

CNCcomp 4.5

CNC pitch compensation program

User Manual

Copyrights:



Tampere University of Technology

Production engineering



Qplus Ltd.

The software is developed by Qplus Ltd:
Qplus Ltd.
PL 70
33721 TAMPERE
FINLAND

Phone: +358 40 504 2693
Email: support@qplus.fi
www: <http://www.qplus.fi>

In no event will software publishing corporation be liable for any damages, including loss of data, lost profits, cost of cover or other special, incidental, consequential or indirect damages arising from the use of the program or accompanying documentation, however caused and on any theory of liability. This limitation will apply even if software publishing corporation or an authorized dealer has been advised of the possibility of such damage.

Table of Contents

1	INSTALLATION INSTRUCTIONS.....	5
2	REGISTRATION.....	6
2.1	SENDING REGISTRATION INFORMATION.....	7
2.2	RECEIVING REGISTRATION KEY FROM QPLUS LTD.....	8
2.3	INPUTTING THE REGISTRATION KEY.....	9
3	COMMON OPERATIONS.....	10
3.1	READING MEASUREMENT FILE.....	10
3.2	SELECTION OF CONTROLLER.....	11
3.3	SENDING FILES TO CONTROLLER.....	11
3.4	RECEIVING FILES FROM CONTROLLER.....	12
3.5	UNIT CONVERSIONS.....	12
3.6	READING PARAMETER FILE.....	13
3.7	SOFTWARE SETTING FILE.....	13
3.8	MEASUREMENT FILE OPTIONS.....	13
3.9	CALCULATION.....	13
3.10	SAVING COMPENSATION FILE.....	14
4	SIEMENS 800.....	15
4.1	PARAMETER OPTIONS.....	16
4.2	SENDING NEW COMPENSATION TABLES FROM SIEMENS 800 SERIES.....	17
5	SIEMENS 840D.....	19
5.1	PARAMETER OPTIONS.....	20
5.2	READING COMPENSATION FILE.....	21
6	FANUC 0/00.....	22
6.1	PARAMETER OPTIONS.....	23
6.2	CALCULATION.....	24
6.3	READING PARAMETER FILE.....	25
6.4	SERIAL COMMUNICATION PARAMETERS.....	25
6.5	SERIAL CABLE.....	25
6.6	READING PARAMETERS FROM CONTROLLER.....	25
6.7	SENDING NEW PITCH COMPENSATION VALUES TO CONTROLLER.....	25
7	FANUC 10/11/12/100/110/120.....	26
7.1	PARAMETER OPTIONS.....	27
7.2	READING PARAMETER FILE.....	28
7.3	SERIAL COMMUNICATION PARAMETERS.....	28
7.4	SERIAL CABLE.....	29
7.5	READING PARAMETERS FROM CONTROLLER.....	29
7.6	SENDING OF NEW PITCH COMPENSATION VALUES TO CONTROLLER.....	29
8	FANUC 15.....	30
8.1	PARAMETER OPTIONS.....	31
8.2	READING PARAMETER FILE.....	33
8.3	SERIAL COMMUNICATION PARAMETERS.....	33
8.4	SERIAL CABLE.....	33
8.5	READING PARAMETERS FROM CONTROLLER.....	33
8.6	SENDING NEW PITCH COMPENSATION VALUES TO CONTROLLER.....	33
9	FANUC 16/18/160/180	34
9.1	PARAMETER OPTIONS.....	35
9.2	READING PARAMETER FILE.....	37
9.3	CALCULATION.....	37
9.4	SAVE COMPENSATION FILE.....	37
9.5	SERIAL COMMUNICATION PARAMETERS.....	38

9.6 CHANGING OF PARAMETERS FOR COMMUNICATION.....	39
9.7 SERIAL CABLE.....	40
9.8 READING PARAMETERS FROM CONTROLLER.....	40
9.9 SENDING NEW PITCH COMPENSATION VALUES TO CONTROLLER.....	41
10 FANUC 30/31/32/300/310/320.....	43
10.1 PARAMETER OPTIONS.....	44
10.2 READING PARAMETER FILE.....	46
10.3 CALCULATION.....	46
10.4 SAVE COMPENSATION FILE.....	47
10.5 SERIAL COMMUNICATION PARAMETERS.....	47
10.6 SERIAL CABLE.....	48
10.7 READING PARAMETERS FROM CONTROLLER.....	48
10.8 SENDING NEW PITCH COMPENSATION VALUES TO CONTROLLER.....	48
11 HEIDENHAIN TNC 426/430, ITNC530.....	49
11.1 PARAMETER OPTIONS.....	50
11.2 SAVING COMPENSATION FILE.....	51
11.3 READING PARAMETER FILE.....	51
11.4 PREPARATION OF CONTROLLER FOR PITCH COMPENSATION.....	52
11.5 READING PARAMETERS FROM CONTROLLER.....	52
11.6 SERIAL CABLE.....	54
11.7 WRITING NEW PITCH COMPENSATION VALUES TO CONTROLLER.....	55
12 HEIDENHAIN ND 2XXB.....	56
12.1 MEASUREMENT FILE OPTIONS.....	57
12.2 PARAMETERS OPTIONS.....	58
12.3 READING PARAMETER FILE.....	59
12.4 SERIAL CABLE.....	59
12.5 MANUALLY RECEIVING COMPENSATION FILE FROM ND 2xxB.....	60
12.6 AUTOMATICALLY RECEIVING COMPENSATION FILE FROM NC 2xxB.....	60
12.7 MANUALLY RECEIVING PARAMETER FILE FROM ND 2xxB.....	61
12.8 AUTOMATICALLY RECEIVING PARAMETER FILE FROM ND 2xxB.....	61
12.9 MANUALLY SENDING COMPENSATION FILE TO ND 2xxB.....	62
12.10 AUTOMATICALLY SENDING COMPENSATION FILE TO ND 2xxB.....	62
12.11 MANUALLY SENDING PARAMETER FILE TO ND 2xxB.....	63
12.12 AUTOMATICALLY SENDING PARAMETER FILE TO ND 2xxB.....	63
13 HEIDENHAIN ND 7XXD.....	64
13.1 MEASUREMENT FILE OPTIONS.....	65
13.2 PARAMETERS OPTIONS.....	66
13.3 READING PARAMETER FILE.....	67
13.4 SERIAL CABLE.....	67
13.5 MANUALLY RECEIVE COMPENSATION FILE FROM ND 7XX.....	68
13.6 AUTOMATICALLY RECEIVE COMPENSATION FILE FROM ND 7XX.....	68
13.7 MANUALLY RECEIVE PARAMETER FILE FROM ND 7XX.....	69
13.8 AUTOMATICALLY RECEIVE PARAMETER FILE FROM ND 7XX.....	69
13.9 MANUALLY SEND COMPENSATION FILE TO ND 7XX.....	70
13.10 AUTOMATICALLY SEND COMPENSATION FILE TO ND 7XX.....	71
13.11 MANUALLY SEND PARAMETER FILE TO ND 7XX.....	71
13.12 AUTOMATICALLY SEND PARAMETER FILE TO ND 7XX.....	72
14 BECKHOFF TWINCAT CNC.....	73
14.1 MEASUREMENT FILE OPTIONS.....	74
14.2 PARAMETER OPTIONS.....	75
14.3 READING COMPENSATION FILE.....	76
14.4 ANNOUNCING THE COMPENSATION TABLE.....	77

This software calculates pitch error compensation table for a numerical controller. The calculation is based on laser interferometer measurement, which is read in from a file produced by a third party program.

1 Installation Instructions

Windows 7

You can just install CNCcomp and it shall work as it is. Windows 7 comes with "Windows Installer 4.5" and "Microsoft .NET Framework Version 4.0" - so all necessary is included.

Windows Vista and XP

When you double-click CNCcomp.msi file, it shall launch a Windows Installer. But if you receive a message about unknown file type, you need first to install "Windows Installer 4.5". You can download it from Microsoft at: <http://www.microsoft.com/downloads> . The Windows Installer 4.5 installation file can also be found in the directory “\Windows Installer 4.5 Windows XP” or “\Windows Installer 4.5 Windows Vista” of the installation CD

CNCcomp runs on "Microsoft .NET Framework Version 4.0" and thus also it must be installed. If CNCcomp doesn't start, you most probably haven't yet installed .NET Framework. Please go to [Microsoft Windows Update](#) and install "Microsoft .NET Framework Version 4.0". The installation file can also be found in the directory “\Microsoft .NET Framework 4.0” of the installation CD.

Windows 2000 and 95

CNCcomp 4.5 doesn't run on Windows 2000 or Windows 95. Please either use previous CNCcomp version 4.4 or upgrade your operating system.

Acrobat Reader

The help file of CNCcomp is made in PDF format. You can download Acrobat Reader from <http://www.adobe.com> to read the PDF file. An installation file of Acrobat Reader can also be found in the directory “\Acrobat Reader 5.1” of installation CD.

2 Registration

After the installation of CNCcomp all users should register their copies of CNCcomp within 60 days.

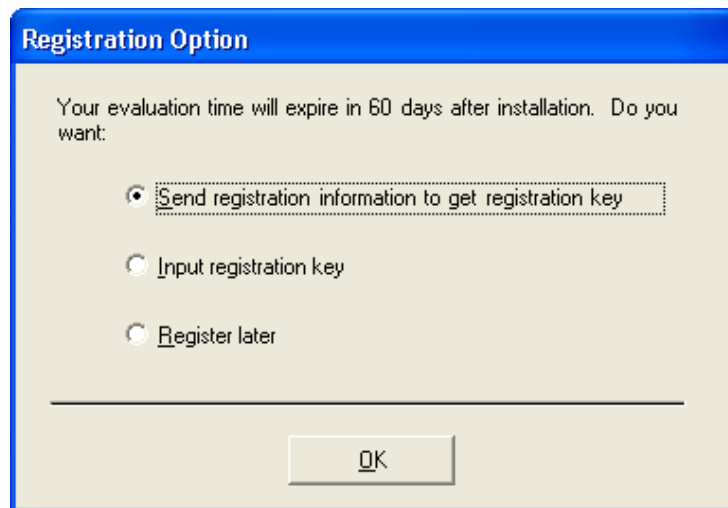
The registration of CNCcomp includes 3 steps:

1. [Sending registration information to Qplus Ltd.](#)
2. [Receiving registration key from Qplus Ltd.](#)
3. [Inputting the registration key.](#)

These 3 steps are described in detail as follows:

2.1 Sending Registration Information

Starting from CNCcomp version 4.0, if the software copy is not registered, a registration dialog box is displayed when the program starts, asking the user to send the registration information or input registration key:



Select *Send registration information* to get registration key, and click *OK*. The Registration Information dialog displays. One can also visit the Registration Information dialog by clicking menu *Help/Send Registration Info* Fill the form with user information and select the control types needed. In the left-lower part of the dialog, the total price will change accordingly. When buying CNCcomp from a dealer, select the option *I have paid to a dealer whose name is:*, and give also the dealer's name. Please notice that *User Name* cannot be empty or "Demo"; the *Company Name* cannot be empty either; at least one control must be selected, and when buying form a dealer, the dealer name must be given. Here is an example:

CNCcomp 4.3 Registration Information

<p>Registration Information</p> <p>User Name: <input type="text" value="Kari Pastila"/></p> <p>Company Name: <input type="text" value="Qplus Oy"/></p> <p>Hard Drive Serial: <input type="text" value="-857691268"/></p>	<p>User Contact Information</p> <p>Please provide the contact information so that we can send the registration key to you:</p> <p>Street Address: <input type="text" value="Hermiankatu 8D"/></p> <p>City: <input type="text" value="Tampere"/></p> <p>State: <input type="text"/></p> <p>Zip Code: <input type="text" value="33720"/></p> <p>Country: <input type="text" value="Finland"/></p> <p>Telephone: <input type="text" value="+358-40-1234567"/></p> <p>Fax: <input type="text" value="+358-3-1234567"/></p> <p>Email: <input type="text" value="kari.pastila@qplus.fi"/></p> <p>VAT ID-Number (for EU users): <input type="text" value="FI12345678-9"/></p>
<p>License Options</p> <p><input type="checkbox"/> Siemens 800</p> <p><input type="checkbox"/> Siemens 840D</p> <p><input type="checkbox"/> Fanuc Q/00</p> <p><input type="checkbox"/> Fanuc 10/11/12/100/110/120</p> <p><input type="checkbox"/> Fanuc 15</p> <p><input type="checkbox"/> Fanuc 15/18/160/180</p> <p><input type="checkbox"/> Fanuc 30/31/32/300/310/320</p> <p><input checked="" type="checkbox"/> Heidenhain TNC425/430, iTNC530</p> <p><input checked="" type="checkbox"/> Heidenhain ND 2xx B</p> <p><input checked="" type="checkbox"/> Heidenhain ND Zxx</p> <p><input type="checkbox"/> Beckhoff TwinCAT CNC</p>	<p>Payment Information</p> <p><input checked="" type="radio"/> I buy via Web. The total price is: <input type="text" value="790 €"/> (VAT not included)</p> <p><input type="radio"/> I have paid to a dealer whose name is: <input type="text"/></p>

NOTE: After you send the registration information to Qplus Ltd, you will soon receive the registration key with the invoice of the CNCcomp software. You can also send query emails to support@qplus.fi.

Save To File Send Email Send Later

One can either click *Save to File* button to save the registration information to a text file and then send it as an email attachment to register@qplus.fi, or, if a user is connected directly to the internet, click *Send Email* button to automatically send email to Qplus Ltd.

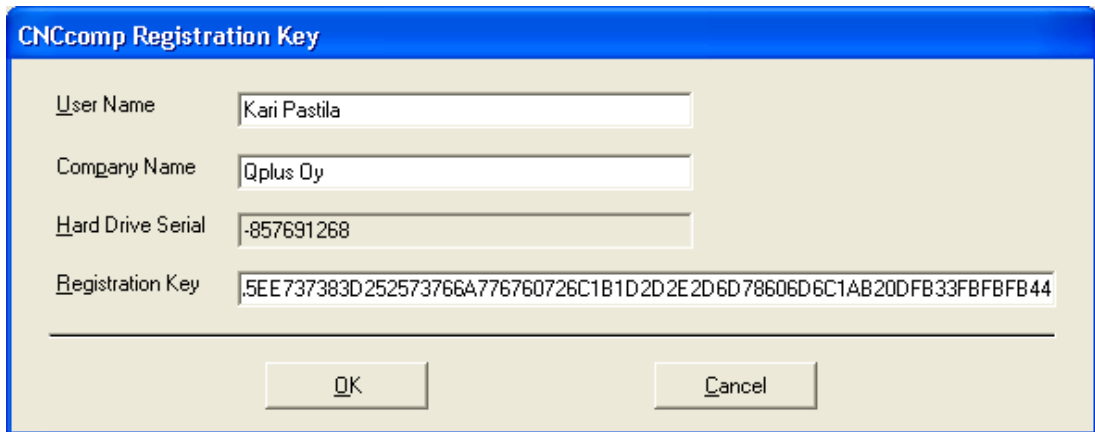
User can also click *Send Later* to quit the dialog.

2.2 Receiving registration key from Qplus Ltd.

After sending the registration information, user will soon receive a letter via Email, Fax or Post, depending on the contact information provided. Inside the letter there are user information (user name, company name, hard disk serial number) and the registration key which is compatible with user's information.

2.3 Inputting the registration key

Starting from CNCcomp 4.0, if the copy is not registered, a registration dialog box is displayed. In the registration dialog, select *Input registration key*. The *Registration Key* dialog is displayed. One can also visit *Registration Key* dialog by clicking menu *Help, Input registration key ...* Input the user name, company name and registration key. Since the registration key is a long hexadecimal string, if a user receives the registration key via e-mail, please use *copy/paste* command to avoid typing mistakes. Please also notice that all inputs are case-sensitive. Here is an example:



The screenshot shows a dialog box titled "CNCcomp Registration Key". It contains the following fields and values:

Field	Value
User Name	Kari Pastila
Company Name	Qplus Oy
Hard Drive Serial	-857691268
Registration Key	5EE737383D252573766A776760726C1B1D2D2E2D6D78606D6C1AB20DFB33FBFBFB44

At the bottom of the dialog are two buttons: "OK" and "Cancel".

Click *OK* button. The registration is done.

3 Common Operations

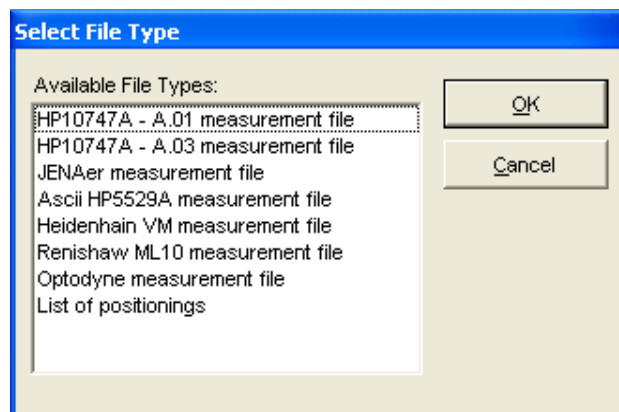
Following operations are common for all controllers.

