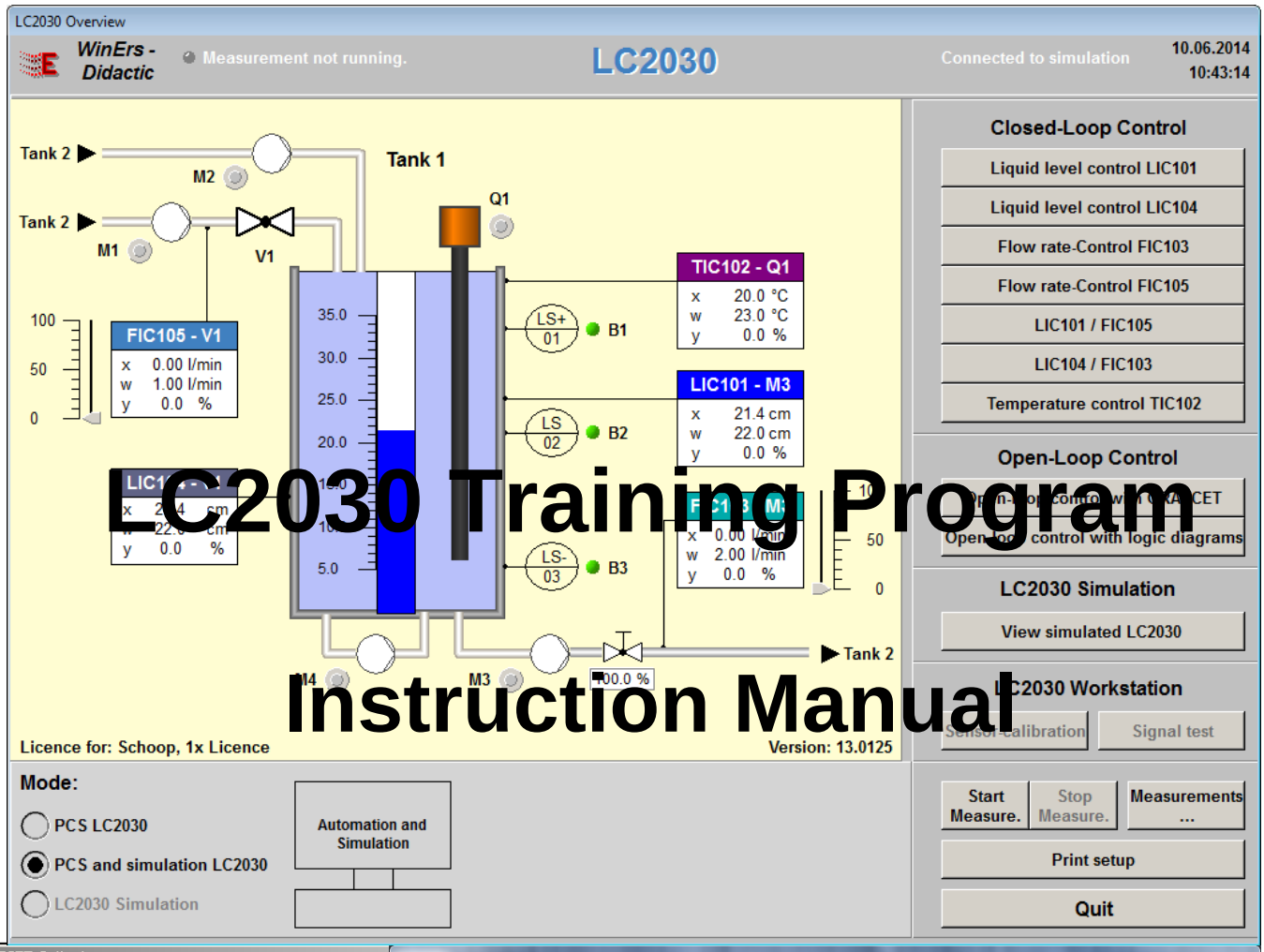
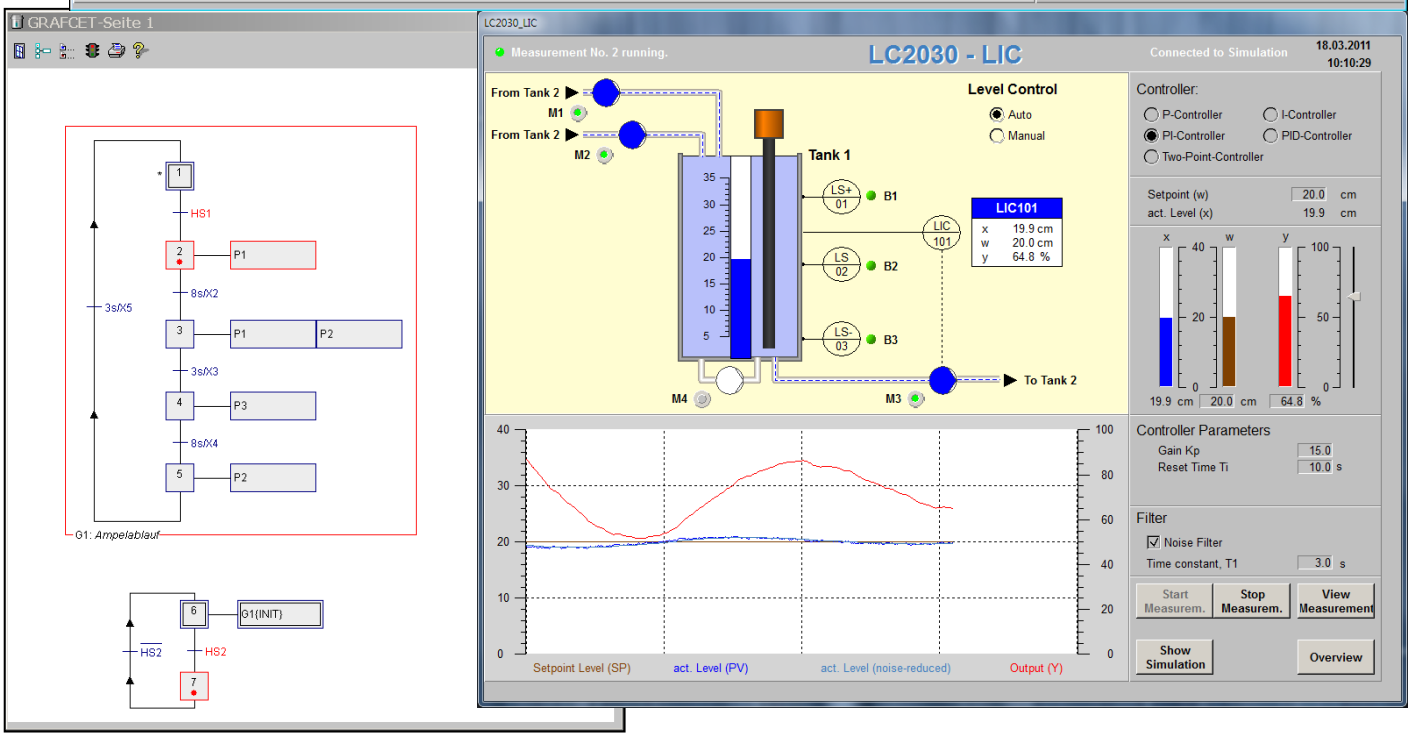
LC2030 Training

WinErs Didactic





2nd Edition 06/2014

Dr.-Ing. Schoop GmbH
www.schoop.de

1 Introduction

With the LC2030 Training program, you can edit and control tasks in the field of open-loop and closed-loop control technology. You can implement open-loop and sequence controls using GRAFCET plans or logic diagrams. Liquid level, flow rate and temperature control are available in closed-loop control engineering. They can be controlled with standard controllers (P, I, PI and PID) and two-position controllers.

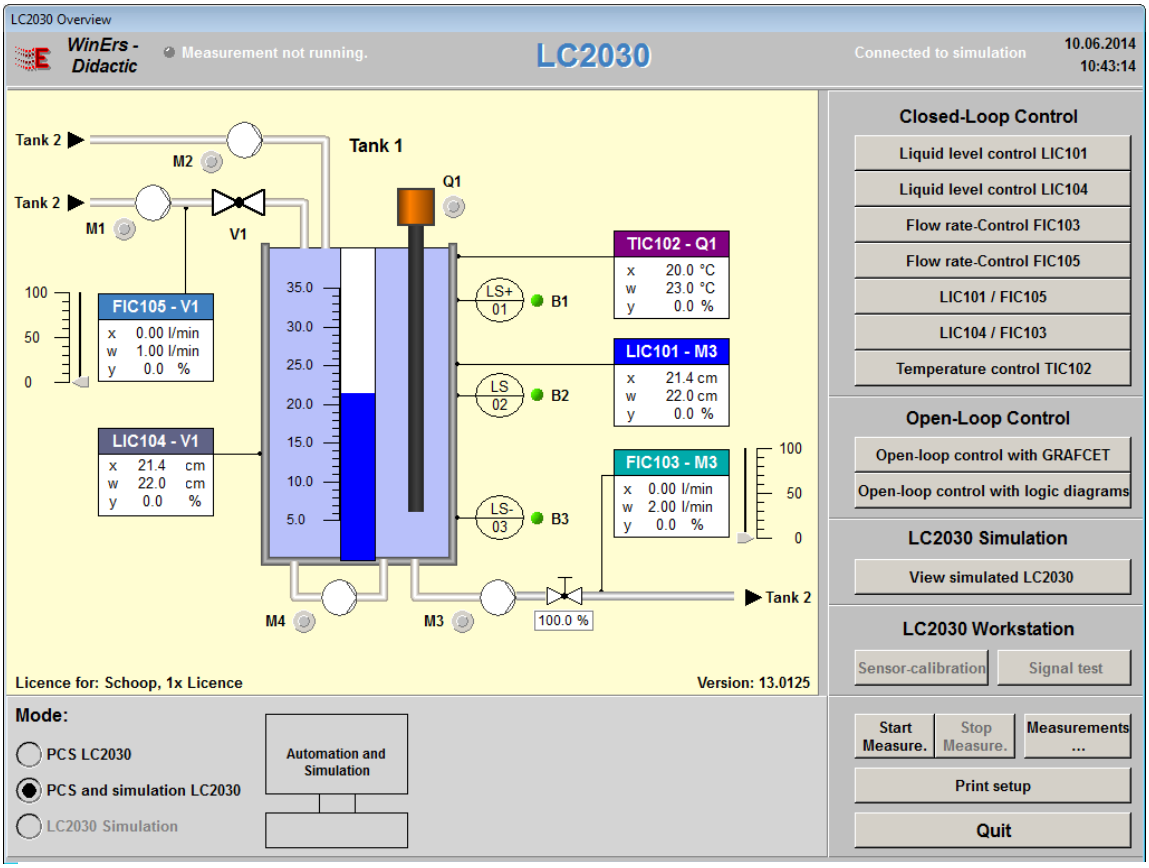


Fig. 1: Overview of LC2030 Training program

LC2030 Training allows you to work with either the real LC2030 workstation or with a simulated system.

