MICRONAS

Application Note

HAL®/HAR® 3970

Programming Environment Description

3D HAL® Technology

master HAL® Technology

Edition Oct. 12, 2020
APN000172_001EN

Contents

1. General Information

This document is intended as guidance for programming HAL/HAR 3970 sensors using TDK-Micronas programming software. Every functionality integrated in the software is described. The different tabs of the programming environment, the register settings and the tools are explained in detail. In combination with the respective data sheets and application notes “HAL/HAR 3970 Programming Guide” and “HAL/HAR 3970 User Manual” it represents the complete customer documentation of the HAL/HAR 3970 3D position sensor.

1.1. Certification

TDK-Micronas GmbH fulfills the requirements of the international automotive standard IATF 16949 and is certified according to ISO 9001. This ISO standard is a worldwide accepted quality standard.

1.2. Support

We kindly ask you to register on https://service.micronas.com in order to obtain access to the workgroups for our various product families. Here you are able to get support by opening a support ticket.

TDK-Micronas GmbH – Application Engineering

Hans-Bunte-Strasse 19

D-79108 Freiburg im Breisgau

2. Main Panel

After successful installation of the HAL/HAR 3970 Programming Environment the application can be started by running the file “HAL_3970.exe” from the Windows start menu.

The **Main** panel (see Fig. 2–1) of HAL/HAR 3970 Programming Environment will appear after the start. The application is already running on startup, which is indicated by a black run button above the **Main** panel. By clicking the red stop button above (see Fig. 2–2) or the grey stop button below the application will be stopped.

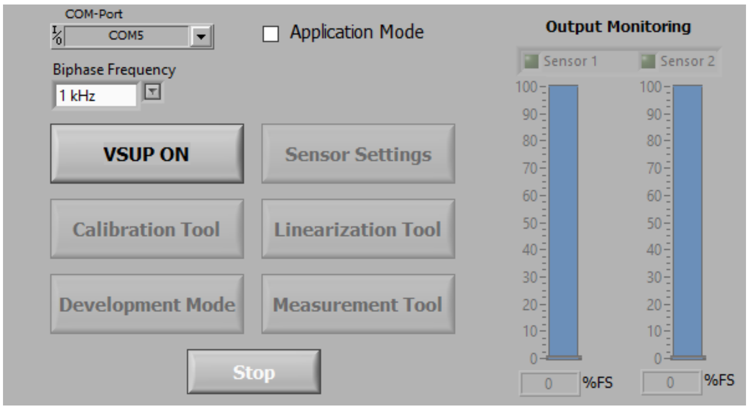

Fig. 2–1: Main Panel of HAL 3970 Programming Environment

To rerun the stopped application the white arrow next to the stop button (see Fig. 2–2) has to be clicked.

Fig. 2–2: Menu Bar at Stopped Application

COM-Port

Once the application is running, as a first step the proper COM-Port connecting the PC to the used programming device (TDK-MSP) has to be selected. A drop-down menu lists all available COM-Ports and offers a functionality to refresh this list, in case a new device has been plugged into the PC. A device which is plugged into the PC with an USB cable can be reached via an automatically created virtual COM-Port address.

Application Mode

If the Application Mode check box is selected, pressing the VSUP ON button will turn on only the supply voltage for the sensor to operate the sensor in applicative mode. There are no other functions available.

VSUP ON/VSUP OFF

Clicking on the VSUP ON button switches on the supply voltage of the sensor connected to the programming device. The software then automatically checks whether the connected sensor type is supported by the HAL/HAR 3970 Programming Environment by reading the HW_ID register. If no supported sensor is connected the Programming Environment changes back to its original state and the supply voltage will switch off automatically. In case the communication with the programming device has been successfully established and the connected sensor is supported, the button label changes to VSUP OFF and all grayed-out buttons on the **Main** panel are activated. The menu for COM-Port and the Application Mode checkbox are disabled and grayed-out as long as the supply voltage is turned on.

Output Monitoring

The Output Monitoring feature indicates with green LEDs, which of the connected sensors (on channel 1, 2 or on both channels) are supported by the HAL/HAR 3970 Programming Environment and visualizes their current output status. It is automatically enabled after an errorless power-on of the supply voltage and stays active as long as the supply voltage is turned on and no functional tool has been selected. This is realized by continuously reading the OUT_1 register of the active sensors and indicating them on a vertical scale bar (in % of full scale). Additionally the clamping levels which are programmed to the sensors are shown as blue areas on the scale bar, indicating the limits of the sensors' outputs. This feature allows the user to get a first impression of the sensor's output behavior resulting from the currently programmed parameters.

Version Indication

In the lower right corner of the panel the currently used version number of the programming environment (SW) and the firmware of the programming device (FW) are displayed in case of a power-up.

Provided an errorless power-on of the supply voltage, the **Main** panel also gives access to the different functional tools of the HAL/HAR 3970 Programming Environment.

– Sensor Settings (see Section 3)

– Calibration Tool (see Section 4)

– Linearization Tool (see Section 5)

– Development Mode (see Section 6)

– Measurement Tool (see Section 7)

In case one of these tools is selected a new panel shows up and the **Main** panel becomes inactive until the new panel is closed.

3. Sensor Settings

Clicking the Sensor Settings button on the **Main** panel of the HAL/HAR 3970 Programming Environment opens the **Sensor Settings** panel. This panel is used to configure the HAL/HAR 3970. It gives access to the customer relevant register settings of the connected sensor (see Fig. 3–3).

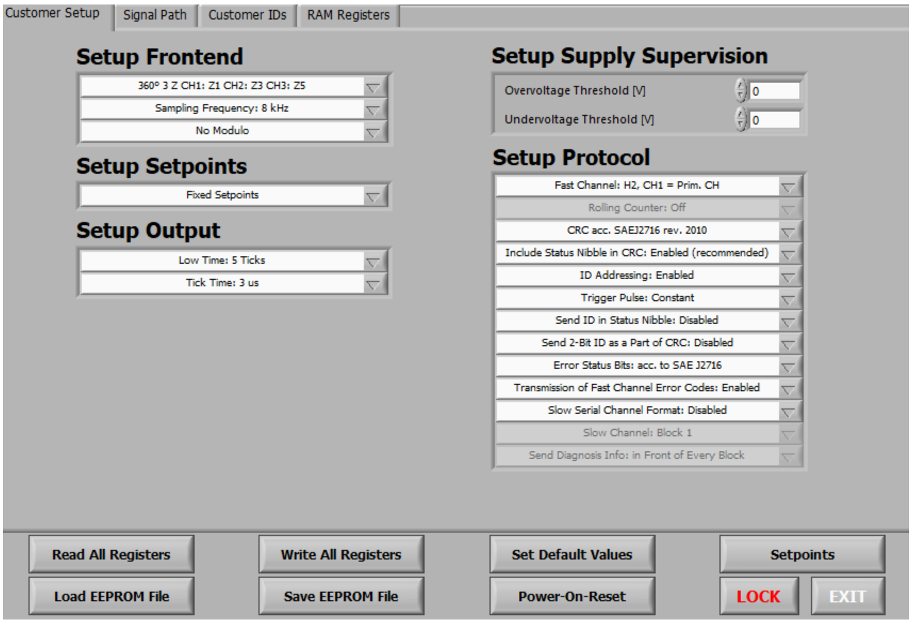

Fig. 3–3: Sensor Settings Panel – Customer Setup Tab

The panel is divided into several function-related areas and buttons, which are described hereinafter. Fundamentally the panel consists of four tabs: the **Customer Setup** tab, the **Signal Path** tab, the **Customer IDs** tab and the **RAM Registers** tab. The **Buttons Area** is located below these four tabs (see Fig. 3–17).

Sensor Selection

On startup the panel dynamically enables the selective sensors in the selection menu on top of the **Sensor Settings** panel (see Fig. 3–4). In case two supported sensors are available Sensor 1 will be chosen by default, indicated by the green circle. The user can switch between communication with Sensor 1 and Sensor 2 by clicking the grey circle of the desired sensor, which will turn green. In case only one supported sensor is available the selection of the other sensor will be disabled and grayed-out.

After switching the selected sensor the sensor type indication in the lower left corner as well as the functions of the **Sensor Settings** panel are updated accordingly.

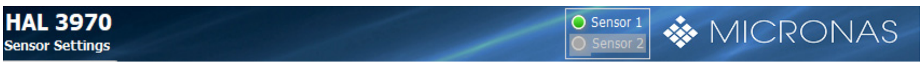

Fig. 3–4: Sensor Selection

In addition on startup all available registers of the sensor are read and entered into the appropriate drop down menus or control elements.

3.1. Customer Setup Tab

The **Customer Setup** Tab is selected by default when the Sensor Settings panel is opened. It is divided into five different areas (see Fig. 3–3).

The drop-down menus in the upper left area show the current setup parameters of the frontend (see Fig. 3–5). With the first drop-down menu the user is able to configure the sensor in different measurement configurations (the constellation of active Hall plates), depending on the required measurement task. SETUP_FRONTEND[3:0] is defining the various available measurement configurations. The second drop-down menu allows the customer to switch the sampling frequency (low-pass filtering) between 2, 4, 8 and 16 kHz. The modulo function can be enabled with the third drop-down menu.

