DATACAL Software

User's Manual

Edition of 10 december 2014

We reserve the right to change all or part of the specifications set out in the present User's Manual without notice.

1. GENERAL

1.1 Overview of DATACAL

Datacal is a software for operation of AOIP field measurers/calibrators, pocket calibrators and pocket thermometers. It operates in the Windows® XP/Vista/Seven graphic operating environment.

It allows acquisition of measuring point results coming from one or more instruments, performance of calculations on the results provided by these instruments and the sending of set point values to output points.

It includes drivers for communication with the following AOIP instruments: Calys 75, Calys 100, Calys 150, Calys 60IS, Calys 80IS, Calys 120IS, TC6621, TC6622, TM6602, TM6612 and TM6630.

1.1.1 Main functions

The main functions of DATACAL are as follows:

. Configure the various instruments connected: measurement and simulation channels, measurement memory parameters, calibrated sensors, management of configurations stored in the devices.

. Read the measurement bursts from these instruments (no limit on the number of measuring points other than the system resources of the host computer), display them in graphic and numeric form, print them in the form of reports and export them to other software such as Microsoft Excel.

. Read the procedures for calibration of the instruments, create new ones, load procedures into the instrument.

. Read the calibration reports and print them in report form.

. Use the instruments connected in real time: display measurements, simulation status, any errors, run a simulation.

1.1.2 Visual appearance

DATACAL presents one type of view. This view is used to manage an instrument completely. Its left-hand panel displays, in its lower part, a menu containing the four major instrument management sections, namely Configuration, Measurement Bursts, Calibration and Virtual Instrument.
The upper part of the left-hand panel is a tree display of items related to the section: the various configurations, the list of local and instrument bursts, the list of calibration procedures and reports, etc.
The right-hand panel displays:
In **Configuration** mode, the various windows including Setup, instrument files and the list of predefined instrument configurations.
In **Measurement Bursts** mode, the list of local bursts and instrument bursts, details of a local burst in numeric form or a graphic view common to all the bursts.
In **Calibration** mode, the list of local and instrument calibration procedures and reports, and details of a procedure or a local report.
In **Virtual Instrument** mode, a partial replica of the screen and the instrument buttons.

2. SETUP

2.1 Required configuration

2.1.1 Hardware Configuration

A PC-compatible microcomputer provided with at least:
. Windows® XP SP3, Server 2003 SP2, Vista SP1, Seven.
. 50 MB of free disk space.
. Minimum display resolution 800x600 (1024x768 recommended).
. An available USB port.

If DATACAL has to communicate with several instruments, the latter can be connected to the various available USB ports.

2.1.2 Software

The USB driver for the instrument must be installed on the system; you can find it on the www.aoip.com website.
The DATACAL application needs an update of the Windows system named ".NET Framework 4.0". This update is automatically installed with DATACAL. If you have to install it manually, you can download it at this url :
http://www.microsoft.com/en-US/download/details.aspx?id=17718 or find it on the installation CD « dotNetFx40_Full_x86_x64.exe »

2.2 Connecting instruments

The instrument is connected to the PC via its USB interface. The USB driver loaded automatically when connecting the instrument emulates a serial link with applications wanting to use the instrument, and the ports for communication with the instruments therefore have the same name as serial ports (COMx).
Concerning Calys 75, Calys 100, Calys 150, TC6621, TC6622, TM6602, TM6612 and TM6630 instruments, communication speed is 115 kBauds.
Concerning Calys 60IS, Calys 80IS and Calys 120IS, it is recommended to limit the communication speed to 9600 Bauds.
Other serial link parameters, common to all the instruments, are: 8 data bits, 1 stop bit, no parity.

2.3 Installing DATACAL

Important notes for user accounts with restricted rights:
. Installation of DATACAL requires a Windows Administrator privilege.

The software is supplied on a CD-ROM.

For installation from the CD-ROM:
. Insert the CD-ROM in the drive.
. Click on DATACAL in the list of proposed software.
. Follow the instructions of the setup program.

The setup creates a group of programs. The document for this group entitled "Notes on DATACAL" may possibly contain additional information to that in this manual, and we invite you to consult it.

2.4 Updating DATACAL

The site **www.aoip.com**, under the **Software** heading, proposes downloading of the most recent version of the software. For installation, follow the instructions.

3. STARTING DATACAL AND LICENSE REQUEST

Open the menu **Start|Programs|AOIP|DATACAL|DATACAL**.

License request:
DATACAL is copy-protected by a user license system which allows software installation on only one computer. To install DATACAL on several computers, several licenses must be purchased.

On first install, DATACAL detects that no license has been registered and grants a trial license that is fully functional for a period of 30 days (from the date of first install).
The number of remaining days is displayed at the bottom of the **About DATACAL** dialog box, available from the **Help|About...** menu bar item.

While the current license is temporary, DATACAL invites you to either request a permanent license from AOIP (**Request permanent license** button), or to register this latter (**Register permanent license** button) if the license text has already been returned by AOIP following a previous request.
The **Next** button ignores the warning and opens a DATACAL session based on the fully functional temporary license.
The **No longer display this message** box instructs DATACAL to open a session directly, without displaying the warning.
In this case, the license registration request can be displayed by selecting **Help|About...** .

So, to save a permanent license :

. Choose the command **Help|About...** then click on the **Save a license...** button.

. A dialog box opens, prompting you to enter the text (key) license: enter the license text received from AOIP and confirm by clicking the **OK** button.

. A new dialog box will appear asking for permission to run the "LicTemp.exe" utility that will actually make the recording of DATACAL license key entered by the user. Click the **Yes** button to allow this operation, which shows a final dialog box indicating the correct result or not to register the DATACAL license. Click the **OK** button to complete the process, the license is registered and operational.

Window size:
When the application is opened, the DATACAL window occupies the whole screen.

It can be reduced so as to occupy only part of the screen by clicking on the **Restore**  box located in the top right-hand or by double-clicking on the title bar again.

It can again be made to occupy the whole screen by clicking on the **Maximize**  box in the title bar or by double-clicking on the title bar.

Finally, it can be reduced to icon form by clicking on the **Minimize**  box in the title bar.

At start-up, the application window is empty. All operations are based on an instrument operating file. It is possible either to create a **new operating file** or to **open an existing operating file**.

4. INSTRUMENT OPERATION

4.1 Creating an instrument operating file

To create a new instrument operating file, perform the following operations:

. Use the **File|New** menu command (or click on the button), then choose the instrument type to create a blank configuration corresponding to the instrument type being selected.