3.1 Reading Measurement file

The measuring file is opened by the menu *File/Open Measurement File...* or toolbar button *Open Measurement File*. Measurements can be recognized either from following file formats:

- HP or Optodyne laser measurement file (*.lin),
- HP measurements saved in ASCII format,
- the ASCII file generated by JENAer Esox software,
- Renishaw ML10 measurement file (*.rtl)
- API measurement file (*.pos)
- Heidenhain VM measurement file.

Program can read also a simple list of positions. Some examples of measurement file are under *SampleTest* subdirectory in application directory: X1.lin, Y1.lin, Z1.lin, and PLIST.txt. If the file cannot be recognized, the following dialog is opened:



If the file is in correct format and the reading is successful, user can proceed in creation of a compensation file. Otherwise an error message appears in screen and the user has to read measurement file again.

Saving measurements in ASCII format

HP-laser software offers a possibility to save measurements also in ASCII format. First open the measurement and then select: *Show Data, Print to File* and *Save*.

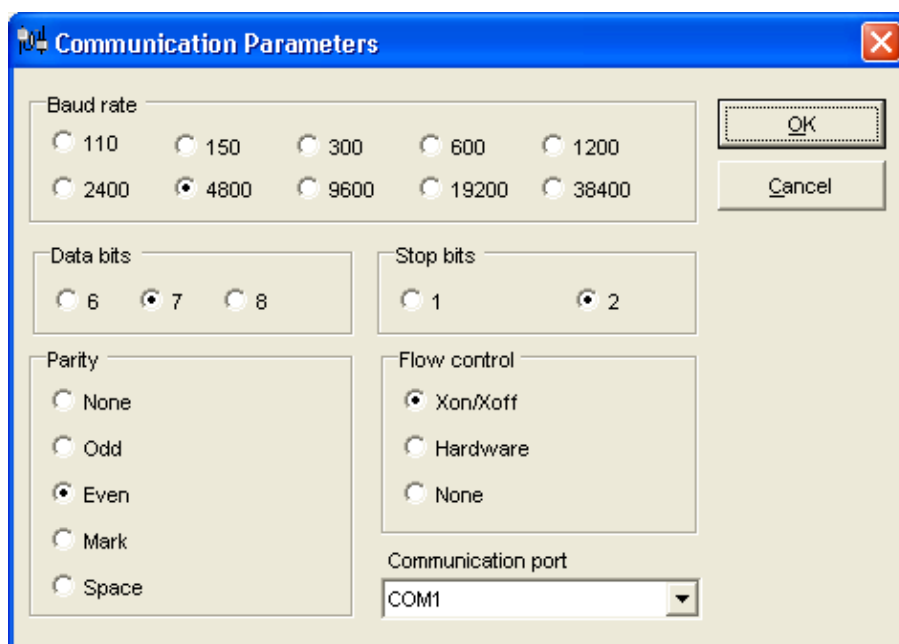
3.2 Selection of Controller

The controller must first be selected from the *Controller* drop down list. After the controller type is selected from the drop down list, controller specific values are read from the initialization file (compensation file header, footer, line end character, and communication settings). The change of controller also changes the fields in *Parameters* frame. CNCcomp currently supports following types of controllers:

- Siemens series 800
- Siemens 840D
- Fanuc 0/00
- Fanuc 10/11/12/100/110/120
- Fanuc 16/18/160/180
- Fanuc 30/31/32/300/310/320
- Heidenhain TNC 426/430, iTNC530
- Heidenhain ND 2xx B
- Heidenhain ND 7xx
- Beckhoff TwinCAT CNC

3.3 Sending Files to Controller

It is possible to send ASCII files to the machine tool controller via RS-232 interface directly from this application. This function is started by selecting the menu *Communication/Send File to Controller Manually...* or toolbar button *Send File to Controller Manually*. A file selection dialog will be opened, after the user select the file to be sent, *Communication Parameters* dialog is displayed.

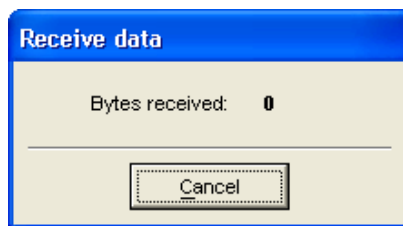


Set the parameters according to the controller settings and RS-cable. Prepare the controller to receive data, and file transfer is started immediately after the *OK* button is pressed.

For Heidenhain ND 2xx B and ND 7xx display units, files can be sent to the devices automatically. Refer to chapters for [ND 2xx B](#) and [ND 7xx](#) for details.

3.4 Receiving Files from Controller

It is possible to receive files from a controller using a serial communication cable. This function can be selected from the menu *Communication/Receive File from Controller manually...* or toolbar button *Receive File from Controller Manually*. Program will ask for the file name. When working with a Heidenhain TNC 426/430 or iTNC 530, use the same file names as in controller. For other controllers a user can use any file name that is convenient for him. After the file name is specified, CNCcomp asks for serial communication settings. These settings are saved individually for each controller type and thus once the right settings have been found for a controller they will stay so. Press *OK* button to let the program wait for data arriving, then prepare controller to send data.



The file is saved as it is received, and NO file formatting work is done. CNCcomp waits for the footer row and if it is found, the communication will terminate automatically. Besides, user can also stop communication at any time by pressing *Cancel* button.

For Heidenhain ND 2xx B and ND 7xx display units, files can be received from the devices automatically. Refer to chapters of [ND 2xx B](#) and [ND 7xx](#) for details.

3.5 Unit Conversions

CNCcomp supports use of millimeters and inches in measuring and compensation files. It attempts to derive the right unit from the reading of a measurement file and switches to the measurement unit mode that is consistent. Users have the possibility to convert measuring data later from inches to millimeters or vice versa. Thus user can, for example, carry out measurements in millimeters and create compensation files in inches.

When creating a compensation file, the program does not check if it has the right units for user's controller. Similarly, if a user has read in old compensation parameters, they can not be converted with *Convert measurement to mm/Inch* function. Thus, users cannot use this function to translate compensations to other units when they switch units in controller (In fact, this is normally done automatically by controllers themselves).

3.6 Reading Parameter File

Menu *File/Read parameters from* reads the CNCcomp parameters into the program, which may have influence to pitch compensation. This functionality is supported for Fanuc 0/00, Fanuc10/11/12/100/110/120, Fanuc 15, Fanuc 16, Heidenhain TNC controllers, and Heidenhain ND units. Operations of this functionality differ slightly.

3.7 Software Setting File

Users normally do not need to change settings described here, but in case of some problems these settings might be helpful. Some controller specific values can be changed in the initialization file. Initialization file named *CNCComp.ini* is located in the main installation directory. It is possible to define header and footer line for each controller type in the file. Header line is always added to the beginning of the compensation file and the footer is added to the end of the compensation file.

For some controllers it is possible to define the line end characters which are used when file is send via serial port. Line end characters can be freely modified by changing the Line end parameter in the initialization file. Line end parameter can include 3 ASCII codes of characters separated by comma. For example:

```
LineEnd = 13, 10, 10      (Carriage Return, Line Feed, Line Feed)
```

Heidenhain TNC has a setting for a decimal separator used to create files. This is '.' by default, but is possible to change using the following key:

```
[Heidenhain TNC 426/430]  
Comma=,
```

3.8 Measurement File Options

Measurement file options are the same for most of the controller types.

Reference point is the machine coordinate value of the reference point.

If measurement file includes several measurement runs (sets) they are all listed in a Measurement set(s) list box. It is possible to select just one or several sets for the compensation. If more than one set is selected the average of all selected measurement sets are used for the compensation.

3.9 Calculation

The calculation is done by pressing the menu *File/Calculate Compensation File*, the toolbar button *Calculate Compensation*, or the arrow sign in the middle of the screen.

Calculation results are displayed on the right side of the screen.

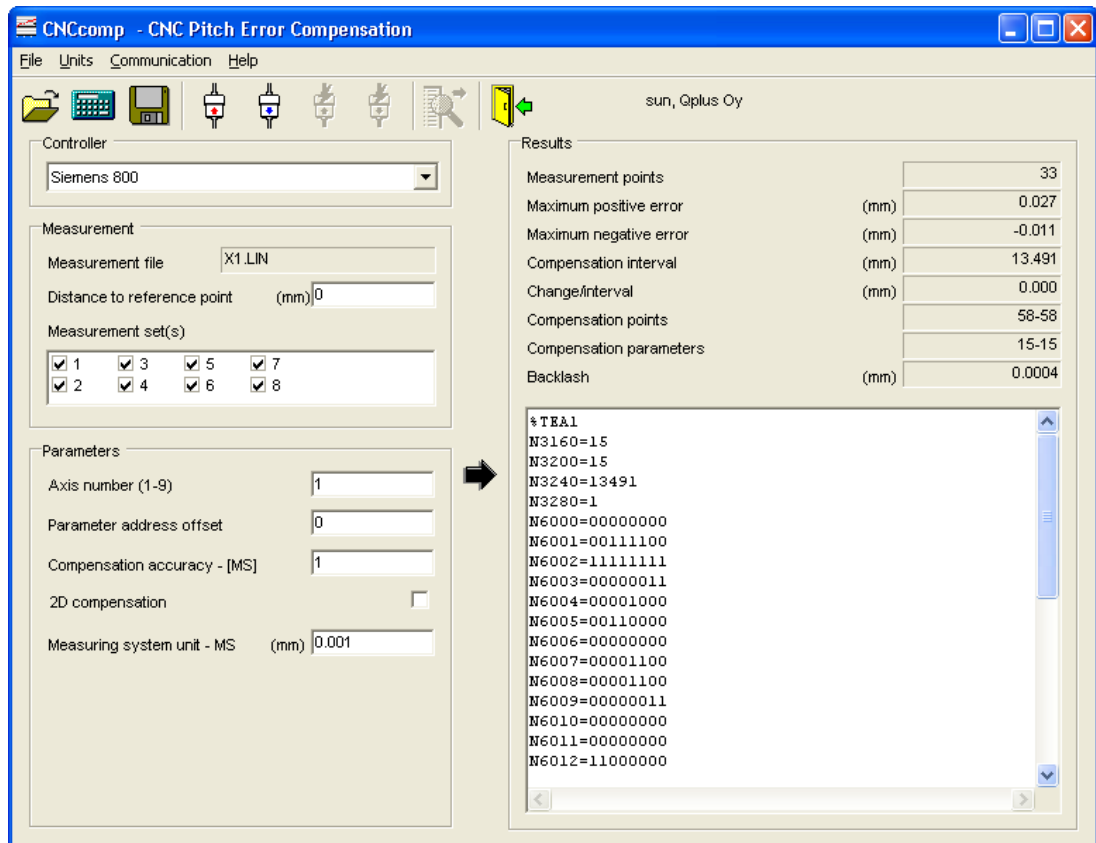
If the measurement is one-directional, the backlash value will not be calculated and hence will be kept empty.

3.10 Saving Compensation File

Compensation file is saved by clicking the menu *File/Save* or toolbar button *Save*, after which the save file dialog is opened.

4 Siemens 800

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Sending new compensation tables to Siemens 800 series*



4.1 Parameter Options

Axis number

Axis number specifies which axis is to be compensated. The compensation file parameter addresses are changed accordingly.

Parameter address offset

Parameter address offset is added to the parameter address starting number. This is how different axis compensation parameter areas are separated.

Compensation accuracy

Compensation accuracy is the value that represents how closely the calculation tries to follow the measurement results. If the accuracy is too small the number of parameters will increase and may crowd out of sensible limits.

2D compensation

With *2D compensation* it is possible to use different measurement sets for opposite compensation directions. Therefore two measuring sets must be selected for 2D compensation.

If the measurement is one-directional, this function will be disabled.

Measuring System Unit [MS]

Measuring system unit defines the position control resolution of a machine tool and it is machine tool specific.

Unit can be set separately for each axis. Unit is defined in machine tool parameter *MD1800* with bits 0-3, which define position control resolution. Standard machine data for parameter *MD1800* bit 0-3 is 0100, which means that MS is 0,001 [mm]. Legal values for MS are 0.1 0.01 0.001 0.0004 0.0001 and 0.00001 according the following table.

Measuring system unit	bit 3	bit 2	bit 1	bit 0
0,01 mm	1	0	0	0
0,001 mm	0	1	0	0
0,0004 mm	1	1	0	0
0,0001 mm	0	0	1	0
0,00001 mm	1	0	1	0

Measuring system unit	bit 3	bit 2	bit 1	bit 0
0,001 inch	0	1	0	1
0,0001 inch	1	1	0	1
0,00004 inch	0	0	1	1
0,00001 inch	1	0	1	1

The compensation accuracy is handled on the given MS unit. For example if the compensation accuracy is 4 and measuring system unit is 0,001 then the calculation uses compensation accuracy of $0,001 \times 4 = 0,004$.

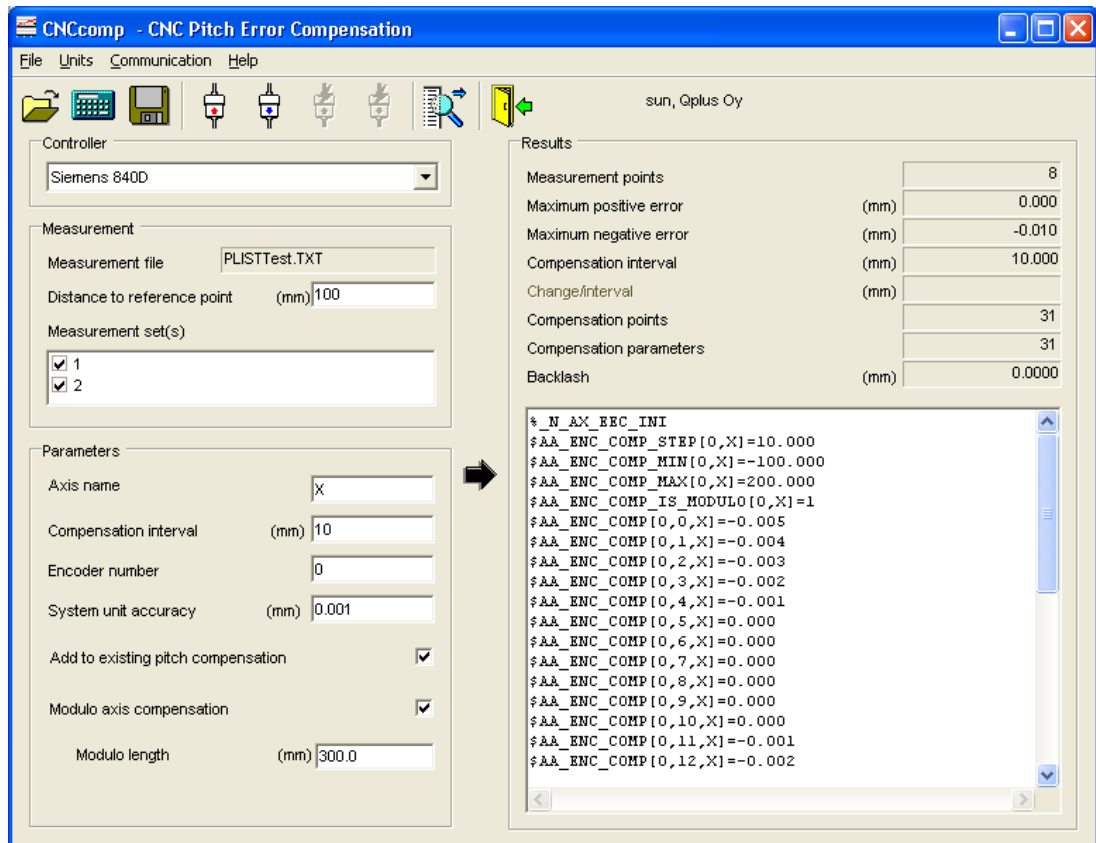
4.2 Sending New Compensation Tables from Siemens 800 Series

1. Check that pitch compensation option is active. Go through menu: *DIAGNOSIS / START-UP / OPTIONS*. Bit 1 in *MD5016* should be set to '1'. If user want to use 2D compensation, he should have option bit 2 in *MD5016* set to '1' too. If user has neither of them, he has not purchased pitch compensation option and thus he cannot proceed in compensation.
2. Give password to allow data input: *DIAGNOSIS / SET PASSWORD*, key in *1111* and press **OK**.

