CARLO GAVAZZI
Automation Components

UWP 3.0 Tool Manual

rev. 8.0.0.3, 14/09/2018

Index

1 IMPORTANT NOTICE 15

1.1 Guidelines 15
1.1.1 System features 15
1.1.2 Best practises 15
1.2 Function status 15
1.3 Legacy products 16
1.3.1 Differences among the four controllers SH2WEB24, SB2WEB24, SA2WEB24, SBP2WEB2416
1.4 Report sent using the email 18

2 INTRODUCTION 20

2.1 Requirements 20
2.1.1 Minimum hardware requirements 20
2.1.2 Software requirements 20
2.1.3 How to read the software version number 20

3 INSTALLATION 21

3.1 How to connect the master unit UWP 3.0 with Ethernet connection 21
3.2 How to connect with a mini-USB cable 22
3.2.1 How to install the mini-USB driver for Windows 7 / Vista / XP 23
3.2.2 How to install the mini-USB driver for Windows 10 / 8.1 / 8 26

4 USER INTERFACE 34

4.1 File Menu 34
4.2 View Menu 35
4.3 Reports menu 36
4.4 Add menu 36
4.5 Program setup menu 37

5 PROJECT STRUCTURE 38

5.1 Wizard 38
5.1.1 Area 1 38
5.1.2 Area 2 39
5.1.3 Area 3 40

6 LOCATIONS 41

6.1 How to add a location 42
6.2 How to manage Locations 45
6.3 Functions and modules view 46
6.4 Delete operation 48
6.4.1 How to delete a Location 48
6.4.2 Delete commands 49
6.4.3 How to delete a module 49
6.4.4 How to delete a function 49
6.5 Copy and Paste operation 50
6.5.1 How to copy and paste a Location 50

7 MODULES 53

7.1 How to manage the filters in the Modules window 53
7.2 How to manage the filters in the Signals window 55
7.3 Import module list from Template file function 58
7.3.1 How to download the Template file 58
7.3.2 Template file overview 59
7.3.3 How to find the barcode to be inserted 60

7.4 Compile the Template file with Microsoft Excel 61
7.4.3 How to enable Excel macros 61
7.4.4 Automatic insertion with a barcode scanner 62
7.4.5 Manual insertion without a barcode scanner 63
7.5 How to import a fulfilled Template file 64
7.5.1 How to fix a Template file with errors 65
7.6 Light switches 66
7.7 Motion detectors 68
7.7.1 How PIRs work and how to configure them 70
7.8 Temperature modules 74
7.8.1 SHxxXLS2TEMDIS modules 76
7.8.2 SHxxXLS2TEMDISU modules 83
7.9 Decentral modules 90
7.9.1 Digital input/output modules: 90
7.9.2 Analogue input/output modules 92
7.10 Cabinet modules 95
7.11 MasterGlass switches 99
7.11.1 How to configure the key as a standard push button 104
7.11.2 How to configure the key as Dimmer interface 105
7.11.3 How to set the feedback LEDs 109
7.11.4 How to set the glass panel properties 110
7.11.5 How to use a dimmer interface in a dimmable light function 111
7.12 Masterglass temperature display 112
7.12.1 How to configure the key as a standard push button 112
7.12.2 How to configure the key as Temdis interface 113
7.12.3 How to set the feedback LEDs 117
7.12.4 How to set the glass panel properties 118
7.12.5 How to show the external temperature in the Masterglass temperature display 120
7.12.6 How to set the temperature range 121
7.13 Wireless modules 122
7.13.1 Wireless flat switch 123
7.13.2 Wireless window sensor modules 124
7.14 How to manage the analogue variables 126
7.14.1 How to manage the analogue variables using the grouping 126
7.14.2 How to define when analogue values have to be sent 129
7.14.3 Wireless input modules 130
7.14.4 Wireless relay modules 133
7.14.5 Wireless dimmer modules 137
7.14.6 Wireless energy meter modules 141
7.14.7 How to manage the routing tree 144
7.14.8 How to connect a module directly to the wireless base unit 150
7.14.9 How to indirectly connect a module to the wireless base unit using one level of routing 152
7.14.10 How to indirectly connect a module to the Wireless base using two levels of routing 154
7.15 How to add an energy meter 157
7.16 How to add the weather station 159
7.17 How to configure a Modbus converter device 160
7.17.1 How to configure a serial device connected to a Modbus converter 162

8 HOW TO ADD MODULES IN A PROJECT 164

8.1 How the UWP 3.0 discovers the devices in the network 164
8.2 Global discovery of the network 164
8.3 How to automatically find the modules – the controller is connected to the modules 164
8.4 How to add the modules manually 166
8.5 How to add the modules offline 167
8.6 How to add the modules manually and then find them in the network 170

9 HOW TO ADD WIRELESS MODULES IN THE PROJECT 172

9.1 Global discovery of wireless modules 172
9.2 How to add the wireless modules manually 174

10 FUNCTIONS.....175

10.1 How to manage the filters on the Functions window 176
10.2 How to manage the highlight functionality 177
10.3 How to set a light function 179
10.3.1 How to switch a light ON/OFF with input commands 180
10.3.2 How to select the output that will control the light.....184
10.3.3 How to set a signal to be a feedback of the status of the function 185
10.3.4 How to manage lights automatically 186
10.3.5 How to select PIR sensors to control lights 186
10.3.6 How to turn on a light with PIR sensors, and turn it OFF manually.....187
10.3.7 How to turn a light on and off with PIR sensors.....188
10.3.8 How to switch the light ON manually and OFF with the PIR sensor 189
10.3.9 How to switch the light ON and OFF according to the daylight 189
10.3.10 How to control light with PIR sensors in combination with lux meters 192
10.3.11 How to switch the light off after a predefined time period.....195
10.3.12 How to reload the energy-save timer.....195
10.3.13 How to disable the energy save timer 196
10.3.14 How to switch the light ON/OFF using the calendar.....198
10.3.15 How to stop the automation 205
10.3.16 How to force the output ON 209
10.3.17 How to force output off.....209
10.3.18 How to enable the function diagnostic.....210
10.3.19 How to change the location of a function.....211
10.3.20 Corridor lighting function.....212
10.4 How to set a Dimmer function 217
10.4.1 How to dim a light with input commands 218
10.4.2 How to select a dimmable output 221
10.4.3 How to change settings and load type of the output 222
10.4.4 How to change settings and load type of the DALI output 223
10.4.5 How to modify the minimum/maximum values of the dimming range 225
10.4.6 How the dimmer works 226
10.4.7 How to set a signal as feedback of the status of the function 227
10.4.8 How to set predefined scenarios with input signals.....228
10.4.9 How to manage lights automatically 234
10.4.10 How to select a PIR sensor to control scenario.....234
10.4.11 How to turn a light ON with PIR sensors, and turn it OFF manually 235
10.4.12 How to turn a light ON with PIR sensors, and change the scenario after a predefined period 236
10.4.13 How to turn a light ON with PIR sensors, and change the light scenario manually when PIR detects movement.....237
10.4.14 How to turn a light on and off with PIR sensors.....238
10.4.15 How to change scenario with a PIR.....239
10.4.16 How to switch the light on manually and off with the PIR sensor 239
10.4.17 How to switch the light on and off according to the daylight.....240
10.4.18 How to control light with PIR sensors in combination with lux sensor.....243
10.4.19 How to switch the light off after a predefined period without lux meter 246
10.4.20 How to switch the light off after a predefined time period with the lux meter 247
10.4.21 How to switch the light on/off using the calendar 247
10.4.22 How to use a MASTER GLASS dimmer interface in a dimmable light function.....248
10.4.23 How to stop an automation 253
10.4.24 How to force the output fully ON.....256
10.4.25 How to force the output OFF 256
10.4.26 How to configure Sequence A 257
10.4.27 How to configure Sequence B 258
10.4.28 How to manage the power supply of a ballast.....259
10.4.29 Corridor lighting function.....260
10.5 How to set a Constant light function 265

10.5.1 How to set a light with input commands 266
10.5.2 How to select a dimmable output 267
10.5.3 How to change settings and the load type of the output 268
10.5.4 How to add the luxsensor 269
10.5.5 How to set a signal as feedback of the status of the function 272
10.5.6 How to set predefined scenarios with input signals 273
10.5.7 How to manage lights automatically 276
10.5.8 How to select a PIR sensor to control scenario 276
10.5.9 How to turn a light ON with PIR sensors, and turn it OFF manually 277
10.5.10 How to turn a light on and off with PIR sensors 279
10.5.11 How to switch the light on manually and off with the PIR sensor 280
10.5.12 How to switch the light off after a predefined period 281
10.5.13 How to set a predefined scenario when the energy save timer expires 282
10.5.14 How to switch the light on/off using the calendar 282
10.5.15 How to stop an automation 286
10.5.16 How to force the output fully ON 289
10.5.17 How to force the output OFF 289
10.5.18 How to configure Sequence A 290
10.5.19 How to configure Sequence B 291
10.5.20 Corridor lighting function 292
10.5.21 How to configure a Constant light zone control 297
10.5.22 Master zone calibration procedure 301
10.5.23 Slave zone calibration procedure 304
10.5.24 Position of the slave zones in relation to the master zone 307
10.6 Group light command function 310
10.6.1 How to set a Group light command function 311
10.6.2 How to switch a Group of light command functions with input commands 313
10.6.3 How to add the Light functions 317
10.6.4 How to change the location of a function 320
10.7 How to set a Smart light function 321
10.7.1 The command structure 322
10.7.2 Configure predefined scenarios S2, S3, S4 and S5 325
10.8 Use cases 326
10.8.1 Toggle the light ON/OFF with a push button 326
10.8.2 Push button - Toggle ON/OFF + Dim UP/DOWN 329
10.8.3 Push button 1,2 - Toggle ON/OFF and Dim UP , Dim DOWN 330
10.8.4 Push Button + ES timer 331
10.8.5 PIR sensor + ES timer 332
10.8.6 PIR sensor + ES timer 334
10.8.7 Push button + PIR sensor + ES timer 335
10.8.8 Push Button + PIR sensor enabled by the Luxmeter sensor 337
10.8.9 PIR sensor + ES timer and Night time condition 339
10.8.10 PIR sensor + ES timer + Day and Night time conditions 340
10.8.11 RTC (Local calendar) 342
10.8.12 PIR + RTC + ES timer 343
10.8.13 Luxmeter sensor turns the light ON to 80% when Night time ON 345
10.8.14 Change the colour temperature automatically according to the colour table 346
10.8.15 Change manually the colour temperature overriding the colour table managed by the RTC. 348
10.9 How to set a Rollerblind function 349
10.9.1 How to move Up/Down blinds using a manual input 350
10.9.2 How to set the reset command 352
10.9.3 How to select and configure the output 353
10.9.4 How to set the running time for Motor output 354
10.9.5 How to set a signal to be a feedback of the status of the function 356
10.9.6 How to manage rollerblinds with automations 357
10.9.7 How to customize the push button behaviour 357
10.9.8 How to use the wind sensor to control blinds 360

10.9.9 How to use the rain sensor to control blinds....362
10.9.10 How to control the blind according to the daylight....364
10.9.11 How to move the blinds up/down and tilt them with the calendar....368
10.9.12 How to use the emergency signals....371
10.9.13 How to stop the automation....373
10.9.14 How to force the safety condition....375
10.9.15 How to force the non-safety condition....376
10.10 How to set a Rollerblind function with tilting slats....377
10.10.1 How to move blinds UP/DOWN using a manual input....378
10.10.2 How to tilt blinds clockwise/anticlockwise using a manual input....381
10.10.3 How to set the reset command....383
10.10.4 How to select and configure the output....384
10.10.5 How to set the running and tilting time....385
10.10.6 How to set a feedback signal....387
10.10.7 How to manage blinds with automations....387
10.10.8 How to customize the push button behaviour....388
10.10.9 How to use the wind sensor to control blinds....390
10.10.10 How to control the blind according to the daylight....393
10.10.11 How to move the blinds up/down and tilt them with the calendar....397
10.10.12 How to use the emergency signals....400
10.10.13 How to stop the automations....402
10.10.14 How to force the safety condition....405
10.10.15 How to force the non-safety condition....405
10.11 How to add a smoke alarm function....406
10.11.1 How to set a simple smoke alarm system....407
10.11.2 How to add a feedback signal....409
10.11.3 How to add the reset signal to inhibit the output status....410
10.11.4 How to use a siren function for the alarm output....412
10.11.5 How to force the alarm status ON....413
10.11.6 How to force the alarm status OFF....414
10.12 How to set a Siren alarm function....415
10.12.1 How to change settings for the siren output....415
10.12.2 How to link an alarm function to the siren....416
10.12.3 How to add the siren output....418
10.12.4 How to use a manual signal to activate the siren....419
10.12.5 How to use more than one alarm with a common siren....419
10.12.6 How to reset the siren....421
10.13 How to add a water alarm function....422
10.13.1 How to set a simple water alarm system....423
10.13.2 How to add a feedback signal....425
10.13.3 How to add the reset signal to inhibit the function status....426
10.13.4 How to use a siren function for the alarm output....428
10.13.5 How to force the alarm status ON....429
10.13.6 How to force the alarm status OFF....430
10.14 How to set an intruder alarm function....431
10.14.1 How to add the alarm signals....434
10.14.2 How to set the arming and disarming time....435
10.14.3 How to arm and disarm the intruder alarm....436
10.14.4 How to manage zones using different arming/disarming signals....438
10.14.5 Output status of the intruder alarm....439
10.14.6 How to set a feedback signal....442
10.14.7 How to reset the alarm....445
10.14.8 How to use the alarm with a siren....447
10.14.9 How to set the stress lights when the alarm is active....447
10.14.10 How to arm and disarm the alarm with the calendar....448
10.14.11 How to force the alarm condition....451
10.14.12 How to force the alarm into disarmed....452
10.15 How to set a temperature function....453

10.15.1 Heating/Cooling control system configuration 454
10.15.2 How to activate/deactivate the control system function....455
10.15.3 How to set the Heating/Cooling control system function always active 457
10.15.4 How to add the heating output....457
10.15.5 How to set a feedback signal....459
10.15.6 How to switch off heating/cooling according to the outdoor temperature....460
10.15.7 How to set the antifreeze temperature 461
10.15.8 How to control heating/cooling with the calendar 462
10.15.9 How to disable the automations 464
10.15.10 How to force the control system function to the on status....466
10.15.11 How to force the control system function to the off status....467
10.16 Zone temperature configuration 468
10.16.1 How to add the temperature signal....468
*10.16.2 How to use a zone temperature function without a heating/cooling control system function*469
10.16.3 How to configure the setpoints 470
10.16.4 How to use the heating and cooling dead-bands 471
10.16.5 How to set the maximum and minimum values for the setpoints 476
10.16.6 How to add the heating output....476
10.16.7 How to set a feedback signal....477
10.16.8 How to use an auxiliary temperature to monitor the temperature in the zone....477
10.16.9 How to enable the display of the external temperature in the TEMDIS display 479
10.16.10 How to set the safe mode....480
10.16.11 How to manage heating/cooling using the calendar automation....481
10.16.12 How to change the set point using the function status 483
10.16.13 How to set the ON/OFF control 484
10.16.14 How to set the PID....485
10.16.15 How to set the PID 1 mode....485
10.16.16 How to set the PID 2 (PID advanced)....486
10.16.17 How to add Analogue output signals....488
10.16.18 How to activate the set point with pushbuttons and functions....489
10.16.19 How to use external signals as Direct setpoints....491
10.16.20 How to set the feedback signals for the selected setpoint 492
10.16.21 How to disable the automations 493
10.16.22 How to force heating/cooling ON....495
10.16.23 How to force heating/cooling OFF....496
10.17 How to set a delay timer function 497
10.18 How to add the start signal....499
10.19 How to add the stop signal....500
10.19.1 How to add the output signal....501
10.20 How to set a recycling timer function....502
10.20.1 How to add the start signal 504
10.20.2 How to add the stop signal 506
10.20.3 How to add the output signal....508
10.21 How to set an interval timer function 509
10.21.1 How to add the start signal 512
10.21.2 How to add the stop signal 514
10.21.3 How to add the output signal 516
10.22 How to set a global calendar function 517
10.22.1 How to use a global calendar in different functions....517
10.22.2 How to use a global calendar in roller blind functions 519
10.22.3 How to use a global calendar in Multigate functions or as a level input signal in any type of function 520
10.22.4 Add a new activity to the calendar....521
10.22.5 Meaning of the actions in the different functions 523
10.22.6 Simultaneous activities 525
10.22.7 How to create a calendar in-between two years 526
10.22.8 How to create a recurring calendar 527
10.23 How to configure a counter function....528

10.23.1 How to add the increasing input 528
10.23.2 How to add the decreasing input 531
10.23.3 How to sum up analogue values 532
10.23.4 How to configure the counting function to count visitors to retail outlets 533
10.23.5 How to reset the counting value 535
10.23.6 How to add an analogue output signal 536
10.23.7 How to add a feedback signal or a relay output 537
10.23.8 How to set the counting options 537
10.23.9 How to set a predefined value using signals 538
10.23.10 How to set the counter function equal to another function 539
10.23.11 How to log the outputs of the counting function 539
10.24 How to set a Multigate function 541
10.24.1 How to select the logic operation 541
10.24.2 How to add the input signals 542
10.24.3 How to add the output signal 543
10.24.4 How to check the status of more functions 544
10.25 How to set the simulated habitation function 545
10.25.1 How to add a function to be recorded 545
10.25.2 How to play/stop the simulated habitation function 548
10.25.3 How to add the feedback signals 551
10.25.4 How to play/stop the function using a calendar 552
10.25.5 How to play/stop Simulated habitation using a Function 554
10.25.6 How to disable the calendar automation 555
10.25.7 How to force the function to play 557
10.25.8 How to force the function stop 558
10.26 How to set the sequence function 559
10.26.1 How to set the start/stop signals 559
10.26.2 How to set the start/pause signals 561
10.26.3 How to add a function to the sequence 562
10.26.4 How to set the action for the functions in the sequence 564
10.26.5 How to modify the sequence 567
10.26.6 How set the delay time between the activation of two functions 568
10.26.7 How to add the feedback signals 570
10.26.8 How to start/stop the sequence using a calendar 571
10.26.9 How to disable the calendar automation 573
10.26.10 How to use an external function to start/stop a sequence 576
10.26.11 How to force the sequence active 577
10.26.12 How to force the sequence stop 578
10.27 How to set the dimmer sequence function 579
10.27.1 How the dimmer sequence works 579
10.27.2 How to set the start/stop signals 582
10.27.3 How to set the start/pause signals 584
10.27.4 How to add a function to the sequence 586
10.27.5 How to set the sequence steps 586
10.27.6 How to modify the sequence steps 587
10.27.7 How to set the delay time between the activation of two steps 588
10.27.8 How to add the feedback 589
10.27.9 How to start/stop the sequence using a calendar 590
10.27.10 How to disable the calendar automation 592
10.27.11 How to use an external function to start/stop a sequence 595
10.27.12 How to force the sequence active 596
10.27.13 How to force the sequence stop 597
10.28 How to set the car heating function 598
10.28.1 How the heating algorithm works 598
10.28.2 How to set the setpoints and times 600
10.28.3 How to set the manual signals 600
10.28.4 How to add the feedback signal 602
10.28.5 How to add the output 603

10.28.6 How add the temperature sensor *604*
10.28.7 How to set the ready time *605*
10.28.8 How to disable the calendar automation *606*
10.28.9 How to force the output on *609*
10.28.10 How to force the output off *610*
10.29 How to set an analogue comparator function 611
10.29.1 Comparator settings *611*
10.29.2 How to set the reference values *613*
10.29.3 How to add the input signals to be compared *614*
10.29.4 How to add the output signal *614*
10.30 How to configure a mathematical function 616
10.30.1 How to add the input signals *617*
10.30.2 How to set the curve *619*
10.30.3 How to generate an output according to the conversion curve *620*
10.30.4 How to use a Mathematical function without a conversion curve *622*
10.30.5 How to add an analogue output signals *623*
10.30.6 How to use an analogue output signal to regulate the valve position *624*
10.30.7 How to use the calculated values *627*
10.30.8 How to log additional function status in the Database *628*
10.30.9 How to change the location of a function *630*
10.31 How to configure an analogue output function 631
10.31.1 How to add the input signal *631*
10.31.2 How to reset the Analogue output value *633*
10.31.3 How to force the Reset output *635*
10.31.4 How to add an analogue output signal *636*
10.31.5 How to enable the function diagnostic *637*
10.31.6 How to change the location of a function *638*
10.32 How to configure an hour counting function 639
10.32.1 How to add the input signal *640*
10.32.2 How to reset the Hour counting value *641*
10.32.3 How to set the Hour counting options *642*
10.32.4 How to add an Alarm status output *643*
10.32.5 How to log the outputs of the Hour counting function *644*
10.33 Astronomical clock function 645
10.33.1 How to add the Astronomical clock function *645*
10.33.2 How to configure the Astronomical clock function *647*
10.33.3 Use cases *649*
10.34 Twin pumps function 655
10.35 How to configure an On/Off pump function 658
10.35.1 How to add the fault signals *658*
10.35.2 How to add the service signals *661*
10.35.3 How to set the function options *662*
10.35.4 How to add the output signals *663*
10.35.5 How to enable the function diagnostic *664*
10.36 How to configure an On/Off pump group function 665
10.36.1 How to activate the regulation for the group of pumps *665*
10.36.2 How to select the controlled On/Off pump functions *667*
10.36.3 How to manually update the priority list based on the working hour counters *668*
10.36.4 How to activate more than one load at the same time *670*
10.36.5 How to disable the automation *674*
10.36.6 How to manually alternate the activation of the loads in a group *676*
10.36.7 How to add the output signals *681*
10.37 How to set the Program function 682
10.37.1 How to set the Function settings *683*
10.37.2 How to command the Program function with Input signals *684*
10.37.3 How to add steps to the Program function *687*
10.37.4 How to add the feedback signals *692*
10.37.5 How to force the sequence active *693*

10.37.6 How to force the sequence inactive *694*
10.37.7 How to start/stop the sequence using a calendar *695*
10.37.8 How to use an external function to start/stop a sequence *696*
10.38 How to manage the Program function from the WEB-App 697
10.38.1 How to add a Program function widget *697*
10.38.2 How to manage the Program function from the advanced menu *699*
10.38.3 How to execute a set of steps one time only *704*
10.39 How to set a Switch function 706
10.39.1 How to turn the function OFF after a predefined time period *707*
10.39.2 How to turn the output ON/OFF with input commands *708*
10.39.3 How to select the output of the Switch function *711*
10.39.4 How to set a signal as feedback of the status of the function *712*
10.39.5 How to force the output ON *715*
10.39.6 How to force the output OFF *716*
10.39.7 How to change the location of a function *717*
10.39.8 How to turn the Switch function ON/OFF using the Local calendar *718*

11 HOW TO CONFIGURE BACNET OBJECTS IN UWP 3.0 720

11.1 How BACnet is implemented in UWP 3.0 720
11.2 BACnet objects related to modules 720
11.3 BACnet objects related to functions 720
11.4 Basic settings for instance numbers 721
11.5 Selection of BACnet objects to be created 722
11.6 BACnet report 723
11.7 How to create an EDE-file 724
11.8 How to Regenerate BACnet map 724

12 HOW TO CONFIGURE A DALI NETWORK WITH A SB2DALIT8230 MODULE 725

12.1 How to interface Dupline to DALI 725
12.2 How many SB2DALIT8230s can be managed 726
12.3 DALI groups 726
12.4 How to add the SB2DALIT8230 DALI master 728
12.5 DALI network manager window 729
12.5.1 Area 1 – Discovery and testing tools *729*
12.5.2 Area 2 – Programming tools *729*
12.5.3 Area 3 - Diagnostic tools *729*
12.6 How to connect to a DALI network 730
12.6.1 How to automatically find and address the ballasts – The network is connected to the master *730*
12.7 DALI addressing 732
12.7.1 Start new discovery procedure *732*
12.7.2 Start/stop discovery procedure *733*
12.8 When *Start new discovery* is to be used 734
12.8.1 New discovery maintaining existing short addresses *734*
12.9 When *Start discovery* can be used 735
12.9.1 Discovery overwriting groups value *735*
12.10 Project design (Pre-programming – How to execute individual commands) 736
12.10.1 Define the position where each ballast is located *737*
12.11 Programming – How to manage the ballast settings 741
12.11.1 Group settings *741*
12.11.2 How to rename the DALI groups *745*
12.12 How to test a specific DALI group 746
12.13 Ballast settings 747
12.13.1 Tuning the fade time / fade rate parameters *748*
12.13.2 How to program the ballast individually *751*
12.13.3 How to program the ballasts together via the Multiediting panel *752*
12.14 Advanced settings 754
12.14.1 How to manually assign an address to a DALI ballast *755*
12.15 Restore ballasts to factory settings 756
12.15.1 How to reset a DALI ballast *756*

12.15.2 How to reset all the DALI ballasts of a DALI network *756*
12.15.3 How to swap two ballasts *757*
12.15.4 How to connect to a SB2DALIT8230 module from the DALI network manager window *760*
12.15.5 How to restore the configuration of a new SB2DALIT8230 module *760*
12.16 How to check the DALI bus status 761
12.16.1 DALI lamp failure *762*
12.16.2 DALI network short-circuited *762*
12.16.3 DALI power failure *762*
12.16.4 How to see diagnostic info in the DALI network manager window *763*

13 HOW TO CONFIGURE A DALI NETWORK WITH A SB2DALI230 MODULE 764

13.1 How to interface Dupline to DALI 764
13.2 How to add the DALI master SB2DALI230 765
13.3 How many SB2DALI230 can be managed 766
13.4 Functions and groups 766
13.5 How to manually add a DALI ballast 768
13.6 How to configure a dimmer function using DALI outputs 770
13.7 How to automatically add a DALI ballast 772
13.8 How to automatically find the ballasts connected to the SB2DALI230 772
13.8.1 How to launch a new network discovery *773*
13.8.2 How to change a DALI address *775*
13.8.3 How to add new modules *777*
13.8.4 How to rename modules *779*
13.9 Advanced setup 780
13.9.1 How to test a module *782*
13.9.2 How to reset a DALI ballast *783*
13.9.3 How to manually assign an address to a ballast *785*
13.9.4 How to swap the addresses of two ballasts *787*
13.10 How to change the settings of the DALI output in a function 788
13.10.1 How to set / restore the DALI ballast output state *789*
13.10.2 How to set the Power on level (%) field *789*
13.10.3 How to set the Light level if the DALI bus is faulty *790*
13.10.4 How to set the Output status if the Dupline bus is faulty *791*
13.11 How to check the DALI bus status 792
13.11.1 DALI lamp failure *793*
13.11.2 DALI network short-circuited *793*
13.11.3 DALI power failure *793*
13.11.4 How to check if the DALI ballast supports the latest state *794*
13.11.5 How to restore the latest value with the Recovery signal option *795*

14 HOW TO CONFIGURE THE WEBSERVER ACCOUNTS 797

15 HOW TO CONFIGURE THE DATABASE 800

15.1 Database files 801
15.1.1 CSV file data format *803*

16 HOW TO CONFIGURE THE UWP 3.0 AS MODBUS SLAVE 805

16.1 How to access the Modbus map 806
16.1.1 How to change the default Modbus ID *807*
16.1.2 How to save the Modbus map *808*
16.2 How to configure a Modbus converter device 811
16.2.1 How to configure a serial device connected to a Modbus converter *813*

17 HOW TO CONNECT TO VMUCS AND UWP 3.0S 815

17.1 How to connect to devices in the same network 817
17.2 How to configure an offline device importing an xml file 820
17.3 How to import the map of variables from a remote controller 824
17.4 How to generate the xml file 827
17.5 How to connect to generic Modbus devices 829
17.6 Modbus frame editor 831

17.6.1 How to add a new Modbus RTU command *832*
17.6.2 How to configure a Modbus output bridge function *837*
17.6.3 How to send the Modbus command manually *837*
17.6.4 How to send the Modbus RTU command automatically *839*
17.6.5 How to add a new Modbus TCP command *841*
17.6.6 Modbus TCP command format *843*
17.6.7 How to configure a Modbus TCP output bridge function *844*
17.6.8 How to send the Modbus TCP command manually *844*
17.6.9 How to send the Modbus TCP command automatically *847*

18 MODBUS DRIVER EDITOR TOOL 850

18.1 How to manage the filters in the Modbus Driver Editor window 851
18.2 Modbus Driver Editor overview 852
18.2.1 List Tab overview *852*
18.2.2 Contextual menu Driver options *853*
18.2.3 Edit tab overview *854*
18.2.4 [A] DRIVER level overview *857*
18.2.5 Things to know *859*
18.2.6 [B] CONFIGURATION level overview *860*
18.2.7 [C] Group level overview *861*
18.2.8 [D] Variable level overview *865*
18.2.9 Things to know *866*
18.3 How to create a Modbus driver – Real Use case (EM24-DIN) 871
18.3.1 Part1 - Creating/editing a Modbus driver *872*
18.3.2 Part2 – Setting the device parameters (A) *873*
18.3.3 Part3 – Adding/editing a Configuration (B) *876*
18.3.4 Part4 – Adding/editing/coping a Group (C) *879*
18.3.5 Part4a - Adding/editing a Trend (or Custom Chart) *883*
18.3.6 Part5 – Adding/editing a variable (D) *886*
18.3.7 How to apply the calculation *888*
18.3.8 Other Variable *891*
18.3.9 Building a Modbus driver *900*
18.3.10 Copying a Modbus driver *900*
18.3.11 Activating a Modbus driver *900*
18.3.12 Testing a Modbus driver *900*
18.3.13 Deleting a driver *900*
18.3.14 Importing a driver *900*
18.3.15 Exporting a driver *901*
18.3.1 How to test a driver *902*

19 LIVE SIGNALS 903

19.1 How live signals work 903
19.1.1 Checking the status of all the analogue and digital values *903*
19.1.2 Tuning the parameters of the functions *903*
19.1.3 Checking the Bus diagnostic *903*
19.1.4 How the quality index works *904*
19.1.5 Checking the status of each module: *907*
19.2 How to enable/disable *live signals* 908
19.2.1 How to allow Live signals to show an analogue value *908*
19.3 How to see values and status of the functions 911
19.4 How to tune the parameters of a function 912
19.5 How to change the status of a function 912
19.5.1 On/off light *913*
19.5.2 Dimmable light *914*
19.5.3 Zone intruder alarm *915*
19.5.4 Main intruder alarm *915*
19.5.5 Smoke alarm *916*
19.5.6 Water alarm *916*

19.5.7 Siren function *917*
19.5.8 Sequence function *917*
19.5.9 Rollerblind function *918*
19.5.10 Window function *919*
19.5.11 Zone Temperature *920*
19.5.12 Heating system temperature *920*
19.5.13 Cooling system temperature *921*
19.5.14 Delay timer *921*
19.5.15 Recycling timer *921*
19.5.16 Interval timer *922*
19.5.17 Simulated habitation *922*
19.5.18 Analogue comparator *922*

20 HOW TO SET THE EMAIL FUNCTION 923

20.1 How to configure the email account 923
20.2 How to manage the accounts in the contacts list 923
20.3 How to configure an outgoing mail 924

21 HOW TO SET THE GSM FUNCTION 925

21.1 GSM configuration 925
21.2 How to set up the smart-house system to send SMS 926
21.3 How to set up the UWP 3.0 to receive an SMS 927

22 GENERAL SETTINGS 930

22.1 How to configure the general settings 930
22.1.1 E-mail /SMS setup *931*
22.1.2 System information *932*
22.1.3 How to set Modbus TCP/IP settings *933*
22.1.4 How to set Modbus RTU properties *934*
22.1.5 How to configure the modem *935*
22.1.6 Advanced settings *938*
22.1.7 How to add module tags *940*
22.1.8 How to create customized locations *941*
22.2 How to configure the Report 942
22.2.1 How to add an FTP account *942*
22.2.2 How to add an email account *943*
22.2.3 How to remove an existing account *944*
22.2.4 How to test the FTP / email connection *944*
22.2.5 How to enable the Report scheduler *945*
22.3 How to configure the Project settings 946
22.3.1 How to test Email and SMS *946*
22.4 How to configure the IP address of the master unit UWP 3.0 949
22.5 How to configure the date and time of the master unit UWP 3.0 950
22.5.1 Internet date and time update *951*
22.6 How to update the firmware in the master unit UWP 3.0 952
22.7 How to set the password protection in the Controller 954
22.8 How to set the DynDNS parameters when using the modem 956
22.8.1 How to connect to the controller by means of a modem *958*
22.8.2 How to see the dynamic IP address *959*
22.9 How to import/export the UWP 3.0 Tool settings 960
22.10 The Compile project function 961
22.10.1 How to compile a project *961*
22.10.2 How to fix an error in the configuration *963*
22.10.3 How to check the network consumption *964*
22.10.4 How to understand when a new compilation is needed *965*
22.11 How to read/write the project in the UWP 3.0 966
22.11.1 How to send a configuration from the Compile manager window *967*
22.11.2 How to manually select the modules/network to program *968*
22.11.3 How to program again the modules in error *970*

22.11.4 How to write a project with Send to controller function.....972
22.11.5 How to manage the writing operations with a modem connection 973
22.12 Programming the wireless modules 973

1 Important notice

In the new software version 7.4.3 we have updated the Modbus drivers of the following products:

EM24
WM14
CPA300
CPT

This means that if a configuration has been generated with 6.5.33 or previous software versions, when it is updated to 7.4.3 or later releases, the energy meters must be deleted and then added again.
If this procedure is not executed, they will show "Updating" in live signal and the communication will fail.

1.1 Guidelines

1.1.1 System features

- Up to 5000 managed signals (including variables, I/Os) shared among Energy Management, Building Automation and Car Park applications.

Note: when the Car Park system is active, there will be 2000 signals available for the other applications (Energy Management and Building Automation).

- up to 128 Modbus devices connected to RS485 ports (64 devices each port)
- up to 5 users concurrently connected to the Web-App
- up to 5 concurrent M2M connections (API connections, BACNET clients, MODBUS masters)
- up to N different products from the CG range can be connected to UWP 3.0.

1.1.2 Best practises

- avoid enabling events logging for analogue variables as they can quickly fill the database
- the use of Park guidance functions together with data push functions is not allowed by UWP 3.0
- avoid sending to UWP 3.0 frequent and/or huge data requests via Rest-API, as they can overload UWP 3.0
- avoid sending to UWP 3.0 frequent and huge data requests via MODBUS or MODBUS/TCP or BACNET, as they can overload UWP 3.0
- avoid mixing in the same UWP 3.0 big Car Park installations with big Building Automation installations; use different controllers for different responsibilities
- remind that direct data connection between UWP 3.0 units or between UWP 3.0 and VMU-C EM or UWP 3.0 and Sx2WEB units should be used for few variables, as the MODBUS/TCP communication is done variable by variable; in the case of multiple variables reading it is strongly suggested to create a driver via the MODBUS DRIVER EDITOR tool.

1.2 Function status

To reduce the number of Dupline channels, from the software release 7.4.3, in the functions listed below, if some fields in the *Advanced* section are used, it is mandatory to enable the relevant status in the *Available output status* field.

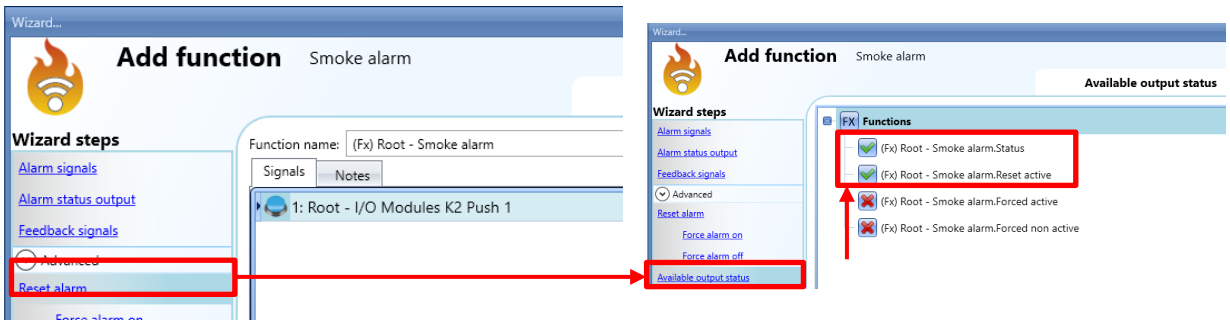
List of functions:

- Dupline Temperature regulation
- Smoke alarm
- Water alarm
- Counter
- Analogue output
- Hour counting

- Car Heating
- Mathematical function
- Zone temperature (when PID advanced is used)
- On/Off pump function

If this is not done, an error will occur during compilation process. The table below shows the status which needs to be enabled.

Function	Field	Output-input link	Additional output to be enabled
Dupline Temperature regulation	Advanced/Heating		Dupline: Temperature regulation.Heating
	Advanced/Cooling		Dupline: Temperature regulation.Cooling
Smoke alarm	Advanced/Reset alarm		Smoke alarm.Reset active
Water alarm	Advanced/Reset alarm		Water alarm.Reset active
Counter	Feedback		Counter function.Feedback status
Analogue output	Output signals		Analogue output.Output value after a reset command
Hour counter	Alarm status output		Hour counter.Alarm status
Car heating	Feedback signals		Car heating.Feedback
Mathematical function	Output signals	Average input value + offset	Mathematical function.Input values average + offset
		Minimum input value	Mathematical function.Minimum input
		Maximum input value	Mathematical function.Maximum input
		A input – B input	Mathematical function.A input – B input
		Average input value + offset, linearized + output offset	Mathematical function.Average input value + offset, linearized + output offset
		Linearized minimum value	Mathematical function.Linearized minimum value
		Linearized maximum value	Mathematical function.Linearized maximum value
Zone temperature (with analogue PID regulation)	Advanced/Analogue output signals/Heating		Zone temperature.Analogue output for heating
	Advanced/Analogue output signals/Cooling		Zone temperature.Analogue output for cooling
On/Off pump function	Output signals		On/Off pump function.Status



1.3 Legacy products

UWP 3.0 integrates all the functions that are available with the four controllers described below.
In order to provide a complete information, the following table describes the differences among the legacy products.

1.3.1 Differences among the four controllers SH2WEB24, SB2WEB24, SA2WEB24, SBP2WEB24

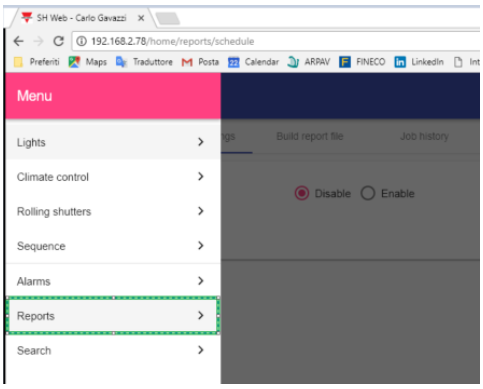
		SA2WEB24	SH2WEB24	SB2WEB24	SBP2WEB24
		Hotel/Flat	Home	Building	Building & Car-park
Database	History (dmd common to all variables)	•	•	•	•
	Event	•	•	•	•
	Scheduler mail report	•	•	•	•
	Scheduler ftp report	•	•	•	•
Remote	Mail notifications	•	•	•	•
	Sms notifications		•	•	•
	Sms commands		•	•	•
Light	light	•	•	•	•
	dimmable light	•	•	•	•
	constant light	•	•	•	•

	dimmer sequence	•	•	•	•
Motor	roller-blinds	•	•	•	•
	tilting blinds	•	•	•	•
	open close window	•	•	•	•
Temperature	zone temperature	•	•	•	•
	heating system temperature	•	•	•	•
	cooling system temperature	•	•	•	•
Alarm	zone intruder alarm	•	•	•	•
	main intruder alarm	•	•	•	•
	smoke alarm	•	•	•	•
	water alarm	•	•	•	•
	siren alarm	•	•	•	•
Timer	delay timer	•	•	•	•
	recycling timer	•	•	•	•
	interval timer	•	•	•	•
Basic	calendar	•	•	•	•
	sequence	•	•	•	•
	multi-gate	•	•	•	•
	counter		•	•	•
	analogue comparator	•	•	•	•
	mathematical		•	•	•
	PID control		•	•	•
	modbus output bridge		•	•	•
	remote modbus variable		•	•	•
	analogue outputs		•	•	•
	hour counting		•	•	•
	on/off pump		•	•	•
	on/off pump group		•	•	•
	analogue command for twin pumps		•	•	•
	analogue command for groups of pump		•	•	•
Car park	Data pointer				•
	Master zone counter				•
	Car park sensor				•
	Car park display				•
	Fast display update				•
	Location Line, Lane, Level				•
Special	Car heating	•	•	•	•
	simulated habitation	•	•	•	•

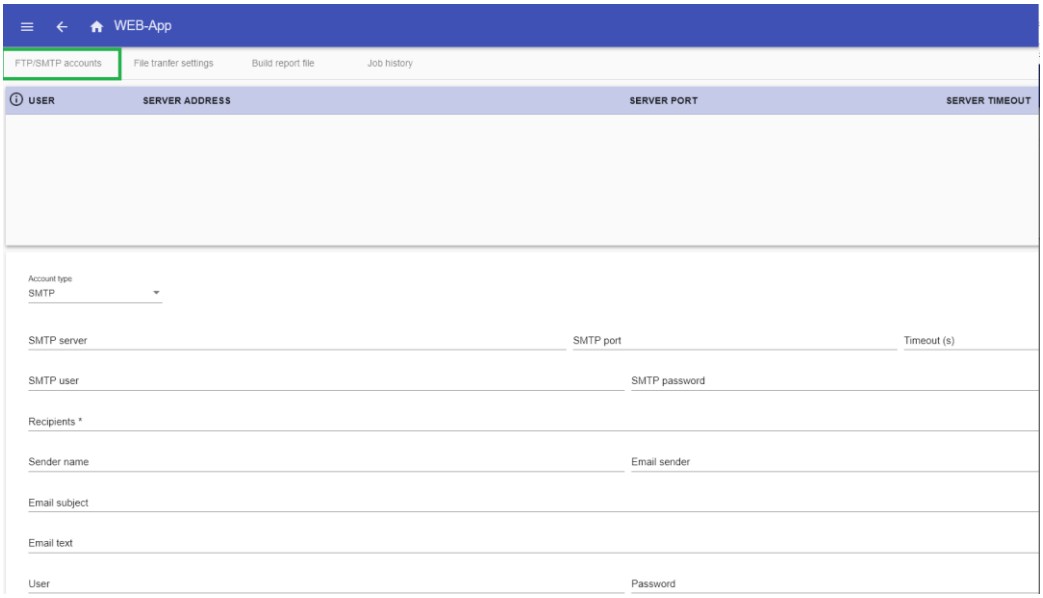
1.4 Report sent using the email

Starting from version 7.4.31, the configuration of emails with reports, can be done ***only*** by means of the web-app.

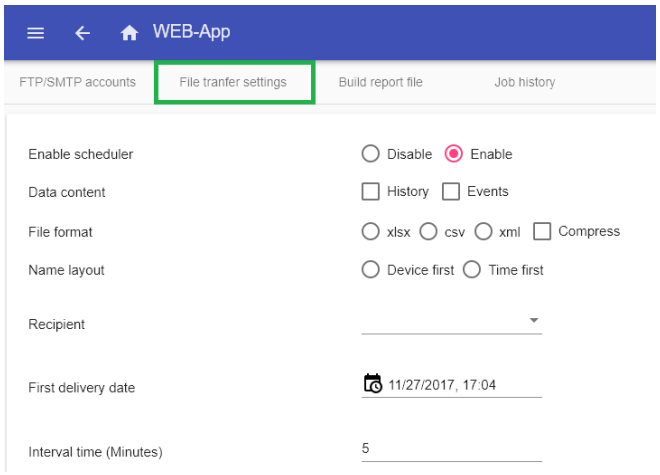
1) Open the report menu

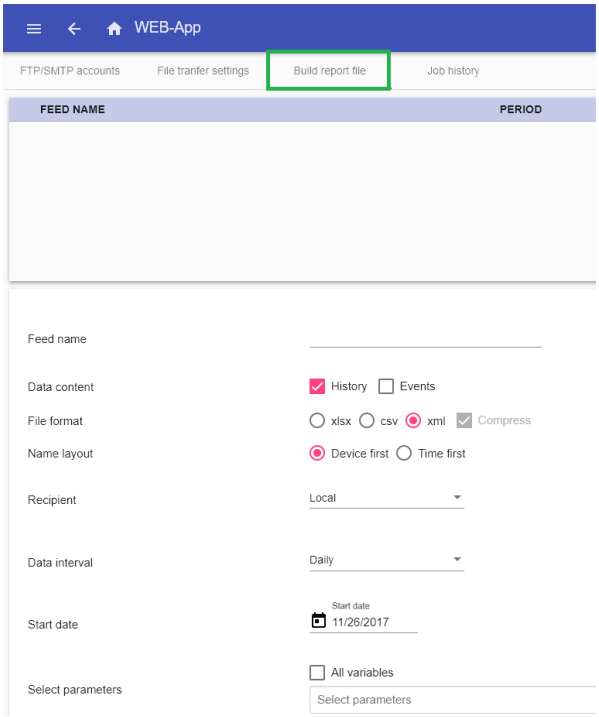


2) Configure the email account if it was not configured using the tool



3) Link the report to be sent





An Email function generated with a former version of the software is not converted, so the user has to do it manually.

2 Introduction

The UWP 3.0 tool has been developed for the configuration of the UWP 3.0, a programmable Linux embedded PC specially designed for home automation applications.
All demotic functions are represented by graphic symbols, and all function related parameters are set up locally in the PC, and then transferred to the UWP 3.0 via Ethernet. Some of the function parameters can be changed later via remote connection to the controller (web server, email, sms, Modbus,..) as described later in this manual.
Likewise, data from the UWP 3.0 can be uploaded and modified.

The PC does not need to be connected to a UWP 3.0 controller in order to make a configuration.

The figures in this manual may differ from the figures on your screen. This is not necessarily an error, but may be caused by revision differences.

The contents of this manual may be altered without notice.

2.1 Requirements

2.1.1 Minimum hardware requirements

- A Microsoft® Windows®-based PC
- Display with a resolution capability of minimum 1024x768 pixels
- 1 GB of disk-space
- An Ethernet-port and cable or SD-card reader or USB port 2.0 or higher

2.1.2 Software requirements

- Microsoft® Windows® 10/8.1/8/7/Vista (32 or 64-bit)
- Microsoft Dot-Net Framework 4.5

2.1.3 How to read the software version number

The UWP 3.0 tool revision number has the following structure:

Major	Minor	Sub minor	Revision

- **Major:** this identifies the main features of the software. It is incremented when new features are added or there are big changes in the existing ones.

- **Minor:** this identifies the version of the relevant *major* release and it is incremented when there are small new functions and bug fixing.

- **Sub minor:** this identifies the version of the relevant *major* release and it is incremented when there are bugs fixing

- **Revision:** this identifies the status of the release.
  1 *beta*: for internal use only
  2 *controlled beta:* to be shared with selected customers for field testing
  3 *final*: available for everybody on the Carlo Gavazzi website

3 Installation

The UWP 3.0 tool can be downloaded from Carlo Gavazzi's Product Selection website. To install the software you just have to double click on the *Setup.exe* file and follow the instructions on the screen.

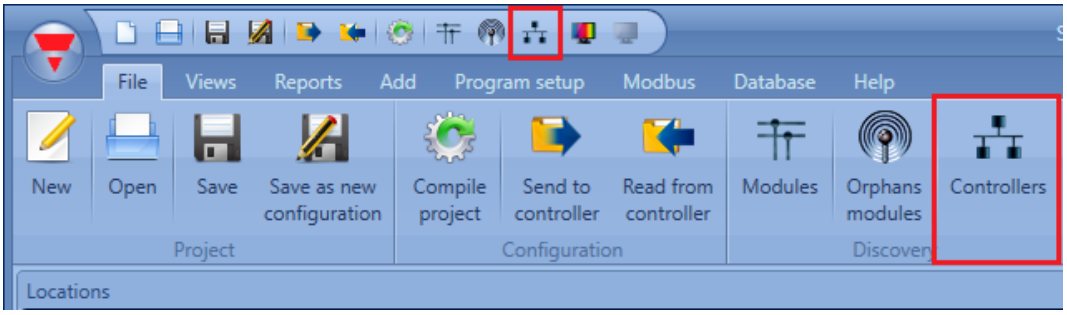
In order to get in contact with the UWP 3.0 controller, the user has several connection modes available:

via Ethernet connection
via Mini USB cable
via Modem connection

After a power on, the UWP 3.0 master unit is ready to work after about 1 minute. Only when the yellow BUS Led starts flashing is the master unit ready.

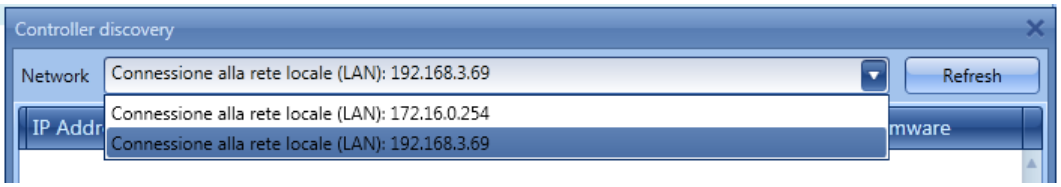
3.1 How to connect the master unit UWP 3.0 with Ethernet connection

To connect to a master unit UWP 3.0, the user has to click on the icon highlighted in red in the picture below: the UWP 3.0 tool will start the discovery of the UWP 3.0 connected to the Ethernet network.



Important note
If the PC is running the Windows Firewall or a Third party Firewall / Antivirus, make sure that the ports 52325, 10000, 10001, 10002, 80 and 443 are not blocked (input/output packets). These ports are used by the UWP 3.0 Tool to search for the master unit in the network and for communication. If a firewall blocks these ports, the UWP 3.0 Tool will not be able to find the controllers in the network or to use the Live Signals.
Be sure that the master unit's IP settings match the IP settings of the PC used: it must have the same IP class and the same net mask address.

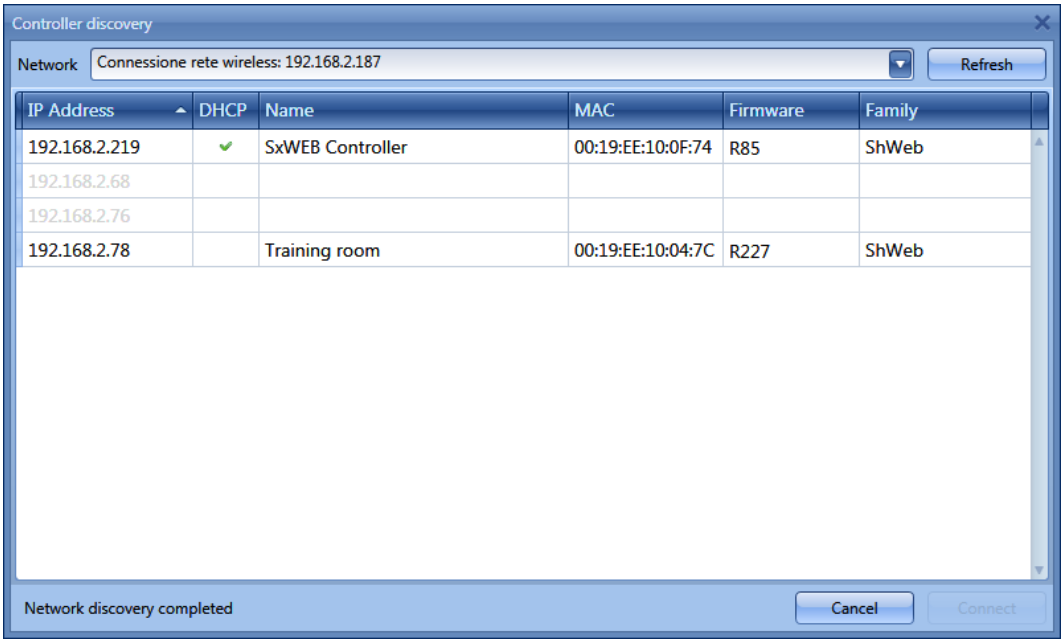
When the PC has more than one network card or has many IP addresses, it is possible to select the right network from the list (see picture below): it must be the same one as the UWP 3.0 is connected to.



When the icon marked in red is clicked, the UWP 3.0 Tool starts looking for the controller(s). If one or more units are found, a window will pop-up, allowing you to make a selection of which controller to connect to.

Select the required master unit on the list or *Cancel* if you do not want to get connected at this time.

After the user has pressed *Connect*, the red Led on the selected UWP 3.0 starts flashing and the connection will be established.



3.2 How to connect with a mini-USB cable

The UWP 3.0 controller can be connected to the PC by means of the *mini-B* port available on the front of the housing, protected by the front cover. Please refer to the *Hardware manual* for technical information.

The following items are required to obtain the connection:

A **USB 2.0 cable-Type-A / Mini-B** (this is not provided with the UWP 3.0 controller).

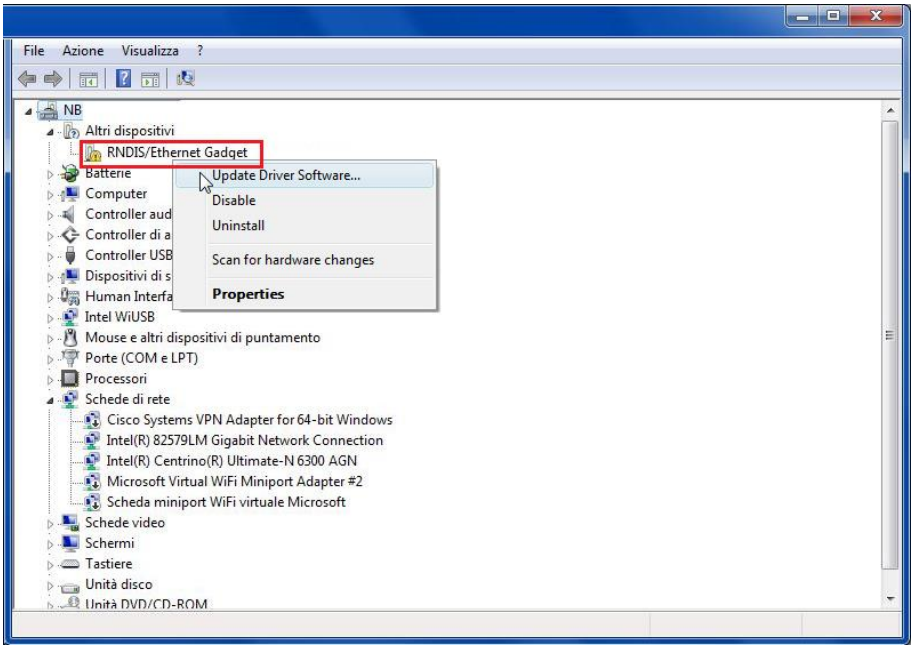
mini-USB driver.zip package available on www.productselection.net website in the UWP 3.0.

The driver installation procedure can change slightly depending on the operating system in the user's PC: please follow the instructions below.

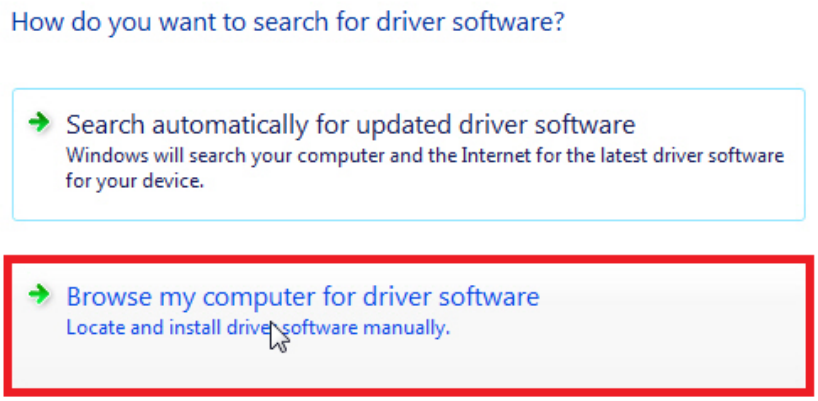
3.2.1 How to install the mini-USB driver for Windows 7 / Vista / XP

Open the archive "*mini-USB driver.zip*" and save the file l*inux.inf* in your PC, then install the driver by following the instructions below:

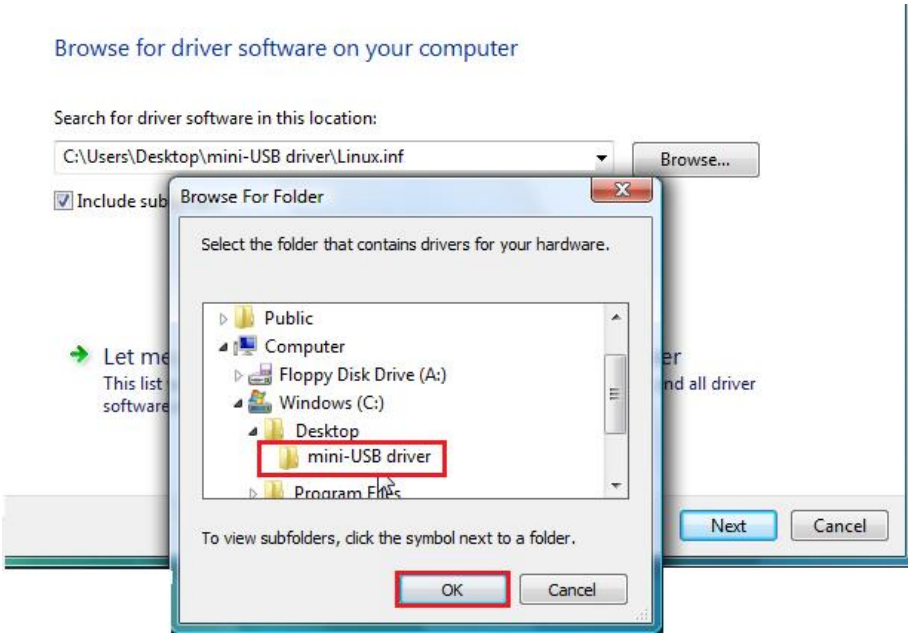
1. Plug the USB connector into a free USB port of the PC and the mini-USB connector into the mini-B port of the UWP 3.0
2. Go to *Control Panel* and open *Device Manager*.
3. Find the device *RNDIS/Ethernet Gadget*, right-click on it and select *Update Driver Software…,* as shown in the picture below.



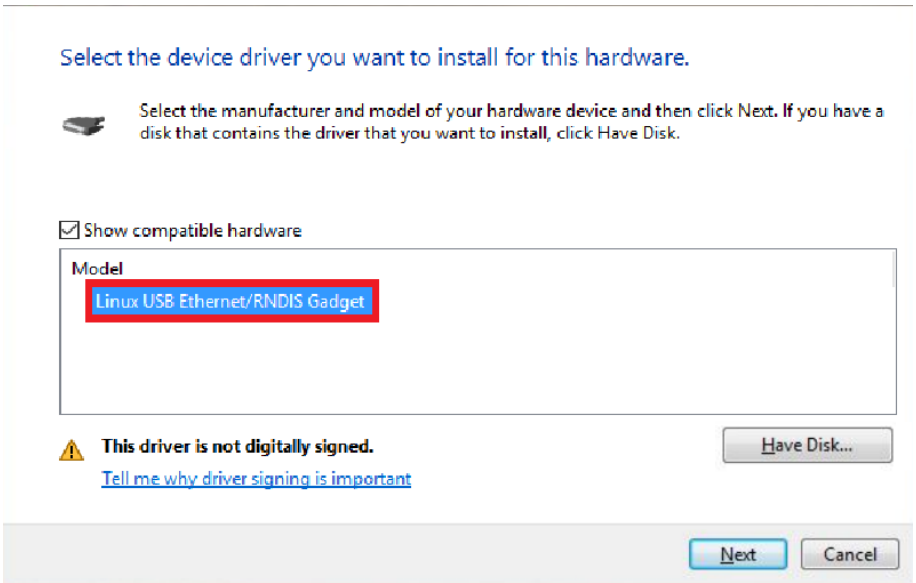
4. Select the *Browse my computer for driver software* option.



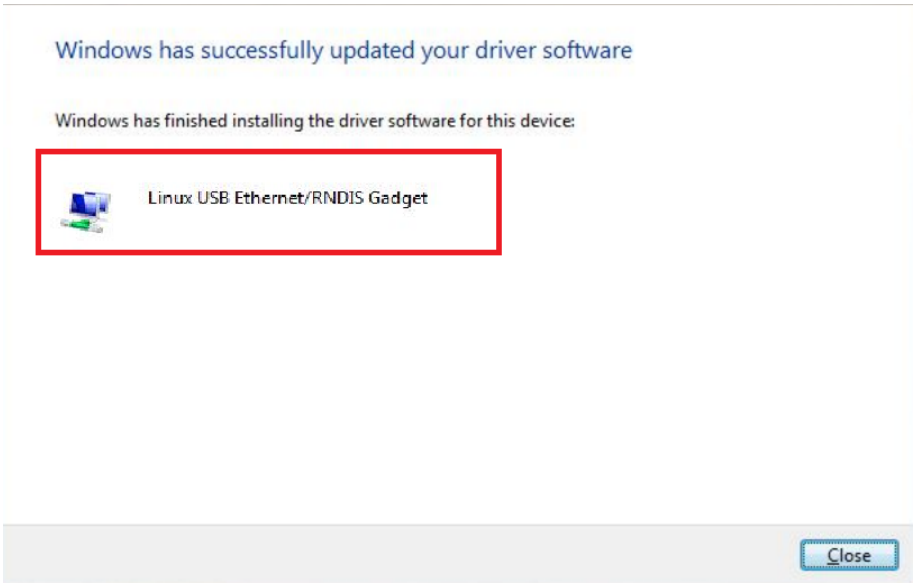
5. Browse for the *linux.inf* driver file and press *OK.*



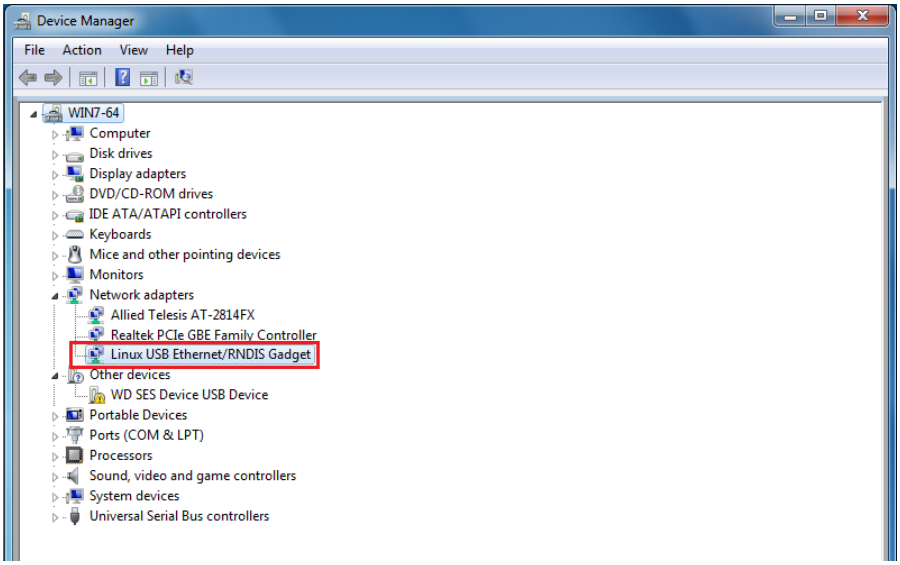
6. The driver *Linux USB Ethernet/RNDIS Gadget* will be detected: press *Next* to proceed.



7. The driver will be installed, as shown in the picture below.

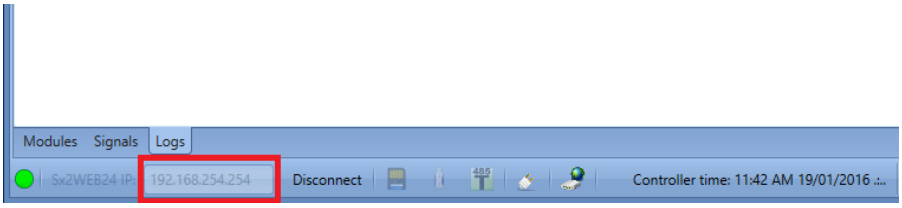


8. When the driver is installed, in the *Network adapter* category a virtual network board named *Linux USB Ethernet/RNDIS Gadget* will be added, as shown in the picture below. The driver automatically gives a dynamic IP address to the Controller/PC according to the actual IP of the PC.



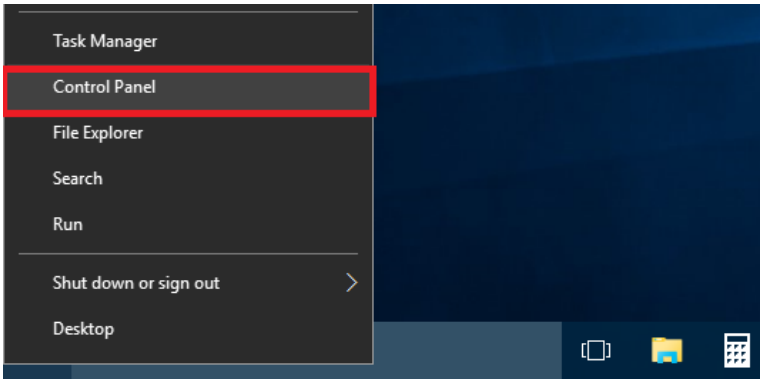
For example, if the PC has the IP address 192.168.0.10, the virtual board will be created with a new address 192.168.254.xxx and the IP address 192.168.254.254 will be assigned to the UWP 3.0 controller.

9. Insert the IP address 192.168.254.254 in the UWP 3.0 Tool and press *Connect,* as shown in the picture below:

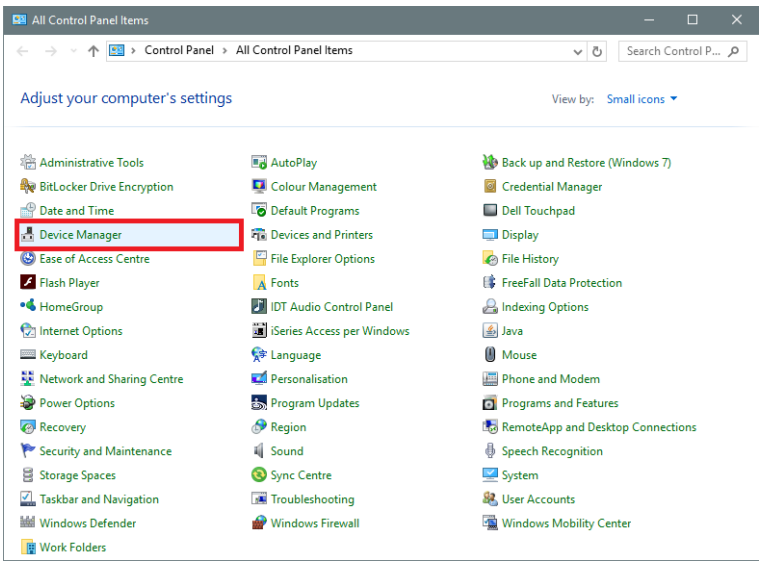


3.2.2 How to install the mini-USB driver for Windows 10 / 8.1 / 8

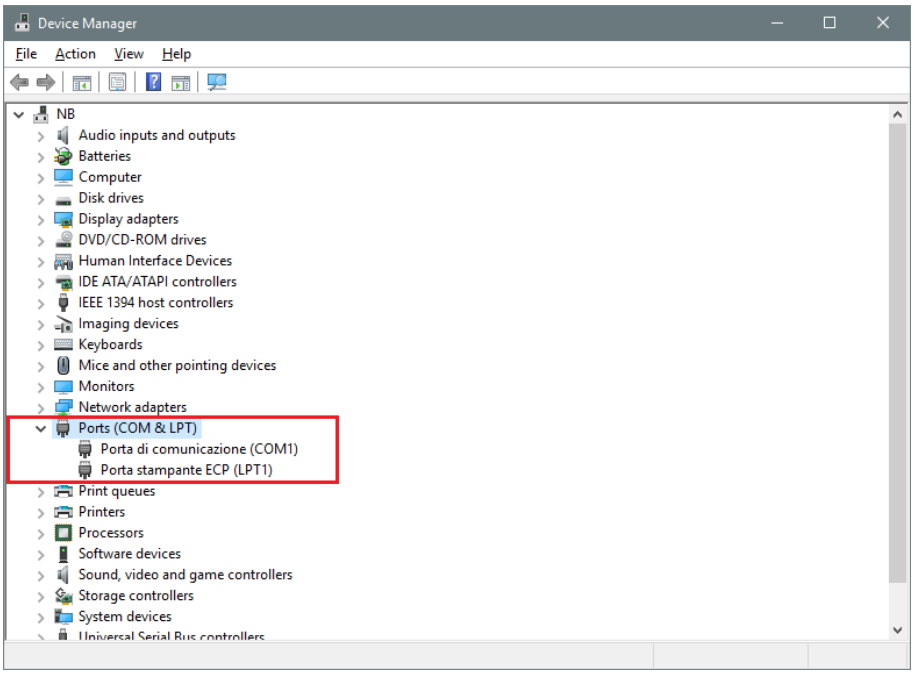
1. Open *Windows 10 Control Panel* by right-clicking on the *Start* button and click on *Control Panel*.



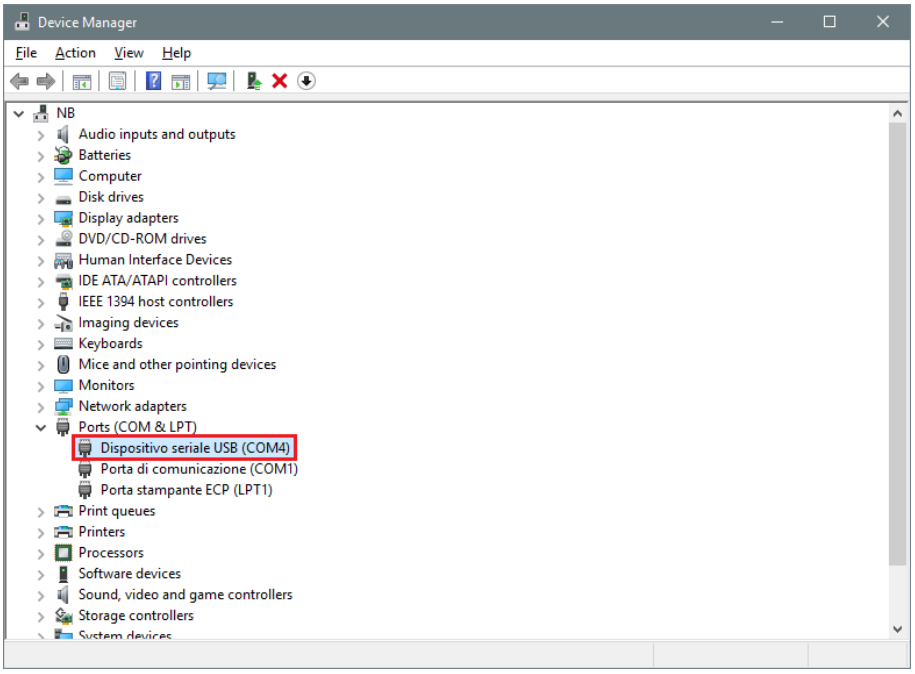
2. In *View by: Small icons*, click on *Device Manager.*



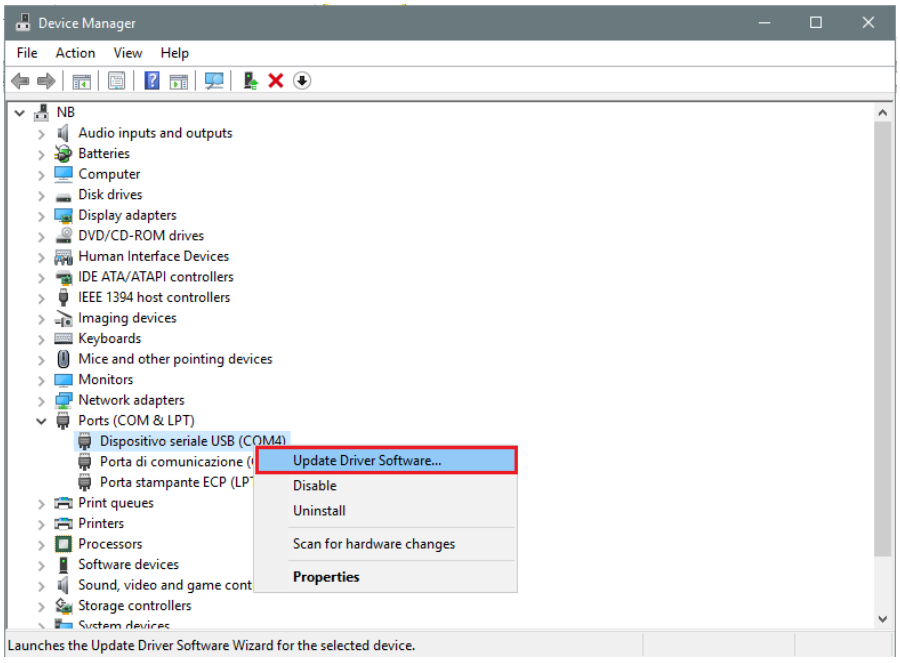
3. Before connecting the mini-USB cable to the PC and to the UWP 3.0 controller, in the list of hardware categories double-click on the category Ports (COM & LPT) and take note of the serial communication (COM) ports in use. In the example shown below, the only serial communication port is ***COM1***.



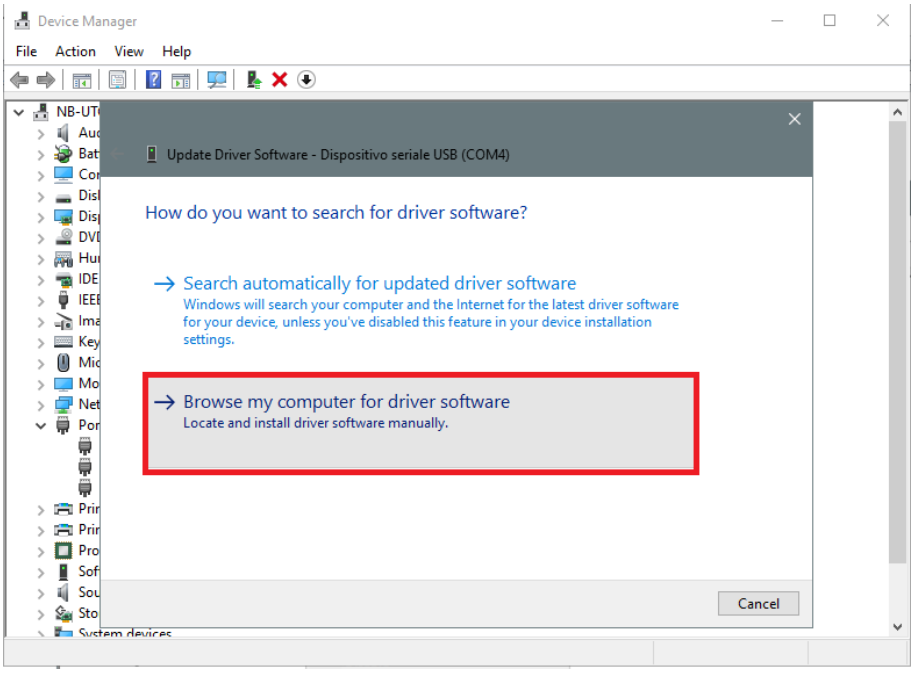
4. Plug the mini-USB cable into the PC and into the UWP 3.0 controller. A new serial communication port (COM*x*) will automatically be added after a few seconds. In the example shown below the new port added is ***COM4.***



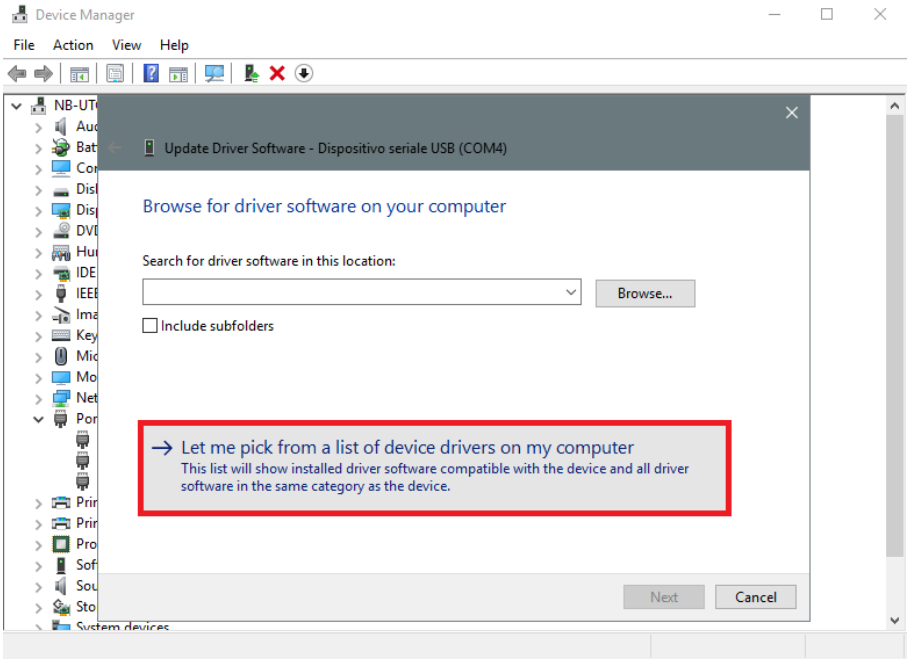
5. Right-click on the device and select *Update Driver Software*…



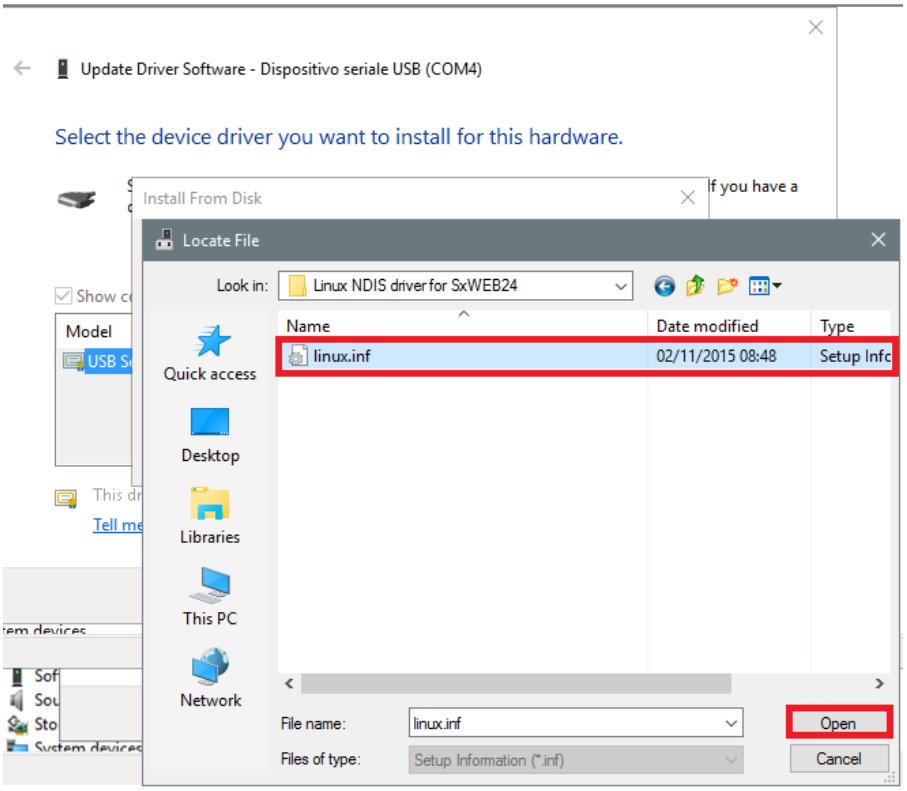
6. In the pop-up window, select the second option, *Browse my computer for driver software*



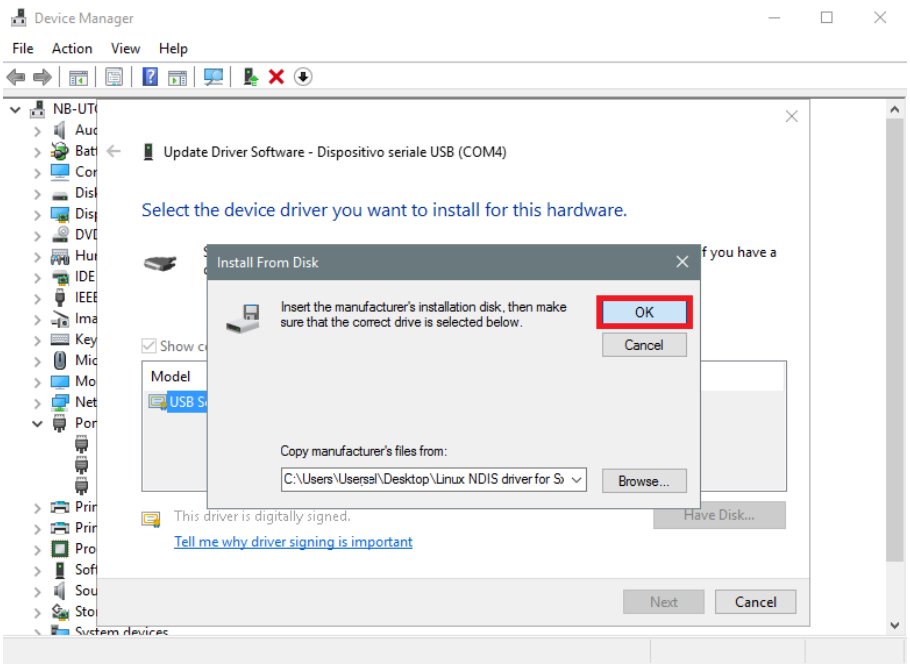
7. In the next page, select the option *Let me pick from a list of device drivers on my computer*



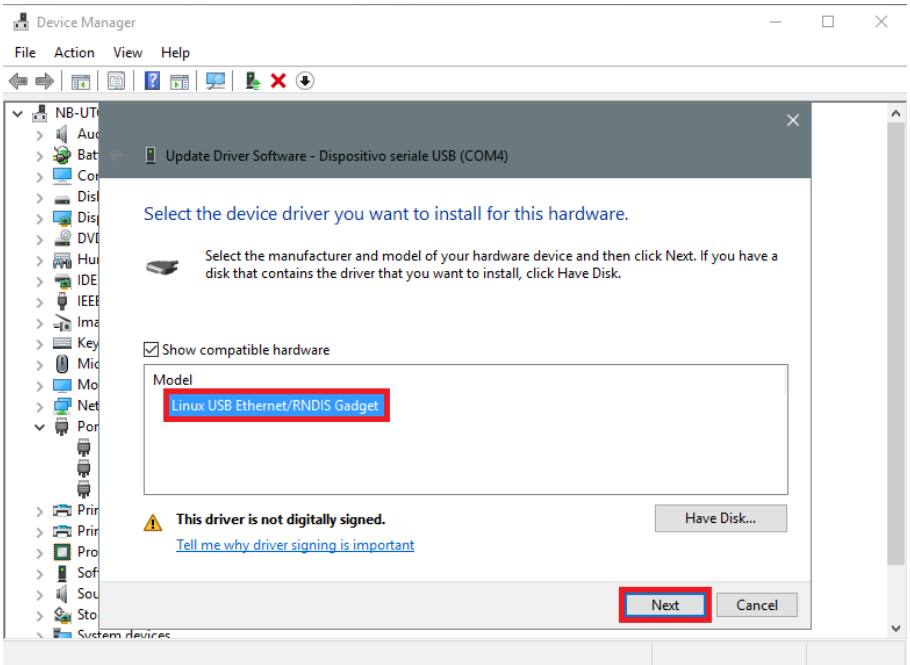
8. Browse for the *linux.inf* file, save it in the PC and press *Open*



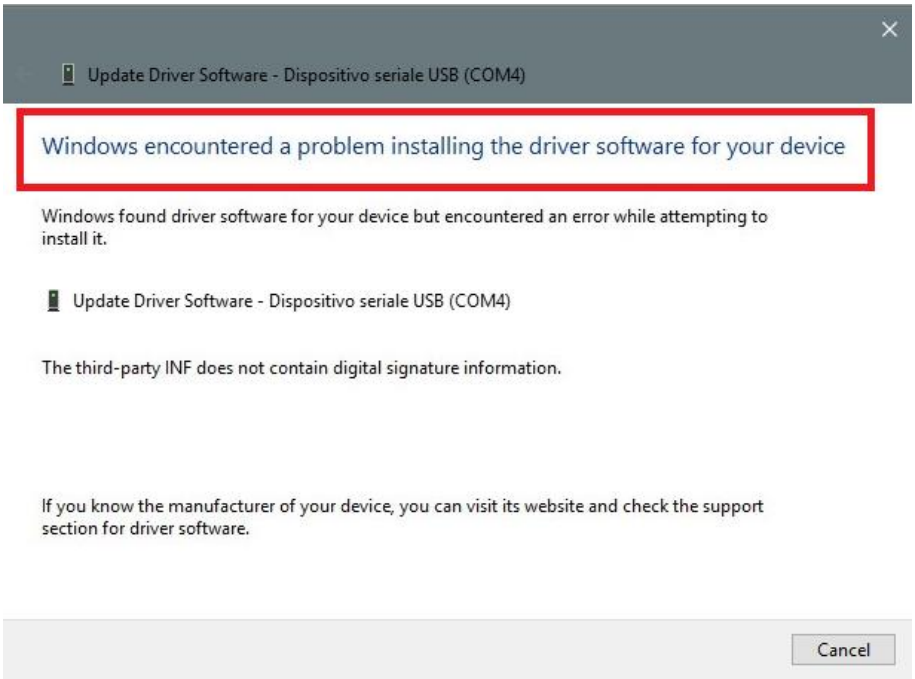
9. Press *OK* in the next window, as shown in the picture below



10. The *Linux USB Ethernet/RNDIS Gadget* is ready to be installed. Click on *Next* to continue the installation.



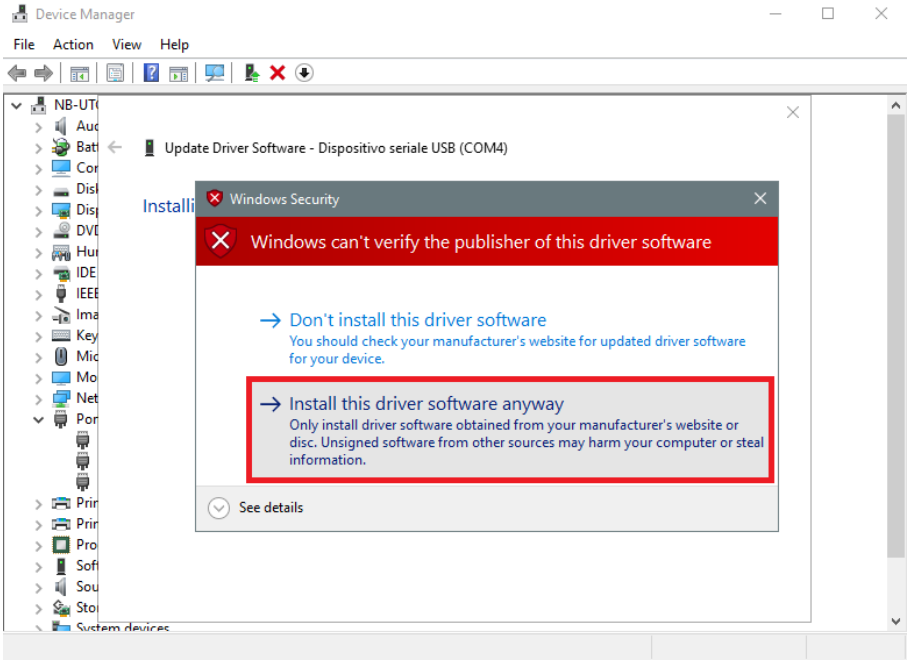
Follow the instructions below to disable driver signature enforcement **ONLY** if the user gets the error shown in the picture below, otherwise go to step 11.



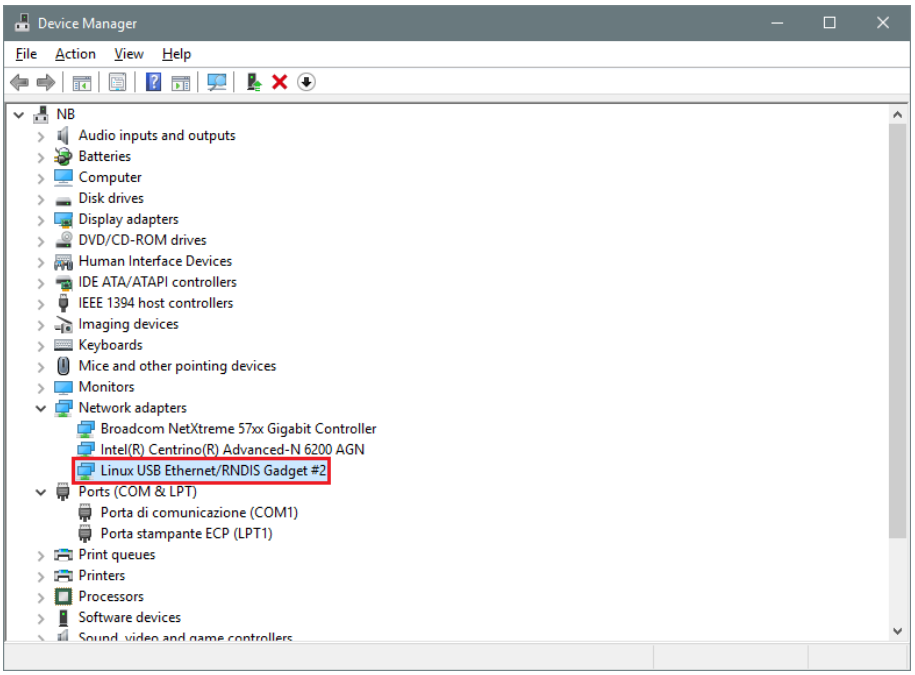
Windows 10 enforces driver signatures by default. This can be disabled to install drivers that are not digitally signed. Please refer to the following steps to disable driver signature enforcement.

1. Click the **Start** **menu** and select **Settings**.
2. Click **Update and Security**.
3. Click on **Recovery**.
4. Click **Restart now** under **Advanced Startup**.
5. Click **Troubleshoot**.
6. Click **Advanced options**.
7. Click **Startup Settings**.
8. Click on **Restart**.
9. On the Startup Settings screen press 7 or F7 to disable driver signature enforcement.
10. The computer will restart and the user will be able to install non-digitally signed drivers.

11. **N.B. Windows 10 does not detect the Digital signature, so to install the driver correctly, click on *Install this driver software anyway.***



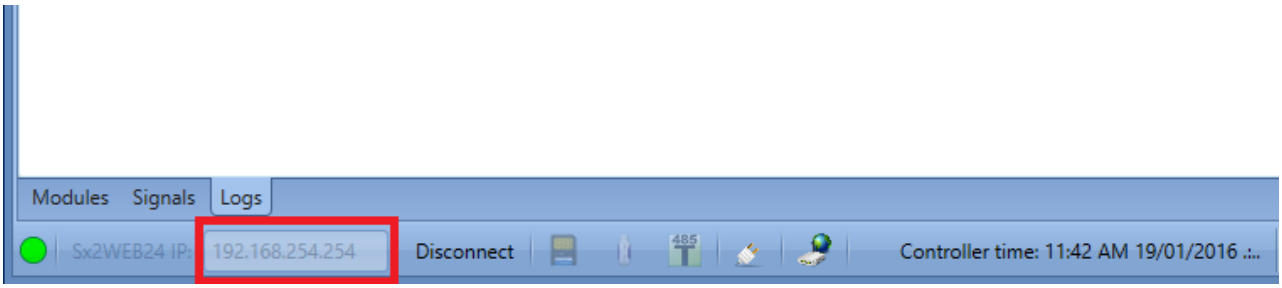
12. When the driver has been installed, in the *Network adapter* category a virtual network board named *Linux USB Ethernet/RNDIS Gadget #2 will* be added, as shown in the picture below.



13. The driver automatically gives a dynamic IP to the Controller/PC according to the actual IP of the PC.

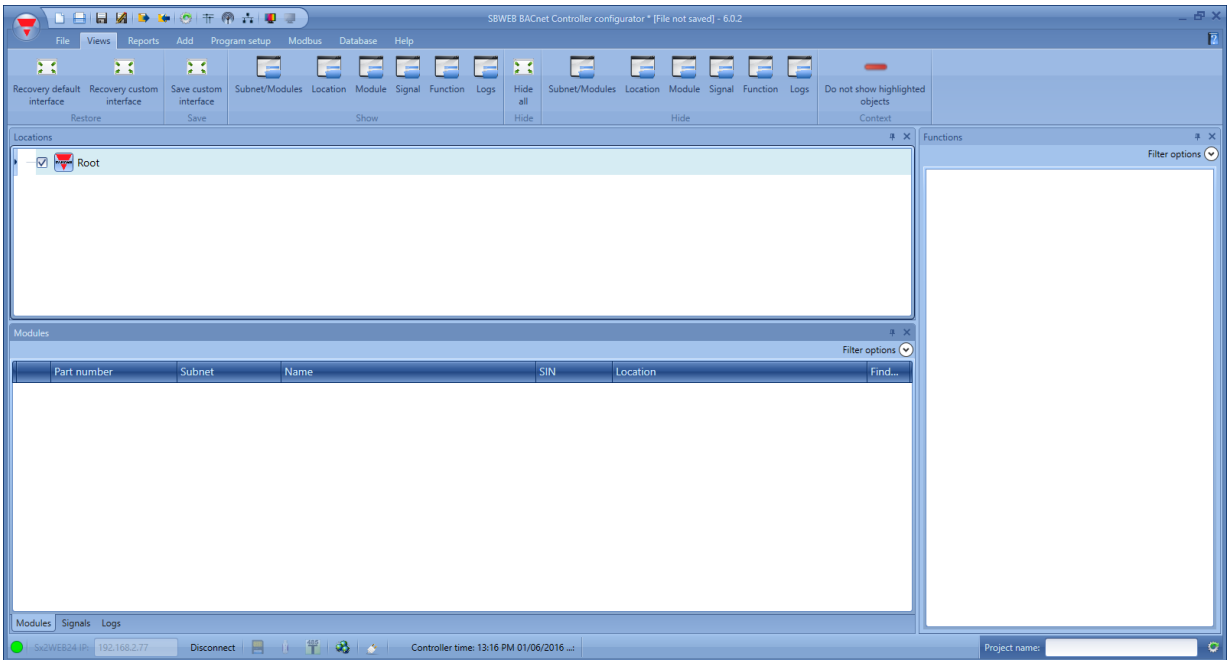
For example, if the PC has the IP 192.168.0.10, the virtual board will be created with a new address 192.168.254.xxx and the address 192.168.254.254 will be assigned to the UWP 3.0 controller.

The user can type the IP address **192.168.254.254** in the address bar and click *Connect*. The green circle icon appears in the left part of the address bar when the connection is established, as shown in the picture below.



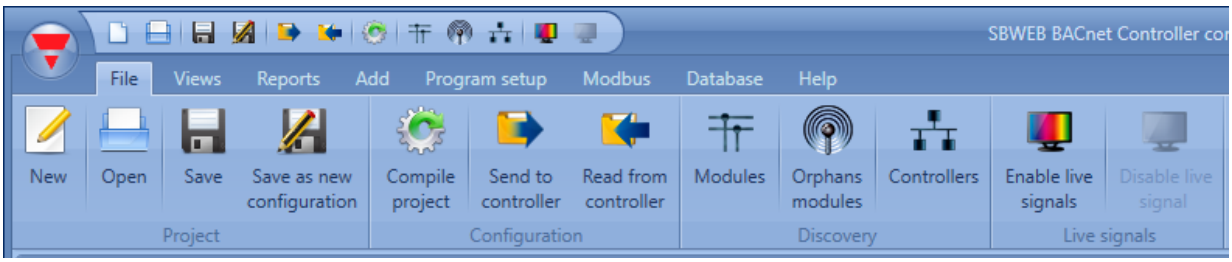
4 User interface

When the UWP 3.0 tool starts, the following window appears:



The user interface uses the standard ribbon tool often used by a lot of software nowadays. To access a ribbon, just click on the relevant menu.

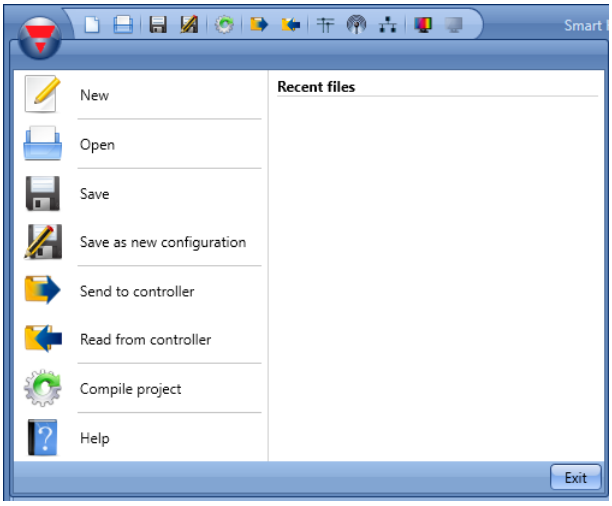
4.1 File Menu



In the File menu the user can create a new project, open an already existing one or save it as in a standard menu File.
In addition, some functions strictly related to the connection with UWP 3.0s are available and listed here:

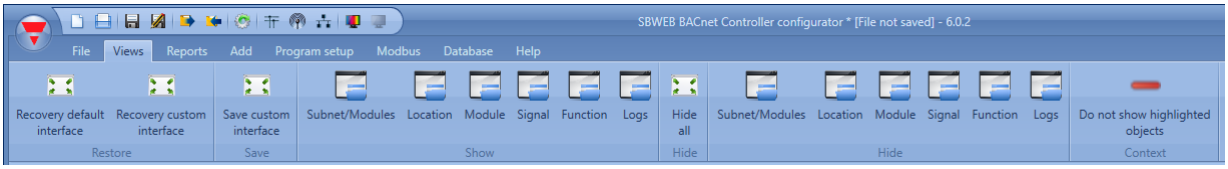
- Compiling a project
- Uploading/downloading of a project
- Discovery functions of the UWP 3.0s and modules connected in the network.
- Enabling/Disabling of Live signal monitoring

The user can access the menu file either with the quick menu in the upper part of the window on the right of the red Carlo Gavazzi triangle or by clicking on the triangle.

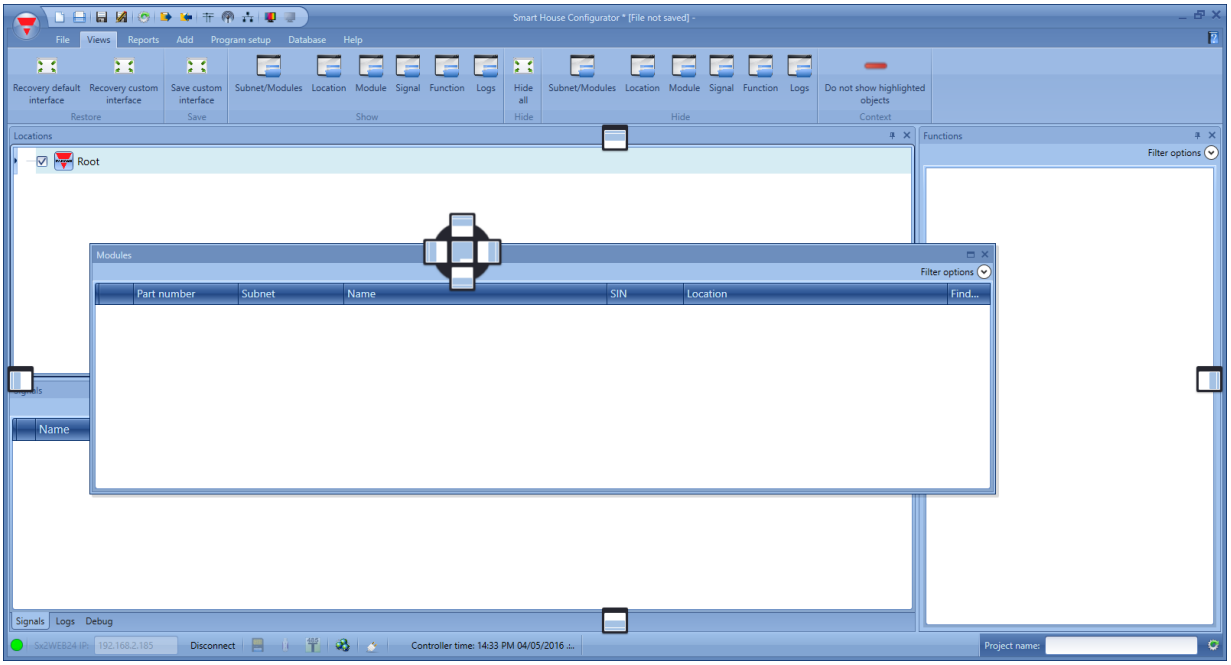


4.2 View Menu

In the View menu, the user can configure the preferred position of the windows relevant to Locations, Modules, Signals, Functions and Subnet. It can also remove the contents view.



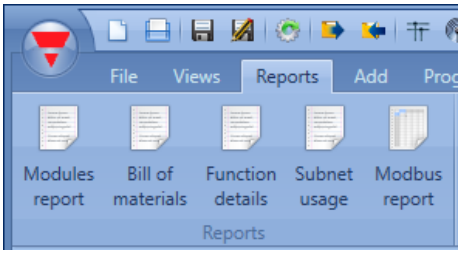
The windows are floating and can be positioned by using the five docking areas shown in the figure below. The position of the windows can be saved.



4.3 Reports menu

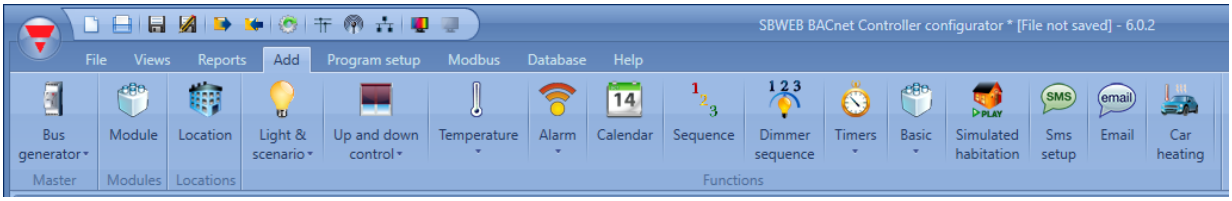
In the reports menu, five different kinds of reports can be created, saved and /or, printed.
The user can select from:

- *Modules list*: the complete list of modules is shown.
- *Bill of materials*: the list of modules is organized by type of modules.
- *Function details*: each function is described with the details of used signals.
- *Subnet usage*: this report indicates the number of used signals and the total current consumption.
- *Modbus TCP/IP report:* in this report the Modbus map of the project is reported.



4.4 Add menu

In the Add menu the user can select what to add: bus extension, modules, locations and functions.

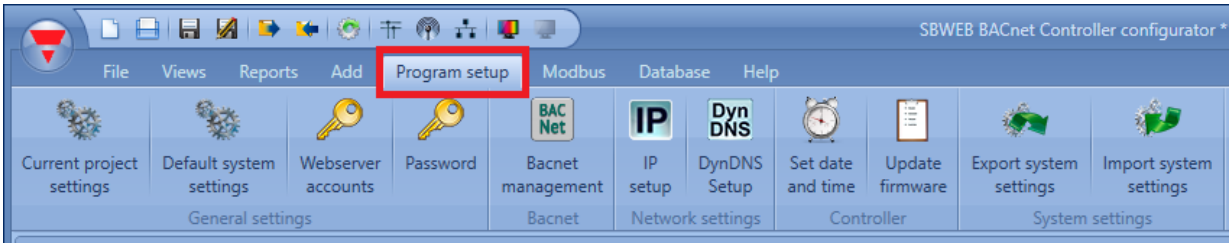


Bus generators: if a new Dupline network is needed, a new bus extension module has to be added.
Module: a new module is added (light switch, PIR sensor, output module, etc…)
Location: The user has to define floors, rooms or any other type of location to have a clear structure of the installation.
Functions: some predefined functions can be defined and configured by means of the wizard tool. The predefined functions are:

- Lights & scenario
- Up and down control: for controlling blinds, curtains, windows
- Temperature System Functions
- Alarm: intruder, smoke, water, siren
- Calendar: can execute activities during the year
- Sequence: executes a list of chosen functions
- Dimmer sequence
- Timers
- Basic functions: this section contains Counter, Logic Gates, Analogue comparator, Mathematical function, Analogue output, Hour counter
- Simulated habitation: records and then plays back the light activations
- SMS setup: manages the SMS for the remote control of functions
- Email
- Car heating

4.5 Program setup menu

In the *Program setup* menu, the user can configure the settings relevant to a specific project, the general settings of the installer, the IP address and time and date of the UWP 3.0. The user can also update the firmware and configure the webserver and the password to access the controller.

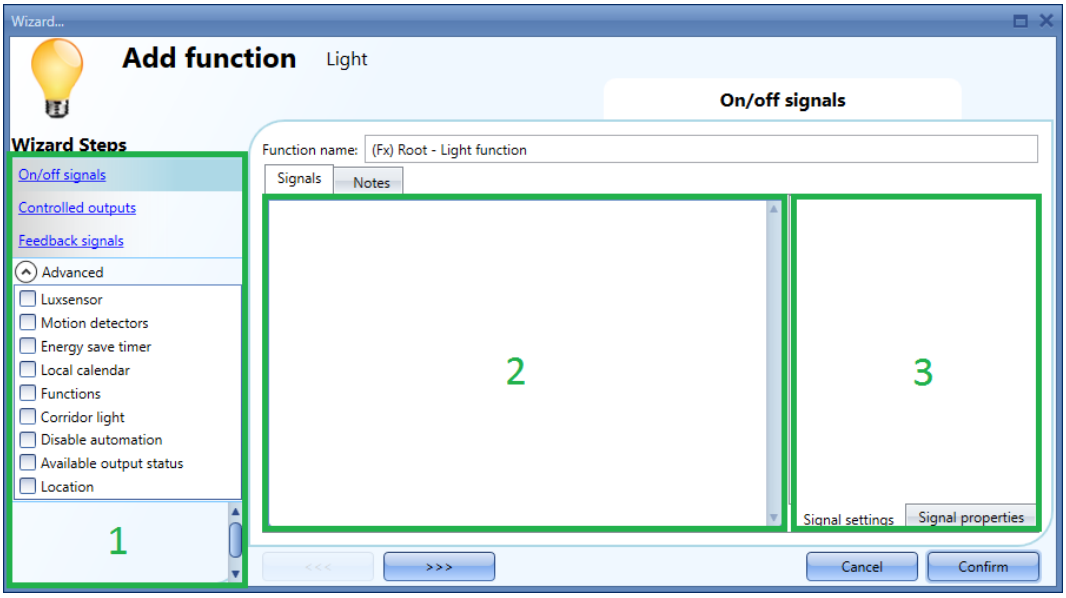


5 Project structure

In a project the user has to define the locations of the house/building, add the required modules (light switches, movement detectors, output modules, …) and place them in the relevant location, then create the functions.
Location, *modules* and *functions* are the pillars of the project structures: they are described in detail in the following paragraphs.

5.1 Wizard

Each object, whichever type it is, is created and configured by means of the *Wizard* tool.



The wizard is a tool that drives the user in the configuration of an object, guiding them in small steps to the complete setup of a module, location or function. The aim of the wizard is to reduce the effort of understanding the complete process of a configuration, making it easy and fast. The different steps can be filled in one by one just by clicking on the *">>>"* button and going through all of them, or by clicking on those required visualized in blue in the area on the right (Area 1).

In the picture above an example of the wizard tool divided into three areas is shown.

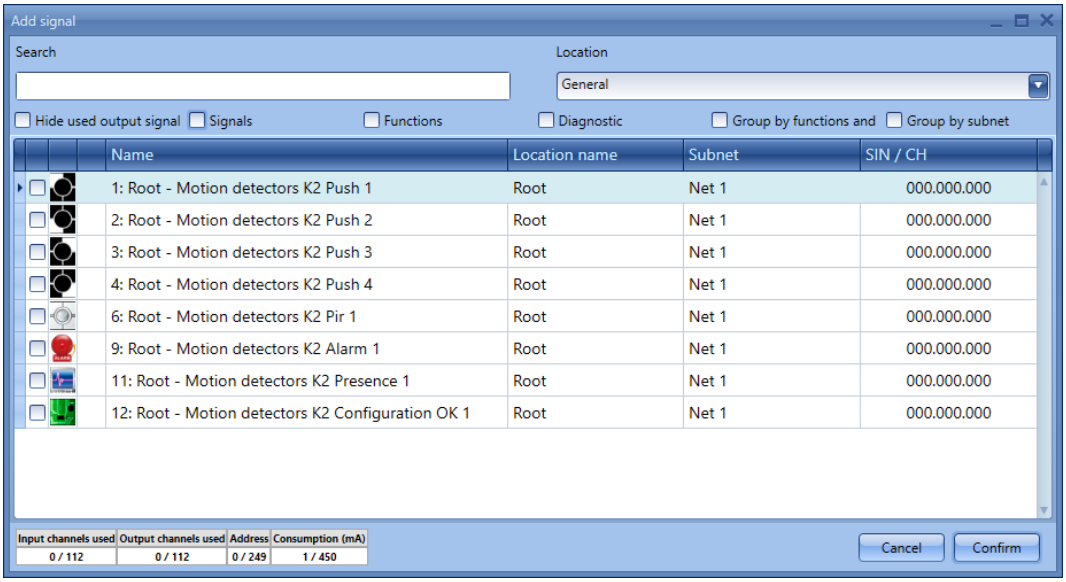
5.1.1 Area 1

Area1 is divided into two sections. The one on top contains the basic signals/settings the user has to insert/define to create the object ('object' is a general word to indicate location, modules or functions). For example, in a light function the basic signals are the input and the output signals, which are the minimum settings required to create this kind of function. In general, most functions show the input and output signals in the basic settings.

In the lower section the "Advanced" functionalities are editable: if they are not enabled they are hidden to the user in order to give an easy user interface to the not-so-skilled installer.
The list of Advanced functionalities will appear: tick the relevant box to enable the one required. For example, if the Lux sensor is to be used, click on this. Once the complete list disappears, by clicking again on *Advanced*, the enabled one/ones will appear for quick access.

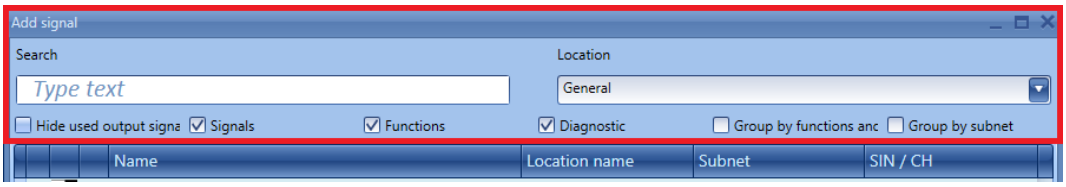
5.1.2 Area 2

This is the area where signals have to be added. By double clicking on this area, the following window will appear. All the available signals and functions will be shown, giving the user the possibility of selecting one or more of them by checking the relevant row and clicking on *Confirm*.



5.1.2.1 Signal window filters

Different filters can be used according to the icons at the top of the window, as shown in the red rectangle in the picture below:

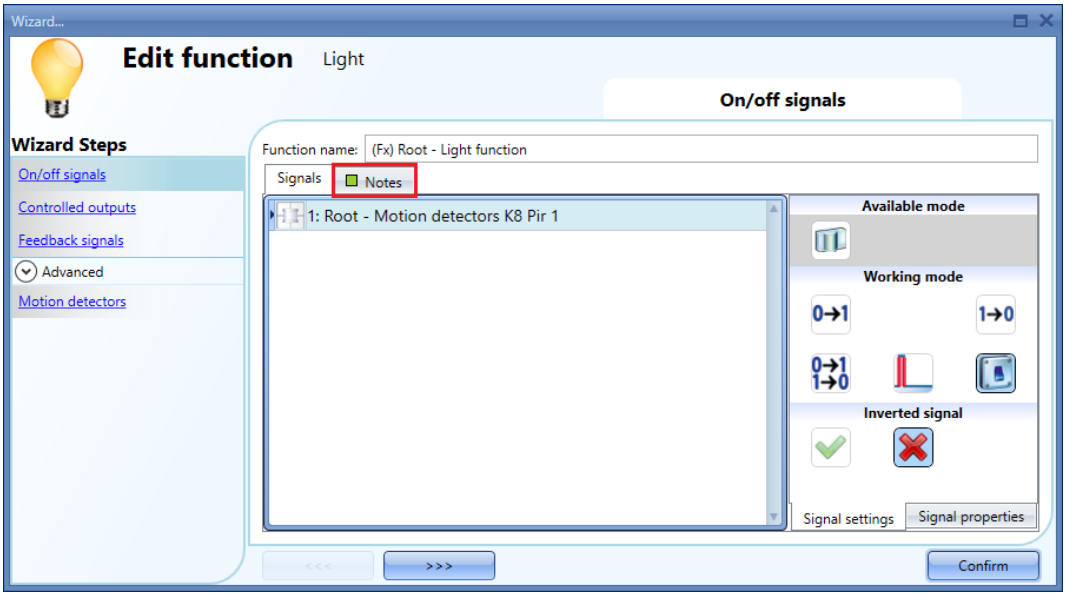


The available filters are listed in the tables below.

Search box	The results will include all words that contain the specified string related to the signals from the modules available in the current project.
Location	Only the modules that are part of the selected location will be shown, i.e. the modules installed in *Cabinet.*
Hide used output signals	The output signals already used in at least one function, will be hidden
Signals	All the signals from the modules available in the current project will be shown in the signals window.
Functions	All the functions available in the current project will be shown in the signals window.
Diagnostics	All the diagnostic signals from the modules available in the current project will be shown in the signals window.
Group by functions and modules	The signals and functions are grouped according to the module they belong to.
Group by subnet	The signals are grouped according to the network they belong to (e.g. SH2MCG24, SH2WBU230N, SH2DUG24 or COM port).

5.1.3 Area 3

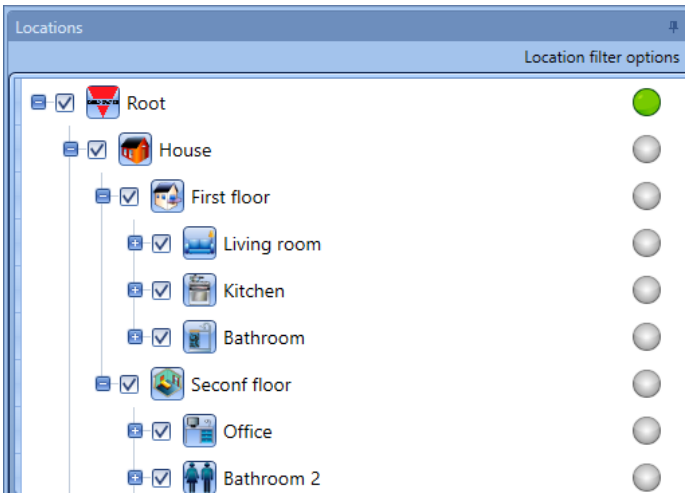
This is the area where signals are configured. The user can modify the signal settings and view the properties of each signal (for example the long activation time of one light switch, or the sensitivity of a motion detector).



If a note is added in the *Notes* tab window, a small green rectangle will appear to indicate this. By selecting this with the mouse, a tool tip is shown with the added note.

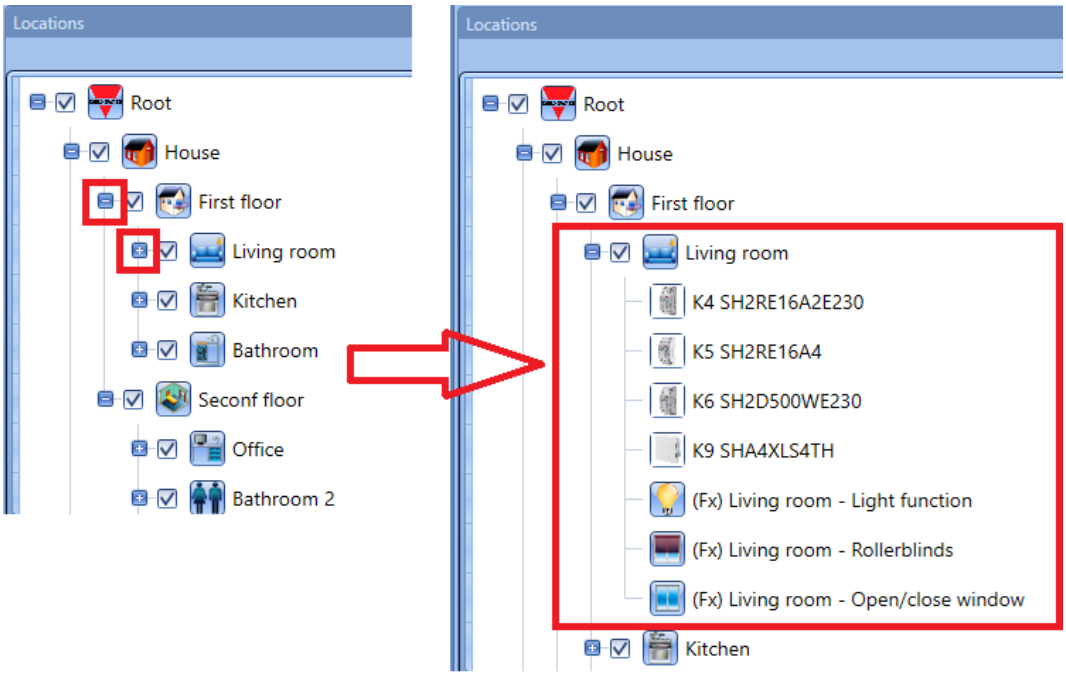
6 Locations

In the *Locations* window the user can define the structure of the project, starting from the locations where modules and functions have to be placed: the user has to define floors, rooms or any other type of location to have a clear structure of the installation.



From the *Locations* window, the user can select which locations have to be shown or not: all the modules and functions related to the locations will be hidden/shown depending on the locations selected in the project tree.

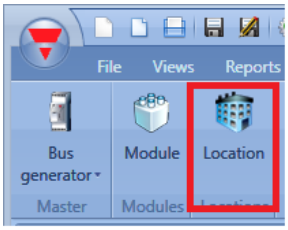
Click on the *plus sign* (+) to expand a Location and to see its sub-locations and modules:

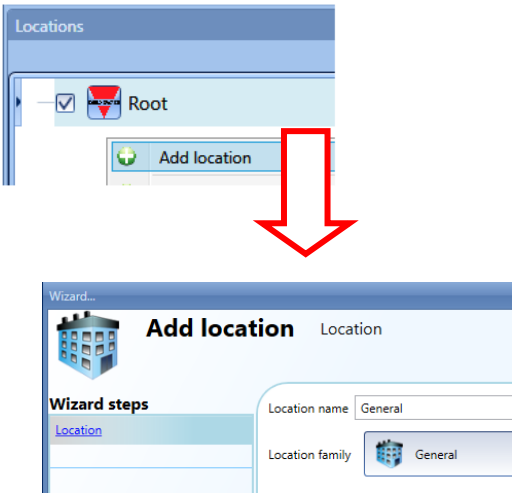


6.1 How to add a location

There are different ways to create a location tree:

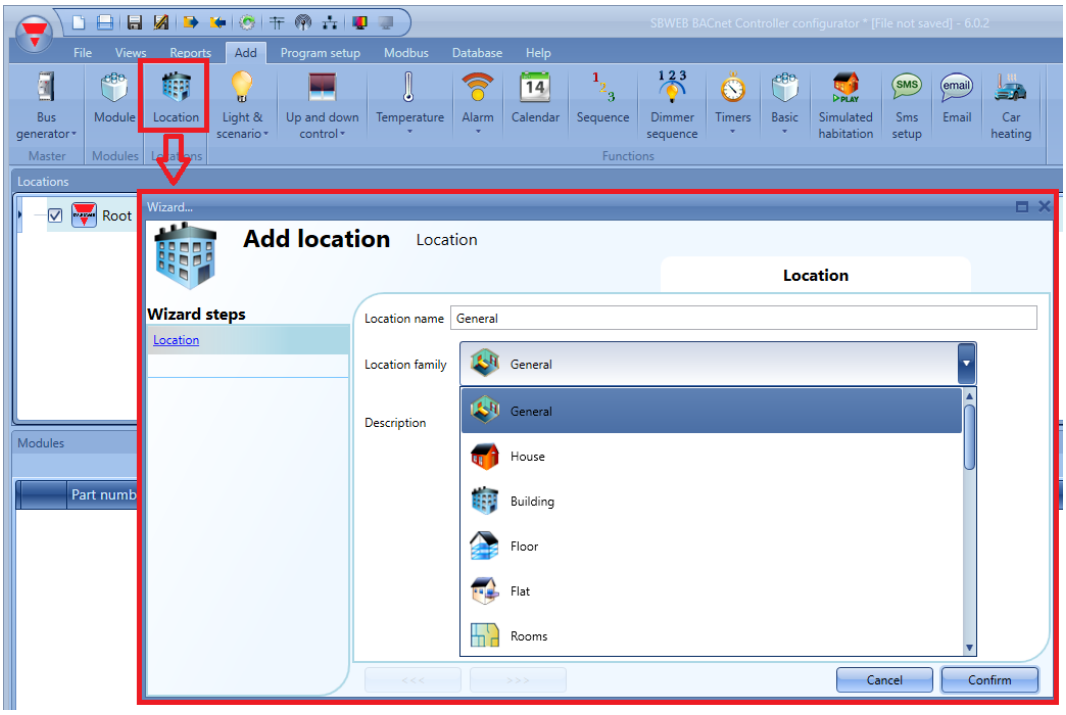
1) Starting from the *Add* menu from the top menu or by right clicking in the *Location* window



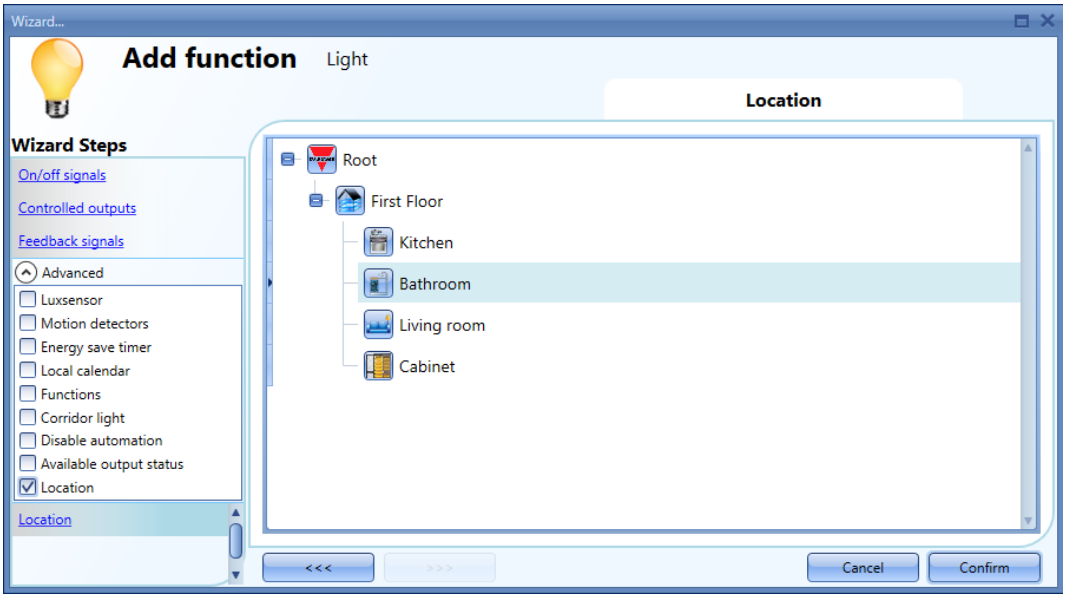


When the user inserts a new location, the following Wizard window will appear.
With this window the user can select the name of the *Location* and the *Location family*, and a short description of the *Location* can be added.

When the *Confirm* button is pressed, the Location is inserted into the *Locations* tree.

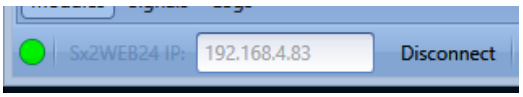


The Location field is available for all functions of the UWP 3.0 Tool, under the *Advanced* menu of the Wizard, as shown below:

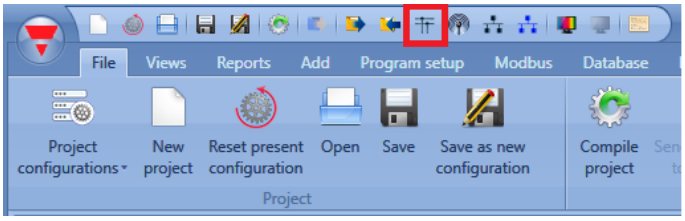


2) In a faster way, the location tree can also be generated starting from the *Discovery manager window:* to open this, follow these steps.

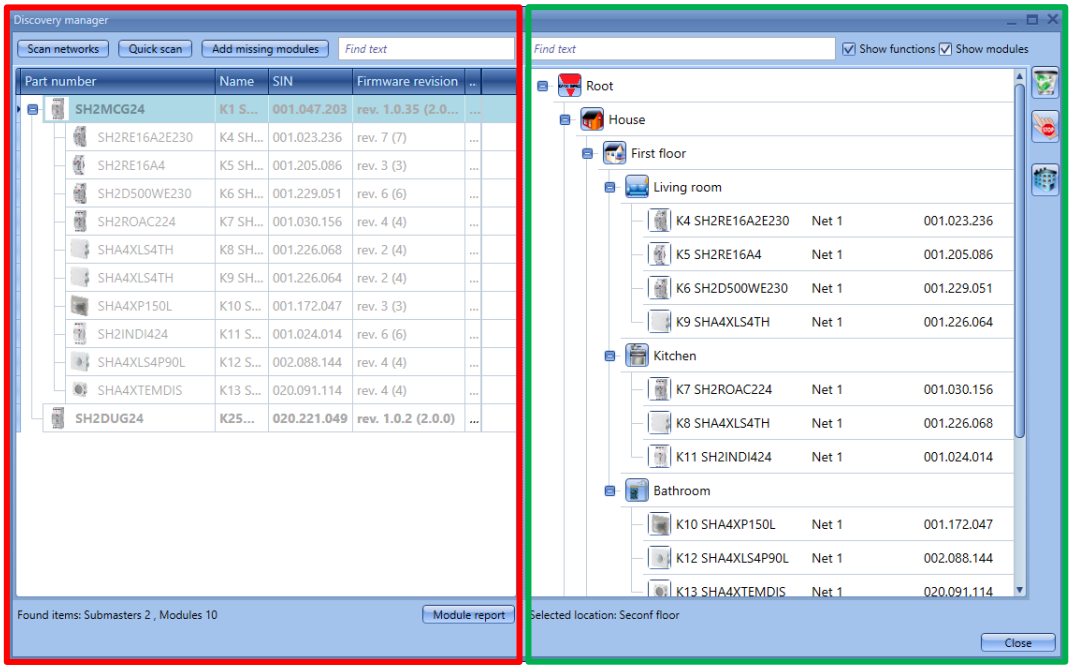
a) Connect to the controller



b) Launch a discovery of the Dupline module



c) The *Discovery manager* window will appear:



On the left of the window (marked in red), all the modules connected to the bus are prompted once the *Scan networks* push button is pressed.