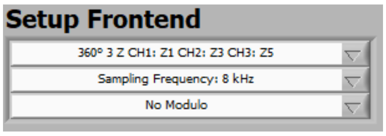

Fig. 3–5: Customer Setup Tab – Setup Frontend

The area directly below the frontend area has an influence on the setup of the setpoints. Fixed or variable setpoints are selectable via the drop-down menu. Therefore HAL/HAR 3970 offers two different possibilities:

1. Fixed setpoints: the customer is able to configure 33 equidistant setpoints.
2. Variable setpoints: 19 non-equidistant setpoints are available, of which 17 are freely selectable.

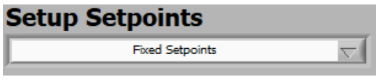

Fig. 3–6: Customer Setup Tab – Setup Setpoints

Setup Output is the third area on the left hand side of the **Customer Setup** tab. The drop-down menus offer the possibility to set the low time between 3 and 6 ticks and the tick time of the SENT output between 1 and 3 µs.

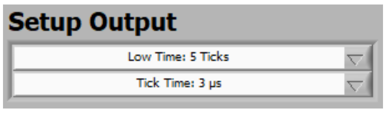

Fig. 3–7: Customer Setup Tab – Setup Output

Setup Supply Supervision is the upper area on the right hand side of the **Customer Setup** tab. The supply supervision offers the possibility to set an over- and an undervoltage threshold for supervision of the external power supply. The thresholds have a length of 8 bits each with a step size of 100 mV/bit.

Fig. 3–8: Customer Setup Tab – Setup Supply Supervision

On the lower right hand side the setup of the SENT output protocol is available. The following protocol configurations are available:

– Change the frame format between H.2 and H.1 - A.7
– select a rolling counter or disable the rolling counter
– CRC type: CRC acc. SAEJ2716 rev. 2016/legacy CRC acc. SAEJ2716 rev. 2008
– include status nibble in CRC: disabled (SENT conform)/enabled
– ID addressing: enabled/disabled
– trigger pulse: constant/variable
– send ID in status nibble: enabled/disabled
– send 2-bit ID as a part of CRC: enabled/disabled
– error status bits: always zero or according to SENT SAEJ2716
– transmission of fast channel error codes: enabled/disabled

In addition some settings for the SENT slow channel can be configured if the 12-bit enhanced serial message format is enabled:

– Select blocks that are sent over the slow channel in addition to block 1
– select SENT SDF mode: diagnosis in front of every block/in front of every ID

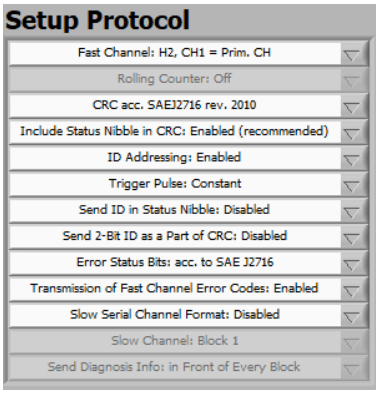

Fig. 3–9: Customer Setup Tab – Setup Protocol

Note: For detailed information about defining the sensor output settings refer to the registers SETUP_OUTPUT and SETUP_PROTOCOL.

Note: It is mandatory to set the lock bit after the sensor was configured and programmed. After the customer lock is activated (by writing and power-on-reset), it is not possible to program the sensor anymore.

3.2. Signal Path Tab

The second tab is the **Signal Path** tab. It contains all parameters used for adjusting the signal path. The current content of the registers (including the registers covered in the other tabs) can be read out by clicking the button Read All Registers. New values can be stored by clicking the button Write All Registers. Above the signal path frontend area the customer gets access to the phase correction, LP-Filter and the registers MAG_LOW and MAG_HIGH. Via the frontend area itself, the customer can configure the offset and gain settings for channel 1, channel 2 and channel 3 and gets access to the Temperature Drift Compensation Tool. Moreover the signal path backend, including e.g. the REF_ANGLE_0...2_CH1 polynomial, SP_OFFSET_CH1 and SP_GAIN_CH1, can be configured. For details about each stage of the signal path and the relation of the parameters and the signal path, please refer to the signal path described in the application note “HAL/HAR 3970 User Manual”.

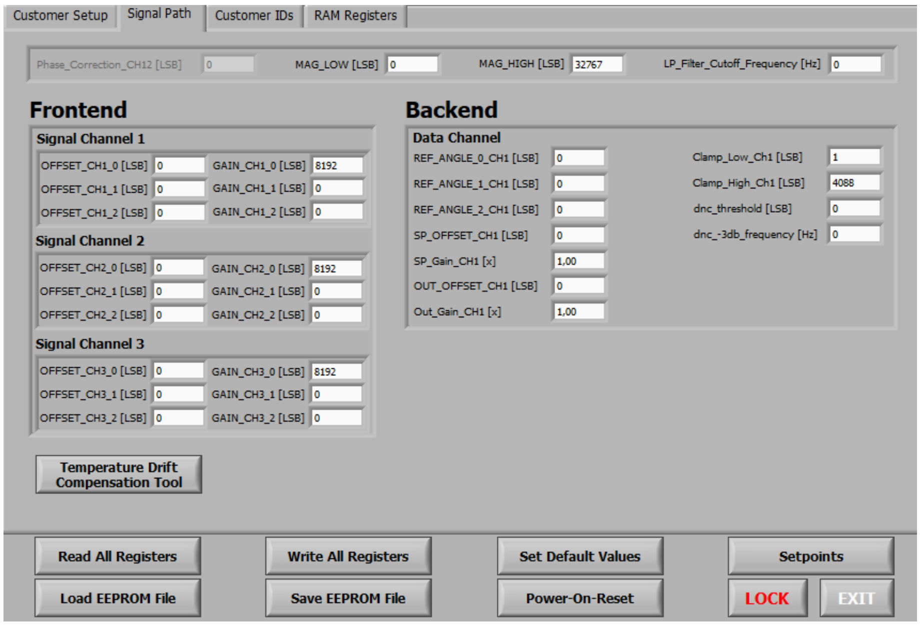

Fig. 3–10: Sensor Settings Panel - Signal Path Tab

Phase Correction

PHASE_CORRECTION_CH12 can be used to compensate a phase-shift of channel 2 in relation to channel 1. The neutral value for the register is zero (no phase-shift correction). The following equation shows how a desired phase error correction is converted into a register value approximately for phase errors less than 10°. In case the phase-shift correction is used, it might be necessary to adapt GAIN_CH2_0...2 according. In the programming environment the phase correction must be entered in LSB.

$$\text{PHASE\_CORRECTION\_CH12} \approx \sin(\beta) \times 32768 \tag{1}$$

$$\beta = \text{phase error in angular degree}$$

Low Pass Filter

The Low Pass (LP) Filter is a first order digital infinite impulse response (IIR) filter to reduce the sampling noise. With the LOW_PASS_FILTER register value it is possible to select different –3 dB (cutoff) frequencies for HAL/HAR 3970. The register is 16-bit organized. In the **Signal Path** tab the –3 dB filter frequency can be directly entered in Hertz. The highest allowed –3 dB frequency is half of the selected sampling frequency. To disable the filter insert “Off”.

Magnetic Field Range

MAG_LOW and MAG_HIGH define the low and the high level for the magnetic field range check function. The magnetic field range check compares the AMPLITUDE register value with an upper and lower limit threshold defined by the registers MAG_LOW and MAG_HIGH. If either the lower or the upper limit is exceeded the sensor indicates it on the sensor’s output (error band active; error can be read out via DIAG_0[5:4]). The MAG_LOW limit is reached in case of a magnet loss. On the other hand the MAG_HIGH limit can be used to detect if the distance between the magnet and the sensor becomes too short. The registers MAG_LOW and MAG_HIGH have a length of 16 bits and are two’s complement-coded (–32768...32767).

An example of how to calculate MAG_LOW for a given magnetic field amplitude in mT is given below (valid if no customer offset is used).

Assuming to set the magnet loss detection threshold to 10 mT the MAG_LOW register value is calculated as:

$$\mathrm{MAG\_LOW} = \frac{\left(\mathrm{Mag\_Low} \times 128 \frac{1}{\mathrm{mT}}\right)^2}{32768} = \frac{\left(10\ \mathrm{mT} \times 128 \frac{1}{\mathrm{mT}}\right)^2}{32768} = 50 \qquad (2)$$

Assuming to set the magnet high level threshold to 90 mT the MAG_HIGH register value is calculated as:

$$\mathrm{MAG\_HIGH} = \frac{\left(\mathrm{Mag\_High} \times 128 \frac{1}{\mathrm{mT}}\right)^2}{32768} = \frac{\left(90\ \mathrm{mT} \times 128 \frac{1}{\mathrm{mT}}\right)^2}{32768} = 4050 \qquad (3)$$

The accuracy of MAG_LOW and MAG_HIGH values depends mainly on the sensitivity. The variation of the sensitivity is specified in the datasheet and is roughly ±8%.

Default values: MAG_LOW = 0 and MAG_HIGH = 32767.

Note: The Mag_Low and Mag_High limits have to consider the temperature drift of the magnet.