. Define the location (the storage directory) and the name of the *operating file*.

. Define its communication parameters.

When these various operations have been performed, you can connect to the instrument with the **Instrument Connection** button in the left-hand part of the window or the button in the toolbar.

4.2 Opening an existing operating file

To open an already existing operating file:

. Select the **File|Open** command (or click on the button).
. Type the name of the file to be opened, or browse through the directories to select it from the list of files.
. Press **OK**.

The name of the operating file is displayed in the title bar of the **DATACAL** window.

4.3 Closing an operating file

A **DATACAL** document is closed automatically when its last window is closed.
To close a document, select the **File|Close** command (or click on the button).

At document closing, DATACAL prompts you to save the changes made to the document before closing it. If a document is closed without being saved, all the changes made since the last backup will be lost.

4.4 Description of the operating window

This window displays all the instrument's configuration parameters and data.
It is divided into two window panes:
. On the left, a tree structure showing the groups of configurable items.
. On the right, the list and details of the items in the group selected in the tree.

The bar separating the two parts can be moved to enlarge one part or the other.

To move the bar:
- Point to it with the mouse;
- When the mouse pointer becomes a fragmenting indicator, drag the bar to the required position.

The left-hand section of the window contains a menu providing access to the main instrument operation sections. There are four such sections: **Configuration**, **Measurement Bursts**, **Calibration** and **Virtual Instrument**; depending on the type of instrument selected and/or type of DATACAL software license, some sections may not be present in the menu.

4.4.1 Configuration section

This section permits access to the communication parameters and the various channel configurations and instrument functionalities. The tree in the left-hand section of the window shows the following items:

4.4.1.1 Setup & Communication

The right-hand section of the window displays the instrument configuration and setup parameters:

Communication port
Indicate the communication port of the computer to which the instrument is connected (see Chapter 2.2).

Identification of the instrument
Indicate whether you want the DATACAL application to check the instrument type at connection. If this option is activated, the instrument's ID is read and compared with the instrument type defined in the instrument operating file; in the event of inconsistency, an error is reported.

Transmission option
Transmission of the input and output channel configuration can be relatively long on some instruments. This option allows you to specify the parameters that are sent during programming: either the application sends all the parameters of a channel irrespective of the measuring or simulation mode chosen, or it sends only the parameters related to the chosen mode. This second operating mode allows faster programming.

Date and Time
The two data entry fields allow you to choose a date and time.
The **Use current time** checkbox allows you to enter the current date and time automatically in the two fields.
The **Set date and time** checkbox indicates whether the instrument's date and time must be set when sending parameters to the instrument.

The **Read Instrument** and **Send to Instrument** buttons allow the above parameters to be reread and sent to the instrument respectively.

4.4.1.2 Configuration files

The right-hand window pane displays the file management interface for the instrument. An instrument can save its current configuration in memory in file form, and it can initialize its current configuration from a file stored in its memory.

The **Open File** button actuates rereading of the configuration stored in the file whose number is entered in the **File to open** area.

The **Save File** button actuates storage of the instrument's current configuration in the file whose number is entered in the **File to save** area.

4.4.1.3 Configurations

This item covers all the predefined instrument configurations saved in the operating file. The operating file can contain several predefined instrument configurations; a predefined configuration can be a configuration reread in the instrument or a new configuration created in Datacal. A predefined configuration can be transmitted to the instrument to become the instrument's current configuration.

A configuration is defined by a name and contains, depending on the instrument type, the parameters setup for input channels, output channels, measurement bursts and calibrated sensors. These parameters are grouped in a tree structure of lower level than the related configuration.

Input channels

The right-hand section of the window displays the parameters of the input channel(s).

In the **Function** area, it is possible, via the **Measuring Function** drop-down list, to select the type of measurement for the instrument channel. Depending on the type of measurement, various options to be selected are displayed below the choice of measurement function: rating, type of probe, unit, type of input, metering time, etc.

The **Scaling** area allows measurement scaling in the instrument to be activated or not via the **Start-up** checkbox.
The **Format** field allows precise selection of the measurement display after scaling.
The **Unit** field can be used to enter the physical unit of measurements after scaling.
The **Measure/Display** table can be used to enter the scaling points. The points are entered directly in the table boxes.

The **Tar** area allows the instrument's Tar function to be activated or not via the **Start-up** checkbox.
The **Tare Value** field can be used to enter the value that will be deducted from the measurement if the Tar function is enabled.

The **Read Instrument** and **Send to Instrument** buttons allow the above input channel parameters to be reread and sent to the instrument respectively.

Output Channels

The right-hand section of the window displays the parameters of the output channel(s).

In the **Function** area, it is possible, via the **Emission Function** drop-down list, to select the type of sending for the instrument channel. Depending on the type of sending, various options to be selected are displayed below the choice of measurement function: rating, type of probe, unit, type of input, metering time, etc.

The **Scaling** area allows measurement scaling in the instrument to be activated or not via the **Start-up** checkbox.
The **Format** field allows precise selection of the measurement display after scaling.
The **Unit** field can be used to enter the physical unit of measurements after scaling.
The **Measure/Display** table can be used to enter the scaling points. The points are entered directly in the table boxes.

In the **Mode** area, it is possible, via the **Emission Mode** drop-down list, to select the sending mode for the output channel. Depending on the sending mode, various options to be selected are displayed below the choice of mode: time, delays, amplitudes, number of repetitions, etc.

The **Channel 2 Mode** area makes it possible, for certain instruments, to choose whether channel 2 acts as an input channel or an output channel.

The **Read Instrument** and **Send to Instrument** buttons allow the above output channel parameters to be reread and sent to the instrument respectively.

Calibrated sensors

The right-hand section of the window displays the list and parameters of the calibrated sensors.

The table presents the list of calibrated sensors, indicating their name, type and calibration date. By selecting a table row it is possible to edit a sensor in the right-hand fields of the table.

The **Designation** field shows the name of the calibrated sensor.

The **Calibration Date** field shows the date of the last sensor calibration.

The two **Sensor Type** fields indicate the sensor family (thermocouple or platinum probe) and the type of probe (Thermocouple K, Pt100, etc.).

The **Input Type** field allows you to choose the unit for entry of real values in the table of sensor calibration points (temperature or voltage).

The **Real value / Measured value** table indicates the sensor calibration points. The real values are represented as a temperature or voltage, while the measured values are always represented as a temperature.

The **Read Instrument** and **Send to Instrument** buttons allow the above calibrated sensor parameters to be reread and sent to the instrument respectively.