3. Connect computer and NC controller with a serial cable. Use connector X151 in NC controller.
4. Choose communication protocol: *SERVICES/ DEVICES/ WS800A/ SET*. Communication settings should be: 9600 baud, 7 data bits, 2 stop bits, Even parity, Hardware flow control
5. Initiate data receiving in the controller by: *SERVICES/ DATA IN/ CLIPBOARD / START INPUT*
6. Send file from PC. Check that communication parameters are the same as above.
7. Data has arrived with name *TEA1* to the temporary directory in NC. Copy the file with new name (e.g. *compx*) to *START-UP DATA* directory: *SERVICES / DATA IN / CLIPBOARD / choose TEA1 file / choose destination path START-UP DATA / PASTE / give new file name e.g. COMPX*
8. Activate new compensations: *DIAGNOSIS / START-UP / MACHINE DATA / FILE FUNCTIONS / choose COMPX.teal / LOAD*
9. Reboot the controller so as to take new compensations into effect. Now machine tool is ready for verification measurements.

5 Siemens 840D

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Read Compensation File*
- *Saving Compensation File*



5.1 Parameter Options

Axis name

Axis name is specified and will be used in compensation files.

Compensation interval

Compensation interval defines how many compensation points are calculated. There is no need to use more compensation points than the measurement points because the controller calculates the compensation points between the measurement points automatically.

Encoder number

It is possible to use different compensations for different encoders. The *encoder number* defines which encoder is used.

System unit accuracy

System unit accuracy defines how many decimals are displayed in the parameter file.

Add to existing pitch compensation

If existing pitch compensation has been read, the new compensation values can be actualized cumulatively to old ones. If user wants to measure and compensate without clearing the old compensations, first carry out measurements with compensation set to

active, then read the old values into CNCcomp, and make the calculation with this selection checked.

Important Note: The range of the new compensation table to be generated is decided by the current measurement file. Therefore, if part of the old compensation range is outside the current measurement range, it will be neglected in the new compensation table, which means that the old compensation values outside the current measurement range are simply lost.

Bidirectional compensation

Bidirectional compensation decides whether bidirectional pitch error compensation is enabled. If bidirectional compensation is enabled, CNCcomp will generate compensation values separately for positive direction and negative direction.

If the measurement is one-directional, this function will be disabled.

Modulo compensation

Modulo compensation decides whether the compensation list will be generated for modulo compensation.

Modulo length

If modulo compensation is enabled, the user must give the modulo length here.

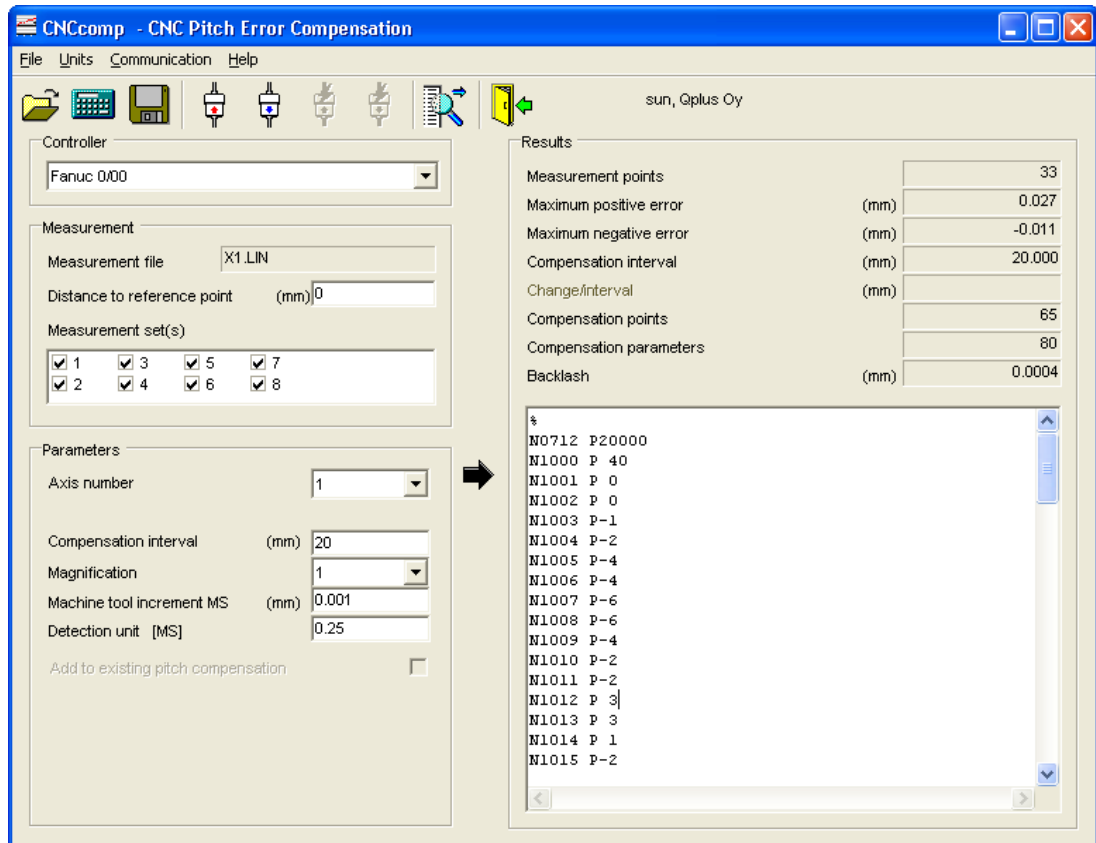
5.2 Reading Compensation File

The program asks for the compensation file name. After the user gives the old compensation file name, the program will read the old compensation list from the file and then inform the user whether the reading operation is successful. If the old compensation file has been read successfully, the program will make the selection of the parameter options according to the old parameter file.

Reading old compensation file makes it possible to measure and compensate machine tools without first clearing pitch compensations and thus speed up the whole measuring process.

6 Fanuc 0/00

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Serial Communication Parameters*
- *Serial cable*
- *Reading Parameters from Controller*
- *Sending New Pitch Compensation Values to Controller*



6.1 Parameter Options

Axis number

Axis number specifies which axis is to be compensated. The compensation file parameter options are changed accordingly.

Compensation interval

Compensation interval defines how many compensation points are needed. Decreasing the interval will make compensation steps smaller.

Magnification

Magnification amplifies the used compensation step. If the magnification is 1 then the compensation step is the same as detection unit. For Fanuc 0/00, magnification values can only be 1, 2, 4, or 8.

Machine tool increment

Machine tool increment is the used linear scale accuracy for the machine tool. Increment can be read from a downloaded parameter file (N0024#1).

Detection Unit

Detection unit is the unit by which compensation is made. So if the detection unit is 0.00025 mm and compensation is 2 and magnification is 4 then compensation at that point is $0.00025 * 2 * 4 = 0.0020$

With this controller type, it is possible to read the current machine tool increment system from the parameter file downloaded from the controller. Parameter N0024 bit 1 defines the machine tool increment system as following:

Value	Least Command Increment
0	0.001 mm/0.0001 inch
1	0.01 mm/0.001 inch

Also the command multiplier (CMR) can be read from the parameter file via N0100 (X Axis), N0101 (Y Axis), N0102 (Z Axis), N0103 (4th Axis).

Setting code	CMR
1	0.5
2	1
4	2
10	5
20	10

Detection Unit = Least Command Increment / CMR

Add to existing pitch compensation

If old pitch compensation has been read from controller, the new compensation values can be actualized cumulatively to old ones. If user wants to measure and compensate without clearing the old compensations, first carry out measurements with compensation set to active, then read the old values into CNCcomp, and make the calculation with this selection checked.

Important Note: The range of the new compensation table to be generated is decided by the current measurement file. Therefore, if part of the old compensation range is outside the current measurement range, it will be neglected in the new compensation table, which means that the old compensation values outside the current measurement range are simply lost.

6.2 Calculation

The calculation is done by pressing the menu *File/Calculate Compensation File*, the toolbar button *Calculate Compensation*, or the arrow sign in the middle of the screen. After the calculation is finished, the calculation results are displayed on the right side of the screen. If a dialog box is displayed asking the user to set parameter N0011#0#1 to

the proper value, please set the parameter bits accordingly in the controller to ensure that the magnification value is correct.

6.3 Reading Parameter File

CNCcomp first asks for the whole parameter file and after that, it asks if user wants to read in also old pitch compensation values. This makes it possible to measure and compensate machine tools without first clearing pitch compensations and thus speed up the whole measurement process.

6.4 Serial Communication Parameters

Before it is possible to read or write any parameters to Fanuc, some communication settings in the controller should be manually checked. Please refer to Fanuc 0/00 manuals for details.

6.5 Serial Cable

Please refer to Fanuc 0/00 manuals for details.

6.6 Reading Parameters from Controller

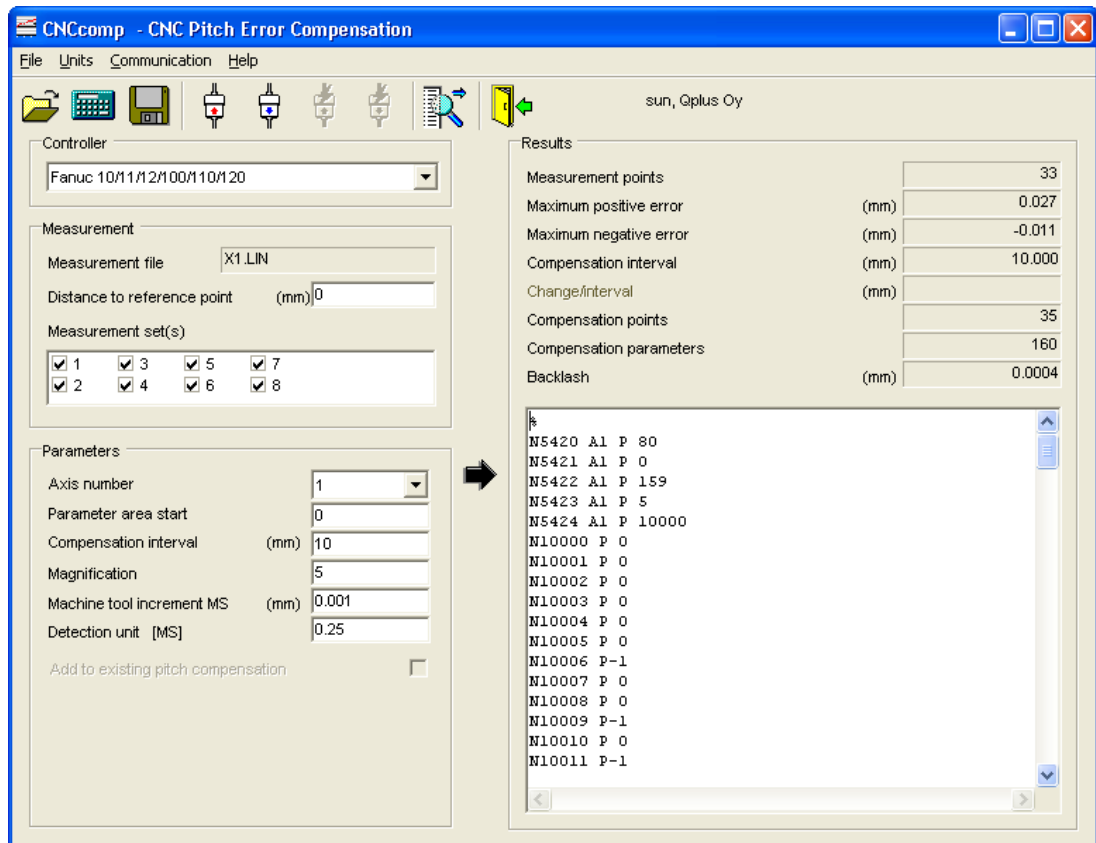
The operations are similar to the [operations of reading parameters from Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 0/00 manuals for detail operations with the controller.

6.7 Sending New Pitch Compensation Values to Controller

The operations are similar to the [operations of sending new pitch compensation values to Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 0/00 manuals for detail operations with the controller.

7 Fanuc 10/11/12/100/110/120

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Serial Communication Parameters*
- *Serial Cable*
- *Reading Parameters from Controller*
- *Sending New Pitch Compensation Values to Controller*



7.1 Parameter Options

Axis number

Axes number specifies which axis is to be compensated. The compensation file parameter options are changed accordingly.

Parameter area start

Parameter area start is the offset value to be added to the address of the compensation value parameters. This is how different axis compensation parameter areas are separated. The maximum compensation parameter address is 11023

Compensation interval

Compensation interval defines how many compensation points are needed. Decreasing the interval will make compensation steps smaller.

Magnification

Magnification value amplifies the used compensation step. If the magnification is 1 then the compensation step is the same as detection unit.

Machine tool increment

Machine tool increment is the linear scale accuracy of the machine tool. Increment can be read from a downloaded parameter file (N1004).

Detection Unit

Detection unit is the unit by which compensation is made. So if the detection unit is 0.00025 mm and compensation is 2 and magnification is 5 then compensation at that point is $0.00025 * 2 * 5 = 0.0025$

With this controller type, it is possible to read the current machine tool increment system from the file downloaded from the controller. Parameter N1004 bit 0 and 1 defines the machine tool increment system as following:

N1004#0	N1004#1	Increment System	Least Command Increment Value
0	0	IS-B	0.001 mm/0.0001 inch
1	1	IS-A	0.01 mm/0.001 inch

Also the command multiplier (CMR) can be read from the parameter file via N1820 and N1816#7:

If N1816#7 = 1, $CMR = 1/N1820$ (N1820 has the valid range of 1 to 27)

If N1816#7 = 0, $CMR = N1820 * 0.5$ (N1820 has the valid range of 1 to 96)

Detection Unit = Command Increment / CMR

Add to existing pitch compensation

If existing pitch compensation has been read from controller, the new compensation values can be actualized cumulatively to old ones. If user wants to measure and compensate without clearing the old compensations, first carry out measurements with compensation set to active, then read the old values into CNCcomp, and make the calculation with this selection checked.

Important Note: The range of the new compensation table to be generated is decided by the current measurement file. Therefore, if part of the old compensation range is outside the current measurement range, it will be neglected in the new compensation table, which means that the old compensation values outside the current measurement range are simply lost.

7.2 Reading Parameter File

First the program asks for the parameter file name and after that, it asks if user wants to read in also the old pitch compensation values. This makes it possible to measure and compensate machine tools without first clearing pitch compensations and thus speed up the whole measuring process.

7.3 Serial Communication Parameters

Before it is possible to read or write any parameters to Fanuc, some communication settings in the controller should be manually checked. Please refer to Fanuc 10/11/12/100/110/120 manuals for details

7.4 Serial Cable

Please refer to Fanuc 10/11/12/100/110/120 manuals for details.

7.5 Reading Parameters from Controller

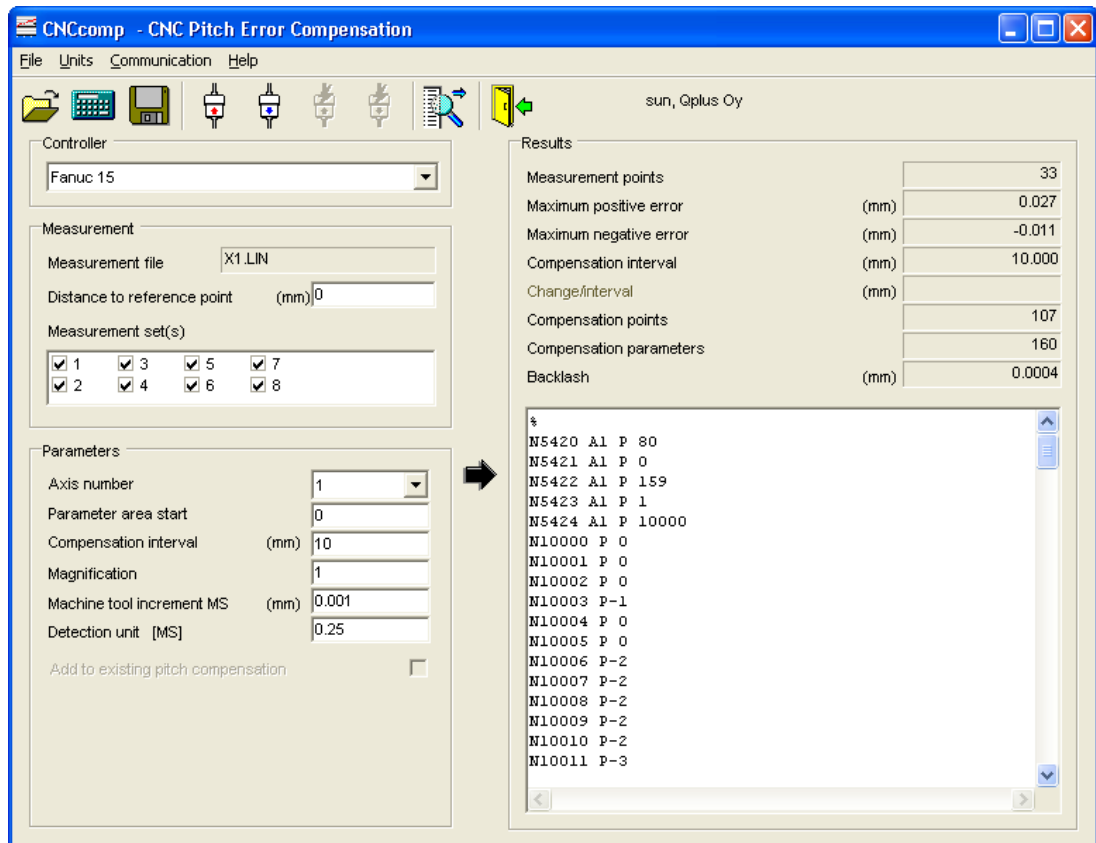
The operations are similar to the [operations of reading parameters from Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 10/11/12/100/110/120 manuals for detail operations with the controller.