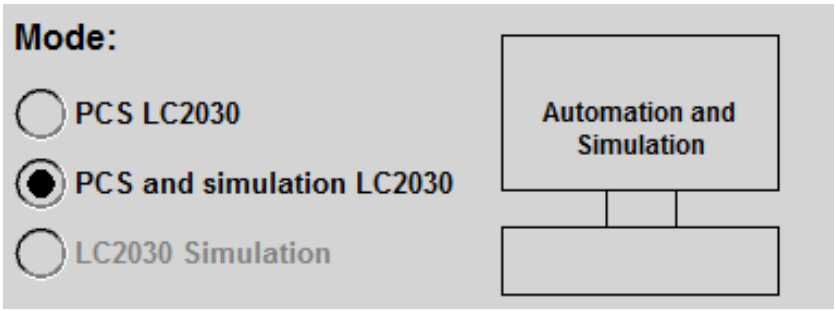


Fig. 2: Select mode

The signals must be connected via I/O box to the plant if the connection to the LC2030 workstation is selected.
All experiments, including closed-loop, open-loop and sequence controls with GRAFCET or logic diagrams, can be performed both with the real system and the simulated system.

In the overview window of the LC2030 Training program, you are able to choose which task you would like to edit.

- Liquid level control via:
    - Outflow (adjustable pump)
    - Inflow (adjustable valve)
- Flow rate control via:
    - Outflow (adjustable pump)
    - Inflow (adjustable valve)
- Combined liquid level and flow rate control via:
    - Outflow (adjustable pump)
    - Inflow (adjustable valve)
- Temperature Control

or

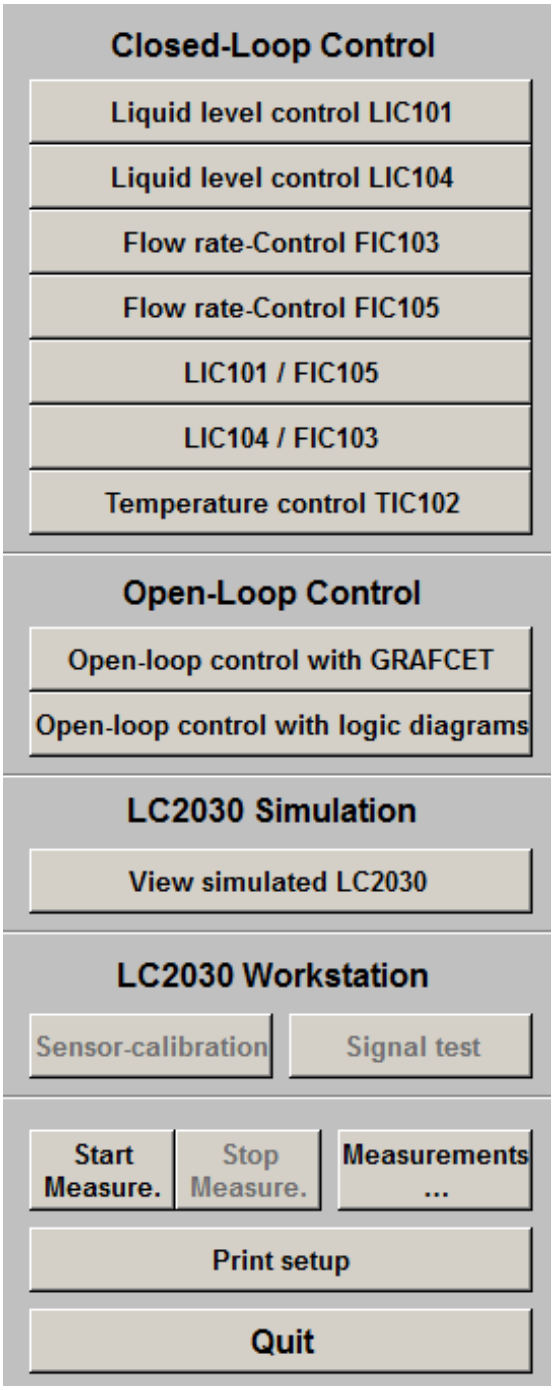
- Open-loop and sequence controls using GRAFCET
- Open-loop and sequence controls using logic diagrams

If you have chosen **PCS LC2030**, meaning the program is connected to a real plant, the sensor calibration menu item is enabled for liquid level, flow rate and temperature.

If you have chosen **PCS and simulation**, meaning the simulated system is selected, you can switch to *View LC2030 Simulation* where you can adjust the parameters for the simulated system.

Clicking *Start Measurement* activates the data logging function for all signals.

The signal sequence can then be examined via *Measurements*….



2 Closed-Loop Control Engineering

The following controlled systems are available in closed-loop control engineering: liquid level, flow rate and temperature.

2.1 Liquid Level Control

The liquid level can be controlled either via outflow using the adjustable pump M3 (*Liquid Level Control LIC101*) or via inflow using the adjustable valve V1 (*Liquid Level Control LIC104*). The control type can be selected in the overview window.

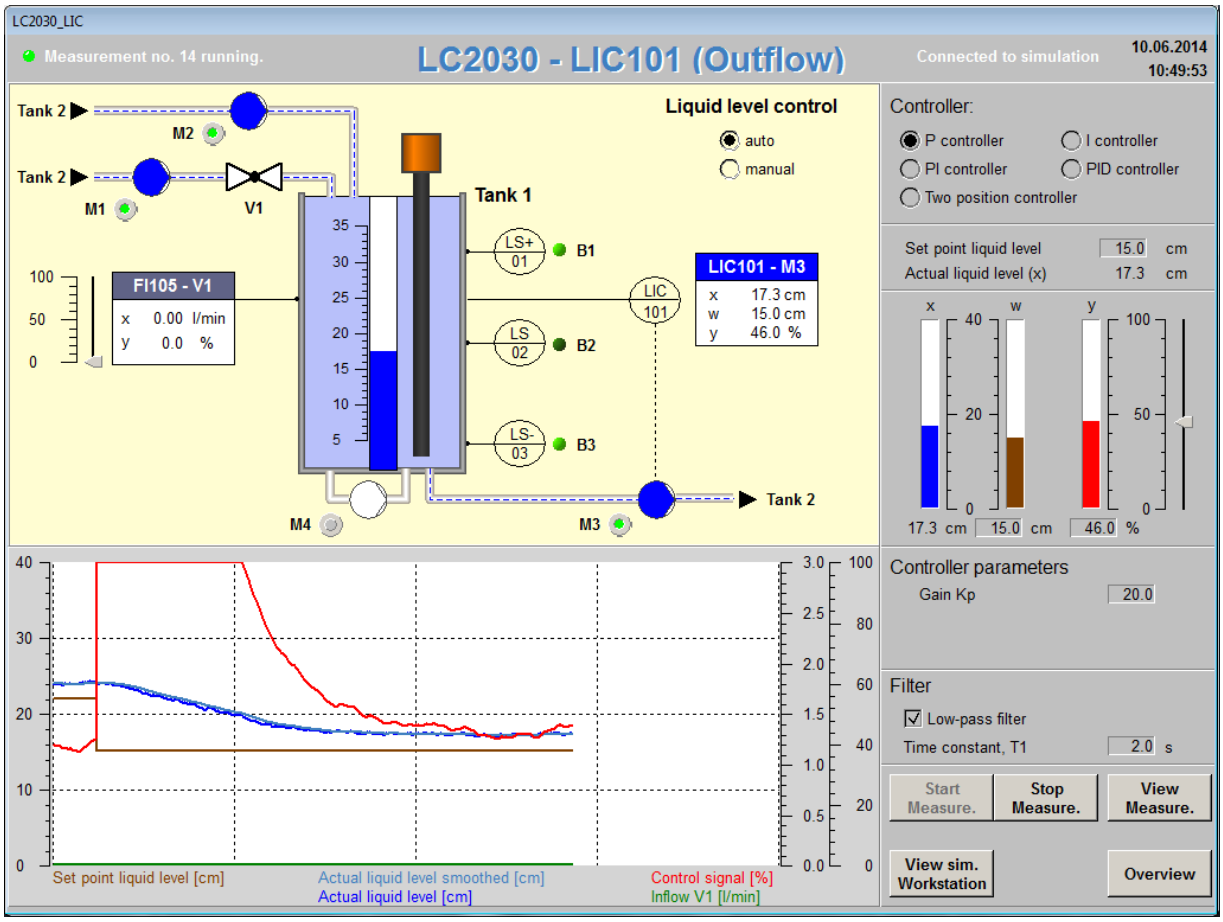


Fig. 3: Liquid level control via outflow

To control the liquid level, pumps *M1* and/or *M2* and *M3* need to be switched on. Press the switch next to the pump symbol to turn on the pump.

Next, select whether the level will be controlled automatically or manually by selecting the appropriate mode.

When *Manual* mode is enabled, you can alter the output signal *y* with the slide controller or by entering an output value numerically.

When *Auto* mode is enabled, you can select *P, I, PI, PID* or *Two-position controller.* Then enter a setpoint and the different controller parameters. The controller parameters that are displayed depend on the chosen controller type (i.e. gain, reset time and derivative time, hysteresis).

To reduce the noise of the level signal, a low pass filter can be activated. The time constant “T1” controls the amount of noise reduction.

Via the *Start Measurement* button, the data logging function for all signals (analog and binary) is activated. The signal sequence can then be examined via the *Measurements*… button. Here you can perform numerical, graphical and statistical analysis of the signal data.

A detailed description of the measurement evaluation can be found in Chapter 2.4.

The *View sim. Workstation* button is activated if you are working with the simulated system. Here you can monitor the state of the simulated workstation and view parameter settings (e.g. reset to the original state or adjust manual valves). See Chapter 2.5 for furthur details.

To return to the main menu click *Overview*.

2.2 Flow Rate Control

The flow rate can either be controlled via outflow using the adjustable pump M3 (*Flow Rate Control FIC103*) or via inflow using the adjustable valve V1 (*Flow Rate Control FIC105*). The control type can be selected in the overview window.