Customer Offset

OFFSET_CH1_0...2, OFFSET_CH2_0...2 and OFFSET_CH3_0...2 support three polynomials of second order and describe the temperature compensation of the offset of channel 1, channel 2 and channel 3 (compensating a remaining offset in each of the three channels). This means a constant, linear and quadratic offset factor can be programmed individually for the three channels (temperature dependent offset). TDK-Micronas delivers precalibrated sensors. Nevertheless it is possible that due to the magnetic circuit an offset in channel 1, channel 2 and channel 3 occurs. This can be compensated with OFFSET_CH1_0...2, OFFSET_CH2_0...2 and OFFSET_CH3_0...2. A temperature compensation can be done in the Temperature Drift Compensation Tool. The registers have a length of 16 bits and are two's complement-coded. The range is –32768 to 32767. Neutral value respectively default value is 0.

Customer Gain

GAIN_CH1_0...2, GAIN_CH2_0...2 and GAIN_CH3_0...2 support three polynomials of second order and describe the temperature compensation of the sensitivity of channel 1, channel 2 and channel 3 (compensating the amplitude mismatches between the channels). This means a constant, linear and quadratic gain factor can be programmed individually for the three channels (temperature dependent gain). TDK-Micronas delivers precalibrated sensors with compensated gain mismatch between channel 1, channel 2 and channel 3. Nevertheless due to the magnetic circuit, a mismatch between the gains of the channels may occur. This can be compensated by GAIN_CH1_0...2, GAIN_CH2_0...2 and GAIN_CH3_0...2. A temperature compensation can be done in the Temperature Drift Compensation Tool. The registers have a length of 16 bits and are two's complement-coded. The range is –32768 to 32767. The value 0 corresponds to 0%. Default values are: GAIN_CHx_0 = 8192, GAIN_CHx_1 = 0 and GAIN_CHx_2 = 0. The number 8192 as default value arises from the processing in the signal path, this is then multiplied by factor 4.

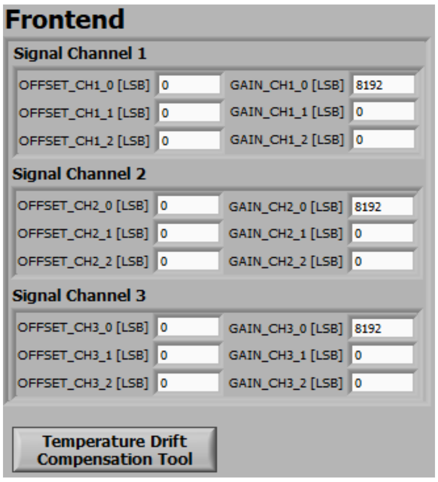

Fig. 3–11: Signal Path Tab – Frontend

3.2.1. Temperature Drift Compensation Tool

By clicking the Temperature Drift Compensation Tool button the user is able to perform a temperature compensation for the parameters Offset, Gain and Ref_Angle in the next window.

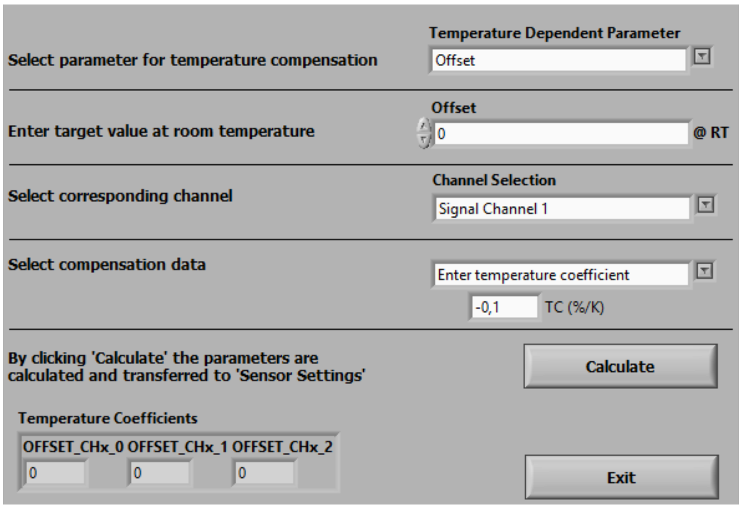

Fig. 3–12: Signal Path Tab - Frontend Temperature Drift Compensation Tool

If an offset, gain and/or reference angle is used in the signal path it may be necessary to adjust this parameter to the temperature behavior of the magnetic system.

After selecting the temperature dependent parameter the user has to enter the target value at room temperature. This is followed by the selection of the corresponding channels.

Then two methods for the compensation data are selectable by clicking the drop-down menu.

Method 1: Enter Temperature Coefficient

In case the temperature behavior of the magnetic system is a linear function and is already known, this method can be chosen. The temperature coefficient TC in %/K can be entered below the drop-down menu of the compensation data selection.

By clicking Calculate, the corresponding EEPROM temperature coefficients are calculated and shown in the lower left corner.

Method 2: Read Temperature Behavior From File

In case the temperature behavior of the complete magnetic system is not sufficiently described by a linear function or not known, it is recommended to measure the parameter change over temperature. For this method a measurement file must be generated.

The file has to fulfill the following requirements:

1. The data format is a text file ".txt".
2. The columns are tab separated.
3. The file has 2 columns:

A header row is mandatory.

Column 1

The first column contains the temperature values (TEMP_ADJ) in LSB (Least Significant Bit). These values can be directly read from the sensor by using the Measurement Tool and selecting the TEMP_ADJ parameter.

It is also possible to convert the ambient temperature from °C into LSB by the following equation:

$$\text{TEMP\_ADJ} = \frac{89.25}{°\text{C}} \times T + 3720 \quad (4)$$

Column 2

The second column includes the parameter values at the temperature values listed in column 1 in relation to the parameter value at room temperature (RT = 25 °C). By using the Measurement Tool the registers COMP_CHx (for Offset and Gain) and ANGLE_OUT_1 (for Ref_Angle) shall be acquired. The best method is to acquire the data over the full temperature range. For example, the following values were measured:

Table 3–1: Example for Offset Change on Channel 1 over Temperature

Temperature in °C	TEMP_ADJ in LSB	Offset of COMP_CH1 in LSB
–40	150	4260
25	5951	4000
150	17108	3500

In this example the unit of the offset is LSB, which is not mandatory. The parameters can be measured in any unit (e.g. in mT). After the measurements the parameter values at the different temperatures are normalized to the parameter value at room temperature. The final file should look like the table below:

$$\text{Normalized Parameter (T)} = \frac{\text{Parameter(T)}}{\text{Parameter(RT)}} \quad (5)$$

Table 3–2: Example for a Temperature Behavior File

TEMP_ADJ in LSB	Normalized Parameter
150	1.065
5951	1.000
17108	0.875

By clicking Calculate the created file has to be selected in the next window. Afterwards the corresponding EEPROM temperature coefficients are calculated and transferred to the **Signal Path** panel.

3.2.2. Backend Data Channel

For a better understanding of the relation between the parameters and the signal path the user is referred to the diagrams of the signal path in the HAL/HAR 3970 User Manual.

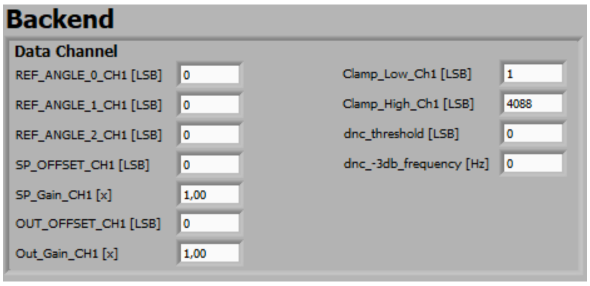

Fig. 3–13: Signal Path Tab – Backend

REF_ANGLE_0...2_CH1

The output signal zero position defines the reference position for the angle output and therefore it is possible to shift the discontinuity in the output characteristics out of the measuring range. These parameters can be set to any value of the angular range. REF_ANGLE_0...2_CH1 defines a polynomial of second order with REF_ANGLE_0_CH1 (constant part), REF_ANGLE_1_CH1 (linear part) and REF_ANGLE_2_CH1 (quadratic part). Each register of REF_ANGLE_0...2_CH1 has a length of 16 bits and is two's complement-coded. REF_ANGLE_0_CH1 is automatically calculated in the Calibration Tool. A temperature compensation of REF_ANGLE_0...2_CH1 can be done in the Temperature Drift Compensation Tool.

SP_OFFSET_CH1

The register SP_OFFSET_CH1 shifts the signal to the desired range of the subsequent Setpoint Linearization blocks. SP_OFFSET_CH1 = 0 is the neutral setting. The register has a length of 16 bits and is two's complement-coded (–32768...32767). It is automatically calculated in the Calibration Tool.

SP_Gain_CH1 (SP_GAIN_CH1 and nmult_1)

The register SP_GAIN_CH1 scales the signal to the desired range of the subsequent Setpoint Linearization blocks. SP_Gain_CH1 = 1 is the neutral setting and equates to 100% of the scaling range for an angular range from 0° to 360° (-1 equates to -100%). The parameter SP_Gain_CH1 consists of the register value SP_GAIN_CH1 multiplied by 2^{nmult_1} to achieve gain factors up to 128 (default factor 1). The register SP_GAIN_CH1 has a length of 16 bits and is two's complement-coded. SP_GAIN_CH1 and nmult_1 are automatically calculated in the Calibration Tool. SP_Gain_CH1 must be entered as a multiplier (not in LSB).

