
   - From the **Interface** tab, select **Test** from the **Interface Type** list.
   - From the **Measure** tab assign a tool number (for example, 1) in the **Tool** area. This number represents the tool stored in the slot with the given number.
   - Either type the command strings or click the **Default** button to insert the commands into the **Controller Commands** boxes and into the **Command** box of the **Measure** area.
   - Click **OK**.
5. Close PC-DMIS NC Server by **File | Exit**.

Step 3: Perform an Offline Test

1. Using Windows Explorer, Create a new folder, name it "Test", and then place it in the CNC sub-folder where you have PC-DMIS/NC installed (for example, C:\PCDMISW\CNC\Test).
2. Start PC-DMIS/NC.
3. Create a small part program and name it "Test".
4. Generate the NC program file by selecting **Operation | Cnc Programming | Create CNC Program** menu option. PC-DMIS displays the **Select CNC Machine** dialog box.
   *If you can't select the **Create CNC Program** menu option, then contact your PC-DMIS/NC sales representative to ensure that your port lock is properly programmed to support the NC functionality*
5. Select **Test** from the **Select CNC Machine** dialog box.
6. Ensure that the **CNC Program File Name** box points to the Cnc\Test subdirectory where you installed PC-DMIS/NC and that it specifies a .cnc file name (Test.cnc).
7. Click **OK** to generate the .cnc program. PC-DMIS displays a processing message box and shows the path lines drawn in the Graphics Display window for the part program.
8. Using a text editor, open Test.cnc and verify that it contains commands. Close the CNC file and PC-DMIS/NC.
9. Start the PC-DMIS NC Server program. You will see the **Test** machine inside the PC-DMIS NC Server.
10. Select **Test** from the list of machines. A **Test 1** dialog box appears.



Test1 dialog box

11. In the **Test 1** dialog box, select the **Send From File** check box.
12. Click the **…** button. An **Open** dialog box appears. Navigate to the directory containing the Test.cnc file. Select the Test.cnc file and click **Open**. PC-DMIS displays the pathway to this file inside the **Test 1** dialog box (for example, C:\PCDMISW\CNC\Test\Test.cnc).
13. Click the **Send** button. The NC simulator will run. PC-DMIS/NC will also run in the background, execute the CNC program, and then exit.
14. Check to see if the program executed.

- Look for a journal subdirectory where you installed PC-DMIS. For example, if the program name if PART1, the full path would be c:\pcdmisw\journal\PART1\Test1<datetimecode>.cnj
- Open the journal file in a text editor such as Notepad. You should see nominal data for the probing points generated by the Test simulator.

Step 4: Set up the Serial Link

1. Open a HyperTerminal session on the server computer:
   - From the **Start** menu, select **Programs | Accessories | Communications**.
   - Select **HyperTerminal**.

   HyperTerminal is a Windows communication accessory that comes with Windows NT, 2000 and XP. You can use any other similar communication program that can talk with a serial interface.
2. Follow HyperTerminal's initial setup wizard to set the parameters of the serial port to be compatible with the port settings on the NC controller:

   For example, COM1, 9600 baud, 8 data bits, 1 stop bit, no parity, hardware flow control.
3. Select a small program on the NC controller.
4. PUNCH out the program to the controller serial port. You should see the program being received in HyperTerminal's window.
5. As an additional test, select **Transfer | Capture Text** from HyperTerminal. A small dialog box appears.
6. Specify a folder and file name for the captured text file.
7. PUNCH out the program again.
8. Select the **Transfer | Capture Text | Stop** from HyperTerminal to stop capture.
9. Exit HyperTerminal and open the captured file in a text editor such as Notepad.
10. You should see a copy of the program sent from the NC controller inside the text editor.

Using PC-DMIS/NC

Once you have successfully completed the steps described in "Testing PC-DMIS/NC" on page 7, then you can use PC-DMIS/NC for real measurements online.

Follow these instructions for general information on how to do this:

Step 1: Calibrate the Probe

Use the Renishaw vector calibration macro to calibrate the probe using a known spherical calibration tool on the NC machine. Follow your existing Renishaw documentation for information on how to do this.

Step 2: Set the Work Offset

Using the **Operation | Cnc Programming | Generate Work Offsets** menu option, set the work offset for the part you will measure into the NC controller. Make a note of its number. PC-DMIS/NC supports standard work offset registers G54-G59. The objective is to match up the initial PC-DMIS/NC alignment with a work offset.

Step 3: Define an NC Machine

1. Start the PC-DMIS NC Server program on the server computer.
2. Click **Analyze Hold** to cause the server to stop running.
3. Define a machine that has the parameters of the NC machine which will perform the measurement.
   - Click **Define Machines**. The **Define Machines** dialog box appears.
   - Click **New**. The **MachineDefinition** dialog box appears.
   - Fill out the tabs for this dialog box with the necessary parameters. See "Creating and Modifying Machines" on page 20 for more information.
   - Include, along with other parameters, the probe changer slot number for the probe and the work offset register.
   - Click **OK** until you return to the main PC-DMIS NC Server window.
4. Click **Exit**, to close down PC-DMIS NC Server and to store the new machine information.

Step 4: Create a NC Program File

1. Using PC-DMIS/NC, create a part program to measure the desired features on the part and generate output.
2. You can use the server computer if the you aren't running the PC-DMIS NC Server program on it. Otherwise, you will need another computer that runs PC-DMIS/NC.
3. Create an unmarked alignment at the beginning of the program. This should include the work offset.
4. Change the mode to DCC mode.
5. Use a probe that has the same nominal ball diameter as the probe in the NC machine.
6. Generate the CNC measurement program from your part program by selecting the **Operation | Create CNC Program** menu item. PC-DMIS/NC generates a file with a .cnc extension. This command also displays the probe's path. Unmarked commands are ignored.

7. Select the machine to use from the **Select CNC Machine** dialog box.
8. Modify the **Macro Number** box, if needed. This is the program number that will be used to identify this measurement sequence within the CNC system. See "Macros" in the Glossary of Terms.
9. If desired, modify where you want to send the CNC program by clicking the **Browse** button and navigating to a different directory; otherwise, accept the default pathway.
10. Click **OK**. PC-DMIS NC generates the CNC program and the probe path is displayed in the Graphics Display window. If you encounter errors, consult the log file inside any text editor application, such as Notepad.

Step 5: Start the PC-DMIS NC Server Program

Start the PC-DMIS NC Server program on the server computer. This is needed for the NC machine to properly capture data from the communications link for the generated NC program file. Click **Analyze Hold** to cause the server to stop running.

Step 6: Load the NC Program File

Using PC-DMIS NC Server, click the **Download Program** button and then select the NC program (.cnc extension) to send to the machine controller for execution.

Note: You may already have your own ways of getting NC programs onto your NC machines and you should use whatever process you're familiar with. The **Download Program** button described here is merely provided for convenience and testing.

Step 7: Execute the NC Program File

1. Change the mode of the controller from Edit to Memory.
2. Start execution of the NC program.
   - During execution, the NC program file will load the Renishaw probe and proceed to measure the features defined in your part program.
   - Measured points are sent to the server computer from the NC controller.
   - PC-DMIS NC Server parses the received lines from the controller. Each line contains a command identifier, a machine identifier, and possibly some other information.

Step 8: Analyze and Report the Data

Since NC controllers cannot do the analysis of the point data, the controller sends the data to PC-DMIS NC Server which loads it into the original PC-DMIS/NC part program for analysis, reporting, statistics and so on.