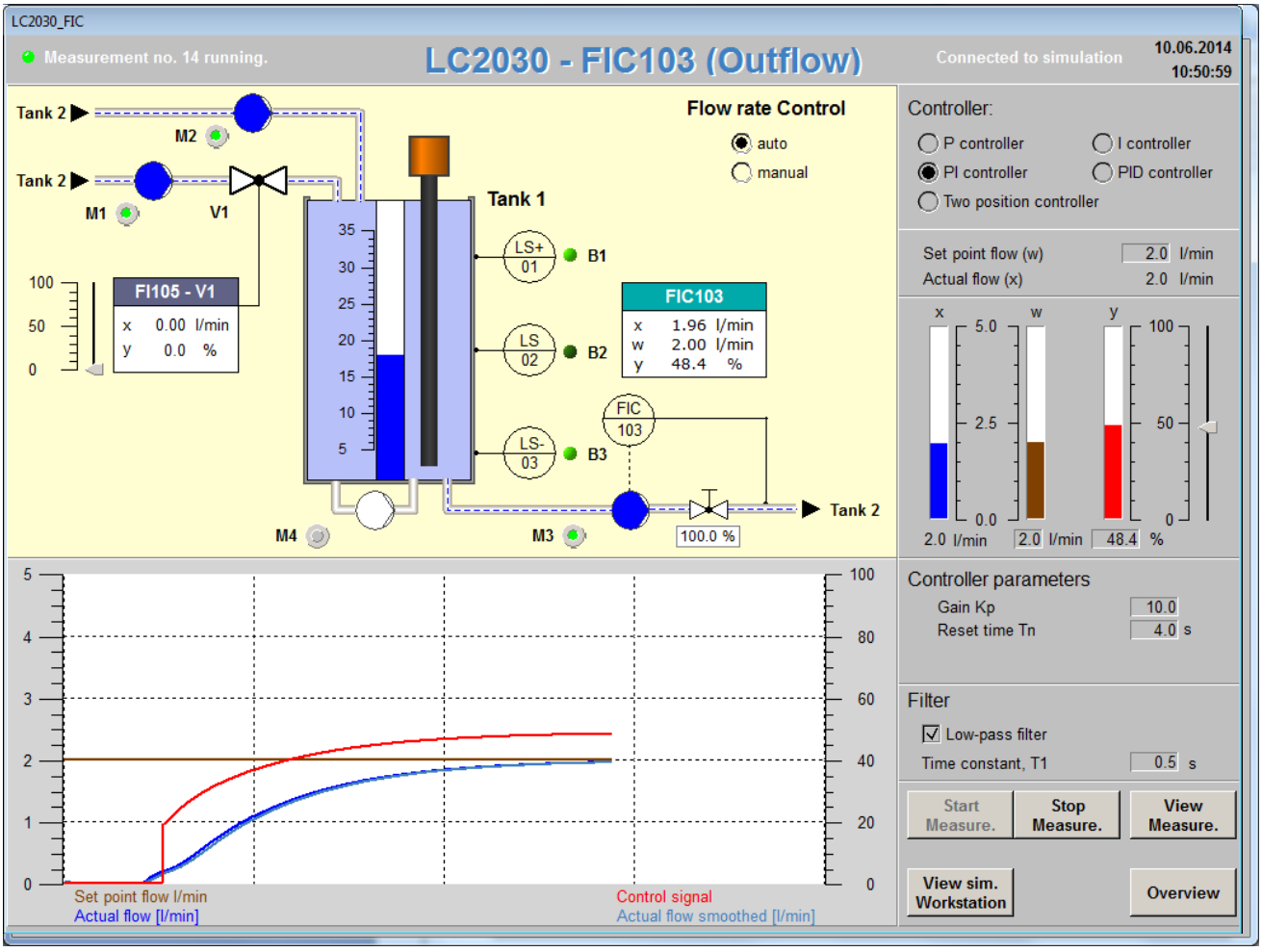


Fig. 4: Flow rate control

To control the flow rate, the pumps *M1* and/or *M2* and *M3* need to be switched on. Press the switch next to the pump symbol to turn on the pump.

Next, select whether the level will be controlled automatically or manually by selecting the appropriate mode.

When *Manual* mode is enabled, you can alter the output signal *y* with the slide controller or you can enter an output value numerically.

When *Auto* mode is enabled, you can select *P, I, PI, PID* or *Two-position controller.* Then enter a setpoint and the different controller parameters. The controller parameters that are displayed depend on the chosen controller type (i.e. gain, reset time and derivative time, hysteresis).

To reduce the noise of the level signal, a low pass filter can be activated. The time constant "T1" controls the amount of noise reduction.

Via the *Start Measurement* button, the data logging function for all signals (analog and binary) is activated. The signal sequence can then be examined via the *Measurements*… button. Here you can perform numerical, graphical and statistical analysis of the signal data.

A detailed description of the measurement evaluation can be found in Chapter 2.4.

The *View sim. Workstation* button is activated if you are working with the simulated system. Here you can monitor the state of the simulated workstation and view parameter settings (e.g. reset to the original state or adjust manual valves). See Chapter 2.5 for furthur details.

To return to the main menu click *Overview*.

2.3 Temperature Control

The temperature control is started with the *Temperature Control TIC102* button.

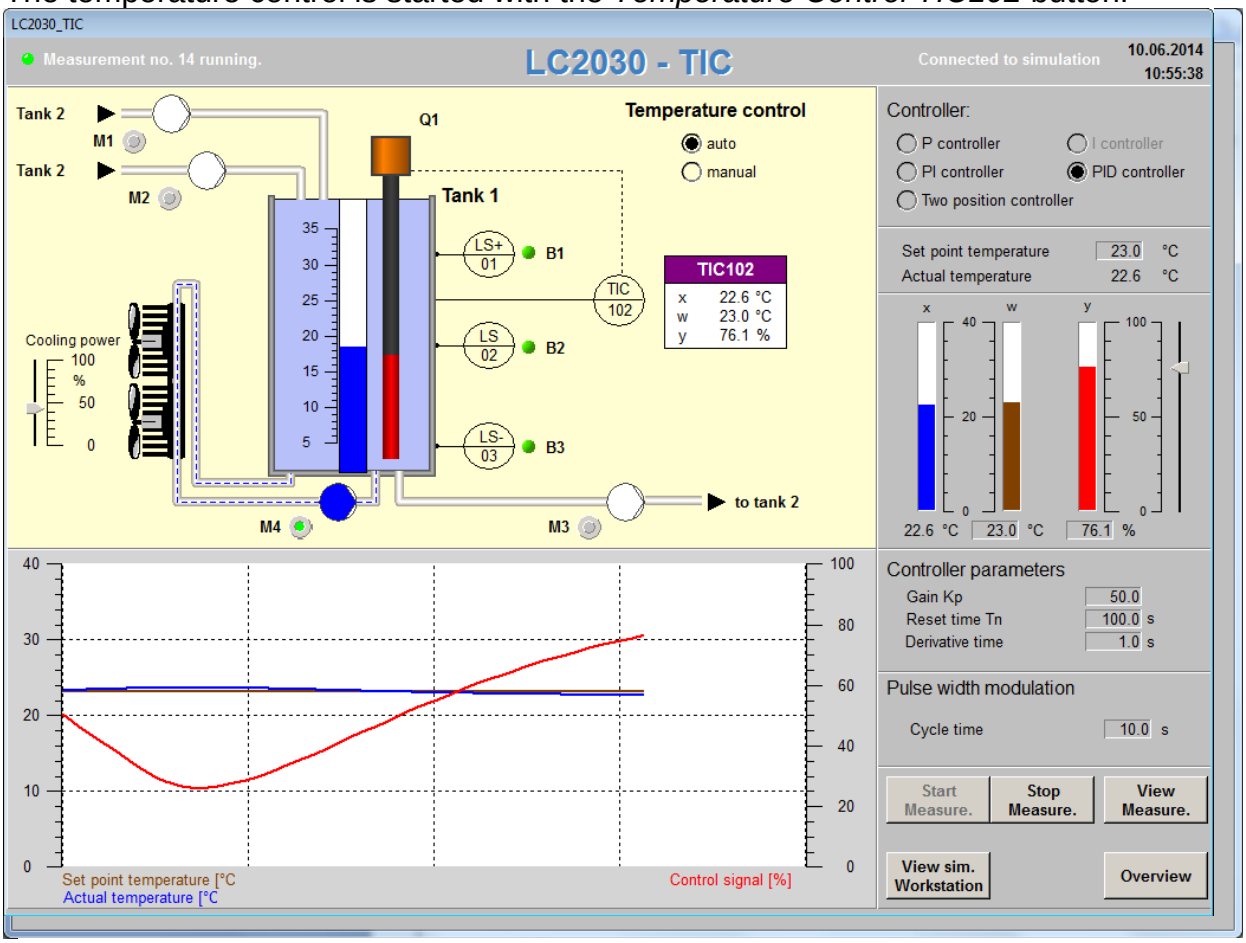


Fig. 5: Temperature control

The pumps *M1*, *M2* and *M3* should be switched off for temperature control. Switch on the pump *M4* for thorough temperature distribution in the tank.

Temperature control is realised through the use of a heating element that can be switched on or off. For quasi-analog control, a pulse-width-modulation (PWM) control is implemented. The PWM cycle time is adjustable.

In pulse-width modulated control, the controller switches the heating element on or off for a particular amount time.

The controller output determines the length of the heating period within the PWM interval (cycle time). You can choose to control the temperature in automatic or manual mode.

When *Manual* mode is enabled, you can alter the output signal *y* with the slide controller or you can enter an output value numerically.

When *Auto* mode is enabled, you can select *P, I, PI, PID* or *Two-position controller.* Then enter a setpoint and the different controller parameters. The controller parameters that are displayed depend on the chosen controller type (i.e. gain, reset time and derivative time, hysteresis).

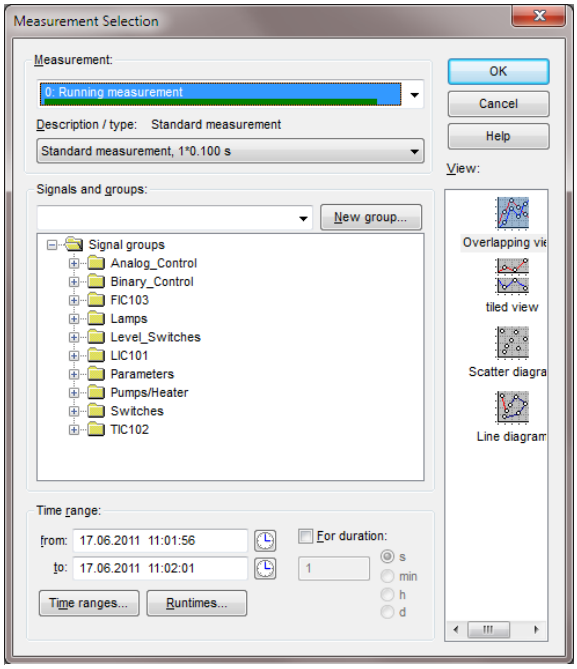
Via the *Start Measurement* button, the data logging function for all signals (analog and binary) is activated. The signal sequence can then be examined via the *Measurements*… button. Here you can perform numerical, graphical and statistical analysis of the signal data.

A detailed description of the measurement evaluation can be found in Chapter 2.4.

The *View sim. Workstation* button is activated if you are working with the simulated system. Here you can monitor the state of the simulated workstation and view parameter settings (e.g. reset to the original state or adjust manual valves). See Chapter 2.5 for furthur details.

When working with the simulated workstation, *M4* influences the temperature of the vessel. It is as if a cooling unit is connected to the circulation pump. This has the advantage that the water in the tank cools down quickly. To return to the main menu click *Overview*.