OUT_OFFSET_CH1 and Out_Gain_CH1

The final customer-programmable scaling block is used to scale the signal according to the desired output signal range. This is realized by the EEPROM registers OUT_OFFSET_CH1 and OUT_GAIN_CH1. The registers have a length of 16 bits and are two's complement-coded. Default value for OUT_OFFSET_CH1 is 0 and default value for OUT_Gain_CH1 is 1 (OUT_GAIN_CH1 = 16348). In case of OUT_Gain_CH1 being smaller than 0 the output signal is inverted. OUT_OFFSET_CH1 must be entered in LSB and Out_Gain_CH1 must be entered as a multiplier. They are automatically calculated in the Calibration Tool.

Clamp_Low_Ch1 and Clamp_High_Ch1

The clamping levels CLAMP_LOW_CH1 and CLAMP_HIGH_CH1 establish the minimum and maximum value of the data channel output signal. Both clamping levels can have values between 1 and 4088. The registers have a length of 16 bits and are two's complement-coded (-32768...32767). The valid range is from -32767...32767.

Note: CLAMP_LOW_CH1 has to be smaller than CLAMP_HIGH_CH1.

DNC Filter

The DNC (Dynamic Noise Cancellation) filter is a non-linear filter and can be used for further noise reduction in addition to the first order low-pass filters after the A/D-converters. It decreases the output noise significantly by adding a low-pass filter with a very low cutoff frequency for signals below a certain signal change threshold (dnc_threshold).

The cutoff frequency dnc_-3dB_frequency of this first order IIR filter (Infinite Impulse Response low-pass filter) can be entered in Hertz (up to half of the sampling frequency) into the programming environment.

Note: To disable the DNC filter the EEPROM setting dnc_threshold or dnc_–3dB_frequency must be programmed to 0.

3.3. Customer IDs Tab

The next tab is the **Customer IDs** tab. The CUSTOMER_IDx registers contain 16 bit words and can be used to store customer specific content/production information, like serial number, project information, OEM codes, etc. The decoded customer IDs will be part of the SENT slow channel in case that the SENT output is activated and transmission via slow channel is selected as well. All configurable customer specific content is defined in CUSTOMER_ID0...9.

The current content of the registers can be read out by clicking the button Read All Registers. New values can be stored by clicking the button Write All Registers. On the left hand side the customer gets access to the Customer IDs register values. On the right hand side the customer gets access to the decoded Customer IDs: sensor type, fast channel information and OEM codes. Entered register values are directly decoded respectively entered decoded values are directly encoded and displayed in the Customer IDs tab.

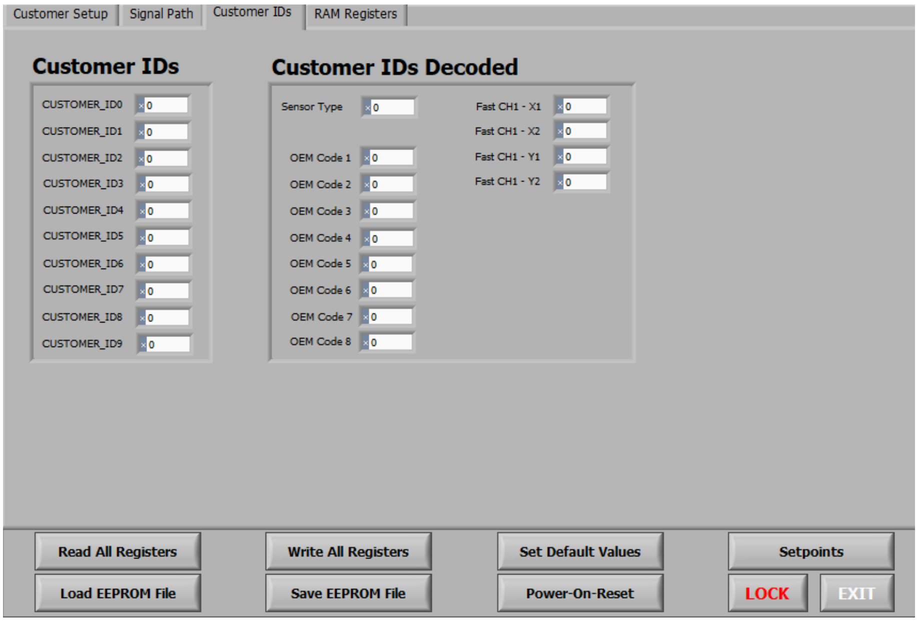

Fig. 3–14: Sensor Settings - Customer IDs Tab

3.4. RAM Registers Tab

Last tab is the **RAM Registers** tab (see Fig. 3–15). In this area certain RAM registers can be read by clicking the Read RAM button (read RAM registers once/continuously).

Fig. 3–15: Sensor Settings – RAM Registers Tab

COMP_CH1...2

The COMP_CH1...2 registers contain the magnetic field information of channel 1 and channel 2 after temperature-drift compensation of the Hall plates. These registers have a length of 16 bits each and are two's complement-coded (–32768...32767).

CUST_COMP_CH1...2

CUST_COMP_CH1...2 contain the customer compensated magnetic field information of channel 1 and channel 2 used for the angle calculation. These registers contain already the customer phase-shift, gain and offset corrected data as well as LP filtering. These registers have a length of 16 bits each and are two's complement-coded (–32768...32767).

ANGLE_OUT_1

The ANGLE_OUT_1 register contain the digital value of the position calculated by the angle calculation algorithm. The register has a length of 16 bits. The full range between –32768 and 32767 is used for the position information. The position can either represent an angular position (angle) or a linear position calculated from two magnetic field components.

SETPOINT_IN_1 and SETPOINT_OUT_1

The SETPOINT_IN_1 register contains the gain and offset corrected position information on the data channel. The SETPOINT_OUT_1 register contains the position information after setpoint adjustment on the data channel. Accordingly, SETPOINT_IN_1 is the input and SETPOINT_OUT_1 is the result of the Setpoint Linearization block.

OUT_1

The OUT_1 register contains the magnetic field data of the data channel after final gain and offset adjustment and clamping. The register has a length of 16 bits and is two's complement-coded (–32768...32767).

AMPLITUDE

The AMPLITUDE register contains the sum of squares of TDK-Micronas compensated magnetic field values on channel 1 to 3. It has a length of 16 bits signed (0...32767).

For updating the RAM Area, click on Read RAM. If the check box continuously is checked, the data will be read in a loop. To leave the continuous reading mode, the continuously check box has to be unchecked.

Decode Diagnosis Bits

Clicking the Decode Diagnosis Bits button enables an overview of the DIAG_0 and DIAG_1 register. The single bits are decoded via green LEDs. If an error occurred the LED flashes green. For detailed information about the single bits, refer to the memory table of the HAL/HAR 3970 User Manual.

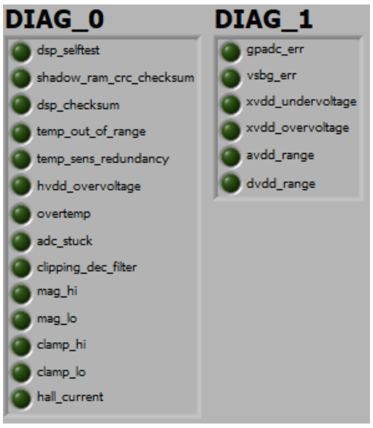

Fig. 3–16: Sensor Settings - RAM Registers Tab Decode Diagnosis Bits

---

Note: Diagnosis bits are latched in programming mode. A power-on-reset is necessary to reset the bits.

3.5. Buttons Area

The **Buttons Area** below the tabs described in the previous sections is always visible and accessible from the **Sensor Settings** panel (see Fig. 3–17). They are referring to all of the sensor settings tabs.

Fig. 3–17: Sensor Settings - Buttons Area

Read All Registers

The signal path registers and customer setup parameters are read from the sensor.

Write All Registers

The signal path registers and customer setup parameters are written and stored in the sensor. After using the Write button a power-on-reset is recommended to boot the sensor with the new customer setup parameters.

Set Default Values

The signal path and customer setup parameters are set to default values. To program the sensor with default values click Write All Registers immediately afterwards.

Setpoints

The **Setpoints** panel is opened for setpoint configuration. For detailed information refer to Section 3.6.

Load EEPROM File

Reads the customer setup and signal path values from an EEPROM dump file (without the Micronas calibration area). The EEPROM dump file must correspond to the form described in Save EEPROM File.

Save EEPROM File

An EEPROM dump of all parameters will be generated and saved to a file. The file contains the register numbers and their contents (clarified by the header). The values in the file are tab separated.

Power-On-Reset

Power-On-Reset (POR) switches the supply voltage off and on again.

LOCK

The customer lock bit can be set by clicking the LOCK button. If the LOCK button has been clicked the CRC checksum over all customer settings (EEPROM) is calculated and stored in CUSTOMER_CHECKSUM. After the customer lock is activated (by writing and power-on-reset), it is not possible to program the sensor anymore.

EXIT

Returns to the **Main** panel (see Fig. 2–1).

