PC-DMIS NC Manual

For PC-DMIS 2013 MR1

By Wilcox Associates, Inc.

Copyright © 1999-2001, 2002-2013 Hexagon Metrology and Wilcox Associates Incorporated. All rights reserved.
PC-DMIS, Direct CAD, Tutor for Windows, Remote Panel Application, DataPage, and Micro Measure IV are either registered trademarks or trademarks of Hexagon Metrology and Wilcox Associates, Incorporated.
SPC-Light is a trademark of Lighthouse.
HyperView is a trademark of Dundas Software Limited and HyperCube Incorporated.
Orbix 3 is a trademark of IONA Technologies.
I-DEAS and Unigraphics are either trademarks or registered trademarks of EDS.
Pro/ENGINEER is a registered trademark of PTC.
CATIA is either a trademark or registered trademark of Dassault Systemes and IBM Corporation.
ACIS is either a trademark or registered trademark of Spatial and Dassault Systemes.
3DxWare is either a trademark or registered trademark of 3Dconnexion.
lp_solve is a free software package licensed and used under the GNU LGPL.
PC-DMIS for Windows uses a free, open source package called lp_solve (or lpsolve) that is distributed under the GNU lesser general public license (LGPL).

```
lpsolve citation data
----------------------
Description: Open source (Mixed-Integer) Linear Programming system
Language: Multi-platform, pure ANSI C / POSIX source code, Lex/Yacc based parsing
Official name: lp_solve (alternatively lpsolve)
Release data: Version 5.1.0.0 dated 1 May 2004
Co-developers: Michel Berkelaar, Kjell Eikland, Peter Notebaert
License terms: GNU LGPL (Lesser General Public License)
Citation policy: General references as per LGPL
Module specific references as specified therein
You can get this package from:
http://groups.yahoo.com/group/lp_solve/
```

PC-DMIS for Windows uses this crash reporting tool:

“CrashRpt”
Copyright © 2003, Michael Carruth
All rights reserved.
Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:
Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
Neither the name of the author nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.
THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Table of Contents

Using PC-DMIS NC (Numerical Control) 1
PC-DMIS NC: Introduction 1
Uses for PC-DMIS NC 2
Differences between PC-DMIS NC and PC-DMIS 2
PC-DMIS NCI Differences 6
Required Components for PC-DMIS NC 7
Files Used by PC-DMIS NC 7
Data Flow Chart for PC-DMIS NC 8
PC-DMIS NCI Components 10
Testing PC-DMIS NC 11
Step 1: Install the Probe and Macros 11
Step 2: Install PC-DMIS NC and Define a Machine 11
Step 3: Perform an Offline Test 12
Step 4: Set up the Machine Link 13
Using PC-DMIS NC 14
Step 1: Calibrate the Probe 14
Step 2: Set the Work Offset 15
Step 3: Define an NC Machine 16
Step 4: Create a NC Program File 16
Step 5: Start the PC-DMIS CNC Server Program 17
Step 6: Load the NC Program File 17
Step 7: Execute the NC Program File 17
Step 8: Analyze and Report the Data 17
Using PC-DMIS NCI 17
Step 1: Install PC-DMIS NCI and Define a Machine 18
Step 2: Generating the Monitor Program 18
Step 3: Executing NCI Programs 19
Working with CNC Edit Window Commands 19
Update Work Offset 20
Use Work Offset 21
Update Tool Offset 22
Set Protection 23
Write CNC Variable 24
Read CNC Variable 24
CNC Pass Through Command 25
CNC Select Table 25
PC-DMIS NC Rotary Tables 26
Overview 26
Step 1: Rotary Table Physical Configuration 26
Step 2: Rotary Table Setup 27
Step 3: Calibrating the Rotary Table 29
Step 4: Verify the Rotary Table Calibration 31
Troubleshooting Rotary Table Problems 31
Working with a Multi-Axis Wrist 32
PC-DMIS Support for Stacked Rotary Tables 33

Part Programs for Stacked Rotary Tables 41
A Note on V42R Compatibility 42
Working with PC-DMIS NC in Offline Mode 42
Using the PC-DMIS CNC Server Application 43
The PC-DMIS CNC Server Menubar 43
The PC-DMIS CNC Server Command Buttons 44
The Machine Status Fields 45
Stopping and Resuming PC-DMIS CNC Server 45
Exiting PC-DMIS CNC Server 45
Moving Programs to the CNC Controller 46
Executing Journal Files 46
Creating and Modifying Machines 47
Working with the MonitorView Window 57
Using PC-DMIS on a Siemens Lathe 71
Calibrating a Siemens NEF400 Lathe 71
Measuring 119
Update Work Offset for a Siemens Lathe 129
Update Tool Length for a Siemens Lathe 131
Macros 135
CncServer Configuration 140
Running Multiple Version of PC-DMIS NC 155
Lathe Upgrade Conclusion 155
Using NC Update 156
Creating Problem Definitions 156
MonitorView Script Options 156
Index 157
Glossary 159

Using PC-DMIS NC (Numerical Control)

PC-DMIS NC: Introduction

Note: The NC product line is not available in the PC-DMIS 64-bit version (x64).

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. PC-DMIS NC part programs may be created in offline mode and used to generate an NC program.

PC-DMIS NCI: PC-DMIS NCI (Numerical Control Interactive) provides an Interactive version of PC-DMIS NC. It enables interactive connection between the core PC-DMIS product and a machine tool similar to that used with a CMM controller. Notes are provided throughout this documentation for exceptions or information that is specific for "PC-DMIS NCI".

In order to measure a part using NC tools, you will need to first fit your NC machine with a probing device and macros. You can then execute the generated part program and return point measurements via a communications interface to the PC-DMIS CNC Server program running on a computer. The PC-DMIS CNC Server application 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.

PC-DMIS NC supports these hardware configurations:

Hardware	Configuration	Supported NC Versions
Fanuc:	FOCAS over Ethernet	NC & NCi
	FOCAS over HSSB	NC & NCi
	Serial line	NC – not interactive
	Shared file drop	NC – not interactive
Siemens:	Requires HMI Advanced Option	
	Shared file drop	NC
	TCP-IP	NCi
Haas:	Serial line	NC – not interactive
Heidenhain:	Feature Content Level 3 (FCL3) recommended for Controller	
	LSV2 protocol over Ethernet	NC
Okuma:	Thinc Controllers OSP100 or later	
	Shared file drop	NC
Mazak:	Fusion 640 or Matrix control	
	Serial line	NC – not interactive
	Shared file drop	NC
Tosnuc 888:	Serial line	NC – not interactive

This documentation discusses the following main topics:

- Testing PC-DMIS NC
- Using PC-DMIS NC

- Using PC-DMIS NCI
- Working with CNC Edit Window Commands
- PC-DMIS NC Rotary Table
- Working with PC-DMIS NC in Offline Mode
- Using the PC-DMIS CNC Server Application

Use this documentation in conjunction with the main PC-DMIS documentation if you come across something in the software that isn't covered here.

Uses for PC-DMIS NC

The following topics describe situations where PC-DMIS NC may be useful:

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 an NC machine you'll have to deal with the following:

- The part may need to be removed from its 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 its 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. Script files can be written to perform process control checks and to react to changes in the data. Script files can be executed manually, or can be made to execute automatically when part measurements enter the system.

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 computations on varied measurement input can be easily compared.

Differences between PC-DMIS NC and PC-DMIS

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

- PC-DMIS NC does not run interactively with the NC machine. However, PC-DMIS NCI does work interactively (See "PC-DMIS NCI Differences"). 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 CNC

Server needs to select the associated PC-DMIS program. The PC-DMIS CNC Server program then loads probing data from the NC machine into PC-DMIS NC and executes the selected program.

- PC-DMIS NC version 1.0 typically cannot send data to the NC controller during execution. Controllers shown with NC or NCi in the "PC-DMIS NC: Introduction" topic are capable of updating work offsets, tool offset, and variables in the control. Controllers for which this is not possible are shown with (NC- not interactive).
- Probing paths are static and cannot be changed for all NC versions. 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. PC-DMIS NCi versions operate interactively with PC-DMIS sending each path point to the control allowing logic operations to control program path execution and iterative alignments.
- Alignments computed from features will only be used during the analysis phase for NC.

The following PC-DMIS topics list particular items as either supported or unsupported in PC-DMIS NC. Where PC-DMIS NCI supports or does not support a particular feature, notation is provided. Otherwise, both NC and NCI are both either supported or unsupported. For information on these specific items, view the main PC-DMIS documentation.

Automatic Features

Common Items

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

Circle

- Circular Move – Supported. The path will be generated with straight line segments by PC-DMIS when only protected moves are allowed. Circular moves are supported in the major machine axis planes XY YZ ZX if the NC controller can perform them. *Unsupported in PC-DMIS NCI which only works with straight line moves.*
- Find Hole – Unsupported. Requires controller based path generation. *Supported in PC-DMIS NCI.*

Cylinder

- Find Hole – Unsupported. *Supported in PC-DMIS NCI.*
- Circular Move – Supported for unprotected motion. *Supported in PC-DMIS NCI.*
- Use Theoretical – Supported.

Cone

- Circular Move – Supported. *Unsupported in PC-DMIS NCI.*

Line

- For CMT Features Only – Unsupported.

Corner Point

- Supported since the 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. *Supported in PC-DMIS NCI.*

High Point

- Unsupported. Supported in PC-DMIS NCI.

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. Supported in PC-DMIS NCI.

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 or *executed directly using PC-DMIS NCI.*
- BASIC Scans – Unsupported. These are controller dependent for generating the scan data.
- Stitch Scanning - Supported when using predefined paths. Supported in PC-DMIS NCI for predefined or adaptive paths.

- 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 – A single continuous table is supported.

Parameter Change

Measurement Speed

- Move Speed – Supported.
- Touch Speed – Unsupported for systems that use a fixed feed rate for measurement moves. Supported for macro sets that allow setting feed rates for measurement moves, for example the Blum macro set.
- 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. Manual operations are not translated.
- Probing Parameters – All are Unsupported. Probing parameters are defined within the Probing macros. *Probing speed, prehit and retract distances are supported in PC-DMIS NCI.*
- Auto-Relative Measure – Unsupported. *Supported in PC-DMIS NCI*
- Prehit and retract distances – Supported.

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.

NC-100 Video Settings

- Unsupported.

Flow Controls

- Supported for PC-DMIS NC if they do not alter the path, measurement sequence, or require measurement results inside a loop. Absolute defined paths are supported (such as loop that measures a point 5 times), however, 'if' statements for example are not supported. *In PC-DMIS NCI, all loop modes and logic are functional.*

Calibrate

- Probes - Supported for single and star tips. The controller resident probing macros may also be used in place of PC-DMIS NC generated calibration. Probe information (tool offset and effective ball diameter) will be requested from the control after every tool change operation. When PC-DMIS probe calibration is used, the probe length and effective ball diameter are managed by PC-DMIS. *PC-DMIS NCI only uses PC-DMIS probe calibration.*
- Rotary Table - Supported.

Basic Scripts

- Supported if they do not affect the measurement sequence or the path which is static. *Supported in PC-DMIS NCI.*

Operations

- Move-To – Unsupported.
- Column Recovery – Unsupported.
- Master/Slave Mode – Unsupported.
- Calibrate/Edit – Active Probe and Rotary Table are supported.
- Rotate Table to Active – Unsupported.

Edit

- Markings – Supported as long as the components of the part program that cause motion when the CNC program was generated remain the same when the program is executed by the server. *Fully supported in PC-DMIS NCI.*

Breakpoints

- Breakpoints are supported. Breakpoints can be used as a debugging tool, but once a program is running on an unattended server all breakpoints should be removed as well as any other commands that require a response to a prompt. Breakpoints will not stop the execution on the NC machine; only the PC-DMIS execution will be affected. *Since PC-DMIS NCI is interactively executing the program on a single user system, operator prompts and responses may be entered as normal.*

Note: Statistics update commands prompt for response. To automatically update the database, select the **Always Update Database** option on the **Dimension** tab of the **Setup** dialog box.
Note: Several variations of COMMENT commands also require response, so they should be removed prior to running the program.

PC-DMIS NCI Differences

The following are fundamental differences of PC-DMIS NCI.

Differences from PC-DMIS:

- DCC operation only
- Calibration Tool Location is definable for DCC probe calibration.
- Initial rough location of the part must be specified in Machine Coordinates (no manual teach-in).

Differences from PC-DMIS NC:

- No post out of a measurement path program (.cnc).
- The active work offset has no effect. All motion is generated in machine coordinates and adjusted to match the active work offset.
- Only one machine is defined using CncServer.exe. The machine must have Machine ID = 1.

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 supported probing and probing macros installed.

Computer with PC-DMIS NC - A Computer running Windows XP 32-bit or Windows Vista 32-bit 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 running PC-DMIS NC or NCI. PC-DMIS NC version 1.0 currently supports serial, Ethernet, File, or HSSB interfaces to communicate between the two computers depending on the type of controller.

Controller Options -

- **Fanuc:** Macro B
- **Fanuc:** Macro memory expansion
- **Siemens:** HMI Advanced
- **Heidenhain**: LSV2
- **Heidenhain:** FCL3 Recommended for 3d probing

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

PC-DMIS NCI: To enable this option, The port lock is programmed for the appropriate NC Interface:

• Fanuc machines with FOCAS option use the **NCFanuc** interface.
• Siemens machines with HMI Advanced option use the **NCSiemens** interface.

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 files 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. Interfaces that do not have direct access to setting work and tool offsets will generate NC programs to update these offsets. Typically the file extension for these programs is .wrk.

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 journal subdirectory of the directory displayed in the **Program** tab of the **MachineDefinition** dialog box of the PC-DMIS CNC Server program.

PC-DMIS NCI: Journal files are not used for PC-DMIS NCI.
Example: If PC-DMIS NC was installed to C:\PCDMISW\, journal files for PC-DMIS program Block.prg would be stored in C:\PCDMISW\Journal\Block\.

Machine Definition Files - These files are stored in the <PCDMISINSTALL>/CncMachineData folder. Sub-directories for each machine contain compens.dat, .PRB files, and the rotary table calibration files.

Machine Related Files - Controller files are located in the following directories that must be copied to the controller:

- **Blum** - <PCDMISINSTALL>/BLUM
- **Fanuc** - <PCDMISINSTALL>/FANUC
- **Haas** - <PCDMISINSTALL>/HAAS
- **Heidenhain** - <PCDMISINSTALL>/HEIDENHAIN
- **Mazak** - <PCDMISINSTALL>/MAZAK
- **Okuma** - <PCDMISINSTALL>/OKUMA
- **Siemens** - <PCDMISINSTALL>/SIEMENS

Monitor Program Set - The Monitor program set consists of NC programs which perform basic move/measure functions and the Monitor program which interfaces with interfac.dll. The NC Programs for move/measure may also be used in PC-DMIS NC as Wilcox probing macros without the Monitor.

PC-DMIS NCI:
The CNCServer.exe and the Ctrltdefaults.dll files are required for PC-DMIS NCI.

NCFanuc.dll - Interface file for the Fanuc controller that is renamed to interfac.dll as needed.

NCSiemens.dll - Interface file for the Siemens controller that is renamed to interfac.dll as needed.

Monitor Program Set - The set is resident on the NC Control and consists of a group of executable NC programs coordinated by a master Monitor program.

The Monitor program is started by the Operator when requested by PC-DMIS NCI to perform move and measure functions. PC-DMIS NCI treats the Monitor as a server and sends requests to it in the form of op codes and attendant data (like target position). When the Monitor completes an operation, the results and status are sent to PC-DMIS NCI and it waits for the next command.

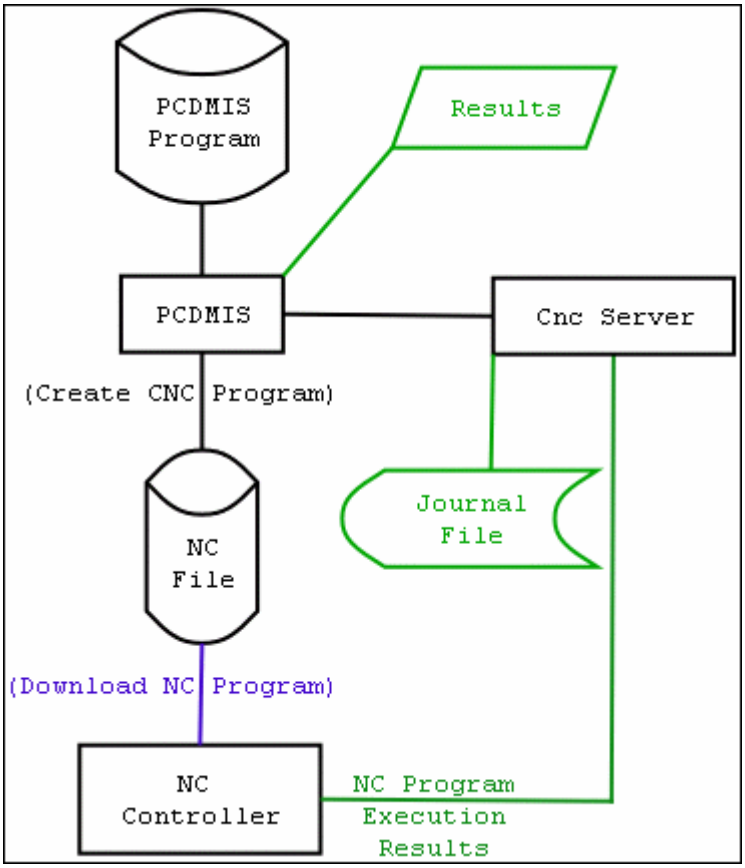
When PC-DMIS NCI goes to Manual mode, it sends an exit command to the Monitor which gracefully terminates. The Cancel button is also active during PC-DMIS execution to immediately stop the Monitor.

Data Flow Chart for PC-DMIS NC

In the chart below, the following process is described:

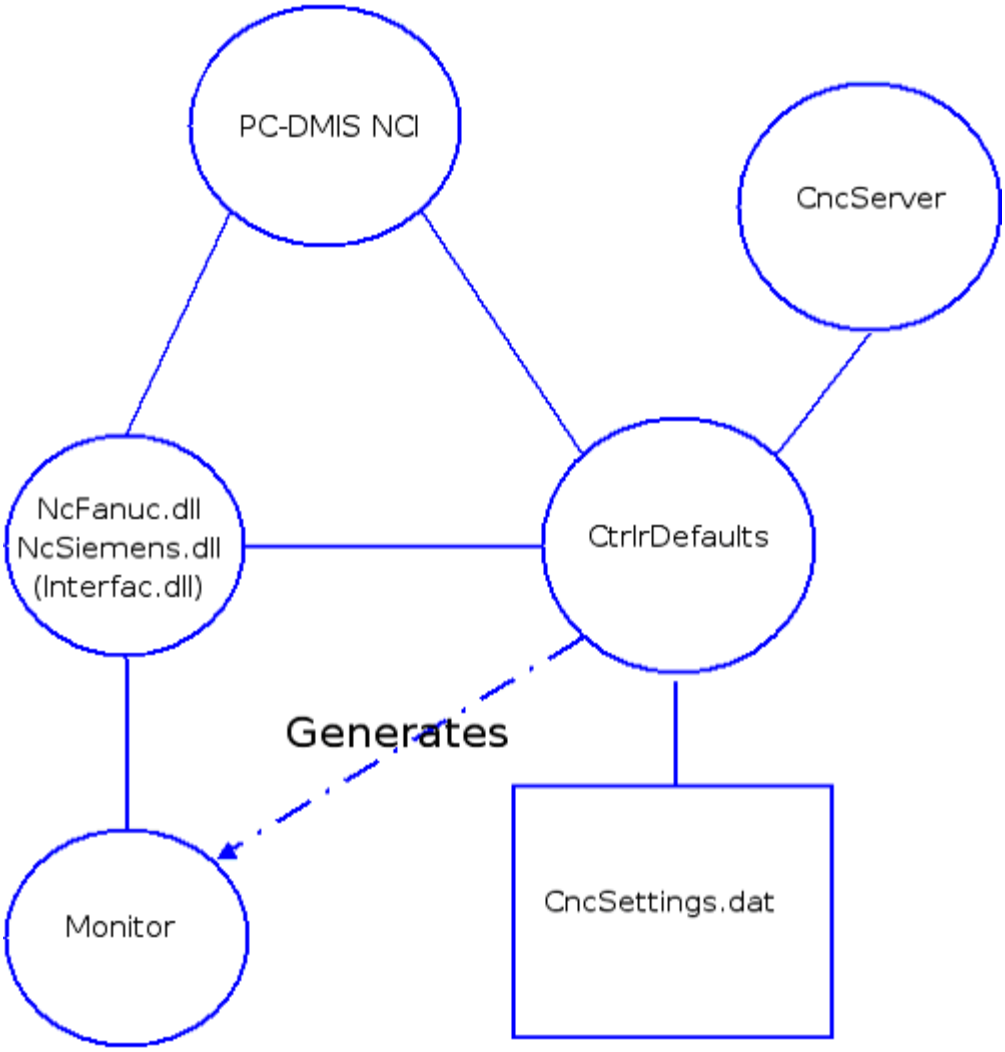
1. Create a PC-DMIS Part Program
2. Open the Part Program in PC-DMIS NC.
3. Select the **Operation | Create CNC Program** menu option. PC-DMIS NC creates the NC program file.
4. Start the PC-DMIS CNC Server program.

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 CNC 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.



PC-DMIS NCI Components

PC-DMIS NCI uses multiple components to generate output for the Monitor application. The components in the PC-DMIS software system are shown in the following diagram:



- **CNCSettings.dat** - Settings contained in this file are managed by CNCServer. This is used by PC-DMIS NC and PC-DMIS NCI to define the machines.
- **CNCServer** - Used to create machine definitions that are used to communicate with the NC controller.
- **PC-DMIS** - Executes NC programs based on the settings and interface information. Measurement results are collected by the Monitor and sent via Interfac.dll to PC-DMIS for evaluation.
- **Interfac.dll** - Either the NCFanuc.dll or the NCSiemens.dll are renamed to interfac.dll in the PC-DMIS install directory. The renamed interfac.dll file works in conjunction with PC-DMIS to interface with the NC Controller.
- **CtrlrDefaults** - Manages the machine interface and the access to set up information for the machine controller.

- **Monitor** - Resident on the NC Control and consists of a group of executable NC programs coordinated by a master Monitor program. Movement and Measurement requests from PC-DMIS NCI are executed by the Monitor returning status and results to PC-DMIS NCI via the Interfac.dll.

Testing PC-DMIS NC

This chapter describes how to install and test PC-DMIS NC with a simulated NC machine.

Note: These steps are not applicable when using PC-DMIS NCI. See the "Using PC-DMIS NCI" section for information

Step 1: Install the Probe and Macros

Use your probe system documentation to:

1. Install your probing system.
2. Install the probing macros.

Verify installation by calibrating the probe using the supplied calibration macros. If calibration macros are not supplied (i.e. Wilcox macros) run a calibration using PC-DMIS.

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. The CNC portlock options include:
    - **CNC Licenses** - Specifies the number of CNC machines that are connected for use with PC-DMIS.
    - **Update Mode** - With this option selected, PC-DMIS is allowed to pass measurement data back to the CNC machine to update work offsets and/or tool offsets. Must also have at least one CNC licenses specified. See "Using NC Update".
    - **NCi** - Authorizes the use of PC-DMIS NC interactive instead of the standard PC-DMIS configuration. See "Using PC-DMIS NCI".
2. Install PC-DMIS. The **CNC Server** application should get installed in the background.
3. Start PC-DMIS CNC Server by double-clicking on the **CNC Server** shortcut.
4. Open the **Define Machines** dialog box by:
    - From the **NC Server** interface: Clicking **Hold** and then selecting the **Define Machine** button
    - From the **PC-DMIS-NC Update** window (NC Update): Selecting the **File | Define Machines** menu item.
5. To define your machine do the following:
    a. Click **New**. The **MachineDefinition** dialog box appears.
    b. Fill out the **Machine Definition** dialog box. See "Creating and Modifying Machines" for more information.
    c. From the **Program** tab, name the machine "Test".

d. Type a value in the **Machine ID** box.

e. **CNC 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.

f. From the **Interface** tab, select **Test** from the **Interface Type** list.

g. Click **OK**.

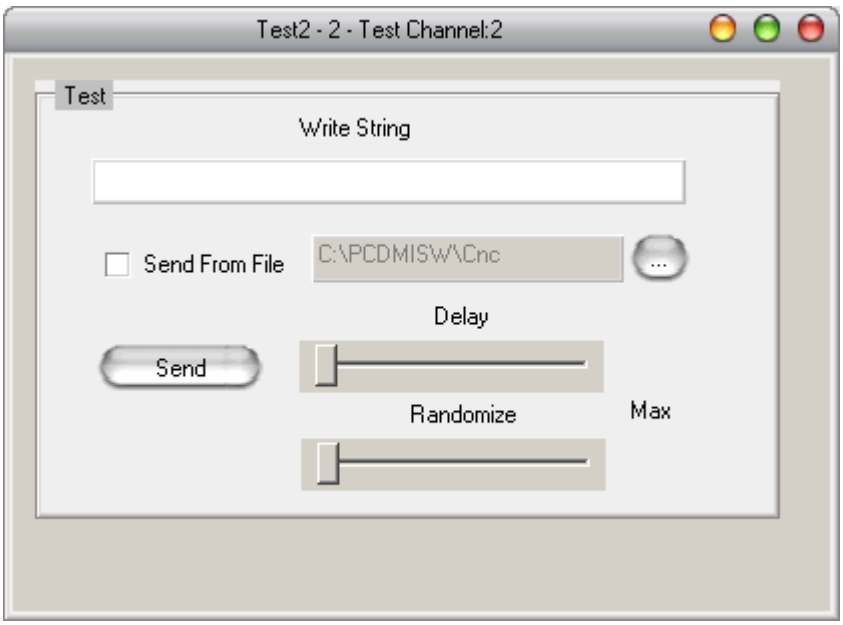
6. Close CNC Server by File | Exit.

Step 3: Perform an Offline Test

1. Start PC-DMIS NC.
2. Create a small part program and name it "Test".
3. Generate the NC program file by selecting **Operation | CNC Programming | Create CNC Program** menu option. PC-DMIS displays the **Select CNC Machine** dialog box.

Note: 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

4. Select **Test** from the **Select CNC Machine** dialog box.
5. 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).
6. Click **OK** to generate the .cnc program. PC-DMIS displays a processing message box and shows the path lines drawn in the Graphic Display window for the part program.
7. Using a text editor, open Test.cnc and verify that it contains commands. Close the CNC file and PC-DMIS NC.
8. Start the **CNC Server** application. You will see the **Test** machine inside the **CNC Server**.
9. Select **Test** from the list of machines. A **Test 1** dialog box appears.



Test dialog box

10. In the **Test** dialog box, select the **Send From File** check box.
11. 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** dialog box (for example, C:\PCDMISW\CNC\Test\Test.cnc).
12. 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.
13. Adjust the **Delay** and **Randomize** sliders to affect the simulated data. **Delay** controls the rate at which simulated data is sent to the CNC server. **Randomize** allows adding random errors to each measured point. Therefore, not all of the data will be represented as nominals. This is useful when doing offline demos and adding data to DataPage.
14. Check to see if the program executed.
15. 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
16. Open the journal file in a text editor such as Notepad. You should see data for the probing points generated by the Test simulator.

Note: The simulator merely returns a point at the end point of the probing move. Since probing moves are usually created to end inside the part surface, the probing points will be inside the surface of the CAD model. To make points on the surface, set the Prehit parameter in PC-DMIS NC to zero.
Note: Only marked features measured in DCC are processed. If the CNC file does not have all the measurements that are expected (and the path lines display is missing features), check that program has all desired features marked and the mode is set as DCC.

Step 4: Set up the Machine Link

This step discusses configuration of a Serial Link which is the most common. Other links would be Ethernet, HSSB and File.

1. Open a HyperTerminal session on the server computer:
2. From the Start menu, select Programs | Accessories | Communications.
3. 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.

4. 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.
5. Select a small program on the NC controller.
6. PUNCH out the program to the controller serial port. You should see the program being received in HyperTerminal's window.
7. As an additional test, select **Transfer | Capture Text** from HyperTerminal. A small dialog box appears.
8. Specify a folder and file name for the captured text file.
9. PUNCH out the program again.

10. Select the **Transfer | Capture Text | Stop** from HyperTerminal to stop capture.
11. Exit HyperTerminal and open the captured file in a text editor such as Notepad.
12. 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", then you can use PC-DMIS NC for real measurements online.

The instructions in this section are specific to PC-DMIS NC, see the "Using PC-DMIS NCI" topic for instructions on using PC-DMIS NCI.

Step 1: Calibrate the Probe

There are two methods for calibrating the probe:

Calibrating Using Macros

This method uses the macros supplied by the probe or NC manufacturer. In this method two macros are normally used:

- The first macro measures the size and location of a known (size and position) hole. The size of the hole is used to determine the diameter of the probe. The location of the hole determines the offsets of the probe center from the spindle axis.
- The second macro determines the probe tool length by touching a flat surface that is perpendicular to the spindle axis and whose location is known with respect to the spindle gage point. Follow your existing documentation for information on how to do this.

Calibration Using the CNC Server program

This method uses standard PC-DMIS calibration techniques. You may use the CNC Server program to calibrate the probe if the probe supports full 3 axis vectoring. You must use PC-DMIS calibration when using 5 axis systems.

To enable PC-DMIS probe calibration:

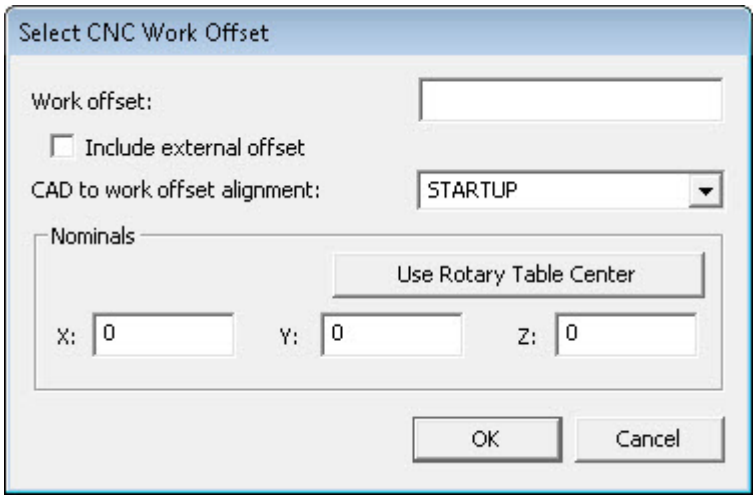
1. Start the **CNC Server** program
2. Click **Hold** and then **Define Machine**. This opens the **Define Machines** dialog box.
3. Select the machine from the list and click **Modify**. This opens the **MachineDefinition** dialog box.
4. Select the **Measure** tab. See the "MachineDefinition – Measure Tab" topic for more information.
5. Select the Use PcDmis_NC Probe Calibration box.
6. On the NC control, set the **Tool Length** offset and any **XY** offsets to zero in the **Tool** area. See NC documentation.
7. Select **Open PC-DMIS** and create a new blank part program.
8. Select the **Insert | Hardware Definition | Probe** menu item to open the **Probe Utilities** dialog box. Set the nominal offsets for the probe tips by building the probe from the components supplied. See the "Defining Probes" topic in the "Defining Hardware" chapter of the core PC-DMIS documentation.
9. Place a Qualification Sphere in the machine volume and define its exact location. Chuck a dial indicator in the spindle and sweep the diameter of the sphere. Jog the spindle until the variation is minimized. The

spindle position now defines two of the sphere axes. The third axis is set by loading a tool length standard in the spindle and bringing it to the top of the sphere with a feeler gage of known size between it and the sphere. Decrease the distance until the feeler gage is tight. The sphere center is the length of the master tool plus the feeler gage plus the known radius of the sphere. The values for the XYZ location of the sphere, it's diameter and stem vector are used to define a qualification tool in the **Measure Probe** dialog box.

10. Click **Add Tool** to define the parameters for calibration and the calibration tool sphere in the **Measure Probe** dialog box. Increasing the **Prehit / Retract** values may be helpful.
11. Press the **Measure** button when the calibration tool and all other parameters are correct. This will insert the measurement commands required to measure the qualification sphere into the current part program. Save the PC-DMIS part program.
12. Select the Operation | CNC Programming | Create CNC Program menu item to create a CNC program.
13. To complete the calibration process from the CNC Server:
    a. Download the newly created calibration program to the NC control.
    b. Select **Analyze** and execute the program. This should cause the NC machine to execute the calibration program and pass the measurements back to the CNC Server which will invoke the PC-DMIS qualification program with the measured data.
    c. The PC-DMIS probe file will be updated to use the calibration data.

Step 2: Set the Work Offset

Using the **Operation | CNC Programming | Use Work Offset** menu option, set the work offset for the part you will measure into the NC controller.



Select CNC Work Offset dialog box

Make a note of its number. PC-DMIS NC supports standard work offset registers. (Example G54-G59 for Fanuc style controls). The objective is to match up the initial PC-DMIS NC alignment with a work offset.

In the case of Heidenhain Controls, modify the Preset Table or Datum Table.

Include External Work Offset

In the case of Fanuc Controls, this will include the value of G53 X, Y and Z as the total offset.

- Case 1: The centers of the rotary tables are defined in machine coordinates, click the **Include External Offset** check box to enable it.

If G53 contains 100, 200, 300 mm and G54, 1P21 contains 1, 2, 3 mm, the total offset used to compute the location of the part in machine coordinates is 101, 202, 303 mm. This location would be used with the Rotary Table center definition in PC-DMIS NC to compute the location of the part when the table(s) are rotated.

- Case 2: The center of the rotary table is put in G53, make sure the Include External Offset check box is unchecked. This is compatible with older versions of PC-DMIS NC.

This way of working puts the rotary table center at 0 (Zero). The standard work offsets (G54,G55, etc.) define an offset from this effective zero. The returned work offset for the part (using the numbers above) is 1, 2, 3 mm.

Step 3: Define an NC Machine

1. Start the PC-DMIS CNC Server program on the server computer.
2. Click **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" for more information.
    - Include, along with other parameters, the tool changer slot number for the probe and the default work offset register.
    - Click **OK** until you return to the main PC-DMIS CNC Server window.
4. Click **Exit**, to close down PC-DMIS CNC 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 you are not running the PC-DMIS CNC Server program on it. Otherwise, you will need another computer that runs PC-DMIS NC.
3. Select the machine to use from the Settings tool bar pull down.
4. Create an unmarked alignment at the beginning of the program. You might define the work offset using features on the CAD model. However, this can be done in a variety of ways.
5. Change the mode to DCC mode.
6. Use a probe that has the same nominal ball diameter as the probe in the NC machine if you are using controller resident probe calibration. If you are using PC-DMIS probe calibration, select the probe and tip you have calibrated.
7. Generate the CNC measurement program from your part program by selecting the **Operation | CNC Programming | 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.

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. This is true for all Fanuc style controls that identify programs by a number like O1234. For other controls, the output file name identifies the program.
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 Graphic Display window. If you encounter errors, consult the log file inside any text editor application, such as Notepad. The log file is located in the "\Documents and Settings\<LoginName>\Local Settings\Temp" directory with an .log extension.

Step 5: Start the PC-DMIS CNC Server Program

Start the PC-DMIS CNC 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 **Hold** to cause the PC-DMIS analysis engine to stop running.

Step 6: Load the NC Program File

Using PC-DMIS CNC 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. Some interfaces have not implemented the **Download Program** button.

Step 7: Execute the NC Program File

1. Start execution of the NC program.
2. During execution, the NC program file will load the probe and proceed to measure the features defined in your part program.
3. Measured points are sent to the server computer from the NC controller.
4. PC-DMIS CNC 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 CNC Server which loads it into the original PC-DMIS NC part program for analysis, reporting, statistics and so on. The PC-DMIS CNC Server must be in Analyze mode to evaluate data.

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

Using PC-DMIS NCI

The steps for using PC-DMIS NCI are different from PC-DMIS NC since these two products are mutually exclusive. PC-DMIS NCI is basically PC-DMIS On-line attached to an NC machine. Communication with the NC Machine occurs by use of the Monitor program which resides on the NC control.

The instructions in this section are specific to PC-DMIS NCI, see the "Using PC-DMIS NC" topic for instructions on using PC-DMIS NC.

Step 1: Install PC-DMIS NCI and Define a Machine

1. Attach the port lock to your computer. Your port lock should already be programmed to enable the PC-DMIS NCI functionality.
2. Install PC-DMIS. The PC-DMIS CNC Server program should get installed in the background.
3. Start PC-DMIS CNC Server by double-clicking on the PC-DMIS CNC Server shortcut.
4. From within PC-DMIS CNC Server, define a machine:

Note: PC-DMIS NCI must **Not** be active while using CncServer.exe.

   a. Click **Hold** to cause the server to stop running.
   b. Click **Define Machines**. The **Define Machines** dialog box appears.
   c. Click **New**. The **Machine Definition** dialog box appears.
   d. Fill out the **Machine Definition** dialog box. See "Creating and Modifying Machines" for more information.
   e. From the **Program** tab, provide the **Machine Name** for your NC machine.
   f. Type a value or "1" in the **Machine ID** box.
   g. The PC-DMIS CNC 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.
   h. From the **Interface** tab, select **Ethernet** from the **Interface Type** list.
   i. Click **OK**.
5. Close PC-DMIS CNC Server by **File | Exit**.

Step 2: Generating the Monitor Program

A Monitor NC Program is used to communicate between PC-DMIS NCI and the NC Controller. The set of NC Programs includes the Monitor program and all of the support macro programs it requires to implement movement and single point measurement. Monitor programs may be custom generated using CncServer.exe.

To Generate a .CNC File:

1. Complete "Step 1: Install PC-DMIS NCI and Define a Machine" All parameters must be correctly set.
2. Verify correct settings in the **MachineDefinition** dialog box and click **OK** to close. Click **OK** to close the **Define Machine** dialog box to exit out to the main CNC Server screen. This will save your machine definition settings.
3. Open the **Measure** tab of the **MachineDefinition** dialog box. Click the **Generate** button to create a .cnc file for the control formatted as “Wilcox”<controller Base name>.cnc (ex. WilcoxFanuc.cnc).
4. Load the newly created .cnc file onto the NC machine. The name of the monitor set is WilcoxFanuc.cnc for Fanuc controls and WilcoxSiemens.cnc for Siemens controls.

Important considerations for generating your .CNC file:

- If the generated program requires modification to run, it is best to use the **MachineDefinition** dialog box to make changes. This will insure that you have a valid parameter set for automatic regeneration. Make the needed changes and exit out to the main screen of the CNC Server application.

- If you have edited the generated .cnc file using an editor such as Notepad, etc. and you do not update the **MachineDefinition** dialog box settings to match, then create backup copies of the modified Monitor program set from the control.

Note: You cannot change the Macro Variable addresses by editing the .cnc file because PC-DMIS NCI is using the information from the Machine Definition to define the addresses of these variables for communication.

Step 3: Executing NCI Programs

Before you run your PC-DMIS part program, you need to change the MODE/MANUAL command at the beginning or your part program to MODE/DCC. PC-DMIS NCI only runs in DCC mode.

Running a Part Program:

Part Programs are run in the same manner as PC-DMIS CMM programs with the inclusion of the Monitor. To run your program using PC-DMIS NCI:

1. Open the .PRG file from PC-DMIS NCI.
2. Place the cursor at the start of the program.
3. Click on Execute or type <CTRL>Q.
4. When prompted to start the Monitor, select the monitor program on the NC Control and Cycle Start. The **Start Monitor** dialog box will close and DCC motion should begin on the NC Machine.

Stopping and Restarting a Part Program:

- The Monitor program will be terminated on the NC control whenever PC-DMIS goes to Manual mode within a program or when the program is completed. When DCC operation is required, the **Start Monitor** dialog will appear.
- Canceling the execution of a program can be done with the **Cancel** button in the PC-DMIS **Execution** dialog box. This will also stop the Monitor program at whatever point it was in its execution loop.
- To restart the program, repeat the steps starting from Step 1 above. Always insure that the Monitor program is started from the beginning. PC-DMIS may be restarted at the current cursor position or the beginning.

Counters

The PC-DMIS counters will be active when the program is not running. The coordinates are defined by the counter setting. It is useful to read the center of probe position for the active tip when jogging around the part. READPOINT can be effectively used to roughly locate the starting point for DCC operation.

Working with CNC Edit Window Commands

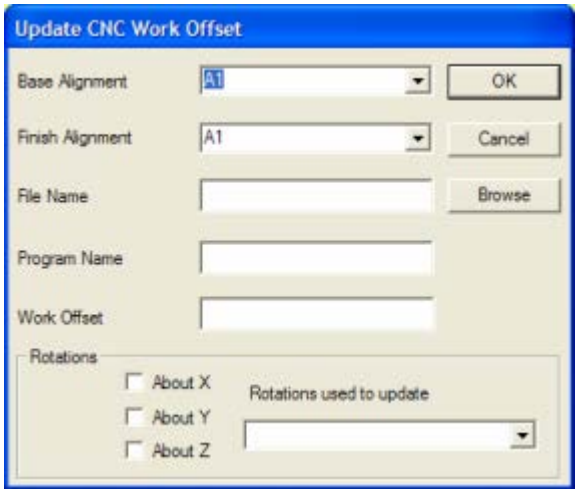
PC-DMIS NC provides you with the following Edit window commands. To modify these commands, press F9 on any command in the Edit window to access its dialog box, or edit them directly inside the Edit window's Command Mode.

To insert these commands into the Edit window 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.

Update Work Offset

Select the Operation | CNC Programming | Update Work Offset menu option to open the Update CNC Work Offset dialog box.



Update CNC Work Offset dialog box

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 a CNC program file that PC-DMIS NC will generate. Some CNC interfaces (for example, Ethernet) 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.

`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 for Fanuc style controllers. This option defines the work offsets that will be enabled in the CNC program.

Rotations

Rotations about base alignment axes may be applied to either a physical rotation axis, like a rotary table, or as a rotation of the NC coordinate system (**Note:** Many controllers do not have this capability). Select the rotation axes to enable: **About X**, **About Y**, and/or **About Z**. The **Rotations Used to Update** field may be set to either:

None: No rotations

Rotary Axes: Rotate the rotary axis associated with the Cartesian axis using the standard rule A about X, B about Y or C about Z. These will show up as rotation offsets in the Work Offset area of the control. This is used to Update Rotations will update the standard work offset rotations G54,G55,G56,G57,G58,G59.

Transformation: Apply rotations to the NC Controller's coordinate system. The changes may show up in a special set of registers in the control (i.e. Fanuc Work Offset Error Compensation registers) or as an extension of the standard work offsets (Siemens 840).

Fanuc - For Fanuc controllers this updates transformations in Fanuc Registers for Work Offset Compensation:

G54 ----> G54.4 P1 .. G59 ----> G54.4 P6

Base Addresses:

G54.4 P1 **X** - 26010 **Y** - 26011 **Z** - 26012 …

G54.4 P2 – P7 X 26020,26030,26040,26050 …

Enabling Work Offset Compensation in an NC Program: (Example From Fanuc Document Work Piece Setting Error)

G55 (Enable std work offset)

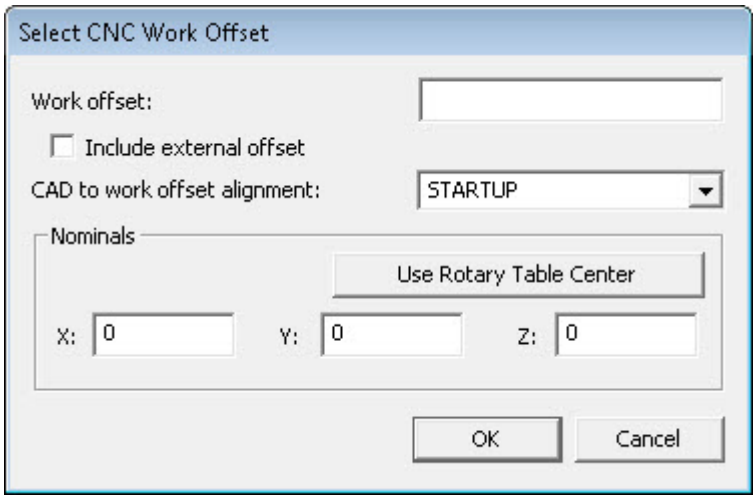
G05.1 Q1 (AICC AI Contouring Control must be active first)

G54.4 P1 (Enable Work Offset compensation #1)

Siemens - The selected User Frames G54 … G57 are updated for rotation and translation.

Use Work Offset

Select the Operation | CNC Programming | Use Work Offset menu option to open the Select CNC Work Offset dialog box.



CNC/USEWORKOFFSET,workOffset,alignmentCommand,nomX,nomY,nomZ

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

workOffset can be a string which represents a basic work offset (example G54-G59). 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 after the first commanded motion until the USEWORKOFFSET command specifies another. The Nominals field in the dialog represent the position in Machine coordinates of the work offset. These fields are required when using a Rotary Table to correctly transform motion and returned point data.

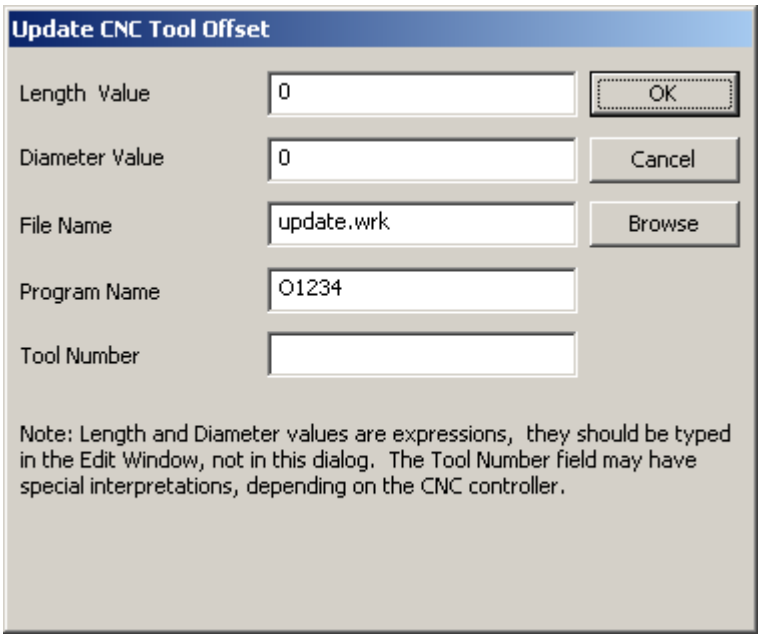
Include external offset check box can be used to include any external work offsets associated with the program, otherwise the external work offsets will not be included.

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.

nomX, nomY and nomZ are the expected values for the X, Y and Z axes of the controller. These values represent the offset values in a perfect world. The X, Y and Z values must be entered if a rotary table is present. These values are used to compensate for table rotation.

Update Tool Offset

Select the Operation | CNC Programming | Update Tool Offset menu option to open the Update CNC Tool Offset dialog box.



Update CNC Tool Offset dialog box

CNC/UPDATETOOLOFFSET,lengthFeature,diameterFeature,fileName,programName,toolNumber

This command calculates changes to a tool-offset (length and/or diameter) and communicates that change to the CNC. Length changes are determined by computing the difference between the nominal and actual locations of a flat or straight feature. Diameter changes are determined by computing the difference between nominal and actual radii of a round feature

lengthFeature should be a PC-DMIS “flat” feature. If tool length is not being set, this can be left blank. The location of this feature relative to its normal vector is compared to the nominal location. The difference defines a

tool length offset. You can select this from the Length Feature list in the Update CNC Tool Offset dialog box for this command. This list contains “flat” PC-DMIS feature commands available to this point in the PC-DMIS part program.

`diameterFeature` should be a PC-DMIS “round” feature. If tool diameter is not being set, this can be left blank. The diameter of this feature is compared to its nominal diameter. The difference defines a tool diameter offset. You can select this from the Diameter Feature list in the Update CNC Tool Offset dialog box for this command. This list contains “round” PC-DMIS feature commands available to this point in the PC-DMIS part program.

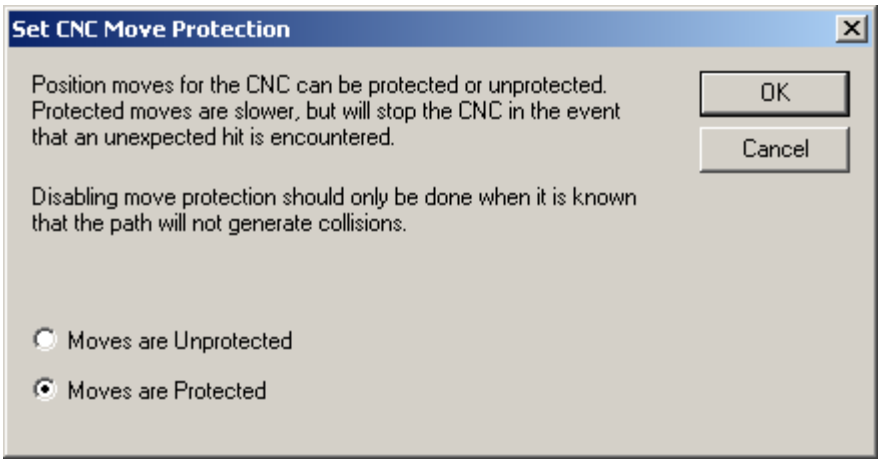
`fileName` is the name of a CNC program file that PC-DMIS/NC will generate. Some CNC interfaces (for example, Ethernet) upload the work-offset changes directly. In these cases, the fileName parameter is not used. If no file extension is provided, the extension, "WRK", will be appended to the CNC file name.

`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.

`toolNumber` is the number of the tool that this command is being applied to.

Set Protection

Select the Operation | CNC Programming | Set Protection menu option to open the Set CNC Move Protection dialog box.



Set CNC Move Protection dialog box

CNC/SETPROTECTION, **`movesProtected`**

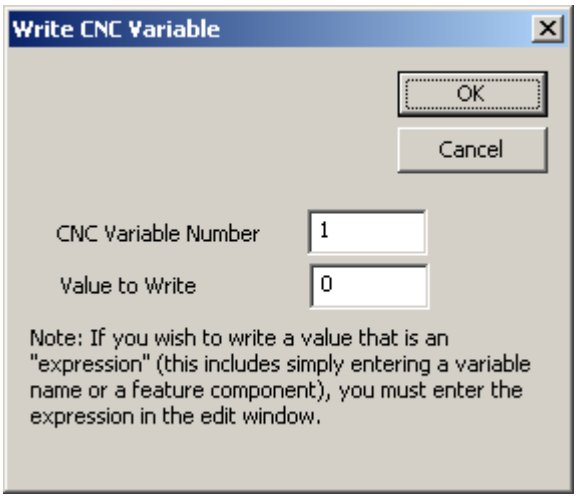
This command sets the state of moves. By default protected moves are safe. This means unexpected probe hits will terminate part program execution. Because of the potential danger of unprotected moves, a second safety lock is in place in the definition of machines in PC-DMIS CNC Server.

`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 be recognized by the controller and execution will be suspended or terminated, depending on how the CNC control deals with alarm conditions. Some controls will have some error recovery abilities, some will not. 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 resulting in damage to the probe.

PC-DMIS NCI: This menu option and Edit window command are not available for PC-DMIS NCI.

Write CNC Variable

Select the Operation | CNC Programming | Write CNC Variable menu option to open the Write CNC Variable dialog box.



Write CNC Variable dialog box

CNC/WRITEVARIABLE,variableNumber,valueToWrite

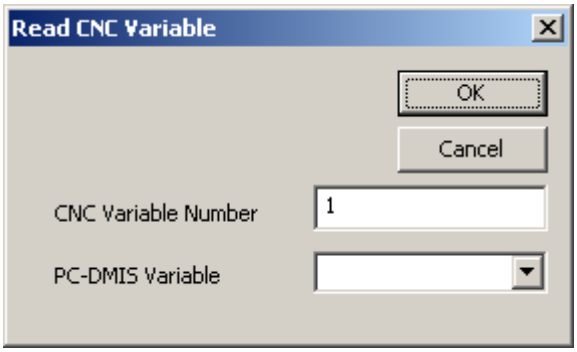
This command writes a value to a CNC register. A block of 10 values is reserved on the CNC control for I/O between the part program and the control.

`variableNumber` should be a value between 1 and 10. This will be mapped to a register on the CNC control.

`valueToWrite` is the value that will be written to the CNC register. This can be either a floating point number or an expression that evaluates to a floating point number. Expressions must be added in the edit window rather than entering it in the dialog. This command is supported only by controls that allow writing directly to registers by outside processes.

Read CNC Variable

Select the Operation | CNC Programming | Read CNC Variable menu option to open the Read CNC Variable dialog box.



Read CNC Variable dialog box

CNC/READVARIABLE,variableNumber,variableName

This command reads a value from a CNC register. A block of 10 values is reserved on the CNC control for I/O between the part program and the control.

`variableNumber` should be a value between 1 and 10. This will be mapped to a register on the CNC control.

`variableName` is a PC-DMIS variable created with the ASSIGN command. Select a previously defined variable name from the PC-DMIS Variable combo box.

Note: Reading of CNC variables from the scratchpad area is now done when the controller executes the program. Previous versions of PC-DMIS (2009 and earlier) used to read them when PC-DMIS did an Analysis. You can no longer expect variables set in a CNC/WRITEVARIABLE command to be picked up in a CNC/READVARIABLE command because they now occur at different phases in the program: READVARIABLE occurs during control execution while WRITEVARIABLE occurs during PC-DMIS Analysis.

CNC Pass Through Command

Select the Operation | CNC Programming | Pass Thru menu option to open the CNC Pass Through Command dialog box.

CNC Pass Through Command dialog box

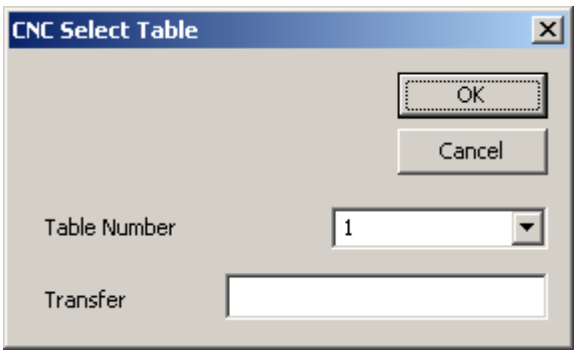
CNC/PASSTHRU,**`CommandString`**

This command string is inserted inline in the generated NC Program. It may be any text string, but should represent valid commands for the controller. No syntax checking is performed in PC-DMIS.

`CommandString` Text string to be inserted into NC program.

CNC Select Table

Select the Operation | CNC Programming | Select Table menu option to open the CNC Select Table dialog box.



CNC Select Table dialog box

CNC/SELECTTABLE,TableNumber,CommandString

This command string is inserted inline in the generated NC Program. It may be any text string, but should represent valid commands for the rotary table. No syntax checking is performed in PC-DMIS.

`TableNumber` Text string to specify the rotary table to which the command string will be sent.

`CommandString` Text string to be transferred to the rotary table for rotation.

PC-DMIS NC Rotary Tables

Overview

PC-DMIS NC rotary table support is based on capabilities within PC-DMIS. The following configurations are supported:

- Only one continuous rotary table.
- Two unconnected rotary tables.
- Indexed rotary tables.
- The typical NC configuration of a pair of stacked rotary tables is also supported. For a machine with this configuration, one of the tables must assume a fixed position for measurement.