7.6 Sending of New Pitch Compensation Values to Controller

The operations are similar to the [operations of sending new pitch compensation values to Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 10/11/12/100/110/120 manuals for detail operations with the controller.

8 Fanuc 15

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Serial Communication Parameters*
- *Serial Cable*
- *Reading Parameters from Controller*
- *Sending New Pitch Compensation Values to Controller*



8.1 Parameter Options

Axis number

Axis number specifies which axis is to be compensated. The compensation file parameter options are changed accordingly.

Parameter area start

Parameter area start is an offset value to be added to the address of the compensation parameters. This is how different axis compensation parameter areas are separated. The maximum parameter address is 13071

Compensation interval

Compensation interval defines how many compensation points are needed. By decreasing the interval smaller compensation steps are needed.

Magnification

Magnification amplifies the used compensation step. If the magnification is 1 then the compensation step is the same as detection unit.

Machine Tool Increment

Machine tool increment is the used linear scale accuracy for the machine tool. Increment can be read from a downloaded parameter file (N1004, N1009).

Detection Unit

Detection unit is the unit by which compensation is made. So if the Machine Tool Increment (MS) value is 0.001 mm, the detection unit is 0.25MS, namely $0.25 * 0.001 = 0.00025$ mm, compensation is 2, and magnification is 5, then compensation at that point is $0.00025 * 2 * 5 = 0.0025$ mm.

With this controller type, it is possible to read the current machine tool increment system from the file downloaded from the controller. Parameter N1004#0, N1004#2, N1004#5, and N1009#1 define the machine tool increment system in a following matter:

N1004 #0	N1004 #1	N1004 #5	N1009 #1	Least Command Increment	Abbreviation
0	0	0	0	0.001 mm/0.0001 inch	IS-B
1	0	0	0	0.01 mm/0.001 inch	IS-A
0	1	0	0	0.0001 mm/0.00001 inch	IS-C
0	0	1	0	0.00001 mm/0.000001 inch	IS-D
0	0	0	1	0.000001mm /0.0000001 inch	IS-E

Also the command multiplier (CMR) can be read from the parameter file via N1820 and N1816#7:

If N1816#7 = 1, $CMR = 1/N1820$ (N1820 has the valid range of 1 to 27)

If N1816#7 = 0, $CMR = N1820 * 0.5$ (N1820 has the valid range of 1 to 96)

Detection Unit = Command Increment / CMR

Add to existing pitch compensation

If existing pitch compensation has been read from controller, the new compensation values can be actualized cumulatively to old ones. If user wants to measure and compensate without clearing the old compensations, first carry out measurements with compensation set to active, then read the old values into CNCcomp, and make the calculation with this selection checked.

Important Note: The range of the new compensation table to be generated is decided by the current measurement file. Therefore, if part of the old compensation range is outside the current measurement range, it will be neglected in the new compensation table, which means that the old compensation values outside the current measurement range are simply lost.

8.2 Reading Parameter File

First the program asks for the parameter file name and after that, it asks if user wants to read in also old existing pitch compensation values. This makes it possible to measure and compensate machine tools without first clearing pitch compensations and thus speed up the whole measuring process.

8.3 Serial Communication Parameters

Before it is possible to read or write any parameters to Fanuc, some communication settings in the controller should be manually checked. Please refer to Fanuc 15 manuals for details

8.4 Serial Cable

Please refer to Fanuc 15 manuals for details.

8.5 Reading Parameters from Controller

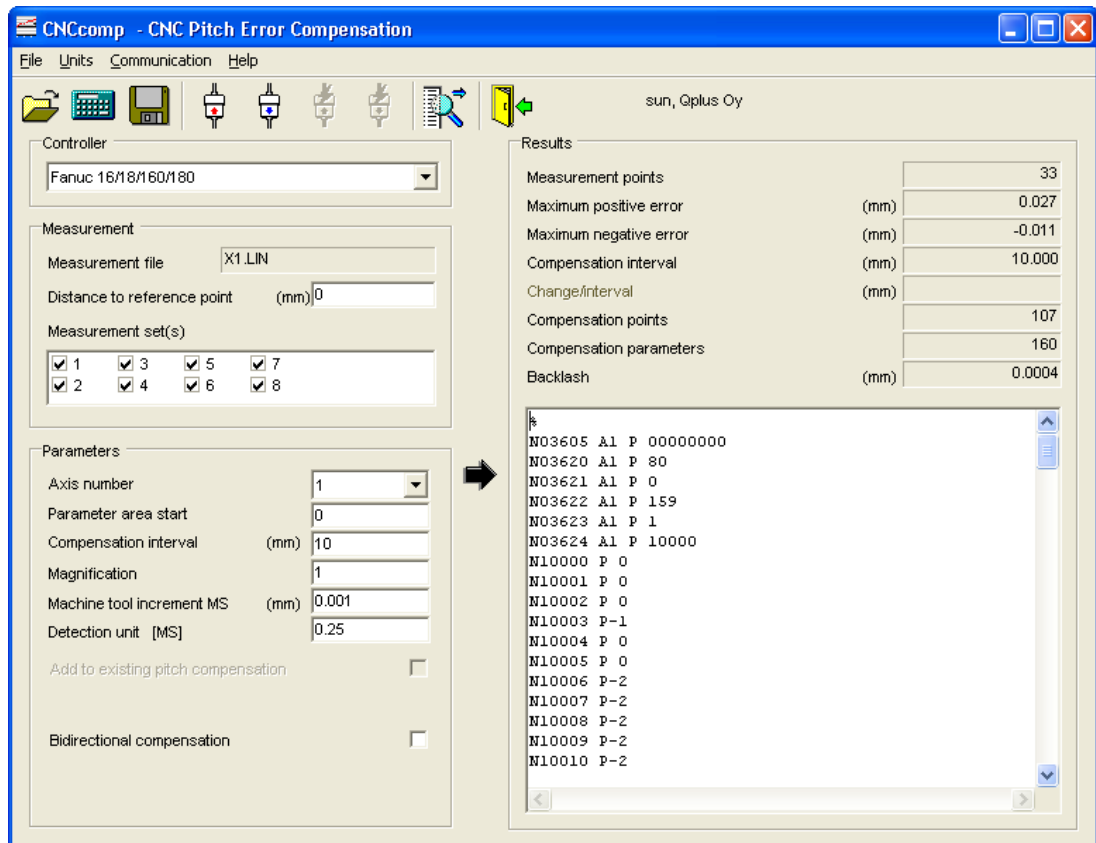
The operations are similar to the [operations of reading parameters from Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 15 manuals for detail operations with the controller.

8.6 Sending New Pitch Compensation Values to Controller

The operations are similar to the [operations of sending new pitch compensation values to Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 15 manuals for detail operations with the controller.

9 Fanuc 16/18/160/180

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Serial Communication Parameters*
- *Changing Parameters for Communication*
- *Serial Cable*
- *Reading of Parameters from Controller*
- *Sending New Pitch Compensation Values to Controller*



9.1 Parameter Options

Axis number

Axis number specifies which axis is to be compensated. The compensation file parameter options are changed accordingly.

Parameter area start

Parameter area start is the offset value to be added to the address of the compensation parameters. This is how different axis compensation parameter areas are separated. The maximum parameter address is 11023

Compensation interval

Compensation interval defines how many compensation points are needed. Decreasing the interval makes the compensation steps smaller.

Magnification

Magnification amplifies the used compensation step. If the magnification is 1 then the compensation step is the same as detection unit.

Machine Tool Increment

Machine tool increment is the used linear scale accuracy for the machine tool. Increment can be read from a downloaded parameter file (N1004).

Detection unit

Detection unit is the unit by which compensation is made. So if the Machine Tool Increment (MS) value is 0.001 mm, the detection unit is 0.25MS, namely $0.25 * 0.001 = 0.00025$ mm, compensation is 2, and magnification is 5, then compensation at that point is $0.00025 * 2 * 5 = 0.0025$ mm.

With this controller type, it is possible to read the current machine tool increment system from a file, if the current parameter area is downloaded from the controller. Parameter N1004 bit 7 and 8 defines the machine tool increment system in a following matter:

Bit	Machine tool Increment	
00	IS-B	0,001
10	IS-C	0,0001
01	IS-A	0,01

Also the Detection unit is read from the file. Detection unit is read from a parameter N1820:

If N1820 is between 2-96, detection unit = $N1004/(N1820/2)$

If N1820 is between 102-127, Detection unit = $N1004/(1/(N1820-100))$

Add to existing pitch compensation

If existing pitch compensation has been read from NC, the new compensation can be calculated cumulatively to old ones. If user wants to measure and compensate without clearing the old compensations, read first the old values to PC check this selection and carry out measurements with compensations active.

Important Note: The range of the new compensation table to be generated is decided by the current measurement file. Therefore, if part of the old compensation range is outside the current measurement range, it will be neglected in the new compensation table, which means that the old compensation values outside the current measurement range are simply lost.

Bidirectional compensation

Bidirectional compensation decides whether bidirectional pitch error compensation is enabled. It relates to the parameter N3605#0. If bidirectional compensation is enabled, CNCcomp will generate compensation values separately for positive direction and negative direction.

If the measurement is one-directional, this function will be disabled.

Parameter area start (- direction)

In case of bidirectional pitch error compensation, this is the offset value to be added to the address of the compensation parameters of negative direction. It relates to the parameter N3626.

Direction of reference position return

This option relates to the parameter N1006#5. It decides whether the direction of reference position return is positive or negative. The sign of N3627 parameter value depends on the direction of reference position return. If this value is read from a machine parameter file, it is read only.

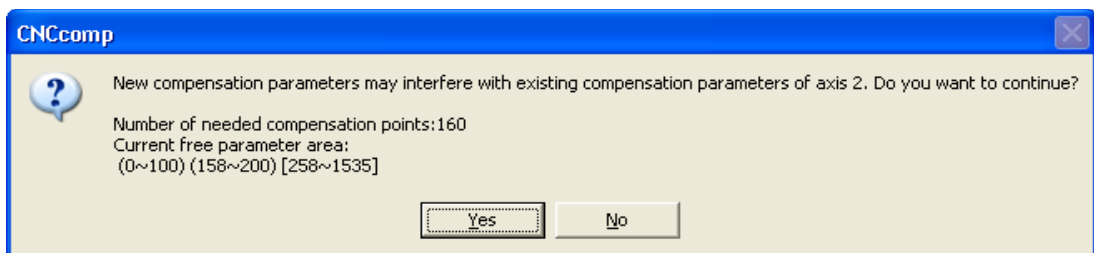
9.2 Reading Parameter File

First the program asks for the parameter file name and after that, it asks if user wants to read in also old existing pitch compensation values. This makes it possible to measure and compensate machine tools without first clearing pitch compensations and thus speed up the whole measuring process.

9.3 Calculation

The calculation is done by pressing the menu *File/Calculate Compensation File*, the toolbar button *Calculate Compensation*, or the arrow sign in the middle of the screen.

If the machine parameter file has been read before calculation, CNCcomp will also check whether there is any interference between the calculated compensation parameters and the existing compensation parameters of other axis on the same machine. If Interference has been found, a dialog box will be shown displaying a warning, the number of needed compensation points and current free parameter areas and the user can decide whether to continue or cancel the calculation. Free parameter areas with round brackets are not big enough to accommodate the current compensation points, while the free parameter areas with square brackets are big enough.



Important Notice: If user selects yes, the overlapped part of the existing parameter values will be overwritten by the new compensation value, and the existing compensation of the related axis will be broken.

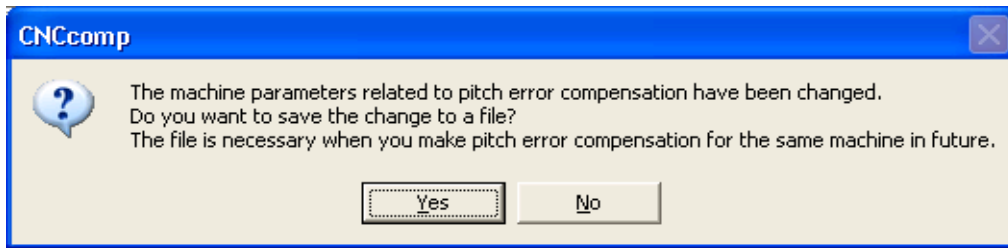
Calculation results are displayed on the right side of the screen.

9.4 Save Compensation File

Compensation file is saved by clicking the menu *File/Save* or toolbar button *Save*, after which the save file dialog is opened.

After the compensation file is saved successfully, if compensation related parameter

values has been changed, CNCcomp will ask you whether you want to save the parameter changes in a file.



If you select “Yes”, a file save dialog box will be displayed to let you specify the file name. Please notice that this parameter file is only necessary when you use CNCcomp to make pitch error compensation of the same machine in future. You should not upload this parameter to your machine.

9.5 Serial Communication Parameters

Before it is possible to read or write any parameters to Fanuc, some communication settings in the controller should be manually checked as follows:

1. Press **SYSTEM** key
2. Press **PARAM** soft key
3. Press **PAGE** ↓ to see a value of a parameter 20. It should be 0 (zero) to allow serial communication to use external RS232C-port.
4. Press again **PAGE** ↓ to see parameter 101. Check the value of bit 0 (SB2), if it is 1 (default) there are 2 stop bits, otherwise just 1 stop bit is used.

Parameter 101:

Value in bit 0 (SB2):	0	1
Stop bits:	1	2

5. Parameter 102 should be set to 0 (zero). This means that RS232C communication protocol is used.
6. Transfer baud rate is set in parameter 103 according the following table (4800baud is default):

Parameter 103:

Value:	1	2	3	4	5	6	7	9	10	11	12
Baud rate:	50	100	110	150	200	300	600	1200	4800	9600	19200

It is recommended not to change the parameters of Fanuc if not absolutely necessary. Instead user can use the valid parameter values in CNCcomp and write down the values of parameters 101 - SB2 and 103. But if there are some other values than zero (0) in parameters 20 or 102, they must be changed.

Summary of actions:

- Change parameters 20 and 102 to zero.
- Check values of parameters:
- Bit 0 (SB2) in parameter 101
- Parameter 103

Parameter values:

Baud rate:	<i>parameter 103</i>
Data bits:	7
Stop bits:	<i>parameter 101</i>
Parity:	even
Data flow control:	xon/xoff
Communication port:	<i>according to the user's cable connection in PC</i>

9.6 Changing of Parameters for Communication

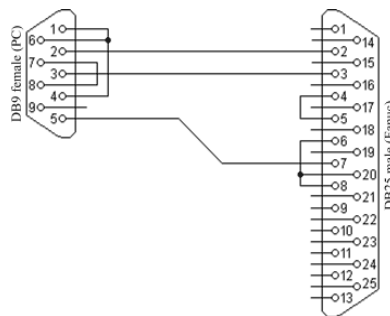
1. Press **Emergency switch** on
2. Press **OFFSET SETTING** key
3. Press **SETTING** soft key
4. Key in value **1** (one) for **PARAMETER WRITE** and press **INPUT** soft key
5. *Parameter write enabled alarm will occur*
6. Press **SYSTEM** key
7. Press **PARAM** soft key
8. Use **PAGE ↓** and **↓** keys to arrive to the row of the parameter 20. Key in value **0** (zero) and press **INPUT** soft key.
9. Use **PAGE ↓** and **↓** keys to arrive to the row of the parameter 102. Key in value **0** (zero) and press **INPUT** soft key.