3.6. Setpoints Panel

The **Setpoints** panel can be called via the button Setpoints on the **Sensor Settings** panel. It gives the user access to the programmable setpoint values, which are part of the setpoint scaling block of the signal path. At startup the programmed setpoint values of the sensor selected in the **Sensors Settings** panel are read out and displayed graphically and numerically according to the setpoints configuration. The desired setpoints configuration has to be stored before opening the **Setpoints** panel.

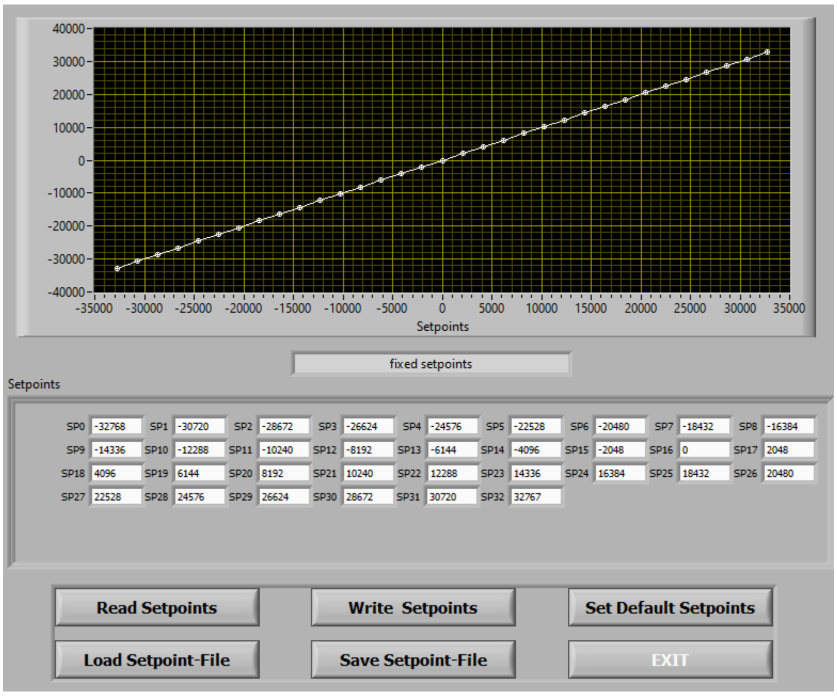

Fig. 3–18: Setpoints Panel - Absolute View of the Setpoint Values

The graph visualizes the transfer curve of the setpoint scaling block by plotting the setpoint values. The user is able to change each setpoint value individually.

The following actions are possible:

– Read Setpoints: the current setpoint values are read out and displayed according to the selected configuration.

– Write Setpoints: the displayed setpoint values are programmed to the sensor.

– Set Default Setpoints: the default setpoint values (neutral) are set and displayed according to the selected configuration.

– Load Setpoint-File: the setpoint values are loaded from an external text file and displayed according to the selected configuration.

– Save Setpoint-File: an EEPROM dump of all parameters (including the setpoint values) will be generated and saved to a file. Information about the structure of the file can be found in Section 3.5, Save EEPROM File.

– EXIT: returns to the **Sensor Settings** panel without additional changes**.**

4. Calibration Tool

By clicking **Calibration Tool** on the **Main** panel (see Section 2) several calibration methods can be selected (see Fig. 4–19).

Note: For the calibration it is necessary to know if the ANGLE_OUT_1 register is increasing or decreasing. This can be found out by reading out the ANGLE_OUT_1 register and moving the magnet from the starting point in the desired direction. ANGLE_OUT_1 can be read e.g. in the **Sensor Settings** panel by clicking Read RAM on the **RAM Registers** tab (see Section 3.4).

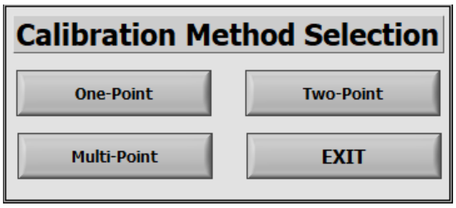

Fig. 4–19: Calibration Methods

The **Calibration Method Selection** panel offers three options: one-point calibration, two-point calibration or multi-point calibration. Inside each **Calibration** panel it is possible to switch between sensor 1 and sensor 2. The one-point calibration can be performed on two sensors in parallel.

Calibration Procedure

All calibration methods start with the dialog to set the setpoints of the active sensor (Sensor 1 and/or Sensor 2) to default values.

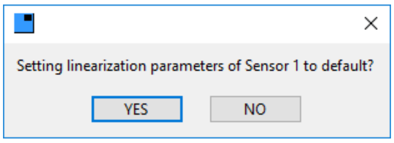

Fig. 4–20: Setting Linearization Parameters to Default

Enter Target Output Values

The first step of a calibration is the selection of the target output values. SP_CP1 is the desired setpoint value at the first calibration point (CP1) and SP_CP2 is the desired setpoint value at the second calibration point (CP2). Target Value 1 is the desired output value at the first calibration point (CP1) and Target Value 2 is the desired output value at the second calibration point (CP2). The target output values are entered in Counts.

Note: For fixed setpoints it is recommended to set SP_CP1 = -31744 and SP_CP2 = 31744 to use the full scaling range.

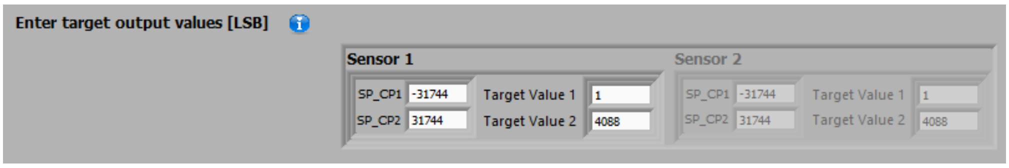

Fig. 4–21: Example Target Output Values

4.1. One-Point Calibration

The one-point calibration can be used when the angular range is known (e.g. 360°).

In case of modulo is activated the discontinuity point is directly set to the calibration point and the modulo 90° (120°, 180° respectively modulo 360°) is mapped to the target values. The difference to the two-point calibration is that the discontinuity point is directly at the first calibration point and no margin is calculated.

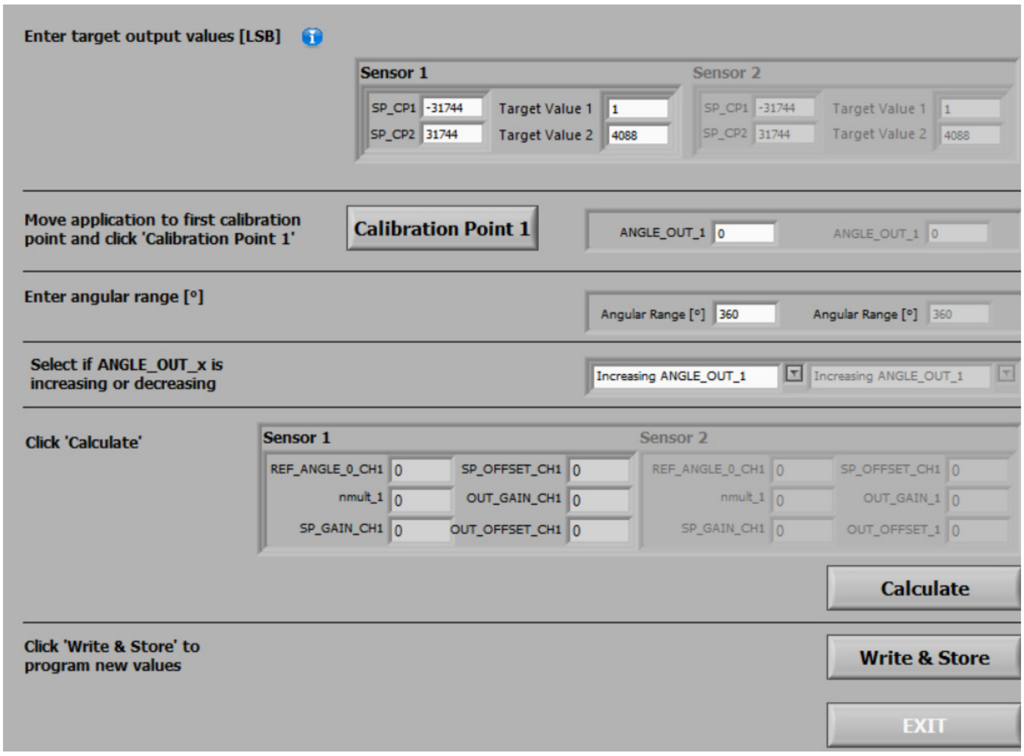

Fig. 4–22: One-Point Calibration Panel

Note: If the modulo function is activated, the angular range has to be set to 360° for a one-point calibration.