Note: If you later execute the part program inside PC-DMIS/NC, PC-DMIS/NC actually accesses the recently created journal file to get the stored point data rather than from the actual NC machine.

Working with CNC Edit Window Commands

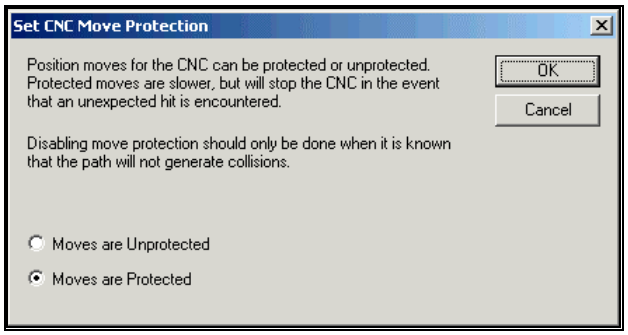
PC-DMIS/NC provides you with the following Edit window commands. To modify these commands, press F9 on any command in the Edit window to access its dialog box, or edit them directly inside the Edit window's Command Mode.

In this version, the only command you can insert by typing is the `CNC/SETPROTECTION` *command.*

To insert these commands into the Edit window, either type them into Command mode, or select an item from the **Operations | CNC Programming** submenu.

In the following topics **`Courier New`** font with bold formatting signifies a parameter that the command can take. These parameters are recognized fields of the command where the value can be set or changed to cause the command to do different things.

Set Protection

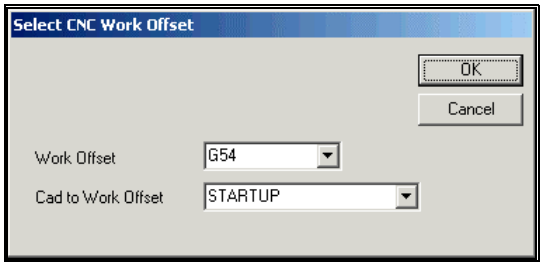


`CNC/SETPROTECTION,` **`movesProtected`**

This command sets the state of moves. By default protected moves are safe. This means unexpected probe hits will terminate part program execution. Because of the potential danger of unprotected moves, a second safety lock is in place in the definition of machines in PC-DMIS NC Server.

`movesProtected` parameter can be either “Yes” or “No”. If you set this to “Yes”, then positional moves in the CNC program will be protected and any unexpected probe hits will terminate execution of the part program. If you set this to “No”, moves will be unprotected, but *only if* the CNC machine definition allows unprotected moves. This double check is in place, because unprotected moves will not stop prematurely, unless the machine “lags out”. This could result in damage to the probe.

Use Work Offset



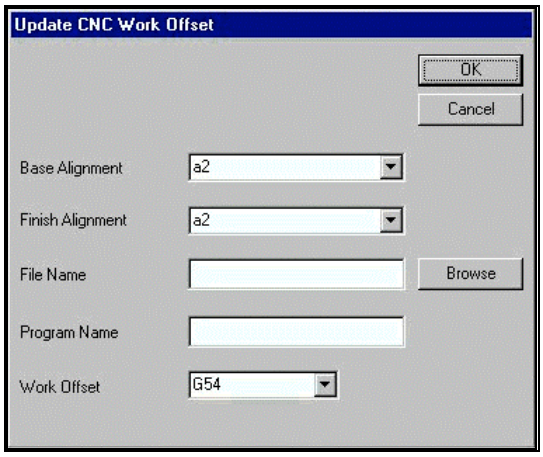
CNC/USEWORKOFFSET,**workOffset**,**alignmentCommand**

This command equates the PC-DMIS alignment with the CNC alignment. CNC alignments are implemented as work-offset coordinate systems.

workOffset can be values from G54 through G59. If an NC Control supports extended work offset registers, G54.1P1 through G54.1.P300 will be included. This parameter defines the work offsets that will be enabled in the CNC program. There is a default work offset defined in the machine's definition that will be used until the USERWORKOFFSET command species another.

alignmentCommand can be any alignment defined previously in the part program. This alignment should represent the work-offset alignment in PC-DMIS. In other words, the coordinates for points in the PC-DMIS part program with this alignment should agree with the coordinates of the same points in the CNC program with the specified work-offset applied.

Update Work Offset



CNC/UPDATEWORKOFFSET,**baseAlignment**,**finishAlignment**,**fileName**,**programName**,**workOffset**

This command calculates changes to a work-offset and communicates that change to the CNC. The changes are determined by computing the shift of the origin between two PC-DMIS alignments. Generally, the first alignment (**baseAlignment**) represents where PC-DMIS/NC assumes the work-offset system is located, and the second alignment (**finishAlignment**) represents the actual location of the work-offset system based on a set of measurements used to generate the second alignment.

`baseAlignment` should be a PC-DMIS alignment that represents where the work-offset system was prior to measurements. You can select this from the **Base Alignment** list in the **Update CNC Work Offset** dialog box for this command. This list contains PC-DMIS alignment commands available to this point in the PC-DMIS part program.

`finishAlignment` should be a PC-DMIS alignment that represents where the work-offset system is after the measurements. You can select this from the **Finish Alignment** list in the **Update CNC Work Offset** dialog box for this command. This list contains PC-DMIS alignment commands available to this point in the PC-DMIS part program.

`fileName` is the name of the CNC program file (.cnc extension) that PC-DMIS/NC generates when you select **Operation | Cnc Programming | Create Cnc Program**. Some CNC interfaces (for example, TCP/IP) upload the work-offset changes directly. In these cases, the `fileName` parameter is not used. If no file extension is provided, the extension, "WRK", will be appended to the CNC file name. This extension informs PC-DMIS CNC Server that the contents of the file define updates to a work-offset system.

`programName` is the CNC macro name of the CNC program that is generated. This parameter is not used for interfaces that upload the work-offset changes directly.

`workOffset` is selected from the **Work Offset** list in the **Update CNC Work Offset** dialog box. This can be values from G54 through G59 and G54.1P1 through G54.1.P300. This option defines the work offsets that will be enabled in the CNC program.

Working with PC-DMIS/NC in Offline Mode

If you execute your part program in offline mode, the **Select Journal Fil**e dialog box appears.

Select Journal File dialog box

If you click **Cancel**, PC-DMIS performs a normal run of the part program in offline mode using the nominal values for the measured values.

If you click **Browse**, select a journal file (.cnj extension), and click **OK**, PC-DMIS pulls the journal file's measurements into PC-DMIS, showing a replay of the measurement program, using those measurement values rather than the nominals.

See "Executing Journal Files" on page 19 for additional information.

Using the PC-DMIS NC Server Application

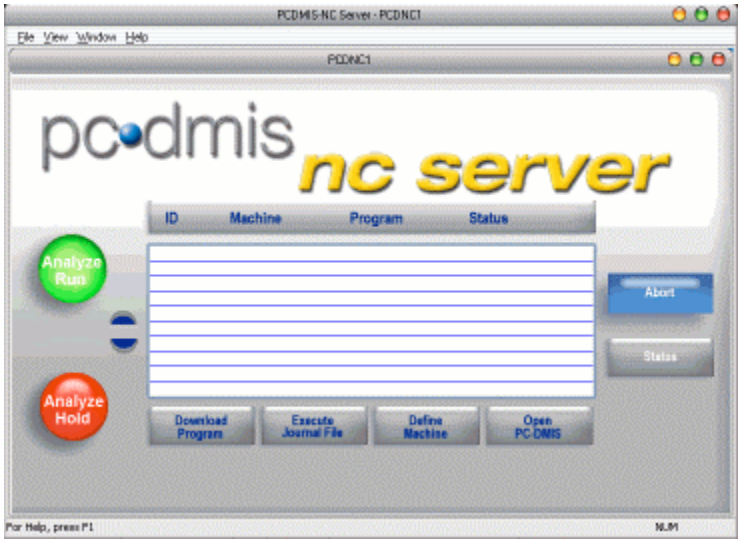
PC-DMIS NC Server is a stand alone program that cooperates with PCDMIS/NC to analyze results.