Rotary tables are calibrated for location and orientation by generating a program using the PC-DMIS rotary table calibrate option in the Rotary Table Setup dialog box. The calibration information is stored by machine and recalled for each program analysis or path generation.

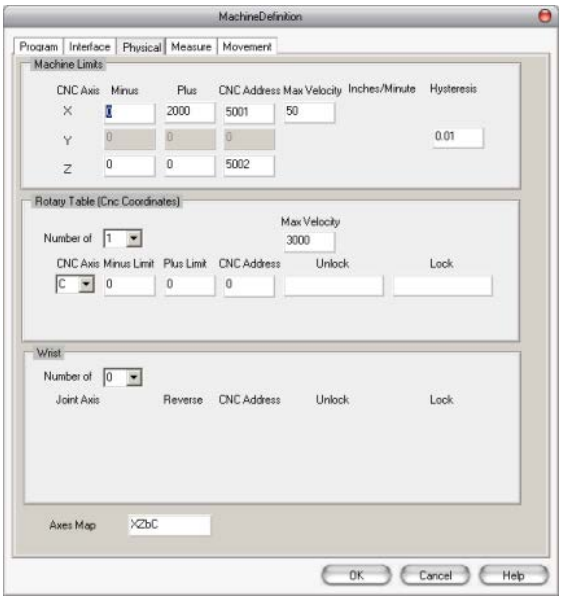
Tips may be calibrated either with the vendor supplied macros (such as Renishaw) or PC-DMIS probe calibration. Tip length and rotary table center position are interrelated when the tip is not parallel to the rotary table axis. This is the typical case represented by a horizontal machining center with a rotary table vector aligned with the Y axis. Consequently, changes in tip length will affect the rotary table calibrated position. Spindle growth is the prominent error for machine tools reflected in Z tip position. Any error on the horizontal configuration will be doubled between zero and 180 degrees rotary table position.

Note: The **Rotary Table** option must be programmed into the port lock as well as at least one **CNC license**. To use a stacked rotary table with PC-DMIS, the **Stacked Rotary Table** option must be programmed as well.

Step 1: Rotary Table Physical Configuration

To define the Physical configuration for your Rotary Table for use with PC-DMIS NC:

1. Click Define Machine in the CNC Server program.
2. Select the needed machine configuration from the list and click **Modify**.
3. Open the **Physical** tab of the **MachineDefinition** dialog box.



MachineDefintion - Physical tab

4. Specify the **Minus Limit** and **Plus Limit** for your Rotary Table. The limits can be set to zero if no limit checking is required. Values less than the minus limit and greater than the plus limit will not be sent to the control.

5. Provide the **CNC Address** for controls to report the rotary table position for measurements.

6. Specify the **Max Velocity** of your Rotary Table. The product of this value and the percentage of maximum speed for PC-DMIS moves is used to define the feed rate of table moves.

7. Specify the **Number of** tables if you are configuring your system to use Dual Rotary or Stacked Rotary tables.

8. Select the **CNC Axis** that corresponds to each rotary table that is used to generate table motion. Select A, B or C. The first item in the list refers to the PC-DMIS W axis and the second line refers to the V axis. PC-DMIS assumes that the W axis is stacked on top of the V axis.

9. Adjust the **Wrist** joint values as needed. See the "Working with a Multi-Axis Wrist" topic for more information on these values.

10. Specify the **Unlock Table** command that is used to unlock the table before rotation can occur. Likewise, provide the **Lock Table** command that is used after rotation.

11. Axes Map maps the controller axis registers to a named axis for Fanuc style controls. The first axis shown is mapped to the first axis register. 'b' symbols are spacers to skip unassigned axes. The example show maps the X,A,Y,Z,C and B axes to position registers 5001,5007 – 5011.

Step 2: Rotary Table Setup

To setup your rotary table:

1. Start PC-DMIS

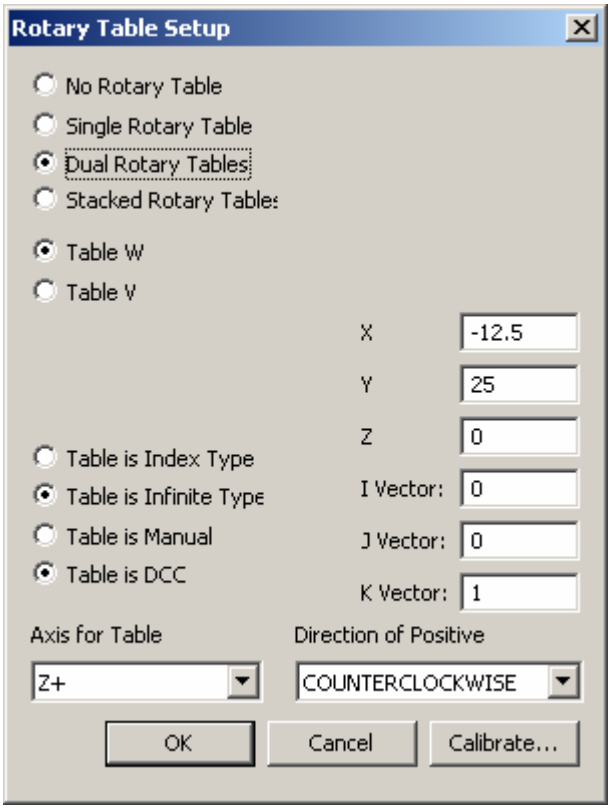
2. Create a new Part Program. It should have the same **Measurement Units** (IN or MM) as your machine for ease in debugging problems, but this is not mandatory.

3. Select the machine of interest from the **Settings** toolbar using the drop-down list. **Note:** PC-DMIS NCI only assigns one machine per PC-DMIS computer, so there is no drop down list.

Important: The correct machine should be selected from the **Settings** toolbar whether you are creating paths for calibration, rotary table configuration or general measurement programs.

4. Select the Operation | Calibrate/Edit | Rotary Table menu option to open the Rotary Table Setup dialog box.

Note: The **Rotary Table Setup** dialog box can also be accessed from the **Edit | Preferences | Rotary Table Setup** menu option



Rotary Table Setup dialog box

5. Select the type of rotary table configuration from:

- **No Rotary Table -** Rotary table is not being used.
- **Single Rotary Table** - For a single rotary table.
- **Dual Rotary Table** - Two rotary tables, not necessarily stacked.
- **Stacked Rotary Tables** - Two rotary table that are stacked.

6. For **Dual** or **Stacked Rotary Table** configurations, you will need to specify the parameters for each table while either the **Table W** or **Table V** option is selected. The settings for each table is independent of the other.

7. For each rotary table, select either the **Table is Index Type** or **Table is Infinite Type** option.

8. For each rotary table, select either the **Table is Manual** or **Table is DCC** option.

9. For each rotary table, select the **Axis for Table** from the drop-down list box. Select the normal vector of the table, this axis is used when defining the **Direction of Positive Rotation**.

Important: Specify the **Axis for Table** before updating the XYZ values.

10. Enter the nominal location of the table center in the **XYZ** boxes. This should be entered in "machine coordinates", that is with no tool or work offsets active.
11. Avoid updating the **IJK vector** values directly.
12. Select either CLOCKWISE or COUNTERCLOCKWISE from the **Direction of Positive Rotation** drop-down box. CNC machines can have a left-handed relationship between the table axis vector and direction of rotation. The direction of rotation defines how the table turns when one is looking "down" the **Axis for Table Vector**.
13. Enter the nominal location of the table center in the **XYZ Center** boxes. This should be entered in "machine coordinates", that is with no tool or work offsets active.
14. Enter the normal vector of the table in the **IJK Vector** boxes.

Note: Work offsets are assumed not to be in effect for Rotary Table calibration.

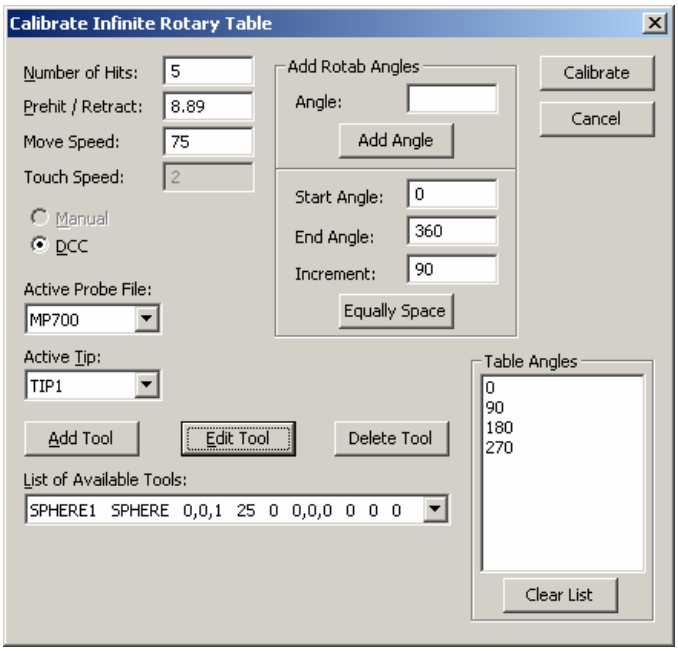
Step 3: Calibrating the Rotary Table

To calibrate your rotary table:

1. Create a new empty PC-DMIS part program.

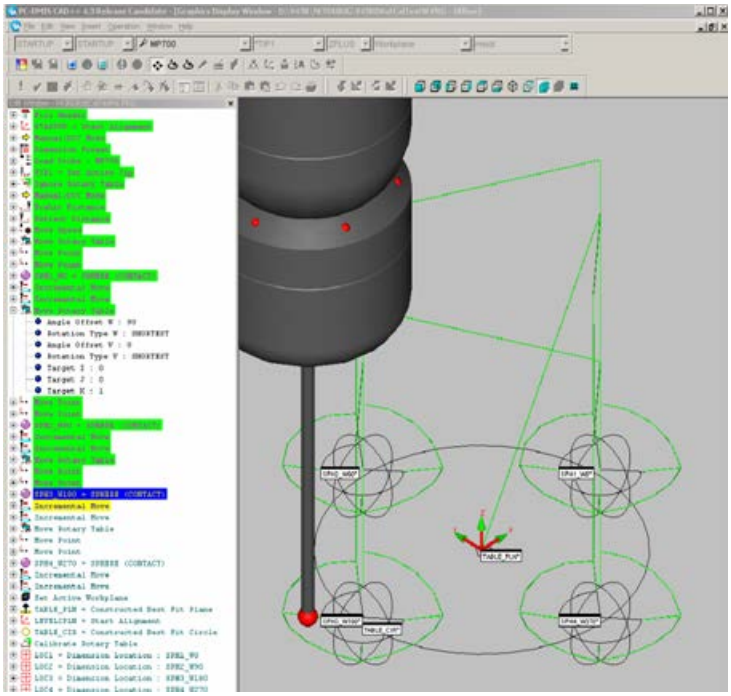
Important: If you are calibrating Dual or Stacked Rotary Tables you will need to select either **Table W** or the **Table V** before clicking **Calibrate** from the **Rotary Table Setup** dialog box. The physical axes for both of these tables should also be set to zero. You will need to complete all of the following steps for both tables.

2. Click the **Calibrate** button on the **Rotary Table Setup** dialog box. This opens the **Calibrate Infinite Rotary Table** dialog box



Calibrate Infinite Rotary Table dialog box

3. Specify the Probing parameters for the calibration process. **Number of Hits** that will be taken on the Qualification tool. The **Prehit/Retract** value will determine the distance from the hit location that the probe moves to before and after the hit is taken. The **Move Speed** value is a percentage of the maximum speed of the probe.
4. Select the **Active Probe File** and the **Active Tip** that will be used for calibration.
5. Click **Add Tool** to define a new the calibration tool or click **Edit Tool** to change an existing calibration tool. See the "Add Tool" topic in the core PC-DMIS documentation for more information. The location of the tool is at zero degrees on the rotary table in machine coordinates. The tool location is determined by the user, by jogging the probe close to the “top” of the probe, reading the counters and adding or subtracting the probe radius to one of the axes. The last step depends on the orientation of the probe with respect to the table.
6. To add a single Rotary Table angle, type the needed angle value above the **Add Angle** button and click **Add Angle**. This will add the specified angle to the **Table Angles** list.
7. To add multiple Rotary Table angles, type the needed **Start Angle**, **End Angle** and **Increment** values. Click **Equally Space** and new Rotary Table angles will be added to the **Tables Angles** list starting with the starting angle and incrementing up to the ending angle.
8. Click **Calibrate.** The commands that will be used to calibrate the rotary table will be written to the current part program. PC-DMIS will generate measurement paths for all the sphere positions.



Rotary Table Calibration Path Lines and Sphere locations.

9. Save the current part program so that it can be executed when the calibration is performed using **CNC Server**.

PC-DMIS NCI: Rotary table calibration will begin immediately with the display of the **Start Monitor** dialog box. Start the Monitor program on the control and PC-DMIS NCI will command the necessary moves and measure commands to calibrate the table.

Step 4: Verify the Rotary Table Calibration

To verify your Rotary Table calibration:

1. Select the **Operation | CNC Programming | Create CNC Program** menu option to post out the calibration program.
2. Open the **CNC Server** program.
3. Click **Download Program** and browse and select the newly created CNC program.
4. Click **Analyze** and execute the Calibration program on the NC machine.
5. Note the updated calibration information in the **Rotary Table Setup** dialog box in PC-DMIS after the calibration is complete. You may wish to add dimensions for the calibration program so as to view the results of the calibration in a report.
6. **Check the Calibration** - Create a sphere measurement program with the rotary table enabled. In this program, the sphere will always be at zero degrees in PC-DMIS, but the location of each measurement will deviate by the error in calibration of the table.

Troubleshooting Rotary Table Problems

Problem	Readings across the part have a large error.
Solution	The tool length and rotary table location are not matched. Rerun the journal file while adjusting the rotary table center. Remember that the rotary table data will be reloaded from CncMachineData/<machine name>/rotab.dat when the machine is selected and restored when unselected. PC-DMIS NCI: For PC-DMIS NCI, the standard rotab.dat file in the installation directory is used. If the results improve with center adjustment, try recalibrating the rotary table. If thermal drift is an issue, a local recalibration of the spindle length (Z) can be included in the PC-DMIS program by measuring a known location.
Problem	The path goes the wrong way
Solution	Clockwise/Counterclockwise is wrong in Rotary table setup.
Problem	Motion is not generated for the axis
Solutions	The rotary table is not enabled in PC-DMIS. You don't have the rotary table option in your port lock. The Axis code in the Machine Definition is wrong.
Problem	Rotary table results do not track table angle for Fanuc controls
Solution	Verify that the wrong table angle is being included with every point in the journal file *(or the machine readout if NCI)*. If it is, position the table at a known location and write a simple MDI program to read the location. For example **#530 = # 5004** If the value of the arbitrary temporary variable #530 is wrong, review the controller / machine documentation to

	find the right address. #5004 is typical for the A axis, but the axes may not be linearly mapped against controller addresses. Change the value in the machine definition, save all, close down and restart PC-DMIS CNC Server.

Working with a Multi-Axis Wrist

PC-DMIS NC must be setup to use a wrist device in the **Physical** tab of the **MachineDefinition** dialog box.

Change the following values on the Physical tab :

1. In the **Wrist** area set the number of wrist axes using the drop-down list.
2. For each axis, define the **Axis** identifier used to move the wrist.
3. The **Reverse** setting may also need to be checked to make the motion of the wrist compatible with PC-DMIS' expected motion.
4. Edit the **Axes Map** which provides the ordering of the machine axes in relation to the logical controller axes. See the "MachineDefinition - Physical Tab" topic for more information on the **Axes Map**.

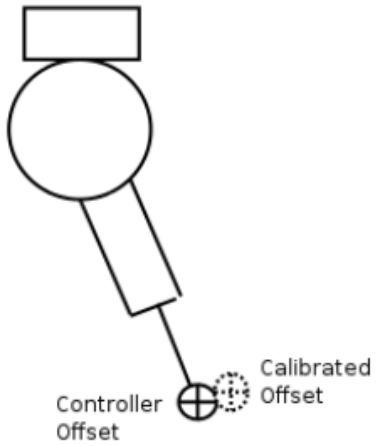
The wrist must be defined in the USRPROBE.DAT file or the PC-DMIS PROBE.DAT file. This file must be supplied on installation with a wrist modeled as a connection with No offsets.

The wrist must always be calibrated at each angle used for measurement using PC-DMIS Probe calibration.

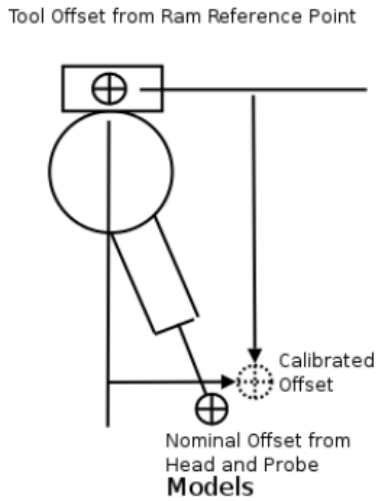
There are two methods in which the wrist may be defined depending on your installation: **Tool Point** or **End of Ram**

Tool Point: This method assumes the controller has the ability to predict the location of the end of the tool regardless of head angles. For example the Siemens 840D control enables this capability with the TRAORI command. The length of each tool, including the measuring probe, is known to a high degree of accuracy, but the remaining errors can be removed by calibration. Do not zero out the tool offsets. Model the probe as a single ball of the same nominal size as the stylus with the No Offset wrist. Only a single stylus may be used. The probe is calibrated at the angles of interest using PC-DMIS probe calibration. The stored calibration values consist of small errors in the XYZ location of the stylus and the stylus diameter.

Tool Offset Known by Controller



End of Ram: This method requires an accurate model of the wrist in the probe database. It is similar to the method used to calibrate CMM wrists. The Probe Offsets maintained in PC-DMIS are now from the end of the machine ram to the ball center of the stylus. The tool offset for the probe in the control, if any, is zeroed before probe calibration. A calibration program is created for the angles of interest, posted out and executed on the NC machine.



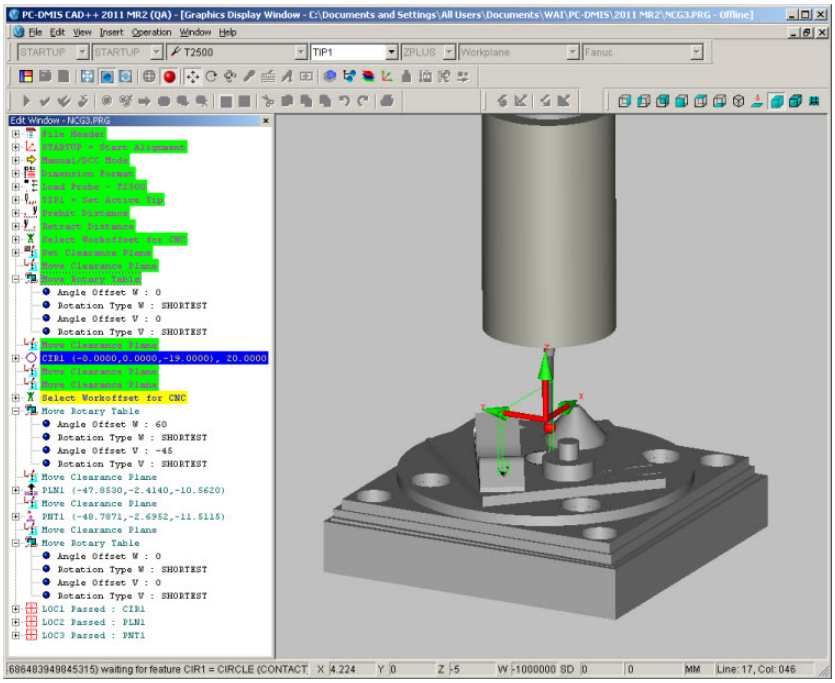
PC-DMIS Support for Stacked Rotary Tables

Support for stacked rotary tables running on CNC machines existed in PC-DMIS v42B but, while the addition of kinematic changes in subsequent releases allowed a picture of a stacked rotary machine, it was unusable on CNC machines. For the first time it is possible to run a CNC machine Using a Machine Model, calibrate probe and rotary tables, properly animate and detect collisions with either the machine or part. PC-DMIS NC now includes Using Work Offsets with a Machine Model.

The MathCAD Model used to validate point data in input and output files for Fanuc, Siemens and Heidenhain controller types is included at the end of this document.

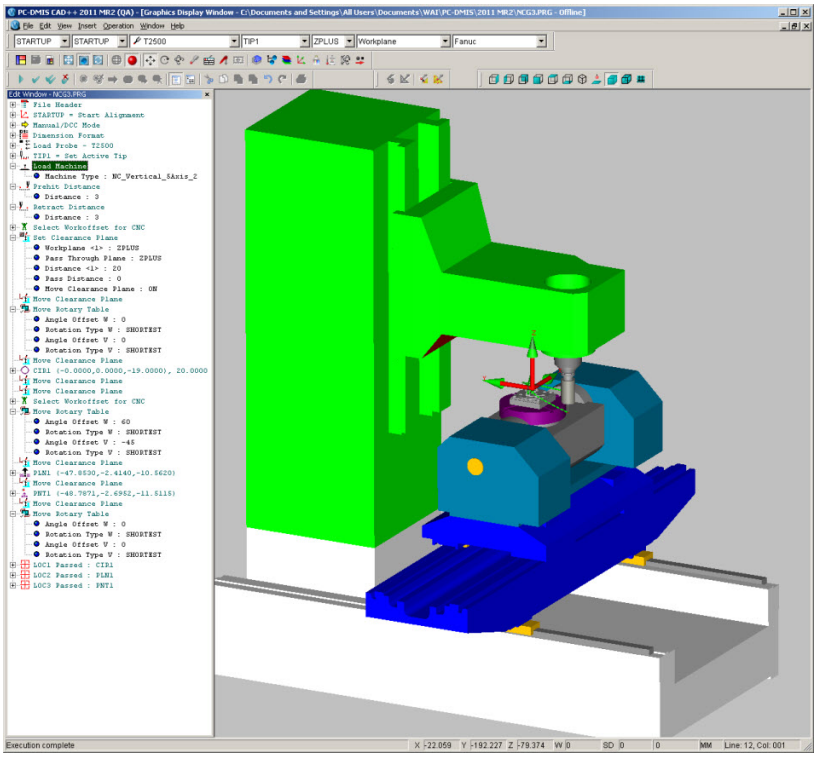
Using a Machine Model

A simple PC-DMIS program (NCG3.PRG) using a stacked rotary table is shown below.



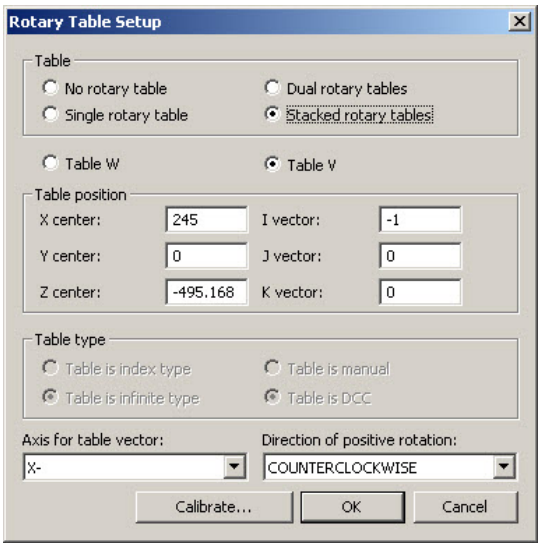
Stacked Rotary Table Program

When animated. the probe moves about and there is no visual indication of the machine or rotary tables. Adding a machine model (NC_Vertical_5Axis_2) provides this feedback as shown below.



Stacked Rotary Table Program with Machine Model

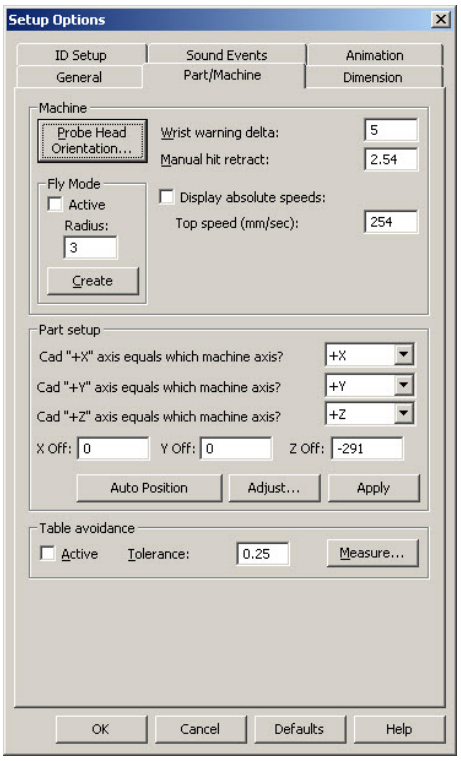
The rotary table setup is now automatically updated from the machine model. Select **Edit | Preferences | Rotary Table Setup** to view the updated values in the **Rotary Table Setup** dialog screen shown below.



Rotary Table Setup Dialog Screen for Stacked Rotary Tables

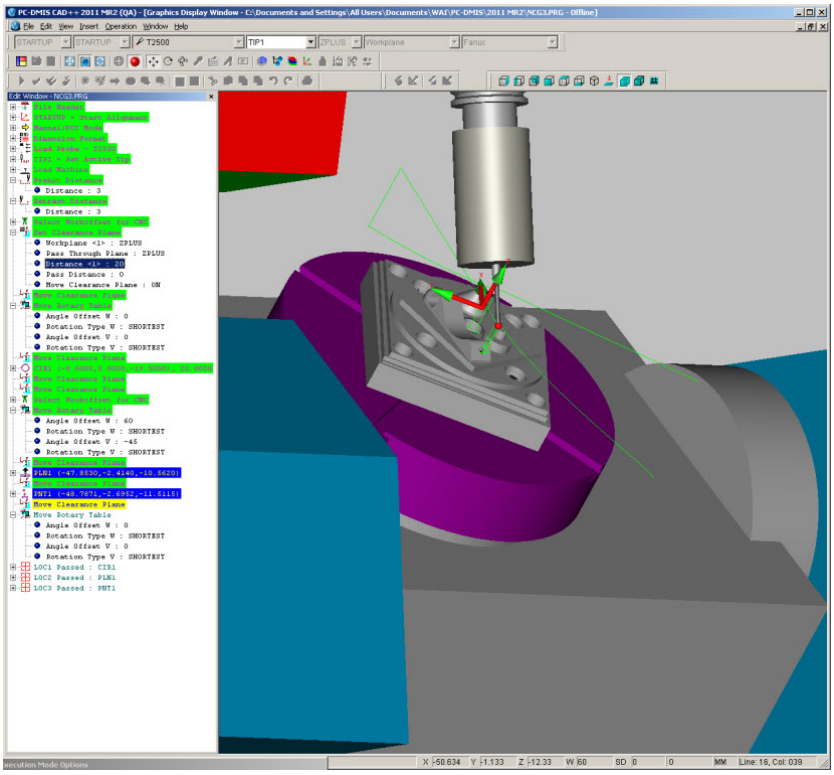
Adjusting the values manually in the dialog now updates the machine model held in memory within PC-DMIS and used by the Pathlines generator. The position of the part in some models can be automatically positioned at the center of the rotary table using the **Auto Position** button in the **Setup Options** dialog. This is analogous to the other option of transforming the CAD.

Press the **F5** key to display the **Setup Options** dialog screen shown below.



Setup Options Dialog Screen for Stacked Rotary Tables

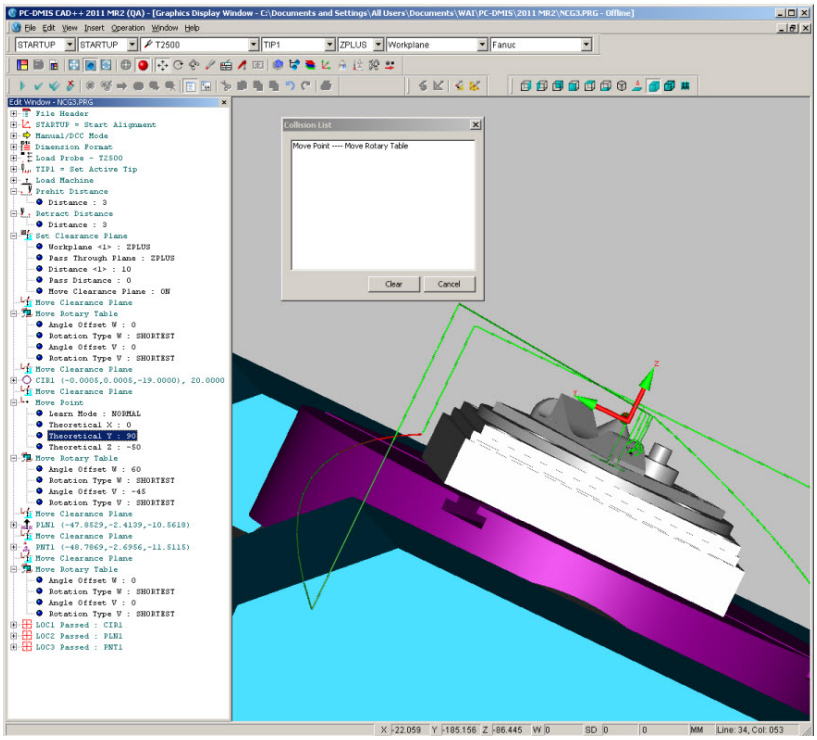
Now it is possible to see accurate Pathlines after successfully calibrating the rotary table. PC-DMIS now animates the machine moving the rotary tables more realistically.



Updated Stacked Rotary Table with Machine Model Program
Collision Detection with part and machine is also possible.

Lowering the Clearance Plane and adding the Move Point (0,90,-50) is an example of a collision with the rotary table when it is rotated.

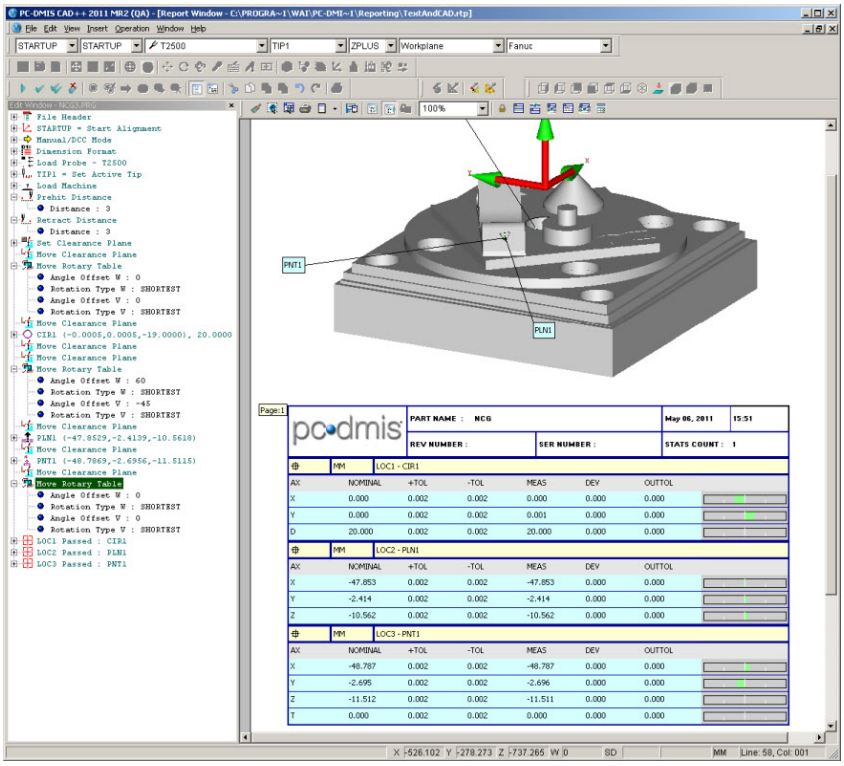
Note: Non-zero prehit and retract distance are needed or collisions will be indicated for every hit.



Updated Stacked Rotary Table with Machine Model Program and Collision Detection
Creating a .cnc file now produces the same Pathlines as used when animating.

Run the Journal file (HU80102011032117295 6.cnj) to produce a perfect report via the Execute function, then navigate to and select the journal file when the dialog screen appears. Click the **Ok** button to generate the report.

When doing this test, the prehit and retract values should be zeroed prior to generating the .cnc file to avoid adding Overtravel distances to the Journal data.

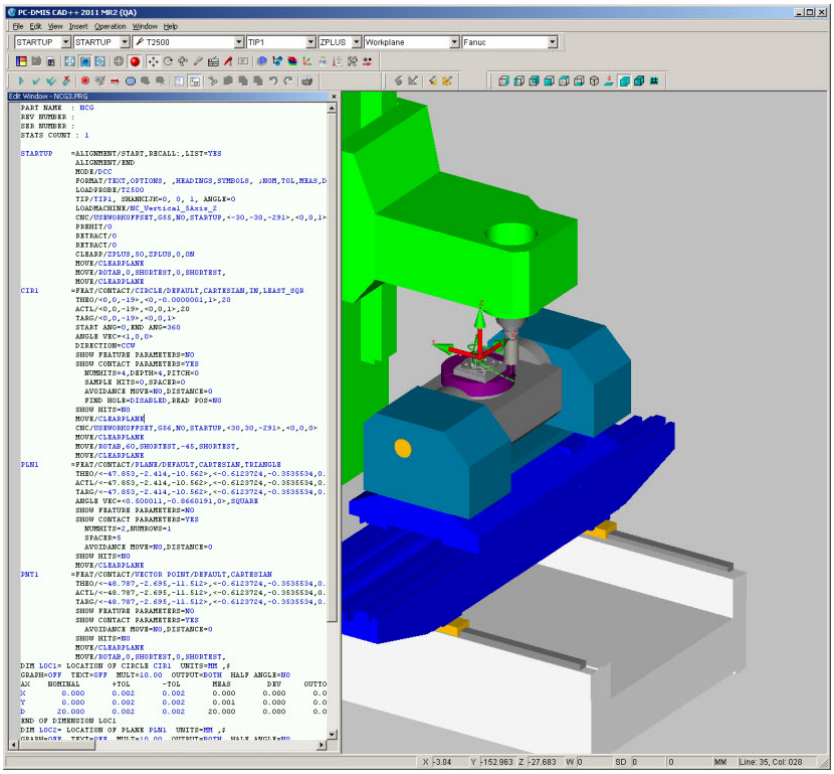


Stacked Rotary Table Program with Report

Using Work Offsets with a Machine Model

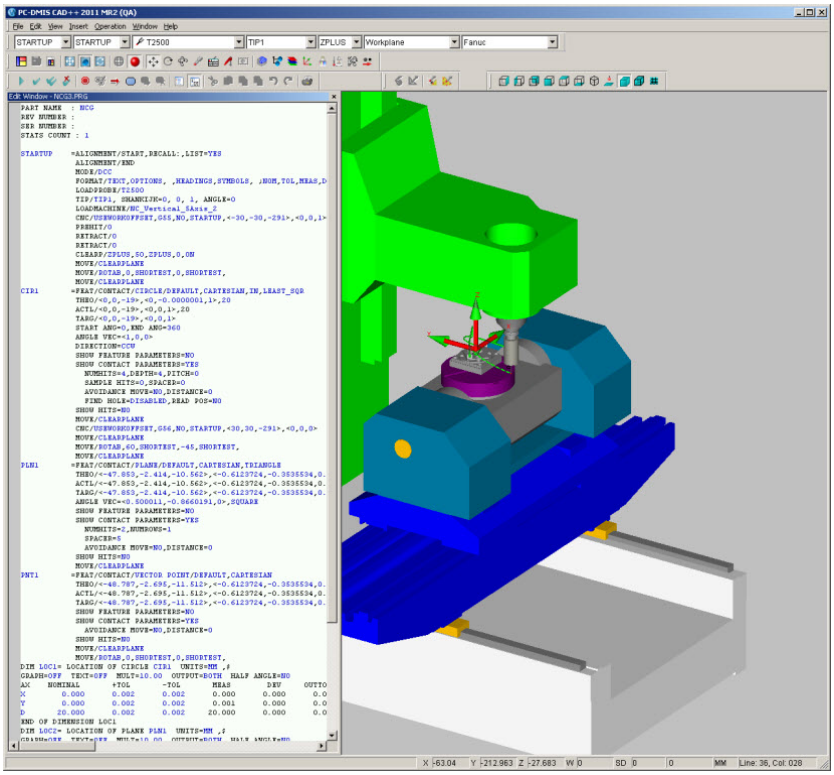
The machine Work Offset needs to be specified by name and value by the user when working with a machine model if Animation and Collision Detection are to work correctly.

Multiple Work Offsets can be specified in this way. Scrolling up and down within the **Edit Window** produces an effect that can be seen graphically as the part moves when the work offset is updated in the same way as any update to a rotary table angle.



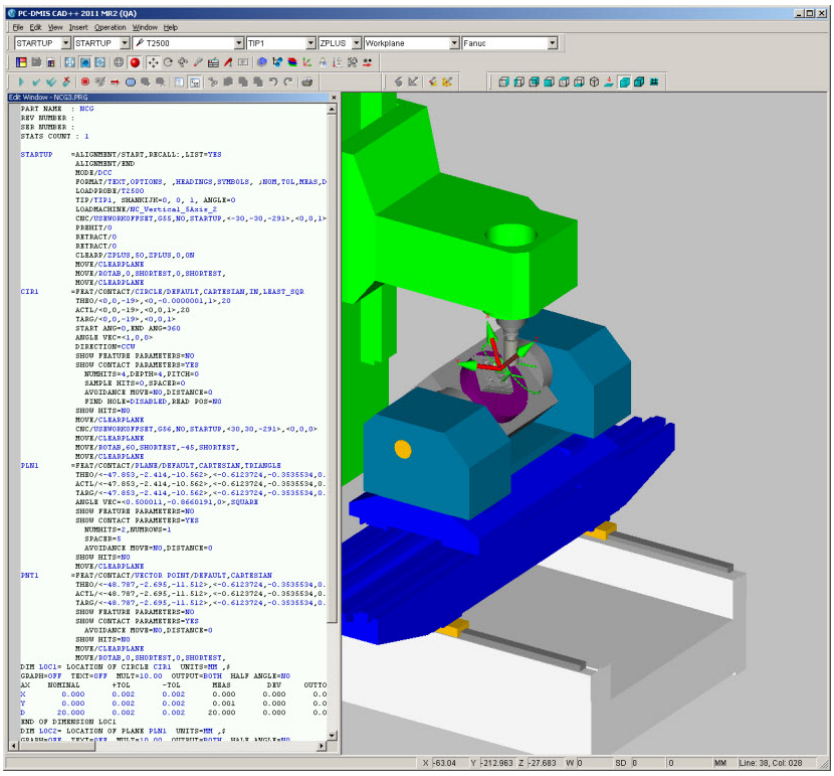
Stacked Rotary Table with Machine Model Program and Work Offset initial line

Scrolling down one line in this case to the Work Offset which is updated from G55 at (-30,-30,-290) to G56 at (30,30,-290), the CAD display shows the part moving (60,60) as follows:



Stacked Rotary Table with Machine Model Program and Work Offset down one line

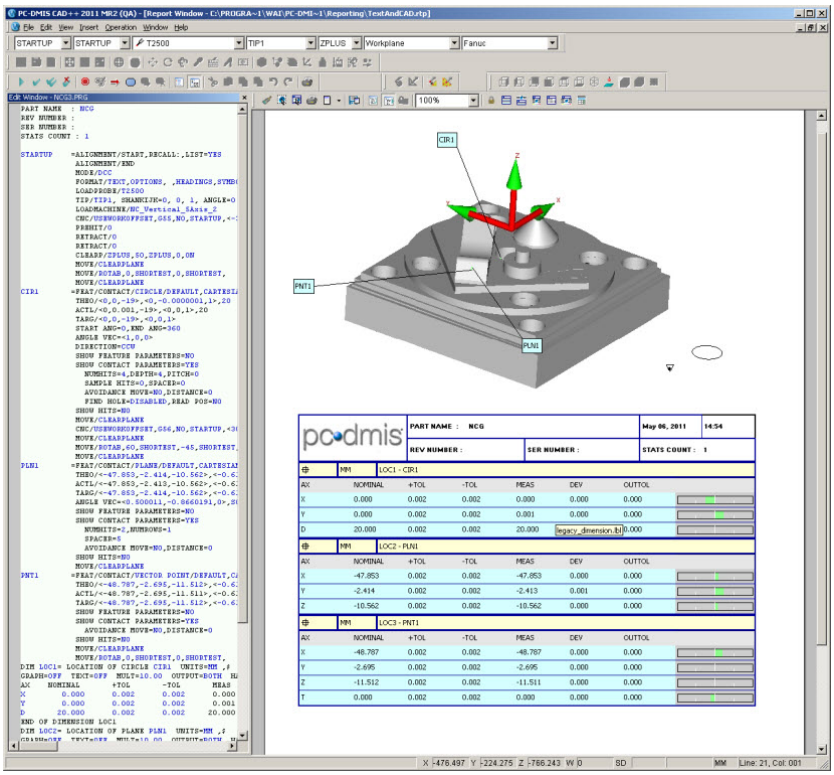
Moving down two more lines and the rotary table angles are updated from (0,0) to (60,-45) and the display is updated as follows:



Stacked Rotary Table with Machine Model Program and Work Offset down two more lines

Of course, when animating the part program, these steps are also seen in sequence. When generating the .cnc part program, the selected Work Offset is selected by name as usual. In this case G55 is followed later by G56.

This can be tested by pasting the same point data from the generated .cnc file, into the .cnj Journal file and adding the lines for each work offset to generate a good report as shown below. When doing this test, the prehit and retract values should be zeroed prior to generating the .cnc file.

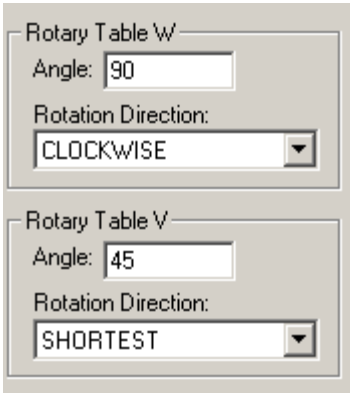


Stacked Rotary Table Program with Report

Part Programs for Stacked Rotary Tables

To create a part program that uses rotary table motion from both of the stacked rotary tables:

1. Select the Edit | Preferences | Rotary Table Setup menu option to open the Rotary Table Setup dialog box.
2. Add a Move Rotary Table command (Insert | Move | Rotary Table) to the part program.
3. Update the **Rotary Table W** and **Rotary Table V** axis positions as needed.



4. Create the CNC (.cnc) file in the usual way to include rotary table commands (see "Step 4: Create a NC Program File"). Rotary Table moves are inserted as G01 commands in the CNC file. PCDMIS will correct the coordinates of each move and measurement point.
5. Start the PC-DMIS NC Server Program (see "Step 5: Start the PC-DMIS NC Server Program").

6. Upload the program (see " Step 6: Load the NC Program File" to the controller as usual.
7. Execute the program (see "Step 7: Execute the NC Program File"). Note the points returned in the journal file are appended with the W axis angle followed by the V axis angle.

A Note on V42R Compatibility

Calibrations for two rotary tables are saved in both PC-DMIS version 4.2 (V42R) and PC-DMIS version 4.3 (V43R). However, V42R only has dual rotary table support and not stacked rotary table support.

Only one single rotary table angle is stored in a V42R part program and the same angle is used in the part program for both rotary tables. On the other hand, V43R has stacked rotary capability and saves both V and W table angles to part programs.

If a V42R program is read into V43R then the single rotary table angle copied to both V and W table angles in order that a part program intended for use dual rotary table will work regardless of the settings of the Rotary Table Setup dialog box.

If a V43R program is saved in V42R format then one of the angles will be lost. PCDMIS will inspect the rotary table setup and if dual rotary and the V table is selected then the V angle is saved, otherwise the W angle is saved. Thus, a part program written for dual rotary can be exported to V42R if required.

If a part program is written in V43R for use on dual rotary with different W and V angles and it is required to export it to V42R then it should be exported to two separate part programs:

- First, with the rotary table settings set to the W table.
- Second, with the settings set to the V table.

Working with PC-DMIS NC in Offline Mode

If you execute your part program in offline mode, the **Select Journal File** 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" for additional information.

Using the PC-DMIS CNC Server Application

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

Before you Begin: You need a copy of PC-DMIS installed, and your port lock *must* be programmed with at least one CNC License in order to use PC-DMIS CNC Server.

To start PC-DMIS CNC 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 CNC Server program appears and PC-DMIS NC runs in the background. When it launches PC-DMIS CNC 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 CNC Server

By default, the **Analyze** button is automatically selected when you launch the PC-DMIS CNC Server program. This means that the server is 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 **Hold** button. Once activated, they will show a blue glow.

PC-DMIS NCI: For PC-DMIS NCI, CNC Server only uses the **Define Machines** function is used to setup the Controller interface. All other functions are not used. Once the NC measurement parameters have been set, they need not be touched once the installation is completed. Securing NC parameters may be accomplished by removing CncServer.exe from the system.

The PC-DMIS CNC Server Menubar

File View Window Help

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".

File | Exit closes PC-DMIS CNC Server.

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

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

The PC-DMIS CNC Server Command Buttons

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

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

Analyze – This button readies the CNC Server to analyze data from an NC machine. See "Stopping and Resuming PC-DMIS CNC Server".

Download Program - This button moves your converted part program (.cnc file) to the NC controller. See "Moving Programs to the NC Controller".

Execute Journal File – This button lets you execute a journal file 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".

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" for more information.

Open PC-DMIS – This button opens PC-DMIS.

Abort – This button aborts any active measurement session for the selected (high lighted) machine. It does not stop the measuring program running on the NC Control. This must be stopped by Feed Hold or Reset controls on the machine control panel.

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 CNC Server provide information on already defined machines. These fields are described below:

ID – This field displays a unique identification number (integer between 1 and 32768) 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. Each name must also be unique.

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. Due to the nature of some machine interfaces the 'receiving data' status may be very brief as all the data is sent at the end of execution.

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 CNC Server

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

Even when stopped, PC-DMIS CNC Server continues to send data to Journal Files so that if you click **Analyze** later, data analysis will continue where it left off. You should not pause the data analysis indefinitely since no measurement results will be created possibly holding up a machine tool waiting for an offset calculation.

Exiting PC-DMIS CNC Server

Exiting the PC-DMIS CNC 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 CNC Server also closes the background instance of PC-DMIS NC that was running.

Moving Programs to the CNC Controller

For some NC controls, you can move programs to the NC controller by using the **Download Program** button. To do this:

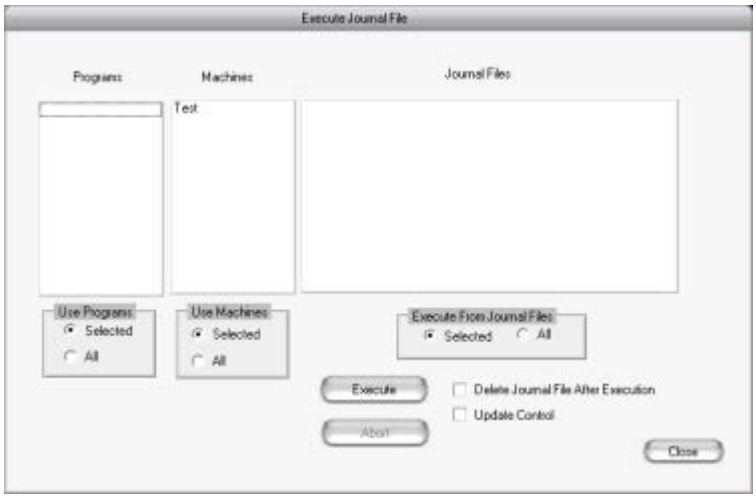
1. Ensure that the NC controller is in a mode that will accept programs. Follow your controller's documentation for information on how to do this.
2. Click the **Hold** button so that the server isn't running
3. Selecting a machine from PC-DMIS CNC Server.
4. Click the **Download Program** button. PC-DMIS CNC 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 CNC Server sends the program to the NC controller.

Executing Journal Files

PC-DMIS CNC 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 **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 CNC Server to delete the journal file after a successful analysis, select the **Delete Journal File After Execution** check box.
5. Select the **Update Control** option to allow PC-DMIS NC to update values on the control when replaying a journal file.

   Important: You may choose to not update if you have a Update Work Offset command or a tool length calculation in your program, since there might be a conflict when you replay the journal file. This can be dangerous if you are applying the update to a machine that is in the middle of cycle as the change will be loaded immediately in most controllers. This only applies to machines with an API that allows direct update (FOCAS, Siemens HMI, Heidenhain LSV2, Okuma Thinc).
6. 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.
7. Click the **Close** button to close the current dialog box and return to the main interface of PC-DMIS CNC Server.

See "Working with PC-DMIS NC in Offline Mode".

Unresolved Journal Files

If the part program on which the journal files are based is removed, renamed, or deleted, you may end up with .cna files that attempt to unsuccessfully reference the missing program. If this occurs, PC-DMIS NC will rename the journal files to have a .no_prog filename extension, and a "No Program"message, informing you that the part program could not be found, will appear in the **Status** area of the PC-DMIS CNC Server for the machine which received the journal file.

Creating and Modifying Machines

PC-DMIS CNC 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.

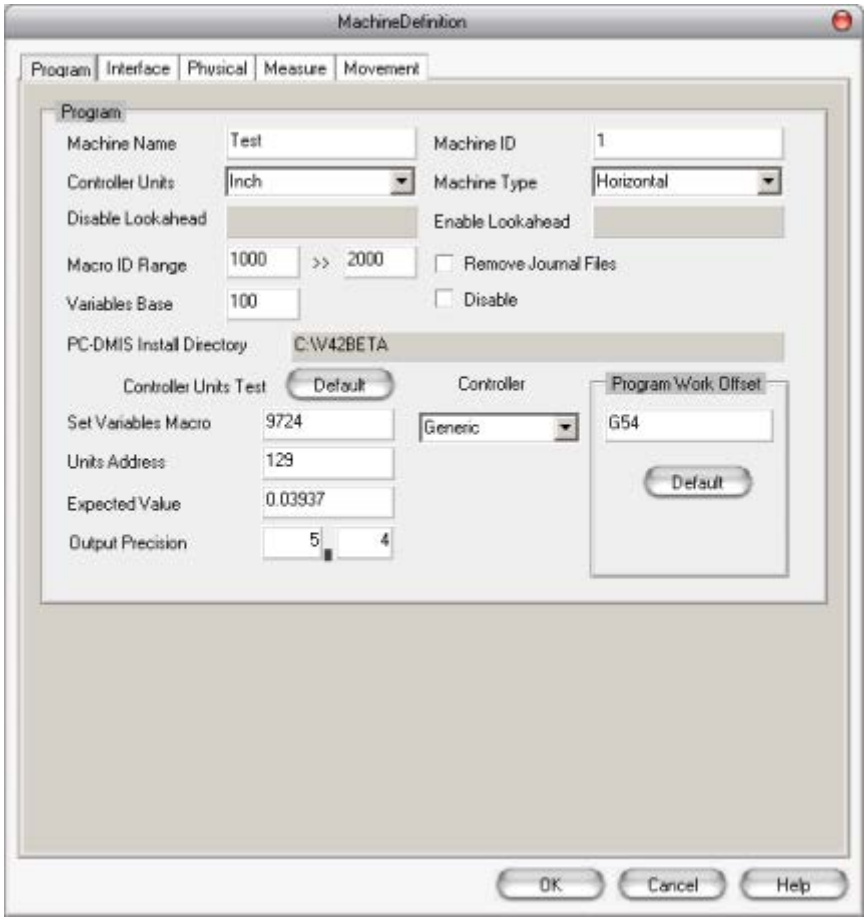
Modify – The **Modify** button modifies the selected machine from the list of defined machines by launching the **MachineDefinition** dialog box. Double-clicking on an existing machine also does this.

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 CNC Server is exited and restarted. The information is held in <pc-dmis install directory>/CncMachineData/CncSettings.dat file.

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 Names should be unique and may not contain any of the following special characters: ~, `, !, @, #, $, %, ^, &, *, (,), +, =, {, [, },], |, \, ", ', :, ;, <, >, ., ?, /, comma or space.

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

PC-DMIS NCI: This value must be "1" for PC-DMIS NCI.

Machine Type – This list allows you specify the machine style. You can select either Horizontal or Vertical. This is defined by the direction of the machine's Z axis when A,B & C axes are at 0.

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.

> PC-DMIS NCI: Use the default range of 1000-2000.

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

> PC-DMIS NCI: 100 is suggested for Fanuc controllers. About 50 variables are used for communication between PC-DMIS NCI and the Monitor Program.

Remove Journal Files – This check box allows you to determine whether or not PC-DMIS CNC 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 NCI: Disable should be unchecked to enable the interface.

PC-DMIS Install Directory – This box displays the complete pathway to the directory where the PC-DMIS/NC executable (pcdlrn.exe) is located. PC-DMIS CNC 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.

> PC-DMIS NCI: Select the Fanuc controller which support FOCAS or the Siemens 840 controller with HMI Advanced.

Set Variables Macro – This defines the Renishaw macro used to set the default variables in the controller. This value is set by default once the **Macro Address** is set on the "**Measure** tab".

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.

Output Precision – This defines the maximum number of digits that will be output to the left and the right of the decimal point. Make it as large as you can for your control to get the greatest precision in measurement. Some controls are limited to a 4.4 format or a maximum of 8 digits.

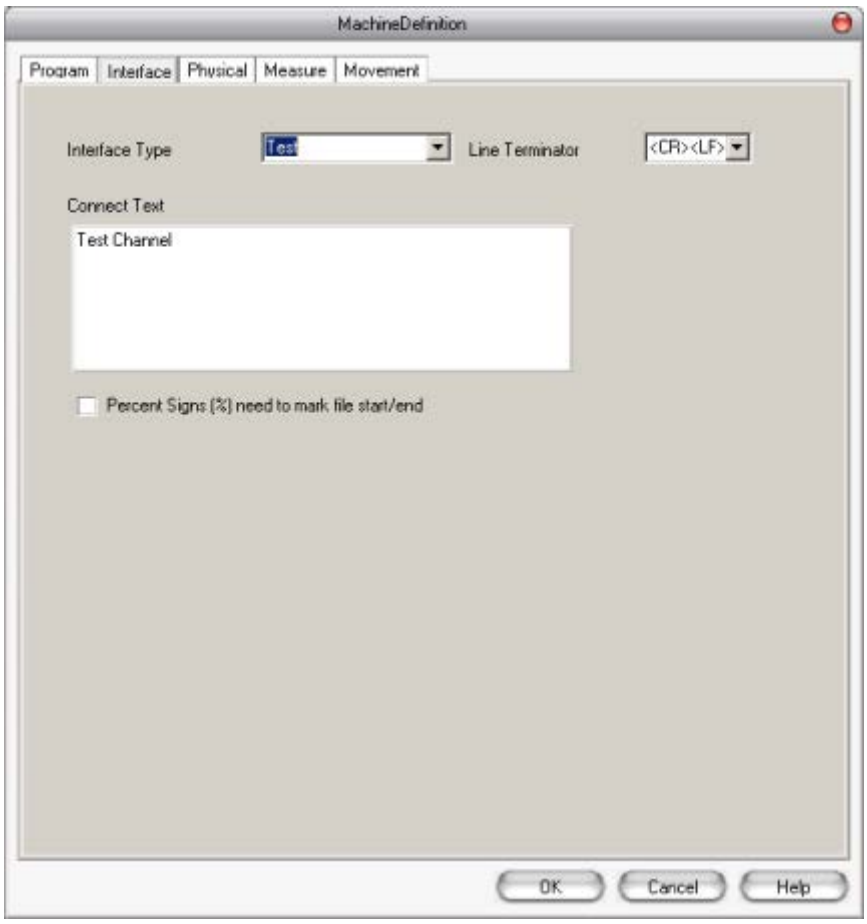
> PC-DMIS NCI: A value of 5.4 or as high as the controller is allowed is recommended.