2.4 Measurements View



The signal sequence can be examined via the *View Measurement* button. Here you can perform numerical, graphical and statistical analysis of the signal data.

The window in Fig. 6 shows up when *View Measurements* is clicked if data logging is deactivated. You can select a measurement number and a group of signals.

When data logging is activated, the window in Fig. 7 shows up. Here you can perform numerical, graphical and statistical analysis of the signal data.

Fig. 6: Select measurement dialog

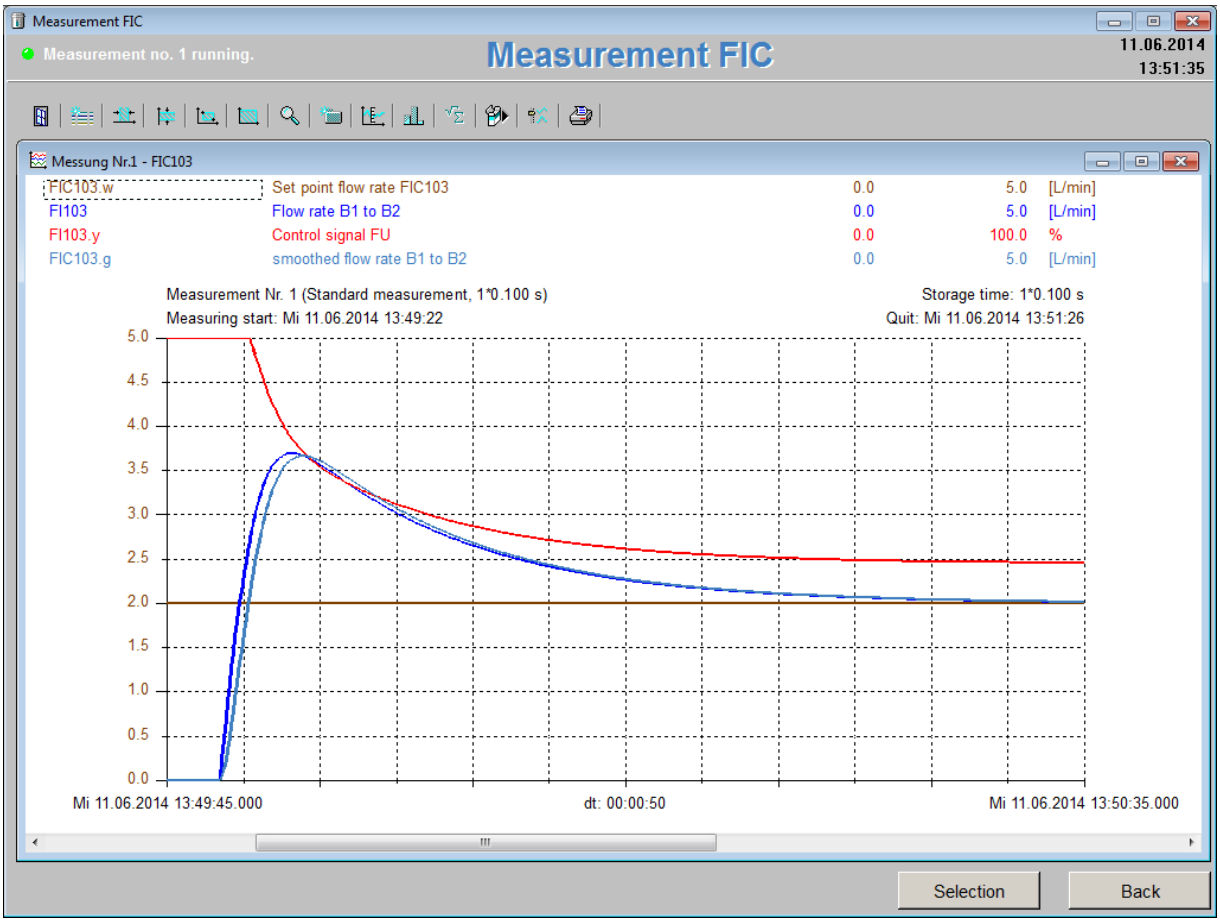


Fig. 6: Measurement view

Via *Selection* the window shown in Fig. opens.

The signal sequence is displayed in Fig. 7. Various operations can be performed by clicking the buttons in the header:

 Add or remove signals

 Change time range numerically

 Change display range numerically

 Select time and display range by click-and-drag

 Restore the original view

 Search

 Apply current settings to group

 Turn ruler on or off

 Statistical analysis

 Statistical evaluation

 Export data displayed in window to a text file (TXT or CSV)

 Adjust setting for representation of measured values

 Print the active subwindow

 Help for the active subwindow

By clicking on a signal name, you can adjust the scaling of the y-axis for analog signals. By double-clicking the signal name, you can adjust/hide the signal squence. Using the “+” key on the numeric keypad toggles the display type for all signals.

Left clicking on the chart displays the value and time stamp of the active signal at the position of the cursor. By holding the left mouse button and moving the cursor you can measure the chart time and value range to determine the corresponding slope.

Click *Return* to go back to the previous window.

2.5 View of the Simulated Workstation

If you work with the simulated system, the *Simulated System* button is activated. Here you can monitor the state of the simulated workstation and view parameter settings. You can adjust the switches and pushbuttons, set valves manually, set the environmental and inlet temperature or reset the unit to its original state via the parameter page. Modifying the tank dimensions and the logic of the switches and pushbuttons can also be done on the parameter page.

Please note: It is always possible to reset the simulation to the original LC2030 workstation parameters.

Click *Back* and *Close* to return to the main window.

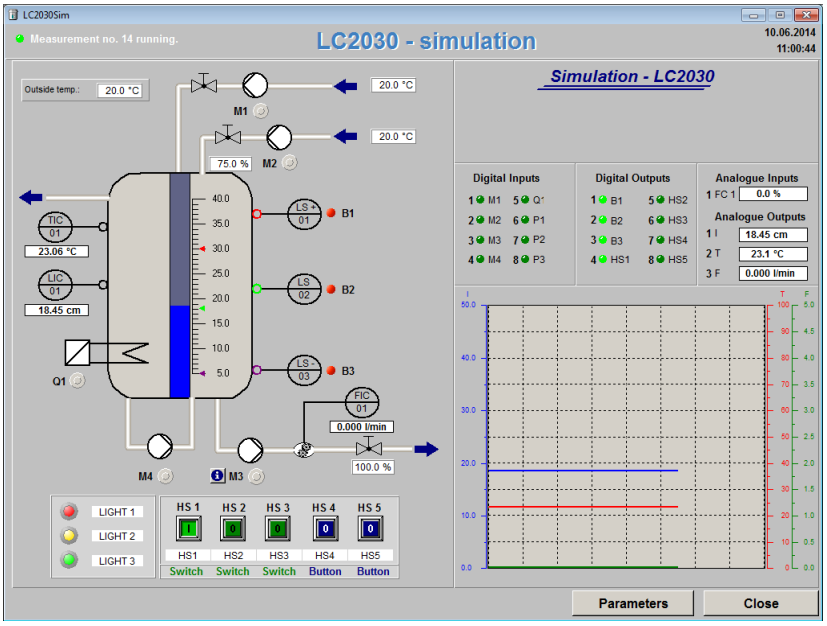


Fig. 7: Simulated workstation

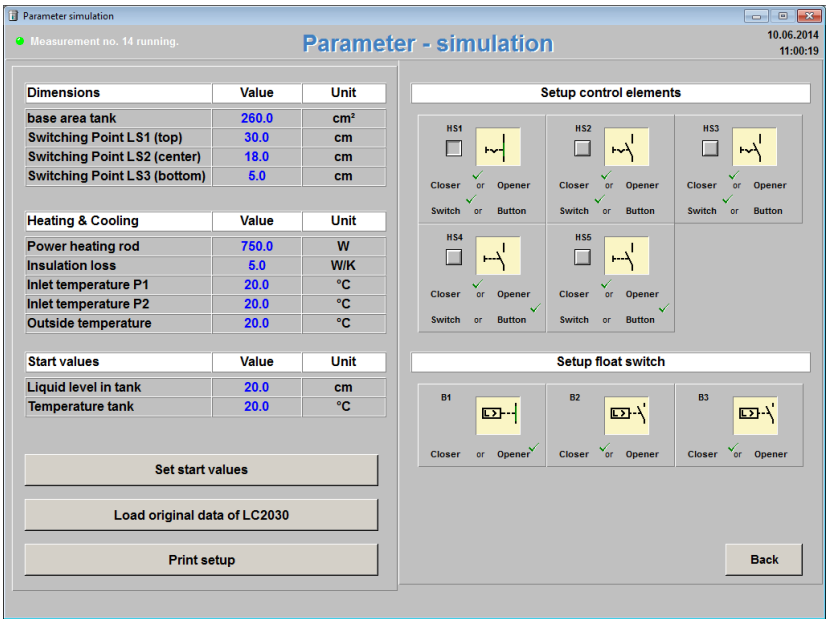


Fig. 8: Parameter page simulated workstation

3 Open-Loop and Sequence Control with GRAFCET

In the LC2030 Training program it is possible to develop open-loop and sequence controls using GRAFCET charts.

On the main page of LC2030 training, click *Open-Loop Control with GRAFCET.* The following window will open:

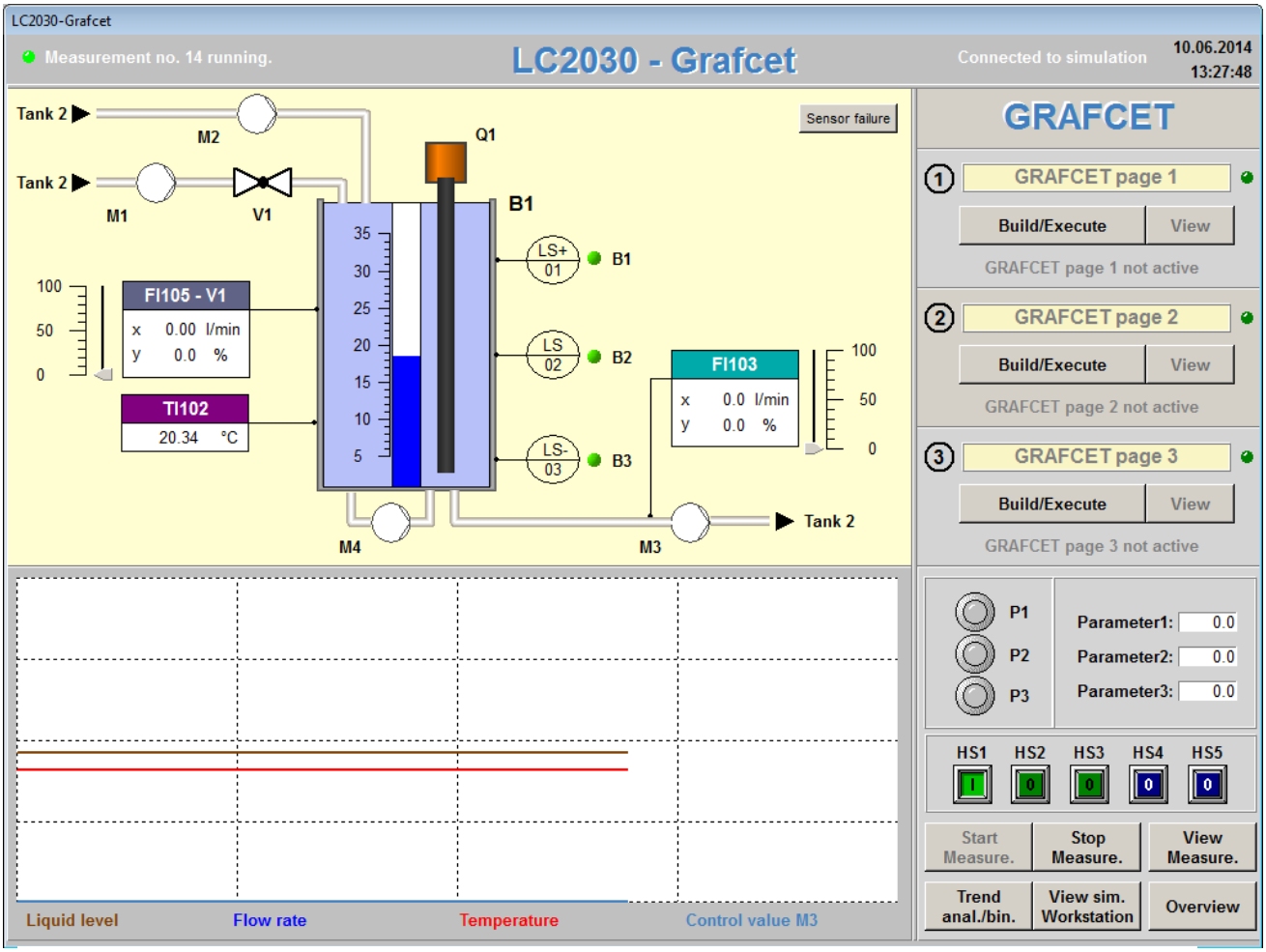


Fig. 9: Develop GRAFCET charts for sequence and open-loop control

Up to three GRAFCET charts can be edited and run simultaneously. Via *Edit/Run* you can edit, modify and test/monitor the GRAFCET structure. To monitor an active structure, press *View*.

Comments can be inserted in the yellow boxes next to the appropriate numbers to describe the function of the GRAFCET pages.

The LED next to the comment box or the text indicates whether the page is activated or not. If a GRAFCET page is active and running, it can be edited or deactivated by pressing the *Edit/Run* button a second time.

3.1 GRAFCET-Editor

By clicking *Edit/Run,* the GRAFCET-Editor window opens. The window will appear empty with only the tool box visible if no GRAFCET charts were created.

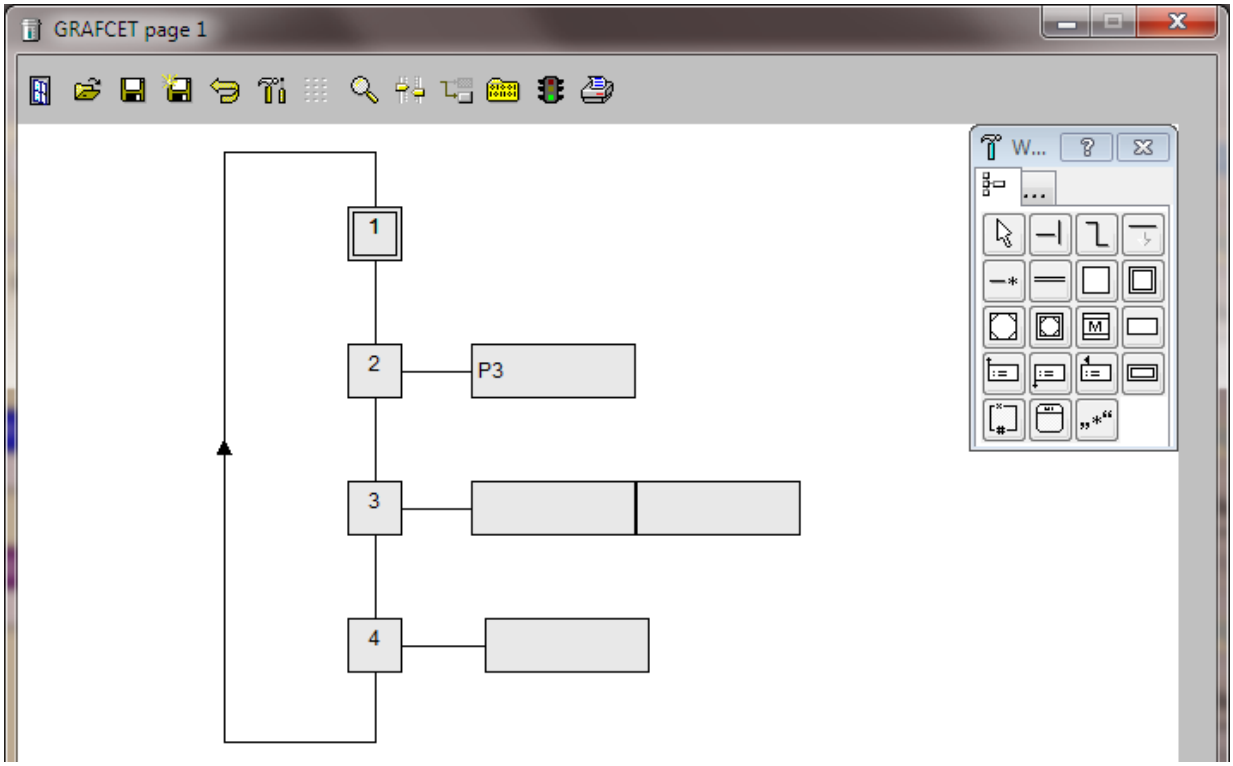
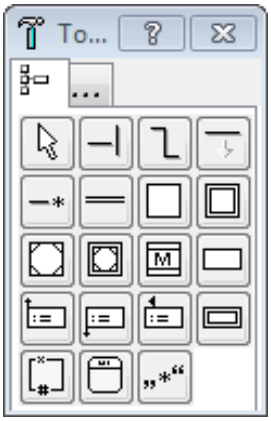


Fig. 10: GRAFCET-Editor



GRAFCET charts are created and modified within the GRAFCET-Editor by using the elements in the tool box.

Select any GRAFCET item in the tool box by left clicking on it. Click in the editor to position one or more elements. Use the Esc key or right click to reset the current tool to the mouse pointer.

The elements on the page are connected by directed lines or directed polygons.

To insert an element to a GRAFCET page, select it from the tool box and click the desired position within the GRAFCET page. Elements with inputs or outputs may be placed at the beginning, middle or end of directed lines. The directed lines will automatically be shortened or split if necessary.

Create directed lines or polygons by dragging with your mouse after selecting the appropriate tool from the tool box.

All inserted elements will be aligned to the current input grid.

Left click on an element to select it. By simultaneously pressing the Control key (Ctrl) and clicking other elements, multiple elements are selected at once. By first selecting one element and clicking on another while holding down the Shift key, all elements lying in this imaginary rectangle are selected. The selected elements can be moved by holding down the mouse button and dragging the elements into position.

By left clicking in an empty part of the screen and dragging the mouse while holding down the button, you can select any elements that are fully enclosed in the rectangle that is created.

All mouse operations can be cancelled by clicking the right mouse button or hitting the Esc key.

Press the Tab or Shift+Tab keys to go through the elements on a GRAFCET page. The settings dialog for the selected element appears by pressing the Enter key or double-clicking the marked elements. Marked items can be moved using the cursor keys. Please note that this can interfere with the alignment of the grid.

When an element from the tool box is selected, it is added by clicking the GRAFCET page. It is possible to position and modify the form using the mouse if the object is resizable.

You can draw connections directly from one element to another if the auto-routing function is enabled (note the tool bar information below). The software will then automatically find a connection between the elements. The auto-routing function only works with the directed line tool. When the directed polygon tool is selected the grid points need to be set manually.

All elements added to the GRAFCET editor will be aligned to a specified grid. You can view the grid or modify the spacing by selecting the grid function in the tool bar at top of the GRAFCET window.

When moving elements, you can choose whether the connections are to be tracked (auto-routing) or not (adjustable on the tool bar). The auto-routing always uses a fixed eight pixel grid, regardless of the alignment grid.

When auto-routing is activated, you can temporarily suppress this function by holding the Alt key when moving elements.

Some elements, such as steps, can only be resized to a certain limit. Other elements, such as comments, are fully resizable.

By holding the F6 key, you can temporarily show the operating direction of the directed lines/polygons. Holding the F7 key will display a crosshair. This may be used to check the alignment of the elements.

Use <Ctrl>+Insert and <Shift>+Insert to copy and paste elements. You can get information on an item by selecting it and pressing the F1 key.

Double-clicking on an element opens the configuration dialog for the item. Here you can select signals, conditions or GRAFCET macros.

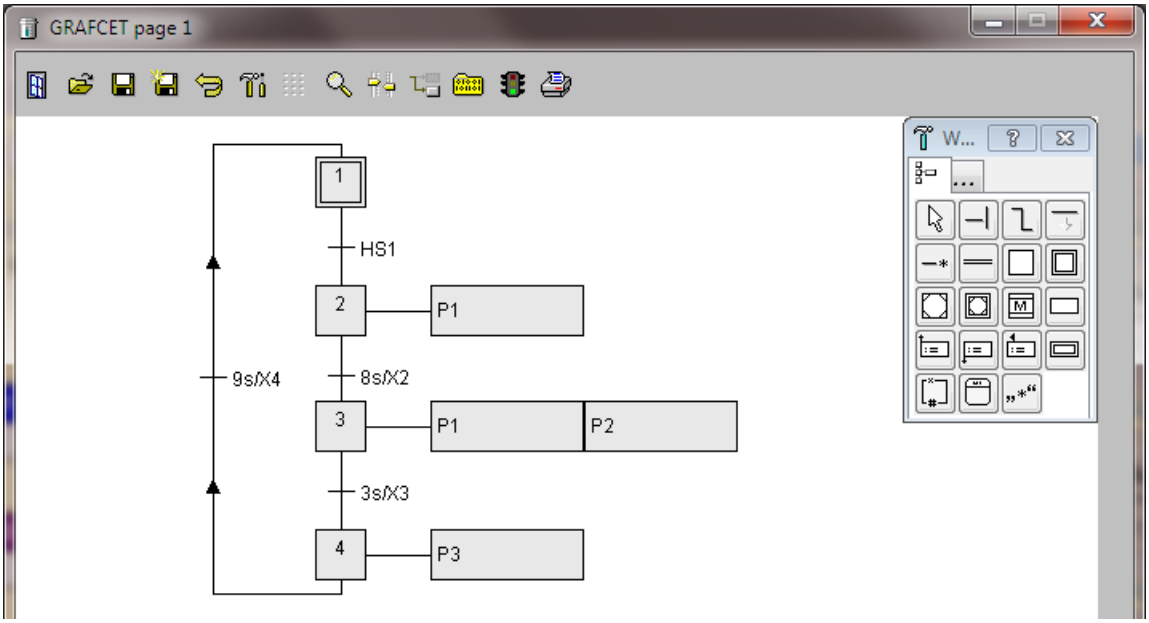


Fig. 11: Example of a GRAFCET structure in the GRAFCET editor

The following functions can be found in the tool bar above the GRAFCET editor:

From left to right:

- Close active subwindow
- Open – insert a file
- Save the current version
- Save all or the selected items to a file
- Restore to last saved version
- Fade in tool box
- Modify grid alignment settings
- General settings for the active window
- Toggle auto-routing on/off
- Compile GRAFCET page
- Activate GRAFCET page
- Print the active subwindow
- Help for the active subwindow

3.2 Compile / Activate GRAFCET Charts

Create the GRAFCET chart using the previous information.