Clicking the info button on the top of the panel gives a short overview of the target values and the calibration points:

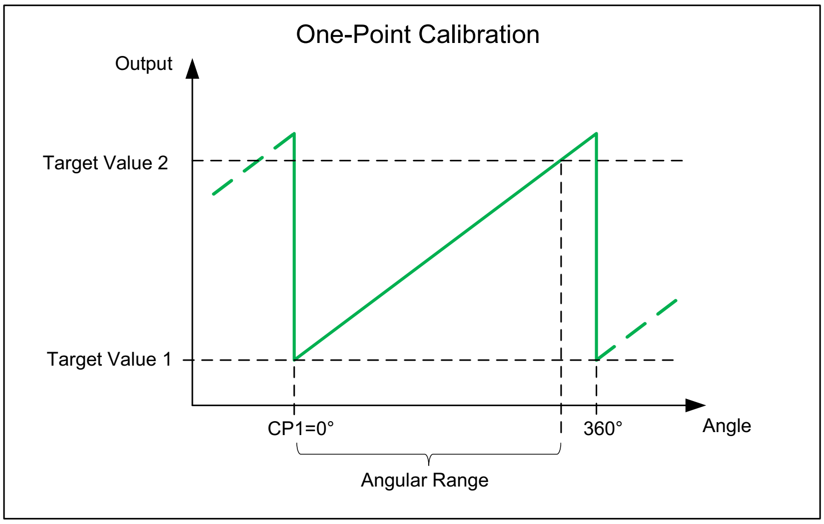

Fig. 4–23: One-Point Calibration Info Button

After SP_CP1, SP_CP2 and the target values have been entered, the start position, where the first target value will be set, the Calibration Point 1 button has to be clicked. Then the angular range must be selected in angular degrees (between 0° and 360°).

The next step is to indicate if the ANGLE_OUT_1 is increasing or decreasing when moving the application from the start position towards the end position.

By clicking Calculate, the register values and nmult_1 for the calibration are calculated. These are REF_ANGLE_0_CH1, nmult_1 SP_GAIN_CH1, SP_OFFSET_CH1, OUT_GAIN_CH1 and OUT_OFFSET_CH1.

By clicking Write & Store the calibrated values are written and stored in the registers.

Exit ends the calibration and returns to the **Main** panel.

4.2. Two-Point Calibration

This method is used to adjust the sensor to the exact angular range of the application. For linear distance measurements and rotation measurements of arbitrary angular ranges (e.g. 90°) this method can be used.

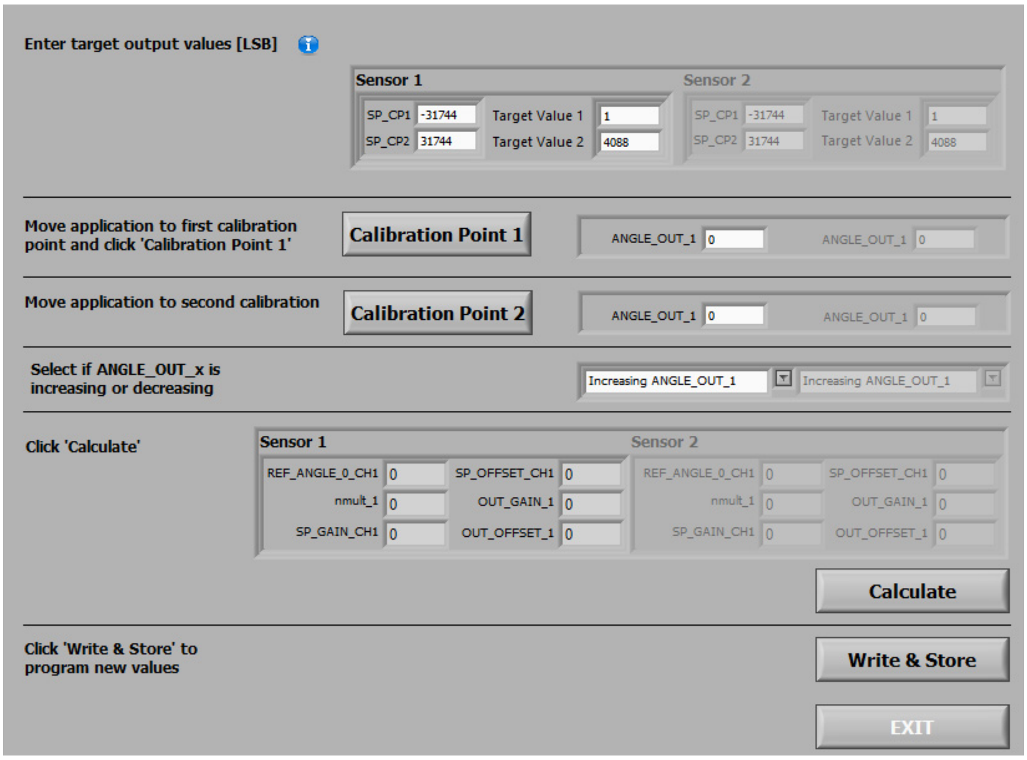

Fig. 4–24: Two-Point Calibration Panel

Clicking the info button on the top of the panel gives a short overview of the target values and the calibration points:

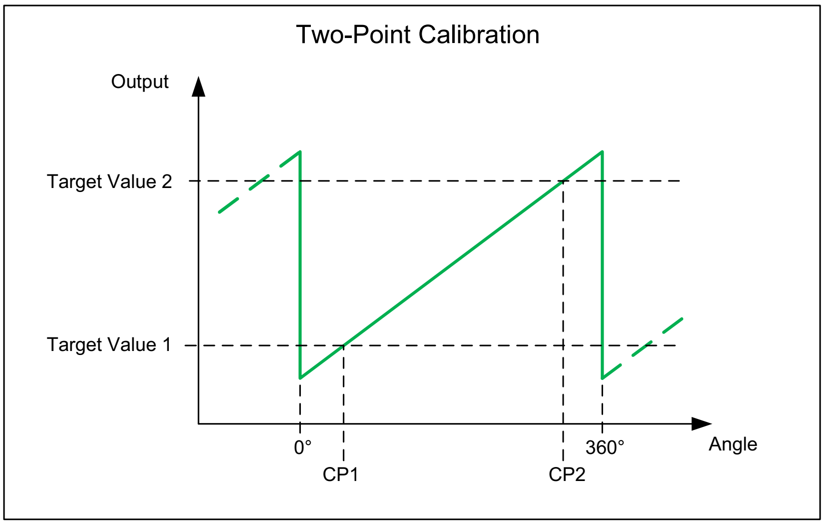

Fig. 4–25: Two-Point Calibration Info Button

Set SP_CP1, SP_CP2 and the target values as described in Section 4. An example for calculating the target values, when they are not known at the calibration points, is given below.

The angular range shall be from 0° (start angle) to 360° and the corresponding output range shall be from 10 %FS (target output value @start angle) to 90 %FS. The calibration points shall be at 30° and at 330°, then the target values for the setpoint and the output range have to be calculated accordingly. The function is:

$$\text{slope} = \frac{\text{output/setpoint range}}{\text{angular range}}$$
$$\text{offset} = \text{target value}_{@\text{start angle}} - \text{slope} \times \text{start angle}$$
$$\text{target value}_{@\text{calibration point}} = \text{slope} \times \text{angle}_{@\text{calibration point}} + \text{offset} \quad (1)$$

In this example the equations give:

$$\text{SP\_CP1} = \frac{(31744-(-31744))}{(360°-0°)} \times 30° + (-31744) = -26453$$
$$\text{SP\_CP2} = \frac{(31744-(-31744))}{(360°-0°)} \times 330° + (-31744) = 26453$$
$$\text{Target Value 1} = \frac{(90\ \%FS - 10\ \%FS)}{(360°-0°)} \times 30° + 10\ \%FS = 16{,}667\ \%FS$$
$$\text{Target Value 2} = \frac{(90\ \%FS - 10\ \%FS)}{(360°-0°)} \times 330° + 10\ \%FS = 83{,}333\ \%FS \quad (2)$$

When the magnet is at the first calibration point click Calibration Point 1. After reaching the second calibration point click Calibration Point 2.

The two-point calibration algorithm tries to have as much margin to the discontinuity point as possible. That means the discontinuity point will be 180° opposite to the center of the measurement range, when the applied angular range is lower then 360°. The angular range is calculated based on the ANGLE_OUT_1 values measured at the calibration points.

The next steps (calculation and programming) should be done as described in Section 4.1.

4.3. Multi-Point Calibration

The multi-point calibration can be used to program any arbitrary output characteristic over the full measurement range using variable setpoints (see Fig. 4–26).

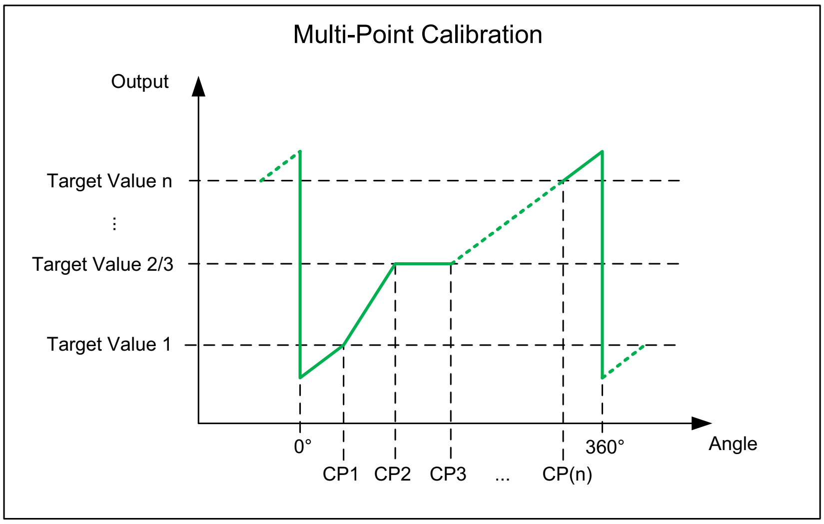

Fig. 4–26: Exemplary Multi-Point Calibration Characteristic

Therefore the panel allows the user to define the desired output behavior by adding calibration points manually or loading a text file (see Fig. 4–27). The file shall contain two tab-separated columns, one counting the number of the calibration points and one with the according output values in percentage of full scale. The maximum number of available calibration points is determined by the number of available variable setpoints (17). At least two calibration points have to be used for a successful calculation. The target values can be set arbitrarily (ascending, steady, or decreasing).