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 are available for each controller type (example: G54 through G59 for Fanuc style controls). 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.

> PC-DMIS NCI: G54 for the default, although this is not used for NCI which ignores the active work offset to drive in machine coordinates.

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
- HSSB (Fanuc proprietary High Speed Bus)
- 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.

PC-DMIS NCI: This value is set to **Ethernet (with the FOCAS or HSSB option**) for Fanuc controllers or **Ethernet (with the HMI Advanced option)** for Siemens controllers.

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. For Ethernet interfaces, the connection string contains the IP address of the control. For HSSB interfaces this contains a string indicating the controller type. For File interfaces, this contains a directory to which some other application drops journal files for processing.

PC-DMIS NCI: Configure the **Connect Text** for ethernet (e.g. “192.168.0.1 8193”) for the same IP address and port number as defined in the control. This is found in the DDE_CONF.txt file for Siemens controllers or in the Parameter pages for Fanuc controllers..

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.

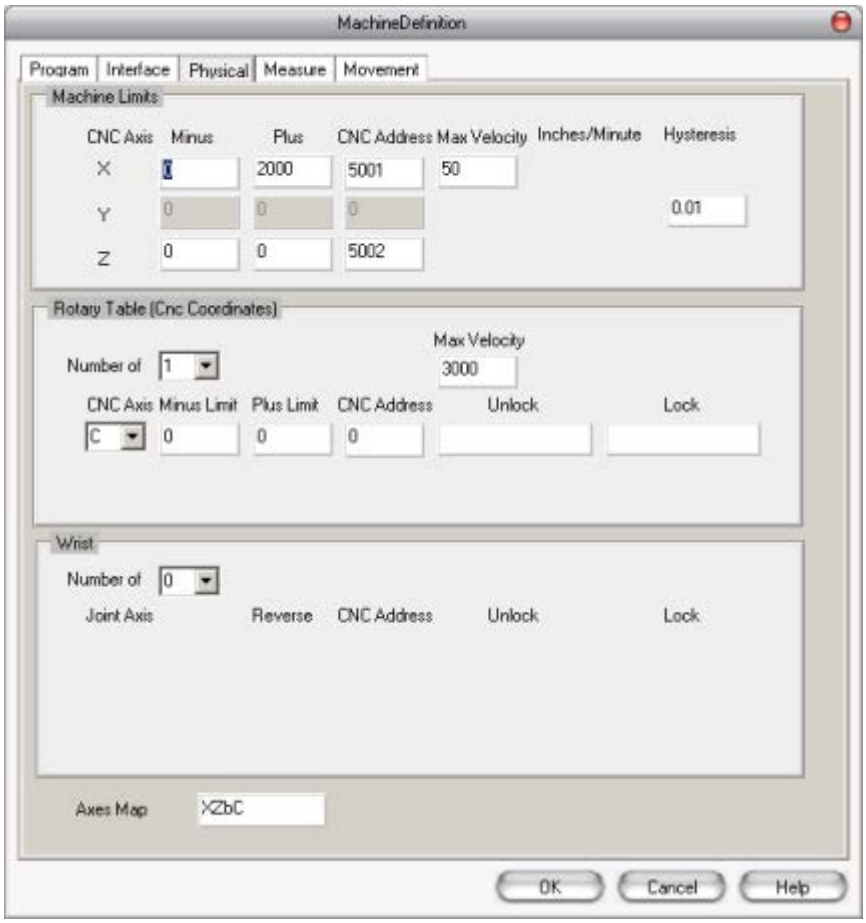
<CR><LF> - A *Carriage Return* and a *Linefeed* are appended to each line. This will cause the cursor to move to the beginning of the next line.

PC-DMIS NCI: Set to **LF**.

Percent Signs (%) need to mark file start/end - If this box is checked the NC programs generated by PC-DMIS will have a percent sign at both the beginning and the end of the file. Some systems require this when files are uploaded to the control.

PC-DMIS NCI: This should be unchecked.

MachineDefinition - Physical Tab



MachineDefinition dialog box – Physical tab

Machine Limits area:

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

Note: These values are in machine coordinates and NC programs are in work offset coordinates, Meaningful range checking can not be performed unless reasonably accurate values for the nominal work offset are supplied in the PC-DMIS command, CNC/USEWORKOFFSET.

CNC Address – These define the location of the controller register containing the limits for each axis. The exact value of the axes limits are stored here when measurement is taken.

Max Velocity – This box specifies the maximum travel velocity in machine units/Minute. The feed rate for positional moves will be the product of this value and the move speed percentage set in PC-DMIS parameter settings.

Hysteresis – The Hysteresis column has been added for lathes in V2011 MR1. An appropriate value entered in this box provides a way to prevent spindle rotation from occurring until a point deviates in the Y coordinate from zero by a user-defined amount. If the Y motion (whether in Y+ direction or Y- direction) is less than the user-defined amount then no Y motion will occur. This is an imaginary Y axis in the case of lathes which do not have a programmable Y axis and Y axis motion is obtained by programming changes in the X and C axes.

The Physical tab image above shows a Fanuc lathe (using inch units) will only rotate the spindle for points that are at least 0.01 inch away from the Y=0 value.

Rotary Table (Cnc Coordinates) area:

Number of – Specifies the number of Rotary tables.

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

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

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.

CNC Address – This box defines the location of the controller register containing the position of the rotary table. The exact value of the rotary table angle is stored here when a measurement is taken.

> PC-DMIS NC: **CNC Address** = 0 will disable this function. For Fanuc controllers, this is the address of the Absolute Position Register if a rotary table is configured.

Unlock – This box allows you to specify a command string required for the controller to unlock the rotary table. This field is left blank if no rotary table is present, or the table does not require unlocking.

Lock – This box allows you to specify a command string required for the controller to lock the rotary table. This field is left blank if no rotary table is present, or the table does not require locking.

Wrist area:

Number of Axes – The number of axes in a machine tool head or wrist

> PC-DMIS NC: Up to two joints can be configured with axis label and direction. The joints are attached to the Z axis ram. A compatible wrist definition is required in Probe.dat or UsrProbe.dat in PC-DMIS.

Joint Axis – The letter used in the NC program to move the wrist axis.

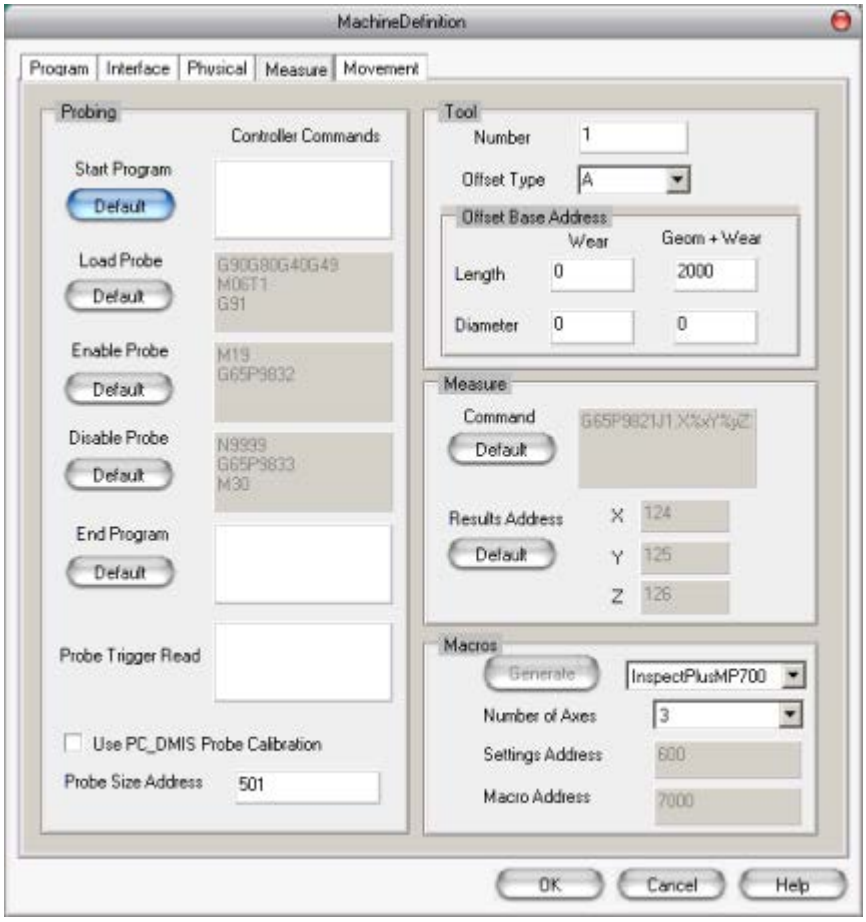
Reverse – Check to reverse the direction of motion to correlate with PC-DMIS.

CNC Address – This box defines the location of the controller register containing information for the Wrist axis. The exact value of the rotary table angle is stored here when a measurement is taken.

Unlock – This box allows you to specify a command string required for the controller to unlock the wrist axis. This field is left blank if no wrist axis is present, or the wrist does not require unlocking.

Lock – This box allows you to specify a command string required for the controller to lock the wrist axis. This field is left blank if no wrist axis is present, or the wrist does not require locking.

MachineDefinition – Measure Tab



MachineDefinition dialog box – Measure tab

Probing area:

Start Program – This box allows you to specify the machine's G/M codes to use to start the program. The **Default** button loads a default value for the associated controller commands.

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 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 probe. The **Default** button loads a default value for the associated controller commands.

End Program – This box allows you to specify the machine's G/M codes to use to end the program. 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. Currently this command is not implemented.

PC-DMIS NC: This is blank. This function is reserved for future manual operation to sense probe deflection and is the PLC address of the probe bit (Ex: X0004 7 H => Bit 7 is the trigger bit active when High).

Use PC-DMIS Probe Calibration – This selection will enable the use of Probe Calibration data from PC-DMIS. PC-DMIS must also be used to calibrate the probe. The tool offset must be set based on the probing system:

- **Renishaw Macros:** The tool offset for the probe must be set to zero. All motion will be commanded from the spindle gage point.
- **Wilcox Macros - Fanuc:** The tool offset is ignored. The motion is commanded from the spindle gage point.
- **Wilcox Macros - Siemens:** The controller's tool offset is active and should be close enough to the calibrated length to allow probe calibration. The machine is driven using the controller tool offset. The calibration values are small deltas from the controller tool offset.

PC-DMIS NCI: This value is checked for PC-DMIS NCI.

Probe Size Address - This value is the macro variable index that contains the size of the probe. This is used to report back to PC-DMIS the size of the probe as it was qualified using the NC native methods for probe calibration. It is not used when using PC-DMIS probe calibration.

PC-DMIS NCI: Defined by Wilcox Macro Selection group.

Tool area:

The **Number** box allows you to specify the 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** boxes allow you to specify the controller address for the wear component of the tool offset for the **Length** and **Diameter**.

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

PC-DMIS NCI: Use the default **Offset Base Address** values.

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 available options will change depending on the **Number of Axes** specified.

PC-DMIS NCI: For Example, the command default is G65P7007X%xY%yZ%z for a 7000 base address.

Note: In some cases (e.g., Blum macros), %i %j and %k act as place holders for the measurement approach vector, %s (or %f) is used as a placeholder for the feedhold and %p and %r represent prehit and retract distances.

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.

PC-DMIS NCI: Default value is defined from the base address from the **Program** tab (e.g. X 130 Y 131 Z 132).

Macros Area:

Macro Type – Select the needed macro type from the drop-down list. Once the macro has been selected you can provide the specific parameters for that macro:

- **InspectionPlus** - Renishaw macros for switching probes. 2.5 D.
- **InspectionPlusMP700** - Renishaw macros for Strain Gage probes. 3D.
- **Blum** - Support BLUM probes
- **Wilcox** - Supports Renishaw probes on Siemens and Fanuc style controllers

PC-DMIS NCI: The Wilcox option must be selected for PC-DMIS NCI.

Depending on the macro type selected, the various settings on the **Measure Tab** will change to reflect settings for that macro.

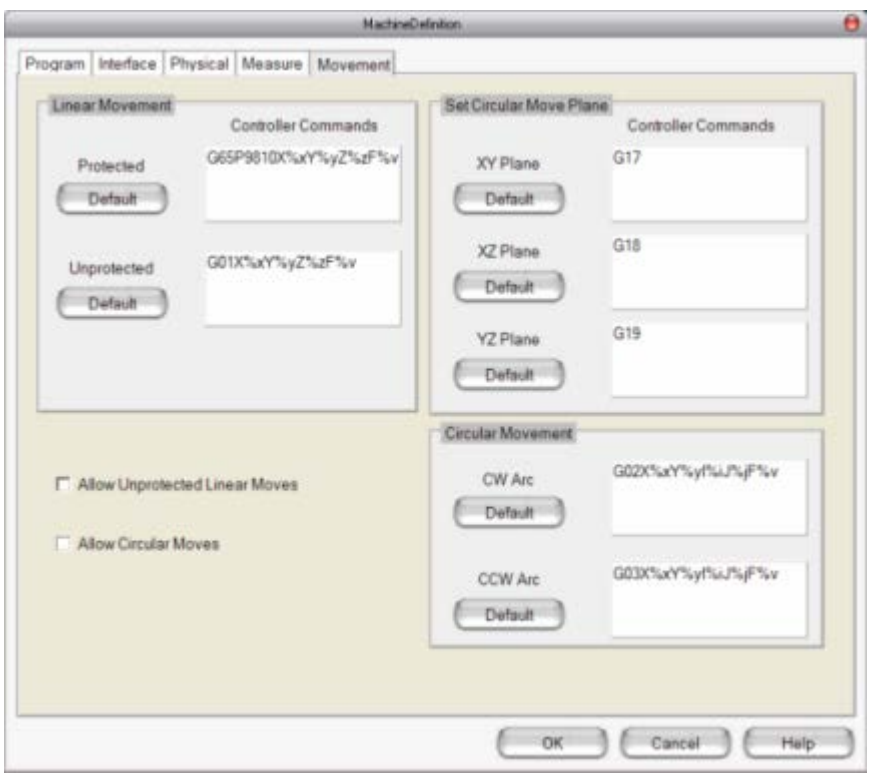
Number of Axes – This defines the number of axes used for a measurement.

3 - Measurements can be performed along any 3D vector.
2 - Measurements are performed either fully within a plane that is perpendicular to the probe axis, or along the probe axis.
1 - Measurements are performed only along one of the machine axes.

Settings Address – This value defines the non-volatile memory area in the controller. It contains about 10 variables which define probing and motion parameters (e.g. 650).

Macro Address – Used for Fanuc controls so you can use the generated probing macros without the Monitor in PC-DMIS NC. this is the starting program number for the Monitor macro programs (example 7000).

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.

PC-DMIS NCI: The values of the **Linear Movement** area set by default from the Wilcox Macro group selected from the "MachineDefinition – Measure Tab".

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.

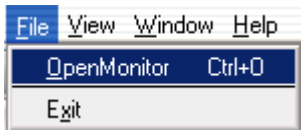
Allow Unprotected Linear Moves: Selecting this check box allows unprotected linear moves. To generate unprotected moves, use the CNC/SETPROTECTION,NO command inside the Edit window of PC-DMIS NC.

PC-DMIS NCI: It is recommended to uncheck this option for all machines. PC-DMIS NCI uses protected moves as defined in the Linear Move macro for all XYZ moves.

Allow Circular Moves: Selecting this check box allows circular movement. The normal motion approximates circular moves with straight line moves. Circular moves are unprotected, so Unprotected Linear Moves must also be checked to allow circular movement.

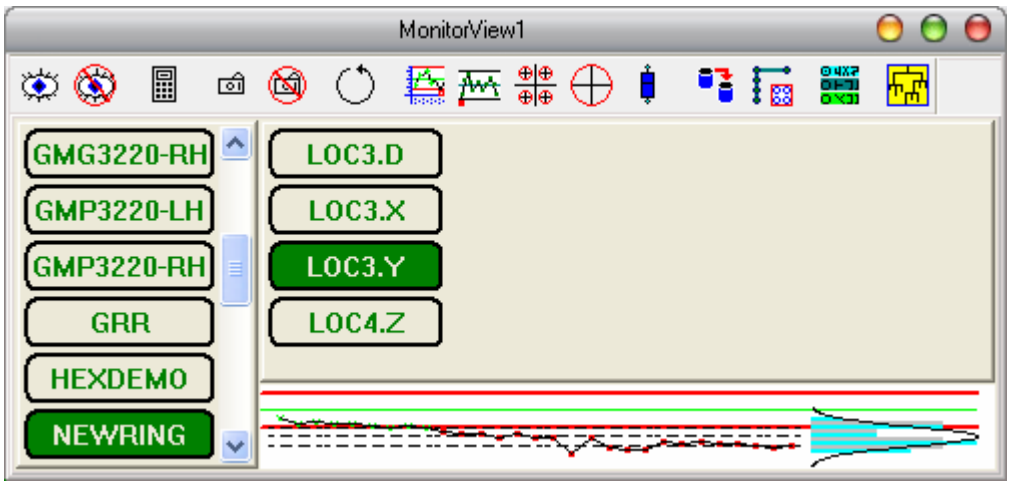
PC-DMIS NCI: Unchecked. PC-DMIS NCI does not use Circular Moves. PC-DMIS will interpolate a circular move as multiple straight line moves.

Working with the MonitorView Window



Menu Bar – Open Monitor

The MonitorView window is opened by selecting **File | OpenMonitor** menu item.



MonitorView window

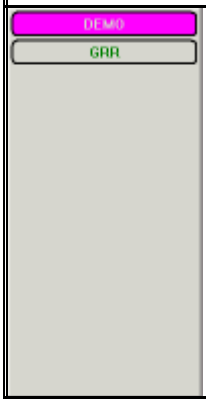
The MonitorView window is a replication of the DataPage Monitor mode and interacts with a DataPage/RT database (.dbd file). If you are familiar with the DataPage Monitor mode, you will find the functionality of the MonitorView window to be similar.

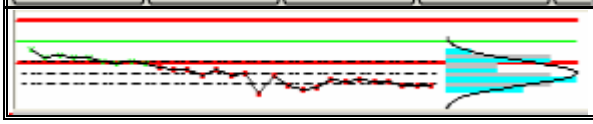
PC-DMIS NCI: The MonitorView window is not available with PC-DMIS NCI because CncServer must not be active concurrently with PC-DMIS NCI. However, the use of DataPage statistics is supported.

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
File View Window Monitor Help	*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 exactly one variable and there is *enough* data, the MonitorView window displays a chart for that variable in the bottom pane.

Sample Chart

The monitor control decides if there is 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" for more information on your statistical settings.

Acknowledging and Unacknowledging Variables

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.

Calculating the Statistical State

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 have not out of tolerance or out of control conditions 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 tolerance.

Purple - Variables that are both out of control and out of tolerance conditions.

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" for more information.

Freezing and Unfreezing 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. Control limits are calculated using the current data and saved 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, the system will recalculate control limits for the selected variables whenever new data is added 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.

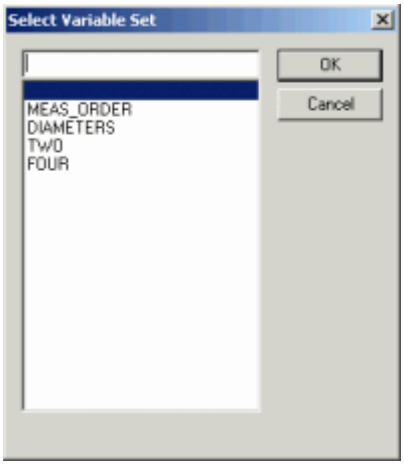
Creating a Whisker Chart – Clicking the **Whisker Chart** icon (or selecting the **Monitor | Create Whisker Chart** menu item) creates a whisker (or capability) plot for the selected features. This chart shows the mean, high, and low values, plus and minus three sigma units.

Changing a Database

To change the current database displayed in the MonitorView window, click the **Database** icon (or select the **File | Set Database** menu item). An **Open** dialog box appears allowing you to select a new DataPage database (.dbd 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

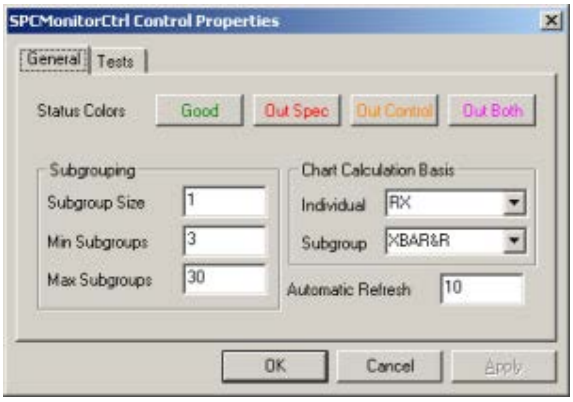
Note: You must have at least one variable set in the database for the current part to use the **Select Variable Set** icon.

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 properties for the MonitorView window.

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.

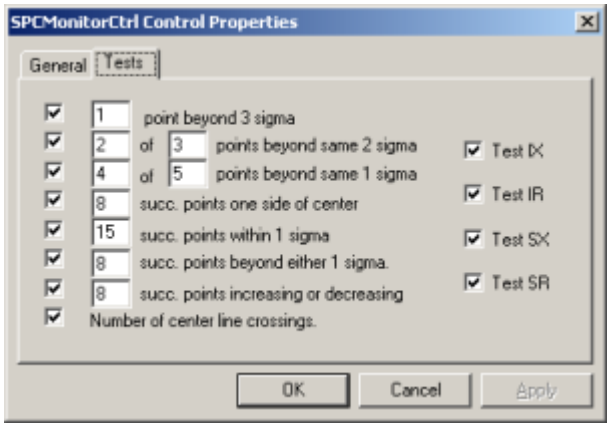
Subgrouping area – This area of the dialog box, contains these boxes Subgroup Size, Min Subgroups, and Max Subgroups. These values define how the MonitorView window groups data for calculations and charting. If subgroup size is 1, the subgroup setting is ignored.

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.

Automatic Refresh – The MonitorView window will update its data according to the number of seconds specified here.

Tests Tab



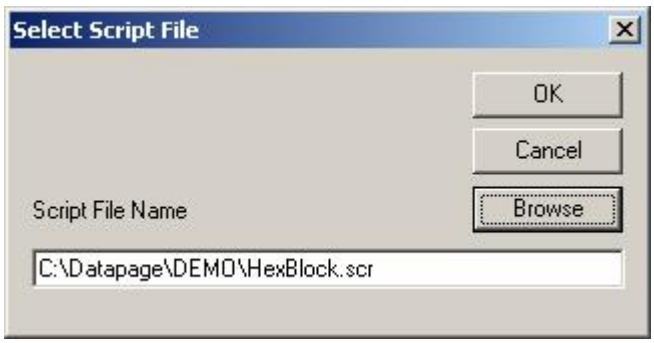
SPCMonitorCtrl Control Properties—Tests tab

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.

Run Scripts

The **Run Scripts** icon invokes a script execution feature. Scripts are text files that allow for the importing of statistical variables, assignment, logical computation, external command execution and limited flow control. Scripts can be used to react to changes of the current statistical state of the measurements.

Clicking on the **Run Scripts** icon opens the **Select Script File** dialog box.



Select Script File dialog box

Browse to the needed script file and click **OK**. This will run the selected script.

Scripting Language

This scripting language is case insensitive and the commands are terminated with semicolons (;). Thus a command can occupy more than one line to improve readability. More than one command can be implemented on a single line.

The commands in this scripting language are:

DECLARE – This command creates one or more variables that must exist before they can be used. If more than one variable is declared with a single DECLARE command then they must be separated by commas. Variables are initialized when they are created by using an equal sign. The DECLARE format is:

```
DECLARE Variable1=Value1, Variable2=Value2, Variable3=Value3, NOValueVariable
```

- Vari*able[1,2,&3]* are variables where the variable is assigned a value.
- *NOValueVariable* is a variable where the value is not initialized with a value.

an example:

```
DECLARE V1=1, V2=2, V3=3, ZERO;
```

DATABASE – This command attaches a database to the script file. By default, the database that is being monitored is used to import data. This command can be used to import data from a different database. The DATABASE format is:

```
DATABASE DatabasePath;
```

- *DatabasePath* is the full path name of the folder containing the database. If DatabasePath is omitted, the database will revert to the database being monitored. The database will be used in any subsequent IMPORTs.

an example:

```
DATABASE C:\DATAPAGE\MYDATA;
```

FILTER – This command defines a filter that will be used when data is loaded using the IMPORT command. Filters are used to define a trace field to limit imported data and to define start and end dates. By default there is no filter. The FILTER format is:

```
FILTER TraceName TraceValue StartDate EndDate;
```

- *TraceName* is the name of a trace field, for example MACHINE. Enter "" as a placeholder if trace field filtering is not being used.
- *TraceValue* is the value of a trace field, for example ROBODRILL. Enter "" as a placeholder if trace field filtering is not being used. If TraceName and TraceValue are defined, Only data that satisfies Tracename=TraceValue will be imported.
- *StartDate* defines the starting point of the data stream based on a date or count criteria. The options are:

LAST N – start at the N-th most recent measurement.

HOURS N – start at the beginning of N hours ago. If N is 0, start at the current hour. If n is 1, start at the beginning of the previous hour.

DAYS N – start at the beginning of N days ago. If N is 0, start at the current day (midnight). If N is 1, start at the beginning or yesterday.

The StartDate may be omitted, in which case no date filtering will be applied.

- *EndDate* defines the ending point of the data stream based on a date or count criteria. The options are:

LAST N – end at the N-th most recent measurement.

HOURS N – end at the beginning of N hours ago. If N is 0, start at the current hour. If n is 1, start at the beginning of the previous hour.

DAYS N – end at the beginning of N days ago. If N is 0, end at the current day (midnight). If N is 1, end at the beginning or yesterday.

The EndDate may be omitted, in which case all data after the start date will be used..

Some examples:

```
FILTER MACHINE ROBODRILL LAST 30 LAST 0; This is the last 30 measurements on the
ROBODRILL machine
FILTER MACHINE ROBODRILL; This is all data made on the ROBODRILL
FILTER "" "" DAYS 0 LAST 0; This is all measured today
FILTER "" ""; This turns off all filtering
```

IMPORT – This command imports a statistical value from the data currently being monitored. The current database and filter are used. The format is:

```
IMPORT Variable=PartName:VariableName:StatVar;
```

- *Variable* is the name of a variable created using the DECLARE command.
- *PartName* is any of the part names in the Part List of monitor mode.

- *VariableName* is any variable that appears in the Variable List of monitor mode of the specified *PartName*.
- *StatVar* is any of the supported statistical calculations. Currently the stats supported for this command are:

MEA (mean)
STD (standard deviation)
MAX (maximum value)
MIN (minimum value)
RNG (range)
NOM (nominal value)
DEV (average deviation)
NUM (number of measurements)
LST (value of last measurement)

an example:

```
IMPORT CIR1DLAST=DEMO:CIR1.D:LST;
```

IF – This command tests a logical (or arithmetic) expression. If that expression evaluates to true (or non zero), the "then" command is executed. The format of the command is:

```
IF LogicalStatement THEN Command;
```

an example:

```
IF ErrorCount > 0 THEN GOTO END;
```

See the "Logical Expressions" topic.

EXECUTE – This command will execute an executable program or batch file with optional command line arguments. The format is:

```
EXECUTE ExecuteProgramPathName [argument[s]];
```

- *ExecuteProgramPathName* is the full path name of the command to execute, including the file extension (exe or bat).
- *argument[s]* are zero or more command line arguments. If the value of a variable is to be used as a command line argument, enclose that argument in square bracers ([…]).

an example:

```
EXECUTE C:\MyStuff\SetAlarm Diameter [Diam];
```

LABEL – This command sets a branching point. It is a reference for the GOTO command. Format is:

```
LABEL LabelName;
```

an example,

```
LABEL END;
```

GOTO – This command branches to a label. The format is:

```
GOTO LabelName;
```

- *LabelName* is the name of a label (defined using the LABEL command).

an example,

```
GOTO END;
```

WRITE – This command writes information to a text file. One possible application for **Write** is to write a batch file on the fly that may be executed. The format is:

```
WRITE [R] FileName "stuff to write";
```

- *[R]* - If the **R** argument is present, then existing contents of the file are deleted before the file is written. If **R** is not specified and the file exists, the file will be opened and appended.
- *FileName* is the name of the file that will be written.
- *"stuff to write"* is what will be written. To write the value of a variable, enclose it in square brackets ([…]).

an example:

```
WRITE R C:\MyStuff\DoIt.BAT SETALARM DIAMETER [DIAM];
```

ATTACH – This command attaches a machine to the script. Machines that support direct modification of work offsets and tool offsets can be attached for this purpose. The format is:

```
ATTACH MachineID;
```

- *MachineID* is the unique numeric identifier (positive integer) of the machine to attach.

an example:

```
ATTACH 1;
```

WORKOFFSET – This command updates a work offset for the currently attached machine. The format is:

```
WORKOFFSET GCode ProgName FileName dx dy dz da db dc;
```

- *GCode* is the name of the work offset that is being set. The format of this depends on the type of machine attached.
- *ProgName* is the name of the macro that will update the work offset. Some interfaces do not require a macro name, in which case ProgName can be "".
- *FileName* is the name of the file that contains the macro program. Some interfaces communicate directly with the CNC controller. These interfaces do not use FileName, which can be set to "". Interfaces that do not communicate directly will create the command necessary to implement the offset update in this file. The user must upload this program to the controller and execute it in this case.
- *dx*, *dy* and *dz* are the changes that will be applied to the X, Y and Z components of the work offset. Note that these are delta changes to the current work offset. dx, dy and dz are required.
- *da*, *db* and *dz* are the change that will be applied to the A, B and Z components of the work offset. Note that these are delta changes to the current work offset.

an example:

```
WORKOFFSET G54 O8888 C:\UPDATE.WRK [DX] [DY] [DZ];.
```

TOOLOFFSET – This command updates a tool offset for the currently attached machine. The format is:

```
TOOLOFFSET ToolName ProgName FileName dl dd;
```

- *ToolName* is the name (number) of the tool for which the offset will be set.
- *ProgName* is the name of the macro that will update the tool offset. Some interfaces do not require a macro name, in which case ProgName can be "".
- *FileName* is the name of the file that contains the macro program. Some interfaces communicate directly with the CNC controller. These interfaces do not use FileName, which can be set to "". Interfaces that do not communicate directly will create the command necessary to implement the offset update in this file. The user must upload this program to the controller and execute it in this case.

- *dl* is the change that will be applied to length offset of the tool. Note that this is a delta change to the current tool offset. This is required, enter zero if the length is not to be modified.
- *dd* is the change that will be applied to diameter offset of the tool. Note that this is a delta change to the current tool offset. This is required, enter zero if the length is not to be modified.

an example:

```
TOOLOFFSET 6 O8889 C:\TOOL6.WRK [DL] [DD];
```

CHAIN – This command will process another script file. After the chained script file is executed, control will revert to the current file. Chained script files inherit the current database, filter and collection of variables. Changes inside a script file are not passed back to the calling script with the exception of variables that existed in the calling script. This means that their values can be modified by the chained script. The format is:

```
CHAIN ScriptPath;
```

- *ScriptPath* is the full path name of the script file that will be executed.

an example:

```
CHAIN C:\MYSUBS\WIDGETLARGE.SCR;
```

PROMPT – This command creates a standard windows message box with text supplied to the command. Yes and No buttons are included in the message box. If the user clicks the yes button, the variable will be set to 1, if the no button is pressed, the variable will be set to 0. The value if this variable can be tested to determine program flow. To include the value of a variable in the prompt, include the name of the variable in square blankets (for example, [DIAM]). The format is:

```
VAR = PROMPT PromptText;
```

- *VAR* is the name of a variable that will accept the choice of the operator.
- *PromptText* should pose a yes/no question to the user.

an example:

```
ANS = PROMPT “Okay to update Work Offsets?”;
If ANS = 0 then GOTO END;
```

CLEARLOG – This command deletes the log file associated with the script. The log file should reside in the same directory as the script file and have the same name, except the log file has the extension LOG. The format is:

```
CLEARLOG;
```

DEBUG – This command writes debug information to the log file. The format is:

```
DEBUG “stuff to write”;
```

- *“stuff to write”* is what will be written. To write the value of a variable, enclose it in square brackets ([…]).

an example:

```
DEBUG The value of DIAM is now [DIAM];
```

DUMP – This command writes the values of all variables to the log file. The format is:

```
DUMP Header;
```

- *Header* will be written to the log file prior to writing the values of all variables.

an example:

```
DUMP Contents of Variables near line 10 is:;
```

= - This symbol is used to assign the results of an arithmetic expression to a variable. Formats of expressions are defined later. The format is:

```
VariableName = expression;
```
an example:

```
AveDiam = (Diam1 + Diam2 + Diam3 + Diam4)/4;
```

Arithmetic expressions

The arithmetic expressions support any depth of parentheses, the standard arithmetic operators for addition (+), subtraction (-), multiplication (*) and division (/). In addition, exponentiation (^) is also supported. Precedence is defined as usual, but parentheses can be used to force the order of computation. Several intrinsic functions are also supported. These functions are:

SIN(x) – compute the sine of x radians.
COS(x) – compute the cosine of x radians.
TAN(x) – compute the tangent of x radians.
COT(x) – compute the cotangent of x radians.
ASIN(x) – compute the arc sine of x.
ACOS(x) – compute the arc cosine of x.
EXP(x) – compute e raised to the x power.
LOG(x) – compute the natural logarithm of x.
SQRT(x) – compute the square root of x.
ABS(x) – compute the absolute value of x.

Logical Expressions

Logical expressions are similar to arithmetic expressions, but use four additional operators. The operators are:

- > - This operator evaluates to true when the expression to the left of the operator is greater than the expression to the right.
- < - This operator evaluates to true when the expression to the right of the operator is less than the expression to the left.
- = - This operator evaluates to true when the expression to the left of the operator is equal to the expression to the right.
- **!** - This operator evaluation reverses the state of the expression to the right of the operator (from true to false or false to true).

Note: A value of zero means false, any other value is considered to be true.

Automatic Script Processing

There are two ways to automatically invoke a script when the CNC Server processes new data. If there is a file with the identical in name to the current PC-DMIS program, but has the extension scr, that script file will be invoked after the program is processed.

1. **Example 1** - the PC-DMIS program is C:\PCDMISW\BLOCK.PRG is executed and the file, C:\PCDMISW\BLOCK.SCR exists, the script file processor will be invoked to execute the file.
2. **Example 2** - If the file described in Example 1 does not exist, the file AUTOMATIC.SCR will be invoked if it exists in the PC-DMIS installation directory (for example C:\PCDMISW\AUTOMATIC.SCR).

When scripts are automatically executed you can pass two additional pieces of information to the script. One is the machine name, the other is the part name. As the string is processed in the script processor they are scanned for the words [MACHINE] and [PART]. If [MACHINE] is found, it is replaced with the machine name. If [PART] is found it is replaced with the part name. These substitutions can be used to customize script files and/or to chain specialized script files.

Sample Script with All Commands
An example of a script file follows:

```
DECLARE X1, X2, X3, X4, Y1, Y2, Y3, Y4, XOFF, YOFF, ANS;

DATABASE "C:\DATAPAGE\DEMO";
FILTER MACHINE ROBODRILL LAST 10;
ATTACH 1;

IMPORT X1 = DEMO:CIR1.X:DEV;
IMPORT X2 = DEMO:CIR2.X:DEV;
IMPORT X3 = DEMO:CIR3.X:DEV;
IMPORT X4 = DEMO:CIR4.X:DEV;
IMPORT Y1 = DEMO:CIR1.Y:DEV;
IMPORT Y2 = DEMO:CIR2.Y:DEV;
IMPORT Y3 = DEMO:CIR3.Y:DEV;
IMPORT Y4 = DEMO:CIR4.Y:DEV;

XOFF = (X1 + X2 + X3 + X4)/4;
YOFF = (Y1 + Y2 + Y3 + Y4)/4;

IF (XOFF > 0.001) THEN GOTO WORKOFFSET;
IF (YOFF > 0.001) THEN GOTO WORKOFFSET;
GOTO NOWORKOFFSET;

LABEL WORKOFFSET;

PROMPT ANS = "Apply Work Offset, X is [XOFF] and Y is [YOFF]";
IF (ANS = 0) THEN GOTO NOWORKOFFSET;

WORKOFFSET G54 "" "" XOFF YOFF 0.0;

LABEL NOWORKOFFSET;

DECLARE DIAMDEV;
IMPORT DIAMDEV = DEMO:CIR1.D:DEV;

IF (DIAMDEV < 0.001) THEN GOTO NOTOOLOFFSET;

PROMPT ANS = "Apply Tool Offset, D is [DIAMDEV]";
IF (ANS = 0) THEN GOTO NOTOOLOFFSET;

TOOLOFFSET 6 "" "" 0.0 DIAMDEV;

LABEL NOTOOLOFFSET;

WRITE R C:\TEMP\MYALARM.BAT "REM Generated by monitor script file";
WRITE R C:\TEMP\DEMO.LOG "";
```

```
DECLARE AVERAGE, LAST, NOM, INDEX, BADCOUNT = 0;

IMPORT AVERAGE = DEMO:CIR1.D:MEA;
IMPORT LAST = DEMO:CIR1.D:LST;
IMPORT NOM = DEMO:CIR1.D:NOM;
INDEX = 1;
CHAIN C:\TESTER.SCR;

IMPORT AVERAGE = DEMO:CIR2.D:MEA;
IMPORT LAST = DEMO:CIR2.D:LST;
IMPORT NOM = DEMO:CIR2.D:NOM;
INDEX = 2;
CHAIN C:\TESTER.SCR;

IMPORT AVERAGE = DEMO:CIR3.D:MEA;
IMPORT LAST = DEMO:CIR3.D:LST;
IMPORT NOM = DEMO:CIR3.D:NOM;
INDEX = 3;
CHAIN C:\TESTER.SCR;

IMPORT AVERAGE = DEMO:CIR4.D:MEA;
IMPORT LAST = DEMO:CIR4.D:LST;
IMPORT NOM = DEMO:CIR4.D:NOM;
INDEX = 4;
CHAIN C:\TESTER.SCR;

IF (BADCOUNT = 0) THEN GOTO ALLISWELL;

EXECUTE C:\TEMP\MYALARM.BAT;
EXECUTE C:\TRANSFER\COPYTOREMOTE C:\TEMP\DEMO.LOG;

LABEL ALLISWELL;
```

Chained Sample Script

The following is a sample chained script file. In the example above, this would be C:\TESTER.SCR.

```
DECLARE SHIFT = FABS(AVERAGE - LAST);
IF (SHIFT < 0.0005) THEN GOTO GOOD1;

WRITE C:\TEM\DEMO.LOG "SHIFT for CIR[INDEX].D is [SHIFT]!!
WRITE C:\TEMP\MYALARM.BAT "LOGGER 1 CIR[INDEX] [SHIFT];

LABEL GOOD1;

DECLARE LASTDEV = FABS(LAST - NOM);
IF (LASTDEV < 0.002) THEN GOTO GOOD2;

WRITE C:\TEMP\DEMO.LOG "ERROR for CIR[INDEX].D is [LASTDEV]!!
```

WRITE C:\TEMP\MYALARM.BAT "LOGGER 2 CIR[INDEX] [LASTDEV];

LABEL GOOD2;

Using PC-DMIS on a Siemens Lathe

This document is intended to describe the use of PCDMIS on a Siemens NEF400 lathe.

The PC-DMIS lathe macros additionally compute suitable spindle speed to effectively use the spindle as a C axis.

Calibrating a Siemens NEF400 Lathe

Calibration is critical to any system in order to ensure valid measurements are being taken. Without a new calibration scheme, only rough measurements are initially possible.

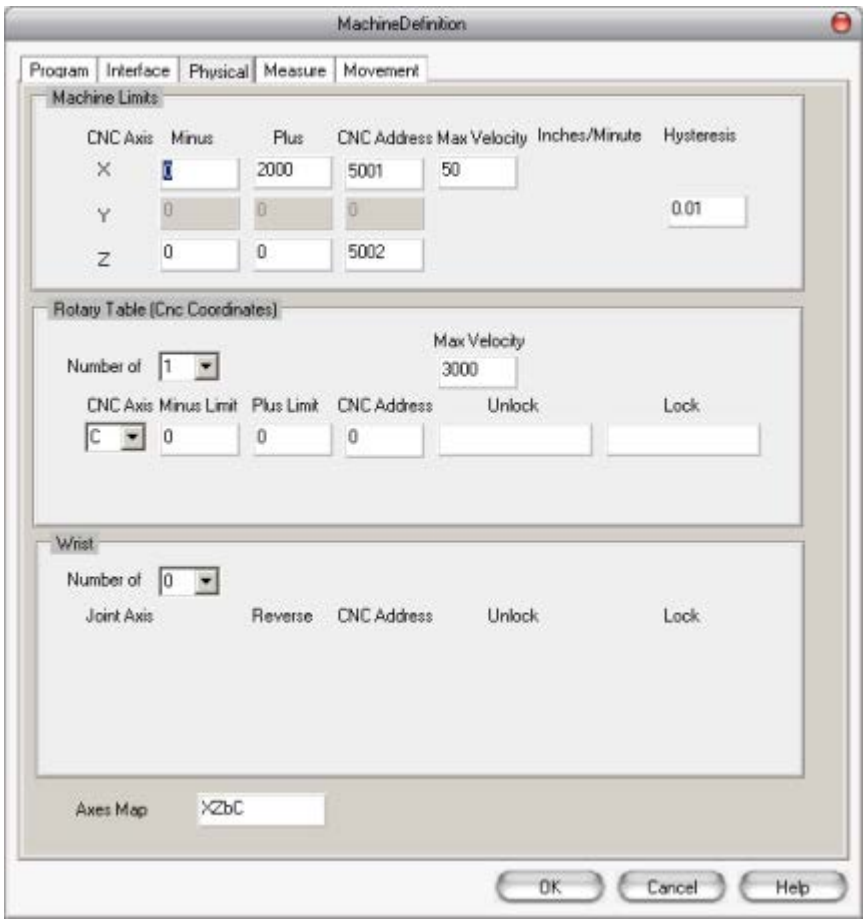
The mounting of a sphere in the chuck jaws by its stalk is feasible, but on the NEF400 movement of the probe is only possible in the XZ axes and with motion restricted to X>0 where X=0 represents the position at the center of the chuck. Consequently, in order to measure four points around the equator of a sphere, the spindle must be rotated with hits taken only in the X+ direction. This means any offset in the X direction of the probe tip from the origin is erroneously included in the computed probe tip radius.

The solution then is to ensure the hits are taken in the X+ and X- directions to cancel out the probe tip offset, therefore obtaining the correct probe tip radius. To do this, a ring or groove calibration is used instead of the standard sphere calibration.

Previously, the registry setting ***HKEY_CURRENT_USER\Software\WAI\PC-DMIS\ProbeCal\ProbeQualRingTool*** is set to **TRUE** to enable ring and groove calibration but this is no longer required.

Calibrating C Axis Lathes

These lathes have a C Axis control. That is, the macros for these lathes can rotate the spindle and can read back the current rotation angle. A C Axis is defined on the Physical tab page of the **MachineDefinition** dialog of the ***CncServer.exe*** application as shown below.



Physical tab of the MachineDefinition dialog

See the Creating and Modifying Machines topic for a more detailed description of each of the **MachineDefinition** tab pages.

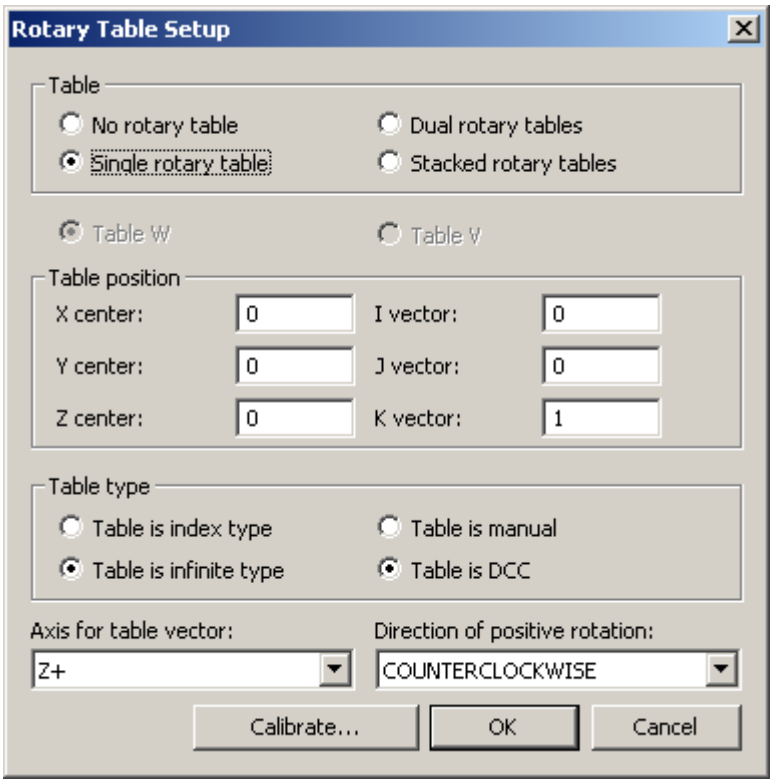
The **Hysteresis** column has been added for lathes in PC-DMIS v2011 MR1 so spindle rotation can be avoided until a point deviates in the Y coordinate from zero by a user-defined amount. The **Physical** tab on the **MachineDefinition** dialog shown above describes a Fanuc lathe using inch units will only rotate the spindle for points that are at least 0.01 inch away from Y=0.

Groove Calibration

PC-DMIS NC has been adapted by building on dialogs found in PC-DMIS Vision in order to obtain a groove calibration where both inside and outside diameters are probed. An example of a calibration groove is shown below.

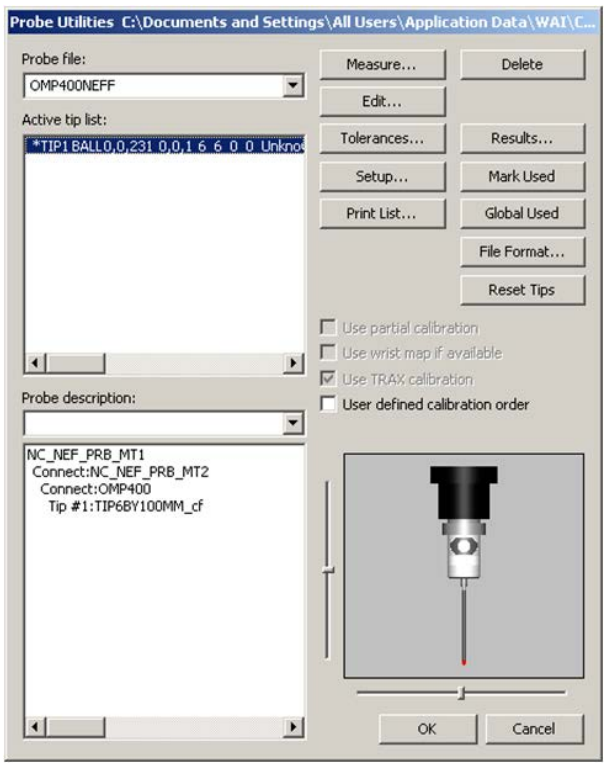
Groove Calibration Example

Select **Edit | Preferences | Rotary Table Setup** and ensure the **Single rotary table** option is selected and clear the offsets if calibrating from scratch as shown below.



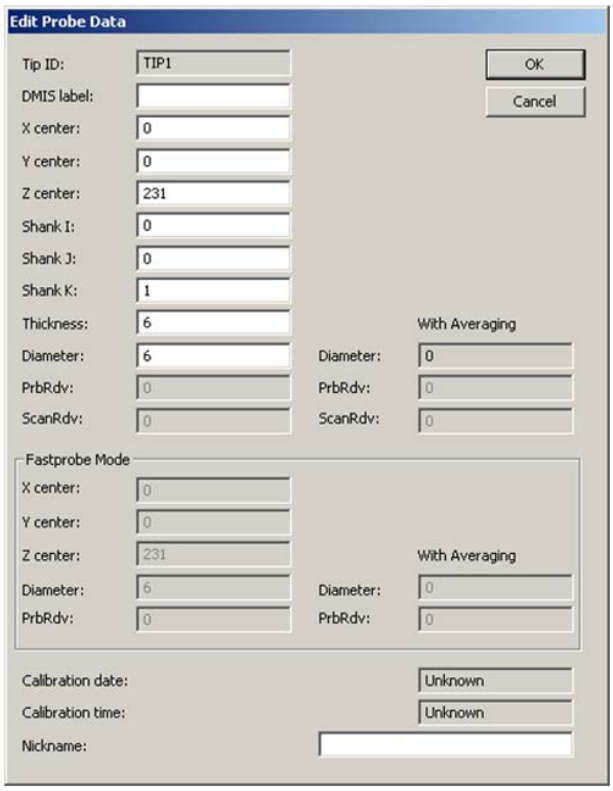
Rotary Table Setup Dialog Screen

Select or create a tip using the **Probe Utilities** dialog:



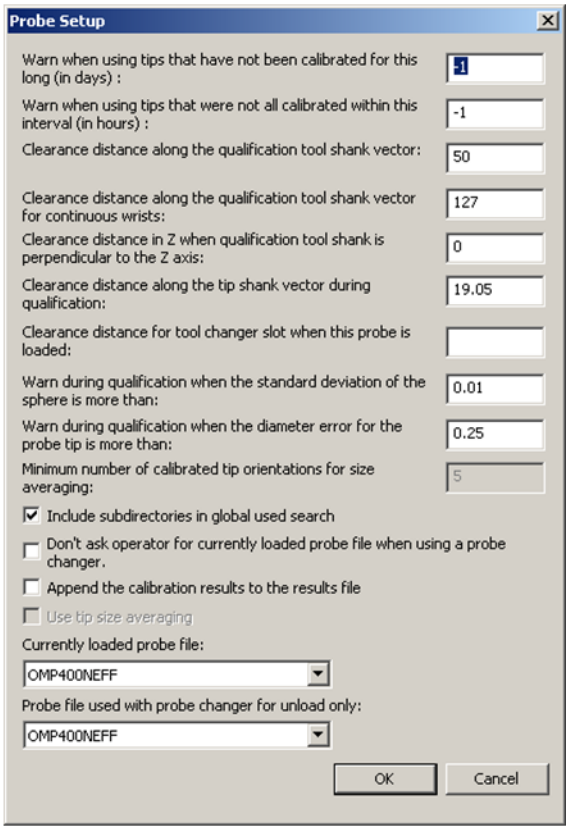
Probe Utilities Dialog Screen

Click on the **Edit** button to view or edit the probe data. Click the **OK** button to accept any changes and return to the previous screen or click the **Cancel** button to return without saving.



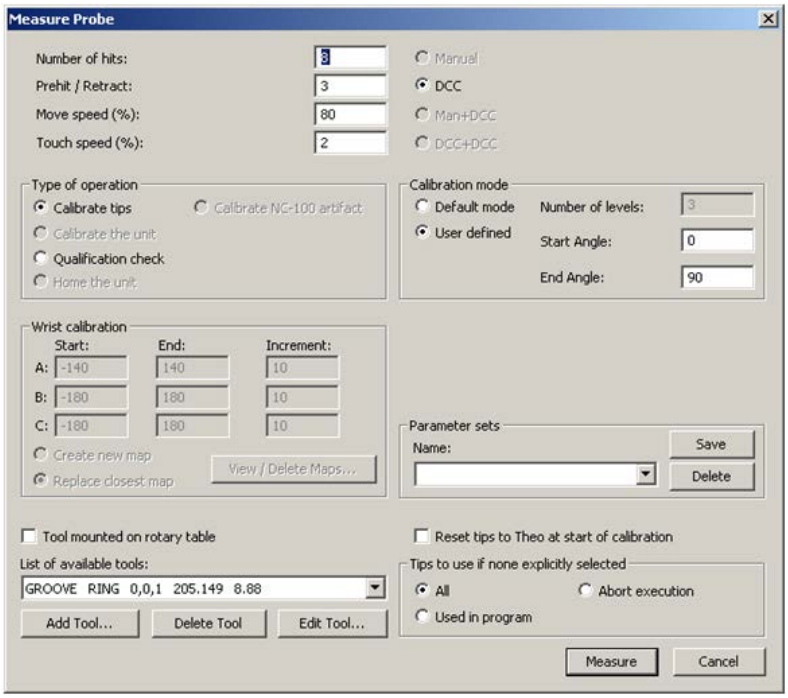
Edit Probe Data Dialog Screen

Click on the **Setup** button to view, edit and/or set the probe setup values as necessary. Click the **OK** button to accept any changes and return to the previous screen or click the **Cancel** button to return without saving.



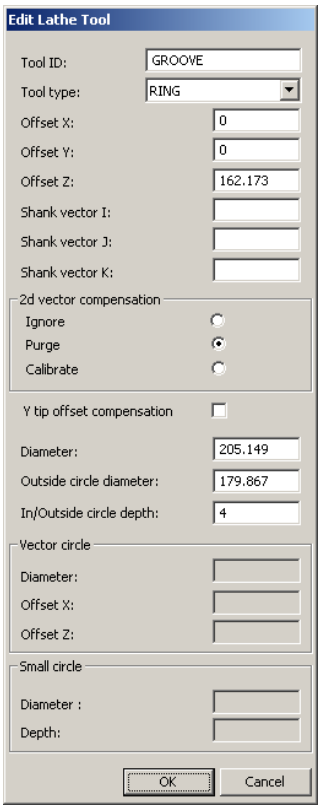
Probe Setup Dialog Screen

Click on the **Measure** button on the PC-DMIS **Probe Utilities** dialog and select a ring tool as shown below.



Measure Probe Dialog Screen

Click on the **Edit Tool...** button to update the groove parameters or to create a new tool setup. The **Edit Lathe Tool** dialog screen appears:

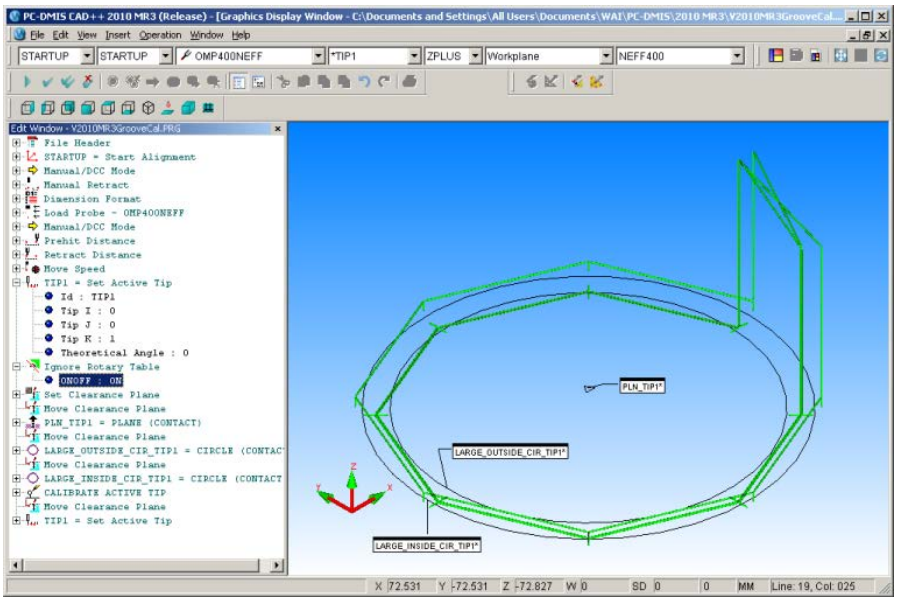


Edit Lathe Tool Dialog Screen

This will measure 4mm deep into a groove at Z=162.173mm with an outside circle of diameter 179.867mm, an inside circle of diameter 205.149mm, a plane with an offset equal to the prehit distance beyond the 205.149 circle. After completion, click the **OK** button to return to the **Measure Probe** dialog then click on the **Measure** button to create the calibration part program in the **Edit** window.