Before you Begin: You need a copy of PCDMIS installed, and your port lock *must* be programmed with the NC option in order to use PC-DMIS NC Server.

To start PC-DMIS NC Server, double-click on the **CNC Server** shortcut icon in the PC-DMIS/NC program group. If this shortcut icon is not available, you may need to re-install PC-DMIS after ensuring that your portlock is properly programmed.

CNC Server

The PC-DMIS NC Server program appears and PC-DMIS/NC runs in the background. When it launches PC-DMIS NC Server automatically closes any pre-existing instances of PC-DMIS/NC so that only one instance of PC-DMIS/NC will open and run in the background.

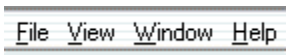


PC-DMIS NC Server

By default, the **Analyze Run** button is automatically selected when you launch the PC-DMIS NC Server application. This means that the server it running and ready to receive data from an NC machine.

Note: The **Download Program**, **Execute Journal File, Define Machines,** and **Open PC-DMIS** buttons are inactive whenever the server is analyzing. To activate these buttons, first click the **Analyze Hold** button. Once activated, they will show a blue glow.

The PC-DMIS NC Server Menubar



The menu bar consists of the **File**, **View**, **Window**, and **Help** menus.

File | OpenMonitor displays the MonitorView window. This window, much like DataPage's Monitor Mode, allows you to view statistical data stored inside DataPage database files (files with a .dbd extension). See "Working with the MonitorView Window" on page 27.

File | Exit closes PC-DMIS NC Server.

View | Status bar shows or hides the status bar at the bottom of PC-DMIS NC Server.

Window | Cascade shows multiple windows on top of one another but keeps the title bars visible so you can easily select a window by clicking on its title bar.

Window | Tile shows multiple windows at the same time without overlapping other windows. Multiple windows size themselves to fit within the available screen space on your monitor.

Window | Arrange Icons automatically arranges icons (windows?)

Help | Help Topics displays the PC-DMIS NC Server help file.

Help | About NC Server displays information about the application. It also displays the version number.

The PC-DMIS NC Server Command Buttons

The following command buttons are often used inside PC-DMIS NC Server:

Analyze Hold – This button stops the NC Server from analyzing data from an NC machine. It also enables other command buttons. See "Stopping and Resuming PC-DMIS NC Server" on page 18. PC-DMIS NC Server continue to collect and analyze data from NC machines once you click the **Analyze Run** button.

Analyze Run – This button readies the NC Server to receive data from an NC machine. See "Stopping and Resuming PC-DMIS NC Server" on page 18.

Download Program - This button moves your converted part program (.cnc file) to the NC controller by using the **Download Program** button1. See "Moving Programs to the NC Controller" on page 19.

Execute Journal File – This button lets you execute a journal files at any time to see an offline replay of the CNC measurement session inside of PC-DMIS/NC for analysis purposes. See "Executing Journal Files" on page 19.

Define Machine – This button opens the **MachineDefinition** dialog box and allows you to create new NC machines, clone existing machines, or edit existing machines. See "Creating and Modifying Machines" on page 20 for more information.

Open PC-DMIS – This button opens or accesses PC-DMIS/NC.

Abort – This button aborts any active measurement session for the machine.

Status – This button displays a window showing all machine input / output (I/O) data. You can observe the status for one machine at a time by first selecting the machine's line in the **Status** column of the main interface. The **Status** button becomes available for selection. Click the **Status** button and the Status window appears.

The Machine Status Fields

The various machine status fields in PC-DMIS NC Server provide information on already defined machines. These fields are described below:

ID – This field displays a unique identification number (positive 16 bit integer) for each machine. Even if you have multiple machines of the same type, each must have a unique ID.

Machine – This field shows the name given for a machine.

Program – This field displays the name of the PC-DMIS part program when the status field displays 'Receiving Data' or 'Analyze Data'.

Status – This field displays the machine's current status. It displays either nothing at all, Receiving Data, or Analyze Data.

If this field is blank, then the machine is not executing a measuring program and it may be running a cutting program.

If this field displays 'Receiving Data', then a measurement program is running on the NC Machine.

If this field displays 'Analyze Data', then the point data has been stored in the journal file and PC-DMIS/NC is in the process of analyzing the data.

Stopping and Resuming PC-DMIS NC Server

You can pause the analysis of data by clicking the **Analyze Hold** button. This also enables the **Download Program**, **Execute Journal File** and **Define Machines** buttons.

Even when stopped, PC-DMIS NC Server continues to send data to its own buffered memory so that if you click **Analyze Run** later, data analysis will continue where it left off. However, since this memory is of limited size, it is possible to overrun it with large amounts of data (thousands of points). You may not be able to pause the data analysis indefinitely.

Exiting PC-DMIS NC Server

Exiting the PC-DMIS NC Server program using the **Exit** button also stops the server but results in data loss if there's an active session running. Exiting the PC-DMIS NC Server also closes the background instance of PC-DMIS/NC that was running.

Moving Programs to the NC Controller

You can move programs to the NC controller by using the **Download Program** button. To do this:

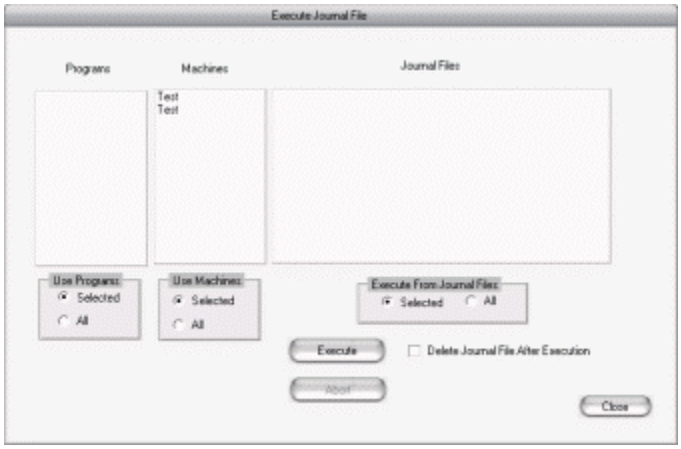
1. Ensure that the NC controller is in a mode that will accept programs. Follow your controller's documentation for information on how to do this.
2. Click the **Analyze Hold** button so that the server isn't running
3. Selecting a machine from PC-DMIS NC Server.
4. Click the **Download Program** button. PC-DMIS NC Server displays an **Open** dialog box.
5. Navigate to where your .cnc files are stored.
6. Select the .cnc program.
7. Click **Open**. PC-DMIS NC Server sends the program to the NC controller.

Executing Journal Files

PC-DMIS NC Server generates a journal file for each session. You can execute these journal files at any time to see a replay of the session inside of PC-DMIS/NC for analysis purposes.

To execute a replay of one of these journal files:

1. Click the **Analyze Hold** button so that the server isn't running.
2. Click **Execute Journal File**. The **Execute Journal File** dialog box appears.



Execute Journal File dialog box

You will see three lists: **Programs**, **Machines** and **Journal Files**.

- The **Programs** list shows all programs for all Machines program directories.
- The **Machines** list shows all defined machines.
- The **Journal Files** list shows all response files for the selected Programs and Machines.