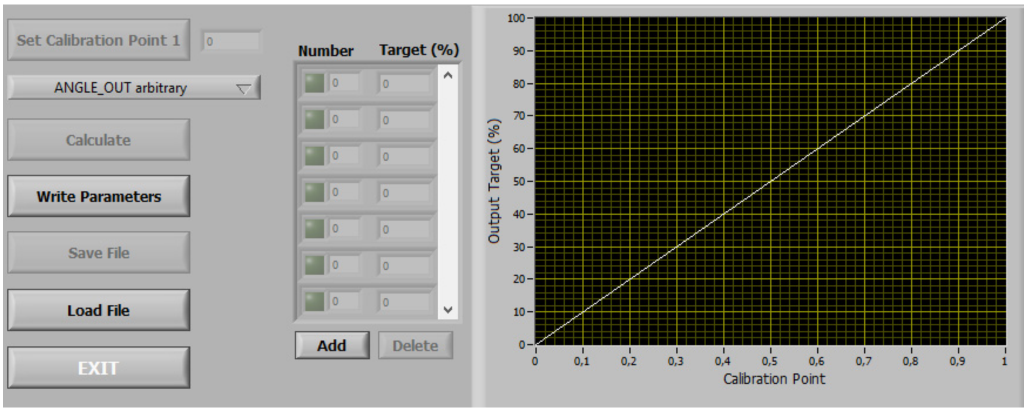

Fig. 4–27: Mutli-Point Calibration Panel

After the desired output characteristic is defined the calibration procedure can be started:

1. Setting the first calibration point: Move the application to the first calibration point and click Set Calibration Point 1. The ANGLE_OUT_1 value is read at this position for further calculation. The green box next to the target value indicates when the reading is finished. The button changes the text to Set Calibration Point 2.

2. Setting the remaining calibration points: Repeat step 1 for all remaining calibration points. If less than five calibration points are defined, the user has to select the ANGLE_OUT_1 order (increasing, decreasing, or arbitrary). For five or more calibration points the SW automatically estimates the ANGLE_OUT_1 order. When all calibration points have been set, the button Calculate is released.

3. Calculating the signal path parameters: Click Calculate for calculation of the appropriate signal path parameters according to the desired output behavior (REF_ANGLE_0_CH1, SP_GAIN_CH1, SP_OFFSET_CH1, SPn, OUT_GAIN_CH1 and OUT_OFFSET_CH1). The algorithm scales the signal path in a way that the variable setpoints are directly mapped to the calibration points. The outer setpoints with fixed setpoint x-values are set to fix values (SP0_Y = -32768 and SP(n)_Y = 32767; n = 18). Unused setpoints are distributed equidistantly on a linear connection between the last calibration point and the outer setpoint SP(n).

4. Programming the calculated parameters: Click Write Parameters to write and store the calculated signal path parameters to the sensor. After programming the registers are readout again and compared with the ideal values. In case of an error during programming the user is notified by a pop-up window.

5. Linearization Tool

The HAL/HAR 3970 features, depending on the setting, a different number of variable or fixed (equidistant/ non-equidistant) setpoints (see Section 3.6) to linearize the output of the sensor. Usually the linearization is necessary when the input magnetic field components cannot be described by an ideal sine and cosine. This is the case mainly for linear distance measurements. The **Linearization Tool** is used to calculate the setpoints. By clicking **Linearization Tool** on the **Main** panel (Fig. 2–1) the **Linearization Tool** panel appears (see Fig. 5–28). The menu on top of the panel allows selecting the sensor (Sensor 1 or Sensor 2), when two sensors are connected.

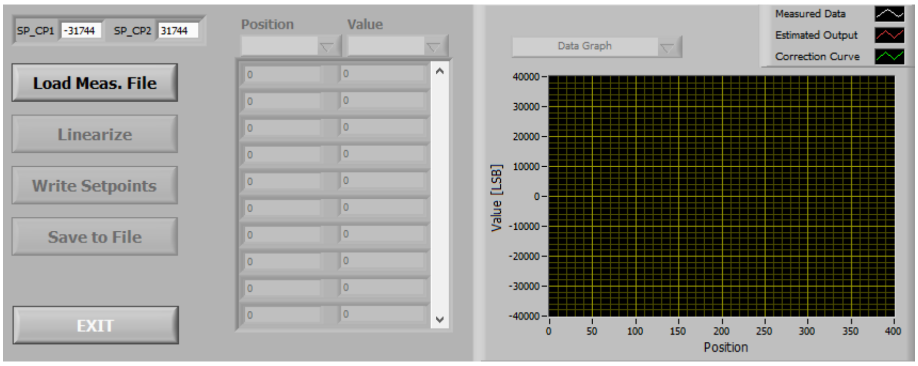

Fig. 5–28: Linearization Tool Panel

At first, a saved text file with angular information over the full application range has to be loaded via the button Load Meas. File. The text file shall contain at least a column with the measurement number or the mechanical position information and a column with the corresponding SETPOINT_IN_1 register values (see Table 5–3). The (arbitrary) naming of the columns shall be located in the first row (header). The data has to be tab-separated. Such measurement file can be created with the **Measurement Tool** described in Section 7. Once the measurement file was successfully loaded, the Linearize button is enabled.

Table 5–3: Example File

Position/ Angle in mm/°	SETPOINT_IN_1 value in LSB
0	-31744
1	-31482
2	-31192
....	....

Linearize

By clicking Linearize the correction function (setpoints) is calculated. The setpoints are scaled in a way that the first value of the input file matches SP_CP1 and the last value of the input file matches SP_CP2.

The user can select between different graphs via the drop-down menu at the lower left corner of the graph: data graph or linearity error graph.

Data Graph (default)

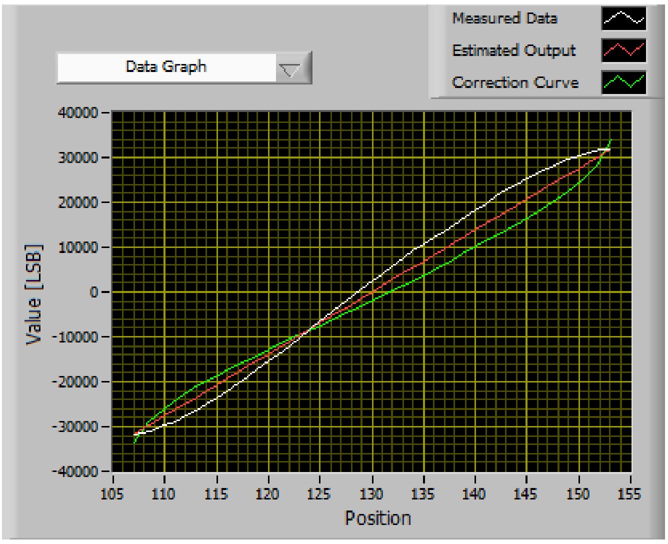

Fig. 5–29: Data Graph Display

With this selection the graph shows three lines: the white line represents the input values read from the file, the green line is the calculated correction curve and the red line is the expected output after the linearization.

Linearity Error Graph

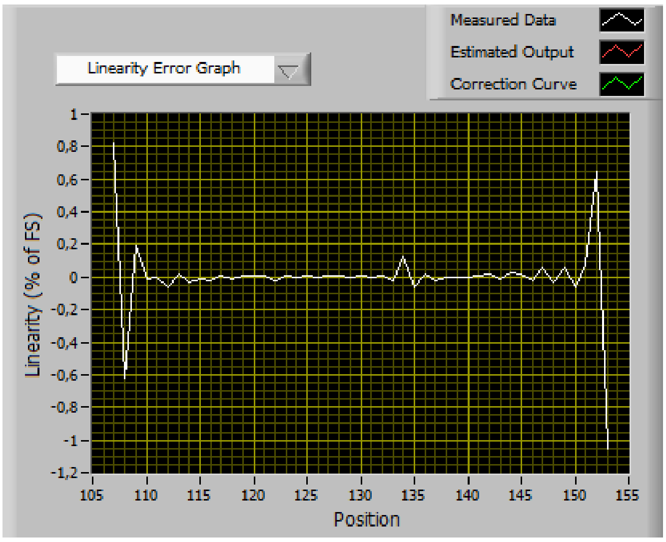

Fig. 5–30: Linearity Error Graph

When linearity error graph is selected, the graph displays the expected linearity error in percentage of the full scale.

After successful calculation of the linearization parameters the Write Setpoints button is released, which triggers writing and storing of the calculated setpoints to the sensor.