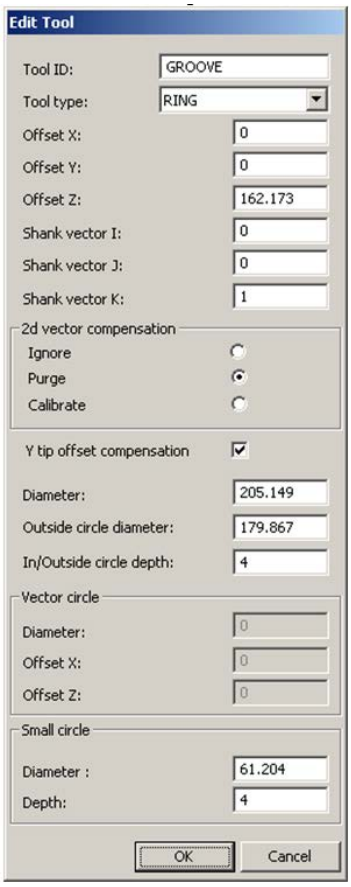
Note: Calibration on a ring instead of a groove is done by making the Outside circle diameter larger than the Inside circle diameter.

View the path lines and check the path in the Graphics Display window shown below.



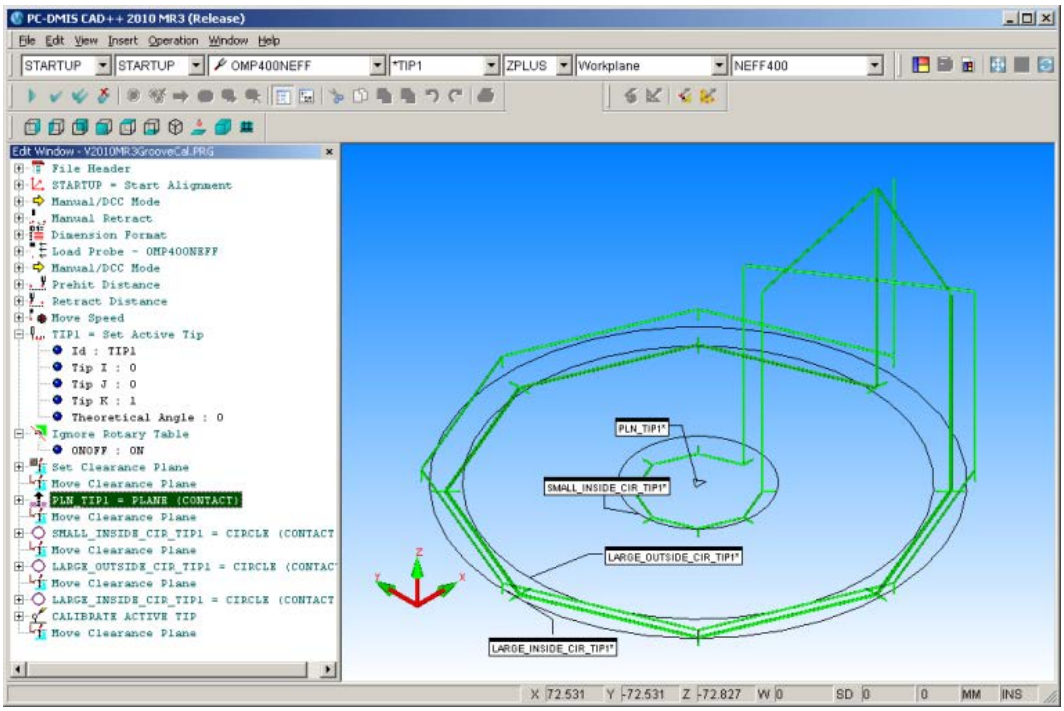
Display of Groove Calibration Path Lines

This calibration ignores any offset due to the probe tip being offset in the Y axis. It is robust in that it allows re-calibration using the current values and in practice has given similar results to the following calibration which adds a further small circle to compensate for the Y offset.



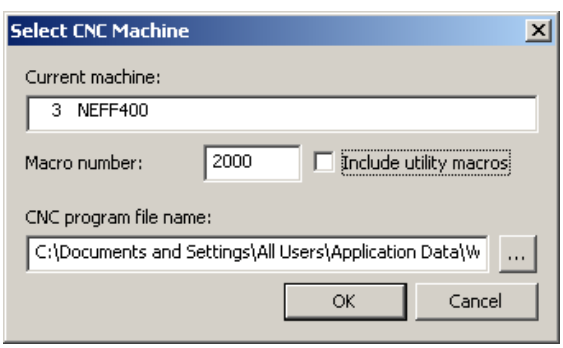
Edit Tool Dialog Screen

This will create a measure 6mm deep into a groove at Z=162.173mm with an outside circle of diameter 179.867mm, an inside circle of diameter 205.149mm, a plane with offset equal to the prehit distance beyond the 205.149 circle, and finally an optional inside circle of diameter 61.204. This latter circle is used if the probe Y offset is required. After completion, click on the **OK** button to return to the **Measure Probe** dialog. Click on the **Measure** button to create the calibration part program in the **Edit** window. View the path lines and check the path in the Graphic Display Window shown below.



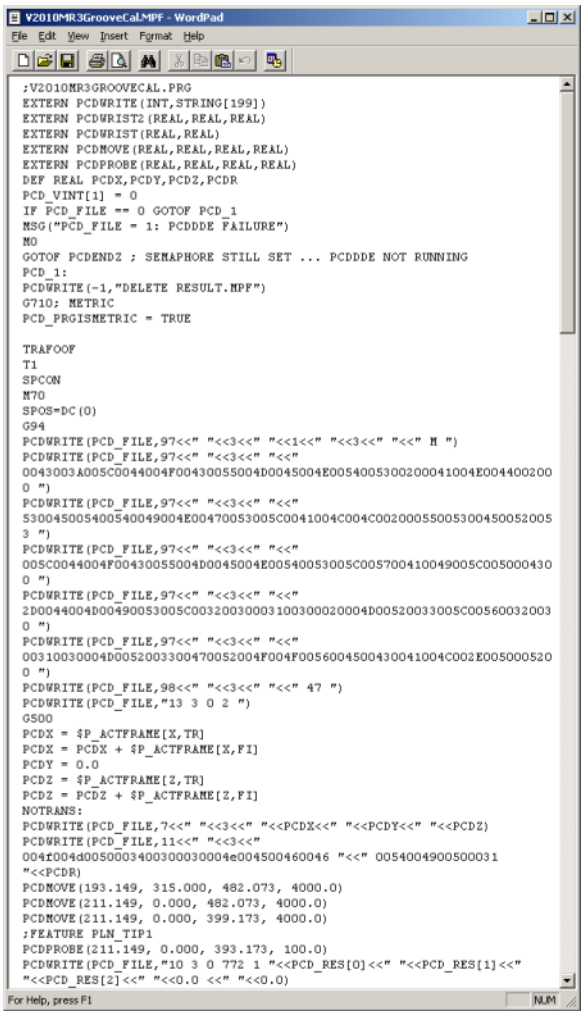
Display of Groove Calibration Screen with Optional Inside Circle

Create the CNC program by selecting Operation | CNC Programming | Create CNC Programming.



Select CNC Machine Dialog Screen

The CNC Program file will display as shown below:



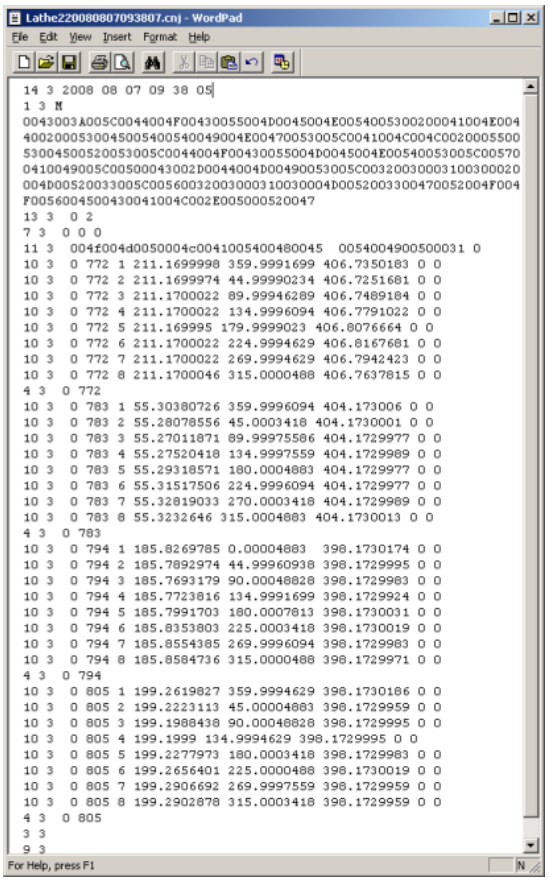
```
;V2010MR3GROOVECAL.PRG
EXTERN PCDWRITE(INT,STRING[199])
EXTERN PCDWRIST2(REAL,REAL,REAL)
EXTERN PCDWRIST(REAL,REAL)
EXTERN PCDMOVE(REAL,REAL,REAL,REAL)
EXTERN PCDPROBE(REAL,REAL,REAL,REAL)
DEF REAL PCDX,PCDY,PCDZ,PCDR
PCD_VINT[1] = 0
IF PCD_FILE == 0 GOTOF PCD_1
MSG("PCD_FILE = 1: PCDDDE FAILURE")
M0
GOTOF PCDENDZ ; SEMAPHORE STILL SET ... PCDDDE NOT RUNNING
PCD_1:
PCDWRITE(-1,"DELETE RESULT.MPF")
G710; METRIC
PCD_PRGISMETRIC = TRUE

TRAFOOF
T1
SPCON
M70
SPOS=DC(0)
G94
PCDWRITE(PCD_FILE,97<<" "<<3<<" "<<1<<" "<<3<<" "<<" M ")
PCDWRITE(PCD_FILE,97<<" "<<3<<" "<<"
0043003A005C0044004F00430055004D0045004E0054005300200041004E004400200
0 ")
PCDWRITE(PCD_FILE,97<<" "<<3<<" "<<"
5300450054005400490049004E00470053005C0041004C004C002000550053004500520055
3 ")
PCDWRITE(PCD_FILE,97<<" "<<3<<" "<<"
005C0044004F00430055004D0045004E0054005300 5C005700410049005C005000430
0 ")
PCDWRITE(PCD_FILE,97<<" "<<3<<" "<<"
2D0044004D0049005300 5C003200300031003000200 04D00520033005C00560032003
0 ")
PCDWRITE(PCD_FILE,97<<" "<<3<<" "<<"
00310030004D005200330047005 2004F004F0056004500430041004C002E005000520
0 ")
PCDWRITE(PCD_FILE,98<<" "<<3<<" "<<" 47 ")
PCDWRITE(PCD_FILE,"13 3 0 2 ")
G500
PCDX = $P_ACTFRAME[X,TR]
PCDX = PCDX + $P_ACTFRAME[X,FI]
PCDY = 0.0
PCDZ = $P_ACTFRAME[Z,TR]
PCDZ = PCDZ + $P_ACTFRAME[Z,FI]
NOTRANS:
PCDWRITE(PCD_FILE,7<<" "<<3<<" "<<PCDX<<" "<<PCDY<<" "<<PCDZ)
PCDWRITE(PCD_FILE,11<<" "<<3<<"
004f004d005000340030003000 4e004500460046 "<<" 0054004900500031
"<<PCDR)
PCDMOVE(193.149, 315.000, 482.073, 4000.0)
PCDMOVE(211.149, 0.000, 482.073, 4000.0)
PCDMOVE(211.149, 0.000, 399.173, 4000.0)
;FEATURE PLN_TIP1
PCDPROBE(211.149, 0.000, 393.173, 100.0)
PCDWRITE(PCD_FILE,"10 3 0 772 1 "<<PCD_RES[0]<<" "<<PCD_RES[1]<<"
"<<PCD_RES[2]<<" "<<0.0 <<" "<<0.0)
```

CNC Calibration Program Output File

Note: The default is now for non-circular moves inside the groove since circular motion is naturally achieved by the lathe.

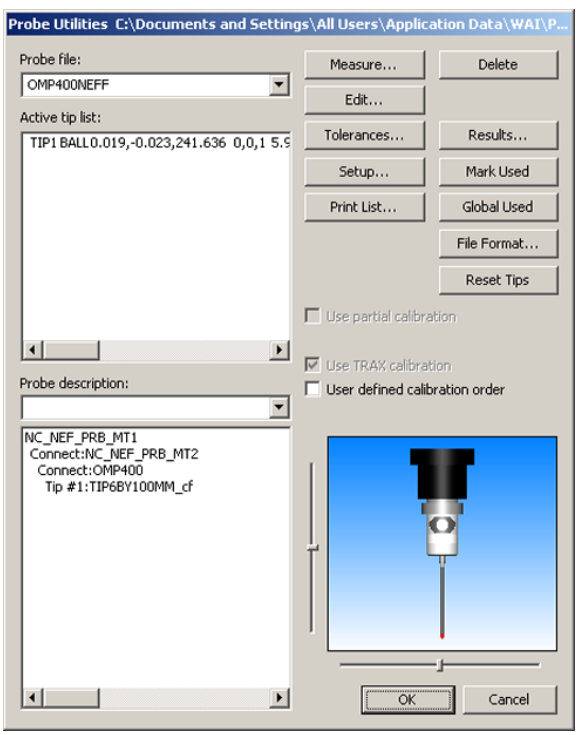
Copy the calibration program to the controller and run it to produce a Journal file as shown below.

Note: This journal file has been adapted from data collected during testing of V43B by copying over the file, machine and feature number identification from the .mpf file generated by PC-DMIS v2010.



CNC Journal Output File

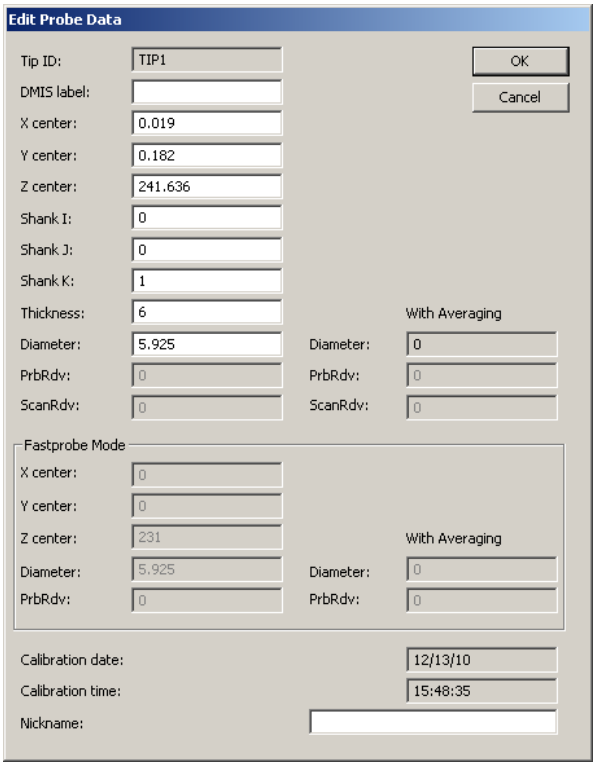
Verify that the probe calibrations have been updated by bringing up the **Probe Utilities** screen.



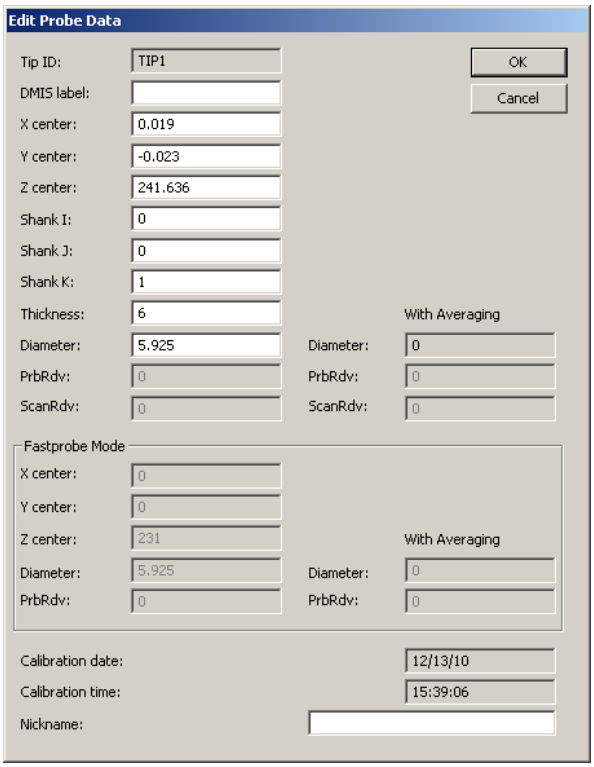
Updated Probe Utilities Dialog Screen

Click the **Edit...** button to see the updated details.

For comparison, the first image below is an example of the calibration that uses all three circles:



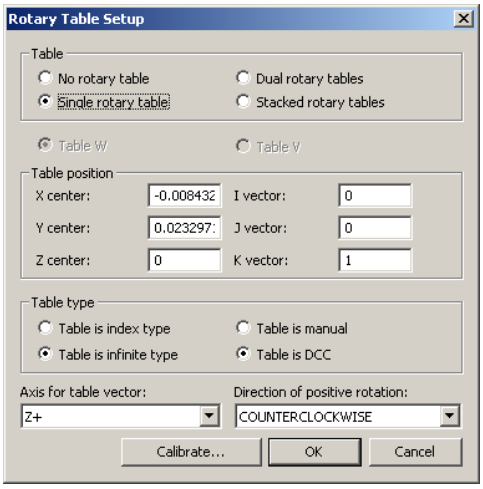
The image below is an example of the difference seen when using just the two circles.



Comparison of Edit Probe Data Dialog Screens

Note: The same Journal file data has been adapted as before to be used in both cases

Navigate to the **Rotary Table Setup** menu option from **Edit | Preferences | Rotary Table Setup** to see the updates to the probe tip offset from the rotation center:



Updated Rotary Table Setup Dialog Screen

It is instructive to dimension the calibration program in PC-DMIS to produce a report. Shown below, the deviations come in red as the nominals supplied are so far from true for this calibration artifact.

pc-dmis

PART NAME : V2010MR3GrooveCal		December 13, 2010	15:52
REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

CALIBRATE ACTIVE TIP WITH FEAT_ID=LARGE_INSIDE_CIR_TIP1

MM	LOC1 - PLN_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
Z	162.173	0.010	0.010	172.771	10.598	10.588

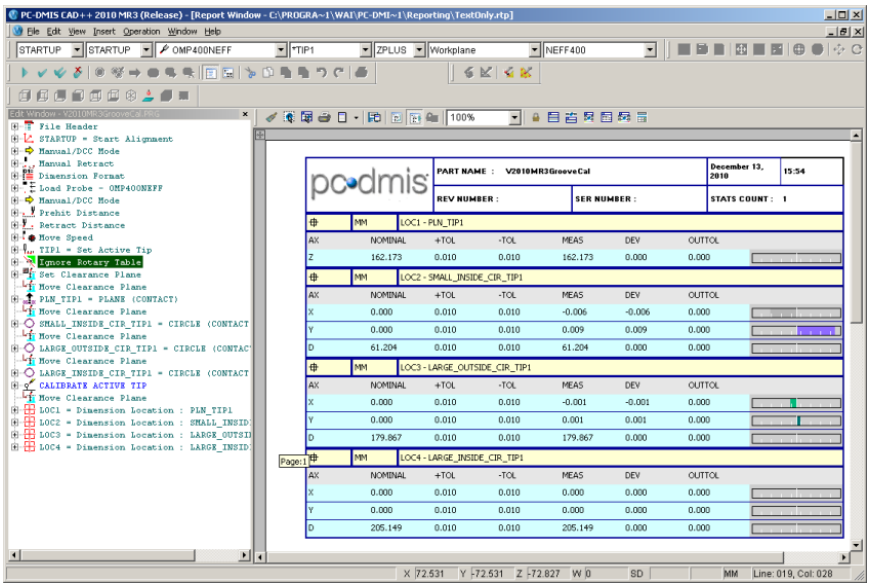
MM	LOC2 - SMALL_INSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.003	0.003	0.000
Y	0.000	0.010	0.010	-0.015	-0.015	-0.005
D	61.204	0.010	0.010	61.299	0.095	0.085

MM	LOC3 - LARGE_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.007	0.007	0.000
Y	0.000	0.010	0.010	-0.022	-0.022	-0.012
D	179.867	0.010	0.010	179.813	-0.054	-0.044

MM	LOC4 - LARGE_INSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.008	0.008	0.000
Y	0.000	0.010	0.010	-0.023	-0.023	-0.013
D	205.149	0.010	0.010	205.245	0.096	0.086

PC-DMIS Calibration Report Screen

A sanity check can be done by turning off **Marking the Probe Calibration** feature in the part program, turning off the **Ignore Rotary Table** feature and re-running the Journal file as if running a normal measurement part program to check if the results are near perfect as shown below.



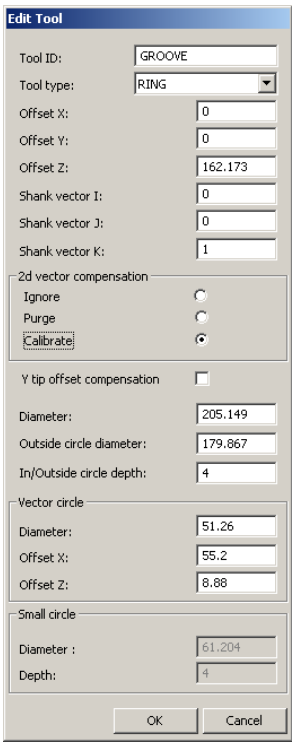
Updated PC-DMIS Report Screen

Note: In practice, no discernible improvement in measurement accuracy has so far been seen by including the small circle and thus creating a Y offset figure in the rotary table settings and bearing in mind the caveat mentioned above it is recommended this option be ignored unless absolutely needed.

Vector Calibration

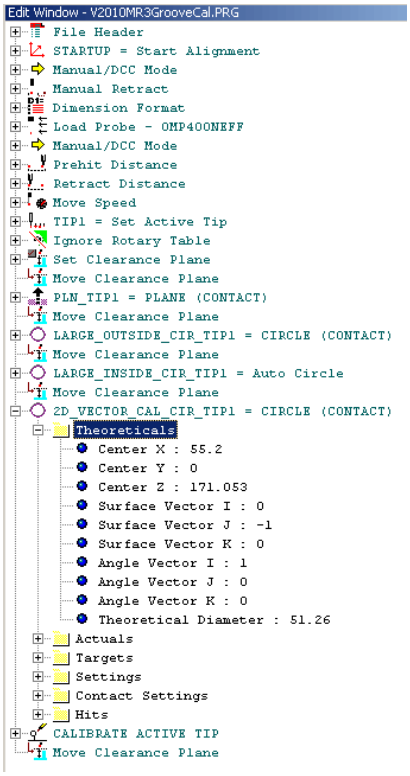
2D Vector Calibration has been added to the Groove calibration to compensate for out of roundness of the tip ball. Groove calibration only measures at three points on the tip so this may be insufficient. Vector Calibration measures a number of points around a circular profile.

As an example, to use the profile of the torus described above, add the diameter of the circle in the **Edit Tool** dialog in the highlighted position as follows and add the X axis offset of the circle center in the last edit box in the dialog.



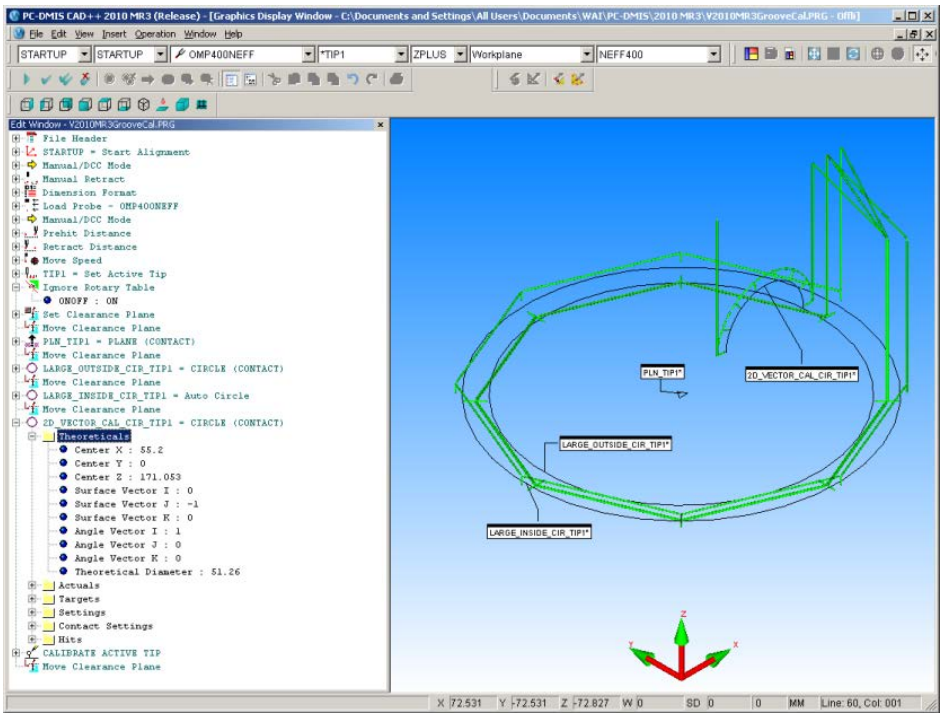
Edit Tool Dialog Screen for Vector Calibration

The generated part program is shown below.



Generated Part Program for Vector Calibration

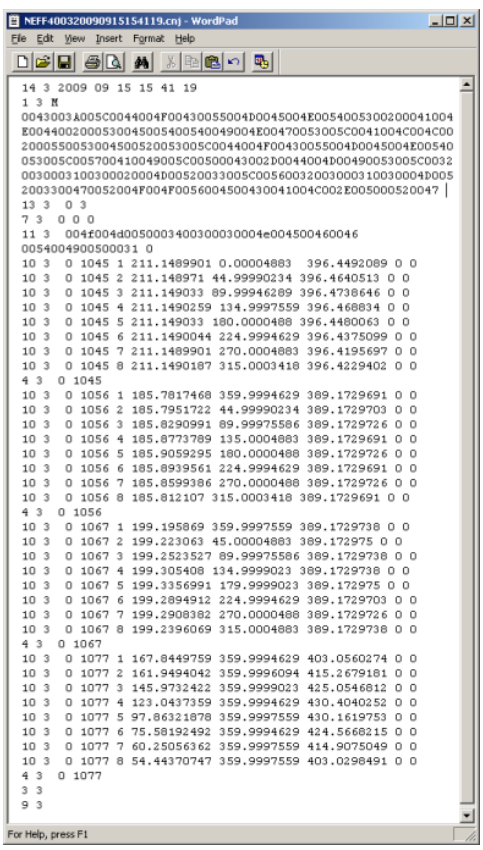
This will create a calibration program as before but with the addition of a profile measurement which is used to create a look-up-table of vector versus tip radius.



Vector Calibration Screen

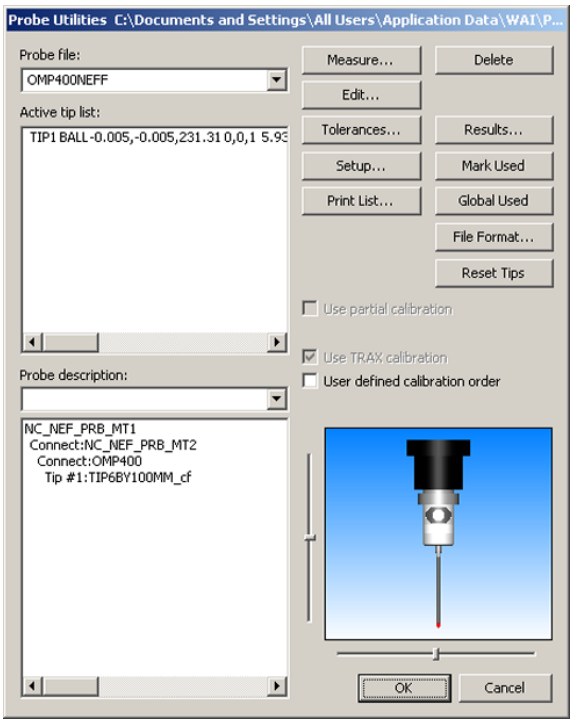
Create the program by selecting Operation | CNC Programming | Create CNC Programming.

Copy the calibration program to the controller and run it to produce a Journal file as shown below.



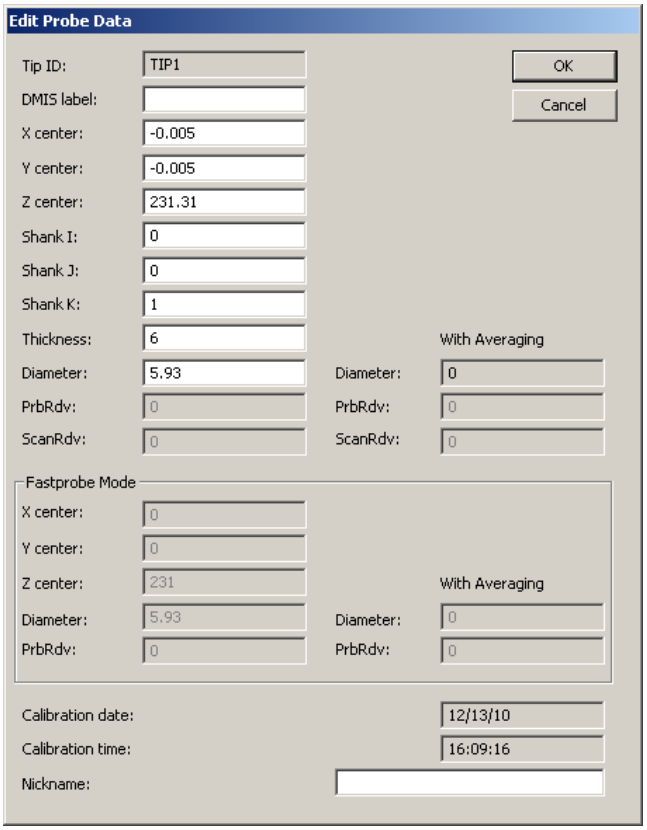
Vector Calibration Journal Output File

Verify that the probe calibrations have been updated by opening the **Probe Utilities** dialog screen.



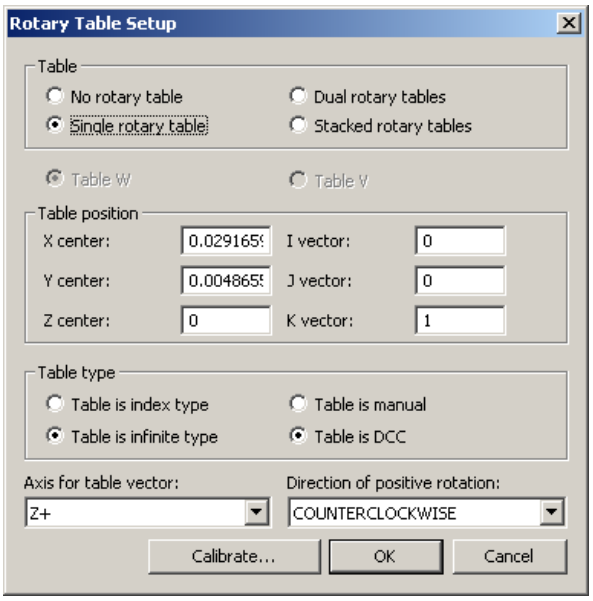
Probe Utilities Dialog Screen after Vector Calibration

Click on the **Edit...** button to view the new settings after Vector Calibration has been completed.



Edit Probe Data Dialog Screen after Vector Calibration

Navigate to the **Rotary Table Setup** dialog screen by clicking on **Edit | Preferences | Rotary Table Setup** from the main menu to see the update to the probe tip offset from the rotation center.



Rotary Table Setup Dialog Screen after Vector Calibration

If the second vector calibration has been done following the simpler groove or ring calibration, you should see no significant difference to the values in either of the above dialogs since the addition of the second vector calibration option only calibrates the profile of the probe tip.

Note: When running saved Journal files to calibrate and recalibrate with different options selected, it is important to make sure the tool dialog matches too. If for example a simple groove calibration part program (with no second vector calibration) and the corresponding Journal file is run but the tool dialog retained the values for the second vector circle, then a bad calibration will result. This scenario does not apply when creating fresh calibrations on a machine as the tool dialog will always match where a fresh calibration program is created.

It is instructive to dimension the calibration program in PC-DMIS to produce a report. In this case, we are also interested in the circularity figure for the profile measurement.

pc-dmis	PART NAME : V2010MR3GrooveCal		December 13, 2010	16:41
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

CALIBRATE ACTIVE TIP WITH FEAT_ID=2D_VECTOR_CAL_CIR_TIP1

	MM	LOC1 - PLN_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
Z	162.173	0.010	0.010	162.448	0.275	0.265	

	MM	LOC2 - LARGE_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	-0.030	-0.030	-0.020	
Y	0.000	0.010	0.010	-0.007	-0.007	0.000	
D	179.867	0.010	0.010	179.844	-0.023	-0.013	

	MM	LOC3 - LARGE_INSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	-0.029	-0.029	-0.019	
Y	0.000	0.010	0.010	-0.005	-0.005	0.000	
D	205.149	0.010	0.010	205.267	0.118	0.108	

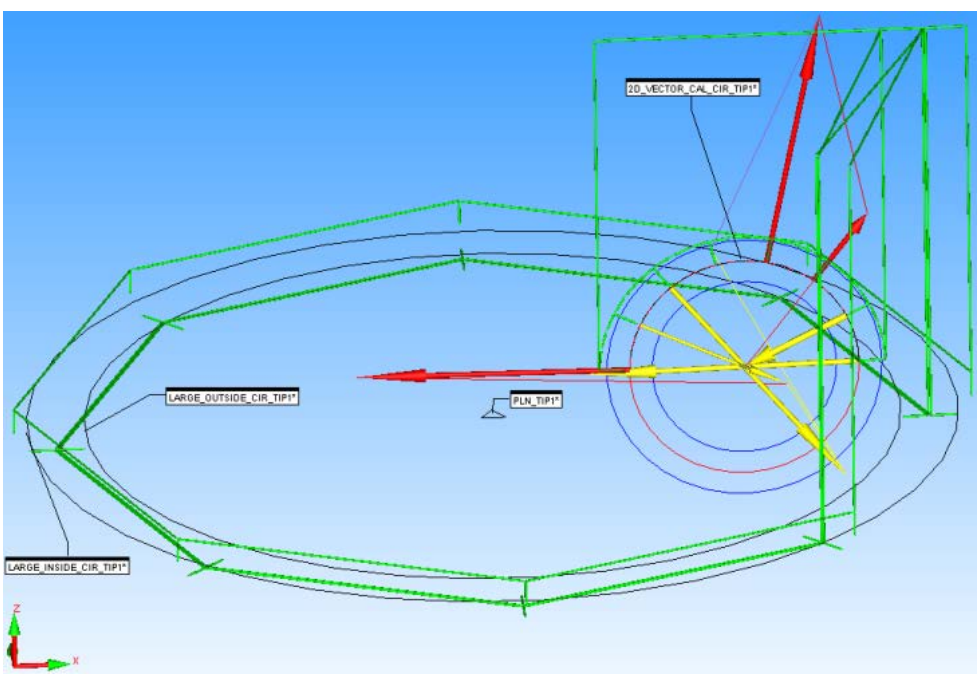
	MM	LOC4 - 2D_VECTOR_CAL_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	55.200	0.010	0.010	55.621	0.421	0.411	
Z	171.053	0.010	0.010	171.631	0.578	0.568	
D	51.260	0.010	0.010	50.672	-0.588	-0.578	

FCFCIR...	MM	○ 0.01					
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
2D_VECTO...	0.000	0.010		0.120	0.120	0.110	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
2D_VECTOR_CAL_CIR_...	1	80.924	-0.001	171.951	1.000	0.000	0.013	-0.060	
	2	78.305	-0.001	182.899	0.896	0.000	0.445	-0.025	
	3	71.151	-0.000	191.681	0.612	0.000	0.791	0.020	
	4	60.869	-0.001	196.476	0.207	0.000	0.978	0.060	
	5	49.585	-0.000	196.234	-0.238	0.000	0.971	0.004	
	6	39.626	-0.000	191.194	-0.633	0.000	0.774	-0.060	MIN
	7	32.795	-0.000	182.539	-0.902	0.000	0.431	-0.036	
	8	30.220	-0.000	171.925	-1.000	0.000	0.012	0.060	MAX

PC-DMIS Report with Dimensioned Calibration Program

Including a graphical view of the Circularity is also instructive as shown below.



Graphical View of Circularity

As before, a sanity check can be done by turning off marking the probe calibration feature in the part program, turning off the **Ignore Rotary Table** feature and re-running the Journal file as if running a normal measurement part program to check if the results are near perfect.

pc-dmis	PART NAME : V2010MR3GrooveCal		December 13, 2010	16:53
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

	MM	LOC1 - PLN_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
Z	162.173	0.010	0.010	162.170	-0.003	0.000	

	MM	LOC2 - LARGE_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	-0.001	-0.001	0.000	
Y	0.000	0.010	0.010	-0.002	-0.002	0.000	
D	179.867	0.010	0.010	179.867	0.000	0.000	

	MM	LOC3 - LARGE_INSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Y	0.000	0.010	0.010	0.000	0.000	0.000	
D	205.149	0.010	0.010	205.149	0.000	0.000	

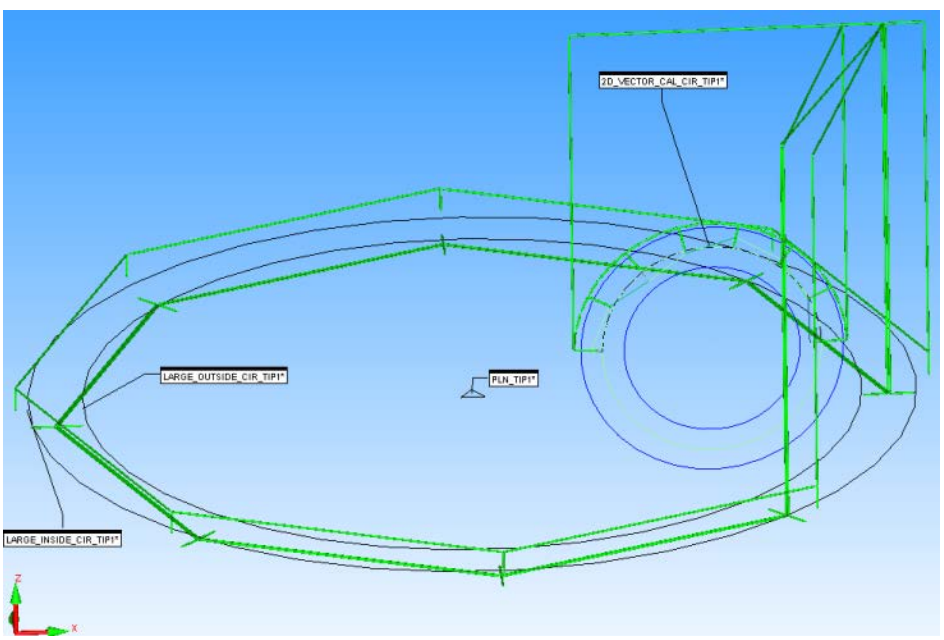
	MM	LOC4 - 2D_VECTOR_CAL_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	55.200	0.010	0.010	55.577	0.377	0.367	
Z	171.053	0.010	0.010	171.342	0.289	0.279	
D	51.260	0.010	0.010	50.747	-0.513	-0.503	

FCFCIR...	MM	○ 0.01					
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
2D_VECTO...	0.000	0.010		0.006	0.006	0.000	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
2D_VECTOR_CAL_CIR_...	1	80.947	0.004	171.642	1.000	0.000	0.012	-0.003	
	2	78.320	0.004	182.594	0.896	0.000	0.443	-0.000	
	3	71.157	0.005	191.371	0.614	0.000	0.789	0.002	
	4	60.873	0.004	196.159	0.209	0.000	0.978	0.003	
	5	49.574	0.005	195.995	-0.237	0.000	0.972	-0.001	
	6	39.540	0.005	191.002	-0.632	0.000	0.775	-0.003	MIN
	7	32.685	0.005	182.288	-0.902	0.000	0.431	0.001	
	8	30.202	0.005	171.616	-1.000	0.000	0.011	0.003	MAX

Updated PC-DMIS Report after Vector Calibration

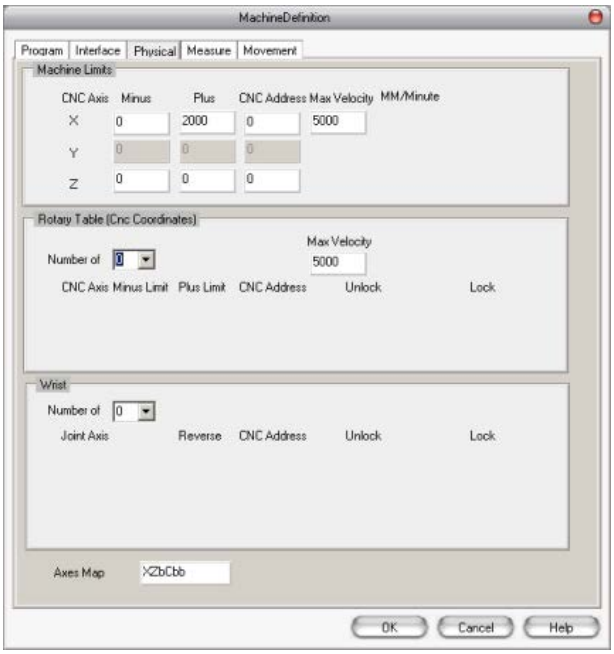
As shown below, the Circularity figure is near perfect.



Updated Graphical View after Vector Calibration

Calibrating Basic Lathes

Basic lathes have no viable C-Axis and cannot measure past the spindle center (X>0). These are defined by setting the Minus limit and the number of Rotary Axes to zero in the **Physical** tab of the **MachineDefinition** dialog box of the ***Cncserver.exe*** application as shown below. Rotary tables should not be selected within PC-DMIS.

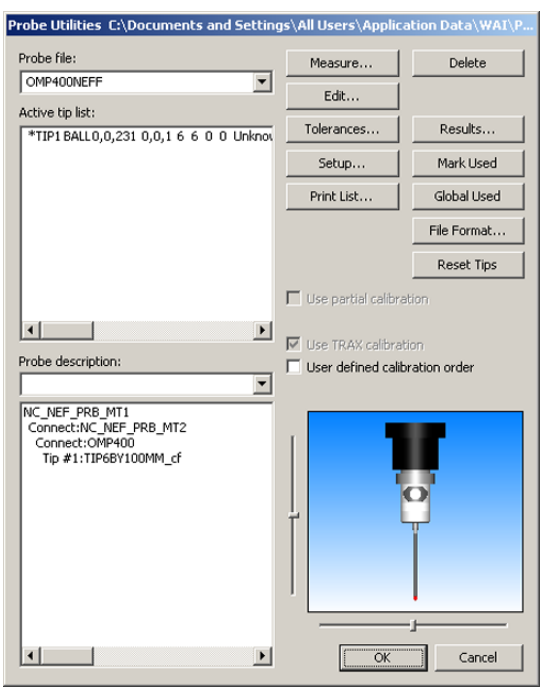


Physical Tab on MachineDefinition Dialog Screen for Basic Lathes

Off-Center Sphere Calibration

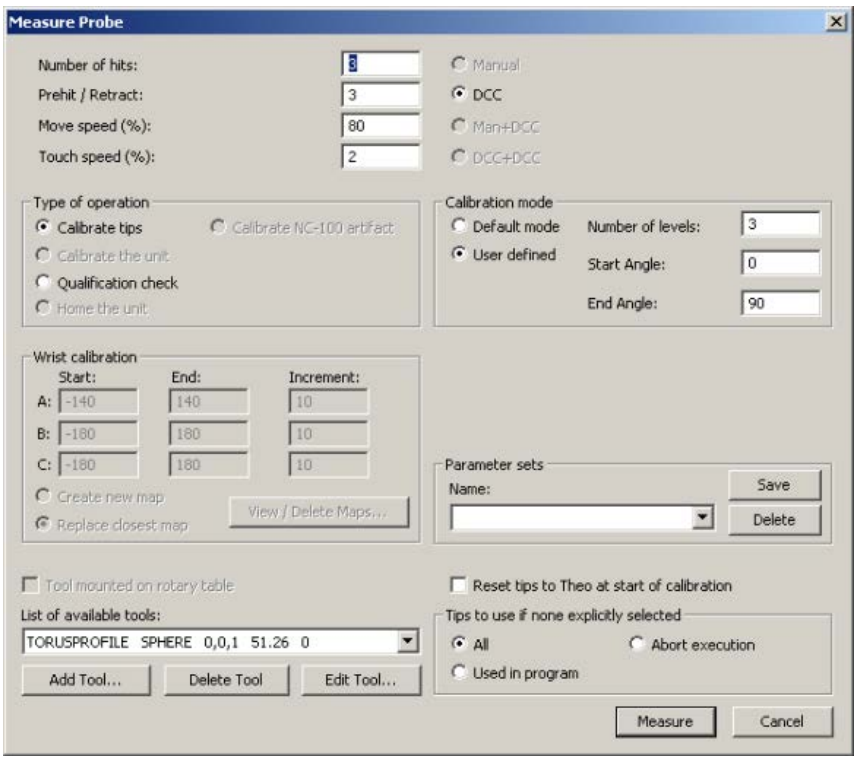
The Off-Center Sphere Calibration method records at least three hits (either side and on top) in the XZ plane.

The probe tip will be built up in the **Probe Utilities** dialog as shown below.



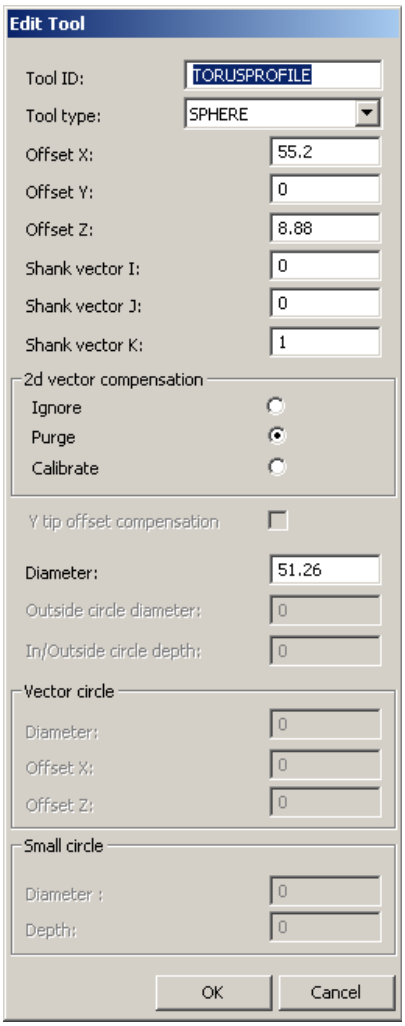
Probe Utilities Dialog Screen for Off-Center Sphere Calibration

The calibration program and number of hits will be selected from the **Measure Probe** dialog as shown below.



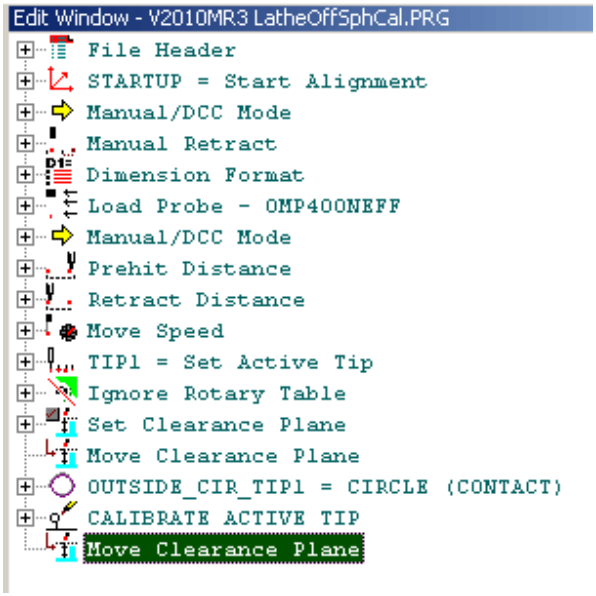
Measure Probe Dialog Screen for Off-Center Sphere Calibration

The details of the sphere calibration will be entered in the **Edit Tool** dialog screen by clicking on the **Edit Tool...** button as shown below.



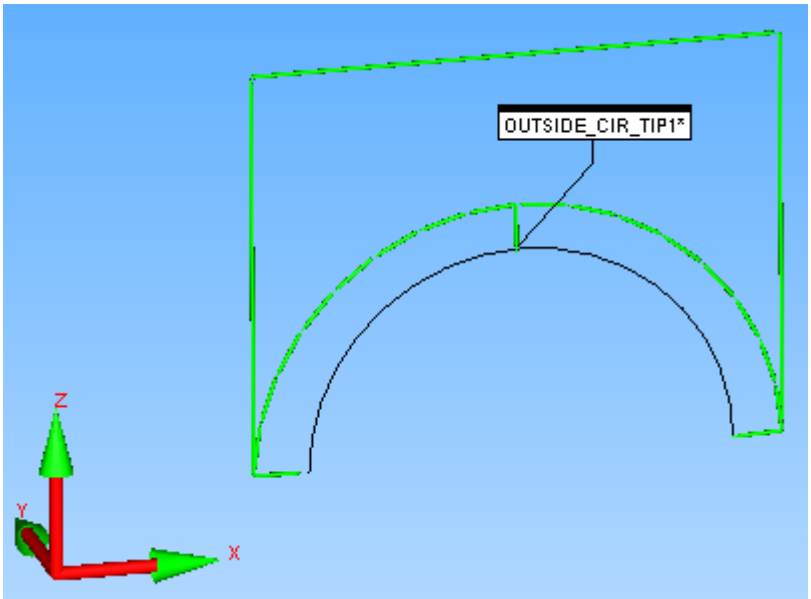
Edit Tool Dialog Screen for Off-Center Sphere Calibration

The calibration part program created will use a PC-DMIS Auto Circle feature as described below.



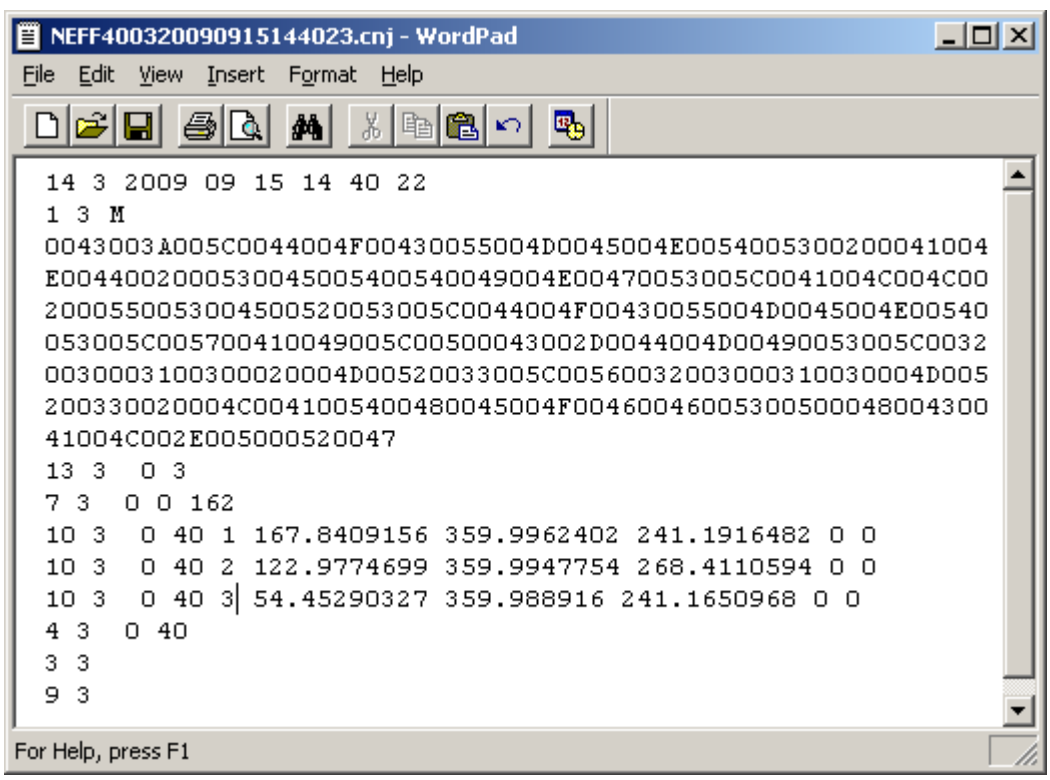
Calibration Part Program for Off-Center Sphere Calibration

PC-DMIS will display the part program graphically as follows:



Off-Center Sphere Calibration Part Program Graphical View

The Journal file for a calibration on a section of the Torus profile is shown below.