Beneath these lists are the **Use Programs**, **Use Machines** and **Execute From Journal Files** areas. In these areas, you can choose to execute **All** or only **Selected** journal files from the list above.

3. Use the list to select one or more journal files.
4. If you want PC-DMIS NC Server to delete the journal file after a successful analysis, select the **Delete Journal File After Execution** check box.
5. Begin analysis of the journal file(s) by clicking the **Execute** button. PC-DMIS/NC opens and 'replays' the journal file. A new Journal File *will not* be generated by this replay process. You can abort the execution at any time by clicking the **Abort** button.
6. Click the **Close** button to close the current dialog box and return to the main interface of PC-DMIS NC Server.

See "Working with PC-DMIS/NC in Offline Mode" on page 15.

Creating and Modifying Machines

PC-DMIS NC Server allows you to define new machines or manage already defined NC machines. To do this, click the **Define Machines** button. The **Define Machines** dialog box appears showing a list of machines.

Define Machines dialog box

This dialog box contains these options:

New – The **New** button creates a new NC machine by launching the **MachineDefinition** dialog box.

Clone – The **Clone** button creates a new machine with the same parameters and settings as the selected machine. Make changes as needed to quickly create similar NC machines.

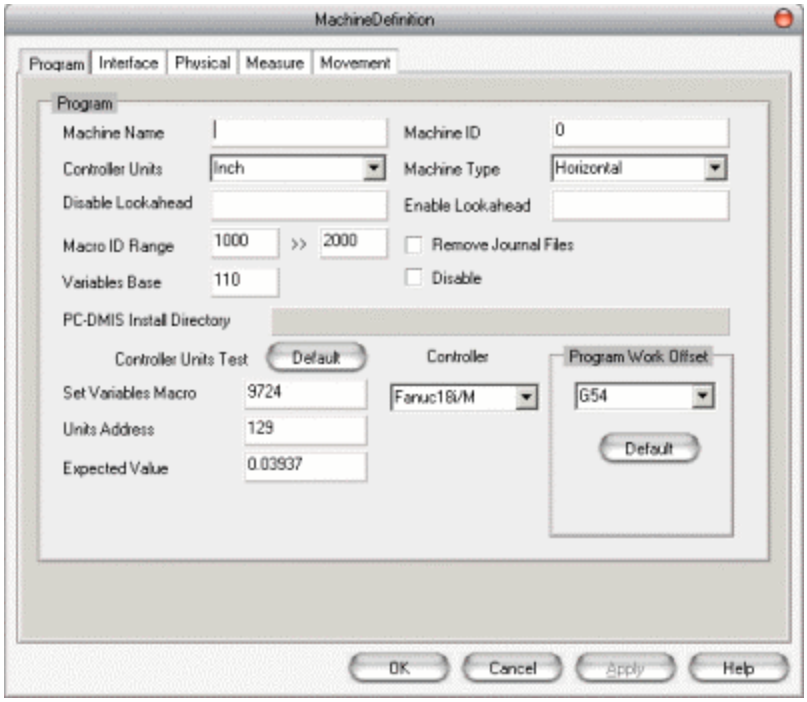
Modify – The **Modify** button modifies the selected machine from the list of defined machines by launching the **MachineDefinition** dialog box. Double-clicking on an existing machine also does this.

Delete – The **Delete** button removes the selected machine from the list of defined machines.

Changes made to a machine or the set of machines will not take place until after PC-DMIS NC Server is exited and restarted. Machine information is saved in the Windows Registry.

The **MachineDefinition** dialog box contains various tabs that allow you to define a new machine or modify an existing machine.

MachineDefinition - Program Tab



MachineDefinition dialog box – Program tab

Machine Name – This box allows you to type a name to identify the machine.

Machine ID – This box allows you to specify a unique positive integer to identify this machine.

Machine Type – This list allows you specify the machine style. You can select either Horizontal or Vertical.

Controller Units – This list specifies the units of measurement to use for controller programs.

Disable/Enable Lookahead – These boxes allow you to specify the NC G/M codes that are enabled or disabled for the controller's Lookahead functionality.

If these boxes don't contain codes (the default), then no codes will be inserted into the NC program.

If these boxes contain codes, then the code in the **Disable Lookahead** box will be sent to the NC program prior to execution of the PC-DMIS/NC program. The code in the **Enable Lookahead** box will be processed at the end of the program to re-enable Lookahead functionality.

Macro ID Range – These boxes allow you to specify a safe range of numerical ID numbers to use for automatically generated macros. This helps ensure that you don't overwrite existing macros with these automatically generated macros.

Variables Base – This box allows you to define the lower limit for assigning variables required by PC-DMIS/NC generated NC programs.

Remove Journal Files – This check box allows you to determine whether or not PC-DMIS NC Server removes the created journal file after a successful PC-DMIS/NC program execution

Disable – This check box allows you to disable a machine interface without removing its machine definition. You may want to use this check box to ignore a problematic machine interface.

PC-DMIS Install Directory – This box displays the complete pathway to the directory where the PCDMIS/NC executable (pcdlrn.exe) is located. PC-DMIS NC Server determines this automatically. It cannot be edited.

Controller Units Test / Default – The **Default** button for Controller Units Test sets the macro program used to set Renishaw defaults, the controller storage register, and value of inch to millimeter conversion.

Controller – This list allows you to select the controller type. The Generic selection is used for the test object and represents Fanuc style controllers.

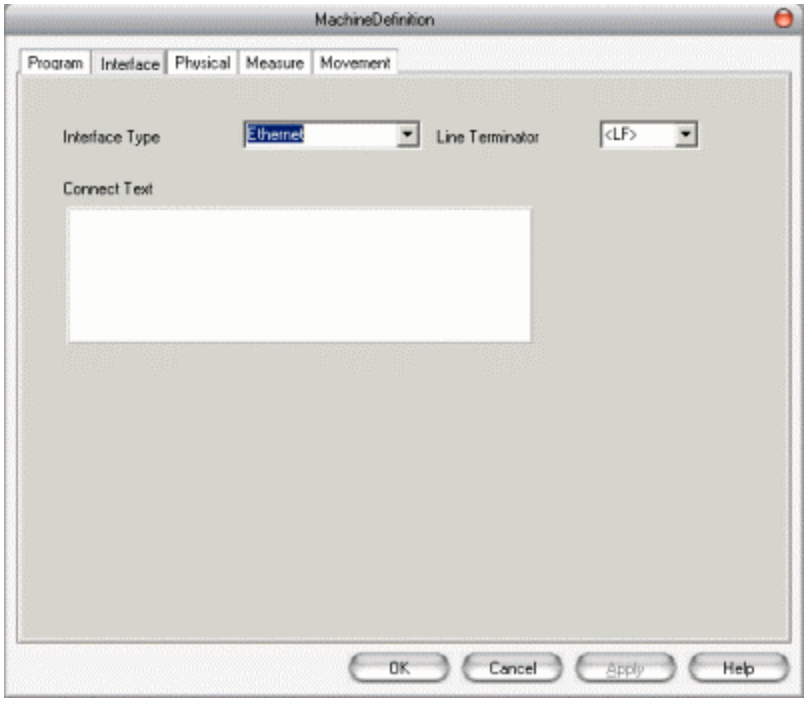
Set Variables Macro – This defines the Renishaw macro used to set the default variables in the controller.

Units Address – This defines the location of the millimeter to inch conversion.

Expected Value – This defines the expected value of the millimeter to inch conversion.