Measurement bursts

The right-hand section of the window displays the list and parameters of measurement bursts. For some instruments, parameters setup can be performed on several channels when several input channels are available.

The **Total block size** field allows you to choose the maximum number of measurements in a burst.

The **Sampling period** field is the time interval between two measurements in seconds.

The **Trigger type** field allows you to choose the starting condition for recording a burst: none (immediate start), manual, upon detecting a high or low threshold.

The **Trigger level** field shows the detection threshold for the trigger.

The **Post-trigger** field allows you to choose the number of measurements to be recorded in the burst after the appearance of the trigger.

The **Read Instrument** and **Send to Instrument** buttons allow the above measurement burst parameters to be reread and sent to the instrument respectively.

Linearization
The right part of the window displays the different linearizations divided into two parts. A part for local linearization functions (**Local Functions** area) and a part for the linearization functions of the instrument **(Instrument Functions** area).

● Linearization Local Functions (**Local Functions** area):

Table **Name / Date** lists the various functions of local linearization by their name and date of creation.

The **Function Name** field allows you to specify the function name of the function local linearization.

The **Input Measure** field allows you to select the unit of the input value to linearize (voltage, current, frequency, pressure, pulse, resistance or temperature).

The **Input Range** field allows you to select the size of the input value to linearize the next unit of this value. (Ex: 200mV gauge for measuring voltage).

The **Number of decimal** places used to select the display format of measures of linearity (without decimals with 1,2,3,4 or 5 decimal places).

The **Output Unit Measure** field allows selecting the unit of the measured value for linearization.

The **Standard** field allows selecting the standard temperature used for linearization.

The **Csf** field allows you to select the type of information the coefficient of junction (This field takes values **Value** (to enter manually) or **Internal** (Csf generated by the instrument).

RTD Mode checkbox allows you to enter if RTD linearization is performed via a linearization table or via a linearization polynomial coefficients for RTD type. If the box is checked, the linearization is generated by the polynomial coefficients.
In this case the field **R0** represents the value of the polynomial at 0 ° C or 0 ° F or 0 Kelvin).
The field **A** is the polynomial coefficient expressed in $°C^{-1}$ or $°F^{-1}$ or K^{-1}.
The field **B** represents the polynomial coefficient expressed in $°C^{-2}$ or $°F^{-2}$ or K^{-2}.
The field **C** is the polynomial coefficient expressed in $°C^{-3}$ or $°F^{-3}$ or K^{-3}.
If the checkbox is unchecked, the linearization is generated via a linearization table.
In this case, the **Point / Input value / Measured Value** table is used to enter the different linearization points with the input value to be linearized and the measured value which unit value is the **Output Unit Measure** field. The measured value is expressed in temperature units (Celsius or Fahrenheit Kelvin) in the case of the use of polynomial RTD.

The **Delete** button, relating to the local linearization functions, allows erasing previously selected local procedures in the **Name / Date** table.

The **Send to Instrument** button allows the instrument to send local procedures previously selected in the **Name / Date** table to the instrument.

The **Add to Local Functions** button allows the file creation of the linearization function which name is specified in the **Function Name** field and with the parameters contained in the various fields listed above. Pressing this button will generate a file of local linearization and the function will appear in the **Name / Date** table.

The **Apply Changes** button allows you to modify an existing local linearization function. Pressing this button will apply the values of the fields in the **Local Functions** area to the procedure selected in the **Name / Date** table.

● Linearization functions of the instrument (**Instrument Functions** area):

The various fields in this area have the same functionality as for the **Local Functions** area, except for

the **Instrument Functions** area fields which are read-only fields (you cannot change them manually writing). Selecting a linearization function of the instrument in the table **Name / Size / Date** displays characteristics of this function in the various fields related to the **Instrument Functions** area.

The **Delete** button allows the deletion of the procedures of the instrument previously selected in the **Name / Size / Date** table.

The **Read from Instrument** button allows sending or previously selected instrument procedures to the **Name / Size / Date** table locally.

Read List button enables reading list linearities in the instrument which are then listed as a list in the **Name / Size / Date** table.

A right-click on the **Configurations** item in the tree structure displays a menu allowing a new configuration to be created.
A right-click on a configuration name in the tree structure displays a menu allowing the current configuration to be deleted or duplicated.

4.4.2 Measurement Bursts section

This section provides access to the instrument's measurement burst data. A measurement burst is acquired by the instrument independently and is then recorded in its internal memory; several bursts can be recorded in the instrument.

The bursts can then be transferred from the instrument to the Datacal application to be displayed in numeric and graphic form. A burst read from the instrument is stored in a file on the hard disk of the computer, and a reference to this burst is inserted in the operating file to be able to access it.

The tree in the left-hand section of the window shows the following items:

4.4.2.1 Local bursts

The right-hand section of the window displays the list of measurement bursts stored locally on the computer. The local bursts are either downloaded from an instrument or calculated (see below). Each line of the list describes a burst by its name, its number of points, its date of saving in the instrument, its measurement function and its unit.
A checkbox at the start of each line allows you to select one or more bursts from the list.

The **Export selected bursts** button allows you to save the data of the selected bursts in files in text format readable by other software. A press on this button brings up a dialog box for choosing locations and filenames to which the bursts will be exported. You must choose as many files as the number of bursts selected. The filename proposed for a burst by default consists of its name and the ".csv" extension.

The **Delete selected bursts** button allows you to erase the selected bursts. When erasing them, only the references to the bursts in the current operating file are erased; the files containing the burst data are kept on the disk.

The **Local bursts** tree node can be opened by clicking on it; a sub-tree then displays the local bursts by their name.
Selection of a local burst in the tree structure displays the burst data in the right-hand section:
- The first text box contains information concerning the burst: Name, number of measurements, date of first measurement, date of last measurement, measurement function, unit, type of recording, scaling active or not, Tar active or not, channel number, name of the file for storage of the burst on the computer's local disk.
- After clicking on the **Display list of points** link, the second text box displays the measuring points. Each point is represented by a time lapse relative to the first point of the burst in seconds, and a measurement value.

A right-click on a local burst name in the tree structure displays a menu allowing the burst to be deleted or displayed on the graphic view.

4.4.2.2 Instrument bursts

The right-hand section of the window displays the list of measurement bursts stored in the instrument. Each line of the list describes a burst by its name, its number of points, its date of saving in the instrument, its measurement function and its unit.
A checkbox at the start of each line allows you to select one or more bursts from the list.
After loading an operating file, the list of instrument bursts is empty; it is not stored in the file. The instrument must be interrogated to view this list.