The *Quick scan* button will list only the modules of the configuration written into the UWP 3.0, while the *Scan networks* button will list all the modules connected to the controller.

On the right, the location tree is shown. To generate this, only three push buttons have to be used:

To delete a location and all its objects, the push button with the basket has to be used

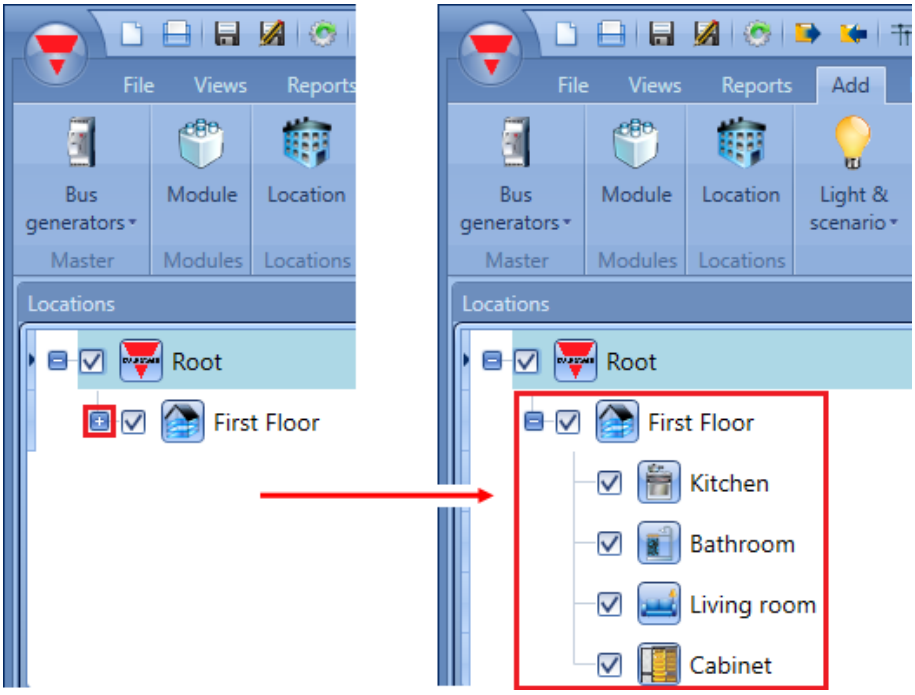
This stops any running activity such as a network discovery or an automatic addressing

This adds any type of locations. It is not enabled if a module is selected.

6.2 How to manage Locations

From the Location window, the user can select which locations have to be shown or not: all the modules and functions related to the locations will be hidden/shown depending on the locations selected in the project tree.

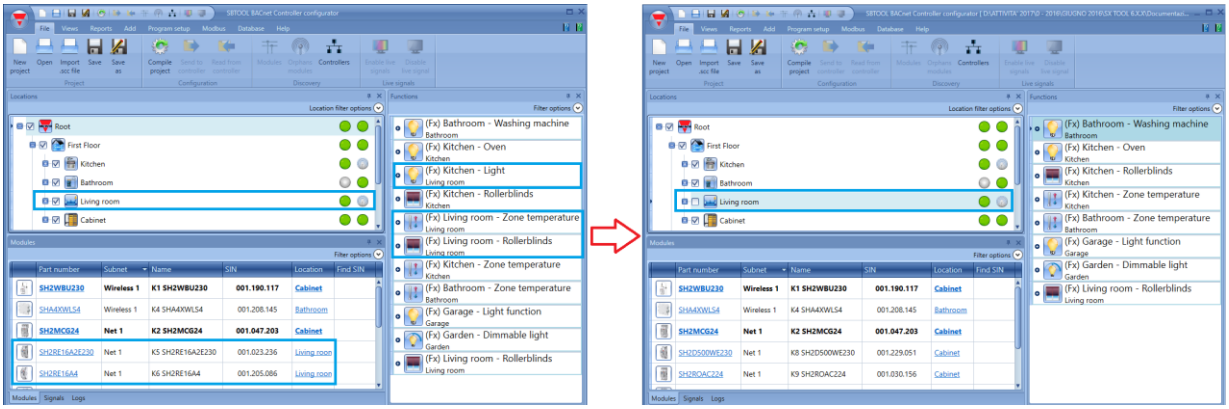
Click on the *plus sign* (+) to expand a Location and to see its sub-locations; in the example shown below, the rooms *Kitchen, Bathroom, Livingroom* and *Cabinet* are parts of the *First Floor*.



The checked box next to each Location indicates that it will be shown by default. The user can choose which Locations to show or to hide by clicking on the relevant symbol.

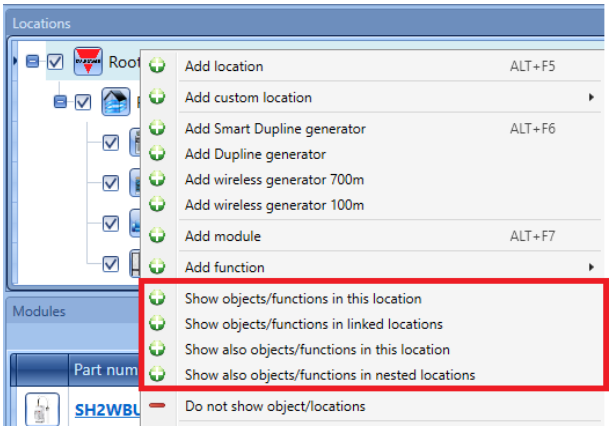
- To show the modules and functions associated to a location, the check box must remain checked
- To hide the modules and functions associated to a location, the user has to uncheck the location. The objects associated to its sub-locations will also be hidden.

Example
In the picture below, the location *Livingroom* is on 'unselect': all the relevant modules and functions are hidden.



6.3 Functions and modules view

By a right click on a location the contextual menu will appear: the user can choose one of the available view options to easily find which are the objects in the selected location.

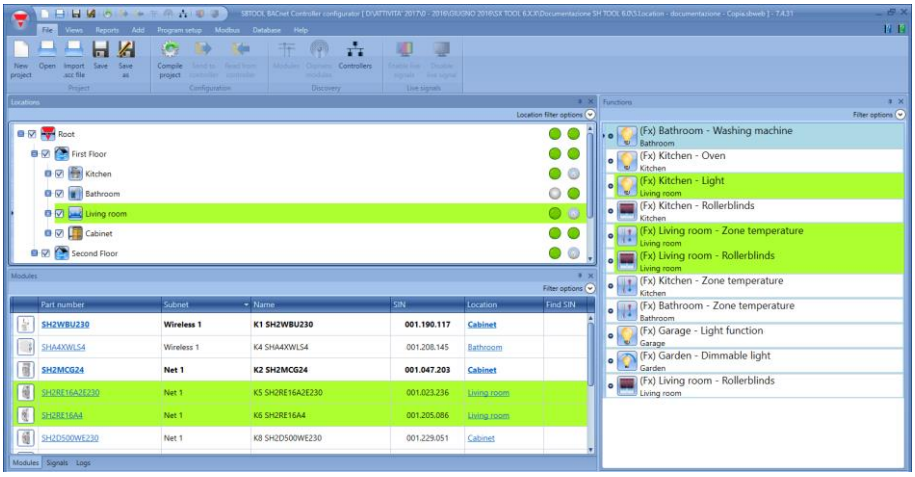


Show objects/functions in this location
The modules and functions in the location are highlighted in green.

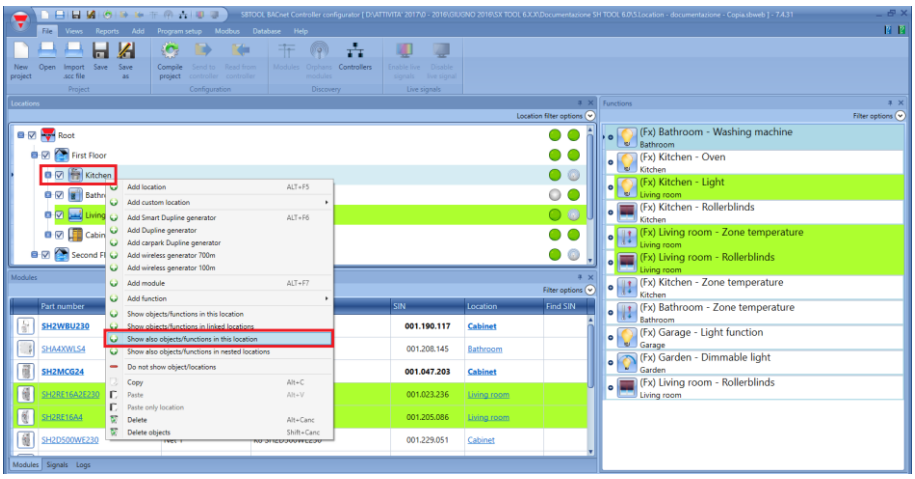
Show objects/functions in linked locations
The modules and functions in all the locations or sub-locations linked to the one selected are highlighted in green.
Show also objects/functions in this location
The modules and functions used in more locations are highlighted in green.

Show also objects/functions in nested locations
The modules and functions in the locations nested into the one selected are highlighted in green.

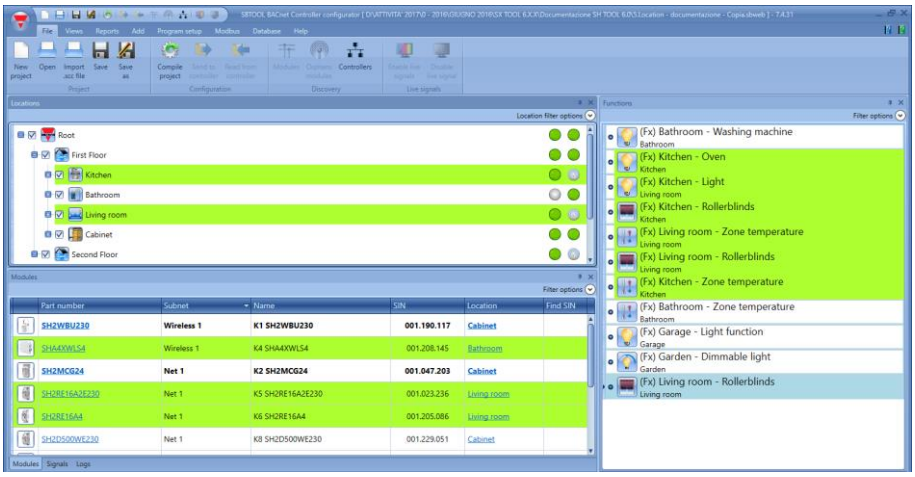
The UWP 3.0 Tool automatically highlights the location and all the modules that are used in the selected location.



Example
When more locations must be highlighted, the user can select the option *Show also objects/functions in this location*. In the example shown below, the objects and functions related to Livingroom are already shown; if the user right-clicks on *Kitchen* and selects *Show also objects/functions in this location*,



All the modules and functions that are part of *Kitchen* will be shown, together with the modules and functions of Livingroom, as shown in the picture below:



6.4 Delete operation

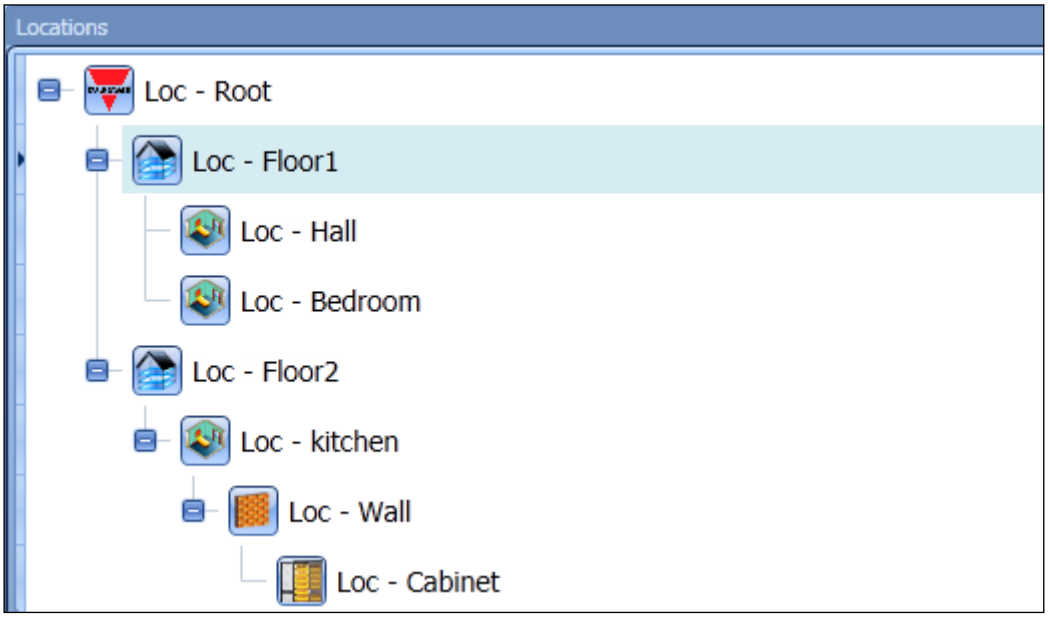
When the user selects the total delete action (delete), a confirm message will be displayed on the screen so that the user can proceed with the deleting operation.

6.4.1 How to delete a Location

Deleting a Location implies deleting also the sub-Locations nested within it, as shown in the example in the picture below:

- By deleting the Wall location, also the Cabinet sub-location nested within it will be automatically deleted.

- By deleting the Floor 1 location, also the Hall and Bedroom sub-locations will be automatically deleted.



When the user selects a location with the aim to delete it using the right mouse button, he can decide between two actions:

- **Delete**: implies deleting the location (and the nested sub-locations) together with all the objects contained within it; all the functions and the modules assigned to the deleted locations will be removed; if the modules/functions being deleted are used as signals in other functions, the signals are to be deleted.
-

  Delete while keeping objects: delete the location and move all the modules and functions being connected to it (and to the nested locations) in the upper location (in the father node). If, as an example, the user decides to delete the Bedroom location, then all the modules and the functions being connected to it will be automatically moved to the father location Floor 1; Whereas, if the user deletes the Floor 1 location, then all the modules and the functions present in the Floor 1 location and in all the locations nested within it, in this case Hall and Bedroom, will be automatically moved to the Root father location (there is no limit in the deepness of the nested Locations, all of them have to be considered).

6.4.2 Delete commands

The two available modes to delete a location are:

1) Right mouse button and select an action to perform between the two actions being listed above.

2) Delete key on the keyboard.

If the user uses the right button, he can decide what kind of action to carry out (deleting all the objects or not); whereas if he uses the Delete key on the keyboard, then the total delete action will be automatically applied. It is also possible deleting only one selected location at a time (together with all its nested Locations).

6.4.3 How to delete a module

The user can delete a module by using the right mouse button or the Delete key on the keyboard. When the user deletes a module, then all the signals coming from the deleted module and being linked to the functions will be automatically removed from the functions. It is possible to delete only one module at a time.

6.4.4 How to delete a function

Deleting a function works like deleting the modules:
The user can delete a function by using the right mouse button or the Delete key on the keyboard. Once deleted, the signals which were used by the function is freely available again; moreover, if the function being deleted was linked to other functions, then it must be removed. It is possible to cancel only one function at a time.

6.5 Copy and Paste operation

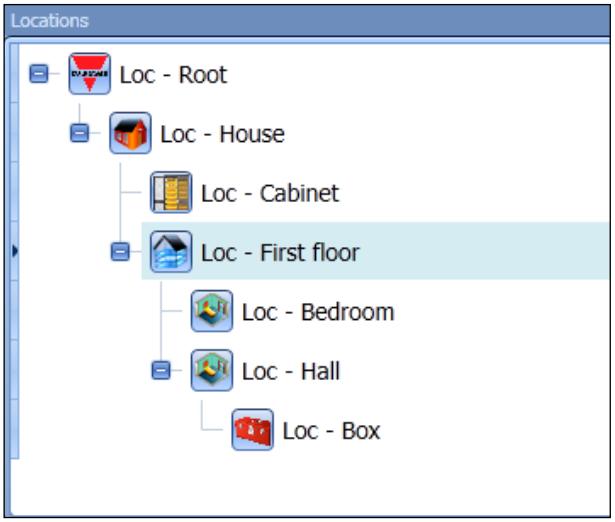
The concept of Location within the program must be explained first in order to better understand the concept of copy and paste operations.
A location is a logic structure able to contain other objects and it must be considered as a folder where other sub-folders can be contained together with files of various kinds.
The main feature for the Locations being defined in the UWP 3.0 Tool must be that they are not linked among each other by any hierarchic order; it must be possible to enter any kind of Location in any point of the installation whatsoever.
This concept guarantees the highest possible flexibility to various installers when they have to create logically structured configurations which may be completely different depending on the case. Should a priority be given to the way of entering the Locations, it would result restrictive to the installers.
There are 12 selectable locations that are respectively:

- General
- House
- Building
- Floor
- Flat
- Rooms
- Room
- Walls
- Wall
- Cabinet
- Box
- Garage

From the user's point of view, all the Locations must differ from each other only for their pre-defined name and for the image of the icon being used.

6.5.1 How to copy and paste a Location

The copy and paste operation of a Location becomes an added value when the user has previously created a part of the project, such as for example a flat, and he wants to copy it instantly to create an apartment block made of flats similar to the starting copy. When the user copies a selected location, he also automatically copies all the sub-locations nested within it. As shown in the example in the picture below, by copying the location First floor, also the sub-locations Bedroom, Hall and Box will be automatically copied. Once the copy has been pasted, also the hierarchy being used by the Locations of the copy are maintained.



When the user copies a location (and the locations being nested within it as a consequence), then a copy of all the objects contained in the Location must be created; after that, the user must copy the name of all the Locations together with the signals linked to it as well as all the modules being present with all the relevant values set in the parameters of each module.

6.5.1.1 Names of the objects being pasted

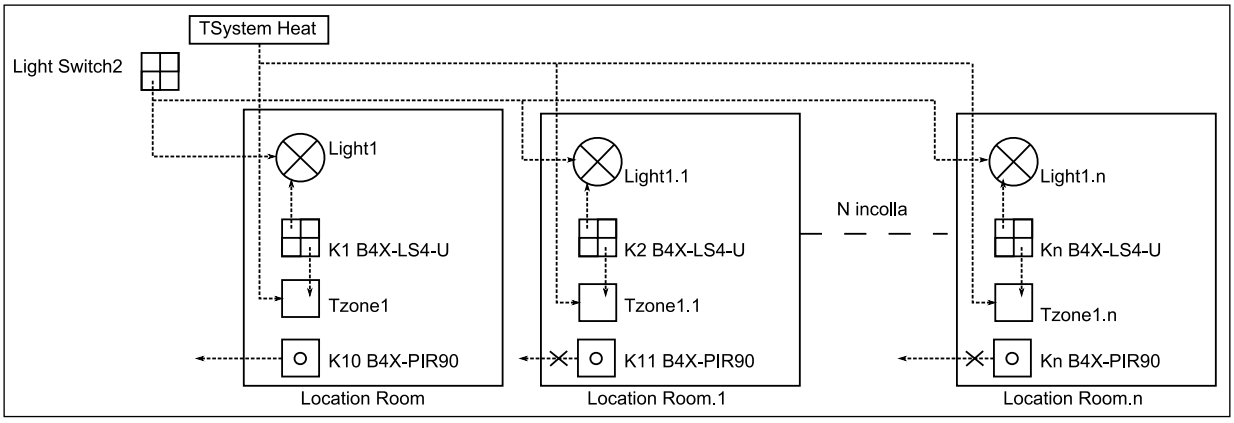
The pasted Locations must be renamed by adding to the name of the location the suffix ".number", and the number will progressively increase according to how many paste operations are being carried out.
The modules must be pasted and renamed using the same method being used to enter more than one module to the "modules" added window, and their names must be of the following kind: *K+number+partNumber*, the number of which will start from the first being available and will progressively increase according to how many paste operations are being carried out.
The pasted functions must be renamed by adding to the name of the function the suffix ".number", and the number will progressively increase according to how many paste operations are being carried out.
See the following example.

Examples of copy and paste

The example shown in the picture below explains how the copy and paste function must be carried out. Let's suppose to have a simple Location named "Location room", including a Light function, a zone temperature function and a light switch module.

"Light1" has as input:

- A signal coming from the first push button of the light switch "K1 B4X-LS4-U" being present in the same Location;
- A signal coming from the second push button of a module external to the copied Location, *Light Switch2*;
- The function Zone Temperature *TZone1* has as input:
- A signal coming from the fourth push-button of the light switch "K1 B4X-LS4-U" that is situated in the same Location;
- A signal coming from the function *TSystem heat* external to the copied Location;

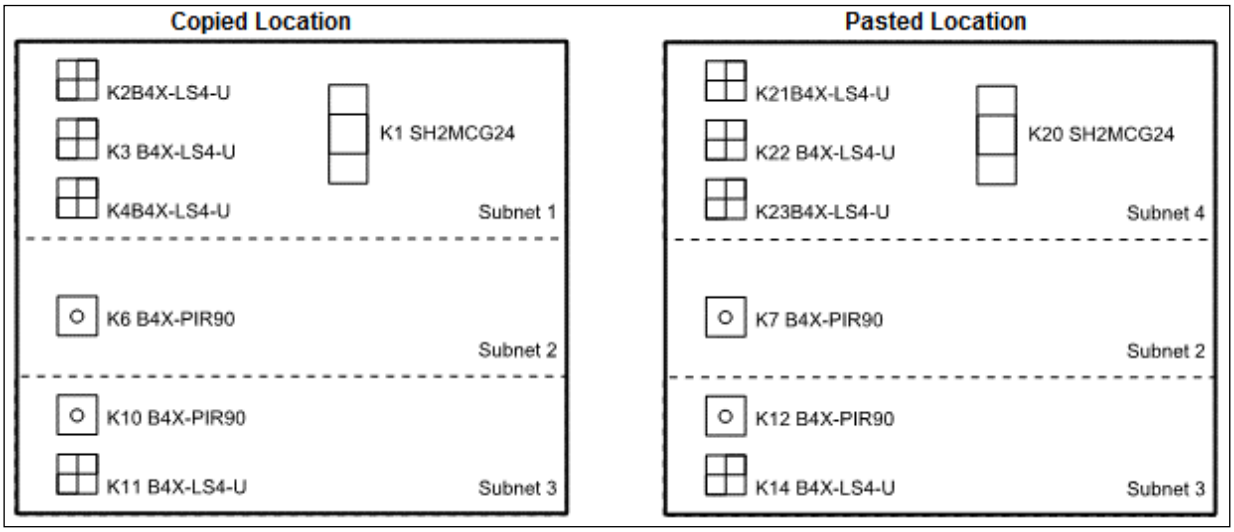


When the new location is pasted, the functions will have the following specifications:
If the user links to the function any signals coming from a module present in the copied Location, a new copy of the function will be generated with another module which will be automatically re-generated.
As for all external links having copied and pasted functions, they are kept always the same.
In the example shown above, therefore, the first paste will generate Location Room1, in which there will be a true copy of the functions *Light1* and *Tzone1*, that will be however linked to a new module internal to the location named "K2 B4X-LS4-U", while the same external links will be kept with the function *TSystem heat* and the module *Light Switch2*.

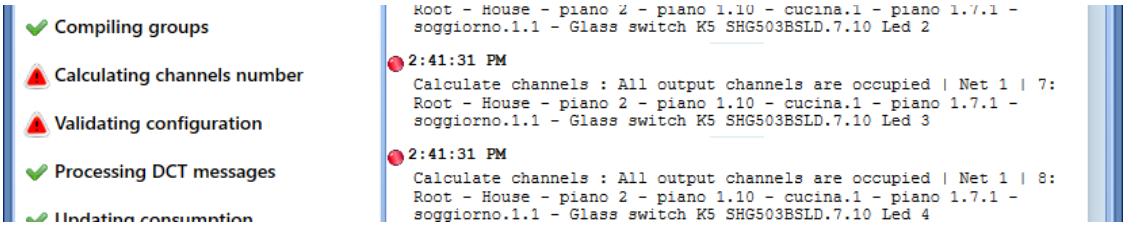
As for the modules not being used in the Location or not being linked to functions external to the copied Location (see K10 B4X-PIR90), they must be treated in the same way, that is when pasting, they must be regenerated without any link (only module without connections).

When a Location is pasted, the pasted modules that are regenerated must occupy the same subnets of the starting modules, modules belonging to different subnets must maintain the same original subnets, moreover if the copied location contains a MasterGenerator, its copy must enter a new master and re-generate all the new modules assigning them to the new MasterGenerator being created.

The example in the figure below shows what has been previously said.
In the location that must be copied, the user can find the modules K2 B4X-LS4-U, K3 B4X-LS4-U, K4 B4X-LS4-U belonging to Subnet1, moreover there is a MasterGenerator also belonging to Subnet 1.
In the location there is also the module K6 B4X-PIR90 that is assigned to Subnet 2 and the modules K11 B4X-LS4-U e K10 B4X-PIR90 that belong to Subnet 3.
When the location is pasted, the MasterGenerator will be substituted with a new MasterGenerator assigned to the first free subnet, in this case Subnet 4, and all the modules that before referred to Subnet 1 are being regenerated with reference to the new subnet, that is Subnet 4.
The modules present at the copied location that refer to Subnets 2 and 3 are generated maintaining always the reference to the same subnet.
When the user pastes a Location having some MasterGenerators, a control must be done before the operation in order to make sure not to enter more than 7 masters. Should there already be 7 MasterGenerators in the configuration, then only the locals will be inserted without adding further masters.

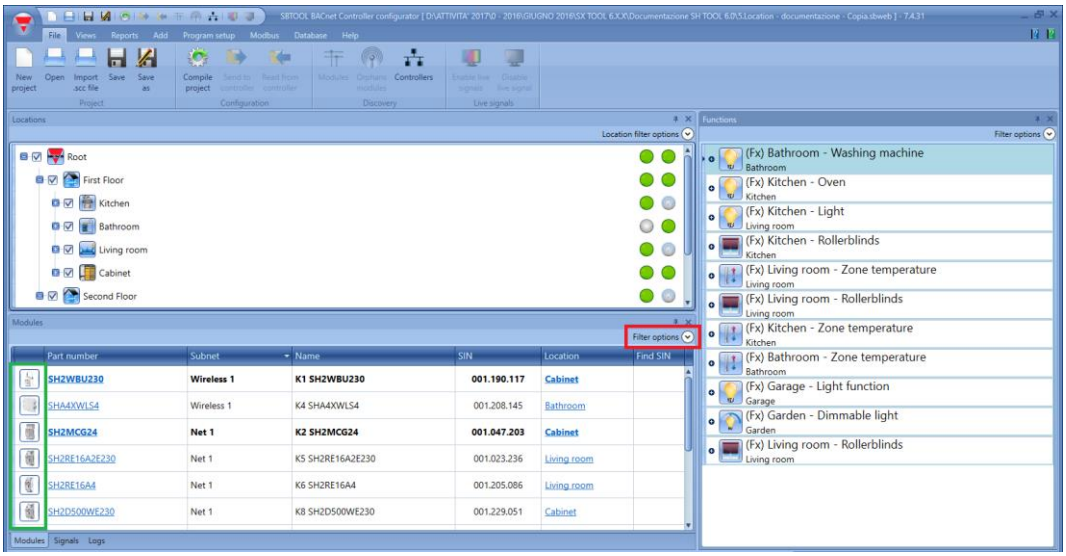


Concerning the number of modules that can be placed in a subnet, when the number of pasted modules exceeds the maximum number, then the user will see the circle on the right of the tree changing colour and becoming red. Regarding all the modules having an invalid address (since they exceed the number of modules being admitted by the subnet), the lack of available channels will be reported during the compiling phase and therefore it is the user who will manually place the subnet for each single module, as shown in the picture below:



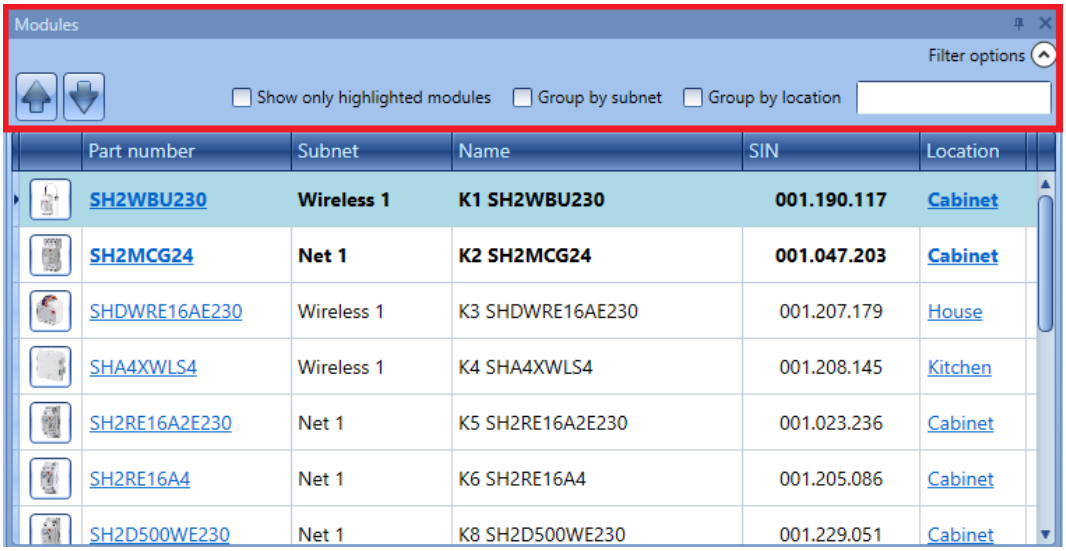
7 Modules

To configure a module, the user has to click on the picture of the module in the *Modules* window of the UWP 3.0 Tool (see the picture below):



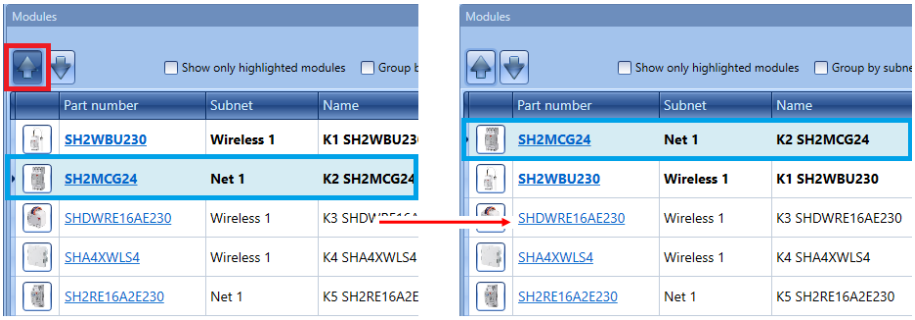
7.1 How to manage the filters in the Modules window

The modules can be sorted or filtered by the *Filter options*: by clicking on the icon, the *Filter options* panel will be opened. The available filters are shown in the top of the *Modules* window, as shown in the red rectangle:

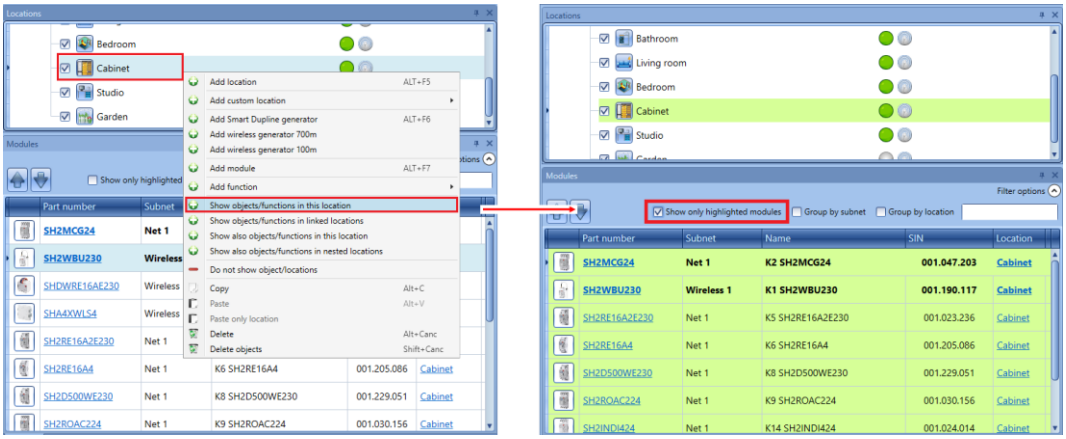


Only the modules specified by the filter will be presented: the filters can be used individually or they can be mixed.

Up/Down arrows:
Once a module is selected, by clicking on the arrows it can be moved up or down.

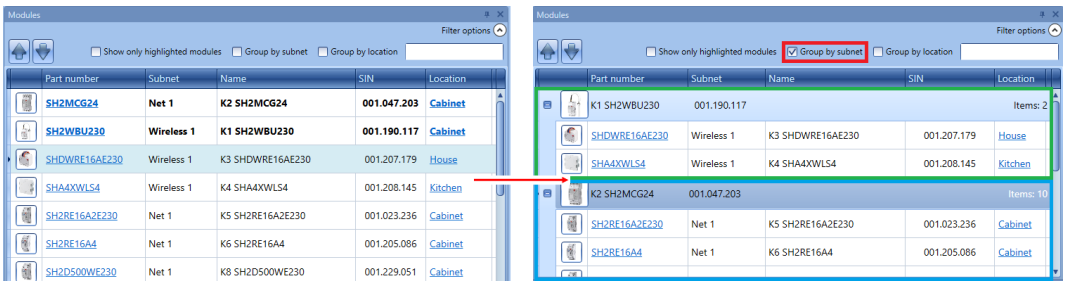


Show only highlighted modules:
When this filter option is checked, only the modules that are highlighted (according to the *highlight* functionality available on *Locations/Functions* window) are presented in the *Modules* window.
In the picture below an example is shown:

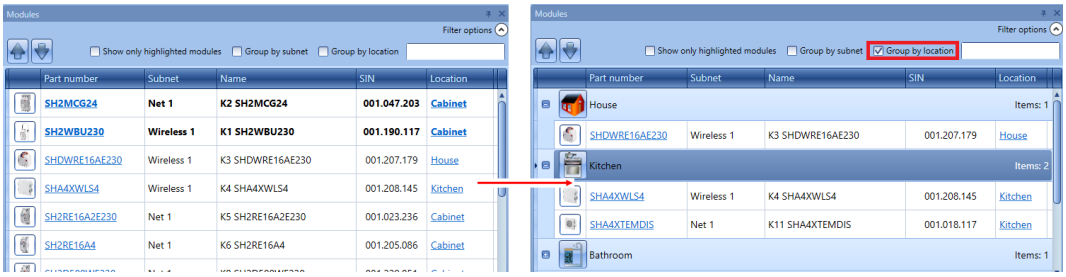


Group by subnet:
The modules are grouped by:

- Mastergenerator (each network generated by SH2MCG24, SH2DUG24 and SH2WBU230N);
- Modbus serial communication port (COM1MASTER or COM2MASTER);
- TCP/IP Modbus connection



Group by location:
The modules are grouped by the locations defined in the project.

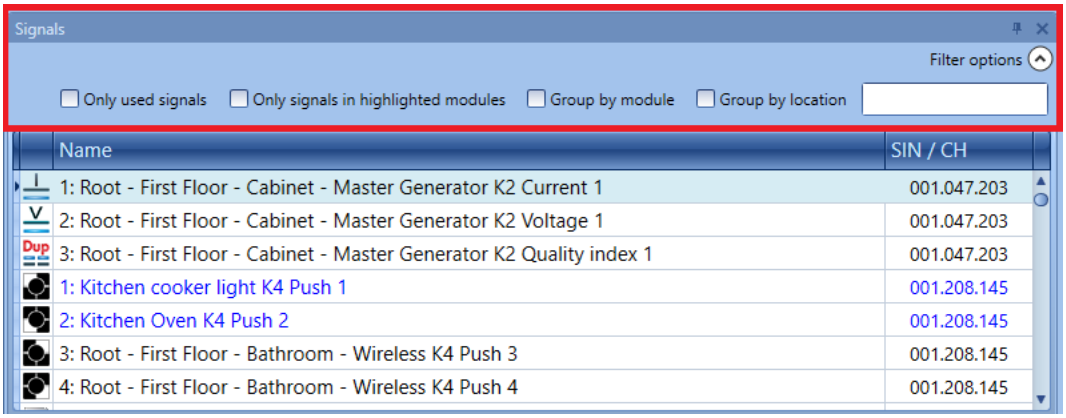


Search box:
The results will include all the words that contain the input string and the search will be carried out on the modules available in the project. The user can insert the full name or part of it.

7.2 How to manage the filters in the Signals window

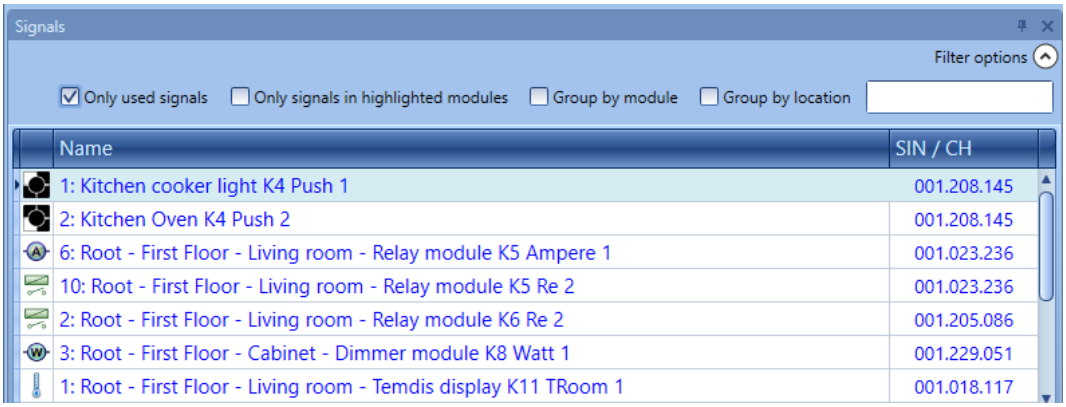
The signals can be sorted or filtered by *Filter options*: by clicking on the icon, the filter options panel will be opened: only the signals specified by the filters will be displayed.

The available filters are shown in the top of the *Signals* window: see the red rectangle in the picture below:

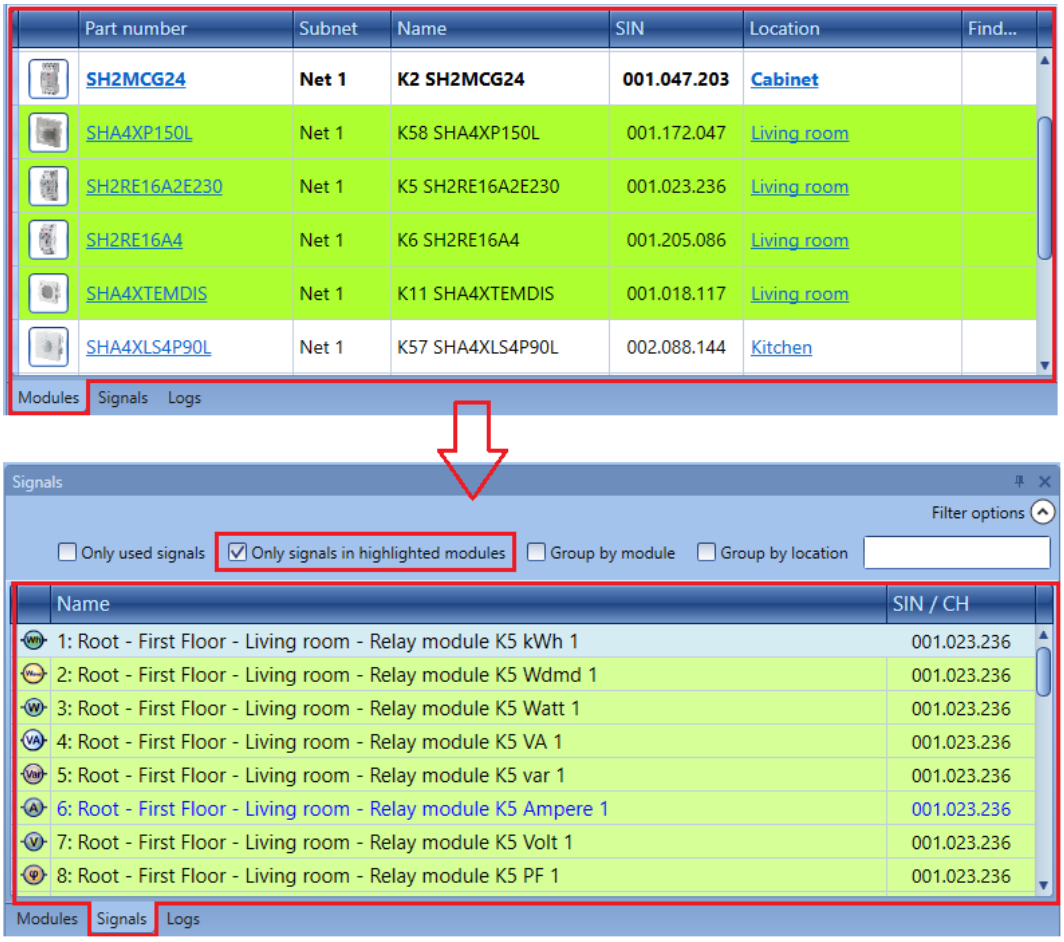


When the option panel is shown, the following filters are available. The filters can be used individually or can be mixed:

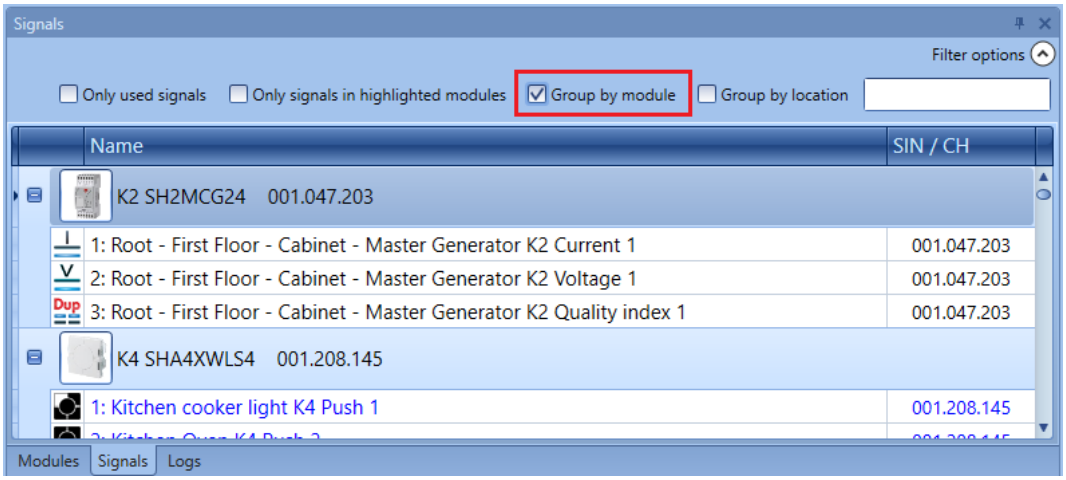
Only used signals:
Only the signals already used in at least one function are presented and they are highlighted in blue:



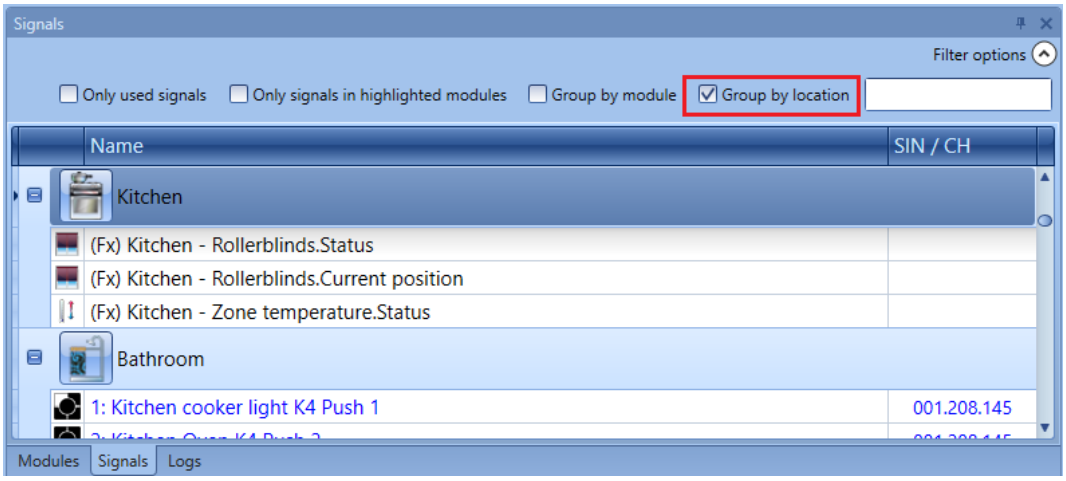
Only signals in highlighted modules:
When this filter option is selected, only the signals belonging to the highlighted modules are shown in the *Signals window*.
See the example below:



Group by module:
The signals are grouped by their own module, as shown in the example below:



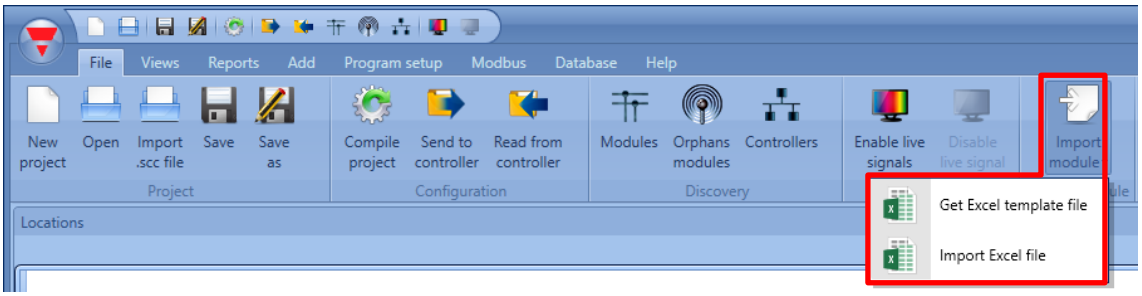
Group by location:
The signals are grouped by location.



Search box:
The results will include all the words that contain the input string and the search will be carried out on the modules available in the project. The user can insert the full name or part of it.

7.3 Import module list from Template file function

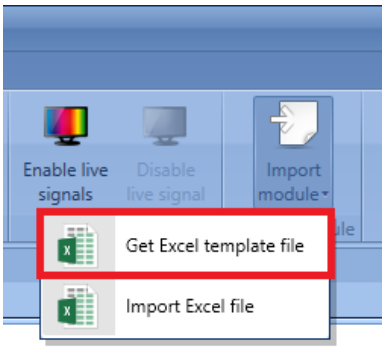
The user can create the UWP 3.0 configuration in a faster way, reducing the time required to create the modules list. The time needed to specify customized names of modules and signals will be reduced. The function is available in the *File* menu of the UWP 3.0 Tool (the function is not available in the SA Tool version).



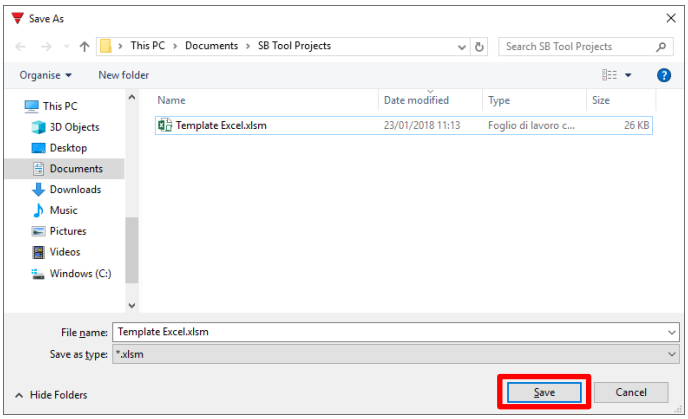
The user defines the modules and signals in a Template file. Customized names for modules and signals can be specified in order to override the default names provided by the UWP 3.0 Tool. The modules and signals list will be imported by an excel file provided by the UWP 3.0 Tool: when a new version of the software is released, the UWP 3.0 Tool embeds a new version of the file to manage new modules.

7.3.1 How to download the Template file

From the *File* menu of the UWP 3.0 Tool, click the *Import module* function, then select the *Get Excel template file* option:



In the *Save As* window that will appear, the user has to locate the target directory to download the template file.



7.3.2 Template file overview

The Template file is in **XLSM** format: the file is editable by the user with Microsoft ® Office 2007 or later. The example below shows the file structure of an inserted module: the cells highlighted in green are editable, while the cells highlighted in red must not be modified, otherwise the file will not be imported successfully:

Barcode	PartNumber	SinA	SinB	SinC	ModuleName	Location path	Type	#	SignalName
00204507800016	B4X-LS4-U	002	045	078			In	1	
							In	2	
							Out	3	
							Dia	4	

The descriptions of all the fields are shown below:

Field name	Description
Barcode	The *barcode* can be scanned by a barcode scanner or typed manually: the fields such as *PartNumber*, *Sin (A, B, C)* and *signals* are automatically filled in with the module's information
PartNumber	The *PartNumber* can be entered manually or automatically: The fields *Sin (A, B, C) and signals* are automatically filled in with the module's information
SinA, SinB, SinC	These fields are automatically filled in when a valid barcode is scanned. When the *PartNumber* is inserted manually, the user must complete the fields *Sin A, B, C* with the SIN of the module
ModuleName	The user can enter a customized name instead of the *Module name* that is automatically assigned by the UWP 3.0 Tool
Location path	The user can enter a customized location path, using the "\" symbol between each nesting location (***e.g. Floor1\ Room1***). The base path "***Root***" will be added automatically to the project when the Template file is imported *Note: the "\" symbol is not a valid character for a location name*
Type (In, Out, Dia)	These fields are automatically filled in according to the number of signals provided by the PartNumber *Note: these fields must not be modified by the user*
# (Number)	
SignalName	The user can enter a customized name instead of the *Signal name* that is automatically assigned by the UWP 3.0 Tool

The fields **marked in bold** are mandatory.

7.3.3 How to find the barcode to be inserted

The barcode is present on different labels attached to the box: this information can be found on the upper-box label and on the inside label, as shown below:

The part numbers that will be managed in the Template file are listed here: other modules will not be recognized by the file and marked as **NONE** part number.

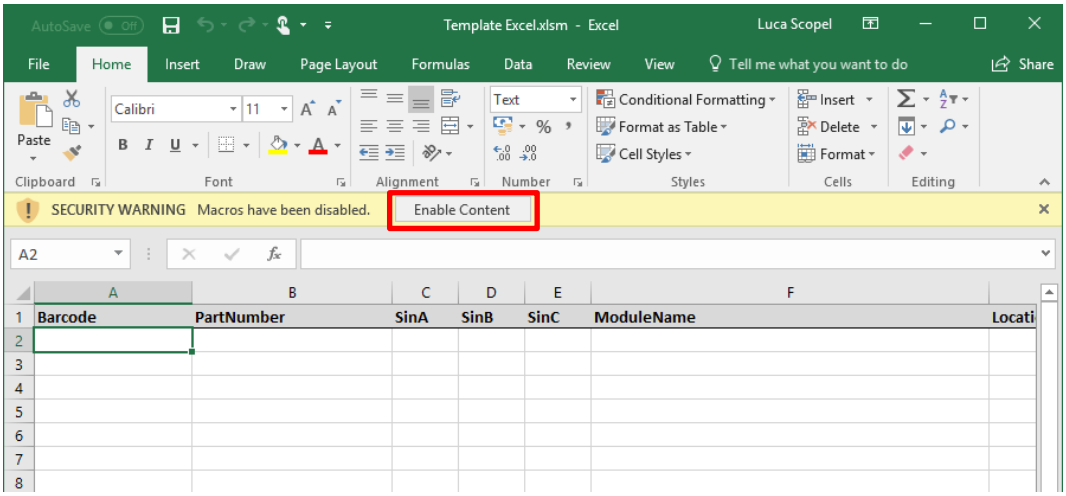
Part Number	Description
SBB4I2O24	*Fire damper I/O module*
SBB4I2O230	*Fire damper I/O module*
BDB-INCON4-U	*Decentral modules with 4 contacts inputs*
BDB-INCON8-U	*Decentral modules with 8 contacts inputs*
BDB-IOCP8-U	*Decentral modules for 4 push buttons and 4 outputs for LED*
BDB-IOCP8A-U	*Decentral modules for 4 push buttons and 4 outputs for LED*
BDA-RE13A-U	*Decentral relay output*
SH2RE16A4	*Output relay module with 4 outputs*
SH2INDI424	*Input module for 4 inputs*
B4X-LS4-U	*Light switch with 4 push buttons and 4 LEDs*
B5X-LS4-U	*Light switch with 4 push buttons and 4 LEDs*
SHA4XLS4TH	*Light switch with 4 push buttons and 4 LEDs with temperature and humidity*
SHE5XLS4TH	*Light switch with 4 push buttons and 4 LEDs with temperature and humidity*
SHSQP360L	*PIR sensor with an integrated Luxmeter*

7.4 Compile the Template file with Microsoft Excel

The Template file is delivered in *macro-enabled workbook* format (.xlsm). After downloading the Template file, open it with Microsoft Office 2007 or a later version. This is necessary to enable the macro content.

7.4.3 How to enable Excel macros

If the yellow Message Bar appears with a shield icon, the user has to click on the *Enable Content* button to enable the macro content, as shown in the red rectangle below:



For more details please visit: Enable or disable macros in Office files

There are two ways to compile the template file with modules, as shown in the following pages:

- Automatic insertion with a barcode scanner;
- Manual insertion by the user (by a keyboard);

7.4.4 Automatic insertion with a barcode scanner

A barcode scanner is used to scan the barcodes and the Template file is automatically filled in with the module information. The procedure is as follows:

System requirements	
Hardware	A barcode scanner ***Note: the barcode scanner has to be configured to send <u>only one CRLF</u> or Enter key after the barcode scan. Please refer to Barcode scanner documentation***
Software	UWP 3.0 Tool 7.5.3 or later (*the SA Tool version is not supported)* Microsoft ® Excel 2007 or later *Note: Excel macros must be allowed by security policies*

1. The user opens the Template file: the active cell in the Excel workbook is focused on the first barcode cell, as shown in the red rectangle below:

Barcode	PartNumber	SinA	SinB	SinC	ModuleName	Location path	Type	#	SignalName

2. A barcode is scanned by a barcode scanner: the barcode is entered in the relevant cell, as shown below:

Barcode	PartNumber	SinA	SinB	SinC	ModuleName	Location path	Type	#	SignalName
AAABBBCCCDDEE									

3. If the barcode is valid, it is recognised and the ***PartNumber*** and the ***Sin*** cells are automatically filled in with the module information (green rectangle below):

Barcode	PartNumber	SinA	SinB	SinC	ModuleName	Location path	Type	#	SignalName
AAABBBCCCDDEE	**PRTNMB001**	**AAA**	**BBB**	**CCC**					

4. Likewise with step 3, the cells Type and # are automatically filled in according to the number of signals provided by the PartNumber (blue rectangle below). The active cell in the excel workbook is moved automatically to the next available cell in the Barcode column (orange rectangle below):

Barcode	PartNumber	SinA	SinB	SinC	ModuleName	Location path	Type	#	SignalName
AAABBBCCCDDEE	PRTNMB001	AAA	BBB	CCC			**In**	**1**	
							In	**2**	
							Out	**3**	
							Dia	**4**	

5. The options fields such as ModuleName, LocationPath and SignalName can be filled in by the user in the relevant cells on the Template file. Repeat steps 1 to 5 for every other module which has to be inserted.

6. Click on the *Save* button in the Excel program to save the changes. The file is now ready to be imported.

7.4.5 Manual insertion without a barcode scanner

The template file can be manually filled in by the user. The requirements are shown below:

System requirements	
Hardware	Keyboard
	UWP 3.0 Tool 7.5.3 or later (the *SA Tool* version is not supported)
Software	Microsoft Excel 2007 or later
	Note: Excel macros must be allowed by security policies

The procedure is as follows:

1. The user opens the Template file: the active cell on the Excel worksheet is the first valid **Barcode** cell (red box below). The barcode can be entered manually by the user: if the barcode is recognised, the module information will be automatically filled in, as in the previous procedure:

Barcode	PartNumber	SinA	SinB	SinC	MouleName	Location path	Type	#	SignalName

2. In a different way, the user can select the first valid **PartNumber** cell and type the part number of the module has to inserted.
Note: the part number must be inserted in capital letter (e.g. SHA5XLS4TH is correct, sha5xls4th is not recognized)

Barcode	PartNumber	SinA	SinB	SinC	MouleName	Location path	Type	#	SignalName
	PRTNMB001								

3. After pressing enter, the cells Type and # are automatically filled in according to the number of signals provided by the PartNumber (blue rectangle below). These fields must not be edited.

Barcode	PartNumber	SinA	SinB	SinC	MouleName	Location path	Type	#	SignalName
	PRTNMB001						**In**	**1**	
							In	**2**	
							Out	**3**	
							Dia	**4**	

4. The user has to insert the Sin number in the Sin (A, B, C) fields (red box below):

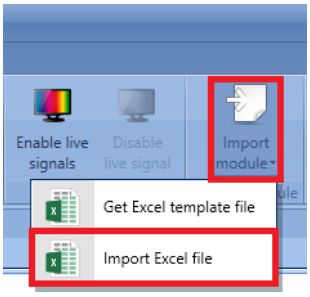
Barcode	PartNumber	SinA	SinB	SinC	MouleName	Location path	Type	#	SignalName
	PRTNMB001	**AAA**	**BBB**	**CCC**			In	1	
							In	2	
							Out	3	
							Dia	4	

5. The user has to select the first active cell in the Barcode column (orange rectangle above) Repeat steps 1 to 7 for every other module which has to be inserted.

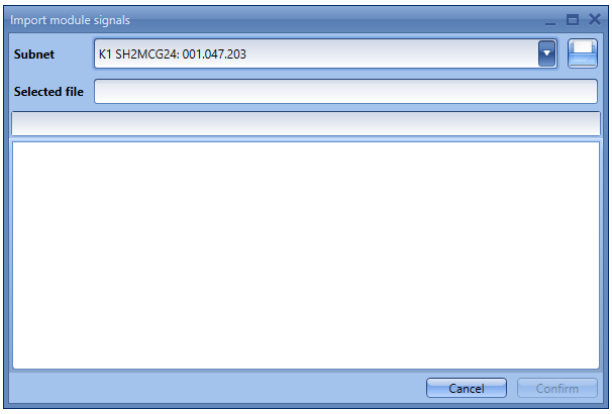
6. Click on the *Save* button in the Excel program to save the changes. The file is now ready to be imported.

7.5 How to import a fulfilled Template file

To import a Template file, the user has to click on the *Import module* button in the *File* menu of the UWP 3.0 Tool and select the *Import Excel file* option (see below):

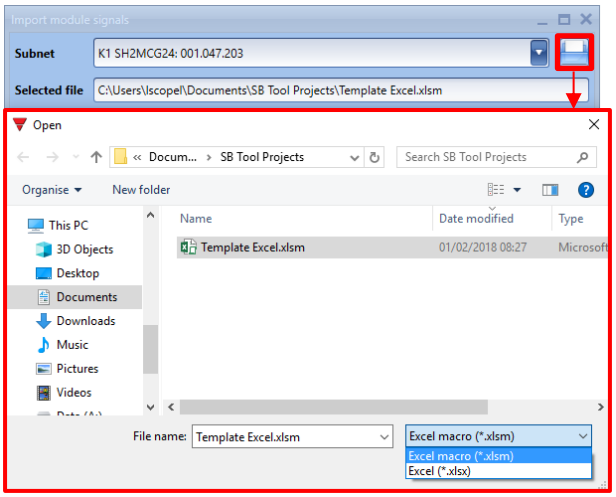


The *Import module signals* window will appear:



In the ***Subnet*** list in the window, the user has to select which Subnet the modules have to be attached to: **only the SH2MCG24** Master Channel Generator is allowed. One file permits specification of the modules attached to only one network. To manage multiple Subnets, one Template file has to be created for each Subnet of the project.

By clicking on the button, the user locates the Template file which has to be imported in the project: **only *XLSX/XLSM*** file extensions are allowed: the *XLS* format will not be supported.



By clicking on the *Confirm* button to process the file: the system starts to check the Template file for data consistency. If there are no errors the user can click on the *Confirm* button to import the modules/locations as defined in the Template file:

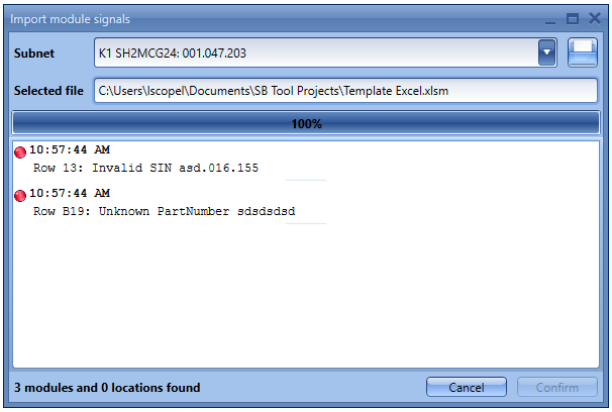
In the UWP 3.0 Tool each part number is added in the Module view and located in the location tree according to the Location path value.

The name of the module is replaced with the one specified in the Template file or automatically assigned by the UWP 3.0 Tool if left blank. The signals names are replaced with the customized values or automatically assigned by the UWP 3.0 Tool.

7.5.1 How to fix a Template file with errors

If there are some errors on the Template file, the user cannot confirm the importing of modules/locations: the user can close the *Import module signals* window or click on the button to open a different Template file.

In the *Import module signals* window, for each row that is not valid, a message will be shown to give information about the row number and the related errors that will guide the user in fixing the error.



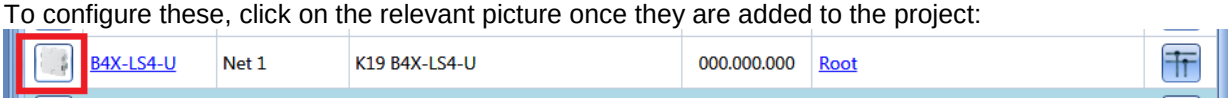
The user can repeat the procedure above to import the fixed Template file.

7.6 Light switches

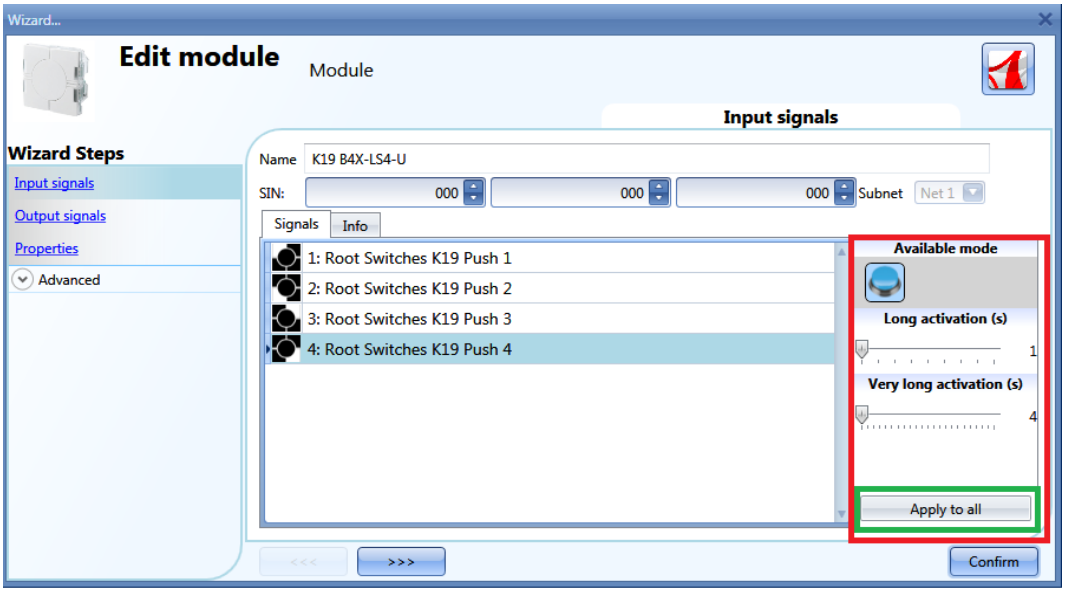
The following items belong to the *Light switches* family:

B4X-LS4-U
B5X-LS4-U (this item also has to be selected for B5B-LS4-U and B5W-LS4-U)
BEW-LS1-U
BEW-LS2-U
BEW-LS3-U
BEW-LS4-U (this item also has to be selected for BEA-LS4-U and BEG-LS4-U)
SHA4XLS4P90L (this is a light switch with integrated PIR sensor and Luxmeter)
SHE5XLS4P90L (this is a light switch with integrated PIR sensor and Luxmeter)

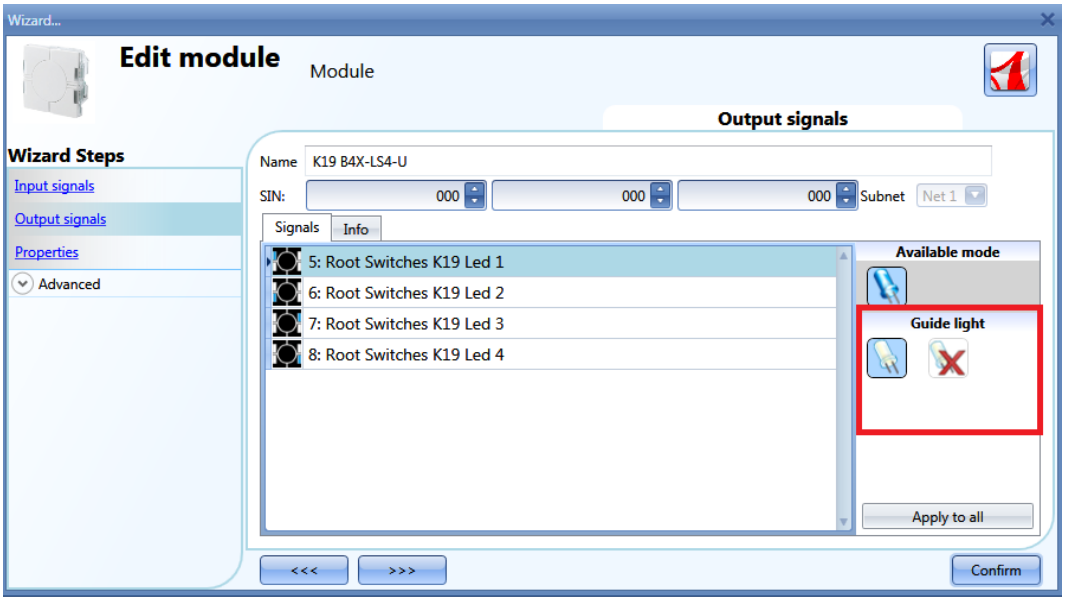
To configure these, click on the relevant picture once they are added to the project:



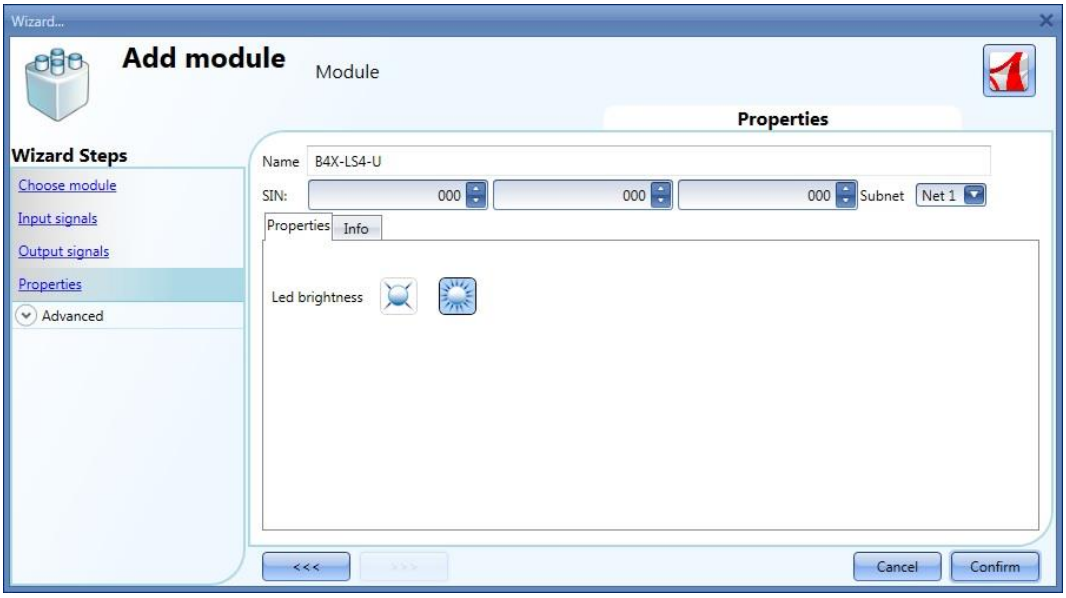
The configuration wizard will appear:



In the *Input signals* field, the *Long activation* and the *Very long* activation time has to be defined.
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and it is always set by the system 3 seconds longer than the *Long activation* time.
The short press and long press are recognized when the push button is released.
The user can configure different times for each push button or can set them to the same values by clicking on *Apply to all.*
If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated to the very long press will be activated as soon as the pushbutton is pressed for the selected time.



In the *Output signals* window, the user can enable the white LED as a guide light (red rectangle): by clicking on the white LED icon, the guide light is enabled, while by clicking on the red cross the guide light is not enabled.
Each LED can have different behaviour.
The guide light is configurable in the following items: B4X-LS4-U, B5X-LS4-U and the brightness can be set in the Properties window.

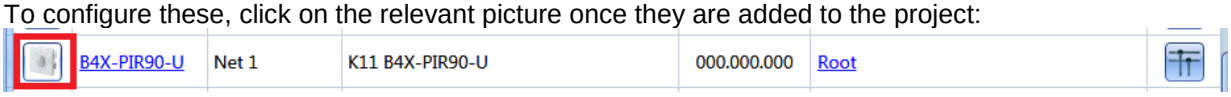


7.7 Motion detectors

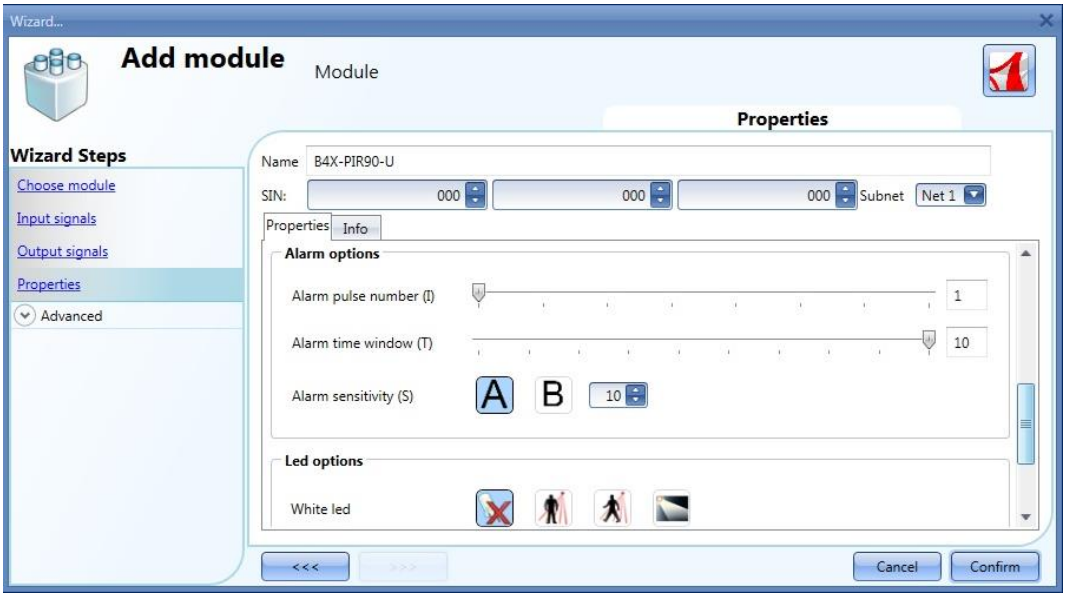
The following items belong to the *Motion detector* family:

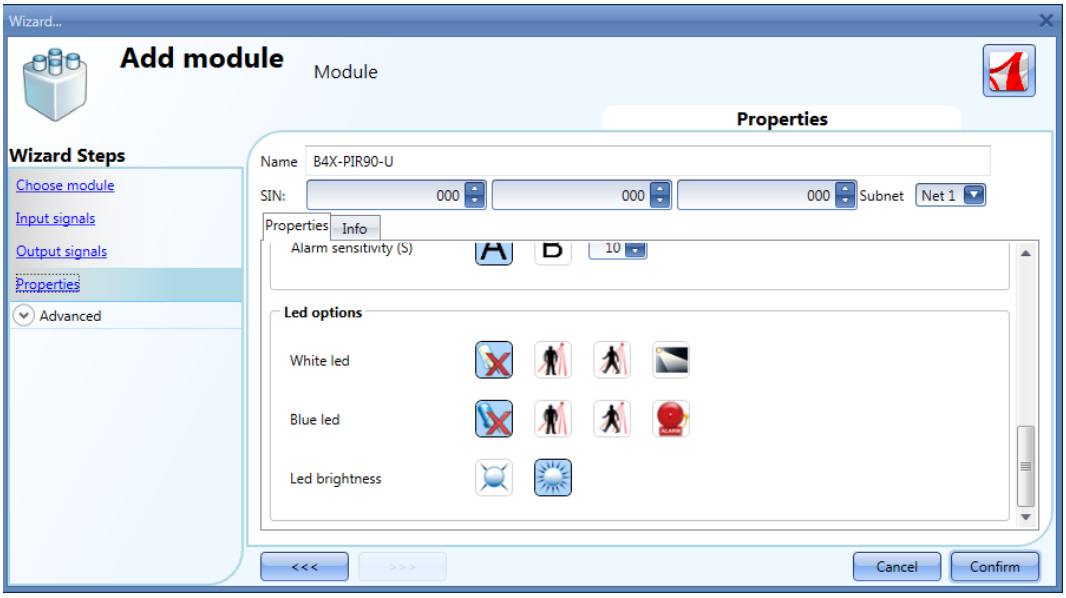
B4X-PIR90-U
B5X-PIR90-U (this item also has to be selected for B5B-PIR90-U and B5W-PIR90-U)
BSB-PIR90-U
BSD-PIR90-U
BSP-PIR90-U
SHA4XP90L
SHE5XP90L
SHSBB90L
SHSBD90L
SHSBP90L
SHSQP360L
SHA4XP150L
SHA4XP150
SHE5XP150L
SHE5XP150

To configure these, click on the relevant picture once they are added to the project:



The configuration wizard will appear: the only configurable field is *Properties*. In *Input signals* and *Output signals* only the available input and output signals are shown.



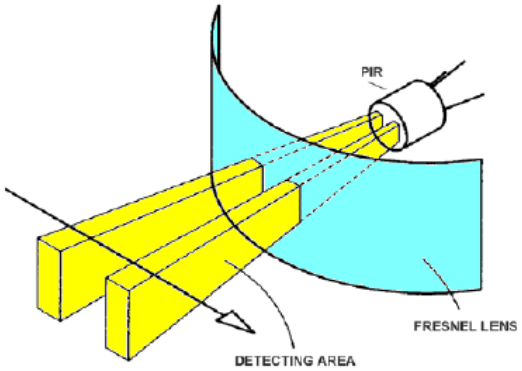


In this window the sensitivity has to be configured, also the behaviour of the LEDs: they could be used as a guide light, feedback on the detection of presence or movement or feedback of an alarm.

7.7.1 How PIRs work and how to configure them

This PIR sensor responds to any fluctuation in infrared heat radiation, so any object or human presence changes the thermal image detected by the sensor when entering its field of vision.

The sensor is equipped with a segmented lens that divides the field of vision into active and passive zones (zones not visible to the sensor, see figure below: active zones in yellow). When a heat source crosses these zones, the sensor detects a change of infrared radiation, then presence and / or movement is detected.



When a warm body moves into the sensor's field of vision, the sensor detects a difference in the measured temperature.

The crossing of the active and passive zones can generate the following conditions:

a) When the body stays still in a passive area, the PIR does not generate any pulses (fig.2).
b) When the body moves into a passive area, the PIR does not generate any pulses (fig.2).
c) When the body stays still in an active area, the PIR does not generate any pulses (fig.2)
d) When the body moves into an active area, the PIR generates a positive pulse (an increase in the temperature is detected) (fig. 3).
e) When the body moves out of an active area the PIR generates a negative pulse (a decrease of the temperature is detected) (fig. 3).
f) When the body moves into an active area, the PIR can generate positive or negative pulses (positive if the body moves towards the sensor, negative if it moves away from the sensor) (fig. 3).
g) When the body moves from a passive area to another passive area by crossing an active area, the PIR generates positive and negative pulses (fig. 3).
h) When the body moves from an active area to another active area by crossing a passive area, the PIR generates negative and positive pulses (fig. 3).

Figure 2.

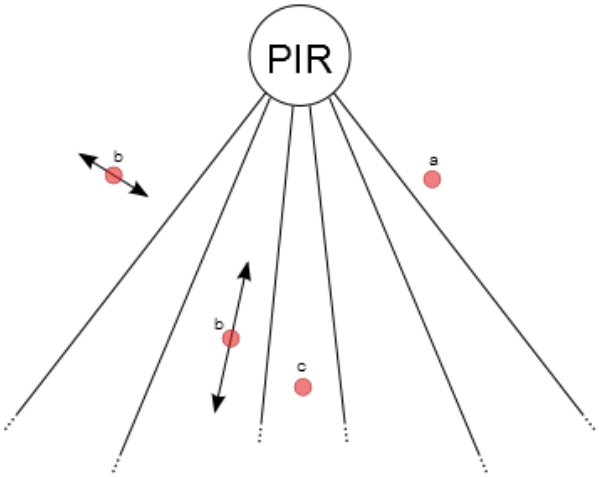
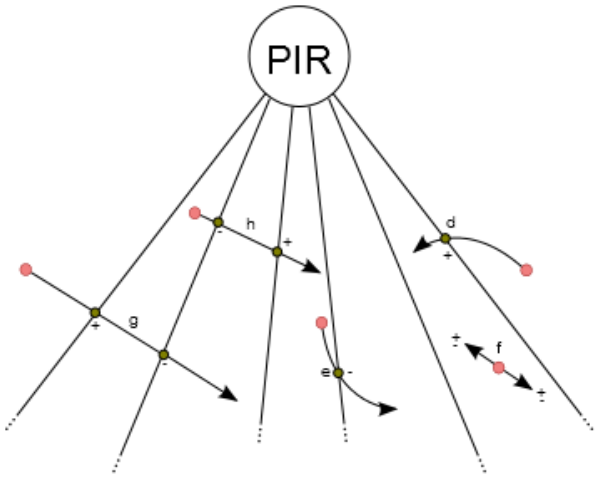


Figure 3.



The pulses generated by the PIR (i.e. the detection of a change in temperature) are compared with a threshold temperature value.

The PIR signal is a value going from 0 to 127 that corresponds to the difference detected in the temperature value; the threshold value can be set from 3 to 100. The higher the threshold is, the bigger the change of temperature.

In the smart-house system, since a PIR sensor can be used both in intruder alarm functions and in light functions at the same time, it generates three kinds of signal: *presence*, *movement and alarm*. The first is used in the light functions to reload the energy-save timer, the second is used to switch the light on in the light functions, the third is used in the intruder alarm functions.

The user defines how the smart-house system detects *presence, movement and the alarm* by setting four different parameters for both the signals by means of the software UWP 3.0 tool.

The parameters are as follows:

1) **Mode of detection**

   A: any type of pulse is valid (positive and negative). This option has to be selected for presence detection and movement, in order to turn the light on as soon as a person moves from an active to a passive area (or vice versa) or into an active area (very quick response). With this setting, any change in temperature causes the sensor to give a detection message to the master unit.
   B: only the sequence positive to negative pulse or negative to positive pulse is valid. The person has to go from an active area to another active area, passing through a passive one, or vice versa.
   This option is recommended for sensors used in the intruder alarm function, in order to avoid false alarm conditions. With this setting the sensor sends a detection message only if it perceives an increase or a decrease in the temperature.

2) **Threshold Value - Sensing**

This is a number that can be set between 3 and 100. The smaller the value, the longer the detection range, but sensitivity to the heat source is greater.
In figures 4 and 5 there are some examples of different sensitivities for the B4X-PIR90-U PIR sensor. Figure 4 represents the covered area of the horizontal sensing plane, while figure 5 represents the covered area of the vertical plane.

a) Threshold =100. With this setting the PIR has the least sensitivity, represented by the white area in figures 4 and 5. Detection extends to 6m and small bodies on the floor cannot be detected. In figure 5 (*vertical sensing area)* you can see that the white area extends to less than 0.5m below the mounting height of the sensor. With this setting small bodies cannot be detected (e.g. small pets)
b) Threshold =42. With this setting the PIR has medium sensitivity, represented by the light blue areas indicated in figures 4 and 5. Detection extends to 7.5m and again small bodies on the floor cannot be detected. In figure 5 (*vertical sensing area)* you can see that the light blue area extends to about 0.5m below the mounting height of the sensor.
c) Threshold =3. With this setting the PIR has the highest sensitivity, represented by the dark blue areas indicated in figures 4 and 5. Detection extends to 8.5m and small bodies on the floor are detected. In figure 5 (*vertical sensing area)* you can see that the dark blue area arrives at floor level, with a sensor mounting height of 1.1m.

3) **Number of pulses**

This is the number of pulses calculated according to mode A or B, which the sensor has to detect in a predefined time interval in order to send a people detection message to the controller. It can be set from 1 to 8.

4) **Time window**

This is the time interval within which pulses are counted. The window can be set between 1 and 10 seconds.

Figure 4: Horizontal sensitivity area.

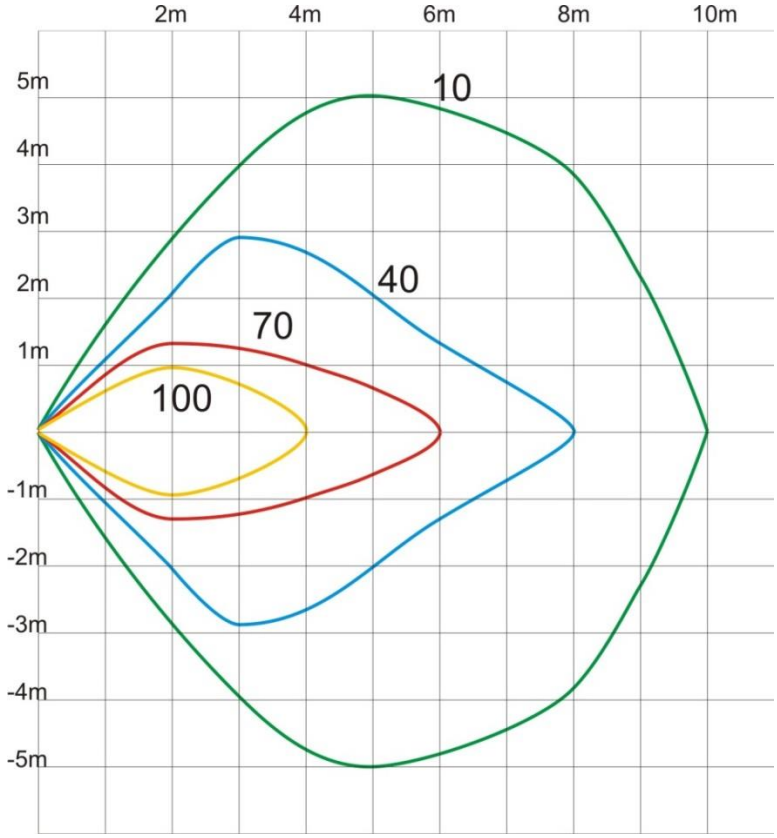
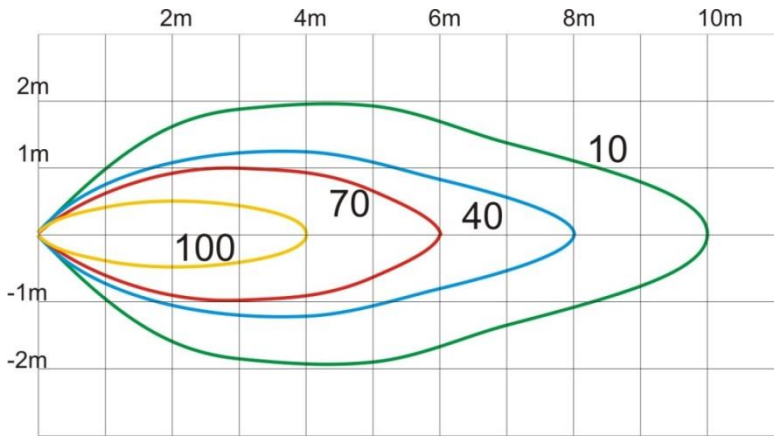


Figure 5: Vertical sensitivity area.



The four parameters have to be tuned during the installation, taking into consideration the environments and the application. Typical adjustments might be the following:

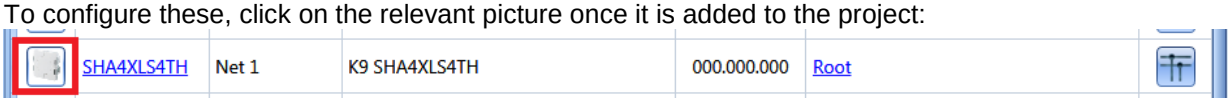
PARAMETER	PRESENCE	MOVEMENT (light fx)	MOVEMENT (alarm fx)
Filter type	A	A	B
Threshold level	10..30	30..70	50..100
Number of pulses	1	1	3
Window time (sec)	10	2	10

7.8 Temperature modules

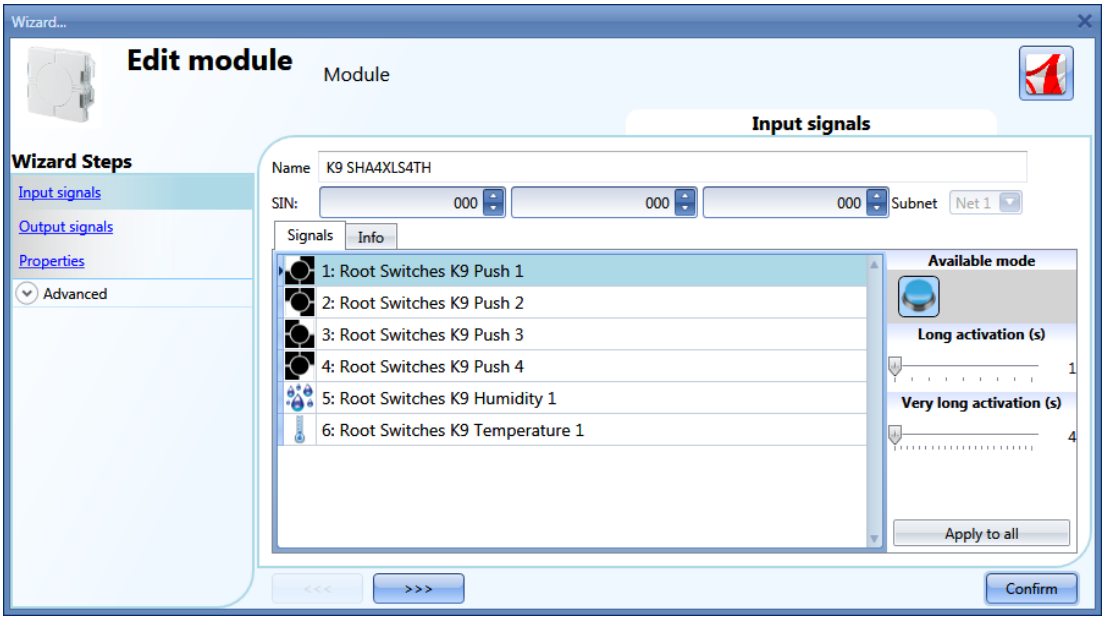
The following items belong to the *Temperature* family:

SHA4XLS4TH
SHE5XLS4TH (this item also has to be selected for SHE5BLS4TH and SHE5WLS4TH)
SHA4XTEMDIS
SHE5XTEMDIS
BSI-TEMANA-U
BSI-TEMANB-U

To configure these, click on the relevant picture once it is added to the project:



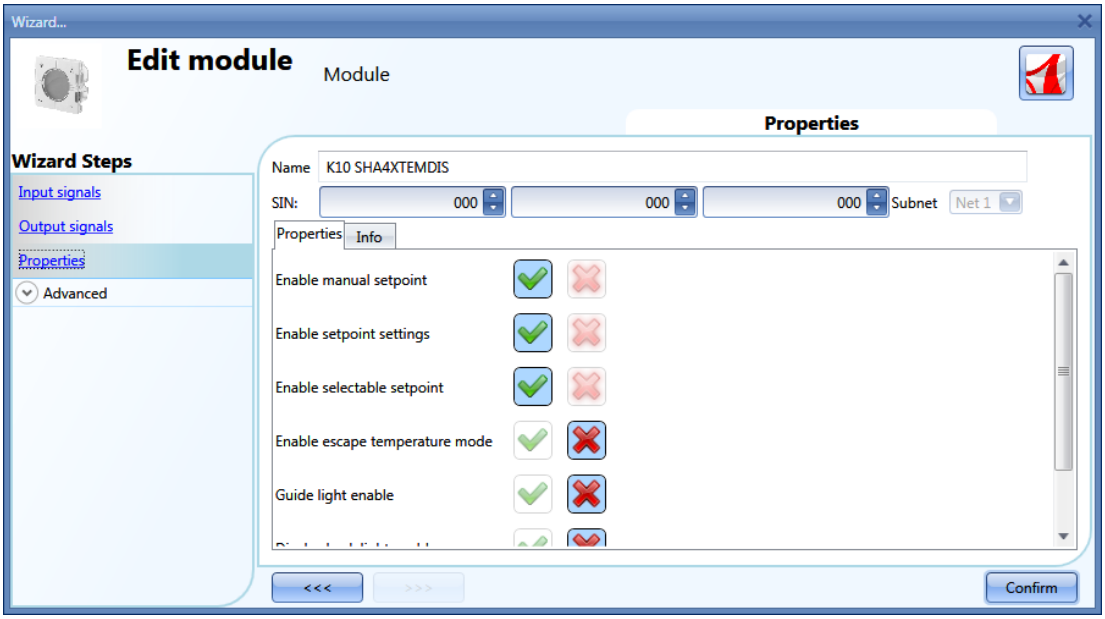
The configuration wizard will appear:



The SHA4XLS4TH and the SHE5XLS4TH are light switches with sensor display, so they can be configured as described in the paragraph related to *Light switches*.

To configure the TEMDIS display, click on the relevant picture once it is added to the project:

The configuration wizard will appear:



In the Input signals field, the available temperature values are shown, and the only window with the configuration settings is *Properties*.

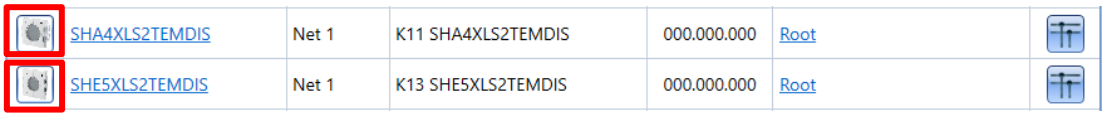
The user can enable the following settings:

1) *Enable manual setpoint*: the user will have access to a manual setpoint in addition to set point 1, set point 2, set point 3.
2) *Enable setpoint setting*: the three set points 1, 2, 3 can be changed by the user.
3) *Enable selectable setpoint*: the user can select the three set points.
4) *Enable escape temperature mode*: at midnight, the regulating setpoint will be updated according to that of the configuration file, and any change of the user is cancelled.
5) *Guide light enable*: the guide light on the push button is enabled.
6) *Display back light enable*: the user can enable the display back light, which will always be on.
7) *Display back light as status indicator*: the back light is on when the zone temperature function is on.
8) *Set resolution to 0.1:* the resolution of the display is 0.1°C instead of 0.5°C
9) *Led brightness:* the user can set the brightness of the blue and white LEDs

7.8.1 SHxxXLS2TEMDIS modules

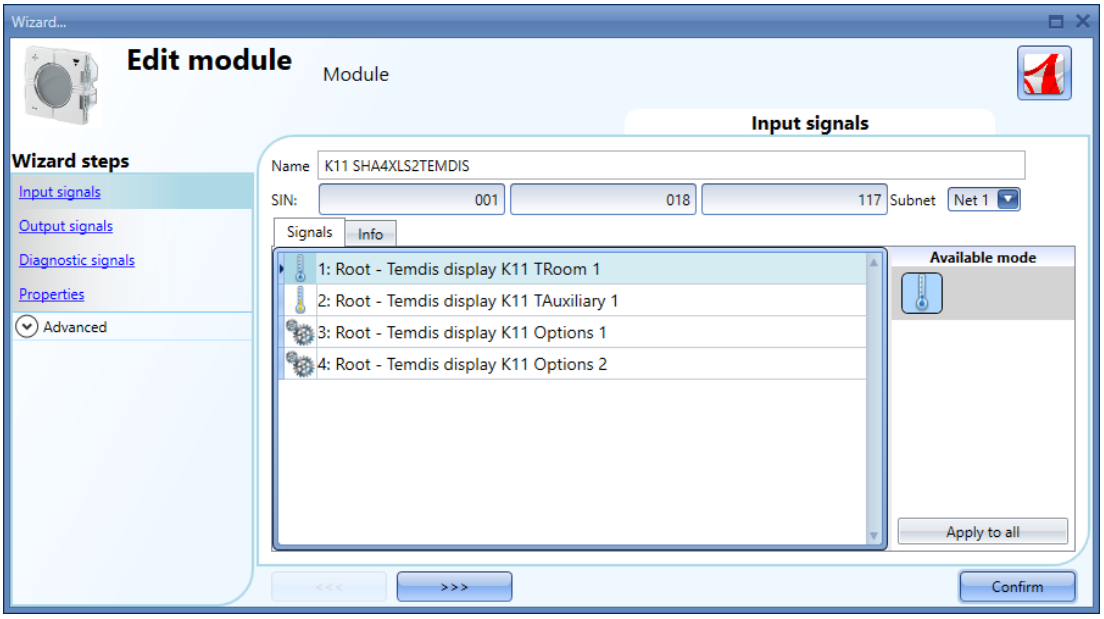
The SHA4XLS2TEMDIS and the SHE5XLS2TEMDIS are simplified TEMDIS displays with two push buttons that can be freely programmed and used in any function/automation:

To configure these, click on the relevant picture once it is added to the project:



SHA4XLS2TEMDIS	Net 1	K11 SHA4XLS2TEMDIS	000.000.000	Root	
SHE5XLS2TEMDIS	Net 1	K13 SHE5XLS2TEMDIS	000.000.000	Root	

The configuration wizard will appear:



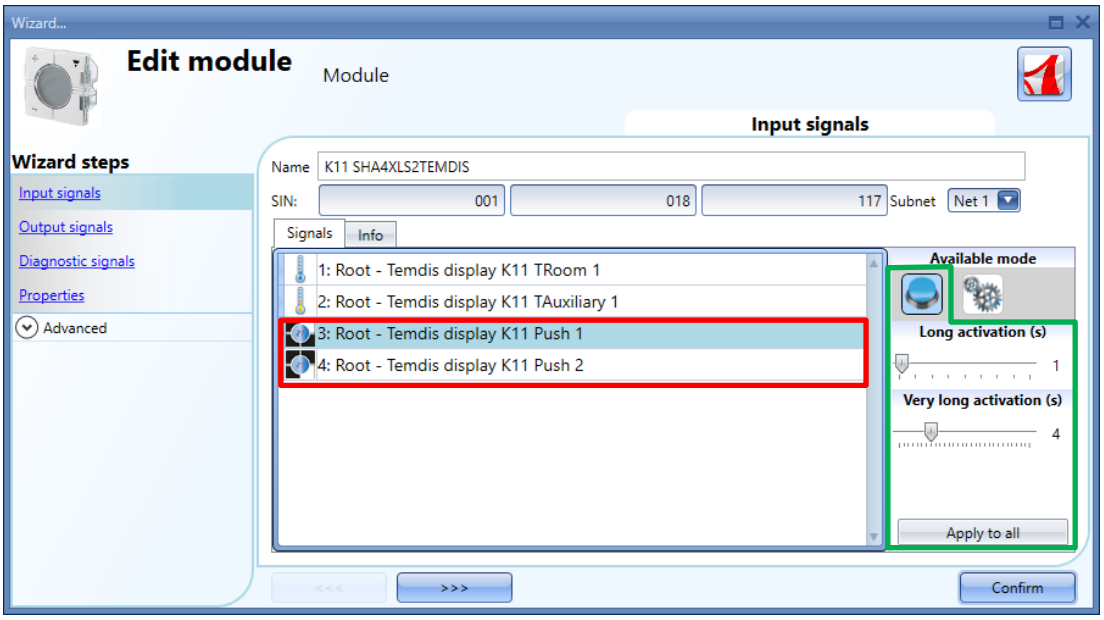
In the *Input signals* field, the temperature sensors, such as the *Troom1* signal and the optional *TAuxiliary1* signal, can be used in any temperature function.

The *Push1* and *Push2* keys are fully programmable and can be used in two different ways:

- as a **Push button** in any function / automation
- as a dedicated **Temdis key**

7.8.1.1 How to configure Push1 and Push2 as standard push buttons

Click on the icon for *Push1* or *Push2* key (see the red rectangle below), then in the *Available mode* window the user has to select the push button icon, as shown in the green rectangle below:



The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*

The *Long activation* and the *Very long activation* times have to be set in the same way as the mechanical push button.

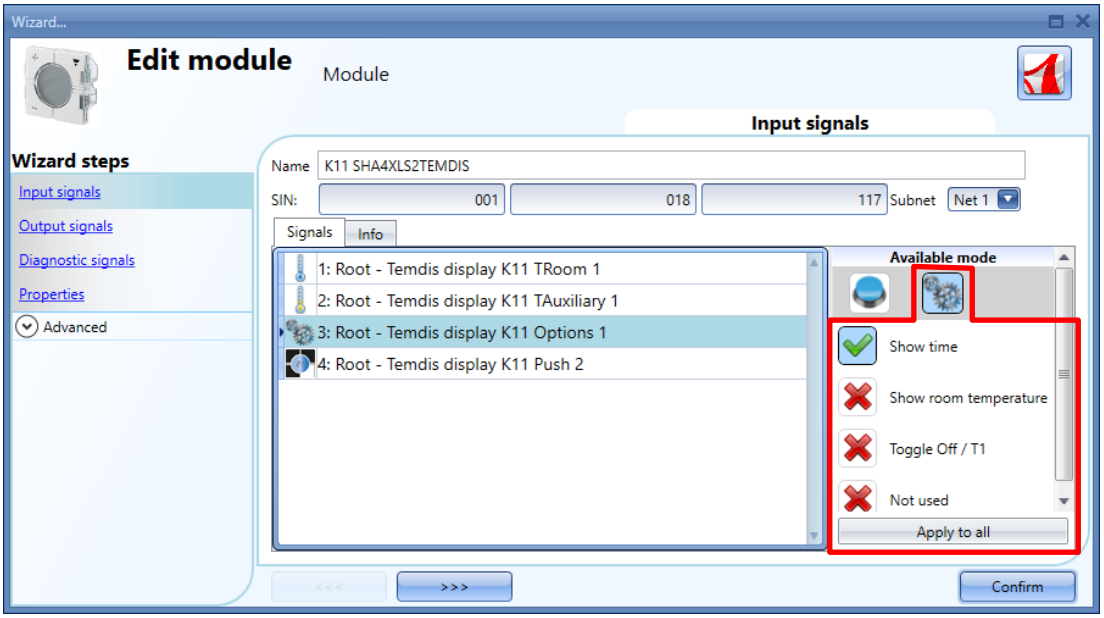
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and this is always set by the system for 3 seconds longer than the *Long activation* time. The short press and long press are recognized when the push button is released.

If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.

7.8.1.2 How to configure Push1 and Push2 as dedicated Temdis keys

Since this temperature display can control a temperature setpoint, with one On/Off command for the temperature function and one command to visualize the current time, each key can be freely associated to one of these functions. More than one key can be associated to the same functionality.

To configure *Push1* or *Push2* as a dedicated *Temdis key*, in the *Available mode* window the user has to click on the gear icon: the available Temdis functions will appear, as shown in the red rectangle below:



By means of the *Available mode* list, the user can associate the key to the required function:

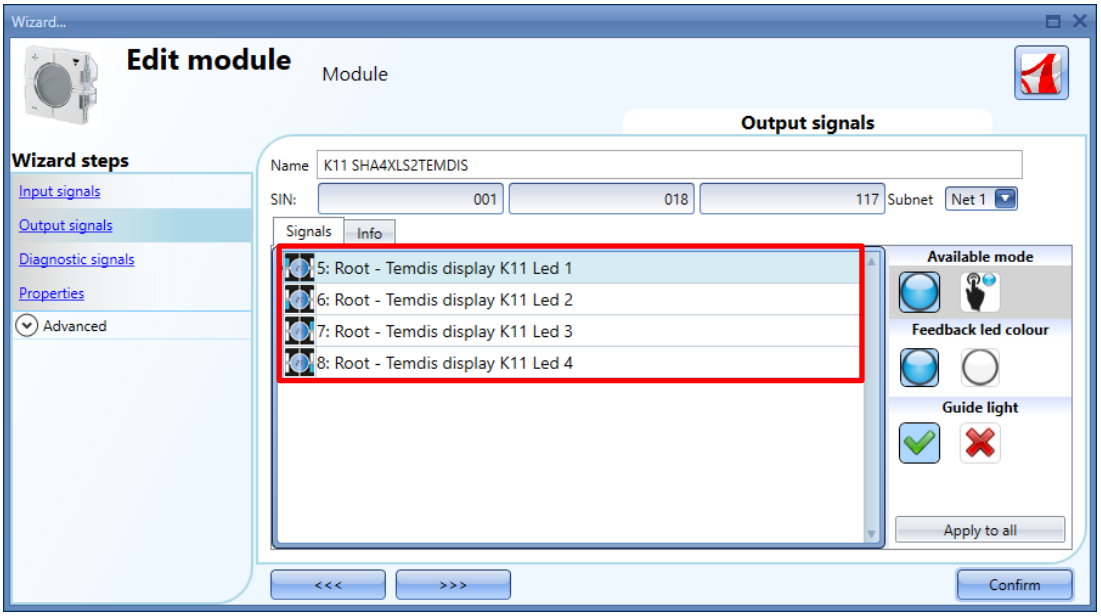
Temdis function	Behaviour
Show time	If the green *V* icon is selected for the *Show time* option, the time will be shown in the Temdis display when the key is pressed. After the *Backlight on time (sec)* value expires, the display will show the main visualization again.
Show room temperature	If the green *V* icon is selected for the *Show room temperature option*, the display shows the current room temperature when the key is pressed. After the *Backlight on time (sec)* value expires, the display will show the main visualization again. (i.e. setpoint t1 value)
Toggle Off/T1	If the green V icon is selected for the *Toggle Off/t1* option, when the key is pressed once, the setpoint t1 value is shown. By pressing the same key again, the temperature control will be switched On/Off. After the *Update time(sec)* value expires, the display will show the main visualization again.
Not used	No action will be executed when this key is pressed

N.B: The temperature display shows the room temperature in Celsius or Fahrenheit Degrees according to the project settings.

7.8.1.3 How to set the output signals

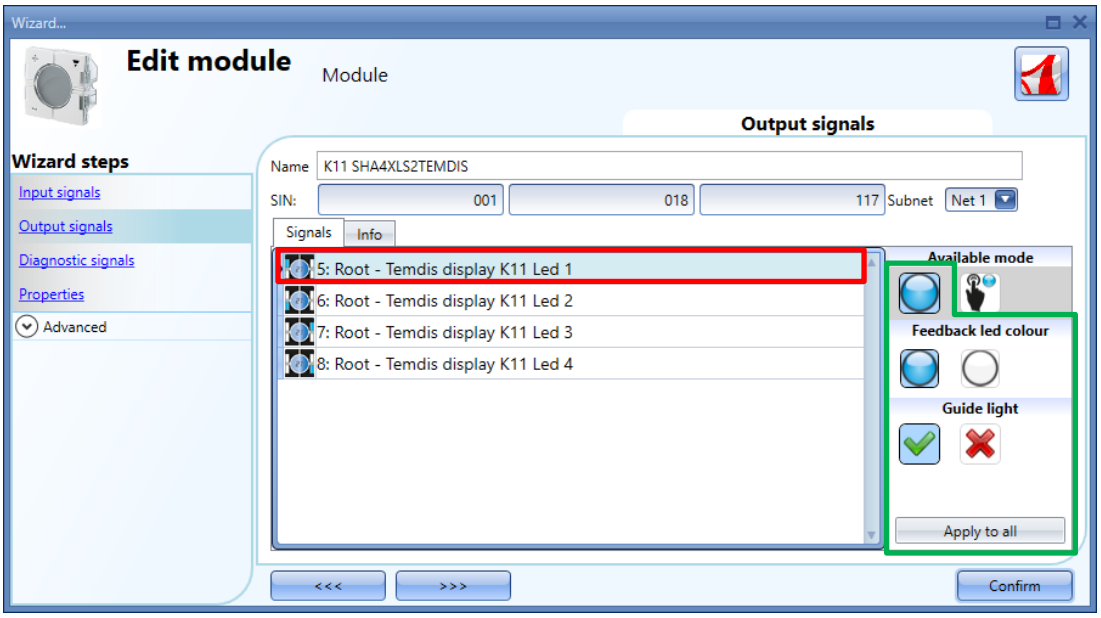
In the *Output signals* window, all the available feedback LEDs are shown: they can be freely programmed as feedback status LEDs for any function and automation or they can be used as feedback LEDs for the push button of the Temdis module. The guide light function is available for both working modes and can be enabled or disabled.

To configure these, click on the relevant picture once it is added to the project:



7.8.1.3.1 How to program a LED as generic feedback

In the *Available mode* window, the user has to click on the first icon at the top, as shown in the green rectangle below:



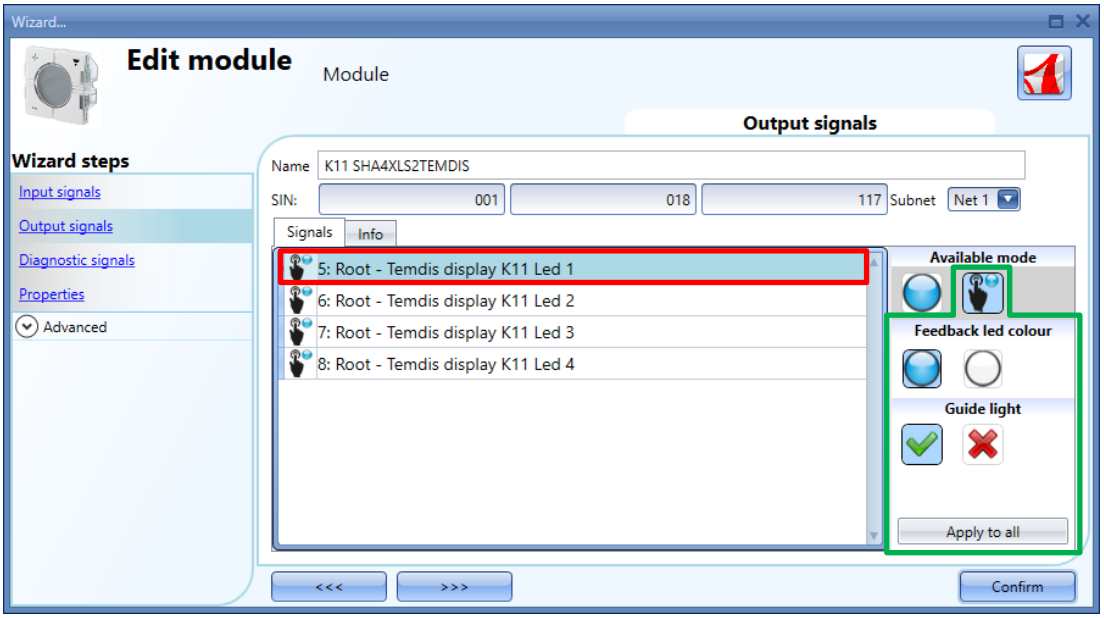
In the *Led colour* field the user can set the colour of the LED for the function feedback:

By clicking on the blue LED icon in the *Led colour* section (see blue rectangle in the picture above), the blue LED is used as feedback and the white LED is used as guide light, while by clicking on the white LED icon in the *Led colour* section, the white LED is used as feedback and the blue LED is used as guide light.

In the Guide light field, the user can enable or disable the guide light option:
By clicking on the white LED icon, the guide light is enabled, while by clicking on the red cross the guide light is disabled.

7.8.1.3.2 How to set a LED as pushbutton feedback

In the *Available mode* window, the user has to click on the second icon at the top, as shown in the green rectangle below:



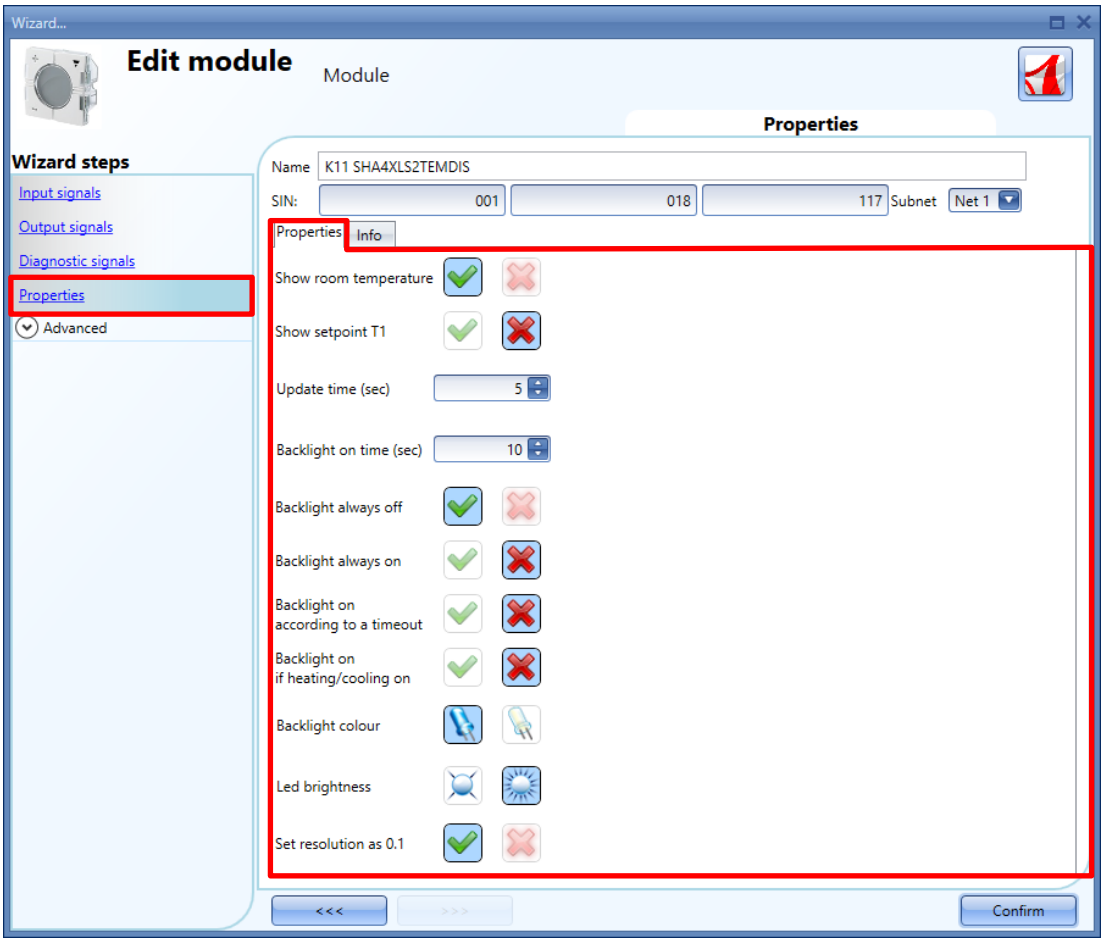
In the *Led colour* field the user can set the colour of the LED for the feedback when a push button is pressed:

By clicking on the blue LED icon in the *Led colour* section (see blue rectangle in the picture above), the blue LED turns On when the key is pressed, and the white LED is used as guide light, while by clicking on the white LED icon in the *Led colour* section, the white LED turns On when the key is pressed and the blue LED is used as guide light.

In the *Guide light* field, the user can enable or disable the guide light option:
By clicking on the white LED icon, the guide light is enabled, while by clicking on the red cross the guide light is disabled.

7.8.1.4 How to set the properties

In the *Properties* window of the module, the user can set the main visualization, the behaviour of the backlight and the timer options, as shown below:



The available options are shown in the table below:

Field name	Description
Show room temperature *Show setpoint T1* *(Main visualization)*	The main visualization can be selected from one of the following options: ▪ if the green *V* icon is selected for *Show room temperature* option, the temperature value is shown as main visualization, or: ▪ if the green *V* is selected for *Show setpoint T1* option, the setpoint t1 value is shown as main visualization *N.B: only one option can be selected at a time*
Update time (sec)	In this field the user can set the adjustment timeout (0 to 15 seconds) to exit from the regulation mode

Backlight on time (sec)	In this field the user can set the delay off timer (0 to 60 seconds) in order to switch the display backlight Off when the timer expires.
Backlight	The display backlight mode can be selected from one of the following options: -*Backlight always off:* when the green *V* is selected, the backlight is always off -*Backlight always on:* when the green *V* is selected, the backlight is always on -*Backlight on according to a timer:* If this property is enabled, the backlight is switched off when the timer set in *Backlight on time(sec)* field expires -*Backlight on if heating/cooling on:* the backlight is on when the zone temperature function is on for heating/cooling *N.B: only one option can be selected at a time*
Backlight colour	In this field the user can set the colour of the display backlight, in blue or in white
LED brightness	In this field the user can set the brightness of the blue and white LEDs
Set resolution as 0.1	If the green *V* is selected, the resolution of the display is 0.1°C instead of 0.5°C

7.8.1.5 How to set the temperature regulation signal

The room temperature shown in the Temdis display is the reference value used for the control of heating/cooling. The temperature signal can be selected from one of the following signals:

- *Temdis display TRoom1* signal;
- *Temdis display TAuxiliary* signal (if the BSO-TEMDIS floor sensor is connected to the Temdis module);

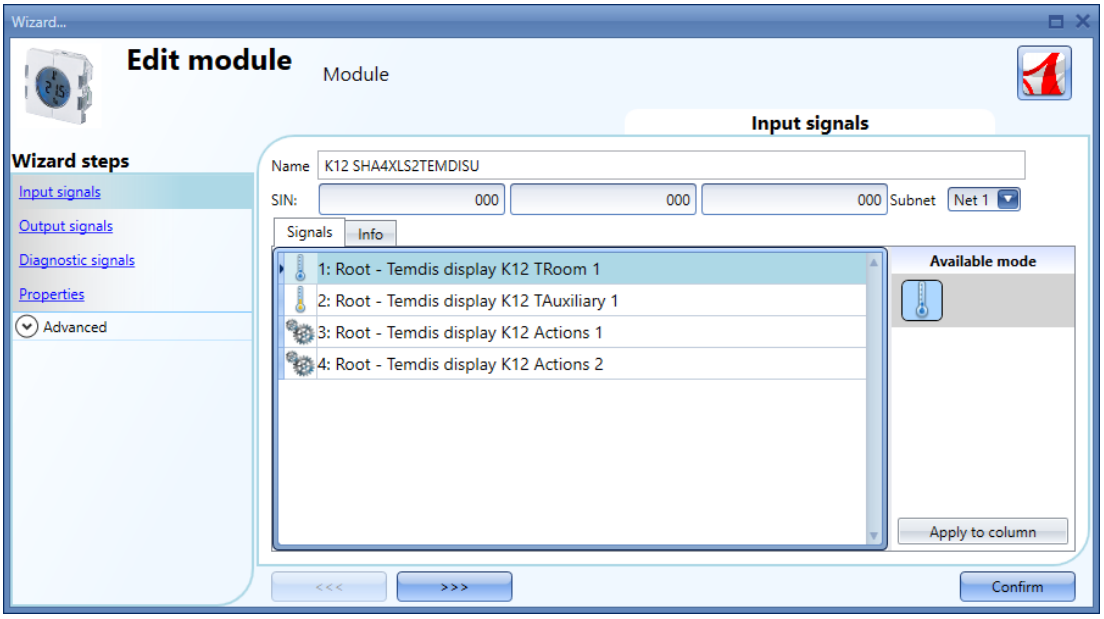
N.B: Refer to the paragraph *How to set a temperature function* in this manual for more details.

7.8.2 SHxxXLS2TEMDISU modules

The SHA4XLS2TEMDISU and the SHE5XLS2TEMDISU are simplified TEMDIS displays with two push buttons that can be freely programmed and used in any function/automation:

To configure these, click on the relevant picture once it is added to the project:

The configuration wizard will appear:



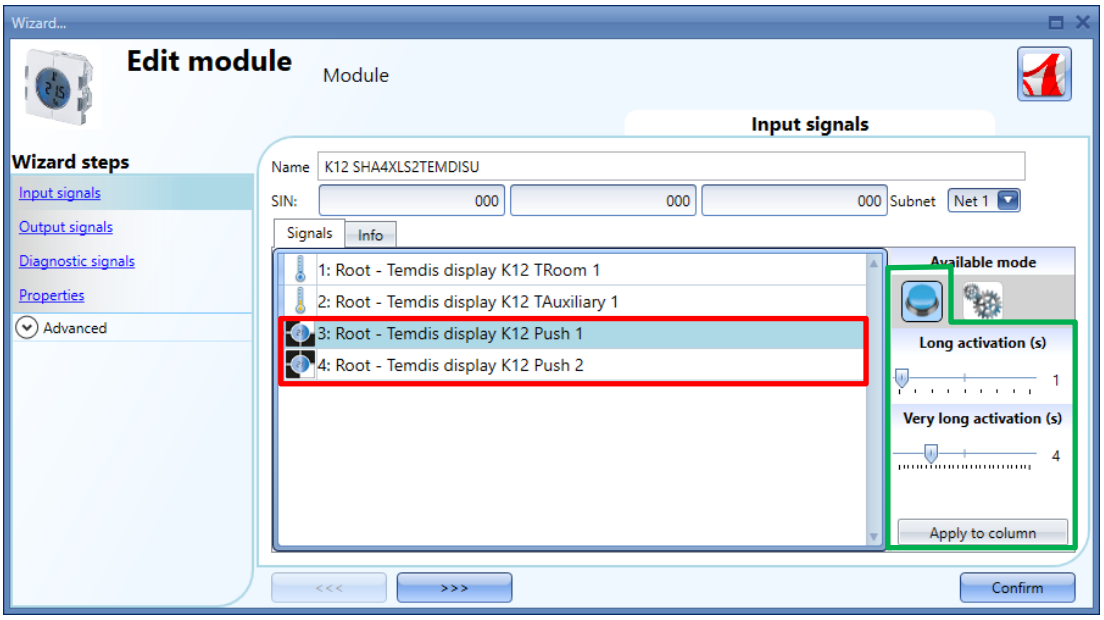
In the *Input signals* field, the temperature sensors, such as the *Troom1* signal and the optional *TAuxiliary1* signal, can be used in any temperature function.

The *Push1* and *Push2* keys are fully programmable and can be used in two different ways:

- as a **Push button** in any function / automation
- as a dedicated **Temdis key**

7.8.2.1 How to configure Push1 and Push2 as standard push buttons

Click on the icon for *Push1* or *Push2* key (see the red rectangle below), then in the *Available mode* window the user has to select the push button icon, as shown in the green rectangle below:



The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*

The *Long activation* and the *Very long activation* times have to be set in the same way as the mechanical push button.

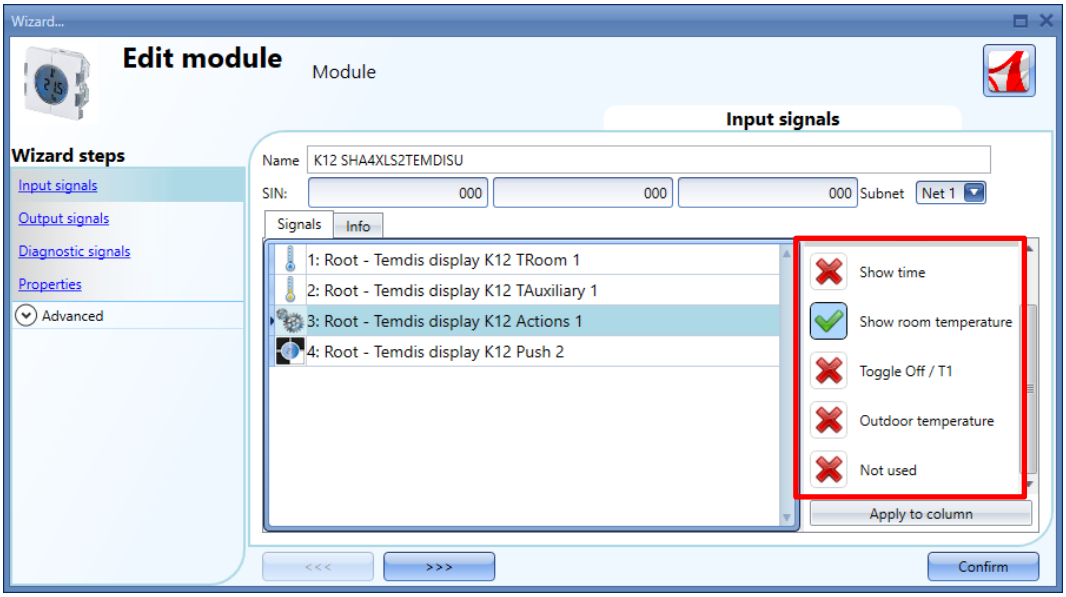
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and this is always set by the system for 3 seconds longer than the *Long activation* time. The short press and long press are recognized when the push button is released.

If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.

7.8.2.2 How to configure Push1 and Push2 as dedicated Temdis keys

Since this temperature display can control a temperature setpoint, with one On/Off command for the temperature function and one command to visualize the current time, each key can be freely associated to one of these functions. More than one key can be associated to the same functionality.

To configure *Push1* or *Push2* as a dedicated *Temdis key*, in the *Available mode* window the user has to click on the gear icon: the available Temdis functions will appear, as shown in the red rectangle below:



By means of the *Available mode* list, the user can associate the key to the required function:

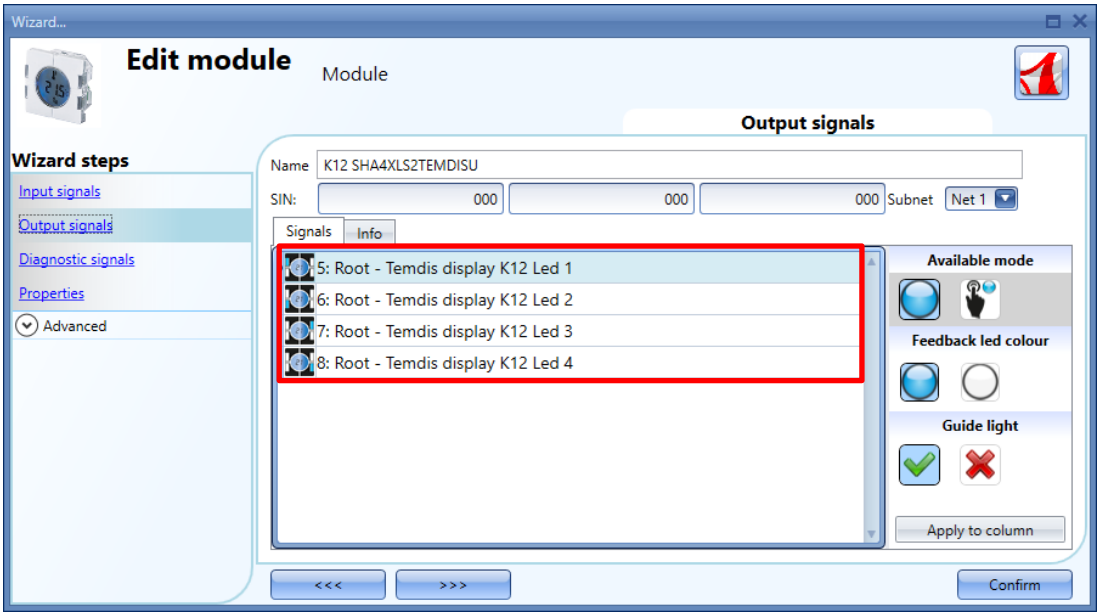
Temdis function	Behaviour
Show time	If the green *V* icon is selected for the *Show time* option, the time will be shown in the Temdis display when the key is pressed. After the *Backlight on time (sec)* value expires, the display will show the main visualization again.
Show room temperature	If the green *V* icon is selected for the *Show room temperature option*, the display shows the current room temperature when the key is pressed. After the *Backlight on time (sec)* value expires, the display will show the main visualization again. (i.e. setpoint t1 value)
Toggle Off/T1	If the green V icon is selected for the *Toggle Off/t1* option, when the key is pressed once, the setpoint t1 value is shown. By pressing the same key again, the temperature control will be switched On/Off. After the *Update time(sec)* value expires, the display will show the main visualization again.
Outdoor temperature	If the green *V* icon is selected for the *Outdoor temperature option*, the display shows the outdoor temperature when the key is pressed. After the *Backlight on time (sec)* value expires, the display will show the main visualization again. (i.e. setpoint t1 value)
Not used	No action will be executed when this key is pressed

N.B: The temperature display shows the room temperature in Celsius or Fahrenheit Degrees according to the project settings.

7.8.2.3 How to set the output signals

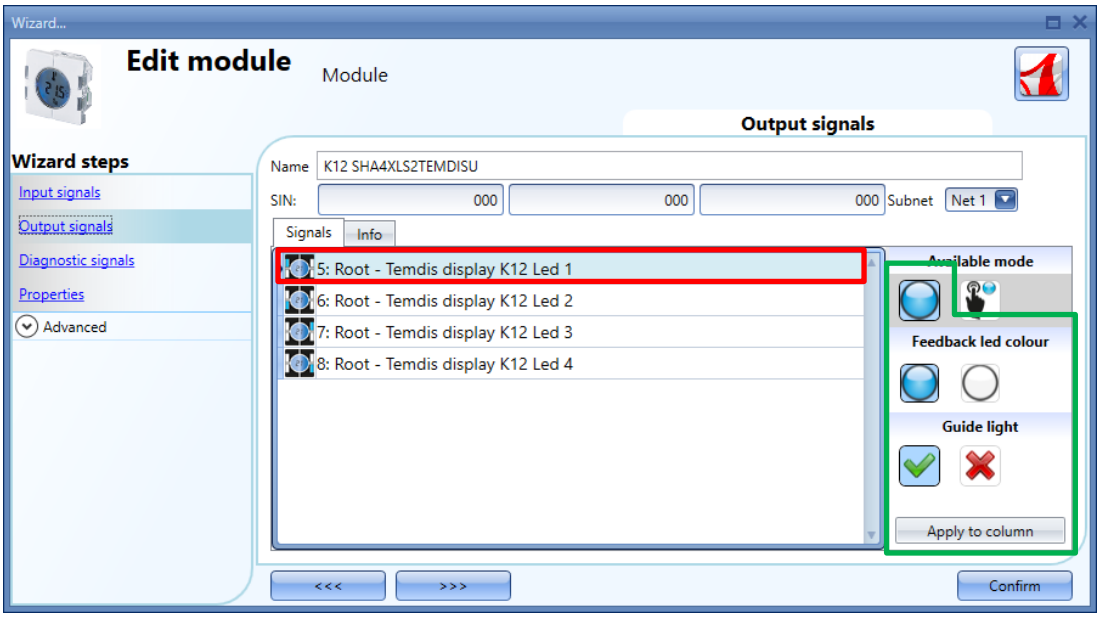
In the *Output signals* window, all the available feedback LEDs are shown: they can be freely programmed as feedback status LEDs for any function and automation or they can be used as feedback LEDs for the push button of the Temdis module. The guide light function is available for both working modes and can be enabled or disabled.