Program Work Offset – This area allows you to define the start location for measurement by selecting a program work offset from the list. The standard controller registers G54 through G59 are available. The **Default** button ensures that the selected work offset is the proper type and is within the allowable ranges for the selected controller type. The selection will not change if it already meets these criteria.

MachineDefinition - Interface Tab



MachineDefinition dialog box – Interface tab

Interface Type – This list allows you to specify the interface type to use when connecting to the NC machine. You can select one of these types:

- Serial
- Ethernet
- RS491
- HSSB (Fanuc proprietary High Speed Bus)
- Focus1
- Test
- File

The interface type is based on the NC machine interface available. You will usually use the same interface type that you use to load cutting programs onto your NC machine. If you don't use a communications interface, then select **Serial** from the list. Most machines support the serial interface type.

Connect Text – This box allows you to specify strings required to connect the server channel to the NC machine. Default text strings are supplied for common interfaces.

For example, for the serial interface, the connection string would read: COM1 9600 N,8,

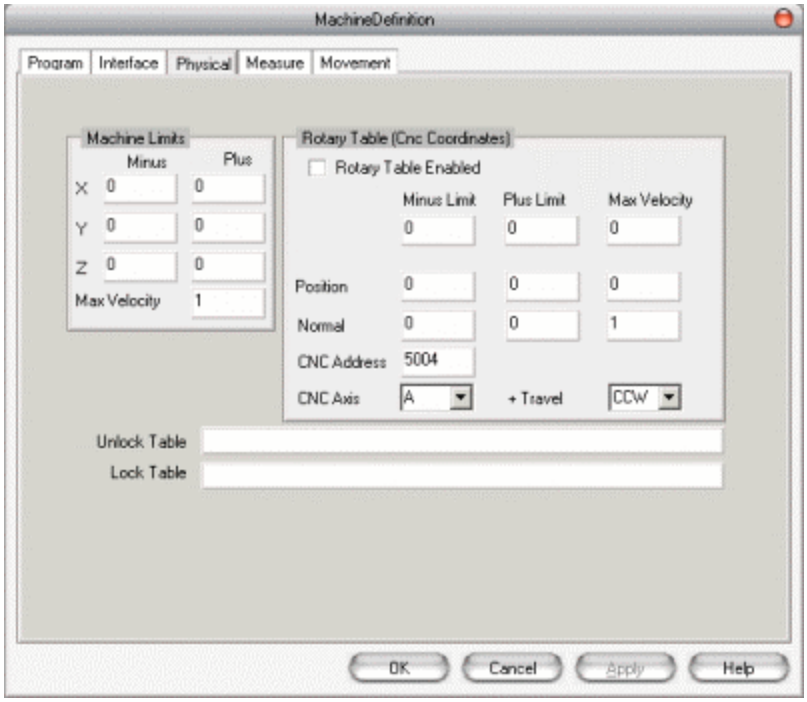
Line Terminator – This list allows you to select the line terminator to use when generating the NC program.

<LF> - An ASCII *Linefeed* character is appended to each line. LF characters advance the cursor one line below the current line without moving the position of the cursor horizontally.

<CR> - An ASCII *Carriage Return* is appended to each line. CR characters cause the cursor to move to the beginning of the current line.

<CR><LF> - A *Carriage Return* and a *Linefeed* are appended to each line. This will cause the cursor to move to the beginning of the next line.

MachineDefinition - Physical Tab



MachineDefinition dialog box – Physical tab

Machine Limits area:

XYZ Minus / Plus – These boxes define the travel limits in machine units. If these boxes contain zeroes, then limit checking will be bypassed at CNC program generation.

Max Velocity – This box specifies the maximum travel velocity in machine units/Minute.

Rotary Table (Cnc Coordinates) area:

Rotary Table Enabled – This check box generates any needed rotary table commands.

Minus / Plus Limits – These boxes specify a rotary table's limits. If these boxes contain zeroes, then limit checking will be bypassed at CNC program generation.

Max Velocity - This box specifies the maximum travel velocity in the rotary table in degrees per second.

Position – This box defines the location of the rotary table in machine coordinates.

Normal – This box defines the orientation vector of the rotary table in machine coordinates

CNC Address – This box defines the location of the controller register containing the position of the rotary table.

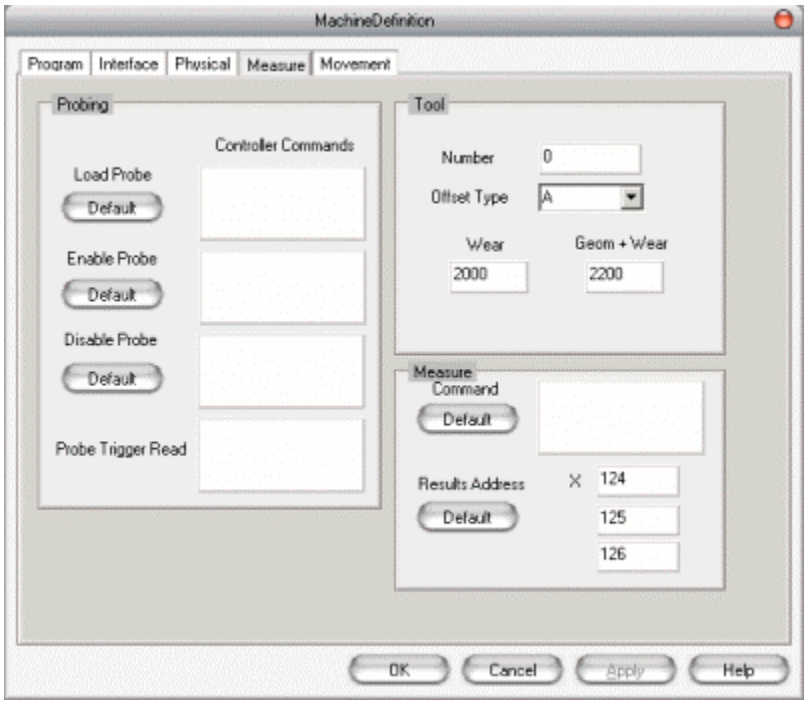
CNC Axis – This box defines the axis indentifier for the rotary table.

Travel – This box defines the positive direction of the rotary table's rotation.

Unlock Table – This box allows you to specify a command string required for the controller to unlock the rotary table.

Lock Table – This box allows you to specify a command string required for the controller to lock the rotary table.

MachineDefinition – Measure Tab



MachineDefinition dialog box – Measure tab

Probing area:

Load Probe – This box allows you to specify the machine's G/M codes to use to load the probe. The **Default** button loads a default value for the associated controller commands.

Enable Probe – This box allows you to specify the machine's G/M codes to use to enable the Renishaw probe. The **Default** button loads a default value for the associated controller commands.

Disable Probe – This box allows you to specify the machine's G/M codes to use to turn off the Renishaw probe. The **Default** button loads a default value for the associated controller commands.

Probe Trigger Read – This box allows you to specify a string to test when the probe is deflected.

Tool area:

The **Number** box allows you to specify the machine tool number where the probe is stored

The **Offset Type** list allows you to specify the offset type used in the control. You can select A, B, or C.

The **Wear** box allows you to specify the controller address for the wear component of the tool offset.