The **Update list** button allows you to read the list of bursts stored in the instrument.

The **Read selected bursts** button allows you to transfer the selected bursts from the instrument to the operating file. Following this transfer, the bursts read are added to the **Local bursts** part of the tree structure.

The **Delete selected bursts** button allows you to erase the selected bursts in the instrument. Once they have been erased, the selected bursts will no longer be able to be transferred from the instrument.

The **Instrument bursts** tree node can be opened by clicking on it; a sub-tree then displays the bursts stored in the instrument by their name.
Selection of an instrument burst in the tree structure displays information concerning the burst in the right-hand section: Name, number of measurements, date of first measurement, date of last measurement, measurement function, unit, type of recording, scaling active or not, Tar active or not, channel number, name of the file for storage of the burst on the computer's local disk.
The measuring points for the burst are not available because, at this stage, they have not been read from the instrument.

4.4.2.3 Calculated burst

A calculated burst does not come from an instrument; it is a burst which points are obtained from a calculation integrating one or many bursts, statistics or logical functions, etc…

To create a new calculation burst, click on the right button on the note **Local bursts** in the tree and select the command **New calculation burst...** The configuration window of a calculation burst will appear which the following choices:

The fields **Name** and **Unit** which allow to key respectively the name of the new burst and a name for the unit. The name must be unique and not correspond to an existing burst name within the local bursts list.

The button **Define calculation...** allows defining the calculation to be done in order to determine the values of the burst points, it displays the screen of the calculation definition. The specified calculation is then done for every burst point taking into account the current time and date as shown below.

The zone N**umber of points to calculate** allows to define the number of measuring points required for the new burst, it can be keyed (**Custom number of points** choice**)** or equal to the number of points of an existing burst (**Burst number of points** choice**)**.

The zone **calculation period for burst points** allows to define intervals between burst points, the interval between two points can be keyed (**Specific period** choice) or the intervals can be equal to those of an existing burst (**Burst period** choice). In the first case, the zone **Delay before first point** enables to key a period of time which will be integrated before the calculation of the 1st burst point, this time will be the timestamp value of the first point.

The button **Burst generation** allows to close the configuration screen and to do the new burst calculation. Once the calculation done, the burst is added to the local bursts list and its contain is displayed on the right part of the instrument operating file window.

The window **Calculation definition** has one main area called **Calculation to be performed** which allows to key the calculation formula.

The **Bursts, Operations** and **Functions** areas contain lists of different operators and functions which can be integrated in the calculation formula. If you select in those lists, a "help text" will be displayed in the lower part of the window. If you double click in a list, or if you use the button **Insert**, the operator or the selected function is inserted in the calculation formula at the current cursor position.

The button **OK** enables to check the calculation formula syntax and to close the window.

The formula specifying the calculation must be keyed with an algebraic expression constituted by operands separated by operators. Parentheses may be used to modify operations evaluation chronology.

The operands can be constants, variables referring to other bursts results or functions. Operators are common algebraic operators.

Example calculation formulae

B(*BurstName1*)- **B**(*BurstName2*)	Calculation of a deviation.
AVE(**B**(*Name1*); **B**(*Name2*); **B**(*Name3*); **B**(*Name4*))	Calculation of an average.
B(*BurstName1*)>80 **OR B**(*BurstName1*)>90	Logical formula.

List of operands

Numerical constants	E.g.: 1.234 **Remark**: The decimal separator is the character defined in the Windows control panel, in the *International - Number formats* section. In principle, this is the dot in English.
X	X is the burst's previous value (0 for the first burst point).
B(<BurstName>)	The value of the burst named <BurstName>.
T	The time elapsed since the start of calculation.
Functions	See the *List of functions* section.
Expression between parentheses	An operand can be comprised of an expression between parentheses. The parentheses are used, in particular, to alter the operation assessment order. See the *Operator precedence* section.

List of operators

+, -, *, /	Addition, subtraction, multiplication and division.
^ or **	Raising to the power of Warning for this operator: it applies only to positive values (as a^x is calculated as e^(x*Log)). (-2)^2 shall therefore raise an error. For integer power, preference should be given to a series of multiplications. (write X*X*X instead of X^3).
MOD	Modulo (remainder of an integer division) E.g.: 13 MOD 3 = 1
DIV	Quotient of the integer division. Exemple :13 DIV 3 = 4
<, <=, =, >=, >, <>	Comparison operators giving a result of 1 or 0.
OR, AND	Logical operators, see example in the *Calculation definition* section.
NOT (<expr>)	Negation of logical expression NOT(1) = 0 ; NOT(0) = 1

Operators precedence

Expressions are assessed from left to right, while respecting usual operator precedence (operations with the highest priority are performed first).

(highest priority operators)
- (unitary minus) and **NOT**
^
*, **/**
+ , **-**
< , <= , = , >= , > , <>
OR , **AND**
(lowest priority operators)

Parentheses are used to alter operator priority.

List of functions

Mathematical functions

ABS <operand>	Returns the absolute value of the operand. ABS B(*BurstName1*) returns the value of burst *BurstName1* if positive, otherwise is returns the opposite of its value. ABS(B(*BurstName1*)* 10.3+B(*BurstName2*)) assesses the expression B(*BurstName1*)*10.3+B(*BurstName2*)) and returns the absolute value of the result.
LOG < operand >	Returns the common logarithm of the operand. LOG 100 returns 2. LOG(B(*BurstName1*)* 10.3+B(*BurstName2*)) assesses the expression B(*BurstName1*)*10.3+B(*BurstName2*) and returns the common logarithm of the result. The operand must be positive.
LN < operand >	Returns the natural logarithm of the operand. (The operand must be positive).
EXP < operand >	Returns the exponential of the operand.
SQR < operand >	Returns the square root of the operand. SQR 4 returns 2. SQR(B(*Name1*)* 10.3+B(*Name2*)) assesses the expression B(*Name1*)*10.3+B(*Name2*) and returns the square root of the result. The operand must be positive or null.
COS < operand >	Returns the cosine of the operand, expressed in radians. COS(1.5708) returns 0 COS(3.1416) returns -1.
SIN < operand >	Returns the sine of the operand, expressed in radians. SIN(1.5708) returns 1 SIN(3.1416) returns 0.
COSH < operand >	Returns the hyperbolic cosine of the operand, expressed in radians.
SINH < operand >	Returns the hyperbolic sine of the operand, expressed in radians.
PI	The number Pi (= 3,1415…)