To configure these, click on the relevant picture once it is added to the project:



7.8.2.3.1 How to program a LED as generic feedback

In the *Available mode* window, the user has to click on the first icon at the top, as shown in the green rectangle below:



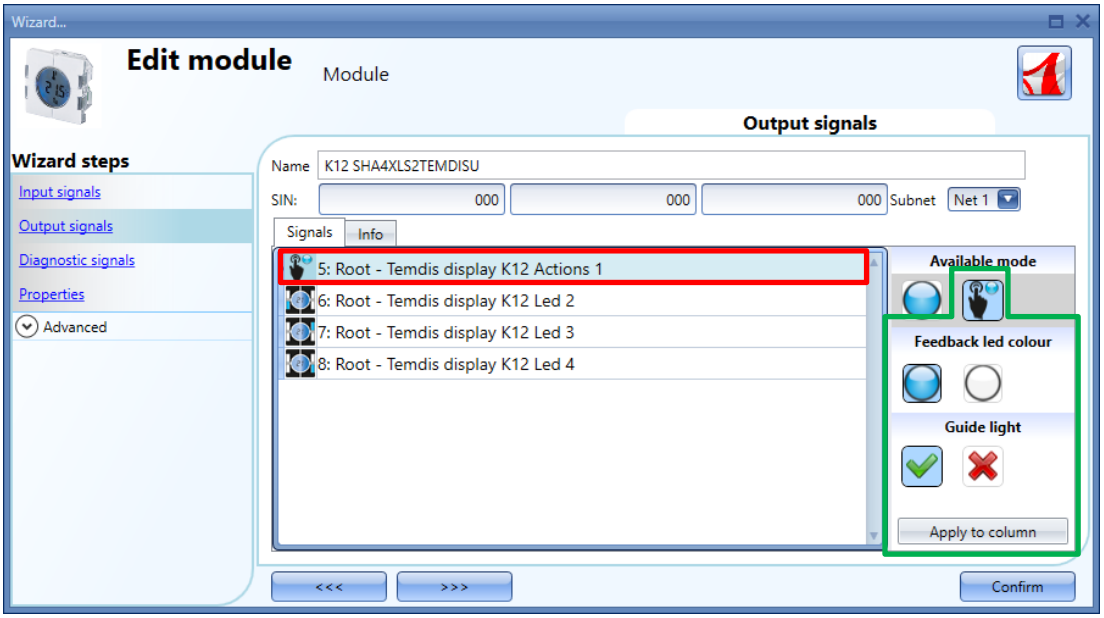
In the *Led colour* field, the user can set the colour of the LED for the function feedback:

By clicking on the blue LED icon in the *Led colour* section (see blue rectangle in the picture above), the blue LED is used as feedback and the white LED is used as guide light, while by clicking on the white LED icon in the *Led colour* section, the white LED is used as feedback and the blue LED is used as guide light.

In the Guide light field, the user can enable or disable the guide light option:
By clicking on the white LED icon, the guide light is enabled, while by clicking on the red cross the guide light is disabled.

7.8.2.3.2 How to set a LED as pushbutton feedback

In the *Available mode* window, the user has to click on the second icon at the top, as shown in the green rectangle below:



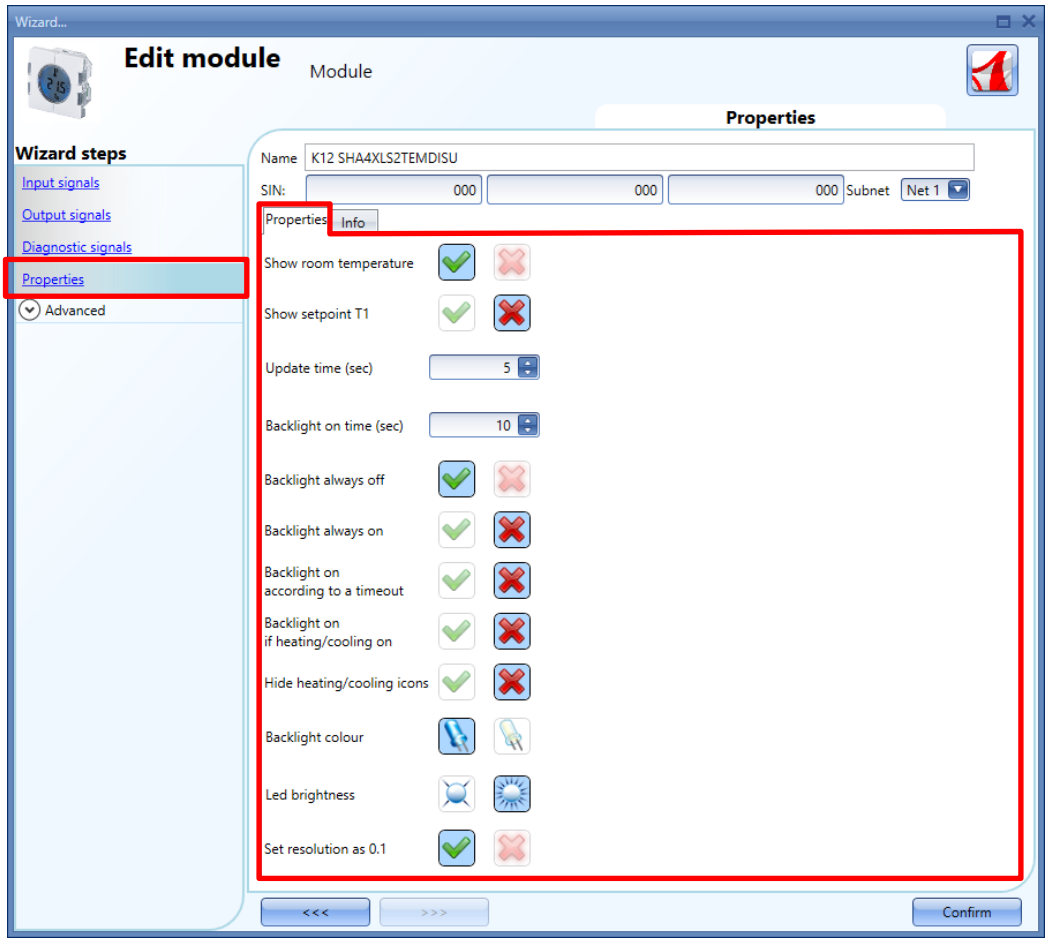
In the *Led colour* field the user can set the colour of the LED for the feedback when a push button is pressed:

By clicking on the blue LED icon in the *Led colour* section (see blue rectangle in the picture above), the blue LED turns On when the key is pressed, and the white LED is used as guide light, while by clicking on the white LED icon in the *Led colour* section, the white LED turns On when the key is pressed and the blue LED is used as guide light.

In the *Guide light* field, the user can enable or disable the guide light option:
By clicking on the white LED icon, the guide light is enabled, while by clicking on the red cross the guide light is disabled.

7.8.2.4 How to set the properties

In the *Properties* window of the module, the user can set the main visualization, the behaviour of the backlight and the timer options, as shown below:



The available options are shown in the table below:

Field name	Description
Show room temperature *Show setpoint T1* *(Main visualization)*	The main visualization can be selected from one of the following options: ▪ if the green *V* icon is selected for *Show room temperature* option, the temperature value is shown as main visualization, or: ▪ if the green *V* is selected for *Show setpoint T1* option, the setpoint t1 value is shown as main visualization *N.B: only one option can be selected at a time*
Update time (sec)	In this field the user can set the adjustment timeout (0 to 15 seconds) to exit from the regulation mode

Backlight on time (sec)	In this field the user can set the delay off timer (0 to 60 seconds) in order to switch the display backlight Off when the timer expires.
Backlight [option]	The display backlight mode can be selected from one of the following options: -*Backlight always off:* when the green *V* is selected, the backlight is always off -*Backlight always on:* when the green *V* is selected, the backlight is always on -*Backlight on according to a timer:* If this property is enabled, the backlight is switched off when the timer set in *Backlight on time(sec)* field expires -*Backlight on if heating/cooling on:* the backlight is on when the zone temperature function is on for heating/cooling *N.B: only one option can be selected at a time*
Hide heating/cooling icons	If the green *V* option is selected, the icons will not be shown on the *Temdis* display even if the heating/cooling functions are working
Backlight colour	In this field the user can set the colour of the display backlight, in blue or in white
LED brightness	In this field the user can set the brightness of the blue and white LEDs

7.8.2.5 How to set the temperature regulation signal

The room temperature shown in the Temdis display is the reference value used for the control of heating/cooling. The temperature signal can be selected from one of the following signals:

- *Temdis display TRoom1* signal;
- *Temdis display TAuxiliary* signal (if the BSO-TEMDIS floor sensor is connected to the Temdis module);

N.B: Refer to the paragraph *How to set a temperature function* in this manual for more details.

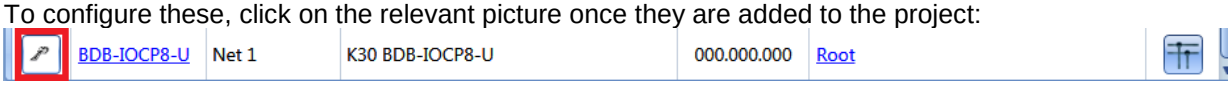
7.9 Decentral modules

The *Decentral module* family can be split into two main groups:

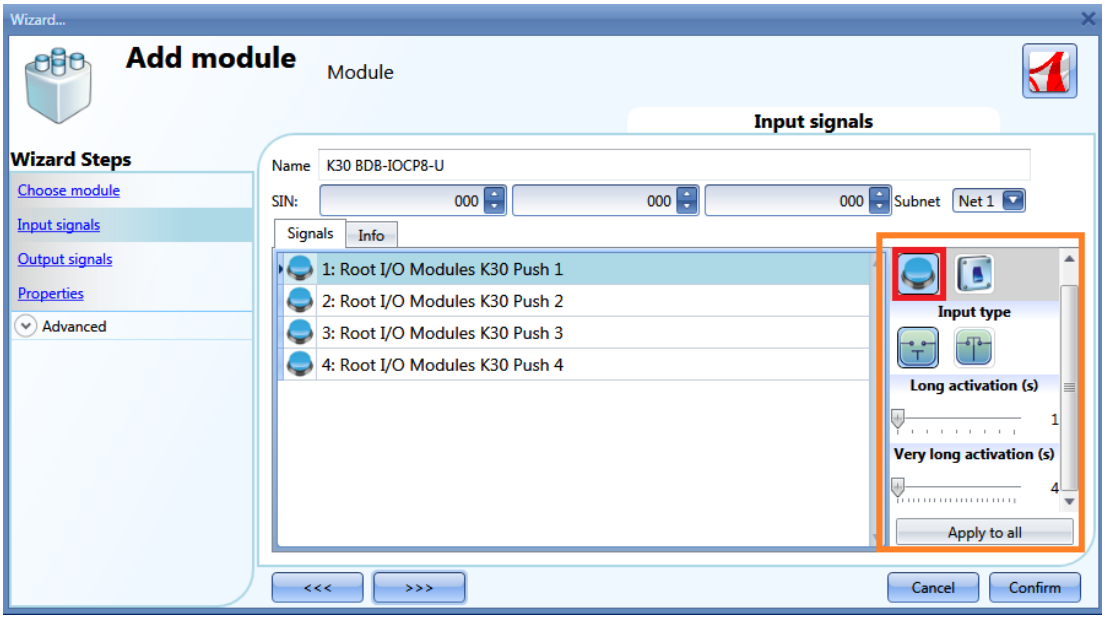
7.9.1 Digital input/output modules:

BDB-IOCP8-U
BDB-IOCP8A-U
BDB-INCON4-U
BDB-INCON8-U
BDD-INCON4-U

To configure these, click on the relevant picture once they are added to the project:

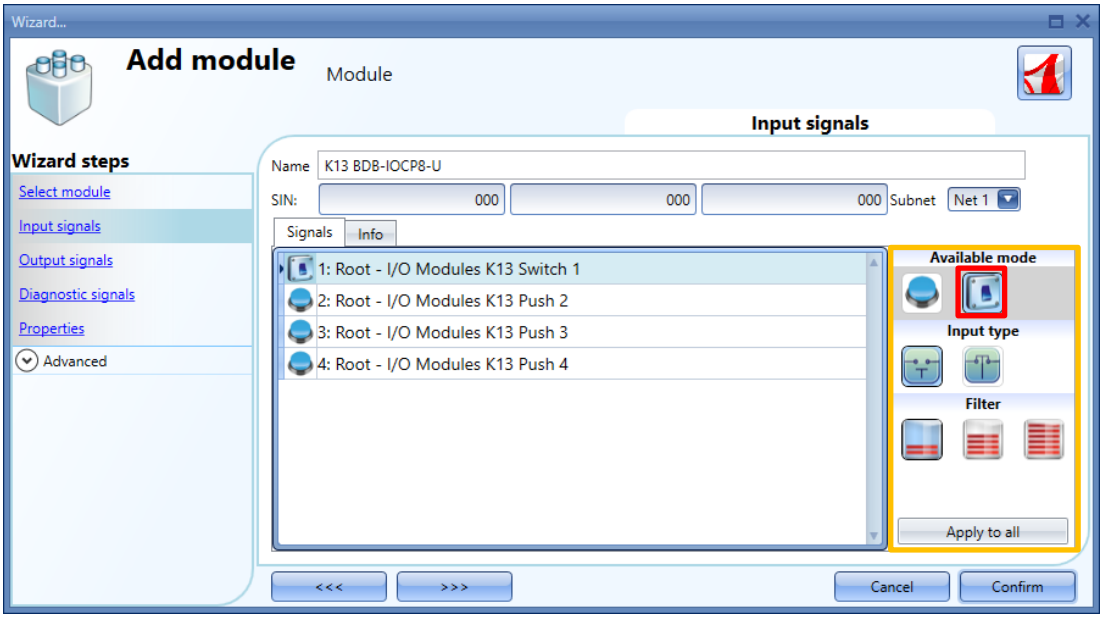


The configuration wizard will appear:



There are configurable settings only in the *Input signals* windows.
Each input can have its own settings, and the first thing to do is to select the behaviour: each input can be configured as a push button or a switch.

1) *Input configured as a pushbutton*: if the behaviour is configured as a push button, such as for light switches, the *Long activation* and *Very long activation* time has to be set. The user can also select if the input is normally closed or normally open, by clicking on the relevant *Input type* icons.
2) *Input configured as a switch*: if the behaviour is configured as a switch, the user can select if the input is normally closed or normally open, by clicking on the relevant *Input type* icons. Then the type of filtering of the signal on the bus can be selected from three options: low, average and high filtering. This should be selected according to how noisy the bus is (see picture below).

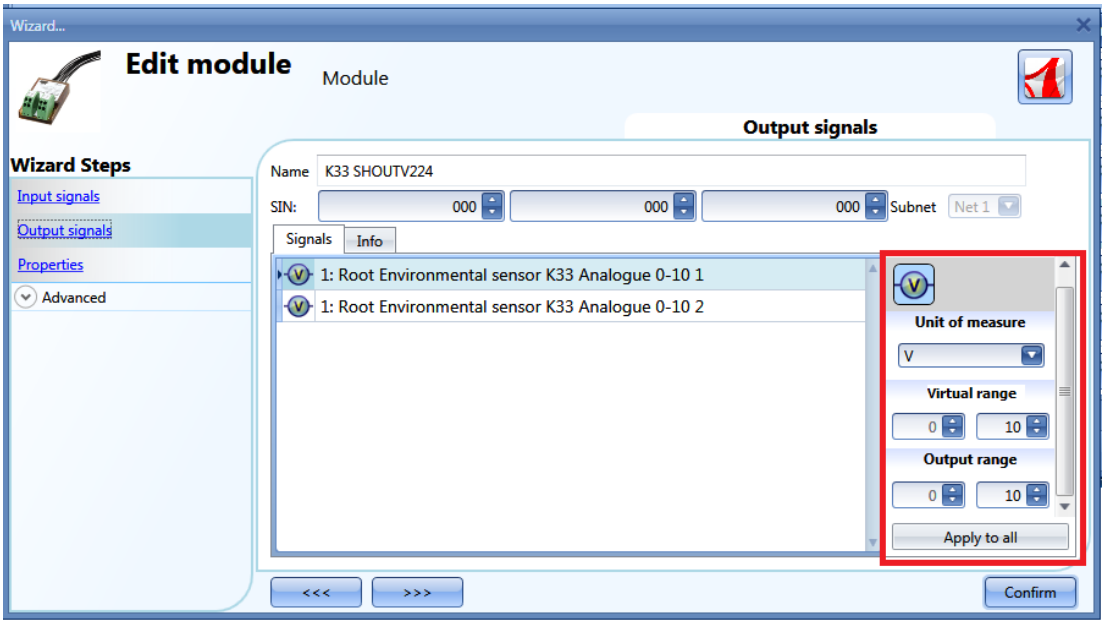

Wizard...
Add module
Module
Input signals
Wizard steps
Select module
Input signals
Output signals
Diagnostic signals
Properties
Advanced
Name
K13 BDB-IOCP8-U
SIN:
000
000
000
Subnet
Net 1
Signals
Info
1: Root - I/O Modules K13 Switch 1
2: Root - I/O Modules K13 Push 2
3: Root - I/O Modules K13 Push 3
4: Root - I/O Modules K13 Push 4
Available mode
Input type
Filter
Apply to all
<<<
>>>
Cancel
Confirm

7.9.2 Analogue input/output modules

SHPOUTV224
SHPINV324
SHPINV2T1P124
SHPINT1P1
SHPINNI2
SHPINA224

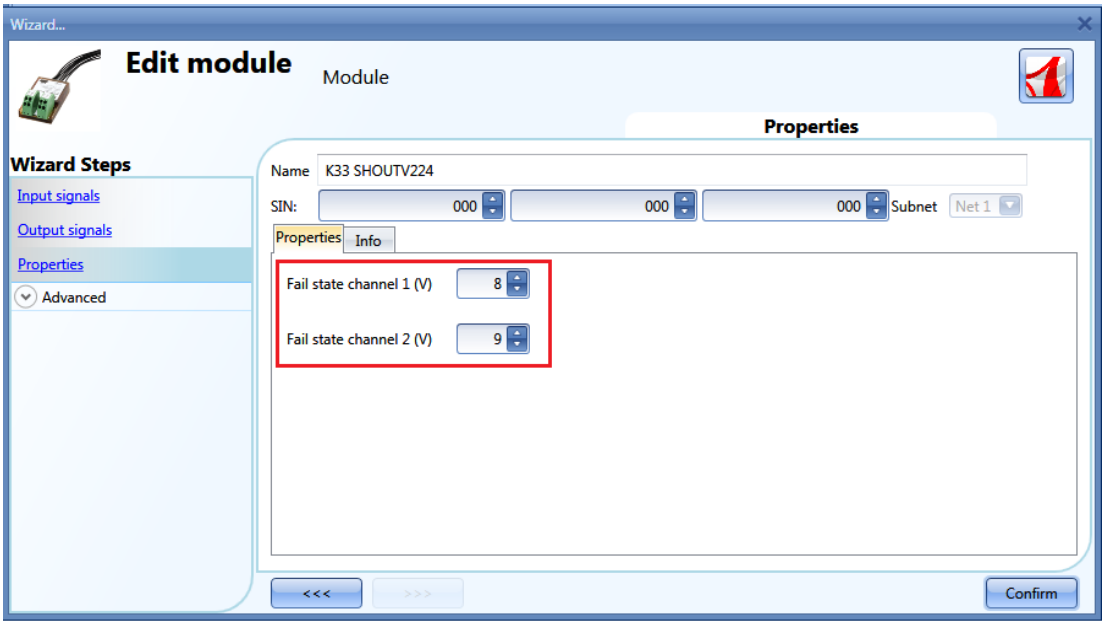
To configure the SHPOUTV224, click on the relevant picture once it is added to the project:

The configuration wizard will appear:



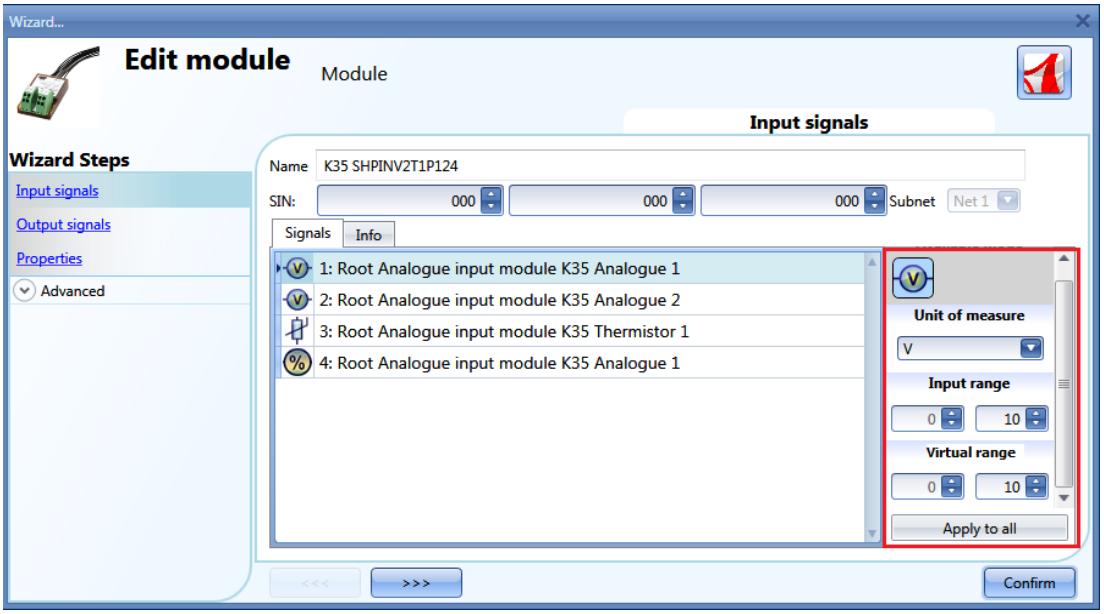
Since this is a 0 to 10V output module, it is a general purpose module and the output can be converted into many different units of measurement.
Those available can be selected from the field *Unit of measurement*: the icon will also be changed accordingly.
The "virtual" range of the selected unit of measurement has to be set in *Virtual range,* while the relevant output value in Volts has to be set in *Output range*. The conversion from the virtual to the voltage output range is linear.

In *Properties*, the user can define the output value if the Dupline bus is faulty or not connected (see picture below).



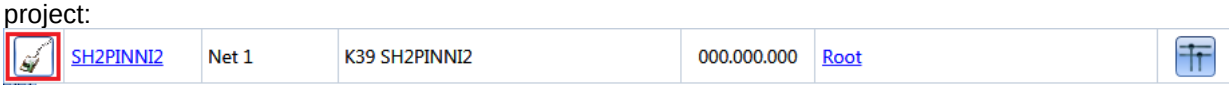
To configure the analogue input module (SHPINV324, SHPINV2T1P124, SHPINT1P1, SHPINA224), click on the relevant picture once it is added to the project:

The configuration wizard will appear:

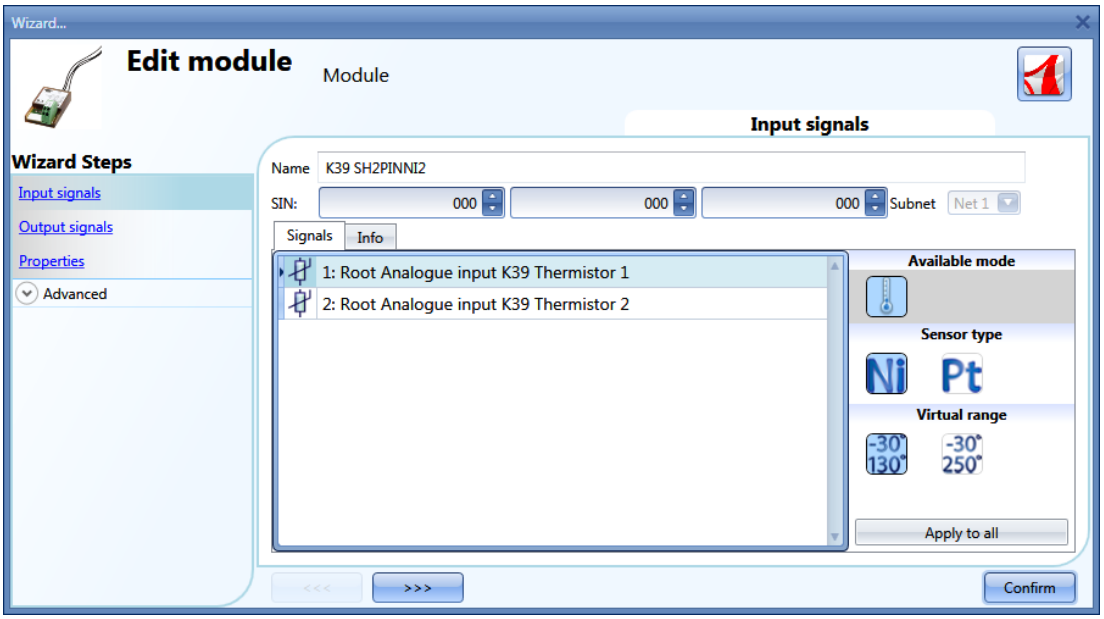


Since this is a general purpose input module, the value can be converted into many different units of measurement.
Those available can be selected from the field *Unit of measurement*: the icon will also be changed accordingly.
The "virtual" range of the selected unit of measurement has to be set in *Virtual range,* while the relevant input value has to be set in *Input range (*it might be a 0-10V value or a percentage according to the selected input*)*. The conversion from the virtual to the real input range is linear. The virtual input can then be used in all the functions, logged into the webserver and read via the Modbus TCP/IP.

To configure the analogue input module SHPINNI2, click on the relevant icon once it is added to the project:



The configuration wizard will appear:



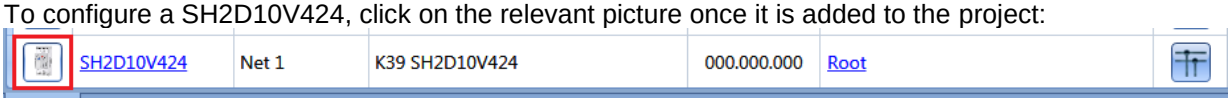
From this window the user can select the type of sensor connected to this input module and the input range.

7.10 Cabinet modules

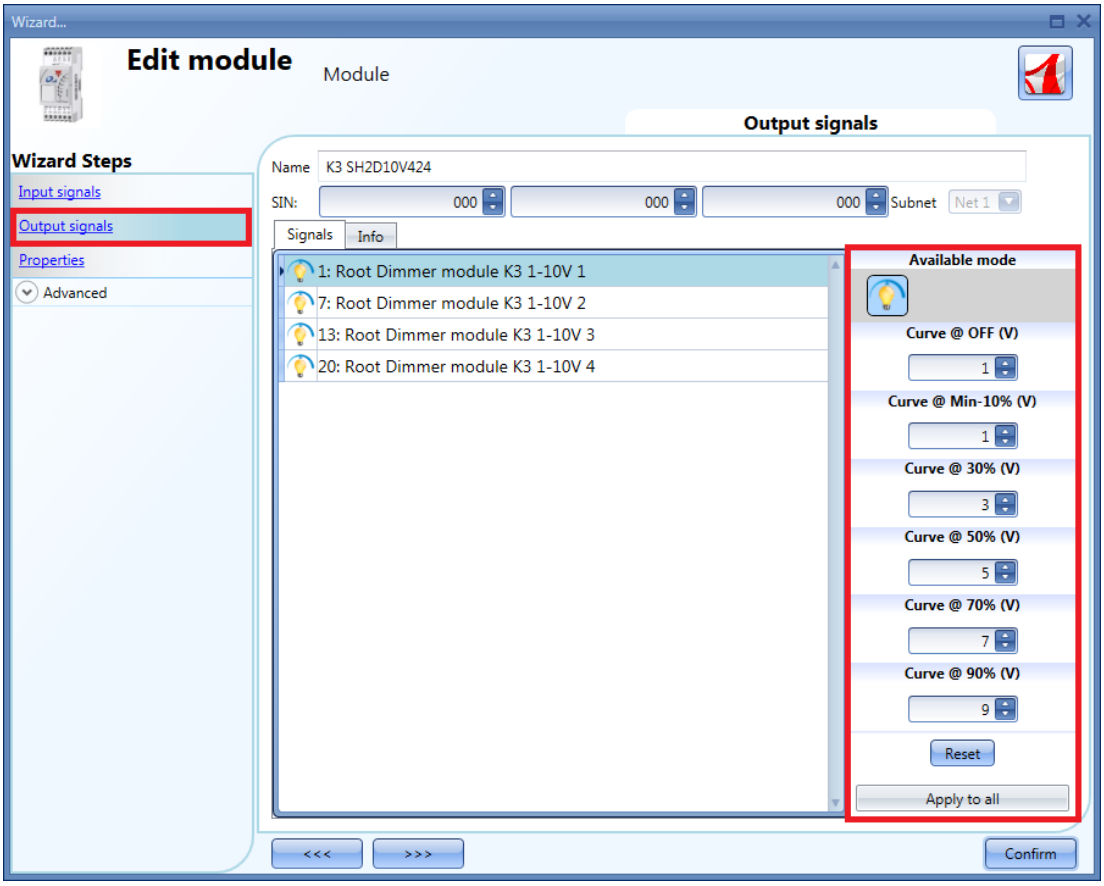
The following items belong to the *Cabinet module* family:

SH2D10V424
SH2D500WE230 (this does not require any configuration)
SH2D500W1230 (this does not require any configuration)
SH2INDI424
SH2RE16A2E230
SH2RE16A4
SH2RE1A424
SH2RODC224
SH2ROAC224
SH2SSTRI424

To configure a SH2D10V424, click on the relevant picture once it is added to the project:

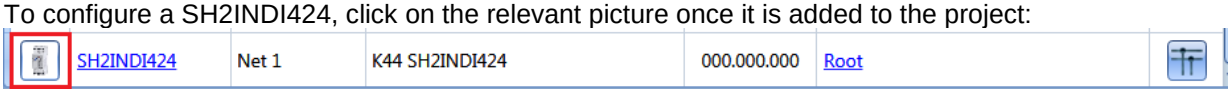


The configuration wizard will appear:

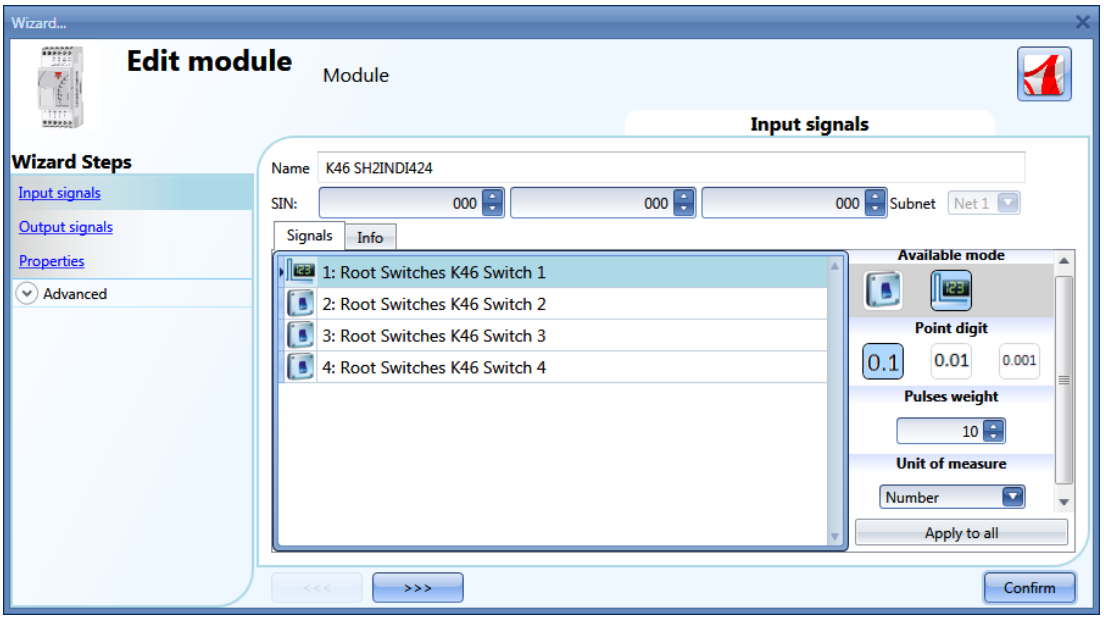


In the *Output signals* field the user can set the output curve according to the input voltage level required by the connected load. The curve is divided into six steps (OFF, 10%, 30%, 50%, 70%, 90%) and for each step the voltage level can be defined.

To configure a SH2INDI424, click on the relevant picture once it is added to the project:



The configuration wizard will appear:

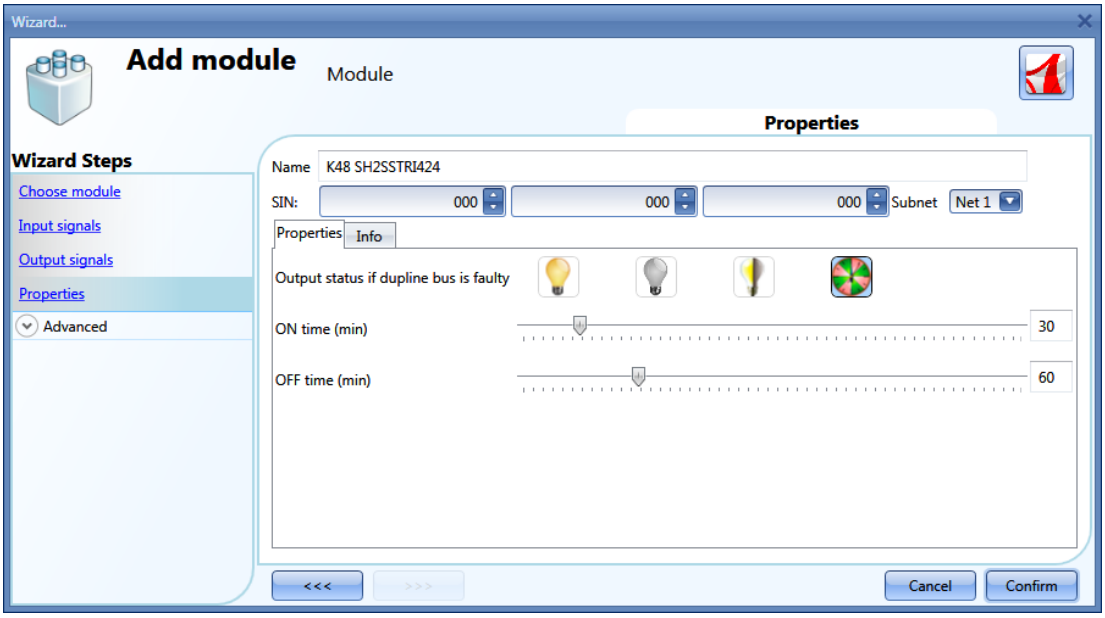


As for the settings of the SH2INDI424, the user has to select the working mode: it might be configured as a switch (the switch icon has to be selected), as a pulse counter by selecting the icon with *123*, or as a push button. With this configuration the weight of each pulse can also be set in the *Pulse weight* field.

See *How to configure a counter function* to set this module for people counting.

To configure the SH2SSTRI424 and the SH2RE16A2E230, click on the relevant picture once they are added to the project:

The configuration wizard will appear:

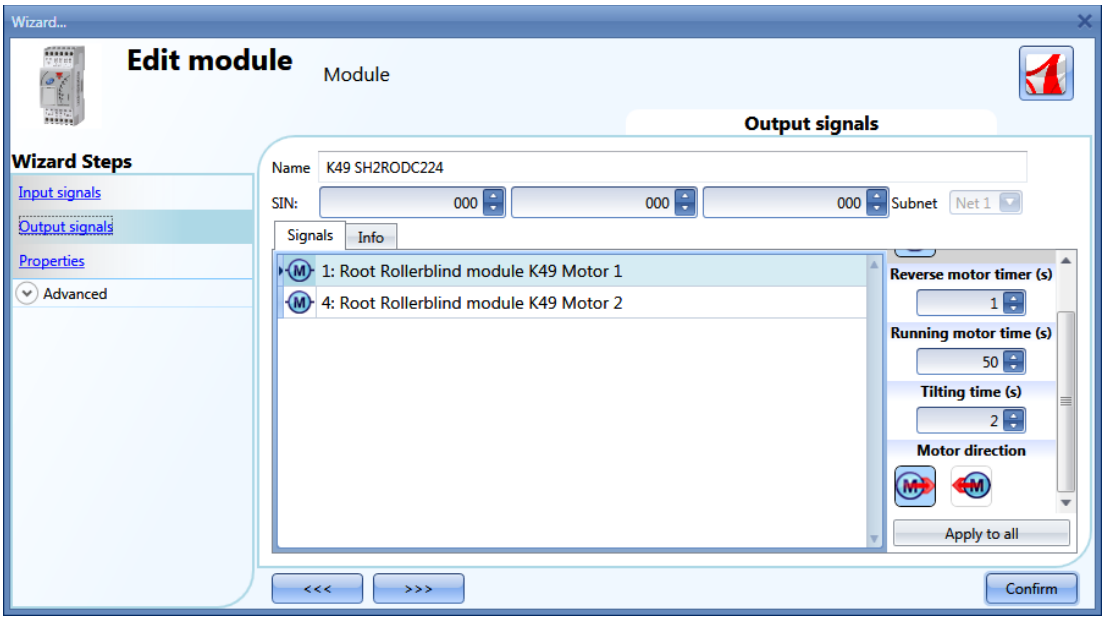


As for the setting of the output modules SH2SSTRI424 and SH2RE16A2E230, the user has to select the fail-safe working mode when the bus is faulty or not connected.
There are 4 available setups:

1) Output always on (yellow bulb)
2) Output always off (grey bulb)
3) The output maintains the status it had before the disconnection of the bus (yellow and grey bulb)
4) The output recycles: it will be on for the *On time*, and off for the *Off time.*

To configure the SH2ROAC224 and the SH2RODC224, click on the relevant picture once they are added to the project:

The configuration wizard will appear:



As for the setting of the roller blind module SH2RODC224 and the SH2ROCC224, the user has to set the *Reverse motor time* (the pause the motor takes before changing its direction), the *Running motor time* (the time the blind takes to go to a fully closed position from a fully open one) and the *Tilting time* (the time the motor needs to do a 180° rotation of the slats).

N.B. The modules with serial numbers higher then MN47xxxx can be programmed with different up/down running times.
All the modules produced before these dates can only be programmed with one running time.

7.11 MasterGlass switches

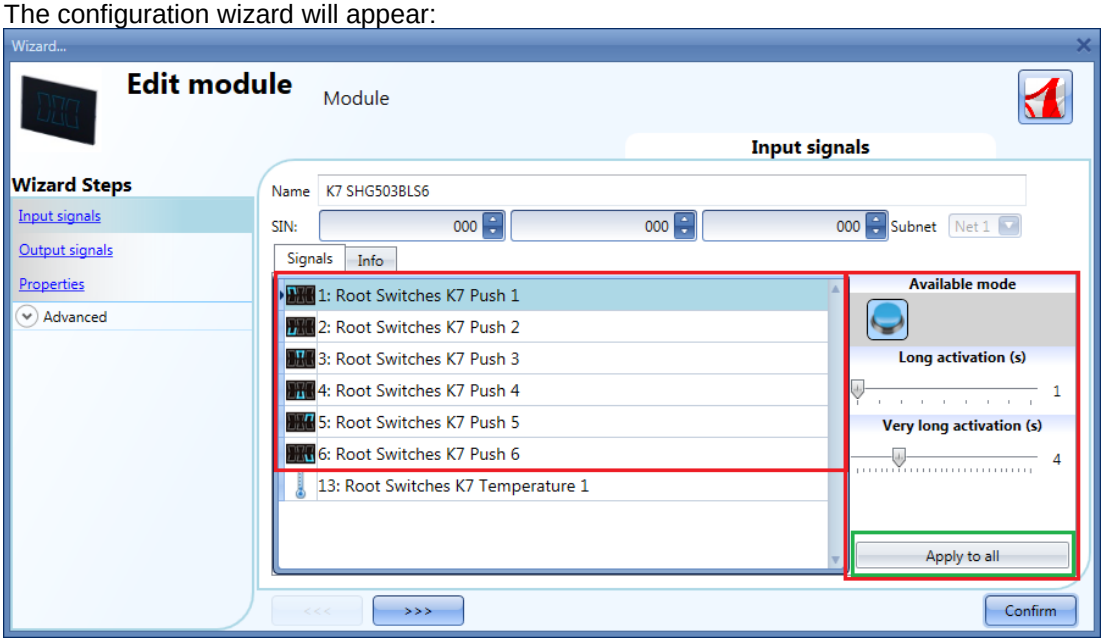
The following items belong to the Master*Glass switches* family:

SHG503WLS6
SHG503BLS6
SHG060WLS4
SHG060BLS4
SHG503WSLD
SHG503BSLD
SHG060WSLD
SHG060BSLD

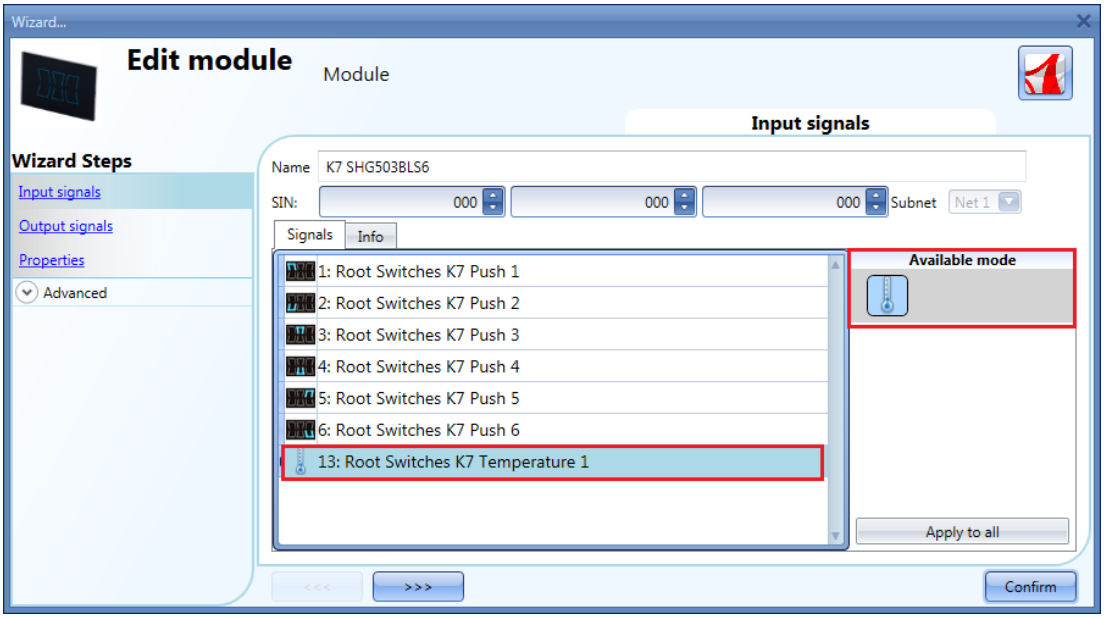
To configure these, click on the relevant picture once they are added to the project:

	SHG503BLS6	Net 1	K7 SHG503BLS6	000.000.000	Root	
	SHG503WLS6	Net 1	K2 SHG503WLS6	000.000.000	Root	

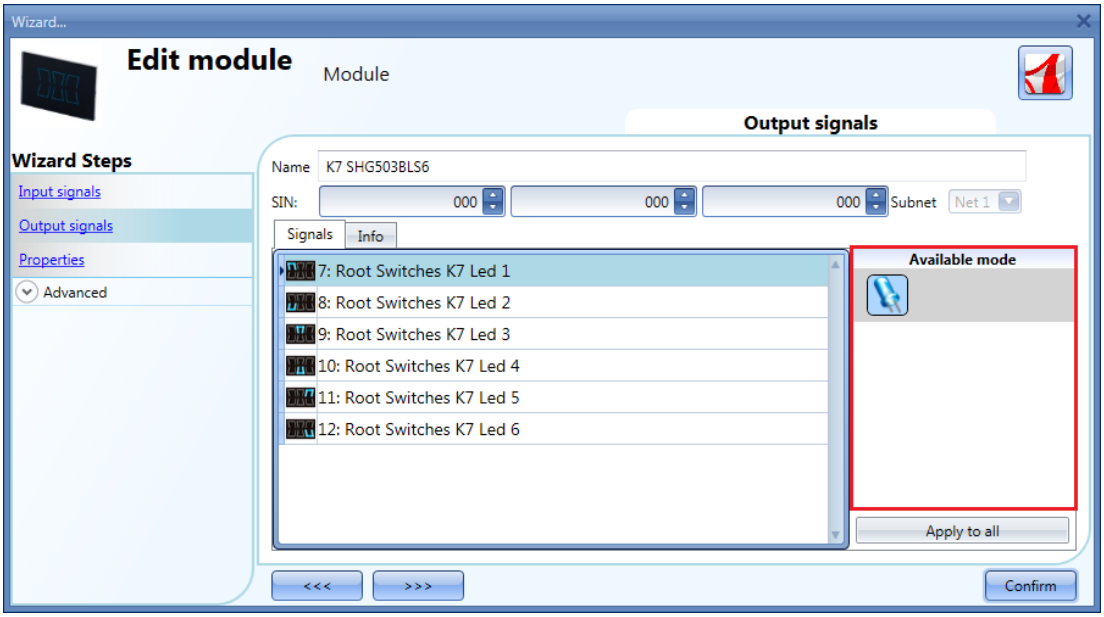
The configuration wizard will appear:



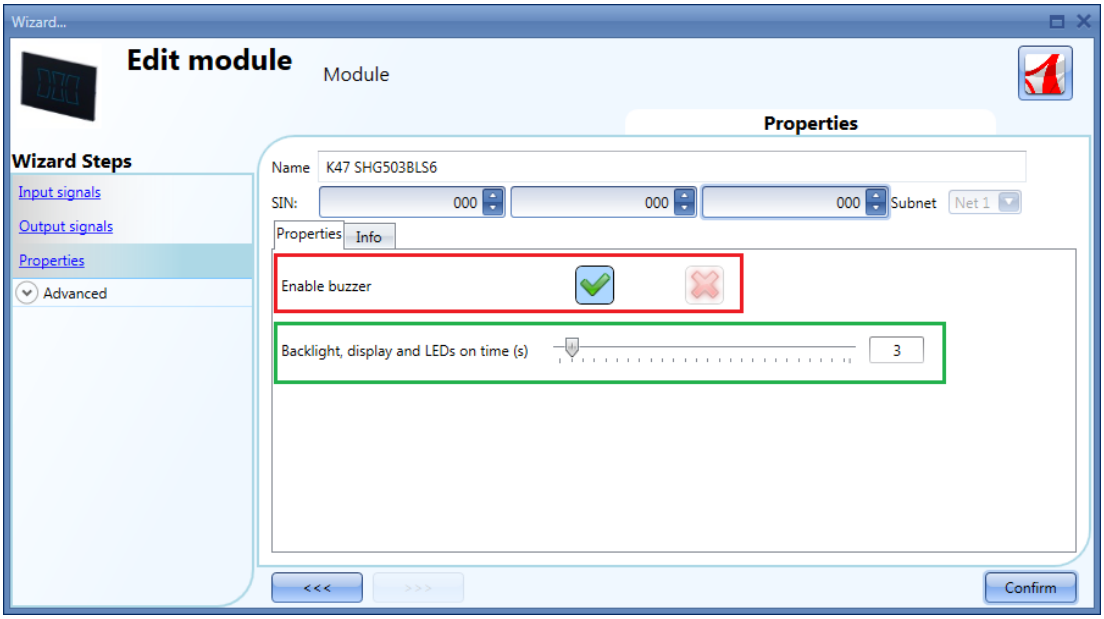
In the *Input signals* field, the *Long activation* and the *Very long* activation time has to be defined.
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and it is always set by the system 3 seconds longer than the *Long activation* time.
The short press and long press are recognized when the push button is released.
The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*
If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.



The SHG503WLS6 and SHG503BLS6 are glass switches with temperature sensors that can be used in any temperature function.



In the *Output signals* window, all the available feedback LEDs are shown: they can be freely programmed as feedback status LEDs for any smart-house function.

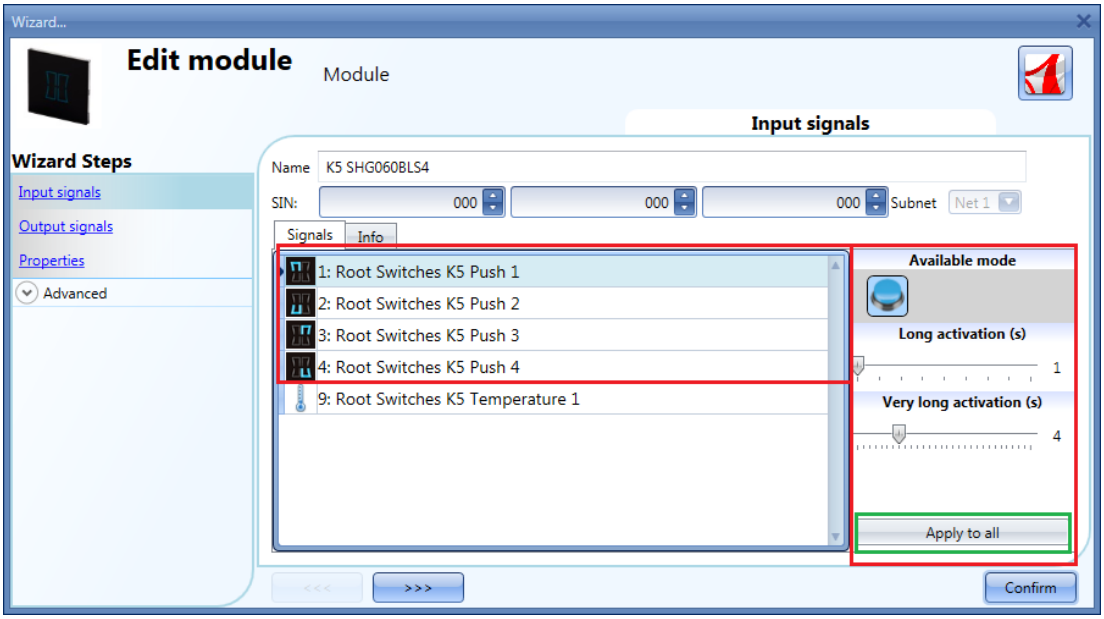


In the *Properties* field, the acoustic feedback and the back light can be set as described here.
The module has a built-in buzzer that can be enabled or disabled by the user (red rectangle). If it is enabled, when a key is pressed, an acoustic feedback will be emitted.
The glass has a backlight that can be configured: it can be set always ON (slider to the far right), always OFF (slider to the far left) or programmed with a delay off timer (once the timer set by the slider expires, the backlight is switched off automatically).

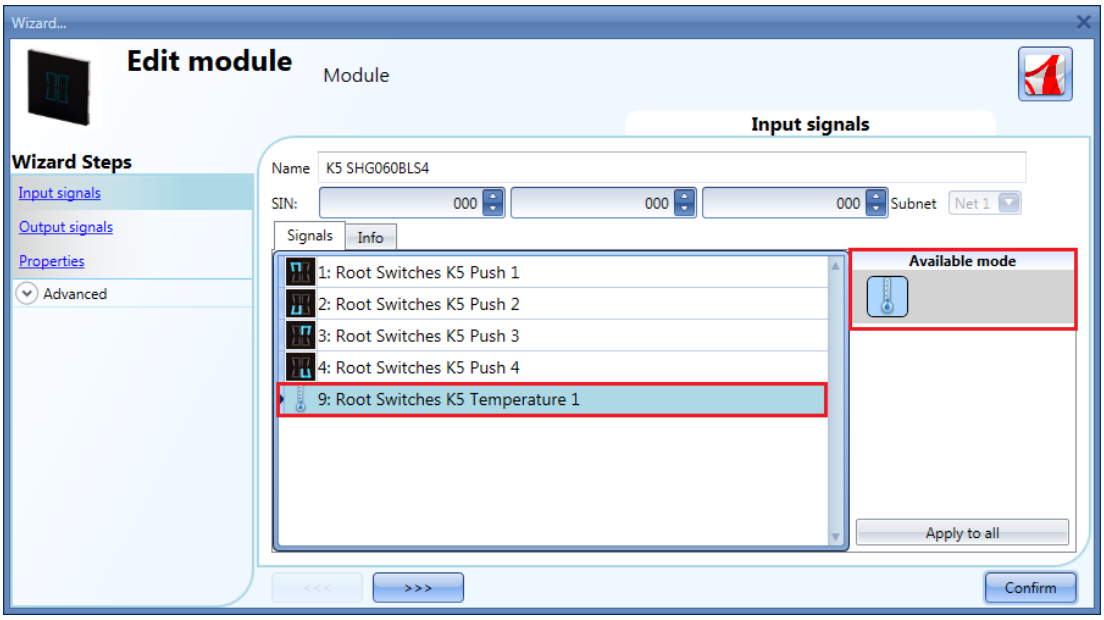
SHG060BLS4 and SHG060WLS4 modules

To configure these, click on the relevant picture once they are added to the project:

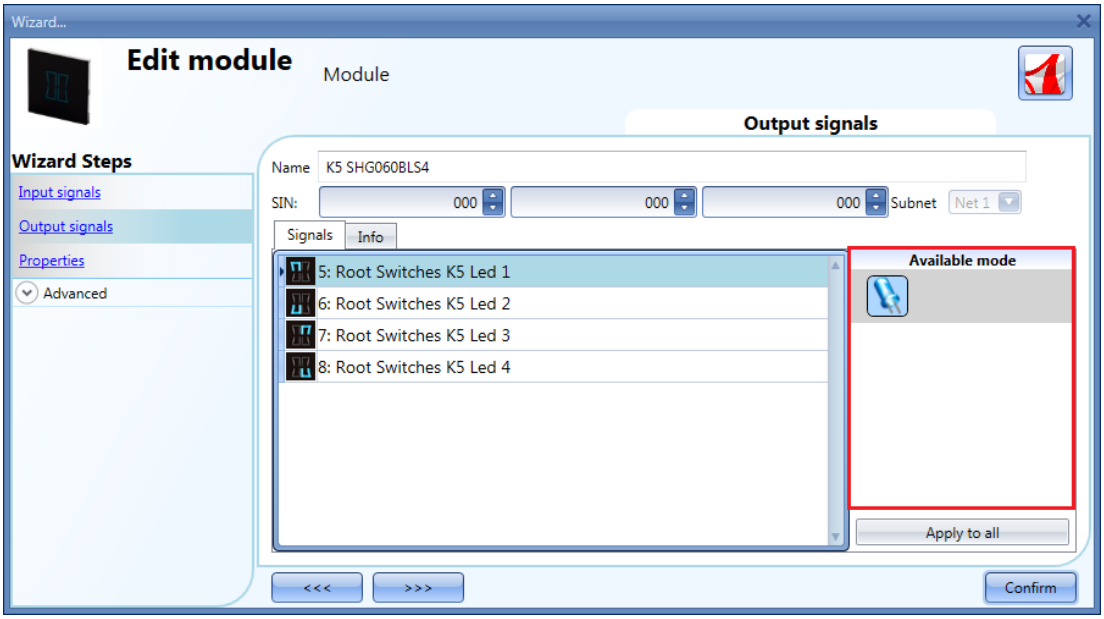
The configuration wizard will appear:



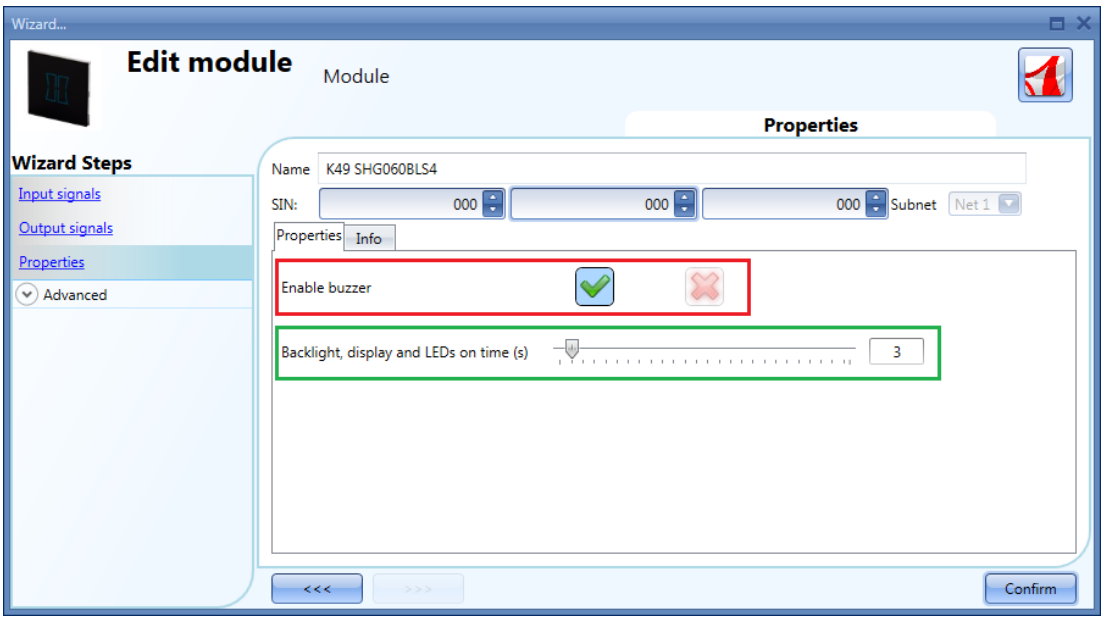
In the *Input signals* field, the *Long activation* and the *Very long* activation time has to be defined.
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and it is always set by the system 3 seconds longer than the *Long activation* time.
The short press and long press are recognized when the push button is released.
The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*
If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.



The SHG060WLS4 and SHG060BLS4 are glass switches with temperature sensors that can be used in any temperature function.



In the *Output signals* window, all the available feedback LEDs are shown: they can be freely programmed as feedback status LEDs for any smart-house function.



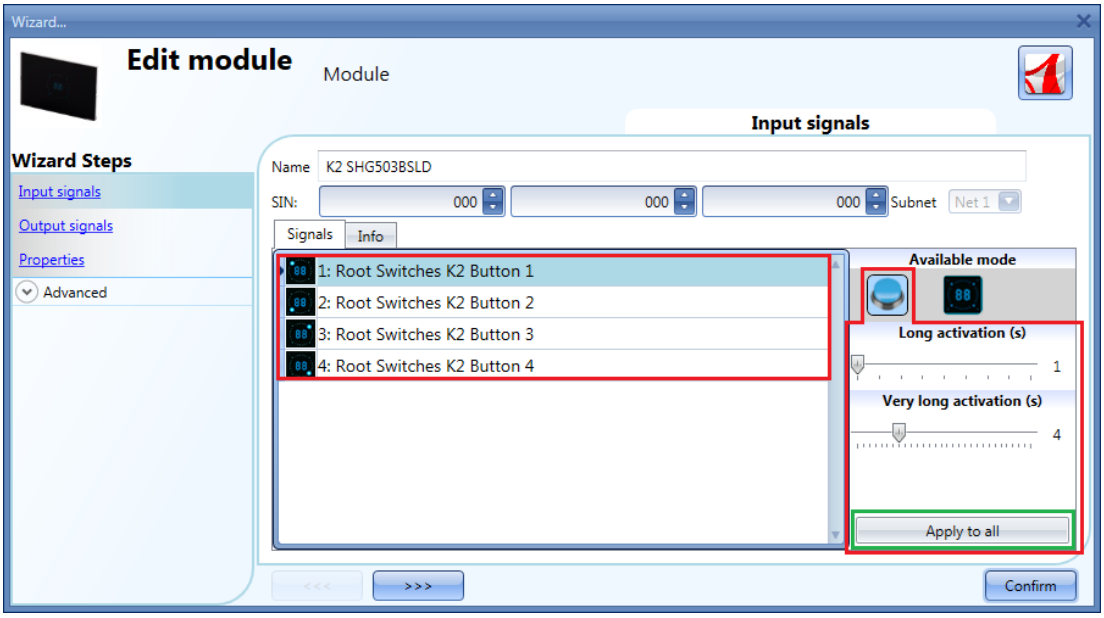
In the *Properties* field, the acoustic feedback and the backlight can be set as described here.
The module has a built-in buzzer that can be enabled or disabled by the user (red rectangle). If it is enabled, when a key is pressed, an acoustic feedback will be emitted.
The glass has a backlight that can be configured: it can be set always ON (slider to the far right), always OFF (slider to the far left) or programmed with a delay off timer (once the time set by the slider expires, the backlight is switched off automatically).

To configure the SHG503BSLD, SHG503WSLD, SHG060BSLD, SHG060WSLD , click on the relevant picture once they are added to the project:

SHG503BSLD	Net 1	K2 SHG503BSLD	000.000.000	Root	
SHG503WSLD	Net 1	K4 SHG503WSLD	000.000.000	Root	

The SHG503BSLD and SHG503WSLD have the same technical specs as the SHG060BSLD and SHG060WSLD: they differ only in front panel dimensions.

To configure these, click on the relevant picture once they are added to the project:



In the *Input signals* field, each key can be used in 2 different ways: as a **Push button** or as a **Dimmer Interface**.

7.11.1 How to configure the key as a standard push button

Click on the icon showing a push button (red rectangle in the figure above).
This can be used in all the on/off functions apart from the rollerblind functions.
If the user wants to change the working mode of a key, the key must not be used in any function. Should it be already being used, the user has to delete it from the function.

If the key is set as *push button*, the *Long activation* and the *Very long activation* time have to be set in the same way as that of the mechanical push button.

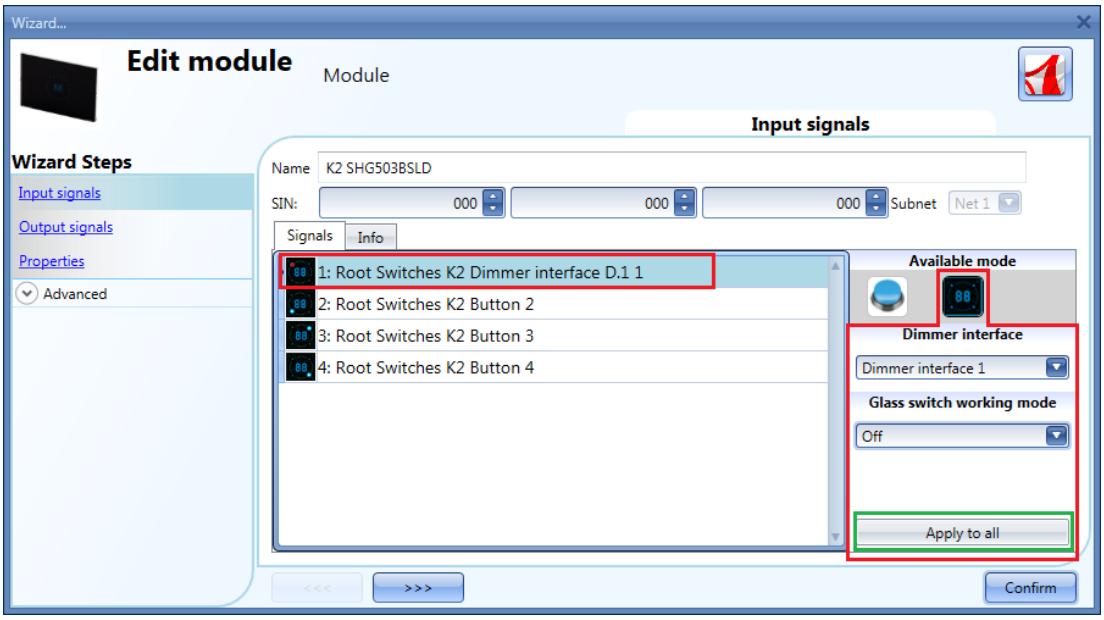
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and it is always set by the system 3 seconds longer than the *Long activation* time. The short press and long press are recognized when the push button is released.

The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*

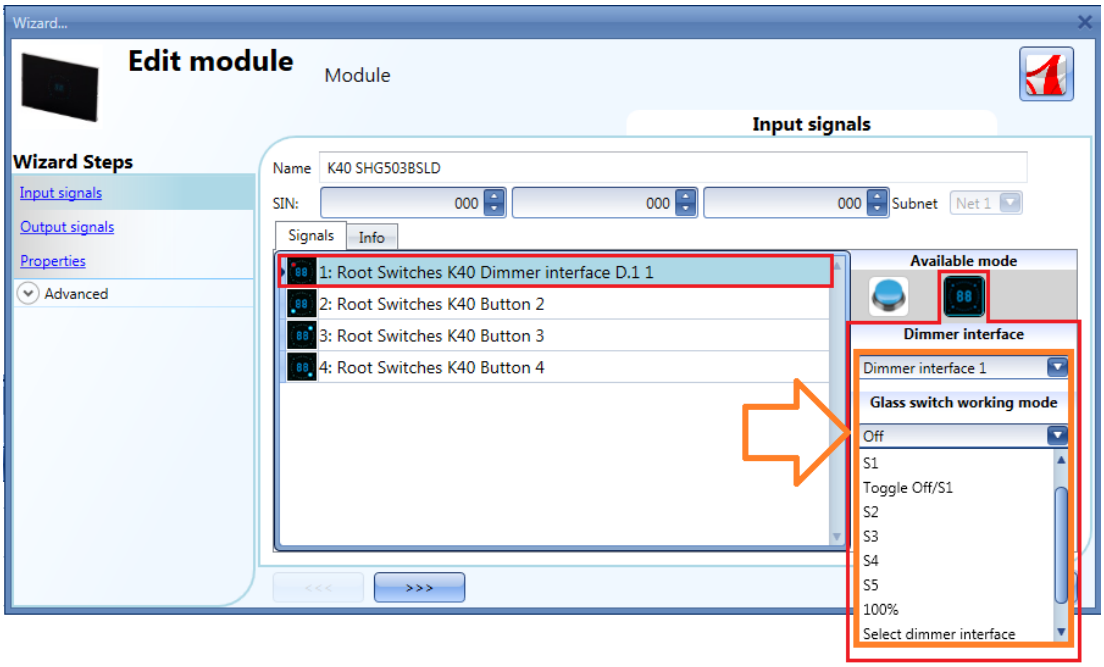
If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.

7.11.2 How to configure the key as *Dimmer interface*

Click on the icon showing the image of the glass switch (red rectangle in the figure below).



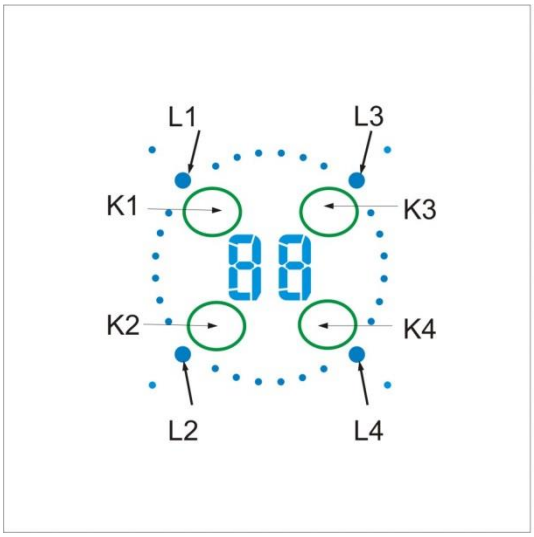
The *Dimmer interface* mode can be configured as shown in the following steps.
To indicate that a key is configured as *Dimmer interface,* the corresponding icon will be marked with a red dot.

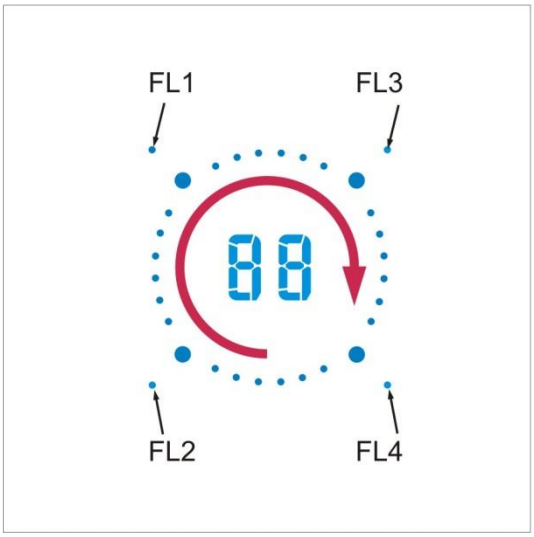


Since this panel can control up to 4 dimming functions, also called dimmer interface, each key can be freely associated to one of them. More than one key can be associated to the same dimmer interface (or dimmer function).
By means of the drop down list *Dimmer interface*, the user can associate the one required to the selected key.

Dimmer Interface	
Dimmer interface 1	It is possible to freely associate each key to a dimmer interface. Press *Apply to all* (green rectangle) to assign the same dimmer interface to all the keys not already used . If a key is already used in a function, the dimmer interface or the working mode cannot be changed.
Dimmer interface 2	
Dimmer interface 3	
Dimmer interface 4	

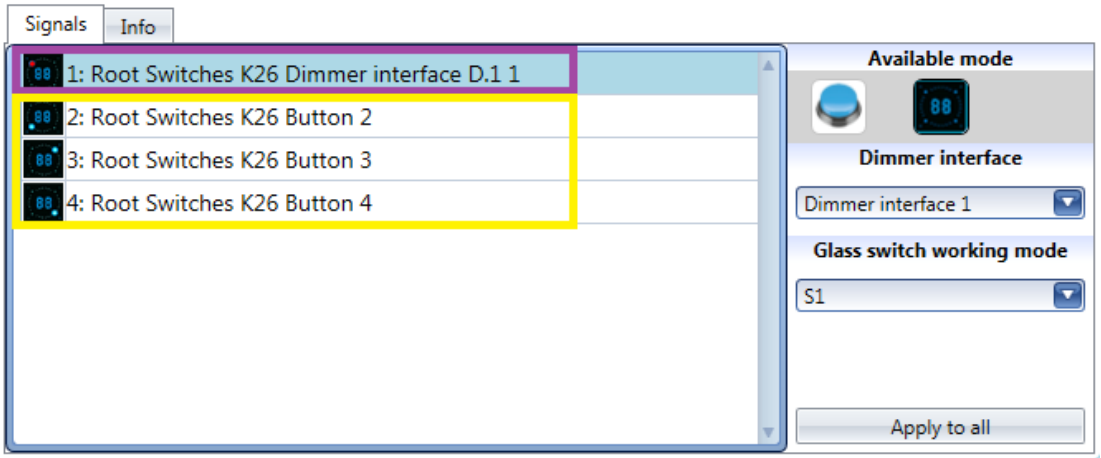
If a key is programmed in dimming mode (dimmer interface), when it is pressed it switches on or selects the relevant dimming group. When a group is selected, moving the finger on the slider will immediately change the corresponding light level. The display shows the dimming percentage of the selected group of lamps.
If the finger rotates clockwise over the slider, the light of the selected group is increased up to 100% (the display shows HI) in 10 steps; if the finger rotates anticlockwise over the slider, the light of the selected group is decreased down to 00% in 10 steps.





Some examples of the configuration of the keys are shown below:

One dimmer interface, three on/off keys
This example is shown below, where push button 1 is set as dimmer interface (purple box), while the other three push buttons are programmed as standard on/off keys (yellow box).
Since only one dimmer interface is programmed, the display and the slider are always associated with only one linked dimming function (see *How to use a dimmer interface in a dimmable light function).*
When K1 is pressed, the action, selected in *Glass switch working mode,* will be carried out. The corresponding FL1 LED will be on if the light level is higher than 0%.
The L*n* LEDs, associated with the other K*n* keys programmed as on/off push buttons, will be off.



Two dimmer interfaces, two on/off keys

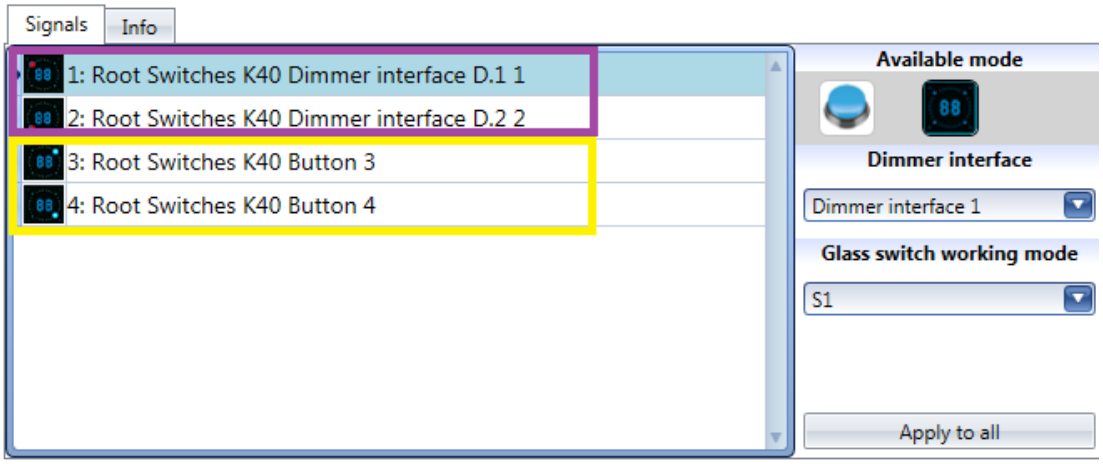
This example is shown below, where keys 1 and 2 are set as dimmer interfaces (purple box), while the other two push buttons are programmed as standard on/off keys (yellow box).

When two dimmer interfaces are programmed the display, the slider and L*n* Led are always associated with the last linked dimming function (see *How to use a dimmer interface in a dimmable light function).*

When K2 is pressed, dimmer interface 2 will be selected, L2 will be on and the display and the slider will be associated with the dimmer function to which K2 is linked. If K1 is pressed, dimmer interface 1 will be selected, L1 will be on and the display and the slider will be associated with the dimmer function to which K1 is linked. The Ln LED indicates to which dimmer interface the slider and the display are linked in that moment.

The push buttons 3 and 4, programmed as on/off, can be used at any time, without changing the association of the slider and the display.

The Ln LEDs, associated with K3 and K4 programmed as on/off push buttons, will be off.

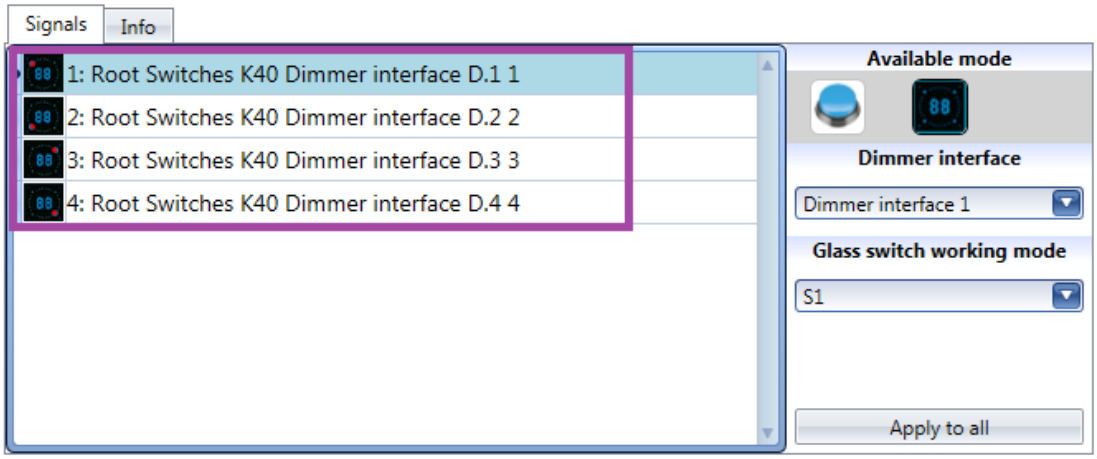


Four dimmer interfaces

This example is shown below, where all keys are set as dimmer interfaces (purple box) and there aren't any push on/off buttons.

When all the keys are programmed as dimmer interfaces, only one dimmer function can be selected at a time: the display and the slider are always associated with the last one in use and the corresponding Ln LED will be on. When, for example, K3 is pressed, dimmer interface 3 will be selected, L3 will be on and the display and the slider will be associated with the dimmer function to which K2 is linked. If K4 is pressed, dimmer interface 4 will be selected, L4 will be on and the display and the slider will be associated with the dimmer function to which K4 is linked.

The Ln LED indicates to which dimmer interface the slider and the display are linked in that moment.

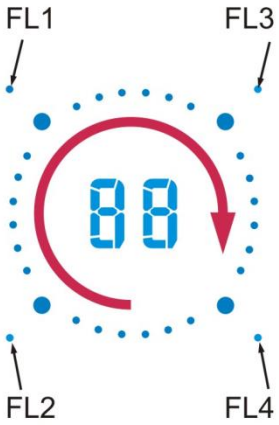


When a dimmer interface is selected, the following working modes are available:

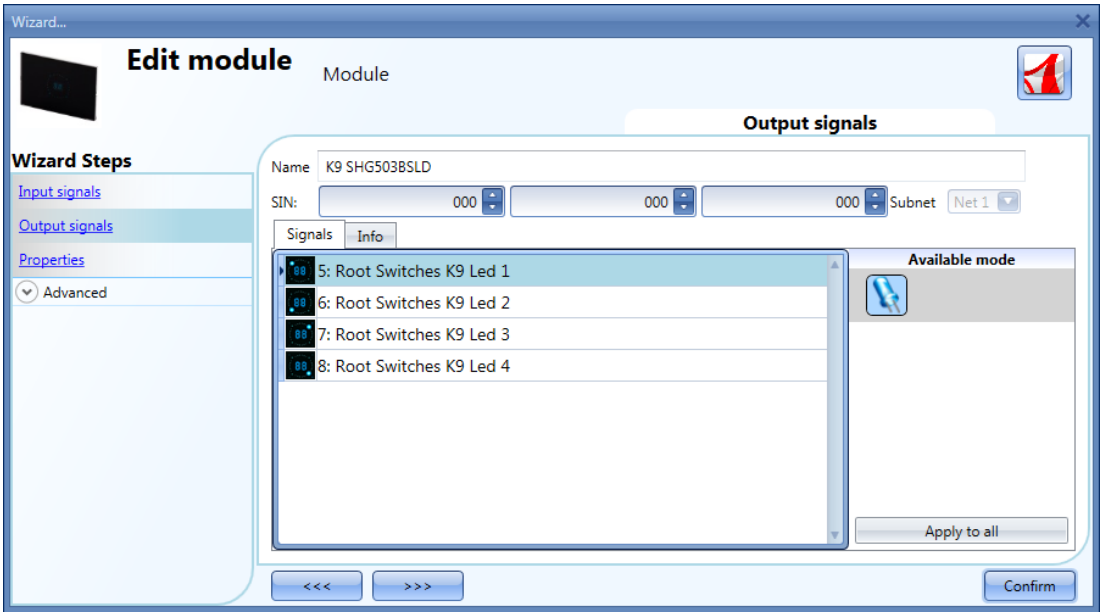
Glass switch working mode	Behaviour
Off	If the key is pressed, it will switch the light off.
S1	If the key is pressed, the light is switched on at the last saved level. If the light level is set down to 0% acting on the slider, and then switched on via a key associated to this working mode, the level is set at 10%.
Toggle Off/S1	If this working mode is selected, the light is toggled on/off. When it is switched on, the light level will be the last saved. If the light level is set down to 0% acting on the slider, and then switched on via a key associated to this working mode, the level is set at 10%.
S2	If the key is pressed, the light is set at Scenario 2 (S2 by default is equal to 20%, and it can freely be changed)
S3	If the key is pressed, the light is set at Scenario 3 (S3 by default is equal to 40%, and it can freely be changed)
S4	If the key is pressed, the light is set at Scenario 4 (S4 by default is equal to 60%, and it can freely be changed)
S5	If the key is pressed, the light is set at Scenario 5 (S5 by default is equal to 80%, and it can freely be changed)
100%	If the key is pressed, the light is set at the highest light level
Select dimmer interface	If the key is pressed, it associates the slider and the display with the linked dimmer function. It will not switch the light on or off.

7.11.3 How to set the feedback LEDs

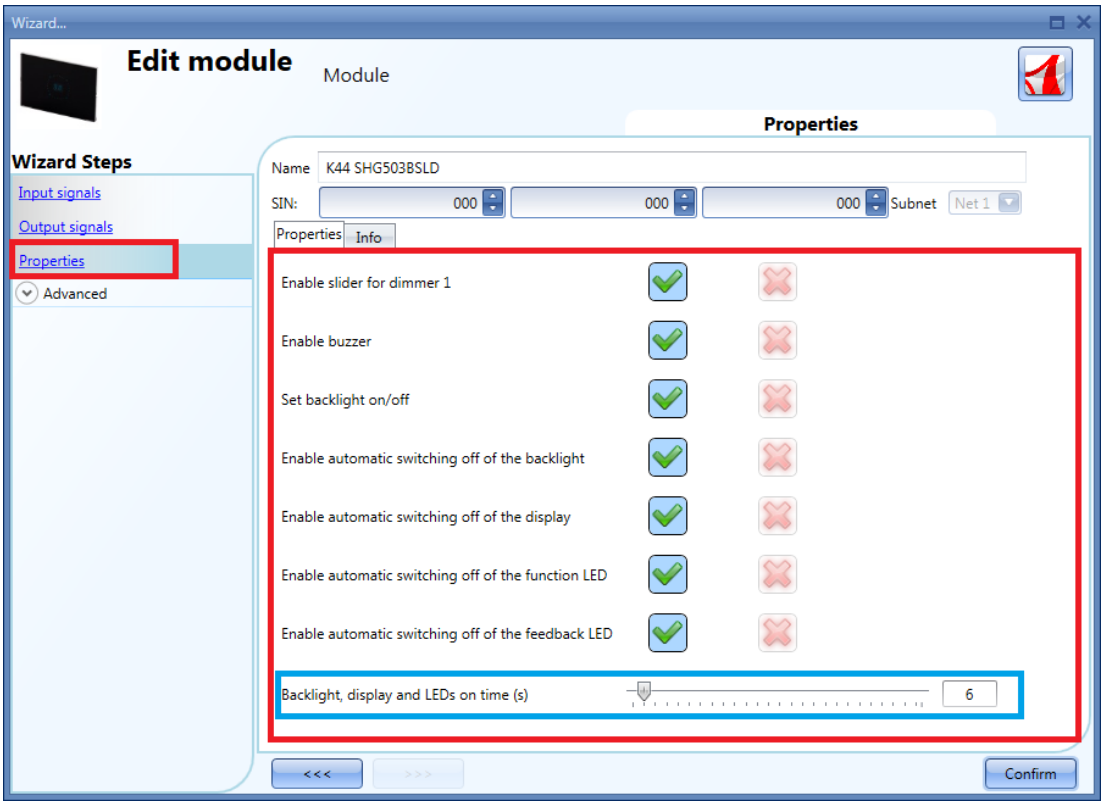
In the glass panel there are four small LEDs that can be freely programmed as feedback of any function. As shown in the figure below, they are called FL1, FL2, FL3, FL4.



In the *Output signals* window, all the available feedback LEDs are shown and they can be used in the corresponding *Feedback* field of any function.



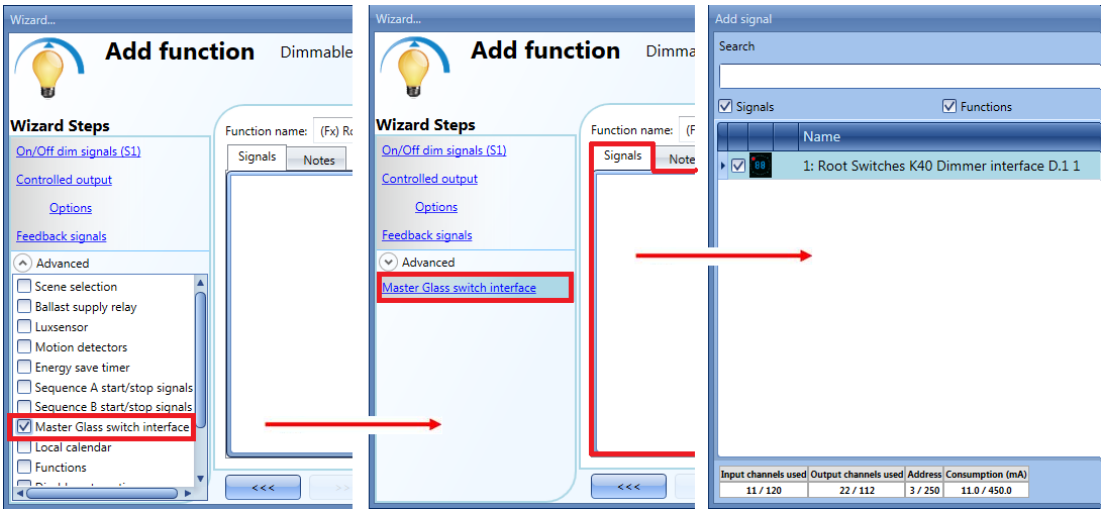
7.11.4 How to set the glass panel properties



In the *Properties* field, the acoustic feedback and the backlight can be set as described here.
The module has a built-in buzzer that can be enabled or disabled by the user: if it is enabled, when a key is pressed or the finger is moved over the slider, an acoustic feedback will be emitted. The glass has a backlight, two 7-digit displays, four function LEDs (L1, L2, L3, L4) and four feedback LEDs (FL1, FL2, FL3, FL4) which can be configured in this field:

Field name	Description
Enable slider for dimmer 1	By default when a key is set as dimmer interface in the *Inputs Signals* field, the slider for the corresponding dimmer is enabled. It can be disabled in order not to use it.
Enable slider for dimmer 2	
Enable slider for dimmer 3	
Enable slider for dimmer 4	
Enable buzzer	The module has a built-in buzzer that can be enabled or disabled by the user. If it is enabled, when a key is pressed, an acoustic feedback will be emitted.
Set backlight on/off	The backlight of the glass can be enabled or disabled.
Enable automatic switching off of the backlight	If this property is enabled, the back light is switched off when the timer, set in *Backlight on time,* expires.
Enable automatic off of the display	If this property is enabled, the display is switched off when the timer, set in *Backlight on time,* expires.
Enable automatic switching off of the function LED	If this property is enabled, the LED function is switched off when the timer, set in *Backlight on time,* expires.
Enable automatic switching off of the feedback LED	If this property is enabled, the feedback LEDs are switched off when the timer, set in *Backlight on time,* expires.
Backlight, display and LEDs on time (s)	This can be set always ON (slider to the far left) or programmed with a delay off timer (once the timer set with the slider expires, the backlight, display, function LED and feedback LEDs are switched off automatically).

7.11.5 How to use a dimmer interface in a dimmable light function



To use a key programmed as dimmer interface in a Dimmable light function, select *Master Glass switch interface* in the *Advanced* field, double click on *Signals* and select the required dimmer interface.

7.12 Masterglass temperature display

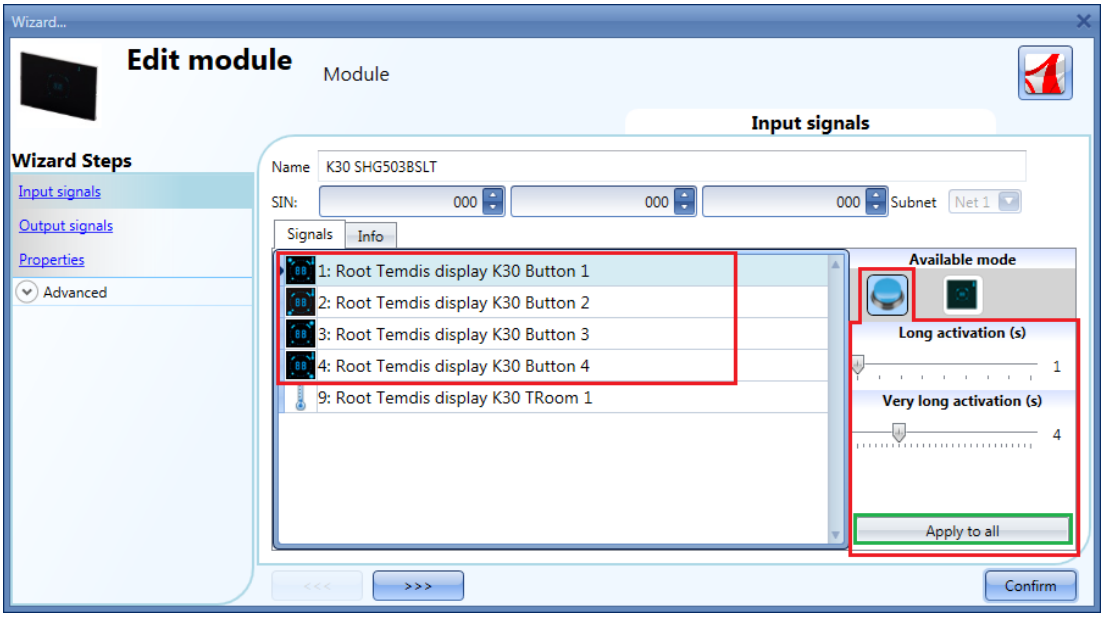
The following items belong to the Masterglass temperature displays:

SHG503WSLT
SHG503BSLT
SHG060WSLT
SHG060BSLT

	SHG503WSLT	Net 1	K29 SHG503WSLT	000.000.000	Root	
	SHG503BSLT	Net 1	K30 SHG503BSLT	000.000.000	Root	
	SHG060WSLT	Net 1	K31 SHG060WSLT	000.000.000	Root	
	SHG060BSLT	Net 1	K32 SHG060BSLT	000.000.000	Root	

The SHG503WSLT and SHG503BSLT have the same technical specs as the SHG060WSLT and SHG060BSLT: they differ only in the dimensions of the front panel.

To configure these, click on the relevant picture once it is added to the project:



In the *Input signals* field, each key can be used in 2 different ways: as a **Push button** or as a **Temdis interface.**

7.12.1 How to configure the key as a standard push button

Click on the icon showing a push button (red rectangle in the figure above).
To indicate that a key is configured as a push button, the corresponding icon will show a red dot. This configuration can be used in all the on/off functions apart from the rollerblind functions.
If the user wants to change the working mode of a key, it should not be in use in any function. Should it be already in use, the user must delete it from the function.

If the key is set as *push button*, the *Long activation* and the *Very long activation* times have to be set in the same way as that of the mechanical push button.

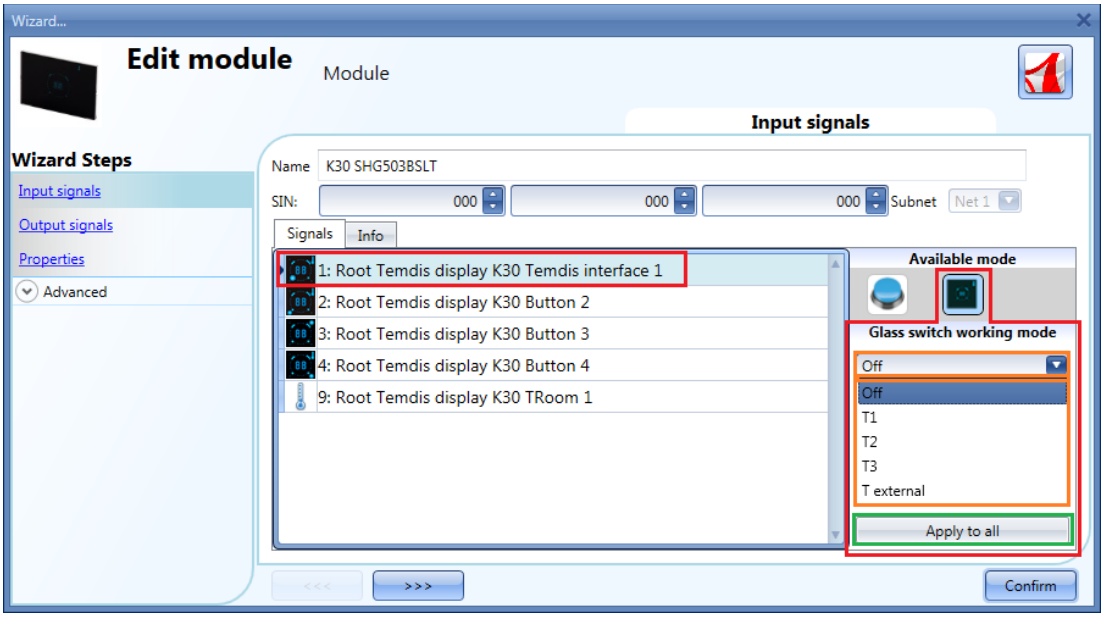
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and this is always set by the system for 3 seconds longer than the *Long activation* time. The short press and long press are recognized when the push button is released.

The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*

If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.

7.12.2 How to configure the key as Temdis interface

Click on the icon showing the image of the glass switch (red rectangle in the figure below).



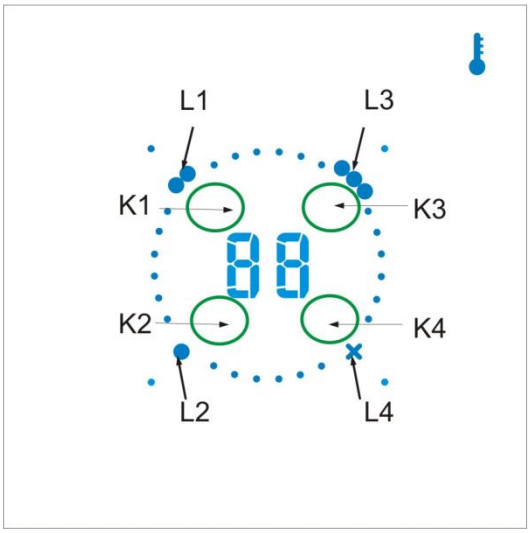
The *Glass switch working mode* can be configured as shown in the following steps.

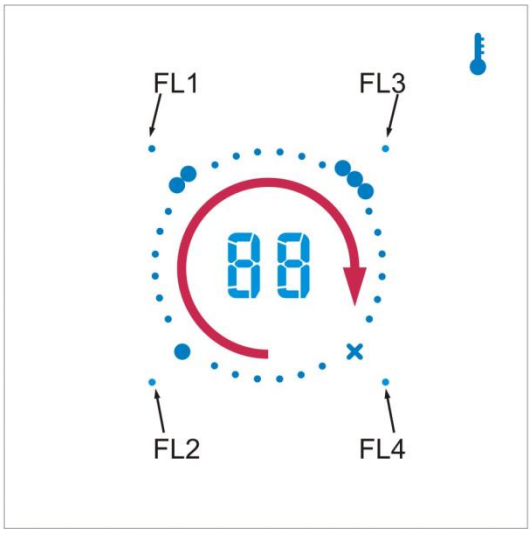
To indicate that a key is configured as *Temdis interface,* the corresponding icon will show a red dot.

Since this panel can control up to three temperature set points, one on/off command for the temperature function and one command to visualize an external temperature, each key can be freely associated to one of these functions. More than one key can be associated to the same functionality.
By means of the drop down list *Glass switch working mode*, the user can associate the key to the required working mode.

Glass switch working mode	Behaviour
Off	If the key is pressed, it will switch the temperature control off
T1	If the key is pressed, the set point 1 (T1) is selected: the temperature function, to which the panel is associated, will regulate the room temperature according to this setpoint.
T2	If the key is pressed, the set point 2 (T2) is selected: the temperature function, to which the panel is associated, will regulate the room temperature according to this setpoint.
T3	If the key is pressed, the set point 3 (T3) is selected: the temperature function, to which the panel is associated, will regulate the room temperature according to this setpoint.
T external	If the key is pressed, the display shows the external temperature for 10 seconds without changing the setpoint.

If a key is programmed as *Temdis interface*, when it is pressed the user can select the required temperature level, change the selected set point with the slider or switch the temperature control on/off. The display shows the room temperature in Celsius or Fahrenheit Degrees according to the UWP 3.0 tool settings.



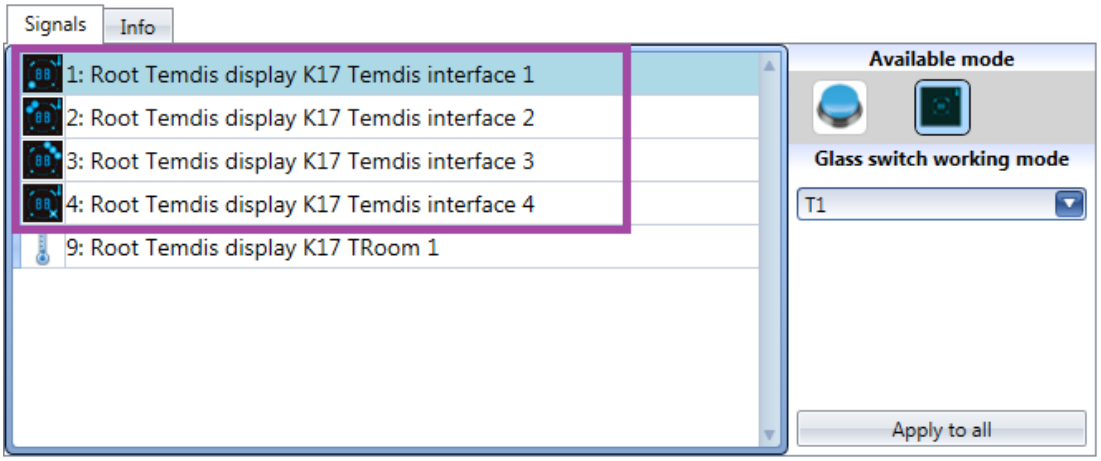


Some examples of the configuration of the keys are shown below:

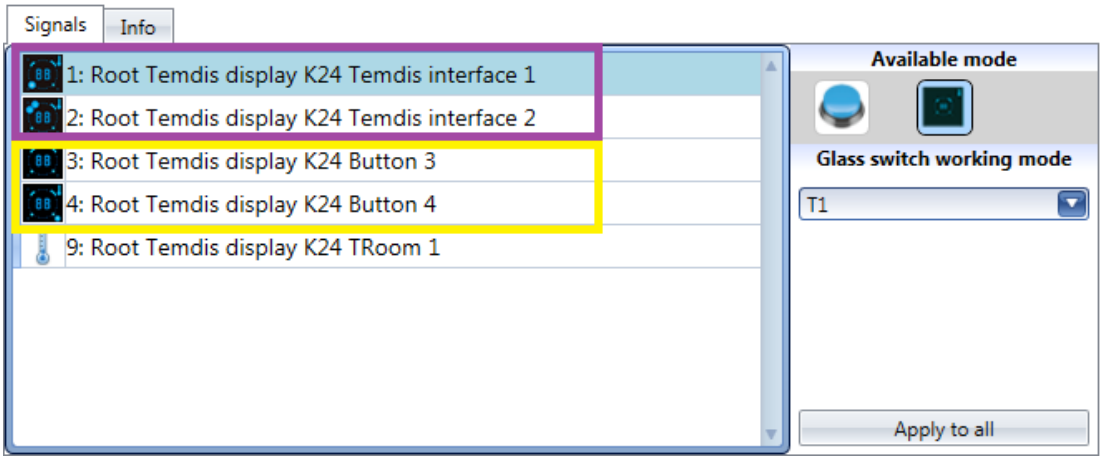
Four Temdis interface keys

This example is shown below, where all the keys are set as *Temdis interface* (purple box) and there aren't any on/off push buttons. The display always shows the room temperature. When K1 is pressed, the display shows *t1* for 1 second, then it shows setpoint T1. L1 will be blinking: if the finger rotates clockwise over the slider, the selected setpoint is increased in 1 or 0.5 degree steps up to the maximum set value (see *How to set temperature range*). If the finger rotates anticlockwise over the slider, the selected setpoint is decreased in 1 or 0.5 degree steps down to the minimum set value (see *How to set temperature range*). If K2 is pressed, the display shows *t2* for 1 second, then it shows setpoint T2. L2 will be blinking: the user can select the required temperature level by means of the slider (as explained above for K1). If K3 is pressed, T3 set point is selected. Finally, when K4 is pressed, the display shows *tE* for 1 second

and then it will show the outdoor temperature for 10 seconds. *(see How to show the external temperature in the Masterglass temperature display).*
The Ln LED indicates the currently selected setpoint

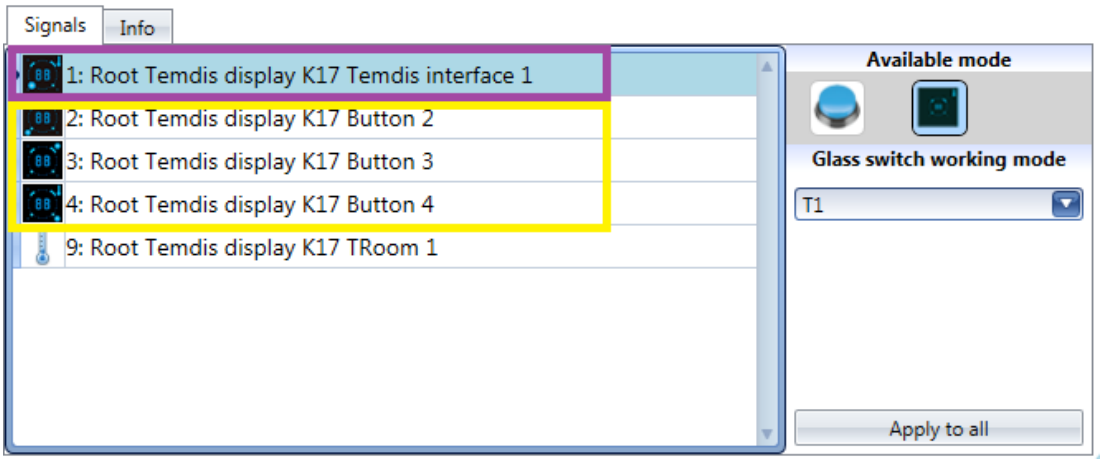


Two Temdis interfaces, two on/off keys
This example is shown below, where key 1 is set as *Temdis interface* (purple box), while the other three push buttons are programmed as standard on/off keys (yellow box).
When K1 is pressed, the display shows *t1* for 1 second, then it shows setpoint T1. L1 will be blinking: if the finger rotates clockwise over the slider, the selected setpoint is increased in 1 or 0.5 degree steps up to the maximum set value (see *How to set temperature range*). If the finger rotates anticlockwise over the slider, the selected setpoint is decreased in 1 or 0.5 degree steps down to the minimum set value (see *How to set temperature range*).If K2 is pressed, the display shows t2 for 1 second, then it shows setpoint T2. L2 will be blinking: the user can select the required temperature level by means of the slider (as explained above for K1). The push buttons 3 and 4, programmed as on/off, can be used at any time. The Ln LEDs associated with K3 and K4 programmed as on/off push buttons will be less bright than the L1 or L2.



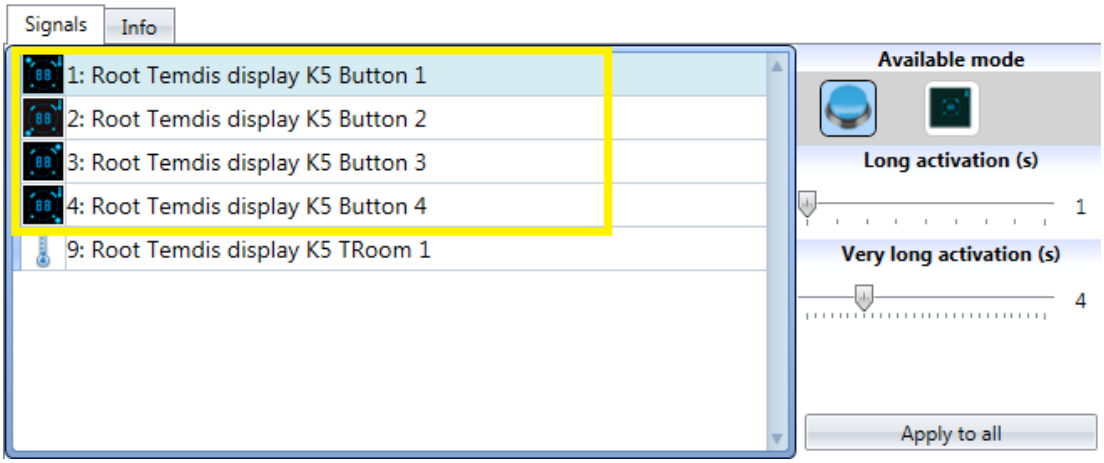
One Temdis interface, three on/off keys
This example is shown below, where key 1 is set as Temdis interface (purple box), while the other three push buttons are programmed as standard on/off keys (yellow box).
When K1 is pressed, the display shows *t1* for 1 second, then it shows setpoint T1. L1 will be blinking: if the finger rotates clockwise over the slider, the selected setpoint is increased in 1 or 0.5 degree steps up to the maximum set value (see *How to set temperature range*). If the finger rotates anticlockwise over the slider, the selected setpoint is decreased in 1 or 0.5 degree steps down to the minimum set value (see *How to set temperature range*). The push buttons 2, 3 and 4, programmed as on/off, can be used at any time.

The Ln LEDs associated with K2, K3 and K4 programmed as on/off push buttons will be less bright than the L1.



Four on/off keys

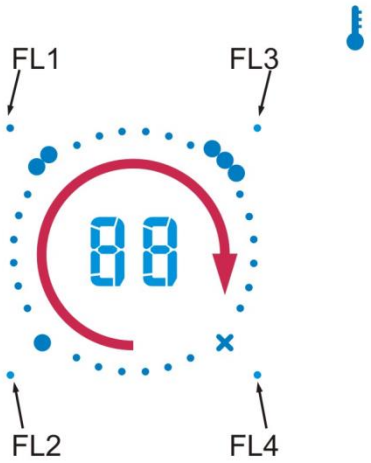
This example is shown below, where all the keys are set as push buttons (yellow box) and there aren't any *Temdis interface* keys. When all the keys are programmed as standard push button on/off, the display always shows the room temperature, but the slider is disabled. The push buttons are programmed as on/off and can be used at any time. All the Ln LEDs are on at the same brightness.



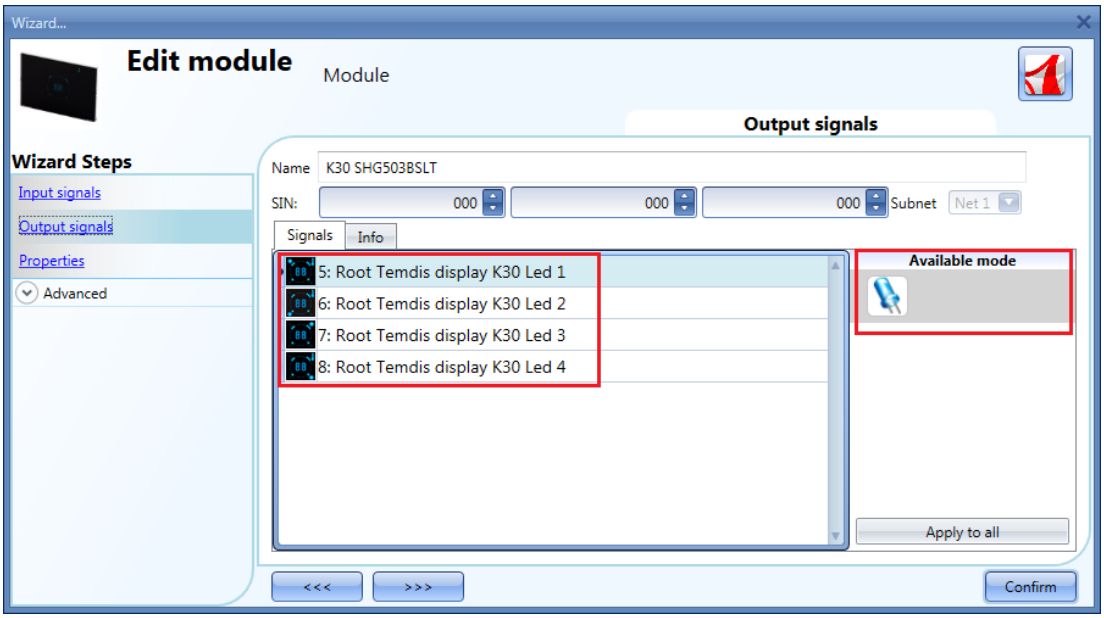
All this is fully programmable via the UWP 3.0 tool, so the user can create any combination.

7.12.3 How to set the feedback LEDs

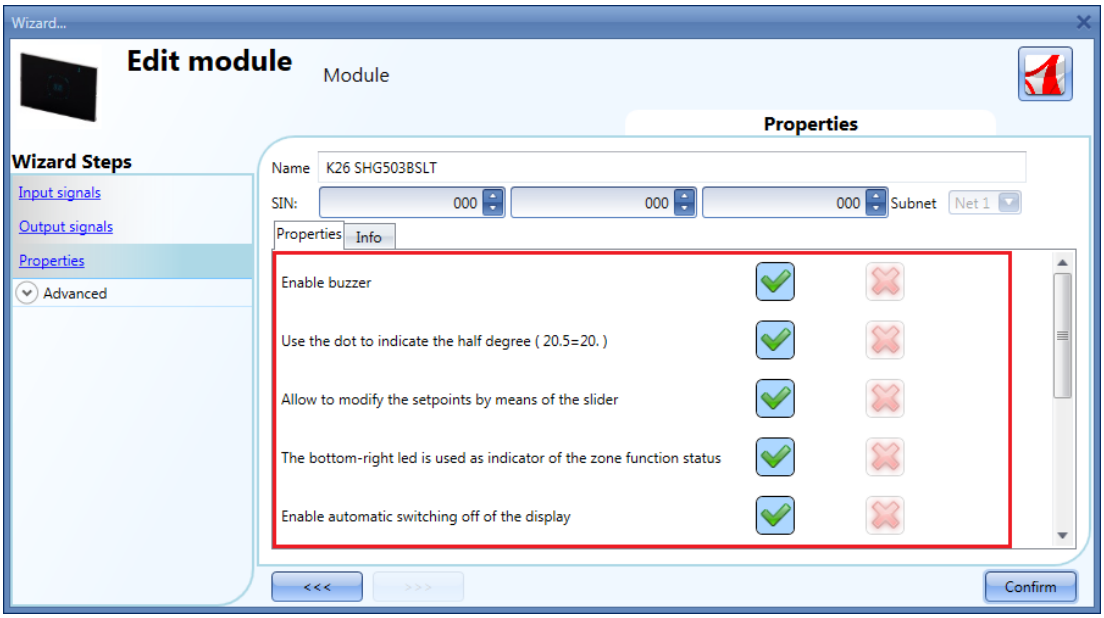
In the glass panel there are four small LEDs that can be freely programmed as feedback of any function. As shown in the figure below, they are named FL1, FL2, FL3, FL4.

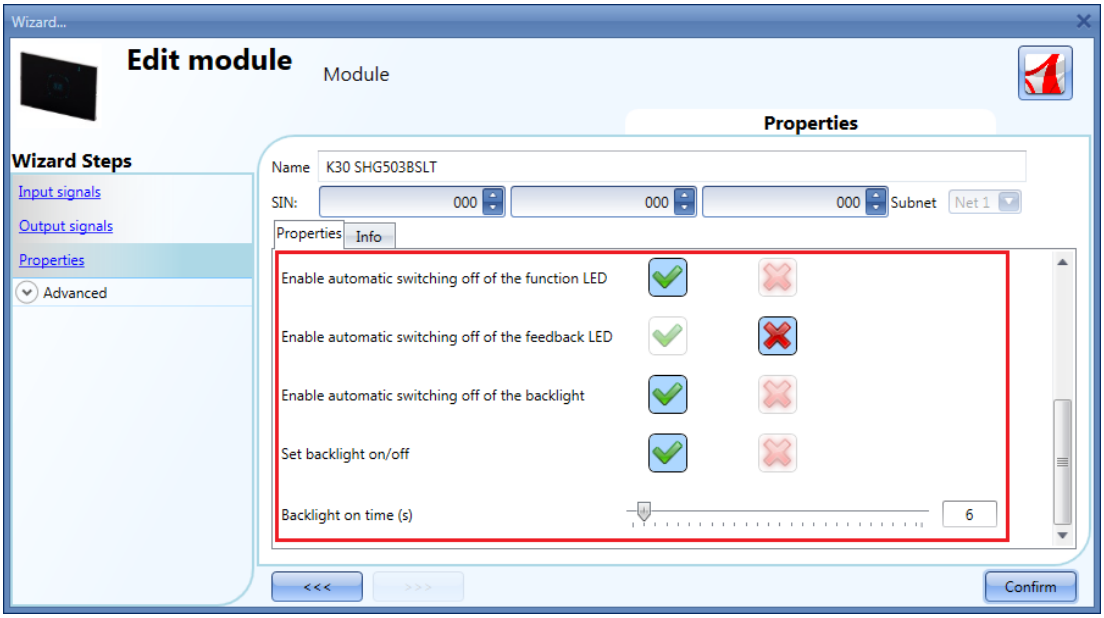


In the *Output signals* window, all the available feedback LEDs are shown and they can be used in the corresponding *Feedback* field of any function.



7.12.4 How to set the glass panel properties

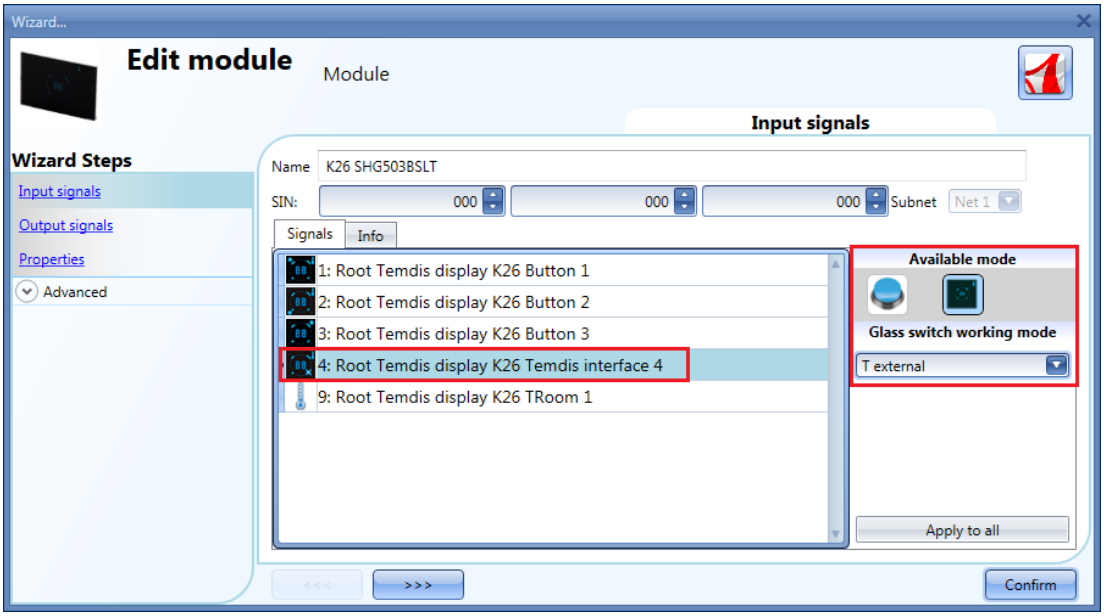




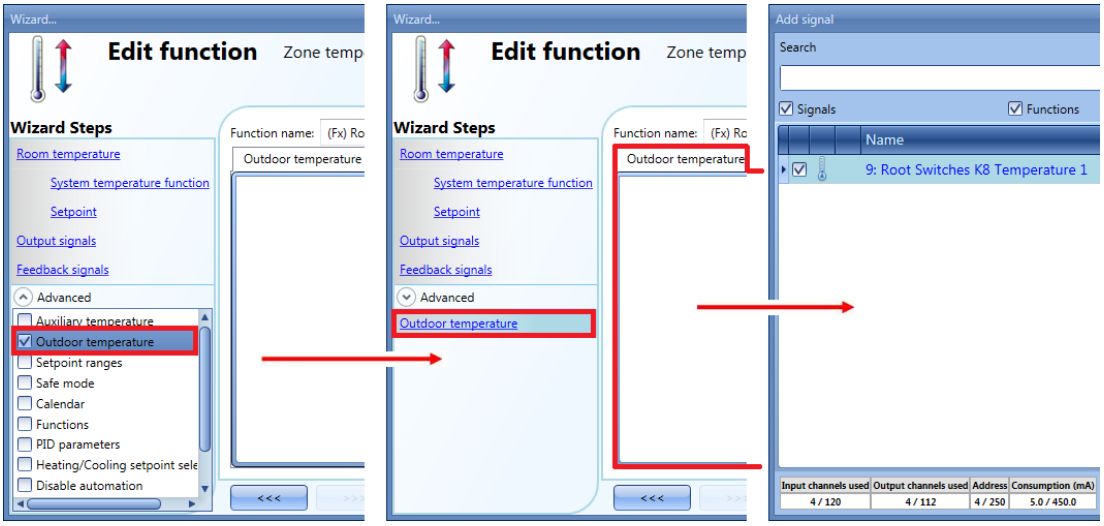
In the *Properties* field, the acoustic feedback and the backlight can be set as described here.
The module has a built-in buzzer that can be enabled or disabled by the user: if it is enabled, when a key is pressed or the finger is moved over the slider, an acoustic feedback will be emitted. The glass has a backlight, two 7-digit displays, four function LEDs (L1, L2, L3, L4) and four feedback LEDs (FL1, FL2, FL3, FL4) which can be configured in this field:

Field name	Description
Enable buzzer	The module has a built-in buzzer that can be enabled or disabled by the user. If it is enabled, when a key is pressed, an acoustic feedback will be emitted.
Use the dot to indicate the half degree (20.5 = 20.)	If this property is enabled, the display shows the dot to indicate the half degree.
Allow to modify of the setpoints by means of the slider	If this property is enabled, the user can select the desired temperature level and change it by means of the slider
The function LED associated to Off is used to indicate the zone function status	If this property is enabled and Kn is associated to the OFF function, the corresponding function LED Ln is ON when the relevant zone temperature function is ON.
Enable automatic switching off of the display	If this property is enabled, the display is switched off when the timer, set in *Backlight on time,* expires.
Enable automatic switching off of the function LED	If this property is enabled, the function LED is switched off when the timer, set in *Backlight on time,* expires.
Enable automatic switching off of the feedback LED	If this property is enabled, the feedback LEDs are switched off when the timer, set in *Backlight on time,* expires.
Enable automatic switching off of the backlight	If this property is enabled, the back light is switched off when the timer, set in *Backlight on time,* expires.
Set backlight on/off	The backlight of the glass can be enabled or disabled.
Backlight, display and LEDs on time (s)	This can be set always ON (slider to the far left) or programmed with a delay off timer (once the timer set with the slider expires, the backlight, display, function LED and feedback LEDs are switched off automatically).

7.12.5 How to show the external temperature in the Masterglass temperature display



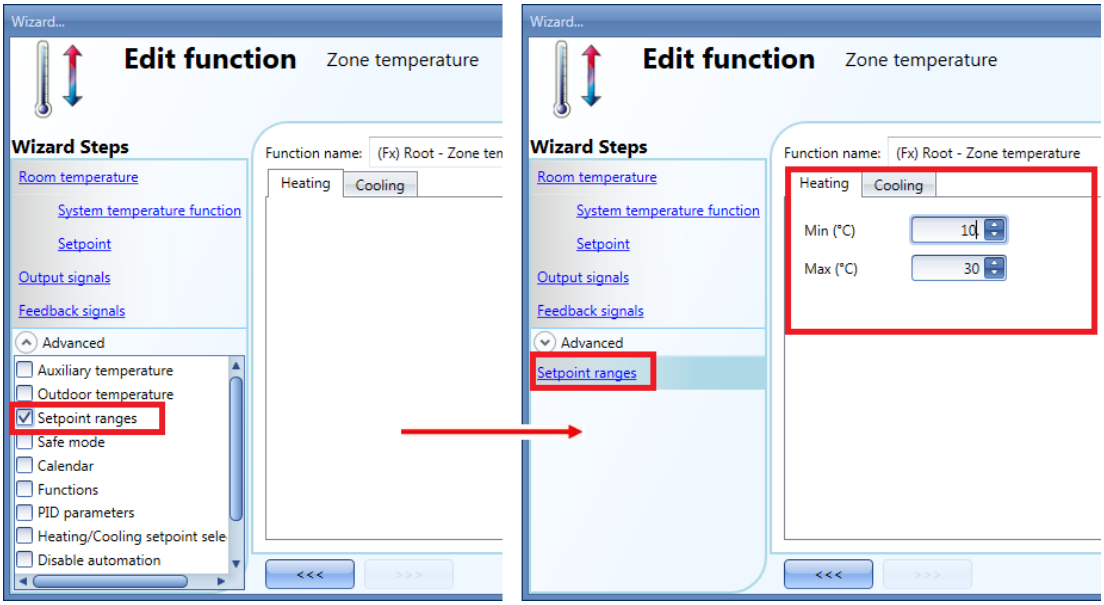
To see an external temperature, the user has to configure a key as *temdis interface* and then select *T external* in *Glass switch working mode*.



To add the external temperature signal, the user should select the relevant section in the wizard of the zone function, then double click on the Signals window and select the *input signal* from the list of those available (see picture below).
In the *Outdoor temperature* signals window, the user can add one or more temperature signals from the sensor modules (e.g. BSI-TEMANA-U, SHA4XLS4TH, SHE5XLS4TH, SHGxxxW-BLSx).

When K4 is pressed in the Masterglass temperature display, the display shows *tE* for 1 second and then it will show the outdoor temperature for 10 seconds. Once this time has expired, the room temperature can be seen.

7.12.6 How to set the temperature range



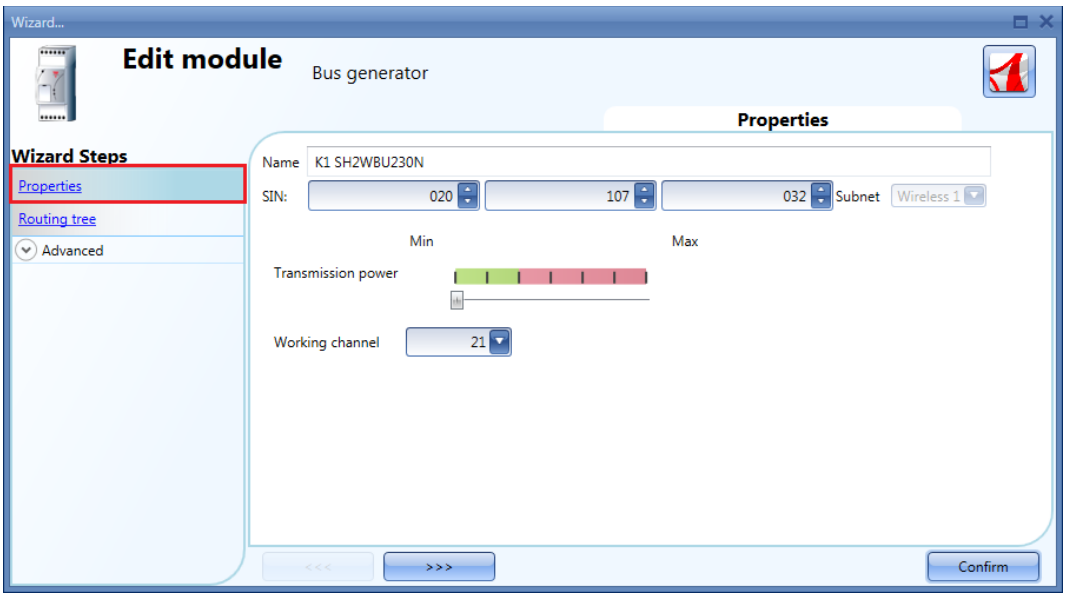
To change the predefined set point values, the user should click on the relevant field in the wizard of the zone temperature function. In the Advanced section, in *Set point ranges*, the user can select the operating range for the set points (min is 10°C and max is 30°C by default).

Refer to - How to configure the set points - for more details.

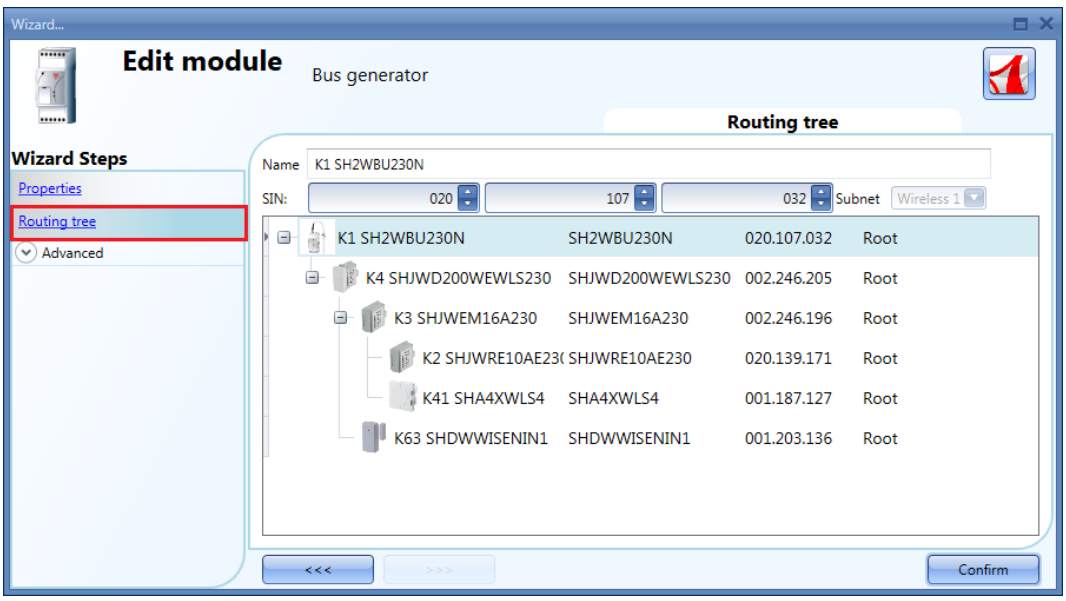
7.13 Wireless modules

In the *Properties* window of the wireless base unit SH2WBU230x, the user can select the working channel of the wireless network. If more than one SH2WBU230x is used, it is advisable to program them to work on different channels so that they don't interfere with each other (for more info see also the manual *How to install the wireless system*).

If a SH2WBU230N wireless base unit is used, the user can set the transmission power of the wireless field by moving the slider in the *Transmission power* field.



In the *Routing tree* window, an overview is shown of how the wireless modules are connected to the wireless base unit, whether directly or indirectly via a repeater. By means of drag &drop, the route of the wireless signals can be changed.