When Save to File is clicked, a file will be generated with the input for the linearization (position/angle and the corresponding SETPOINT_IN_1 value), the ideal output, the calculated error and the calculated setpoints.

6. Development Mode

The Development Mode can be used to address single registers and to set some programmer board functions.

Note: The lock bit can be accidentally set in the Development Mode by writing to the SETUP_FRONTEND register (EXT EEPROM: 0x0A). After the customer lock is activated (by writing and power-on-reset), it is not possible to program the sensor anymore.

Fig. 6–31: Development Mode Panel

The menu on top of the panel allows selecting the sensor (Sensor 1 or Sensor 2), when two sensors are connected to the programming device. When only one sensor is connected, no selection is possible.

The HAL/HAR 3970 has 96 EEPROM registers. They are split into 80 EEPROM registers and 16 EXT EEPROM registers (extended EEPROM registers). For a complete list of registers (including RAM and hardware registers), the user is referred to the memory tables of the HAL/HAR 3970 User Manual.

A register can be selected via entering the register address in the Address field or clicking on the Register field and selecting one register. In case of an extended EEPROM register is to be read or written, the EXT button has to be activated.

Read

Clicking Read will read the register value. The register value is shown at the right hand side of the Read button.

Write

Clicking Write will write the data entered at the right hand side of the Write button into the desired register and read back the register. The received register value is shown at the right hand side of the Read button.

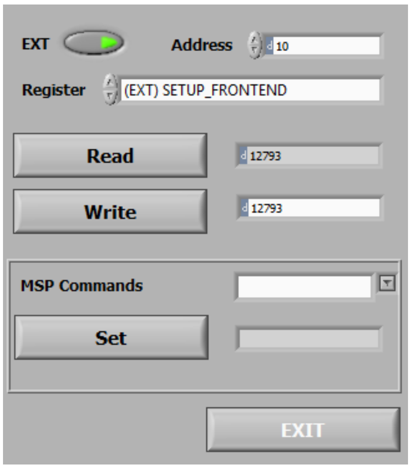

Fig. 6–32: Write 12793 into EXT Register 10

Note: After using the Write button a power-on-reset is recommended to boot the sensor with the new customer setup parameters.

Left from the indicated or entered data a small d is visible. By clicking d the data can easily be converted into different number formats: hex (x), dec (d), octal (o), binary (b) and SI-spelling (p).

When entering a register value at the right hand side of the Write button it shall be taken care that the indicated format matches to the format of the entered value. Otherwise the wrong bits can be programmed.

Set

Via the drop-down menu the user is able to insert MSP commands himself or select one of the implemented MSP commands. Clicking the Set button after writing or selecting a command will send the command to the sensor. The response is then shown below the drop-down menu. For detailed information about all MSP commands, please refer to the Magnetic Sensor Programmer V1.x application note.

The following commands are implemented:

– Get Firmware Version reads out the firmware version of the TDK-MSP
– Get Bit Time reads out the current bit time for the Biphase communication protocol
– Set Bit Time sets the desired bit time for the Biphase communication protocol
– Measure VOUT measures the output voltage
– Measure VSUP measures the supply voltage
– Power-On-Reset switches the supply voltage off and on again
– Pull-up OFF/ON enables or disables the pull-up resistor (pull-up = 3.6 kΩ) of the programming device between output and supply voltage.

7. Measurement Tool

The **Measurement Tool** panel (see Fig. 7–33) can be accessed from the **Main** panel (see Fig. 2–1). This tool enables the user to stepwise read out certain RAM register values of the sensor's signal path, show the measurement result on a plot and save it to a text file. The created measurement data file can be used in the **Linearization Tool** (see Section 5).

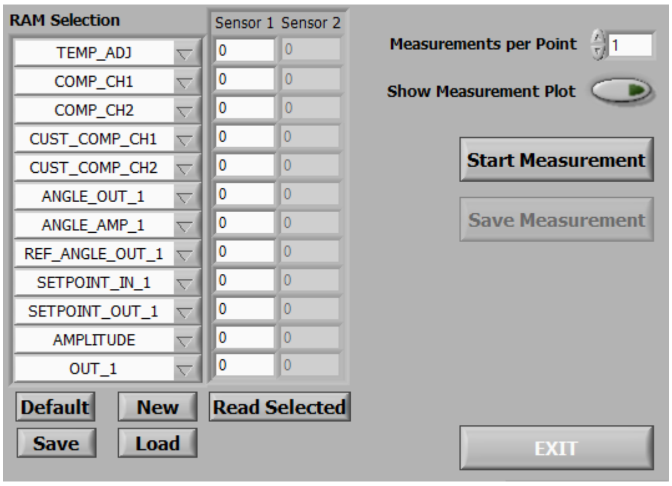

Fig. 7–33: Measurement Tool Panel with One Connected Sensor

The sensor selection menu on top allows the user to choose between the connected sensors for the measurements. Available and selected sensors are marked with a green circle. A sensor can be excluded from the measurements by clicking the green circle, which will turn grey.

The **Measurement Tool** panel offers the following functionality:

– RAM Selection for Measurement: The user is able to select the desired RAM registers which have to be included into the measurement via the drop-down menus. To deselect a RAM Selection the drop-down menu has to be empty. Then the indication field next to the drop-down menu is disabled. For selected RAM registers the indication fields are enabled corresponding to the sensor selection.

– The RAM Selection for Measurement can be saved to a text file by clicking the Save button, set to default by clicking the Default button. A new RAM selection can be selected after clicking the New button and with the Load button it is possible to load an already saved RAM Selection Measurement file into the RAM Selection.

– The SETPOINT_IN_1 value can be used in the **Linearization Tool** (see Section 5). The other parameters can be used for analyzing the input field components.

– Measurements per Point: This number defines how often the selected RAM register values are read out before averaging during a measurement sequence or a single read-out. This is helpful to avoid noise errors during the measurement and leads to more reliable results whereas a higher number (e.g. 10) of measurement points also increases the measurement time.

– Read Selected: With this button a single read-out of the selected RAM registers will be triggered. The number of Measurements per Point determines how often each RAM register value is read out. The averaged results are displayed in the corresponding fields in the middle of the RAM selection area example. This function can be used to determine if ANGLE_OUT_1 is increasing or decreasing, when moving the application from the start position towards the end position, which is a required information for a calibration of the sensor (see Section 4).

– Show Measurement Plot: This option displays the Measurement Plot. During the measurement procedure the plot shows the activated RAM register values (see Fig. 7–34).

– Start Measurement: This button starts the measurement procedure. The user is asked to move the application stepwise through the application range and to confirm each position, which triggers several read-outs (defined by the number of measurements per point) of the selected RAM registers. Instead of confirming the next position the user can also decide to stop the measurement procedure or to repeat the last measurement point. If Show Measurement Plot has been enabled the measurement plot shows the selected RAM register values, which have been read out at every position and updates the plot after each confirmed position. For a following linearization of the output characteristic with the **Linearization Tool** (see Section 5) the first and last measurement point should match the first and second calibration point used during two-point calibration (see Section 4.2). The total number of positions for the measurement can be arbitrary.

– Save Measurement: After the measurement procedure it is possible to save the measured values into a text file by clicking the Save Measurement button. The user is asked to select a text file for the storage of the measured data. The text file containing the measurement results is created properly formatted for the **Linearization Tool** (see Section 5).

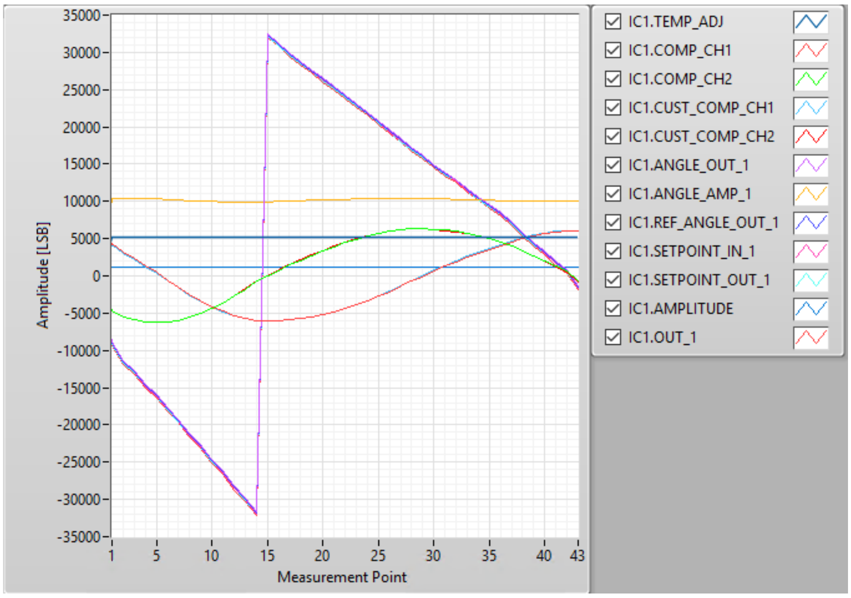

Fig. 7–34: Activated Measurement Plot During a Typical Measurement Procedure

8. Application Note History

1. HAL/HAR 3970 Programming Environment Description, Oct. 12, 2020; APN000172_001EN. First release of the application note.