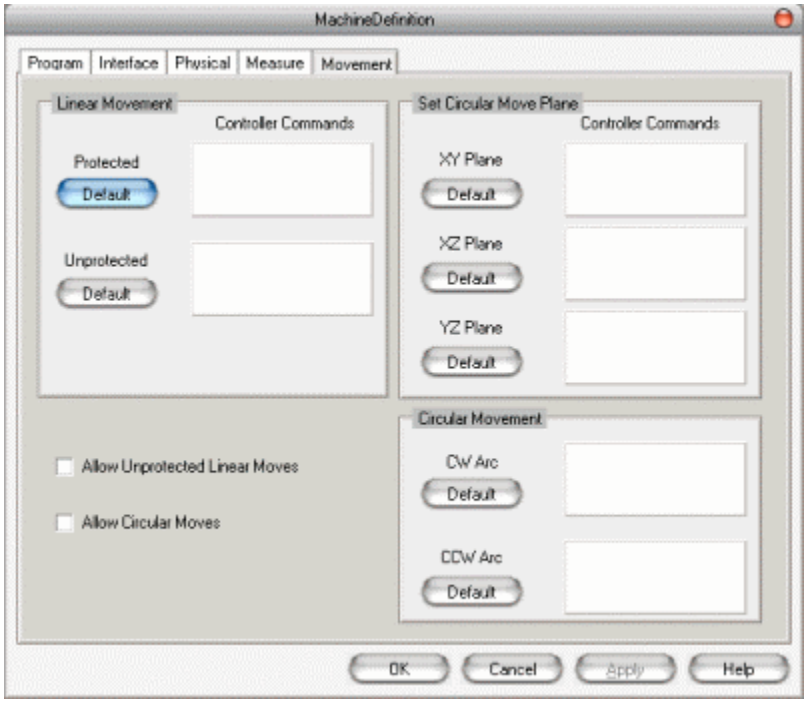
The **Geom + Wear** box allows you to specify the contoller address for the geometry and wear component of the tool offset.

Measure area:

The **Command** box allows you to specify the parameterized command used to measure a point. The %x %y and %z act as place holders where the associated X, Y and Z probing values will be substituted. The **Default** button loads a default value for the associated controller commands.

The **Results Address** boxes (**X**, **Y**, and **Z** boxes) allow you to specify where to find the probing results. These addresses are used to generate the text used in the **Command** box above. The **Default** button loads a default value for the associated controller commands.

MachineDefinition – Movement Tab



MachineDefinition dialog box – Movement tab

The **Movement** tab of the **MachineDefinition** dialog box, allows you to define the codes needed to specify whether or not the NC machine can perform unprotected moves, set work planes, or enable circular moves.

Note: The **Default** button for each item below sets the appropriate NC codes in the boxes for the selected controller. You can then modify or replace these codes as needed.

Linear Movement area:

Protected – This box allows you to define the controller codes required to perform a move so that the probe detects collisions.

Unprotected – This box allows you to define the controller codes required to perform a move so that the probe does not detect collisions.

Set Circular Move Plane area:

XY Plane – This box lets you define the controller codes that set the XY workplane (for example, G17).

XZ Plane – This box lets you define the controller codes that set the XZ workplane (for example, G18).

YZ Plane – This box lets you define the controller codes that set the YZ workplane (for example, G19).

Circular Movement area:.]

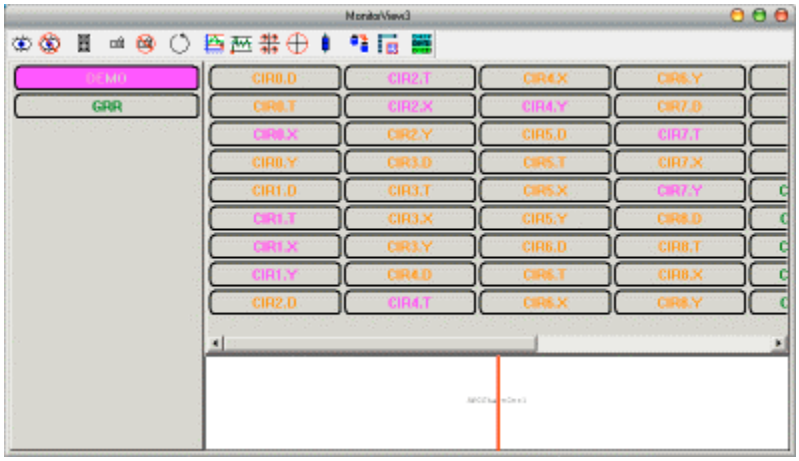
CW Arc – This box lets you define the controller codes that allow movement in a clockwise direction.

CCW Arc – This box lets you define the controller codes that allow movement in a counterclockwise direction.

Allow Unprotected Linear Moves: Selecting this check box allows unprotected linear moves. To generate unprotected moves, use the CNC/SETPROTECTION,NO command inside the Edit window of PC-DMIS/NC.

Allow Circular Moves: Selecting this check box allows circular movement. The normal motion approximates circular moves with straight line moves. Circular moves are unprotected.

Working with the MonitorView Window



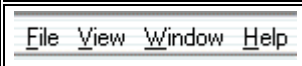
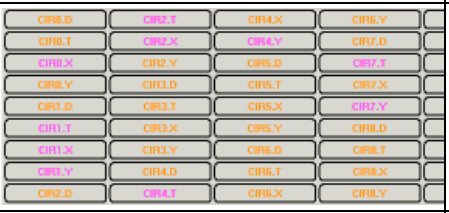
MonitorView window

The MonitorView window is a replication of the DataPage Monitor mode and interacts with a DataPage/RT database (.dbd file). If you are familiar with the DataPage Monitor mode, you will find the functionality of the MonitorView window to be similar.

For in-depth information on DataPage, see your DataPage help file or printed documentation.

Understanding the MonitorView Window Interface

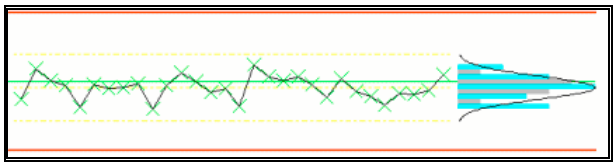
The MonitorView window is made up of a toolbar, a left pane, a right pane, and a bottom pane.

Window Area	Description
	*The menu bar* contains commands to manipulate the display of data in the MonitorView window; it has the same commands available on the MonitorView toolbar.
	The toolbar contains all the commands needed to work with the statistical data in the database. You can also select these commands from the **Monitor** submenu.
	The left pane displays the parts in the database. When you select a part from the left pane, the right pane is updated to show the variables associated with the selected part.
	*The right pane* displays the variables for a selected part.
	The bottom pane automatically displays a chart for a single variable you select from the right pane.

Working with Variables

Selecting Variables – To select multiple variables next to each other, click on the first variable in the range, press the SHIFT key and then click last variable in the range. To select multiple variables out of sequence, press the CTRL key and then click the desired variables.

Displaying a Chart – If you select only one variable and there's *enough* data, the MonitorView window displays a chart for that variable in the bottom pane.



Sample Chart

It decides if there's sufficient data to draw a chart based on the amount of data in the database, your statistical settings, and the last time that data was “acknowledged”. See "Changing Properties" on page 31 for more information on your statistical settings.

Calculating the Statistical State – You can calculate the statistical state of variables by selecting multiple variables and clicking the **Calculate Status** icon from the toolbar or by selecting **Monitor | Calculate Alarms**. The calculation process determines whether or not the selected variables are in or out of tolerance, and what their control limits are. The statistical state of the variables is then displayed by showing each variable in one of the following default colors.

Green - Uncalculated variables, or those that are in tolerance after calculation.

Orange - Variables that failed at least one control test are considered out of control.

Red - Variables that have at least one measurement that is out of spec.

Purple - Variables that are both out of control and out of spec.