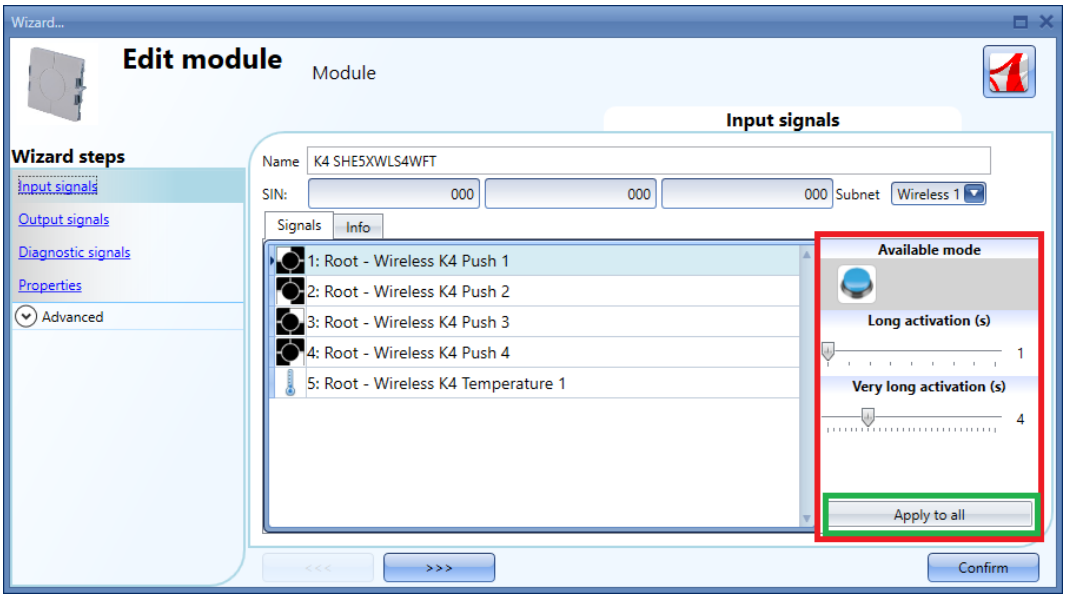
7.13.1 Wireless flat switch

The following items belong to the *wireless light switches modules*:

SHA4XWLS4
SHE5XWLS4
SHE5XWLS4BF
SHE5XWLS4BFT (Wireless flat switch with temperature sensor)
SHE5XWLS4WF
SHE5XWLS4WFT (Wireless flat switch with temperature sensor)

To configure them, click on the relevant picture once they are added to the project:

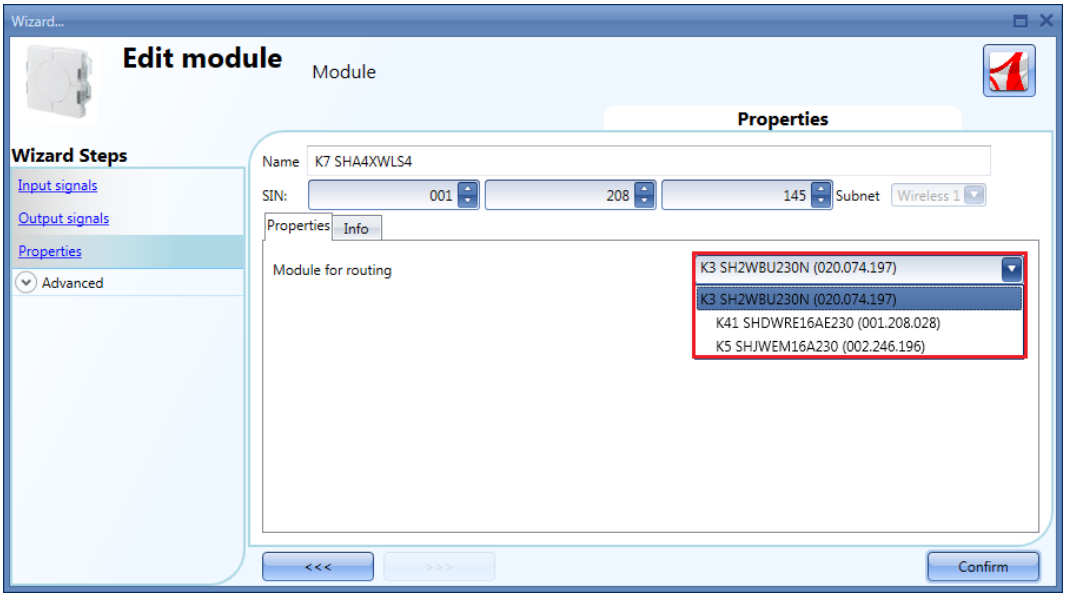
The configuration wizard will appear:



In the *Input signals* field, the *Long activation* and the *Very long* activation time has to be defined.
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and it is always set by the system 3 seconds longer than the *Long activation* time. The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*

The *Output signals* field is blank because the wireless light switches do not manage the feedback LEDs, in order to increase the lifetime of the battery.

In the *Properties* window, the user can choose in the *Select module for routing* field, the module to use for routing if the light switch is placed in an area not covered by the SH2WBU230x: in this situation it is possible to extend the operating distance using a SHJWD200WExxxx, a SHJWEM16Axxx, a SHJWRE10AExxx , a SHJWINS04xxx and a SHDWRE16AE230 as router/repeater.

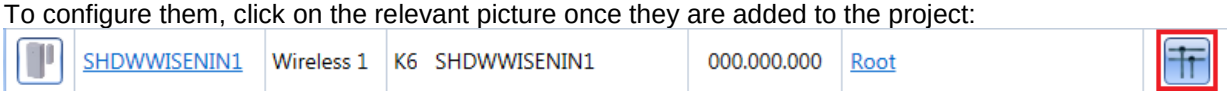


7.13.2 Wireless window sensor modules

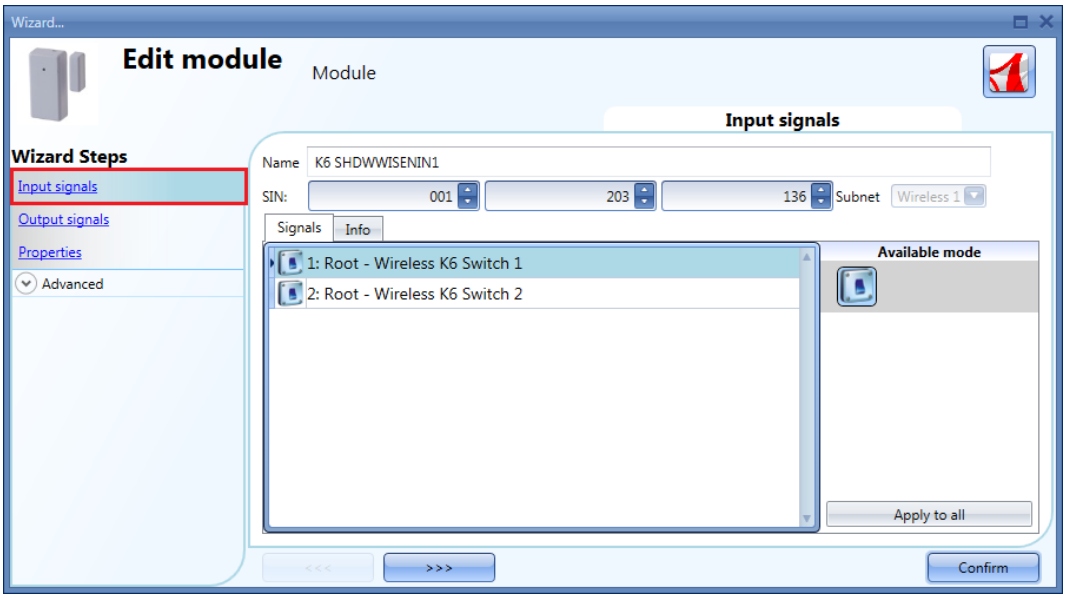
The following items are part of the *Wireless window sensor modules*:

SHDWWISEN
SHDWWISENIN1

To configure them, click on the relevant picture once they are added to the project:



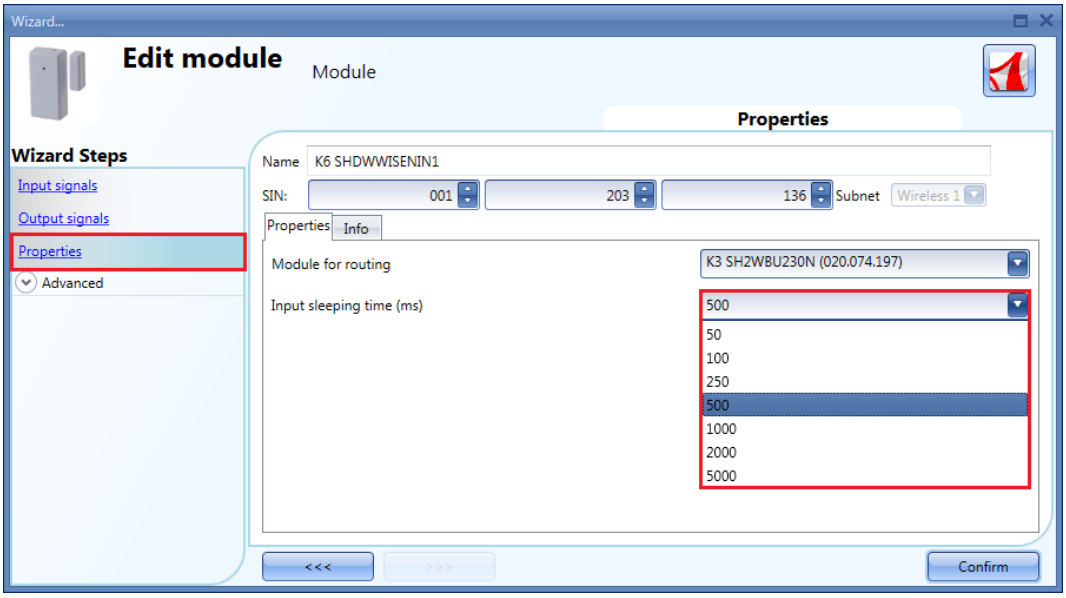
The configuration wizard will appear:



In the *Input signals* field, for the SHDWWISEN one input signal is shown, while for the SHDWWISENIN1 two input signals are shown because it includes a voltage free input as well.

In the *Properties* window, the user can choose the module to use for routing the signal if the w*ireless window sensor* is placed in an area not covered by the SH2WBU230x: in this situation it is possible to extend the operating distance by using a SHJWD200WExxx, a SHJWRE10AExxx, a SHJWEM16Axxx, a SHJWINS04xxx or a SHDWRE16AE230 as router/repeater.

In the *Input sleeping time (ms)* field, the user can set the refresh time for the switch status to be sent to the UWP 3.0: if this time is short, the sensor will send its status very often causing the battery to be discharged quickly. The right balance has to be found between the battery life time and the status update.



Example
In the example shown above, a sleeping time of 500 ms is set by the user: the switch status of the window sensor module will be sent every 500ms to the UWP 3.0 controller.

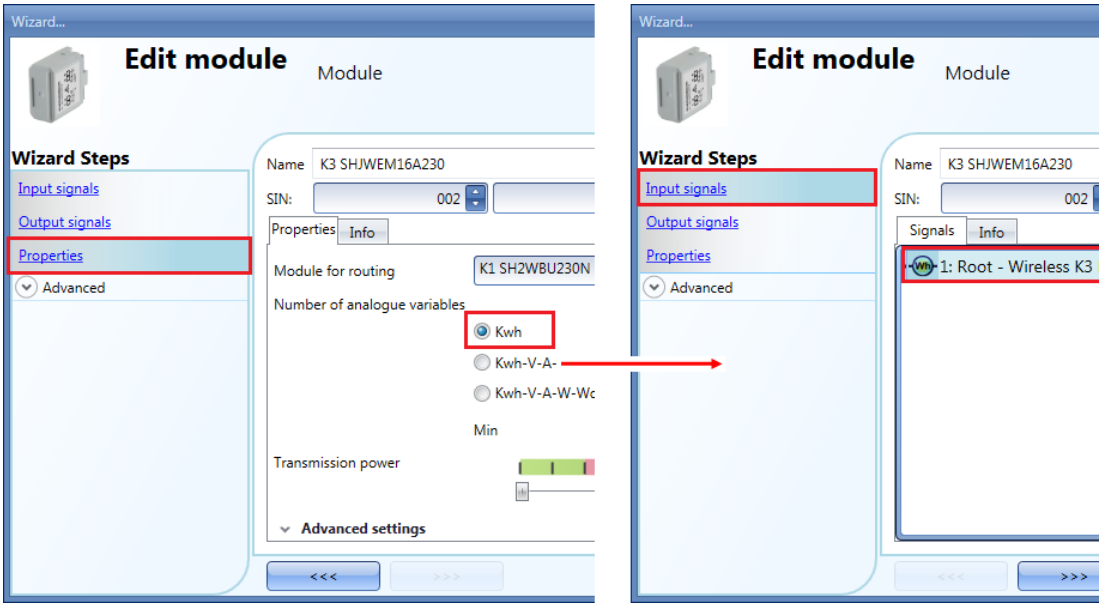
7.14 How to manage the analogue variables

7.14.1 How to manage the analogue variables using the grouping

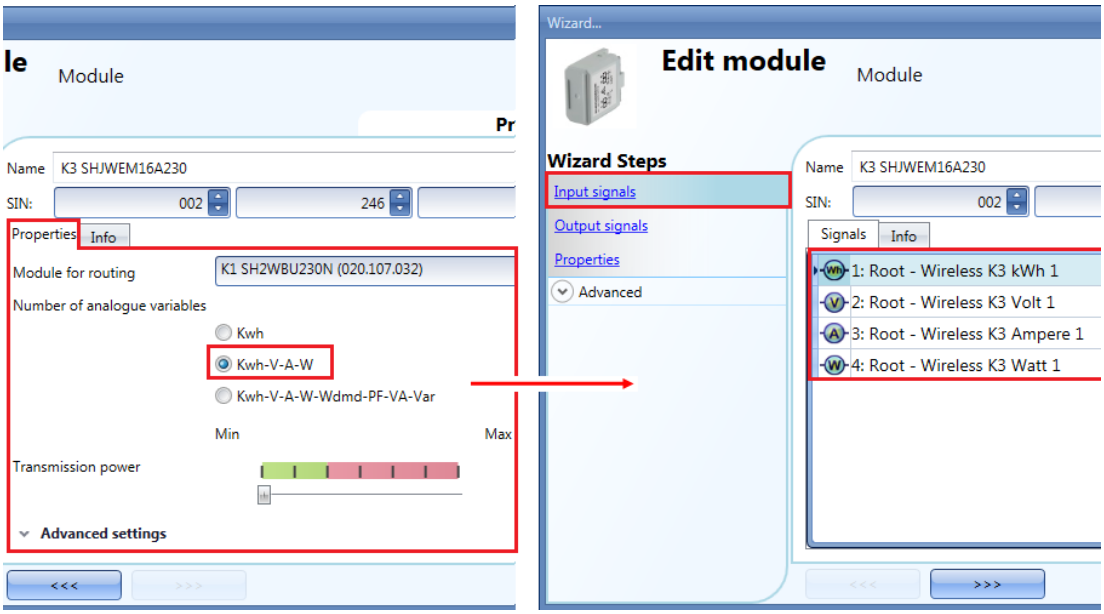
In the *Properties* window of a wireless module with readout feature, the user can select the relevant group of analogue variables that will be transmitted to the UWP 3.0. The table below shows the analogue variables for each group.

Group of analogue variables	Type of variables
None	The analogue variables are not sent
Kwh	kWh
Kwh-V-A-W	kWh, Volt, Ampere, Watt
Kwh-V-A-W-Wdmd-PF-VA-Var	kWh, Volt, Ampere, Watt, Wdmd, Power factor, Volt*Ampere, Var

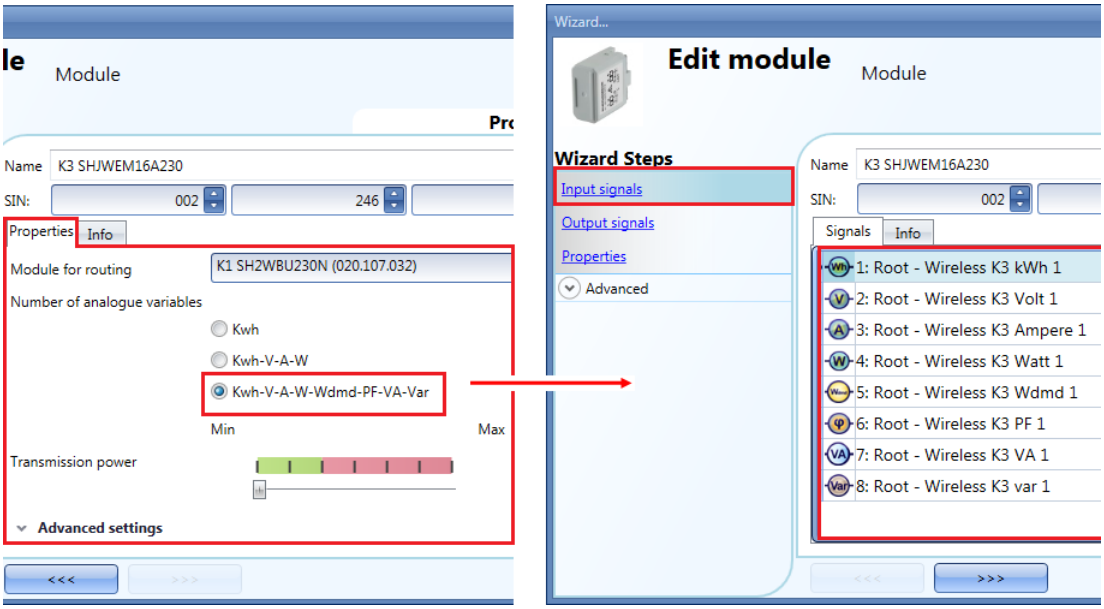
In the *Number of analogue variables* field, if the kWh icon is selected, the module will transmit only the *kWh* value: in the *Input signals* window a dynamic list shows the analogue variables related to the selected group, as shown in the picture below.



When the kWh-V-A-W icon is selected, the module will transmit all the variables related to this group: in the *Input signals* window a dynamic list shows all the analogue variables related to the selected group, as shown in the picture below.

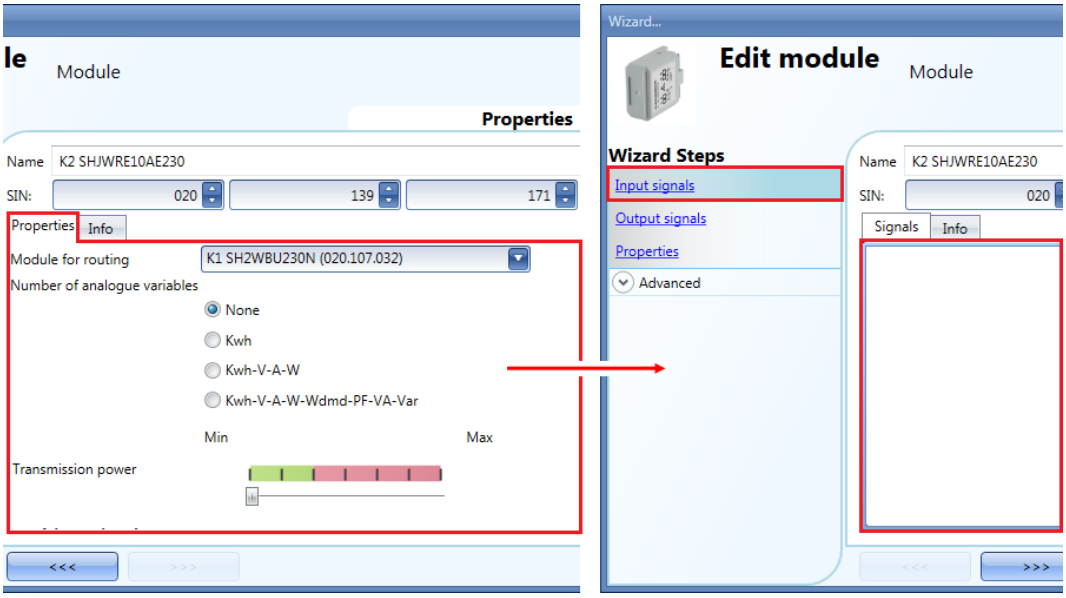


If the kWh-V-A-W-Wdmd-PF-VA-Var icon is selected by the user, the module will transmit all the variables related to this group: in the *Input signals* window a dynamic list shows all the analogue variables related to the group selected, as shown in the picture below.



Apart from the SHJWEM16Axxx module, in all the other wireless modules with energy readout feature, the user can disable the analogue data transmission.

In the *Number of analogue variables* field, when the None is selected, the wireless modules will not transmit any read data to the UWP 3.0.

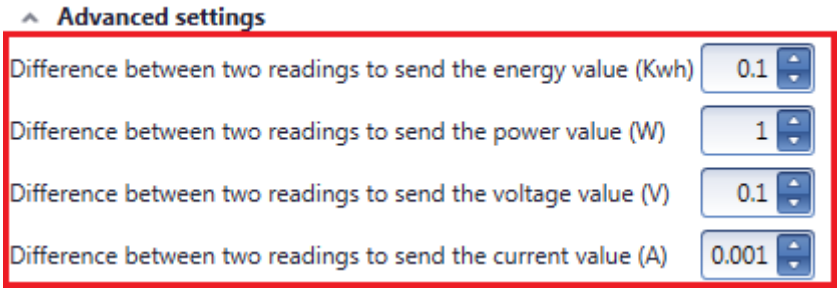


7.14.2 How to define when analogue values have to be sent

In order to limit the traffic of data between the wireless base unit and the slave modules, the user can set the value of the change that result in new analogue values being sent.

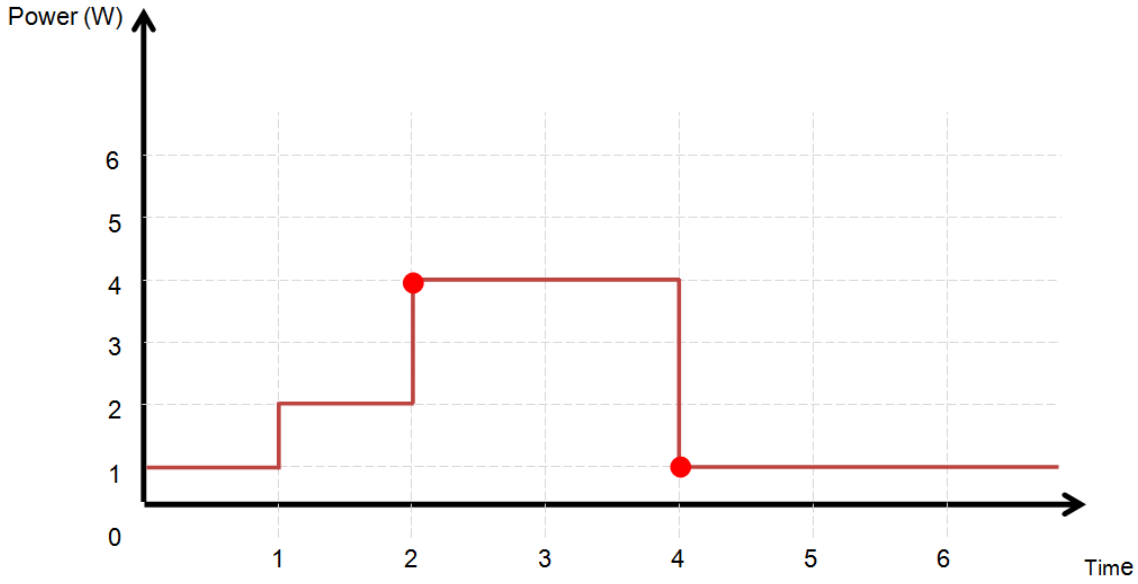
In the *Trigger window* fields, the user can set the range of variation: if the value read differs from the last value sent by a number bigger than the one defined in these fields, the new value is sent.

In the *Properties* window of a wireless module (SHDWWISENxxx, SHJWD200WExxx, SHJWRE10AExxx, SHJWEM16Axxx, SHJWINS04xxx, SHDWRE16AE230) the *Advanced settings* fields are available, where the user can set the difference between two readings to send the energy value for KWh, Watt, Volt and Ampere, as shown in the red rectangle in the picture below:



Example
In the example shown below, the *Difference between two readings to send the energy value* (*W)* field is set to 2 W: every time the analogue value (power) has a variation bigger than 2W from the last sent value, it is sent to the UWP 3.0. In the example below, a new sample is sent at point 2, because the power value goes from 2 W to 4 W (the variation equals 2 W), then another sample is sent at point 4 when the power value goes down from 4 W to 1 W (the variation equals 3 W).

● Points where the analogue values are sent to the UWP 3.0 controller.

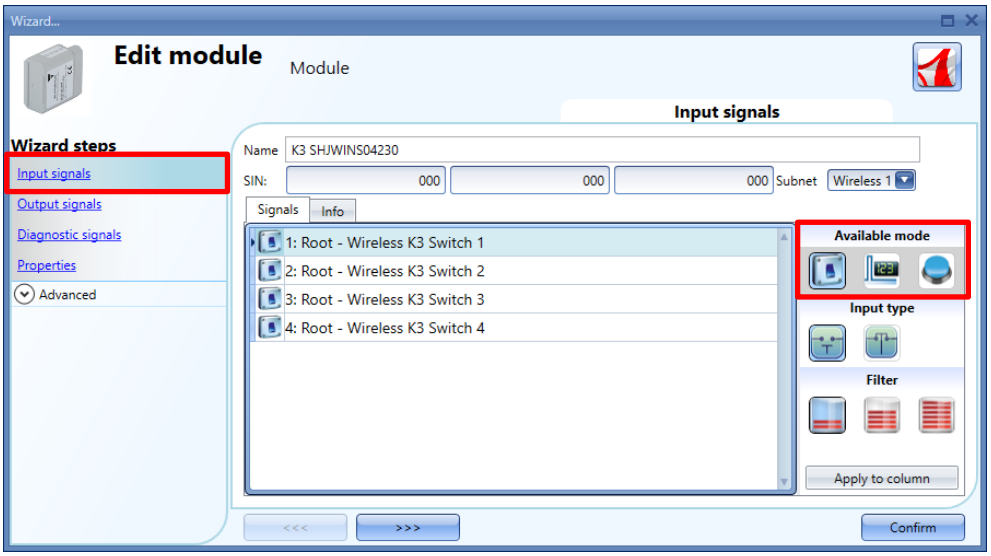


7.14.3 Wireless input modules

The SHJWINS04115 and the SHJWINS04230 are input modules for counting pulses from energy meters, water meters, gas meters etc...and also include a people-counting functionality. It is also possible to use the inputs as standard digital inputs. These can be configured via the UWP 3.0 tool for each of the inputs.

To configure them, click on the relevant icon once they are added to the project:

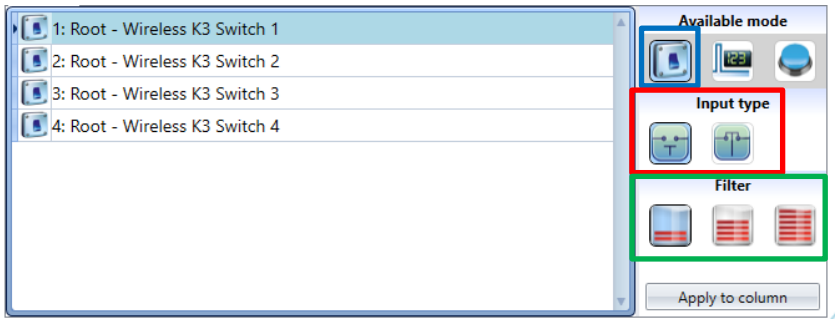
The configuration wizard will appear:



As for the settings of the SHJWINS04xxx, in the *Input signals* field the user has to select the working mode: this can be configured as a switch (the switch icon has to be selected), as a pulse counter by selecting the icon with *123,* or as a push button (the third icon has to be selected).

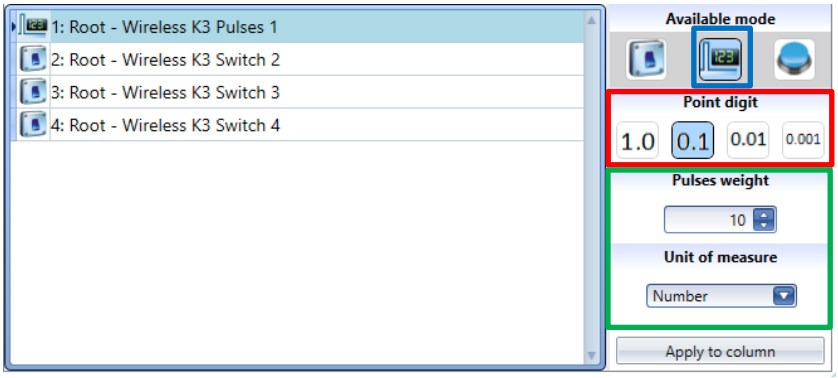
Input configured as a switch

If the behaviour is configured as a switch, the user can select whether the input is normally closed or normally open by clicking on the relevant *Input type* icon (see red rectangle below). Then the type of filtering of the signal on the bus can be selected from three options: low, medium or high filtering (see green rectangle below). This should be selected according to how noisy the bus is.



Input configured as a pulse counter

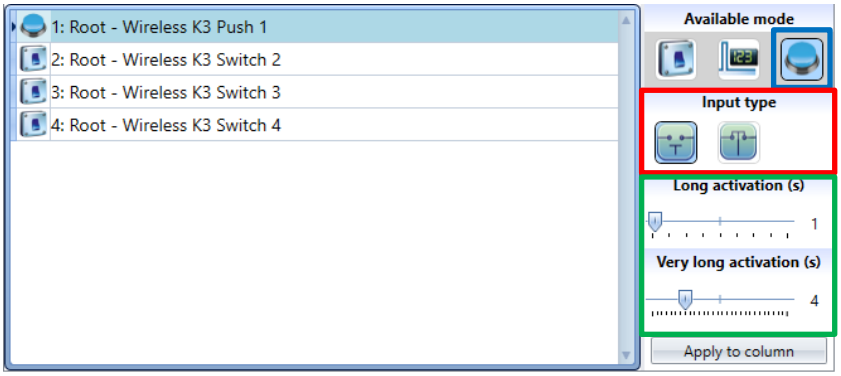
If the behaviour is configured as a pulse counter, the user can select the weight of each pulse in the *Pulse weight* field and the *Unit of measurement* (see green rectangle below). The user has to set the point digit position which to be applied by the system when the pulse counter value is read from the wireless input module (see red rectangle below)



Note: If the wireless input module has to be used in a people-counting function, please see *How to configure a counter function* section in this manual.

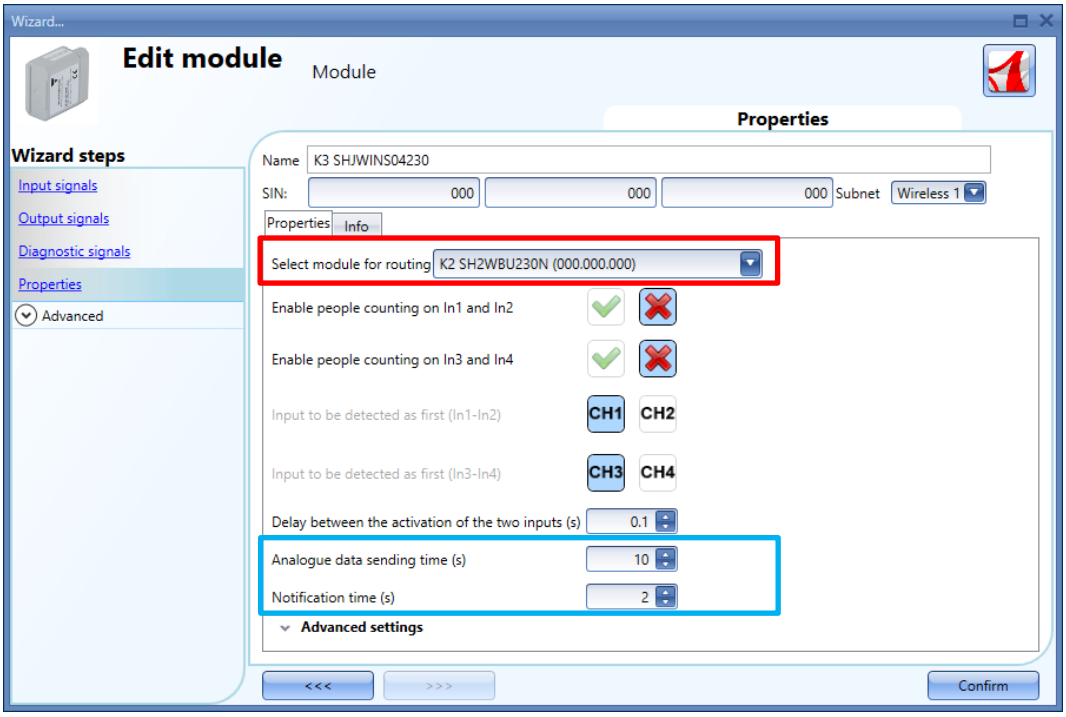
Input configured as a push button

If the behaviour is configured as a push button, such as for light switches, the *Long activation* and *Very long activation* time has to be set: The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds, it is always set by the system 3 seconds longer than the *Long activation* time (see green rectangle below). The user can also select if the input is normally closed or normally open by clicking on the relevant *Input type* icons (see red rectangle below).



The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to column button.*

In the *Properties* window, the user can choose in the *Select module for routing* field the module is which to be used for routing if the location (of the module being configured) is in an area not covered by the SH2WBU230N: in this situation it is possible to extend the operating distance by using a SHJWD200WExxxx, a SHJWEM16Axxx, a SHJWRE10AExxx, a SHDWRE16AE230 or a SHJWINS04xxx as a router/repeater.



In order to limit the traffic of data between the wireless base unit and the Wireless input module, in the *Analogue data sending time (s) field* the user can set the period in seconds in which the analogue values are sent to the UWP 3.0 Tool.

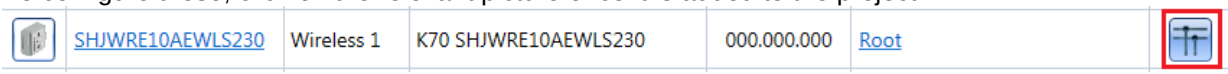
In the *Notification time (s)* field the user can set the period in seconds in which the Diagnostic signals have to be updated on the UWP 3.0 tool when the *Live Signals* are enabled.

7.14.4 Wireless relay modules

The following items belong to the *wireless relay modules*:

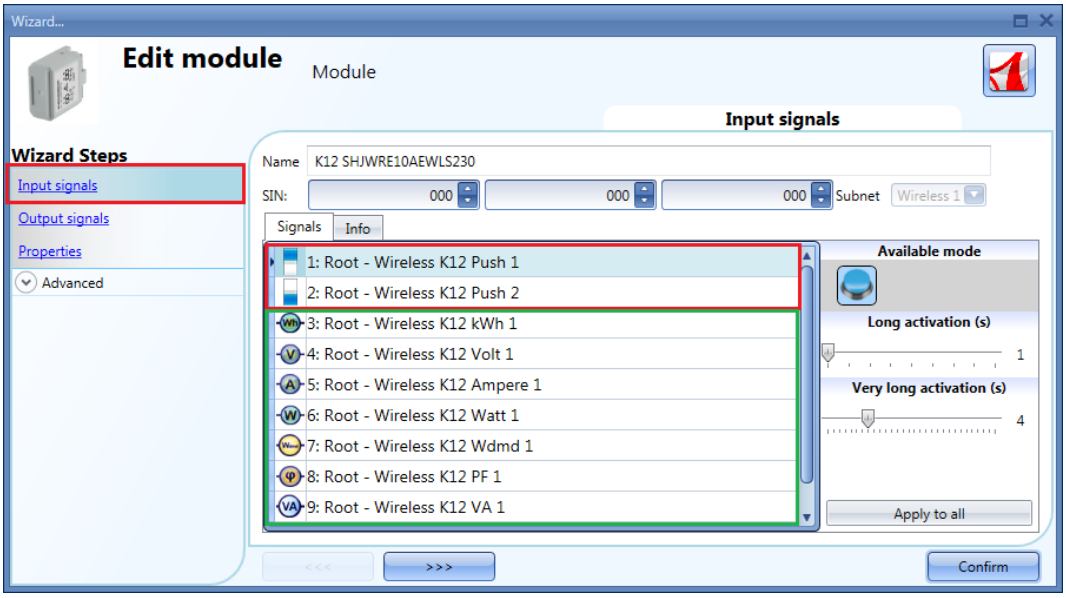
SHJWRE10AE230
SHJWRE10AE115
SHJWRE10AEWLS230 (with two push buttons)
SHJWRE10AEBLS230 (with two push buttons)
SHDWRE16AE230

To configure these, click on the relevant picture once it is added to the project:



The configuration wizard will appear:

In the *Input Signals* window, all the analogue variables added in the *Properties* window will be shown.

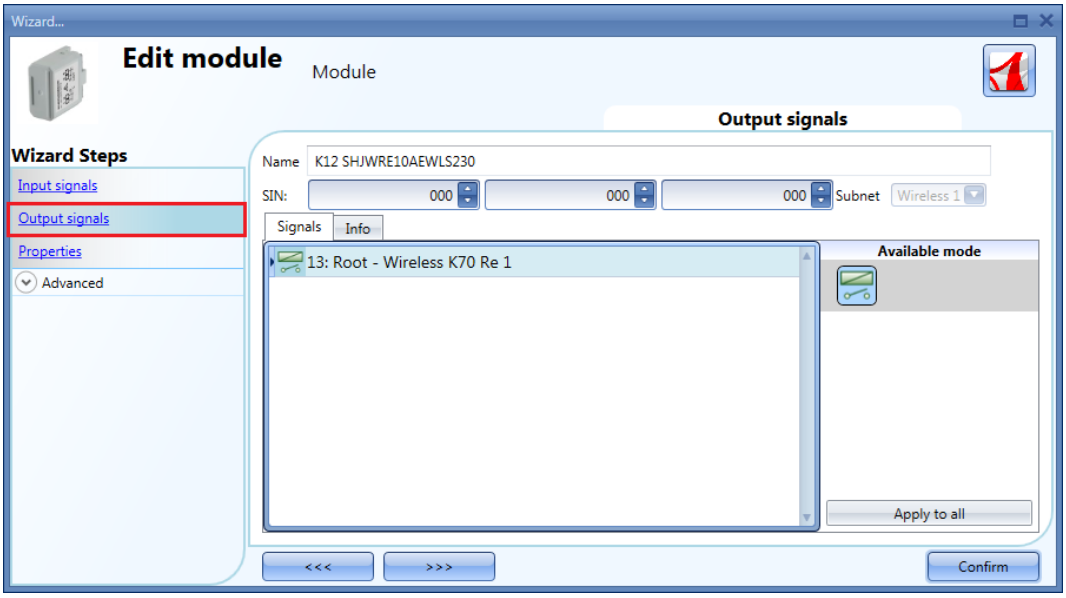


The SHJWRE10AExLS230 module integrates two programmable capacitive push-buttons and can be used in all the UWP 3.0 tool functions, as shown in the red rectangle in the picture above.

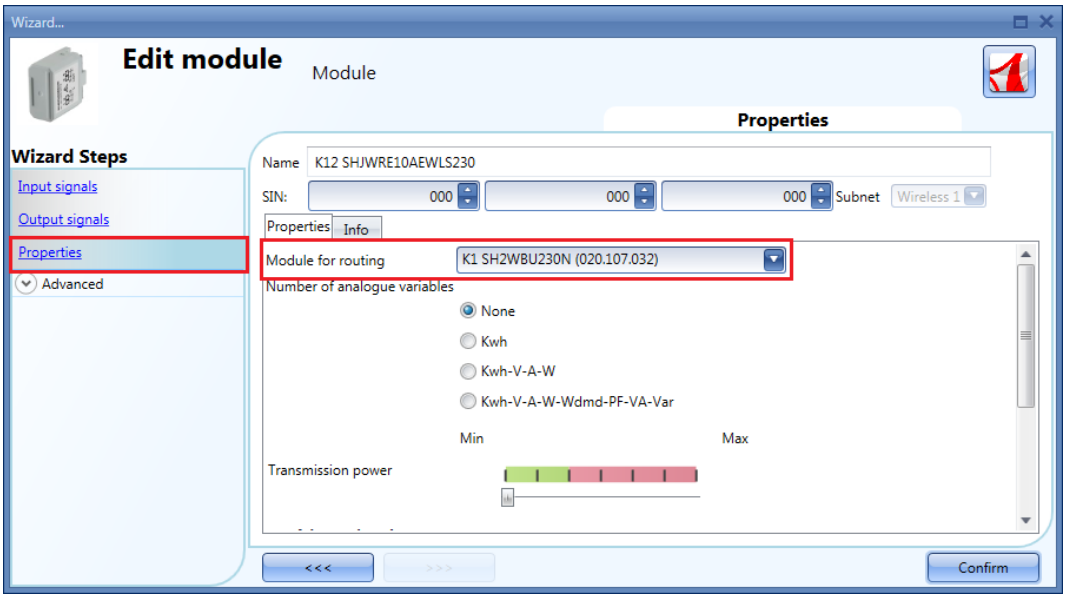
In the *Input signals* field, the *Long activation* and the *Very long* activation time has to be defined.
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds; it is always set 3 seconds longer than the *Long activation* time. The short press and long press are recognized when the push button is released.
The user can configure different times for each push button, or he can set them to the same values by clicking on *Apply to all.*

If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.

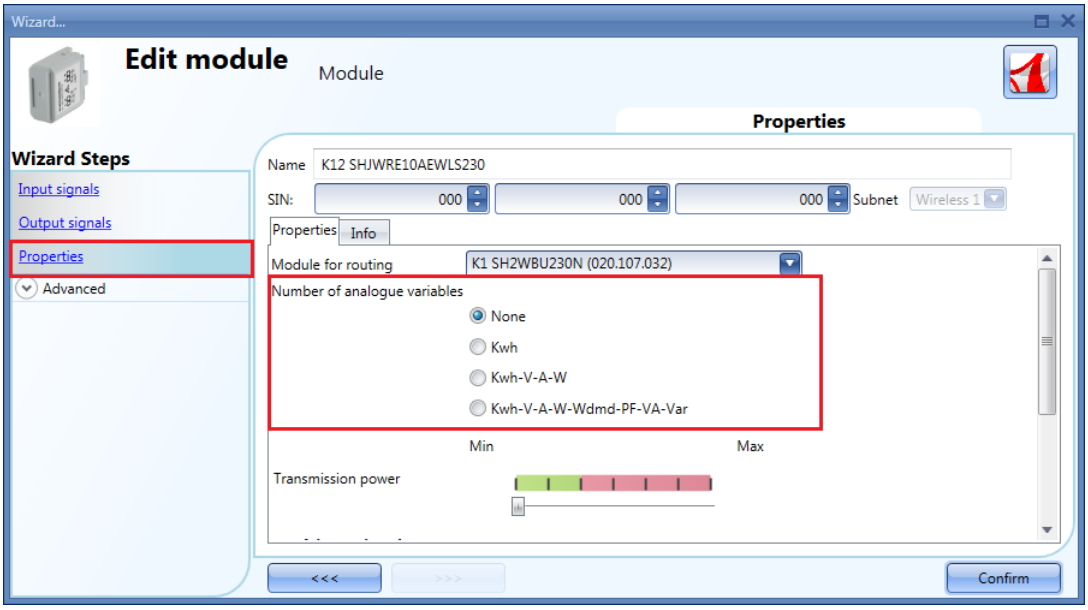
In the *Output Signals* window, the relay output is shown.



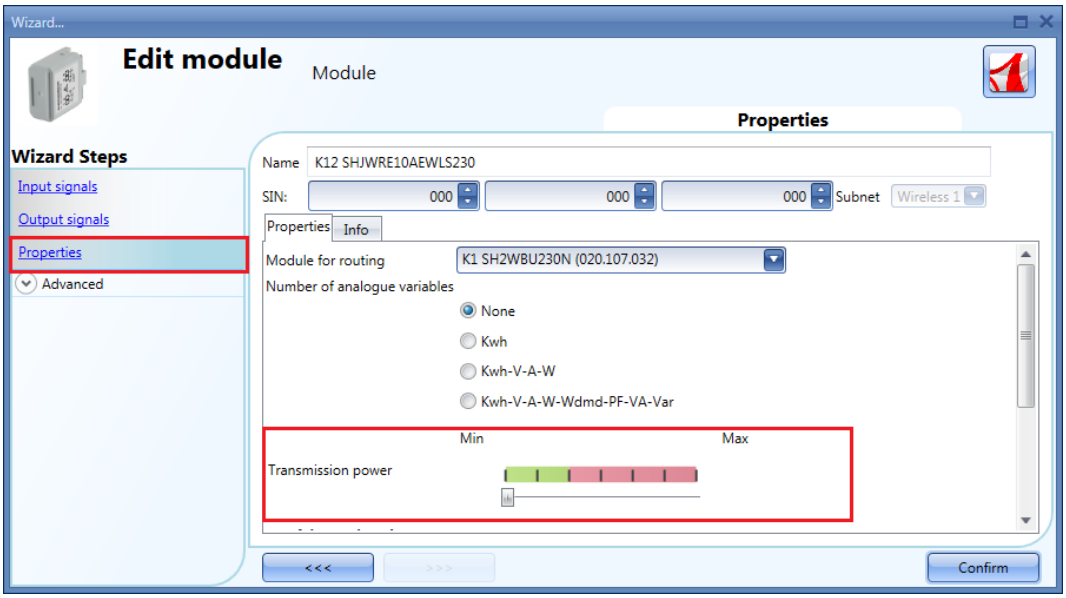
In the *Properties* window, the user can choose the module to be used for routing if the wireless relay module is placed in an area not covered by the wireless base unit. In the *Module for routing* field, the user can select the way the modules connect to the SH2WBU230x: either directly or via a router, as shown in the red rectangle in the picture below.



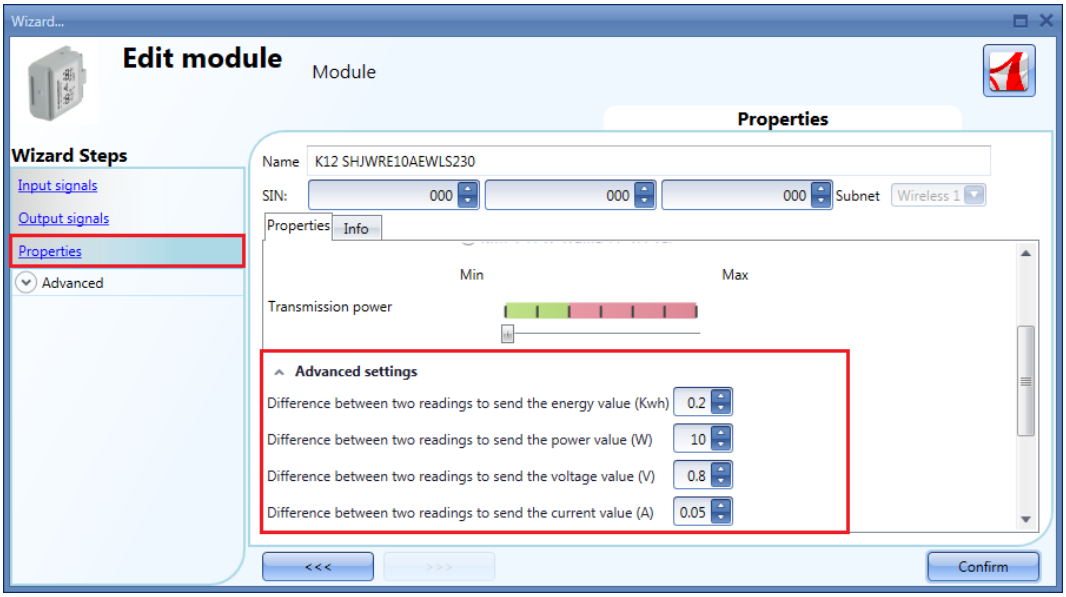
In the *Number of analogue variables* field, the user can select the relevant group of analogue variables, as shown in the picture below.



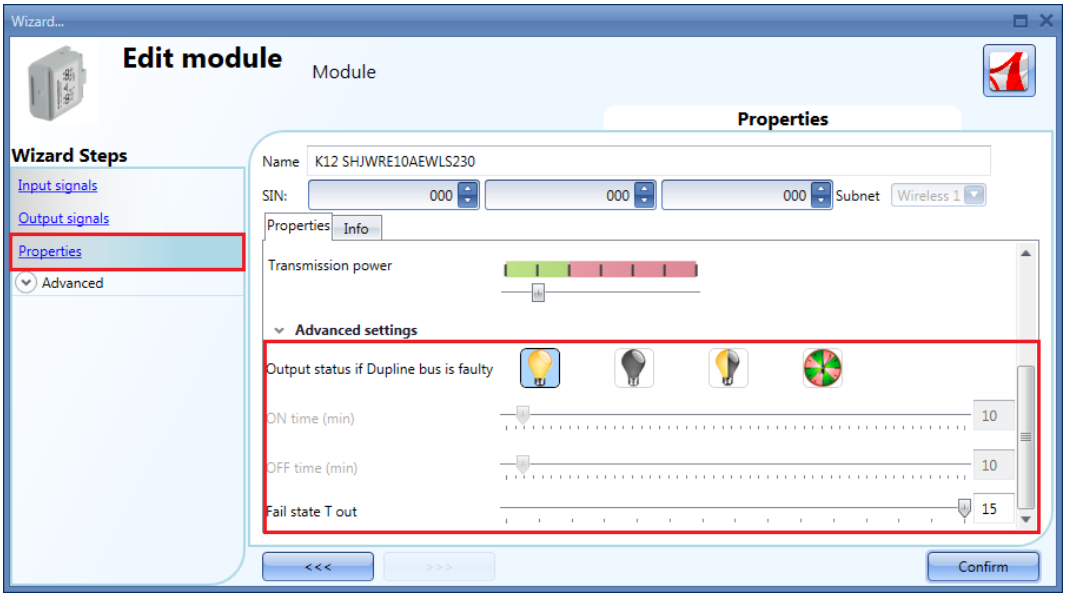
In the *Transmission power* field, the user can also manage also the transmission power of the module by moving the slider. This field is not available for the SHDWRE16AE230 module.
If there are metallic parts shielding the wireless field or if a longer distance is needed, the strength of the wireless field can be increased.



In the *Advanced settings* fields, the user can set the range of variation: if the value read differs from the last value sent by a number bigger than the one defined in these fields, then the new value is sent, otherwise it will not be.



In the *Output status if Dupline bus is faulty* field, the user has to select the fail-safe working mode if the connection with the wireless base unit is faulty.



There are four available setups:

1) Output always on (yellow bulb)
2) Output always off (grey bulb)
3) The output maintains the status it had before the disconnection (yellow and grey bulb)
4) The output recycles: it will be on for the *On time*, and off for the *Off time.*

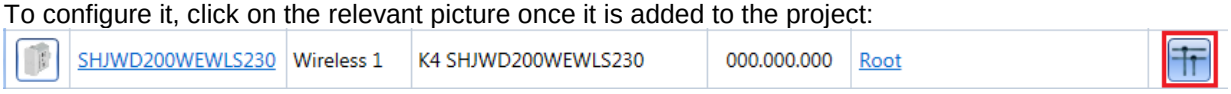
Fail state T out: when the module loses communication with the wireless base unit for a time value longer than the entered value, the fail state condition is activated (it is possible to set the time from 1 to 15 minutes).

7.14.5 Wireless dimmer modules

The following items belong to the *wireless dimmer module*s with push buttons:

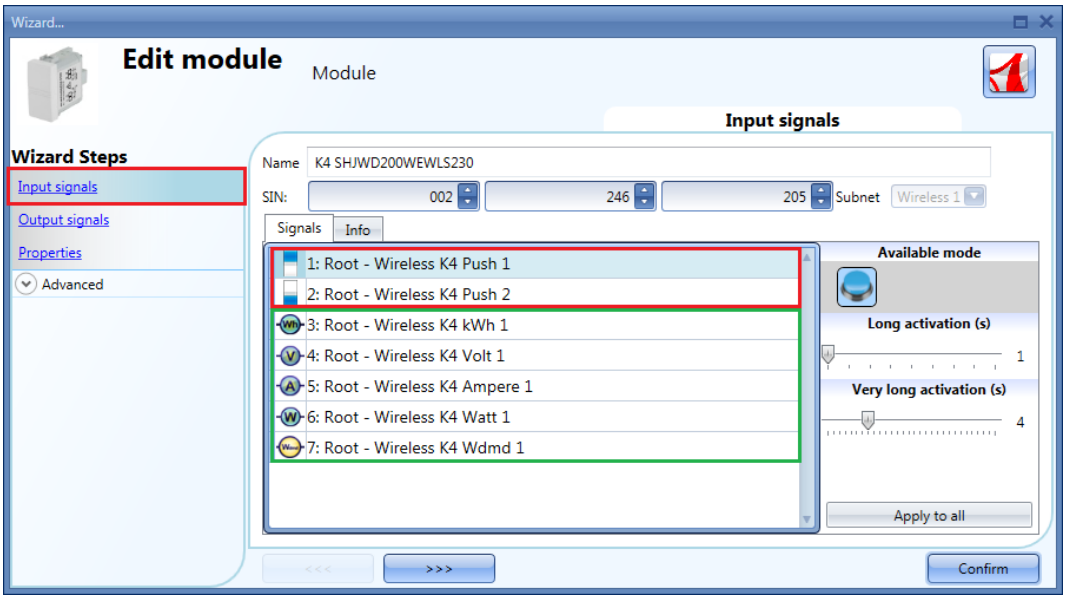
SHJWD200WE230
SHJWD200WE115
SHJWD200WEWLS230
SHJWD200WEBLS230

To configure it, click on the relevant picture once it is added to the project:



The configuration wizard will appear:

In the *Input Signals* window, all the analogue variables added in the *Properties* window will be shown.

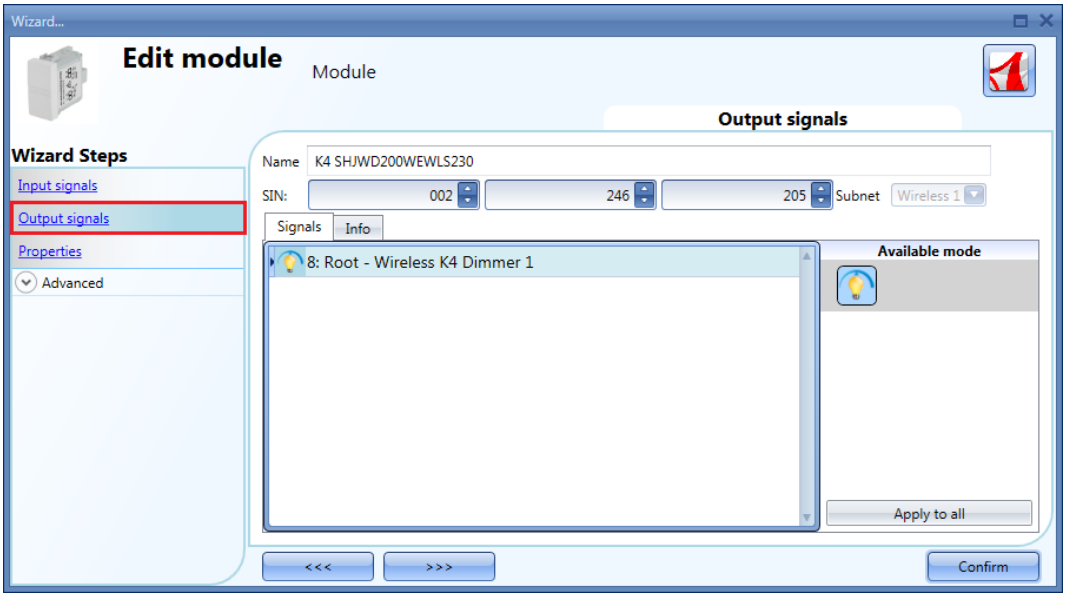


The SHJWD200WExLS230 module integrates two programmable capacitive push-buttons and can be used in all the UWP 3.0 tool functions, as shown in the red rectangle in the picture above.

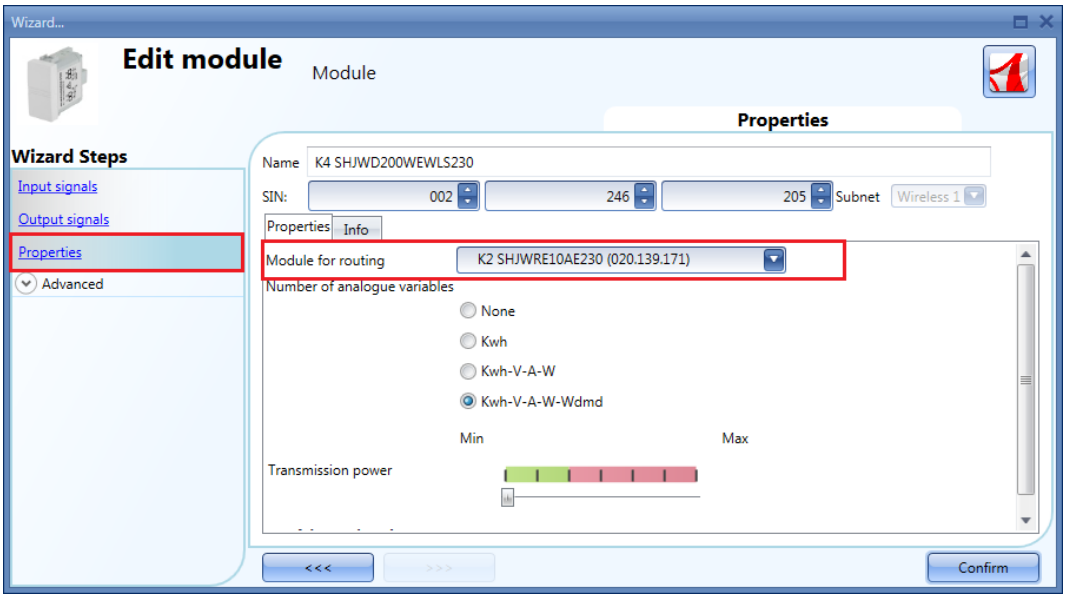
In the *Input signals* field, the *Long activation* and the *Very long* activation time has to be defined.
The *Long activation* time can be set from 1 to 5 seconds, while the *Very long activation* time can be set from 0.5 to 15 seconds and it is always set by the system 3 seconds longer than the *Long activation* time.
The short press and long press are recognized when the push button is released.
The user can configure different times for each push button, or can set them to the same values by clicking on *Apply to all.*

If the *Very long activation time* is set shorter than 4 seconds, the long press is disabled: in this situation, the function associated with the very long press will be activated as soon as the push button is pressed for the selected time.

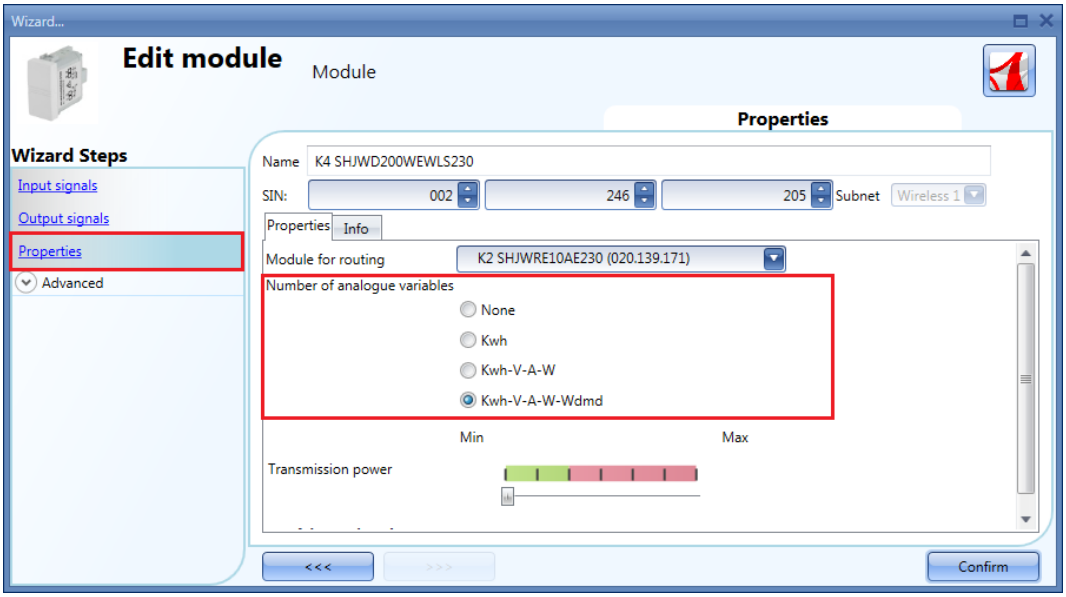
In the *Output Signals* window, the relay output is shown.



In the *Properties* window, the user can choose the module to be used for routing if the wireless dimmer module is placed in an area not covered by the wireless base unit. In the *Module for routing* field, the user can select the way the modules connect to the SH2WBU230x: either directly or via a router, as shown in the red rectangle in the picture below.

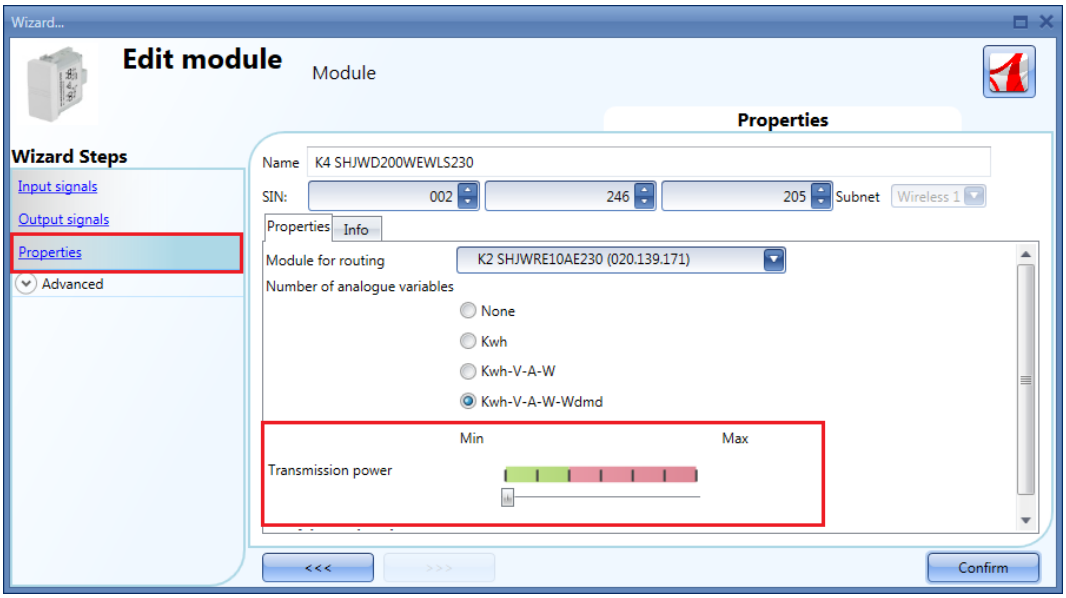


In the *Number of analogue variables* field, the user can select the relevant group of analogue variables, as shown in the red rectangle in the picture below.

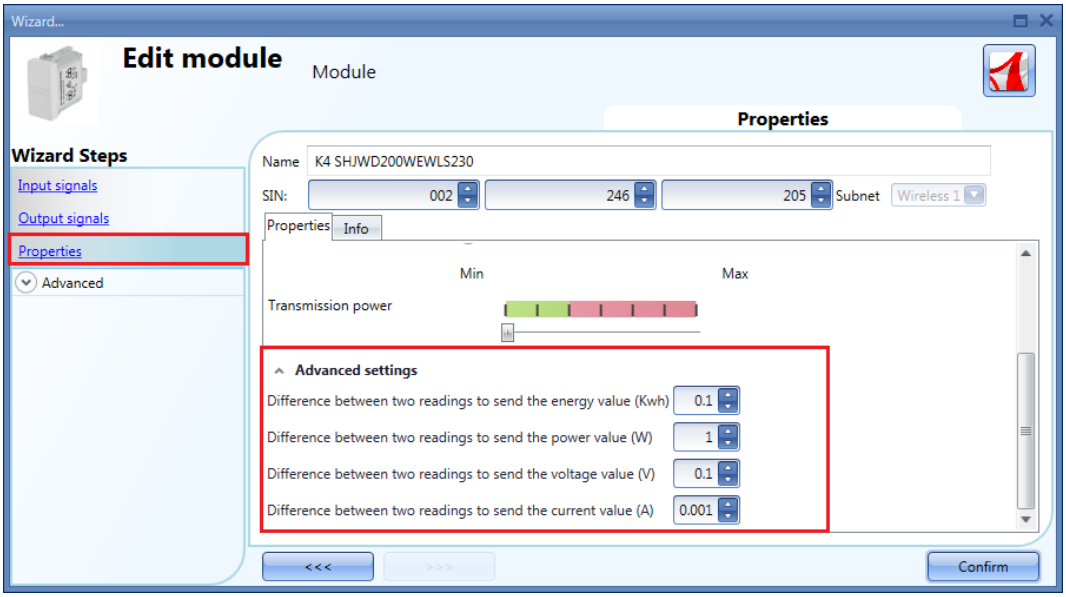


N.B. If the fourth option is selected in the field *Number of analogue variables*, the module will transmit the *kWh, V, A, W, Wdmd* values.

In the *Transmission power* field, the user can also manage the transmission power of the module by moving the slider.
If there are metallic parts shielding the wireless field or if a longer distance is needed, the strength of the wireless field can be increased.



In the *Advanced settings* fields, the user can set the range of variation: if the value read differs from the last value sent by a number bigger than the one defined in these fields, then the new value is sent, otherwise it will not be.

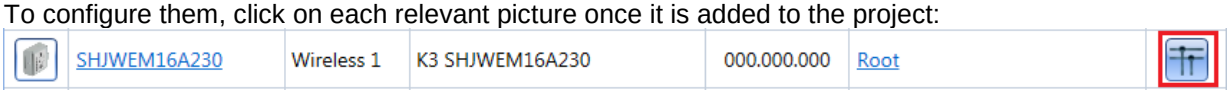


7.14.6 Wireless energy meter modules

The following items belong to the *wireless energy meter modules*:

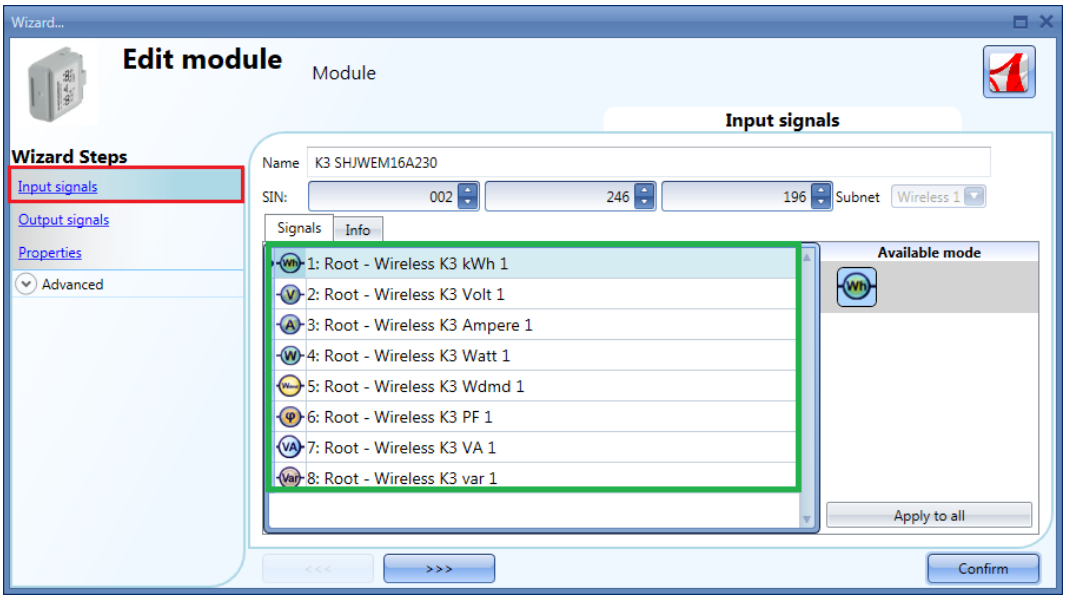
SHJWEM16A230
SHJWEM16A115

To configure them, click on each relevant picture once it is added to the project:

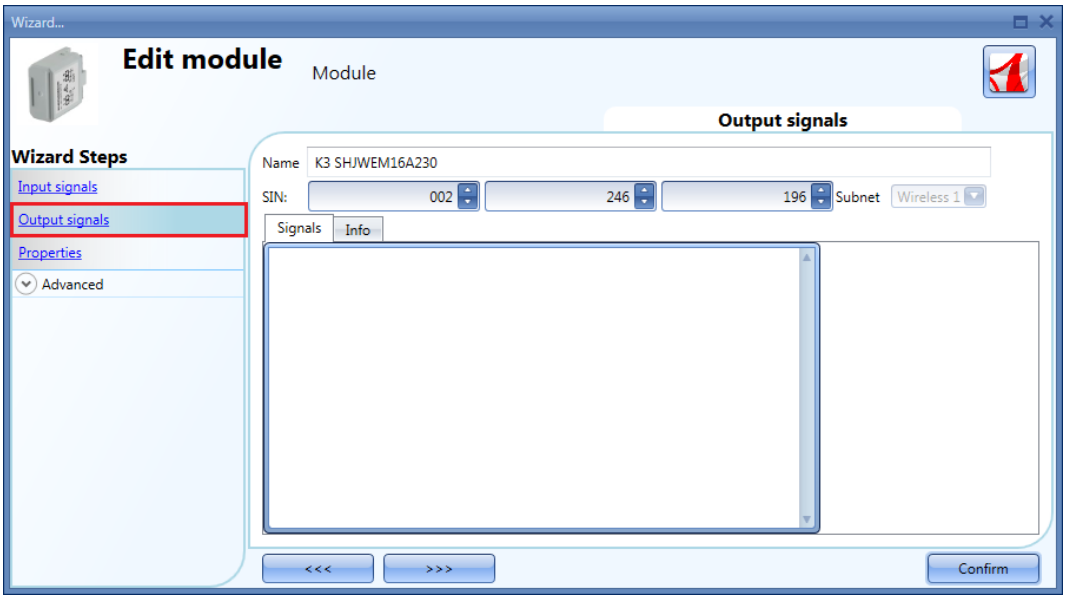


The configuration wizard will appear:

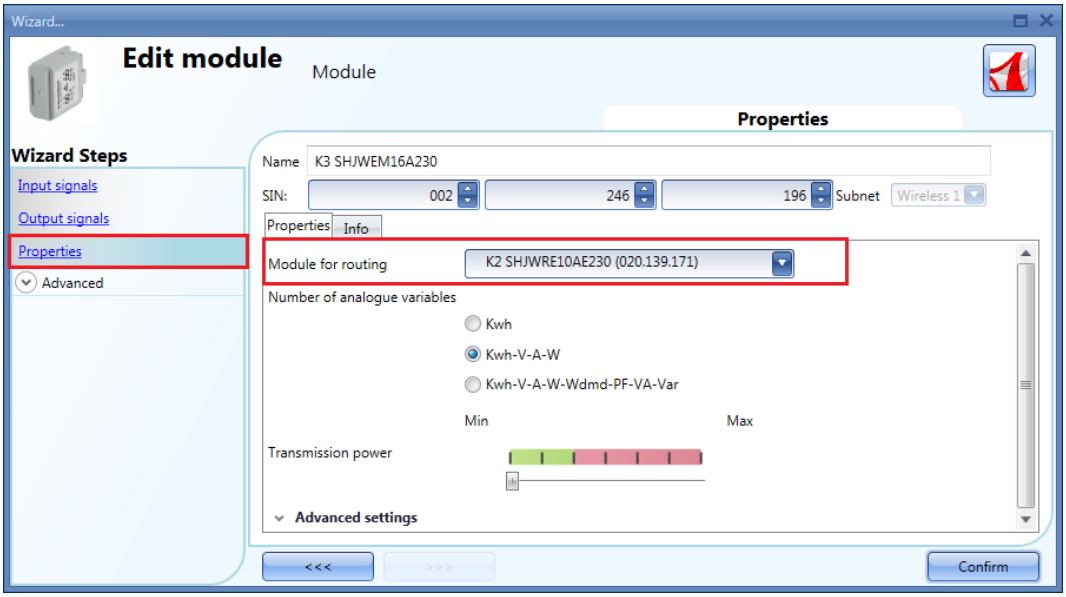
In the *Input Signals* window, all the analogue variables added in the *Properties* window will be shown.



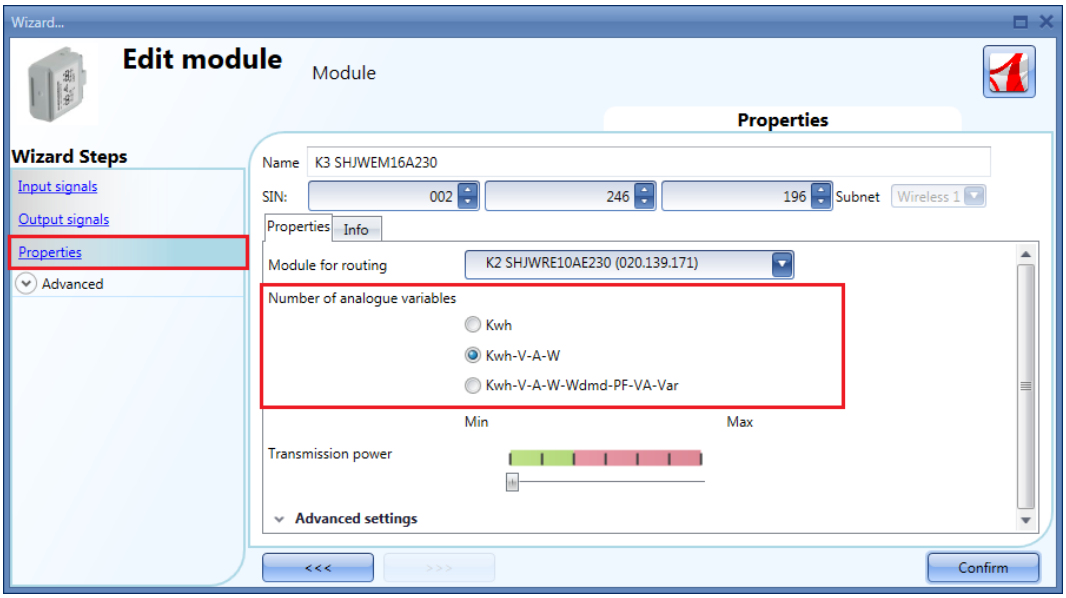
The wireless energy meter has no output signals, so the *Output signals* window is blank.



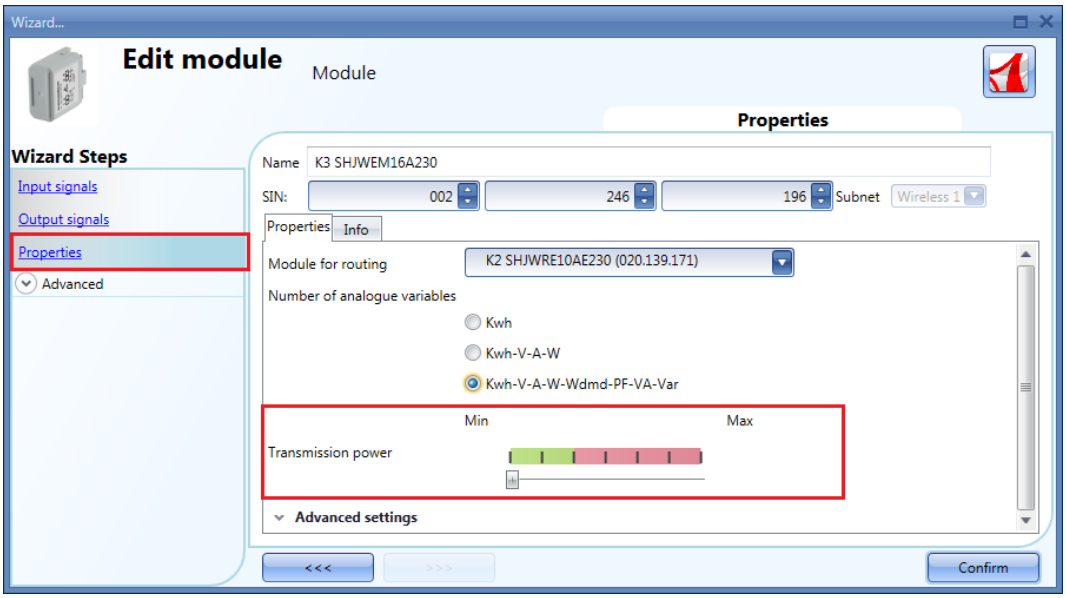
In the *Properties* window, the user can choose the module to be used for routing if the wireless meter module is placed in an area not covered by the wireless base unit. In the *Module for routing* field, the user can select the way the modules connect to the SH2WBU230x: either directly or via a router, as shown in the red rectangle in the picture below.



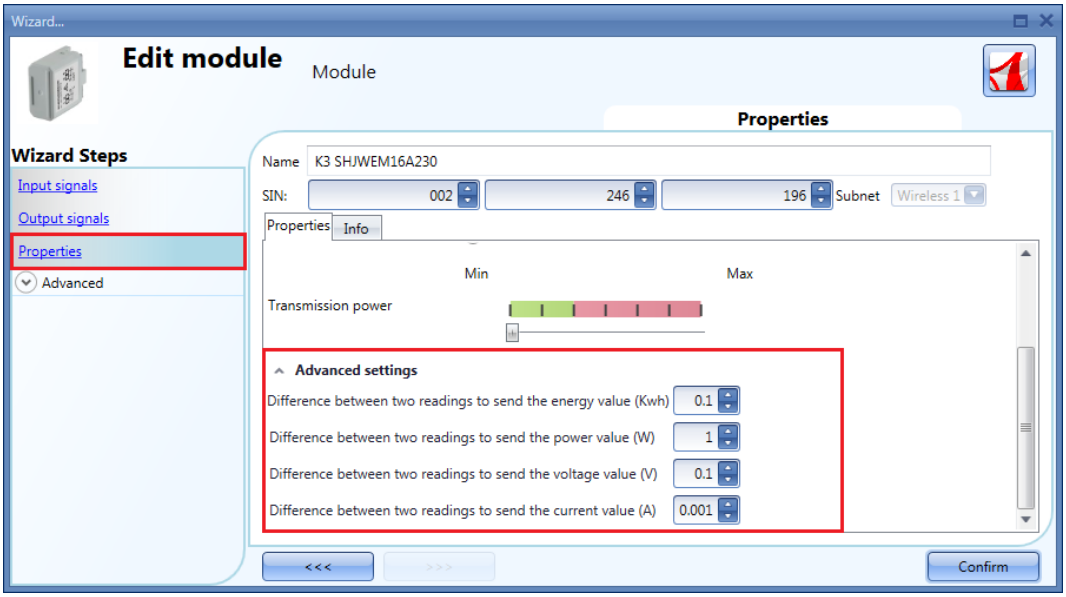
In the *Number of analogue variables* field, the user can select the relevant group of analogue variables, as shown in the red rectangle in the picture below.



In the *Transmission power* field, the user can also manage the transmission power of the module by moving the slider.
If there are metallic parts shielding the wireless field or if a longer distance is needed, the strength of the wireless field can be increased.



In the *Advanced settings* fields, the user can set the range of variation: if the value read differs from the last value sent by a number bigger than the one defined in these fields, the new value is sent otherwise it will not be.



7.14.7 How to manage the routing tree

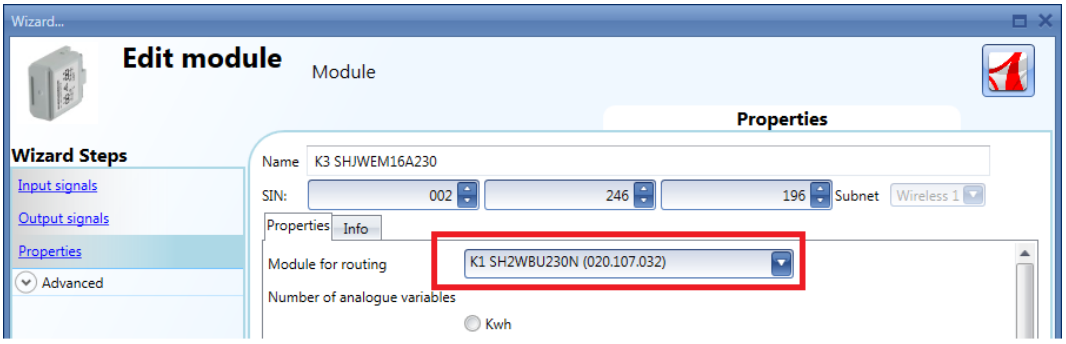
The wireless modules can be assigned directly to the wireless base unit, or a module with routing capability can be used as router/repeater between the base unit and the module to be reached, if this is placed in an area not covered by the SH2WBU230x: in this situation it is possible to extend the operating distance by using up to two modules as routers/repeaters. A wireless module with routing capability must be used, such as SHJWRE10AExxx, SHJWD200WExxx, SHJWEM16Axxx, SHJWINS04xxx or SHDWRE16AE230.

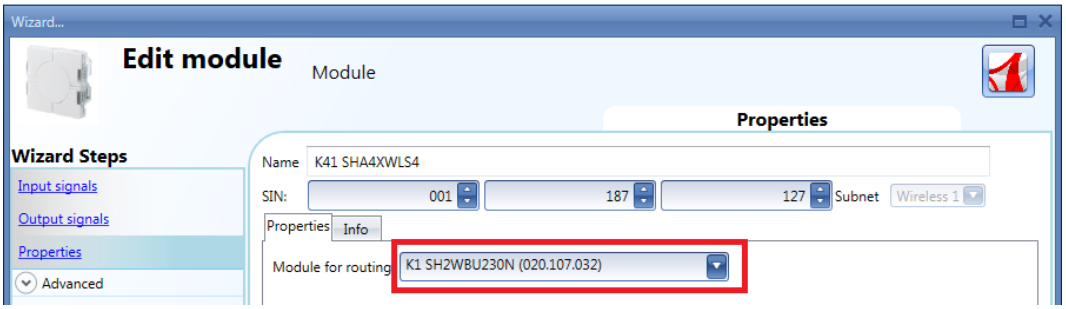
The new generation SH2WBU230N can manage up to two levels of routing, whereas the SH2WBU230 and SHDWRE16AE230 can manage only one level of routing.

There are two ways to generate the Routing tree.

7.14.7.1 In the *Properties* window of the module

In the *Properties* window, in the *Select module for routing* field, the user can select the way the wireless module connects to the SH2WBU230x: either directly or via a router.



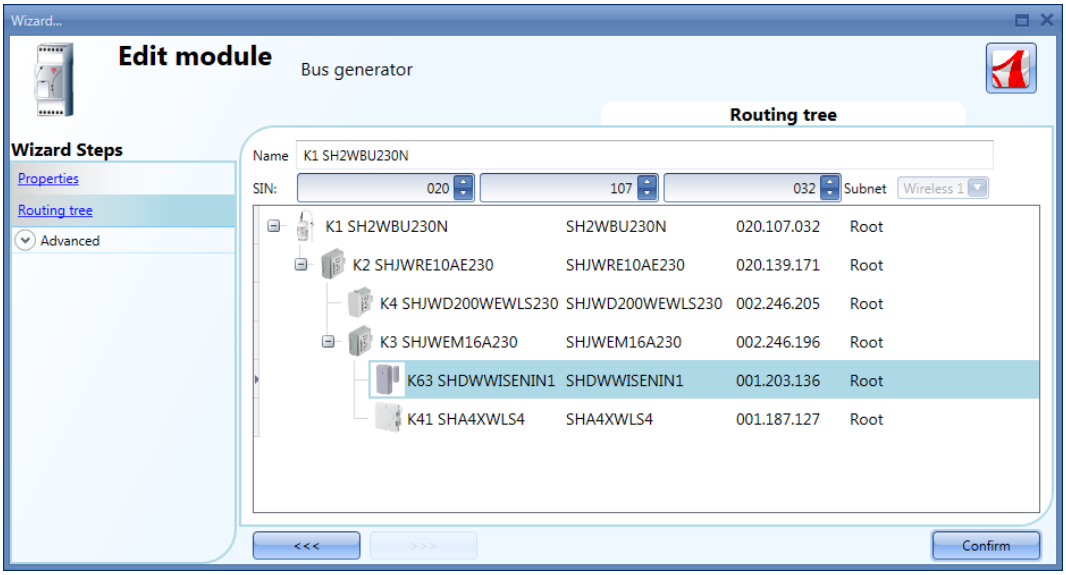


7.14.7.2 In the *Routing tree* window of the wireless base unit

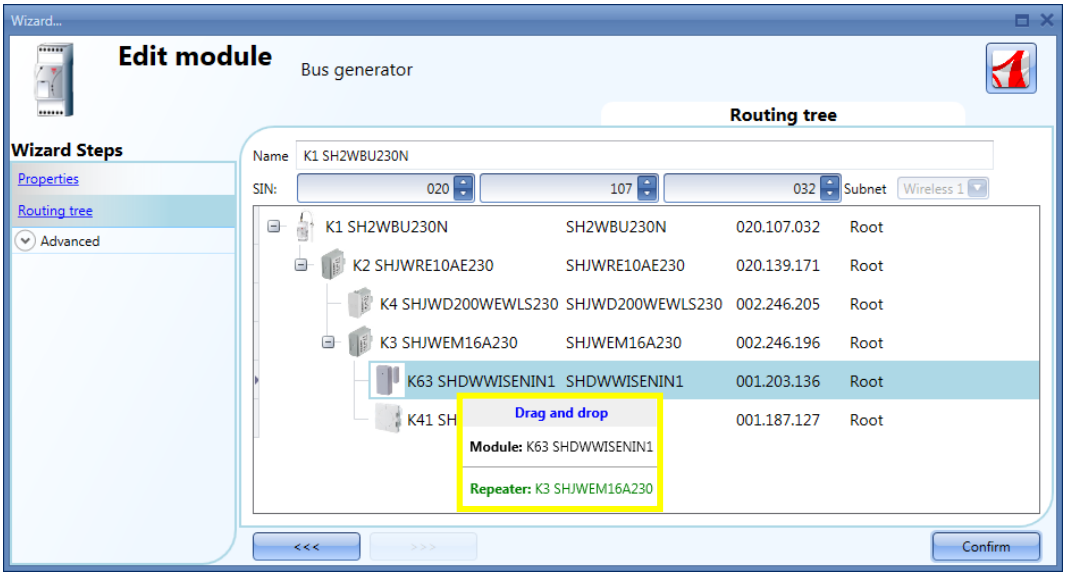
In the *Routing tree* window, an overview of the wireless connection tree is shown. The user can change the associations of the modules by means of drag-and-drop actions.

In the example shown below, the window sensor (K63 SHDWWISENIN1) is connected indirectly to the SH2WBU230N and it uses a wireless energy meter and a wireless relay (K3 SHJWEM16A230 and K2 SHJWRE16AE230) as routers to communicate to the wireless base unit. The wireless dimmer (K4 SHJWD200WEWLS230) has only one router to communicate to the wireless base unit (K2 SHJWRE16AE230). The positioning of the modules in the right locations has to be done using *drag & drop,* according to the following steps:

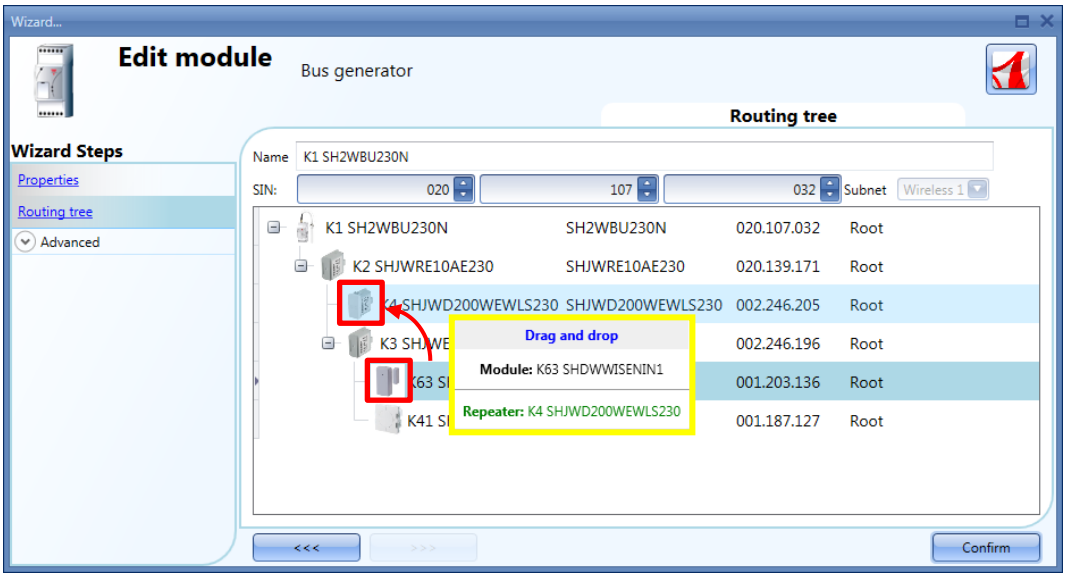
1. Move the mouse pointer until it is directly over the module you want to move.



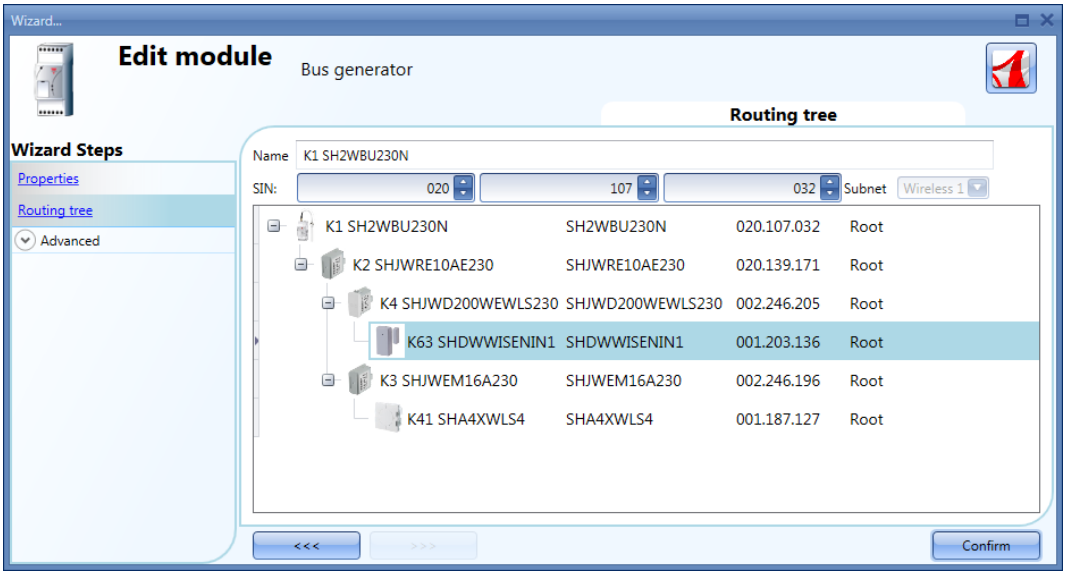
2. Press and hold down the left mouse button. When you start moving the module, a tool tip will appear (see the yellow box in the picture below): in the *Module* field the name of the module to be moved is shown, while in the *Repeater* field there is the name of the module with routing capability which the source module is associated to.



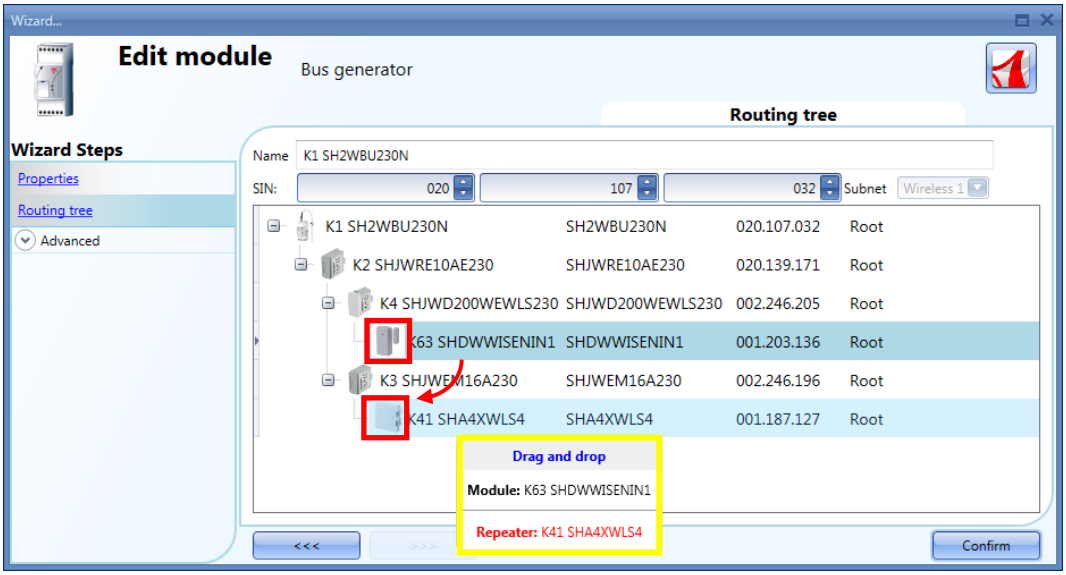
3. With the left mouse button held down, drag the object to the desired location (over the K4 SHJWD200WEWLS230 module as shown in the example below).



4. Drop the module by releasing the left mouse button: the window sensor (K63 SHDWWISEN1) module is now assigned to the wireless dimmer (K4 SHJWD200WEWLS230), as shown in the picture below.

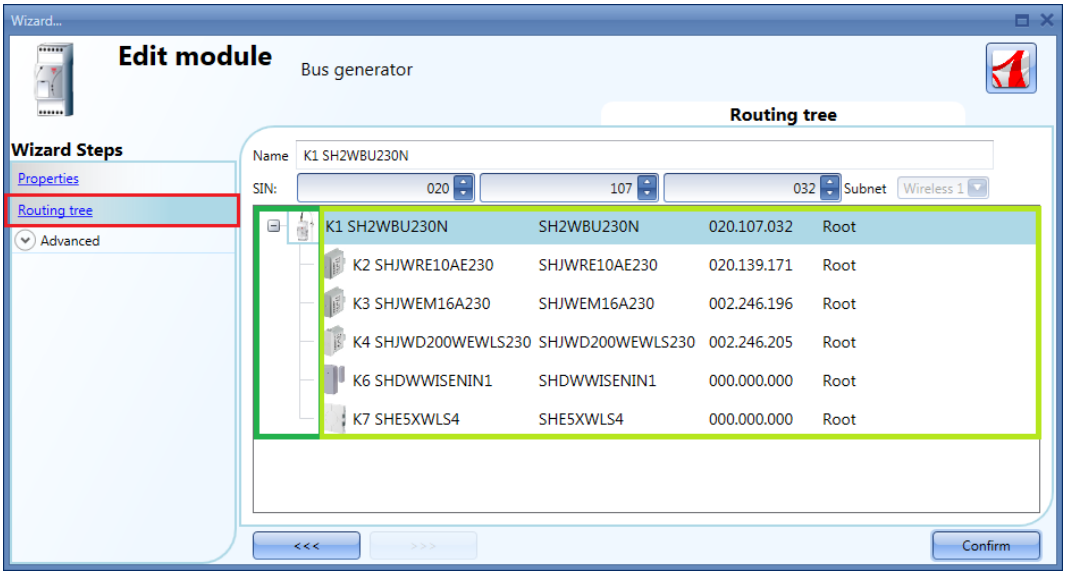


N.B. The *Repeater* module must be a wireless base unit or a wireless module with routing capability: when a *drag & drop* action is performed on a not-allowed *repeater* module, the *Repeater* field shown by the tool tip is highlighted in red: in the example below, the module SHA4XWLS4 can't be used as a router module for the SHDWWISENIN1, since it does not have routing capabilities.

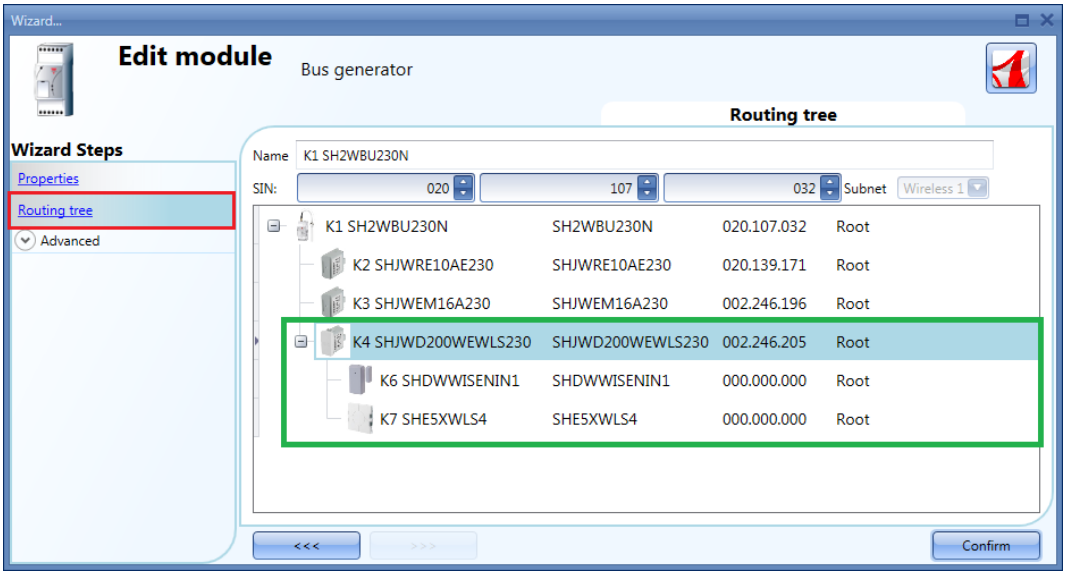


The following are some examples of routing trees.

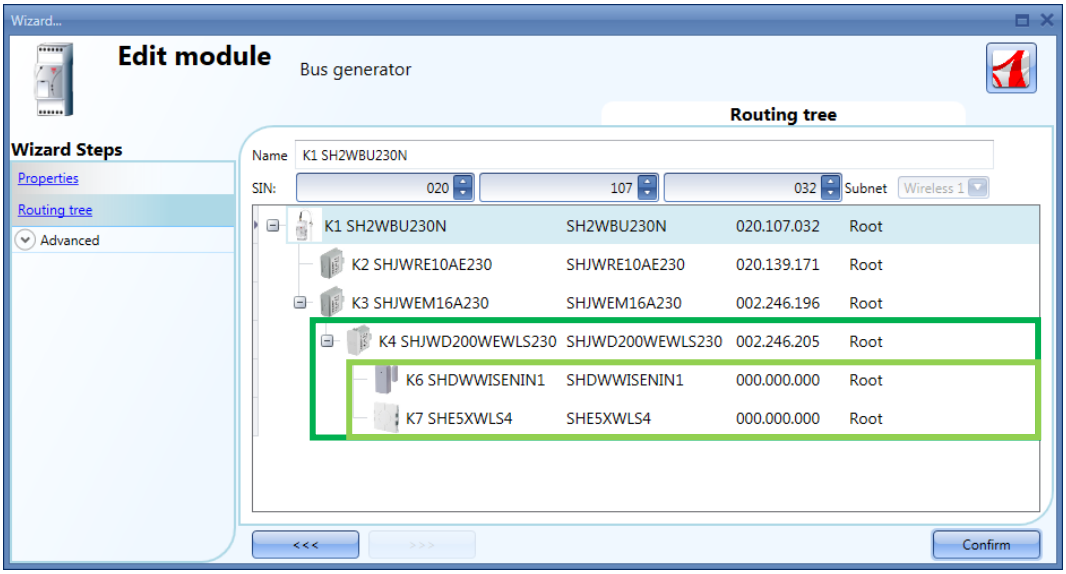
Example 1: all the wireless modules are directly connected to the wireless base unit (please see the green rectangle).



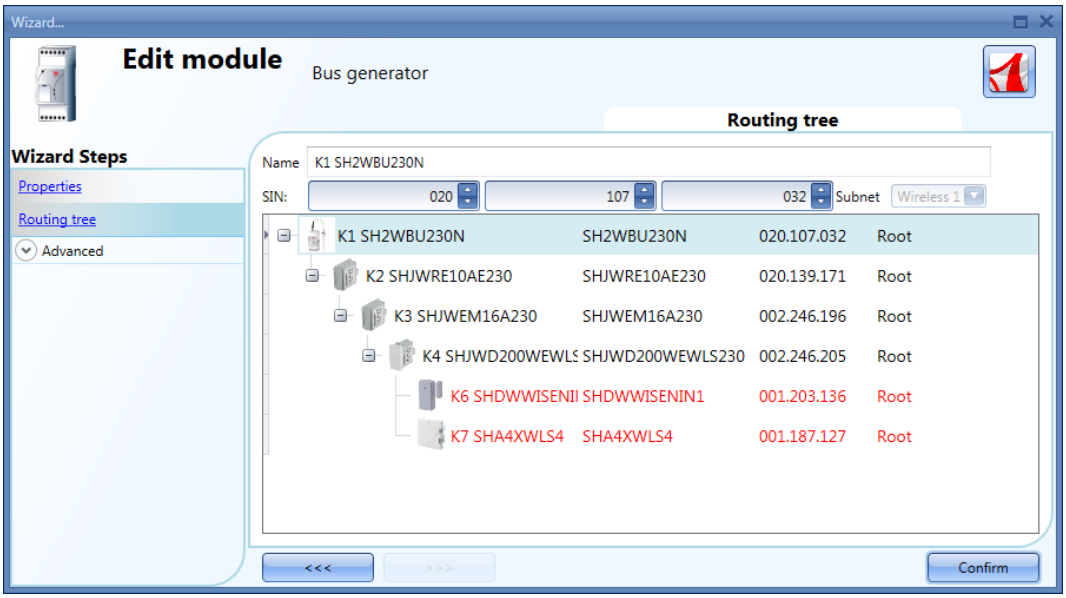
Example 2: SHDWWISENIN1 and SHE5XWLS4 modules are connected indirectly and they use the SHJWD200WEWLS230 module as a router to communicate with the wireless base unit (see the green rectangle in the picture below). The SHJWRE10AE230 and the SHJWEM16AE230 are directly connected to the wireless base unit.



In the picture below the SHDWWISENIN1 and the SHE5XWLS4 modules are indirectly connected to the wireless base unit through two routers: the first router is the SHJWEM16AE230 module and it is directly connected to the wireless base unit; the second one is the SHJWD200WEWLS230 and it is between the first router and the other two slave modules (SHDWWISENIN1 and the SHE5XWLS4), as shown in the green rectangle in the picture below.



In the example shown below the two levels of routing have been exceeded and since this is not possible, the modules are marked in red.

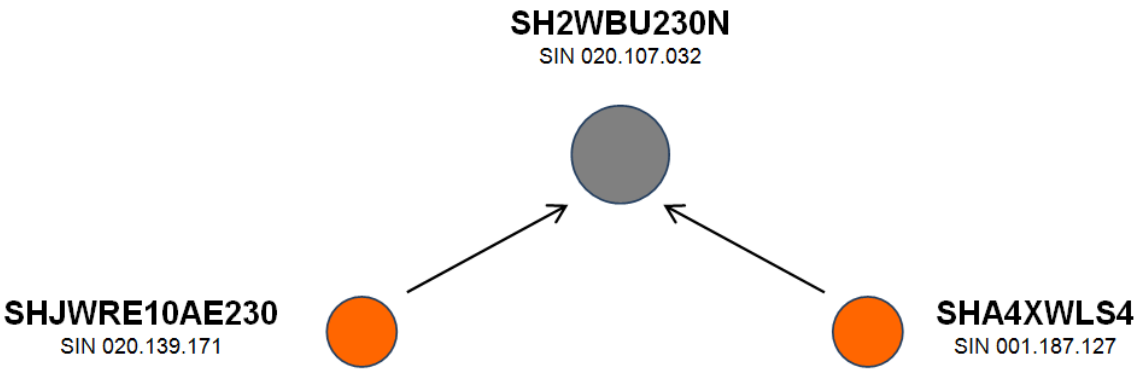


The table below shows the routing capability and the maximum operating distance in the open air for each wireless module when it is connected to the wireless base unit SH2WBU230N:

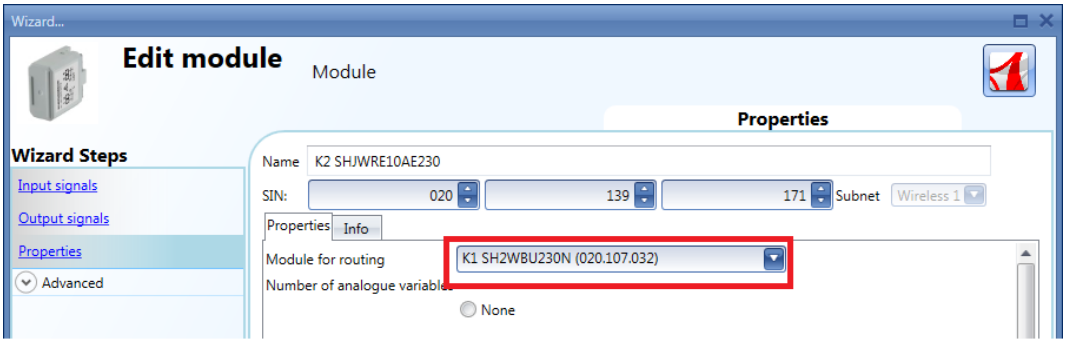
WiDup module	Routing capability	Operating distance
SHJWRE10Axxx	YES, 2 levels	700 mt. open air
SHJWRE10AExLS230	YES, 2 levels	700 mt. open air
SHJWD200WExxx	YES, 2 levels	700 mt. open air
SHJWD200WExLS230	YES, 2 levels	700 mt. open air
SHJWEM16Axxx	YES, 2 levels	700 mt. open air
SHJWINS04xxx	YES, 2 levels	700 mt. open air
SHDWRE16AE230	YES, 1 level	100 mt. open air
SHA4XWLS4	NO	100 mt. open air
SHE5XWLS4	NO	100 mt. open air
SHDWWISENxxx	NO	100 mt. open air

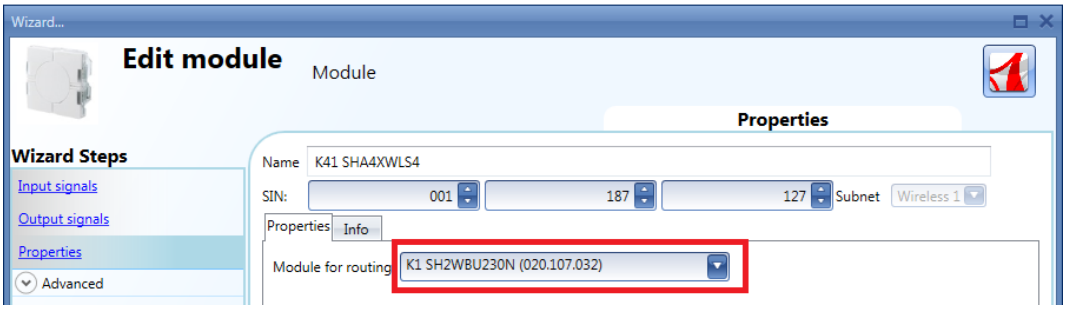
7.14.8 How to connect a module directly to the wireless base unit

The picture below shows a simple example of a direct connection of two modules to the wireless base unit SH2WBU230x.

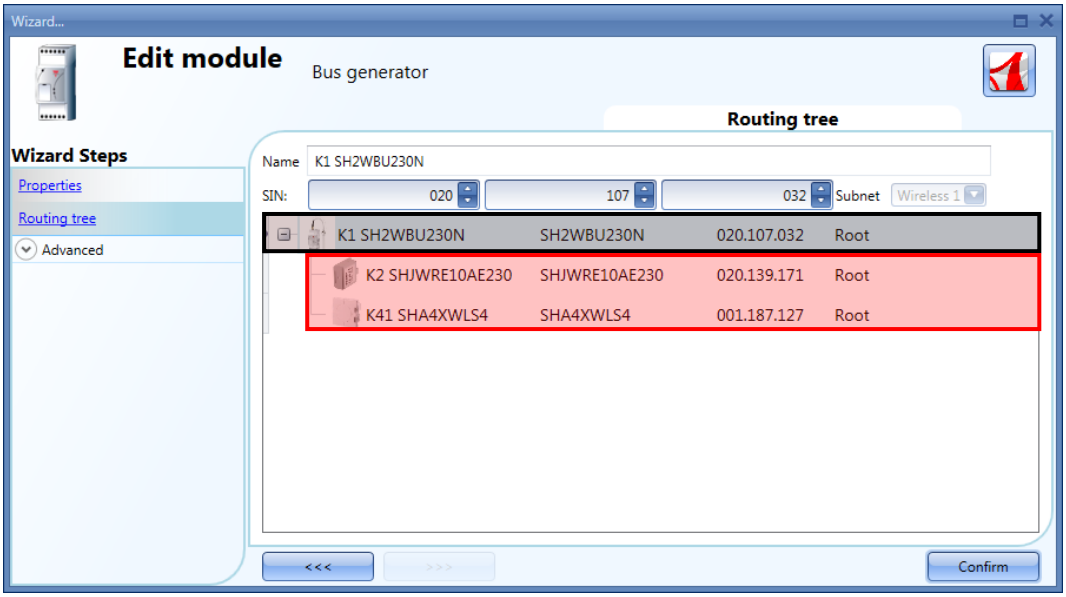


The picture below shows how to manage the *Properties* in the two modules in order to perform the direct connection of the example above (both modules use the wireless base unit as router).





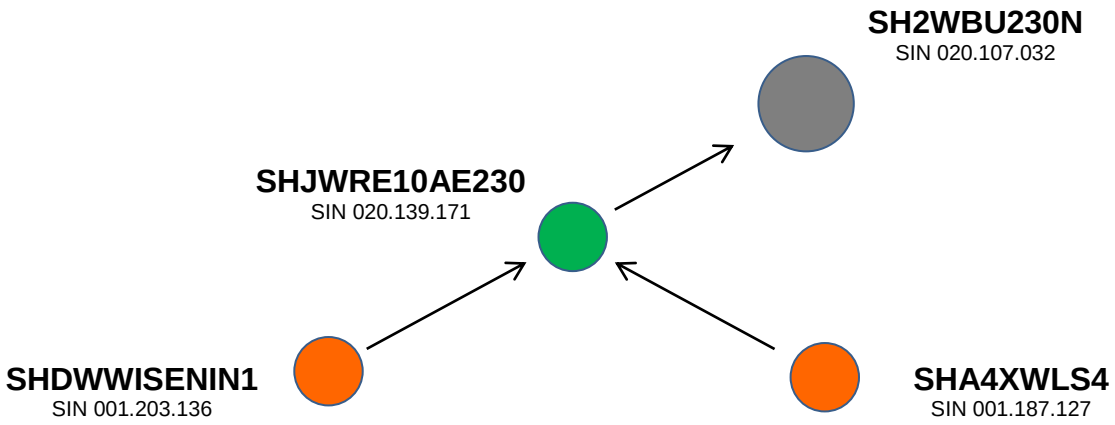
The picture below shows an example of a direct connection of two modules to the wireless base unit SH2WBU230x.



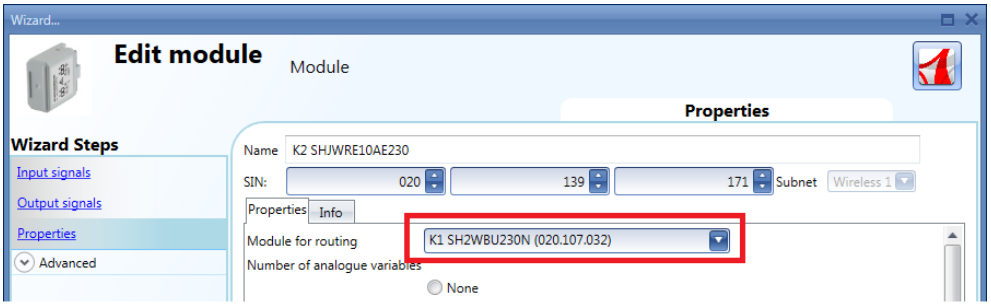
7.14.9 How to indirectly connect a module to the wireless base unit using one level of routing

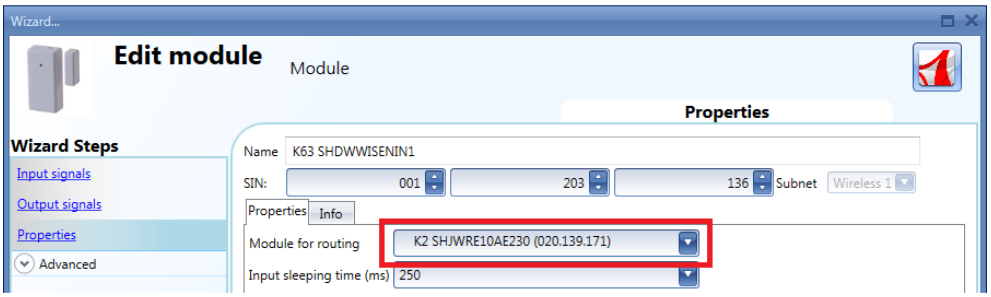
The picture below shows a simple example of an indirect connection of two modules to the wireless base unit SH2WBU230x. One relay module (green dot, SHJWRE16AE230) is used as router and it is directly connected to the wireless base unit. The pushbutton (SHA4XWLS4) and the window sensor (SHDWWISENIN1) are connected indirectly and they use the relay to communicate to the wireless base unit.

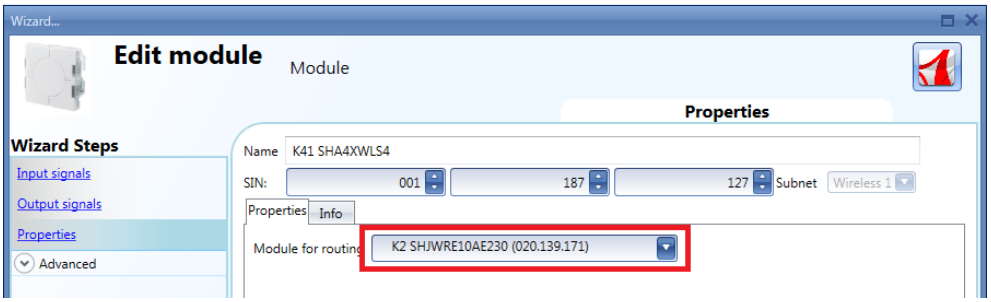
NOTE: The modules that use a router might not be seen by the SH2WBU230 so they will not appear in the orphan network discovery. In this situation they have to be added manually.



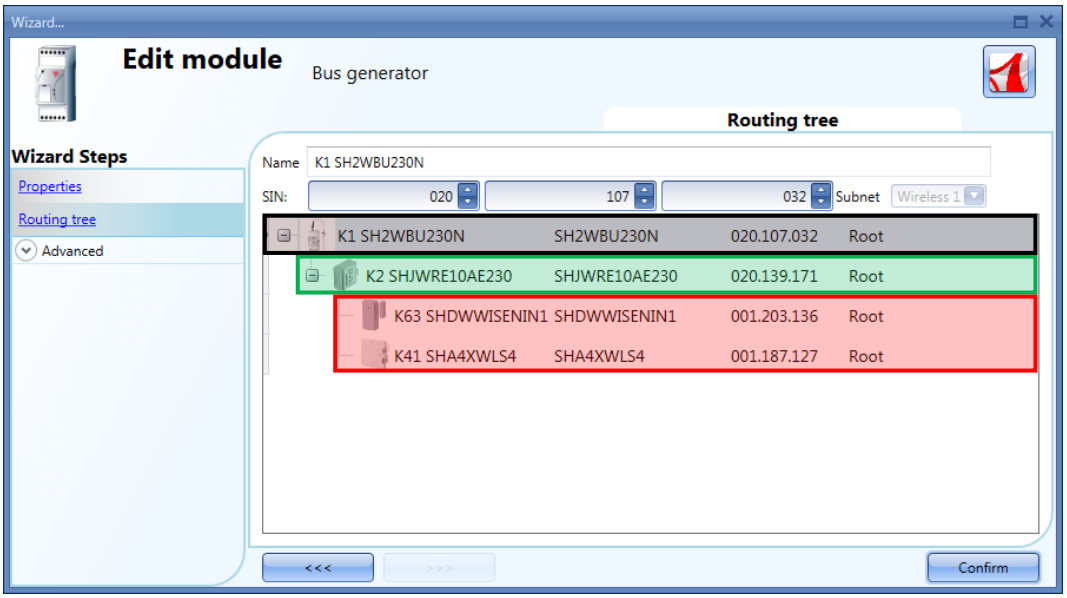
The picture below shows how to set the *Properties* of the two modules in order to perform the indirect connection of the above example, where a relay module is used as router.







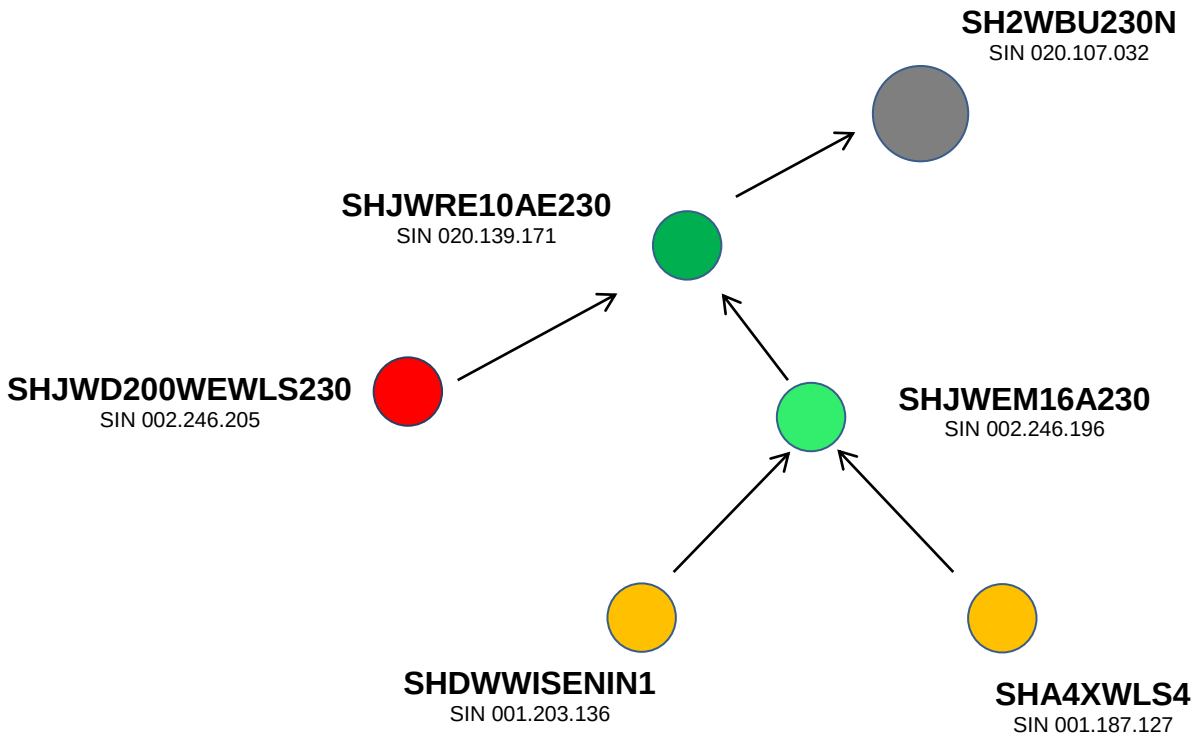
The picture below shows an example of an indirect connection where a wireless relay module is used as router.



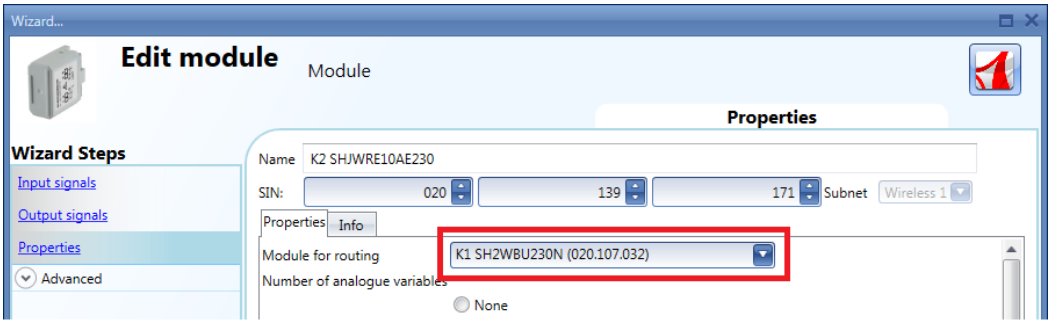
7.14.10 How to indirectly connect a module to the Wireless base using two levels of routing

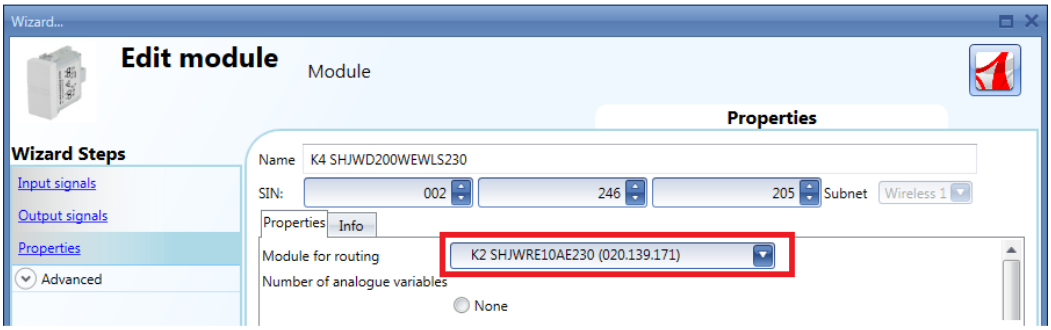
The picture below shows a simple example of an indirect connection of two modules to the wireless base unit SH2WBU230N.
One relay module (green circle, SHJWRE10AE230) is used as a router and it is directly connected to the wireless base unit (grey circle, SH2WBU230N). The wireless dimmer module (red circle, SHJWD200WEWLS230) and the wireless energy meter (light green circle, SHJWEM16A230) are connected indirectly and they use the relay to communicate to the wireless base unit. The wireless energy meter is used as a second router and it is directly connected to the first router. The window sensor module and the pushbutton (orange circles, SHDWWISENIN1 and SHA4XWLS4) are connected indirectly and they use the wireless energy meter as a second router to communicate to the wireless base unit.

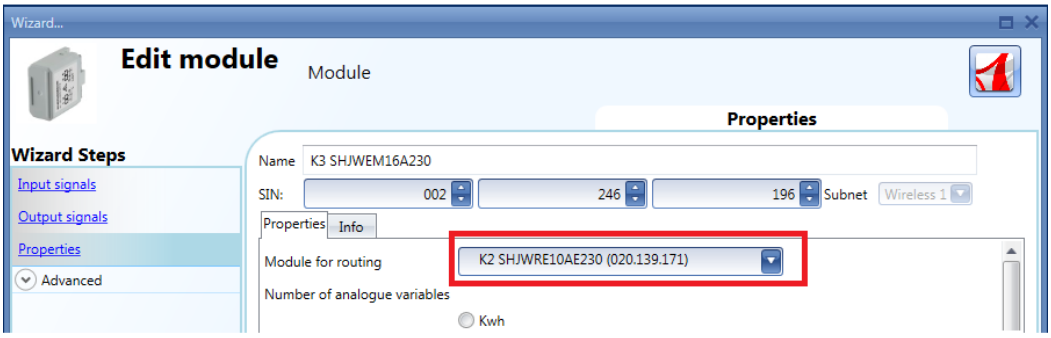
NOTE: Only one level of routing is allowed for the SH2WBU230 base unit. The modules that use a router might not be seen by the SH2WBU230 so they will not appear in the orphan network discovery. In this situation they have to be added manually.

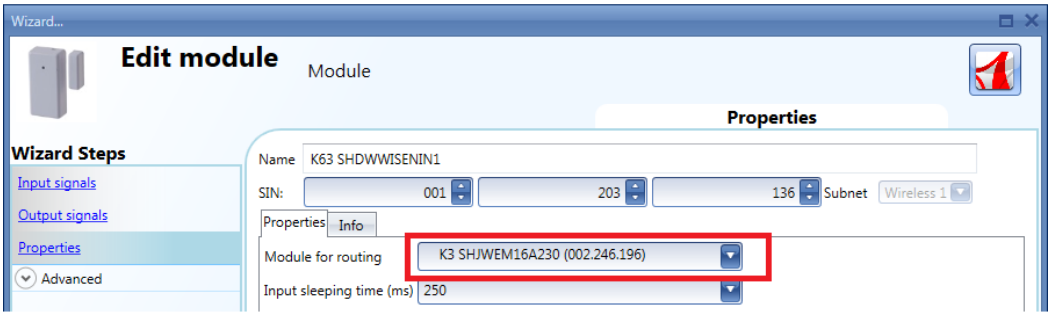


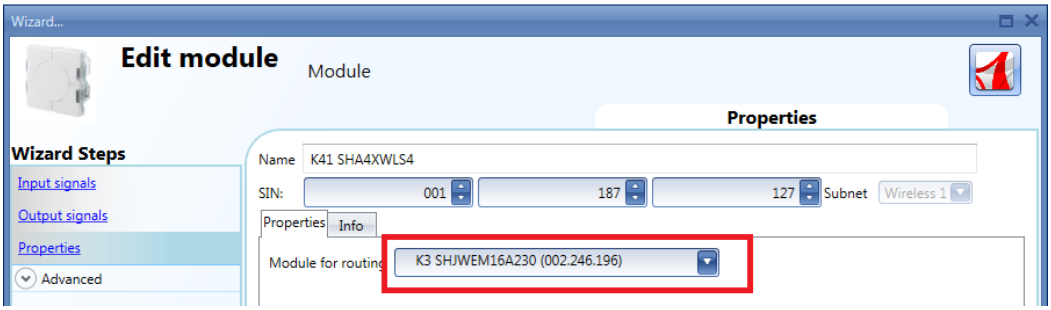
The picture below shows how to set the *Properties* of the modules in order to perform the indirect connection of the above example, where two levels of routing are used.