NEFF40032009091514402 3.cnj - WordPad

File Edit View Insert Format Help

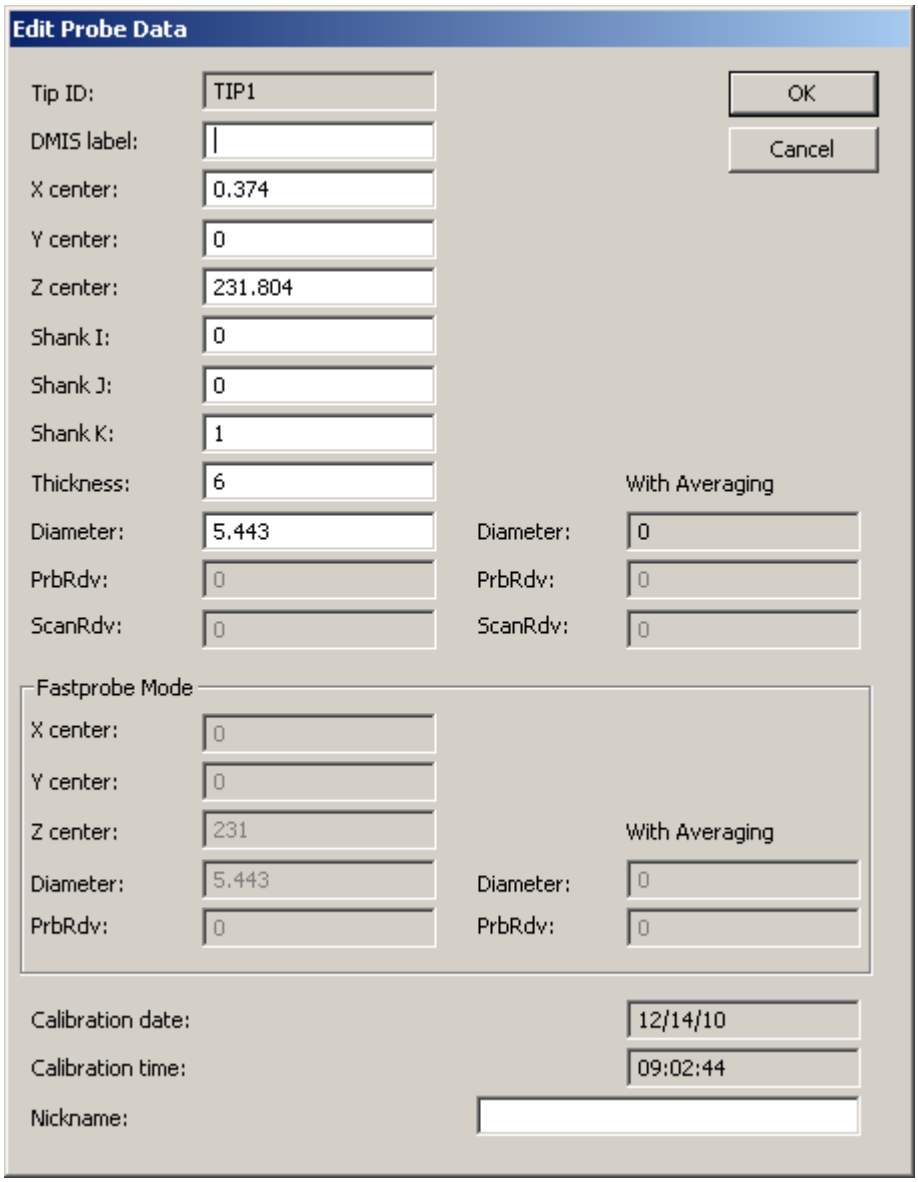
```
14 3 2009 09 15 14 40 22
1 3 M
0043003A005C0044004F0043005500 4D0045004E0054005300200041004
E0044002000530045005400540049004E0047005300 5C0041004C004C00
2000550053004500520053005C0044004F0043005500 4D0045004E00540
053005C0057004100490 05C00500043002D0044004D0049005300 5C0032
0030003100300020004D0052003300 5C0056003200300031003000 4D005
200330020004C0041005400480045004F0046004600530050000480043 00
41004C002E005000520047
13 3  0 3
7 3  0 0 162
10 3  0 40 1 167.8409156 359.9962402 241.1916482 0 0
10 3  0 40 2 122.9774699 359.9947754 268.4110594 0 0
10 3  0 40 3 54.45290327 359.988916 241.1650968 0 0
4 3  0 40
3 3
9 3
```

For Help, press F1

Off-Center Sphere Calibration Journal Output

This is used to compute and update the Probe Tip Radius, Probe Length and Probe Tip X Offset.

Select the **Edit** button from the **Probe Utilities** dialog to display the **Edit Probe Data** dialog screen shown below.

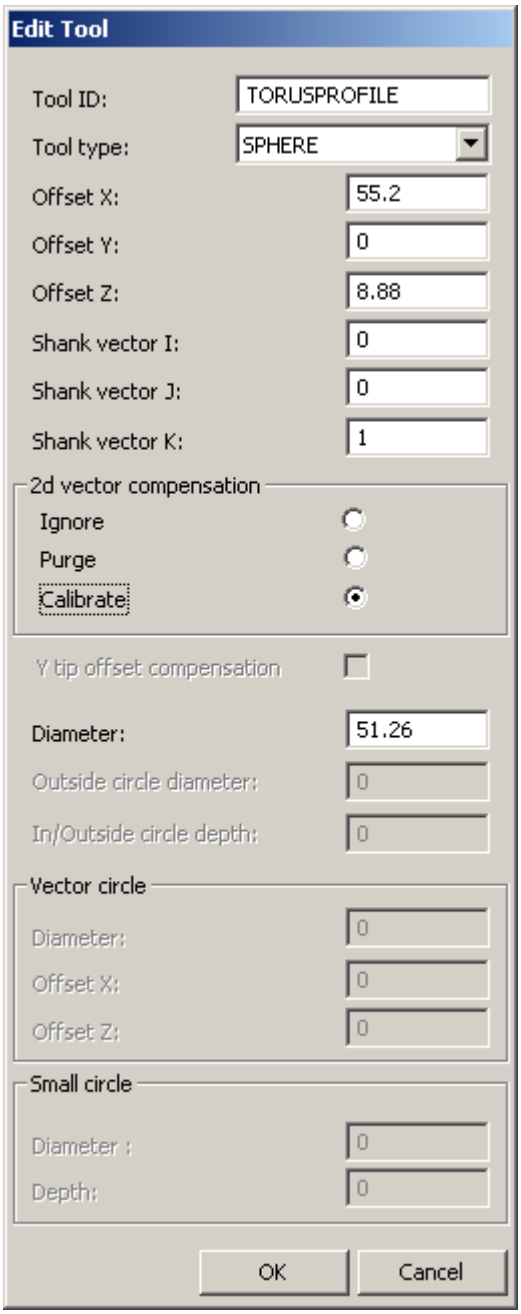


Edit Probe Dialog Screen

Offsets in the Y axis will be zeroed as it is impossible to compute Y offset information with this form of calibration. The sphere and probe tip must be mounted accurately in the XZ plane - no spindle rotation is needed.

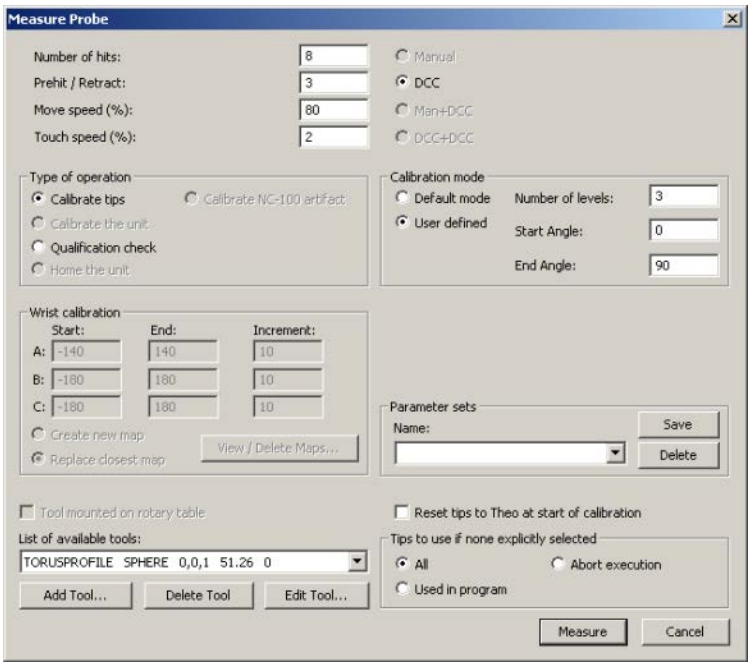
2D Vector Calibration

The 2D Vector Calibration measures the profile of a semi-circle in the XZ plane and generates a look-up table to compensate for lobing effect. Selection of eight or more hits and selection of **2D Vector Calibration** in the **Edit Tool** dialog screen will invoke the Vector Calibration.



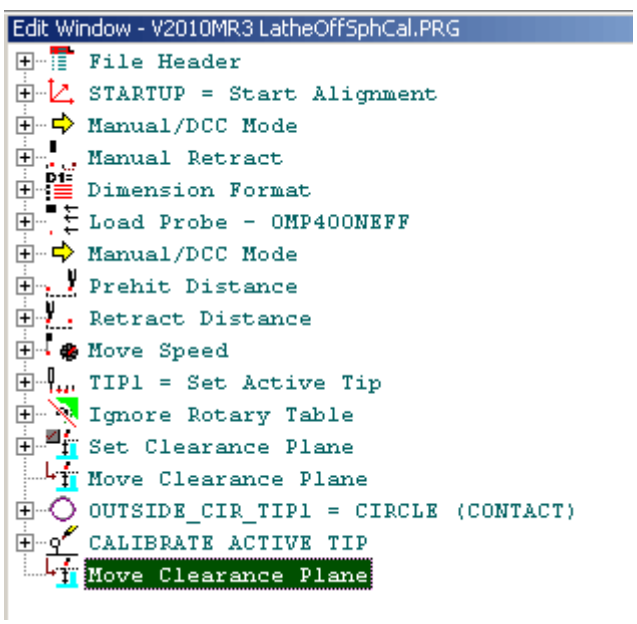
Edit Tool for 2D Vector Calibration

The **Measure Probe** dialog screen is shown below for reference.



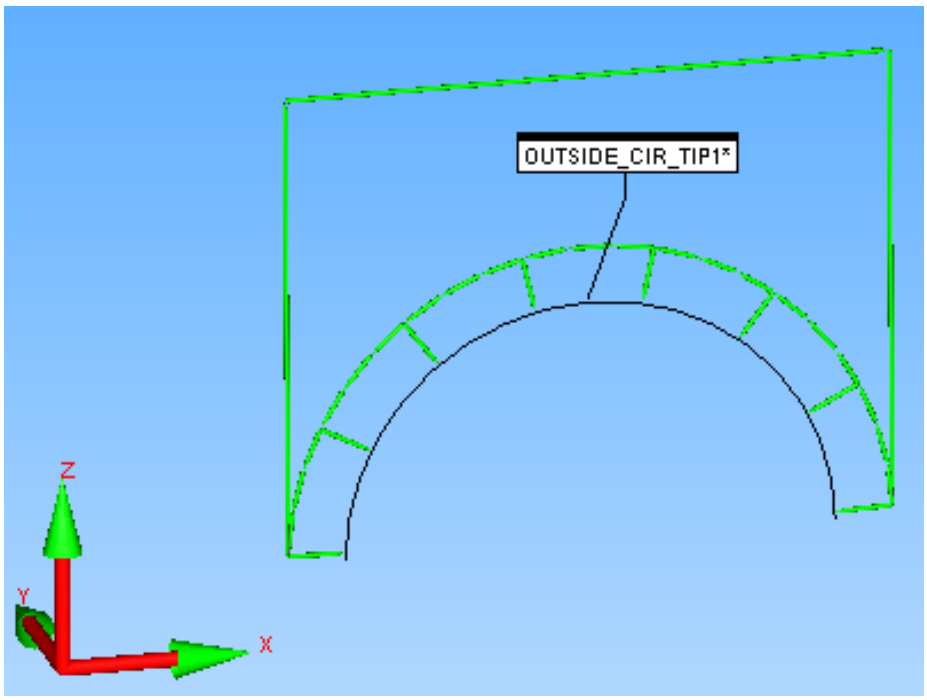
Measure Probe Dialog Screen for 2D Vector Calibration

The Sphere Calibration for the 2D Vector Calibration part program appears unchanged from the Off-Center Sphere Calibration method.



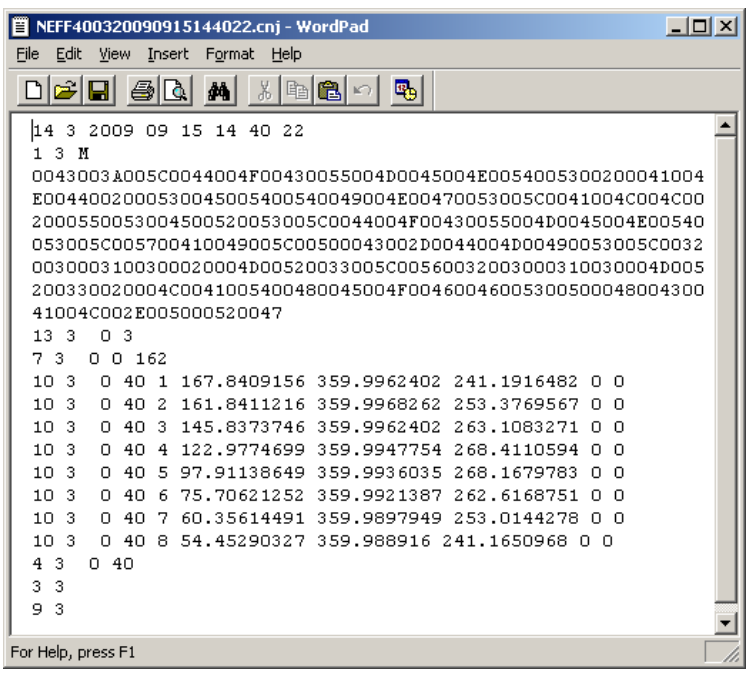
Part Program for 2D Vector Calibration

The Graphical View shows the eight hit points as programmed:



2D Vector Calibration Part Program Graphical View

The Journal file is shown below:



NEFF40032009091514402 2.cnj - WordPad

File Edit View Insert Format Help

```
14 3 2009 09 15 14 40 22
1 3 M
0043003A005C0044004F00430055004D0045004E005400530020004100 4
E004400200053004500540054004900 4E00470053005C0041004C004C00
200055005300450052005300 5C0044004F00430055004D0045004E00540
053005C0057004100490 05C00500043002D0044004D00490053005C0032
0030003100300020004D00520033005C00560032003000310030004D005
200330020004C0041005400480045004F00460046005300500048004300
41004C002E005000520047
13 3  0 3
7 3  0 0 162
10 3  0 40 1 167.8409156 359.9962402 241.1916482 0 0
10 3  0 40 2 161.8411216 359.9968262 253.3769567 0 0
10 3  0 40 3 145.8373746 359.9962402 263.1083271 0 0
10 3  0 40 4 122.9774699 359.9947754 268.4110594 0 0
10 3  0 40 5 97.91138649 359.9936035 268.1679783 0 0
10 3  0 40 6 75.70621252 359.9921387 262.6168751 0 0
10 3  0 40 7 60.35614491 359.9897949 253.0144278 0 0
10 3  0 40 8 54.45290327 359.988916 241.1650968 0 0
4 3  0 40
3 3
9 3
```

For Help, press F1

2D Vector Calibration Journal File

The Report is shown below.

pc-dmis	PART NAME : V2010MR3 LatheOffSphCal	December 14, 2010	09:33
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1

CALIBRATE ACTIVE TIP WITH FEAT_ID=OUTSIDE_CIR_TIP1

⌖	MM	LOC1 - OUTSIDE_CIR_TIP1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	55.200	0.050	0.050	55.622	0.422	0.372
Z	8.880	0.050	0.050	9.632	0.752	0.702
D	51.260	0.050	0.050	50.670	-0.590	-0.540

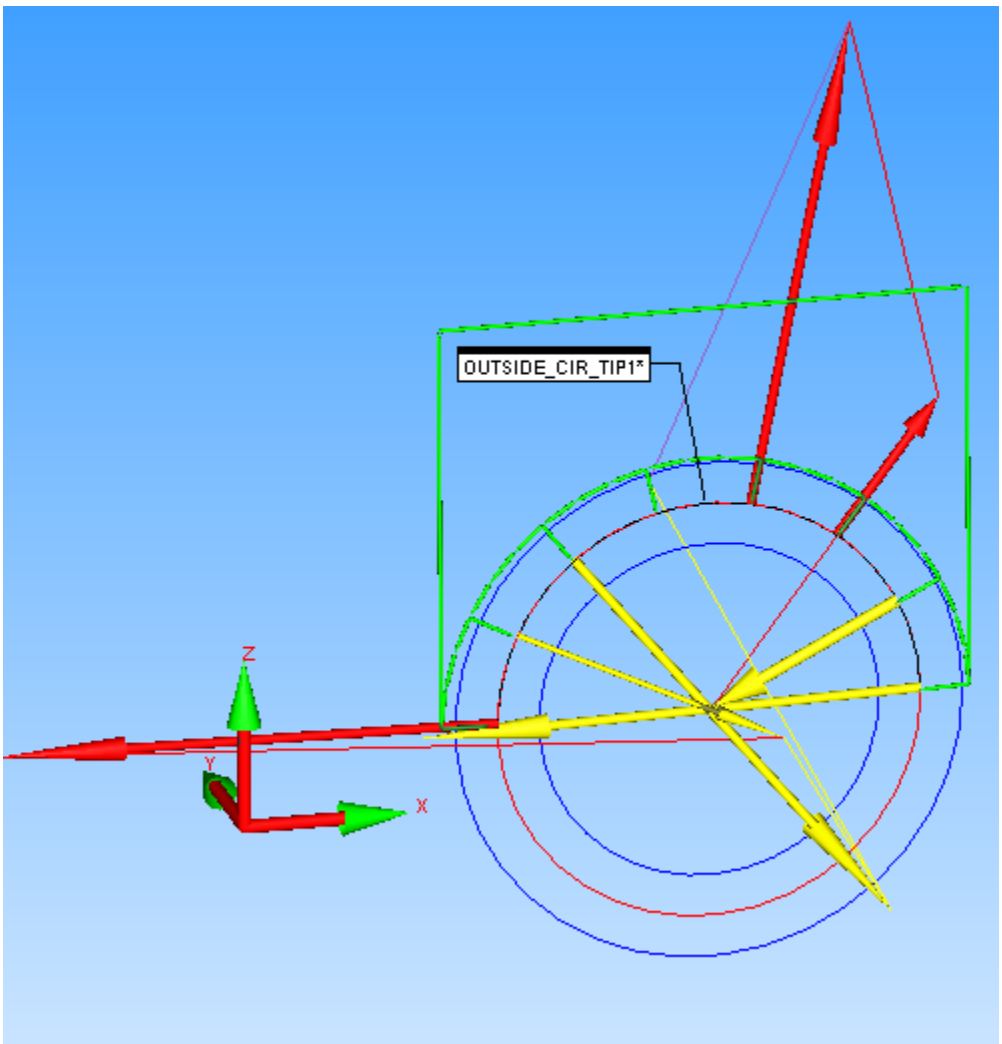
FCFCIR...	MM	◯ 0.01

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
OUTSIDE_...	0.000	0.010		0.119	0.119	0.109	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
OUTSIDE_CIR_T...	1	80.922	-0.006	10.087	1.000	0.000	0.018	-0.060	
	2	78.251	-0.004	21.008	0.893	0.000	0.449	-0.025	
	3	71.084	-0.005	29.735	0.610	0.000	0.793	0.020	
	4	60.836	-0.006	34.483	0.205	0.000	0.979	0.060	
	5	49.609	-0.005	34.240	-0.237	0.000	0.971	0.004	
	6	39.688	-0.005	29.244	-0.631	0.000	0.776	-0.060	MIN
	7	32.848	-0.005	20.646	-0.900	0.000	0.435	-0.035	
	8	30.225	-0.005	10.060	-1.000	0.000	0.017	0.060	MAX

2D Vector Calibration Report

Graphically, the Out of Roundness is drawn in the Graphics Display window as shown below.



2D Vector Calibration Out Of Roundness Graphical View

Running the same Journal file after the completion of the Vector Calibration shows an improvement to Circularity due to the 2D Vector Compensation.

pc-dmis	PART NAME : V2010MR3 LatheOffSphCal	December 14, 2010	09:36
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1

⌖	MM	LOC1 - OUTSIDE_CIR_TIP1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	55.200	0.050	0.050	55.151	-0.049	0.000
Z	8.880	0.050	0.050	8.902	0.022	0.000
D	51.260	0.050	0.050	51.264	0.004	0.000

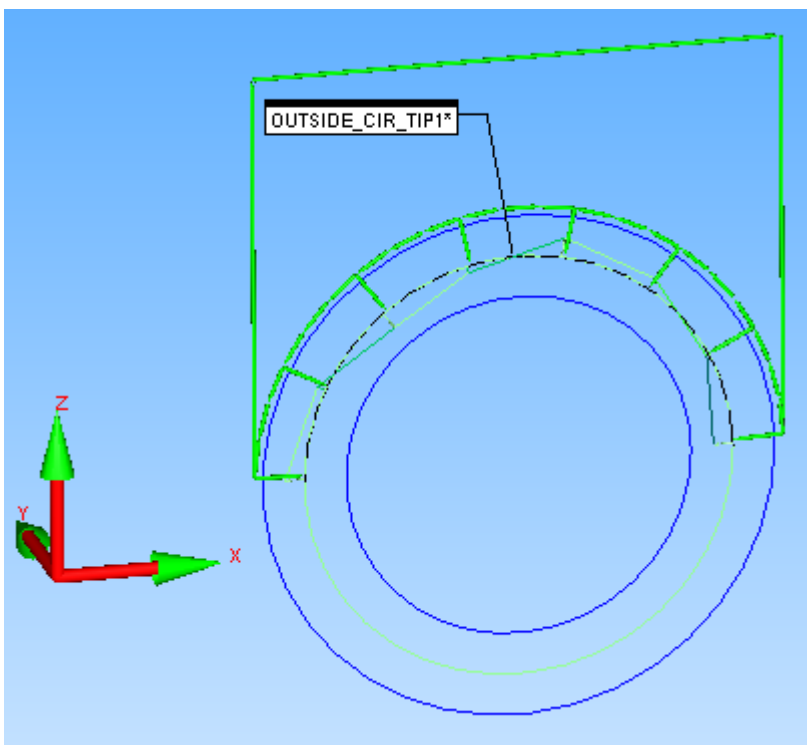
FCFCIR...	MM	◯ 0.01

Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
OUTSIDE_...	0.000	0.010		0.005	0.005	0.000	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
OUTSIDE_CIR_T...	1	80.778	-0.005	9.344	1.000	0.000	0.017	-0.002	
	2	78.070	-0.004	20.379	0.894	0.000	0.448	-0.000	
	3	70.820	-0.005	29.188	0.611	0.000	0.791	0.001	
	4	60.469	-0.006	33.978	0.207	0.000	0.978	0.002	
	5	49.114	-0.005	33.812	-0.236	0.000	0.972	-0.001	
	6	39.016	-0.005	28.815	-0.630	0.000	0.777	-0.002	MIN
	7	32.079	-0.005	20.071	-0.900	0.000	0.436	0.001	
	8	29.520	-0.005	9.318	-1.000	0.000	0.016	0.002	MAX

Report after 2D Vector Calibration

The Graphical View then would be drawn as shown below:



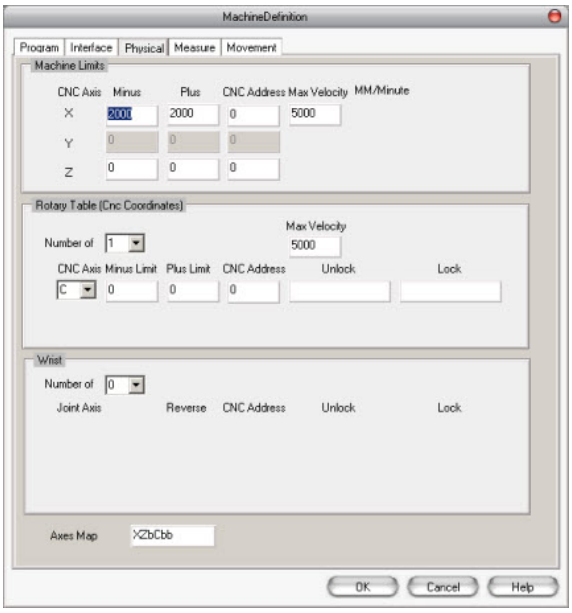
Graphical View after 2D Vector Calibration

Calibrating Extended Lathes

Extended lathes can measure past the spindle centre i.e. X<0. Hembrug lathes are examples of such machines. Calibration of offsets for these machines can be simplified with a single outside circle feature centred at the origin rather than the inside and outside circle as done previously. Simpler still, a spindle mounted sphere can be used and this can also be used to incorporate a 2d Vector Compensation too if required.

The calibration scheme chosen must manage the transition between taking hits at different sides of the spindle center to avoid collisions. To do this, the circle feature centerd at the spindle center is broken up into two semi-circles and a move ClearPlane interposed between the two. This scheme is only adopted to simplify calibration and measurement programs will measure only with X>0.

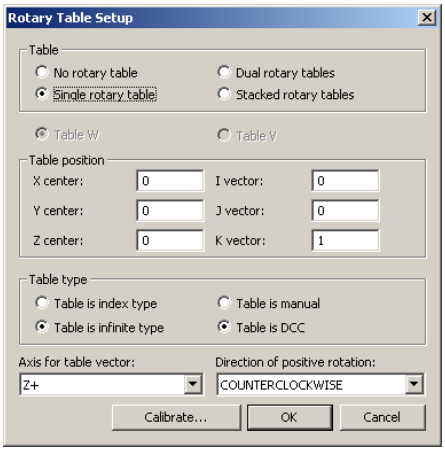
Extended lathes are setup by indicating a non-zero Minus limit for the X axis and a Rotary Table axis in the **Physical** tab of the **MachineDefinition** dialog of the ***CncServer.exe*** as shown below.



Physical Tab of MachineDefinition Dialog Screen for Extended Lathes

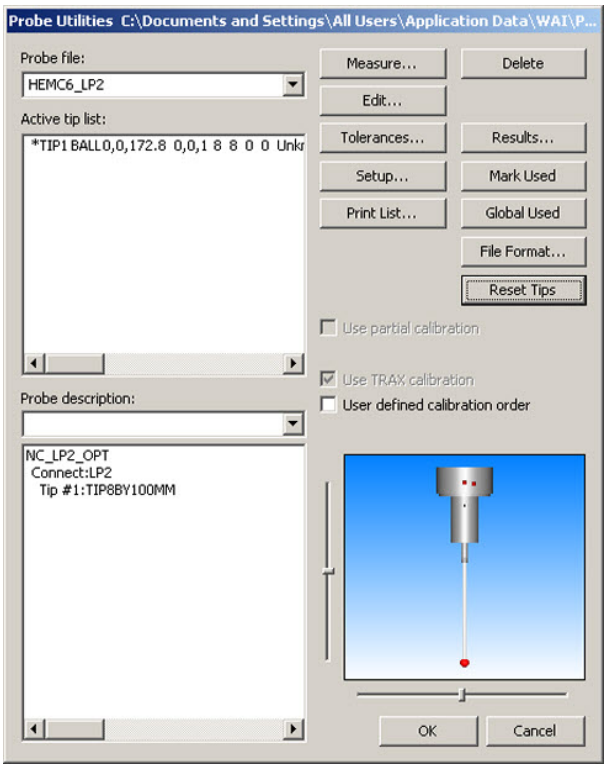
Ring Calibration

Select **Edit | Preferences | Rotary Table Setup** and ensure the **Single rotary table** is selected and clear the offsets if calibrating from scratch as shown below.



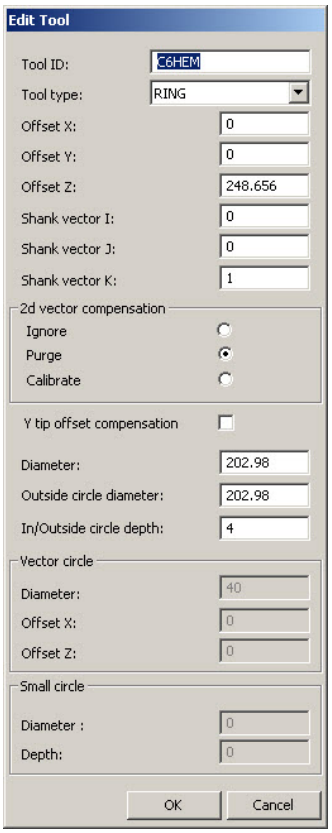
Rotary Table Setup Dialog Screen

Construct the probe as normal by navigating to the **Insert | Hardware Definition | Probe...** option from the main toolbar.



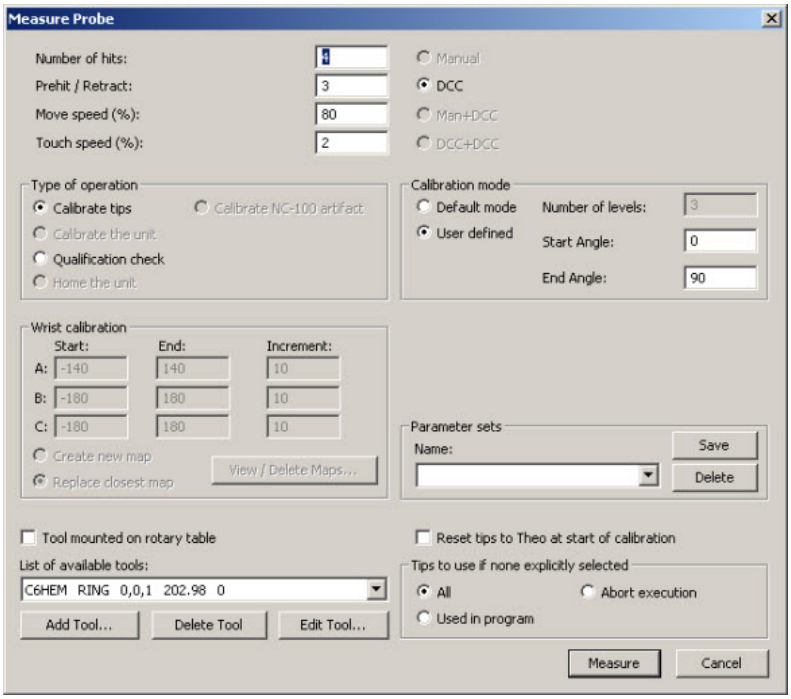
Probe Utilities Dialog Screen for Extended Lathes

Once the probe is defined, click on the **Edit...** button and add a new Ring Tool. Enter the ring diameter in both Diameter and Outside Circle Diameter edit boxes since this will be used to create the circle features - one on the right hand (X>0) side of the spindle and the other on the left hand side (X<0).



Edit Tool Dialog Screen for Ring Calibration

The 2d Vector Compensation is shown to be turned off and the Y Tip Compensation omitted although these options are still available if required. Click the **OK** button and then create the calibration program by clicking the **Measure** button from the **Probe Utilities** dialog screen to display the **Measure Probe** screen.



Measure Probe Dialog Screen for Extended Lathe

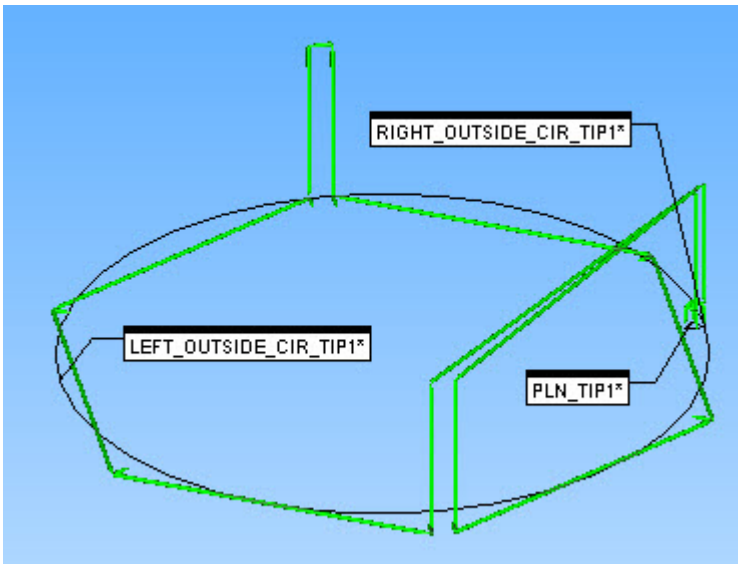
Click the **Measure** button to create the Calibration Part Program shown below:

```
Edit Window - V2010MR3RingCalHembrug.PRG
File Header
STARTUP = Start Alignment
Manual/DCC Mode
Dimension Format
Load Probe - HEMC6_LP2
Manual/DCC Mode
Prehit Distance
Retract Distance
Move Speed
TIP1 = Set Active Tip
Ignore Rotary Table
    ONOFF : ON
Set Clearance Plane
Move Clearance Plane
PLN_TIP1 = PLANE (CONTACT)
Move Clearance Plane
RIGHT_OUTSIDE_CIR_TIP1 = CIRCLE (CONTACT)
Move Clearance Plane
LEFT_OUTSIDE_CIR_TIP1 = CIRCLE (CONTACT)
CALIBRATE ACTIVE TIP
Move Clearance Plane
```

Calibration Part Program for Extended Lathe

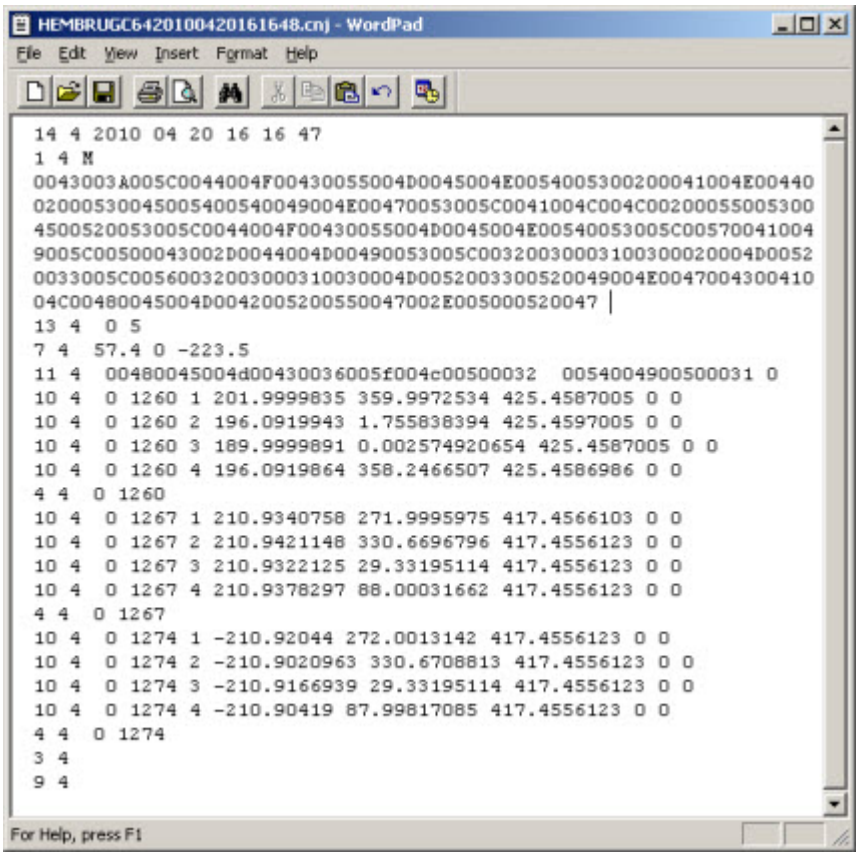
A single outside circle and a plane are needed on the artifact used for calibration. The part program generated splits the circle into 2 semi-circles.

- The right hand circle will be measured by rotating the artifact - hits are taken along the X axis at X>0.
- The left hand circle will be measured by rotating the artifact - hits are taken along the X axis at X<0.



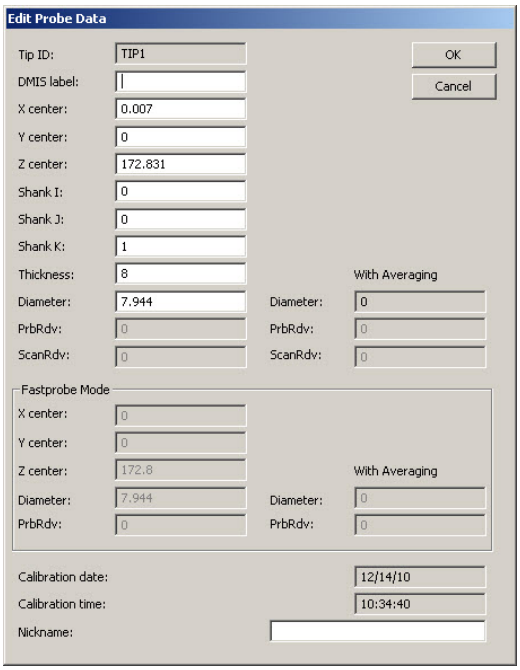
Calibration Part Program Graphical View

Create and run the part program as normal to generate the Journal file shown below:



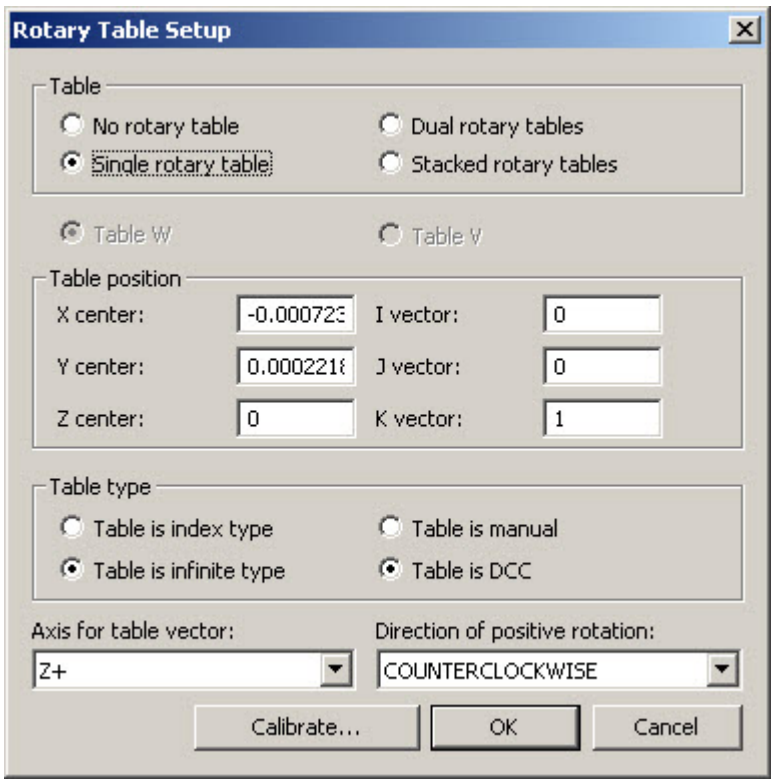
Extended Lathe Journal File

From the **Probe Utilities** dialog screen, click on the **Active Tip** from the list, then click on the **Edit...** button to open the **Edit Probe Data** dialog screen shown below to view the updated values.



Edit Probe Data Dialog Screen for Extended Lathe

Select **Edit | Preferences | Rotary Table Setup** to view the updated Rotary Table values.



Updated Rotary Table Setup Dialog Screen for Extended Lathe

Dimensioning the calibration program is instructive as shown below:

pc-dmis	PART NAME : V2010MR3RingCalHembrug				December 14, 2010	10:45
	REV NUMBER :		SER NUMBER :		STATS COUNT : 1	

CALIBRATE ACTIVE TIP WITH FEAT_ID=LEFT_OUTSIDE_CIR_TIP1

MM	LOC1 - PLN_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
Z	248.656	0.010	0.010	248.659	0.003	0.000
MM	LOC2 - RIGHT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.001	0.001	0.000
Y	0.000	0.010	0.010	0.000	0.000	0.000
D	202.980	0.010	0.010	202.936	-0.044	-0.034
MM	LOC3 - LEFT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.002	0.002	0.000
Y	0.000	0.010	0.010	0.002	0.002	0.000
D	202.980	0.010	0.010	202.912	-0.068	-0.058

Extended Lathe Dimensioned Calibration Program

The post-calibration sanity check is done by switching off the **Ignore Rotary Table** in the part program and un-checking the **Calibrate Active Tip** line and rerunning the Journal file (or better still, run the part program on the machine) to see if a near perfect result is achieved.

pc-dmis	PART NAME : V2010MR3RingCalHembrug		December 14, 2010	10:47
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

	MM	LOC1 - PLN_TIP1				
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
Z	248.656	0.010	0.010	248.656	0.000	0.000

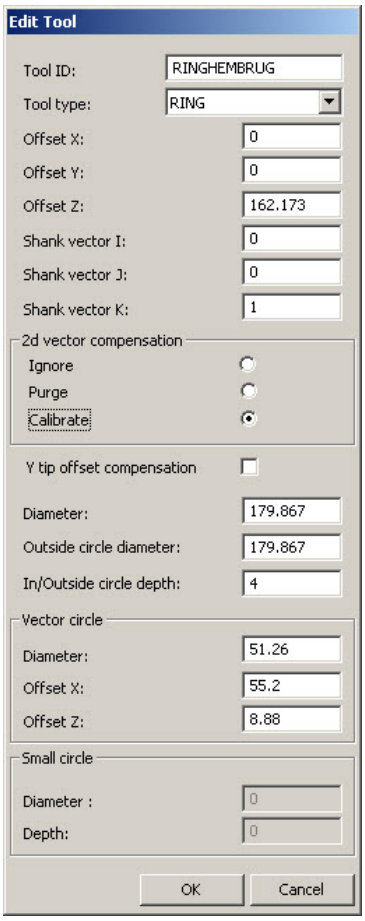
	MM	LOC2 - RIGHT_OUTSIDE_CIR_TIP1				
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.000	0.000	0.000
Y	0.000	0.010	0.010	0.000	0.000	0.000
D	202.980	0.010	0.010	202.980	0.000	0.000

	MM	LOC3 - LEFT_OUTSIDE_CIR_TIP1				
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.001	0.001	0.000
Y	0.000	0.010	0.010	0.002	0.002	0.000
D	202.980	0.010	0.010	202.980	0.000	0.000

Extended Lathe Updated Report

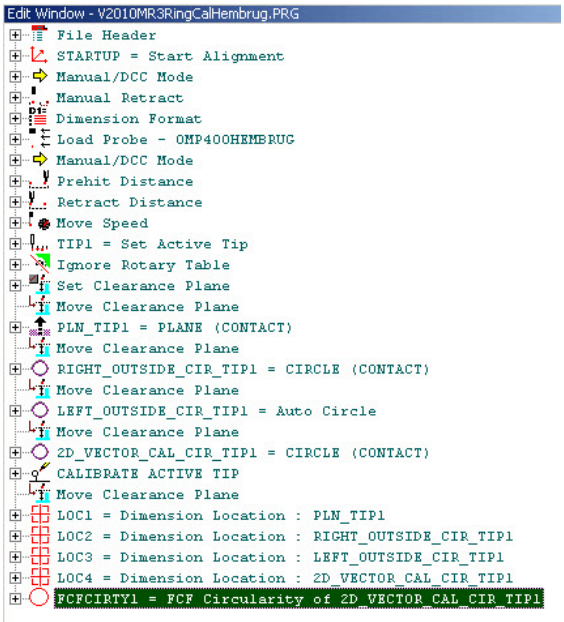
Ring Calibration with 2d Vector Compensation

2d Vector and Y Tip Offset Compensation can be added by enabling it in the **Edit Tool** dialog. Click the **Edit...** button from the **Probe Utilities** dialog to access this screen.



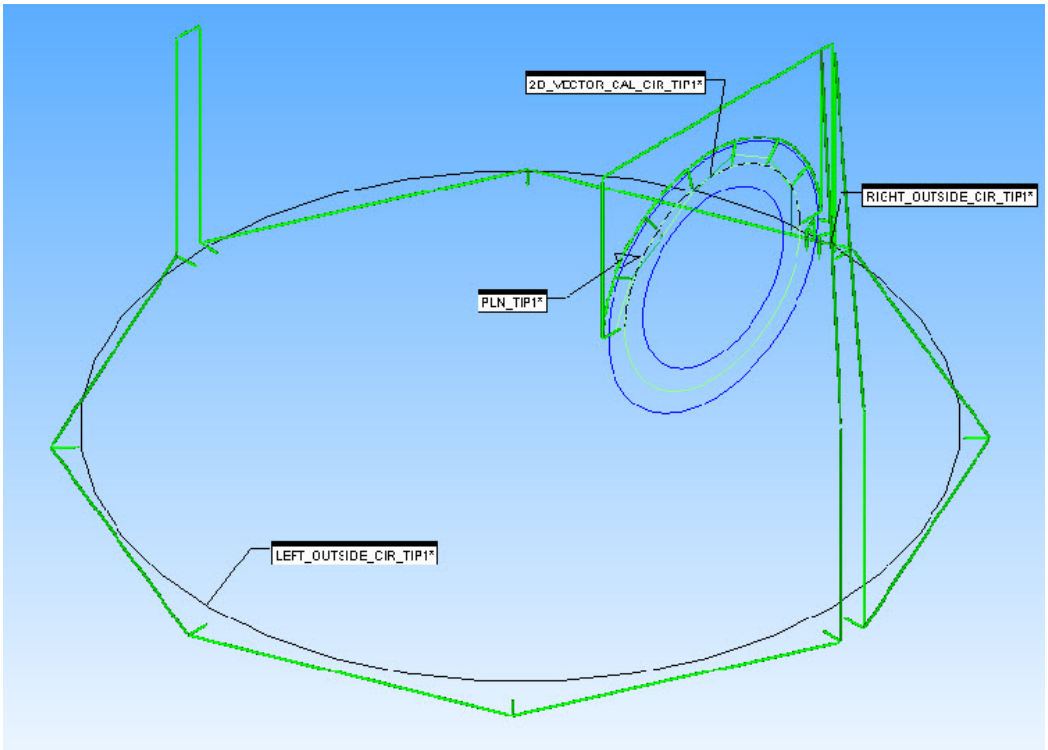
Edit Tool Dialog Screen for Ring Calibration with 2d Vector Compensation

Create the calibration part program as before. In this case, the 2d Vector Calibration circle has been edited to measure eight hits instead of five to match to saved Journal file data.



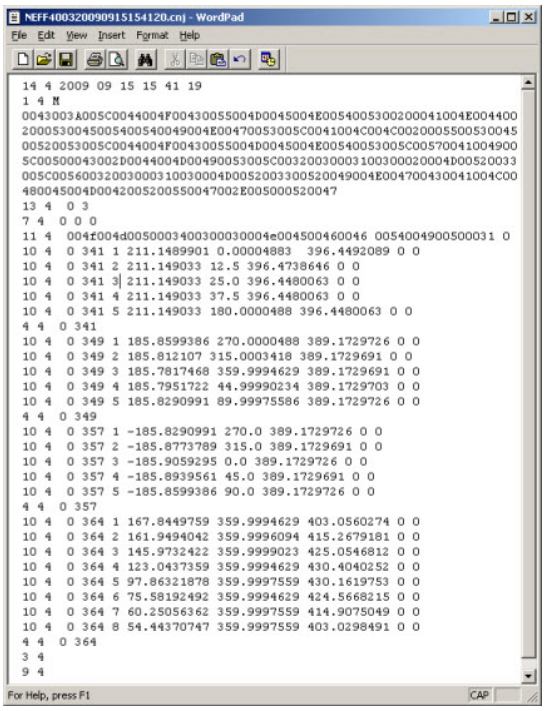
Part Program for Ring Calibration with 2d Vector Compensation

Running the program will display the following in the Graphic Display window:



Part Program for Ring Calibration with 2d Vector Compensation

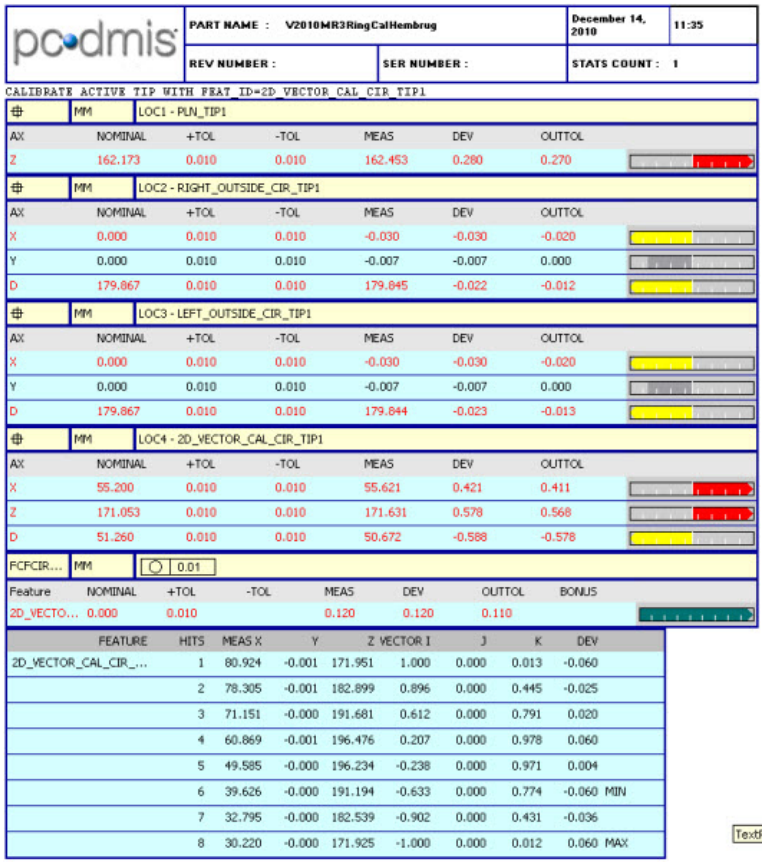
This part program generates a Journal file as shown below. In this case the Journal file has been created from parts of a Journal file from outside circle measurements on a Siemens NEF400 machine using the non-extended lathe groove calibration.



```
NEFF40032009091515412 0.cnj - WordPad
File Edit View Insert Format Help
14 4 2009 09 15 15 41 19
1 4 M
0043003A005C0044004F00430055004D0045004E005400530020004100 4E004400
2000530045005400540049004E00470053005C0041004C004C0020005500530045
0052005300 5C0044004F00430055004D0045004E005400530 05C00570041004900
5C00500043002D0044004D00490053005C0032003000310030002000 4D00520033
005C005600320030003100300 04D00520033005200 49004E00470043004100 4C00
480045004D0042005200550047002E005000520047
13 4  0 3
7 4  0 0 0
11 4  004f004d005000340030003004e004500460046 0054004900500031 0
10 4  0 341 1 211.1489901 0.00004883  396.4492089 0 0
10 4  0 341 2 211.149033 12.5 396.4738646 0 0
10 4  0 341 3 211.149033 25.0 396.4480063 0 0
10 4  0 341 4 211.149033 37.5 396.4480063 0 0
10 4  0 341 5 211.149033 180.0000488 396.4480063 0 0
4 4  0 341
10 4  0 349 1 185.8599386 270.0000488 389.1729726 0 0
10 4  0 349 2 185.812107 315.0003418 389.1729691 0 0
10 4  0 349 3 185.7817468 359.9994629 389.1729691 0 0
10 4  0 349 4 185.7951722 44.99990234 389.1729703 0 0
10 4  0 349 5 185.8290991 89.99975586 389.1729726 0 0
4 4  0 349
10 4  0 357 1 -185.8290991 270.0 389.1729726 0 0
10 4  0 357 2 -185.8773789 315.0 389.1729691 0 0
10 4  0 357 3 -185.9059295 0.0 389.1729726 0 0
10 4  0 357 4 -185.8939561 45.0 389.1729691 0 0
10 4  0 357 5 -185.8599386 90.0 389.1729726 0 0
4 4  0 357
10 4  0 364 1 167.8449759 359.9994629 403.0560274 0 0
10 4  0 364 2 161.9494042 359.9996094 415.2679181 0 0
10 4  0 364 3 145.9732422 359.9999023 425.0546812 0 0
10 4  0 364 4 123.0437359 359.9994629 430.4040252 0 0
10 4  0 364 5 97.86321878 359.9997559 430.1619753 0 0
10 4  0 364 6 75.58192492 359.9994629 424.5668215 0 0
10 4  0 364 7 60.25056362 359.9997559 414.9075049 0 0
10 4  0 364 8 54.44370747 359.9997559 403.0298491 0 0
4 4  0 364
3 4
9 4
For Help, press F1
CAP
```

Journal File for Ring Calibration with 2d Vector Compensation

Starting again from nominals for the tip and rotary table and using the dimensioning shown above, PC-DMIS produces the following report:



pc-dmis

PART NAME : V2010MR3RingCalHembrug		December 14, 2010	11:35
REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

CALIBRATE ACTIVE TIP WITH FEAT_ID=2D_VECTOR_CAL_CIR_TIP1

MM	LOC1 - PLN_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
Z	162.173	0.010	0.010	162.453	0.280	0.270

MM	LOC2 - RIGHT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	-0.030	-0.030	-0.020
Y	0.000	0.010	0.010	-0.007	-0.007	0.000
D	179.867	0.010	0.010	179.845	-0.022	-0.012

MM	LOC3 - LEFT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	-0.030	-0.030	-0.020
Y	0.000	0.010	0.010	-0.007	-0.007	0.000
D	179.867	0.010	0.010	179.844	-0.023	-0.013

MM	LOC4 - 2D_VECTOR_CAL_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	55.200	0.010	0.010	55.621	0.421	0.411
Z	171.053	0.010	0.010	171.631	0.578	0.568
D	51.260	0.010	0.010	50.672	-0.588	-0.578

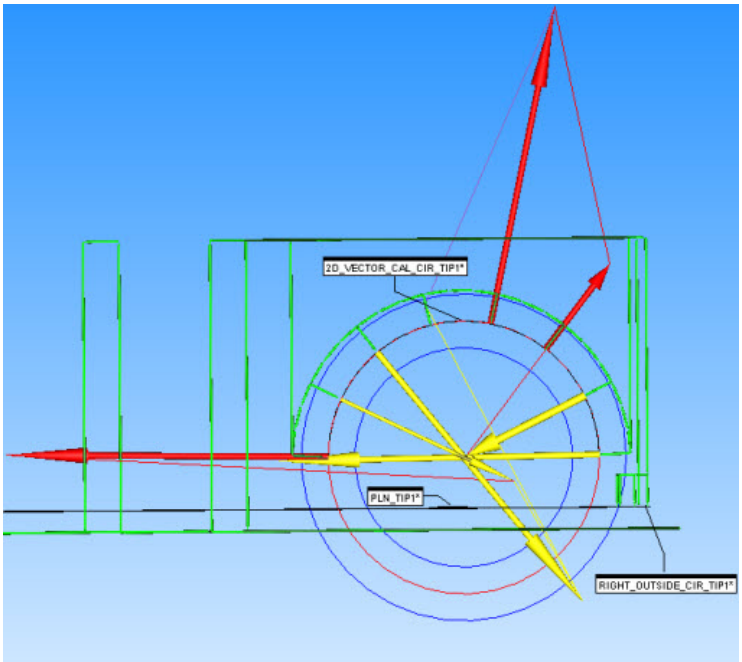
FCFCIR...	MM	0.01					
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
2D_VECTO...	0.000	0.010		0.120	0.120	0.110	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
2D_VECTOR_CAL_CIR_...	1	80.924	-0.001	171.951	1.000	0.000	0.013	-0.060	
	2	78.305	-0.001	182.899	0.896	0.000	0.445	-0.025	
	3	71.151	-0.000	191.681	0.612	0.000	0.791	0.020	
	4	60.869	-0.001	196.476	0.207	0.000	0.978	0.060	
	5	49.585	-0.000	196.234	-0.238	0.000	0.971	0.004	
	6	39.626	-0.000	191.194	-0.633	0.000	0.774	-0.060	MIN
	7	32.795	-0.000	182.539	-0.902	0.000	0.431	-0.036	
	8	30.220	-0.000	171.925	-1.000	0.000	0.012	0.060	MAX

Textf

Updated Calibration Report for Ring Calibration with 2d Vector Compensation

Graphically, the Out of Circularity found during calibration is shown below:



Updated Graphical View of Ring Calibration with 2d Vector Compensation

The post-calibration sanity check is done as before, generating the following report:

pc-dmis

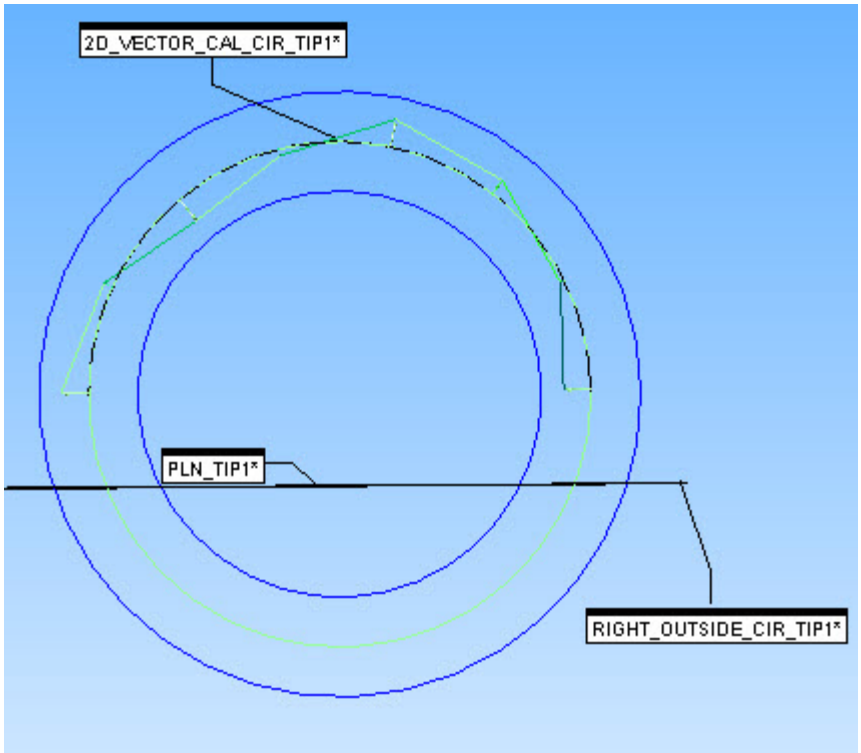
PART NAME :	V2010MR3RingCalHembrug	December 14, 2010	11:39
REV NUMBER :	SER NUMBER :	STATS COUNT :	1

MM	LOC1 - PLN_TIP1						
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
Z	162.173	0.010	0.010	162.170	-0.003	0.000	
MM	LOC2 - RIGHT_OUTSIDE_CIR_TIP1						
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Y	0.000	0.010	0.010	0.000	0.000	0.000	
D	179.867	0.010	0.010	179.867	0.000	0.000	
MM	LOC3 - LEFT_OUTSIDE_CIR_TIP1						
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Y	0.000	0.010	0.010	0.000	0.000	0.000	
D	179.867	0.010	0.010	179.867	0.000	0.000	
MM	LOC4 - 2D_VECTOR_CAL_CIR_TIP1						
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	55.200	0.010	0.010	55.602	0.402	0.392	
Z	171.053	0.010	0.010	171.361	0.308	0.298	
D	51.260	0.010	0.010	50.699	-0.561	-0.551	
FCFCIR...	MM	○ 0.01					
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
2D_VECTO...	0.000	0.010		0.006	0.006	0.000	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
2D_VECTOR_CAL_CIR_...	1	80.948	0.006	171.659	1.000	0.000	0.012	-0.003	
	2	78.323	0.007	182.601	0.896	0.000	0.443	-0.000	
	3	71.167	0.007	191.371	0.614	0.000	0.789	0.002	
	4	60.893	0.007	196.155	0.209	0.000	0.978	0.003	
	5	49.604	0.007	195.990	-0.237	0.000	0.972	-0	textualanalysis.lbl
	6	39.580	0.007	191.001	-0.632	0.000	0.775	-0.003	MIN
	7	32.731	0.007	182.296	-0.902	0.000	0.431	0.001	
	8	30.251	0.007	171.634	-1.000	0.000	0.011	0.003	MAX

Ring Calibration with 2d Vector Compensation Post Calibration Report

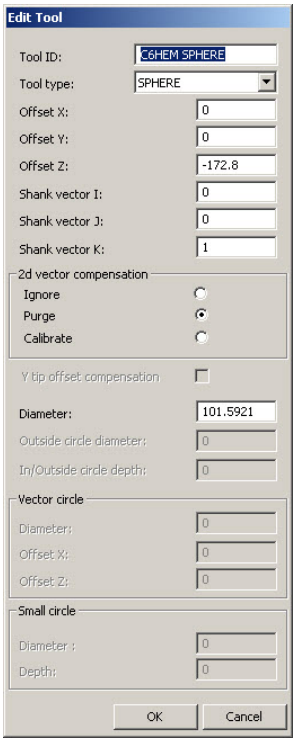
Graphically the results for Circularity are near perfect as shown below:



Ring Calibration with 2d Vector Compensation Post Calibration Graphical View

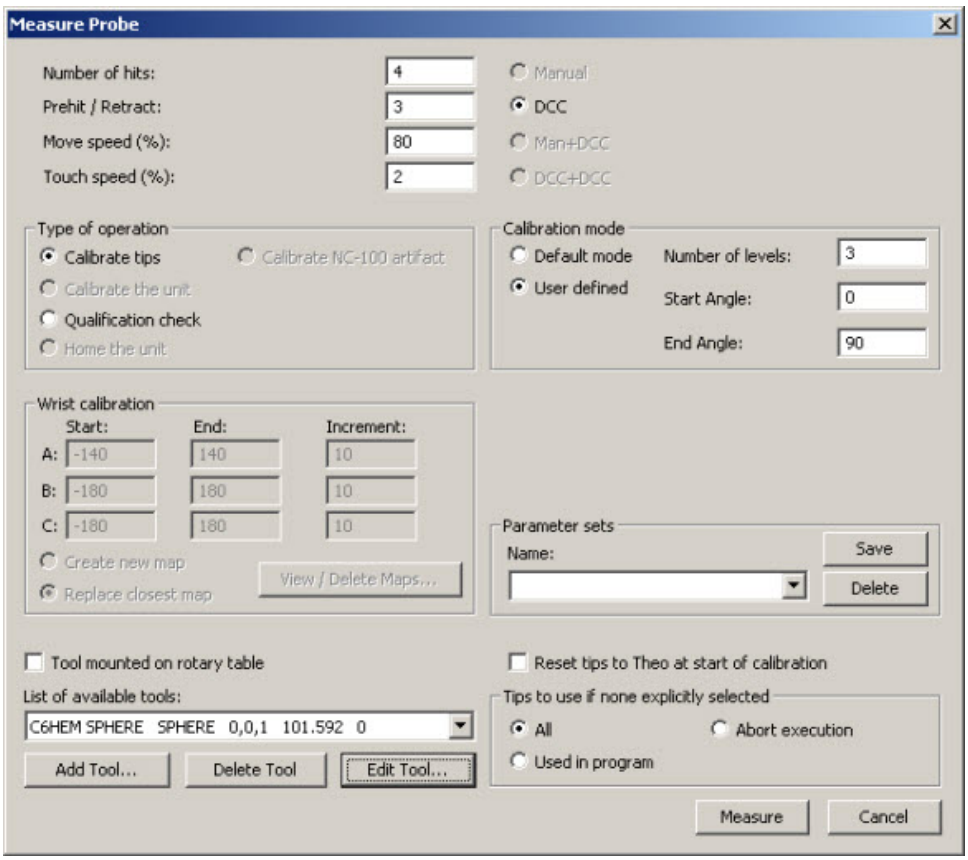
Sphere Calibration

Add a Sphere Tool from the **Probe Utilities** dialog screen by clicking on the **Edit...** button. Click the **OK** button when done.