The serial communication settings are now ready for file transfer. *Parameter Write* should be disabled for the time of measurements:

10. Press **OFFSET SETTING** key
11. Press **SETTING** key
12. Key in value **0** (zero) for *PARAMETER WRITE* and press **INPUT** soft key
13. Parameter write enabled alarm will disappear
14. Release **Emergency switch**

9.7 Serial Cable

RS232 cable shouldn't be longer than 15 meters. The pins of the cable should be connected according the following diagram. Fanuc has usually a 25-pin connector and a 9-pin connector is more commonly used in PCs.



9.8 Reading Parameters from Controller

Put first CNCcomp software ready to wait for data and give a name for parameter file. Then prepare Fanuc 16 controller to send the parameter table using the following sequence:

1. Press **SYSTEM** key
2. Press **PARAM** soft key
3. Press (**OPRT**) soft key
4. Press [**+**] soft key
5. Press **PUNCH** soft key
6. Press **EXEC** soft key

Reading of all the parameters to the computer will take approximately 5 minutes, so please be patient. Current pitch compensation table should also be sent to the computer: First make CNCcomp ready for receiving file from controller and the press following keys:

1. Press **SYSTEM** key
2. Press *PARAM* soft key
3. Press *[+]* soft key
4. Press *PITCH* soft key
5. Press (*OPRT*) soft key
6. Press *[+]* soft key
7. Press *PUNCH* soft key
8. Press *EXEC* soft key

After these two files have been read into the computer, they can be read into CNCcomp via *Read parameters from file* function.

9.9 Sending New Pitch Compensation Values to Controller

First 'Parameter write' setting has to be enabled:

1. Press **Emergency switch** on
2. Press **OFFSET SETTING** key
3. Press *SETTING* soft key
4. Key in value **1** (one) for *PARAMETER WRITE* and press *INPUT* soft key
5. *Parameter write enabled* alarm will occur

Next, get Fanuc ready to wait for new compensation values:

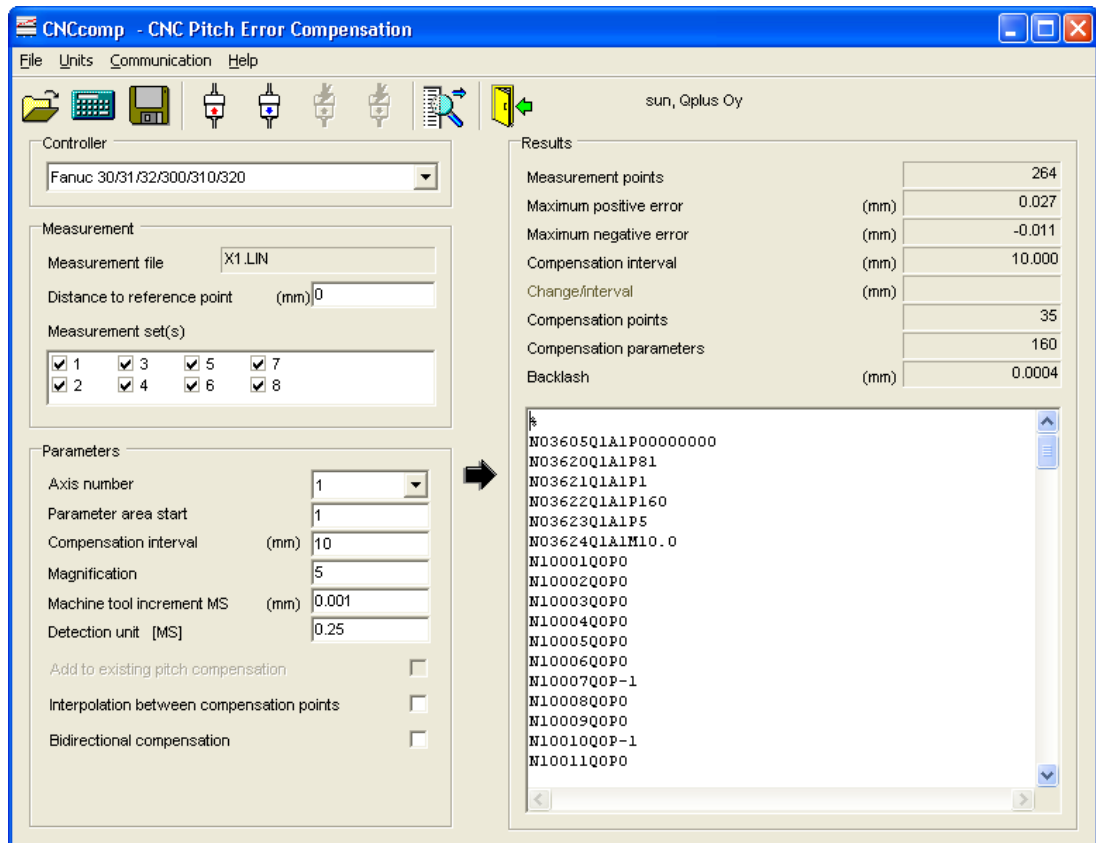
6. Press **SYSTEM** key
7. Press *PARAM* soft key
8. Press *[+]* key
9. Press *PITCH* soft key
10. Press (*OPRT*) soft key
11. Press *[+]* soft key
12. Press *READ* soft key
13. Press *EXEC* soft key
14. Send the new compensation file from CNCcomp software

It takes approximately half a minute to send one compensation file. *Parameter write* should be disabled after the file transfer to protect a controller against changes made by accident:

1. Press OFFSET SETTING key
2. Press SETING soft key
3. Key in value 0 (zero) for PARAMETER WRITE and press INPUT soft key.
4. Parameter write enabled alarm will disappear
5. Release Emergency switch
6. The Fanuc controller should be restarted to activate new compensation values.

10 Fanuc 30/31/32/300/310/320

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Serial Communication Parameters*
- *Serial Cable*
- *Reading Parameters from Controller*
- *Sending New Pitch Compensation Values to Controller*



10.1 Parameter Options

Axis number

Axis number specifies which axis is to be compensated. The compensation file parameter options are changed accordingly. The valid range for is 1 to 32 for Fanuc 30/300, 1 to 20 for Fanuc 31/310, and 1 to 9 for Fanuc 32/320.

Parameter area start

Parameter area start is an offset value to be added to the address of the compensation parameters. This is how different axis compensation parameter areas are separated. The valid range is 0 to 1535

In case of bidirectional compensation, it is the offset value for positive directional compensation. The valid range is 0 to 1535. In case of extended bidirectional compensation, the valid range is from 0 to 2559.

Compensation interval

Compensation interval defines the distance between two neighboring compensation points. It decides how many compensation points are needed. By decreasing the interval smaller compensation steps are needed.

Magnification

Magnification amplifies the used compensation step. If the magnification is 1 then the

compensation step is the same as detection unit.

Machine Tool Increment

Machine tool increment is the used linear scale accuracy for the machine tool. Increment can be read from a downloaded machine parameter file. If this value is read from a machine parameter file, it is read only.

With this controller type, it is possible to read the current machine tool increment system from the file downloaded from the controller. Parameter N01013#0, N01013#1, N01013#2, and N01013#3 define the machine tool increment system in the following way:

N1013 #0	N1013 #1	N1013 #2	N1013 #4	Least Command Increment	Abbreviation
0	0	0	0	0.001 mm/0.0001 inch	IS-B
1	0	0	0	0.01 mm/0.001 inch	IS-A
0	1	0	0	0.0001 mm/0.00001 inch	IS-C
0	0	1	0	0.00001 mm/0.000001 inch	IS-D
0	0	0	1	0.000001mm /0.0000001 inch	IS-E

Detection Unit

Detection unit is the unit by which compensation is made. If this value is read from parameter file, then it is read only.

So if the Machine Tool Increment (MS) value is 0.001 mm, the detection unit is 0.25MS, namely $0.25 * 0.001 = 0.00025$ mm, compensation is 2, and magnification is 5, then compensation at that point is $0.00025 * 2 * 5 = 0.0025$ mm.

A parameter called command multiplier (CMR) can be read from the parameter file N01820. It indicates the ratio of the lease command increment to the detection unit for each axis.

Least command increment = detection unit \times command multiplier.

The setting of a command multiplier is as follows:

- (1) When command multiplier is 1 to 1/27, the value of N01820 = 1/command multiplier + 100, hence has the valid range 101 to 127
- (2) When command multiplier is 0.5 to 48, the value of N01828 = 2 \times command multiplier, hence has the valid range 1 to 96

Add to existing pitch compensation

If existing pitch compensation has been read from controller, the new compensation values can be actualized cumulatively to old ones. If user wants to measure and

compensate without clearing the old compensations, first carry out measurements with compensation set to active, then read the old values into CNCcomp, and make the calculation with this selection checked.

Important Note: The range of the new compensation table to be generated is decided by the current measurement file. Therefore, if part of the old compensation range is outside the current measurement range, it will be neglected in the new compensation table, which means that the old compensation values outside the current measurement range are simply lost.

Interpolation between compensation points

The option decides the interpolation type of pitch error compensation and relates to the value of parameter N03605#1. In interpolation type pitch error compensation, a compensation value at each point in each error completion point interval is divided for output of one pulse at equally spaced intervals.

Bidirectional compensation

Bidirectional compensation decides whether bidirectional pitch error compensation is enabled. It relates to the parameter N03605#0. If bidirectional compensation is enabled, CNCcomp will generate compensation values separately for positive direction and negative direction.

If the measurement is one-directional, this function will be disabled.

Parameter area start (- direction)

In case of bidirectional pitch error compensation, this is the offset value to be added to the address of the compensation parameters of negative direction. It relates to the parameter N03626.

Direction of reference position return

This option relates to the parameter N01006#5. It decides whether the direction of reference position return is positive or negative. The sign of N03627 parameter value depends on the direction of reference position return. If this value is read from a machine parameter file, it is read only.

10.2 Reading Parameter File

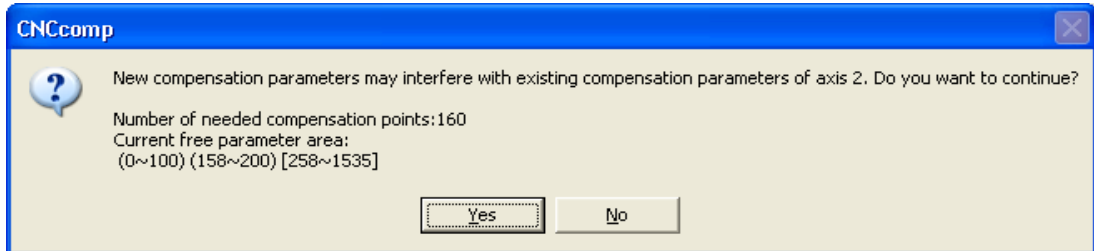
First the program asks for the parameter file name and after that, it asks if user wants to read in also the existing pitch compensation values. This makes it possible to measure and compensate machine tools without first clearing pitch compensations and thus speed up the whole measuring process.

10.3 Calculation

The calculation is done by pressing the menu *File/Calculate Compensation File*, the toolbar button *Calculate Compensation*, or the arrow sign in the middle of the screen.

If the machine parameter file has been read before calculation, CNCcomp will also

check whether there is any interference between the calculated compensation parameters and the existing compensation parameters of other axis on the same machine. If Interference has been found, a dialog box will be shown displaying a warning, the number of needed compensation points and current free parameter areas and the user can decide whether to continue or cancel the calculation. Free parameter areas with round brackets are not big enough to accommodate the current compensation points, while the free parameter areas with square brackets are big enough.



Important Notice: If user selects yes, the overlapped part of the existing parameter values will be overwritten by the new compensation value, and the existing compensation of the related axis will be broken.

Calculation results are displayed on the right side of the screen.

10.4 Save Compensation File

Compensation file is saved by clicking the menu *File/Save* or toolbar button *Save*, after which the save file dialog is opened.

After the compensation file is saved successfully, if compensation related parameter values has been changed, CNCcomp will ask you whether you want to save the parameter changes in a file.



If you select “Yes”, a file save dialog box will be displayed to let you specify the file name. Please notice that this parameter file is only necessary when you use CNCcomp to make pitch error compensation of the same machine in future. You should not upload this parameter to your machine.

10.5 Serial Communication Parameters

Before it is possible to read or write any parameters to Fanuc, some communication settings in the controller should be manually checked. Please refer to Fanuc 30/31/32/300/310/320 manuals for details

10.6 Serial Cable

Please refer to Fanuc 30/31/32/300/310/320 manuals for details.

10.7 Reading Parameters from Controller

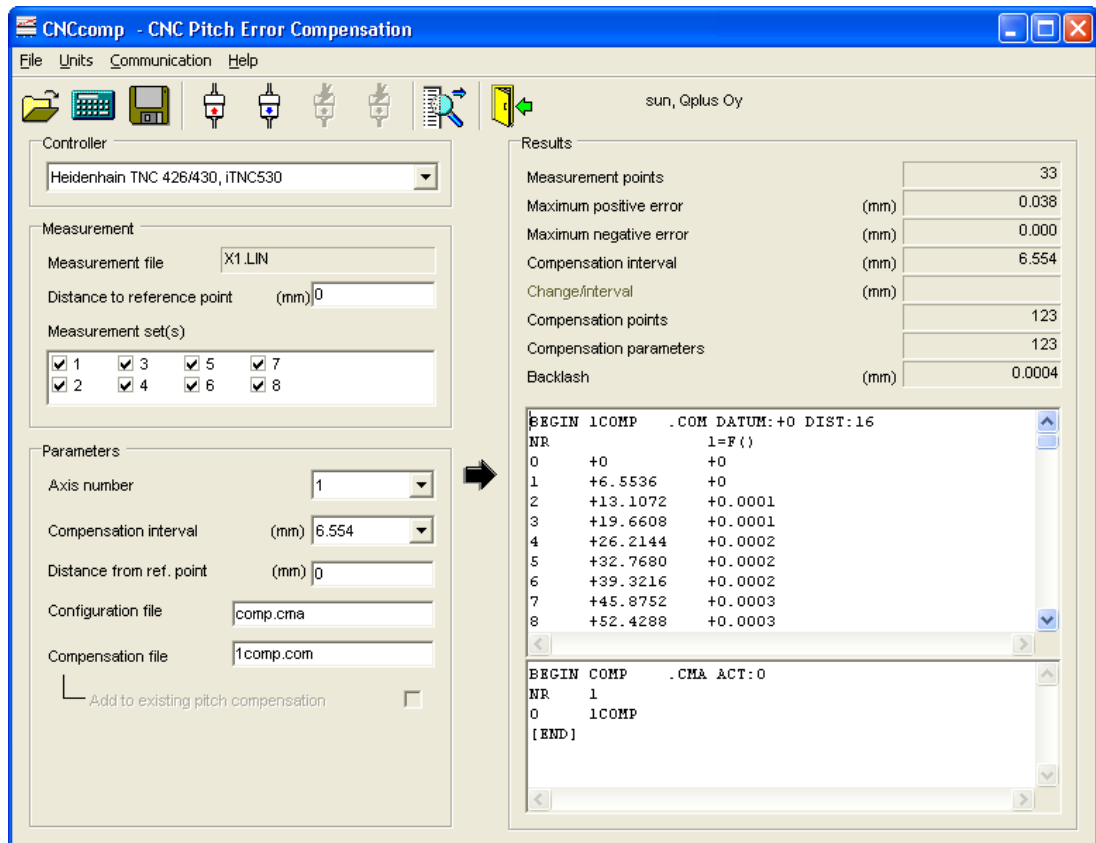
The operations are similar to the [operations of reading parameters from Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 30/31/32/300/310/320 manuals for detail operations with the controller.

10.8 Sending New Pitch Compensation Values to Controller

The operations are similar to the [operations of sending new pitch compensation values to Fanuc 16/18/160/180 controller](#). Please refer to Fanuc 30/31/32/300/310/320 manuals for detail operations with the controller.

11 Heidenhain TNC 426/430, iTNC530

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Preparation of Controller for Pitch Compensation*
- *Reading Parameters from Controller*
- *Serial Cable*
- *Sending New Pitch Compensation Values to Controller*



11.1 Parameter Options

Axis number

Axis number specifies which axis is to be compensated. This changes the name of the compensation file and the column that it affects. If current compensation settings have been read in earlier, a change of axis will modify *Compensation interval* and *Distance of table from datum* to active settings.

Compensation interval

Compensation interval defines distance between consecutive compensation points. Intervals can only be fixed sizes ($2^n / 10000$) and thus users have to select a suitable step from predefined choices. Because the amount of compensation points per table is restricted to 256, sometimes it might be necessary to increase step size. CNCcomp will show a message if the step size is too small.

Distance of table from reference point

This setting determines the start of compensation table from machine tool datum. When creating a new table, this value is by default the negative of *Distance to reference point* of measurement. However if old compensations are already active, it might be better to use the old setting.