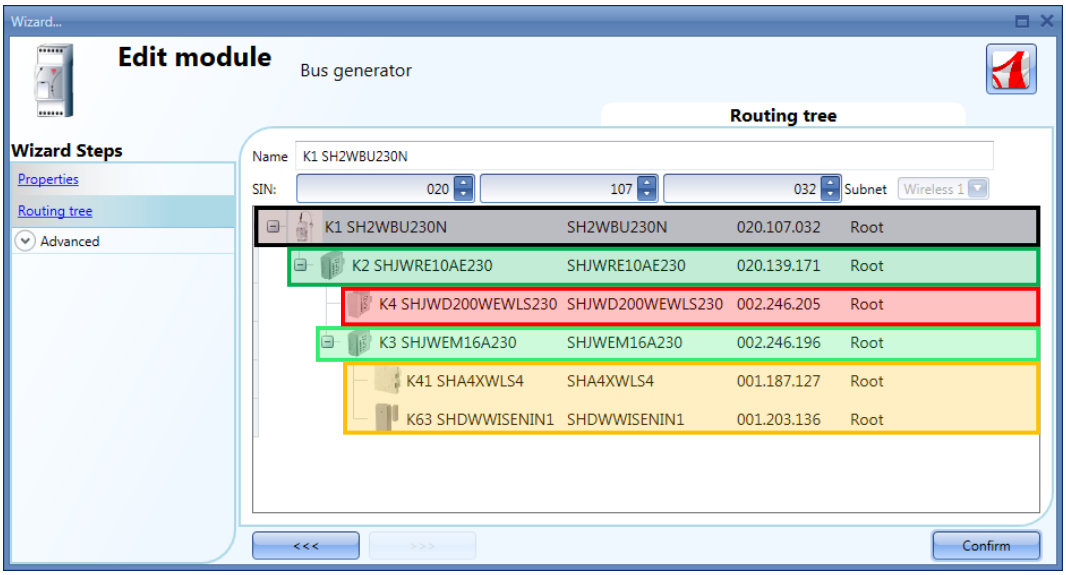






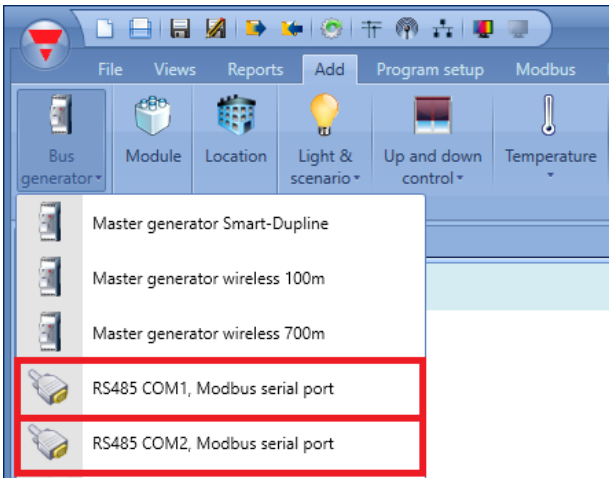


The picture below shows an example of an indirect connection where two levels of routing are used.

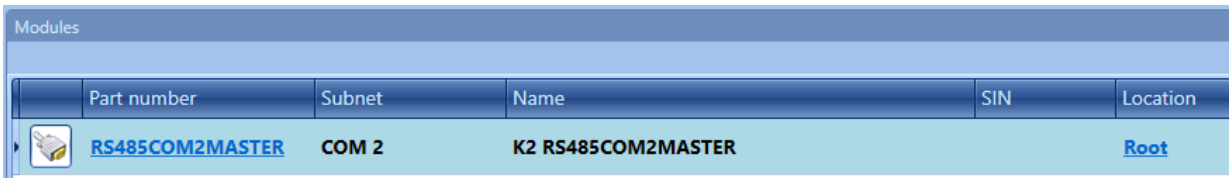


7.15 How to add an energy meter

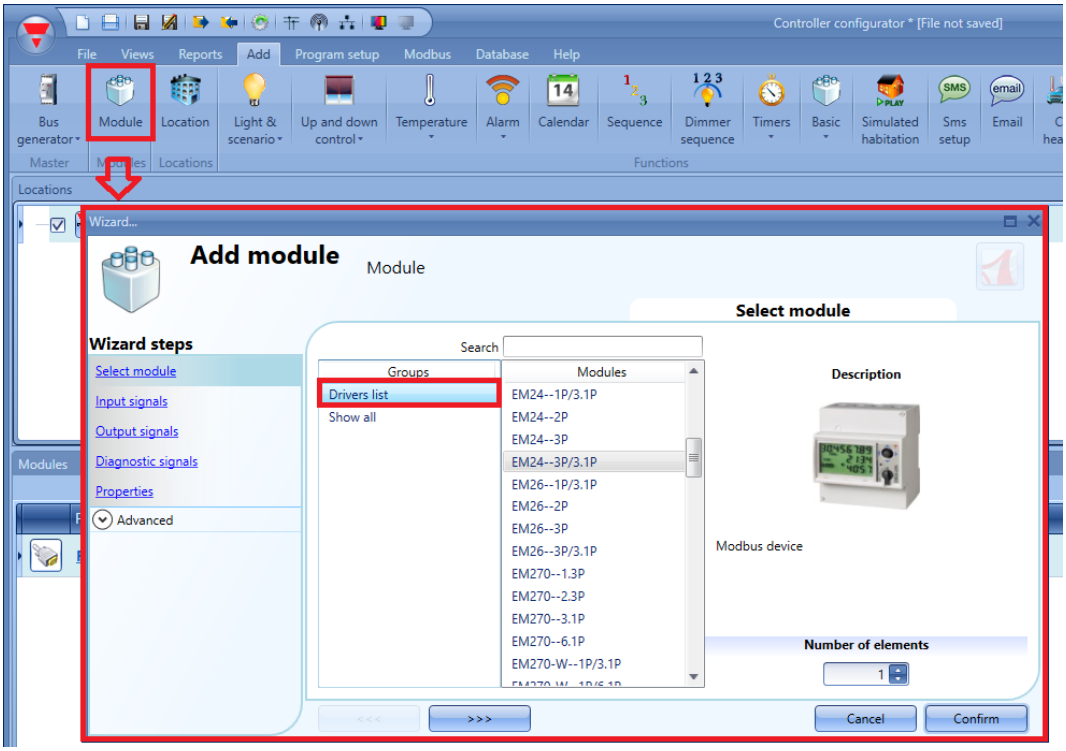
To add an energy meter, the Modbus serial port RS485 COM1 or COM2 has to be added.



The virtual device RS485COMxMASTER is added in the *Modules* window.

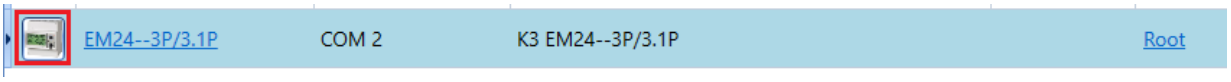


To select the energy meter needed, the user has to click on *Add* and then to *Module.* In the *Add module* window a new line *Drivers list* will appear, as shown in the picture below:

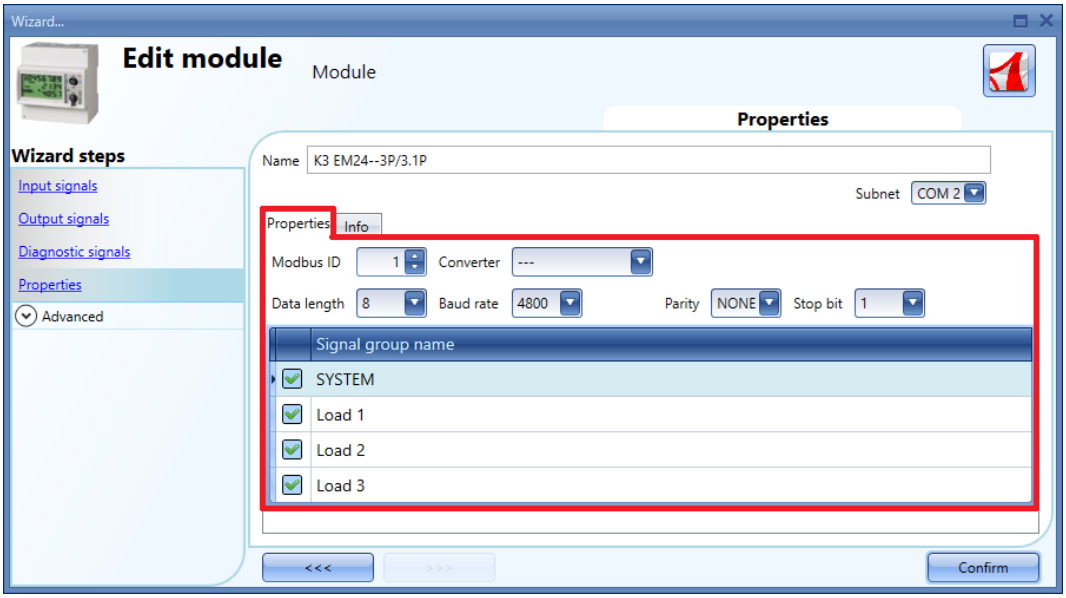


By clicking on this, the list of available energy meters will be shown.

To configure the serial parameters of an energy meter, click on the relevant picture once it is added to the project:



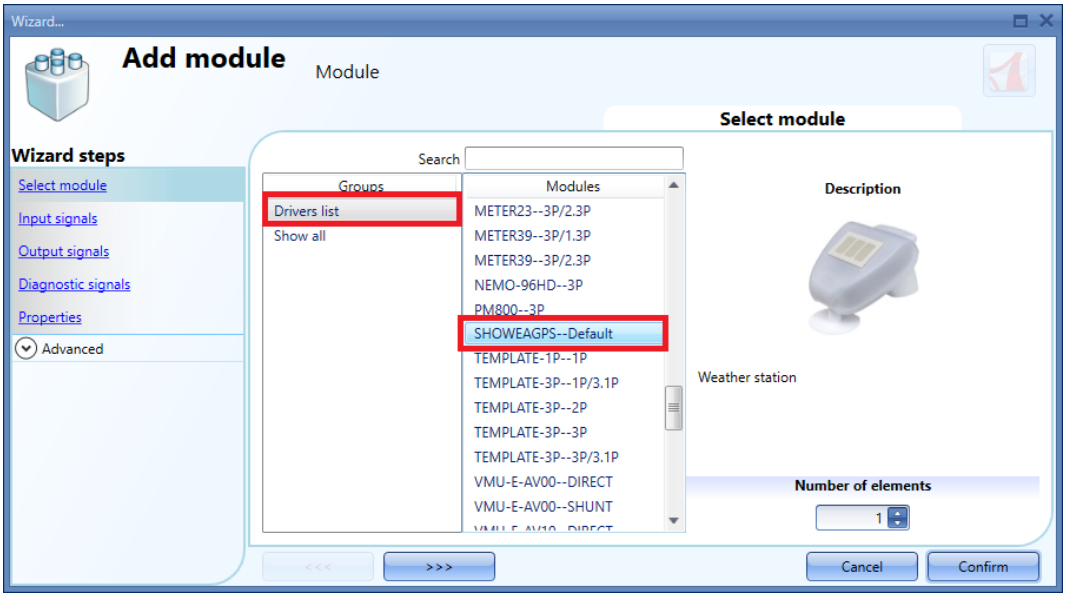
The configuration wizard will appear, where the serial parameters can be configured: Modbus ID, Converter*, Data length, Baud rate, Parity, number of Stop bits.



* Please refer to the *How to configure a Modbus converter device* for additional information.

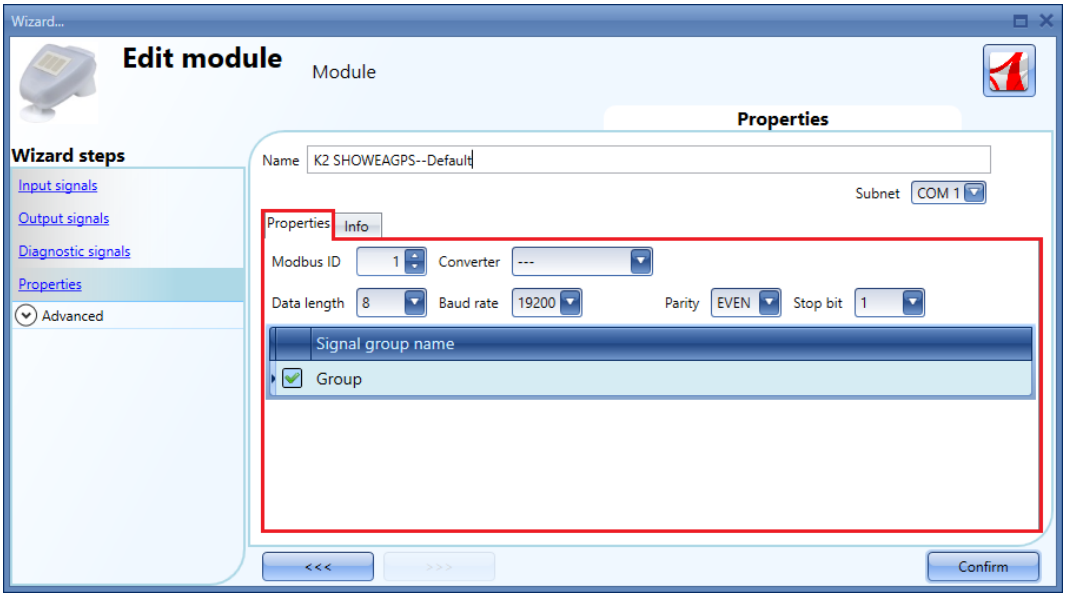
7.16 How to add the weather station

To add the weather station SHOWEAGPS, the user should first add the Modbus serial port RS485 COM1 or COM2, as described in the previous paragraph. In the *Add module* window on the *Drivers list* the user has to select the *SHOWEAGPS-Default*, then click on the *Confirm* button.



To configure the serial parameters of the weather station, click on the relevant picture once it is added to the project:

The configuration wizard will appear where the serial parameters can be configured: Modbus ID, Converter*, Data length, Baud rate, Parity, number of Stop bits.

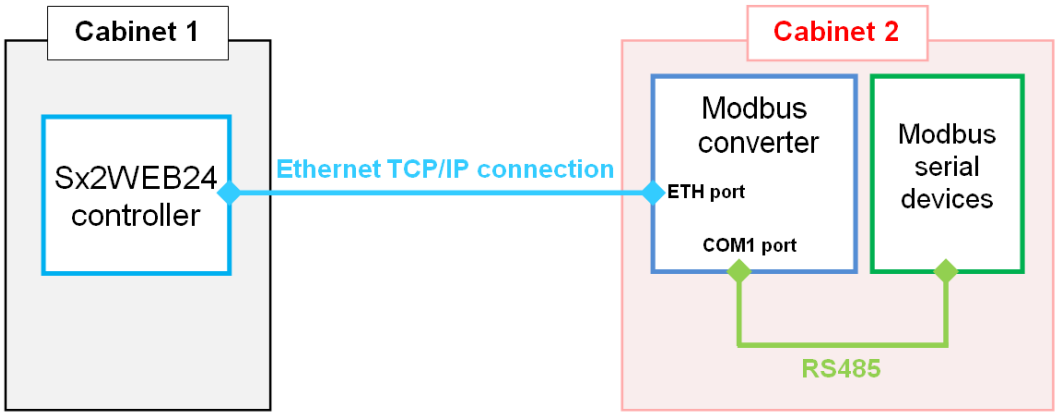


* Please refer to the *How to configure a Modbus converter* device for additional information.

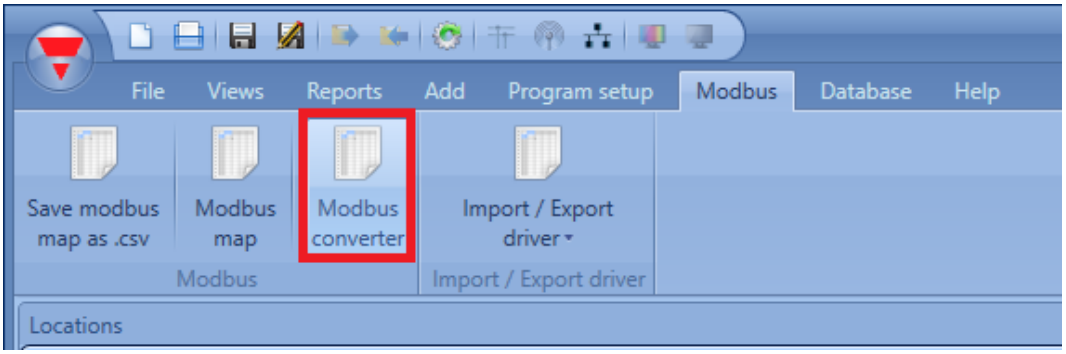
7.17 How to configure a Modbus converter device

The *Modbus converter* can be used to connect Modbus serial devices, such as energy meters, weather station, etc… to a UWP 3.0 controller via IP networks. In a building installation several Modbus serial devices can be located in cabinets that are far away from where the UWP 3.0 is placed. In this scenario a converter can be used as a gateway between the RS485 port and the Ethernet port.

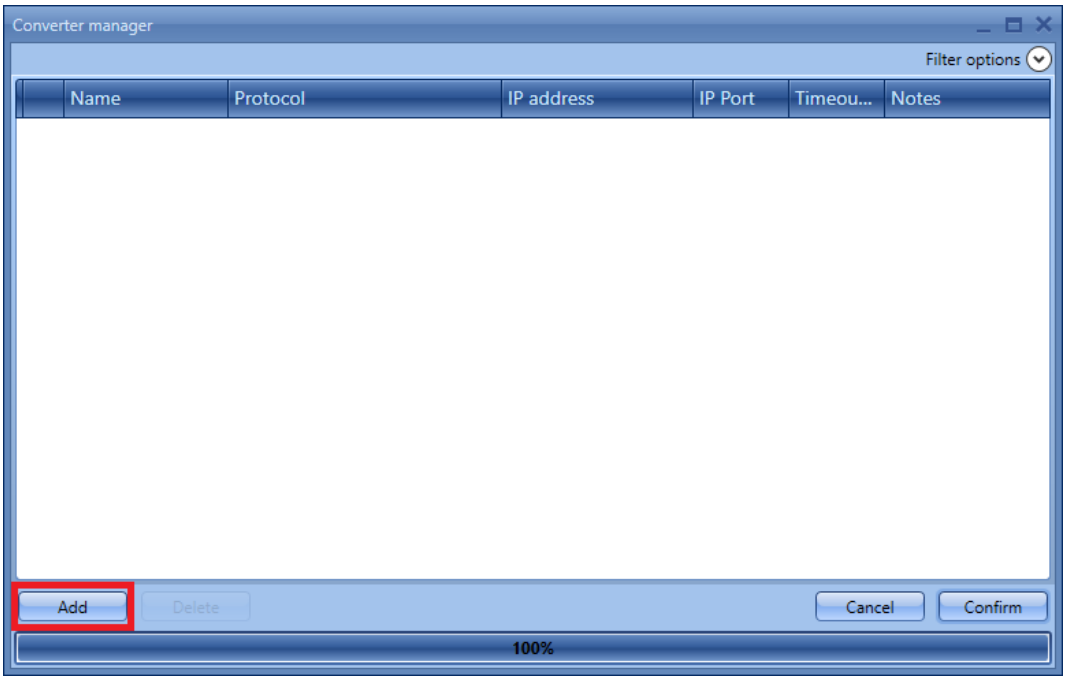
Example
In *Cabinet1* (black square), the installer has placed the UWP 3.0, while in *Cabinet2* (red square), the installer has placed the serial devices connected via RS485 to the Modbus converter. The UWP 3.0 controller and the Modbus converter are connected to the same local IP network.



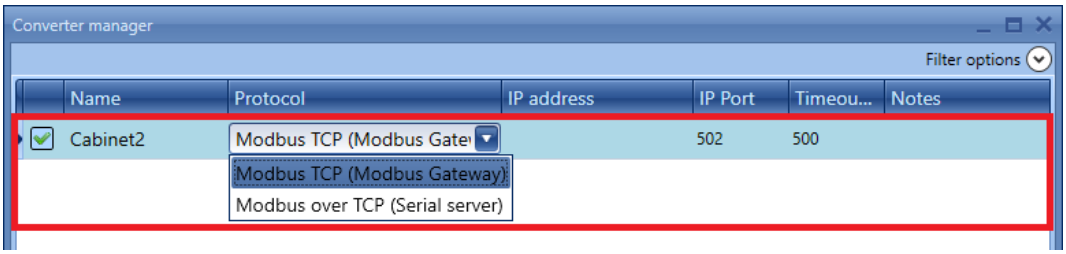
To configure the Modbus converter into the UWP 3.0 tool, the user has to click on *Modbus converter* in the *Modbus* menu, as displayed in the picture below:



The *Converter manager* window will appear, as presented in the picture below:



To configure a Modbus converter device, the user has to click on *Add* and then fill in the necessary settings:



Name: in this field the user has to add the name of the converter that will be used in the UWP 3.0 tool to identify it

Protocol: in this field the user has to select the model of the Modbus converter used in the installation

Please refer to the table below for the device association:

Modbus converter	
Modbus TCP (Modbus Gateway)	CUTCP-3
Modbus over TCP (Serial server)	CUTCP-2

IP address, IP Port and Timeout: in these fields the user has to fill in the IP address, port and timeout of the Modbus converter.
N.B. Please refer to the user manual of the converter for the configuration instructions.

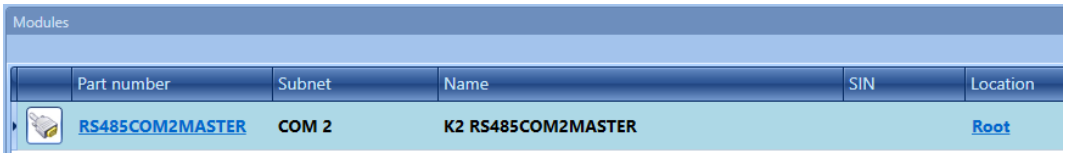
Notes: in this field the user can insert a text information about the Modbus converter, i.e. the place where it is installed.

By clicking on *Confirm*, the Modbus converter configuration will be saved.

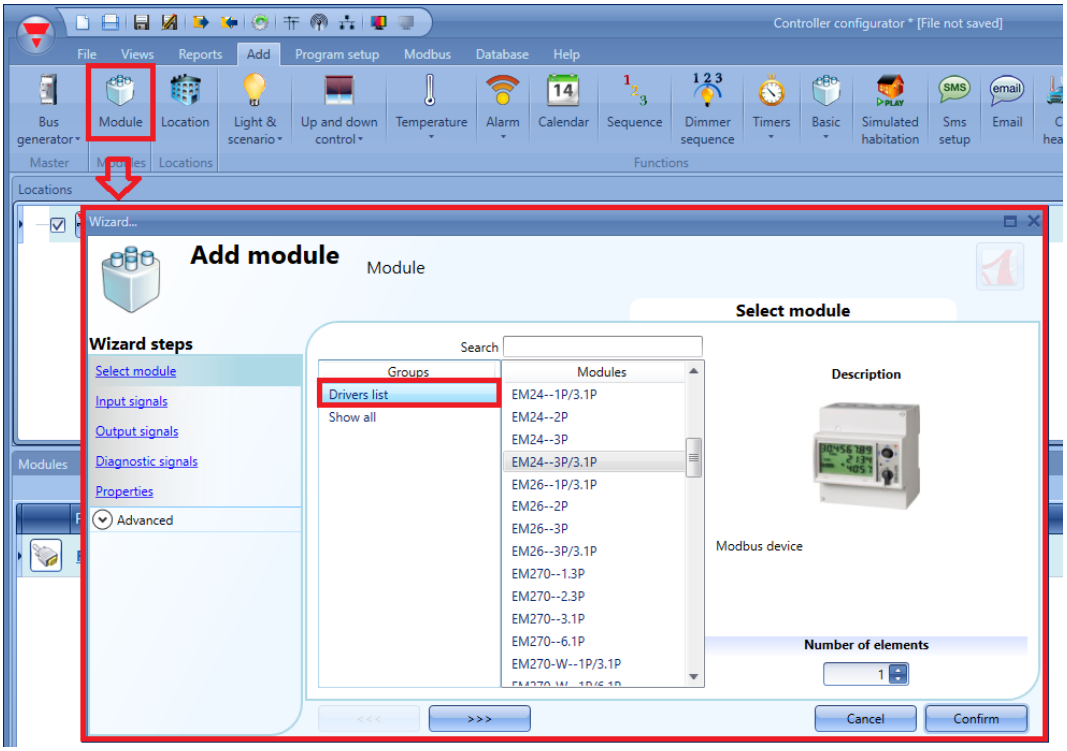
If the user selects one converter and clicks on *Delete*, the selected module will be removed from the *Converter manager* window.

7.17.1 How to configure a serial device connected to a Modbus converter

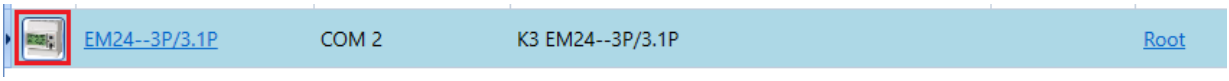
To configure a Modbus serial device, a bus generator RS485COMxMASTER has to be added in the *Modules* window.



Then, the user has to click on *Add* and then on *Module* and select the serial module from the *Driver list* menu, as shown in the picture below:

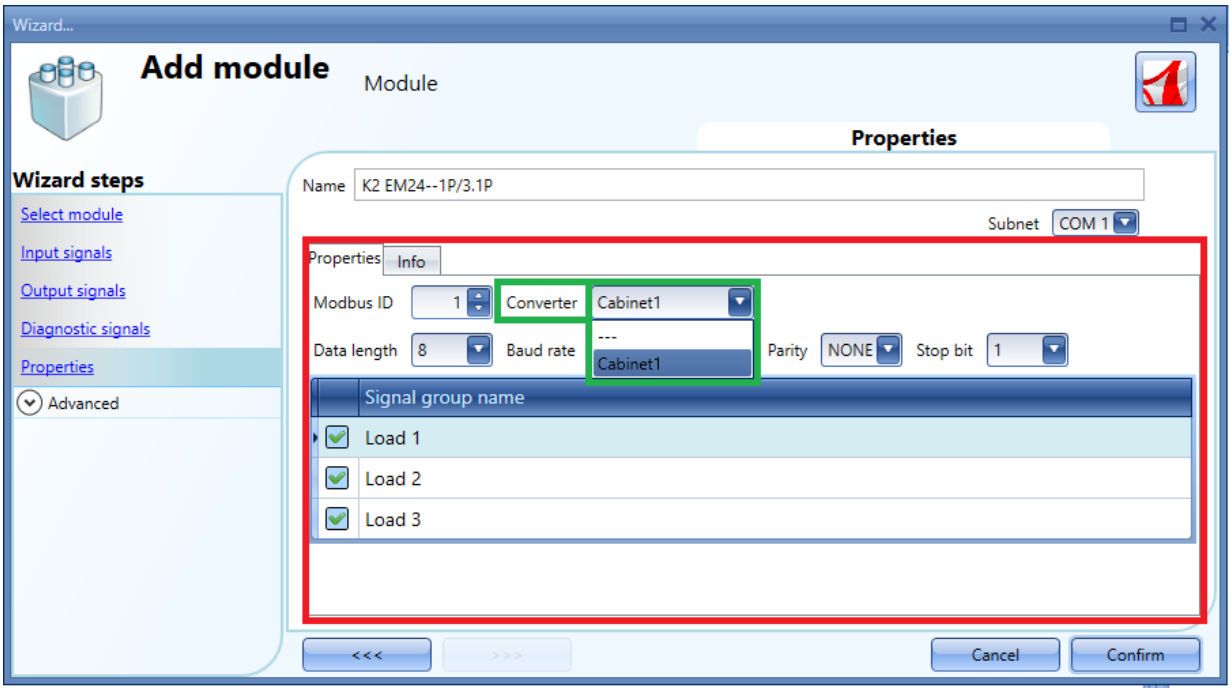


To select the Modbus converter connected to the Modbus serial device, click on the picture of the module:



The configuration wizard will appear, where the serial parameters can be configured: Modbus ID, Data length, Baud rate, Parity, number of Stop bits.

In the *Converter* field of the *Properties* window, the user can choose the Modbus converter, as displayed in the green rectangle in picture below:



8 How to add modules in a project

8.1 How the UWP 3.0 discovers the devices in the network

One of the most innovative features in the smart-house system is that no addressing of modules is needed: the installer only has to mount all the modules, launch the network scan and the system will find and automatically recognise the connected devices without needing to go around the whole installation to make association or to address.

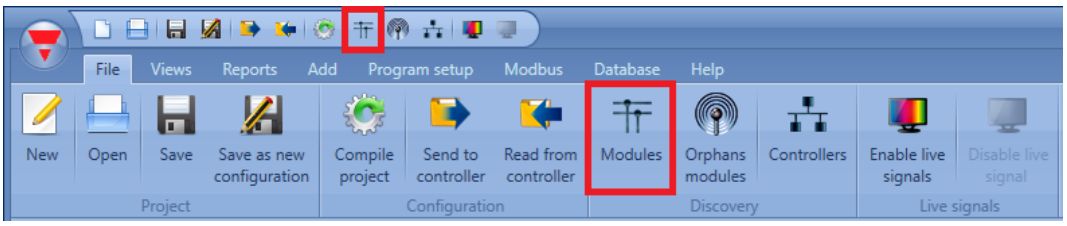
Three different approaches can be used to add the modules to the project:

- The first is to connect to the UWP 3.0, launch the network scan to find all the modules and then create the configuration, adding the modules in the relevant functions and locations.
- The second is to add them manually and also fill in the SIN manually.
- The third is to manually insert all the modules in the project with SIN 000.000.000, then create the project by linking the modules to the functions and locations. Once the project is finished, connect to the UWP 3.0 controller, launch a network analysis and then associate all the modules.

8.2 Global discovery of the network

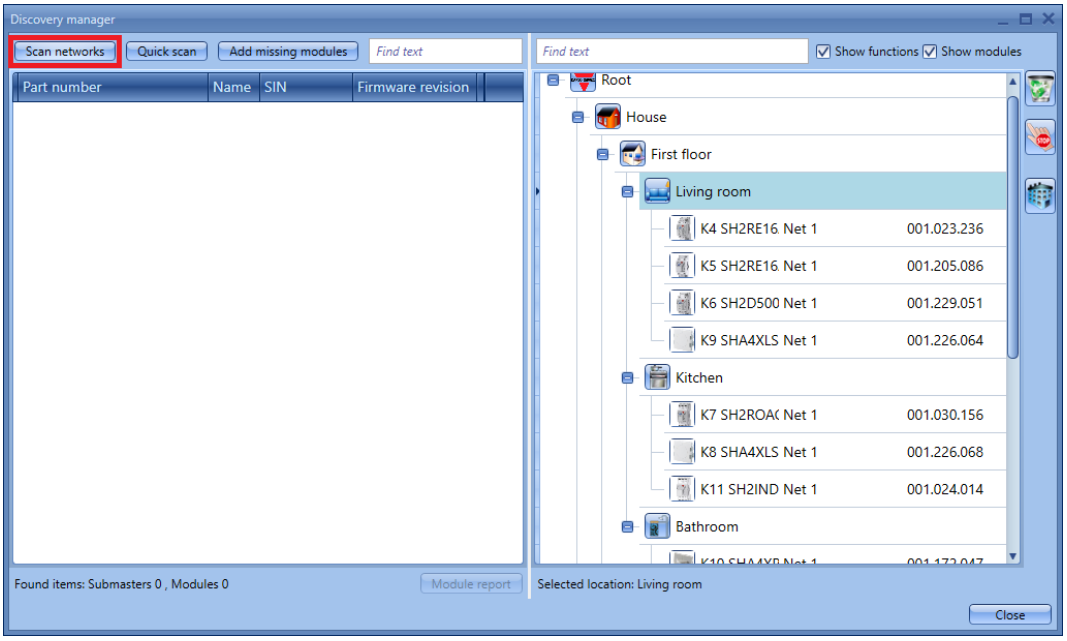
The global discovery of the network is used to find all the modules connected to the Dupline bus. Each time a new module is added to the bus, the global scan must be used to find the new module.

The user can proceed to add the modules: there are two ways of doing this, an automatic scan of the network launched by the *Discovery manager* window, and a manual mode. To launch the global scan of the network, the user should click on the icon at the top (see picture below).

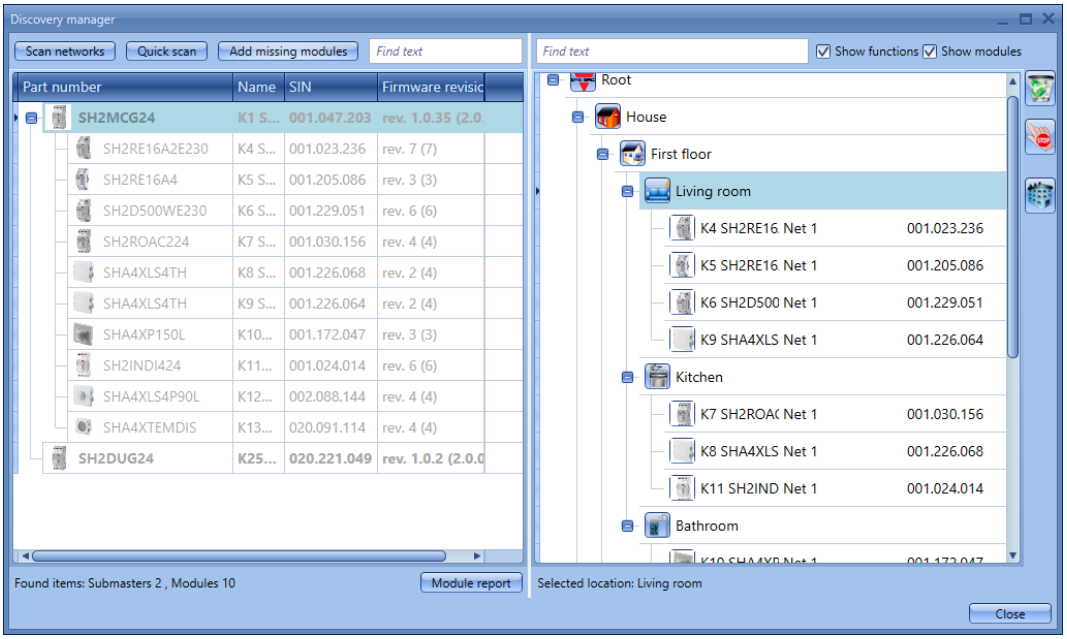


8.3 How to automatically find the modules – the controller is connected to the modules

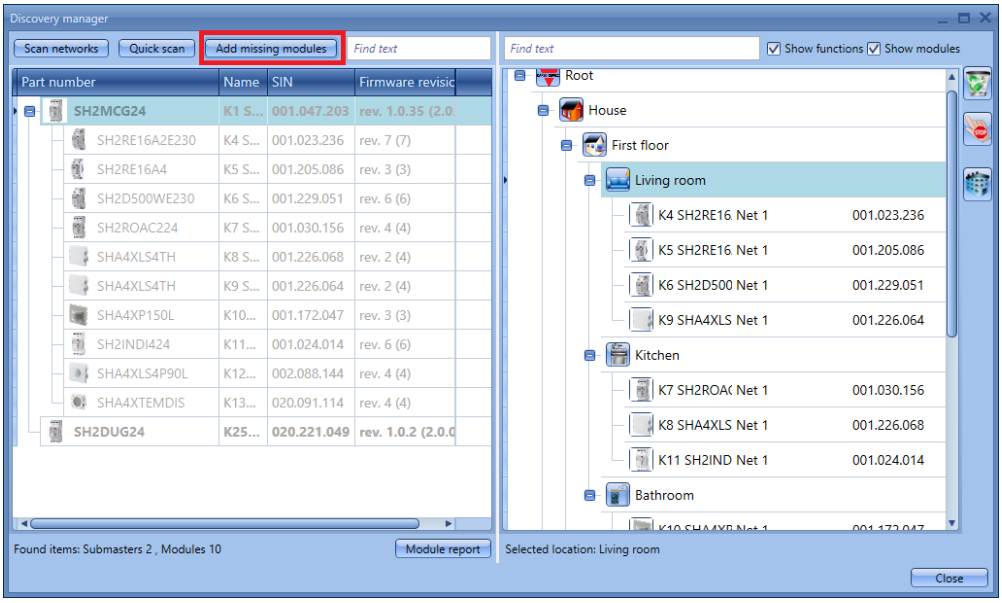
In the *Discovery manager* window, click on *Scan networks*: all the modules present on the bus will be prompted.



Once the scan is finished, the UWP 3.0 tool will present the list of devices found, grouped according to the Dupline network they are connected to. This means that the system also shows which Dupline generator SH2MCG24 each module is connected to.



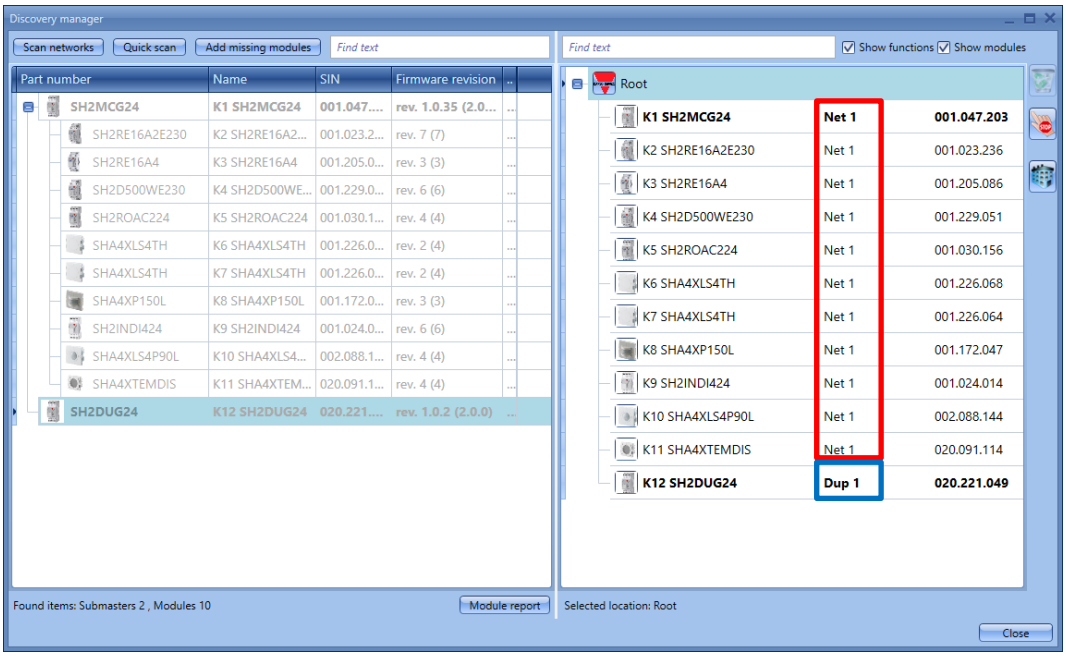
The user can decide to add them manually with just one click on the *Add missing modules* button at the top of the window.



8.4 How to add the modules manually

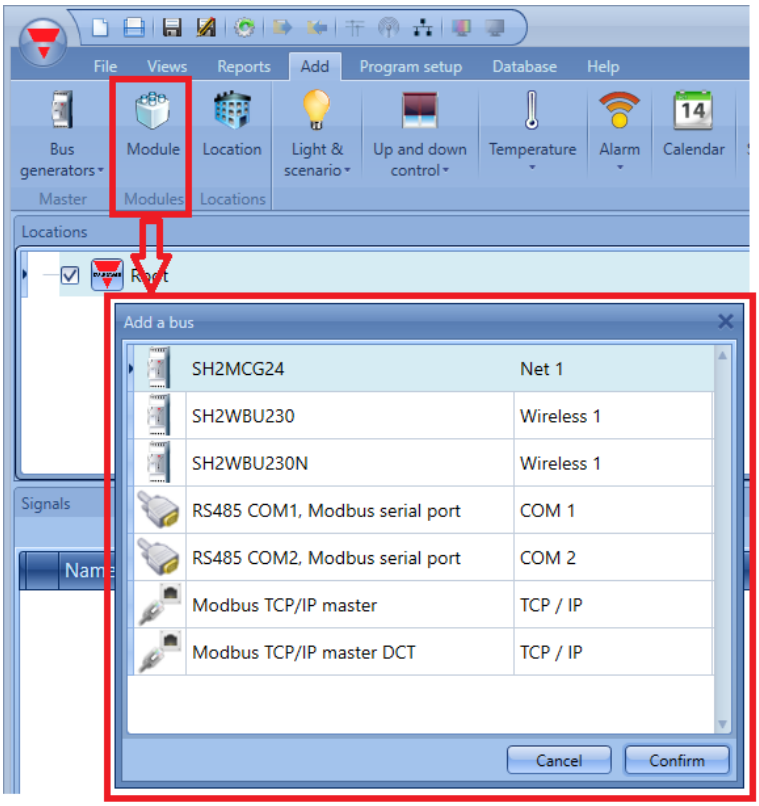
If the user wants to add the modules manually, the first thing to do is to add the Dupline generator SH2MCG24: when presenting the devices found, the Dupline sub-networks are identified as “undefined” since their number will depend on how they are added in the project, so that the user can decide the order.

The first SH2MCG24 added will be assigned to the sub-network 1 (Net 1), the second one to sub-network 2 (Net 2) and so on until all the Dupline generators are added.
The sub-networks will automatically be assigned to the slave Dupline devices. (see picture below)

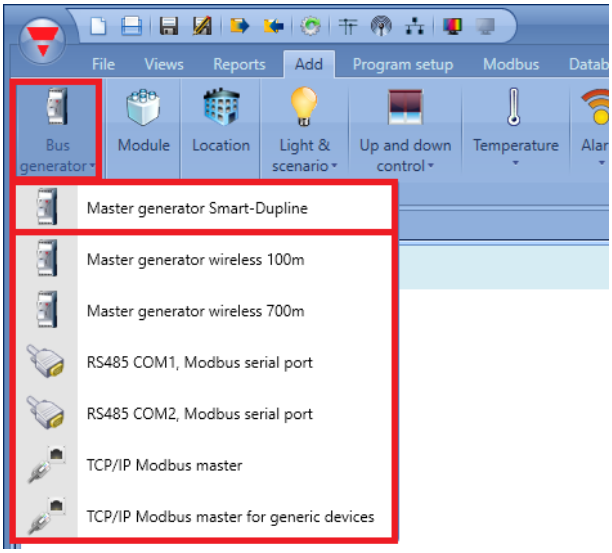


8.5 How to add the modules offline

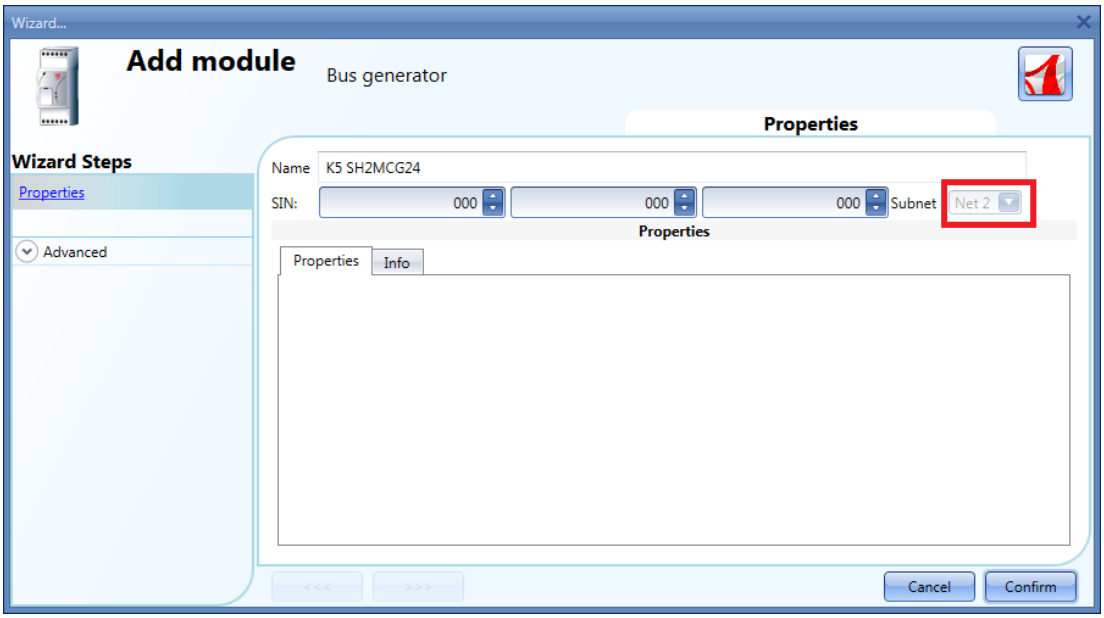
When the user creates the project without using the automatic discovery, the first thing to do is to add and configure the SH2MCG24 Dupline generators and then add all the other modules, selecting the sub-network they belong to. In the figures below, how to add a new bus generator is shown.



Each added SH2MCG24 will be assigned to a **new sub-network,** up to 7 Master generators/wireless Master generators.
To add a master generator the user should select *Bus generator* from the Add menu, then select *Master generator* (see picture below). The new module will be added into the selected location.

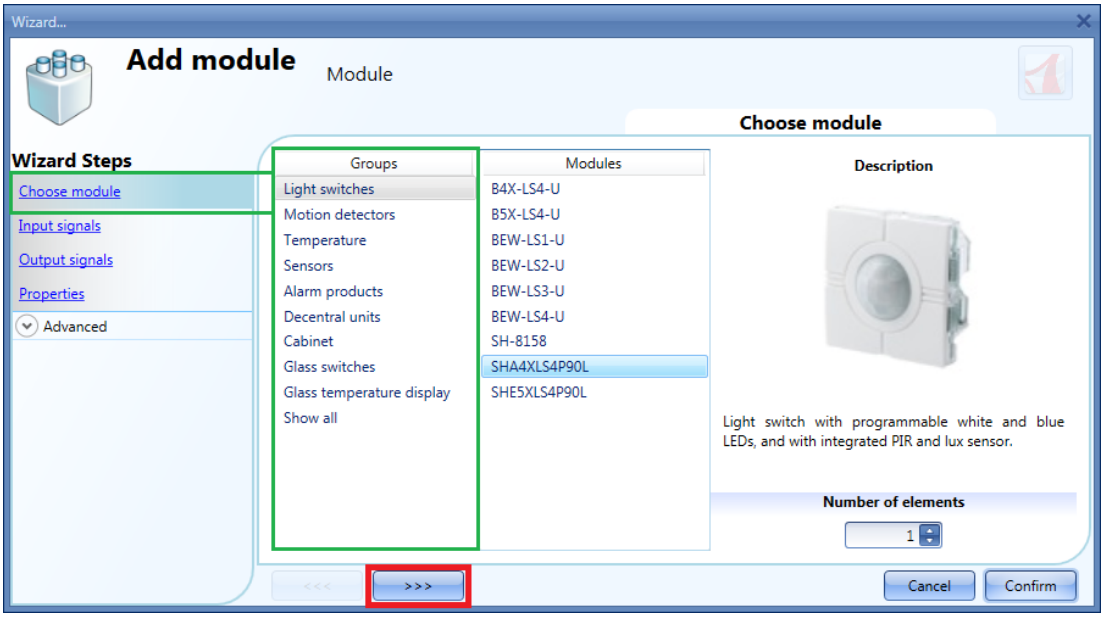


The wizard relevant to a Master generator can also be opened by typing Alt+F6. (See table of short cuts).

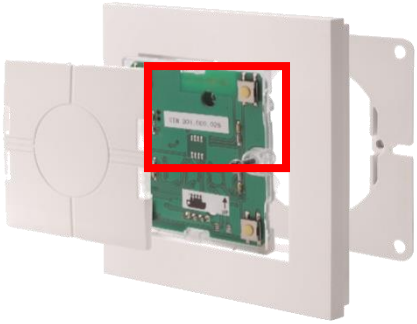


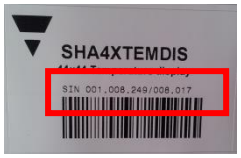
In the picture above, a new module is inserted with its associated new network (e.g. Net 2)
As shown, each new generator will be assigned to the next free network (e.g. Net 2)

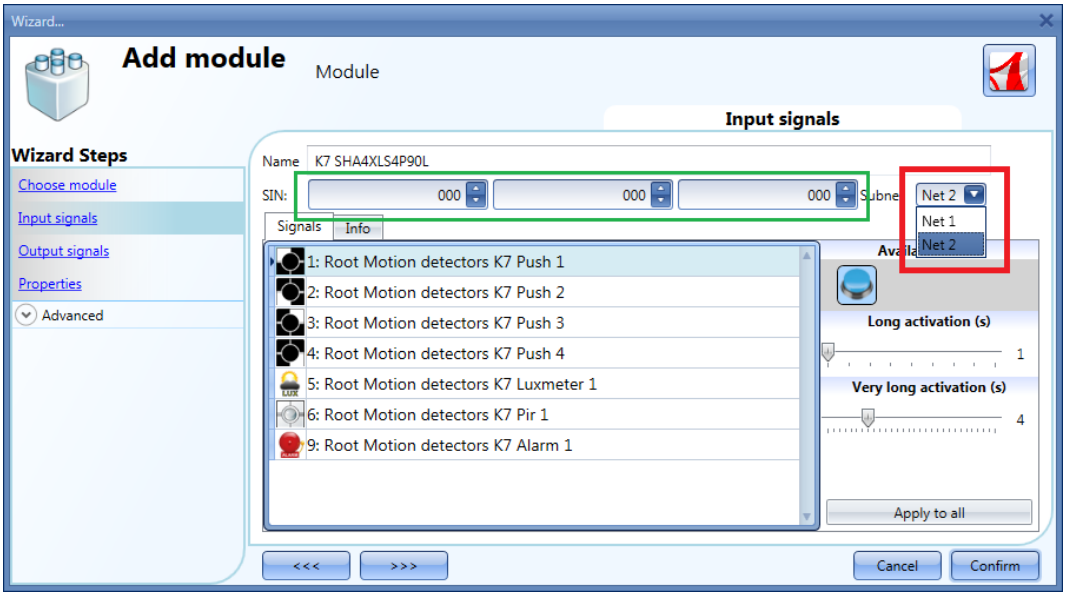
If the project has more than one SH2MCG24 Dupline generator, the user must be careful to assign the correct network to each Dupline module according to the network where it is physically placed: to do this press the *triple arrow* button (see picture below) highlighted with a red rectangle. If *Confirm* is pressed directly, the module is added to the first free network.



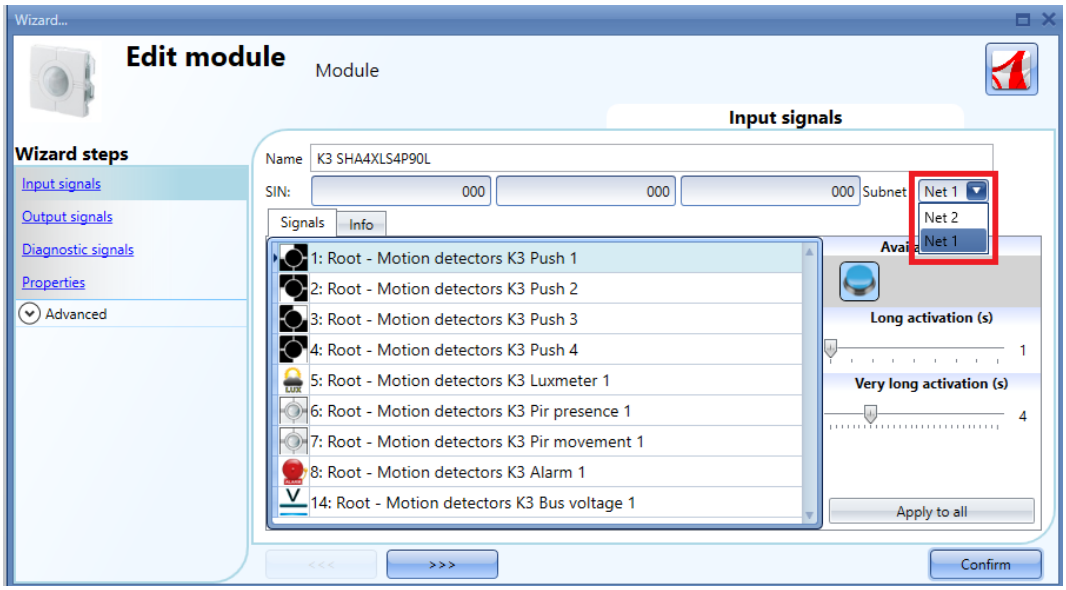
All the modules are added to the project with SIN 000.000.000, which can be set manually: this means that the installer can fill in the SIN by hand in the fields marked with a green rectangle. The SIN can be found on the module, on the box and on the label delivered with each device, it could also be attached to the manual or in the location where the module is placed.







If a module is entered with a wrong Subnet, e.g. *Net1* instead of *Net2*, the user can click on the relevant picture from the Modules window, then the Subnet can be changed from the *Subnet* field of the *Edit module* window, as displayed in the red rectangle below.

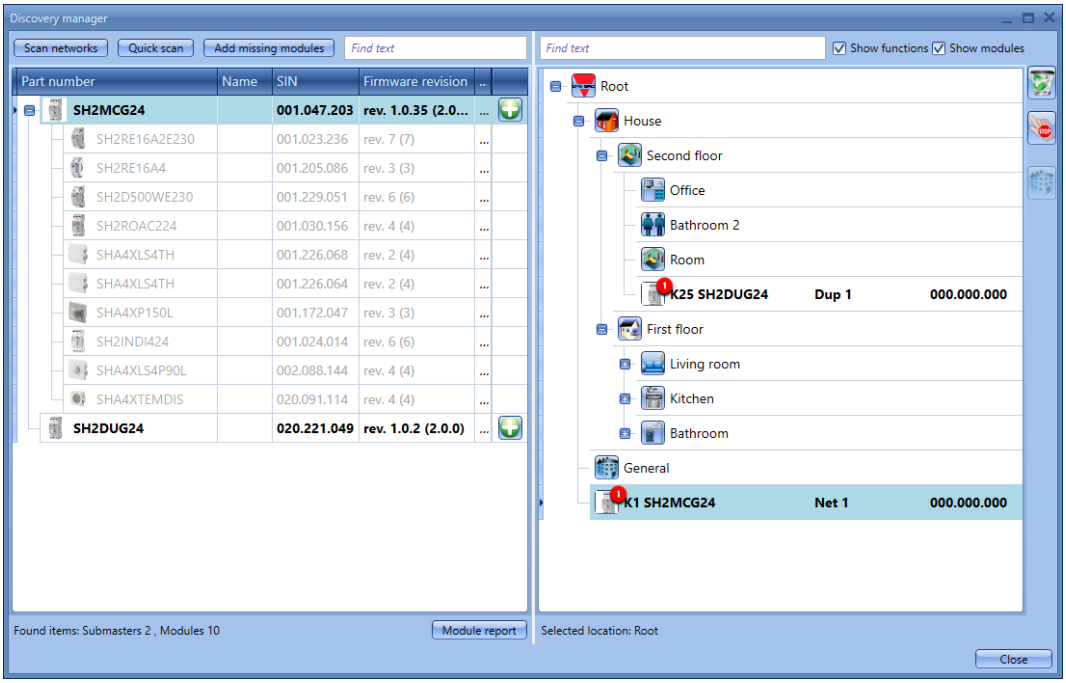


8.6 How to add the modules manually and then find them in the network

The user can prepare the configuration in the office without doing a network scan, and can then associate the right SIN once he is at the installation site. Once the configuration is ready with all the required modules, there are two ways to proceed:

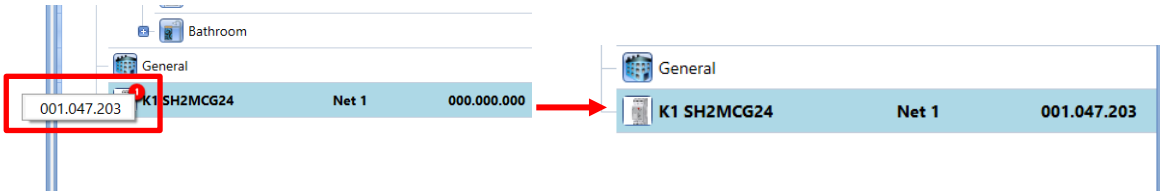
1) The user can launch a network scan by clicking on the *Scan networks* button.

All the modules available will be presented in the list on the left side of the *Discovery manager* window, as shown below:

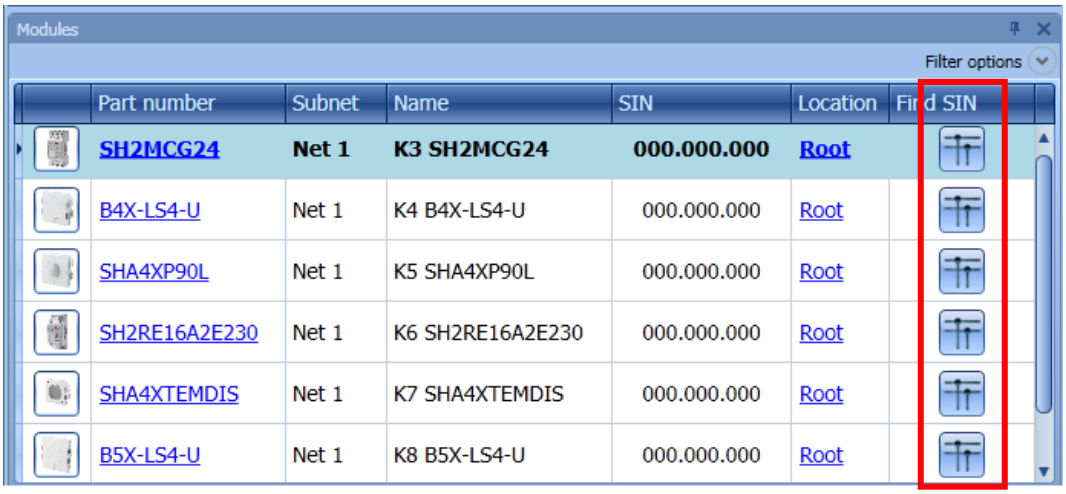


For each Master-generator, such as the SH2MCG24, SH2DUG24 or SH2WBU230N that have been found, the system shows a small red icon (1) over a module where an association can be made.

By clicking on the red icon with the right mouse button, the module will be added to the project, as shown below:

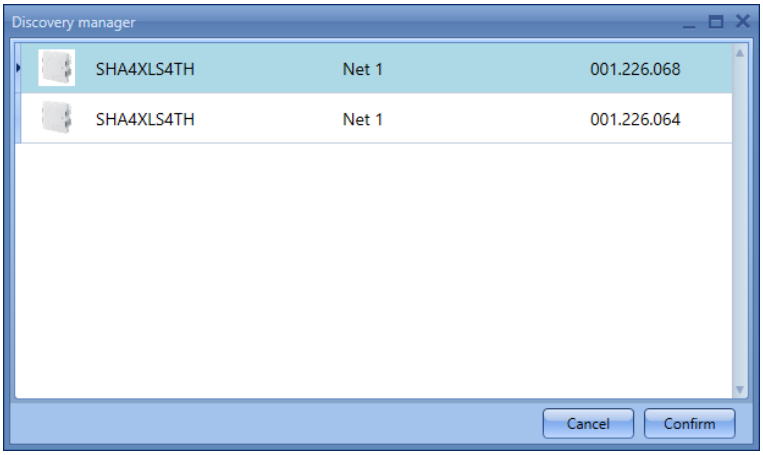


2) From the *Modules* window, clicking on the icons marked with the red rectangle as shown below:



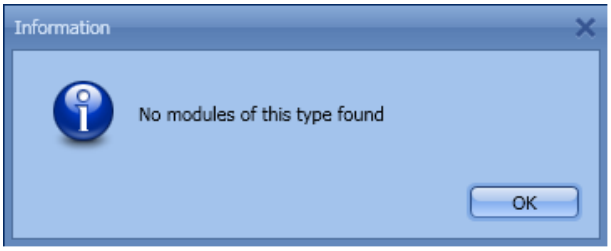
If the association is carried out from the *Modules* window, when clicking on , only the modules of the same type as the selected item are shown in order to simplify the association.

For example, by clicking on the network scan icon of the SHA4XLS4TH, the following window appears only with the SHA4XLS4TH found.



By clicking on the required module, the association is carried out.

If the user tries to match a module that has been added in the project by selecting a wrong network, the UWP 3.0 tool will show a message indicating that the module has not be found (see picture below).



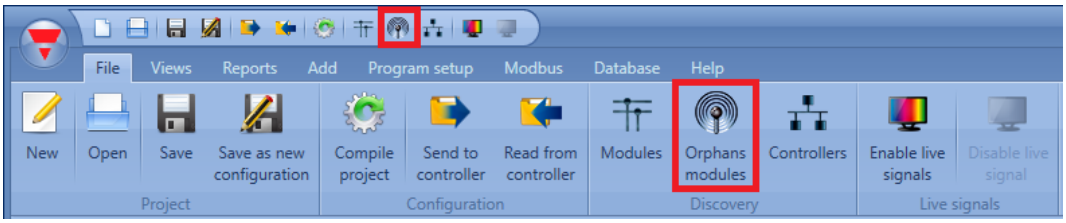
9 How to add wireless modules in the project

9.1 Global discovery of wireless modules

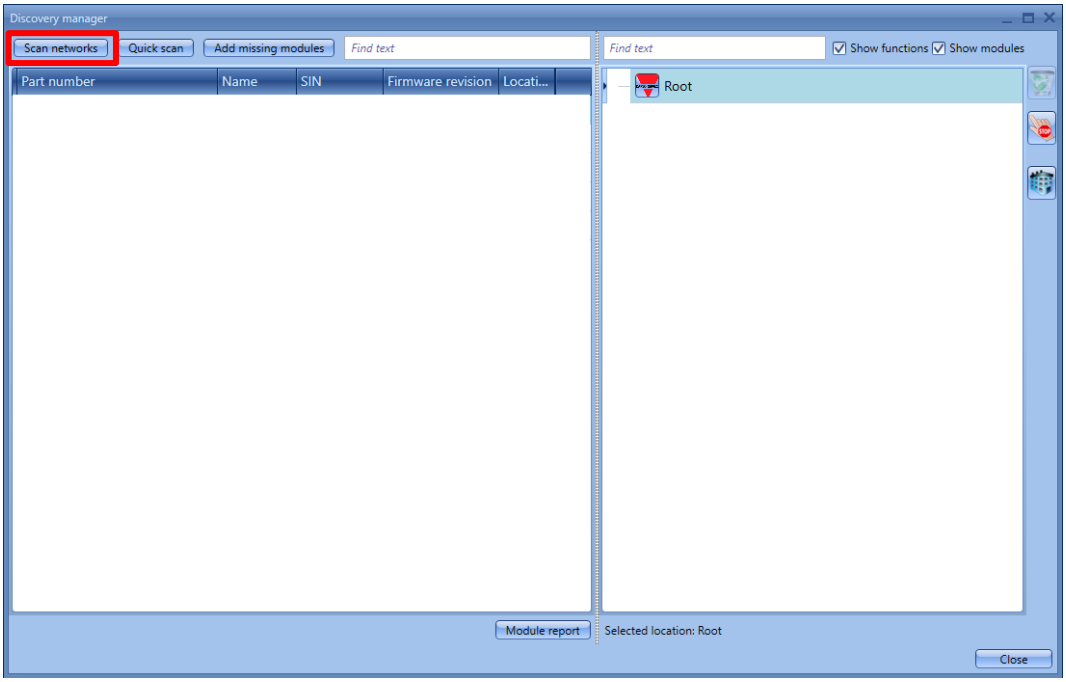
The global discovery finds all the wireless modules present in the wireless field covered by a SH2WBU230N base unit: it automatically shows only the “orphan” modules (an “orphan” module is a module that has never been programmed before as it is only just out of the factory).
Once a module is used in an installation (this means that it has been programmed at least once), it is no longer possible to find it using the discovery: if the user needs to add it in another system, it has to be done manually.

To launch the wireless global scan of the network, the user has to click on the icon at the top (see below). To save the battery, the wireless modules automatically send an acknowledgement every three minutes, so it might happen that if a wireless discovery is launched before an acknowledgement is received, the relevant module is not seen. The user can manually make a module send the acknowledgement in the following ways:

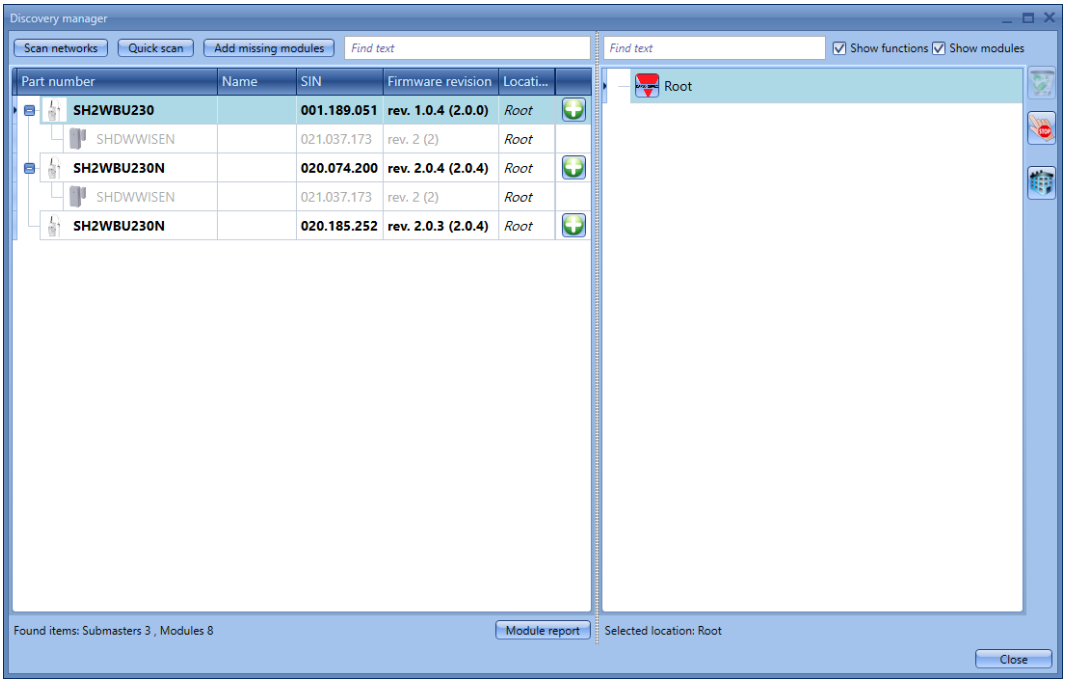
- By pressing a button in a light switch
- By switching off and then on again a wireless relay module



In the *Discovery manager* window the user has to click the *Scan networks* button to start the discovery.

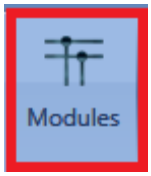


When the scan of the network is finished, a window will appear showing all the modules found.
The user can decide to add them manually, using the + green icon at the right side of each device, or add all the modules with just one click on the *Add missing modules* button at the top of the window.



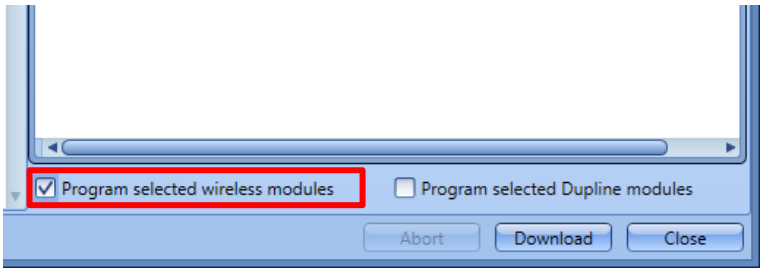
If more wireless base units are installed, the UWP 3.0 tool will show all the modules found in each network. If two or more SH2WBU230Ns cover the same area, they might find the same modules: it will then be up to the installer to associate the “orphan” modules to the required SH2WBU230N.

As already explained, once the wireless modules are programmed they can no longer be considered “orphans”, so they are not found by clicking on the icon *Orphans modules.*



They are shown together with the associated SH2WBU230N when a global network scan is launched by clicking on the icon *Modules*.

To program the wireless modules, the configuration has to be sent to the master unit UWP 3.0 by selecting the option *Program selected wireless modules*.



If a wireless module is no longer present, the system needs three minutes to recognise this (this is the time that has to expire between two consecutive acknowledgments).

9.2 How to add the wireless modules manually

The user can manually add the wireless modules in the same way as described for the wired modules. Should the user create the project without using the automatic discovery, the first thing he has to do is to add and configure the SH2WBU230N base unit and then add all the other modules, selecting the wireless-network they belong to.

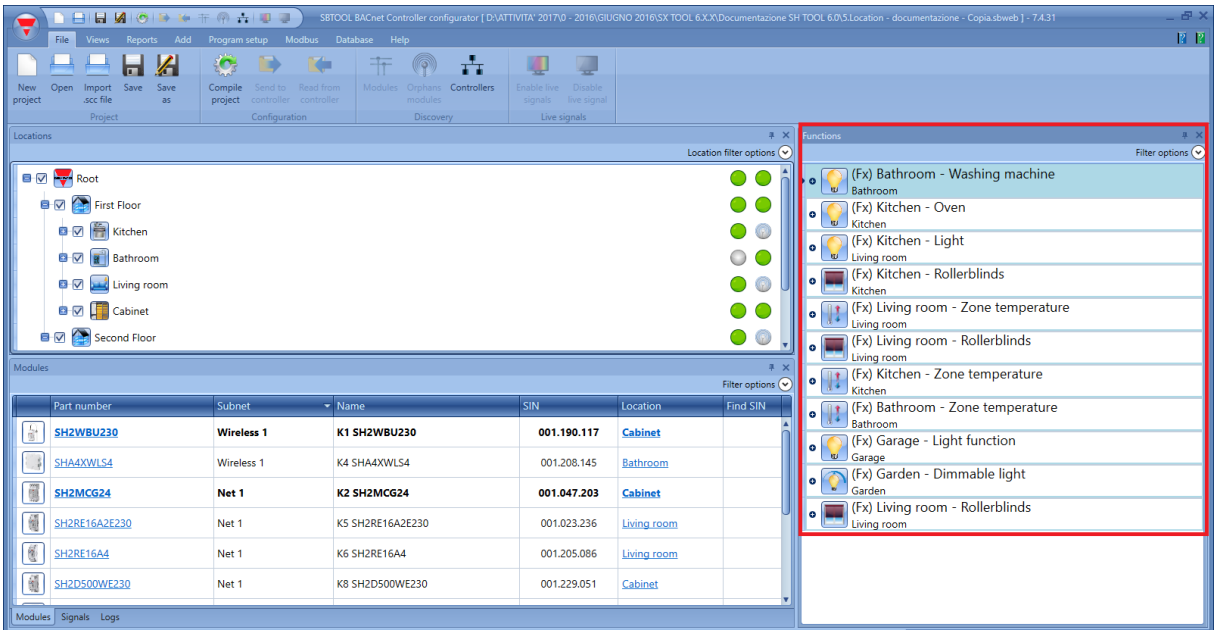
If the project has more than one wireless generator SH2WBU230N, the user must be careful to assign the correct network to each module.

10 Functions

A function is defined as a series of instructions that, given some inputs, generates one or more outputs. Every predefined function available in UWP 3.0 is seen as a set of steps, some of which are mandatory, others optional, defining the behaviour of the function itself.
Some objects are “public” and, being defined as a function, they can be used in other functions as well. Each function automatically manages a status signal, which can be used by other functions.

Functions are divided into basic and predefined ones.
Basic functions are all the simple functions that the user can link together to create more complex functions, or that can be used to customize some of the predefined functions (e.g. two light functions are an input of one OR function).
The predefined functions are used to manage a whole series of automations, from lights to roller blinds.

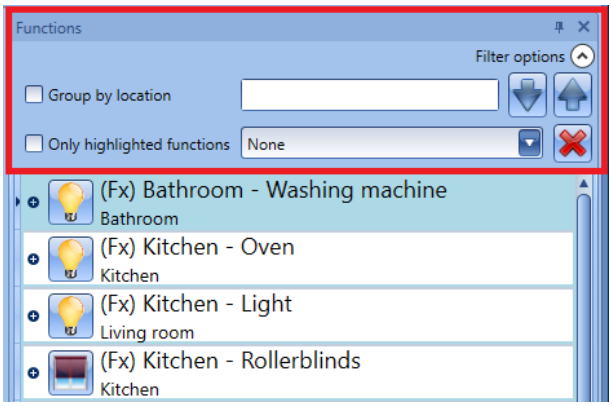
The ***Functions*** window on right side of the UWP 3.0 Tool, as shown in the red rectangle in the picture below, contains the list of the functions created for the project.



When a new function is created by the user, it will be added into this area; to edit a function already made, the user has to click on the picture of the function, the wizard will be opened.

10.1 How to manage the filters on the Functions window

The functions can be sorted or filtered by *Filter options*, click on the icon, the filter options panel will be opened: only the functions specified by the filters will be displayed.



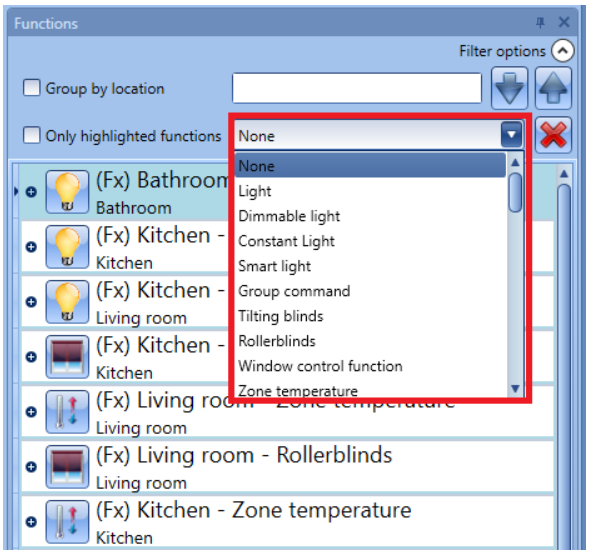
Group by location:
The functions are grouped by the locations defined in the project.

Search box:
The results will include all the words that contain the input string and the search will be carried out on the functions available in the project. The user can insert the full name or part of it.

Up and Down arrows:
Once a function is selected, by clicking on the arrows it can be moved up or down.

Only highlighted functions:
When this filter option is checked, only the function that are highlighted (according to the *highlight* functionality available on *Locations/Modules* window) are presented in the *Functions* window.

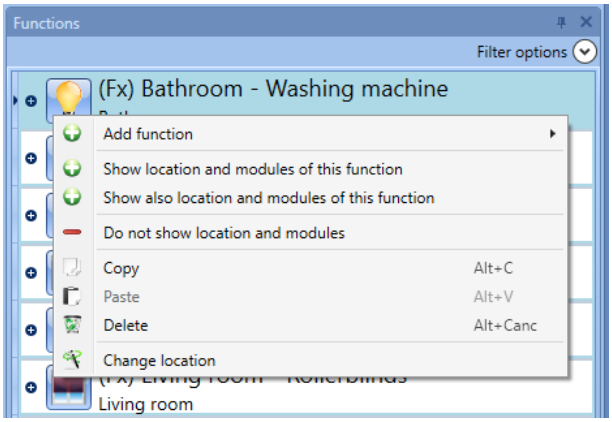
Combo box:
Only the functions according to the type selected will be shown. See the picture below.



The user can remove the filters applied by clicking the red *X* icon.

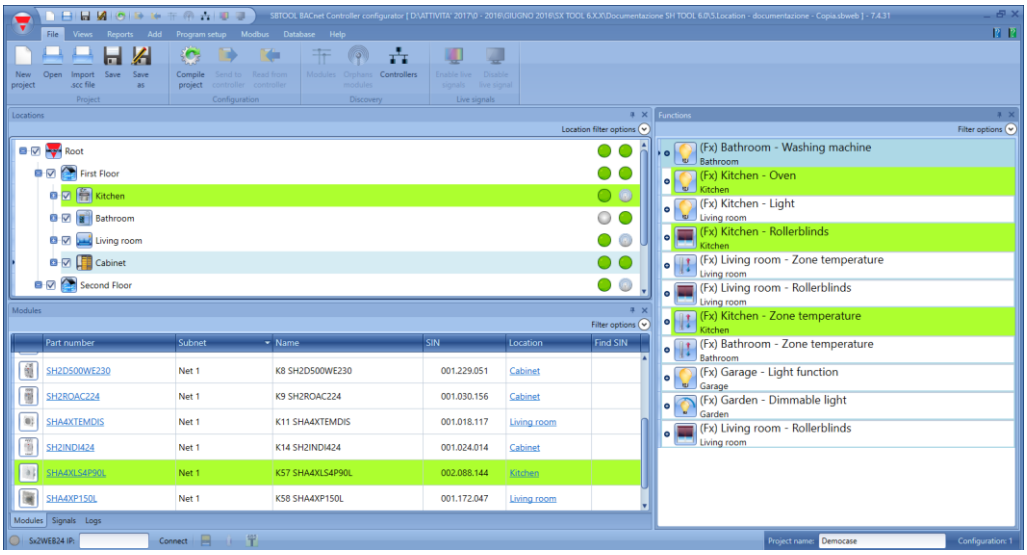
10.2 How to manage the highlight functionality

The functionality *Show location and modules* allows the user to easily find the signals used in each function.

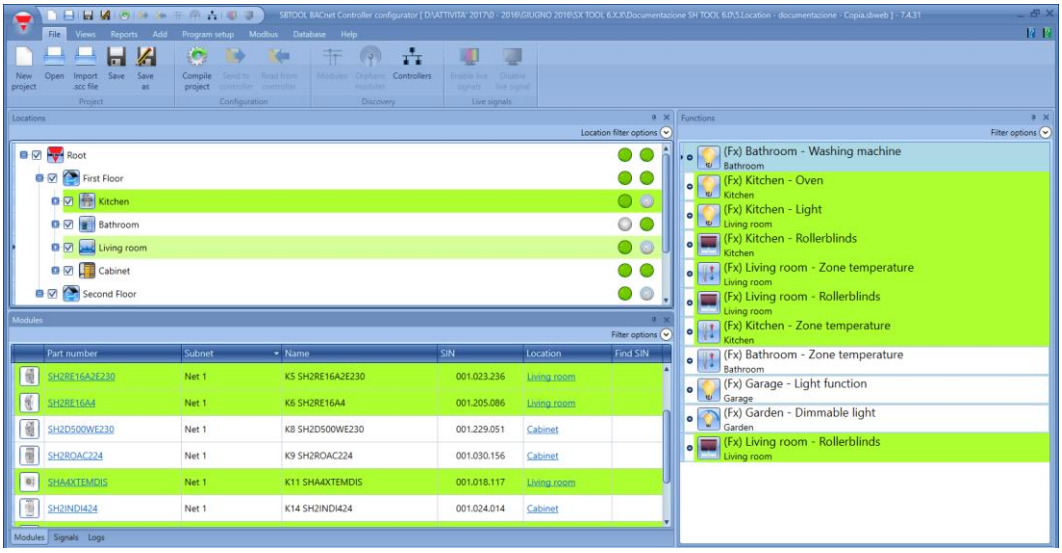


By a right click on the function it is possible to select the option *Show location and modules of this function*.

The UWP 3.0 Tool automatically highlights the location and all the modules that are used in the function selected.



When more functions must be highlighted the user can select the option *Show also objects/functions in this location*. all the modules and functions that are used also for an additional one will be shown:

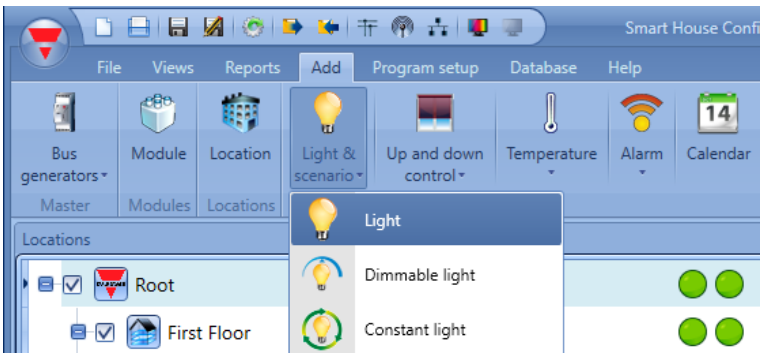


In the picture above the modules and functions part of the *Kitchen* and *Livingroom* are shown.

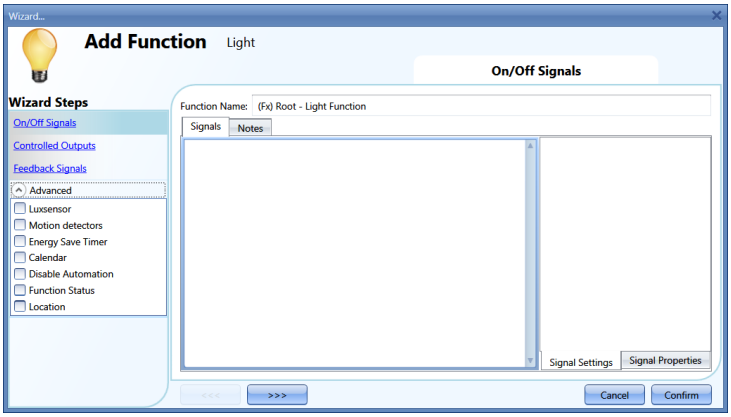
10.3 How to set a light function

This function allows the operator to manage one or more lights at the same time. The user can either configure a basic function to switch the light on manually or implement an automatic system by programming the relevant objects of the function.

To set up a light function the user has to select *Light Functions* from the Add menu, then select *Light* (see figure below). The new function will be added into the selected location.
The wizard relevant to a Light function can also be opened by typing Alt+L. (See table of short cuts).

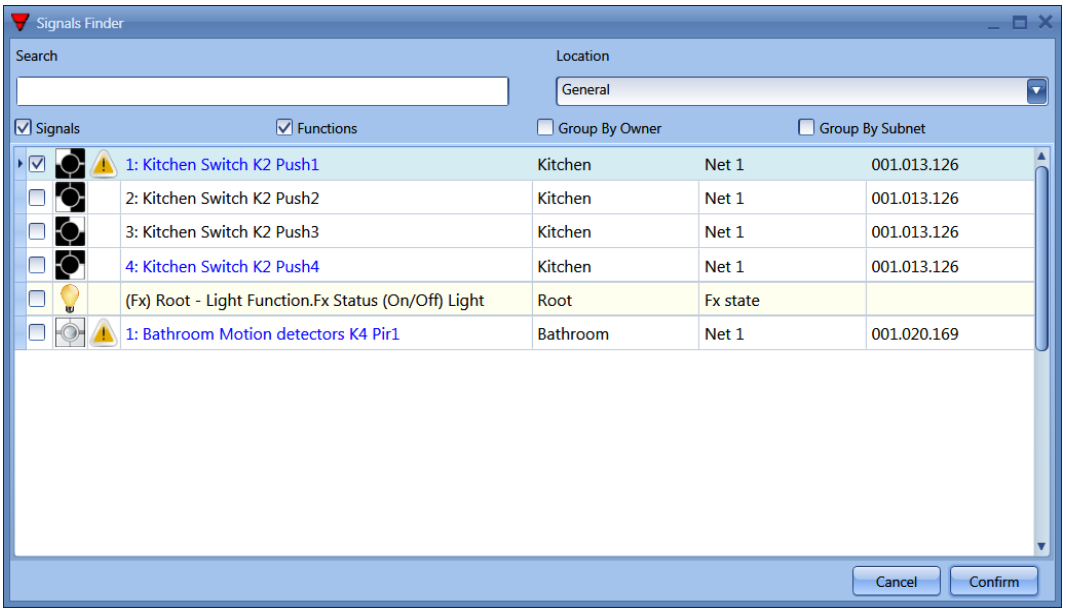


This function manages the ON/OFF switching of one or more outputs by means of one or more input commands. The input command might be a real signal, a function or a remote command (Webserver, sms, email, Modbus TCP/IP).
The automation of the light function is managed by accessing the *Advanced* section (see fig. below).
In the Advanced section the user can select different ways of controlling the light: according to the ambient light or the presence of people, with timers and/or schedulers.



10.3.1 How to switch a light ON/OFF with input commands

First, the user has to add the input signals to control the light in the *ON/OFF Signals* section.
By double clicking in the *Signals* area, a window will appear with all the available signals.



Select the signal/signals you want to control your light with. If more than one signal is selected in this window (you can chose up to 50 signals), the function will be activated/deactivated if at least one signal is active (the smart-house system does the logical OR).
The different colours have the following meanings:

- black signal → not used in any function
- blue signal → the signal is already used in another function
- blue signal + warning signal (yellow triangle)→ the signal is already used in this function

The available input signals are classified as:

- Push button
- Switch
- Function
- Remote

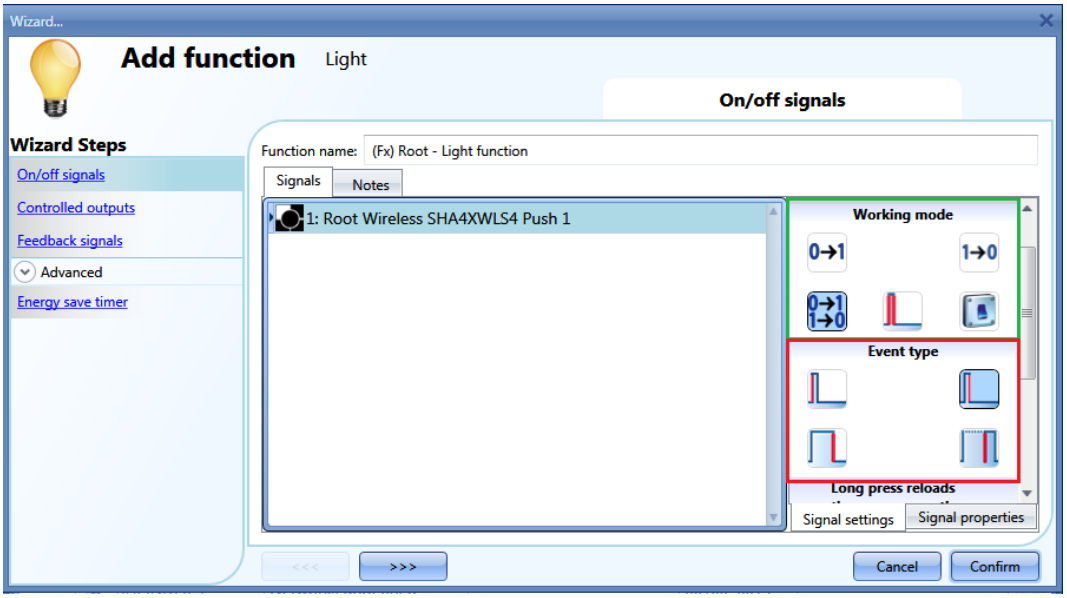
Push button:
This is a signal active for a short time following a human action (for example, a switch pressed or kept pressed).

Five different actions (*Working mode)* can be chosen: switching the function ON only , switching the function OFF only , toggle mode (every pulse toggles the function), level mode (a level signal such as a global calendar programmed as "level signal" will activate the light function when it is on and deactivate it when it is off), every change (every time the level of the signal changes, the function is toggled).
See figure below, green rectangle.
If one of the first five actions is selected, three different types of event are available to perform it, according to the duration of the activation of the push button: short pulse, long pulse, very long pulse.
See figure below, red rectangle.
The *short pulse* is predefined and corresponds to the short pressure people use to switch the light ON/OFF (shorter than 1 second). The user can select from two different types of short pulses: when the button is pressed or when the button is released . If the first option is selected, the long and very long pulses cannot be used.

The *long pulse* corresponds to the push button being kept pressed for a few seconds (from 1 to 8 seconds: this has to be predefined in the wizard relevant to the specific module).

The *very long pulse* corresponds to the push button being kept pressed for a time longer than the long press (from 0.5 to 15 seconds: this has to be predefined in the wizard relevant to the specific module).

Both these times can been seen by clicking on the *Signal Properties* tab window.



Here are some examples:

	Event type			
Working mode				
0→1	As soon as the push button is pressed, the function will be activated.	After a short press (less than 1 second), the function will be activated at the release of the push button.	After a long press, the function will be activated at the release of the push button.	After a very long press, the function will be activated at the release of the push button.
1→0	As soon as the push button is pressed, the function will be deactivated.	After a short press (less than 1 second), the function will be deactivated at the release of the push button.	After a long press, the function will be deactivated at the release of the push button.	After a very long press, the function will be deactivated at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the function will be toggled.	After a short press (less than 1 second), the function will be toggled at the release of the push button.	After a long press, the function will be toggled at the release of the push button.	After a very long press, the function will be toggled at the release of the push button.
	The function is toggled when the push button is pressed and toggled again when the push button is released.			
	The function is switched on when the signal goes ON, and it is switched off when the signal goes OFF.			

Of course, only one of the event types can be selected to perform the required working mode.
If the combination marked in green is selected, i.e. toggle mode with short press acting on the release of the push button, and the energy-save timer is enabled in the *Advanced* section, the long press can be used to reload the timer and the very long press to disable it. This enabling can be carried out by using the icons in the purple rectangle.

An example could be: in a walk-in wardrobe, the light is switched ON/OFF by means of a light switch, but I want to be sure that after 5 minutes it is switched off automatically if I forget to do it manually. To achieve this, I have to enable the energy-save timer. But if I need to stay in the wardrobe room (see above), I would also like to reload the timer: if I keep the push button pressed, the timer will reload automatically and the light will be switched OFF 5 minutes after my long press (if I do not reload it again, of course). I might also stay there for a longer time and need to disable the timer: to do this I have to keep the push button pressed for a very long time, and the timer will be disabled. In this situation the light has to be switched off manually. The timer will be enabled at the next switching ON.
This kind of automation can also be used in some other applications, such as stairs or corridors.

The long and very long activation times can be set directly from this wizard, in the *Signal properties* tab window, without the need to re-open the wizard of the module.

In the Signal setting window, the user can also enable the reversing of the signal. See the yellow rectangle in the figure below.

Switch and Function:
The signals of this type perform the selected action when they change level, according to the table below:

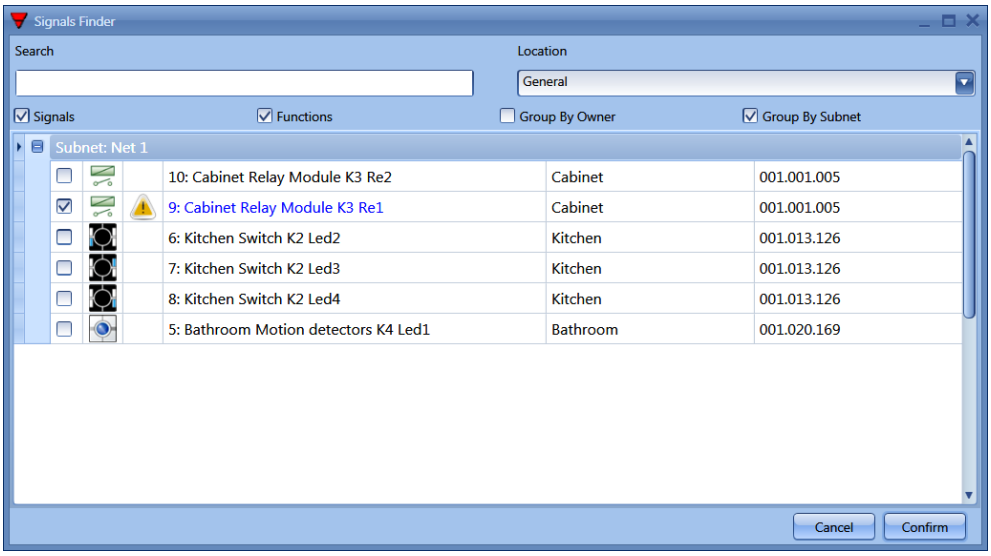
	Event type	
Working mode	**Signal activated**	**Signal deactivated**
0→1	The function is activated	No action
1→0	The function is deactivated	No action
0→1 1→0	The function is toggled	No action
	The function is toggled	The function is toggled
	The function is activated	The function is deactivated

If *Inverted* is enabled, the following table applies:

	Event type	
Working mode	**Signal activated**	**Signal deactivated**
1→0	The function is deactivated	No action
0→1	The function is activated	No action
0→1 1→0	The function is toggled	No action
	The function is toggled	The function is toggled
	The function is deactivated	The function is activated

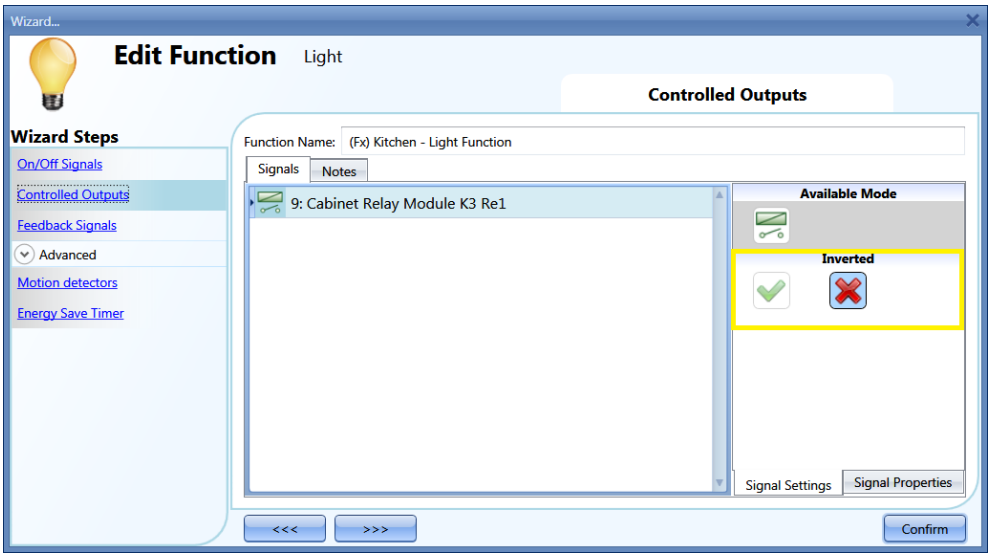
10.3.2 How to select the output that will control the light

To select the output signal that is connected to the light, click on *Controlled outputs* and then double click on the Signal window.



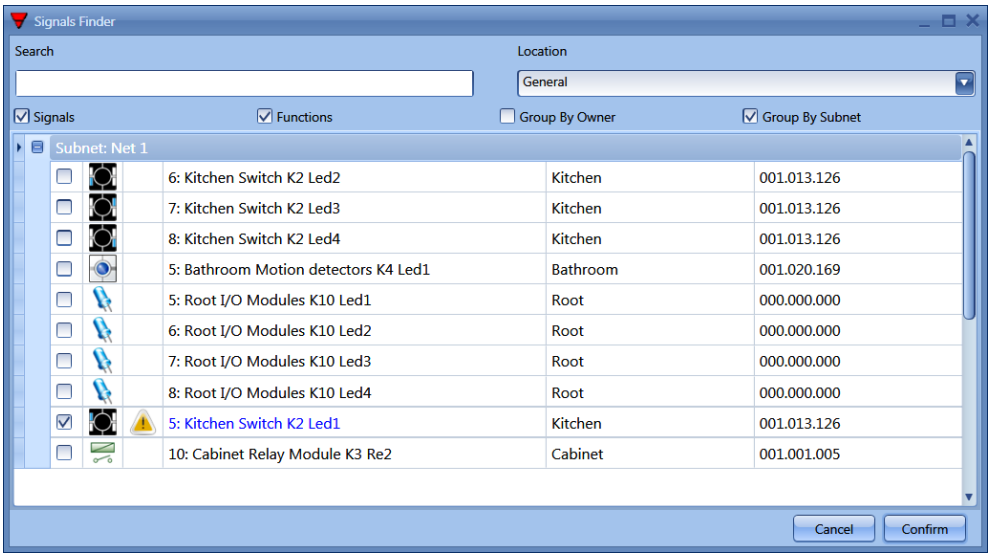
The signals available in this window are relays and LEDs.
Up to 50 signals can be chosen and they are managed in parallel.

The logic of each output signal can be set as normal or inverted (see yellow rectangle).



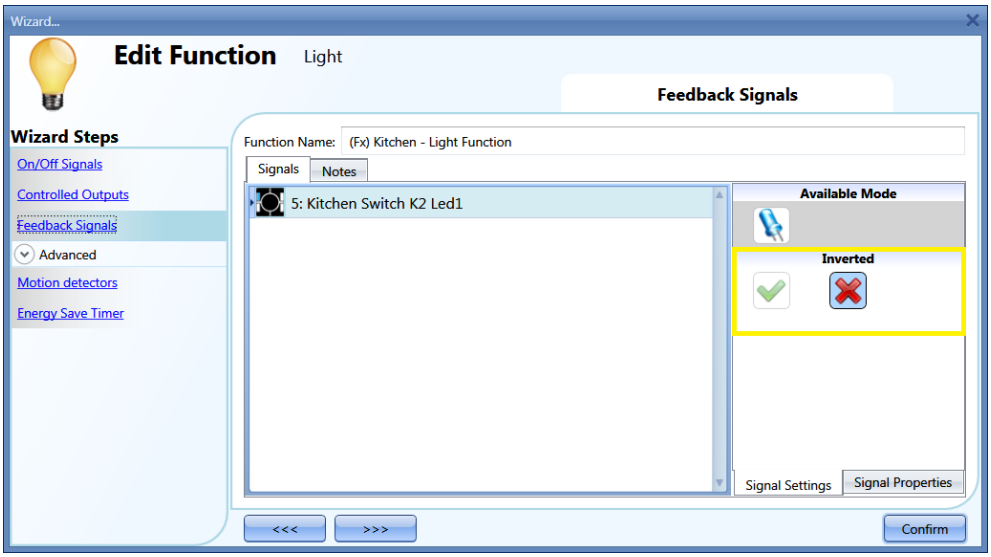
10.3.3 How to set a signal to be a feedback of the status of the function

To select the feedback signal that indicates the status of a function, click on *Feedback signals* and then double click on the Signal window.



The signals available in this windows are LEDs and relays.
Up to 50 signals can be chosen and they are managed in parallel.

The logic of each output signal can be set as normal or inverted (see yellow rectangle).

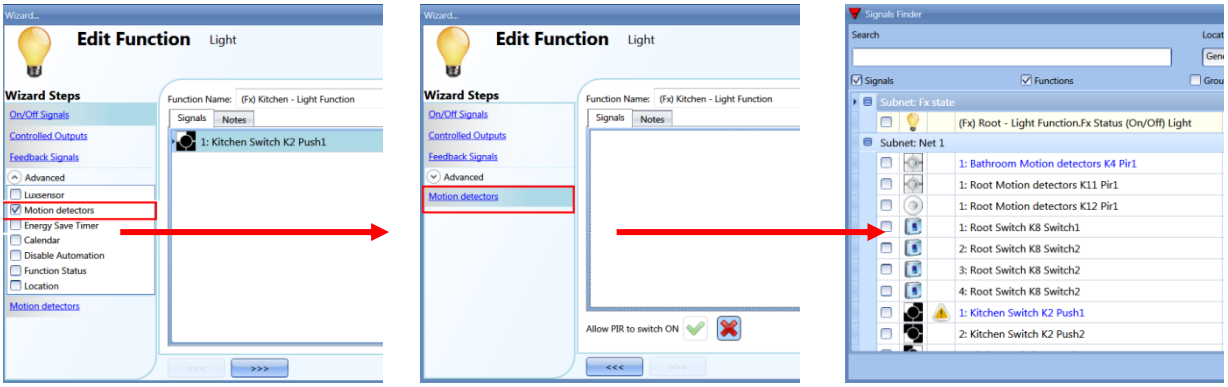


10.3.4 How to manage lights automatically

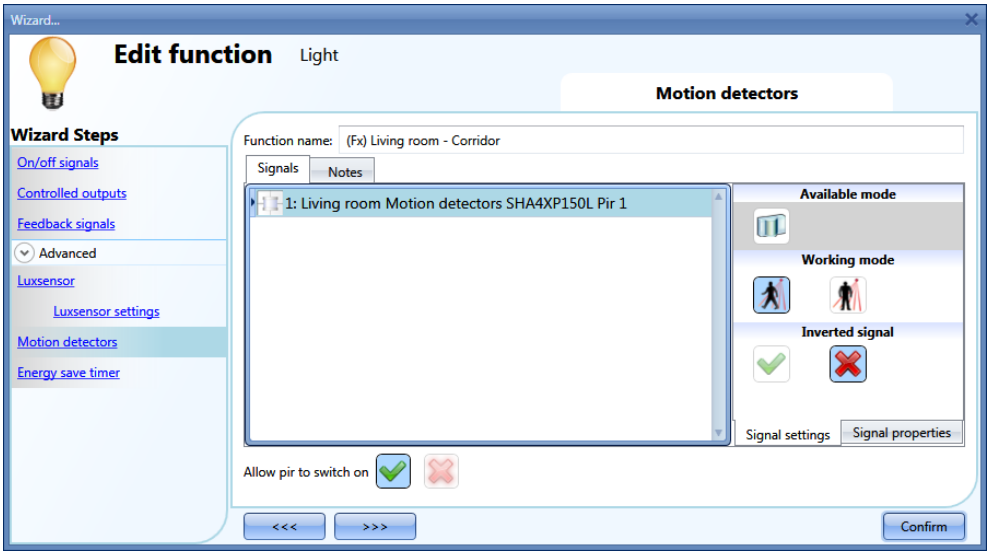
The automatic on/off switching of the light can be managed by PIR sensors (the light will be switched ON when the PIR sensor detects people movement/presence), by calendar functions (calendar to switch the light on/off at pre-defined time intervals) or by luxmeters which can switch the light on or off according to the levels of ambient light.
All the above have to be enabled in the Advanced section.

10.3.5 How to select PIR sensors to control lights

To correctly set and mount a PIR sensor, please refer to the Modules paragraph, in the PIR section..
The first thing to do when you need to use a PIR sensor in a light function is to enable it in the *Advanced* section.



The *Motion detector* menu will appear. After selecting it, with a double click on the Signal window, the list of available signals will appear.
Select the required signal/signals and click on *Confirm*. Up to 50 signals can be selected and the system will do a logical OR of all of them.



You can also select to invert the signal, by checking the green *V* below *Inverted*, in the Signal Setting tab window.

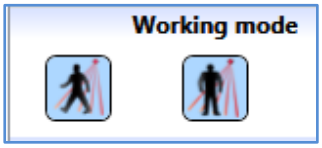
10.3.6 How to turn on a light with PIR sensors, and turn it OFF manually

After selecting the required PIR signal as described above, to turn the light on as soon as movement is detected, you need only to enable *Allow PIR to switch ON*.

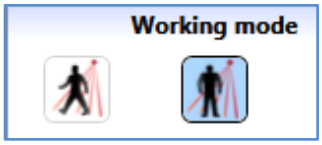
In this case the light will not be switched OFF if no presence is detected.
The energy-save timer should not be enabled.

10.3.6.1 Working mode of PIR signals

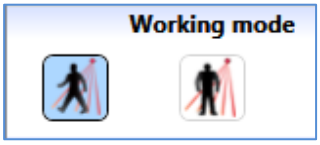
In this field, the user can define the action that any signal added in this will perform:



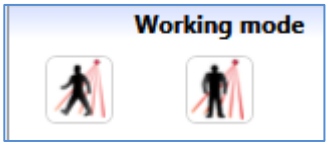
The PIR signal will switch the light ON if *Allow PIR to switch ON* is enabled, if movement is detected, and will reload the energy save timer if presence is detected.



The PIR signal will **not** switch the light ON even if *Allow PIR to switch ON* is enabled if movement is detected, and will reload the energy save timer if presence is detected.



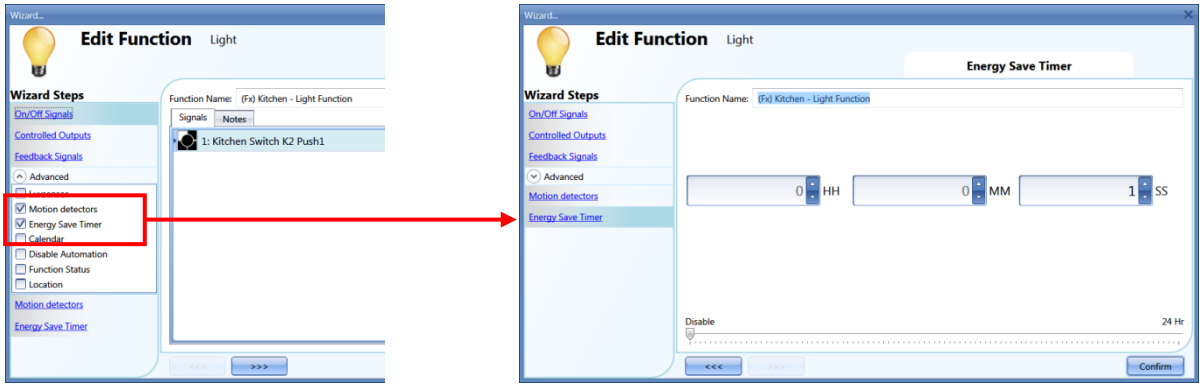
The PIR signal will switch the light ON if *Allow PIR to switch ON* is enabled, if movement is detected, and will **not** reload the energy save timer if presence is detected.



The PIR signal will do nothing.

10.3.7 How to turn a light on and off with PIR sensors

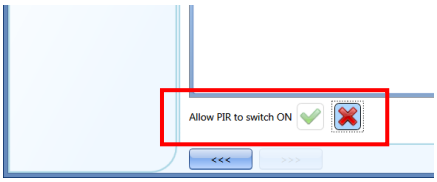
To allow the PIR sensor to turn the light on, just follow the previous paragraphs.
To turn the light off immediately or with a certain delay after the last presence has been detected, the energy-save timer has to be enabled.



A minimum time of 1 second has to be set to switch the light OFF. You can set the desired delay off timer by moving the slider or by typing the hours, minutes and seconds.

10.3.8 How to switch the light ON manually and OFF with the PIR sensor

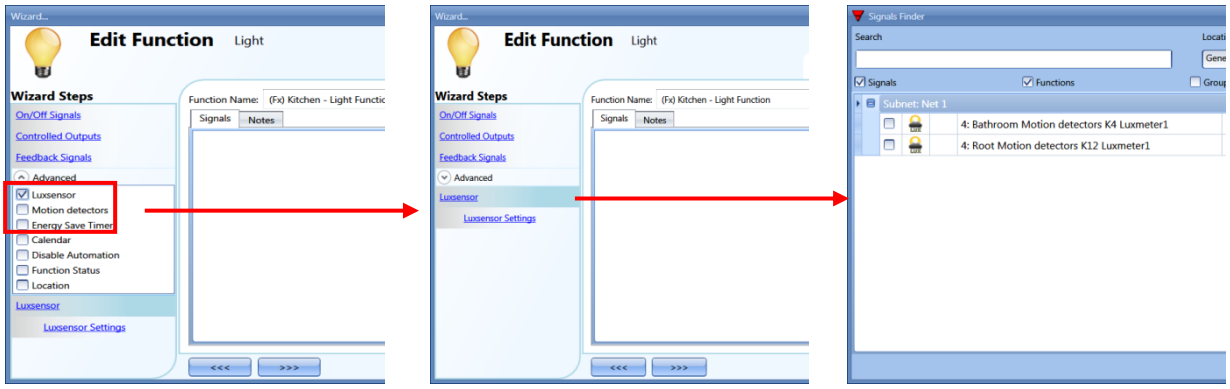
In order not to switch on the light with the PIR sensor, *Allow PIR to switch on* must not be enabled.



To allow the PIR to switch the light off, the energy-save timer has to be enabled.
Once this is done, by following the instruction described in the previous paragraph, you can set the required time with a minimum of 1 second.

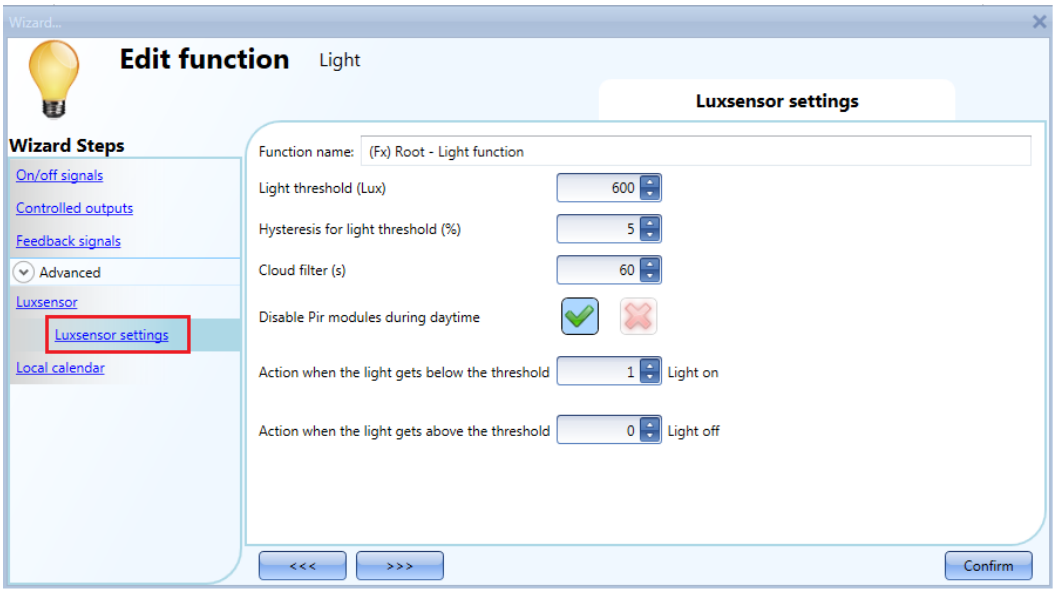
10.3.9 How to switch the light ON and OFF according to the daylight

The light can be managed in an automatic way by using lux meters: you can set a threshold in order to have the smart-house system switch the light ON if the daylight goes below this threshold, and switch the light OFF if the daylight goes above the threshold.
Since this is an advanced functionality, it has to be enabled in the *Advanced* section.



Once the *Luxsensor* is enabled, by clicking on it, the standard window to add signal will appear.
Up to 10 signals can be added: the smart-house system will calculate the average value.

As a second step, the threshold has to be set: click on Luxsensor settings.
The following window will appear.



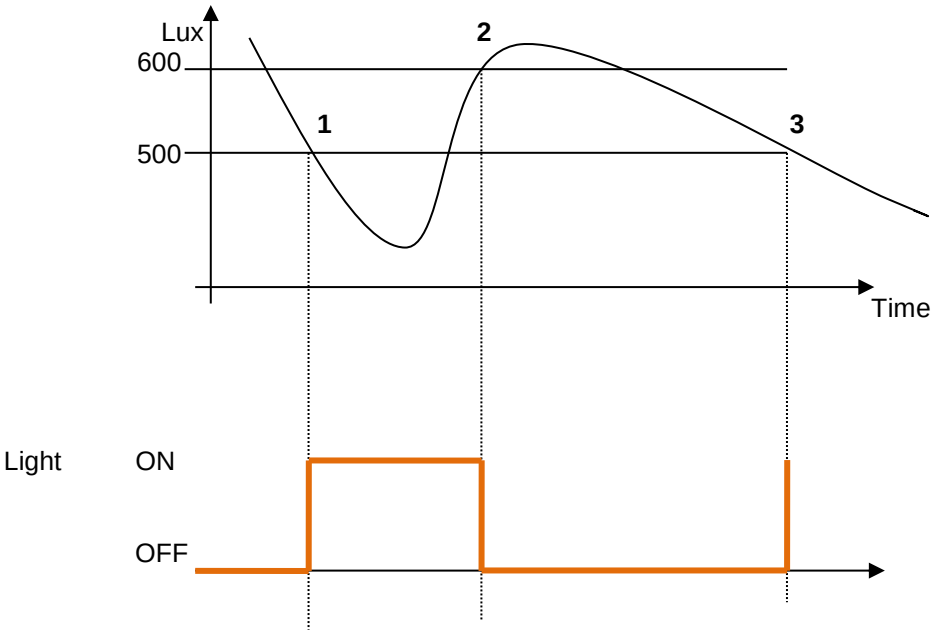
In the field *Light threshold (lux)*, the desired daylight level below which you want your light ON must be entered. A value between 0 and 5000 lux can be set.

In the field *Hysteresis for light threshold (%)*, the hysteresis you need to have the light level OFF threshold must be entered, that is the threshold above which the light has to be switched OFF.
This field is a percentage of the *Light level ON* threshold. A value between 5 and 50 % can be set.

In the example shown in fig. 1, the level OFF threshold can be calculated in the following way:

Level OFF= level ON+ 20% of Level ON= 500 + 100 = 600 lux

If the daylight goes above the threshold of 600 lux, the light will be switched OFF.
This example is also explained in the diagram below.
Fig. 1.



Due to clouds that might change the daylight level for a short period, we suggest setting the Cloud filter for at least 60 seconds, in order not to have the light turning ON/OFF continuously.
The field *Disable PIR modules during day time* must be considered if the lux meter is used in combination with PIR sensors.

If you want to switch the light ON when the light level goes below the threshold (points 1 and 3 in fig. 1), the field *Action when the light gets below the threshold* should be set to action 1 (light on).
If you want to switch the light OFF when the light level goes below the ON level threshold (points 1 and 3 in fig. 1), the field *Action when the light gets below the threshold* should be set to action 0 (light off).

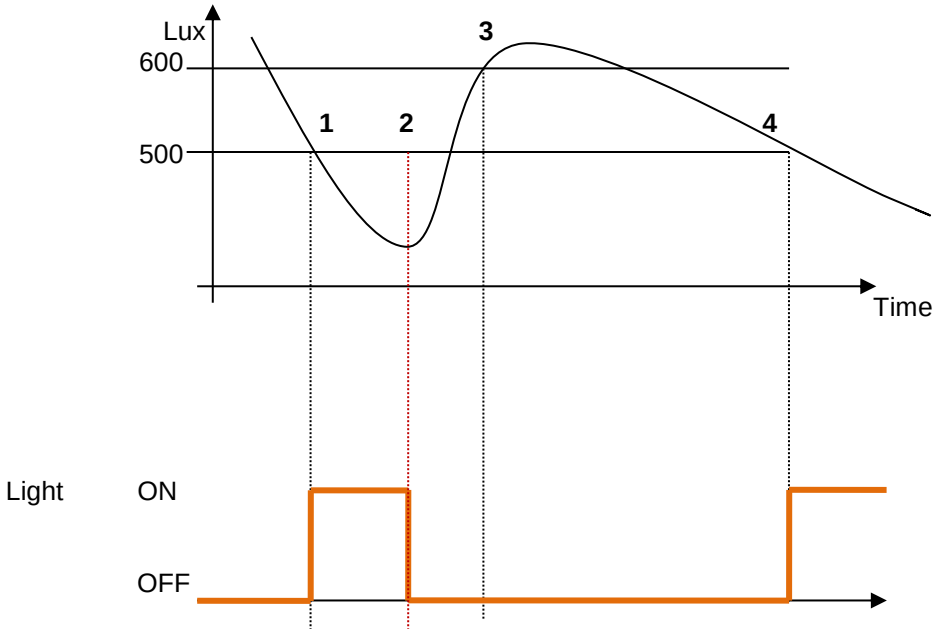
If you want to switch the light OFF when the light level goes above the threshold + hysteresis (point 2 in fig. 1), the field *Action when the light gets above the threshold* should be set to action 0 (light off).
If you want to switch the light ON when the light level goes above the ON level threshold + hysteresis (point 2 in fig. 1), the field *Action when the light gets above the threshold* should be set to action 1 (light on).

If these two green Vs are not checked, the system will not do anything when the thresholds are surpassed. This might be needed when lux meters are used in combination with PIR sensors or calendar.

The ON and OFF switching is done in the event of the thresholds being surpassed, so if the light status is changed manually by means of a light switch, or automatically e.g. by a scheduler, the automation of the lux sensor will not change it unless thresholds are surpassed another time.

Please see the example below: the light is switched ON automatically by the lux meter in point 1, then if it is switched OFF, for example manually, in point 2, it will be turned ON again by the lux meter only in point 4, when the daylight level again goes below the level ON threshold.

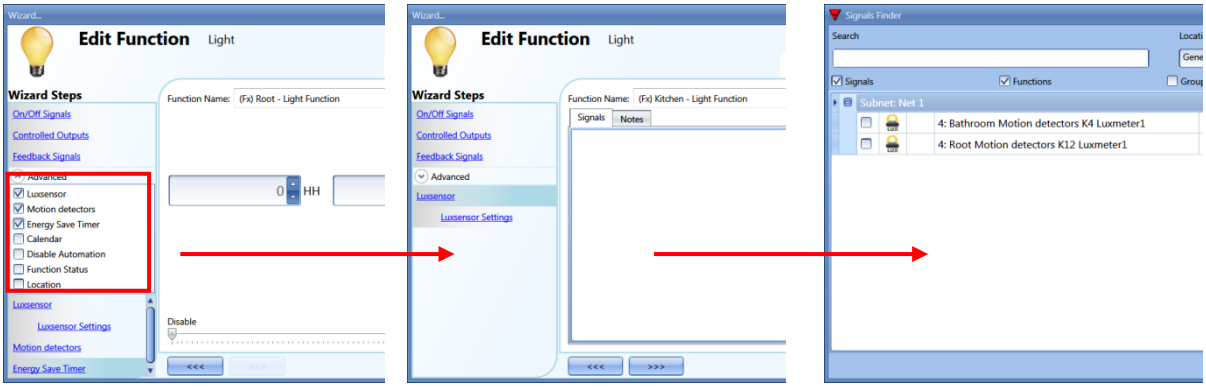
Fig.2



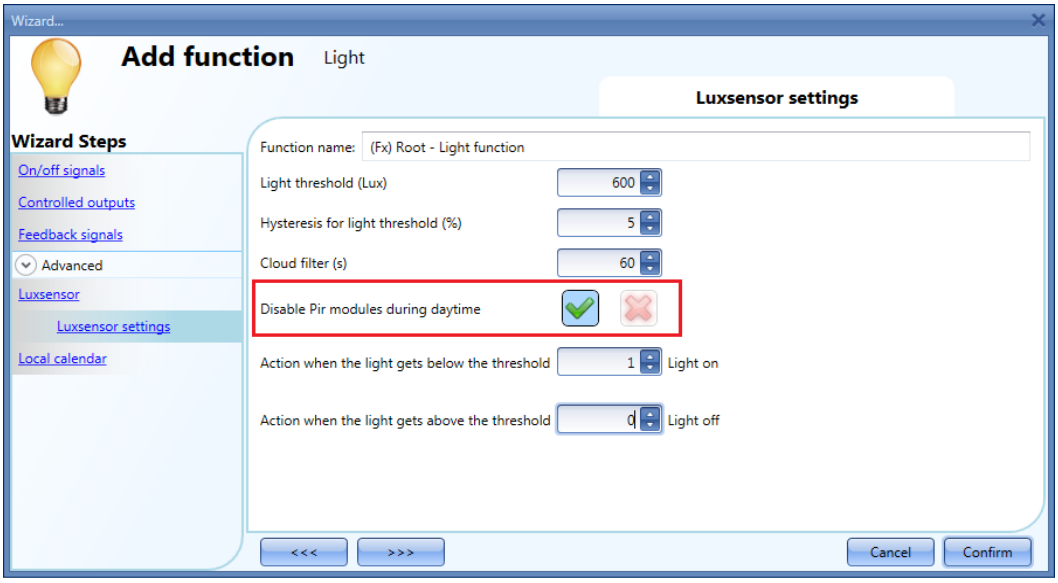
10.3.10 How to control light with PIR sensors in combination with lux meters

If PIR sensors are used in combination with lux meters, it means that if a people presence is detected, the light has to be switched ON only if the daylight level is below the OFF level threshold.
To achieve this, a PIR sensor and a luxsensor have to be added.
The steps are summarised below:

1. Enable Luxsensor, motion detectors and energy-save timer in the advanced section. Add the required luxmeter.

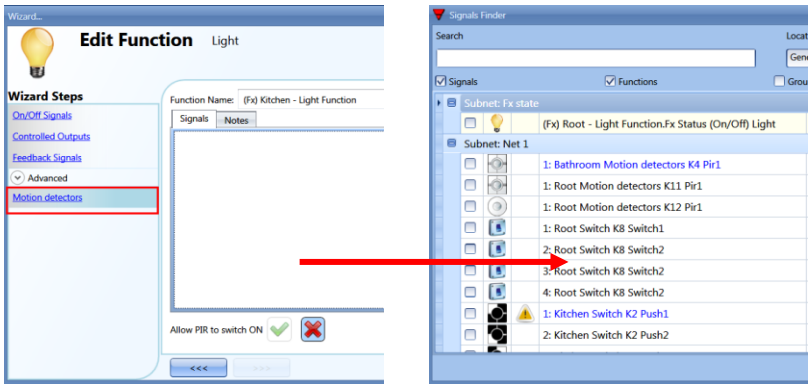


2. Set the required light threshold as described above.

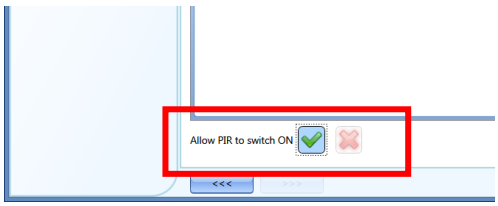


If for any reason the PIR sensor is enabled to switch the light ON also during day time, the red X in the red rectangle should be checked.

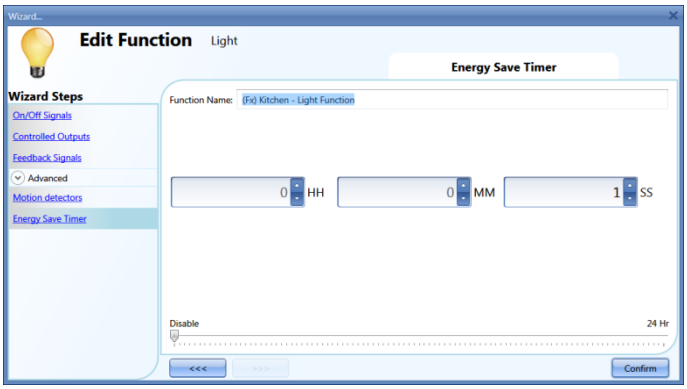
3. Add the PIR sensor



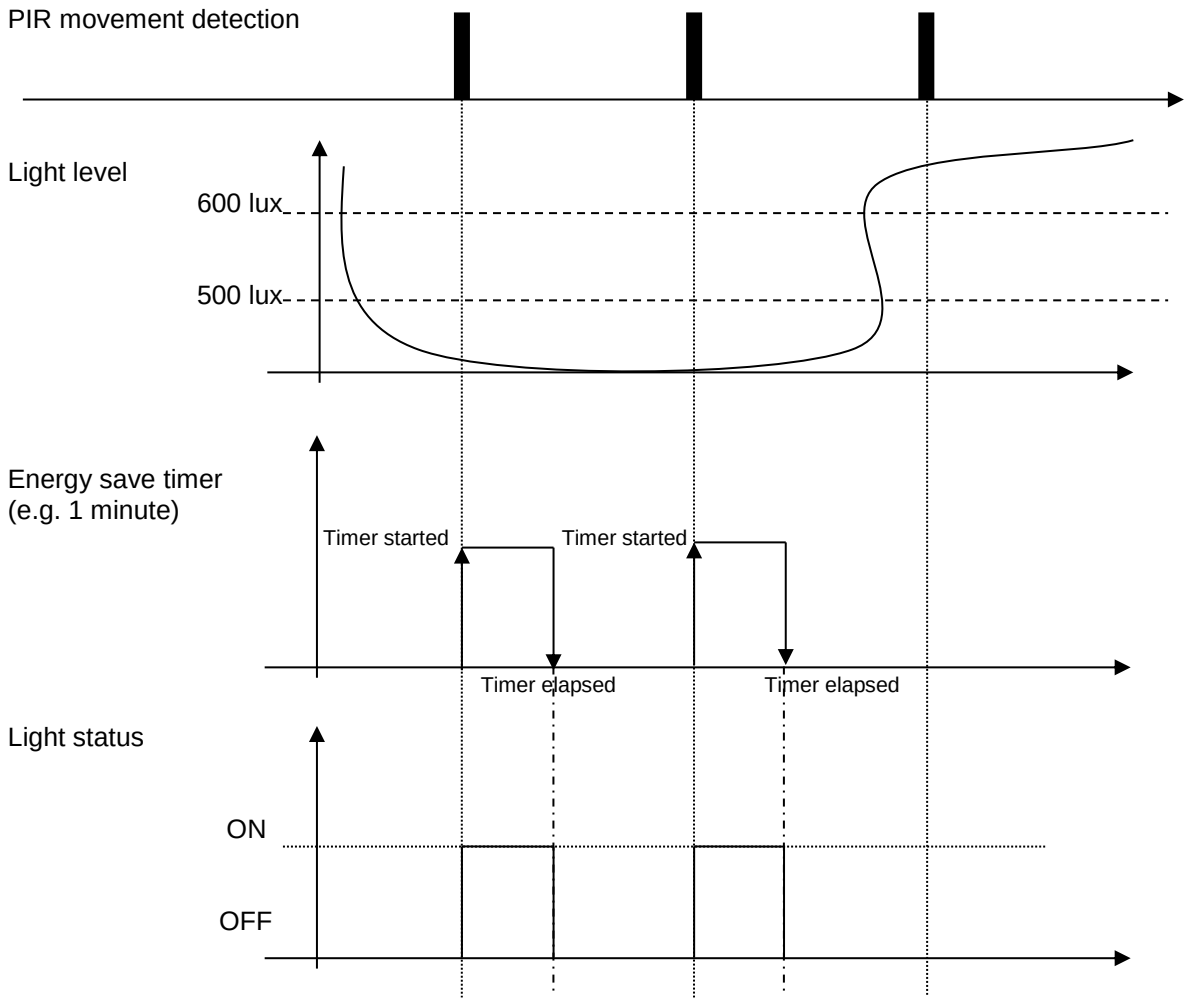
4. Enable it to switch the light ON



5. Set the time after which you want the light to turn off if no more presence is detected. Click on energy-save timer.

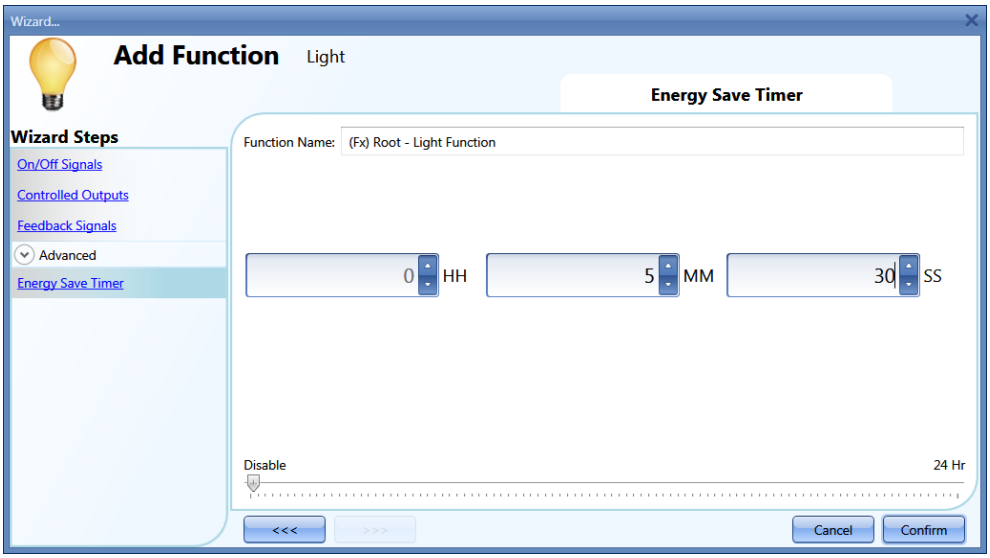


The diagram below shows what the smart-house system will do if the previous steps are followed:



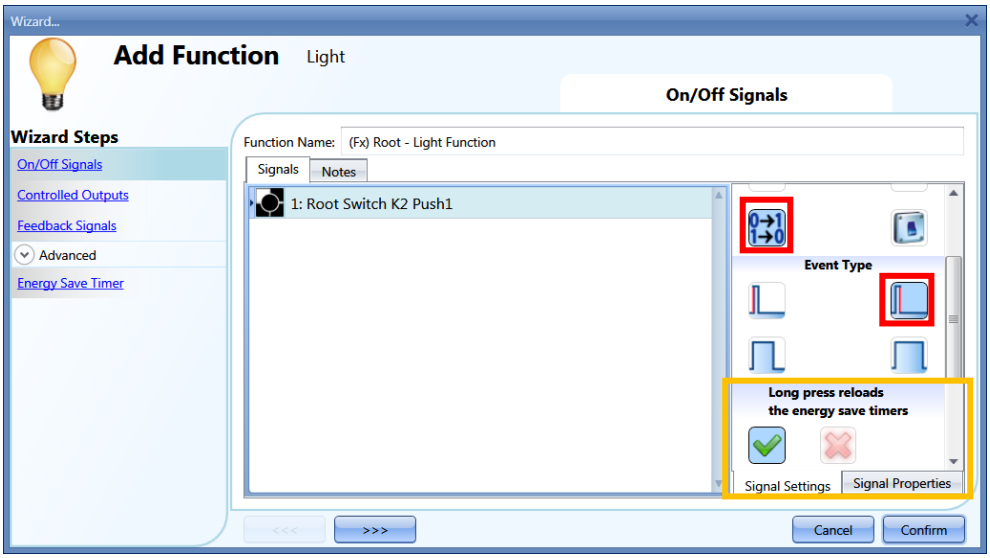
10.3.11 How to switch the light off after a predefined time period

If the user needs to be sure that the light is switched off to save energy, the energy-save timer should be selected in the advanced functionalities (the energy-save timer is selected by default when the user creates a new light function).
The energy-save timer starts every time the light is switched On, causing the light to go off automatically when the set time has expired.
The energy-save timer can be set up to a maximum value of 24 hours, while when it is set to 0 it does nothing. So the shortest time that can be set is 1 second.
The set timer value can be set and changed by means of the webserver.