Click the index card symbol to check whether the GRAFCET chart is syntactically correct.

Click the traffic light symbol to compile and activate the GRAFCET chart. The chart will only be activated if it is syntactically correct.

If the GRAFCET chart was not created properly, an error message is displayed and you will be asked if the compilation messages should be shown. By clicking on the error, the erroneous part in the GRAFCET chart is highlighted.

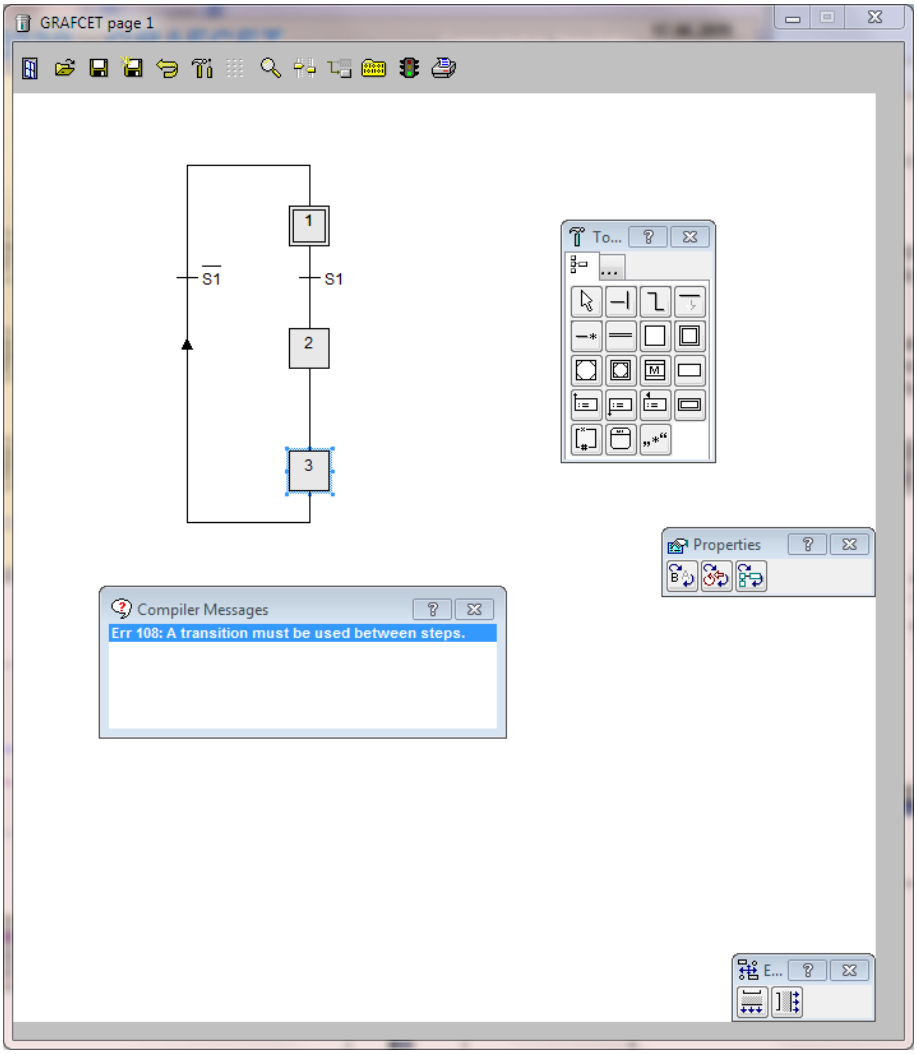


Fig. 12: Error messages after compilation / activation of GRAFCET chart

If the GRAFCET chart was created properly, it can be activated by clicking the traffic light symbol. All initial steps will be activated instantly.

The following window (GRAFCET view) will appear. Here you can observe the sequence of the GRAFCET chart.

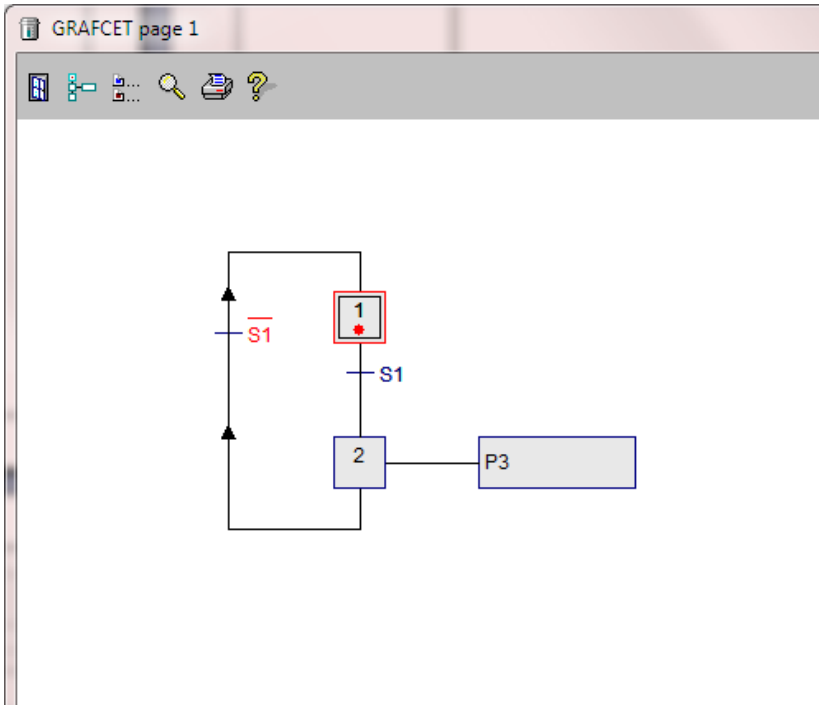


Fig. 13: Activated GRAFCET chart

The initial step in the GRAFCET chart shown in Fig. 14 is active. When the signal *S1* changes from Low to High, the condition of the transition is fulfilled and the following step will be activated. Hence, the signal *L1* will be set (due to the *continuous effective action*).

It is possible to specifically initiate, activate or reset certain steps. To perform the initialisation, click the initialisation symbol in the tool bar.

The following dialog is shown:

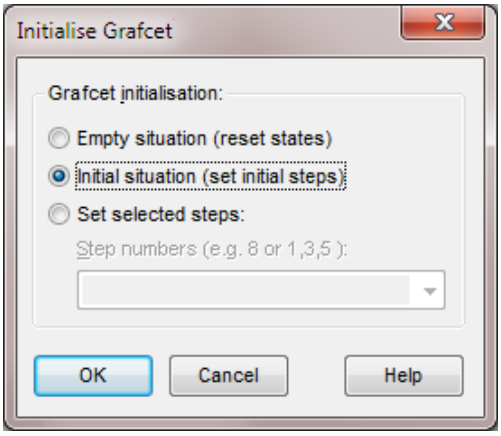


Fig. 14: Initialise GRAFCET dialog

These options are available:

- Empty situation (reset and stop the whole structure)
- Initial situation (all initial steps will be activated)
- Set selected steps (enter the step numbers you want to activate)

3.3 GRAFCET Elements

You can find all the elements that are necessary to create a GRAFCET chart in the tool box of the GRAFCET editor

When a GRAFCET item is selected from the tool box, the mouse cursor transforms into a symbol resembling its function.

By clicking the arrow icon, pressing the ESC key or right clicking the mouse, the current tool selection will be reset to pointer mode.

To keep the tool box clearly structured, not all tools are displayed simultaneously. The two tabs shown in the top of the box represent the tools for:

Standard GRAFCET elements, provided in DIN EN 60848.

Extendend GRAFCET elements, not defined in DIN EN 60848.

Standard GRAFCET Elements (as found in DIN EN 60848)

Directed line (connecting line) for directed links

Polygon line (multiple point connecting line) for directed links

Link label for linking to other structures

Transition condition

Synchronisation

Step

Initial step

Enclosing step

Initial enclosing step

Macro step

 Continuous action

 Action on activation

 Action on deactivation

 Action on event

 Constraining command

 Enclosing structure (Sub-GRAFCET)

 Macro structure (Sub-GRAFCET)

 Comment

Extended GRAFCET elements are not part of DIN EN 60848, hence they are not specified further here.

4 Open-Loop Control with Logic Diagrams

In the LC2030 Training program it is possible to develop open-loop and sequence controls using GRAFCET charts or logic diagrams.