You can easily configure these colors by clicking the **Properties** icon and using the **General** tab inside the dialog box that appears. See "Changing Properties" on page 31 for more information.

Acknowledging and Unacknowledging Variables – MonitorView can easily let you know when your part is out of tolerance or out of control. Parts containing out of tolerance or out of control variables are displayed in the colors already discussed.

Clicking the **Acknowledge** icon (or selecting the **Monitor | Acknowledge Alarms** menu item) turns the selected variables green, records the date and time you acknowledged them, and recalculates the control limits using data from the acknowledged date/time forward. Previous data is disregarded.

Clicking the **Unacknowledge** icon (or selecting the **Monitor | Unacknowledge Alarms** menu items) removes the acknowledgement from previously acknowledged variables and recalculates the current state of the variables. The system uses all available data to calculate control limits.

Freezing and Unfreezing Limits – You can freeze or unfreeze control limits for selected variables.

Clicking the **Freeze Limits** icon (or selecting the **Monitor | Freeze Control Limits** menu item) makes it so that the database does not recalculate control limits when new data gets added to the database.

Clicking the **Unfreeze Limits** (or selecting the **Monitor | Unfreeze Control Limits** menu item) icon unfreezes the control limits to their

previous state, allowing the system to recalculate control limits for the selected variables each time it adds new data to the database.

To view the state of the limits, select **View | Variables | Form Editor** within the DataPage/RT Editor. See your DataPage/RT Editor documentation for more information.

Refreshing Data

If you add, remove, or change data from an outside process, you can use the **Refresh** icon (or selecting the **Monitor | Refresh Alarms** menu item) to re-read the database, changing the output in the MonitorView window accordingly. If data has changed, and you click this button, MonitorView also updates the current chart.

Working with DataPage to Display Charts and Plots

Important: The following chart and plot processes require that you have DataPage installed on your computer. When you click on the icon for these processes, DataPage launches and displays the selected chart or plot. If you need information beyond the scope of this documentation, view your DataPage documentation.

Creating Run Charts – Clicking the **Run Charts** icon (or selecting the **Monitor | Create Run Charts** menu item) creates run charts for each selected variable. A run chart is an ordered plot of raw data points used to study a variable's behavior and variance over time. Points are plotted against specification and/or control limits.

Creating Process Charts – Clicking the **Charts** icon (or selecting the **Monitor | Create Process Charts** menu item) creates a process chart for each variable. A process chart contains a selection of statistical charts that provides a complete definition of a variable's process: its distribution, control, capability characteristics, and assigned cause codes. These can be viewed in different ways, such as an individual chart, a group chart, a histogram chart, and so forth.

Creating Target Plots – Clicking the **Target** icon (or selecting the **Monitor | Create Target Plots** menu item) creates target plots for each selected feature. A target plot displays the positions of measured points in relation to the nominal values in two dimensions and the feature's upper tolerance. To select a "feature" you must select the T (true position) axis and two positional axes (among, X, Y and Z). You can select more variables for each feature, but they will be ignored.

Creating a Group Target Plot – Clicking the **Group Target** icon (or selecting the **Monitor | Create Group Target Plots** menu item) creates a group target plot for the selected features. A group target plot displays the positions of several features. DataPage will display a single plot with multiple symbols; each symbol representing the average location of that feature. To select a "feature" you must select the T (true position) axis and two positional axes (among, X, Y and Z). You can select more variables for each feature, but they will be ignored.

Creating a Whisker Chart – Clicking the **Whisker Chart** icon (or selecting the **Monitor | Create Whisker Chart** menu item) creates a whisker (or capability) plot for the selected features. This chart shows the mean, high, and low

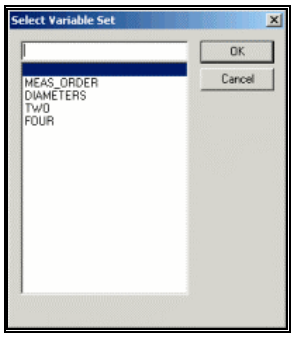
values, plus and minus three sigma units. To select a “feature” you must select the T (true position) axis and two positional axes (among, X, Y and Z). You can select more variables for each feature, but they will be ignored.

Changing a Database

To change the current database displayed in the MonitorView window, click the **Database** icon. An **Open** dialog box appears allowing you to select a new DataPage database (.dpd file).

Monitoring Variable Sets

To monitor a set of critical features (or any DataPage variable set) click the **Variable Set** icon (or select the **File | Select Variable Set** menu item). This displays the **Select Variable Set** dialog box.



Select Variable Set dialog box

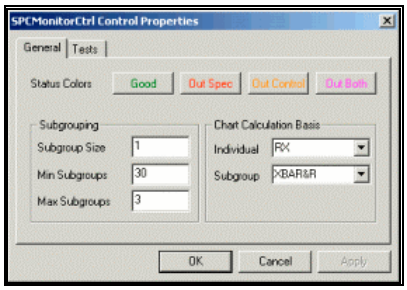
Note: You must have at least one variable set in the database for the current part to use the **Select Variable Set** icon.

When you select a variable set from this dialog box, MonitorView displays only the variables in that set. MonitorView obeys the order of the variables in the variable set. To show all the variables in the part again, select the blank line at the top of the list in the **Select Variable Set** dialog box.

Changing Properties

Clicking the **Properties** icon (or selecting the **Monitor | Properties** menu item) displays the **SpcMonitorCtrl Control Properties** dialog box. This dialog box contains two tabs, **General** and **Tests**, that allows you to change various MonitorView window's properties.

General Tab



SPCMonitorCtrl Control Properties—General tab

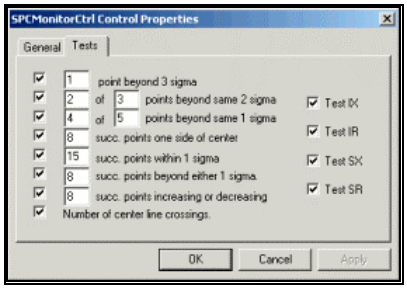
Status Colors buttons– Click these buttons to invoke the standard windows color selection dialog box and change the colors for the statistical rules: Good, Out Spec, Out Control and Out Both.

Subgrouping area – This area of the dialog box, contains these boxes Subgroup Size, Min Subgroups, and Max Subgroups. These values define how the MonitorView window groups data for calculations and charting. If subgroup size is 1, the subgroup setting is ignored.

Note: You must have at least Subgroup Size * Min Subgroups measurements in order to make control calculations or display the feedback chart. If you have more than Subgroup Size * Max Subgroups, then the MonitorView window only uses the last complete MaxSubgroups in the calculation.

Chart Calculations Basis area– This area contains two lists: **Individual** and **Subgroup**. These lists determine the calculations made when you click the **Calculate** icon to determine the statistical state and the calculations made when you display charts.

Tests Tab



SPCMonitorCtrl Control Properties—Tests tab

The **Tests** tab contains a number of edit boxes and check boxes that allow you to customize and enables various tests. It also determines whether a test is applied to the X component and/or R component of the charts, for both individual and subgroup charts. The subgroup chart settings are ignored when the **Subgroup Size** box in the **General** tab is 1.

Glossary of Terms

Here are some common terms and definitions used with the PC-DMIS/NC application.

Analysis phase

A phase where PC-DMIS/NC runs the part program (.prg file) to analyze the data returned from the NC machine has after it has taken all the probing hits.

Channel

This is a communications interface from an NC machine to the PC-DMIS NC Server.

CNC File

NC machine measurement program generated from a PC-DMIS/NC part program with a .cnc extension. This file contains commands for probe's movement and measure path. It however does not contain any analysis, reporting or statistics capability.