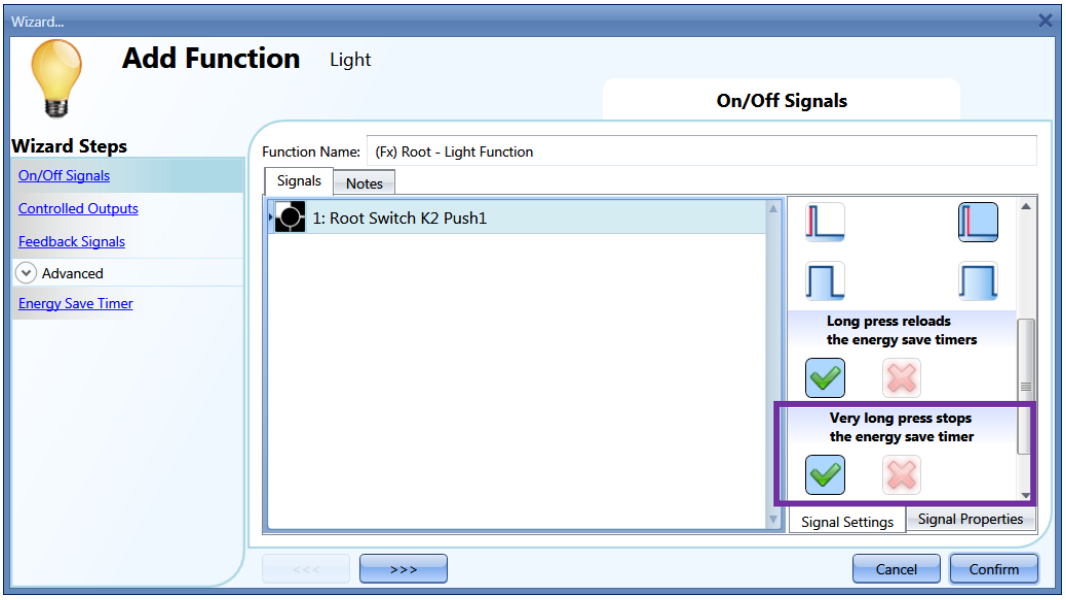
10.3.12 How to reload the energy-save timer

The reload of the timer is possible if the light is switched on/off in toggle mode and if the toggle event is set on the release of the push button (see red rectangles).
If these two settings are selected, a further field appears (see the orange rectangle) saying *Long press reloads the energy-save timer:* if the green V is checked, the timer is reloaded by means of a long press and it starts counting again (to see 'long press' click here).

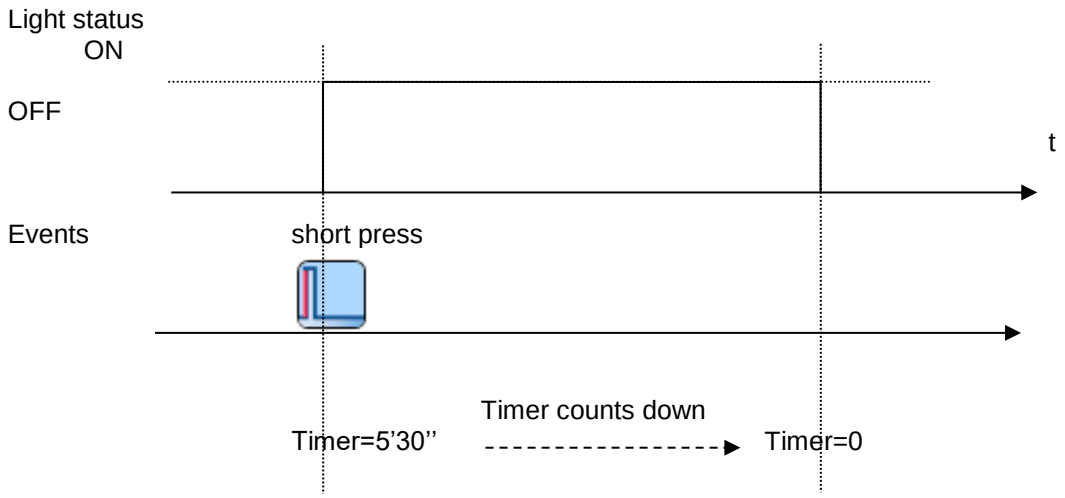


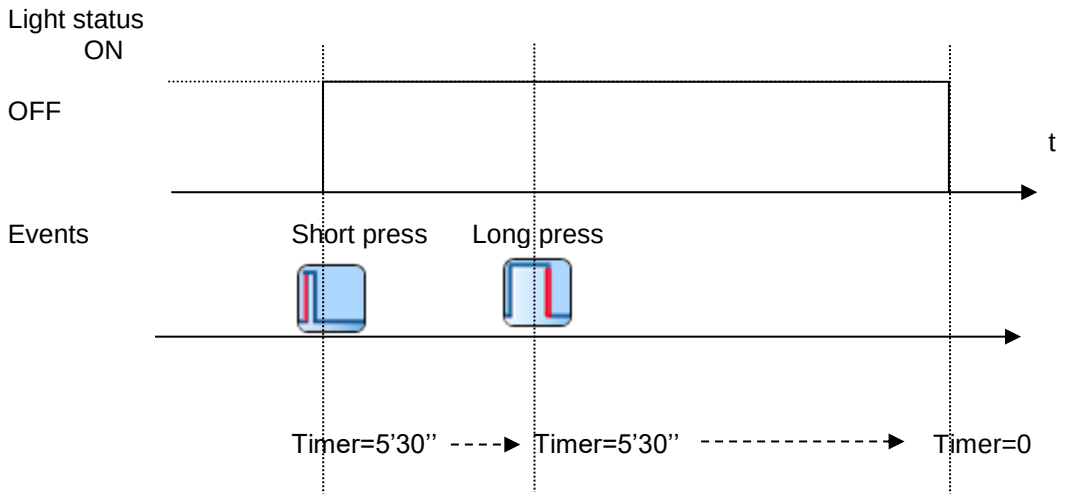
10.3.13 How to disable the energy save timer

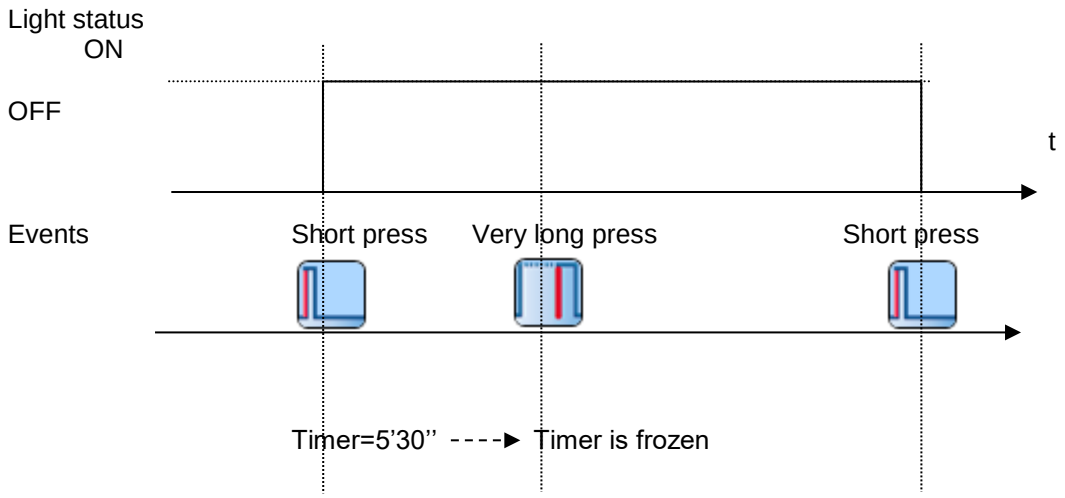
If the user wants to disable the energy-save timer, once the behaviour of the push button is set according to the figure below, the green V should be checked in the field *Very long press stops the energy save timer* (violet rectangle).
In this situation, with a very long press the timer will be disabled and the only way to switch the light off is manually, with a short press on the push button. The timer will be activated at the next switching on.



To understand better the behavior of the energy-save timer, if the light function is set according to the previous picture 5, the table below is valid:





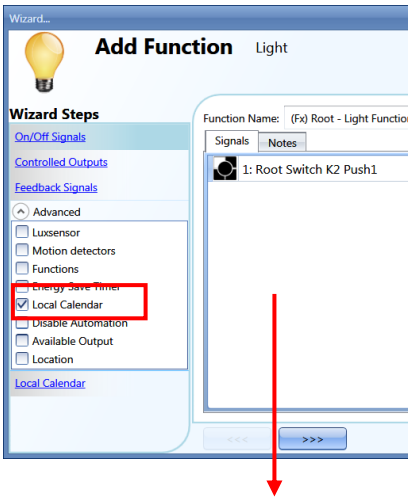


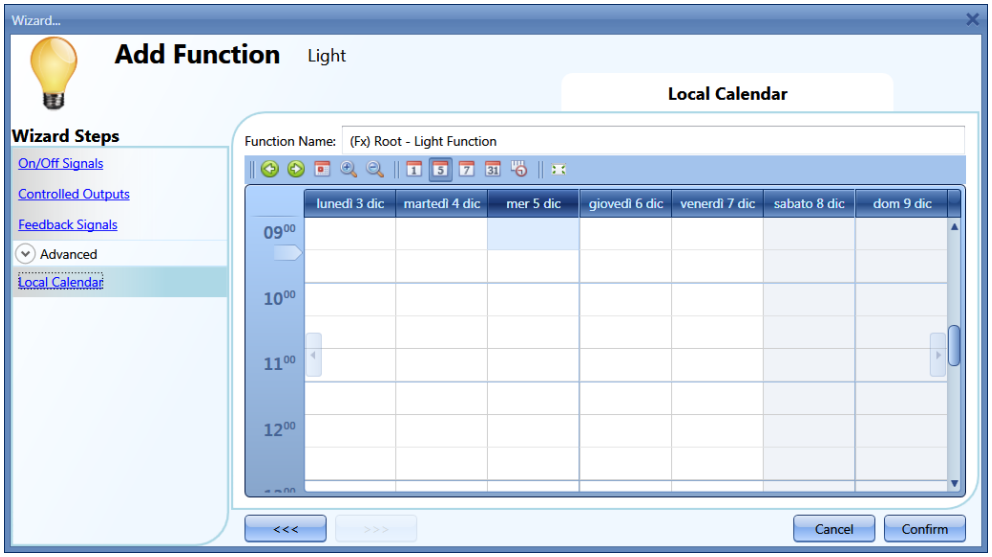
10.3.14 How to switch the light ON/OFF using the calendar

There are two ways of managing the lights with the calendar: by setting the “local” calendar inside the function or using a “global” calendar function.

10.3.14.1 The local calendar function

The relevant menu should be enabled in the Advanced section.





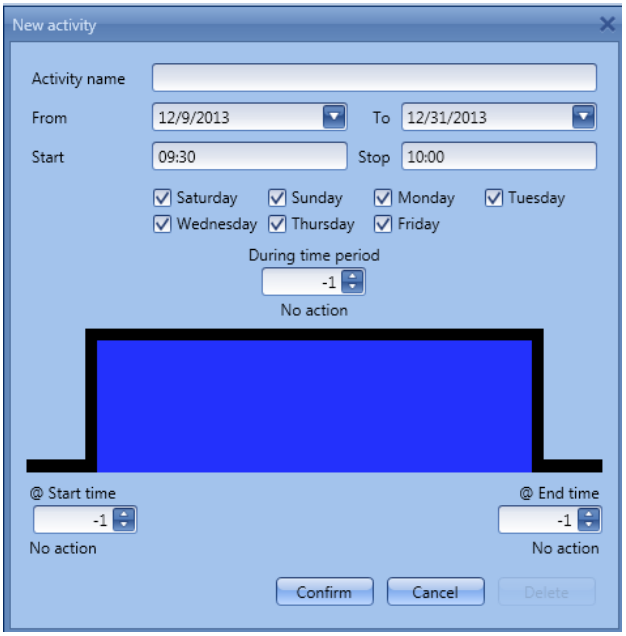
The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons :

	One step back in the Calendar. e.g: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. e.g: if a week is visualized, this button allows the user to go to the next week.
	Go immediately to the current day.
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally.
	The calendar visualizes 31 days horizontally
	The calendar visualizes 7 days vertically
	Full screen view

10.3.14.2 Calendar activity

Once the user has selected his preferred type of visualization, he should double click to enter a time band on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that will appear on the calendar: this is a mandatory field.

From*:* Date at which the event will start.

To*:* Date at which the event will finish.

Start*:* Time at which the event will start.

Stop: Time at which the event will finish.

The activity will be repeated in the following years, using the same dates and times.

The action managed by the calendar can be of two types:

1) Event activity: the system will perform the selected action only at start time and end time and they are managed as an event
2) Level activity: during the time band, all the automation can be disabled.

If ***Event activity*** is selected, the user can decide what action he wants when the time band starts and finishes.

@ ***start time***: in this field the user selects the action he wants performed when the period starts. The available actions are:

- (-1) No action: nothing will be done at start time
- (0) Light off: the light will be switched off at start time
- (1) Light on: the light will be switched on at start time

If ***Level activity*** is selected, the user can disable the automation of the PIR sensors and luxmeters (see paragraph Disable automation).

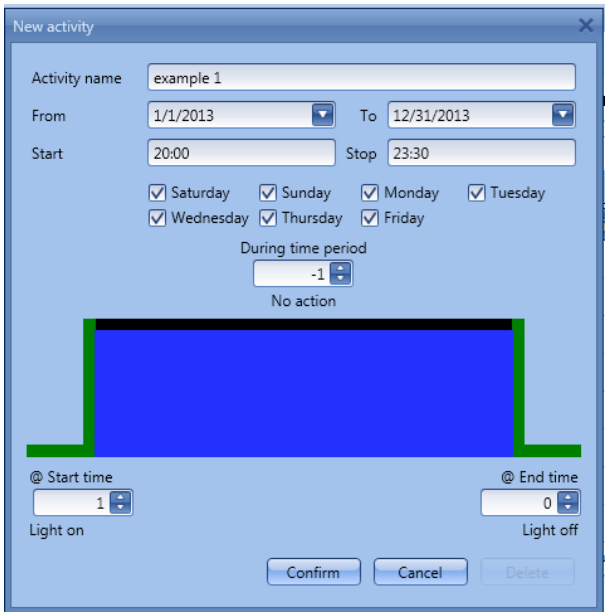
During time period: in this field the user selects the action he wants performed during the time band.

- No action
- Disable automations

When the user chooses a Level activity, the actions at start time is done first, then the automations are disabled and at the end time the selected action is performed.

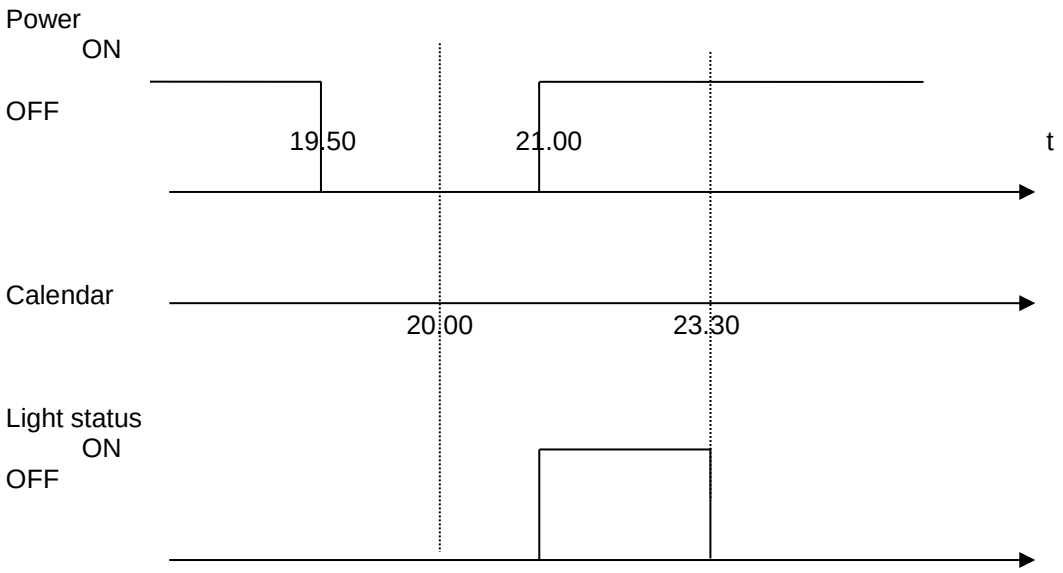
Here are some examples.

- Example 1
  A light has to be switched ON at 8 o'clock in the evening and then switched off at 23.30 every day of the year.



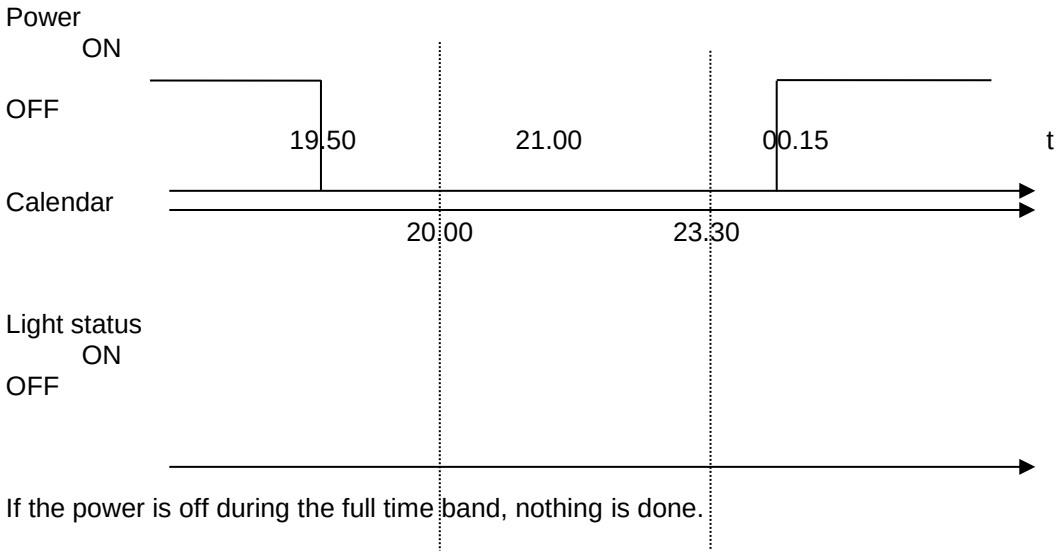
Since this action is to be performed every day of the year, the dates 1st January and 31st December have to be inserted in the fields **From** and **To**.
With the settings in figure above, every day of the year the light will be switched on at 20.00, and off at 23.30, from Sunday to Saturday.
If the light is switched off manually at 21.00, it will not be turned on again by the calendar until 8 o' clock the following day.

If a power-down occurs at the start time or at the end time, the system will behave according to the diagram below.



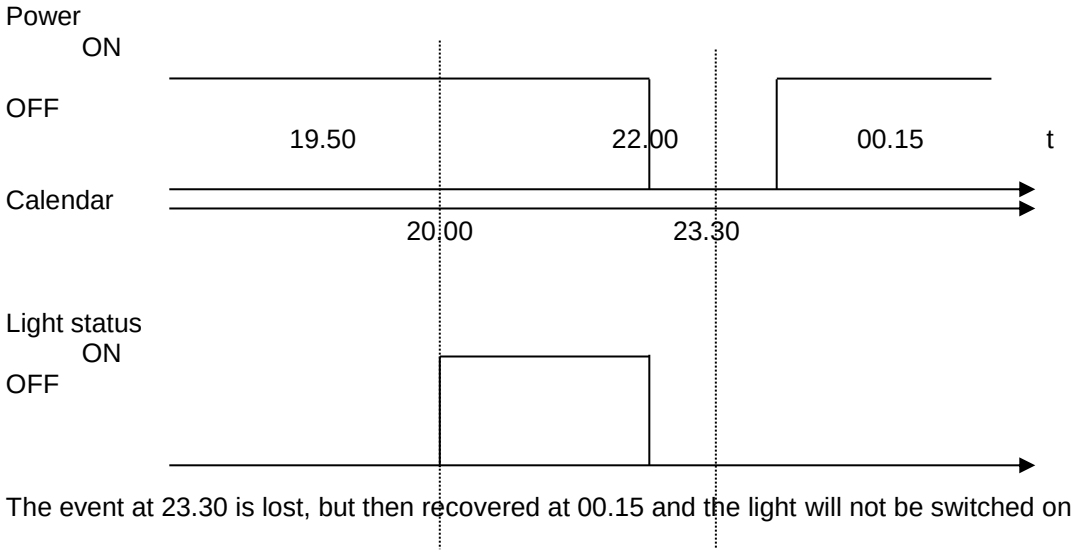
The event at 20.00 is lost but then recovered at 21.00. In general, if power is off during a period where the light is switched on by the calendar, if power returns before the activity has expired, the light is switched on again.

1)



If the power is off during the full time band, nothing is done.

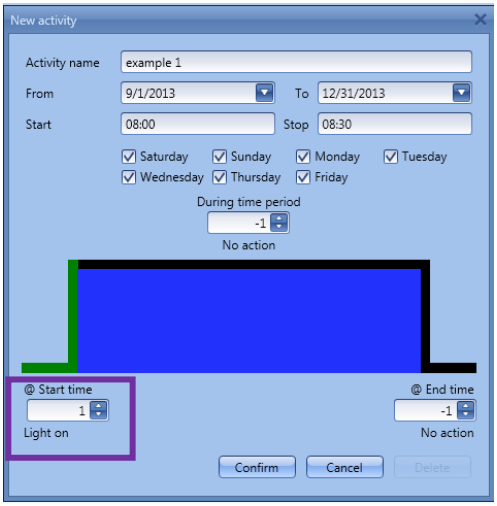
2)

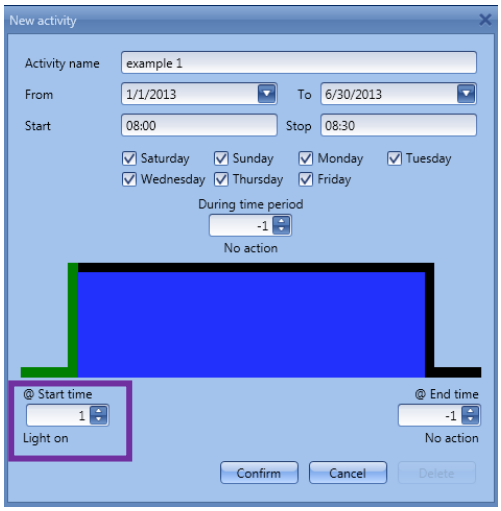


The event at 23.30 is lost, but then recovered at 00.15 and the light will not be switched on again.

- Example 2
  The light is switched on at 8 o clock in the morning, from September to June, from Monday to Friday..

  In this example, if an activity starts in one year and ends in the following year, two activities have to be created: the first one from September to December, the second one from January to June.



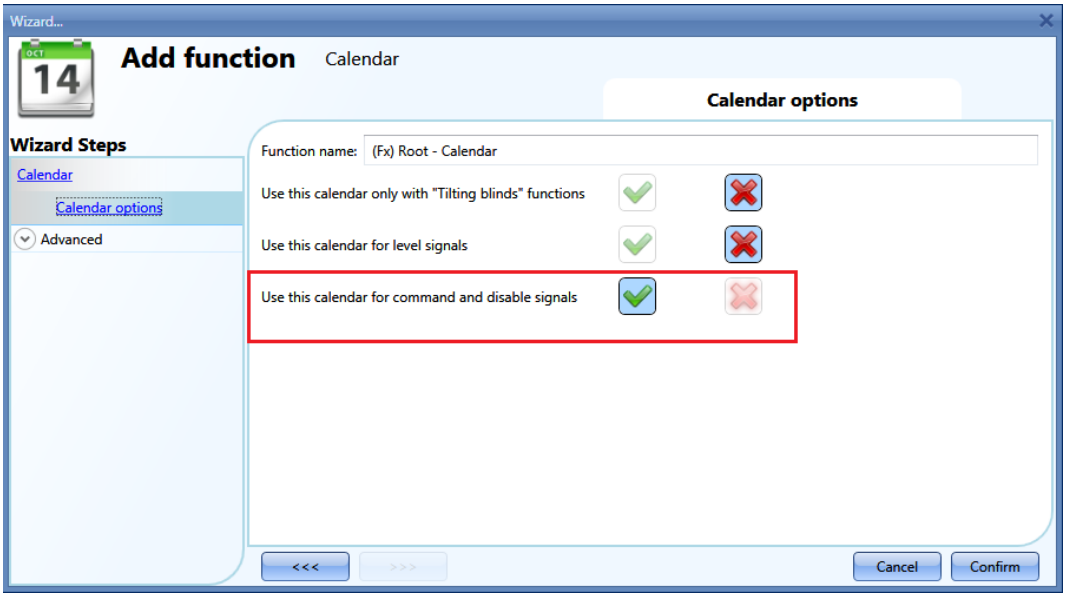


As shown in the violet rectangle, since no end action is needed, ‘No action’ is selected (-1).

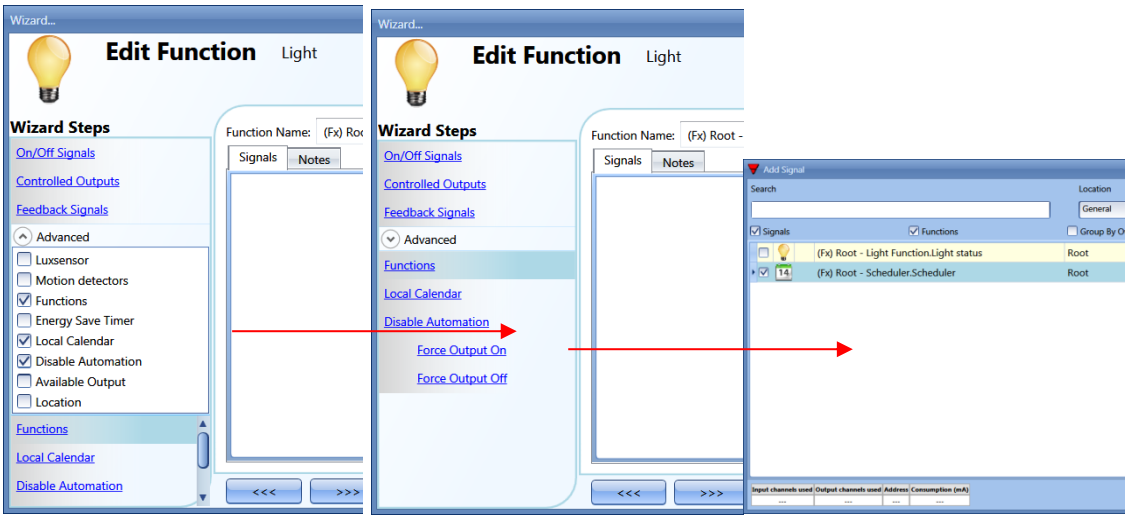
If a power-down happens at the start time, the event is lost, and the light will be switched on automatically at 8 o’ clock the following day.

10.3.14.3 The global calendar

Before using a global calendar programmed as "*a calendar for command and disable signals*", it has to be defined as a function (See **How to set a global calendar**).



The field function then has to be selected in the *Advanced* menu.
By clicking on Functions and then double clicking in the *Signal* window, the *Add signal* window will appear. The required calendar function can be selected.
The behaviour will be the same as the one described for the local calendar.



If the calendar is programmed as "*a calendar for level signal*", it has to be added in the On/Off signal field.

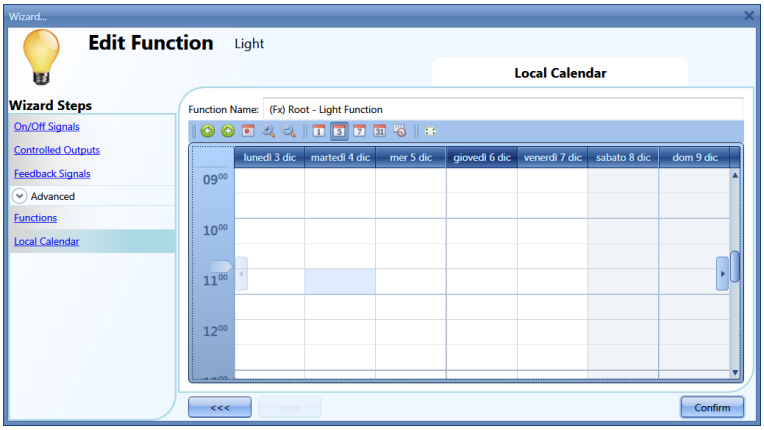
10.3.15 How to stop the automation

There are two ways of disabling the automation of the PIR sensors, Luxmeter and energy-save timer: the first one with the calendar, the second one with signals.
If an automation carried out with the energy-save timer is disabled, the energy-save timer is frozen and it will finish counting when the disabling is removed.

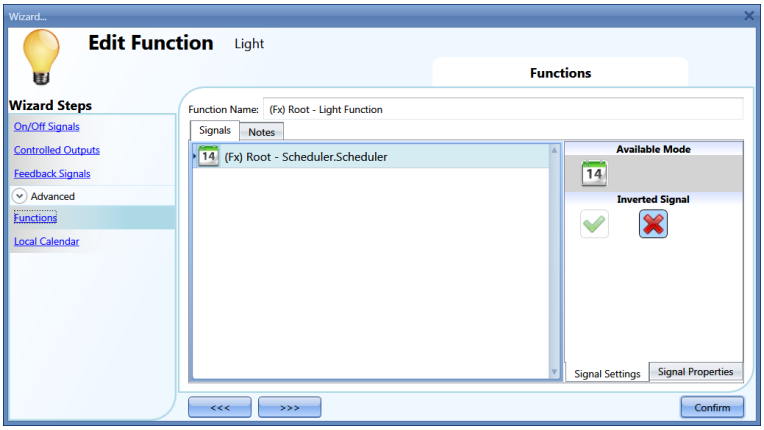
10.3.15.1 Disabling automation using the calendar

To disable the automation, the user can also select the local or the global calendar.
The local calendar has to be enabled, using the advanced functionality *Local calendar*, while the global calendar has to be added as a signal in the *Functions* menu of the advanced functionalities.

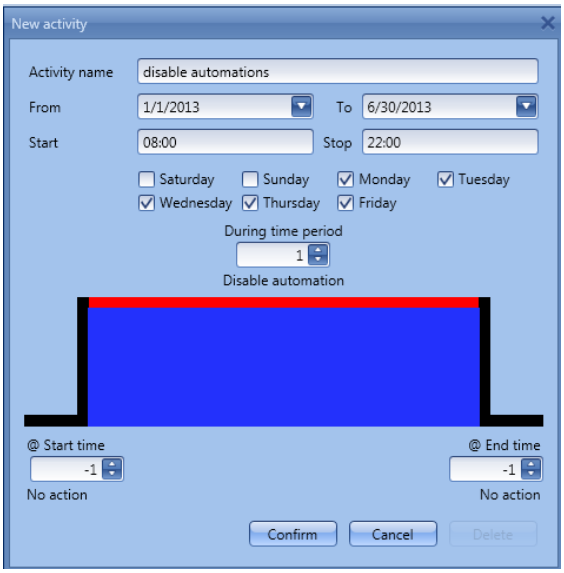
Local calendar



Global calendar



Once the type of calendar is set, the activity has to be defined.

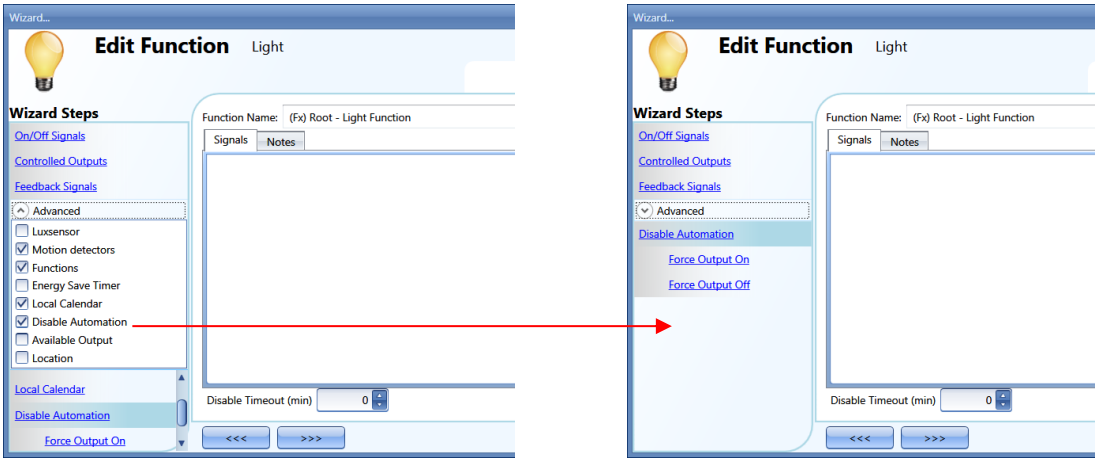


Level activity has to be selected.

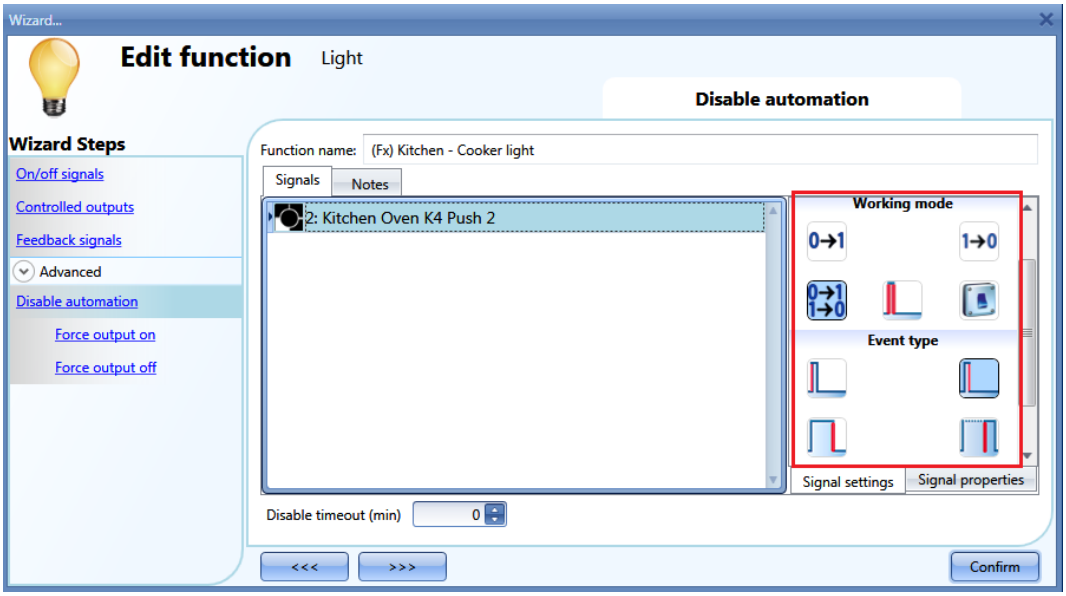
According to the activity set in the figure above, the automation of the light by PIR sensors and luxmeter will be stopped in December, from 8 to 10 o'clock in the evening, from Monday to Friday. During this period the light can be switched on/off only manually.

10.3.15.2 Disabling automation using signals

The second way of disabling automation is by using signals: please check *Disable automation* in the *Advanced* section.



The required signal has to be added by double clicking in the Signal window.



If a push button type is selected, the working mode, marked by the red rectangle, follows the table below.

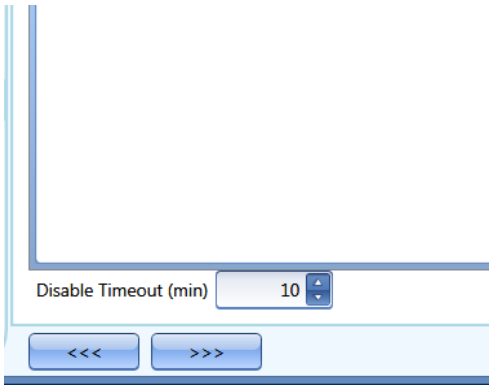
	Event type			
Working mode				
0→1	As soon as the push button is pressed, the automation is disabled.	After a short press (less than 1 second), the automation is disabled at the release of the push button.	After a long press, the automation is disabled at the release of the push button.	After a very long press, the automation is disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automation is enabled again.	After a short press (less than 1 second), the automation is enabled again at the release of the push button.	After a long press, the automation is enabled again at the release of the push button.	After a very long press, the automation is enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the automation will be enabled/disabled in toggle mode.	After a short press (less than 1 second), the automation will be enabled/disabled in toggle mode at the release of the push button.	After a long press, the automation will be enabled/disabled in toggle mode at the release of the push button.	After a very long press, the automation will be enabled/disabled in toggle mode at the release of the push button.
	The automation will be enabled/disabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the settings of the working mode are as follows:

	Event type	
Working mode	**Signal activated**	**Signal deactivated**
0→1	The automation is disabled	No action
1→0	The automation is enabled again	No action
0→1 1→0	The automation is disabled/enabled in toggle mode	No action
	The automation is disabled/enabled in toggle mode	The automation is disabled/enabled in toggle mode
	The automation is disabled	The automation is enabled

The user might need to set a time after which the automation is enabled again even if the selected signal is still active: to achieve this the Disable timeout field should be set.

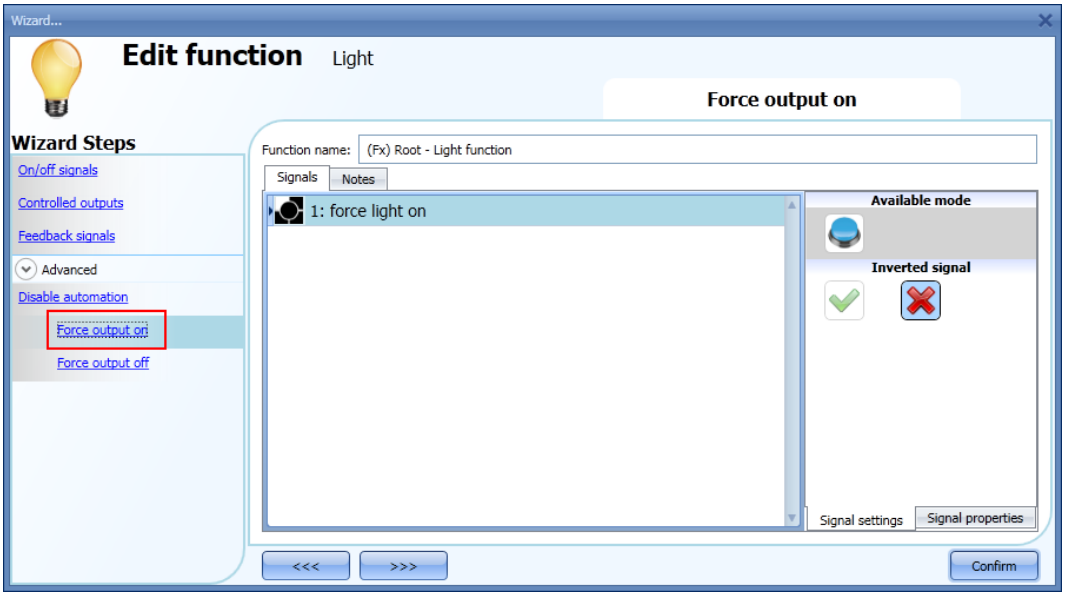
To access the stop automation functionalities, they must be enabled in the Advanced section field.
If the user needs to be sure that 'disable' is automatically removed, the *disable automation timer* should be set.



The timer starts every time the disable status becomes activated: the disable status is automatically deactivated as soon as the time has elapsed.
In the figure above a disable timeout of 10 minutes is set. The maximum delay is 59minutes.

10.3.16 How to force the output ON

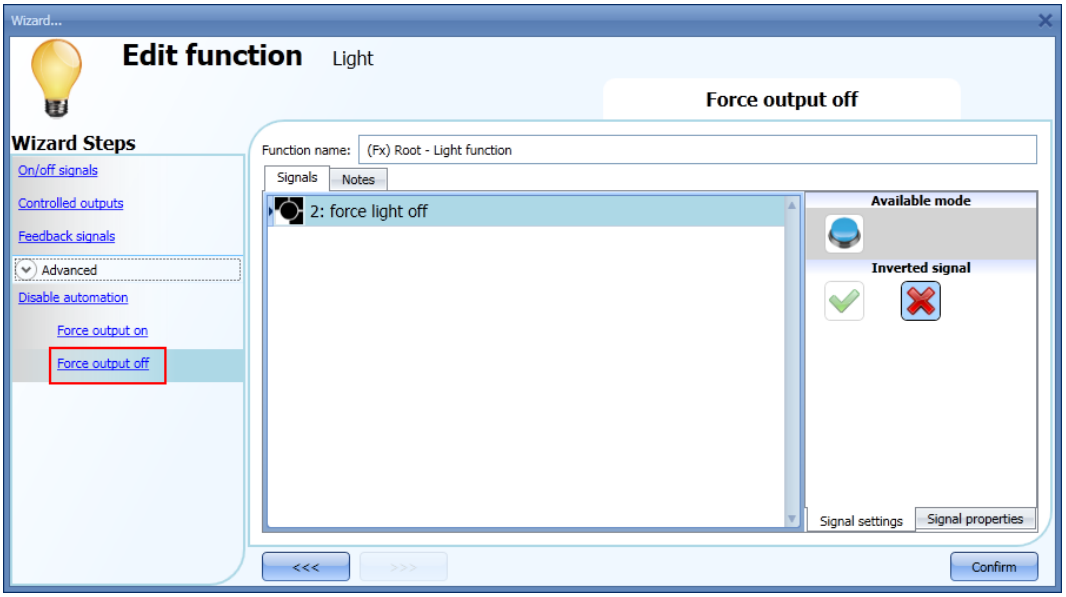
If the user wants to force the output on regardless of all other signals used in the function, the *Force output on* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output on*, then double click on the signal window and select the right signal to use.



Each signal used in the *Force output on* window works as a level signal: until the signal is active, the light function is forced on.
When the *force output on* signal is activated and the *force output off* signal is activated, the force to output on signal has the priority.

10.3.17 How to force output off

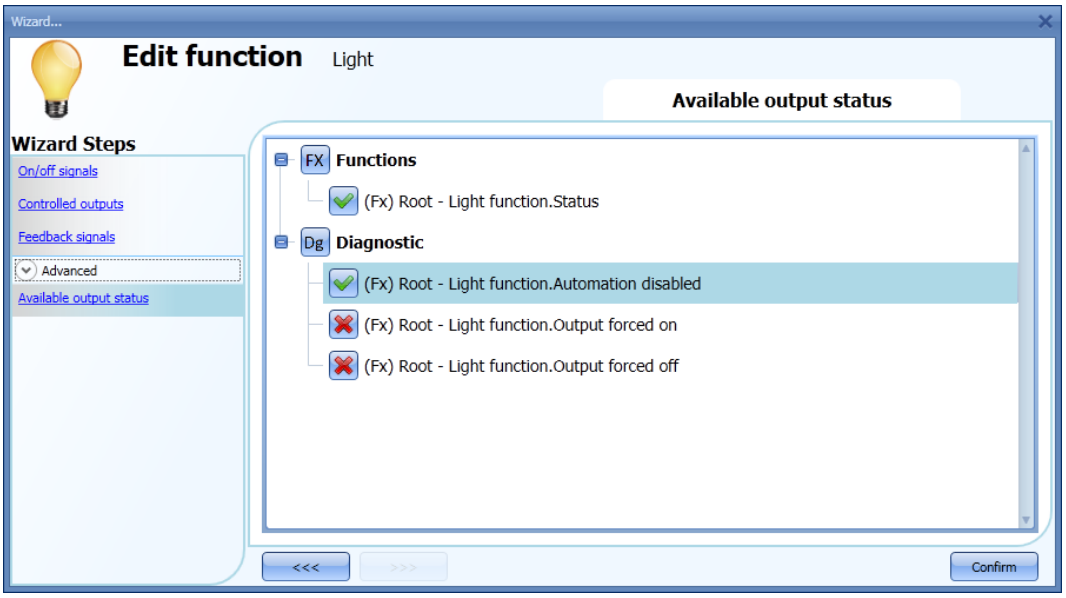
If the user wants to force the output off regardless of all other signals used in the function, the *Force output off* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output off*, then double click on the signal window and select the right signal to use.



Each signal used in the *Force output off* window works as a level signal: until the signal is active, the light function is forced off. When the *force output on* signal is activated and the *force output off* signal is activated, the force to output on signal has the priority.

10.3.18 How to enable the function diagnostic

In the *Available output status* the user can decide which information from the light function he needs to use in the configuration of his system, for example as an input in another function or in a sequence. Like all the other advanced functionalities, this one also has to be enabled.

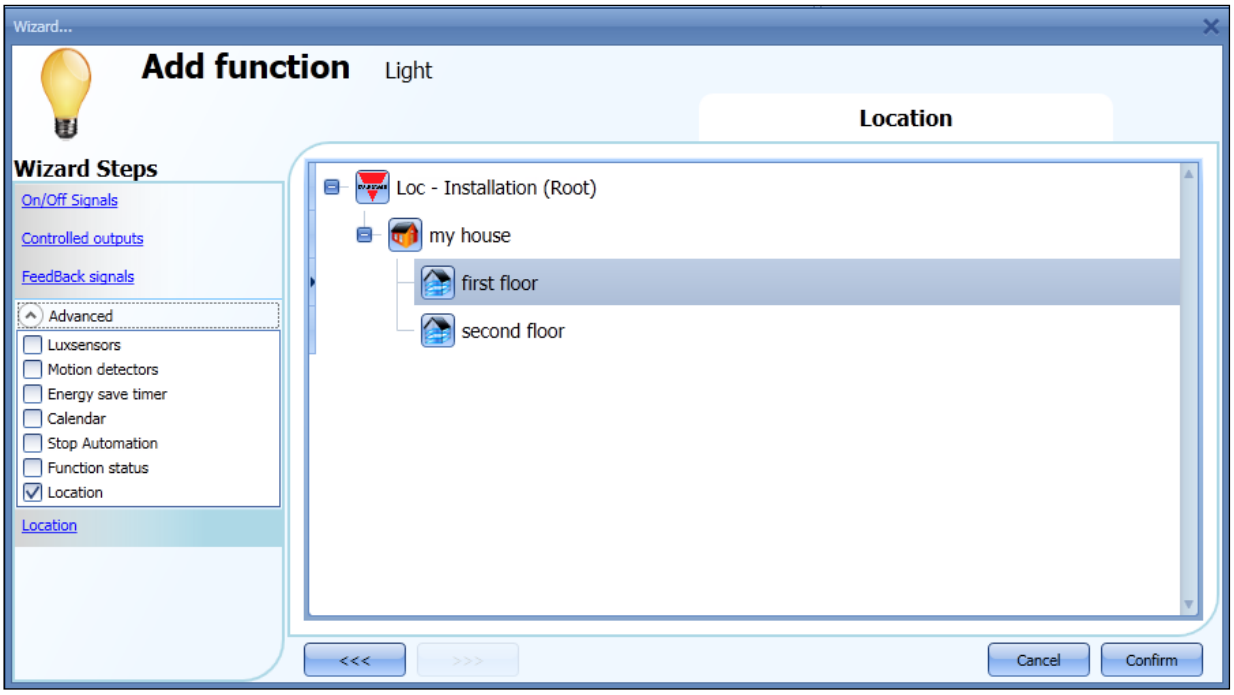


The statuses that can be enabled and used in other functions are:

- **Automation disable**: indicates if the automations are disabled.
- **Output forced on**: indicates if the light is forced in the On status.
- **Output forced off**: indicates if the light is forced in the off status .

10.3.19 How to change the location of a function

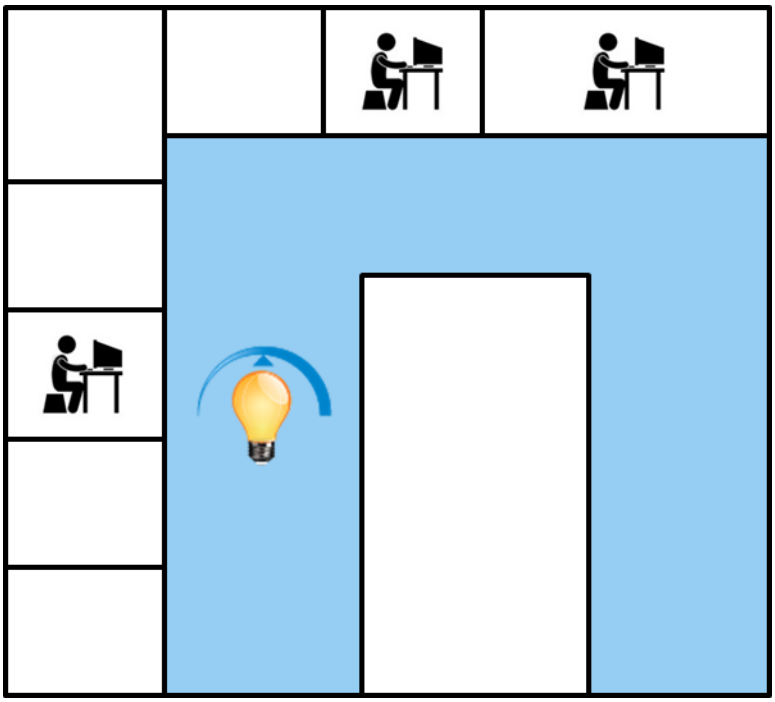
To access the *Location* menu, the user should enable it in the advanced functionalities.
From this window the location where the function is placed can be changed by simply selecting and clicking on the required one.



10.3.20 Corridor lighting function

This function is also an enhancement (additional option) to the *Light* functions.

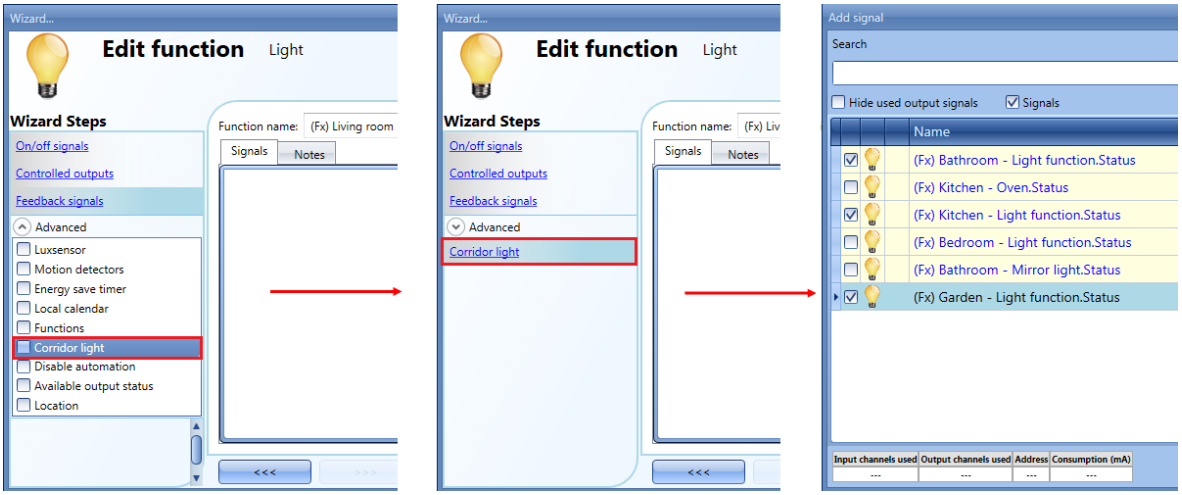
As shown in the picture below, the main aim of the Corridor lighting function is to control the lighting in the corridor, based not only on the presence of people in the corridor, but also on the presence of people in the adjacent offices. If there are people present in one or more offices, the corridor light is not allowed to switch OFF.



10.3.20.1 How to set the Corridor lighting function

To set up a *Corridor lighting* function the user should select one of the existing *Light Functions* present in the project.

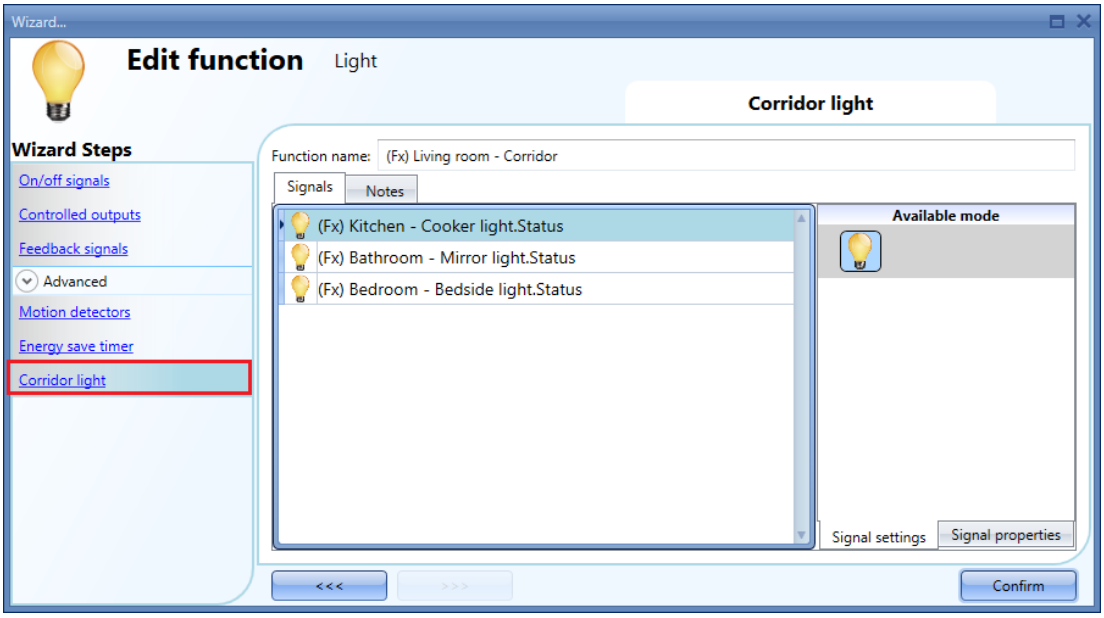
Since this is an Advanced functionality, the *Corridor light* field has to be enabled in the A*dvanced* section.



The Corridor light menu will appear. After selecting it, by double clicking on the Signals window the list of available signals will appear.

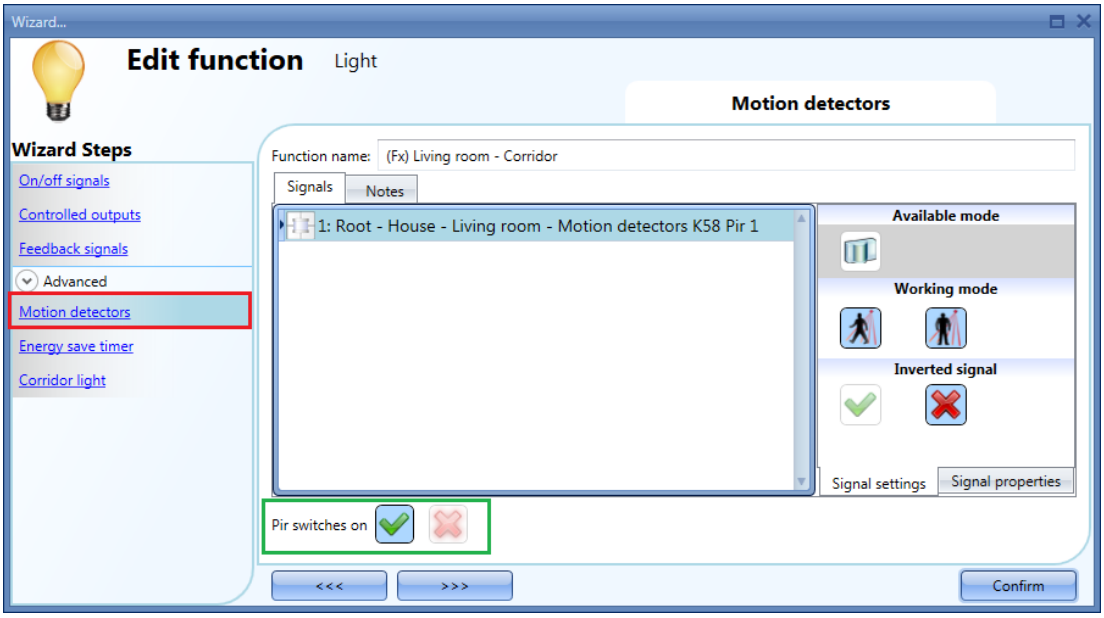
As shown in the picture above, the user may choose which offices he wants to include in the function, by selecting the relative signals: they can be function status or physical signals.
If more than one signal is selected in this window (you can choose up to 50 signals), the function will be activated/deactivated if at least one signal is active (the system does the logical OR).

The control of lighting in the corridor is based not only on the presence of people in the corridor, but also on the presence of people in the adjacent offices. If there are people present in one or more offices, the corridor light is not allowed to switch OFF.



10.3.20.2 How to turn a light ON with PIR sensors combined with Corridor light function

If the *Corridor light* option is selected, the user must select at least one signals which he wants to include in the function in the Advanced section.
This will be used by the *corridor light* function to detect if there is “people presence” in the adjacent offices. The signals added in this field are used to keep the light ON but they are not used to switch the light ON. Anyway as usual, a PIR signal can be used to switch the light ON in the *Motion detector* field and the user should select the green “V” in the option “Allow the PIR to switch on”. In this way, each time the PIR detects movement, the light will automatically be switched ON.



If the office lighting is not PIR-based, the user should use "*light function output*", as shown in the previous paragraph.

10.3.20.3 How the Corridor light function works

There are different possible scenarios, depending on the presence conditions in the offices adjacent to the corridor.

10.3.20.3.1 Presence of people in offices is TRUE

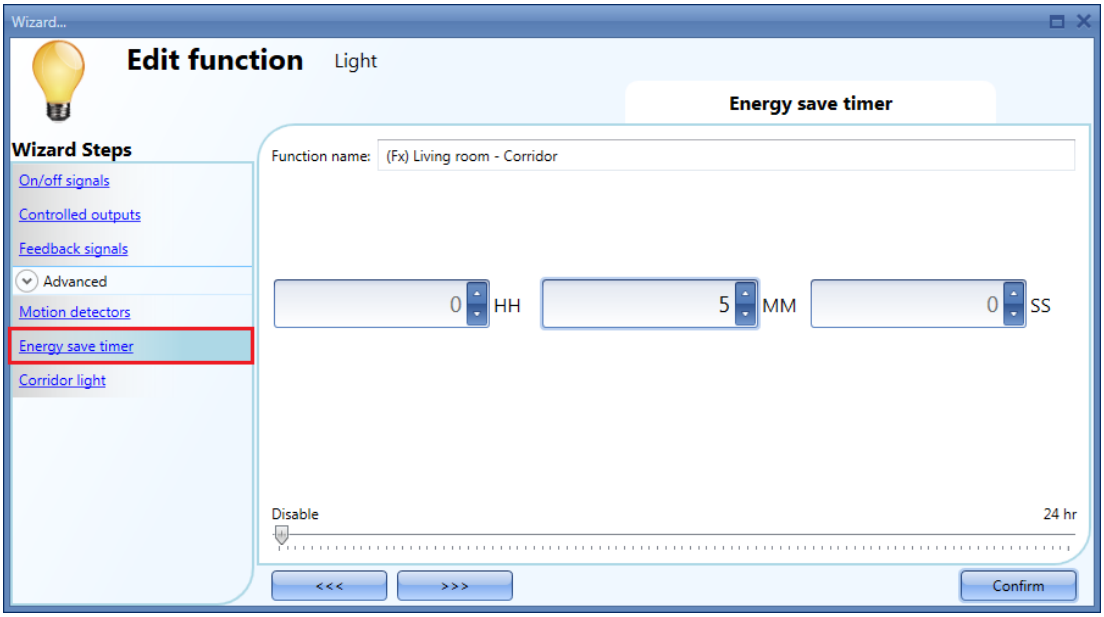
In this situation, one or more of the selected *PIRs* are ON or one or more of the selected *light functions* are ON

As long as "people presence" is TRUE, the corridor light function still ON.

10.3.20.3.2 Presence of people in offices is FALSE

If an energy save timer is used to switch the light OFF it starts counting down when all the signals in the field *Motion detector* and *Corridor light* are OFF: this means people presence neither in the corridor nor in the adjacent offices.

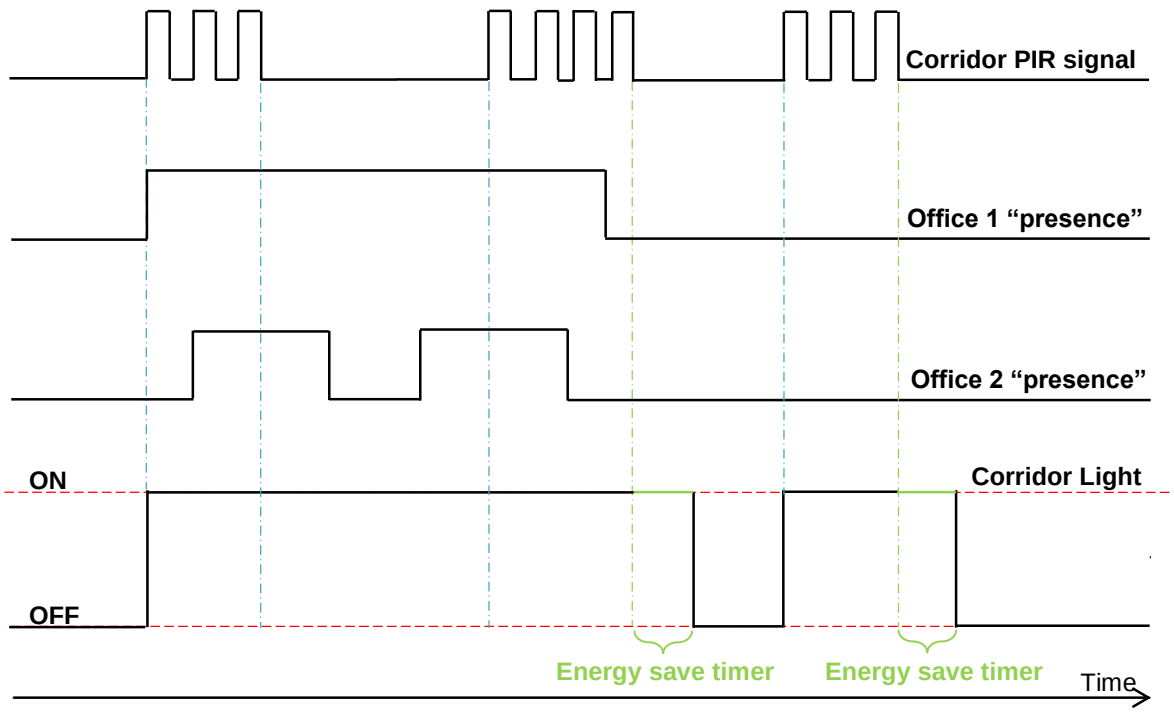
In the example shown below, while the user is in the corridor the light stays ON and the energy-save timer is continuously reloaded; when no more people are detected in the corridor and in the adjacent offices, the energy-save timer starts counting and the light is switched OFF. In the example shown below an energy save timer of 5 minutes is set by the user.



In the example shown below, a PIR is present in the corridor: when it detects any movement, the corridor light switches ON and remains until the PIR detects any movement in the corridor.

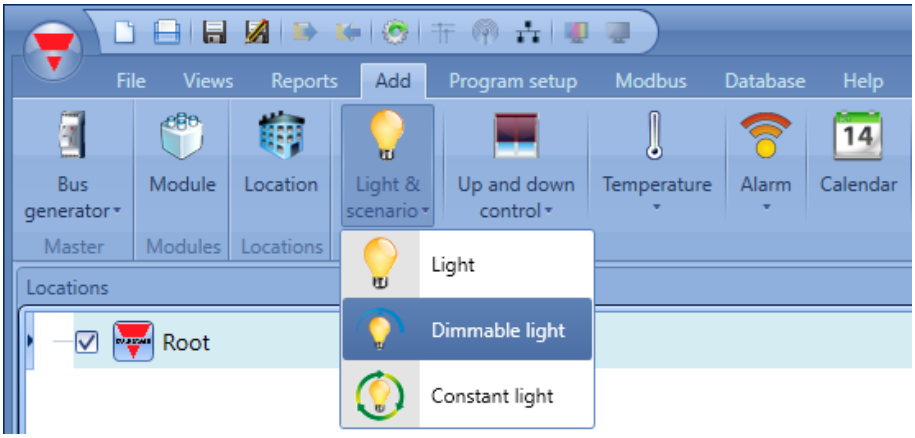
If the PIR doesn't detect any movement and the light statuses of all adjacent offices are OFF, the corridor light goes OFF as set by the user in the *Energy saver timer* field.

When the PIR in the corridor detects any movement, the corridor light switches ON and the "people present" operation resumes.



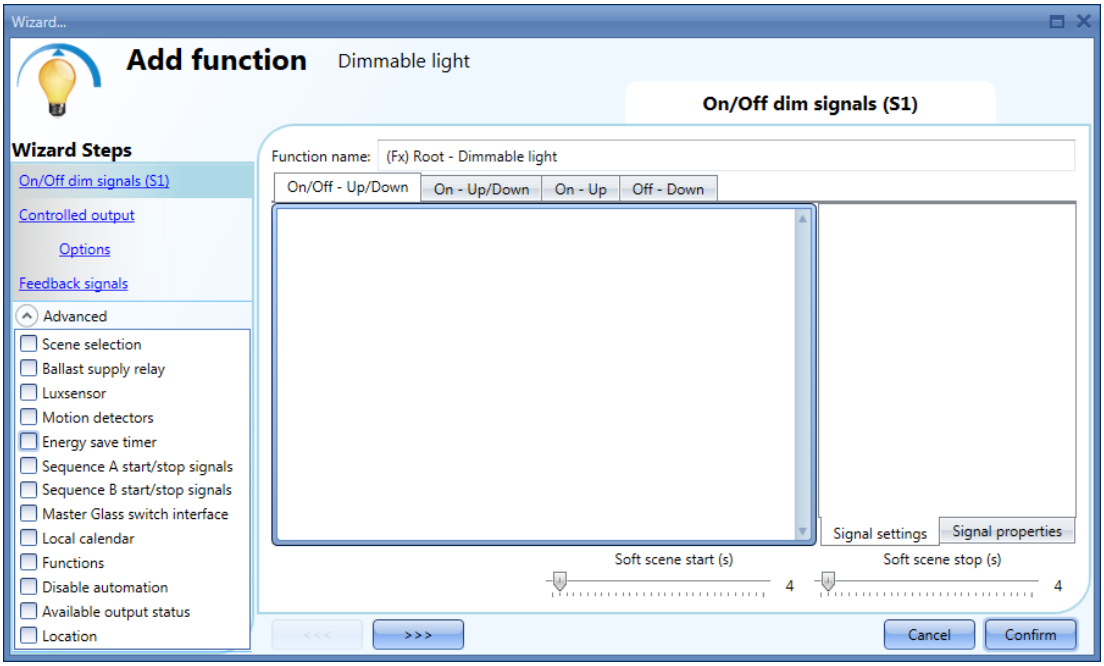
10.4 How to set a Dimmer function

This function allows the installer to manage one or more lights at the same time.
The user can either configure a basic function to switch the light on /off and adjust the light intensity, or implement an automated system by programming the relevant objects of the function by using the *Advanced* section.
To set up a dimmable light function the user should select *Light Functions* from the Add menu, then select *Dimmable Light* (see picture below). The new function will be added into the selected location.
The wizard relevant to a Dimmable Light function can also be opened by typing Alt+D. (See table of short cuts).



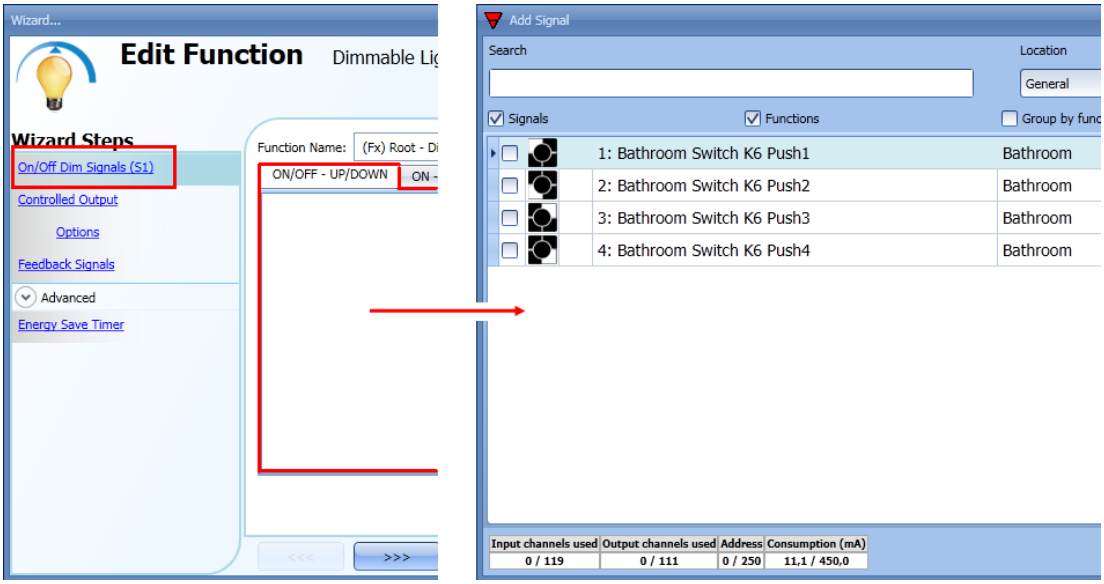
This function manages one or more dimmer outputs controlled by one or more input commands. The command might be a real signal such as a pushbutton, a function or a remote command (Webserver, sms, email, Modbus TCP/IP)

The automation of the Dimmable light function is managed by accessing the *Advanced* section (see picture below).
In the Advanced section the user can select different ways of controlling the dimmable light: according to the ambient light, the presence of people, with timers and/or schedulers. Up to 5 different predefined scenarios can be set.

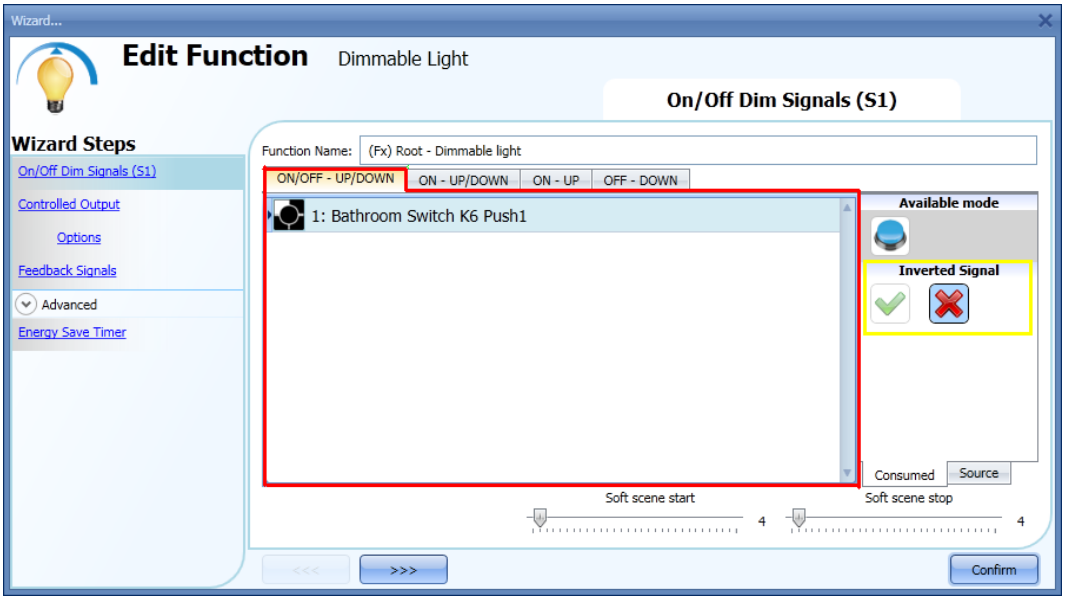


10.4.1 How to dim a light with input commands

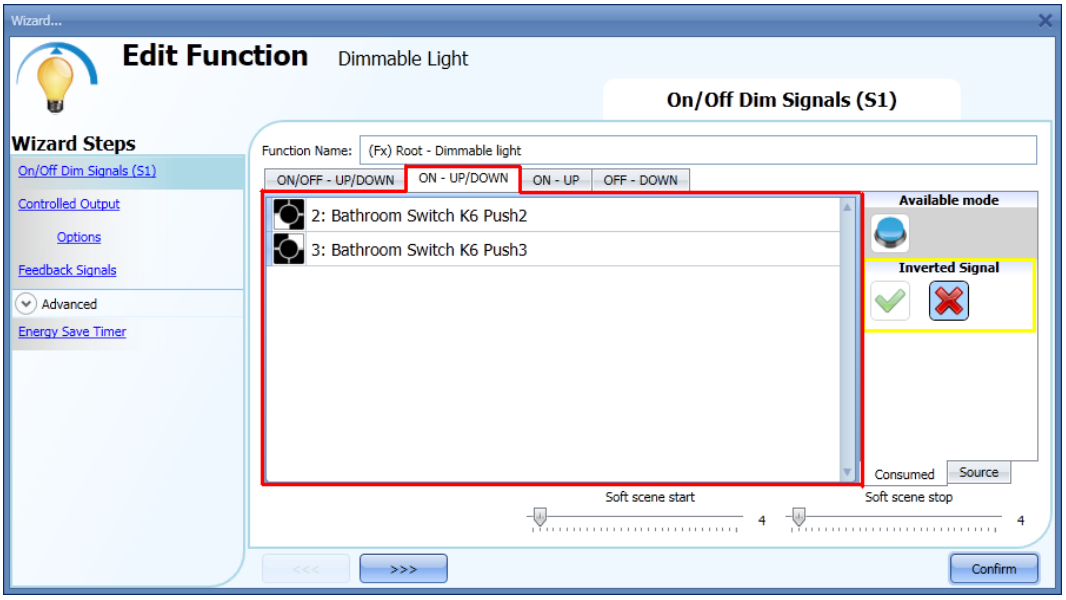
First of all, the user has to add the input signals to control the dimmable light in the *On/Off Dim Signals (S1)* field.
The input signal can be a pushbutton or a switch and can manage both the ON/OFF switching and the light level adjustment in different ways, according to the field where it is added.
The input signal window is split into four sections: each section has different ways to manage the output. The user should select the required one, then double click on it and add the input signal from the list.



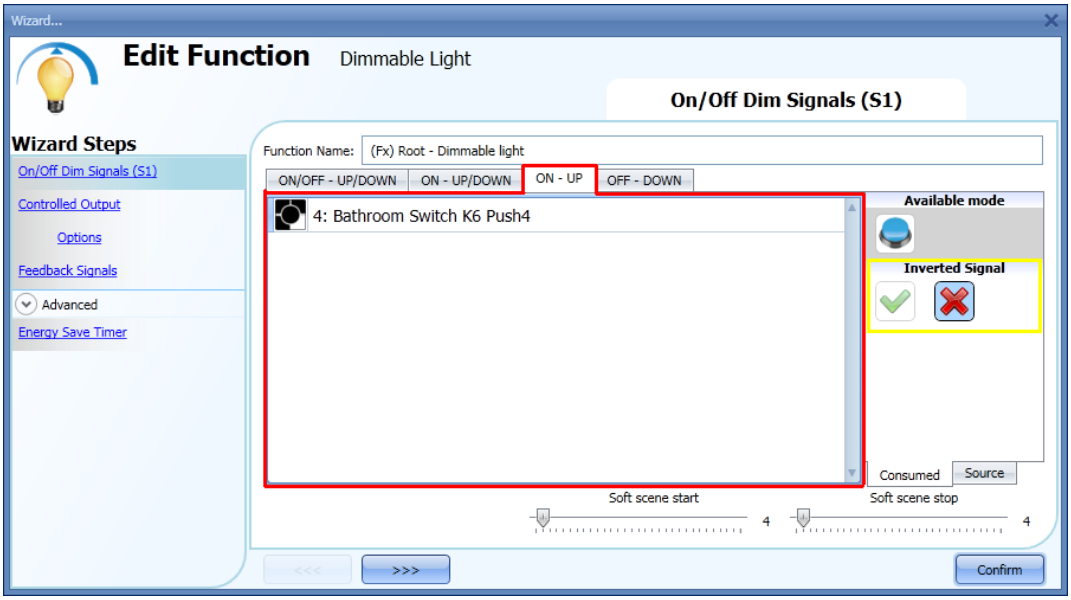
Section 1: ON/OFF-UP/DOWN
The light is switched On/Off with a short click (input active for a period shorter than 1 second). When the light is on, by keeping the input active for more than 1 second, the light level starts going up/down.
Each time the light reaches the maximum/minimum level, the ramp is inverted. In the Signal setting window, the user can also enable the reversing of each added signal (See yellow rectangle).



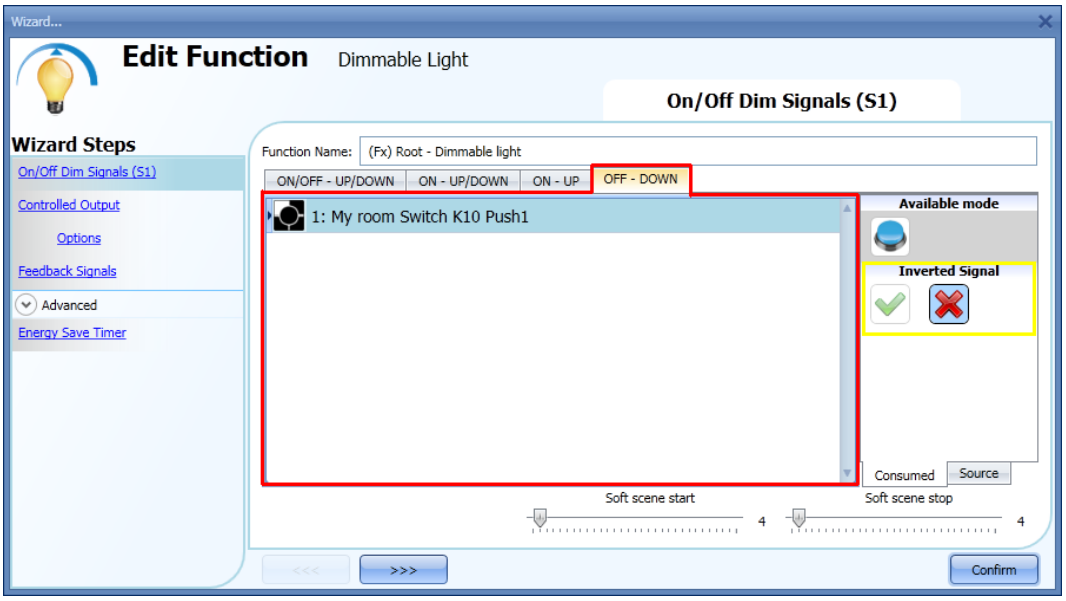
Section 2: ON-UP/DOWN
The short click only switches on the light (input active for a period shorter than 1 second). When the light is on, by keeping the input active for more than 1 second, the light level starts going Up/Down. Each time the light level reaches the maximum/minimum level, the ramp is inverted. In the Signal setting window, the user can also enable the reversing of each added signal (See yellow rectangle)



Section 3: ON-UP
The short click only switches on the light (input active for a period shorter than 1 second). When the light is On, by keeping the input active for more than 1 second, the light level will start going up until the maximum level is reached. In the Signal setting window, the user can also enable the reversing of each added signal (See yellow rectangle).



Section 4:OFF-DOWN
The short click only switches off the light (input active for a period shorter than 1 second). When the light is On, by keeping the input active for more than 1 second, the light level will start going down until the minimum level is reached. In the Signal setting window, the user can also enable the reversing of each added signal (See yellow rectangle)

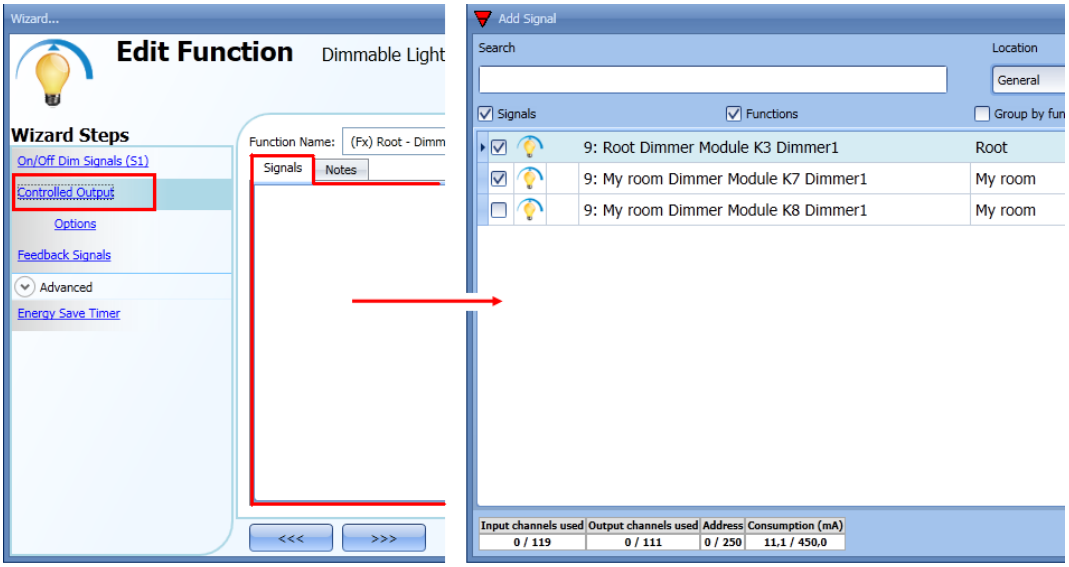


The user can set the soft scene start/stop: this is a common setting used for all input signals.

Soft scene start (s):
the user can define the ramp time for switching on the light (see Scene selection paragraph for more details).
Soft scene stop (s):
the user can define the ramp time for switching off the light (see Scene selection paragraph for more details).

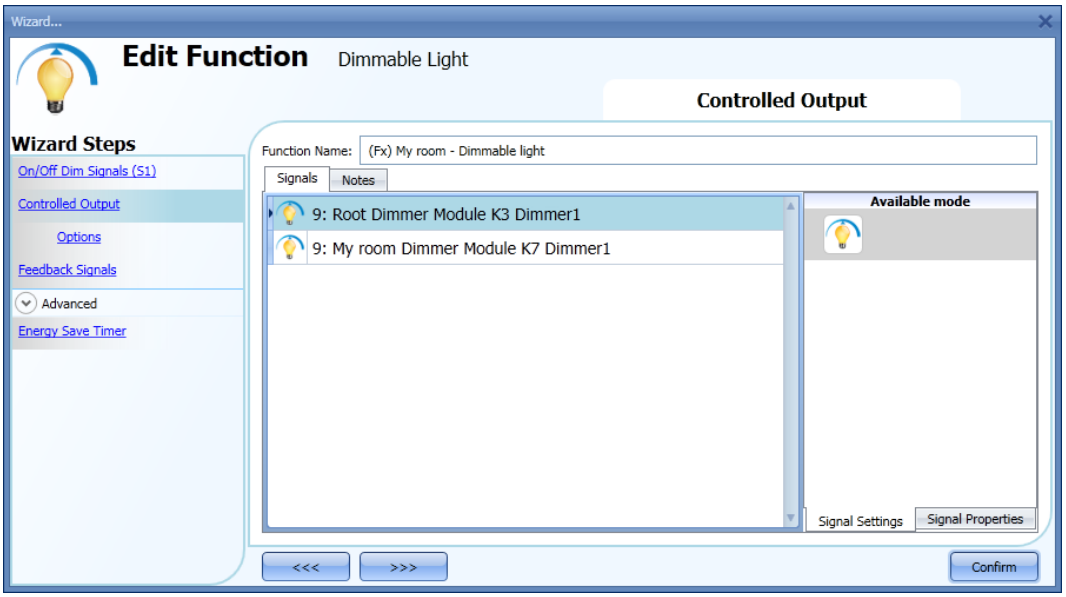
10.4.2 How to select a dimmable output

To select the output signal that is controlled by the dimmable light function, click on *Controlled output* and then double click on the Signal window. Once the output window is opened, select the relevant outputs from the list.



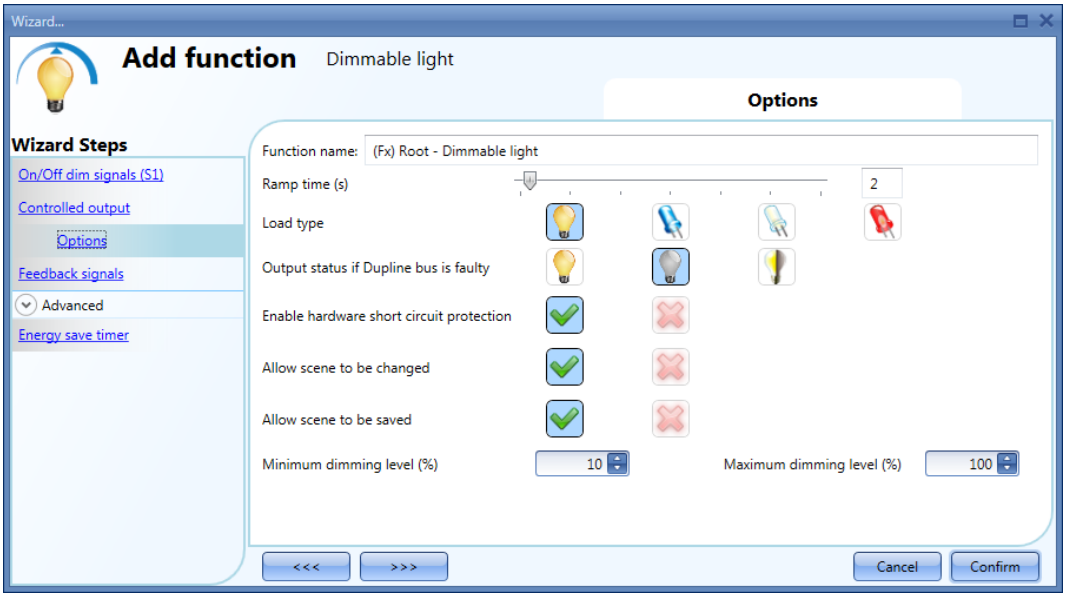
The signals available in the output window are only Dimmer output type.
Up to 100 signals can be chosen and they are managed in parallel.
If an output from a 1-10V dimmer is selected, a relay output has to be connected to the relevant ballast to control ON and OFF. The relay output should be added in the advanced field *Ballast supply relay*.

N.B: If more outputs are selected, they should be placed in the same Dupline network.
Dimmer outputs placed in different networks cannot be added together in the same Dimmable function.



10.4.3 How to change settings and load type of the output

To select the settings of the output, the user has to click on *Options*.
If more outputs are selected, the settings are applied to all of them.



Ramp Time (s):
This is the required time to go from 0% to 100% when at least one input is active.
For example, if 10 seconds is set, once the light is on, this is the time the light will take to go from 0% to 100% and from 100% to 0% when the push button is kept pressed.
The ramp time is the same for all the dimmer outputs belonging to the same function.

Load type:
In this field the type of load connected to the dimmer module has to be selected. The load can be a standard RLC one (halogen lamp, inductive or electronic transformer, …), or a LED lamp.
The user can select from three different types of LEDs: please see the dimmer datasheet for more details on how to select the correct LED load.

Output status if the Dupline bus is faulty:
In this field the user can select the status of the output if the bus is faulty. The possible actions are: force the output into on status, force the output into the off status and memory: when the user selects memory, the output remains in the status it had before the loss of connection with the Dupline Bus. Each selected fail state action will be kept active until the Dupline bus signal returns.

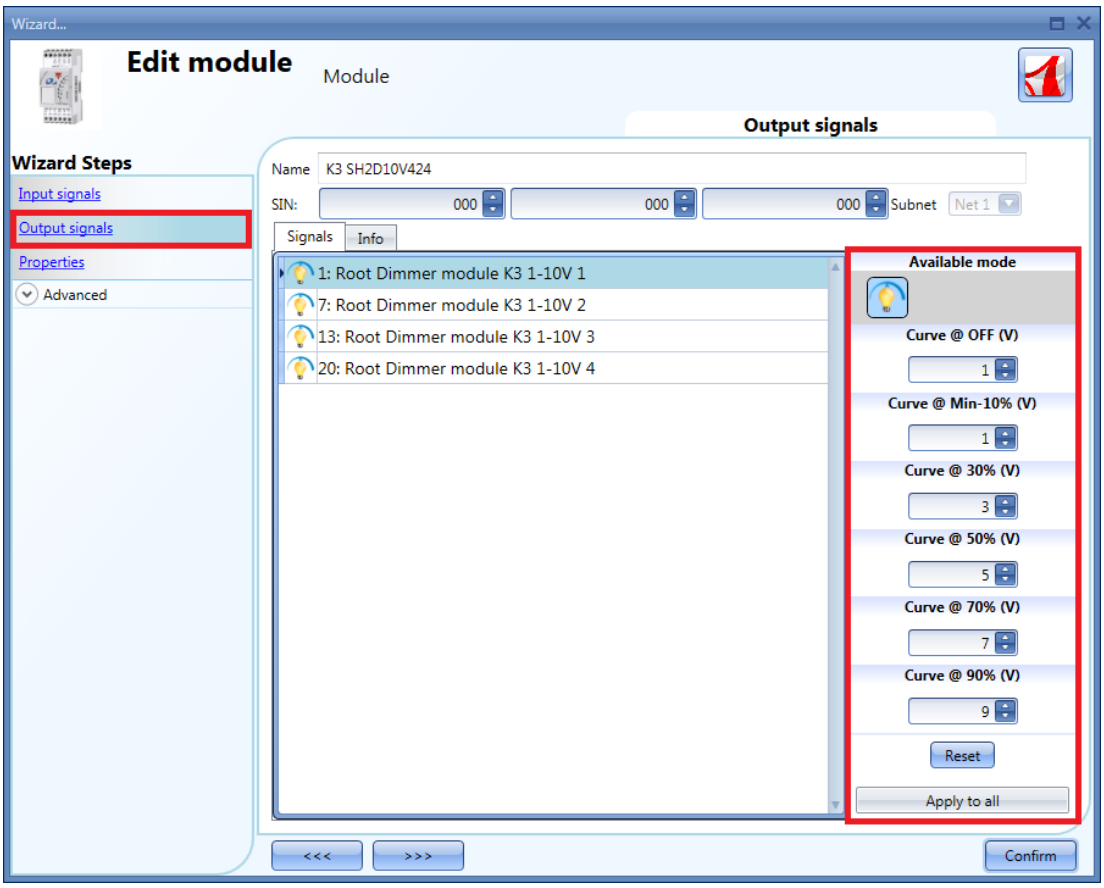
Enable hardware short circuit protection:
When this option is enabled (the green “V” is selected), the user will enable the hardware short circuit protection of the output Dimmer module. Otherwise (when the red “cross” is chosen), the user will disable the hardware protection.

Allow scene to be changed:
When this option is enabled (the green “V” is selected), the light level in all the scenarios will be modifiable when the input signal is activated. Otherwise (when the red “cross” is chosen), the user will not be able to change it (with the exception of scenario S1).

Allow scene to be saved:
When this option is enabled (the green “V” is selected), every time the dimmer is powered OFF, the current light level value is saved in the memory before the switching off of the module. Otherwise (when the red “cross” is chosen) the scenario is not saved and the power is switched OFF.

Minimum dimming level (%) and Maximum dimming level (%):
In the graph below the dimming curves are shown according to different type of loads.
By default, the minimum value that the dimmer can manage is 10% (values from 0 to 9 are reserved for scenarios), while the maximum value is 100%. These settings are valid for the 500 W dimmer.

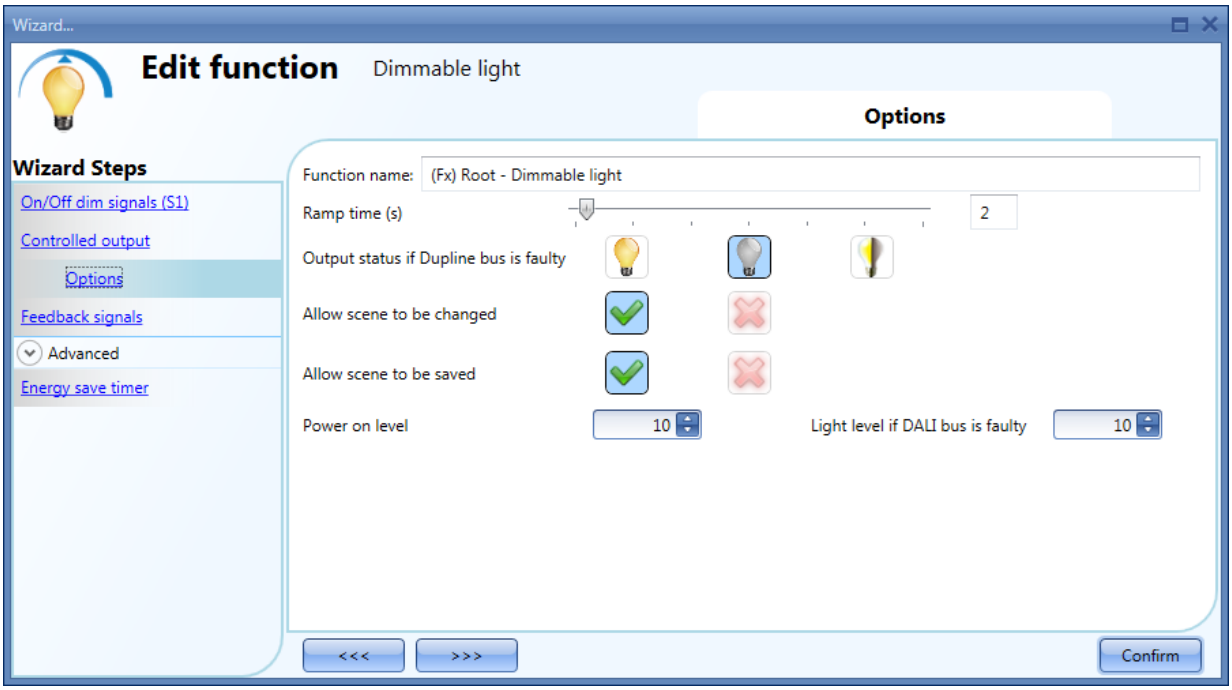
N.B. If the SH2D10V424 dimmer is used, the relevant options of load type, enable hardware short circuit protection, min/max value, are not shown in the field Options of the dimmer function. The minimum-maximum value has to be managed in the wizard of the module, in the menu Output signals (see picture below):



10.4.4 How to change settings and load type of the DALI output

If the SB2DALI230 dimmer is used, the settings *Load type*, *Enable hardware short circuit protection*, *Min/max value* are not shown in the field *Options* of the dimmer function.

The available settings are shown in the picture below:



Ramp Time (s):
This is the required time to go from 0% to 100% when at least one input is active.
For example, if 10 seconds is set, once the light is on, this is the time the light will take to go from 0% to 100% and from 100% to 0% when the push button is kept pressed.
The ramp time is the same for all the dimmer outputs belonging to the same function.

Output status if the Dupline bus is faulty:
In this field the user can select the status of the output if the bus is faulty. The possible actions are: force the output into on status, force the output into the off status and memory: when the user selects memory, the output remains in the status it had before the loss of connection with the Dupline Bus. Each selected fail state action will be kept active until the Dupline bus signal returns.

Allow scene to be changed:
When this option is enabled (the green “V” is selected), the light level in all the scenarios will be modifiable when the input signal is activated. Otherwise (when the red “cross” is chosen), the user will not be able to change it (with the exception of scenario S1).

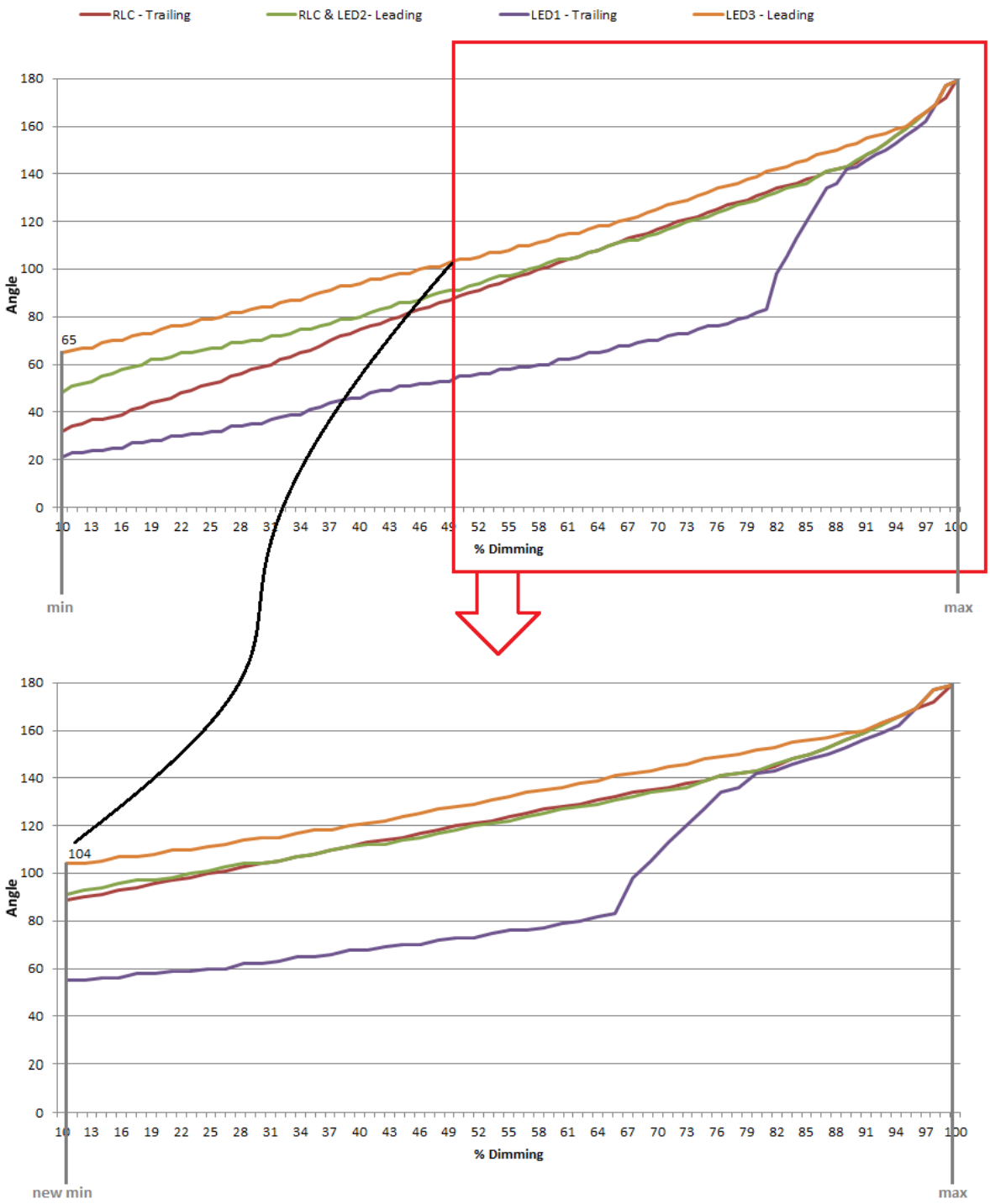
Allow scene to be saved:
When this option is enabled (the green “V” is selected), every time the dimmer is powered OFF, the current light level value is saved in the memory before the switching off of the module. Otherwise (when the red “cross” is chosen) the scenario is not saved and the power is switched OFF.

Power on Level:
In this field the user can select the value of the light when the ballast turns on waiting for the first DALI command.

Light level if DALI bus is faulty:
In this field the user can select the value of the output if the DALI bus is faulty. The output will be kept at the value set until the DALI bus signal returns.

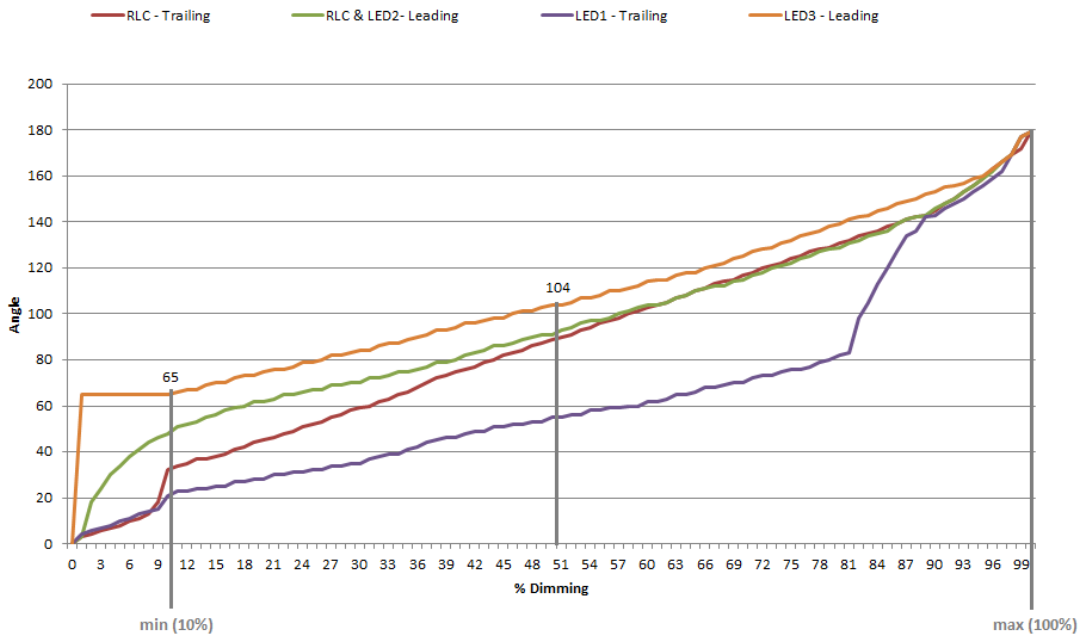
10.4.5 How to modify the minimum/maximum values of the dimming range

In the graph below, the minimum value has been changed to 50% and the new dimming output range is adapted according to the new minimum-maximum values; in this example the previous 50% now corresponds to the lower value the dimmer can manage (new min 10%), while the maximum level is still the same (100%).



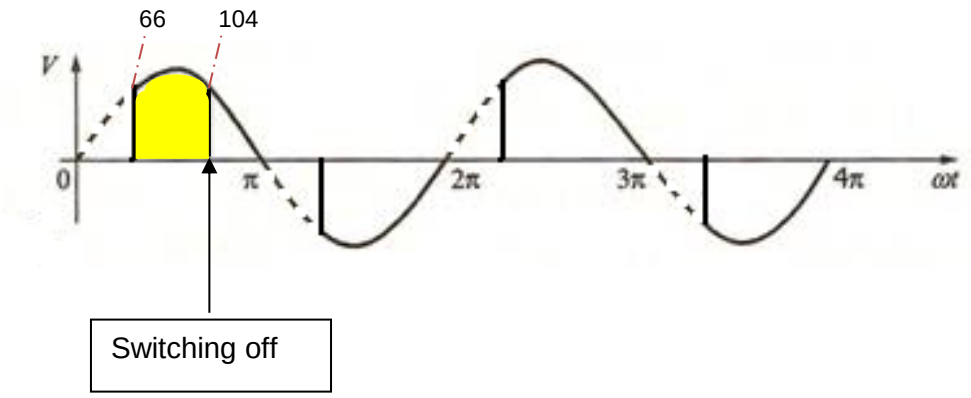
10.4.6 How the dimmer works

The technology used by SH2D500Wx230 is called phase cutting technology, which works by cutting off part of the mains voltage to modify the RMS voltage fed to the lamp. The brightness of the lamp can thus be controlled by increasing or decreasing the RMS voltage and thereby the power supplied to the lamp.



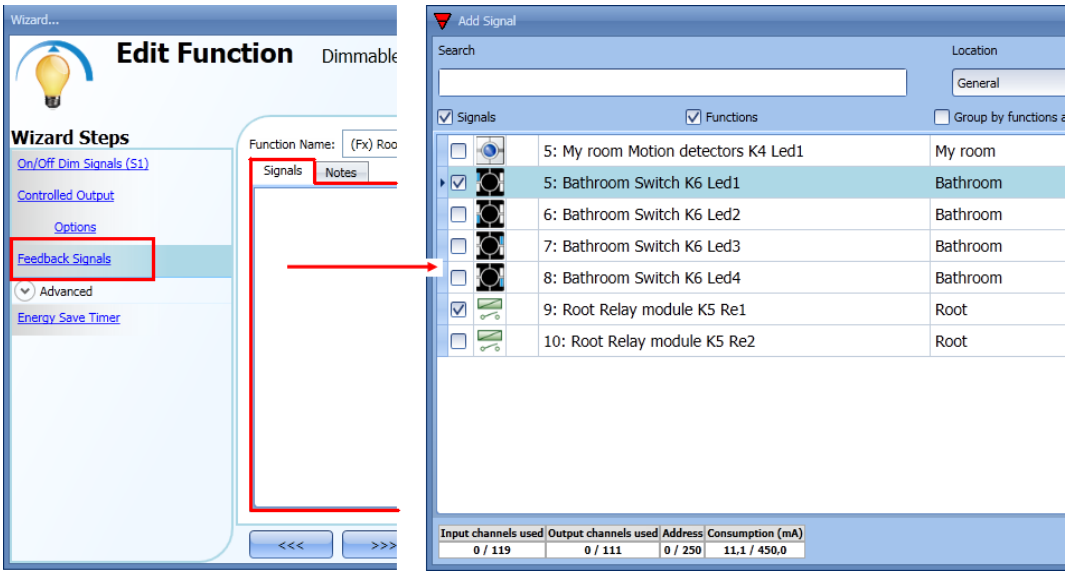
In this example the lighting circuit turns on at a phase angle of 66° (if 60° is the minimum angle to switch the lamp on, 66=60+10% is the minimum angle to switch it on without any flickering). Afterwards, if the minimum dimmer output is set at 50%, the minimum switching on angle will be 104° (see the graph above).
The maximum output is at 180°.

LEADING EDGE CUT OFF (L, R LOADS)

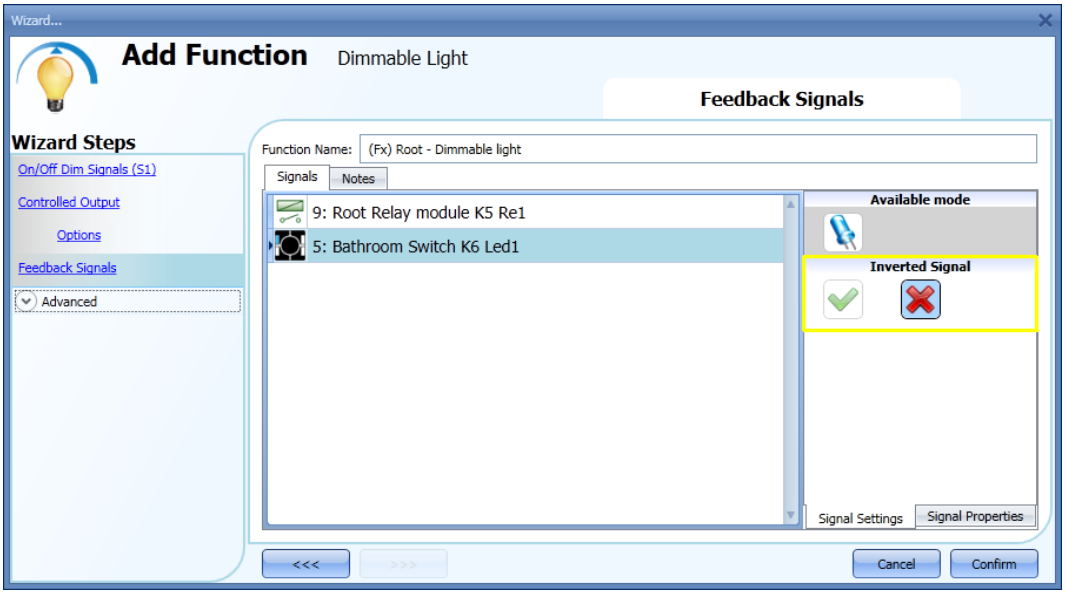


10.4.7 How to set a signal as feedback of the status of the function

To select the feedback signal that indicates the status of a function, click on *Feedback signals* and then double click on the Signals window.



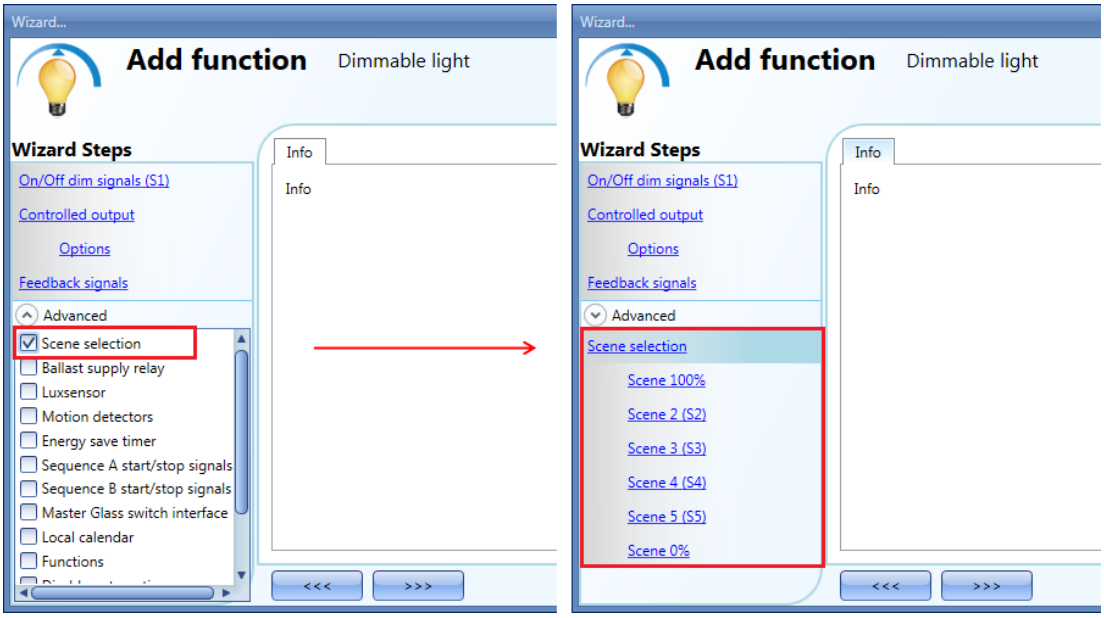
The signals available in this window are LEDs and relays.
Up to 50 signals can be chosen and they will be managed in parallel.
The logic of each feedback signal can be set as normal or inverted (see yellow rectangle).



10.4.8 How to set predefined scenarios with input signals

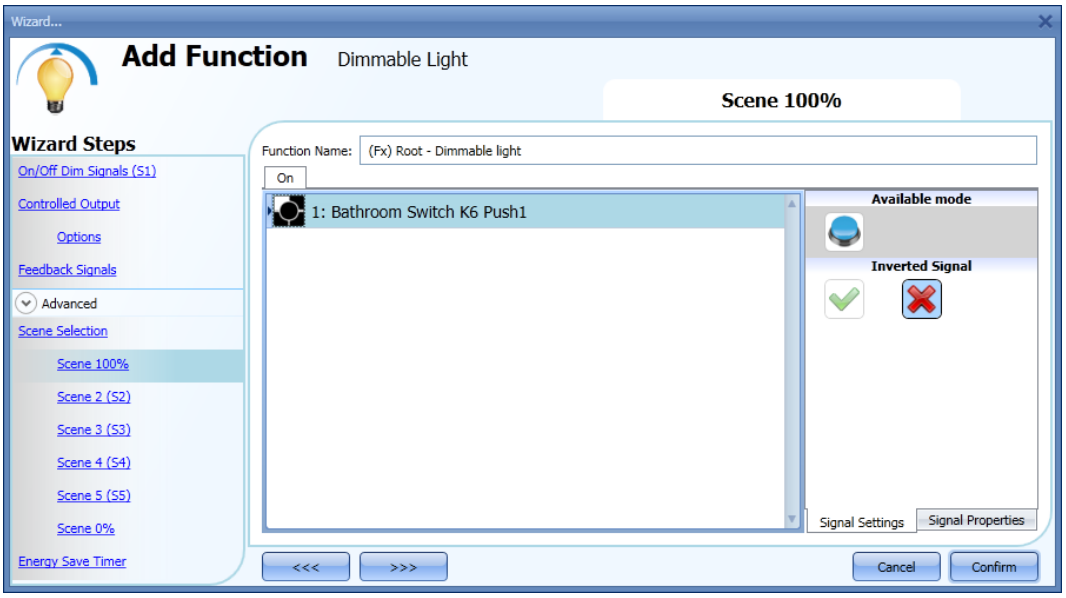
In this section, the user can customize different scenarios and select different input signals to activate them. There are two scenarios that are not editable (Scene 100% and scene 0%). Here the user can only set the signals that activate them.
All other scenarios are editable. The user can set different output percentages and different timings for the soft start/stop.
Each scenario can be activated manually by adding an input signal in the signals window, or it can also be activated by different automations such as PIR sensors, Lux sensors or Calendar.

The first thing to do to use different Scenarios is to enable them in the *Advanced* section.
The user should go to the advanced steps of the function and then check *Scene selection*. The user can select the scenario from the list, then with a double click in the signals field can add one or more inputs to activate the scenario.



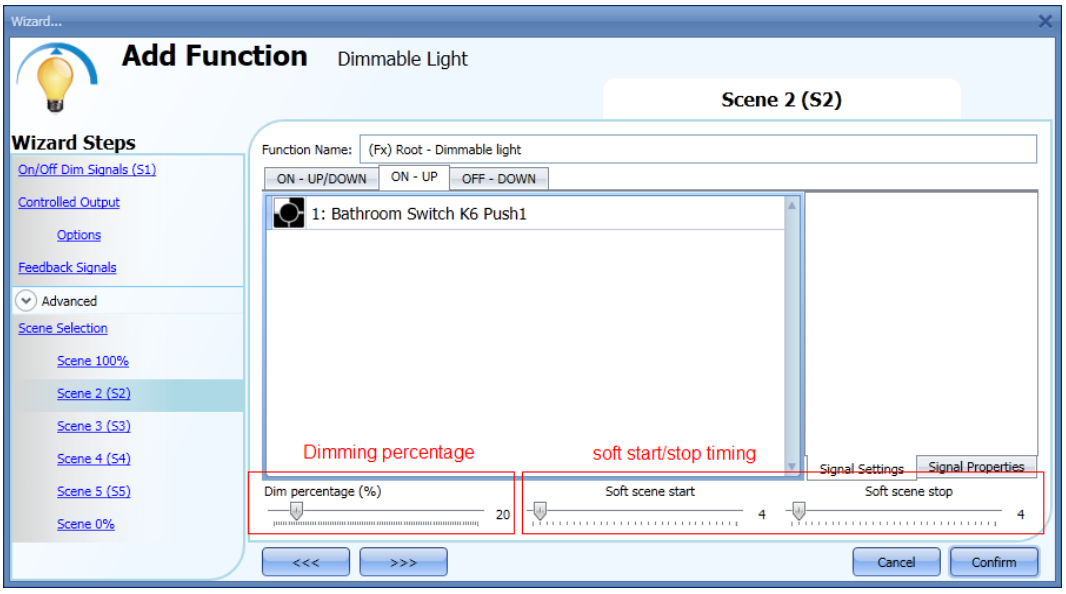
10.4.8.1 Scene 100%

When the scenario scene 100% is selected, all signals added in the window 'signals' will switch the light ON at the highest light level with a short press (input activated and deactivated within 1 s).
It is not possible to dim this scenario and the soft start and soft stop times are the same as scenario S1.
In the picture below, a simple example is shown of how to set a pushbutton to activate the scenario 100%.
Each time the pushbutton 1 is activated, the light level will be increased to the maximum value.



10.4.8.2 Scene 2 (S2)

The *Scene 2* window is split into three sections: each section has a different way to manage the output. The user should select the relevant sections, then add an input signal with a double click on the *Signal* window and then select the input from the list of available signals.



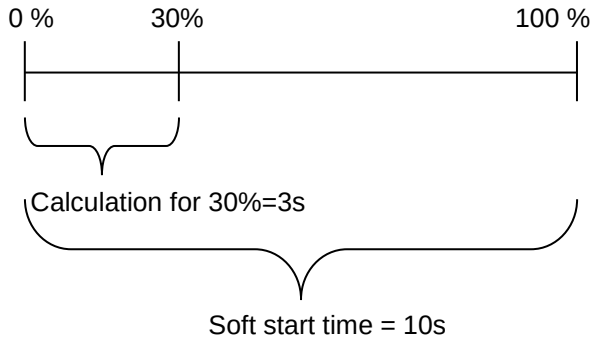
Section 1: ON-UP/DOWN
The light is switched On with a short press (input activated and deactivated within 1 s). When the light is on, by keeping the input active for more than 1 second, the light level will start going Up/Down until the input signal is active. In the Signal setting window, the user can also enable the reversing of each added signal.

Section 2: ON-UP
The light is switched On with a short press (input activated and deactivated within 1 s). When the light is on, by keeping the input active for more than 1 second, the light level will start going up. When the light level reaches the maximum level (100%), the dimming is stopped.

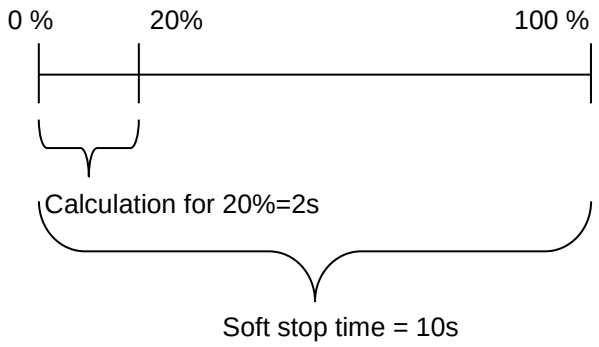
Section 3: OFF-DOWN
The light is switched Off with a short press (input activated and deactivated within 1 s). When the light is on, by keeping the input active for more than 1 second, the light level will start going down. When the light level reaches the minimum level (10%), the dimming is stopped.

Dim percentage (%)
The user can set the dimming percentage value when scenario 2 is activated.
This value is stored in the dimmer module but can be overwritten with a long press on the input signal (if the option “Allow to change scenes” is enabled in the Controlled output options).

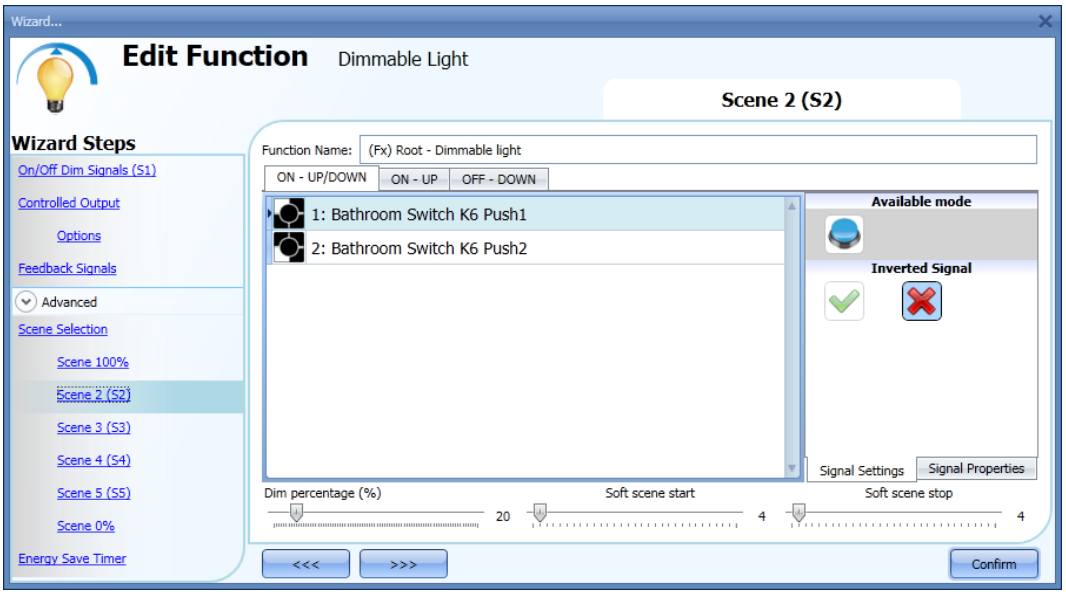
Soft scene start (s)
The soft start time is the time the dimmer output takes to change the output from a value of 0% to a value of 100%. This time is adapted according to the percentage value set in the scenario.
For example, if the soft start time of scene 2 is set to 10s and the percentage is set to 30%, the light will be switched on at 30% after 3s (calculation 100:10=30:x).



Soft scene stop(s)
The soft stop time is the time the dimmer output takes to change the output from a value of 100% to a value of 0%. This time is adapted according to the percentage value set in the scenario.
For example, if the soft stop time of scene 2 is set to 10s and the percentage is set to 20%, the light will be switched off after 2s (calculation 100:10=20:x).

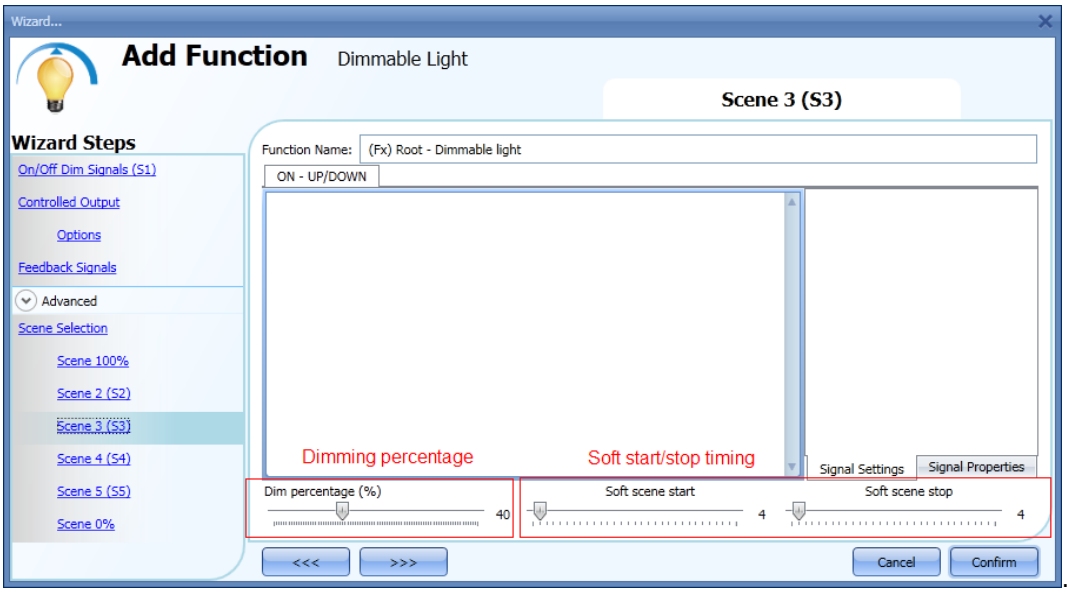


In the example below, the user has set two pushbuttons for the manual activation of scenario 2, the dimming percentage is set to 20% and both soft start and stop are set to 4 s.
Each time the pushbuttons 1 or 2 are activated with a short press, the light level will be set to 20 %. If they are activated with a long press, the light level starts dimming Up/Down.



10.4.8.3 Scene 3 (S3)

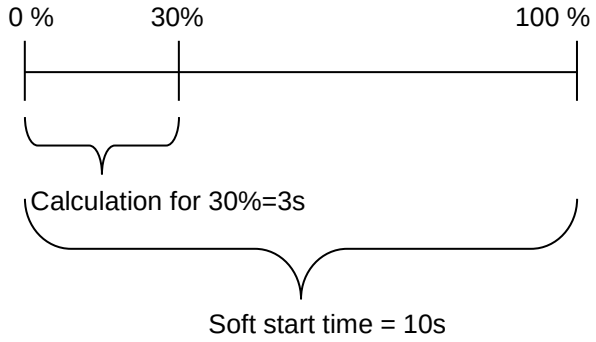
The *Scene 3* window has a single section. The user can add one or more input signals with a double click on the *Signal* window and then select the input from the list of available signals.



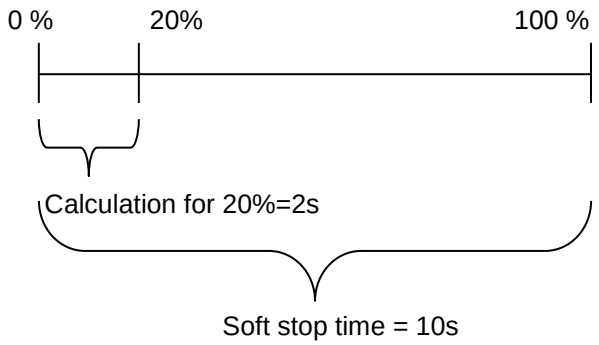
Section 1: ON-UP/DOWN
The light is switched On with a short press (input activated and deactivated within 1 s). When the light is on, by keeping the input active for more than 1 second, the light level will start going UP/Down. Every time the light level reaches the maximum/minimum level, it starts going UP/Down again until the input signal is active. In the Signal setting window, the user can also enable the reversing of each added signal.
Dim percentage (%)

The user can set the dimming percentage value when scenario 3 is activated.
This value is stored in the dimmer but can be overwritten with a long press on the input signal (if the option *Allow to change scenes* is enabled in the Controlled output options).