On the main page of the LC2030 program, click *Open-Loop Control with Logic Diagrams.* The following window will open:

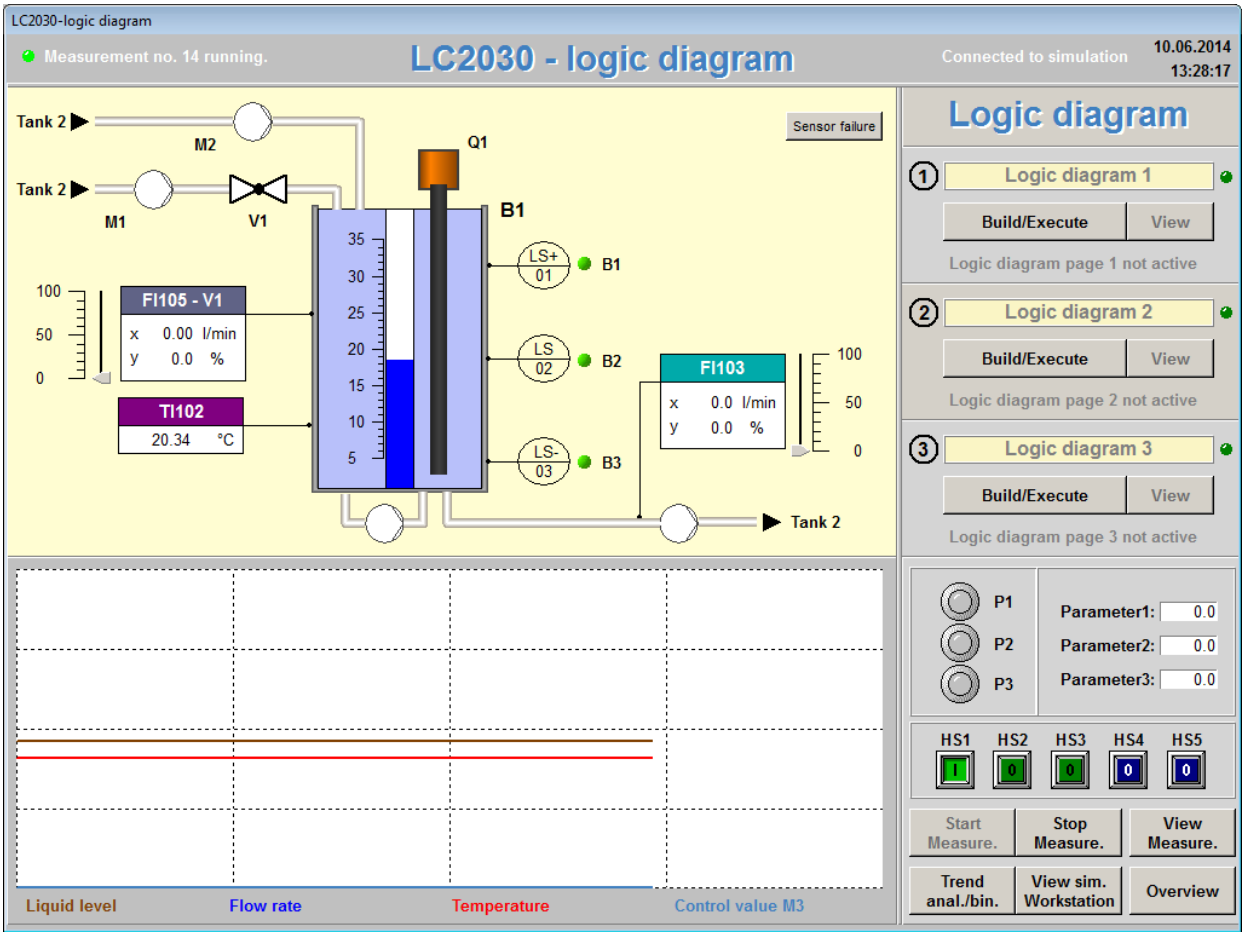


Fig. 15: Open-loop control with logic diagrams

Up to three logic diagrams can be edited and run simultaneously. They can be used to control the real or simulated LC2030 workstation. Via *Edit/Run* you can edit, modify and test/monitor the logic diagram structure. To monitor an activated structure press *View*.

You can insert comments in the yellow boxes next to the numbers to describe the function of your logic diagram pages.

The LED next to the comment box or the text indicates whether or not the page is active. If a GRAFCET page is active and running, it can be edited or deactivated by pressing the *Edit/Run* button a second time.

4.1 Logic Diagram Editor

By clicking *Edit/Run* the logic diagram editor window opens. The window will appear empty with only the tool box visible if no logic diagram charts were created.

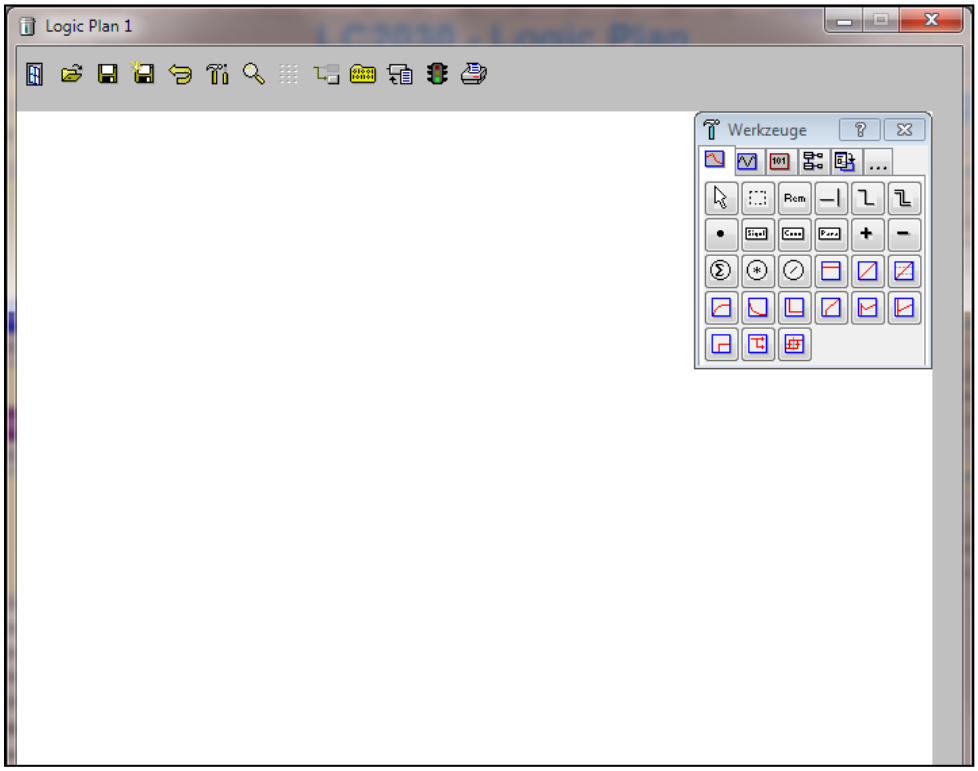


Fig. 16: Logic diagram editor

Within the logic diagram-editor logic diagrams are created and modified by using the elements in the tool box.

You can select any item in the tool box by left clicking it. Position the element by clicking in an open space in the editor. Use the Esc key or right click with the mouse to reset the current tool to the standard mouse pointer.

The elements on the page are connected by directed lines or directed polygons.

To insert an element into a logic diagram, select it from the tool box and click the desired position within the logic diagram page. Elements with inputs or outputs may be placed at the beginning, middle or end of directed lines. The directed lines will automatically be shortened or split if necessary.

Create directed lines or polygons by dragging with your mouse after selecting the appropriate tool from the tool box.

All inserted elements will be aligned to the current input grid.

Left click on any element to select it. By simultaneously pressing the Control key (Ctrl) and clicking other elements, multiple elements are selected at once. By first selecting one element and clicking on another while holding down the Shift key, all elements lying in this imaginary rectangle are selected. The selected elements can be moved by holding down the mouse button and dragging the elements into position.

By left clicking in an empty part of the screen and dragging the mouse while holding down the button, you can select any elements that are fully enclosed in the rectangle that is created.

All mouse operations can be cancelled by clicking the right mouse button or hitting the Esc key.

Press the Tab or Shift+Tab keys to go through the elements on a logic diagram page. The settings dialog for the selected element appears by pressing the Enter key or double-clicking the marked elements. Marked items can be moved using the cursor keys. Please note that this can interfere with the alignment to the grid.

When an element from the tool box is selected, it is added by clicking the logic diagram page. It is possible to arrange and position it by modifying the form using the mouse if the object is resizable.

You can draw connections directly from one element to another if the auto-routing function is enabled (note the tool bar information below). The software will then automatically find a connection between the elements. The auto-routing function only works with the directed line tool. When the directed polygon tool is selected the grid points need to be set manually.

All elements added to the logic diagram editor will be aligned to a specified grid. You can view the grid or modify the spacing by selecting the grid function in the tool bar at top of the logic diagram window.

When moving elements, you can choose whether the connections are to be tracked (auto-routing) or not (adjustable on the tool bar). The auto-routing always uses a fixed eight pixel grid, regardless of the alignment grid.

You can temporarily suppress this function by holding the Alt key when moving elements as long as auto-routing is activated.

Some elements, such as steps, can only be resized to a certain limit. Other elements, such as comments, are fully resizable.

Holding the F7 key will display a crosshair. This may be used to check the alignment of the elements.

Use <Ctrl>+Insert and <Shift>+Insert to copy and paste elements. You can get information on an item by selecting it and then pressing the F1 key.

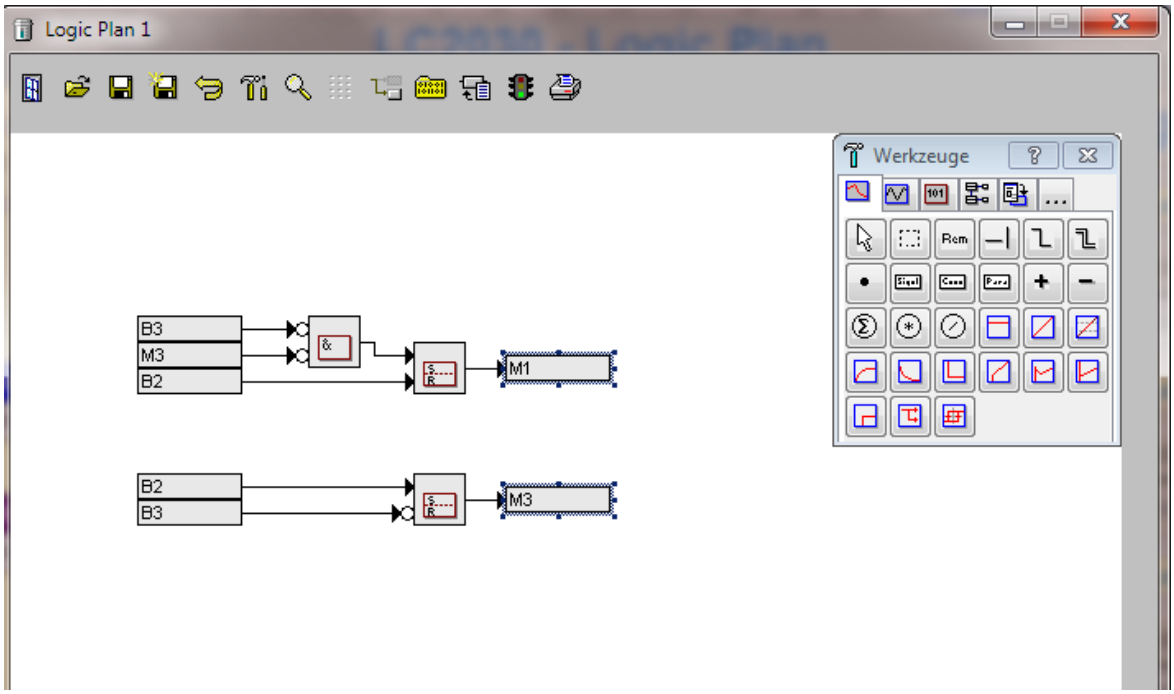


Fig. 17: Example of Logic Plan

The following functions can be found in the tool bar above the logic diagram editor:

From left to right:

- Close active subwindow
- Open – insert a file
- Save current version
- Save all or the selected items to a file
- Restore to last saved version
- Show tool box (if hidden)
- Modify grid alignment settings
- Toggle auto-routing on/off
- Compile logic diagram
- Switch parameter mode on/off after compilation
- Activate logic diagram
- Print the active subwindow
- Help on active subwindow

4.2 Logic Diagram Compilation and Activation

You can find all the elements that are necessary to create a logic diagram in the toolbox.