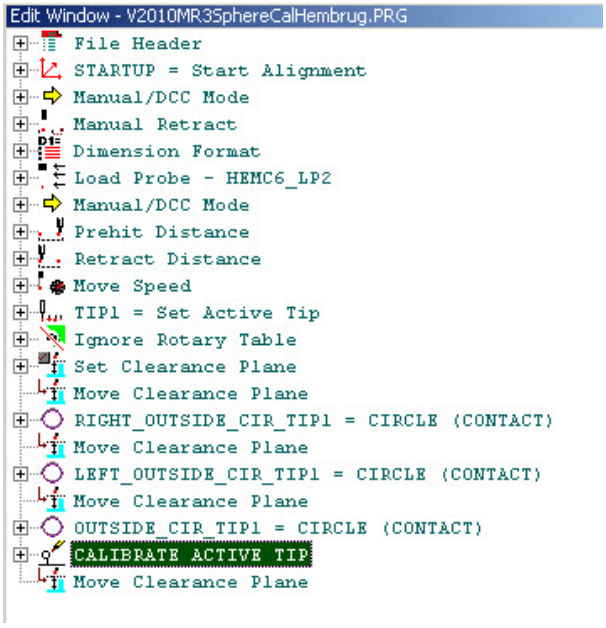
Edit Tool for Adding a Sphere Tool

From the **Probe Utilities** dialog screen, click the **Measure** button to display the **Measure Probe** dialog screen.



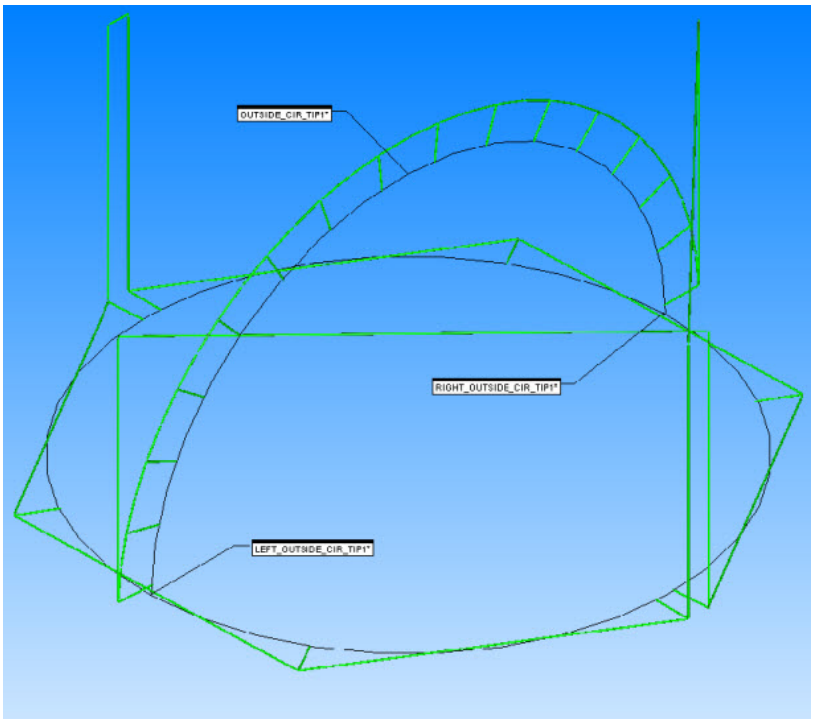
Measure Probes Dialog Screen for Sphere Calibration

Make the necessary change then click on the **Measure** button to create the Calibration Part Program as shown below.



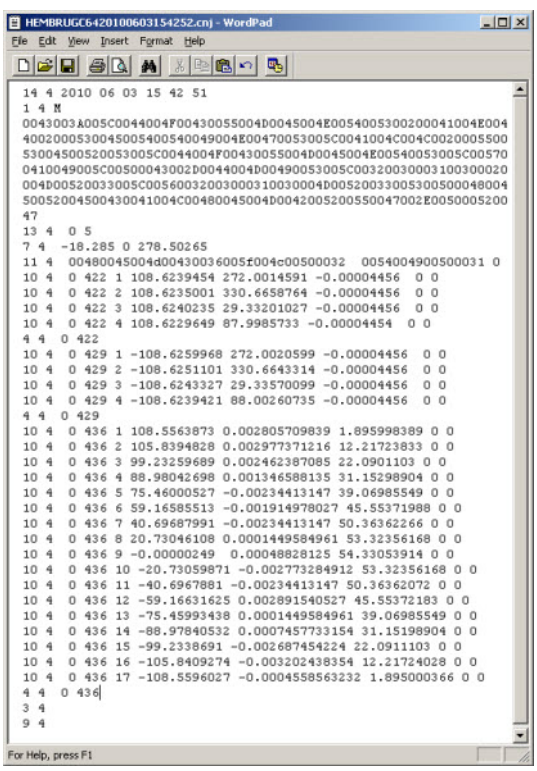
Sphere Calibration Part Program

The Graphical View of this program is shown below. Note that the number of hits for the outside circle has increased.



Graphical View of Sphere Calibration Part Program

Create and run the program as normal and a Journal file will be created similar to the following:



HEMBRUGC6420100603154252.cnj - WordPad

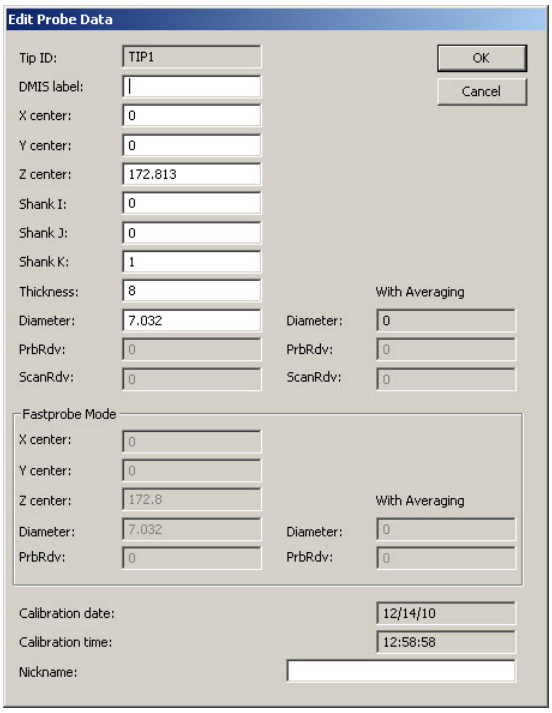
File Edit View Insert Format Help

```
14 4 2010 06 03 15 42 51
1 4 M
0043003A005C0044004F00430055004D0045004E00540053002000410 04E004
4002000530045005400540049004E00470053005C0041004C004C0020005500
530045005200530 05C0044004F00430055004D0045004E00540053005C00570
0410049005C00500043002D0044004D00490053005C0032003000310030002 0
004D00520033005C00560032003000310030004D00520033005300500048004
500520045004300410 04C00480045004D0042005200550047002E00500052 00
47
13 4  0 5
7 4  -18.285 0 278.50265
11 4  00480045004d00430036005f004c00500032  0054004900500031 0
10 4  0 422 1 108.6239454 272.0014591 -0.00004456  0 0
10 4  0 422 2 108.6235001 330.6658764 -0.00004456  0 0
10 4  0 422 3 108.6240235 29.33201027 -0.00004456  0 0
10 4  0 422 4 108.6229649 87.9985733 -0.00004454  0 0
4 4  0 422
10 4  0 429 1 -108.6259968 272.0020599 -0.00004456  0 0
10 4  0 429 2 -108.6251101 330.6643314 -0.00004456  0 0
10 4  0 429 3 -108.6243327 29.33570099 -0.00004456  0 0
10 4  0 429 4 -108.6239421 88.00260735 -0.00004456  0 0
4 4  0 429
10 4  0 436 1 108.5563873 0.002805709839 1.895998389 0 0
10 4  0 436 2 105.8394828 0.002977371216 12.21723833 0 0
10 4  0 436 3 99.23259689 0.002462387085 22.0901103 0 0
10 4  0 436 4 88.98042698 0.001346588135 31.15298904 0 0
10 4  0 436 5 75.46000527 -0.00234413147 39.06985549 0 0
10 4  0 436 6 59.16585513 -0.001914978027 45.55371988 0 0
10 4  0 436 7 40.69687991 -0.00234413147 50.36362266 0 0
10 4  0 436 8 20.73046108 0.0001449584961 53.32356168 0 0
10 4  0 436 9 -0.00000249  0.00048828125 54.33053914 0 0
10 4  0 436 10 -20.73059871 -0.002773284912 53.32356168 0 0
10 4  0 436 11 -40.6967881 -0.00234413147 50.36362072 0 0
10 4  0 436 12 -59.16631625 0.002891540527 45.55372183 0 0
10 4  0 436 13 -75.45993438 0.0001449584961 39.06985549 0 0
10 4  0 436 14 -88.97840532 0.0007457733154 31.15198904 0 0
10 4  0 436 15 -99.2338691 -0.002687454224 22.0911103 0 0
10 4  0 436 16 -105.8409274 -0.003202438354 12.21724028 0 0
10 4  0 436 17 -108.5596027 -0.0004558563232 1.895000366 0 0
4 4  0 436
3 4
9 4
```

For Help, press F1

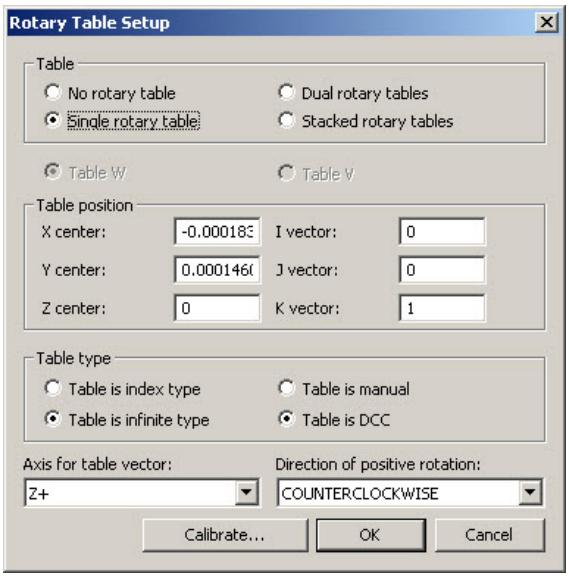
Sphere Calibration Journal File

From the **Probe Utilities** dialog screen, click on the Active Tip from the list, then click on the **Edit...** button to open the **Edit Probe Data** dialog screen shown below to view the updated values.



Sphere Calibration Edit Probe Data Dialog Screen

Select **Edit | Preferences | Rotary Table Setup** to view the updated Rotary Table values.



Sphere Calibration Rotary Table Setup Dialog Screen

Dimensioning the calibration program is instructive as shown below:

pc-dmis	PART NAME : V2010MR3SphereCalHembrug		December 14, 2010	13:45
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

CALIBRATE ACTIVE TIP WITH FEAT_ID=OUTSIDE_CIR_TIP1

	MM	LOC1 - RIGHT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Y	0.000	0.010	0.010	0.000	0.000	0.000	
D	101.592	0.010	0.010	100.623	-0.969	-0.959	

	MM	LOC2 - LEFT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Y	0.000	0.010	0.010	0.000	0.000	0.000	
D	101.592	0.010	0.010	100.625	-0.967	-0.957	

	MM	LOC3 - OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Z	-172.800	0.010	0.010	-172.787	0.013	0.003	
D	101.592	0.010	0.010	100.615	-0.977	-0.967	

FCFCIR...	MM	○ 0.01					
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
OUTSIDE_...	0.000	0.010		0.011	0.011	0.001	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
OUTSIDE_CIR_T...	1	50.281	0.003	-171.044	0.999	0.000	0.035	0.005	MAX
	2	49.034	0.003	-161.532	0.975	0.000	0.224	0.002	
	3	46.005	0.002	-152.430	0.914	0.000	0.405	-0.002	
	4	41.304	0.001	-144.065	0.821	0.000	0.571	-0.004	
	5	35.102	-0.002	-136.746	0.698	0.000	0.716	-0.005	MIN
	6	27.623	-0.001	-130.733	0.549	0.000	0.836	-0.005	
	7	19.139	-0.001	-126.249	0.380	0.000	0.925	-0.005	
	8	9.956	0.000	-123.455	0.198	0.000	0.980	-0.003	
	9	0.409	-0.000	-122.449	0.008	0.000	1.000	0.005	
	10	-9.156	0.001	-123.289	-0.182	0.000	0.983	-0.003	
	11	-18.389	0.001	-125.924	-0.365	0.000	0.931	-0.005	
	12	-26.955	-0.001	-130.262	-0.535	0.000	0.845	-0.005	
	13	-34.544	-0.000	-136.149	-0.686	0.000	0.728	-0.005	
	14	-40.878	-0.001	-143.368	-0.812	0.000	0.584	-0.005	
	15	-45.731	0.002	-151.659	-0.908	0.000	0.419	-0.001	
	16	-48.923	0.003	-160.722	-0.971	0.000	0.239	0.003	

Sphere Calibration Dimensioned Calibration Program

Turn the **Ignore Rotary Table** off, un-check the **CALIBRATE ACTIVE TIP** and then re-run the Journal file. The report is near perfect as shown below.

pc-dmis

PART NAME :	V2010MR3SphereCalHembrug	December 14, 2010	13:46
REV NUMBER :	SER NUMBER :	STATS COUNT :	1

MM	LOC1 - RIGHT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.000	0.000	0.000
Y	0.000	0.010	0.010	0.000	0.000	0.000
D	101.592	0.010	0.010	101.592	0.000	0.000

MM	LOC2 - LEFT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.000	0.000	0.000
Y	0.000	0.010	0.010	0.001	0.001	0.000
D	101.592	0.010	0.010	101.592	0.000	0.000

MM	LOC3 - OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.010	0.010	0.001	0.001	0.000
Z	-172.800	0.010	0.010	-172.800	0.000	0.000
D	101.592	0.010	0.010	101.583	-0.009	0.000

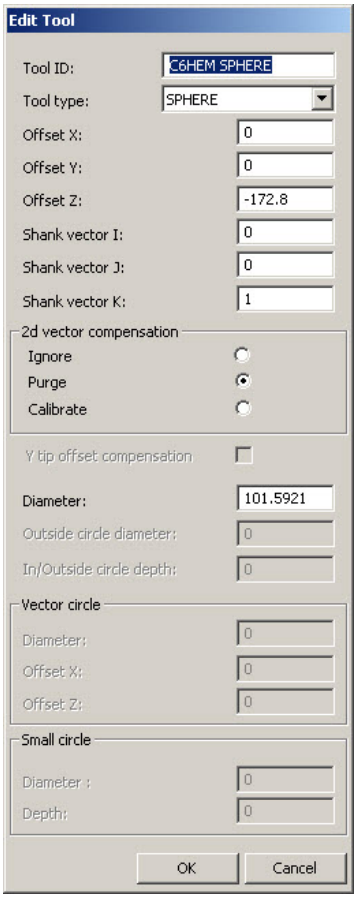
FCFCIR...	MM	0.01					
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
OUTSIDE_...	0.000	0.010		0.011	0.011	0.001	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
OUTSIDE_CIR_T...	1	50.764	0.003	-171.040	0.999	0.000	0.035	0.005	MAX
	2	49.504	0.003	-161.431	0.975	0.000	0.224	0.002	
	3	46.442	0.002	-152.235	0.914	0.000	0.405	-0.002	
	4	41.690	0.001	-143.786	0.821	0.000	0.571	-0.004	
	5	35.420	-0.001	-136.394	0.697	0.000	0.717	-0.005	MIN
	6	27.861	-0.001	-130.325	0.548	legacy_dimension.lbl		-0.005	
	7	19.285	-0.001	-125.801	0.380	0.000	0.925	-0.005	
	8	10.006	0.000	-122.988	0.197	0.000	0.980	-0.003	
	9	0.360	0.000	-121.981	0.007	0.000	1.000	0.005	
	10	-9.302	0.001	-122.841	-0.183	0.000	0.983	-0.003	
	11	-18.626	0.001	-125.515	-0.367	0.000	0.930	-0.005	
	12	-27.273	-0.001	-129.911	-0.537	0.000	0.844	-0.005	
	13	-34.929	0.000	-135.869	-0.687	0.000	0.727	-0.005	
	14	-41.315	-0.000	-143.173	-0.813	0.000	0.583	-0.005	
	15	-46.201	0.002	-151.557	-0.909	0.000	0.418	-0.001	
	16	-49.406	0.003	-160.719	-0.971	0.000	0.238	0.003	

Updated Sphere Calibration Report

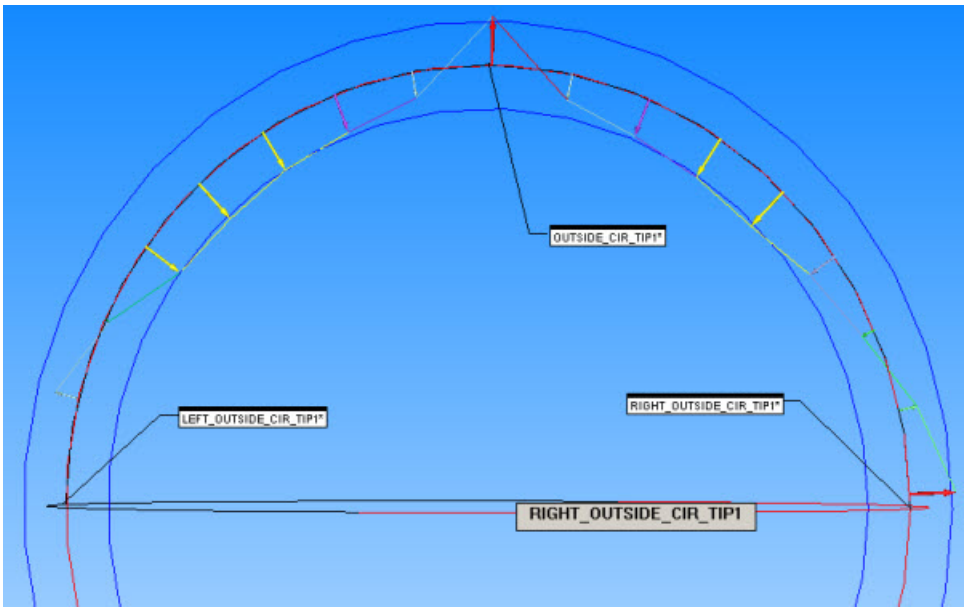
Sphere Calibration with 2D Vector Compensation

Add the Sphere Tool with 2d Vector Compensation as shown below. Click on the **Edit...** button from the **Probe Utilities** dialog to access this screen.



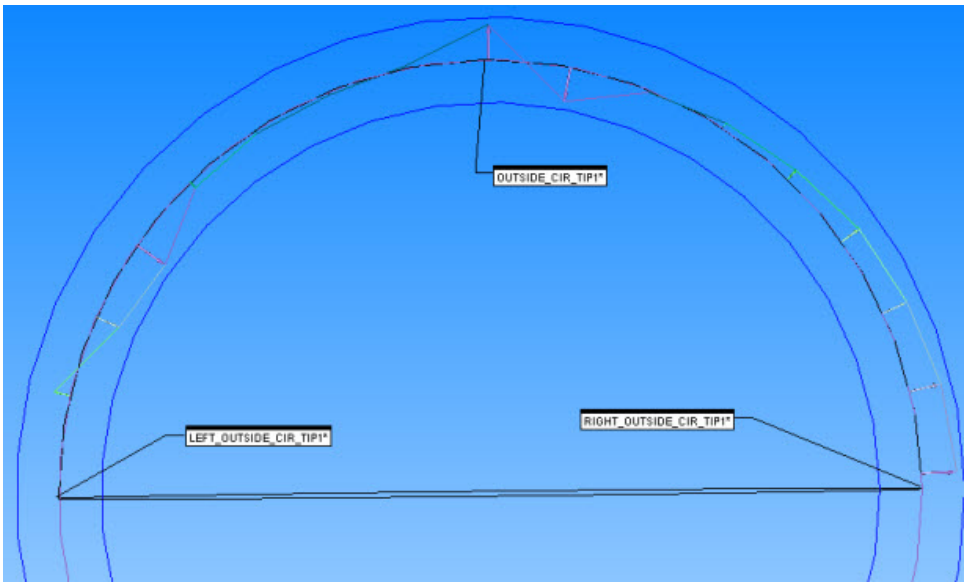
Edit Tool Dialog Screen for Sphere Calibration with 2d Vector Compensation

From the **Probe Utilities** dialog screen, click the **Measure** button to display the **Measure Probe** dialog screen. Make any changes as needed then click the **Measure** button to created the part program which will by default look the same as a standard Sphere Calibration routine.



Initial Graphical View of Sphere Calibration Part Program with 2d Vector Compensation

Re-running the new Journal file after this kind of calibration emphasizes the effect of 2d Vector Compensation as shown below.



Updated Graphical View of Sphere Calibration Part Program with 2d Vector Compensation

Dimensioning the calibration program is instructive as shown below:

pc-dmis	PART NAME : V2010MR3SphereCalHembrug		December 14, 2010	13:49
	REV NUMBER :	SER NUMBER :	STATS COUNT : 1	

	MM	LOC1 - RIGHT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Y	0.000	0.010	0.010	0.000	0.000	0.000	
D	101.592	0.010	0.010	101.592	0.000	0.000	

	MM	LOC2 - LEFT_OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL		DEV	OUTTOL	
X	0.000	0.010	0.010	0.000	0.000	0.000	
Y	0.000	0.010	0.010	0.001	0.001	0.000	
D	101.592	0.010	0.010	101.592	0.000	0.000	

TextReportObject1

	MM	LOC3 - OUTSIDE_CIR_TIP1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	
X	0.000	0.010	0.010	0.002	0.002	0.000	
Z	-172.800	0.010	0.010	-172.794	0.006	0.000	
D	101.592	0.010	0.010	101.581	-0.011	-0.001	

FCFCIR...	MM	○ 0.01					
Feature	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL	BONUS
OUTSIDE_...	0.000	0.010		0.008	0.008	0.000	

FEATURE	HITS	MEAS X	Y	Z	VECTOR I	J	K	DEV	
OUTSIDE_CIR_T...	1	50.762	0.003	-171.040	0.999	0.000	0.035	0.004	MAX
	2	49.505	0.003	-161.431	0.975	0.000	0.224	0.004	
	3	46.446	0.002	-152.233	0.914	0.000	0.405	0.003	
	4	41.695	0.001	-143.782	0.821	0.000	0.571	0.002	
	5	35.425	-0.001	-136.389	0.697	0.000	0.717	0.001	
	6	27.864	-0.001	-130.319	0.548	0.000	0.836	0.001	
	7	19.287	-0.001	-125.796	0.380	0.000	0.925	-0.000	
	8	10.006	0.000	-122.987	0.197	0.000	0.980	-0.004	MIN
	9	0.360	0.000	-121.980	0.007	0.000	1.000	0.004	
	10	-9.303	0.001	-122.837	-0.183	0.000	0.983	0.000	
	11	-18.629	0.001	-125.510	-0.367	0.000	0.930	-0.000	
	12	-27.277	-0.001	-129.906	-0.537	0.000	0.844	-0.000	
	13	-34.934	0.000	-135.866	-0.687	0.000	0.726	-0.001	
	14	-41.317	-0.000	-143.172	-0.813	0.000	0.583	-0.004	
	15	-46.200	0.002	-151.557	-0.909	0.000	0.418	-0.003	
	16	-49.406	0.003	-160.719	-0.971	0.000	0.237	0.002	

Dimensioned Part Program Report for Sphere Calibration with 2d Vector Compensation

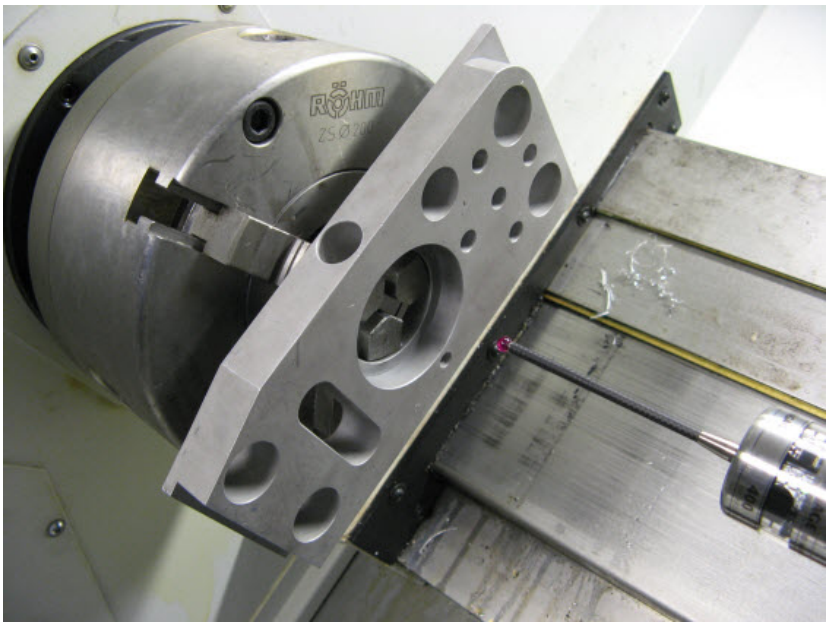
Measuring

The following two cases of 3D measurement with a C-Axis Lathe are detailed in this chapter.

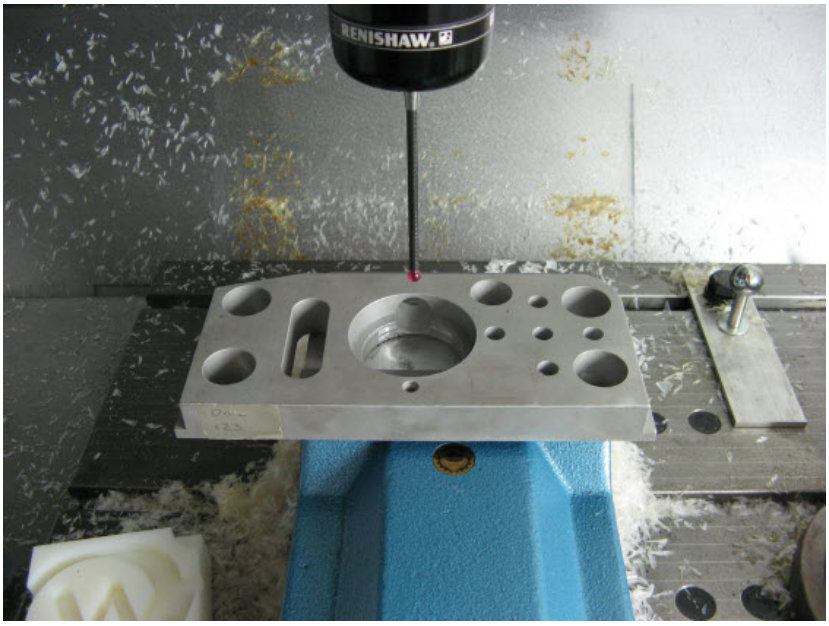
Measuring a Brown & Sharpe Block

An example of a 3D measurement on a lathe is described below.

1. Mount a Brown and Sharpe test block in the lathe chuck via the large center hole in the block.
2. Run a normal PC-DMIS test program on it as on a CMM or a milling machine using PC-DMIS NC.



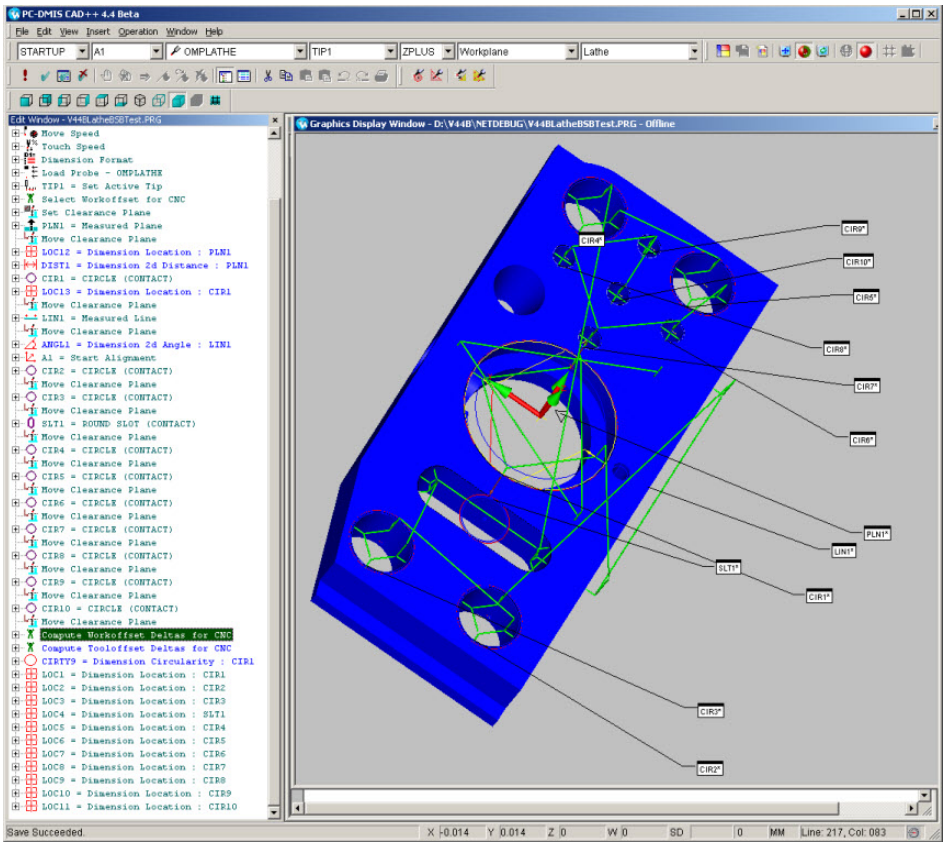
Measurement Example on a Fanuc 21i/M Mill



Measurement Example on a Siemens NEF400 Lathe

PC-DMIS NC runs the same part program used on a Fanuc21i/M Mill and gives results on the features on the block to within 10 microns. The Siemens NEF400 Lathe has a OMP400 probe installed while the Fanuc 21i/M Mill has an MP700 probe installed.

In either case, there are very few differences between the two PC-DMIS program setups with only the selection of the Lathe required - the Probe, CAD Transform and Alignment of the part in the part program remains unchanged.

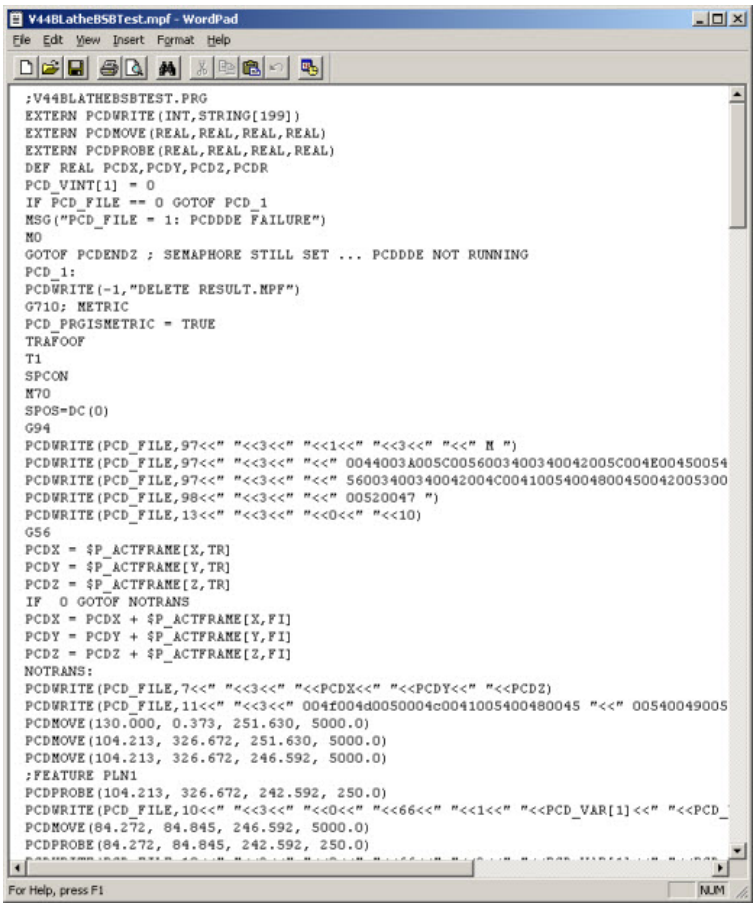


Brown & Sharpe Measurement Part Program

To access points on the left hand side of the block (X<0), the rectangular to polar conversion results in the block being spun around are required in order to ensure the point lies on the X axis (where X>0). A further modification to the Siemens macro computes the spindle speed such that the spindle move takes the same time as the XZ axis move resulting in something that looks like an XYZ axis interpolated move despite the lathe have no Y or C axes.

It should be noted however that on a proper XYZ axis machine (e.g. the Fanuc mill) the 3D moves will be vectoring in on points on a straight line instead of an arc. However, for measurement moves with a small prehit, the error is usually insignificant. The arcing of straight path lines will also mean collision detection cannot always be relied on. Of course, for measurement moves where no spindle motion is required, there is no such measurement error, but to achieve this in the above part, there would need to be some modifications to the part program copied from the milling machine.

The part program generated by PC-DMIS is only slightly different to a normal 3D measurement using a Siemens 840D machine.



PC-DMIS Part Program

Here the results PC-DMIS returned in the PCD_VAR[] variables but v2010 and later versions avoid using these user variables and instead returns results in the new UGUD variables PCD_RES[] instead.

Comparing the Siemens NEF400 Lathe measurement results with a CMM measuring the same part in the same way shows reasonable agreement:

pc-dmis | PART NAME : V438LathebSBTest | August 07, 2008 | 11:18
REV NUMBER : | SER NUMBER : | STATS COUNT : 1

MM	LOC1 - CIR1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.050	0.050	0.000	0.000	0.000
Y	0.000	0.050	0.050	0.000	0.000	0.000
D	63.500	0.050	0.050	63.394	-0.106	0.056
RN	0.000	0.050	0.000	0.019	0.019	0.000

MM	LOC2 - CIR2					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.687	-0.057	0.007
Y	-25.400	0.050	0.050	-25.270	0.130	0.080
D	25.400	0.050	0.050	25.549	0.149	0.099

MM	LOC3 - CIR3					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.606	0.024	0.000
Y	25.400	0.050	0.050	25.527	0.128	0.078
D	25.400	0.050	0.050	25.555	0.155	0.105

MM	LOC4 - SLT1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-52.578	0.050	0.050	-52.569	0.009	0.000
Y	-3.239	0.050	0.050	-3.143	0.096	0.046
Z	0.000	0.050	0.050	0.000	0.000	0.000
D	19.050	0.050	0.050	19.113	0.063	0.013
L	60.325	0.050	0.050	60.326	0.001	0.000

MM	LOC5 - CIR4					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.405	0.032	0.000
Y	25.400	0.050	0.050	25.189	-0.210	0.160
D	25.400	0.050	0.050	25.554	0.154	0.104

MM	LOC6 - CIR5					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.307	-0.066	0.016
Y	-25.400	0.050	0.050	-25.615	-0.214	0.164
D	25.400	0.050	0.050	25.549	0.149	0.099

MM	LOC7 - CIR6					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	64.933	-0.040	0.000
Y	-25.400	0.050	0.050	-25.627	-0.227	0.177
D	9.525	0.050	0.050	9.539	0.014	0.000

MM	LOC8 - CIR7					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	41.148	0.050	0.050	41.109	-0.039	0.000
Y	0.000	0.050	0.050	-0.180	-0.180	0.130
D	9.525	0.050	0.050	9.542	0.017	0.000

MM	LOC9 - CIR8					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.014	0.041	0.000
Y	25.400	0.050	0.050	25.139	-0.260	0.210
D	9.525	0.050	0.050	9.544	0.019	0.000

MM	LOC10 - CIR9					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.364	-0.009	0.000
Y	0.000	0.050	0.050	-0.306	-0.306	0.256
D	9.525	0.050	0.050	9.540	0.015	0.000

MM	LOC11 - CIR10					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.008	0.034	0.000
Y	0.000	0.050	0.050	-0.248	-0.248	0.198
D	9.525	0.050	0.050	9.530	0.005	0.000

pc-dmis | PART NAME : V438DBSBTest | June 04, 2008 | 10:53
REV NUMBER : | SER NUMBER : | STATS COUNT : 1

MM	LOC1 - CIR1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.050	0.050	0.000	0.000	0.000
Y	0.000	0.050	0.050	0.000	0.000	0.000
D	63.500	0.050	0.050	63.403	-0.097	0.047

MM	LOC2 - CIR2					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.697	-0.067	0.017
Y	-25.400	0.050	0.050	-25.236	0.164	0.114
D	25.400	0.050	0.050	25.555	0.155	0.105

MM	LOC3 - CIR3					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.590	0.040	0.000
Y	25.400	0.050	0.050	25.562	0.162	0.112
D	25.400	0.050	0.050	25.559	0.159	0.109

MM	LOC4 - SLT1					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-52.578	0.050	0.050	-52.576	0.002	0.000
Y	-3.239	0.050	0.050	-3.129	0.110	0.060
Z	0.000	0.050	0.050	0.000	0.000	0.000
D	19.050	0.050	0.050	19.126	0.076	0.026
L	60.325	0.050	0.050	60.330	0.005	0.000

MM	LOC5 - CIR4					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.408	0.034	0.000
Y	25.400	0.050	0.050	25.167	-0.233	0.183
D	25.400	0.050	0.050	25.557	0.157	0.107

MM	LOC6 - CIR5					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.298	-0.075	0.025
Y	-25.400	0.050	0.050	-25.634	-0.234	0.184
D	25.400	0.050	0.050	25.556	0.156	0.106

MM	LOC7 - CIR6					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	64.934	-0.039	0.000
Y	-25.400	0.050	0.050	-25.640	-0.240	0.190
D	9.525	0.050	0.050	9.547	0.022	0.000

MM	LOC8 - CIR7					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	41.148	0.050	0.050	41.111	-0.037	0.000
Y	0.000	0.050	0.050	-0.187	-0.187	0.137
D	9.525	0.050	0.050	9.542	0.017	0.000

MM	LOC9 - CIR8					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.002	0.029	0.000
Y	25.400	0.050	0.050	25.128	-0.271	0.221
D	9.525	0.050	0.050	9.541	0.016	0.000

MM	LOC10 - CIR9					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.363	-0.011	0.000
Y	0.000	0.050	0.050	-0.329	-0.329	0.279
D	9.525	0.050	0.050	9.540	0.015	0.000

MM	LOC11 - CIR10					
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.003	0.030	0.000
Y	0.000	0.050	0.050	-0.250	-0.250	0.200
D	9.525	0.050	0.050	9.540	0.015	0.000

Single Circle Measurement Comparison - Siemens Lathe results on the left, CMM results on the right

Note: The above results were obtained after using Update Work Offset to correct the rotational alignment of the part.

By comparison, using two circles for calibration compares quite well and shows very little difference:

pc-dmis | PART NAME : V430LatheBSBTest | August 07, 2008 | 14:29
REV NUMBER : | SER NUMBER : | STATS COUNT : 1

MM LOC1 - CIR1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.050	0.050	0.000	0.000	0.000
Y	0.000	0.050	0.050	0.000	0.000	0.000
D	63.500	0.050	0.050	63.397	-0.103	0.053
RN	0.000	0.050	0.000	0.020	0.020	0.000

MM LOC2 - CIR2

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.701	-0.071	0.021
Y	-25.400	0.050	0.050	-25.224	0.176	0.126
D	25.400	0.050	0.050	25.552	0.152	0.102

MM LOC3 - CIR3

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.590	0.040	0.000
Y	25.400	0.050	0.050	25.572	0.172	0.122
D	25.400	0.050	0.050	25.555	0.155	0.105

MM LOC4 - SLT1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-52.578	0.050	0.050	-52.584	-0.006	0.000
Y	-3.239	0.050	0.050	-3.189	0.050	0.000
Z	0.000	0.050	0.050	0.000	0.000	0.000
D	19.050	0.050	0.050	19.196	0.146	0.096
L	60.325	0.050	0.050	60.310	-0.015	0.000

MM LOC5 - CIR4

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.416	0.043	0.000
Y	25.400	0.050	0.050	25.137	-0.262	0.212
D	25.400	0.050	0.050	25.556	0.156	0.106

MM LOC6 - CIR5

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.294	-0.079	0.029
Y	-25.400	0.050	0.050	-25.668	-0.268	0.218
D	25.400	0.050	0.050	25.550	0.150	0.100

MM LOC7 - CIR6

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	64.941	-0.032	0.000
Y	-25.400	0.050	0.050	-25.617	-0.217	0.167
D	9.525	0.050	0.050	9.543	0.018	0.000

MM LOC8 - CIR7

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	41.148	0.050	0.050	41.106	-0.042	0.000
Y	0.000	0.050	0.050	-0.090	-0.090	0.040
D	9.525	0.050	0.050	9.538	0.013	0.000

MM LOC9 - CIR8

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.005	0.032	0.000
Y	25.400	0.050	0.050	25.156	-0.244	0.194
D	9.525	0.050	0.050	9.537	0.012	0.000

MM LOC10 - CIR9

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.370	-0.003	0.000
Y	0.000	0.050	0.050	-0.351	-0.351	0.301
D	9.525	0.050	0.050	9.538	0.013	0.000

MM LOC11 - CIR10

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.006	0.033	0.000
Y	0.000	0.050	0.050	-0.213	-0.213	0.163
D	9.525	0.050	0.050	9.542	0.017	0.000

pc-dmis | PART NAME : V430DBSBTest | June 04, 2008 | 10:53
REV NUMBER : | SER NUMBER : | STATS COUNT : 1

MM LOC1 - CIR1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	0.000	0.050	0.050	0.000	0.000	0.000
Y	0.000	0.050	0.050	0.000	0.000	0.000
D	63.500	0.050	0.050	63.403	-0.097	0.047

MM LOC2 - CIR2

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.697	-0.067	0.017
Y	-25.400	0.050	0.050	-25.236	0.164	0.114
D	25.400	0.050	0.050	25.555	0.155	0.105

MM LOC3 - CIR3

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-87.630	0.050	0.050	-87.590	0.040	0.000
Y	25.400	0.050	0.050	25.562	0.162	0.112
D	25.400	0.050	0.050	25.559	0.159	0.109

MM LOC4 - SLT1

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	-52.578	0.050	0.050	-52.576	0.002	0.000
Y	-3.239	0.050	0.050	-3.129	0.110	0.060
Z	0.000	0.050	0.050	0.000	0.000	0.000
D	19.050	0.050	0.050	19.126	0.076	0.026
L	60.325	0.050	0.050	60.330	0.005	0.000

MM LOC5 - CIR4

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.408	0.034	0.000
Y	25.400	0.050	0.050	25.167	-0.233	0.183
D	25.400	0.050	0.050	25.557	0.157	0.107

MM LOC6 - CIR5

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.298	-0.075	0.025
Y	-25.400	0.050	0.050	-25.634	-0.234	0.184
D	25.400	0.050	0.050	25.556	0.156	0.106

MM LOC7 - CIR6

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	64.934	-0.039	0.000
Y	-25.400	0.050	0.050	-25.640	-0.240	0.190
D	9.525	0.050	0.050	9.547	0.022	0.000

MM LOC8 - CIR7

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	41.148	0.050	0.050	41.111	-0.037	0.000
Y	0.000	0.050	0.050	-0.187	-0.187	0.137
D	9.525	0.050	0.050	9.542	0.017	0.000

MM LOC9 - CIR8

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.002	0.029	0.000
Y	25.400	0.050	0.050	25.128	-0.271	0.221
D	9.525	0.050	0.050	9.541	0.016	0.000

MM LOC10 - CIR9

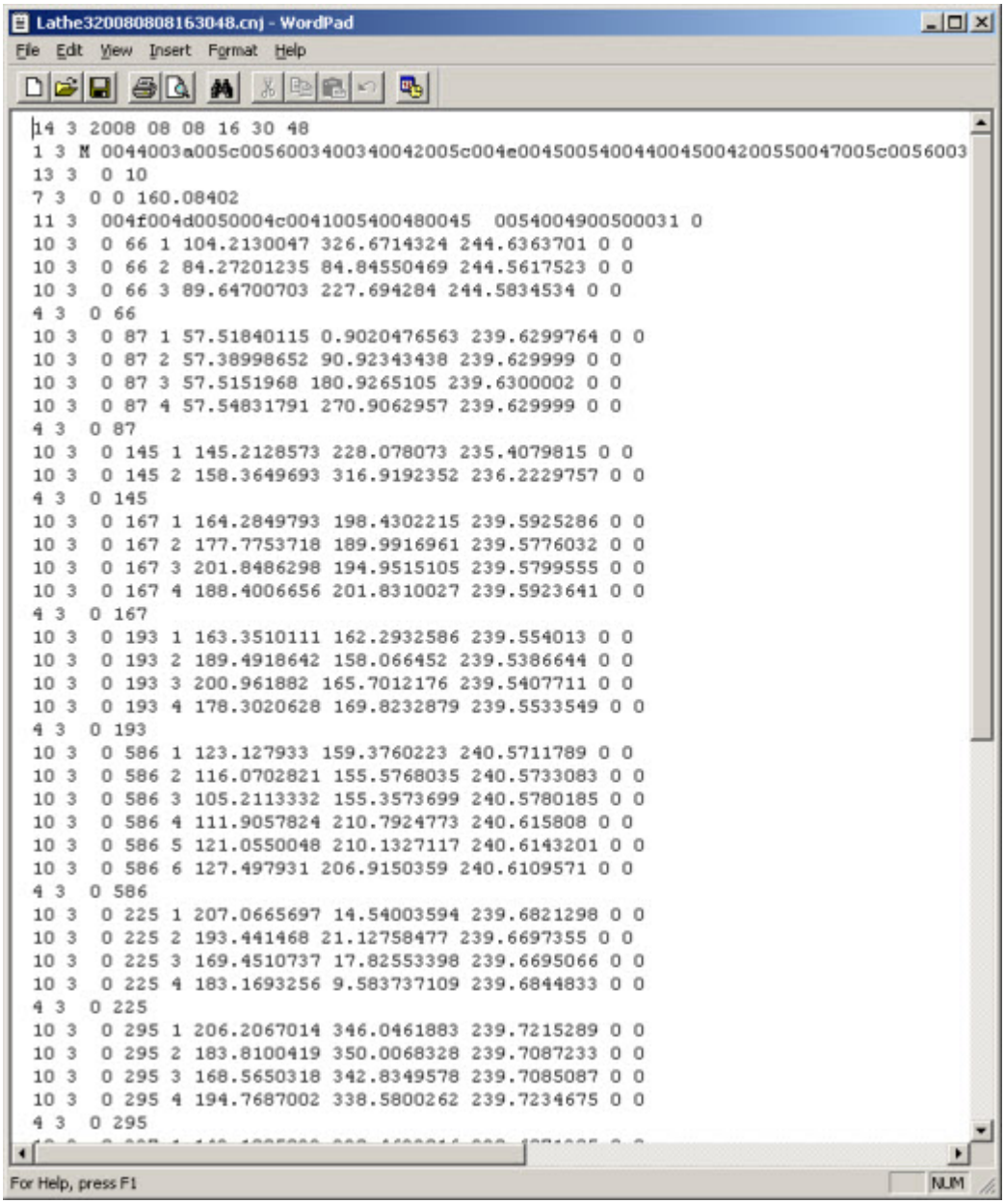
AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	90.373	0.050	0.050	90.363	-0.011	0.000
Y	0.000	0.050	0.050	-0.329	-0.329	0.279
D	9.525	0.050	0.050	9.540	0.015	0.000

MM LOC11 - CIR10

AX	NOMINAL	+TOL	-TOL	MEAS	DEV	OUTTOL
X	64.973	0.050	0.050	65.003	0.030	0.000
Y	0.000	0.050	0.050	-0.250	-0.250	0.200
D	9.525	0.050	0.050	9.540	0.015	0.000

Two Circle Measurement Comparison - Siemens Lathe results on the left, CMM results on the right

The Journal file is shown below.



```
14 3 2008 08 08 16 30 48
1 3 M 0044003a005c005600340034004200 5c004e0045005400440045004200550047005c0056003
13 3  0 10
7 3  0 0 160.08402
11 3  004f004d0050004c0041005400480045  0054004900500031 0
10 3  0 66 1 104.2130047 326.6714324 244.6363701 0 0
10 3  0 66 2 84.27201235 84.84550469 244.5617523 0 0
10 3  0 66 3 89.64700703 227.694284 244.5834534 0 0
4 3  0 66
10 3  0 87 1 57.51840115 0.9020476563 239.6299764 0 0
10 3  0 87 2 57.38998652 90.92343438 239.629999 0 0
10 3  0 87 3 57.5151968 180.9265105 239.6300002 0 0
10 3  0 87 4 57.54831791 270.9062957 239.629999 0 0
4 3  0 87
10 3  0 145 1 145.2128573 228.078073 235.4079815 0 0
10 3  0 145 2 158.3649693 316.9192352 236.2229757 0 0
4 3  0 145
10 3  0 167 1 164.2849793 198.4302215 239.5925286 0 0
10 3  0 167 2 177.7753718 189.9916961 239.5776032 0 0
10 3  0 167 3 201.8486298 194.9515105 239.5799555 0 0
10 3  0 167 4 188.4006656 201.8310027 239.5923641 0 0
4 3  0 167
10 3  0 193 1 163.3510111 162.2932586 239.554013 0 0
10 3  0 193 2 189.4918642 158.066452 239.5386644 0 0
10 3  0 193 3 200.961882 165.7012176 239.5407711 0 0
10 3  0 193 4 178.3020628 169.8232879 239.5533549 0 0
4 3  0 193
10 3  0 586 1 123.127933 159.3760223 240.5711789 0 0
10 3  0 586 2 116.0702821 155.5768035 240.5733083 0 0
10 3  0 586 3 105.2113332 155.3573699 240.5780185 0 0
10 3  0 586 4 111.9057824 210.7924773 240.615808 0 0
10 3  0 586 5 121.0550048 210.1327117 240.6143201 0 0
10 3  0 586 6 127.497931 206.9150359 240.6109571 0 0
4 3  0 586
10 3  0 225 1 207.0665697 14.54003594 239.6821298 0 0
10 3  0 225 2 193.441468 21.12758477 239.6697355 0 0
10 3  0 225 3 169.4510737 17.82553398 239.6695066 0 0
10 3  0 225 4 183.1693256 9.583737109 239.6844833 0 0
4 3  0 225
10 3  0 295 1 206.2067014 346.0461883 239.7215289 0 0
10 3  0 295 2 183.8100419 350.0068328 239.7087233 0 0
10 3  0 295 3 168.5650318 342.8349578 239.7085087 0 0
10 3  0 295 4 194.7687002 338.5800262 239.7234675 0 0
4 3  0 295
```

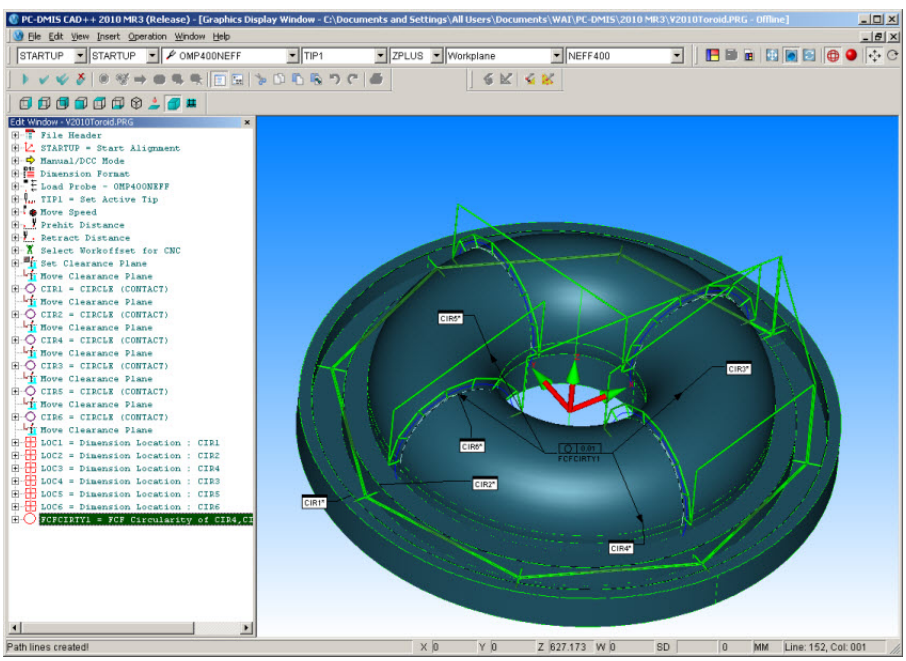
Measurement Journal File

Measuring a Torus

An example of measurement with the 2d Vector Calibration invoked is to measure profiles around the Toroid pictured below and also used for calibration.