Rounding functions

FLOOR < operand >	Returns the integer value immediately inferior to the operand. FLOOR(2.9) returns 2 FLOOR(-2.9) returns –3
CEIL < operand >	Returns the integer value immediately greater than the operand. CEIL(2.9) returns 3 CEIL(-2.9) returns –2
ROUND <oprand>	Rounds the operand's value to the nearest integer. ROUND(2.4) returns 2 ROUND(2.6) returns 3

Statistical functions

MAX (<expr 1>;…;<expr n>)	Assesses the expressions and returns the maximum. MAX(1;2;3) returns 3. MAX(B(*Name1*);B(*Name2*);B(*Name3*);B(*Name4*)) returns the maximum value of the four bursts *Name1* to *Name4* at the current timestamp. MAX(0;B(*Name1*)-B(*Name2*)) returns the value of bursts *Name1*-*Name2* if positive, otherwise returns 0.
MIN (<expr 1> ;…;<expr n>)	Assesses the expressions and returns the minimum (see MAX function).
AVE (<expr 1>;…;<expr n>)	Assesses the expressions and returns the average. AVE(B(*Name1*);B(*Name2*);B(*Name3*);B(*Name4*)) returns the average of the four bursts *Name1* to *Name4* at the current timestamp.

History-based statistical functions

MAXB (<B(BurstName>;<HH:MM:SS>)	Returns the maximum value of burst <BurstName> for the indicated period. MAXB(B(Temperature); 12:00:00) returns the maximum temperature for the past 12 hours.
MINB (<B(BurstName>;<HH:MM:SS>)	Returns the minimum value of burst <BurstName> for the indicated period. MINB(B(Temperature); 12:00:00) returns the minimum temperature for the past 12 hours.
AVEB (<B(BurstName>;<HH:MM:SS>)	Returns the average value of burst <BurstName> for the indicated period. AVEB(B(Temperature); 12:00:00) returns the average temperature for the past 12 hours.
DEVB (<B(BurstName>;<HH:MM:SS>)	Returns the standard deviation value of burst <BurstName> for the indicated period. DEVB(B(Temperature); 12:00:00) returns the standard deviation temperature for the past 12 hours.
HISTB (<B(BurstName>;<HH:MM:SS>)	Returns the value of burst <BurstName> at the start of the indicated period. HISTB(B(Temperature); 12:00:00) returns the value of the temperature 12 hours previously.

Miscellaneous functions

CT	Returns the hour of the current day in the form of a decimal number between 0 and 24. In other words, 0 <= CT < 24. For example, at half past noon, the CT function returns a value of 12.5.

HOUR (<HH:MM:SS>)	Conversion to decimal of a time expressed in HH:MM:SS format. E.g.: HOUR(12:30:00) returns a value of 12.5. The following condition: CT > HOUR(08:30:00) AND CT < HOUR (18:30:00) is true between 8:30am and 6:30pm (it can also be written as follows: CT > 8.5 AND CT < 18.5).
DW	Returns the current day of the week, between 1 (Sunday) and 7 (Saturday). For example, the DW function will return a value of 4 on Wednesday.
DM	Returns the current day of the month, between 1 and 31.
MONTH	Returns the current month number, between 1 and 12
YEAR	Returns the current year (2000 to 2100). E.g.: 2014

Linearization and correction functions

CORRSEG (<expr>;<FileName>;<RowName>)	Segment-based correction <expr>: X or B (<BurstName>) <FileName>: File containing the correction data (see format below) <RowName>: Name of the file row to use.
CORRPOL (<expr>;<FileName>;<RowName>)	Polynomial correction <expr>: X or B (<BurstName >) <FileName>: File containing the correction data (see format below) <RowName>: Name of the file row to use.

Correction data file format

This is generally a file containing points generated by a calibration program giving, for each calibrated instrument and each measurement point, the correspondence between the read value and the actual value.

The expected format is as follows:
. 1 row per calibrated instruments,
. Row format

```
"InstrumentName\tNameChannel\tActualValue1;ReadValue1\tActualValue2;ReadValue2\t
...
  ...\tActualValueN;ReadValueN\r\n"
```

`\t`: "tab" character
`\r`: "carriage return" character
`\n`: "new line" character

Thus, to define a correction using such a file, you must enter the name of the file and locate the row of points by stating the calibrated instrument name: <InstrumentName>.

4.4.2.4 Graphic view

The right-hand section of the window displays a graph on which the local measurement bursts can be shown graphically.

The burst measuring points are displayed in accordance with the time-and-date stamping of each point; the scale of ordinates is automatic and common to the various bursts present in the graph. A legend below the graph indicates, for each burst, its name, its unit and the colour for its plotting.

To view the measuring points for a burst on the graph, there are two methods:

- Click with the right mouse button on a local burst in the tree structure and choose **Display in graphic view** in the context-sensitive menu.
- Select a local burst in the tree structure and do a drag-and-drop directly on the graph or on the **Graphic view** node of the tree structure.

To erase the plotting of a burst on the graph, you must do a right-click on the burst in the legend and select **Delete plotting** in the context-sensitive menu.

The **Cursor** button allows a cursor to be shown on the graph. The cursor can be moved freely on the graph; it indicates the x-axis and y-axis coordinates pointed to.

4.4.3 Calibration section

This section provides access to the instrument calibration procedures and reports. The calibration process starts with the definition of a procedure. This procedure describes the calibration method, the channels used and their measurement or simulation functions, the measuring points, sequencing mode and conditions of acceptance. The procedure can be defined using the Datacal application or directly on the instrument. One can then start running the procedure in the instrument, which then performs measurements in accordance with the procedure. At the end of the procedure, the instrument generates a report containing the main calibration information and measuring points. This report is linked to the procedure to be able to retrieve the conditions in which a calibration was performed, and as a result a procedure containing at least one report can no longer be modified.

Several procedures can be saved in the instrument. Several reports can be linked to a procedure and stored in the instrument.

The calibration procedures can be transferred from the instrument to the Datacal application and vice versa. The reports can be transferred from the instrument to the application.

The tree in the left-hand section of the window shows the following items:

Local procedures

The right-hand section of the window displays the list of calibration procedures and reports saved in the instrument operating file. Each line of the list describes either a procedure, indicating the name and manufacturer of the instrument, or a report, indicating the serial number of the instrument and the calibration date.
The list is organized in a tree structure. A procedure is listed in bold letters. Below can be found the reports linked to that procedure, listed in standard text type and with a start of line offset relative to the procedure.