Configuration file

The name of compensation configuration file can be changed. If old compensation settings have been read in, all those settings are written to the new file as well.

Compensation file

This is the file where actual compensations of an axis are written. Notice that in one file there can be many columns. Each column compensates one axis, a row in the file is chosen based on the position of the current axis. CNCcomp compensates only positioning and thus a row is selected based on the location of the same axis it affects.

Add to existing pitch compensation

If the old pitch compensation file has been read from controller, the new compensation values can be actualized cumulatively to old ones. If one wants to measure and compensate without clearing the old compensations, first carry out measurements with compensation set to active, then read the old values into CNCcomp, and make the calculation with this selection checked.

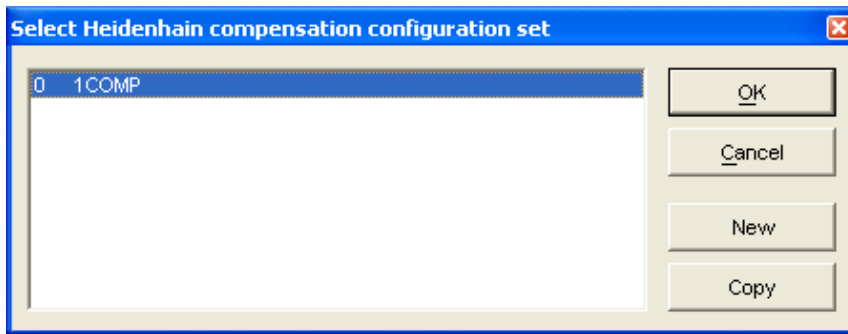
11.2 Saving Compensation file

Compensation file is saved by clicking the menu item *File/Save* or toolbar button *Save*. The *save file* dialog is opened, asking for the name of compensation file. User can give the same name as *Compensation file* setting in screen, but this is not compulsory. Different name just causes configuration file to have wrong file name entry. However one should choose the right file extension: .COM or .VNC. The link program might require specific extension (TNCremo works with both extensions, but old link programs require .VNC extension).

Program saves configuration file into the same directory without asking. It will use file extension .CMA if user choose .COM for compensation file. Otherwise extension will be .SNC. A message window will appear to tell the file name that has been used.

11.3 Reading Parameter file

Program asks for *Heidenhain configuration file*. This file determines collection of actual compensation files used in the controller. Hence if the configuration file and corresponding compensation files have been downloaded from the controller, CNCcomp can read all the information. Configuration file has the file extension '.cma' or '.snc' and compensation files have the extension '.com' or '.vnc'. If the configuration file consists of more than one configuration sets, CNCcomp will ask the user to select a set to edit. One has to select the set that was active in laser measurement. Based on the selection the program reads active compensation files and creates later new compensation files on top of them. It is also possible to create a new set at that moment (press button *New*), which causes the new tables to be created on empty basis. *Copy* duplicates the selected set. It also copies the compensation files of the source set. After the selection of compensation set, the program reads corresponding compensation files automatically (they have to be located in the same directory as configuration file and have the same names as they have in the controller).



It is not necessary to download all compensation files from controller (if those axes are not going to be compensated). However, it is necessary to download the configuration file. *Read parameters from file* functionality doesn't work without a configuration file. However, users can still create compensation files (and a new configuration file). In that case no information of active compensations in the controller is available. Thus when new compensation files are uploaded to the controller, old compensation values on corresponding axes are lost.

11.4 Preparation of Controller for Pitch Compensation

First of all, a nonlinear compensation of axis must be activated in the controller. This is set with machine parameter 730. This setting can be edited as follows:

1. Set the controller to programming (*Program Edit /Save*) mode
2. Press **MOD** button
3. Key in access code **807667**, press **ENT** button.
4. Press *MP EDIT* soft key
5. Browse through machine parameter area with *PAGE DOWN* and *PAGE UP* soft keys so that the cursor moves to the row of machine parameter 730
6. Set bits to one (1) for the axes to be compensated. Bit 0 correspond to axis 1 and so on.
7. Press **END** button to finish editing.

If nonlinear pitch compensation is enabled, it is highly probable that some files have been used for pitch compensation. Thus those files should be transferred to the computer for the current setting to be read.

11.5 Reading Parameters from Controller

It is recommended to use existing data link to user's Heidenhain TNC controller instead of the functionality of CNCcomp. The instruction below assumes that TNCCremo software has been set up already.

1. Set the controller to programming (*Program Edit /Save*) mode.

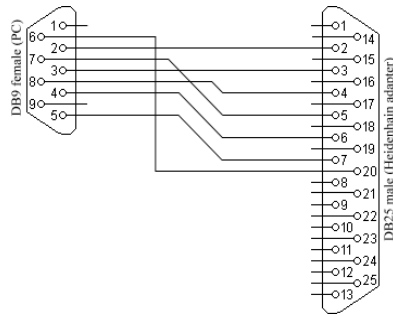
2. Press **MOD** button.
3. Key in access code **807667**, press **ENT** button.
4. Press **PGM MGT** button.
5. Using up and down arrow buttons to select **PLC:** in the left column. Press **ENT** button.
6. The main directory of TNC's PLC data is displayed. Press **right arrow** button so that **OEM.SYS** file can be selected. Press **ENT** button.
7. Look if this file already has a row starting with a key text: **TABCMA=**. The file name following this key the text that indicates current compensation configuration file (***.CMA**). Write down the name of the file. If this row cannot be found, exit from this chapter - no compensation has been activated previously. Be sure not to make changes in this file, they might be fatal! Press **END** button.
8. Browse to the configuration file and view its contents (by pressing **ENT** button). The actual compensation files can be seen, which are now active in the controller. Write down the names of all those files. (Compensation files have an extension **.COM**, but not shown here) Press **END** button.
9. Transfer all those compensation related files to **PC**. First press **WINDOW** soft key.
10. If necessary use **PATH** soft key to change path of the right side column to "**RS232:**" TNCremo program should be ready on the computer by this time.
11. Select files to be transferred in left column and press **COPY** soft key.
12. When all necessary files have been transferred, one can exit from *program management* of controller.
13. If necessary one can copy the files to a different computer so that CNCcomp can access them.
14. After these files have been read into the computer, they should be read into the CNCcomp using *Read parameters from file* function. CNCcomp first asks for the name and location of configuration file (extension **.CMA** or **.SNC**). After the configuration file has been read successfully, it will continue until all the compensation files (with extension **.COM** or **.VNC**) defined in the configuration file are read.

Note:

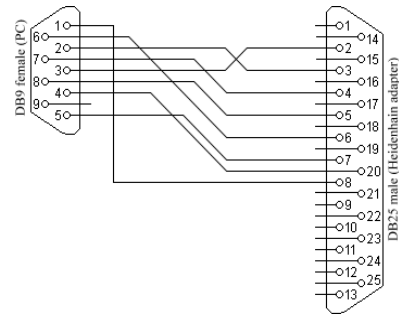
TNCremo software can be downloaded free of charge from Heidenhain web site at <http://www.heidenhain.com/techdata.html> . Follow the link *Document/Downloads* and then *Download Area*.

11.6 Serial Cable

RS232 cable shouldn't be longer than 3 meters. The pins of the cable should be connected according the following diagram. Heidenhain TNC has a 25-pin connector. A Computer commonly has a 9-pin connector. The diagram on the left side illustrates the cable schema to connect to the Heidenhain TNC when it has a connector in a chassis of a machine tool (adapter block). This is the most common case.



PC to TNC adapter



PC to X21 connector in TNC

If the computer have to be connected directly to a controller unit, cable schema illustrated on the right side should be used. More information on cables can be found in the User Manual of Heidenhain TNC.

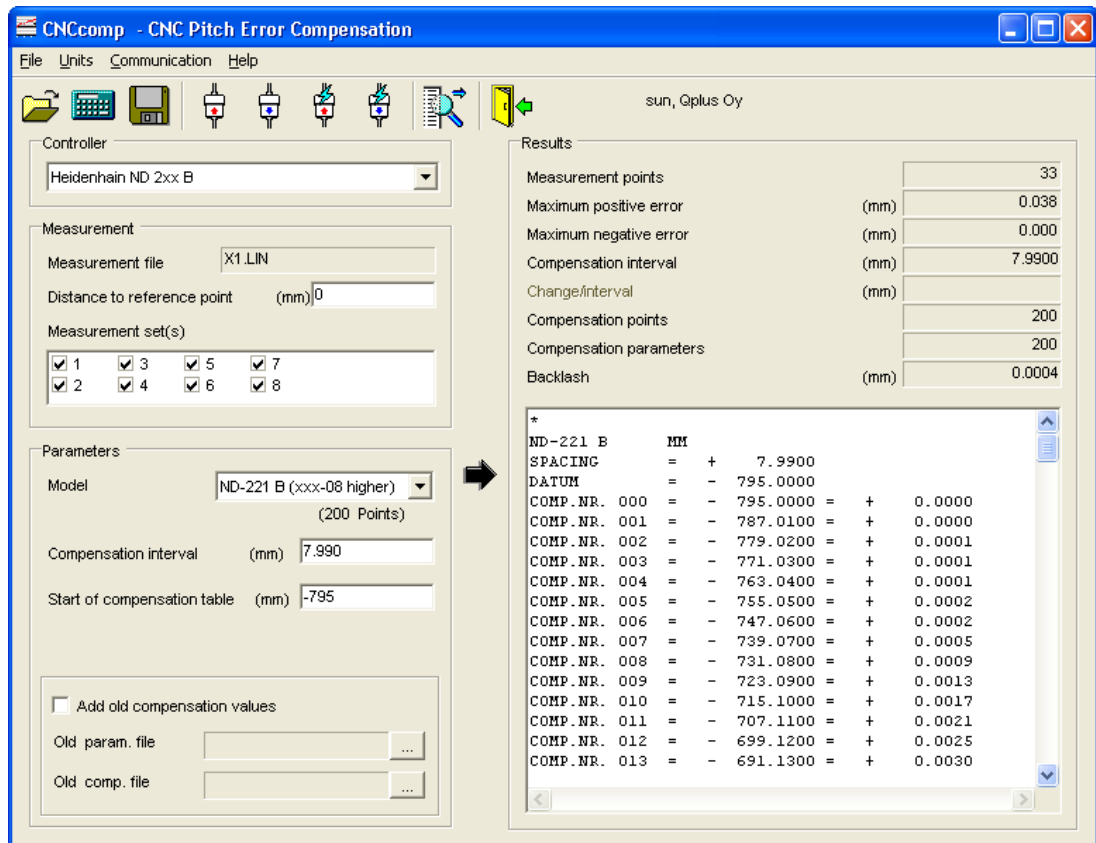
11.7 Writing New Pitch Compensation Values to Controller

When new compensation files are ready, they can be transferred to Heidenhain TNC for activation. It is recommended to transfer those files using Heidenhain TNCremo software.

1. Copy compensation files to a directory accessible from link software.
2. Set the controller to programming (*Program Edit /Save*) mode.
3. Press **MOD** button.
4. Key in access code **807667**, press **ENT** button.
5. Press **PGM MGT** button.
6. Using up and down arrow buttons select **PLC:** in the left column. Press **ENT** button.
7. Press **WINDOW** soft key.
8. If necessary use **PATH** soft key to change path of the right side column to **RS232:** TNCremo program should be ready on a PC this time.
9. Select files to be transferred (configuration file has the extension **.CMA** or **.S** and compensation files have the extension **.COM** or **.V**) in a right column and press **COPY** soft key.
10. When all necessary files have been transferred, close transfer mode by pressing **WINDOW** soft key.
11. Select new compensation configuration file in a **PLC:** directory of a controller and press **ENT** button to view it. One can close it right away using **END** button. This activates new configuration file (by writing a new "TABCMA="... entry in OEM.SYS file).
12. Exit from *program management* of a controller. New compensations are activated immediately and the machine tool is ready for verification measurements.

12 Heidenhain ND 2xxB

- *Reading Measurement File*
- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Serial Cable*
- *Manually Receiving Compensation File from ND 2xxB*
- *Automatically Receiving Compensation File from ND 2xxB*
- *Manually Receiving Parameter File from ND 2xxB*
- *Automatically Receiving Parameter File from ND 2xxB*
- *Manually Sending Compensation File to ND 2xxB*
- *Automatically sending Compensation File to ND 2xxB*
- *Manually Sending Parameter File to ND 2xxB*
- *Automatically sending Parameter File to ND 2xxB*



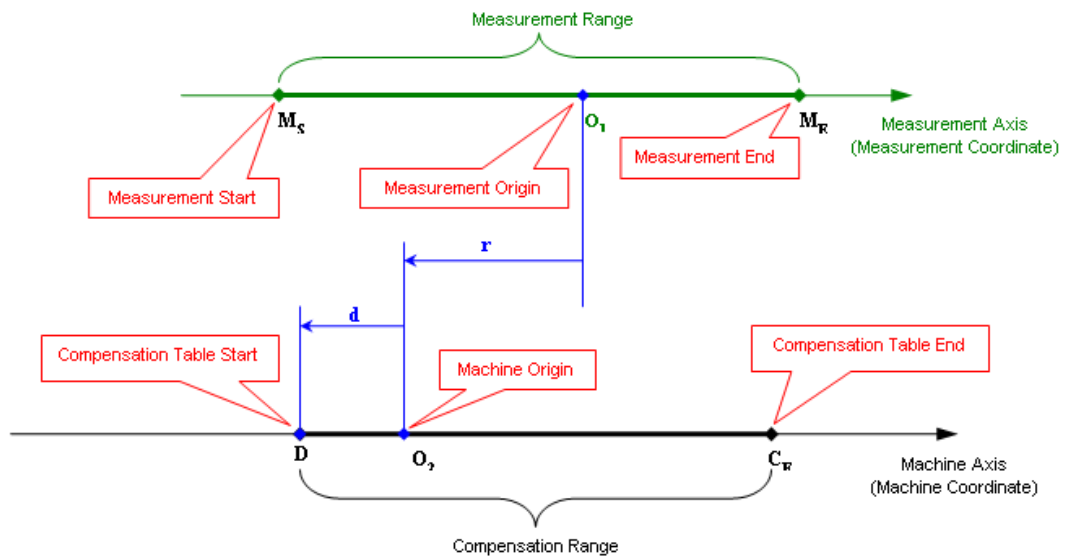
12.1 Measurement File Options

Distance to reference point

Distance to reference point controls the location of the measurement range in machine.

If the position data in the measurement file are taken in machine coordinates, this value should be 0. If not, as shown in the figure below, this value is the distance from origin of the machine O_2 to the origin of measurement O_1 , which is value r . Please also notice that if the machine origin O_2 locates at the left of the measurement origin O_1 , then $r < 0$. If the machine origin O_2 locates at the right of the measurement origin O_1 , then $r > 0$.

By default, the measurement range is aligned with the machine so that the machine origin O_2 is overlapped with measurement origin O_1 . Please notice that users may have cases which are different from the default setting.



Measurement and Machine Coordinates

If measurement file includes several measurement runs (sets), they all are listed in *Measurement set(s)* list box. It is possible to select just one or several sets for the compensation. If more than one set is selected, the average values of all selected measurement sets are used for the compensation.

12.2 Parameters Options

Model

Model specified the model type of the display unit.

Important Note: Depending on the software version, the display units can have either 64 compensation points with fixed intervals or 200 compensation points with free interval. Please check the backside of the display units for the software version. If the software version is 349797-08 or higher, then *ND2x1 B (xxx-08 higher)* should be selected. Otherwise select *ND2x1 B (xxx-07 lower)*.

Compensation interval

Compensation interval defines distance between consecutive compensation points.

For the units with software version 349797-07 or lower, interval can only be a fixed size ($2^n/1000$) and thus one has to select a suitable step from predefined choices. Because amount of compensation points per table is restricted to 64, sometimes it might be necessary to increase step size. Program will show a message if the step size is too small.

For the units with software version 349797-08 or higher, interval can be decided freely. Program will give warning if the compensation table cannot reach the end of the measurement range with the specified interval (Referring to the figure above, the end point of compensation table C_E will locate at the right of measurement end point M_E).

Start of compensation table

This value determines the start of compensation table in machine coordinate, which is d in the figure above. If compensation start point D locates at the right of machine origin O_2 , then $d > 0$. If compensation start point D locates at left of machine origin O_2 as in the figure above, then $d < 0$. This value is also the datum in compensation table.

Add old compensation values

If existing pitch compensation file has been read in, the new compensation can be actualized cumulatively to old ones. If one wants to measure and compensate without clearing the old compensations, first carry out measurements with compensations enabled in the machine tool, then read in the old parameter file and compensation file and make calculation with this selection checked.

If this item is checked, the compensation values in the new compensation/parameter files are the sum of old and new compensation values. Otherwise, only new compensation values are saved in the new compensation/parameter files.