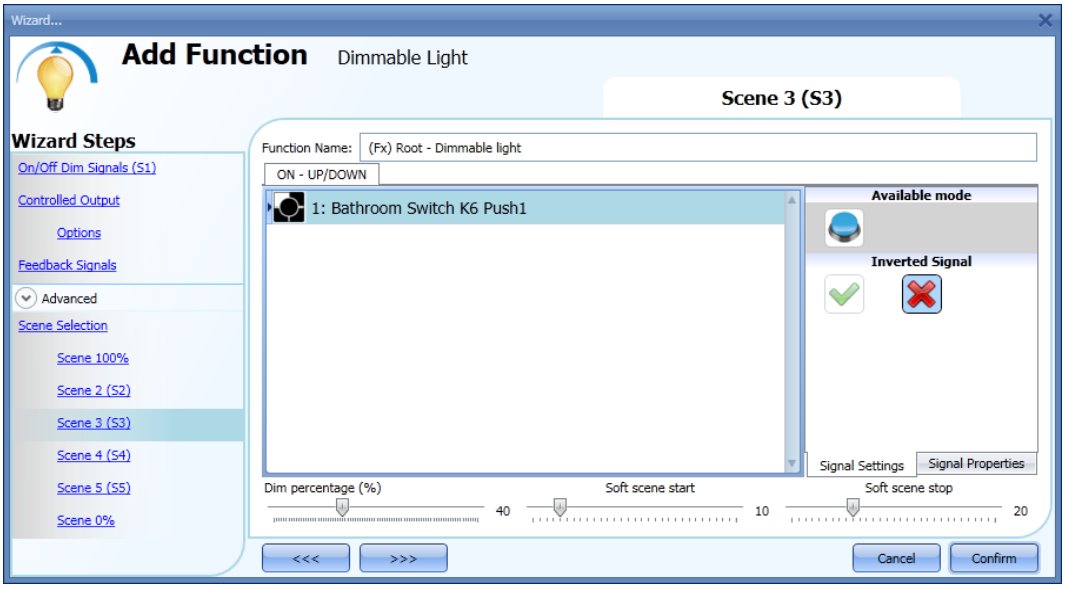
Soft scene start(s)
The soft start time is the time that the dimmer output takes to change the output from a value of 0% to a value of 100%. This time is adapted according to the percentage value set in the scenario.
For example, if the soft start time of Scene 3 is set to 10s and the percentage is set to 30%, the light will be switched on at 30% after 3s (calculation 100:10=30:x).



Soft scene stop(s)
The soft stop time is the time that the Dimmer output takes to change the output from a value of 100% to a value of 0%. This time is adapted according to the percentage value set in the scenario.
For example, if the soft stop time of Scene 3 is set to 10s and the percentage is set to 20%, the light will be switched off after 2s (calculation 100:10=20:x).



In the example below, the user has set one pushbutton for the manual activation of scenario 3. The output dimming percentage is set to 40%, the soft start is 10s and the soft stop is 20s.
Each time pushbutton 1 is activated with a short press, the light level will be set to 40%. If the light is on and the input signal is kept activated, the light level starts dimming Up/Down until the pushbutton is released.



10.4.8.4 Scene 4 (S4)

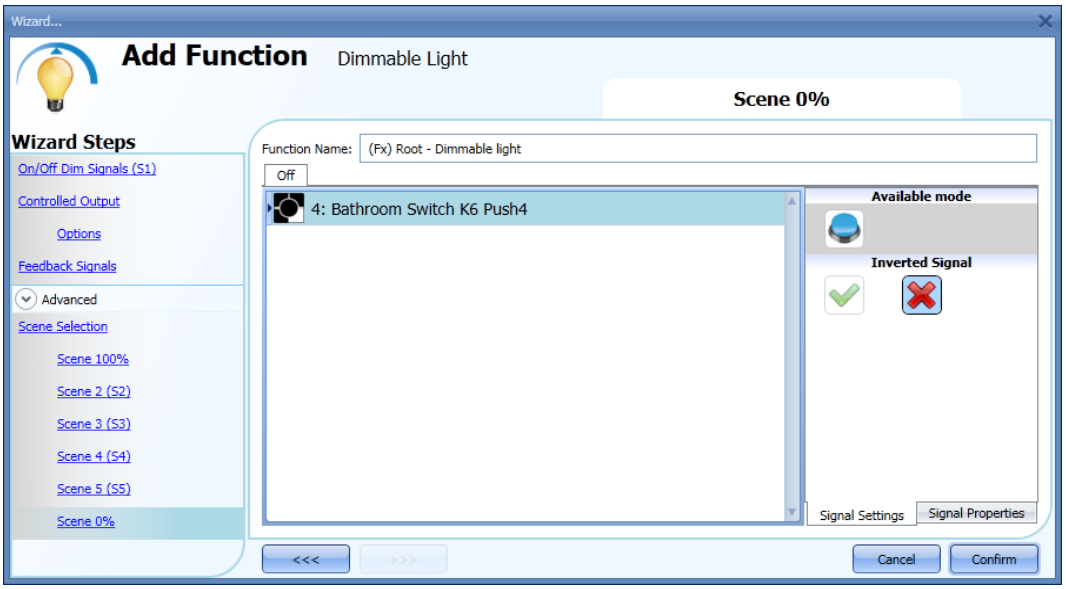
This scenario is managed in the same way as scenario 3.

10.4.8.5 Scene 5 (S5)

This scenario is managed in the same way as scenario 3.

10.4.8.6 Scene 0%

When scenario *Scene 0%* is selected, all signals added in the window Signals will switch the light off with a short press (input activated and deactivated within 1 s).
It is not possible to dim this scenario. The soft start and soft stop timing used are the same as scenario S1.
In the screen below, a simple example is shown of how to set a pushbutton to activate the scenario 0%. Each time pushbutton 4 is activated, the light is switched off.



10.4.9 How to manage lights automatically

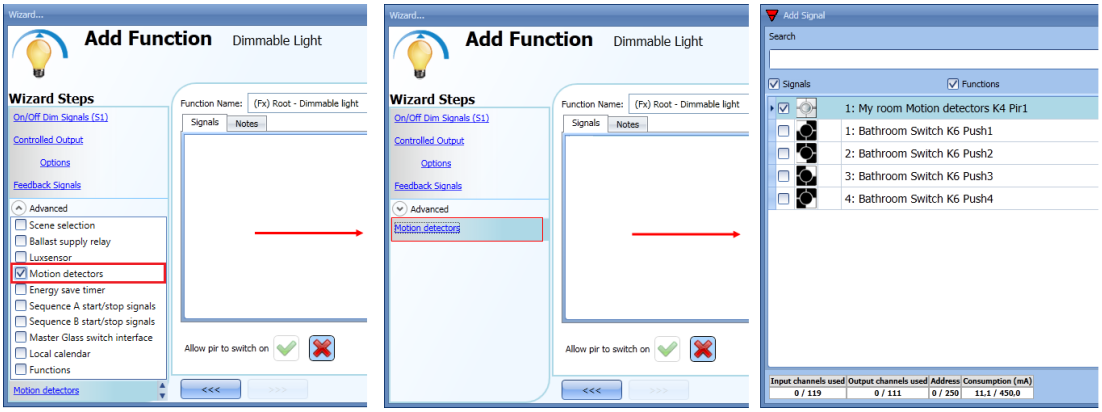
The automatic on/off switching of the light can be managed by PIR sensors (the light will be switched ON when the PIR sensor detects people movement/presence), by calendar functions (calendar to switch the light on/off at pre-defined time intervals) or by lux meters switching the light on or off according to the level of ambient light.
All these automations can be enabled in the *Advanced* section.

10.4.10 How to select a PIR sensor to control scenario

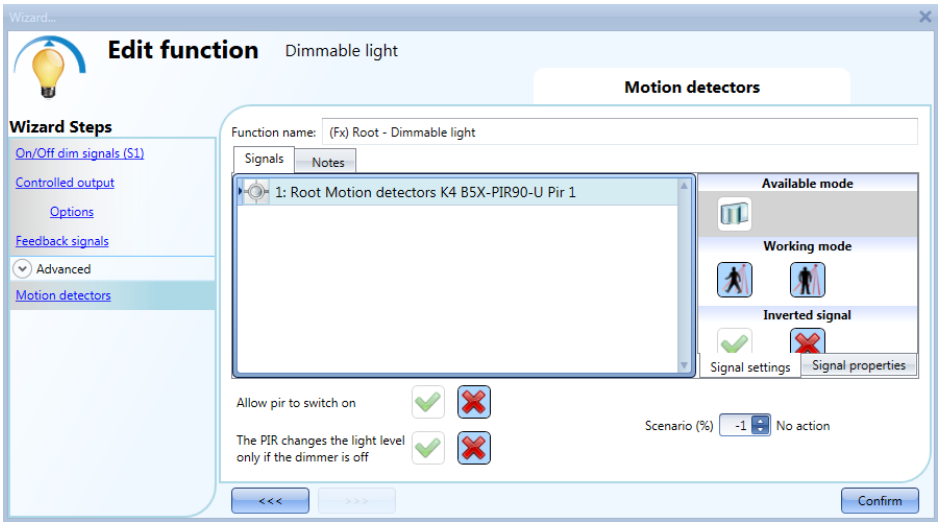
The motion detector sensor can be used to perform different functions:

- switching the light on, on movement detection.
- adjusting the light level to a predefined value.
- switching the light off if no presence is detected within a time interval.
- Starting a sequence that manages the activation of different scenarios

To set and mount a PIR sensor correctly, please read here.
The first thing to do when a PIR sensor needs to be used in a dimmable function is to enable it in the *Advanced* section.
The *Motion detector* menu will appear. After selecting it, with a double click on the Signal window, the list of available signals will appear.
Select the required signal/signals and click on *Confirm*. Up to 50 signals can be selected and the system will perform a logical OR of all of them.



Once the PIR sensor is added, the user can also choose to invert the signal by selecting the green *V* below *Inverted signal*, in the Signal Setting tab window.

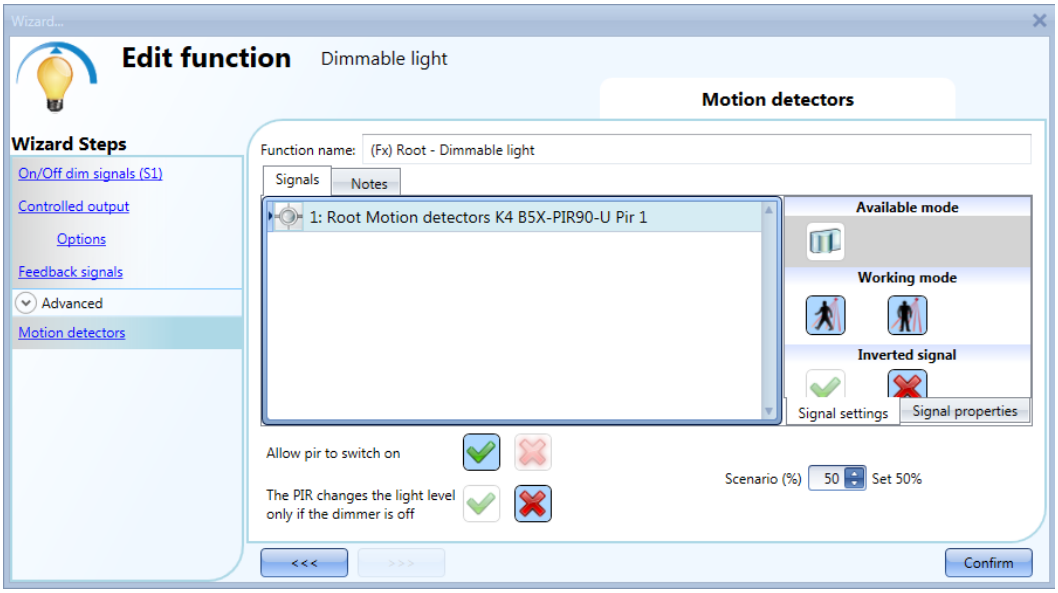


10.4.11 How to turn a light ON with PIR sensors, and turn it OFF manually

In order to create this easy automation, the user must select at least one input signal for the manual command, one output signal to control the load and then a PIR sensor in the Advanced section.
In this case the light will not be switched OFF automatically if no presence is detected, so the energy-save timer must not be used in the Advanced section.

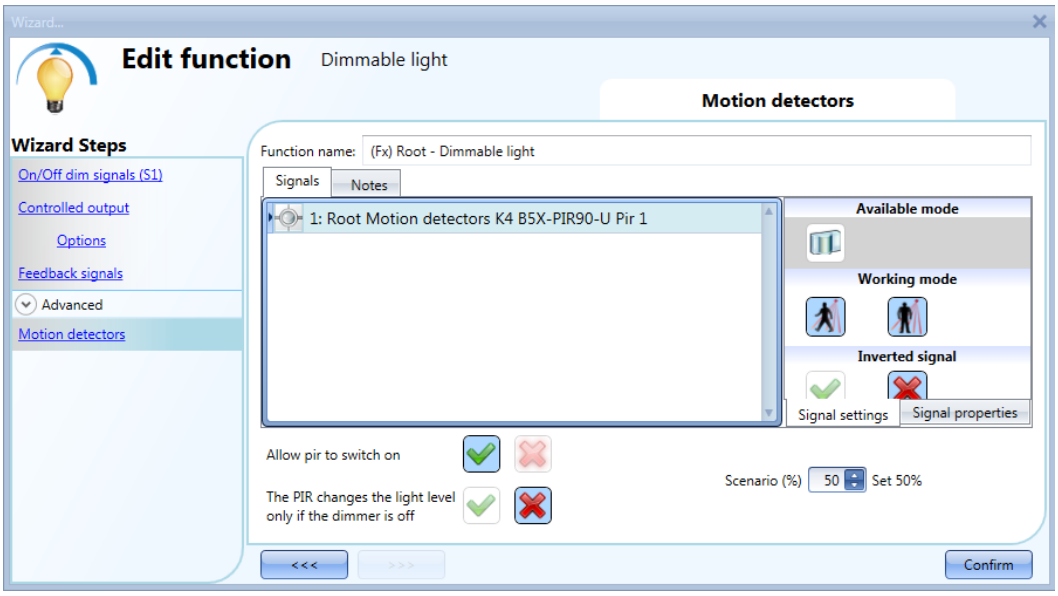
Once the PIR signal is entered, the user should select the green “V” in the option “Allow the PIR to switch on”. In this way, each time the PIR detects movement, the light will automatically be switched ON.
The user can also select the light percentage value of the output when the PIR detects movement and switches the light on. The event command can be selected from: no action (-1), switch the light Off (0), set scenario S1 (1), set scenario S2 (2), set scenario S3 (3), set scenario S4 (4), set scenario S5 (5), set a specific level from 10% to 100% (6-7-8-9 set to 10% that is the minimum value in order not to have the LED bulb flickering).

In the example below, the PIR is set to switch the light on at 50%.



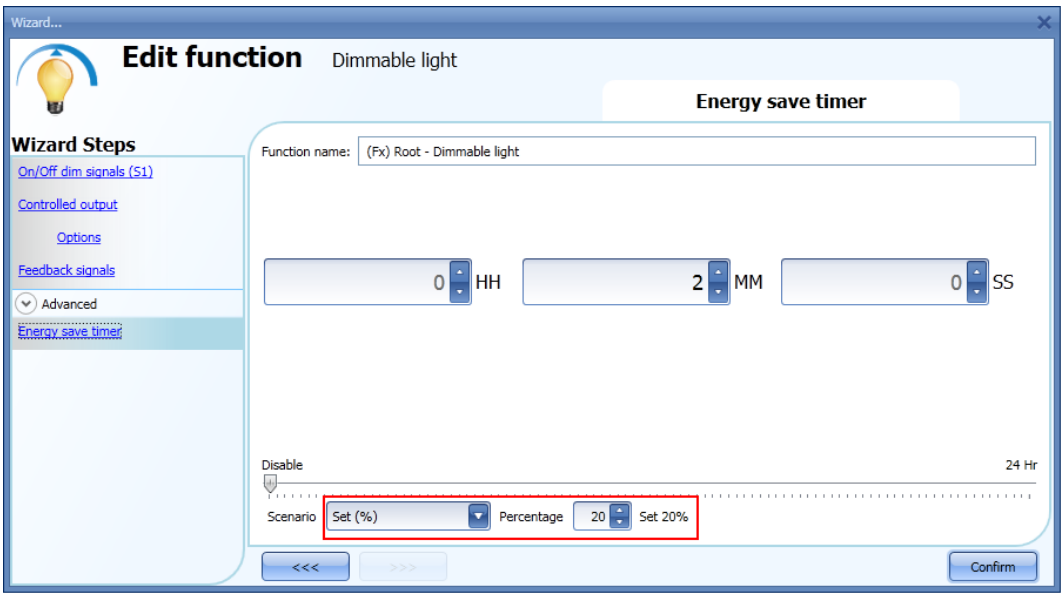
10.4.12 How to turn a light ON with PIR sensors, and change the scenario after a predefined period

The first step is to add the output signal, then to add a PIR sensor with the option “Allow PIR to switch on” enabled, in the picture below the PIR is set to switch the light on at 50% when movement is detected.



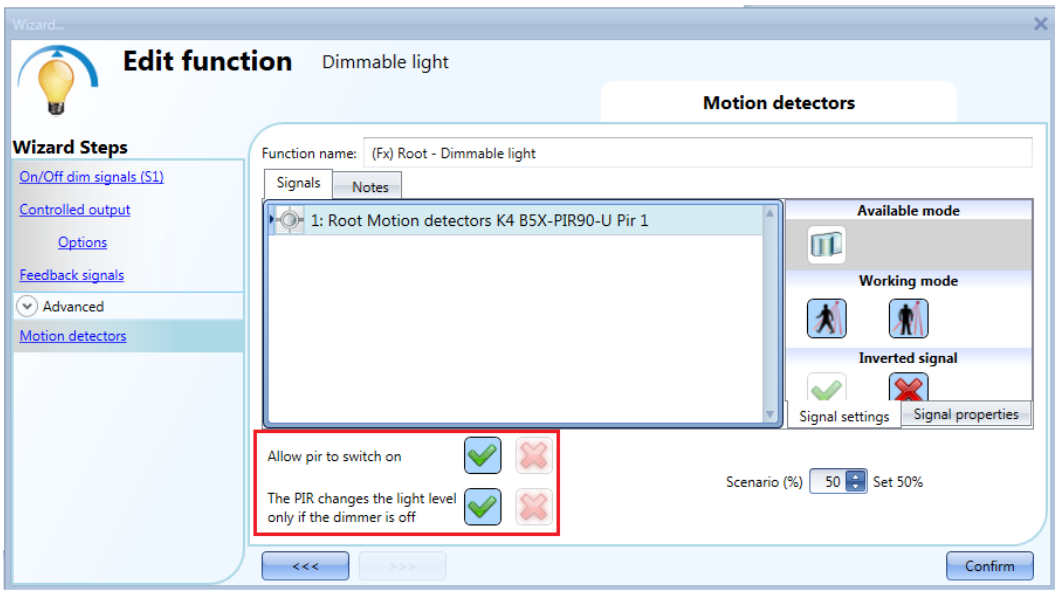
Every time the PIR detects movement, the light level is set to 50%, regardless the previous level reached by the function.

When the PIR is entered the energy-save timer must be set in order to change the current scenario when time elapses.
To set an energy-save timer, select *Energy save timer* from the Advanced section, change the timing by moving the slider or typing hours, minutes and seconds, select the action to be performed at the end of the timer (Set %), select the percentage and then press confirm.
In the picture below the energy save timer is configured to set the light level to 20% 2 minutes after the PIR stop detecting presence.

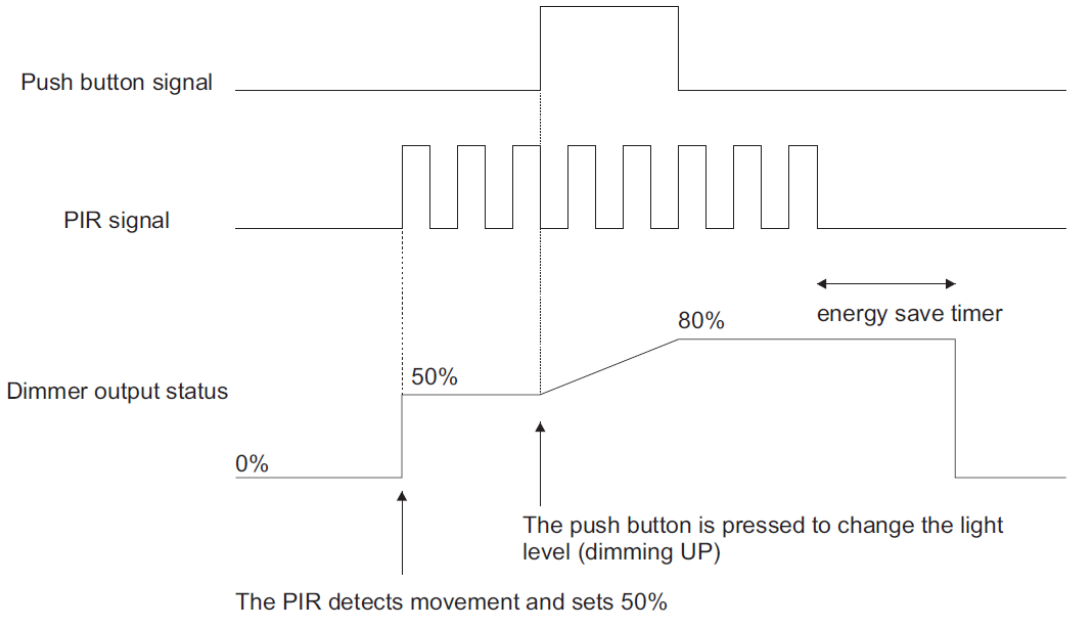


10.4.13 How to turn a light ON with PIR sensors, and change the light scenario manually when PIR detects movement

The first step is to add the output signal, then to add a PIR sensor with the options “Allow to switch on” and “The PIR changes the light level only the dimmer is OFF” enabled, in the picture below the PIR is set to switch the light on at 50% when movement is detected.



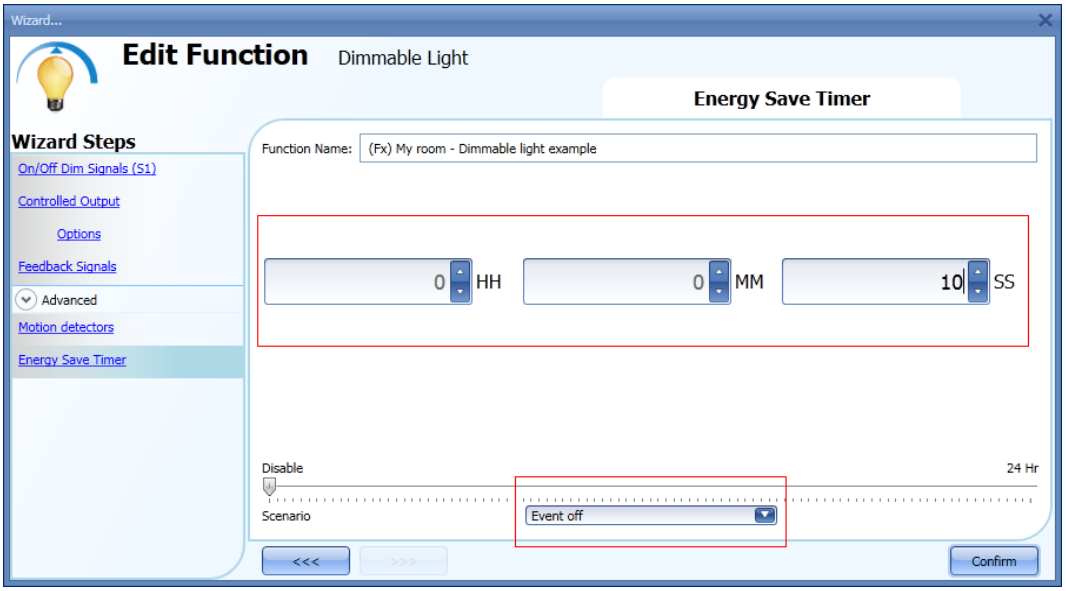
By using the above settings, the Dimmer function will work in this way: if the light is OFF and the PIR detects the movement, the light level is set to 50%, then it is possible to adjust the light level of the function using the pushbuttons, for example to 80%. When the light is ON and the light level is changed, the PIR doesn’t set the 50% when the movement detected.
If an energy save action is used, the light is switched off automatically when time elapses.



10.4.14 How to turn a light on and off with PIR sensors

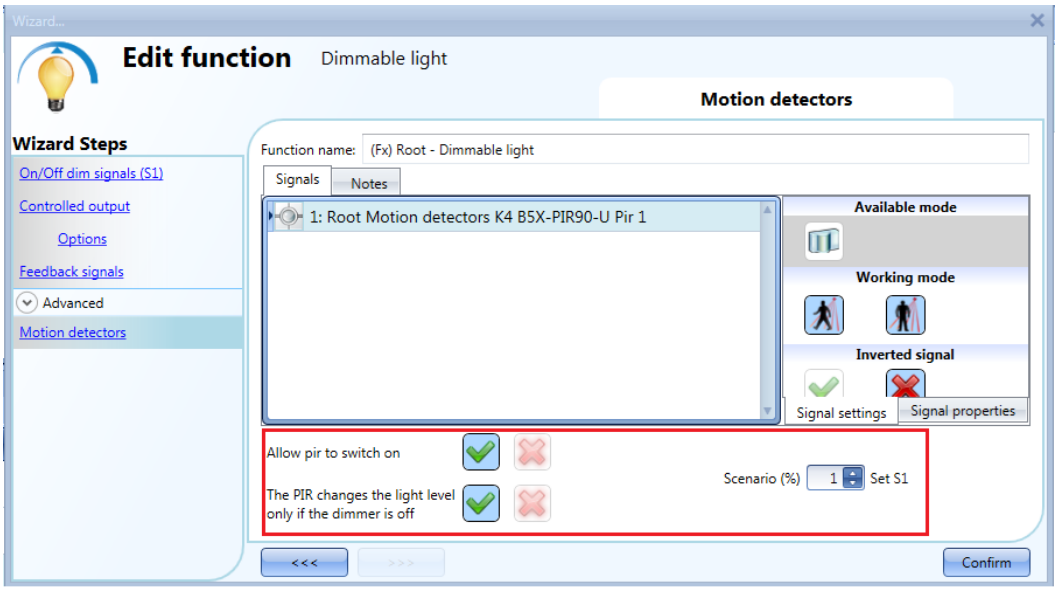
An example of this automation could be in a corridor: when the user enters the corridor, the light is switched on and the energy-save timer starts counting; while the user is in the corridor the light stays on and the energy-save timer is continuously reloaded; when the user leaves the corridor and the timer expires, the light is switched off automatically.

The first step is to add the output signal, then to add a PIR sensor with the option “Allow to switch on” enabled, and then to set an energy-save timer to switch the light off automatically when no presence is detected.
To set an energy-save timer, select *Energy save timer* from the Advanced section, change the timing by moving the slider or typing hours, minutes and seconds, select the action to be performed at the end of the timer (event off), then press confirm.
The energy-save timer starts counting when no presence is detected. Each time presence is detected the timer is reloaded. When the timer expires the light is switched off.



10.4.15 How to change scenario with a PIR

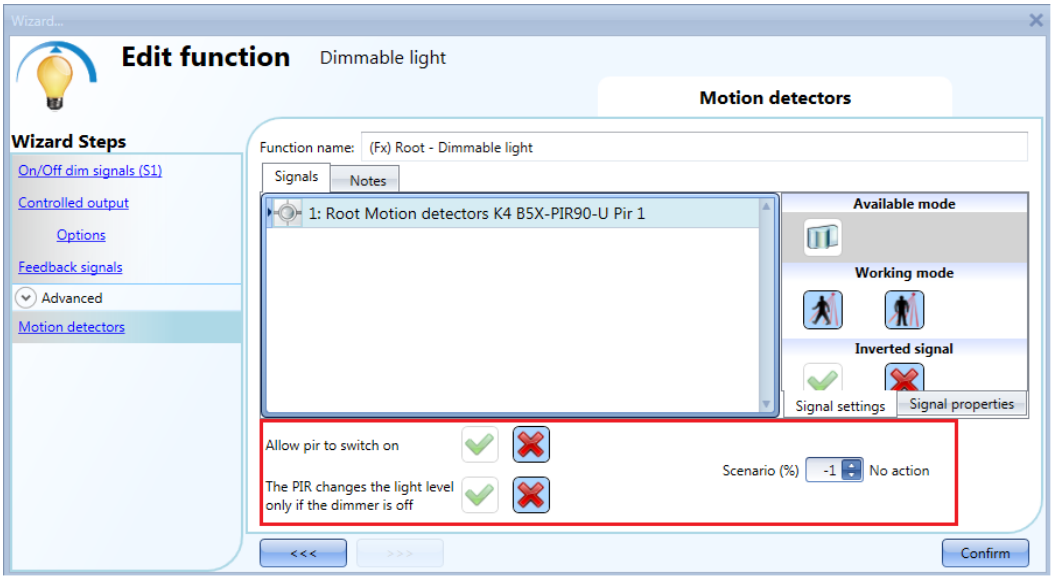
The PIR sensor can change the current scenario when movement is detected. The user should type the value of the scenario in the *Scenario %* field: no action (-1), switch the light Off (0), set scenario S1 (1), set scenario S2 (2), set scenario S3 (3), set scenario S4 (4), set scenario S5 (5), set a specific level from 6% to 100%.
In the example below, the user has set the PIR K4 to switch the light on at scenario S1: each time the PIR switches the light On, the light level of the output is the last valid value stored in scenario S1. The same rule is valid for all other scenarios.



10.4.16 How to switch the light on manually and off with the PIR sensor

An example of this automation is an office where the employees sometimes forget to switch the light off when they leave the office. When the PIR stops detecting presence in the office, the energy-save timer starts counting and when it expires, the light is turned off.

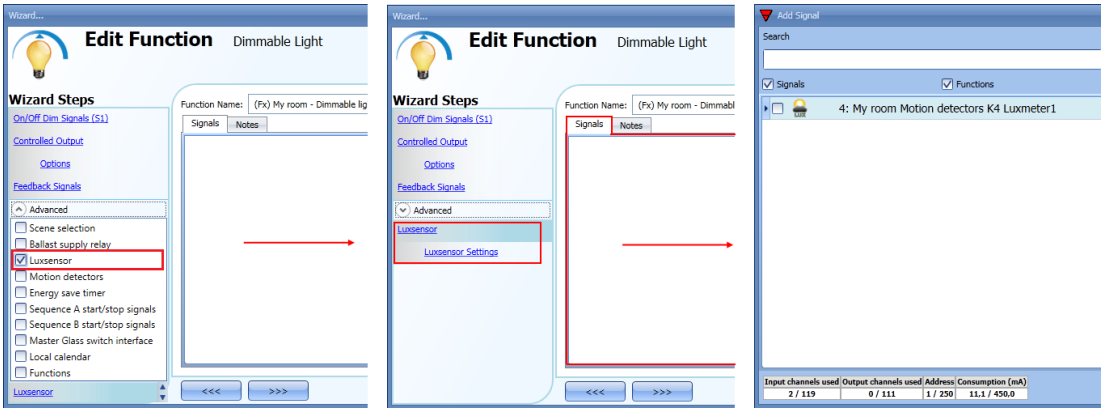
If the user wants to use the PIR sensor to switch the light off automatically, the following settings should be used.
Add a PIR sensor and an energy-save timer from the Advanced section.
In the PIR settings, the red cross in the *Allow to switch on* field must be selected because the light has to be switched on manually. In the *Scenario %* field, the value -1 must be selected (-1 means no action when movement is detected); with these two settings the PIR cannot switch the light on and cannot set a different light percentage when it detects movement.
The energy-save timer must be set differently from zero, and will be triggered when the light is switched on. While the PIR detects presence, the timer is reloaded. When the PIR stops detecting presence, the timer starts counting down and when it expires the light is switched off accordingly.



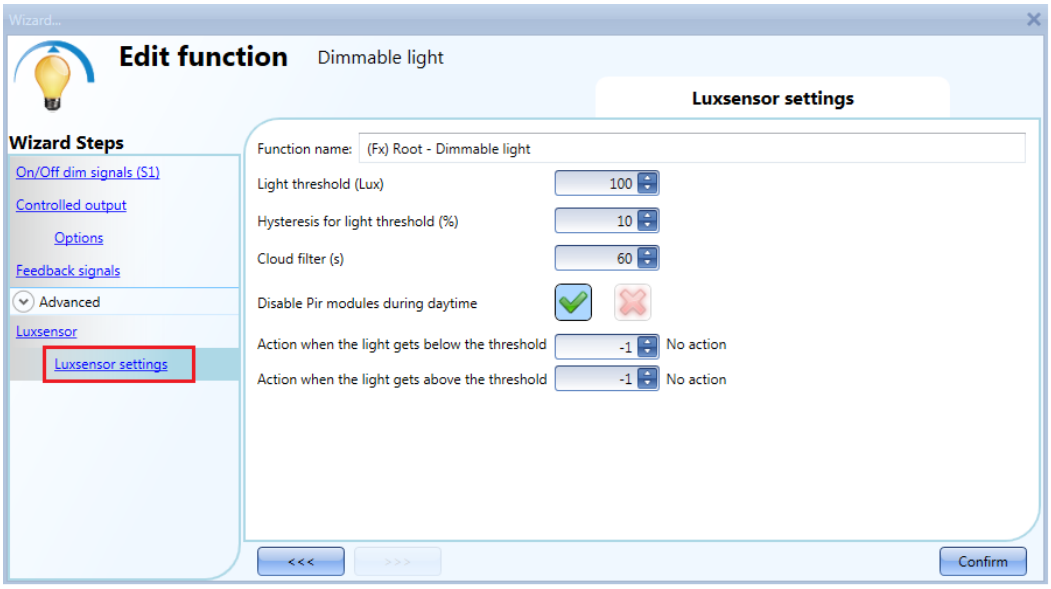
10.4.17 How to switch the light on and off according to the daylight

An example of this automation could be an outdoor light that has to be switched on at sunset and stay on during the night.

To create this kind of automation, lux meters have to be used: the user can set a threshold value so that the smart-house system switches the light ON when the measured daylight goes below this threshold, and switches the light OFF when the daylight goes above this threshold.
Since this is an Advanced functionality, the *Luxsensor* field has to be enabled in the *Advanced* section.



The user can add up to ten lux sensors: if more lux signals are linked to the function, the average lux value is calculated and then used.
Once a lux sensor is selected, the user can change the settings as shown in the picture below:



Light threshold (Lux)
In this field the user can set the threshold for the night time condition. When the measured light level goes below the night time threshold, the night time condition is activated. The day condition is activated when the measured light level goes above the night threshold + the hysteresis value.

Hysteresis value for light thresholds (%)
In this field the user should enter the hysteresis value needed to establish the light level OFF threshold, that is the threshold above which the night time condition ends and the day time condition starts. (A value between 5 and 50% can be set).

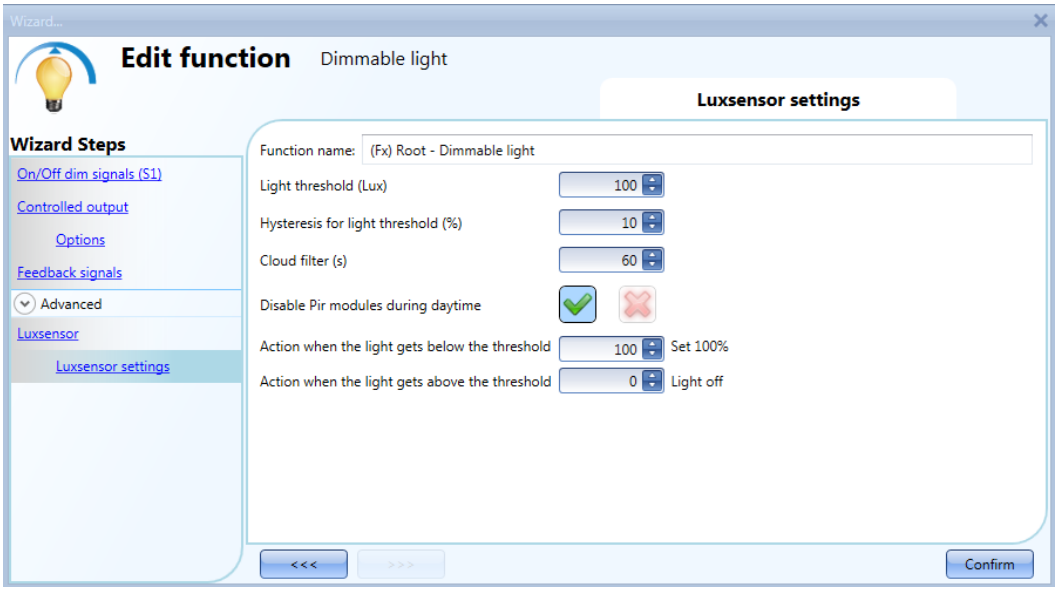
Cloud filter (s)
In this field the user can set up a delay (in seconds) to avoid the turning on/off of the light if a cloud shields the sun for a short time. If the time is set to 0, the filter is disabled.
We suggest setting the Cloud filter at at least 60 seconds in order not to have the light turning ON/OFF continuously.

Disable PIR during day time
If the green V is selected, this functionality is enabled and the PIR sensors are disabled to switch the light on and off during day time. The field *Disable PIR modules during day time* must be used if the lux meter is used in combination with PIR sensors.

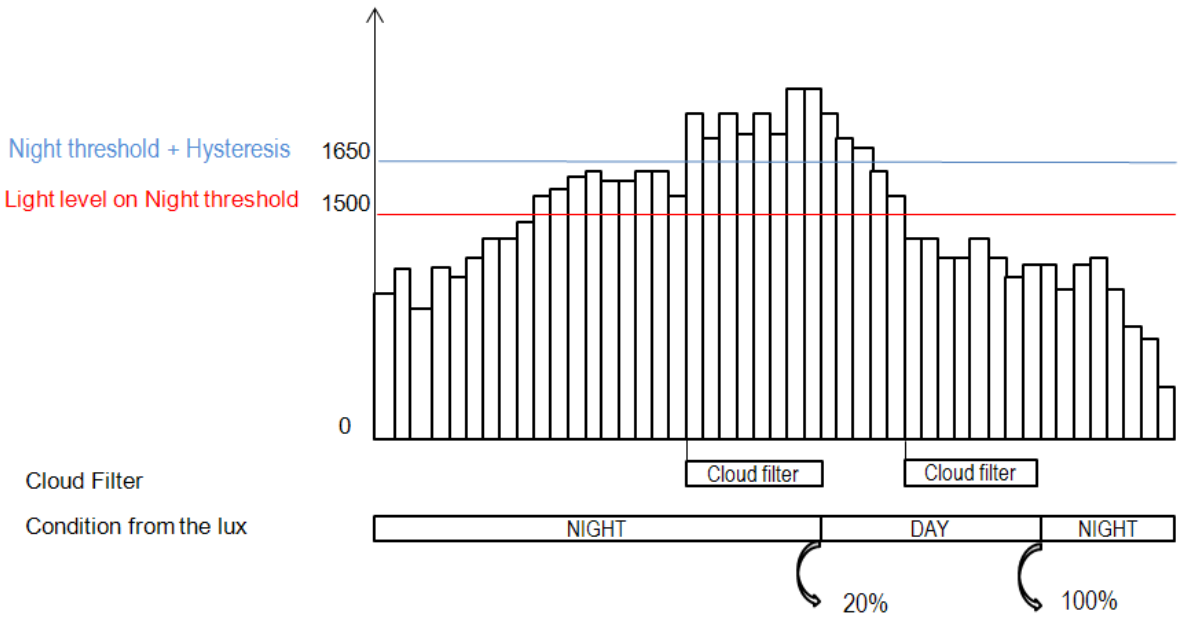
Action when the light gets below the threshold
The user can select the action to be performed when the light level goes below the threshold. The available actions are: no action (-1), switch the light Off (0), set scenario S1 (1), set scenario S2 (2), set scenario S3 (3), set scenario S4 (4), set scenario S5 (5), set a specific level from 6% to 100%.
If the value -1 is selected, the system will not do anything at the surpassing of the threshold. This might be needed when lux meters are used in combination with PIR sensors or calendar.

Action when the light gets above the threshold
The user can select the action to be performed when the goes above the threshold. The available actions are: no action (-1), switch the light Off (0), set scenario S1 (1), set scenario S2 (2), set scenario S3 (3), set scenario S4 (4), set scenario S5 (5), set a specific level from 6% to 100%.
If the value -1 is selected, the system will not do anything at the surpassing of the threshold. This might be needed when lux meters are used in combination with PIR sensors or calendar.

The ON and OFF switching is done in the event of the thresholds being surpassed, so if the light status is changed manually by means of a light switch or automatically e.g. by a scheduler, the automation of the lux sensors will not change it unless thresholds are surpassed another time.



Please see the example in the picture above:
The night threshold is set to 1500 Lux, the hysteresis is 10 %, the level OFF threshold is calculated in the following way: Level OFF= level ON+ 10% of Level ON, which means Level OFF= 1500 + 150 = 1650 lux.
The cloud filter is set to 60 seconds, and the action set for the night time condition is “Set the dimmer output at 100%”. The action for the day time condition is “Set the dimmer output at 20%”.

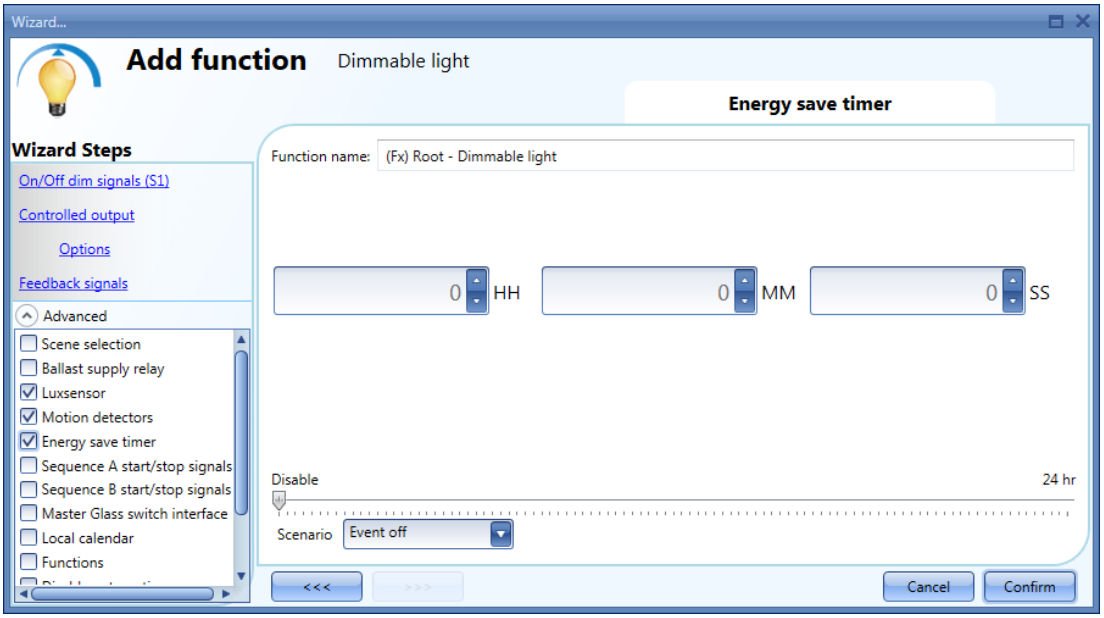


10.4.18 How to control light with PIR sensors in combination with lux sensor

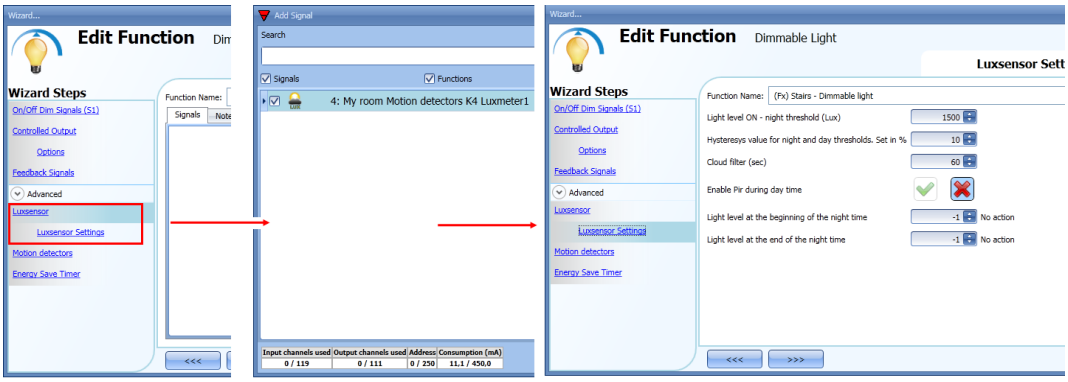
An example of this automation could be a corridor with a lot of windows.
During the day the sunlight enters the corridor and it is not necessary to switch the light on: in the evening, when the sun goes down, the light level decreases and is it is necessary to switch the light on. When the light level is low, the PIR switches the light on automatically and the energy-save timer is started: when the corridor is no longer in use and the time expires, the light is switched off automatically.
If PIR sensors are used in combination with lux meters, it means that if a people presence is detected, the light is switched ON according to the daylight level. To achieve this, a PIR sensor and a lux sensor have to be added.

The steps are summarised below:

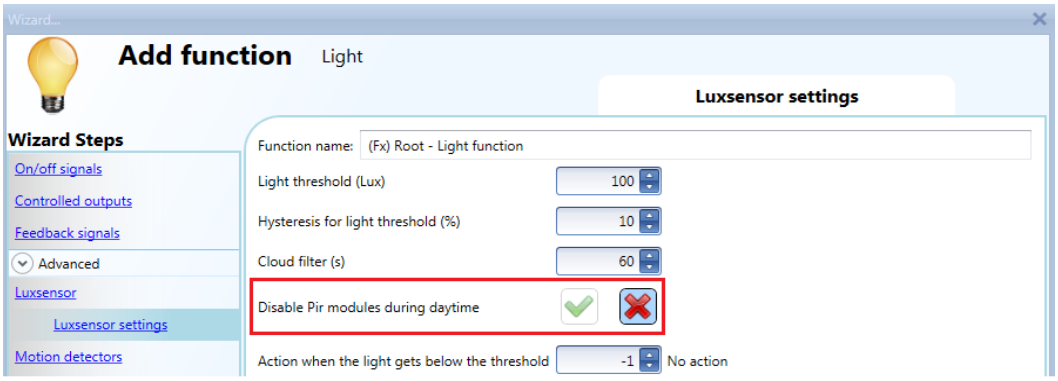
A) Enable Lux sensor, Motion detectors and energy-save timer in the advanced section



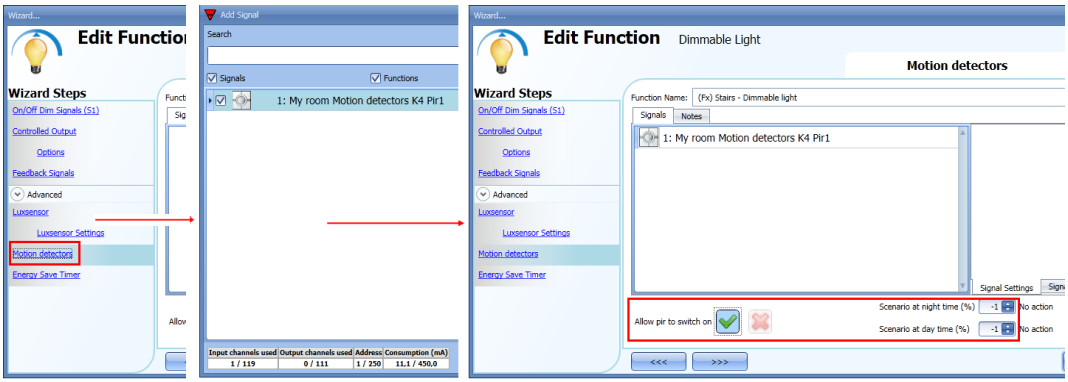
B) Add the lux sensor and set the threshold values for light



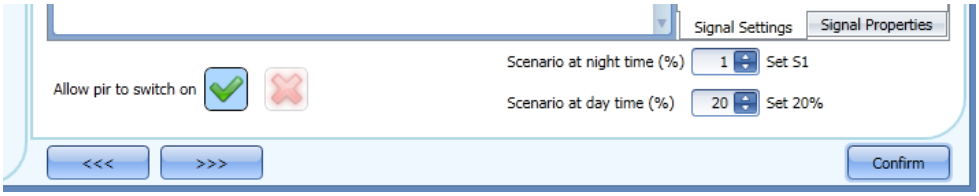
If the user wants to enable the PIR during day time, the green "V" should not be selected in the relevant setting (see picture below). If this option is not enabled, the PIR will work only during night time (when the lux level is below the night threshold).



C) When the lux sensor is entered, add the PIR sensor

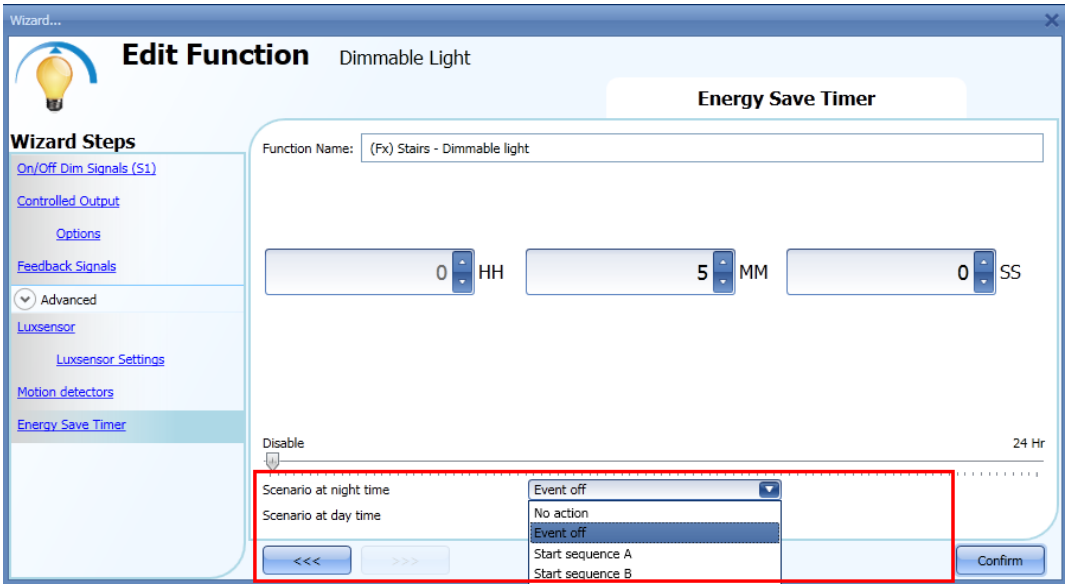


When the PIR sensor is entered, the user must select the green "V" in *Allow PIR to switch on*, then he can set two different actions according to the time of day: one light level for night time and another for day time.
In the *Scenario at night time,* the user can set the action to be performed when the PIR detects movement during the night time.
In the *Scenario at day time*, the user can set the action to be performed when the PIR detects movement during the day time.



The picture above shows an example of a PIR sensor that switches the light on at Scenario 1 when the night time condition is active and switches the light on at 20% when the day time condition is active.

D) The last thing to set is the energy-save timer
When the lux sensor is combined with other automations, the energy-save timer window changes and offers the possibility of selecting two different actions to be performed when the timer expires.

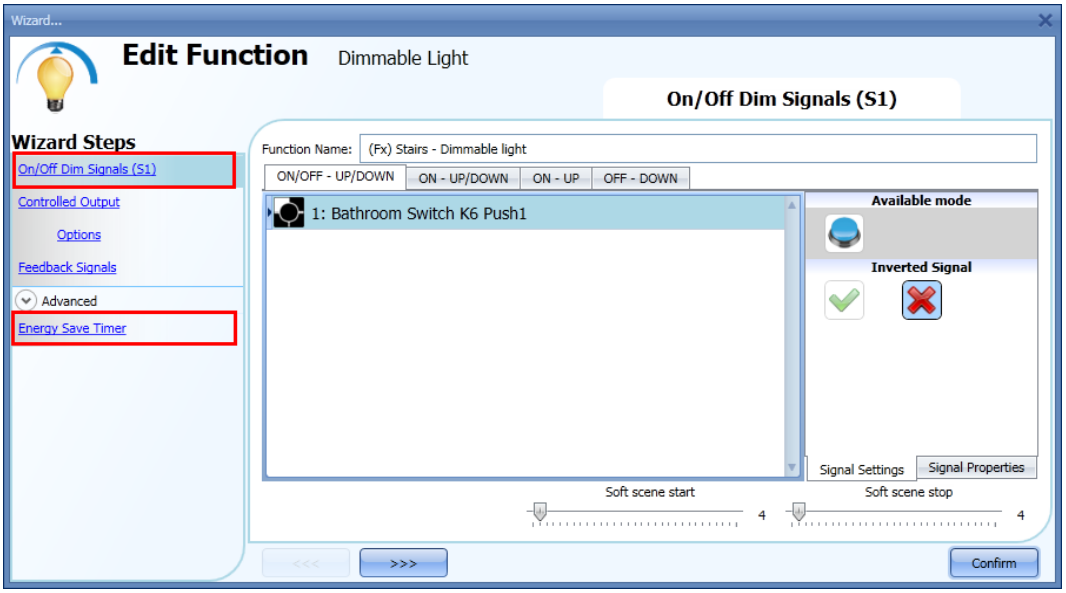


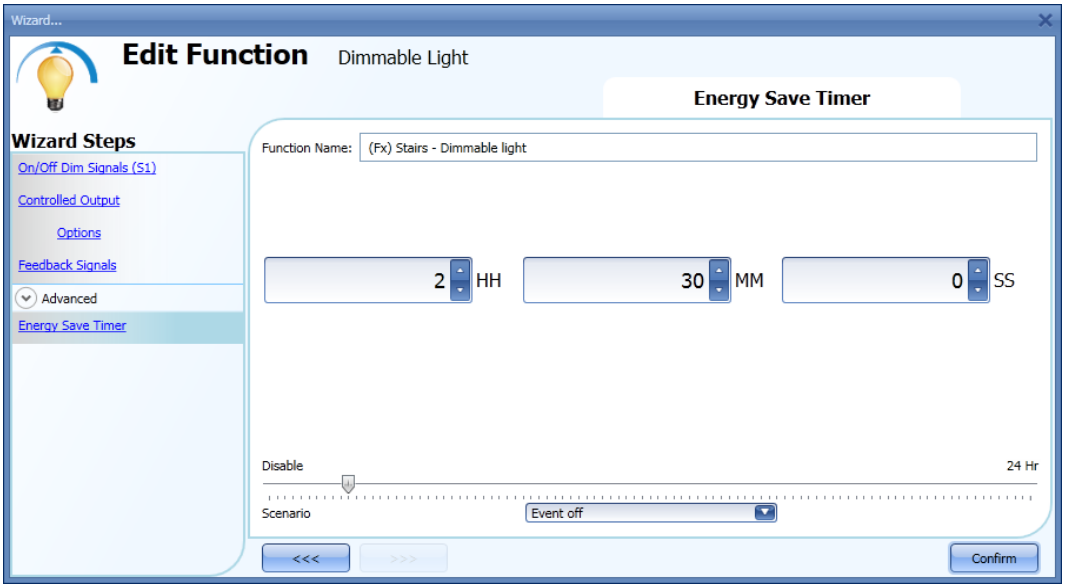
The user can set an action to be performed when the timer expires during the daytime, and another one when the timer expires during the night-time.
If the option *No action* is selected, the light level is not changed when the timer expires; if *Event off* is selected, the light is switched off when the timer expires.
The user can also choose to start a sequence A or B or set a predefined scenario with the selection *Set (%).*

10.4.19 How to switch the light off after a predefined period without lux meter

If the user needs to be sure that the light is switched off to save energy, the energy-save timer should be selected in the advanced functionalities (the energy-save timer is selected by default when the user creates a new Dimmer function).
The energy-save timer starts counting each time the light is switched ON manually or by using the PIR sensor, causing the light to go off automatically when it expires.
The energy-save timer can be set up to a maximum value of 24 hours, whilst when it is set to 0 it is disabled. For this reason, the shortest time that can be set is 1 second.
The energy-save timer value can be changed by means of the webserver.

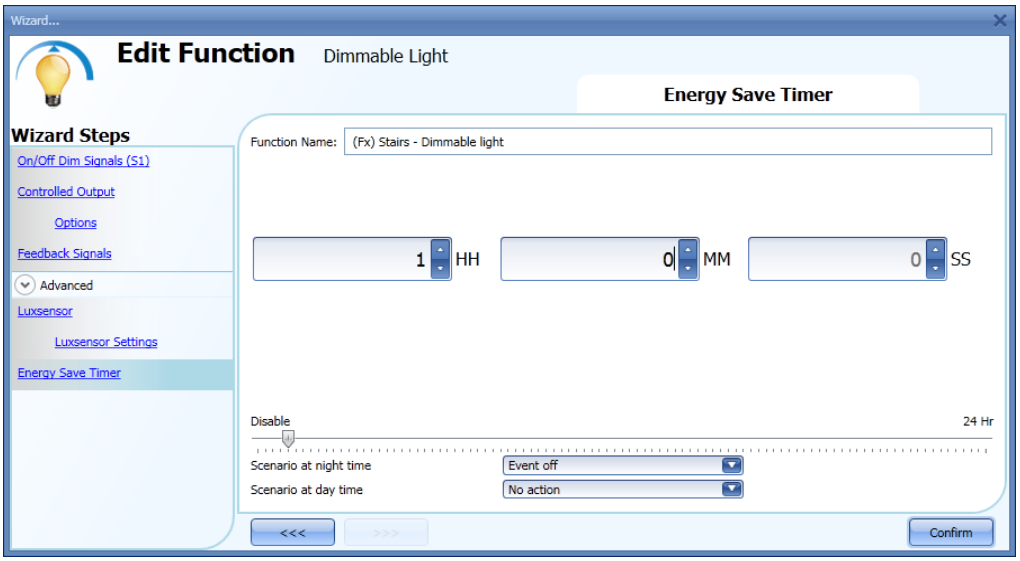
In the example below, the user has set a function with a pushbutton to switch the light on and an energy-save timer of 2 hours 30 minutes to switch the light off automatically (*Event off* is selected as the action to be performed).





10.4.20 How to switch the light off after a predefined time period with the lux meter

If the lux sensor needs to be used combined with the energy-save timer, the energy-save timer window changes and offers the possibility of selecting two different actions to be performed when the timer expires, according to day or night time condition.
The example below shows an application of this: during the day the light is controlled manually, and it is not necessary to switch it off automatically if the user forgets to do so (no action is selected in the energy-save timer actions).
If the light is manually (with a push button) switched on during the night and the user forgets to switch it off, the energy-save timer automatically turns it off when the time expires.

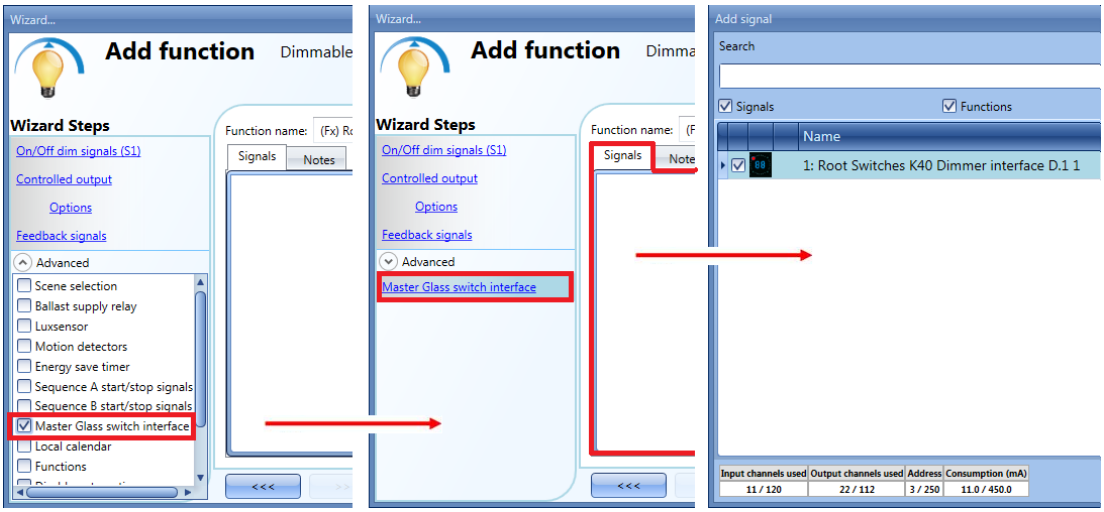


10.4.21 How to switch the light on/off using the calendar

An example of this automation could be a calendar that switches all lights off at a pre-set time in the night.

There are two ways of managing dimmable lights with the calendar: by setting the “local” calendar inside the function or using a “global” calendar function.
If the function uses an energy-save timer, the calendar does not affect it in anyway.

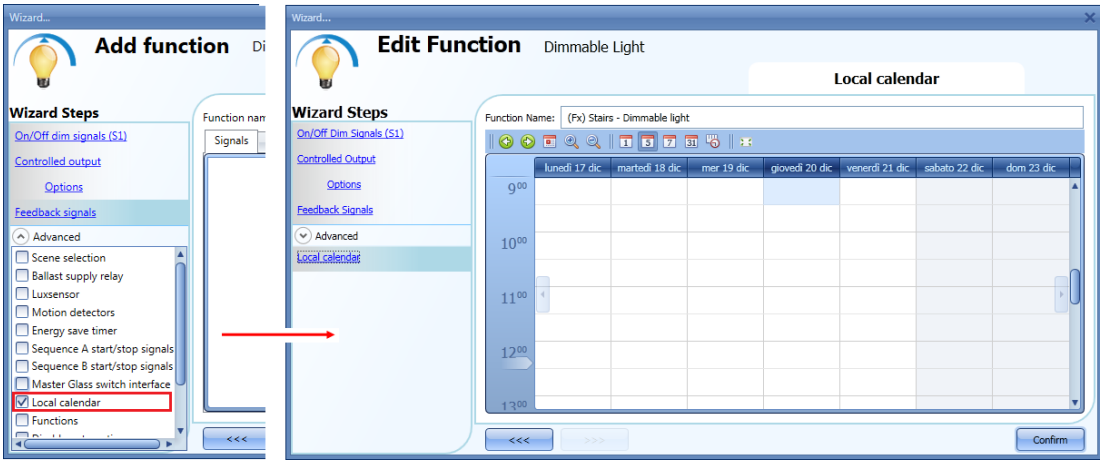
10.4.22 How to use a MASTER GLASS dimmer interface in a dimmable light function



To use a key programmed as dimmer interface in a Dimmable light function, select *Masterglass switch interface* in the *Advanced* field, double click on *Signals* and select the required dimmer interface.

10.4.22.1 The local calendar

To enable it, the relevant menu has to be enabled in the Advanced section.



The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* shown below:

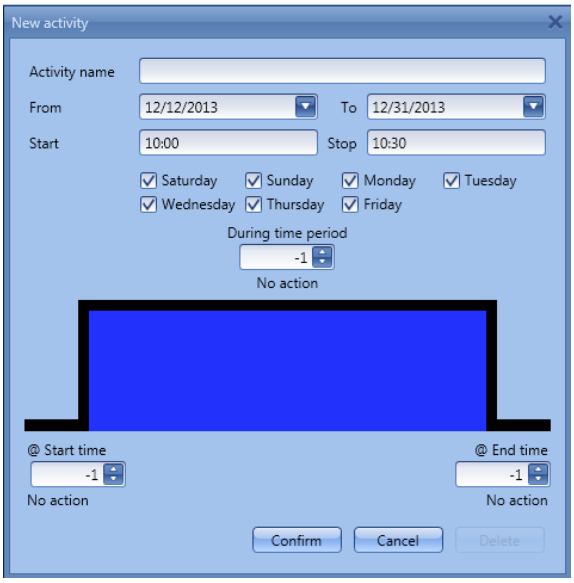
Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week
	One step forward in the calendar. E.g: if a week is visualized, this button allows the user to go to the next week
	Go to the current day
	Magnifying lens to see more/fewer time bands
	The calendar visualizes one day horizontally
	The calendar visualizes 5 days horizontally
	The calendar visualizes 7 days horizontally
	The calendar visualizes 31 days horizontally
	The calendar visualizes 7 days vertically
	Full screen view

Calendar activities:

Event activity

Once the user has selected his preferred type of visualization, to enter a time band he should double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that is to appear on the calendar. This is a mandatory field.

From*:* This is the starting date for the activity.

To*:* This is the end date of the activity.

Start*:* This is the start time of the activity.

Stop: This is the end time of the activity.

@ ***start time***: in this field the user can select the action to be performed at the start time.
The selectable actions are:

- No action (-1)
- Switch the light off (0)
- Set the light to Scenario 1 (1)
- Set the light to Scenario 2 (2)
- Set the light to Scenario 3 (3)
- Set the light to Scenario 4 (4)
- Set the light to Scenario 5 (5)
- Set the light level to a fixed value between 6% and 100% (6-100)

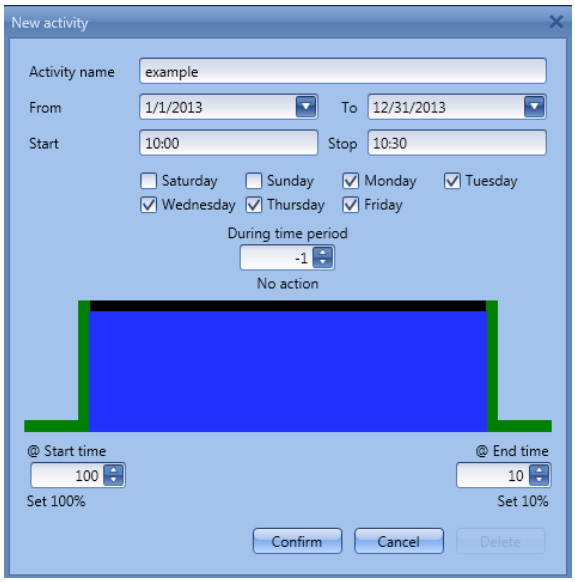
@ ***end time***: in this field the user can select the action to do at the end time.
The selectable actions are:

- No action (-1)
- Switch the light off (0)
- Set the light to Scenario 1 (1)
- Set the light to Scenario 2 (2)
- Set the light to Scenario 3 (3)
- Set the light to Scenario 4 (4)
- Set the light to Scenario 5 (5)
- Set the light level to a fixed value between 6% and 100% (6-100)

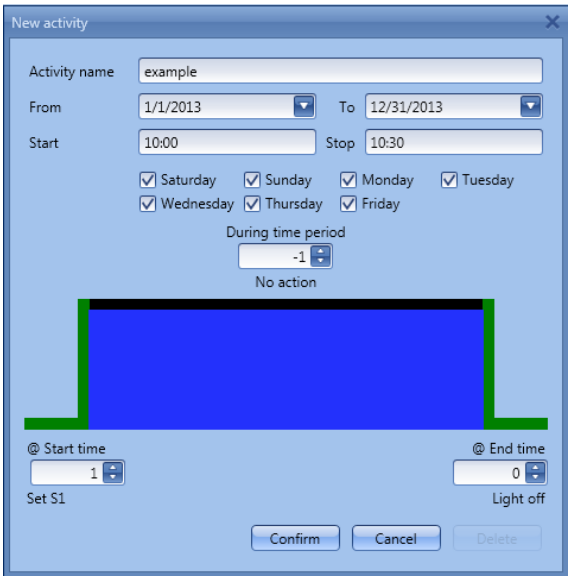
Days: The user should select the week days when the actions are to be performed.

If ***During time period*** is selected, the user go to disable all the automations of the function (PIR sensors, lux meters, calendars).

- No action (-1)
- Disable automations (1)



In the first example, the calendar is set to work all the year round (from 1st January to 31st December) . The working days are: Monday, Tuesday, Wednesday, Thursday and Friday (the calendar does not work on Saturday and Sunday)
At the start time (10:00) the light is switched on at 100%: at the stop time the light level goes down to 10%.
All the activities are repeated automatically each year, so looking at the example below, once 2013 is finished, the activity will go on in the same way into 2014 and so on.

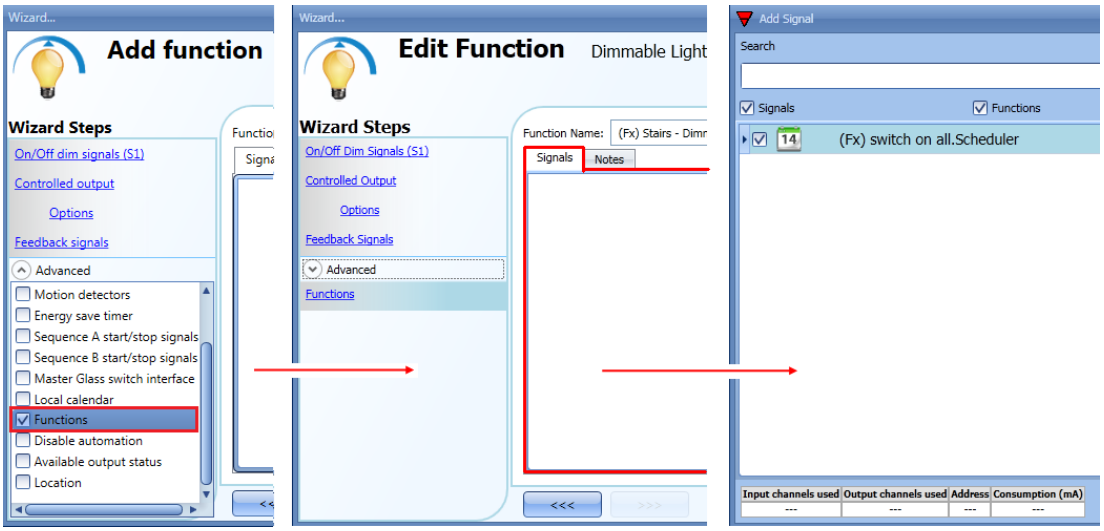


In the second example, the calendar is set to work all the year round (from 1st January to 31st December), every day.
At the start time (10:00) the light is switched on at scenario 1: at the stop time of 10:30 the light is switched off.

10.4.22.2 The global calendar function

Before using a global calendar, the user should define it as a function (See How to set a global calendar).
The field function then has to be selected in the *Advanced* menu.
By clicking on Functions and then double clicking in the *Signal* window, the *Add signal* window will appear.
Select the required calendar function.

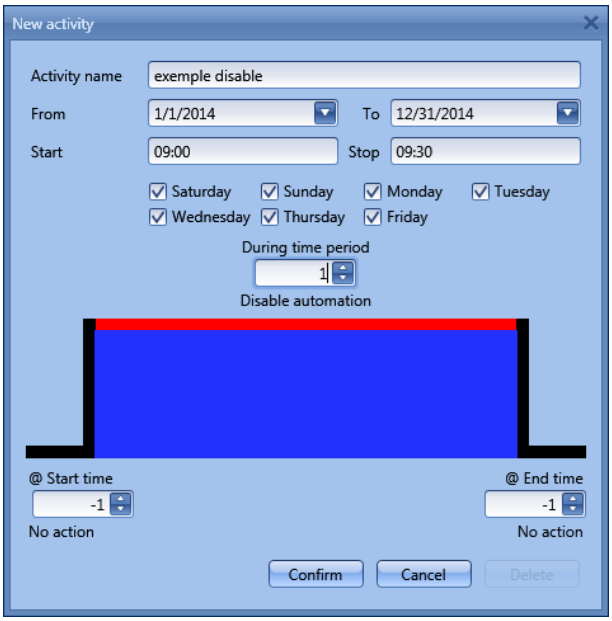
In the example below, the global calendar added is a calendar created for switching on all the lights.
Its behaviour is the same as that described for the local calendar.



10.4.23 How to stop an automation

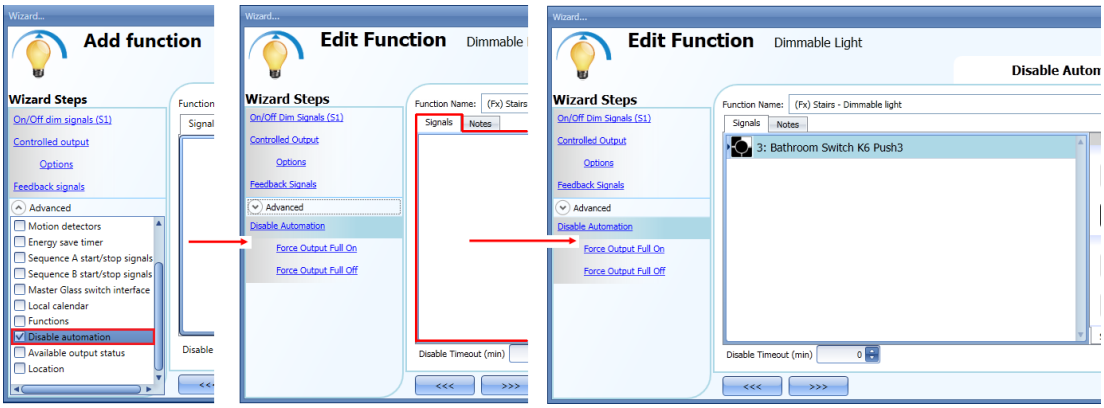
There are two ways of disabling the automation performed by PIR sensors, lux meter, calendar and energy-save timer: the first is with the calendar, the second with signals.

Disabling automation using the calendar
To disable the automation, the user can select the local or the global calendar.
The local calendar should be enabled, using the advanced functionality *Local calendar*, while the global calendar should be added as a signal in the *Functions* menu of the *Advanced* functionalities.
Regardless of the type of calendar added, the settings to disable all the automations are shown below.

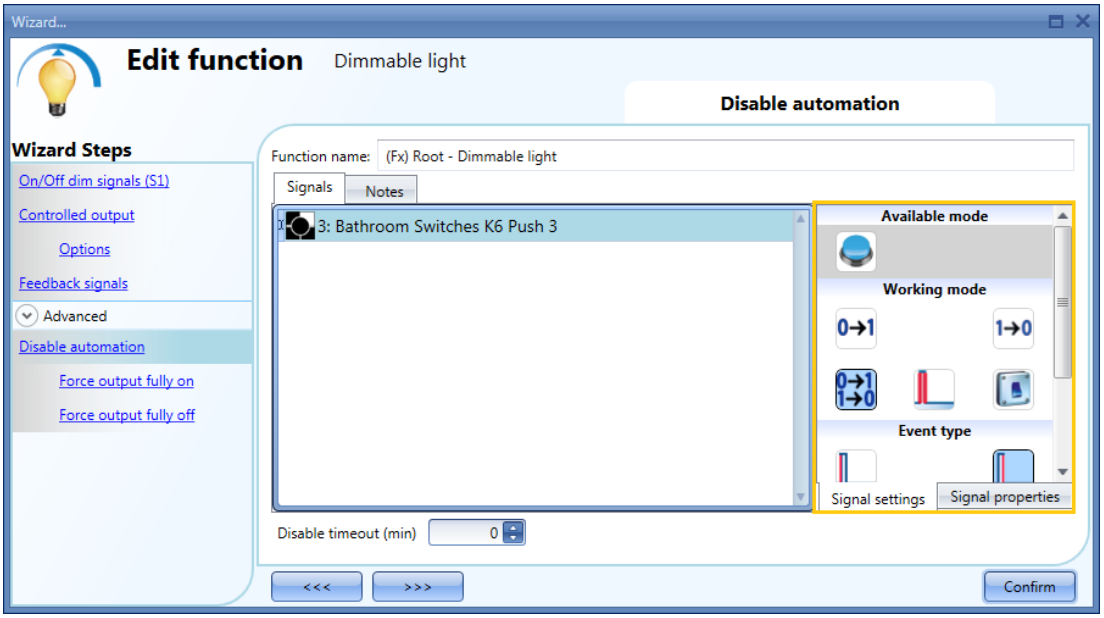


In the level activity the disabling of the automation is selected by entering the value 1. In this way, during the entire period when the calendar is active, all automations are disabled.

Disabling automation using signals
The second way of disabling an automation is by using signals: to enable it, select *Disable automation* in the *Advanced* section, double click on the signal window and select the right signal to use.



When the signal is added, the user should select the working mode and the type of the event for the signal:



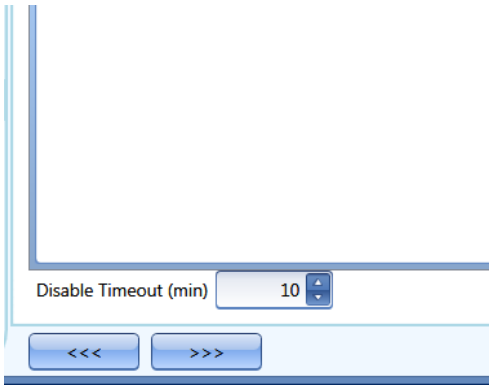
If a push button type is used, the way of working marked by the yellow rectangle should be selected following the table below.

	Event type			
Working mode				
0→1	As soon as the push button is pressed, the automation is disabled.	After a short press (less than 1 second), the automation is disabled at the release of the push button.	After a long press, the automation is disabled at the release of the push button.	After a very long press, the automation is disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automation is enabled again.	After a short press (less than 1 second), the function automation is enabled again at the release of the push button.	After a long press, the automation is enabled again at the release of the push button.	After a very long press, the automation is enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the automation will be enabled/disabled in toggle mode.	After a short press (less than 1 second), the automation will be enabled/disabled in toggle mode at the release of the push button.	After a long press, the automation will be enabled/disabled in toggle mode at the release of the push button.	After a very long press, the automation will be enabled/disabled in toggle mode at the release of the push button.
	The automation will be enabled/disabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode has to be carried out following the table below:

	Event type	
Working mode	Signal activated	Signal deactivated
1→0	The automation is enabled again	No action
0→1	The automation is disabled	No action
0→1 1→0	The automation is disabled/enabled in toggle mode	No action
	The automation is disabled/enabled in toggle mode	The automation is disabled/enabled in toggle mode
	The automation is disabled	The automation is enabled

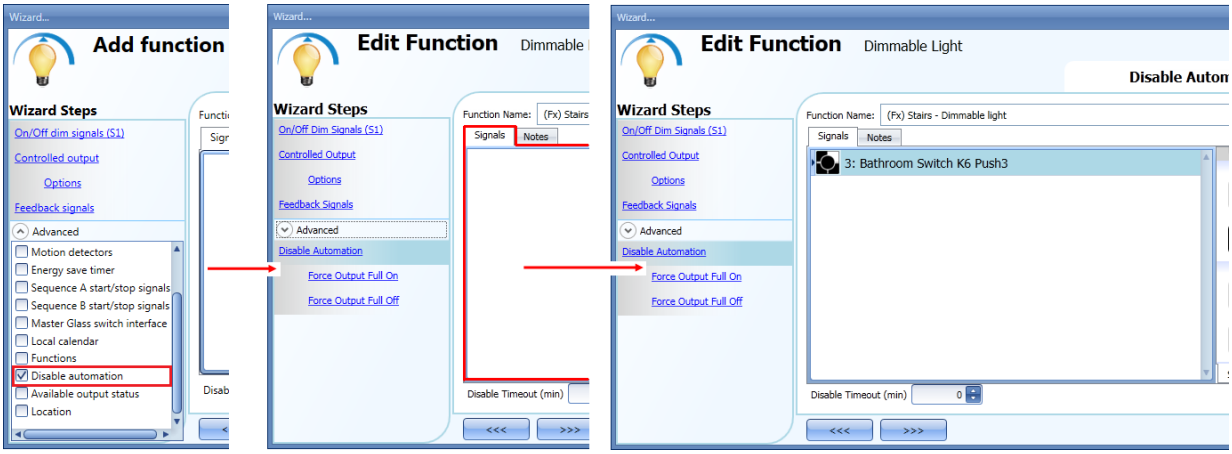
The user might need to set a time after which the automation is to be enabled again, even if the selected signal is still active. To do this the *Disable timeout* field has to be set.



In figure above, a disable timeout of 10 minutes is set. The maximum delay is 59minutes.

10.4.24 How to force the output fully ON

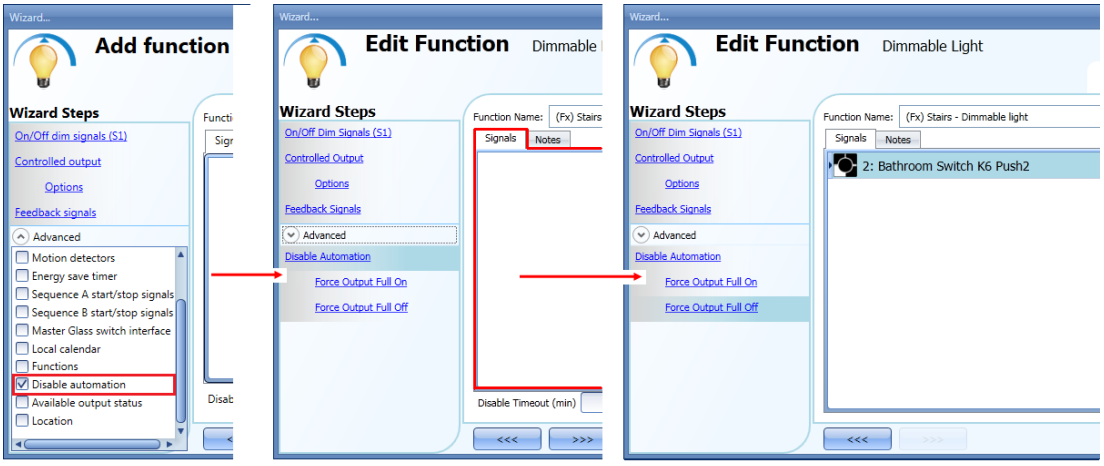
If the user wants to force the output On regardless of all other signals used in the function, the *Force output fully on* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output fully on*, then double click on the signal window and select the right signal to use.



Each signal used in the “Force output fully on” window works in level mode. Until the signal is active, the light is forced to the fully on status. When the signal is non-active, the light returns to the previous status. When the *force to fully on* signal is activated and at the same time the *force fully off* signal is activated, the *force to on* status has priority.

10.4.25 How to force the output OFF

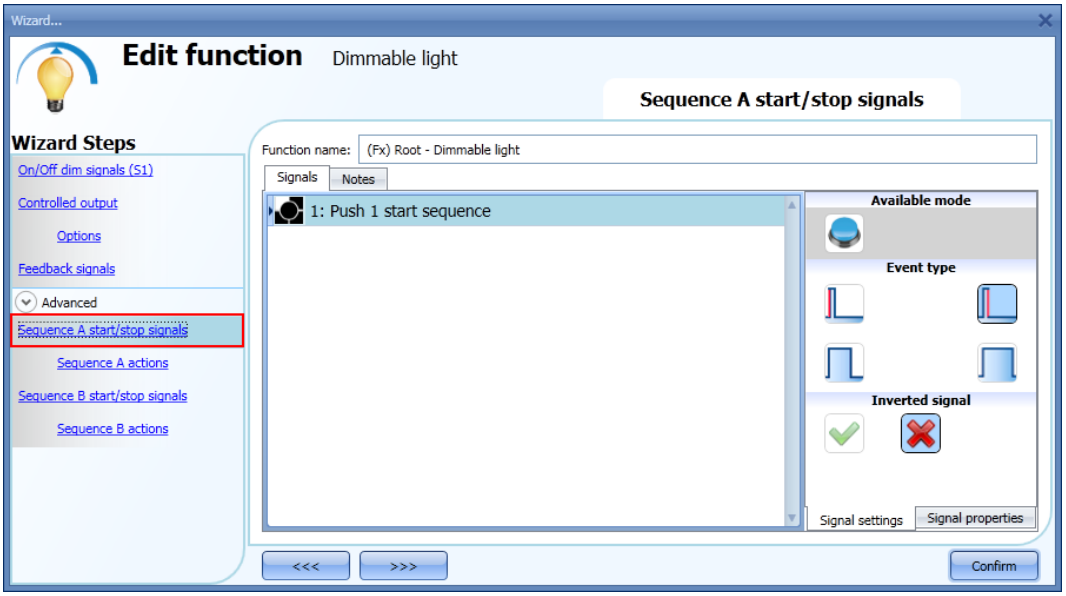
If the user wants to force the output Off regardless of all other signals used in the function, the *Force output fully off* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force the output fully off*, then double click on the signal window and select the right signal to use.



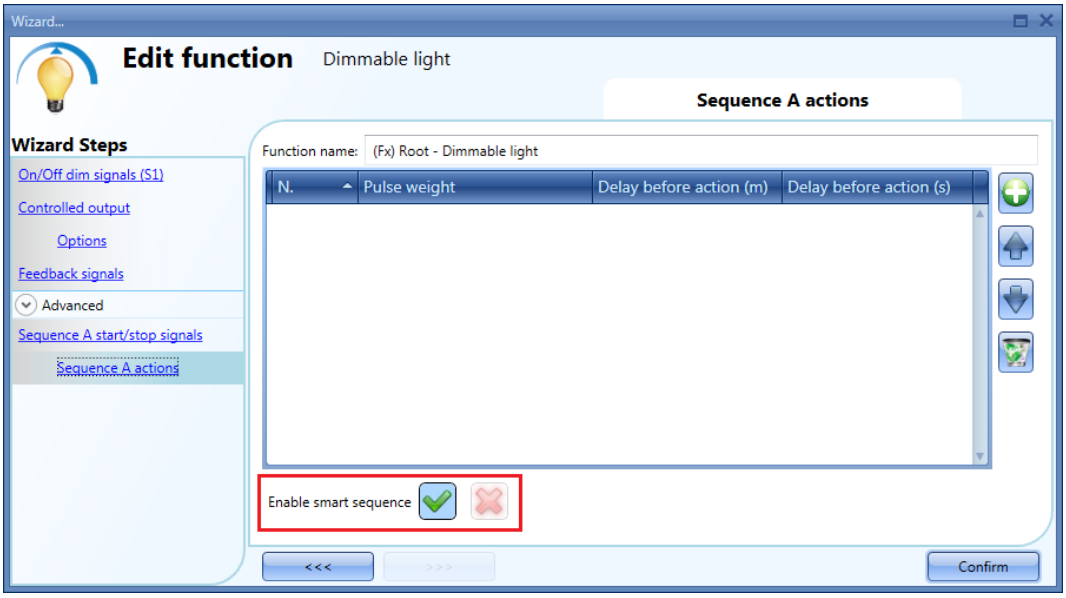
Each signal used in the *Force output fully* off window works in level mode.
Until the signal is active, the light is forced to the off status; when the signal is non-active, the light returns to the previous status.
When the force to fully on signal is activated and at the same time the force to fully off signal is activated, the *force to on status* has priority.

10.4.26 How to configure Sequence A

Sequence A automation is a list of commands for changing the light level and it can be used to create dynamic scenarios.
First it has to be enabled in the *Advanced* section.
The sequence is started when an On event occurs: the start/stop signal can be a pushbutton, a function or a remote command. The user can select short, long and very long activation of the input to start the sequence (see picture below).



The sequence is stopped when any dimmer command is generated.
For each signal the inverted logic can be enabled.

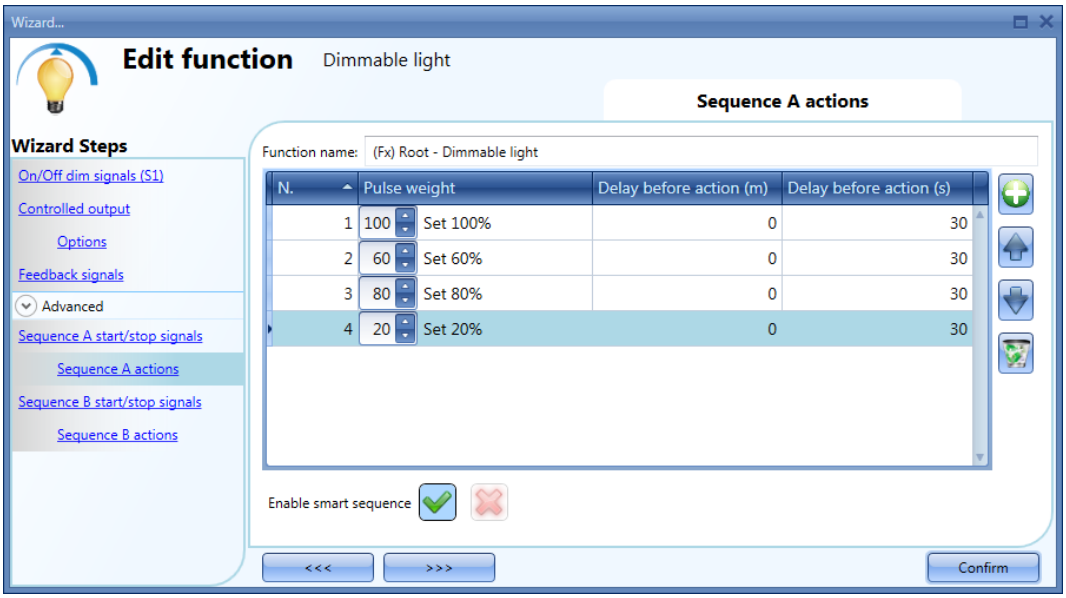


In the bottom of the window, the user can select to run the sequence as a “standard sequence” (executing all the steps added one after the other), or can select to run it as a “smart” sequence (see the dimmer sequence function behavior).
N.B. In the “smart” sequence the added scenarios must all make the light level go in the same direction either up or down. If the light has to go up and down, different sequences must be created.

10.4.26.1 Sequence A actions

In the sequence options window the user can manage each scenario of the sequence list.
In the window the user can set:

- The order of the scenario in the list.
- The percentage of dimming of each single scenario in the list.
- The pause to next time of each scenario in the list.



Sequence actions icons :

	Add a new Scenario step to the sequence list
	Move the selected Scenario step UP one position in the list
	Move the selected Scenario step DOWN one position in the list
	Delete the selected Scenario step from the sequence list

Delay before action: in this field the user can set the delay between one scenario and the following one.

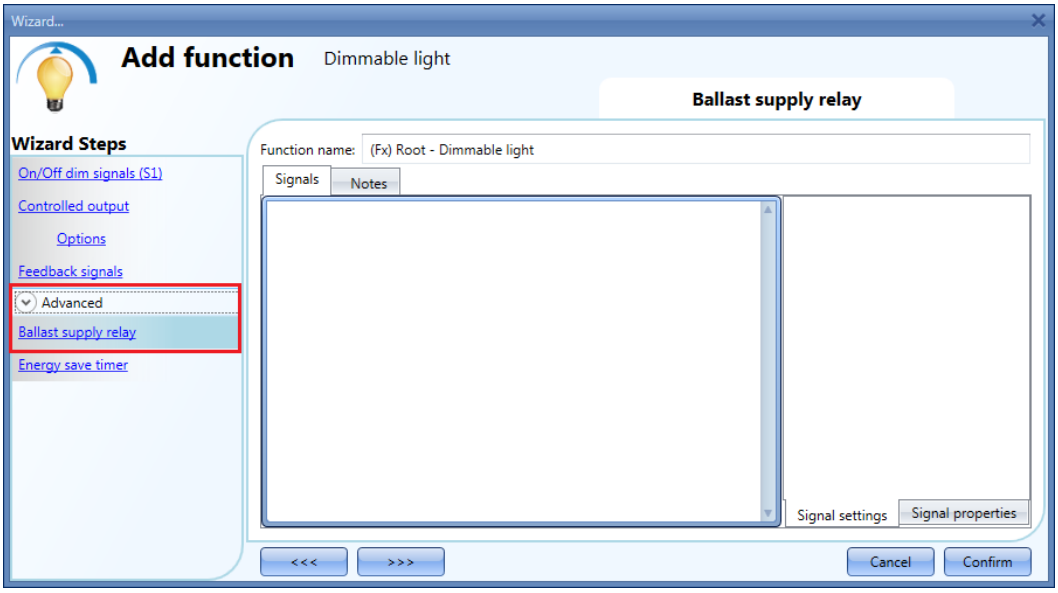
Scenario (Set%): in this field the user can select the percentage of dimming of each command. (0=OFF, 100%=fully ON) can be selected.

10.4.27 How to configure Sequence B

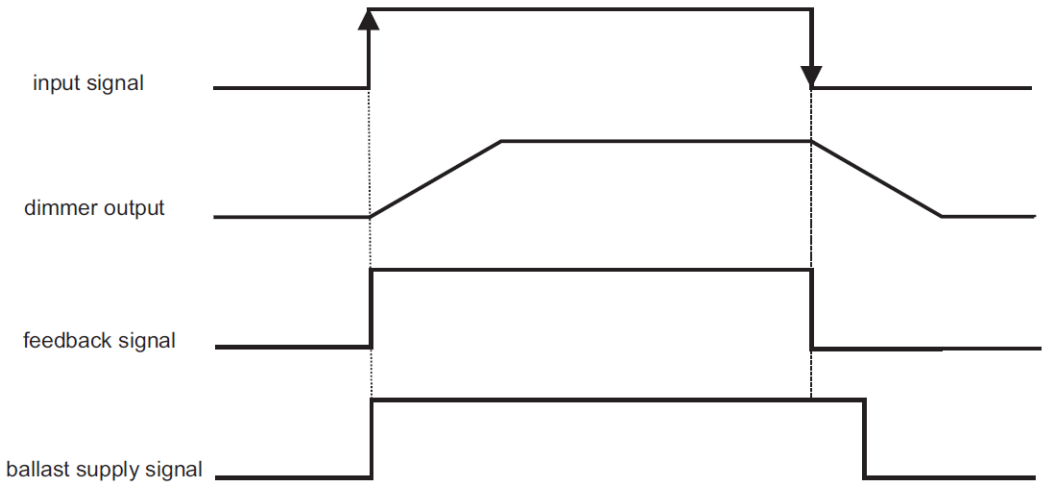
This sequence B is managed in the same way as sequence A.

10.4.28 How to manage the power supply of a ballast

If a 1-10 V dimmer is selected, a relay output has to be connected in serial of the power supply of the ballast to switch it off, since the 1-10V just dimmer it.
The relay output has to be added in the advanced field *Ballast supply relay*, in this way the system will manage automatically the switching of the relay.



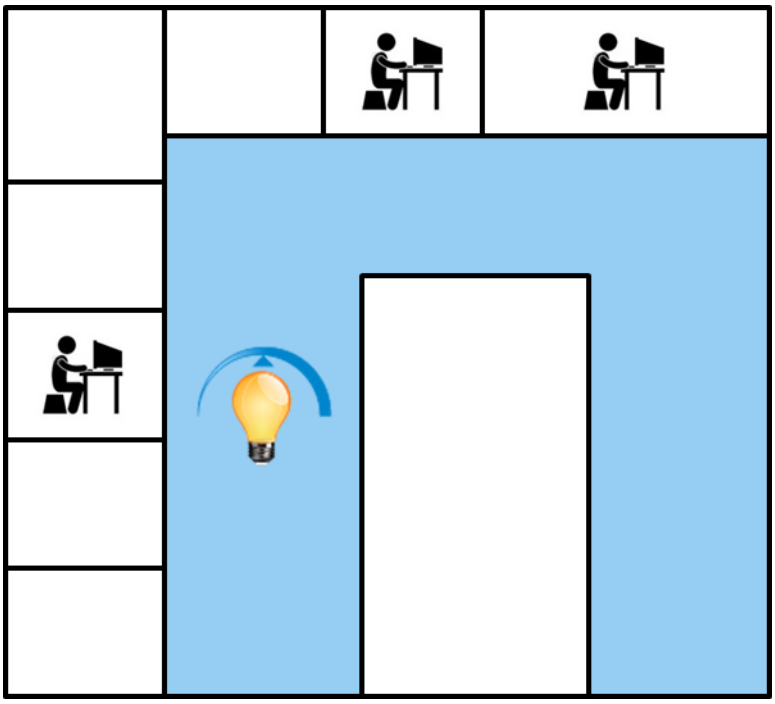
In the picture below it is showed the behavior of the signals according to the status of the dimmer function.



10.4.29 Corridor lighting function

This function is also an enhancement (additional option) to the dimming functions (*Dimmable light, Constant light, Light ON/OFF*).

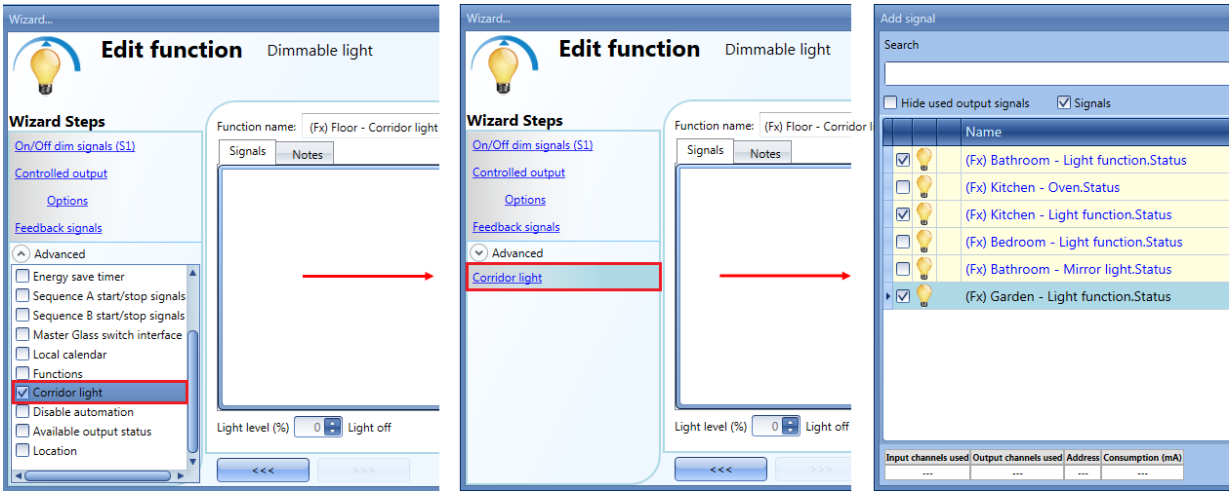
As shown in the picture below, the main aim of the Corridor lighting function is to control the lighting in the corridor, based not only on the presence of people in the corridor, but also on the presence of people in the adjacent offices. If there are people present in one or more offices, the corridor light is not allowed to switch OFF but has to be at a certain level.



10.4.29.1 How to set the Corridor lighting function

To set up a *Corridor lighting* function the user should select one of the existing *Light Functions* present in the project.

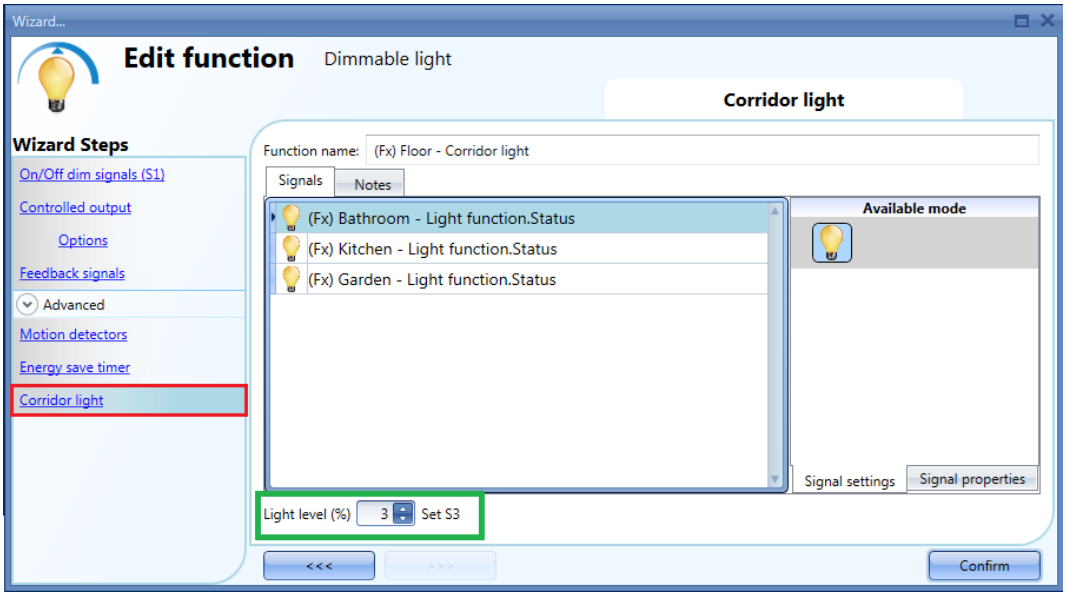
Since this is an Advanced functionality, the *Corridor light* field has to be enabled in the A*dvanced* section.



The Corridor light menu will appear. After selecting it, by double clicking on the Signals window the list of available signals will appear.

As shown in the picture above, the user may choose which offices he wants to include in the function, by selecting the relative signals: they can be function status or physical signals.
If more than one signal is selected in this window (you can chose up to 50 signals), the function will be activated/deactivated if at least one signal is active (the system does the logical OR).

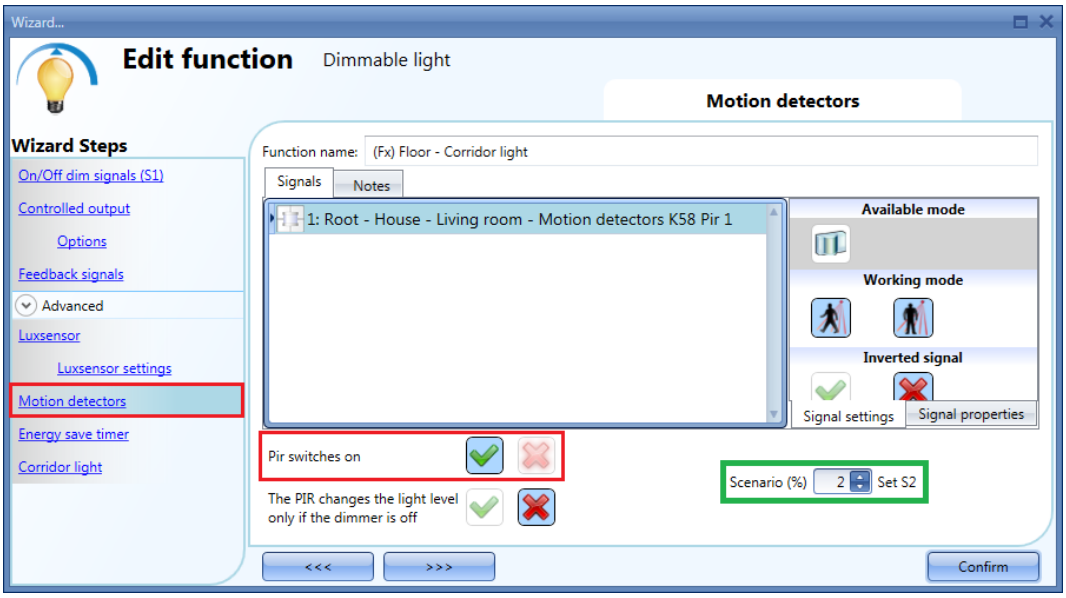
The control of lighting in the corridor is based not only on the presence of people in the corridor, but also on the presence of people in the adjacent offices. If there are people present in one or more offices, the corridor light is not allowed to switch OFF: it has to be at a certain level (e.g. 20%), in this case S3 scenario. See the green rectangle in the picture below.



10.4.29.2 How to turn a light ON with PIR sensors combined with Corridor light function

If the *Corridor light* option is selected, the user must select at least one signals which he wants to include in the function in the Advanced section.
This will be used by the *corridor light* function to detect if there is "people presence" in the adjacent offices. The signals added in this field are used to keep the light ON at the level defined in the field *Scenario (%)* (green rectangle, picture above, S2 scenario), but they are not used to switch the light ON.

Anyway, as usual, a PIR signal can be used to switch the light ON in the *Motion detector* field and the user should select the green "V" in the option "Allow the PIR to switch on". In this way, each time the PIR detects movement, the light will automatically be switched ON to a certain Scenario. In the green rectangle in the example shown below, the S2 scenario has been selected.



When the PIR does not detect any movement in the corridor, the light returns to the light level set in the Corridor light options (S3 scenario).

If the office lighting is not PIR-based, the user should use "*light function output*", as shown in the previous paragraph.

10.4.29.3 How the Corridor light function works

There are different possible scenarios, depending on the presence conditions in the offices adjacent to the corridor.

10.4.29.3.1 Presence of people in offices is TRUE

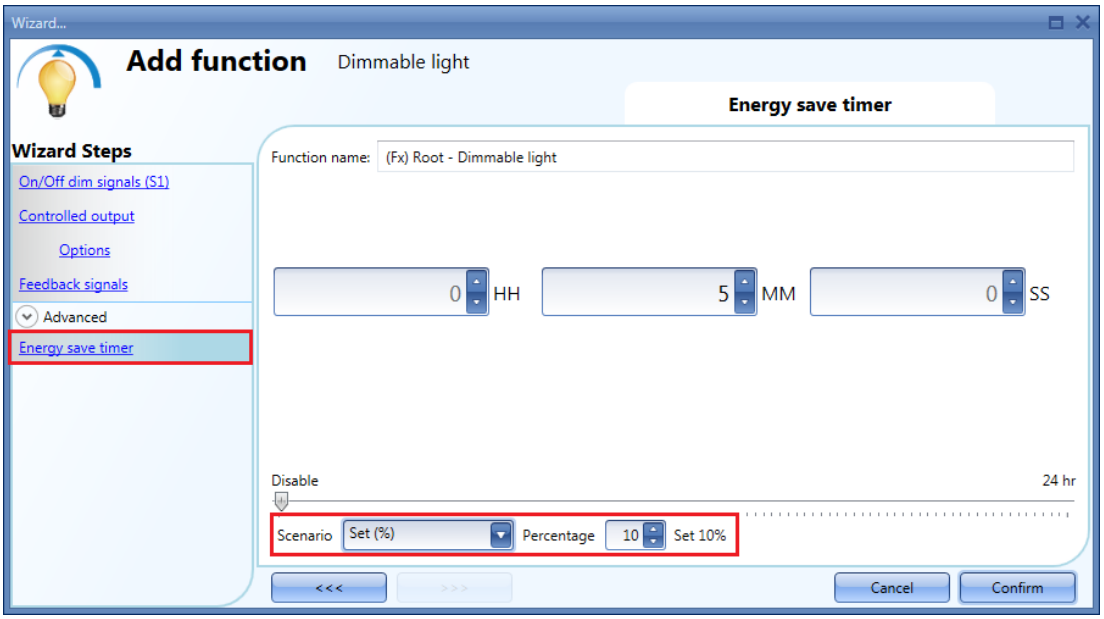
In this situation, one or more of the selected *PIRs* are ON or one or more of the selected *light functions* are ON

As long as "people presence" is TRUE, the corridor light function works as a PIR-based scenario control, where the light switches between, for example, S2 scenario and S3 scenario depending on whether the PIR is ON or OFF.

10.4.29.3.2 Presence of people in offices is FALSE

If an energy save timer is used to switch the light OFF or to set the light at a lower level, it starts counting down when all the signals in the field *Motion detector* and *Corridor light* are OFF: this means people presence neither in the corridor nor in the adjacent offices.

In the example shown below, while the user is in the corridor the light stays on S2 scenario and the energy-save timer is continuously reloaded; when no more people are detected in the corridor and in the adjacent offices, the energy-save timer starts counting and the light is switched to a lower level, e.g. 10%. In the example shown below an energy save timer of 5 minutes is set by the user.

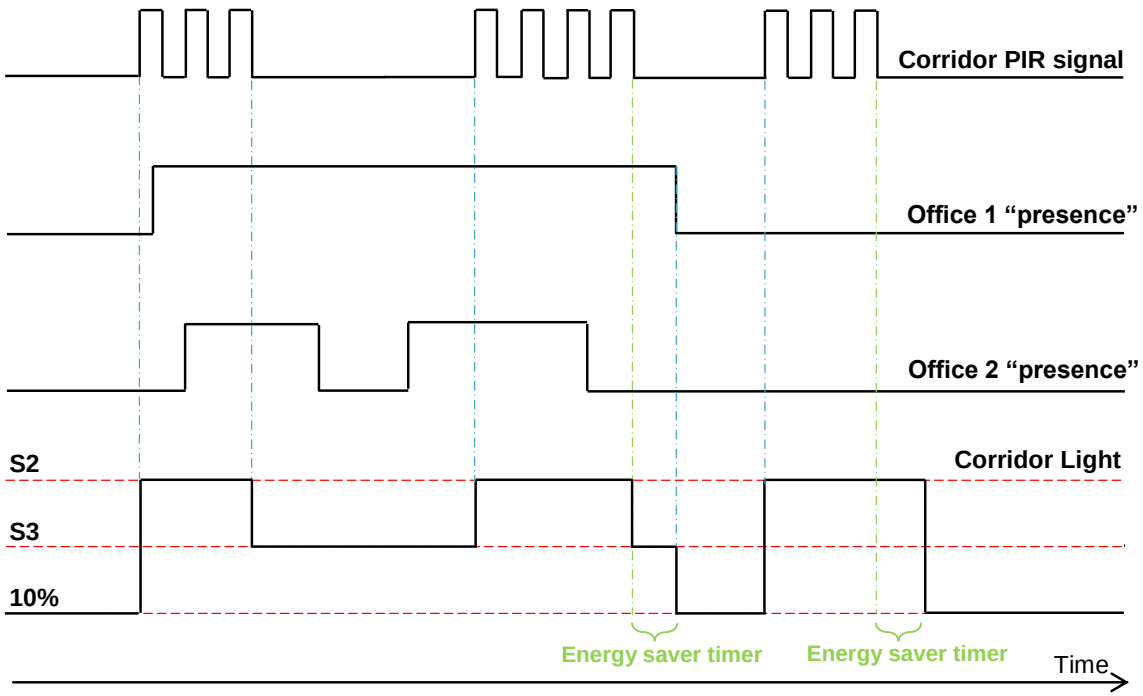


In the example shown below, a PIR is present in the corridor: when it detects any movement, the corridor light switches ON to S2 scenario and remains at this level until the PIR detects any movement in the corridor.

As long as at least one or more adjacent offices are occupied, and the PIR doesn't detect any movement in the corridor, the corridor is set to the S3 scenario.

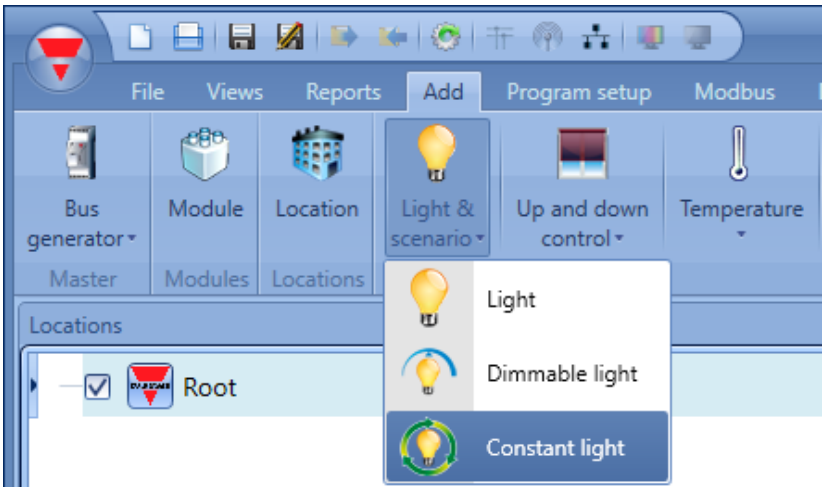
When the PIR redetects any movement in the corridor, the corridor light returns ON to S2 scenario. If the PIR doesn't detect any movement and there is no people presence in the adjacent offices, the corridor light goes to energy saver light level, e.g. 10%, as set by the user in the *Energy saver timer* field.

When the PIR in the corridor detects any movement, the corridor light switches ON to S2 scenario and the "people present" operation resumes.



10.5 How to set a Constant light function

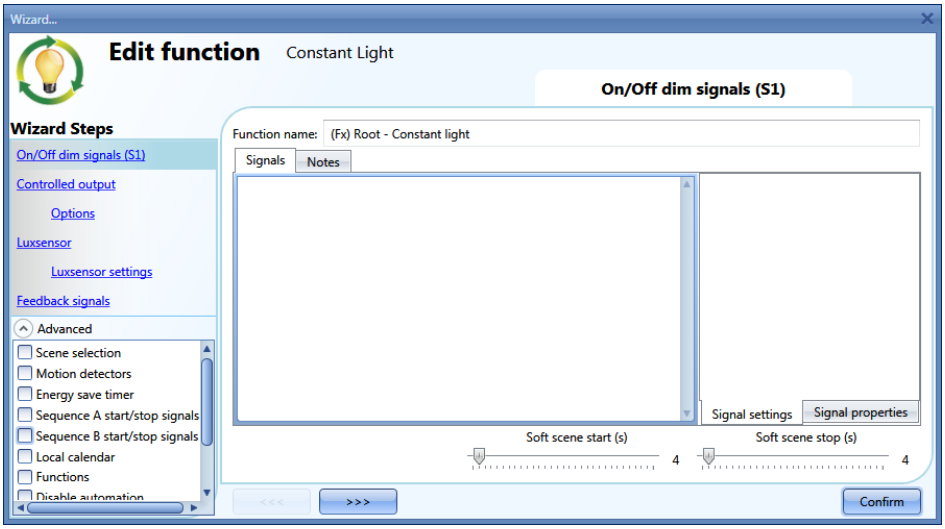
This function automatically regulates a constant light level using dimmers. This function needs a luxmeter to measure the light level.
To set up a constant light function the user should select *Light Functions* from the Add menu, then select *Constant light* (see picture below). The new function will be added into the selected location.



This function manages one or more dimmer outputs controlled by one or more input commands. The command might be a real signal such as a pushbutton, a function or a remote command (Webserver, sms, email, Modbus TCP/IP)

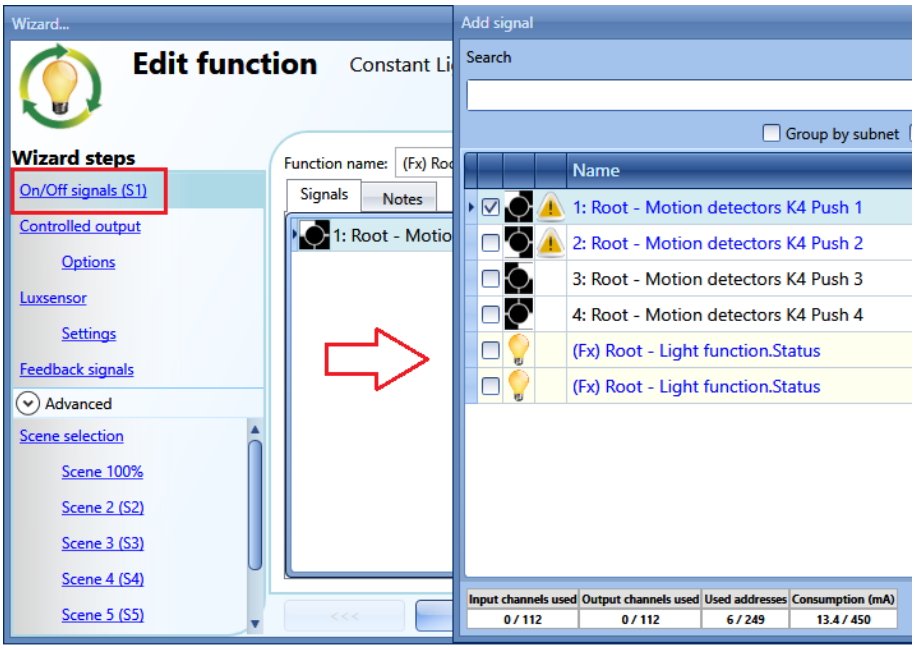
The algorithm to maintain the light constant can be stopped to remain at a predefined dimming output, the manual mode. The system can exit this status by means of a timeout, by pressing a button again or by switching off the light.

The automation of the Constant light function is managed by accessing the *Advanced* section (see picture below).
In the Advanced section the user can select different ways of controlling the constant light: according to the presence of people, with timers and/or schedulers. Up to 5 different predefined scenarios can be set.

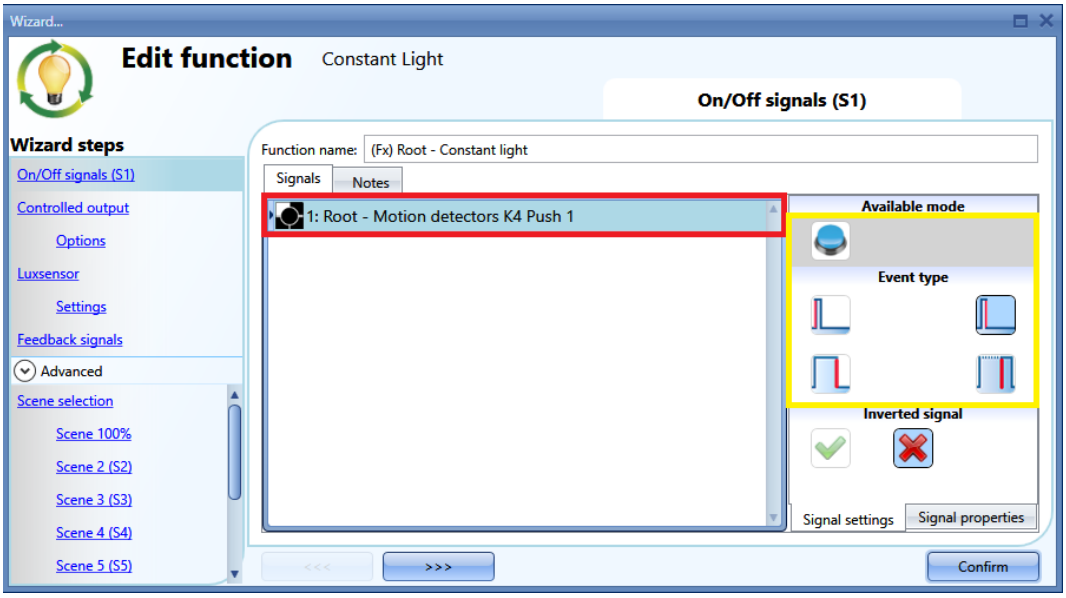


10.5.1 How to set a light with input commands

First of all, the user has to add the input signals to control the dimmable light in the *On/Off Dim Signals (S1)* field.
The input signal can be a pushbutton or a switch and can manage both the ON/OFF switching and the light level adjustment.

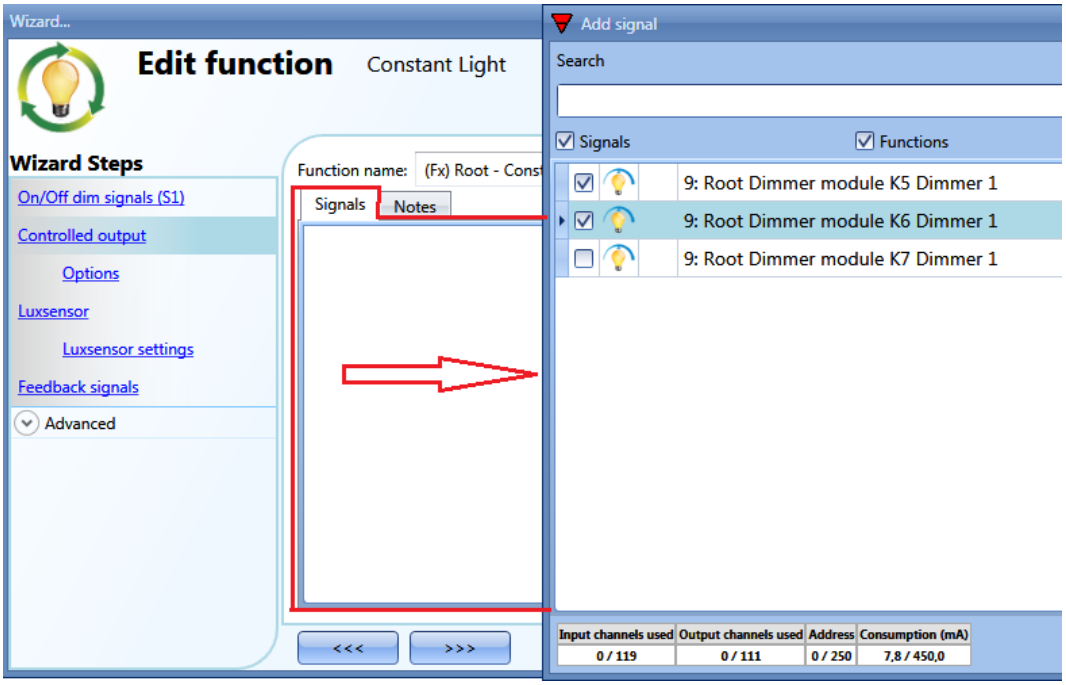


The light is switched On/Off according to the selected even type.



10.5.2 How to select a dimmable output

To select the output signal that is controlled by the dimmable light function, click on *Controlled output* and then double click on the Signals window. Once the output window is opened, select the relevant outputs from the list.

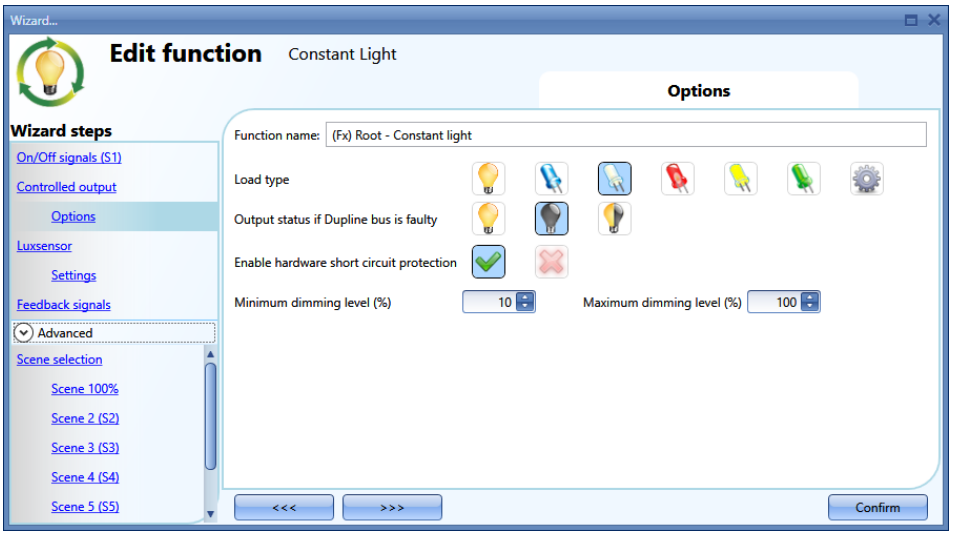


The signals available in the output window are only Dimmer output type.
Up to 100 signals can be chosen and they are managed in parallel.
If an output from a 1-10V dimmer is selected, a relay output has to be connected to the relevant ballast to control ON and OFF. The relay output should be added in the *Ballast supply relay* field.

N.B.: If more outputs are selected, they should be placed in the same dupline network and must be of the same type.
Dimmer outputs placed in different networks cannot be added together in the same Dimmable function.

10.5.3 How to change settings and the load type of the output

To select the settings of the output, the user has to click on *Options*.
If more outputs are selected, the settings are applied to all of them and to the slave zone outputs as well.



Load type:
This option is present only if the dimming output is 230V.
In this field the type of load connected to the dimmer module has to be selected.
If the module is a 230V dimmer, the load can be a standard RLC one (halogen lamp, inductive or electronic transformer, …), or a LED lamp. The user can select from three different types of LEDs: please see the dimmer datasheet for more details on how to select the correct LED load.

Enable hardware short circuit protection:
This option is present only if the dimming output is 230V.
When this option is enabled (the green "V" is selected), the user will enable the hardware short circuit protection of the output Dimmer module. Otherwise (when the red "cross" is chosen), the user will disable the hardware protection.

Minimum dimming level
This is the minimum dimming level that can be configured for the outputs of the functions: some LEDs flickers at low dimming level, so this avoids that the LED doesn't work properly.

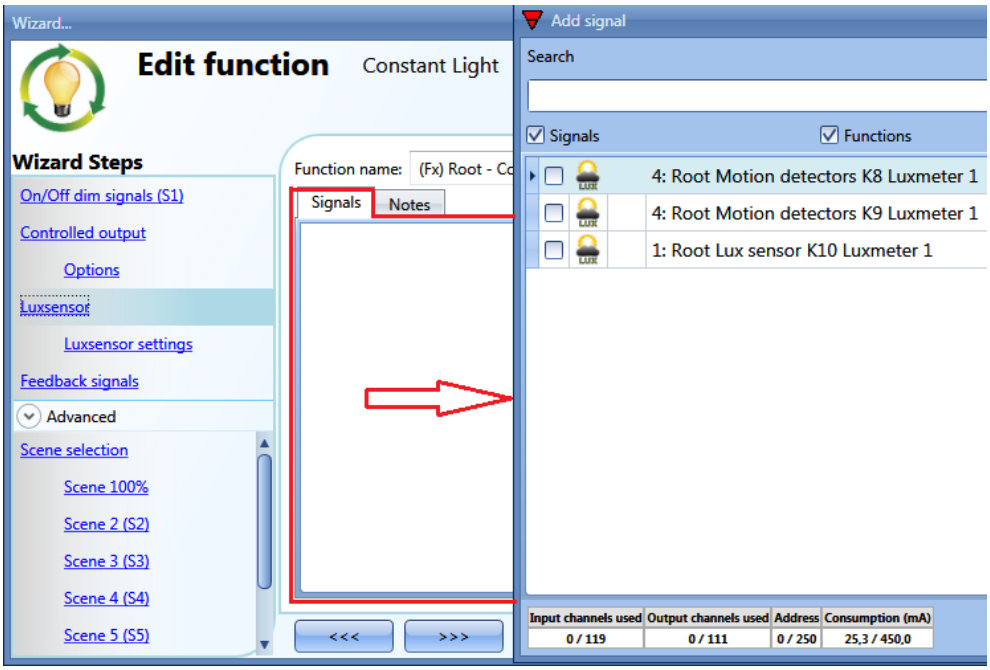
Maximum dimming level
This is the maximum dimming level that can be configured for the outputs of the functions: sometimes the lamp is over dimensioned, so reducing the dimming level gives a better energy saving.

Power on level
This field is present only if the dimming output is a DALI one.
It indicates the light level of the DALI ballast when it is powered and before it receives any command from the controller.

Light level if DALI bus is faulty
This field is present only if the dimming output is a DALI one.
This field indicates the light level of the ballast when the DALI bus is down.

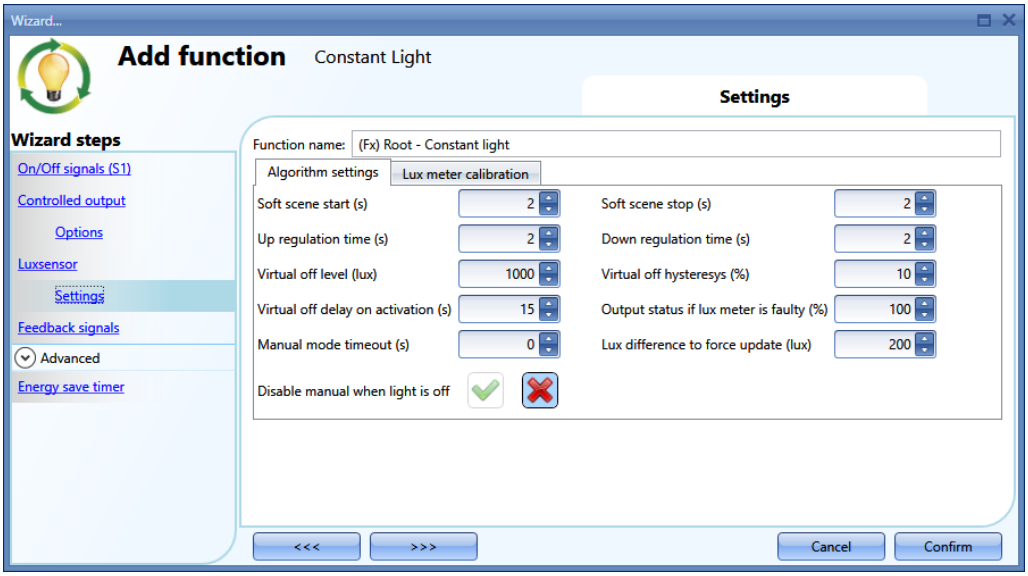
10.5.4 How to add the luxsensor

Since the light level has to be measured in the constant light function, the user must add a luxmeter in the relevant field Luxsensor.