A checkbox at the start of each line allows you to select one or more procedures and/or reports from the list.

The **Export selected reports** button allows you to save the data of the selected reports in files in text format readable by other software. A press on this button brings up a dialog box for choosing locations and filenames to which the reports will be exported. You must choose as many files as the number of reports selected. The filename proposed for a report by default consists of the name of the procedure followed by the serial number of the instrument, followed by the calibration date, and the ".txt" extension.

The **Send Selected Procedures to Instrument** button allows you to send the selected procedures to the instrument.

The **Delete selected procedures and reports** button allows the selected reports and procedures to be erased from the current operating file.

The **Local procedures** tree node can be opened by clicking on it; a sub-tree then displays the local procedures by their name and the local reports by their instrument serial number and calibration performance date.
Selection of a local procedure in the tree structure displays the characteristics of the procedure in the right-hand section (see Chapter 8).

Selection of a local report in the tree structure displays the report data in the right-hand section:
- The first text box contains information concerning the report: Name, serial number and manufacturer of the calibrated instrument; model, serial number and dates of calibration of the instrument; serial number of the pressure sensor; name of the operator; comments; stage and date of calibration; verdict.
- The second text box displays the calibration measuring points. Each point is represented by a set point value, an instrument measurement, possibly a reference measurement and a deviation. The deviation is calculated, depending on the type of calibration, between the set point value and the instrument measurement or between the set point value and the reference measurement.
- The graph contains 3 plots: 2 discontinuous line curves which represent the limits of acceptance as defined in the procedure, and a curve of the deviation between the set point value and the measurement. Depending on the type of calibration, the x-axis represents the instrument measurement or the reference measurement.

A right-click on a local procedure name in the tree structure displays a menu allowing a new procedure to be created, or the current procedure to be duplicated or deleted.

Instrument procedures

The right-hand section of the window displays the list of calibration procedures and reports stored in the instrument. Each line of the list describes either a procedure, indicating its number in the instrument and the name and manufacturer of the instrument, or a report, indicating its number in the instrument, the serial number of the instrument and the calibration date.

The list is organized in a tree structure. A procedure is listed in bold letters. Below can be found the reports linked to that procedure, listed in standard text type and with a start of line offset relative to the procedure.

A checkbox at the start of each line allows you to select one or more procedures and/or reports from the list.

After loading an operating file, the list of instrument procedures and reports is empty; it is not stored in the operating file. The instrument must be interrogated to view this list.

The **Update list** button allows you to read the list of procedures and reports stored in the instrument.

The **Read selected procedures and reports** button allows you to transfer the selected items from the instrument to the operating file. Following this transfer, the procedures and reports read are added to the **Local procedures** part of the tree structure. It should be noted that the report can be read from the instrument to the Datacal application only if the procedure on which it depends is already in the list of local procedures. If this is not the case, the procedure must be read at the same time as the report.

The **Delete selected procedures and reports** button allows the selected procedures and reports to be erased from the instrument. Once they have been erased, the selected items will no longer be able to be transferred from the instrument.

The **Instrument procedures** tree node can be opened by clicking on it; a sub-tree then displays the instrument procedures by their name and the associated reports by their instrument serial number and calibration performance date.

4.4.4 Virtual Instrument section

This section provides access to certain commands and the instrument status.

The right-hand section of the window displays a screen copy of the instrument. After loading an operating file, the virtual screen of the instrument is empty; channel polling must be activated, which will poll the instrument at regular intervals and display its status on screen. The various buttons around the screen are used to adjust the display and the sending part of the instrument.

The **Channel Scanning** button allows you to start or stop polling of the instrument. When it is active, the information displayed on the screen copy of the instrument is updated regularly.

The **Measurement** area contains the **Pause** and **Read** buttons which allow the measurement cycles on the input channel(s) to be suspended and the cycles to be resumed respectively.

The **Emission** area contains the following:

- The **Emission Mode** drop-down list allows the sending mode to be selected for the output channel.
- The **Value** data entry field allows entry of the value to be sent in the case of a manual sending mode.
- The **Ascending** and **Descending** buttons allow simulation to be started in the case of the Step, Ramp, Cyclic Ramp and Synthesizer modes.
- The **Pause**, **Resume** and **Stop** buttons allow simulation to be suspended, resumed or stopped in the case of the Step, Ramp, Cyclic Ramp and Synthesizer modes.
- The **Previous point** and **Next point** buttons allow single-step operation when simulation is suspended in the case of the Step and Synthesizer modes.
- The **Repetitions** data entry field allows you to seize the number of times the waveform will be sent in the case of a Cyclic Ramp or Synthesizer sending mode.
- The **First Point** and **Last Point** data entry fields allow you to define the range of values to be sent in the case of a Synthesizer sending mode.

4.5 Printing reports

Printing a report involves printing measurement results with a particular formatting, i.e., in general, a page header containing a title, a date and a logo, a text box containing information on the measuring conditions and instruments, a table containing the measurements and any calculation results (deviations), a graph showing a curve of measurements versus time or the various measurements relative to one another.

There are two types of reports that can be printed in Datacal, a report on a measurement burst and a calibration report.

To print a report on a measurement burst, it must first be selected in the **Measurement Bursts** section in the **Local bursts** tree structure.

To print a calibration report, it must first be selected in the **Calibration** section in the **Local procedures** tree structure (that is, click the **Calibration** button, then select the desired calibration report in the **Local procedures** tree structure).

The **File|Print** command can then be used to print directly, or **File|Print Preview** to preview the report.

The **File|Print** option displays a window for choice of the printer and printing options. Printing is started via the OK button in this window.

The **File|Print Preview** option allows the appearance of the report to be viewed before printing properly speaking. The preview window is displayed, containing an overview of the report in "Plan" mode, i.e. without page breaks.
This window contains a toolbar whose elements provide the following functions:

Navigation in the various pages of the report when displayed in Page Mode.


Stopping and restarting report generation.


Report printing.


Overview of the report in "Page" mode, i.e. similar to its printed appearance.


Page setting for the report.


Export of the report to a file in “Portable Document Format” (pdf) or Microsoft Excel format.

Zoom on the report.

Text search in the report in Plan mode.

5. OPTIONS

The **Tools|Options** command allows certain performance options to be selected for the Datacal application.

It displays the **Options** dialog box which contains two tabs concerning the application language and printing.