Old param. file

Load the old parameter file and display its file name.

Old comp. file

Load the old compensation file and display its file name.

12.3 Reading Parameter File

Click menu item *File/Read Parameters from File*, the program asks the user to select the parameter file from file open dialog. The loaded parameter file will be displayed in *Old param. file* box.

12.4 Serial Cable

The RS-232-C/V.24 interface of Heidenhain ND 221B/231B/281B display unit is used to transfer measured data in ASCII format between PC and the display unit. One can use a connecting cable with full wiring or simplified wiring. The wiring schemas can be found in the User's Manual of the display unit. A cable with full wiring is also available from HEIDENHAIN. On this type of cable, pin 6 and pin 8 are additionally connected over a jumper.

The maximum cable length is 20 m (66 ft).

The wiring schema can be found on Page 29 (ND221B), Page 37 (ND231B), and Page 46 (ND281B) of the User Manuals.

12.5 Manually Receiving Compensation File from ND 2xxB

1. CNCcomp: Select menu *Communication/Receive File from Controller Manually...* or click toolbar button *Receive File from Controller Manually*, a dialog is displayed to let the user give the name of the compensation file to be saved.
2. CNCcomp: Give the file name and confirm with *Save* button, *Communication Parameters* dialog is displayed.
3. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press *OK*. CNCcomp is then ready to receive data. A dialog is displayed showing the number of bytes received.
4. ND 221B/231B/281B: Press **CL** key together with **MOD** key.
5. ND 221B/231B/281B: Press **I/2** key multiple times until *P00 CODE* is displayed.
6. ND 221B/231B/281B: Enter the code number **486153** and confirm with **ENT** key. *"TRANSFER"* is displayed.
7. ND 221B/231B/281B: Proceed by pressing the **ENT** key, then press - key if necessary until *"SEND CMP."* is displayed.
8. ND 221B/231B/281B: Press the **ENT** key, the compensation file is sent over the RS-232-C/V.24 interface. After the sending is completed, the display returns to *"TRANSFER"*.
9. CNCcomp: After the file is completely received, the message *"File received and saved"* is displayed.
10. ND 221B/231B/281B: Press - key to proceed with other data transfer tasks, or exit transfer function by press **CL** key.

12.6 Automatically Receiving Compensation File from NC 2xxB

Note: This function is only available with software version 349797-08 or higher. Please check the backside of the display units for the software version.

1. ND 221B/231B/281B: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Receive File from Controller Automatically ...* or click toolbar button *Receive File from Controller Automatically*.
3. CNCcomp: In the dialog *Select File Type to Be Received*, select *X compensation file*, and press *OK*.
4. CNCcomp: A dialog is displayed which let the user give the name of the compensation file to be saved. Give the file name and confirm with *Save* button.
5. CNCcomp: If the transmission of compensation file is successful, the message *"File received and saved"* is displayed. Otherwise, an error message is presented.

12.7 Manually Receiving Parameter File from ND 2xxB

1. CNCcomp: Select menu *Communication/Receive File from Controller manually ...* or click toolbar button *Receive File from Controller Manually*, a dialog is displayed to let the user give the name of the parameter file to be saved.
2. CNCcomp: Give the file name and confirm with *Save* button, *Communication Parameters* dialog is displayed.
3. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press OK. CNCcomp is then ready to receive data, a dialog is displayed showing the number of bytes received.
4. ND 221B/231B/281B: Press **CL** key together with **MOD** key.
5. ND 221B/231B/281B: Press **I/2** key for multiple times until “*P00 CODE*” is displayed.
6. ND 221B/231B/281B: Enter the code number **486153** and confirm with **ENT** key. “*TRANSFER*” is displayed.
7. ND 221B/231B/281B: Proceed by pressing the **ENT** key, and then press - key multiple times until “*SEND PARAM.*” is displayed.
8. ND 221B/231B/281B: Press the **ENT** key, the parameter file is sent over the RS-232-C/V.24 interface. After output is completed, the display returns to “*TRANSFER*”.
9. CNCcomp: After the file is completely received, the message “*File received and saved*” is displayed.
10. ND 221B/231B/281B: Press - key to proceed with other data transfer tasks, or exit transfer function by press **CL** key.

12.8 Automatically Receiving Parameter File from ND 2xxB

Note: This function is only available software version 349797-08 or higher. Please check the backside of the display unit for the software version.

1. ND 221B/231B/281B: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Receive File from Controller Automatically ...* or click toolbar button *Receive File from Controller Automatically*.
3. ND 221B/231B/281B: In the dialog *Select File Type to Be Received*, select *Receive parameter file*, and press **OK**.
4. CNCcomp: A dialog is displayed which let the user give the name of the parameter file to be saved. Give the file name and confirm with *Save* button.
5. CNCcomp: If the transmission of parameter file is successful, the message *File received and saved* is displayed. Otherwise, an error message is presented.

12.9 Manually Sending Compensation File to ND 2xxB

1. ND 221B/231B/281B: Press **CL** key together with **MOD** key.
2. ND 221B/231B/281B: Press **I/2** key for multiple times until “*P00 CODE*” is displayed.
3. ND 221B/231B/281B: Enter the code number **486153** and confirm with **ENT** key. “*TRANSFER*” is displayed.
4. ND 221B/231B/281B: Proceed by pressing the **ENT** key, and then press - key multiple times until “*REC. CMP.*” is displayed.
5. CNCcomp: Select menu *Communication/Send File to Controller Manually ...* or click toolbar button *Send File to Controller Manually*, a dialog is displayed to let the user select the compensation file to be sent.
6. CNCcomp: Give the file name and confirm with *Open* button, *Communication Parameters* dialog is displayed.
7. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press **OK**. CNCcomp then sends the compensation file. After CNCcomp finishes sending the compensation file, the message “*The file has been sent successfully*” is displayed.
8. ND 221B/231B/281B: After the file is completely received, the display unit returns to “*TRANSFER*”.
9. ND 221B/231B/281B: Press - key to proceed with other data transfer tasks, or exit transfer function by press **CL** key.

12.10 Automatically Sending Compensation File to ND 2xxB

Note: This function is only available with software version 349797-08 or higher. Please check the backside of the display unit for its software version.

1. ND 221B/231B/281B: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Send File to Controller Automatically...* or click toolbar button *Send File to Controller Automatically*.
3. CNCcomp: A dialog is displayed which let the user select the compensation file to be sent. Select the file and confirm with *Open* button.
4. CNCcomp: If the transmission of compensation file is successful, the message “*Sending file completed*” is displayed. Otherwise, an error message is presented.

12.11 Manually Sending Parameter File to ND 2xxB

1. ND 221B/231B/281B: Press **CL** key together with **MOD** key.
2. ND 221B/231B/281B: Press **I/2** key multiple times until **P00 CODE** is displayed.
3. ND 221B/231B/281B: Enter the code number **486153** and confirm with **ENT** key. “**TRANSFER**” is displayed.
4. ND 221B/231B/281B: Proceed by pressing the **ENT** key, and then press - key multiple times until “**REC. PARAM.**” is displayed.
5. CNCcomp: Select menu *Communication/Send File to Controller Manually ...* or click toolbar button *Send File to Controller Manually*, a dialog is displayed to let the user select the compensation file to be sent.
6. CNCcomp: Give the file name and confirm with **Open** button, *Communication Parameters* dialog is displayed.
7. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press **OK**. CNCcomp then sends the compensation file. After CNCcomp finishes sending the compensation file, the message “*The file has been sent successfully*” is displayed.
8. ND 221B/231B/281B: After the file is completed, the display unit restarts automatically.
9. ND 221B/231B/281B: Press - key to proceed with other data transfer tasks, or exit transfer function by press **CL** key.

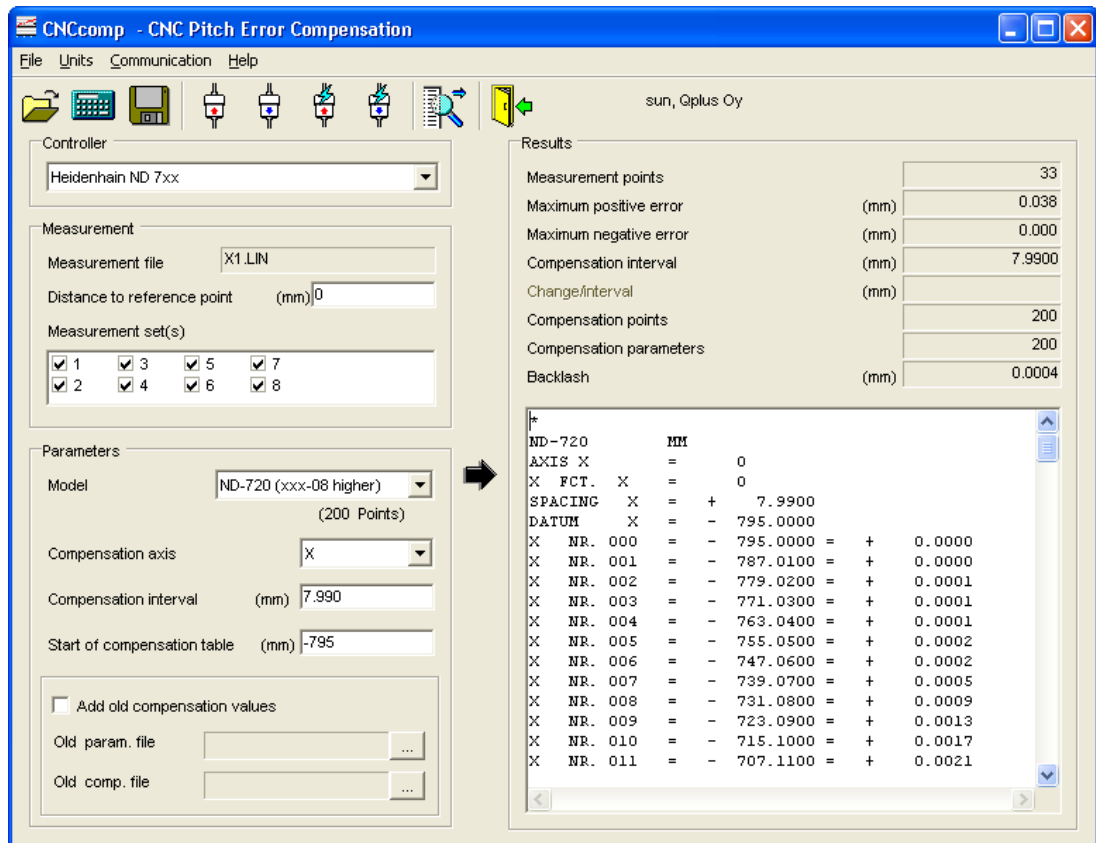
12.12 Automatically Sending Parameter File to ND 2xxB

Note: This function is only available with software version 349797-08 or higher. Please check the backside of the display units for the software version.

1. ND 221B/231B/281B: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Send File to Controller Automatically ...* or click toolbar button *Send File to Controller Automatically*.
3. CNCcomp: A dialog is displayed which let the user select the parameter file to be sent. Select the file and confirm with **Open** button.
4. CNCcomp: If the transmission of parameter file is successful, the message “*Sending File Completed*” is displayed. Otherwise, an error massaged is presented.

13 Heidenhain ND 7xxd

- *Reading Measurement File*
- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Parameter File*
- *Serial Cable*
- *Manually Receiving Compensation File from ND 7xx*
- *Automatically Receiving Compensation File from ND 7xx*
- *Manually Receiving Parameter File from ND 7xx*
- *Automatically Receiving Parameter File from ND 7xx*
- *Manually Sending Compensation File to ND 7xx*
- *Automatically sending Compensation File to ND 7xx*
- *Manually Sending Parameter File to ND 7xx*
- *Automatically sending Parameter File to ND 7xx*



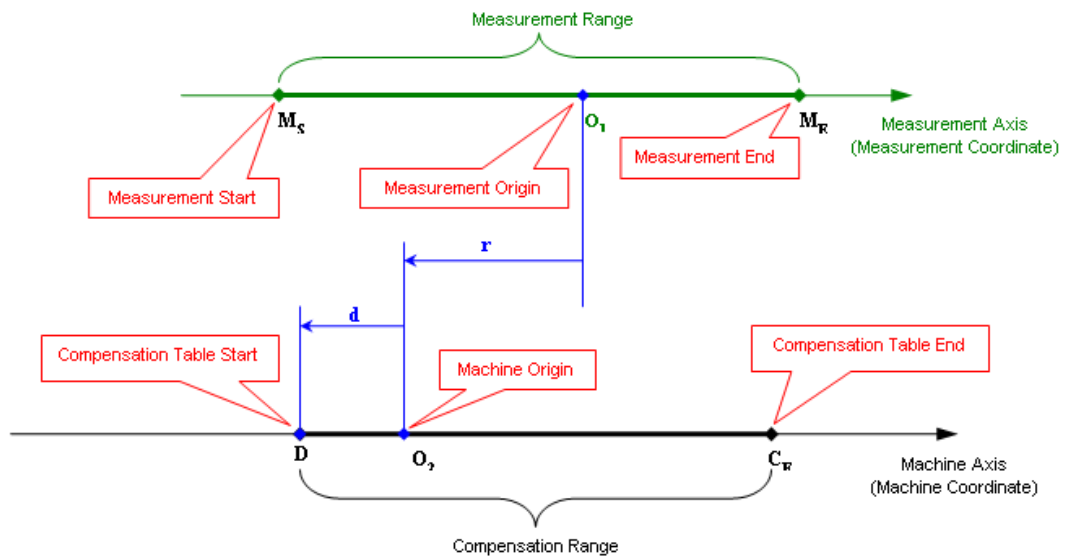
13.1 Measurement File Options

Distance to reference point

Distance to reference point controls the location of the measurement range in machine.

If position data in the measurement file are taken in machine coordinates, this value should be 0. If not, as shown in the figure below, this value is the distance from origin of the machine O_2 to the origin of measurement O_1 , which is value r . Please also notice that if the machine origin O_2 locates at the left of the measurement origin O_1 , then $r < 0$. If the machine origin O_2 locates at the right of the measurement origin O_1 , then $r > 0$.

By default, the measurement range is aligned with the machine so that the machine origin O_2 is overlapped with measurement origin O_1 . Please notice that users can have cases which are different from the default setting.



Measurement and Machine Coordinates

If measurement file includes several measurement runs (sets) they are all listed in *Measurement set(s)* list box. It is possible to select just one or several sets for the compensation. If more than one set is selected, the average values of all selected measurement sets are used for the compensation.

13.2 Parameters Options

Model

Model specifies the model type of the display unit (ND 720 or ND 760). The selection of model decides the possible selection of compensation axis. ND 720 supports 2 axes: X and Y. ND 760 supports 3 axes: X, Y and Z.

Important Note: Depending on the software version, the display units can have either 64 compensation points with fixed intervals or 200 compensation points with free interval. Please check the backside of the display units for the software version. If the software version is 246271-08 or higher, then *ND7x0 (xxx-08 higher)* should be selected. Otherwise select *ND7x0 (xxx-07 lower)*.

Compensation axis

Compensation axis specifies which axis is to be compensated. If current compensation settings have been read in earlier, Compensation Axis is set accordingly.

Compensation interval

Compensation interval defines distance between consecutive compensation points.

For the units with software version 246271-07 or lower, interval can only be a fixed size ($2^n / 1000$) and thus one has to select a suitable step from predefined choices. Because amount of compensation points per table is restricted to 64, sometimes it might be

necessary to increase step size. Program will show a message if the step size is too small.

For the units with software version 246271-08 or higher, interval can be decided freely. The program will give a warning if the compensation table cannot reach the end of the measurement range with the specified interval (the end point of compensation table C_E locates at the right of measurement end point M_E).

Start of compensation table

This value determines the start of compensation table in machine coordinate, which is d in the figure above. If compensation start point D locates at the right of machine origin O_2 , then $d > 0$. If compensation start point D locates at left of machine origin O_2 , then $d < 0$. This value is also the datum in compensation table.

Add old compensation values

If existing pitch compensation file has been read in, the new compensation can be actualized cumulatively to old ones. If one want to measure and compensate without clearing the old compensations, first carry out measurements with compensations active in the machine tool, then read in the old parameter file and compensation file, and make calculation with this selection checked.

If this item is checked, the compensation values in the new compensation/parameter files are the sum of old and new compensation values. Otherwise, only new compensation values are saved in the new compensation/parameter files.

Old param. file

Load the old parameter file and display its file name.

Old comp. file

Load the old compensation file and display its file name.

13.3 Reading Parameter File

Click menu item *File/Read Parameters from File*, the program asks the user to select the parameter file from file open dialog. The loaded parameter file will be displayed in *Old param. file* box.