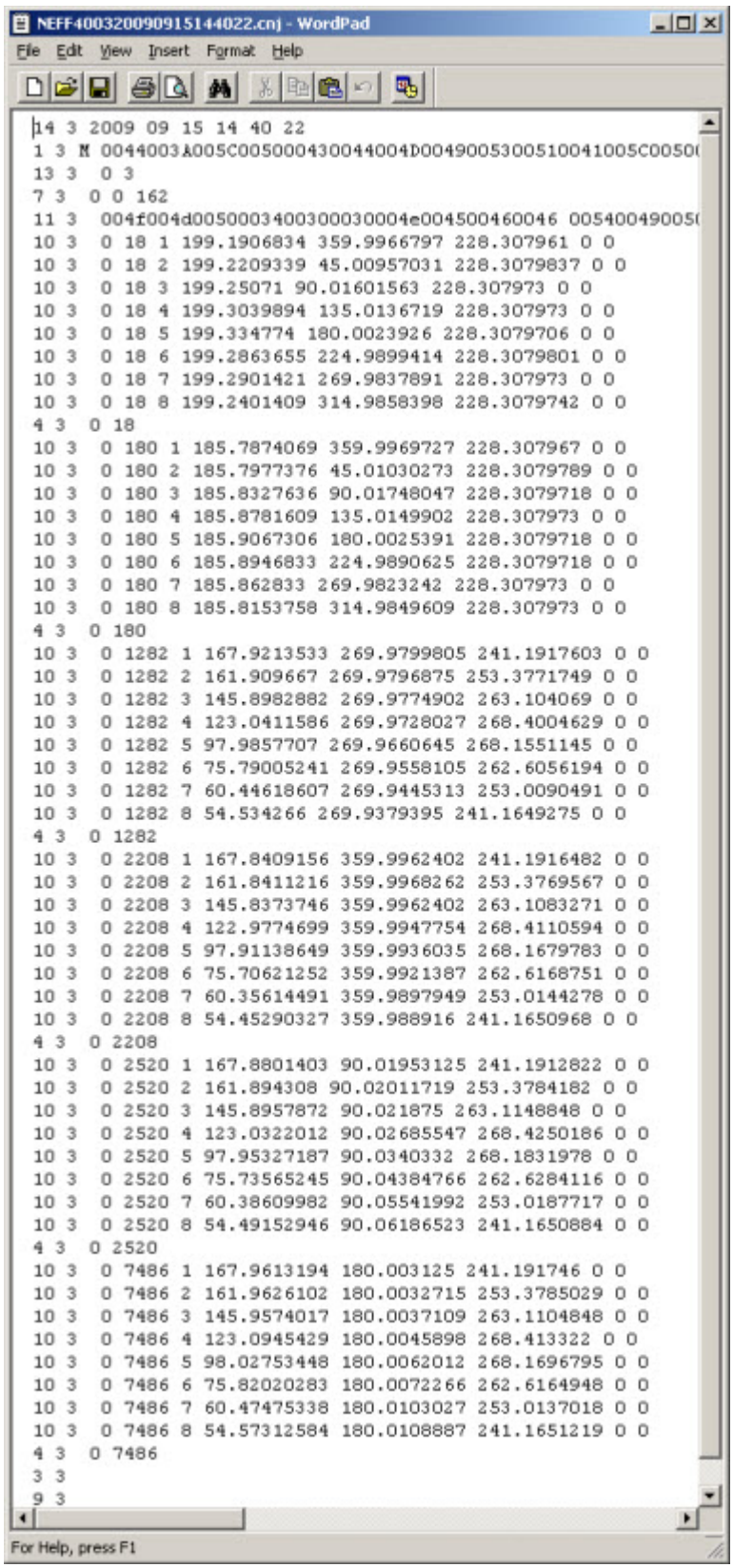
Measurement of a Torus Feature

In this case a separate measurement program is created and the CAD model of the toroid is used.



PC-DMIS Measurement Program and CAD Model for Torus

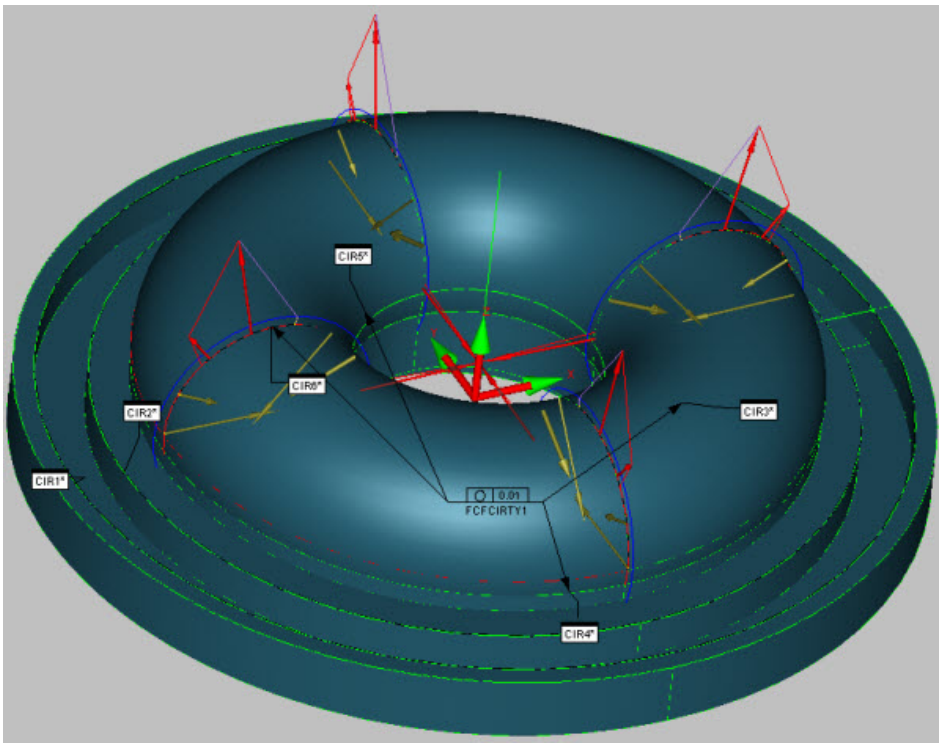
The Journal file produced is shown below.




```
14 3 2009 09 15 14 40 22
1 3 N 0044003A005C005000430044004D004900530051004100 5C0050
13 3  0 3
7 3  0 0 162
11 3  004f004d00500034003000030004e004500460046 00540049005
10 3  0 18 1 199.1906834 359.9966797 228.307961 0 0
10 3  0 18 2 199.2209339 45.00957031 228.3079837 0 0
10 3  0 18 3 199.25071 90.01601563 228.307973 0 0
10 3  0 18 4 199.3039894 135.0136719 228.307973 0 0
10 3  0 18 5 199.334774 180.0023926 228.3079706 0 0
10 3  0 18 6 199.2863655 224.9899414 228.3079801 0 0
10 3  0 18 7 199.2901421 269.9837891 228.307973 0 0
10 3  0 18 8 199.2401409 314.9858398 228.3079742 0 0
4 3  0 18
10 3  0 180 1 185.7874069 359.9969727 228.307967 0 0
10 3  0 180 2 185.7977376 45.01030273 228.3079789 0 0
10 3  0 180 3 185.8327636 90.01748047 228.3079718 0 0
10 3  0 180 4 185.8781609 135.0149902 228.307973 0 0
10 3  0 180 5 185.9067306 180.0025391 228.3079718 0 0
10 3  0 180 6 185.8946833 224.9890625 228.3079718 0 0
10 3  0 180 7 185.862833 269.9823242 228.307973 0 0
10 3  0 180 8 185.8153758 314.9849609 228.307973 0 0
4 3  0 180
10 3  0 1282 1 167.9213533 269.9799805 241.1917603 0 0
10 3  0 1282 2 161.909667 269.9796875 253.3771749 0 0
10 3  0 1282 3 145.8982882 269.9774902 263.104069 0 0
10 3  0 1282 4 123.0411586 269.9728027 268.4004629 0 0
10 3  0 1282 5 97.9857707 269.9660645 268.1551145 0 0
10 3  0 1282 6 75.79005241 269.9558105 262.6056194 0 0
10 3  0 1282 7 60.44618607 269.9445313 253.0090491 0 0
10 3  0 1282 8 54.534266 269.9379395 241.1649275 0 0
4 3  0 1282
10 3  0 2208 1 167.8409156 359.9962402 241.1916482 0 0
10 3  0 2208 2 161.8411216 359.9968262 253.3769567 0 0
10 3  0 2208 3 145.8373746 359.9962402 263.1083271 0 0
10 3  0 2208 4 122.9774699 359.9947754 268.4110594 0 0
10 3  0 2208 5 97.91138649 359.9936035 268.1679783 0 0
10 3  0 2208 6 75.70621252 359.9921387 262.6168751 0 0
10 3  0 2208 7 60.35614491 359.9897949 253.0144278 0 0
10 3  0 2208 8 54.45290327 359.988916 241.1650968 0 0
4 3  0 2208
10 3  0 2520 1 167.8801403 90.01953125 241.1912822 0 0
10 3  0 2520 2 161.894308 90.02011719 253.3784182 0 0
10 3  0 2520 3 145.8957872 90.021875 263.1148848 0 0
10 3  0 2520 4 123.0322012 90.02685547 268.4250186 0 0
10 3  0 2520 5 97.95327187 90.0340332 268.1831978 0 0
10 3  0 2520 6 75.73565245 90.04384766 262.6284116 0 0
10 3  0 2520 7 60.38609982 90.05541992 253.0187717 0 0
10 3  0 2520 8 54.49152946 90.06186523 241.1650884 0 0
4 3  0 2520
10 3  0 7486 1 167.9613194 180.003125 241.191746 0 0
10 3  0 7486 2 161.9626102 180.0032715 253.3785029 0 0
10 3  0 7486 3 145.9574017 180.0037109 263.1104848 0 0
10 3  0 7486 4 123.0945429 180.0045898 268.413322 0 0
10 3  0 7486 5 98.02753448 180.0062012 268.1696795 0 0
10 3  0 7486 6 75.82020283 180.0072266 262.6164948 0 0
10 3  0 7486 7 60.47475338 180.0103027 253.0137018 0 0
10 3  0 7486 8 54.57312584 180.0108887 241.1651219 0 0
4 3  0 7486
3 3
9 3
```

Torus Journal File

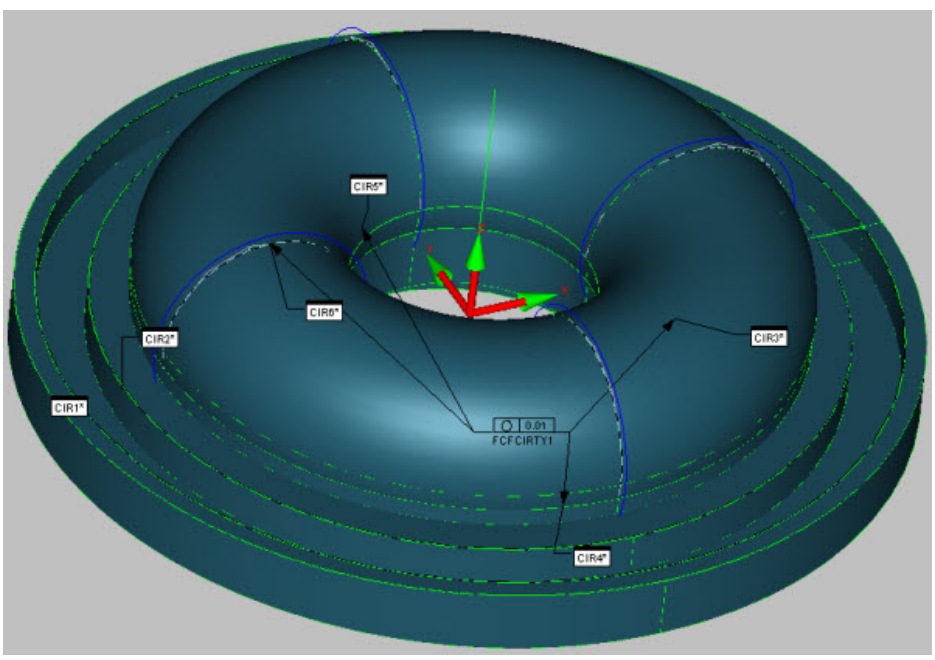
When calibrated without the use of 2d Vector Calibration, the Circularity of the four profiles measured are imperfect as shown graphically below.



Torus Circularity Profile with no 2d Vector Compensation

Note: Refer to the caveat mentioned in the calibration section when running Journal files to switch between 2d Vector Calibration and non-2d Vector Calibrations.

When using 2d Vector Calibration, the Circularity deviation of all four profiles is reduced from 119 microns to 5 microns as shown below.

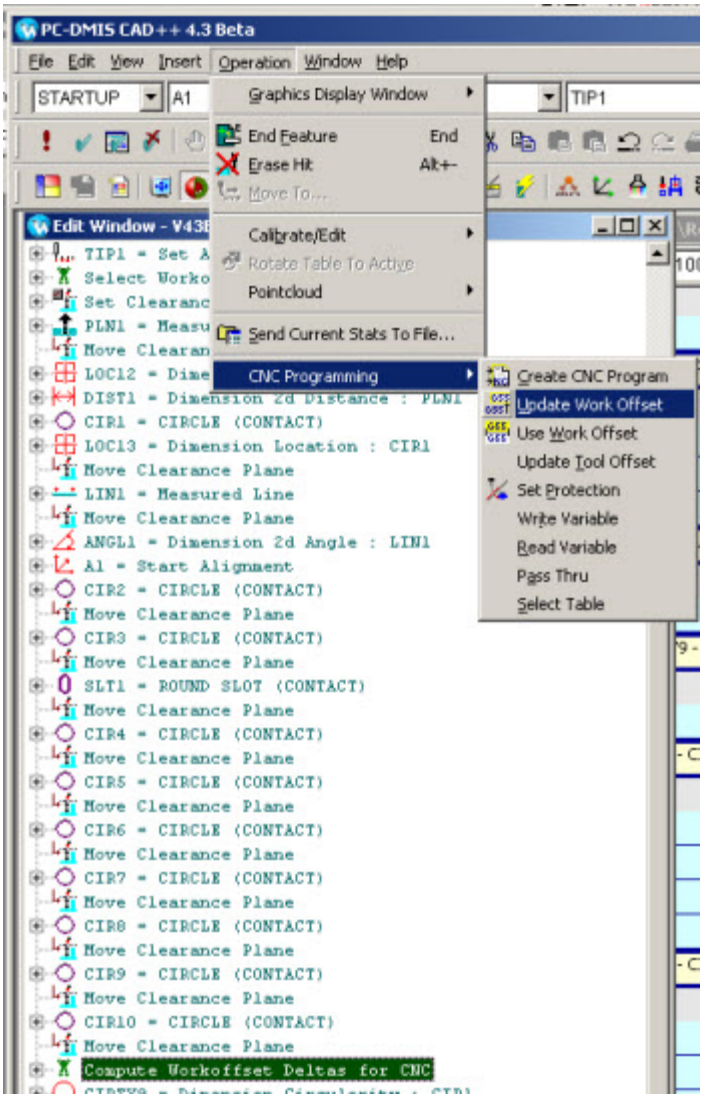


Torus Circularity Profile with 2d Vector Compensation

Update Work Offset for a Siemens Lathe

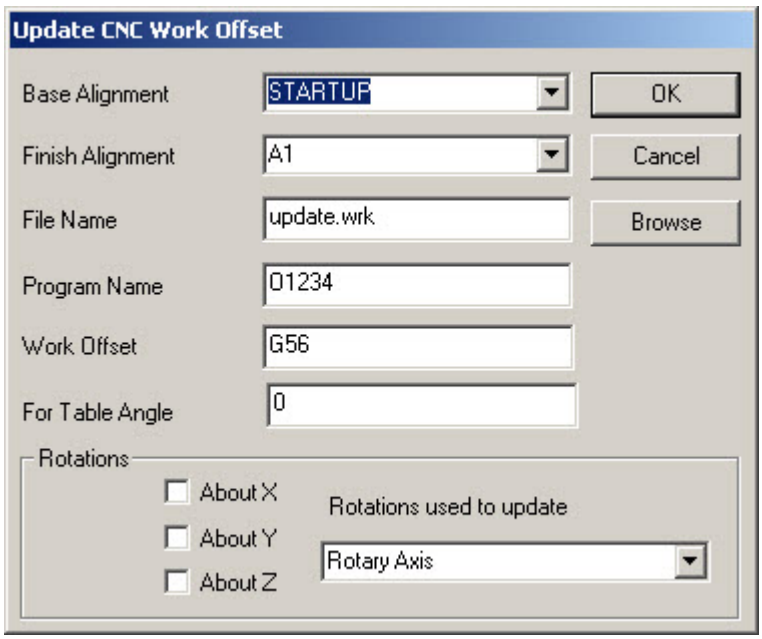
Rotational alignment by eye to better than 0.4 degree is good enough to run the part program described in the Measuring section initially. Update Work Offset can be used to adjust G56 to make it more accurate.

To update a Work Offset, first add the feature to the part program as shown below.



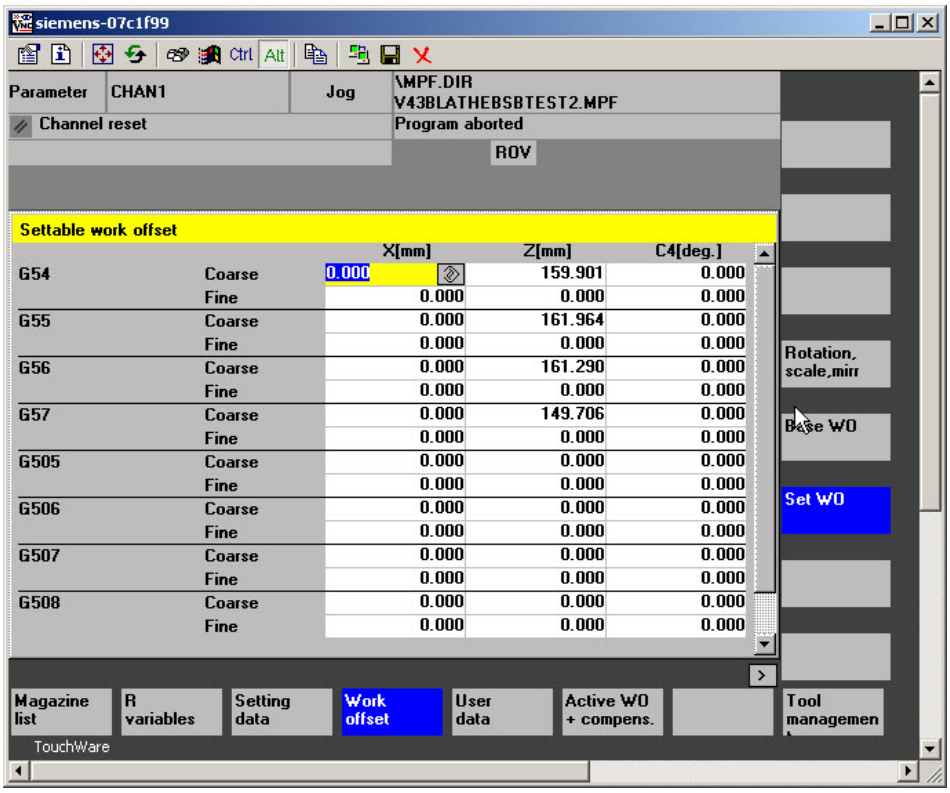
Menu Navigation for Update Work Offset option

Enter the necessary information into the **Update CNC Work Offset** dialog screen.



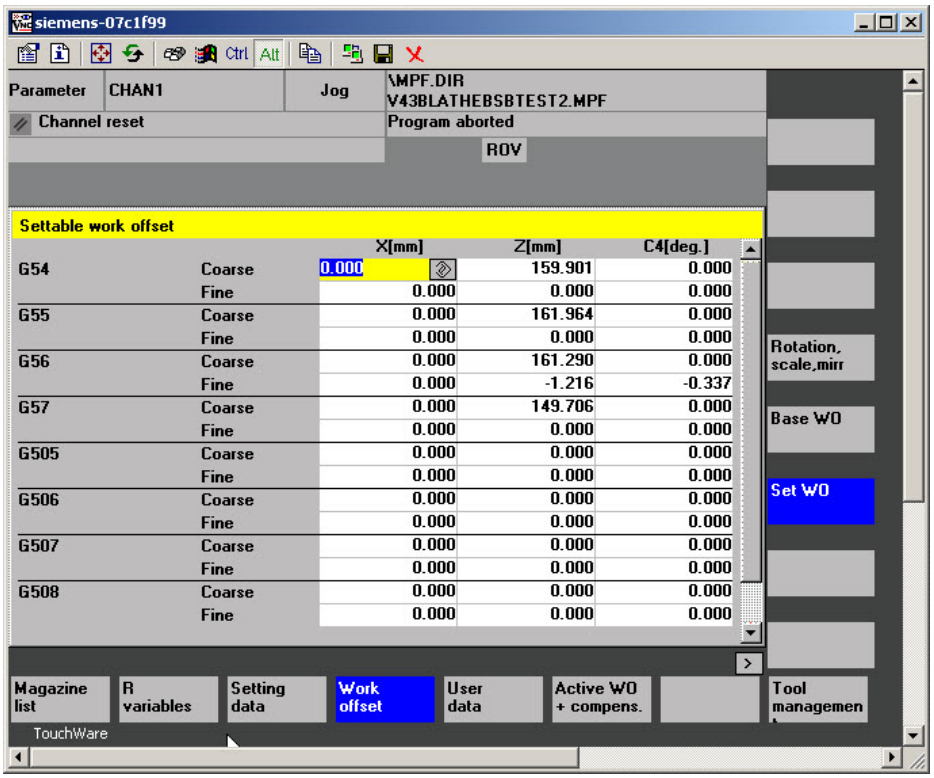
Update CNC Work Offset Dialog Screen

This will update G56 on the Siemens 840D which is initially setup as shown below.



Initial Work Offset Settings for the Siemens 840D Lathe

After running the part program, the Work Offset setting are updated as shown below.

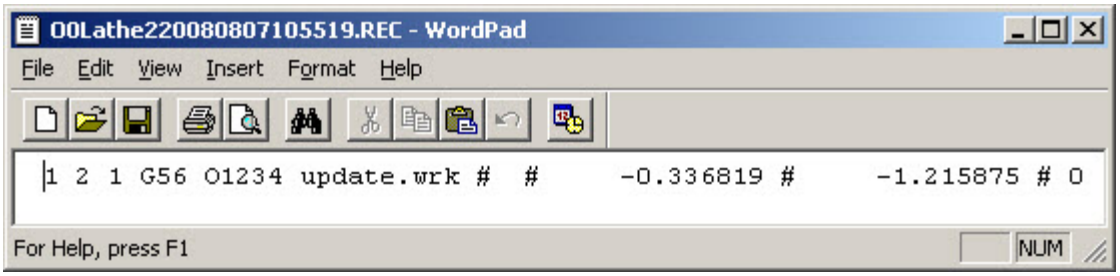


Resulting Work Offset Settings for the Siemens 840D Lathe

Note: The X offset is unchanged as this would compromise the probe tip calibration.

PC-DMIS does this by writing a request file (.req) to the **\pcdmis\req** folder on the 840D controller where a separate application is waiting to receive it.

On success, the request file is renamed to a .rec file - in this case the file looks as follows:



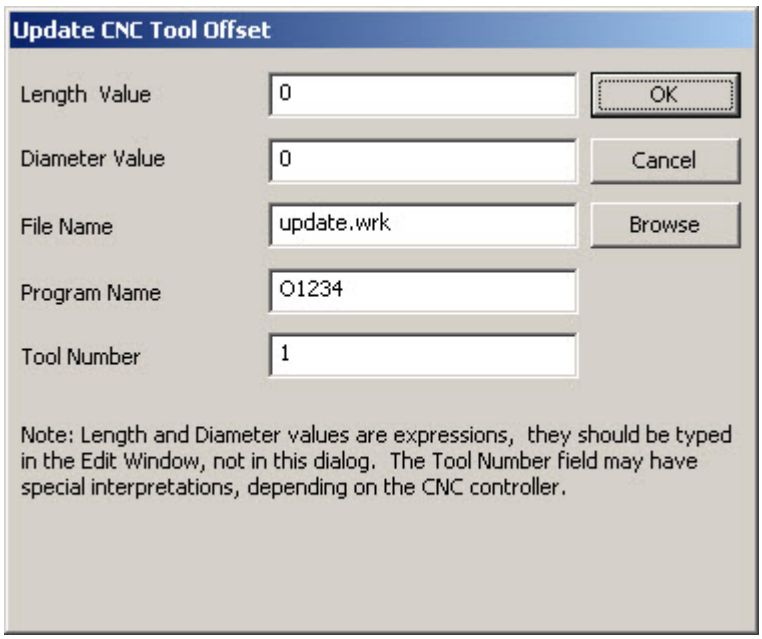
Siemens 840D Update Work Offset Request File

Ignoring the right-most 0 (zero), the six right-most numbers (with # placeholders where omitted) denote the updates to X, Y, Z, A, B, C axes in reverse order. This is modified in the above case by the axis map of XZbC so it becomes X, Z, blank, C axes in reverse order.

Update Tool Length for a Siemens Lathe

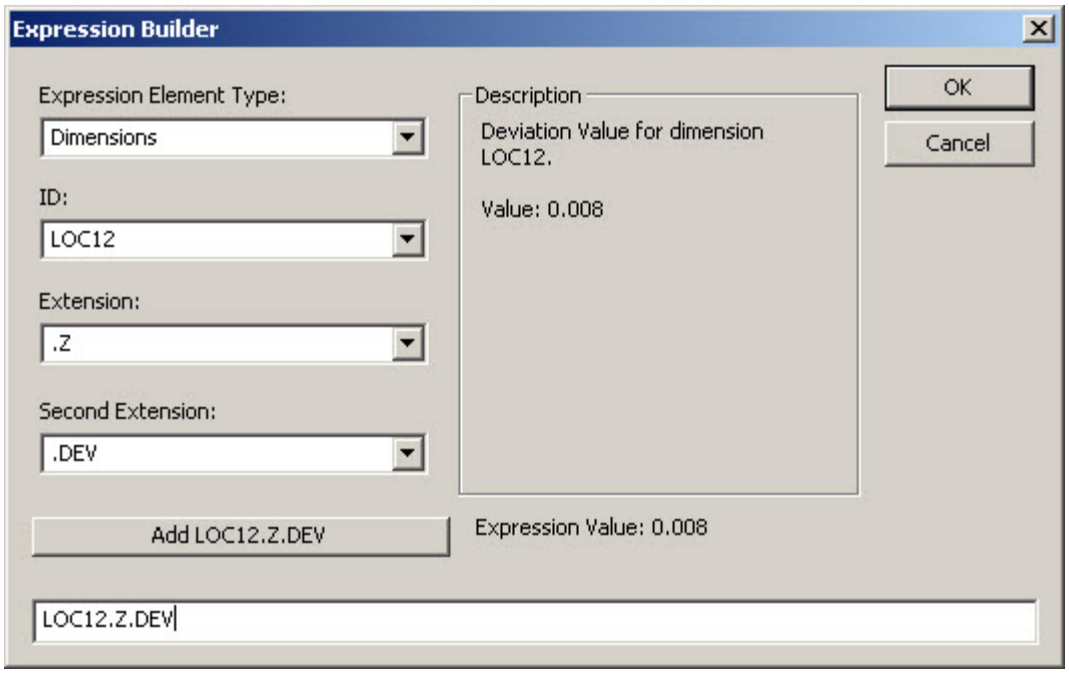
A similar mechanism in the Update Work Offset is employed to do Update Tool Offset.

Open up the Update CNC Tool Offset dialog screen by navigating to Operations | CNC Programming | Update Tool Offset menu option. The Update CNC Tool Offset dialog screen is presented.



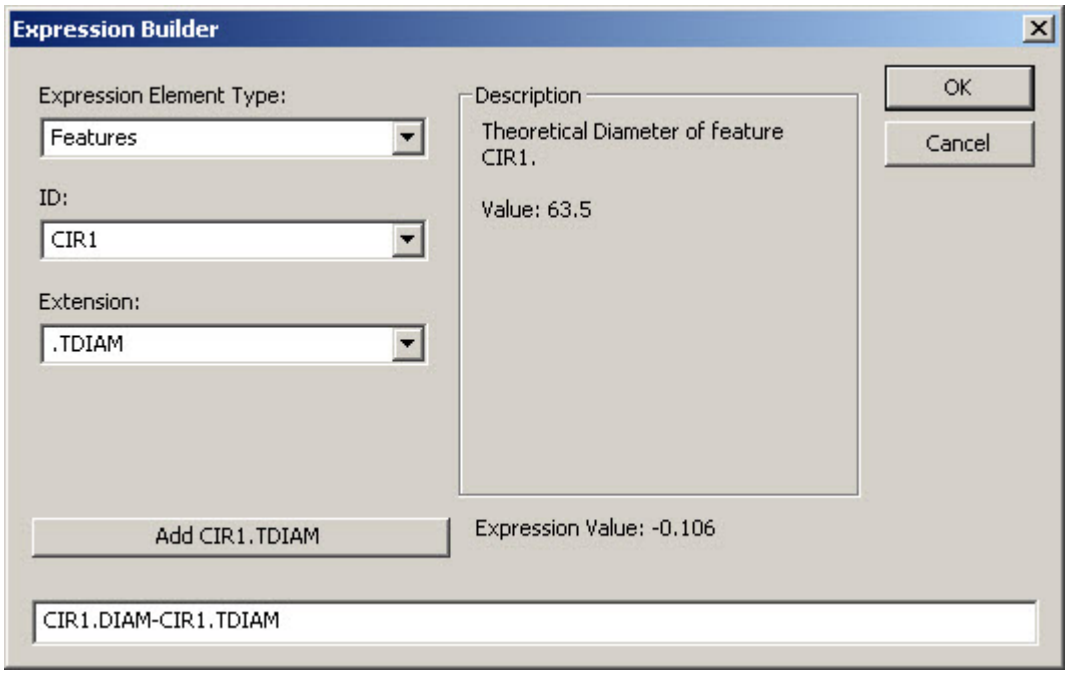
Update CNC Tool Offset Dialog Screen

In this case, the Length and Diameter values are filled in outside of the dialog. Fill in the tool number and click on the **OK** to create the **Update Tool Offset** feature in the part program. Use **\View\Command** mode and navigate to the **Length** field in the feature and invoke the ***Expression Builder*** by clicking on the **F2**. Select the required length feature such as the measured plane and create an expression that computes the difference between nominal and measured values as shown below.



Expression Builder Dialog Screen for Length

Repeat the above procedure for the Diameter as shown below.

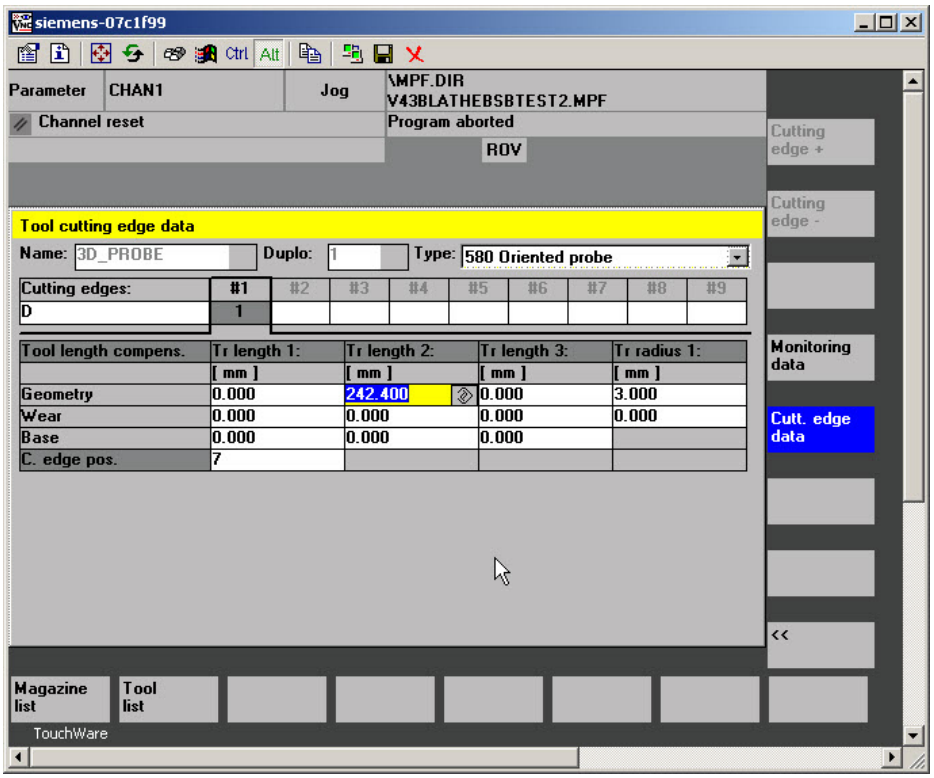


Expression Builder Dialog Screen for Diameter

This results in the following line added in to the part program:

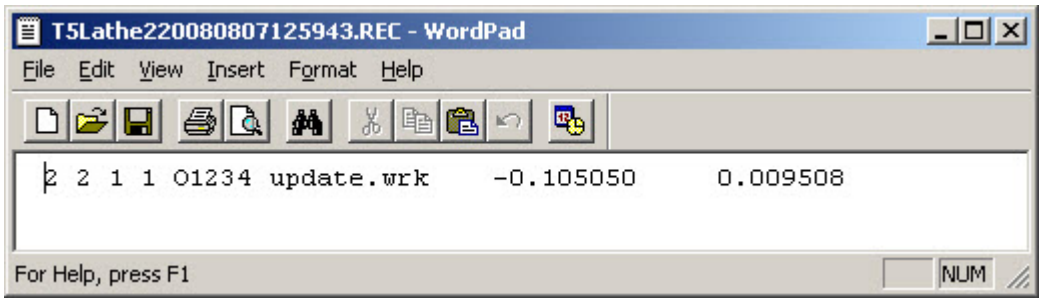
```
CNC/UPDATETOOLOFFSET,LOC12.Z.DEV,CIR1.DIAM-CIR1.TDIAM,update.wrk,O1234,1
```

The Tool Table for Tool 1 which is the OMP400 probe on the 840D is shown below.



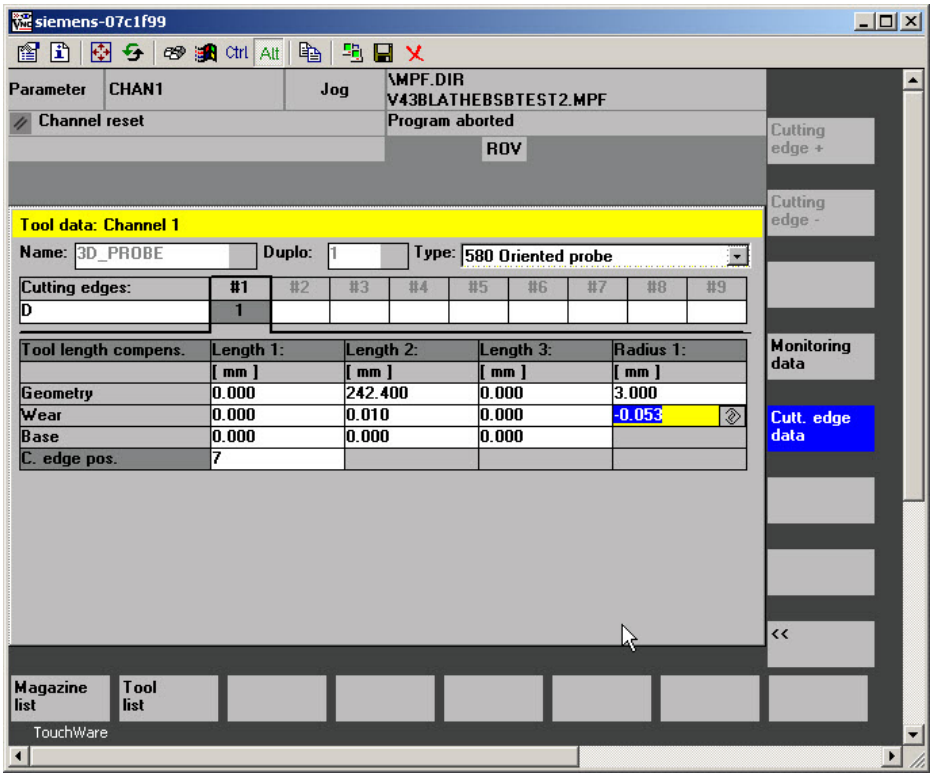
Siemens 840D Initial Tool Offset Tool Table

When executed, the following request file is generated. In this case, the right-most figure is the Tool Length update derived from the Plane Deviation and the Penultimate is the Tool Radius update derived from Circle 1 Diameter Deviation.



Siemens 840D Update Tool Offset Request File

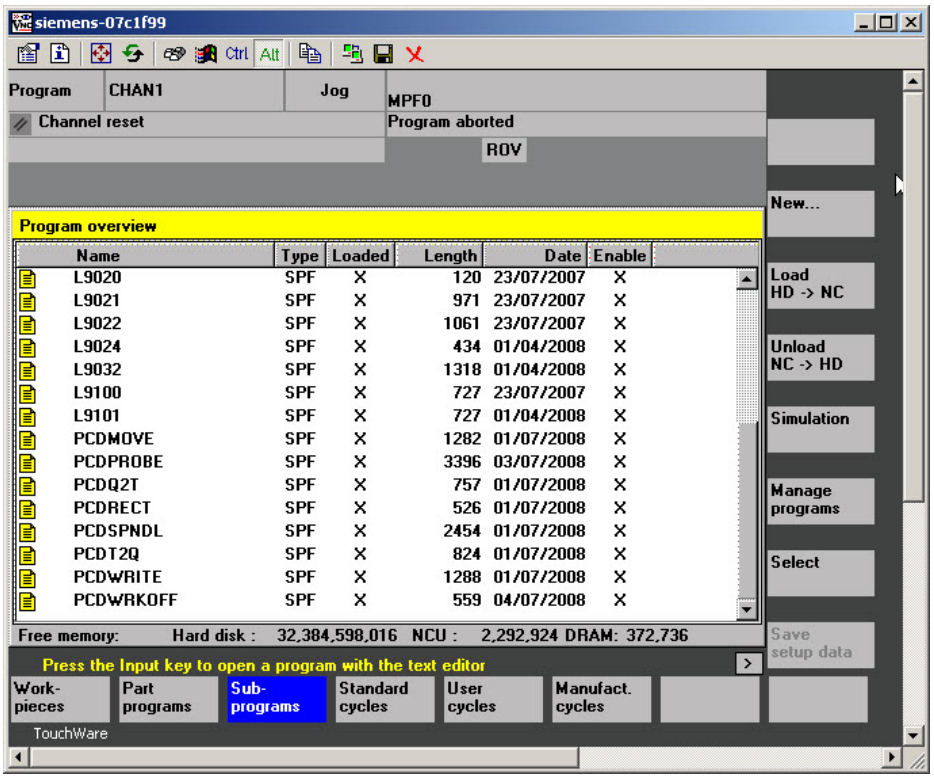
The tool table is updated as shown below.



Siemens 840D Updated Tool Offset Tool Table

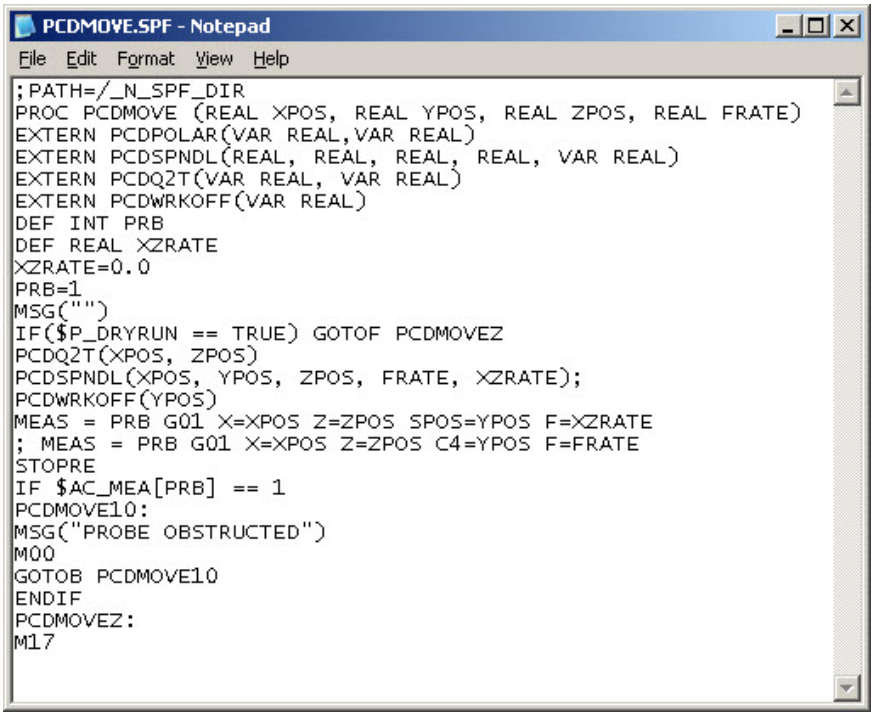
Macros

Map the 840D F-drive to X: on the PC, the macros are downloaded to ***X:\dh\spf.dir***.



Siemens 840D Interface Showing Drive Contents

PCDMOVE.SPF is displayed in a text editor as shown below.



PCDMOVE.SPF File Contents

Note: The reference to the Y axis has been removed. New functions PCDQ2T and PCDSPNDL subtract the XZ Tool Offset values found in Tool Table and work out the correct spindle speed.

The contents of PCDPROBE.SPF is shown below.

PCDPROBE.SPF - Notepad

File Edit Format View Help

```
;PATH=/_N_SPF_DIR
PROC PCDPROBE (REAL XPOS, REAL YPOS, REAL ZPOS, REAL FRATE)
EXTERN PCDT2Q(VAR REAL,VAR REAL)
EXTERN PCDPOLAR(VAR REAL,VAR REAL)
EXTERN PCDRECT(VAR REAL,VAR REAL)
EXTERN PCDSPNDL(REAL, REAL, REAL, REAL, VAR REAL)
EXTERN PCDQ2T(VAR REAL, VAR REAL)
EXTERN PCDWRKOFF(VAR REAL)
EXTERN PCDWRITE(INT,STRING[199])
DEF INT PRB
DEF REAL XPRE, YPRE, ZPRE, MEASFRATE
DEF REAL MEASRATE
DEF REAL XZRATE
XZRATE=0.0
MSG("")
; Set PRB equal to MEAS +/- <channel>
PRB=1
; Set MEASFRATE equal to Measuring velocity for This machine in mm/min
MEASFRATE = 200
IF(PCD_PRGISMETRIC == FALSE)
MEASFRATE = MEASFRATE / 25.4
ENDIF
IF($P_DRYRUN == TRUE)
PCDQ2T(XPOS, ZPOS);
G0 X=XPOS Z=ZPOS F=FRATE
$AA_MW[X]=$AA_IW[X]
$AA_MW[S]=$AA_IW[S]
$AA_MW[Z]=$AA_IW[Z]
GOTOF PCDPROBEZ
ENDIF
STOPRE
XPRE = $AA_IW[X];
YPRE = $AA_IW[S];
ZPRE = $AA_IW[Z];
STOPRE
PCDQ2T(XPOS, ZPOS);
PCDSPNDL(XPOS, YPOS, ZPOS, MEASFRATE, XZRATE);
PCDWRKOFF(YPOS)
;PCDWRITE(PCD_FILE,20<<" "<<5<<" "<<0<<" "<<9999<<" "<<9<<" "<<XPOS<<" "<<YPOS<<" "<<ZPOS<<"
"<<MEASFRATE<<" "<<XZRATE)
MEAS = PRB G01 X=XPOS Z=ZPOS SPOS=YPOS F=XZRATE
; MEAS = PRB G01 X=XPOS Z=ZPOS C4=YPOS F=FRATE
;PCDWRITE(PCD_FILE,20<<" "<<XPOS<<" "<<ZPOS<<" "<<YPOS)
STOPRE
IF $AC_MEA[PRB] == 0
PCDPROBE10:
MSG("PROBE FAILURE")
M00
GOTOB PCDPROBE10:
ELSE
PCD_VAR[1]=$AA_MW[X]
PCD_VAR[2]=$AA_MW[S]
PCD_VAR[3]=$AA_MW[Z]
;PCDWRITE(PCD_FILE,20<<" "<<5<<" "<<0<<" "<<9999<<" "<<9<<" "<<PCD_VAR[1]<<" "<<PCD_VAR[2]<<"
"<<PCD_VAR[3])
PCDT2Q( PCD_VAR[1], PCD_VAR[3])
PCDSPNDL(XPRE, YPRE, ZPRE, 10*MEASFRATE, XZRATE);
G01 X=XPRE Z=ZPRE SPOS=YPRE F=XZRATE
; G01 X=XPRE Z=ZPRE C4=YPRE F=FRATE
ENDIF
PCDPROBEZ:
M17
```

PCDPROBE.SPF File Contents

Once again reference to the Y axis has been removed and PCDQ2T and PCDSPNDL have been used. The results are currently stored in a couple of PCDMIS reserved variables PCD_VAR[] and the PCDMIS part program (.mpf) is modified to copy these to the Journal file.

PCDQ2T is as follows. It subtracts the XZ Tool Offset values found in the Tool Table.

PCDQ2T.SPF - Notepad

File Edit Format View Help

```
;PATH=/_N_SPF_DIR
PROC PCDQ2T(VAR REAL XMEAS, VAR REAL ZMEAS)


DEF REAL XPOS
DEF REAL ZPOS
XPOS=XMEAS-$TC_DP3[$P_TOOLNO,$P_TOOL]-$TC_DP12[$P_TOOLNO,$P_TOOL]
ZPOS=ZMEAS-$TC_DP4[$P_TOOLNO,$P_TOOL]-$TC_DP13[$P_TOOLNO,$P_TOOL]
XMEAS=XPOS
ZMEAS=ZPOS
; XMEAS=XMEAS-PCD_VAR[4]
M17
```

PCDQ2T.SPF File Contents

PCDRECT is as follows. It converts from Polar to Cartesian coordinates

PCDRECT.SPF - Notepad

File Edit Format View Help

```
;PATH=/_N_SPF_DIR
PROC PCDRECT(VAR REAL RAD1, VAR REAL RTHETA)



DEF REAL XPOS
DEF REAL YPOS
XPOS=RAD1*COS(RTHETA)/2
YPOS=RAD1*SIN(RTHETA)/2
RAD1=XPOS
RTHETA=YPOS
M17
```

PCDRECT.SPF File Contents

PCDSPNDL is as follows. It calculates Calc Spindle and XZ Feedrate to complete in the same time as the Cartesian XYZ move.

PCDSPNDL.SPF - Notepad

File Edit Format View Help

```
;PATH=/_N_SPF_DIR
PROC PCDSPNDL (REAL XPOS, REAL YPOS, REAL ZPOS, REAL FRATE, VAR REAL XZRATE)
EXTERN PCDRECT(VAR REAL,VAR REAL)
EXTERN PCDWRITE(INT,STRING[199])



DEF REAL MTIME
DEF REAL MANGLE
DEF REAL SPNDLSPD
DEF REAL XCUR, YCUR, ZCUR
DEF REAL XZDIST
XCUR = $AA_IW[X]
YCUR = $AA_IW[S]
ZCUR = $AA_IW[Z]
MANGLE=ABS(YPOS-YCUR)
XZDIST=SQRT(POT((XPOS-XCUR)/2)+POT(ZPOS-ZCUR))
PCDRECT(XPOS, YPOS)
PCDRECT(XCUR, YCUR)
MTIME=SQRT(POT((XPOS-XCUR))+POT(YPOS-YCUR)+POT(ZPOS-ZCUR))/FRATE
IF (MTIME>0)
IF (MANGLE>180.0)
MANGLE=ABS(MANGLE-360.0)
ENDIF
SPNDLSPD=MANGLE/MTIME
FA[S]=SPNDLSPD
XZRATE=XZDIST/MTIME
ENDIF
IF (XZRATE<=0.0)
XZRATE=FRATE
ENDIF
M17
```

PCDSPNDL.SPF File Contents

PCDT2Q is as follows. It adds back in the XZ Tool Offset values found in the Tool Table.

PCDT2Q.SPF - Notepad

File Edit Format View Help

```
;PATH=/_N_SPF_DIR
PROC PCDT2Q(VAR REAL XMEAS, VAR REAL ZMEAS)


DEF REAL XPOS
DEF REAL ZPOS
XPOS=XMEAS+$TC_DP3[$P_TOOLNO,$P_TOOL]+$TC_DP12[$P_TOOLNO,$P_TOOL]
ZPOS=ZMEAS+$TC_DP4[$P_TOOLNO,$P_TOOL]+$TC_DP13[$P_TOOLNO,$P_TOOL]
XMEAS=XPOS
ZMEAS=ZPOS
;XMEAS=XMEAS-PCD_VAR[4]
; XMEAS=XMEAS+2*PCD_VAR[4]
M17
```

PCDT2Q.SPF File Contents

PCDWRITE.SPF is not modified. It displays the Journal file contents.

PCDWRITE.SPF - Notepad

File Edit Format View Help

```
;PATH=/_N_SPF_DIR
PROC PCDWRITE (INT FNUM,STRING[199] FSTG)



DEF STRING[100]FRES
DEF INT _FWERR = 0
FRES="/_N_WKS_DIR/_N_PCDMIS_WPD/_N_RESULT"
MSG("")
IF FNUM >=0 GOTOF PCDWRITE1
DELETE(_FWERR,FRES)
IF _FWERR == 0 GOTOF E_PCDWRITE
IF _FWERR == 3 GOTOF E_PCDWRITE ; File not found, ok
MSG("FILE DELETE ERROR " << _FWERR)
GOTOF PCDWRITE2
PCDWRITE1:

WRITE(_FWERR,FRES,FSTG)
IF _FWERR == 0 GOTOF E_PCDWRITE
MSG("FILE WRITE ERROR " << _FWERR)
PCDWRITE2:
G4F3
E_PCDWRITE:
M17
```

PCDWRITE.SPF File Contents

PCDWRKOF.SPF is as follows. It adds the C Axis Angle to the Work Offset since the Siemens NEF400 system will not.