5.1 Language

The **Language** tab allows you to choose the display language for the various application windows.
The **Language most similar to the system** checkbox, when activated, allows the operating system to choose from the application languages that which is most appropriate by comparison with one's own language. The default language is English.
If this box is not activated, the **Force the language** drop-down list allows a language to be chosen from those proposed.

5.2 Printing

The **Page Header Logo|File** entry field contains the path to an image file that will be inserted in the header of the printed reports. If this file is not specified, a default logo is printed.
The **Browse...** button allows a file to be selected by navigating in the tree structure of the computer's disk.

6. TOOLBAR

6.1 Toolbar commands

The toolbar is displayed at the top of the application window, just below the menu bar. It allows rapid access to certain DATACAL functionalities, using the mouse.

Click on	In order to
	Create a new document.
	Open an existing document. DATACAL displays the **Open** dialog box, in which a file can be found and opened.
	Save active document under its current name.
	Close active document.
	Delete the selected data from the document and place them in the Clipboard.
	Copy selection to Clipboard.
	Insert the content of the Clipboard at the insertion point.
	Print active document.
	Print Preview of active document.
	Opens communication with the instrument.
	Closes communication with the instrument.
	Access to the DATACAL online help facility.

7. STATUS BAR

The status bar is displayed at the bottom of the DATACAL application window.

The left-hand part of the status bar shows the ID and serial number of the connected instrument.

The right-hand part of the status bar shows:
- which of the following keys is locked:

Indicator	Description
CAP	The Capital Lock key is locked.
NUM	The Number Lock key is locked.

8. APPENDIX A: CALIBRATION PROCEDURE

8.1 Principle

Calibration involves determining, in specified conditions, the relationship between the values of a quantity indicated by a measuring instrument and the corresponding values of the quantity indicated by a standard. Calibration allows characterization of the response of an instrument, i.e. its measurement deviation relative to a reference measurement.

8.2 Calibration procedure

A calibration procedure defines the actions to be performed to calibrate an instrument: calibration method (by comparison, standard generator), allocation of instrument channels, measurement and simulation functions, calibration point numbers and values, automatic or manual measurement sequencing, measurement stability detection criterion, determination of a verdict.

8.2.1 Transmitter (or indicator) calibration function

Transmitters other than pressure transmitters

The instrument's output channel is used as 'Standard' generator and connected to the transmitter's measurement input.
The 4-20mA or 0-20mA output of the transmitter is measured on the instrument's input channel.

Pressure transmitters

The instrument reads the pressure by means of a KELLER sensor considered as reference sensor, and displays it in the instrument's output channel window.
The 4-20mA or 0-20mA output of the transmitter is measured on the instrument's input channel.
What is involved in this case, therefore, is calibration by comparison with a reference chain.

8.2.1.1 Parameters setup for the procedure in the instrument

The terms used refer to the transmitter:
'Input' to refer to the transmitter input, hence the instrument output channel.
'Output' for the transmitter's current output, measured by the instrument input channel.

8.2.1.2 Parameters setup for the procedure in DATACAL

NB: The terms used do not have the same meaning as in the instrument; they refer to the instrument itself:
'Input' to refer to the instrument input, hence the transmitter output channel.
'Output' for the instrument output, hence the transmitter input channel.

The **DUT Name** field allows you to enter the name of the device to be calibrated; this name is also the name of the procedure.

The **Manufacturer** field allows you to enter the manufacturer of the device to be calibrated.

The **Set Point** table allows you to enter the calibration points.

The **Min. Stability Time** field allows you to define the waiting time between sending a calibration point value and the corresponding measurement.

The **Generation Mode** field allows you to choose the type of measurement sequencing during execution of the procedure. In **Manual** mode, there is a pause between each calibration point, and on the instrument the user must confirm the transition to the following point; he may possibly change the values of the points. In **Down** and **Down/Up** mode, the calibration points are sequenced automatically, complying with the minimum stability time lag between each point; **Down** mode goes through the points of the table in the descending direction, while Down/Up mode goes through the table in one direction and then the other.

The **Acceptable deviation** field allows you to define the acceptance condition (OK or NOT OK) on the difference between the set point value and the measurement in the form of a relative deviation and an absolute deviation.

The **Channel 1 Measure** area allows you to select the mode for measurement of the signal coming from the transmitter and the measurement scaling.

The **Channel 2 Generator** area allows you to select the mode for measurement of the signal coming from the transmitter and the measurement scaling.

8.2.2 Generic Calibration function

The terms used have the same meaning in the instrument and in DATACAL; they refer to the instrument itself:
'Input' to refer to the instrument input, hence the output channel of the device to be calibrated.
'Output' for the instrument output, hence the input channel of the device to be calibrated.

8.2.2.1 Parameters setup for the procedure in DATACAL

General parameters

The **DUT Name** field allows you to enter the name of the device to be calibrated; this name is also the name of the procedure.

The **Manufacturer** field allows you to enter the manufacturer of the device.

The **Calibration Method** selection list allows you to define the way in which the instrument is connected, either to a standard reference chain (**Comparison** method), or to a standard generator (**Standard Generator** method). In the latter case, the reference chain is unnecessary.

The **DUT Measurement** selection list allows you to choose the channel that will be used for measuring the device being calibrated; it can be:
. either instrument channel 1, or
entered via the keyboard, in which case, when running the procedure, entry of the values will be requested for each measurement cycle.

The **Reference Measurement** selection list is used in the case of the calibration method with a reference chain; the reference measurement can be:
. either instrument channel 2, or
entered via the keyboard, in which case, when running the procedure, entry of the values will be requested for each measurement cycle.

The **Generator** selection list allows you to choose the generation mode; this can be:
. either manual. In this case, when running the procedure, for each measurement cycle you will be asked to set the generator to the current set point value;
. or instrument channel 2, or
. an external generator (Isotech furnace, pressure generator).

Hart parameters

The **Hart procedure type** field indicates if the device to be calibrated is compatible with the Hart protocol and if this is the case displays the type of measures to be carried out on the device.

Le **Current loop** calibrates the Hart transmitter current loop. The current set points will be numerically sent through the Hart protocol whereas the measures will be carried out analogically on the current loop. In this mode, the calibration protocol is of "standard generator".

The **Detector** type calibrates the detector part (input point) of the Hart transmitter, the set points are generated by the instrument or by an external generator and sent to the transmitter's input channel. The measures are numerically carried out through the Hart procedure.

The **Loop and Detector** type calibrates the whole of the Hart transmitter, the set points and the current loop. The set points are generated by the instrument or through an external generator and applied at the input channel of the transmitter. The measures are analogically carried out on the current loop.