Click the index card symbol to check whether the logic diagram is syntactically correct.

This button enables parameter mode. You can set the block parameters (e.g. timer) after they have been compiled correctly and verified. Double-click the block you want to adjust and a dialog with the corresponding parameters will open.

Click the traffic light symbol to compile and activate the logic diagram. The logic diagram will only be activated if it is syntactically correct.

If the plan was not properly created, an error message is displayed and you will be asked if the compilation messages should be shown. By clicking on the error, the erroneous part in the the logic diagram is highlighted.

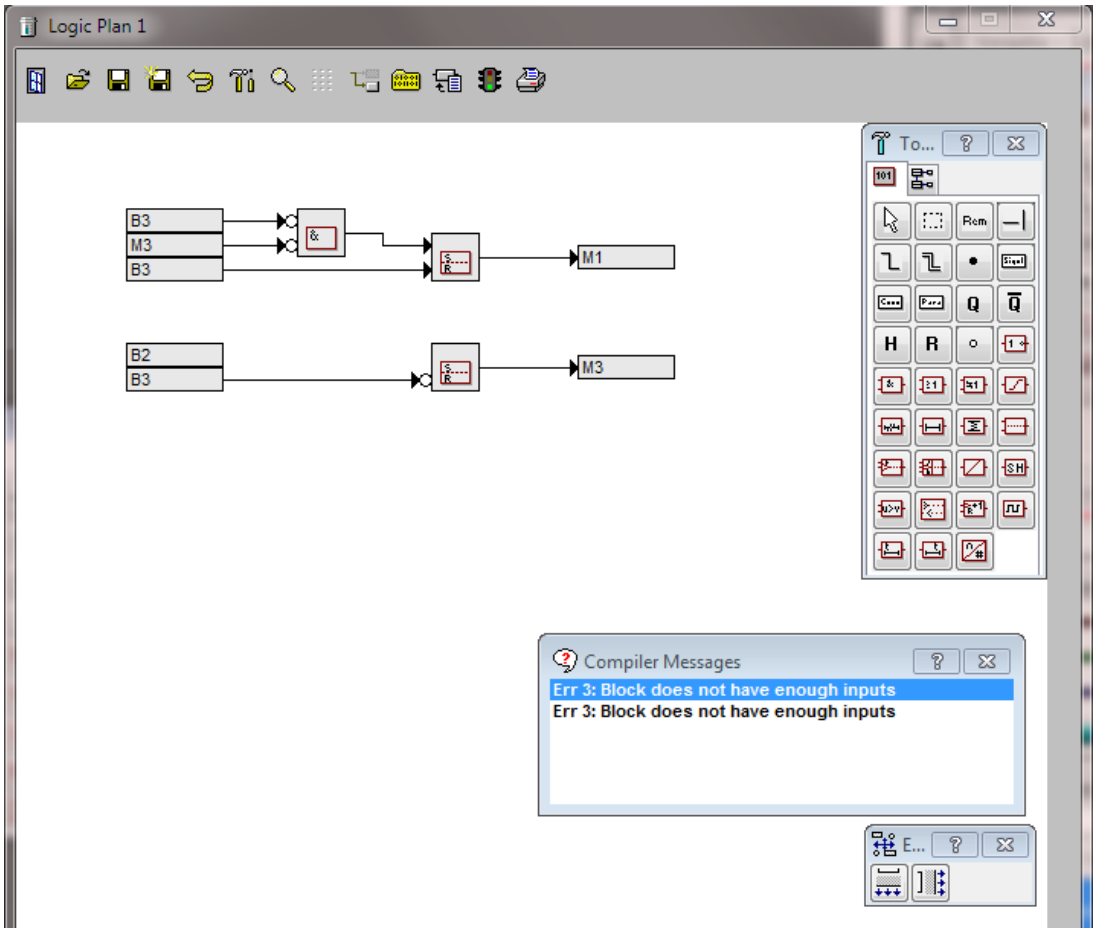


Fig. 18: Error messages after compilation of the logic diagram

If the plan was created properly, you can switch to parameter mode to further configure blocks or you can activate the logic diagram by clicking the traffic light symbol.

The following window (logic diagram view) will appear. Here you can observe the sequence of the logic diagram chart or change parameters of logic blocks by double-clicking them with the left mouse button.

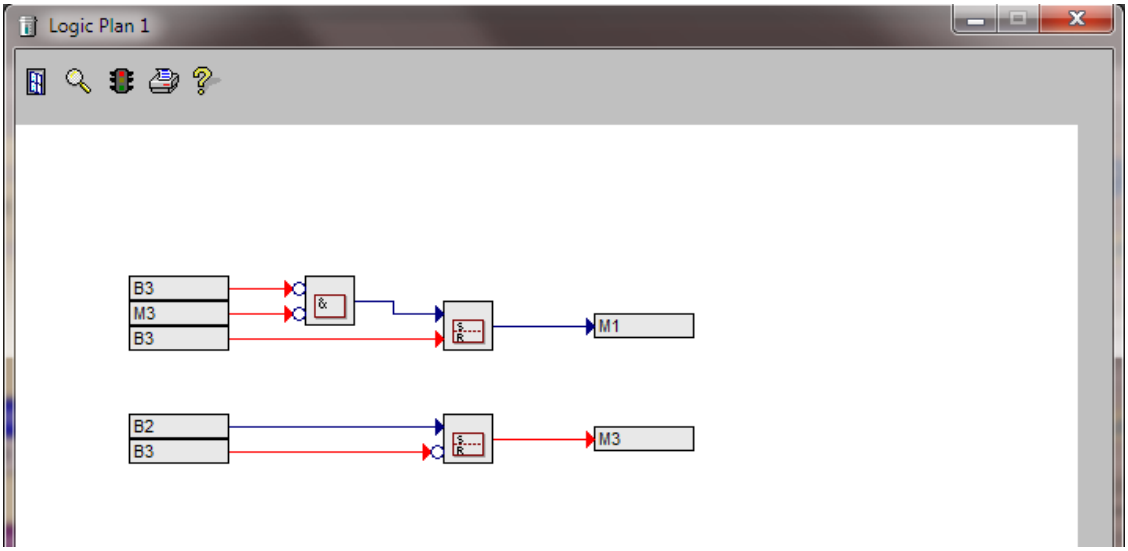


Fig. 19: Activated logic diagram in view mode

4.3 Logic Diagram Elements

Please find a short description of the logic plan elements below.

	Switch back to pointer mode	Turn off current tool
	Frame	Add a frame to the logic diagram
Rem	Comment display	Add a comment box to the logic diagram
	Flow or connecting line	Use lines to connect various elements
	Flow or connecting polygon	Use polygons to connect various elements
	Flow or connecting vector polygon	Use vector polygons to create multiple connecting lines in just one polygon. You must set vector dimension (i.e. the number of combined lines
	Branching point	Click on a line to add a branching point for a second (or third) line from the original line
	Signal block	Double-click with the left mouse button for signal selection
	Constant block	Set as a binary or analog constant. The value cannot be changed while the logic diagram is active
	Parameter block	Set as a binary or analog parameter. The value can be changed while the logic diagram is active
Q	Identifier for non-negative binary outputs of blocks	These output identifiers specify which outputs are negative and which are non-negative
$\overline{Q}$	Identifier for negative binary outputs of blocks	
H	Identifier for hold input	Some blocks may be halted, use this identifier to determine the hold input
R	Identifier for reset input	Some blocks may be reset, use this identifier to determine the reset input
	Negation point	Invert binary inputs or outputs to blocks
	Negation block	Invert the input signal. Needs one input and one output
	AND-block	AND function, 2-8 input signals possible
	OR-block	OR function, 2-8 input signals possible

	XOR-block	XOR function, 2 input signals
	Binary trigger-block	Set to “Positive edge”, “Negative edge” or “Positive and negative edge”. When the input signal changes from low to high, the output of the “Positive edge” trigger will switch from 0 to 1 for one program cycle (0.1s)
	Binary break-block	Not required for the tasks in this program, delays the input signal for one cycle (0.1s)
	Binary delay block	Delays the input for an adjustable number of cycles
	Binary debouncer block	Use to debounce input signals, signal state must be constant for the configured time span to toggle the output signal
	RS-flip-flop	Reset or set-flip-flop, reset input has priority. You can use one or two outputs. When only one output is used, it will be the non-negative. When two outputs are used, output identifiers must be used. You can use the identifier “R” for the reset input. If you do not want to use the reset identifier, the first input is defined as the set command and the second as the reset command.
	D-flip-flop	The D-flip-flop has two inputs and one output. The first input is the signal, the second is the clock/trigger signal. You can use one or two outputs. When one output is used, it will be the non-negative. When two outputs are used, output identifiers must be used.
	JK-master-slave-flip-flop	The JK-master-slave-flip-flop element has three inputs (J signal, clock, K signal). You can use one or two outputs. When one output is used, it will be the non-negative. When two outputs are used, output identifiers must be used.
	Relay-block	The relay block can be used as a toggle or switch relay. If you use two inputs, the function resembles the function of an AND-block. When using three inputs, the first two are the inputs and the third is the switch signal. If any of the other two signals should be the switch signals, use the “S” identifier (on the second tab in the tool box). The switch signal assigns either input 1 or input 2 to the output signal.
	Sample & hold-block	Sample & hold blocks have one analog input, one binary input and one analog output. As long as the state of the binary input is low, the analog input is assigned to the output. When the binary input is high, the last value of the analog input is held.
	Comparision block	Two analog inputs can be compared. Double-click to select mode.
	Limit indicator	The limit indicator can have three analog inputs (upper limit, analog input signal, lower limit) or one analog input. When the value of the analog input is within the limit, the binary output signal is low. When only one analog input is assigned to the block, parameters for upper and lower limit must be set in parameter mode.
	Impulse-counter-block	The impulse counter can be configured as an up counter (2 inputs: count, reset) or as an up/down counter (3 inputs: count, direction, reset). Double-click to select the counter mode. A reset identifier can be used.
	Binary clock generator	Double-click in parameter mode to set the time period.
	Timer block, starting delayed	Uses one (input) or three (input, hold, reset) binary input signals. A “high” input signal will be assigned to the output signal when the timer has expired.
	Timer block, stopping delayed	Uses one (input) or three (input, hold, reset) binary input signals. A “low” input signal will be assigned to the output signal when the timer has expired.
	Analog-to-binary function	Converts analog values greater than zero to the “high” binary signal state

Information regarding errors, inaccuracies and expansion options are highly appreciated!

Please send an e-mail to: info@schoop.de

If you would like further information on the process control and simulation system WinErs, please contact us at:

Ingenieurbüro Dr.-Ing. Schoop GmbH
Riechelmannweg 4
D-21109 Hamburg
Telephone number: 040 / 754 922 30
www.schoop.de
Email: info@schoop.de