DataPage

DataPage™ is a statistical processing software package which links seamlessly with PC-DMIS/NC and with the MonitorView window inside the PC-DMIS NC Server.

DNC

Direct Numerical Control. This allows large NC cutting or measurement programs to be stored on an external device (such as your computer's internal hard disk drive) and then moved to the controller as required rather than storing the entire program in the NC controller's limited memory. This ability to use the NC programs stored on the external device is some times referred to as a "drip feed". PC-DMIS/NC does not provide DNC capabilities.

G/M Codes

These are standard codes (or commands) for NC controllers. *G codes* control motion, work offsets, and so forth. *M codes* control status, calling subroutines, programs, and program control.

Journal File

Journal files are created by the CNC server when a part is measured on the CNC machine (or the test machine). Journal files contain the measurement points, probe diameter, work offset alignment, etc. It stores all transactions between PC-DMIS NC Server and the NC machine for a session. The journal file's name is a product of the Machine Name, Machine ID, and the Year, Month, Day, and time, followed by either a .cnj or .cnp extension (.cnj for a successful completion of the entire measurement program or .cnp for an unsuccessful completion). For a successful completion, PC-DMIS NC Server must see the End command from the controller and there can't be any parsing errors on the received data.

Log File

This log file stores the status of all sessions for all NC machines. This file has the same name as your part program with a .log extension.

Lookahead

This functionality allows the controller to begin processing the next command even before the existing command finishes. Typically, motion commands take time to process, so if the controller does not need the results of the current motion command, it can 'look ahead' at the NC program file and begin processing additional commands. Since the controller's logic commands, such as IF tests, do not generate motion, these may be ignored. If this is the case with your NC controller, you may disable Lookahead by inserting G/M codes provided by the controller manufacturer. Insert these codes in the **Disable Lookahead** box in the **Program** tab of the **DefineMachines** dialog box.

Macros

These are Renishaw probing subroutines in NC language. They are typically called NC programs or macros.

Offset Register

These are locations in the NC controller where the work offset is stored. The active work offset is set by one of these NC G code commands: G54 through G59.

PCDMIS/NC

This is a subset of the PC-DMIS application that supports NC operation.

PC-DMIS NC Server

This is a server program which interfaces between PCDMIS/NC and the NC machines. It serves the NC machines and provides PC-DMIS/NC analysis for all connected machines. This is sometimes called the *CncServer* application

Protected

Protected moves terminate the program when an unexpected probing takes place. This can help prevent damage to the probe.

Session

A "session" is an active measurement program running on an NC machine. The session begins when you execute a .CNC program on the NC machine. The session terminates after PC-DMIS/NC finishes execution of the part program file.

Serial interface

This is a communications interface between the NC machine and the PC-DMIS NC Server. Typical serial interfaces on a computer are COM1 or the USB ports.

Unprotected

Unprotected do not terminate the part program when an unexpected probing takes place. If you aren't careful, this can cause damage to the probe.

Work Offset

This common NC control term defines the start machining point from the NC machine zero. This is much like an alignment translation. Storage areas for Work Offsets are activated in an NC program by using codes G54, G55…G59. All commanded motion and reported positions are relative to the active Work Offset.

Index

A

Analyze and report data 12

C

Calibrate the probe 10
Clockwise 27
Clone an NC machine 20
CNC file
  create a 11
  execute 12
  loading 12
CNC/SETPROTECTION 13
CNC/UPDATEWORKOFFSET 14
CNC/USEWORKOFFSET 14
Computer Numeric Control machines 1
Connect text 23
Controller 22
Controller Units Test 22
Counterclockwise 27
Create a cnc file 11
Create an NC machine 20

D

Data
  analyze and report 12
Define a machine 8, 11
Define an NC machine 20
Delete an NC machine 20
Directory
  of programs 22
Disable probe
  G/M clodes 25
Download program 19

E

Edit window commands
  CNC/SETPROTECTION 13
  CNC/UPDATEWORKOFFSET 14
  CNC/USEWORKOFFSET 14
  with CNC 13
Enable probe
  G/M clodes 25
Execute cnc file 12
Execute journal files 19
Exit the PC-DMIS NC Server 19
Expected value 22

G

G/M codes
  disable probe 25
  enable probe 25
  load probe 25
Geometry and wear 26

H

Horizontal machine 21

I

Install
  PC-DMIS/NC 8
  Renishaw probe 7
Interface type 23

J

Journal files 15
  executing 19
  remove 22

L

Line terminator
  carriage return 23
  linefeed 23
Load cnc file 12
Load probe
  G/M clodes 25
Lock rotary table 25
Lookahead 21

M

Machine
  clone an NC 20
  define a 8, 11
  define an NC 20
  delete an NC 20
  ID 21
  modify an NC 20
  Name 21

offset 26
tool number 26
travel limits 24
type 21
velocity 24
Machine status fields 18
Macro 4, 5, 6, 8, 10
Macro ID range 22
Macros 12
Measure
command 26
Modify an NC machine 20
MonitorView 28
calculate statistical state 29
change database 31
charts and plots 30
creating a group target plot 30
creating a whisker chart 30
creating process charts 30
creating run charts 30
creating target plots 30
colors 29, 32
displaying charts 28
interface 28
properties 31
refresh 30
setup 31
tests 31
variable sets 31
variables 28
Move program to NC controller 19

N

NC 1
NC controller
measurement units 21
receive a program 19
NC program file
create a 11
executing 12
loading 12
Numeric Control 1

O

Offline mode
PC-DMIS/NC 15
Offline test 9
Offset 26

P

PC-DMIS NC Server 16
clone an NC machine 20
command buttons 17
define NC machine 20
execute journal files 19
exit from 19
machine status fields 18
menubar 16
monitor view 17
moving programs to controller 19
starting 12, 16
stop and resume 18
using DataPage Monitor 17
PC-DMIS/NC 1
analyze and report data 12
calibrate probe 10
create cnc file 11
data flow 6
define a machine 8
define machine 11
differences from PC-DMIS 2
Edit window commands 13
execute the NC program file 12
files used with 6
load cnc file 12
PC-DMIS supported functionality 2
perform a test offline 9
renishaw probe 7
required components for 6
set the serial link 10
set work offsets 11
start PC-DMIS NC Server 12
testing 7
use offline 15
use PC-DMIS NC Server 16
uses for 1
using 10
Perform an offline test 9
Probe calibration 10
probe deflection
string 25
probe trigger
string 25
Probing results 26
Program
move to NC controller 19
Program directory 22
Protected 27
Protected moves 13

R

Replay session 19
Report data 12
Resume the PC-DMIS NC Server 18
Rotary table
controller register 25
defining axis 25
direction of rotation 25

enabling 24
limits 24
lock 25
orientation vector 25
position 24
unlock 25
velocity 24

S

Serial link
set up 10
Session replay 19
Set the serial link 10
Set the work offset 11
Set Variables Macro 22
Start PC-DMIS NC Server 12, 16
Statistics
viewing within PC-DMIS NC Server 17
Status fields for a machine 18
Stop the PC-DMIS NC Server 18

T

Test offline 9
Tool number 26
Tool offset
geometry and wear 26
wear 26
Travel limits 24

U

Units Address 22
Units of measurement 21
Unlock rotary table 25
Unprotected 27
Unprotected moves 13

V

Variables base 22
Velocity 24
Vertical machine 21

W

Wear 26
Work offset 11, 22
update 14
use 14
Work plane
YZ 27
Workplane
XY 27
XZ 27