13.4 Serial Cable

The RS-232-C/V.24 interface of Heidenhain ND 720/760 display unit is used to transfer measured data in ASCII format between PC and the display unit.

A connecting cable with full wiring or simplified wiring can be used. The wiring schemas can be found respectively in Page 47 of N720/760 User's Manual.

13.5 Manually Receive Compensation File from ND 7xx

1. CNCcomp: Select menu *Communication/Receive File from Controller Manually ...* or click toolbar button *Receive File from Controller Manually*, a dialog is displayed to let the user give the name of the compensation file to be saved.
2. CNCcomp: Give the file name and confirm with *Save* button, *Communication Parameters* dialog is displayed.
3. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press *OK*. CNCcomp is then ready to receive file, a dialog is displayed showing the number of bytes received.
4. ND 720/760: Select *special function* with **SPEC FCT** key.
5. ND 720/760: Press *I/2* key for multiple times until *Parameter* displayed.
6. ND 720/760: Select **ENT** key and *I/2* key, “*P00 CODE*” is displayed.
7. ND 720/760: Enter the code number **486153** and confirm with **ENT** key. “*TRANSFER*” is displayed.
8. ND 720/760: Proceed by pressing the **ENT** key.
9. ND 720/760: Depending on the target axis, press - key until “*SEND CMP. X*”, “*SEND CMP. Y*” or “*SEND CMP. Z*” is displayed (ND760 has 3 axes X, Y and Z. ND 720 has only two axes: X and Y).
10. ND 720/760: Press the **ENT** key, the parameter list is output over the RS-232-C/V.24 interface. After output is completed, the display unit returns to “*TRANSFER*”.
11. CNCcomp: After the file is completely received, the message “*File received and saved*” is displayed.
12. ND 720/760: Press - key to proceed with other data transfer tasks, or exit transfer function by press **CL** key.

13.6 Automatically Receive Compensation File from ND 7xx

Note: This function is only available with software version 246271-08 or higher. Please check the backside of the display units for the software version.

1. ND 720/760: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Receive File from Controller Automatically ...* or click toolbar button *Receive File from Controller Automatically*.
3. ND 720/760: In the dialog *Select File Type to Be Received*, select *Receive X compensation file* or *Receive Y compensation file* or *Receive Z compensation file*. (For ND 720, *Receive Z compensation file* is disabled.)

4. CNCcomp: A dialog is displayed which let the user give the name of the compensation file to be saved. Give the file name and confirm with *Save* button.
5. CNCcomp: If the transmission of compensation file is successful, the message “*File received and saved*” is displayed. Otherwise, an error message is presented.

13.7 Manually Receive Parameter File from ND 7xx

1. CNCcomp: Select menu *Communication/Receive File from Controller Manually...* or click toolbar button *Receive File from Controller Manually*, a dialog is displayed to let the user give the name of the compensation file to be saved.
2. CNCcomp: Give the file name and confirm with *Save* button, *Communication Parameters* dialog is displayed.
3. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press OK. CNCcomp is now ready to receive data, a dialog is displayed showing the number of bytes received.
4. ND 720/760: Select ***SPEC FCT*** key.
5. ND 720/760: Press ***I/2*** key multiple times until “*Parameter*” is displayed.
6. ND 720/760: Select ***ENT*** key and ***I/2*** key, “*P00 CODE*” is displayed.
7. ND 720/760: Enter the code number ***486153*** and confirm with ***ENT*** key. “*TRANSFER*” is displayed.
8. ND 720/760: Proceed by pressing the ***ENT*** key.
9. ND 720/760: Press **-** key multiple times until “*SEND PARAM.*” is displayed.
10. ND 720/760: Press the ***ENT*** key, the parameter list is output over the RS-232-C/V.24 interface. After output is completed, the display returns to “*TRANSFER*”.
11. CNCcomp: After the file is completely received, the message *File received and saved*” is displayed.
12. ND 720/760: Press **-** key to proceed with other data transfer tasks, or exit transfer function by press ***CL*** key.

13.8 Automatically Receive Parameter File from ND 7xx

Note: This function is only available with software version 246271-08 or higher. Please check the backside of the display units for the software version.

1. ND 720/760: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Receive File from Controller Automatically ...* or toolbar button *Receive File from Controller Automatically*.

3. ND 720/760: In the dialog *Select File Type to Be Received*, select *Receive parameter file*.
4. CNCcomp: A dialog is displayed which let the user give the name of the parameter file to be saved. Give the file name and confirm with *Save* button.
5. CNCcomp: If the transmission of parameter file is successful, the message "*File received and saved*" is displayed. Otherwise, an error message is presented.

13.9 Manually Send Compensation File to ND 7xx

1. ND 720/760: Press **SPEC FCT** key.
2. ND 720/760: Press **I/2** key until "*Parameter*" is displayed.
3. ND 720/760: Select **ENT** key and **I/2** key, "*P00 CODE*" is displayed.
4. ND 720/760: Enter the code number **486153** and confirm with **ENT** key. "*TRANSFER*" is displayed.
5. ND 720/760: Proceed by pressing the **ENT** key.
6. ND 720/760: If the display is not "*REC. CMP.*", press - key multiple times until "*REC. CMP.*" is displayed.
7. CNCcomp: Select menu *Communication/Send File to Controller Manually ...* or click toolbar button *Send File to Controller Manually*, a dialog is displayed to let the user select the compensation file to be sent.
8. CNCcomp: Select the file and confirm with *Open* button, *Communication Parameters* dialog is displayed.
9. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press *OK*. CNCcomp now sends the file to ND720/760.
10. CNCcomp: After CNCcomp finishes sending the file, the message "*The file has been sent successfully.*" is displayed.
11. ND 720/760: After the file transmission is completed, the display unit returns to "*TRANSFER*".
12. ND 720/760: Press - key to proceed with other data transfer tasks, or exit transfer function by press **CL** key.

13.10 Automatically Send Compensation File to ND 7xx

Note: This function is only available with software version 246271-08 or higher. Please check the backside of the display units for the software version.

1. ND 720/760: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Send File to Controller Automatically ...* or click toolbar button *Send File to Controller Automatically*.
3. CNCcomp: A dialog is displayed which let the user select the compensation file to be sent. Select the file and confirm with *Open* button.
4. CNCcomp: If the transmission of compensation file is successful, the message "*Sending file completed*" is displayed. Otherwise, an error message is presented

13.11 Manually Send Parameter File to ND 7xx

1. ND 720/760: Press **SPEC FCT** key.
2. ND 720/760: Press **I/2** key until "*Parameter*" is displayed.
3. ND 720/760: Press **ENT** key and **I/2** key, "*P00 CODE*" is displayed.
4. ND 720/760: Enter the code number **486153** and confirm with **ENT** key. "*TRANSFER*" is displayed.
5. ND 720/760: Proceed by pressing the **ENT** key.
6. ND 720/760: If the display is not "*REC. PARAM.*", press – key multiple times until "*REC. PARAM.*" is displayed.
7. CNCcomp: Select menu *Communication/Send File to Controller Manually ...* or click toolbar button *Send File to Controller Manually*, a dialog is displayed to let the user select the parameter file to be sent.
8. CNCcomp: Select the file and confirm with *Open* button, *Communication Parameters* dialog is displayed.
9. CNCcomp: Set the appropriate Baud rate and select the serial port (COM1, COM2 or other port), then press **OK**. CNCcomp now sends the file to ND720/760.
10. CNCcomp: After CNCcomp finishes sending the file, the message "*The file has been sent successfully*" is displayed.
11. ND 720/760: After the file transmission is completed, the display unit restarts automatically.
12. ND 720/760: Press - key to proceed with other data transfer tasks, or exit transfer function by press **CL** key.

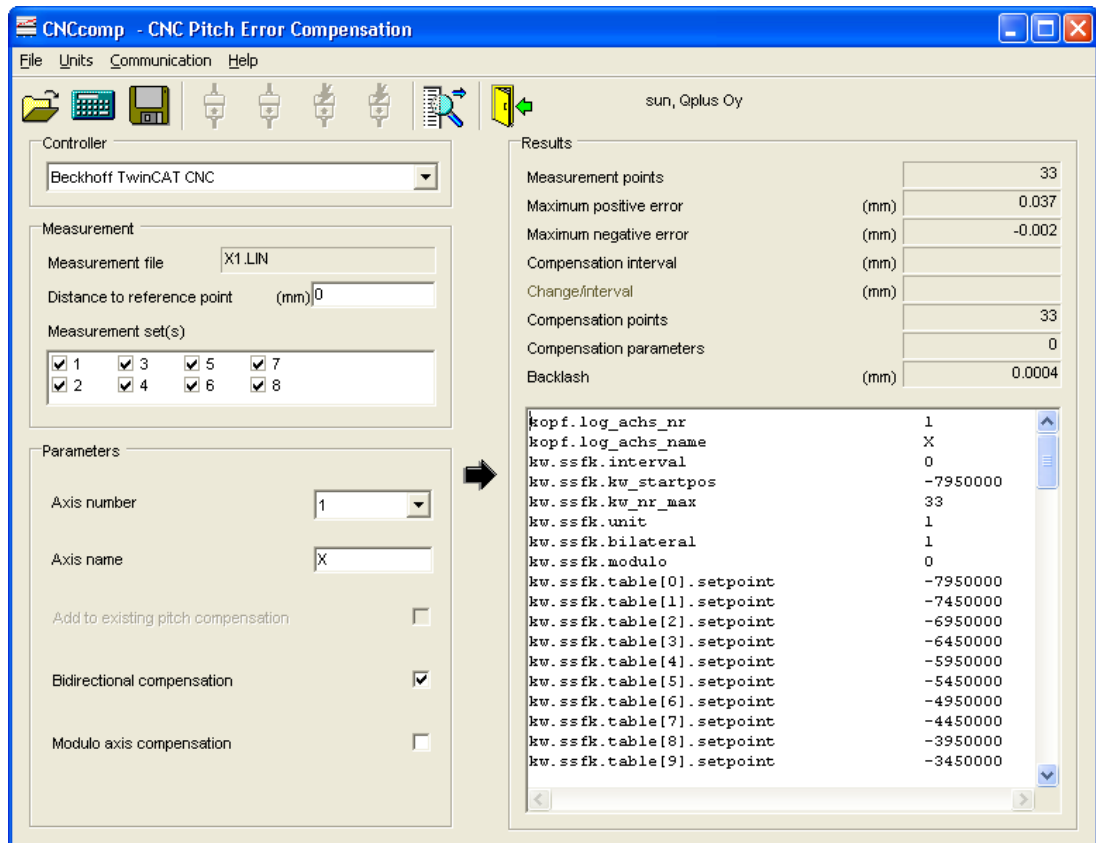
13.12 Automatically Send Parameter File to ND 7xx

Note: This function is only available with software version 246271-08 or higher. Please check the backside of the display units for its software version.

1. ND 720/760: Check that the display unit is switched on.
2. CNCcomp: Select menu *Communication/Send File to Controller Automatically ...* or click toolbar button *Send File to Controller Automatically*.
3. CNCcomp: A dialog is displayed which let the user select the parameter file to be sent. Select the file and confirm with *Open* button.
4. CNCcomp: If the transmission of parameter file is successful, the message "*Sending file completed*" is displayed. Otherwise, an error message is presented.

14 Beckhoff TwinCAT CNC

- *Measurement File Options*
- *Parameter Options*
- *Calculation*
- *Saving Compensation File*
- *Reading Compensation File*
- *Announcing the Compensation Table*



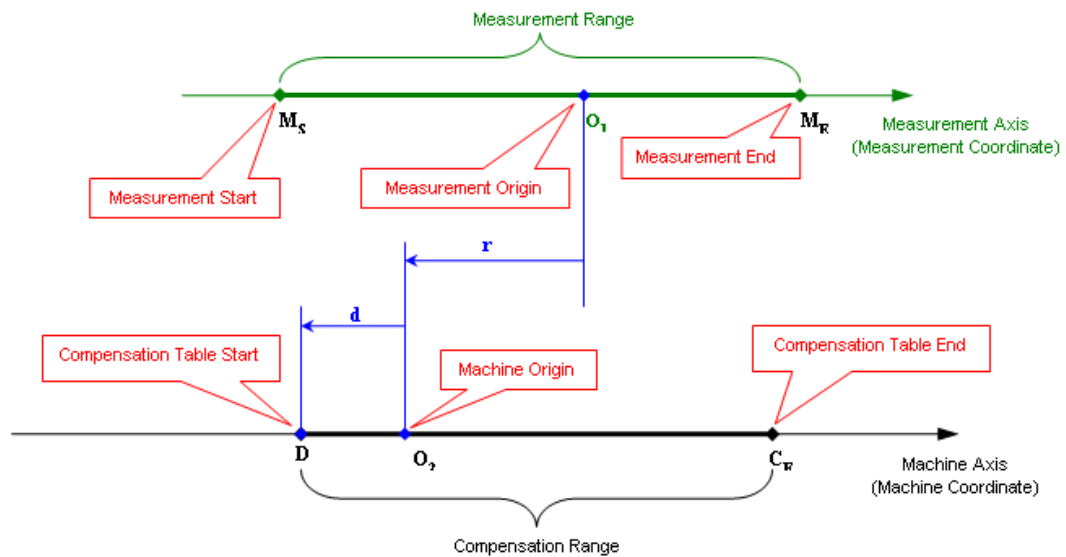
14.1 Measurement File Options

Distance to reference point

Distance to reference point controls the location of the measurement range in machine.

If position data in the measurement file are taken in machine coordinates, this value should be 0. If not, as shown in the figure below, this value is the distance from origin of the machine O_2 to the origin of measurement O_1 , which is value r . Please also notice that if the machine origin O_2 locates at the left of the measurement origin O_1 , then $r < 0$. If the machine origin O_2 locates at the right of the measurement origin O_1 , then $r > 0$.

By default, the measurement range is aligned with the machine so that the machine origin O_2 is overlapped with measurement origin O_1 . Please notice that users can have cases which are different from the default setting.



Measurement and Machine Coordinates

If measurement file includes several measurement runs (sets) they are all listed in *Measurement set(s)* list box. It is possible to select just one or several sets for the compensation. If more than one set is selected, the average values of all selected measurement sets are used for the compensation.

14.2 Parameter Options

Axis number

Axis number specifies which axis is to be compensated. The compensation file parameter options are changed accordingly. The program supports 16 axis for Beckhoff CNC.

Axis name

Axis name specified the name of the axis to be compensated. This is only used for diagnostic purpose. When the user change the axis number to 1, 2 or 3, the program will gives "X", "Y", "Z" as default axis name. User can input axis name manually.

Add to existing pitch compensation

If existing pitch compensation has been read, the new compensation values can be actualized cumulatively to old ones. If user wants to measure and compensate without clearing the old compensations, first carry out measurements with compensation set to active, then read the old values into CNCcomp, and make the calculation with this selection checked.

Important Note: The range of the new compensation table to be generated is decided by the current measurement file. Therefore, if part of the old compensation range is outside the current measurement range, it will be neglected in the new compensation

table, which means that the old compensation values outside the current measurement range are simply lost.

Bidirectional compensation

Bidirectional compensation decides whether bidirectional pitch error compensation is enabled. If bidirectional compensation is enabled, CNCcomp will generate compensation values separately for positive direction and negative direction.

If the measurement is one-directional, this function will be disabled.

Modulo compensation

Modulo compensation decides whether the compensation list will be generated for modulo compensation.

Modulo length

If modulo compensation is enabled, the user must give the modulo length here.

14.3 Reading Compensation File

The program asks for the compensation file name. After the user gives the old compensation file name, the program will read the old compensation list from the file and then inform the user whether the reading operation is successful. If the old compensation file has been read successfully, the program will make the selection of the parameter options according to the old parameter file.

Reading old compensation file makes it possible to measure and compensate machine tools without first clearing pitch compensations and thus speed up the whole measuring process.

14.4 Announcing the Compensation Table

According to the TwinCAT CNC manual, the following entries must be present in the startup description in order to announce the compensation table to the control system.

Variable Name	Type	Meaning
Zahl_kw	Unsigned short integer	Number of the compensation value lists
Achs_kw[i]	String	Name of the data file
Achs_kw_log_ax_nr[i]	Unsigned short integer	Logical axis number of the axis to which the compensation value list is to apply

Note: If the compensation value list is configured by the TwinCAT System Manager, these entries are made automatically in the start list.

Example:

Excerpt from start list *hochlauf.lis*:

```
...
zahl_kw          3
#
achs_kw_log_ax_nr[0]  1
achs_kw[0]        tc_...\achskw1.lis
#
achs_kw_log_ax_nr[1]  2
achs_kw[1]        tc_...\achskw2.lis
#
achs_kw_log_ax_nr[2]  6
achs_kw[2]        tc_...\achskw6.lis
...
```