PCDWRKOF.SPF - Notepad

File Edit Format View Help

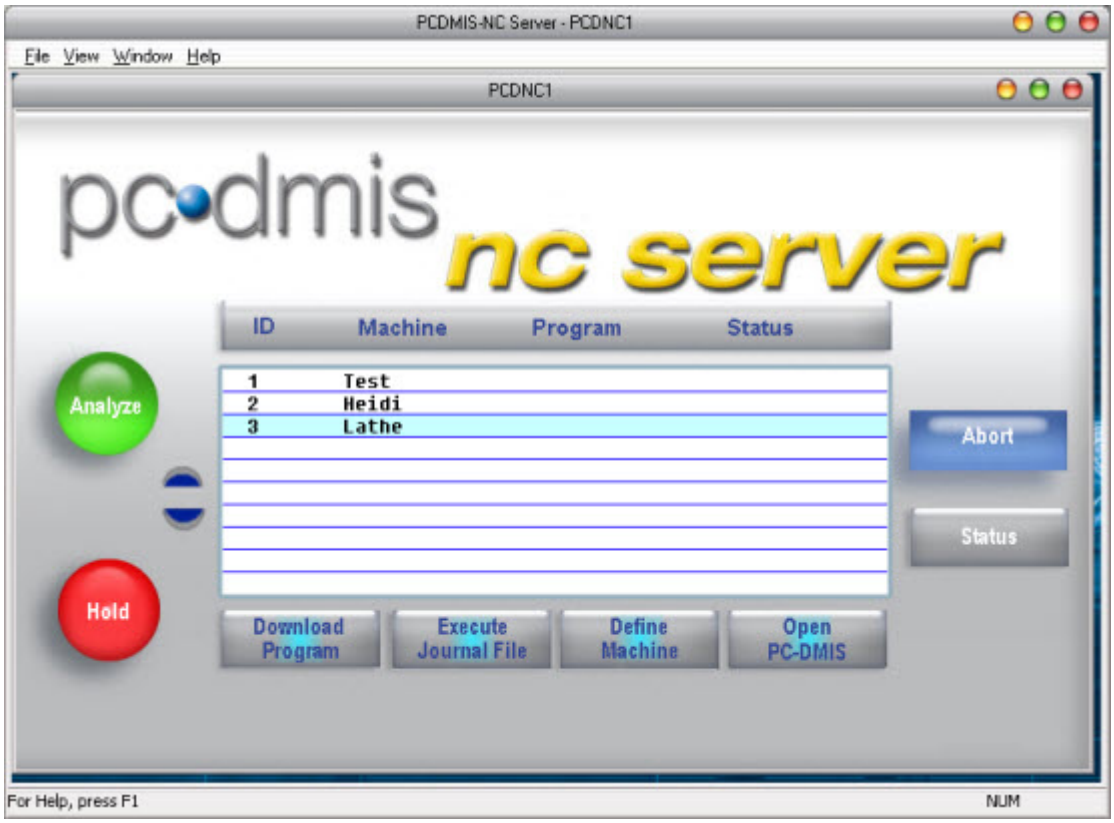
```
;PATH=/_N_SPF_DIR
PROC PCDWRKOFF(VAR REAL RTHETA)


DEF REAL WRKOFF
WRKOFF = $P_ACTFRAME[C44,TR]
WRKOFF = WRKOFF + $P_ACTFRAME[C44,FI]
RTHETA = RTHETA+WRKOFF
IF (RTHETA<0.0)
RTHETA = RTHETA+360.0
ELSE
IF (RTHETA>=360.0)
RTHETA = RTHETA-360.0
ENDIF
ENDIF
M17
```

PCDWRKOF.SPF File Contents

CncServer Configuration

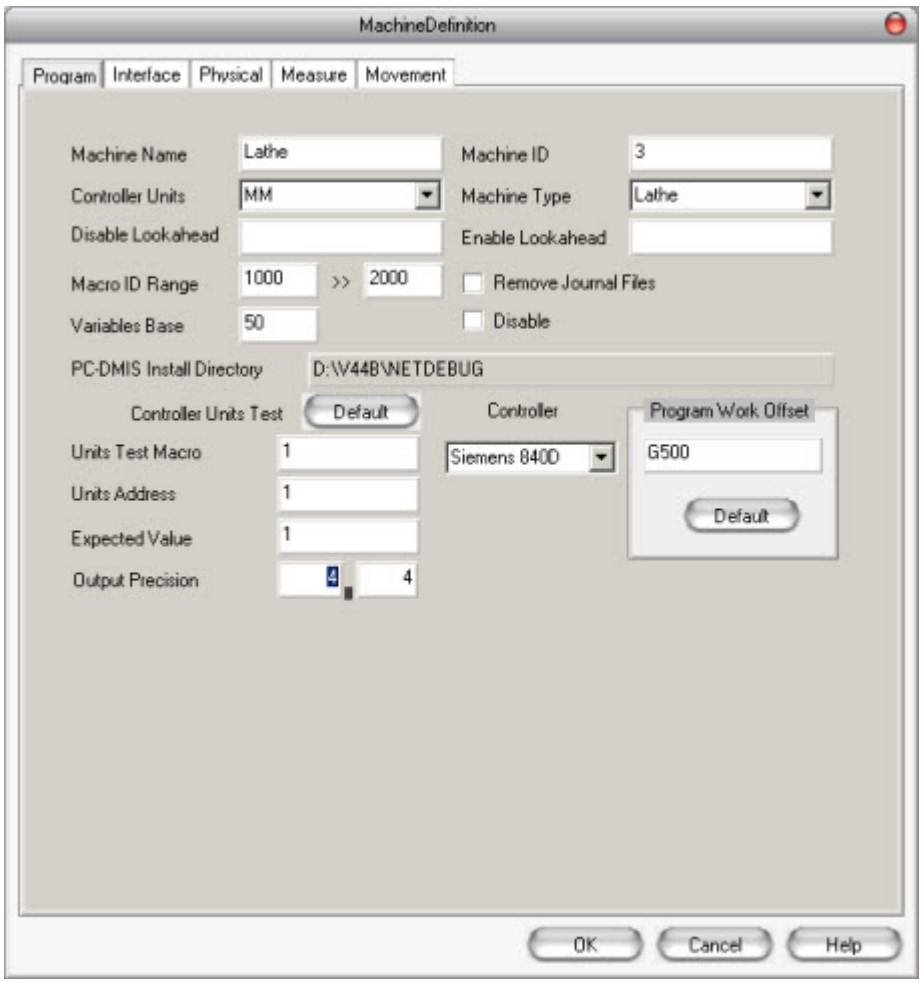
C Axis Lathe (NEF400)

A single lathe entry is made in **CncServer** as shown below. Select the Machine from the list.

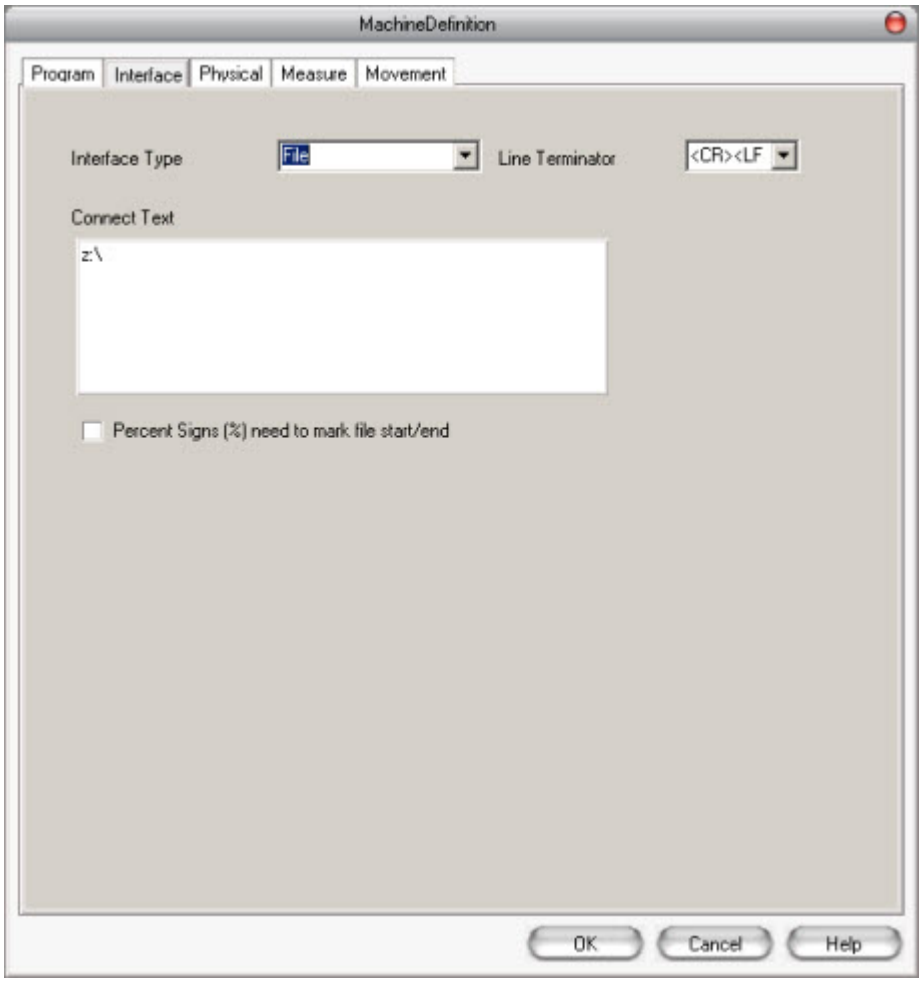


CncServer Dialog Screen

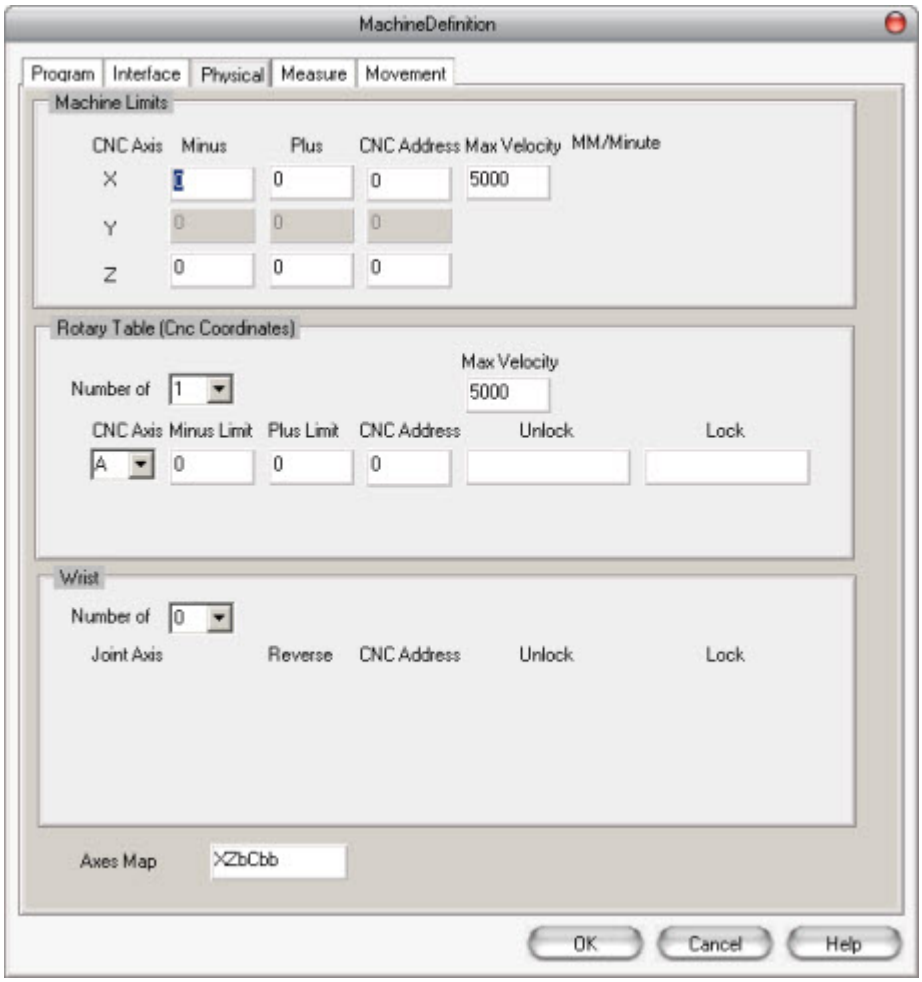
Click the **Define Machine** button to open the lathe definitions as shown in the following **MachineDefinition** dialog screens. See the Creating and Modifying Machines topic for a more detailed description of each of the **MachineDefinition** tab pages.



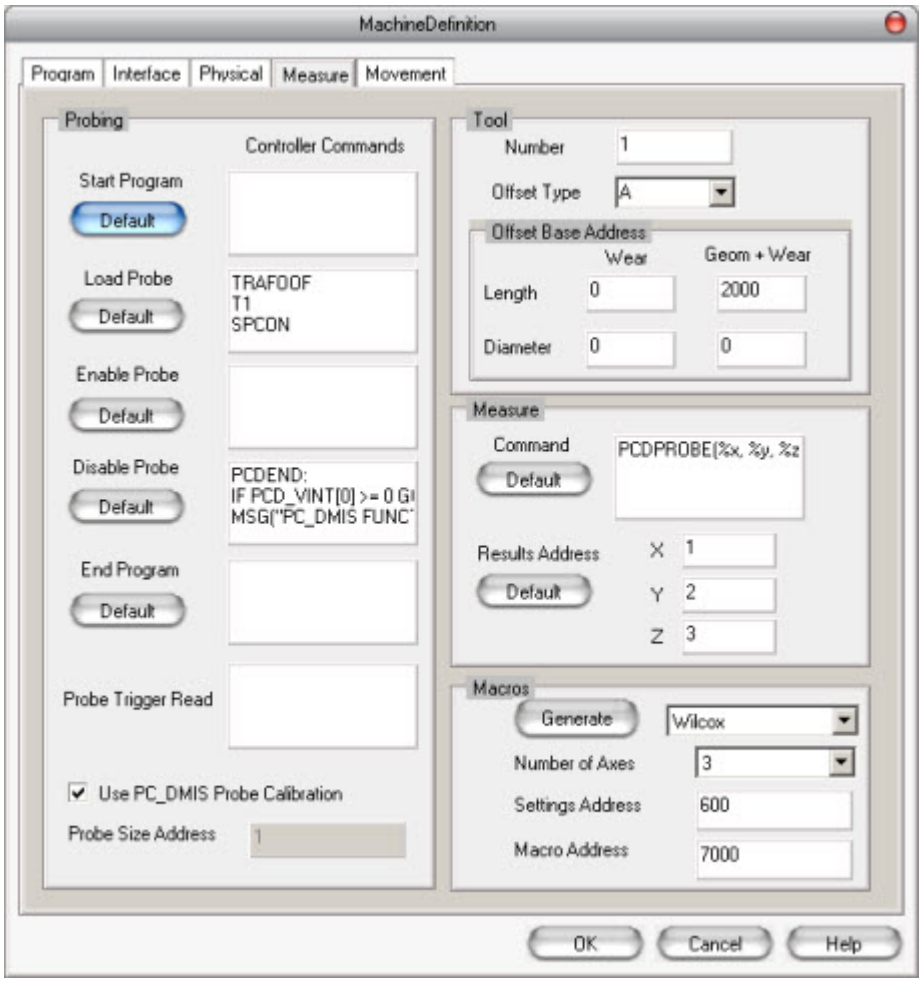
MachineDefinition Program Tab



MachineDefinition Interface Tab



MachineDefinition Physical Tab



MachineDefinition Measure Tab

Details of the Load and Disable sections of the **Measure** tab are described below.

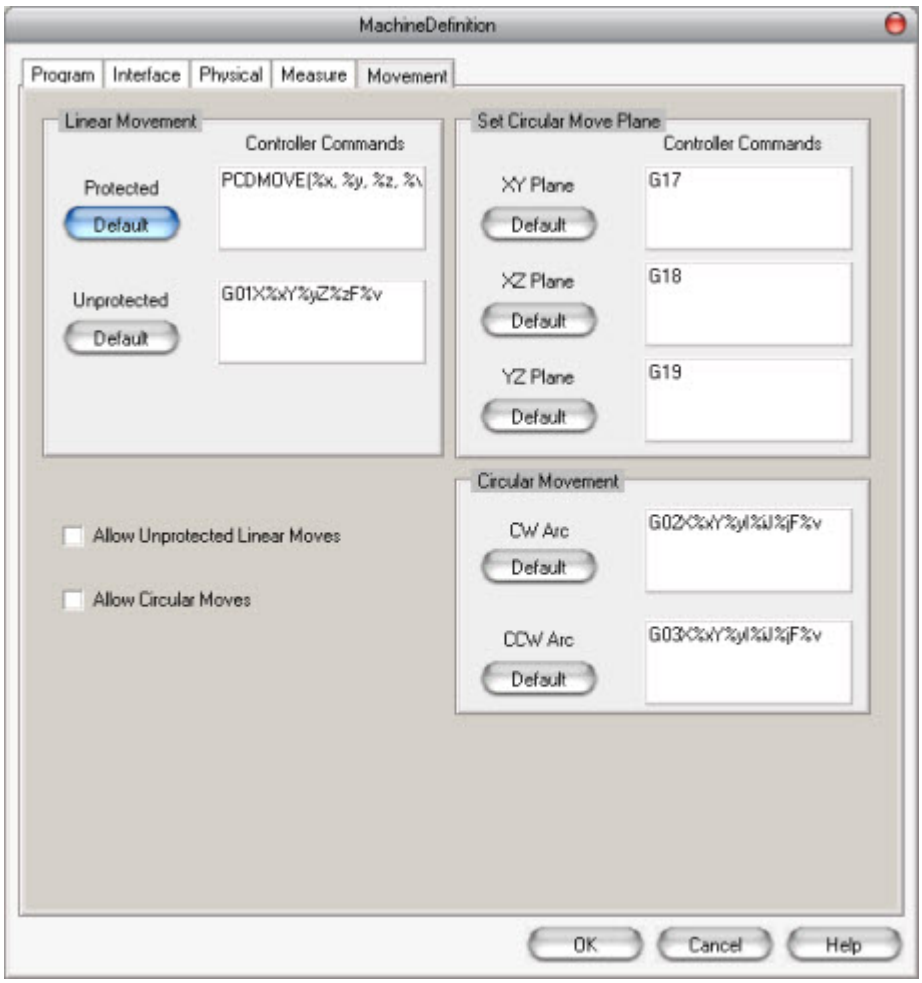
Load Probe Section

```
TRAFOOF
T1
SPCON
M70
SPOS=DC(0)
G94
```

Disable Probe Section

```
PCDEND:
IF PCD_VINT[0] >= 0 GOTOF PCDENDX
MSG("PC_DMIS FUNCTION ERROR "<<PCD_VINT[0])
M0
PCDENDX:
PCDENDZ:
M30
```

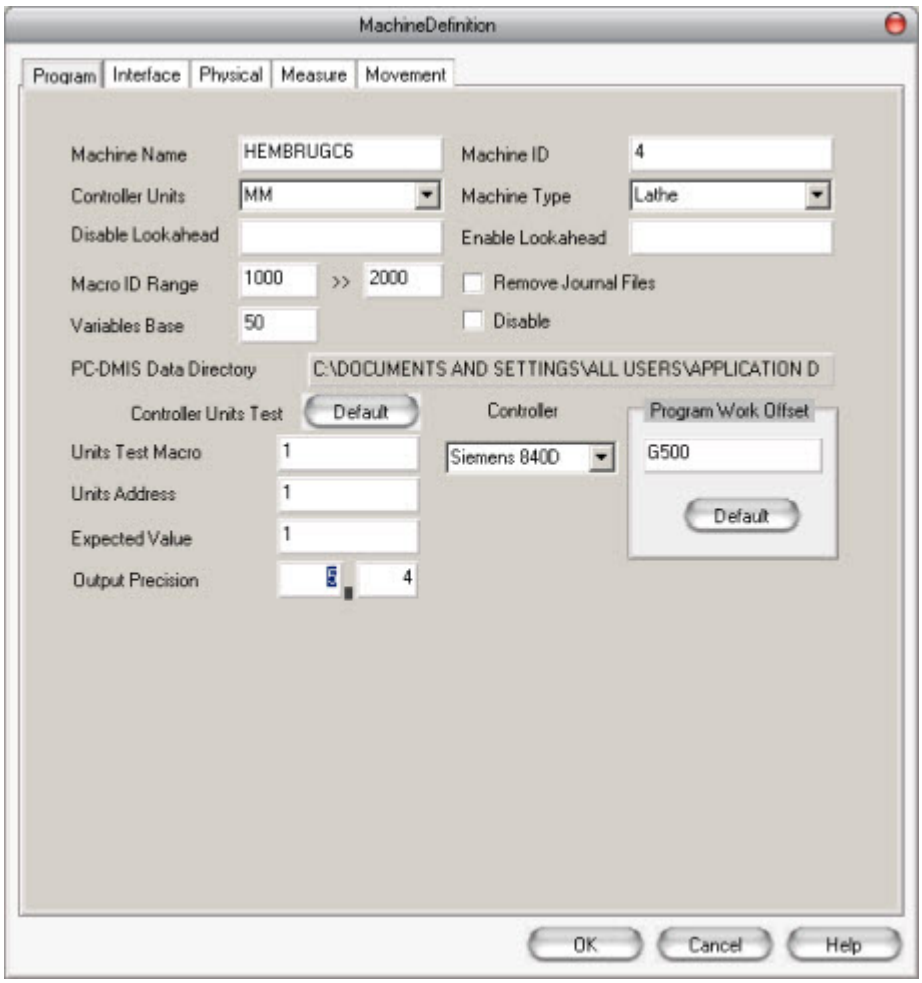
If an error message prevents the above changes from being accepted, go back to the **Program** tab and select “Vertical” instead of “Lathe”. Make the necessary changes then save them and only then select “Lathe”.



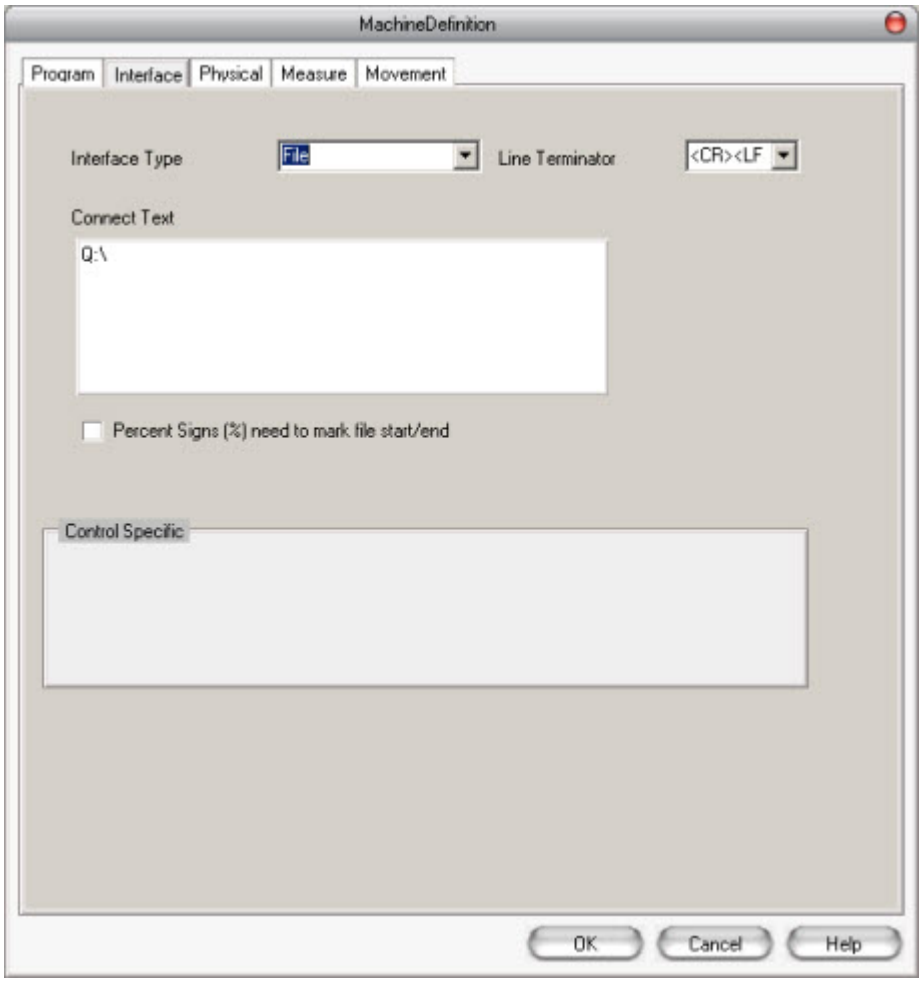
MachineDefinition Movement Tab

Extended C Axis Lathe (Hembrug)

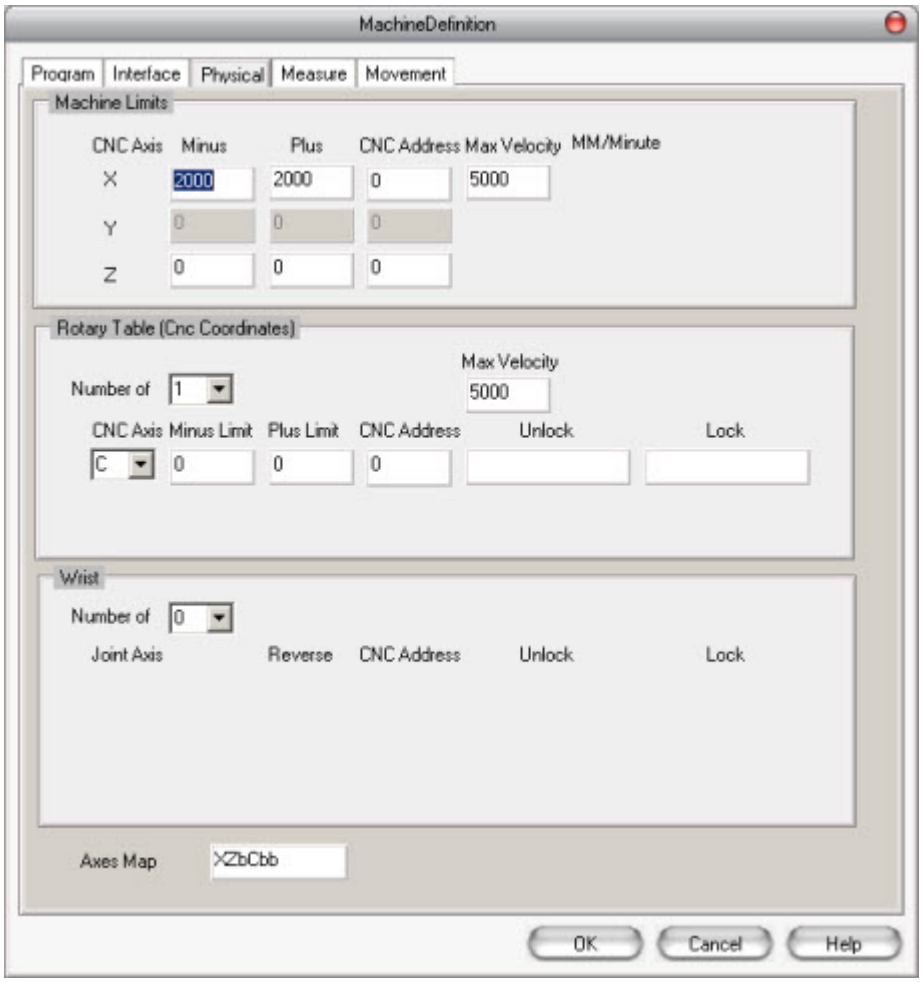
The following **CncServer MachineDefinition** screen shots can be used to define the settings when using an Extended C Axis Hembrug Lathe as an example.



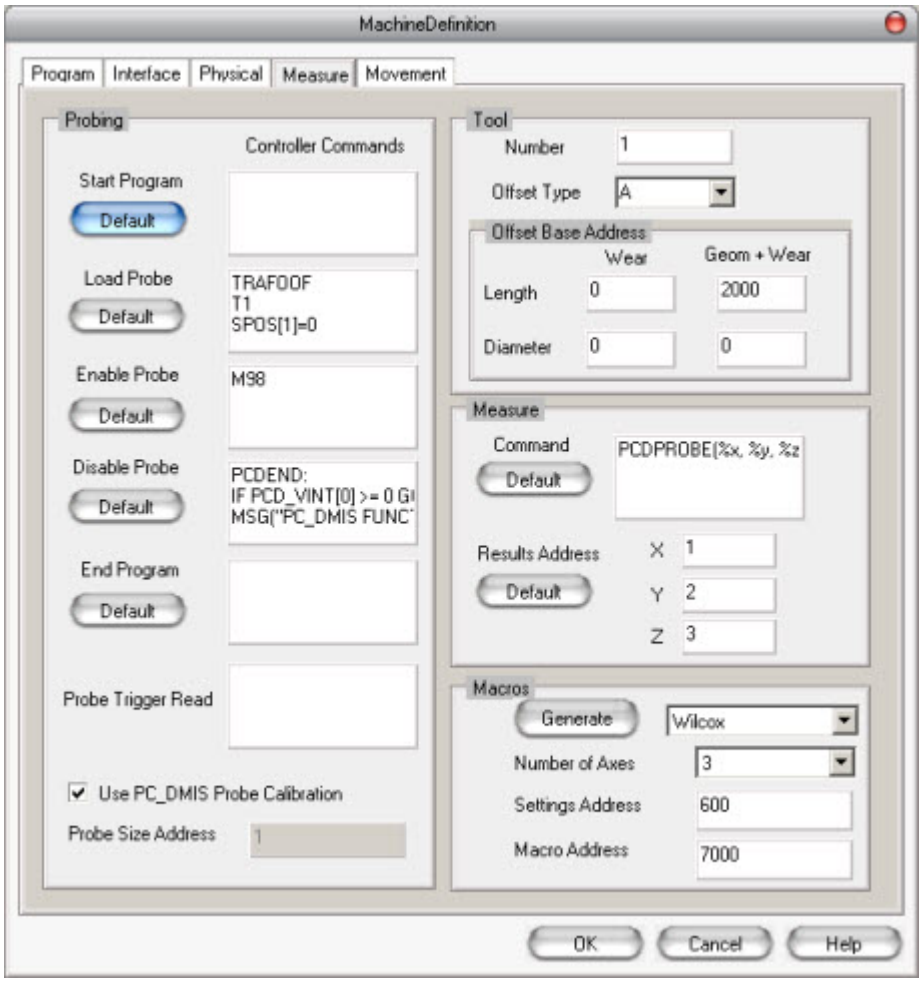
MachineDefinition Program Tab for Hembrug Lathe



MachineDefinition Interface Tab for Hembrug Lathe



MachineDefinition Physical Tab for Hembrug Lathe



MachineDefinition Measure Tab for Hembrug Lathe

Details of the Load, Enable and Disable sections of the **Measure** tab are described below.

Load Probe Section

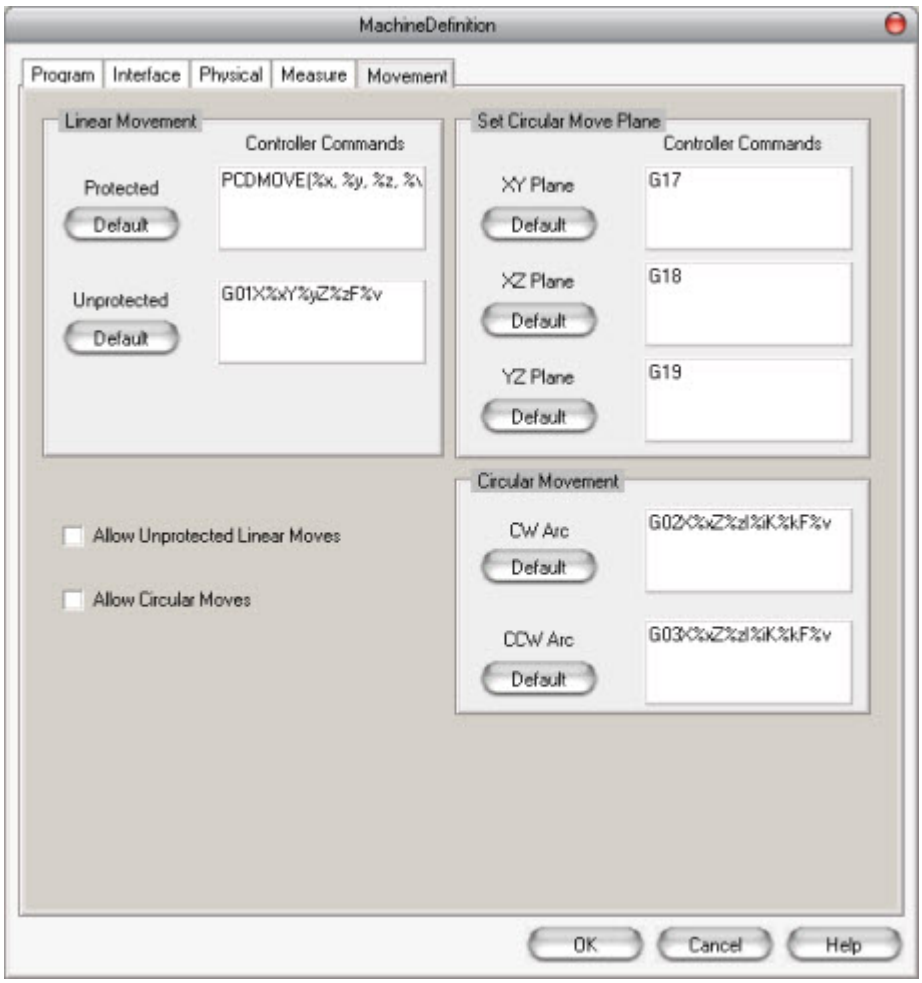
```
TRAFOOF
T1
SPOS[1]=0
DIAMON
G94
```

Enable Probe Section

```
M98
```

Disable Probe Section

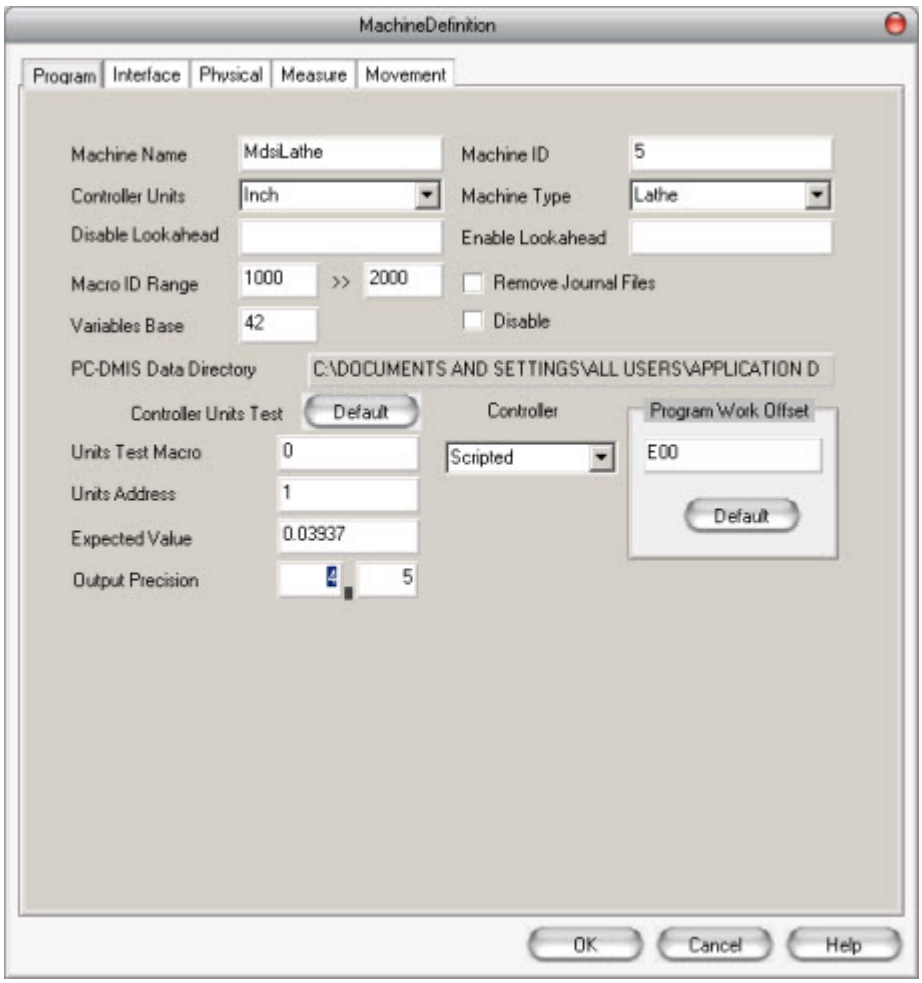
```
PCDEND:
IF PCD_VINT[0] >= 0 GOTOF PCDENDX
MSG("PC_DMIS FUNCTION ERROR "<<PCD_VINT[0])
M0
PCDENDX:
PCDENDZ:
M05
M99
M30
```



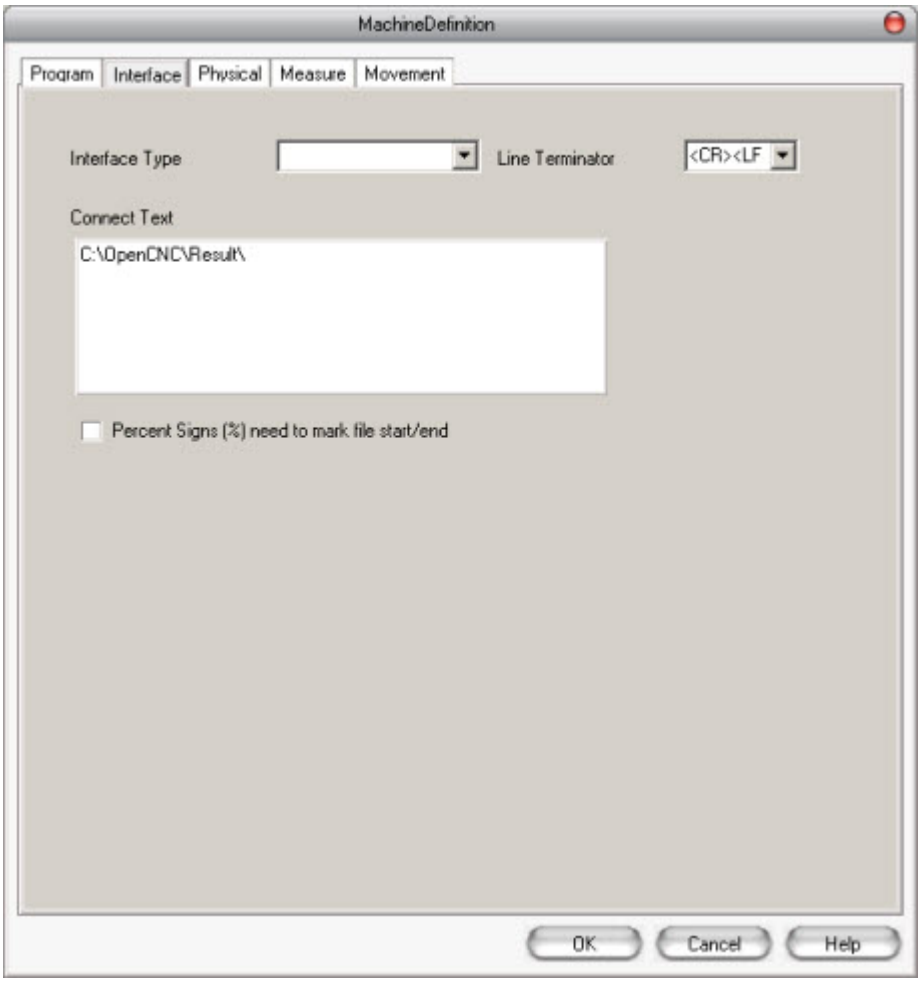
MachineDefinition Movement Tab for Hembrug Lathe

Extended Basic C Axis Lathe (MDSI)

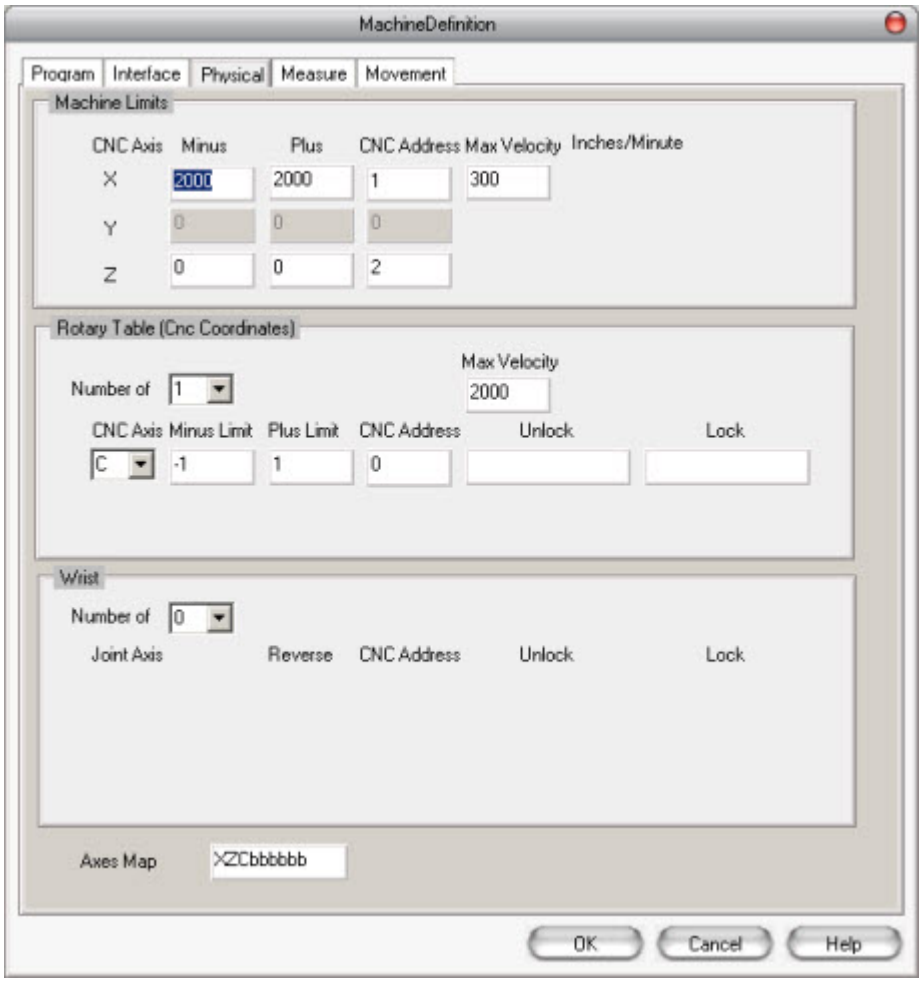
The following **CncServer MachineDefinition** screen shots can be used to define the settings when using an Extended Basic C Axis MDSI Lathe as an example.



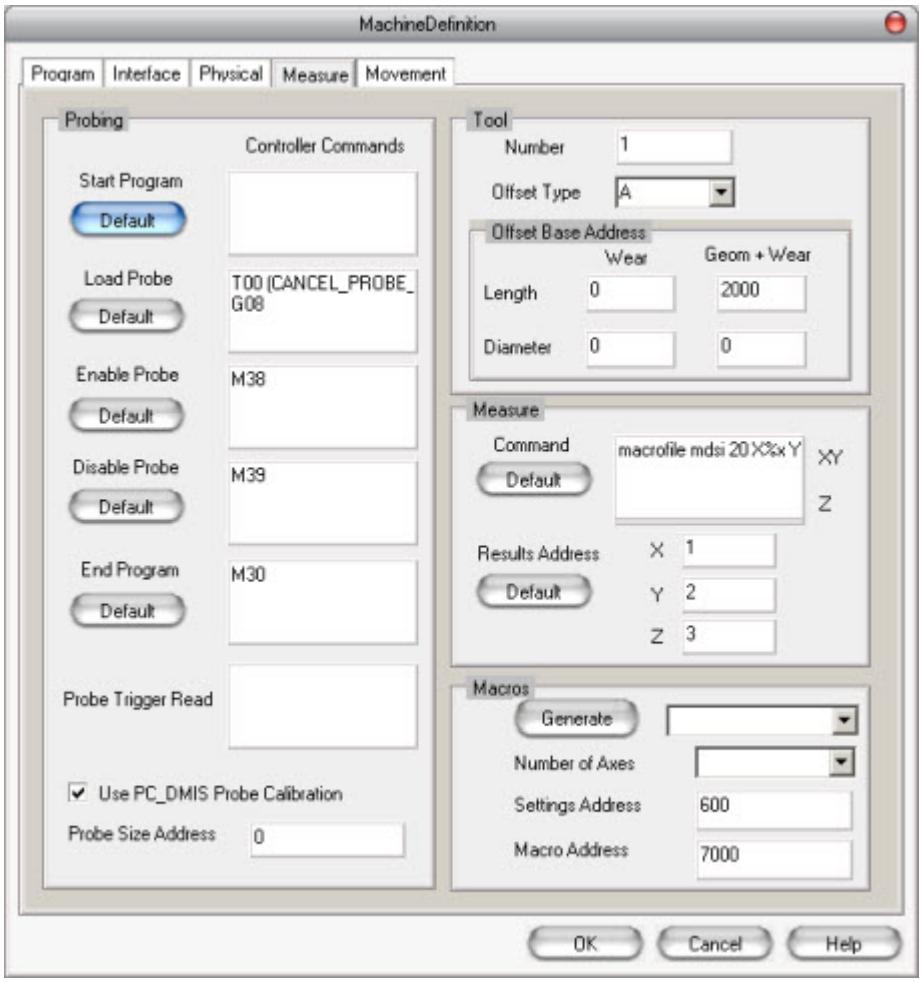
MachineDefinition Program Tab for an Extended Basic MDSI Lathe



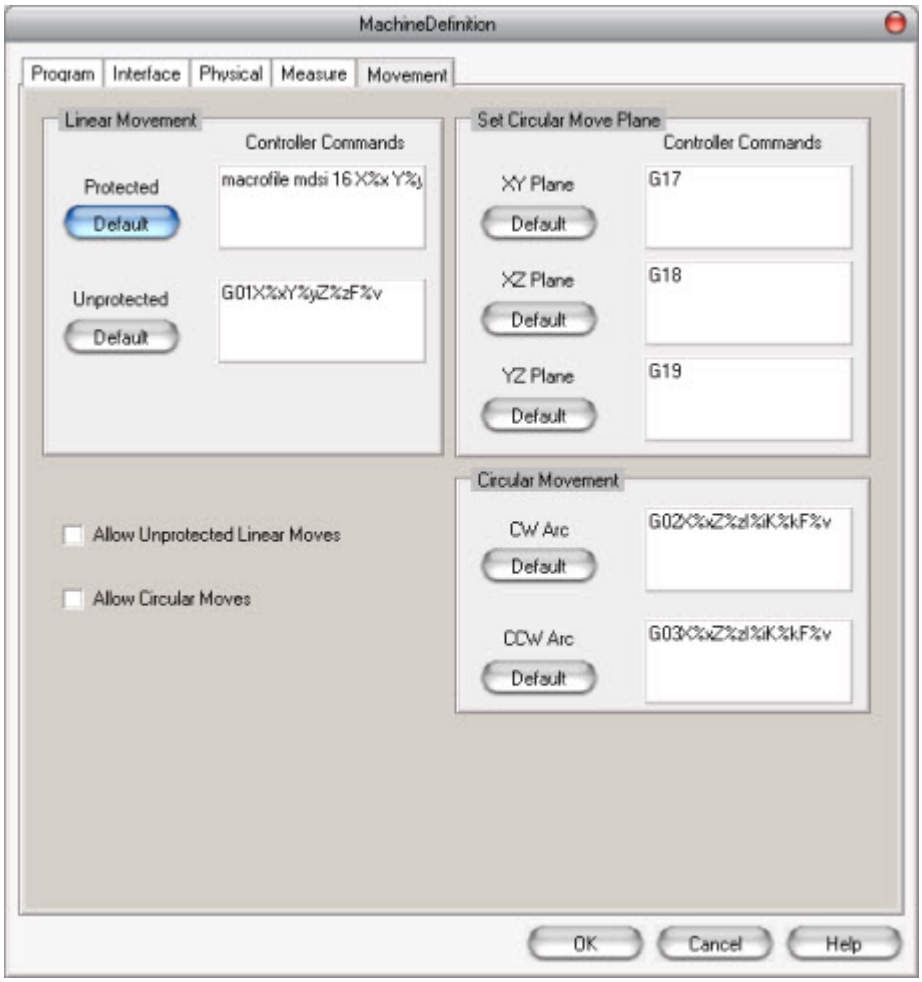
MachineDefinition Interface Tab for an Extended Basic MDSI Lathe



MachineDefinition Physical Tab for an Extended Basic MDSI Lathe



MachineDefinition Measure Tab for an Extended Basic MDSI Lathe



MachineDefinition Movement Tab for an Extended Basic MDSI Lathe

Running Multiple Version of PC-DMIS NC

To switch between installed versions of PC-DMIS NC, users should first register the NC dll as follows from the required PCDMIS folder: **regsvr32 ctrlrdefaults.dll**

Following this, PC-DMIS should be run prior to any attempt to run the ***CncServer.exe*** application.

CncServer may then be run as normal.

Lathe Upgrade Conclusion

The lathe measures in 3d to 10 micrometer accuracy by Polar to Rectangular conversion and Spindle Speed calculation in the macros and with the calibration of the probe tip on inner and outer ring surfaces.

Calibration is a main part of the lathe upgrade which successfully compensates for offset of the rotational position in XY and for tip offset in XZ using Groove Calibration. An optional small inside circle in the Groove Calibration can compensate for Y axis tip offset but in practice it has been found to make little improvement and carries with it caveats in how the calibration is done.

Using NC Update

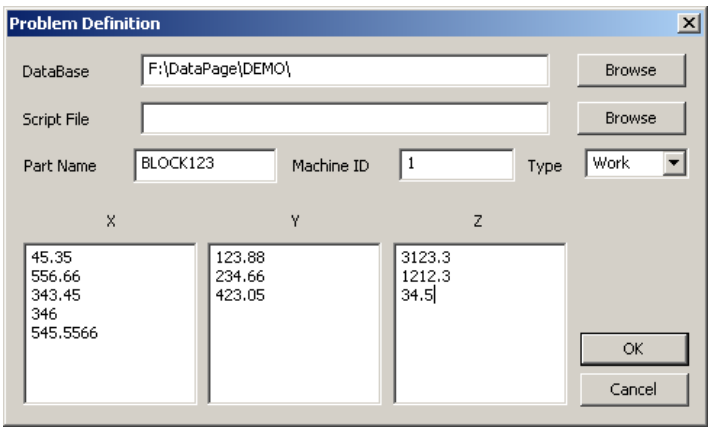
When PC-DMIS NC is in Update mode, you can pass measurement data back to the CNC machine to update work offsets and/or tool offsets. To enable this mode you must have Update Mode and at least one CNC licenses programmed in your portlock.

NC Update uses "Problem Definitions" to pass measurement data.

Creating Problem Definitions

To create a problem definition:

1. Launch CNC Server. With **Update Mode** programmed into your portlock, the **PC-DMIS nc Update** interface will appear.
2. Select the **File | New Problem** menu option to open the **Save As** dialog box.
3. **Browse** to the desired directory and provide the **File Name** to which the problem definition will be saved. Click **Save**. The **Problem Definition** dialog box appears.



Problem Definition dialog box

4. **Browse** to and select the Datapage database to which data will be sent.
5. **Browse** to and select the script (*.scr) that will be used for processing statistical information. See "Run Scripts".
6. Specify the **Part Name** and **Machine ID** for the entries in the database.
7. Specify the offset **Type** to update: **Tool** or **Work**.
8. Type one or more values in the **XYZ** list boxes for Work offsets.
9. Click **OK**. The **MonitorView** window opens. See "Working with the MonitorView Window". There are additional MonitorView options when opening from Update mode having to do with scripts. See "MonitorView Script Options".

MonitorView Script Options

When using **PC-DMIS nc Update**, the following additional option is available for scripts:

Edit File - Opens the **Problem Definition** dialog box. See "Creating Problem Definitions".

Index

=
= 81
A
Acknowledging Variables 70
Attach 79
Automatic Script Processing 82
B
Blum Control 9
C
Calculating the Statistical State 71
Chain 80
Chained Sample Script 86
Clearlog 80
D
Data Flow Chart for PC-DMIS NC 10
Database 76
  Changing 73
DataPage 72
Debug 80
Declare 76
Differences between PC-DMIS NCI and PC-DMIS 3
  Automatic Features 4
  Basic Scripts 7
  Breakpoints 7
  Calibrate 6
  Edit 7
  Flow Controls 6
  Moves 5
  Operations 7
  Parameter Change 6
  Read Point 5
  Regular Measured Features 5
  Scans 5
Dump 81
E
Edit Window Commands 22
  Pass Through Command 29
  Read CNC Variable 28
  Set Protection 27
  Update Tool Offset 26
  Update Work Offset 23
  Use Work Offset 25
  Write CNC Variable 27
Execute 78
Expressions
  Arithmetic 81
  Logical 81
F
Fanuc Control 8, 9, 11
Filter 76
Freezing Limits 71
G
goto 78
H
Haas Control 9
I
If 78
Import 77
J
Journal Files, Executing 55
L
Label 78
M
Machine Definition
  Interface Tab 60
  Measure Tab 64
  Movement Tab 67
  Physical Tab 62
  Program Tab 57
Machines, Creating and Modifying 57
Monitoring Variable Sets 73
MonitorView Script Options 206
MonitorView Window 68
  Understanding 69
Moving Programs to the NC Controller 55
Multi-Axis Wrist 36
N
NC Update 205
O
Offline Mode 50
P
Pass Through Command 29
PC-DMIS NC 16
  Analyze and Report the Data 20
  Calibrate the Probe 16
  Create NC Program 19
  Define a Machine 13, 18
  Executing the NC Program 20
  Files Used 9
  Install 13
  Install the Probe and Macros 13
  Introduction 1
  Load the NC Program 20
  Machine Link 15
  Offline Mode 50
  Offline Test 14
  Required Components 8
  Start CNC Server Program 19
  Testing PC-DMIS NC 13
  Work Offset 17
PC-DMIS NC Server 51
  Command Buttons 53
  Exiting 55
  Machine Status Fields 54

Menubar 52
Resuming 54
Stopping 54
PC-DMIS NCI..... 20
Components 11
Define a Machine..... 20
Differences 8
Executing..... 22
Generating the Monitor Program 21
Install 20
Problem Definitions..... 205
Prompt 80
Properties, Changing 73
General Tab..... 74
Tests Tab..... 74
R
Read CNC Variable 28
Refreshing Data..... 72
Rotary Table 30, 50
Calibrating 33
Rotary Table Physical Configuration..... 31
Setup 32
Troubleshooting..... 36
Verify Calibration 35
Run Scripts 75
S
Scripting Language..... 75
= 81
Attach 79
Chain 80
Clearlog 80
Database 76
Debug 80
Declare 76
Dump 81
Execute..... 78
Filter..... 76
goto..... 78
If 78
Import 77
Label..... 78
Prompt 80
Sample Script 82
ToolOffset..... 79
WorkOffset..... 79
Write 78
Select Table..... 29
Set Protection 27
Siemens Control 8, 9, 11
Stacked Rotary Tables..... 49
Statistical State, Calculating the..... 71
T
ToolOffset 79
U
Unacknowledging Variables..... 70
Unfreezing Limits 71
Update Tool Offset..... 26
Update Work Offset 23
Use Work Offset..... 25
Uses for PC-DMIS NC 2
Common Metrology Tool 3
Difficult Setups..... 2
Large Parts 2
Process Monitoring..... 3
V
V42R Compatibility 50
W
Working with Variables 70
WorkOffset..... 79
Write 78
Write CNC Variable..... 27

Glossary

C

CNC: Computer Numerical Control

J

Journal File: A script that works with PC-DMIS/NC

N

NC: Numerical Control

NCI: Numerical Control Interactive