The **Manufacturer ID** and **Transmitter ID** fields are the Hart transmitter's identifiers. They define the type of model to which the procedure applies.

The **Read Default Hart Procedure from Instrument** button downloads the default parameters of the chosen Hart procedure and input them in the current procedure. To download these parameters, the Hart device must be physically and logically connected to the instrument. DATACAL must be connected to the instrument. This download sends the manufacturer and Hart device transmitter's identifier as well as the default configurations parameters of the input and output channels, the scale and units.

Set Point values

The **Set Point** table allows you to enter the calibration points.

The **Generation Mode** field has the same meaning as in transmitter calibration.

The **Rest Point** field represents the set point value sent to the generator at the end of running the procedure.

Stability detection

The **Detection** field allows you to choose the **Manual** or **Automatic** sequencing mode for calibration points.

The **Delay before** field allows you to enter the time between sending the set point value to the generator and the instrument's output measurement.

Confirmation of calibration point stability can be **Manual**, in which case you must press a key on the instrument to confirm stability, or **Automatic**, in which case the reference channel is monitored and stability will be confirmed if the reference measurement:

. does not leave the range of values defined by the **Admissible Variation** field (in absolute value);
. does not deviate from the set point value by more than the value defined in the **Max. deviation with setpoint** field (in absolute and relative values);
. all this during the time defined in the **Minimum stability time** field.

Acceptance condition

The **Display verdict** field defines whether or not measurement deviations relative to the acceptable deviation for the device must be tested.

The **Acceptable deviation** field allows you to define the acceptance condition (OK or NOT OK) on the difference between the set point value and the measurement in the form of a relative deviation and an absolute deviation.

Channels

The **Channel 1 Device Measure** area allows you to select the mode for measurement of the signal coming from the device and the measurement scaling.

The **Channel 2 Reference Measure** area allows you to select the mode for measurement of the signal coming from the reference and the measurement scaling.

The **Channel 2 Generator** area allows you to select the generator simulation mode to the device's measurement input and scaling of the set point values.

8.2.3 Calys 60IS / 80IS / 120IS Calibration Function

Procedures for calibrating a Calys 60IS / 80IS / 120IS instrument are defined by two separate files: a procedure file and a file called Tag. The procedure file contains generic parameters of the procedure while the file Tag contains the elements of a calibration as the name of the device under test.
Tag file refers to a procedure file. This file will be selected in the menu to launch the instrument calibration and to generate a calibration report file.
Multiple Tag files can refer to the same procedure.

8.2.3.1 Setting procedure in DATACAL

General Settings

The **Procedure Name** field allows you to enter the name of the procedure which may be the name of the device to be calibrated.

The **Manufacturer** field allows you to enter the manufacturer of the device to be calibrated.

The **Model** field Camera model captures the model of the device to be calibrated.

The **Serial Numb**er field allows you to enter the serial number of the device to be calibrated.

The **Description** box allows you to add comments on the procedure.

Tag's Number field allows you to select the number of Tag files to be associated with the procedure (1 file Tag = 1 PV generated calibration). It is through these Tag files that the procedure can be initiated into the instrument and thus generate a PV calibration.

Stabilization time field is used to select the measurement settling time of the output of the device to be calibrated. This time is expressed in seconds.

Reference channel (Channel 2)

The **Measure** field is used to select the mode of measuring the reference.

The **Range** field to select the measurement range of the reference.

The **Direction** field to select the direction (input or output) of the reference channel.

The **Decimals** field allows you to select the display format of the data reference channel (0 to 5 decimal places to display for measurements).

The **Channel Pressure** field can select the channel pressure to be used in case the selected metering mode is the mode pressure.

Tc Type field allows to select the type of thermocouple (type J, K ...) for reference. This field appears if the selected metering mode is the mode Thermocouple.

CSF Type field allows to select the method of using the coefficient of junction used. CSF can be defined in two ways (internal or external). This field appears if the selected metering mode is the mode Thermocouple.

Standard field allows to select the standard temperature used for the reference channel in case the selected metering mode is the mode Thermocouple or RTD.

RTD Type field to select the type of RTD sensor used for the reference channel in case the selected metering mode is the mode RTD.

The **Connection** field allows you to select the type of connection used for RTD sensor selected.

Output Level field is used to select the output level of the pulse in the event that the measurement mode selected for the reference channel is the user frequency, and in the case the pulse size is selected.

The **Trx** field selects the mode of transmitter used. The use of a transmitter mode to automatically perform a scaling of the measured input signal, the latter may be a voltage or a current. This one is available only if the direction of the channel is configured as input.

The **High** and **Low Scale** fields can respectively learn the high and low values of the scaling of the signal when the transmitter mode is used. These fields only appear when a transmitter mode is selected.

Measurement channel (Channel 1)

Fields previously presented are identical for the measurement channel.

Steps

The table allows you to enter different stages of values the calibration procedure and the accepted error tolerance at each stage.

Step Value column allows you to enter the value of the step in the procedure.

Error Base column to select the method of calculating the error tolerance (on scale or reading).

Relative Error column captures the value of the relative error, which is used for both methods of calculating error tolerance.

Absolute Error column captures the value of the absolute error is used for the calculation of reading error tolerance. In the case of the calculation of scale, this field is automatically set to 0, there is no need to learn in this case.

Minimum Scale column can enter the minimum value of the scale in the case of calculating the error tolerance level. In the case of calculation method of reading, this field is automatically set to 0, there is no need to learn in this case.

Maximum Scale column captures the maximum value of the scale in the case of calculating the error tolerance level. In the case of calculation method of reading, this field is automatically set to 0, there is no need to learn in this case.

9. APPENDIX B: STARTING DATACAL VIA THE COMMAND LINE

It is possible to specify one or two arguments in the command line to start running DATACAL.

The first argument is the name of the operating file for instrument (with **.DCC** extension) to be loaded.
The second is the key word "-connect" which forces automatic connection to the instrument, as soon as the application is started.

These arguments can be defined via the **Program Properties** dialog box accessible via the **File|Properties** command in the Program Manager menu.

The general syntax of the command line is therefore as follows:

<DATACAL directory>\DATACAL <access path to file .DCC> **-connect**

Example:

The command line:

C:\Program Files\DATACAL\DATACAL C:\Instruments\Instr1.dcc -connect

starts running the *DATACAL.EXE* program located in the *C:|Program Files|DATACAL* directory by asking it to load the operating file for instrument *Instr1.dcc* located in the *C:|Instruments* directory, and to automatically perform connection to the instrument.