The user can add up to ten lux sensors: if more lux signals are linked to the function, the average lux value is calculated and then used.

Once a lux sensor is selected, the user can change the settings, as shown below:



Soft scene start/s
This is the time the function will take to arrive at 100% starting from 0. We suggest to set 4 as the longest time.

Soft scene stop/s
This is the time the function will take to arrive at 0 starting from 100%. We suggest to set 4 as the longest time.

Up regulation time/s
Up regulation time must be >= than *Soft scene start*
The constant light algorithm will increase the light level by 1% every n second, defined in this field, to reach the setpoint.
Example: if the constant light function is regulating at 25% and a new setpoint is required that corresponds to 45%, it will take 20 x 2 =40 seconds to arrive at the new target

Down regulation time/s
Down regulation time must be > =than *Soft scene stop*
The constant light algorithm will decrease the light level by 1% every n second, defined in this field, to reach the setpoint.
Example: if the constant light function is regulating at 75% and a new setpoint is required that corresponds to 60%, it will take 15 x 2 =30 seconds to arrive at the new target

Virtual off level (Lux)
In this field the user can set the light threshold, above which the output is switched off, to save energy (of course the status of the function stays on and regulation will start again as soon as the measured level goes below this threshold).
This threshold should be set at a level corresponding at least to the lux measured at 100% of the dimming output, plus 30% to avoid on/off switching during the regulation.

Virtual off hysteresis (%)
In this field the user should enter the hysteresis value needed to switch the light ON again when its level goes below the Threshold – Hysteresis value. (A value between 5 and 50% can be set).

Virtual off delay on activation/s
In this field the user can set a delay (in seconds) to avoid the turning on/off of the light if a cloud shields the sun for a short time. If the time is set to 0, the filter is disabled.
We suggest setting this at a minimum of 60 seconds in order not to have the light turning ON/OFF continuously.

Output status if lux meter is faulty (%)
In this field, the user can set the output value, should the lux sensor be faulty.

Manual mode timeout/s
If the manual mode has been selected by using the pushbuttons programmed in the scenarios field, the algorithm will be stopped and the light will stay at the level defined with the manual command. The constant light algorithm will start again after the delay defined in this field. This counter is not subject to any disable automation commands.
If 0 is entered, the manual mode has to be disabled manually or by switching the function off.

Disable manual mode when light is off
If this option is enabled, every time the constant light function is switched off the manual mode is disabled and once the function is activated again, it will be regulated according to the constant light algorithm.

Lux difference to force update (lux)

When the difference between the measured value and the light setpoint is higher than that which is entered in this field, the algorithm will work in *fast* mode, while if the difference is smaller than this value, the algorithm will work in *slow* mode.

The constant light algorithm
This works in *fast* mode when the |*setpoint – measured value*| < the *Lux difference to force update*.
It works in *slow* mode when |*setpoint – measured value*| > the *Lux difference to reach the setpoint.*

Fast mode: the output value is increased/decreased according to the following table

Fast mode algorithm			
Increment		***Decrement***	
Present output	*New Output*	*Present output*	*New Output*
<10	*15*	*>10*	*Actual - 10*
<=15	*Actual + 5*	*>=15*	*Actual - 5*
<=30	*Actual + 10*	*>=30*	*Actual - 3*
<=60	*Actual + 15*	*>=60*	*Actual - 2*
<=100	*Actual + 20*	*>=100*	*Actual - 1*

The output is changed according to this table every *Up/Down regulation time*

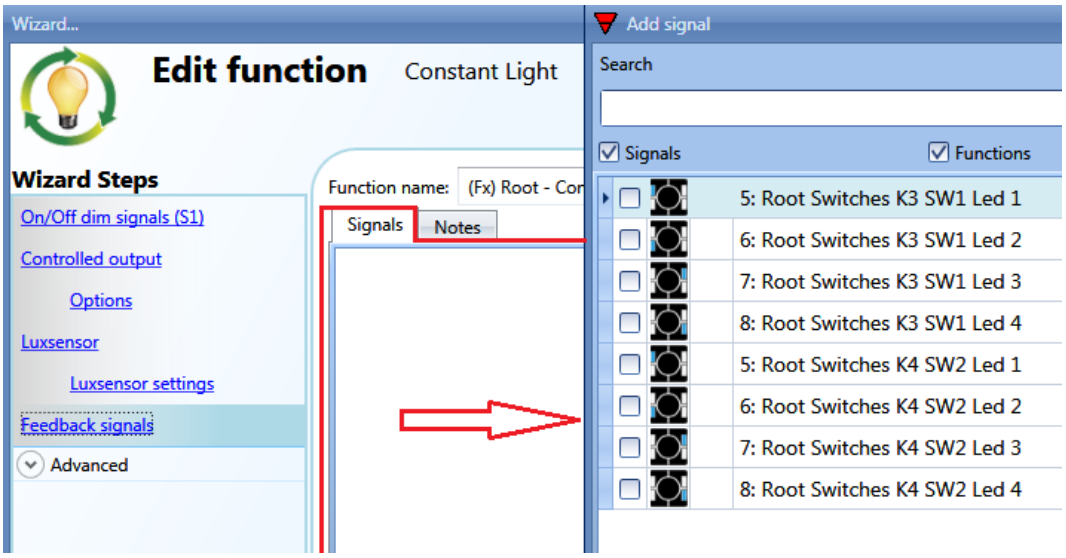
Slow mode: the output is adjusted by a step of +/-1%. The first ten updates, after the activation of *slow* mode, are executed every *UP/Down regulation time*, then the update is carried out every *UP/Down regulation time* x 10 (up to 120sec).

The algorithm consists of the following rules:

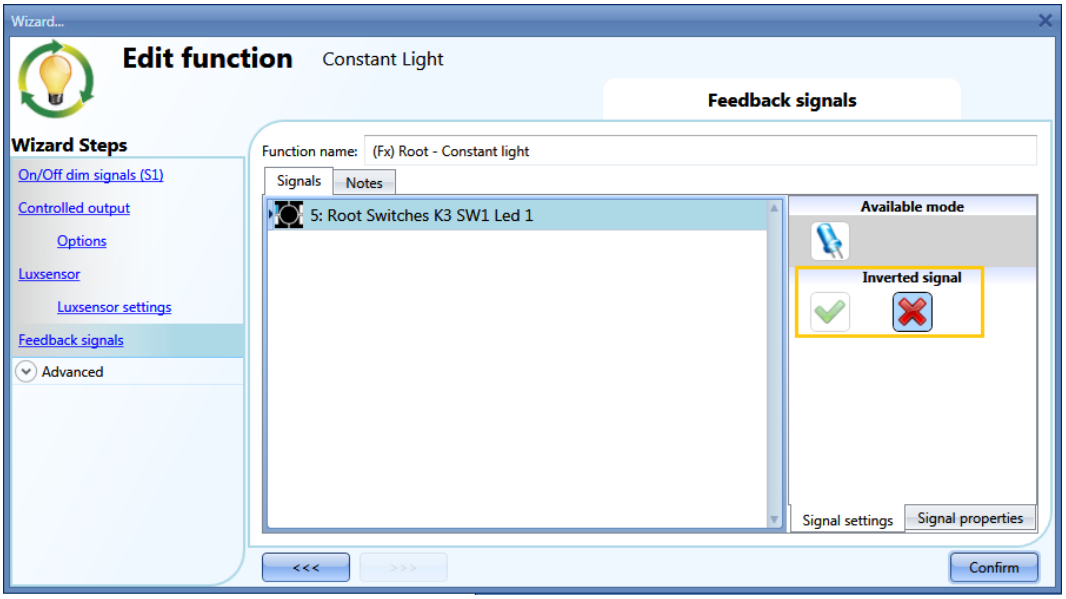
- Each time the scenario is changed, the *fast* mode regulation is activated
- In *fast* mode, the output is updated every *UP/Down regulation time*:
  - If the option *Start from percentage* is enabled, the first output value, generated after the scenario has been changed, is taken from the calibration table and the *fast* mode is switched immediately to *slow* mode.
  - If the option *Start from percentage* is *not* enabled, the output value is increased/decreased according to the present output value (see table) until the setpoint has been reached, then the fast mode is switched to slow mode.

10.5.5 How to set a signal as feedback of the status of the function

To select the feedback signal that indicates the status of a function, click on *Feedback signals* and then double click on the Signals window.



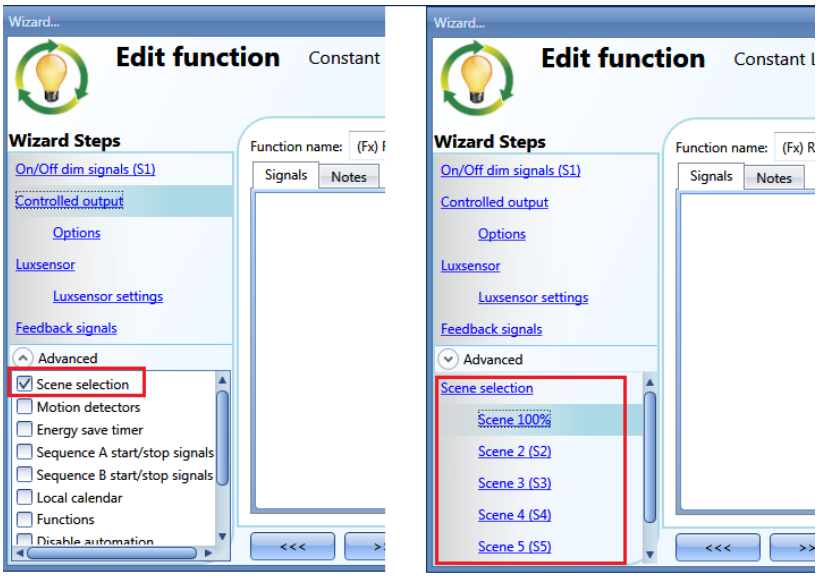
The signals available in this window are LEDs and relays.
Up to 50 signals can be chosen and they will be managed in parallel.
The logic of each feedback signal can be set as normal or inverted (see yellow rectangle).



10.5.6 How to set predefined scenarios with input signals

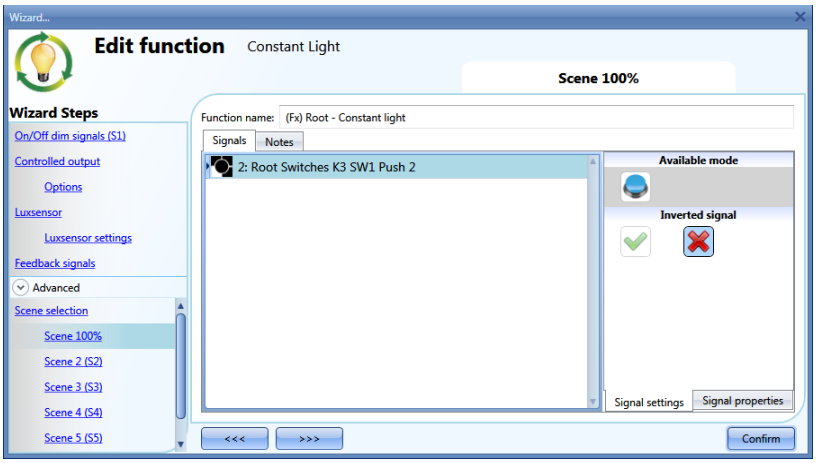
In this section, the user can customize different scenarios and select different input signals to activate them.
The user can set different output percentages and different timings for the soft start/stop.
Each scenario can be activated manually by adding an input signal in the signals window, or it can be activated by different automations such as PIR sensors and calendar.

The first thing to do to use different Scenarios is to enable them in the *Advanced* section.
The user should go to the advanced steps of the function and then check *Scene selection*. The user can select the scenario from the list, then with a double click in the signals field can add one or more inputs to activate the scenario.



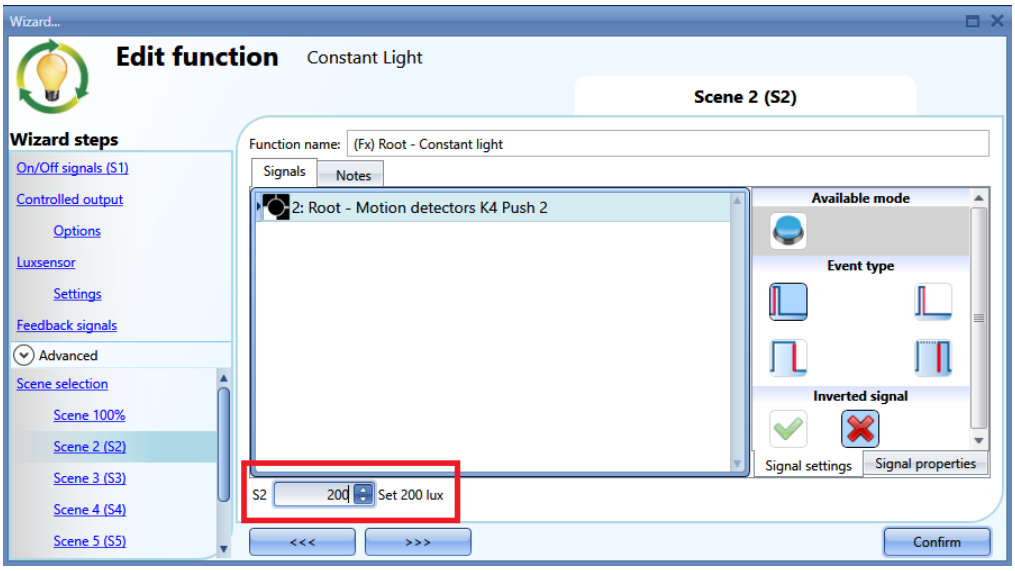
10.5.6.1 Scene 100%

When the scenario scene 100% is selected, all signals added in the window ‘signals’ will switch the light ON at the highest light level.
It is not possible to dim this scenario.
In the picture below, a simple example is shown of how to set a pushbutton to activate the scenario 99%.
Each time the pushbutton 1 is activated, the light level will be increased to the maximum value in manual mode triggering the timeout.
N.B. If this scenario is used, the Off threshold has to be set correctly in order not to have the light switching on and off continuously.



10.5.6.2 Scene 2 (S2)

In the Scene 2 configuration window, once the push button is added, the user has to select the option in the field marked in red to define the action when the button is pressed.



The available actions are described in the table below:

Value	Description of the action	Note
7	Toggle Manual / Automatic (set % / Lux)	Light function control is toggled Auto / Manual. When it is switched to Auto the last lux level is applied; when it is switched to manual the present dimmer output is maintained. Command is executed only when the function status is ON. The timeout to go back to automatic mode is **not** activated
8	Toggle Manual / Automatic (set % / Lux)	Light function control is toggled Auto / Manual. When Fx is switched to Auto the last Lux set is applied; when Fx is switched to manual the present dimmer output is maintained. Command is executed only when the function status is ON. The timeout to go back to automatic mode is activated
9	Manual @ present % +10%	Light function control is switched to Manual and the dimmer is increased by 10% up to 99% and then decreased by 10% down to10%. If this command is executed when the light is OFF, the first output value will be 10%. The timeout to go back to automatic mode is activated
10..99	Manual @ setpoint (%)	Light function control is switched to Manual mode and the dimmer output is set to the selected value (%). The timeout to go back to automatic mode is activated
100..30000	Automatic @ setpoint (lux)	Light function control is switched to Automatic mode and the algorithm will regulate the selected lux value
30001	Start sequence A	Sequence A is triggered
30002	Start sequence B	Sequence B is triggered

10.5.6.3 Scene 3 (S3)

This scenario is managed in the same way as scenario 2.

10.5.6.4 Scene 4 (S4)

This scenario is managed in the same way as scenario 2.

10.5.6.5 Scene 5 (S5)

This scenario is managed in the same way as scenario 2.

10.5.6.6 Scene 0%

When scenario *Scene 0%* is selected, all signals added in the window Signals will switch the light off.

10.5.7 How to manage lights automatically

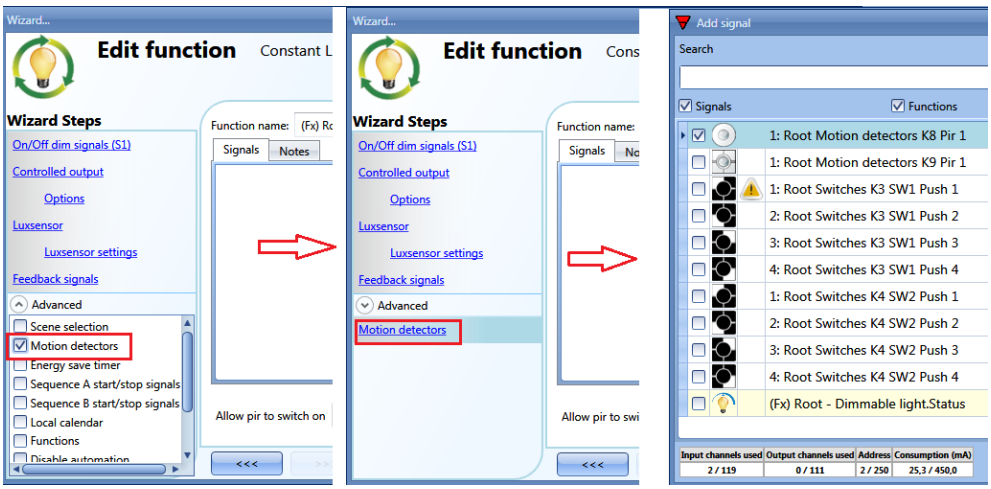
The automatic on/off switching of the light can be managed by PIR sensors (the light will be switched ON when the PIR sensor detects people movement/presence), by calendar functions (calendar to switch the light on/off at pre-defined time intervals) or by general functions.
All these automations can be enabled in the *Advanced* section.

10.5.8 How to select a PIR sensor to control scenario

The motion detector sensor can be used to perform different functions:

- switching the light on, on movement detection
- adjusting the light level to a predefined value in manual mode
- activating the automatic control at a lux level
- switching the light off if no presence is detected within a time interval
- starting a sequence that manages the activation of different scenarios

To set and mount a PIR sensor correctly, please read paragraph *How PIRs work and how to configure them*.
The first thing to do when a PIR sensor needs to be used in a constant light function is to enable it in the *Advanced* section. The *Motion detector* menu will appear. After selecting it, with a double click on the Signals window, the list of available signals will appear.
Select the required signal/signals and click on *Confirm*. Up to 50 signals can be selected and the system will perform a logical OR of all of them.

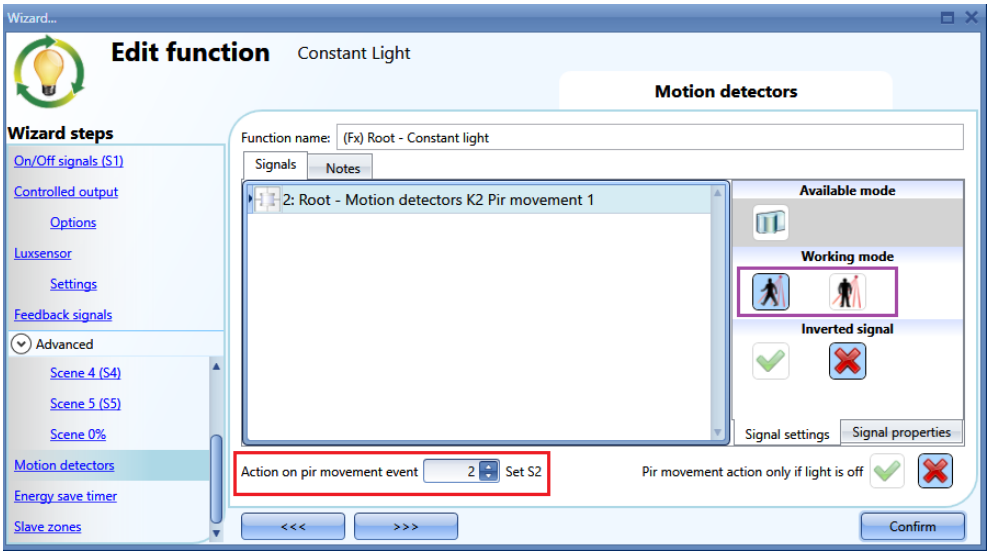


Once the PIR sensor is added, the user can also choose to invert the signal by selecting the green *V* below *Inverted signal*, in the Signal Setting tab window.

10.5.9 How to turn a light ON with PIR sensors, and turn it OFF manually

In order to create this easy automation, the user must select at least one input signal for the manual command, one output signal to control the load and then a PIR sensor in the *Advanced* section.
In this case the light will not be switched OFF automatically if no presence is detected, so the energy-save timer must not be used in the *Advanced* section.
To turn the light off manually, a push button has to be added in the *On/Off signals(S1)* or in one of the Scenarios in the *Advanced* section.

Once the PIR signal is entered, the user should select the action the system has to perform when the PIR is activated, also if it has to be executed only when the light is off or at any time.



The action has to be selected in the field *Action on PIR movement event* according to table A:

Table A:

Value	Description of the action	Note
-1	No action	
0	Off	Light is switched off
1	Toggle ON/OFF	Light is toggled OFF / last ON value. Last ON can be Manual (value in the range 10..99) or Auto (value in the range 100..30000)
2	S2	Light set @ scenario S2
3	S3	Light set @ scenario S3
4	S4	Light set @ scenario S4
5	S5	Light set @ scenario S5
6	Manual 100%	Dimmer output is set to 99%. The timeout to go back to automatic mode is activated
7	Toggle Manual / Automatic (set % / Lux)	Light function control is toggled Auto / Manual. When it is switched to Auto the last lux level is applied; when it is switched to manual the present dimmer output is maintained. Command is executed only when the function status is ON. The timeout to go back to automatic mode is **not** activated
8	Toggle Manual / Automatic (set % / Lux)	Light function control is toggled Auto / Manual. When Fx is switched to Auto the last Lux set is applied; when Fx is switched to manual the present dimmer output is

		maintained. Command is executed only when the function status is ON. The timeout to go back to automatic mode is activated
9	Manual @ present % +10%	Light function control is switched to Manual and the dimmer is increased by 10% up to 99% and then decreased by 10% down to10%. If this command is executed when the light is OFF, the first output value will be 10%. The timeout to go back to automatic mode is activated
10..99	Manual @ setpoint (%)	Light function control is switched to Manual mode and the dimmer output is set to the selected value (%). The timeout to go back to automatic mode is activated
100..30000	Automatic @ setpoint (lux)	Light function control is switched to Automatic mode and the algorithm will be regulated at the selected lux value
30001	Start sequence A	Sequence A is triggered
30002	Start sequence B	Sequence B is triggered

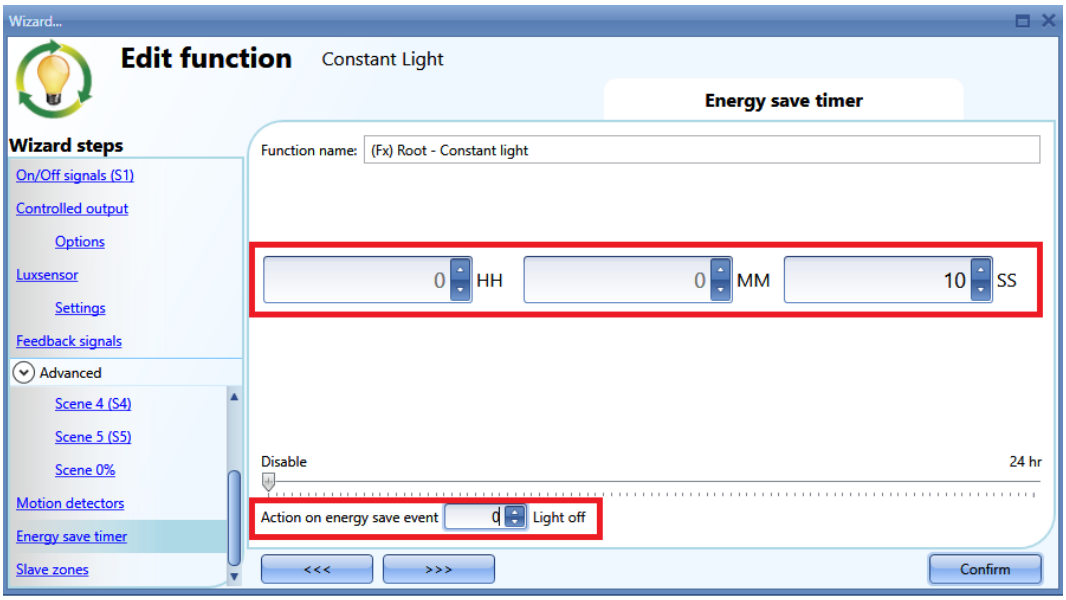
If the field *Pir movement action only if light is off* is enabled, the action will be carried out only if the light is already off. If this field is not checked, every time the PIR sensor detects a movement, the selected action will be performed.

Note
The action selected in the field *Pir movement action only if light is off* is executed by default only if the movement signal is selected. Should the user need to add the presence signal, they have to select the icon **.**

The reload of the energy save is done by default only if the presence signal of a PIR sensor is selected. Should the user need to add the movement signal, they have to select the icon

10.5.10 How to turn a light on and off with PIR sensors

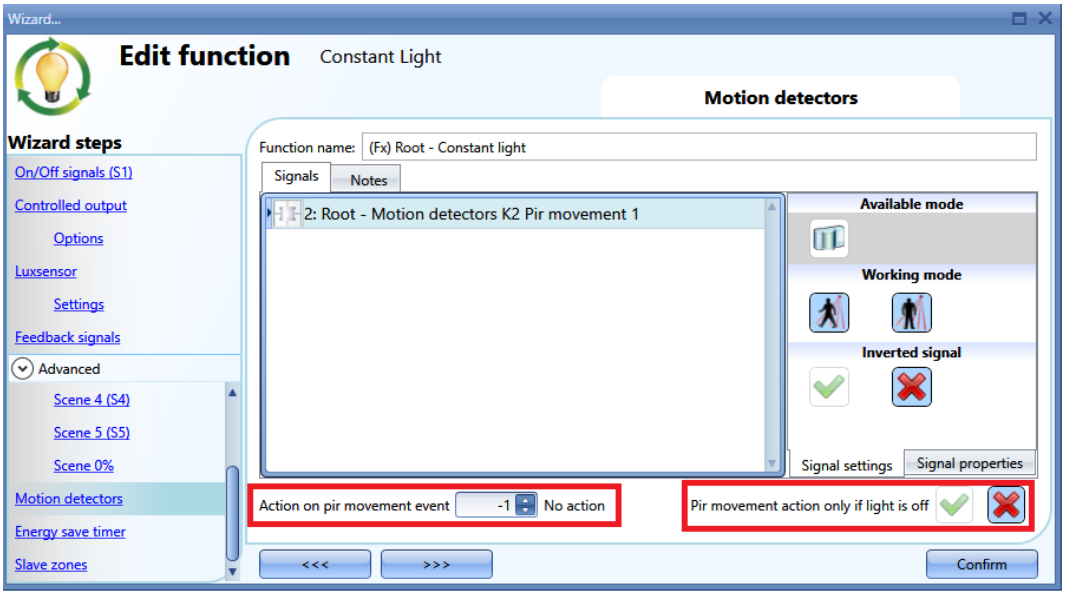
The first step is to add the output signal, then to add a PIR sensor with an action that makes the light switched ON, and then to set an energy-save timer to switch the light off automatically when no presence is detected.
To set an energy-save timer, select *Energy save timer* from the Advanced section, change the timing by moving the slider or typing hours, minutes and seconds, select the action to be performed at the end of the timer (event off), then press confirm.
The energy-save timer starts counting when no presence is detected. Each time presence is detected the timer is reloaded. When the timer expires, the light will be set according to the action set in the field *Action on energy save event* (see Table A).



10.5.11 How to switch the light on manually and off with the PIR sensor

An example of this automation is an office where the employees sometimes forget to switch the light off when they leave the office. When the PIR stops detecting presence in the office, the energy-save timer starts counting and when it expires, the light is turned off.

If the user wants to use the PIR sensor to switch the light off automatically, the following settings should be used.
Add a PIR sensor and an energy-save timer from the Advanced section.
In the PIR settings, the red cross in the *PIR movement action only if the light is off* field must be selected because the light has to be switched on manually. In the *Action on PIR movement event* field, the value -1 must be selected (-1 means no action when movement is detected); with these two settings the PIR cannot switch the light on and cannot set a different light percentage when it detects movement.
The energy-save timer must be set differently from zero, and will be triggered when the light is switched on. While the PIR detects presence, the timer is reloaded. When the PIR stops detecting presence, the timer starts counting down and when it expires the light is switched off accordingly.

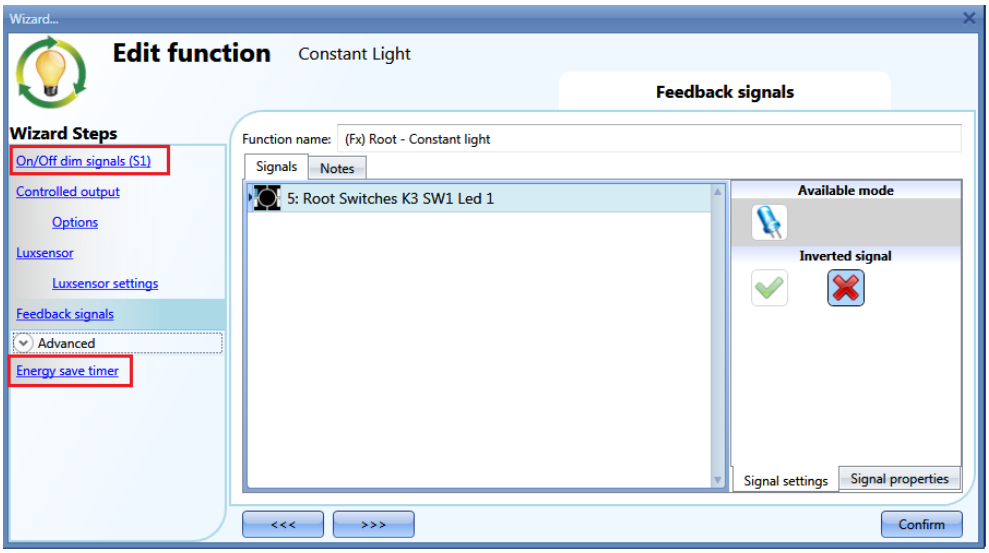


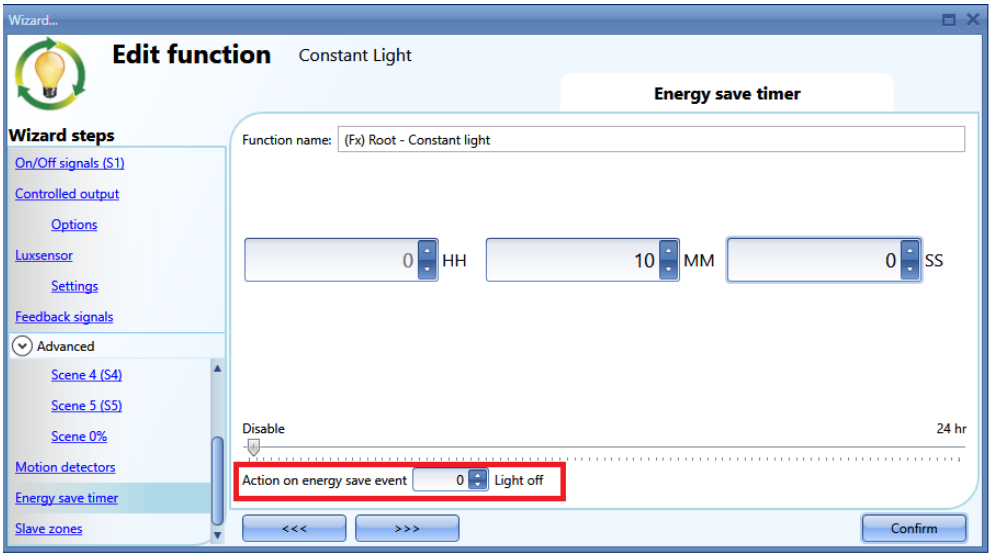
If a movement signal is chosen, to reload the timer the icon must be selected.

10.5.12 How to switch the light off after a predefined period

If the user needs to be sure that the light is switched off to save energy, the energy-save timer should be selected in the advanced functionalities (the energy-save timer is selected by default when the user creates a new Dimmer function).
The energy-save timer starts counting each time the light is switched On manually or by using the PIR sensor, causing the light to go off automatically when it expires.
The energy-save timer can be set up to a maximum value of 24 hours, whilst when it is set to 0 it is disabled. For this reason, the shortest time that can be set is 1 second.
The energy-save timer value can be changed by means of the webserver.

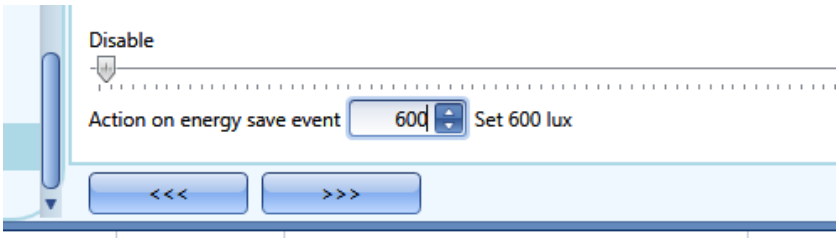
In the example below, the user has set a function with a pushbutton to switch the light on and an energy-save timer of 10 minutes to switch the light off automatically (*Light off* is selected as the action to be performed).





10.5.13 How to set a predefined scenario when the energy save timer expires

To select a different scenario when the timer expires, the user has to set the required light level in the *Energy save timer* field as shown below.



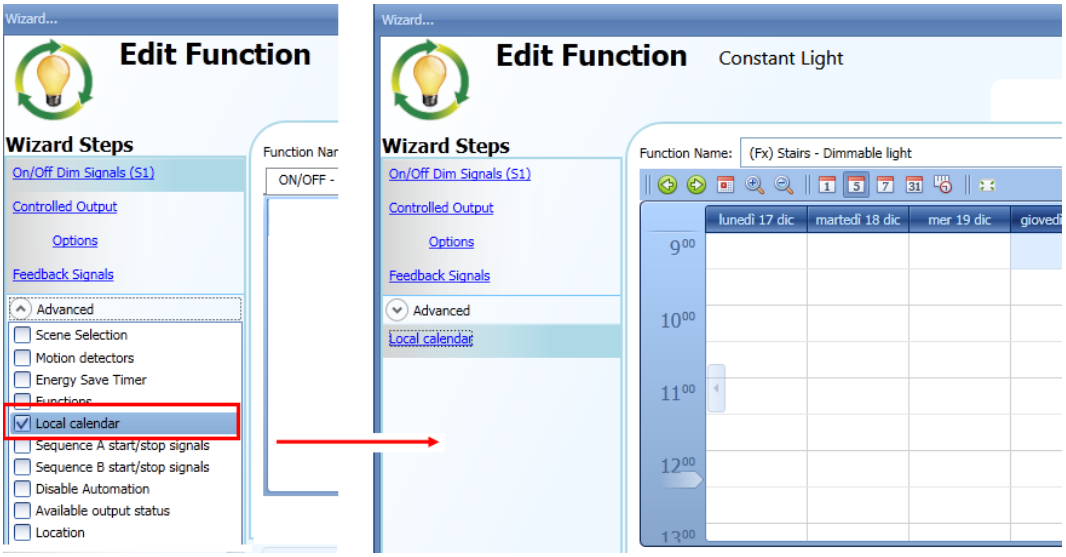
10.5.14 How to switch the light on/off using the calendar

An example of this automation could be a calendar that switches all lights off at a pre-set time in the evening.

There are two ways of managing constant lights with the calendar: by setting the “local” calendar inside the function or using a “global” calendar function.
If the function uses an energy-save timer, the calendar does not affect it in anyway.

10.5.14.1 The local calendar.

To enable this, the relevant menu has to be enabled in the Advanced section.



The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* shown below:

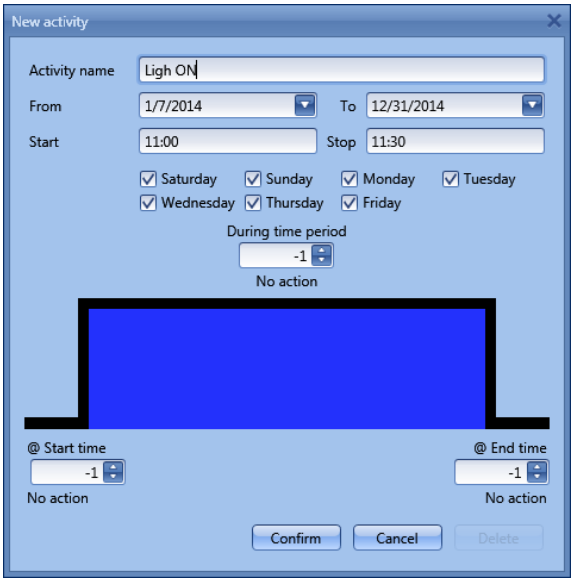
Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week
	One step forward in the calendar. E.g.: if a week is visualized, this button allows the user to go to the next week
	Go to the current day
	Magnifying lens to see more/fewer time bands
	The calendar visualizes one day horizontally
	The calendar visualizes 5 days horizontally
	The calendar visualizes 7 days horizontally
	The calendar visualizes 31 days horizontally
	The calendar visualizes 7 days vertically
	Full screen view

Calendar activities:

Event activity

Once the user has selected their preferred type of visualization, to enter a time band they should double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that is to appear on the calendar. This is a mandatory field.

From*:* This is the starting date for the activity.

To*:* This is the end date of the activity.

Start*:* This is the start time of the activity.

Stop: This is the end time of the activity.

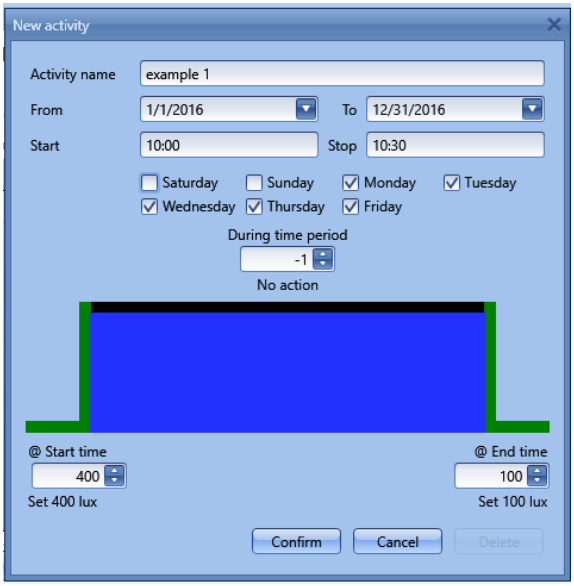
@ ***start time***: in this field the user can select the action to be performed at the start time.
The selectable actions are the ones listed in table A

@ ***end time***: in this field the user can select the action to be performed at the end time.
The selectable actions are the ones listed in table A

Days: The user should select the week days when the actions are to be performed.

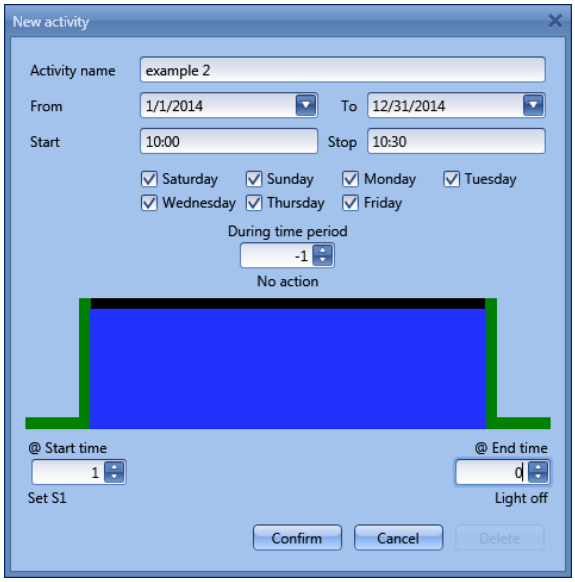
If ***Level activity*** is selected, the user can disable all the automations of the function (PIR sensors, lux meters, calendars).

- No action (-1)
- Disable automations (1)



In the first example, the calendar is set to work all the year round (from 1st January to 31st December) . The working days are: Monday, Tuesday, Wednesday, Thursday and Friday (the calendar does not work on Saturday and Sunday)
At the start time (10:00) the light is switched on at 400 lux: at the stop time the light level goes down to 100 lux.
All the activities are repeated automatically each year, so looking at the example below, once 2016 is finished, the activity will go on in the same way into 2017 and so on.

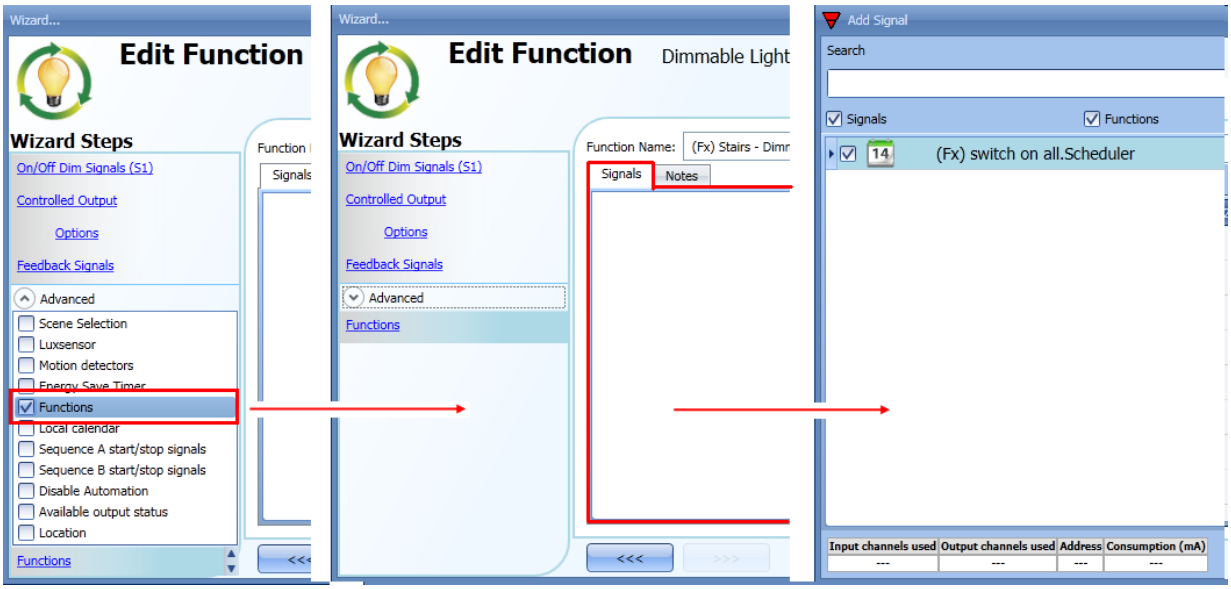
In the second example, the calendar is set to work all the year round (from 1st January to 31st December), every day.
At the start time (10:00) the light is switched on at scenario 1: at the stop time of 10:30 the light is switched off.



10.5.14.2 The global calendar function

Before using a global calendar, the user should define it as a function (See How to set a global calendar). The field function then has to be selected in the *Advanced* menu.
By clicking on Functions and then double clicking in the *Signal* window, the *Add signal* window will appear. Select the required calendar function.

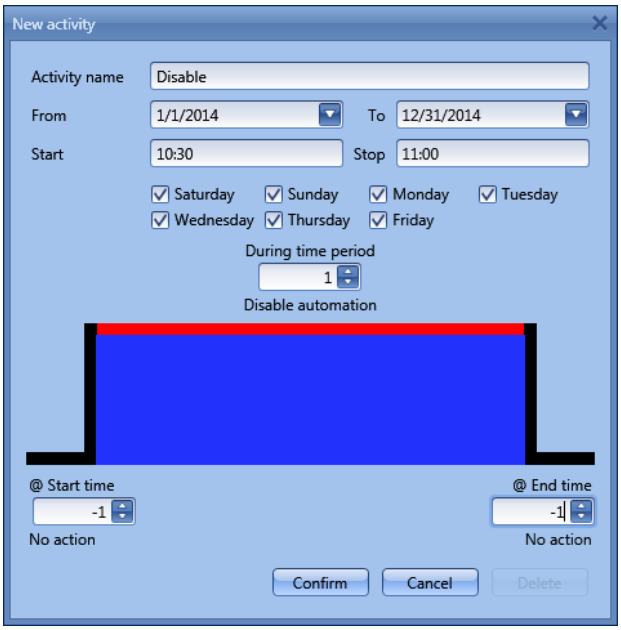
In the example below, the global calendar added is a calendar created for switching on all the lights. Its behaviour is the same as that described for the local calendar.



10.5.15 How to stop an automation

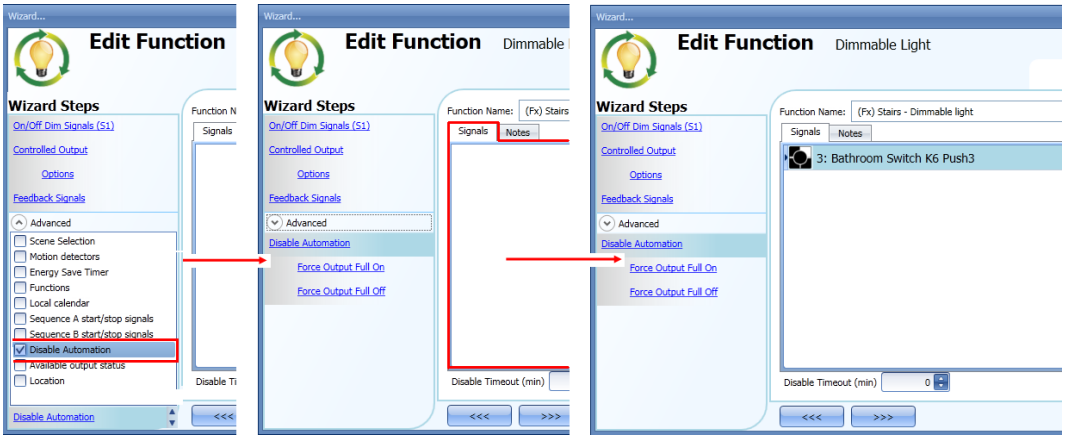
There are two ways of disabling the automation performed by PIR sensors, lux meter, calendar and energy-save timer: the first is with the calendar, the second with signals.

Disabling automation using the calendar
To disable the automation, the user can select the local or the global calendar.
The local calendar should be enabled, using the advanced functionality *Local calendar*, while the global calendar should be added as a signal in the *Functions* menu of the *Advanced* functionalities.
Regardless of the type of calendar added, the settings to disable all the automations are shown below.

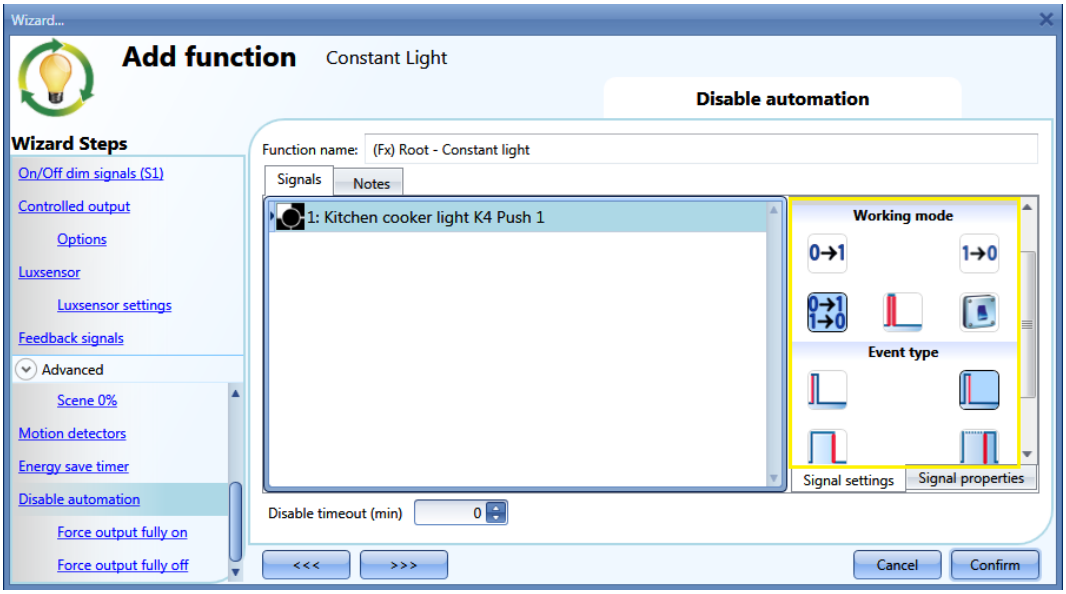


In the level activity the disabling of the automation is selected by entering the value 1. In this way, during the entire period when the calendar is active, all automations are disabled.

Disabling automation using signals
The second way of disabling an automation is by using signals: to enable this, select *Disable automation* in the *Advanced* section, double click on the signals window and select the right signal to use.



When the signal is added, the user should select the working mode and the type of the event for the signal:



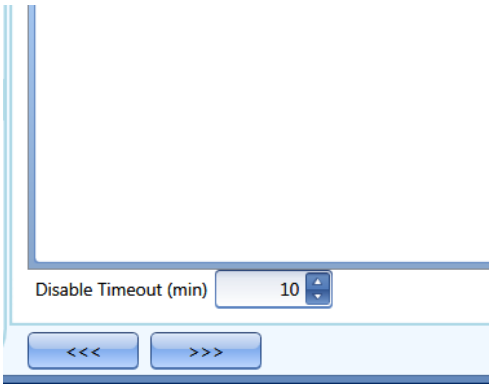
If a push button type is used, the way of working marked by the yellow rectangle should be selected following the table below.

	Event type			
Working mode				
0→1	As soon as the push button is pressed, the automation is disabled.	After a short press (less than 1 second), the automation is disabled at the release of the push button.	After a long press, the automation is disabled at the release of the push button.	After a very long press, the automation is disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automation is enabled again.	After a short press (less than 1 second), the function automation is enabled again at the release of the push button.	After a long press, the automation is enabled again at the release of the push button.	After a very long press, the automation is enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the automation will be enabled/disabled in toggle mode.	After a short press (less than 1 second), the automation will be enabled/disabled in toggle mode at the release of the push button.	After a long press, the automation will be enabled/disabled in toggle mode at the release of the push button.	After a very long press, the automation will be enabled/disabled in toggle mode at the release of the push button.
	The automation will be enabled/disabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode has to be carried out following the table below:

	Event type	
Working mode	Signal activated	Signal deactivated
1→0	The automation is enabled again	No action
0→1	The automation is disabled	No action
0→1 1→0	The automation is disabled/enabled in toggle mode	No action
	The automation is disabled/enabled in toggle mode	The automation is disabled/enabled in toggle mode
	The automation is disabled	The automation is enabled

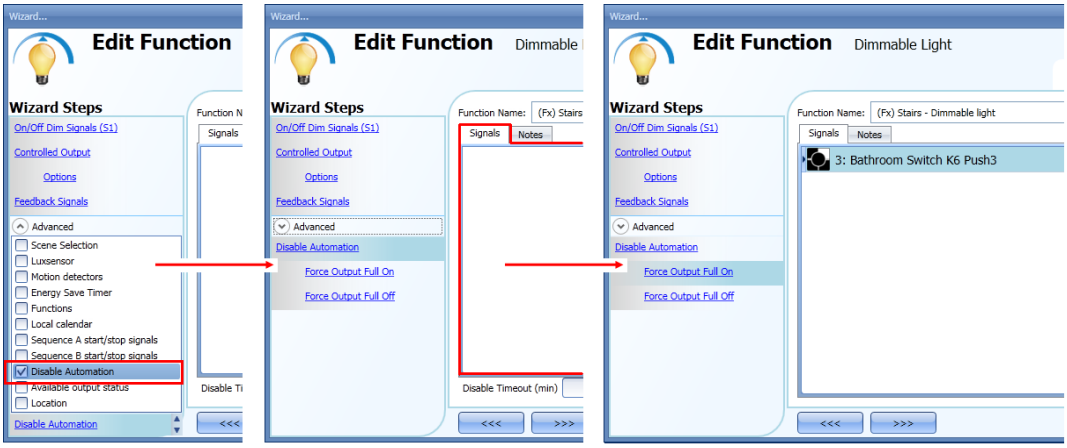
The user might need to set a time after which the automation is to be enabled again, even if the selected signal is still active. To do this the *Disable timeout* field has to be set.



In the figure above, a disable timeout of 10 minutes is set. The maximum delay is 59 minutes.

10.5.16 How to force the output fully ON

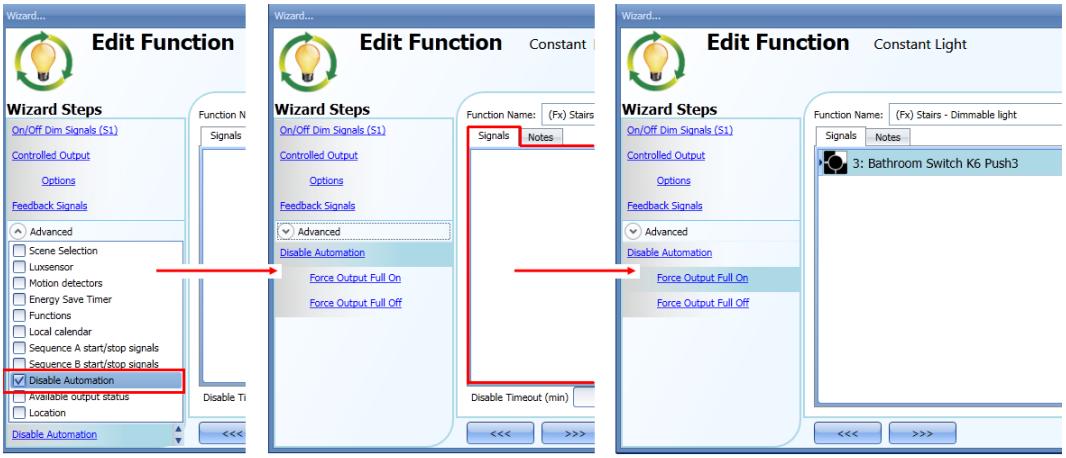
If the user wants to force the output On regardless of all other signals used in the function, the force Output field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output fully on*, then double click on the signals window and select the right signal to use.



Each signal used in the “Force output fully on” window works in level mode. Until the signal is active, the light is forced to the fully on status. When the signal is non-active, the light returns to the previous status. When the *force to fully on* signal is activated and at the same time the *force fully off* signal is activated, the *force to on* status has priority.

10.5.17 How to force the output OFF

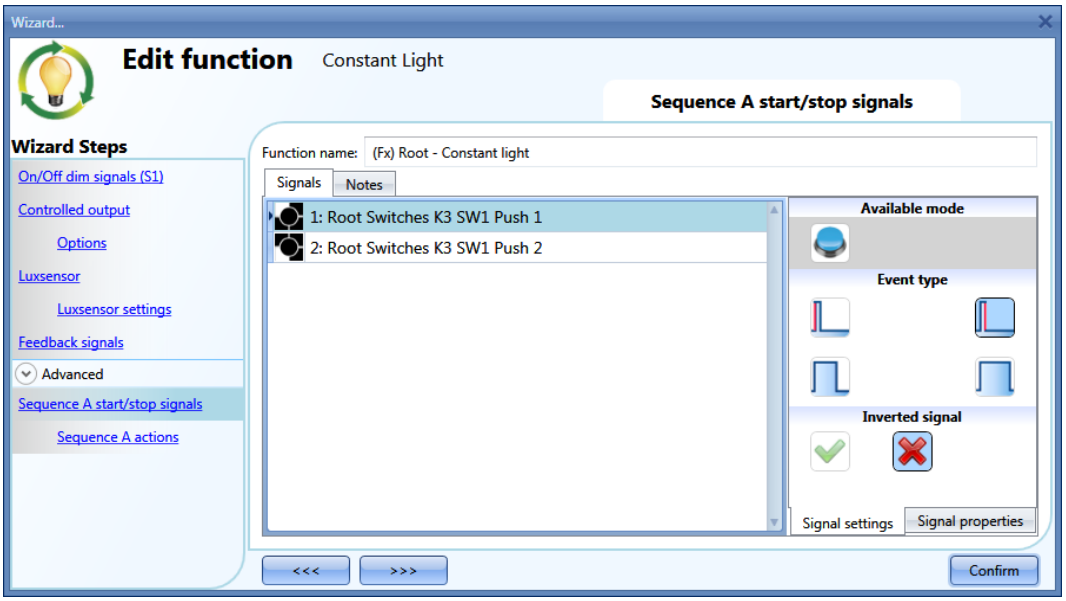
If the user wants to force the output Off regardless of all other signals used in the function, the *Force output fully off* field has to be used: to enable this, select *Disable automation* in the *Advanced* section, select *Force the output fully off*, then double click on the signals window and select the right signal to use.



Each signal used in the *Force output fully* off window works in level mode.
Until the signal is active, the light is forced to the off status; when the signal is non-active, the light returns to the previous status.
When the force to fully on signal is activated and at the same time the force to fully off signal is activated, the *force to on status* has priority.

10.5.18 How to configure Sequence A

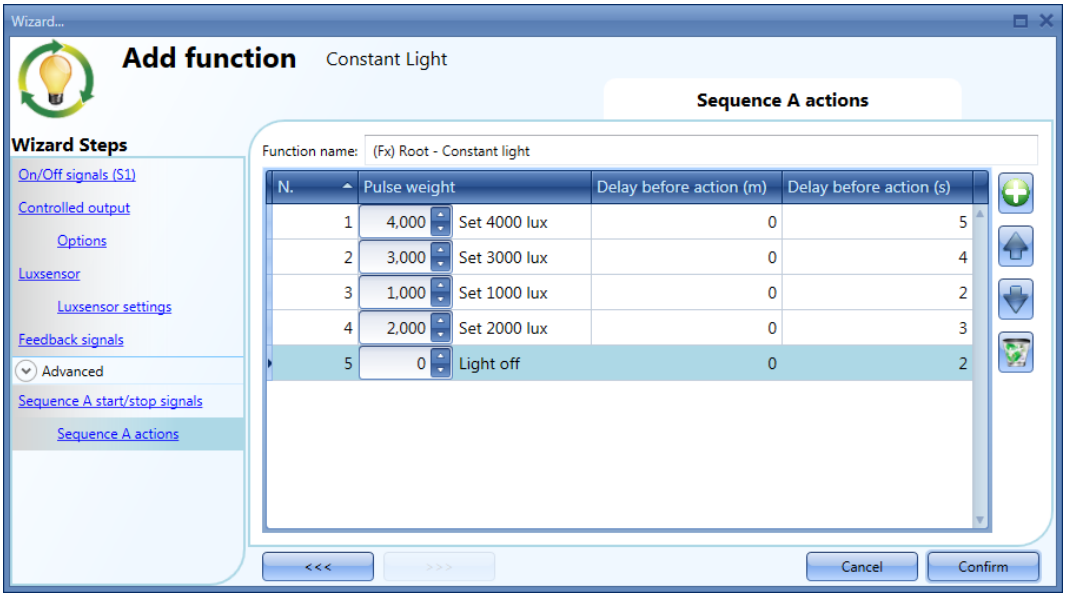
Sequence A automation is a list of commands for changing the light level and it can be used to create dynamic scenarios.
First it has to be enabled in the *Advanced* section.
The sequence is started when an On event occurs: the start/stop signal can be a pushbutton, a function or a remote command.
The user can select from short, long and very long activation to start the sequence.
The sequence is stopped when any dimmer command is generated.
For each signal the inverted logic can be enabled.



10.5.18.1 Sequence A options

In the sequence options window the user can manage each step of the sequence list.
In the window the user can set:

- The order of the scenario in the list.
- The level of the light of each single scenario in the list.
- The pause time of each scenario in the list.



Sequence options icons :

	Add a new Scenario step to the sequence list
	Move the selected Scenario step UP one position in the list
	Move the selected Scenario step DOWN one position in the list
	Delete the selected Scenario step from the sequence list

Delay before action: in this field the user can set the delay between one scenario and the following one.

Scenario (Set lux): in this field the user can select the action according to table A.

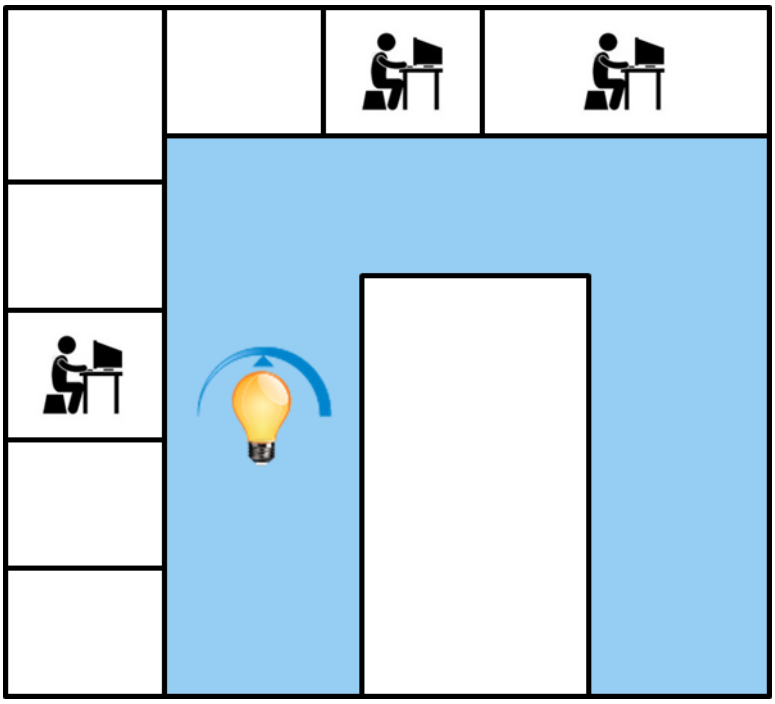
10.5.19 How to configure Sequence B

Sequence B can be configured in the same way as Sequence A.

10.5.20 Corridor lighting function

This function is also an enhancement (additional option) of the dimming functions (*Dimmable light, Constant light, Light ON/OFF*).

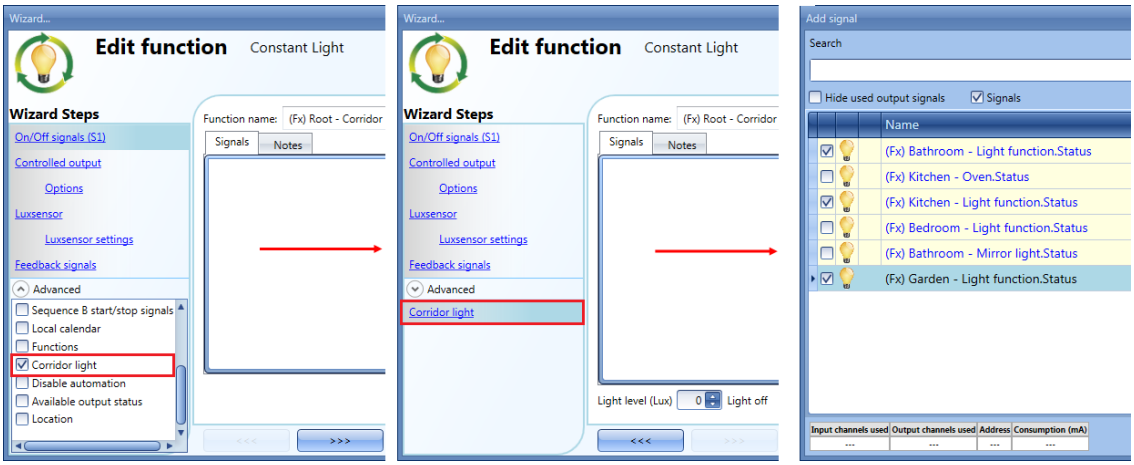
As shown in the picture below, the main aim of the Corridor lighting function is to control the lighting in the corridor, based not only on the presence of people in the corridor, but also on the presence of people in the adjacent offices. If there are people present in one or more offices, the corridor light is not allowed to switch OFF but has to be at a certain level.



10.5.20.1 How to set the Corridor lighting function

To set up a *Corridor lighting* function the user should select one of the existing *Light Functions* present in the project.

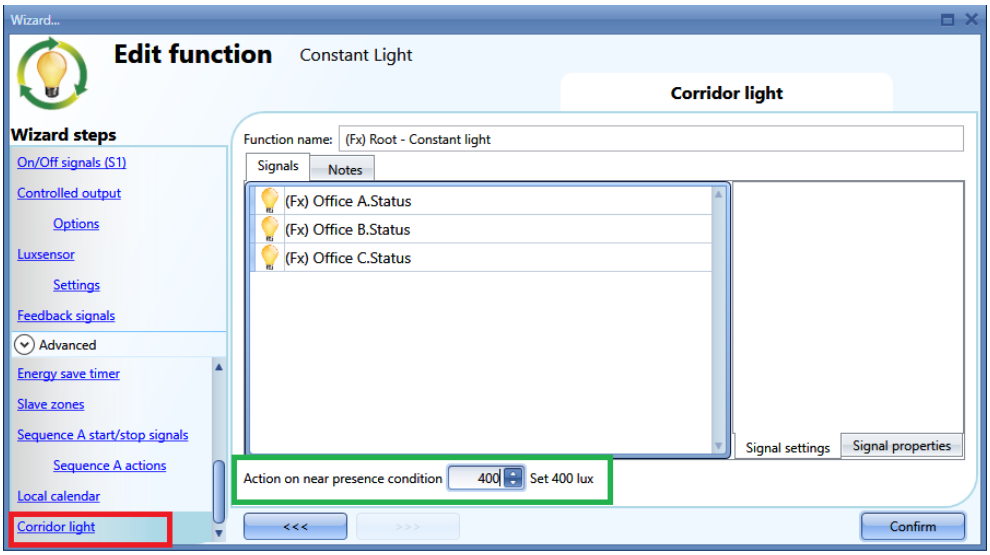
Since this is an Advanced functionality, the *Corridor light* field has to be enabled in the A*dvanced* section.



The Corridor light menu will appear. After selecting this, by double clicking on the Signals window the list of available signals will appear.

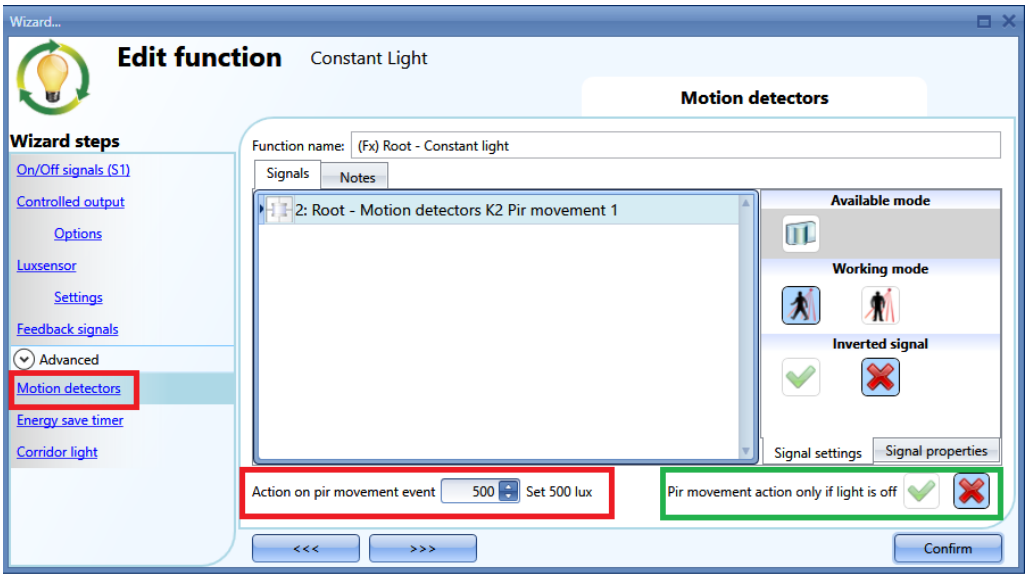
As shown in the picture above, the user may choose which offices he wants to include in the function, by selecting the relative signals: they can be function status or physical signals.
If more than one signal is selected in this window (you can choose up to 50 signals), the function will be activated/deactivated if at least one signal is active (the system performs the logical OR).

The control of lighting in the corridor is based not only on the presence of people in the corridor, but also on the presence of people in the adjacent offices. If there are people present in one or more offices, the corridor light is not allowed to switch OFF: it has to be at a certain light level (e.g. 400 lux). See the green rectangle in the picture below.



10.5.20.2 How to turn a light ON with PIR sensors combined with Corridor light function

If the *Corridor light* option is selected, the user must select at least one signal which they want to include in the function in the Advanced section.
This will be used by the *corridor light* function to detect if there is “people presence” in the adjacent offices. The signals added in this field are used to keep the light ON at the level defined in the field *Action on PIR movement event (Lux)* (green rectangle, picture above, 400 lux), but they are not used to switch the light ON.
However, a PIR signal can be used to switch the light ON in the *Motion detector* field and the user should select the red “X” in the option *PIR movement action only if light is OFF*. In this way, each time the PIR detects movement, the light will automatically be switched ON to a certain light level. In the red rectangle in the example shown below, a light level of 500 lux has been selected.



When the PIR does not detect any movement in the corridor, the light returns to the level set in the Corridor light options.

If the office lighting is not PIR-based, the user should use “*light function output*”, as shown in the previous paragraph.

10.5.20.3 How the Corridor light function works

There are different possible scenarios, depending on the presence conditions in the offices adjacent to the corridor.

10.5.20.4 If people are present in the office next to the corridor

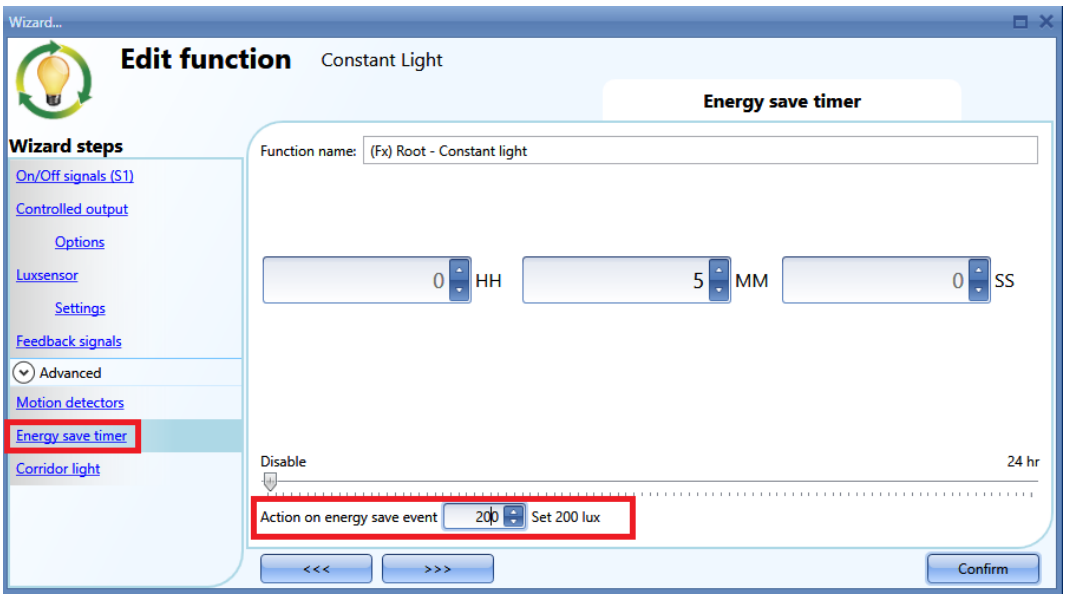
In this situation, one or more of the selected *PIRs* are ON or one or more of the selected *light functions* are ON

As long as “people presence” is TRUE, the corridor light function works as a PIR-based scenario control, where the light switches between, for example, 500 lux and 400 lux light level, depending on whether the PIR is ON or OFF.

10.5.20.5 If people are not present in offices next to the corridor

If an energy-save timer is used to switch the light OFF or to set the light at a lower level, it starts counting down when all the signals in the field *Motion detector* and *Corridor light* are OFF: this means people presence neither in the corridor nor in the adjacent offices.

In the example shown below, while the user is in the corridor the light stays at 500 lux and the energy-save timer is continuously reloaded; when no more people are detected in the corridor and in the adjacent offices, the energy-save timer starts counting and the light is switched to a lower level, e.g. 200 lux. In the example shown below an energy-save timer of 5 minutes is set by the user.

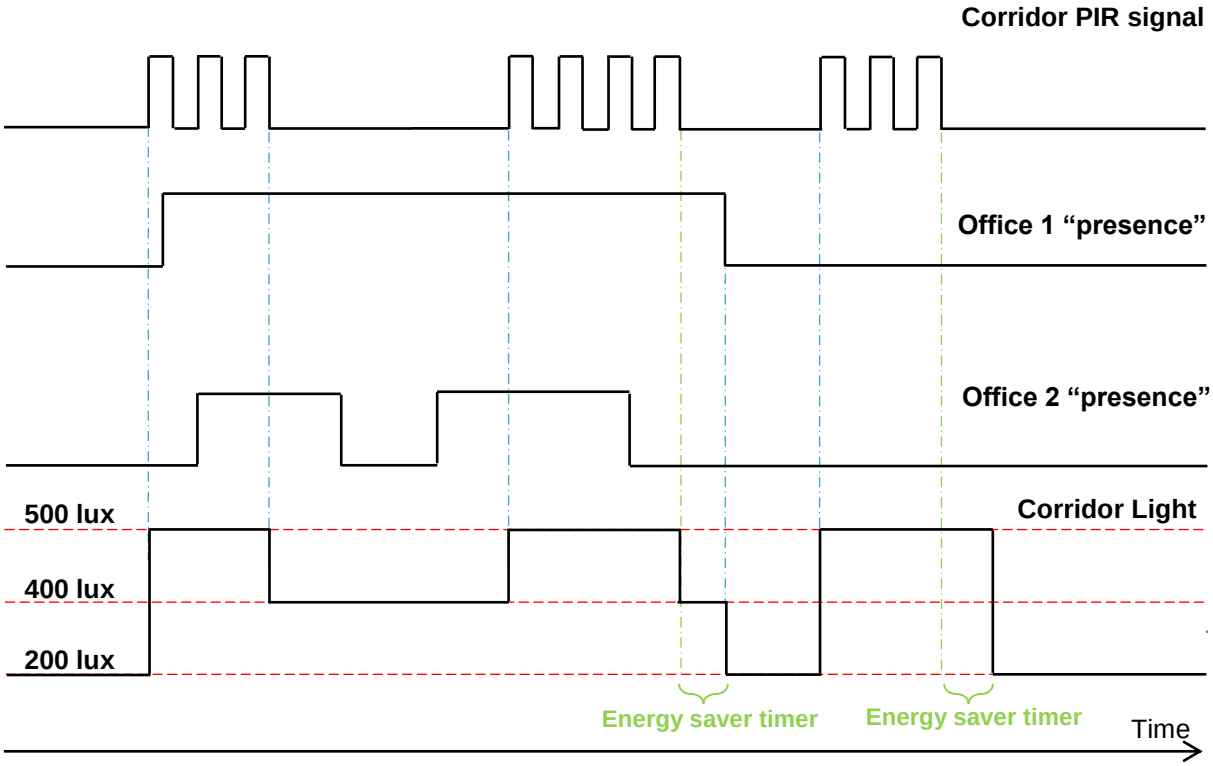


In the example shown below, a PIR is present in the corridor: when it detects any movement, the corridor light switches ON to 500 lux light level and remains at this level until the PIR detects any movement in the corridor.

As long as at least one or more adjacent offices are occupied, and the PIR doesn't detect any movement in the corridor, the corridor is set to 400 lux light level.

When the PIR redetects any movement in the corridor, the corridor light returns ON to 500 lux light level. If the PIR doesn't detect any movement and there is no people presence in the adjacent offices, the corridor light goes to energy-save light level, e.g. 200 lux, as set by the user in the *Energy saver timer* field.

When the PIR in the corridor detects any movement, the corridor light switches ON to 500 lux and the "people present" operation resumes.

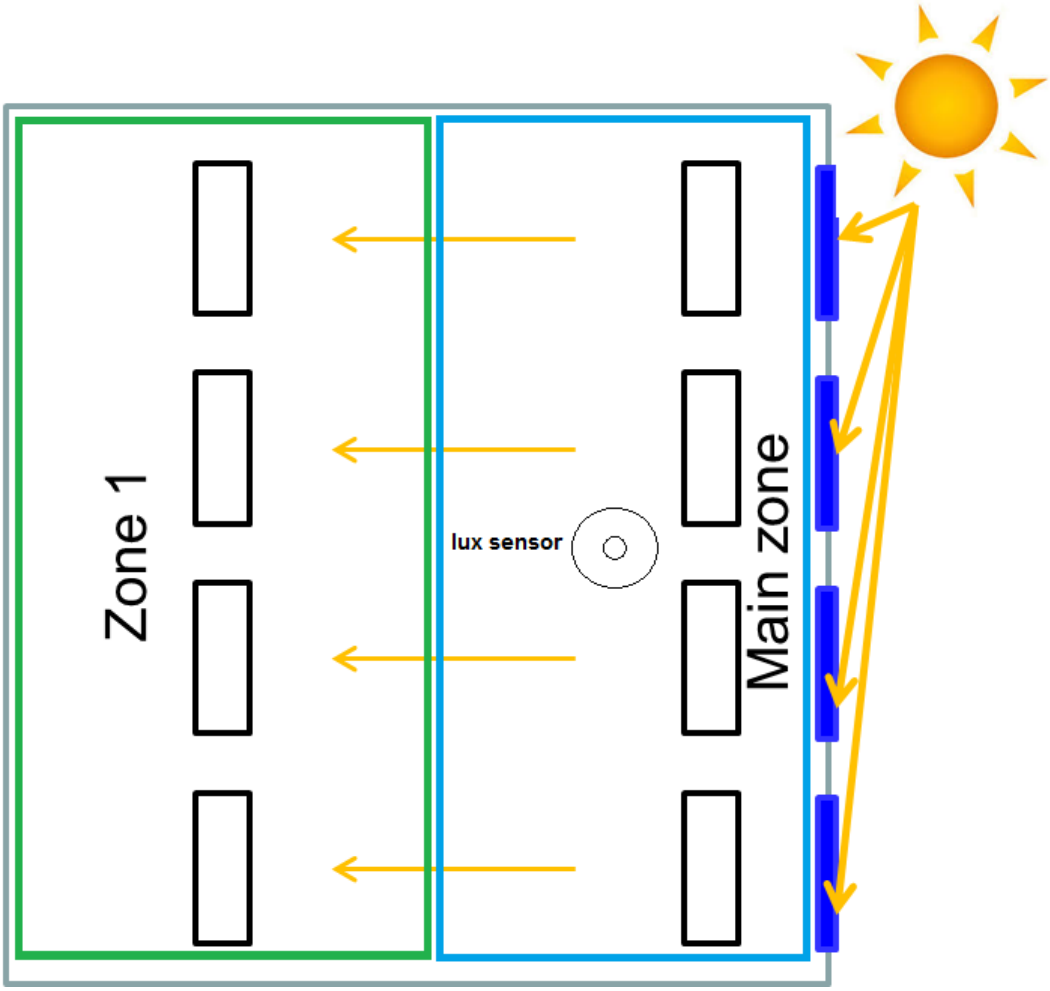


10.5.21 How to configure a Constant light zone control

This feature is typically used in open office environments where the influx of natural light is stronger close to the windows than it is deeper into the office away from the windows. Typically, the office can be divided into two or three lighting zones with different dimming level requirements to maintain the desired lux setpoints. It is not an option to simply create a separate constant light function for each zone, because then the zones would influence each other and there would be the risk of fluctuating lights.

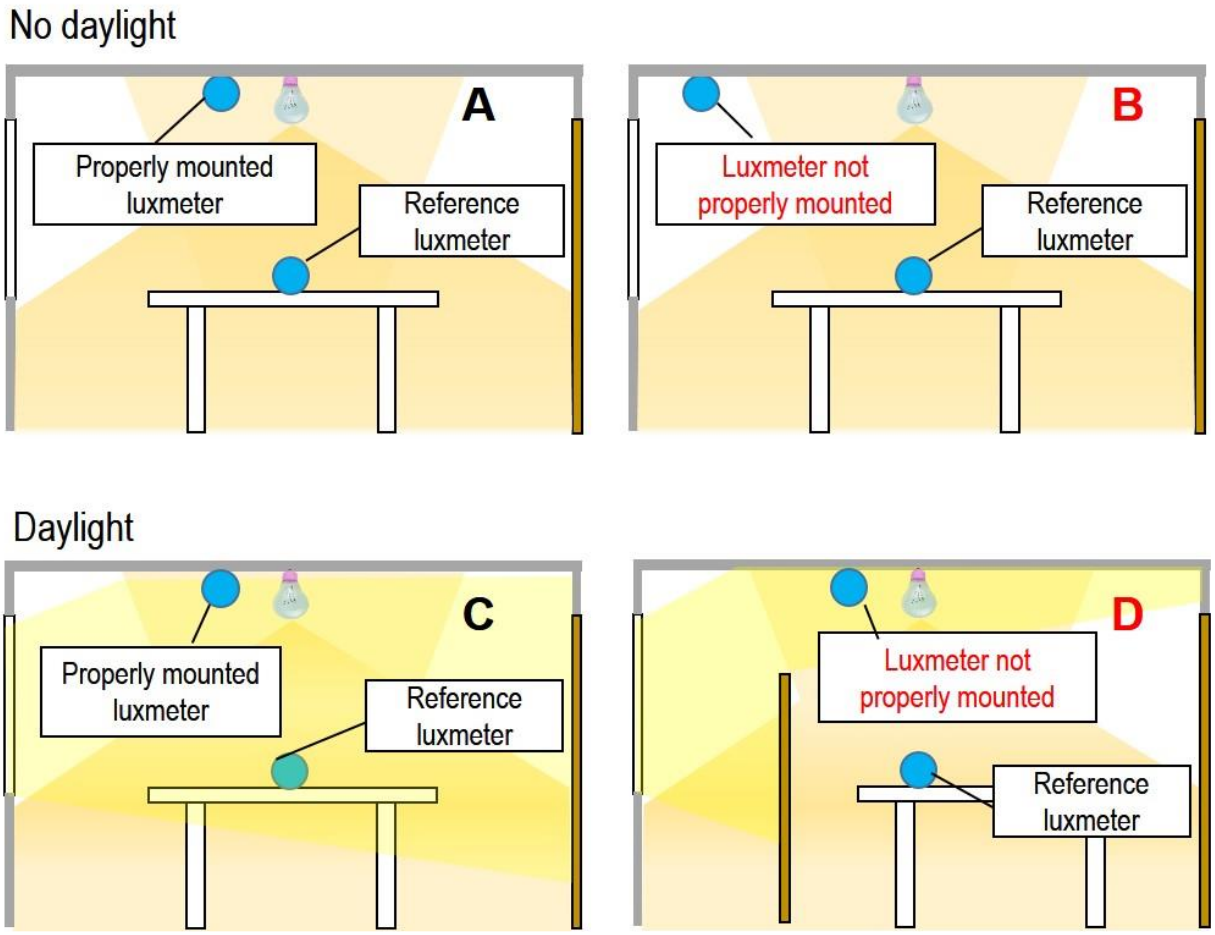
The solution is to create slave zones to the main constant light function. The dimming level of the slave zones can be calculated based on the dimming level of the main constant light function. This approach is also used in order to minimize the number of lux meters installed.

The *Main zone* is the line of lighting fixtures closest to the windows, *Zone 1* is the line furthest away from the windows. The lux sensor (if only one) would typically be located in the *Main zone.* A fundamental step in order to have the algorithm working properly is to install the lux sensor in a place where the lux measurement is not influenced by the artificial lights of the other slave zones.
The master zone can be placed next to the window or internally in the room.



10.5.21.1 How to mount the light sensor

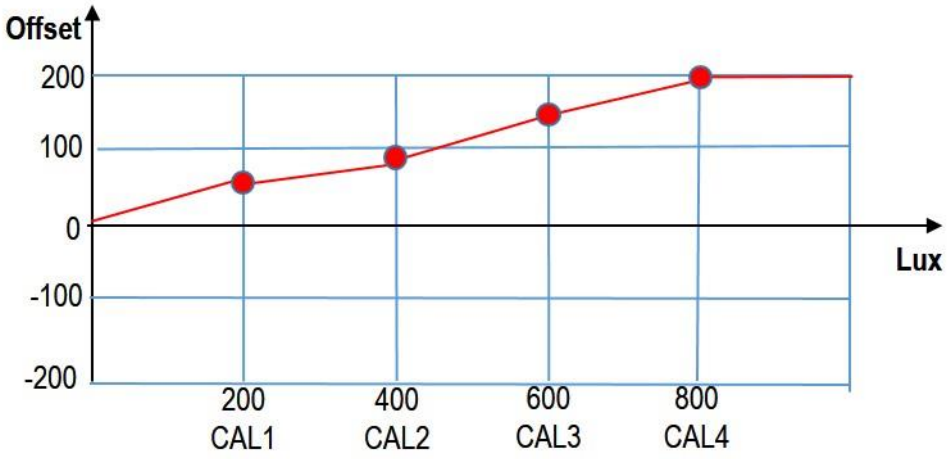
To have the constant light working correctly, it is very important that the lux sensor is mounted in the right place/s.
Below there is some advice about how it should be positioned.



1) The lux meter adjustment function is needed because the lux sensor can't be installed on the illuminated desk and it receives only the reflected light from the ceiling.. Thus an offset has to be calculated
2) The offsets must be calculated at the worst conditions, with no daylight influence
3) The calibration has to be carried out on four points.
4) The unused calibration points should be calibrated at 0 lux
5) Usually the *measured lux* is smaller than the scenario setpoint (positive offset = (scenario setpoint-measured lux)). If the offset is negative, it should be smaller than (*measured lux)* / 2.
   If this is not the case, the sensor might be wrongly positioned
6) The use of a wall-mounted lux sensor can reduce the difference between the measured lux and the reference lux

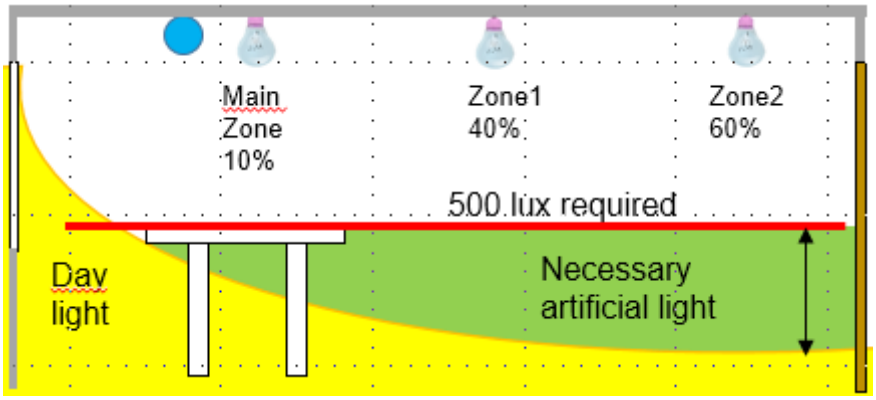
10.5.21.2 Why calibration is needed

The constant light function is needed to maintain constant light on desks in office buildings. For this reason, lux sensors are needed to control the light level, but since they are usually not mounted on the desk but on the ceiling, a calibration procedure is needed: if the sensor is correctly positioned, it receives only the light reflected by the desk, while the desk receives the direct light from the bulb. This means that on the desk, when the light is on, there is much more light than on the ceiling and an offset has to be calculated to make the algorithm work well.
The calibration procedure of the main zone consists of four steps, during which four offsets have to be calculated according to four scenarios.

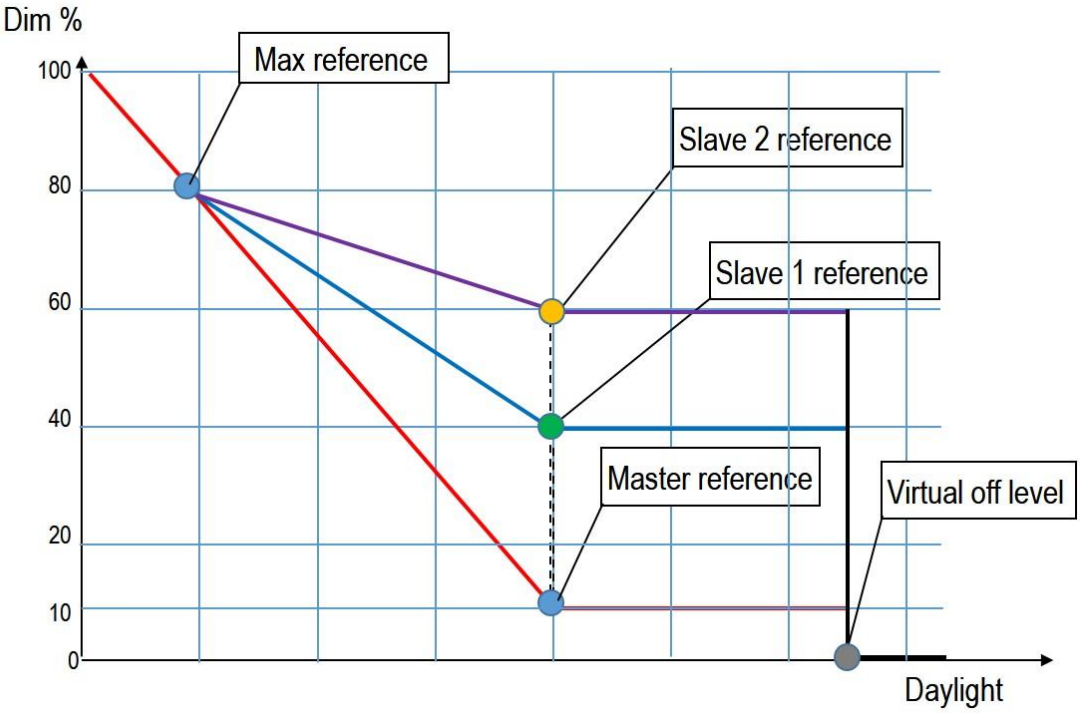


In the example shown above, the calibration has been carried out according to the four scenarios set at 200, 400, 600 and 800 lux, and for each of them the relevant offset of 50, 90, 140 and 200 have been calculated. From these four points a line is built and this is used by the constant light algorithm to calculate the correct setpoint at each light level. For this reason, it is important to calibrate the function at the light levels that will be used in the office.

Once the main zone is calibrated, the slave zones need to be worked out too. This operation is required because, if the main zone is next to the window, the two slaves zones positioned internally in the room will need more artificial light (see picture below).



During the calibration of the slave zones, the light level of the slave zones in terms of dimming percentage will be defined when the main zone is dimming at a predefined level. In the example shown above, when the main zone is dimming at 10%, slave zone 1 will dim at 40% and zone 2 at 60%.



To summarize, the calibration of the master zone is needed to create its dimming curve (red line), while the calibration of the slaves zones is needed to create their dimming curve (blue and purple lines).

10.5.22 Master zone calibration procedure

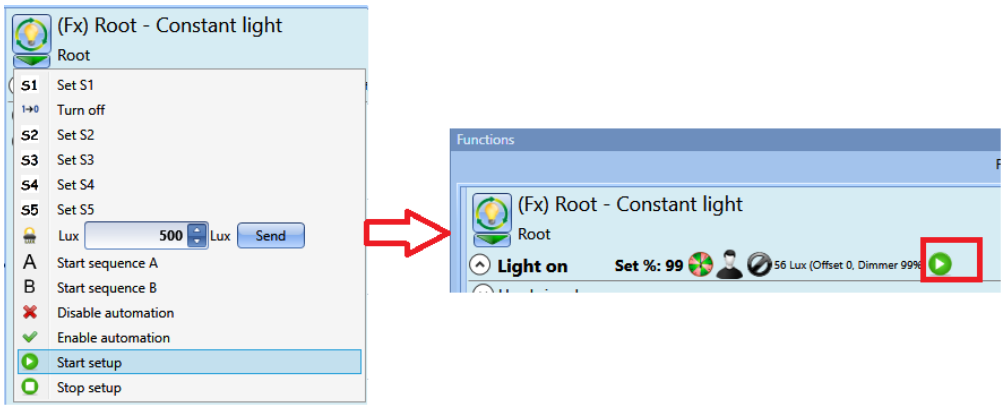
For a correctly executed calibration process, it is important to follow these steps:

1) The room must be in the dark: this means that no natural light should affect this process

2) Place a reference lux meter in the area that has to be calibrated

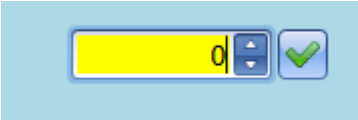
3) Write into the controller the configuration with the new constant light function

4) Enable the *live signals* and enter the set-up mode as shown below:

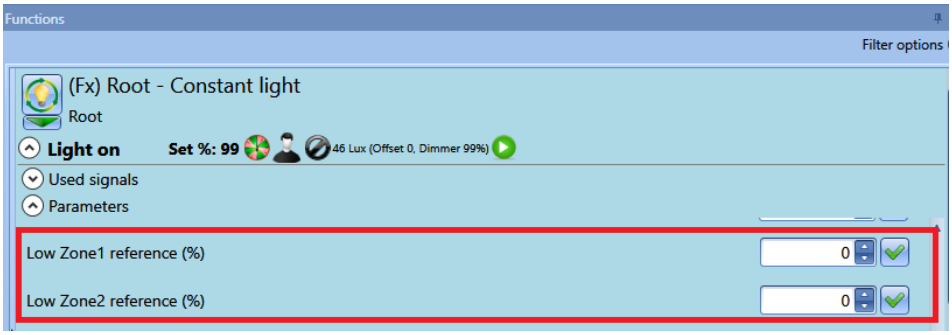


Once the set-up mode is enabled, the green circle with the “play” symbol will appear in the live signal window (red rectangle).
In set-up mode the constant light algorithm is not running, so the user can manually enter the dimming output to reach the four calibration setpoints.

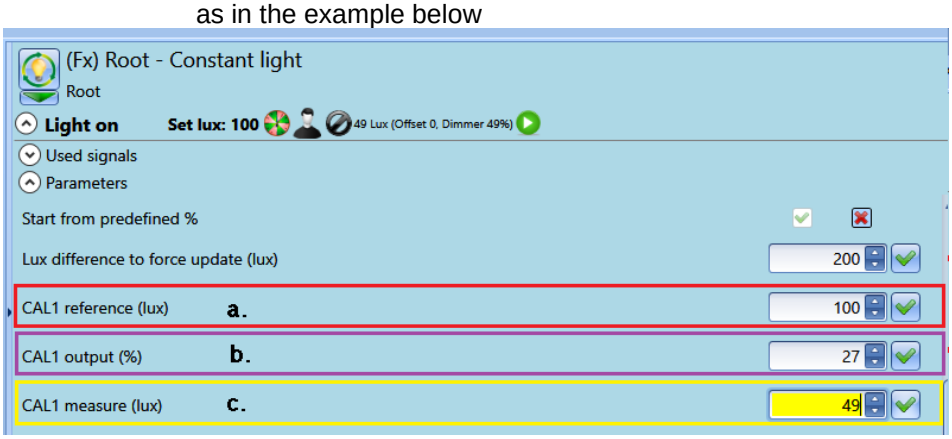
NB: to change a parameter, type in the new value in the relevant box (it will turn yellow) and then click on the green V to send the new parameter to the controller.



5) Set the parameters *Low Zone 1 reference* and *Low Zone 2 reference* to 0*:* this means switching off the lights in the two slave zones so that they do not influence the master zone.

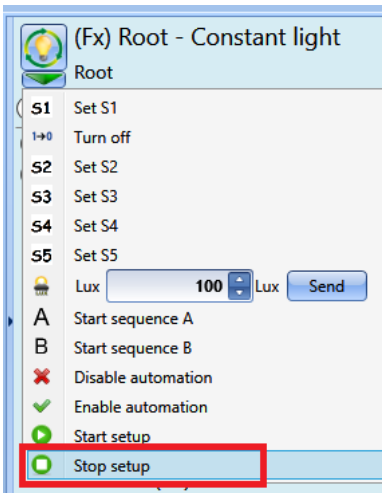


6) Adjust the first calibration point:
   a. Type in the lux level and click on the green V.
   b. Increase/decrease the dimming output until the reference luxmeter has reached the light level in *CAL1 reference*
   c. In *CAL1 measure* type in the value read by luxmeter used in the constant light function, as in the example below



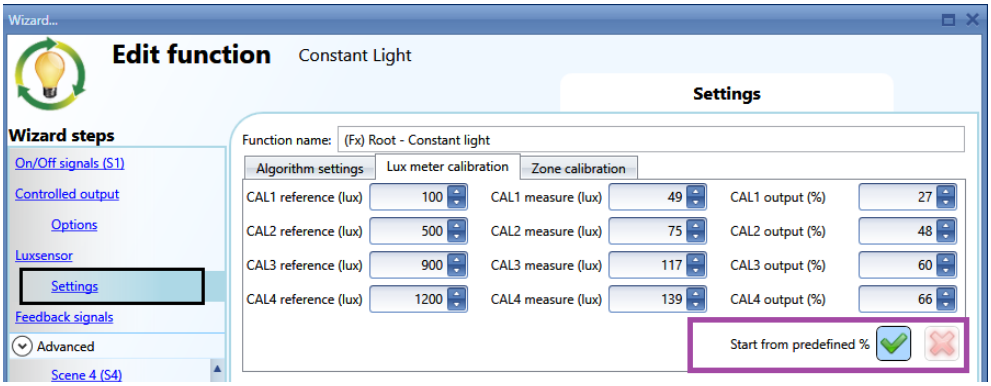
7) Follow the same procedure for the other 3 calibration points

8) Exit from the set-up mode: the system doesn't exit automatically from this mode



If the calibration parameters are already known, the user can set them manually, skipping the calibration procedure: they are accessible in the *Settings* of the luxmeter.

If the field *Start from predefined %* is enabled, once a scenario is changed, the dimming output will be set at CALx output(%) corresponding to the selected light level. If this option is not enabled, the dimming output will change according to the constant light algorithm.



These calibration parameters can also be changed when the set-up mode is not active.

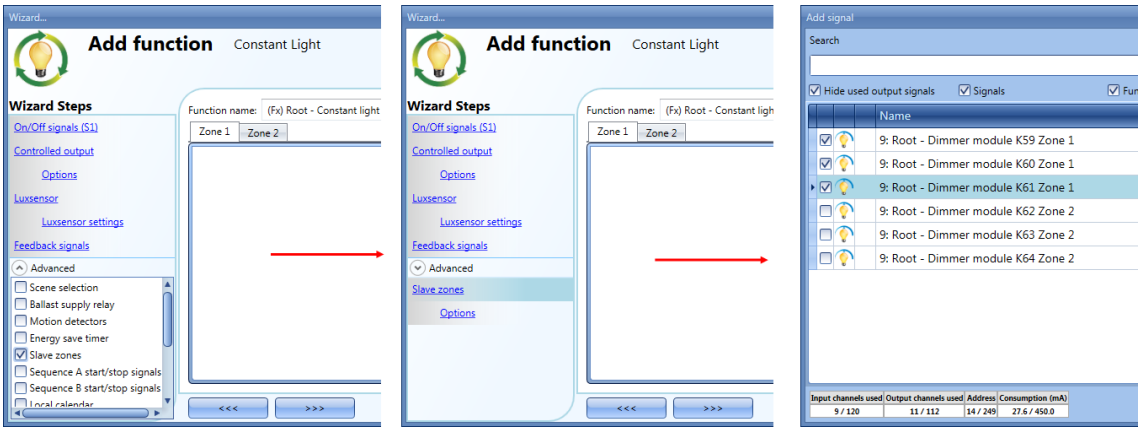
To reset the offset, write 0 in *CALx measure (lux).*

10.5.23 Slave zone calibration procedure

In a constant light function there might be no slave zones, or one or two. If they are present, they have to be calibrated in order to set the right light level in daylight conditions.
The calibration of the slave zones has to be carried out after the calibration of the master zone, following these steps:

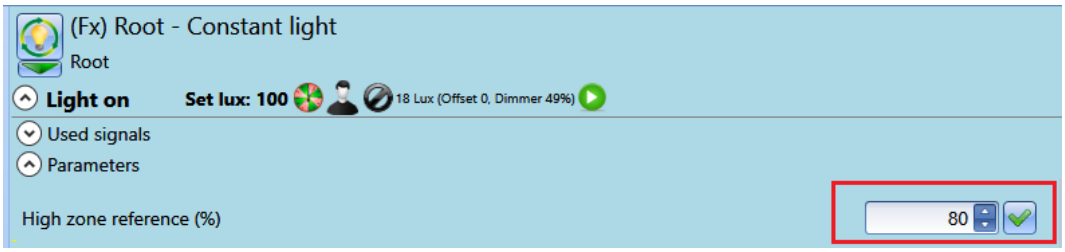
1) The procedure has to be carried out in daylight conditions

2) Add the dimming output of the slave zones in the relevant field



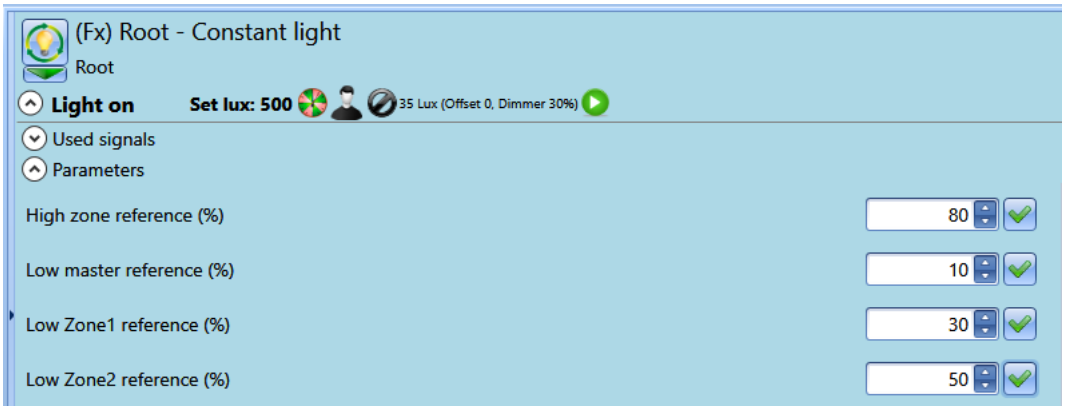
3) Write the configuration into the controller and start the set-up procedure

4) Define the maximum dimming level in the main zone, *High master reference*, above which the slave zones have to follow. This is the point where the daylight doesn't influence the slave zones.

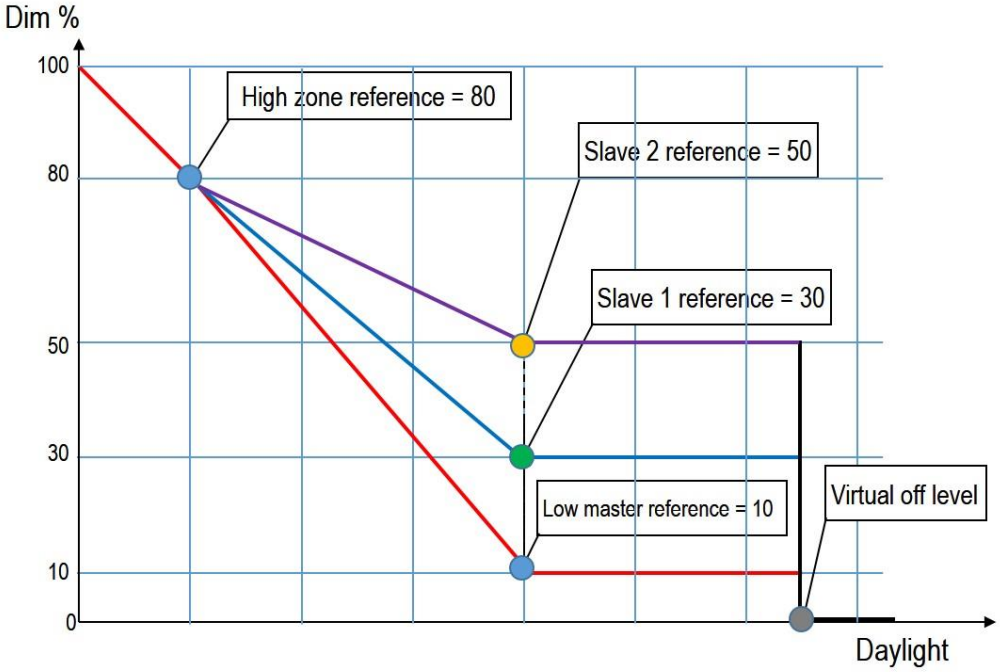


This gives the high point in calculating the dimming curve of the slave zones.

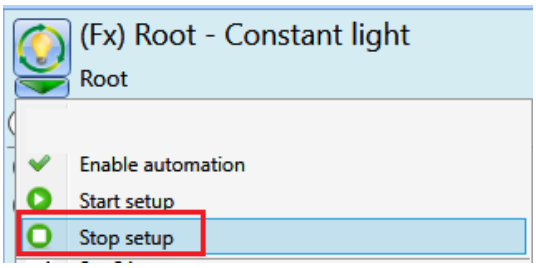
5) Define the reference point in the master zone to be used to calculate the second point in the slave zones.



In the example shown above, when the master zone output is set to 10%, slave zone 1 will be set at 30%, while zone 2 will be at 50%.
According to this example, the dimming curves will be as follows:



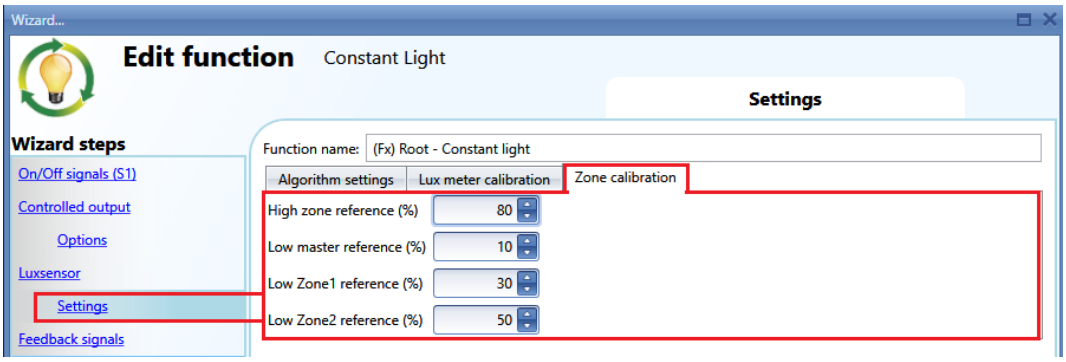
6) Exit from the set-up mode



Some notes about the calibration of the slave zones

a) The slave zone fixtures should not influence the luxmeter in the master zone and all the controlled lamps in a function should be of the same type
b) If the curves of the slave zones are higher than the curve of the master zone (this means that the master zone is the one next to the window), they will never reach the minimum dimming value
c) All the zones will go off when the virtual off threshold is reached (black line in the figure above)
d) The virtual off threshold should be set at least 30% higher than the light level reached with the output set at 100%, to avoid on/off switching during the automatic control.

Should the calibration parameters be known already, the user can set them manually, skipping the calibration procedure: they are accessible in the *Settings* of the luxmeter.

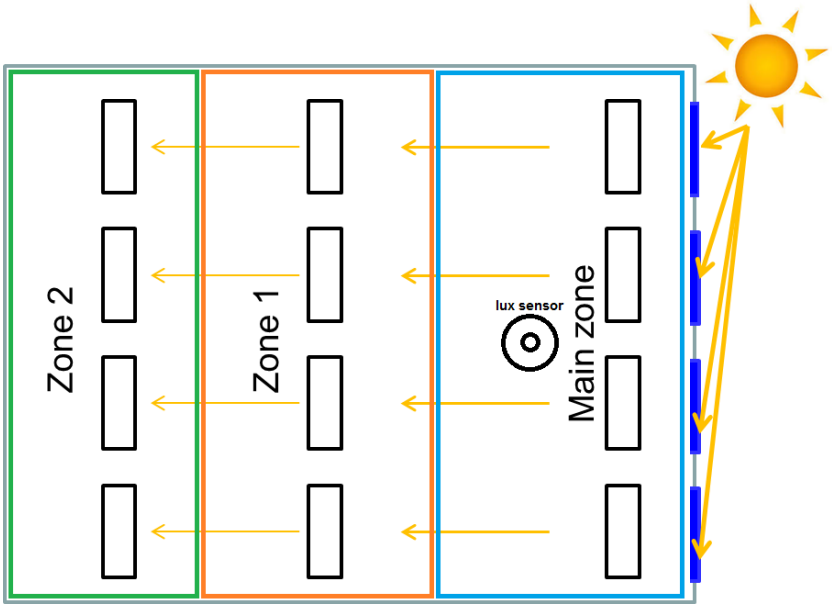


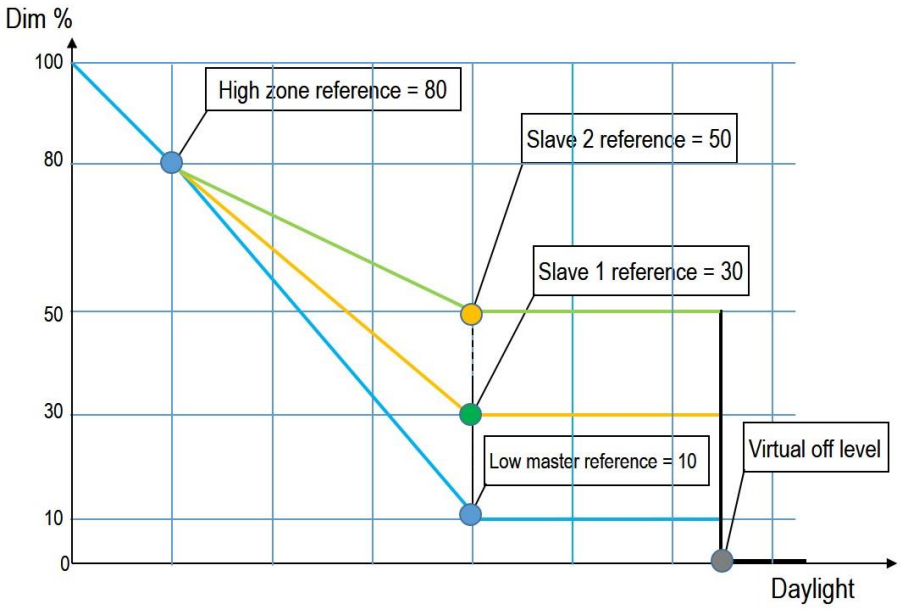
***High zone reference (%)* must always be higher than the other parameters (by at least 10 points (%)).**
In manual mode zone outputs will be the same at the same level of the master zone outputs.

10.5.24 Position of the slave zones in relation to the master zone

The master zone can be positioned either next to the window or inside the room, according to the needs of the user.
In the examples below, three different set-ups with the relevant dimming curve diagrams are shown.

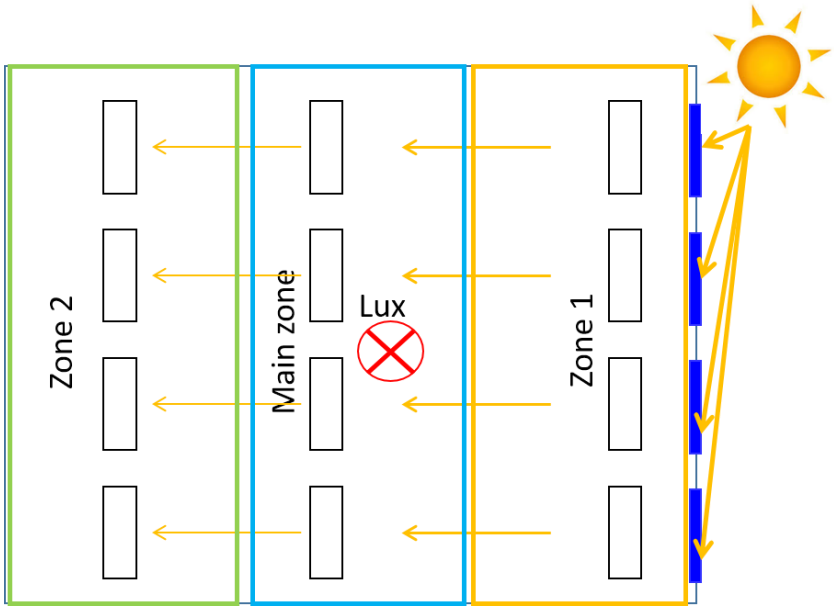
a) Master zone next to the window

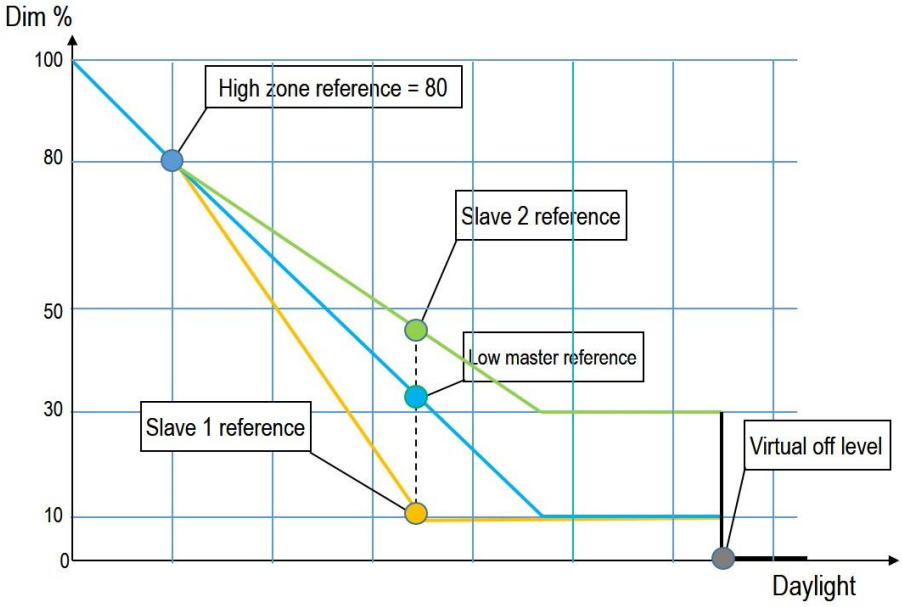




In this case the slave zones will never reach the minimum dimming level.

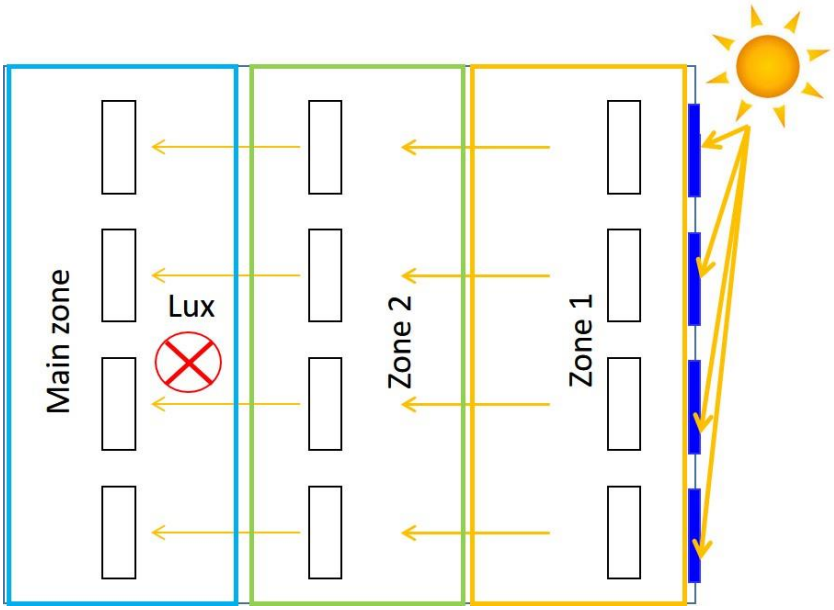
b) Master zone between the two slaves zones

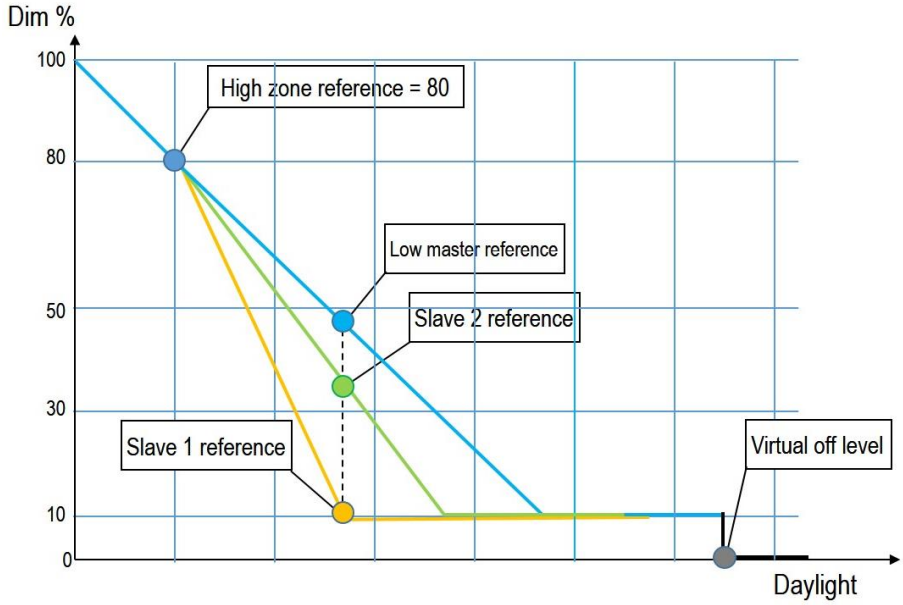




If a slave zone is nearer to the window than the master zone, its dimming curve will be below the master one and it will reach the minimum dimming value.

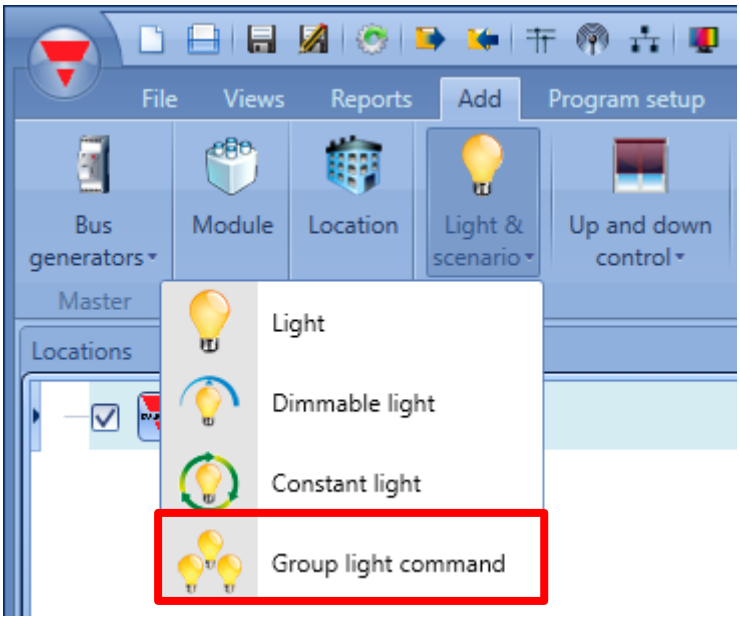
c) The master zone can also be placed in the internal area, furthest from the window



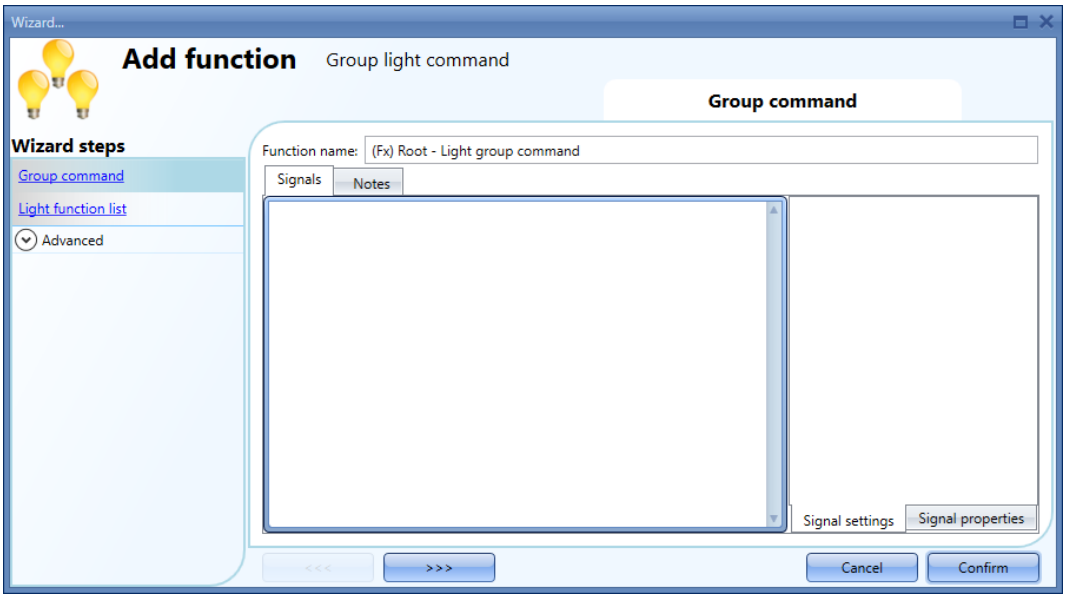


10.6 Group light command function

To set up a *Group light command* function the user has to select *Light & scenario* from the Add menu, then select *Group light command* (see figure below). The new function will be added into the selected location.
The wizard relevant to a *Group light command* can also be opened by typing Alt+A+I+G. (see table of short cuts).



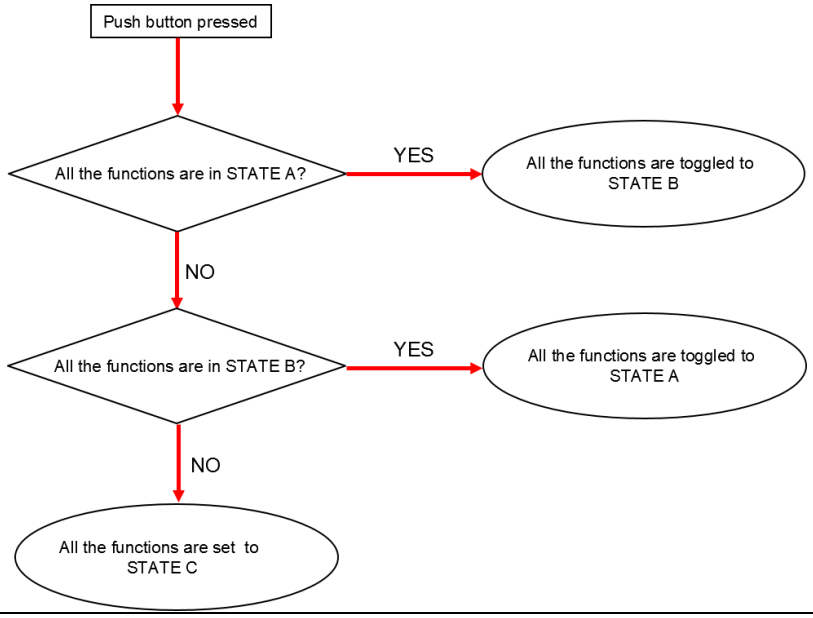
This function manages the ON/OFF switching of one or more light functions by means of one or more Group commands. The Group command might be a real signal, a function or a remote command (Webserver, Modbus TCP/IP).



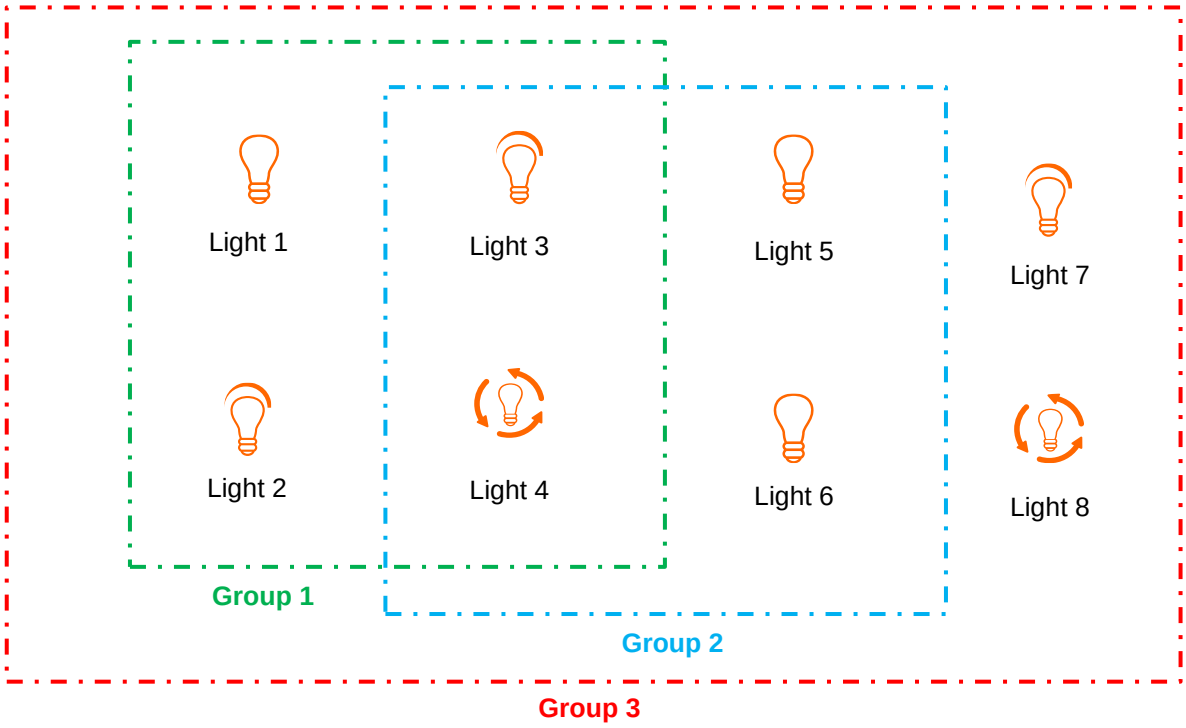
10.6.1 How to set a Group light command function

The *Group light command* function is used to change a group of light functions, whichever type they are, such as Light On/Off, Dimmable light and Constant light, between three statuses. For each light function belonging to the *Group light command* function, the user has to define the values the light function will be set to for *State A*, *State B* and *State C*.

The table below shows the description of each state according to the light function status:

State	State description
State A (All the functions are OFF)	If all the lights are in **State B** (i.e. all the light functions are OFF) when the Group command signal is triggered, all the light functions belonging to the Group command function will be turned ON (i.e. the value set for **State A**)
State B (All the functions are ON)	If all the lights are in **State A** (i.e. all the light functions are ON) when the Group command signal is triggered, all the light functions belonging to the Group command function will be turned OFF (i.e. the value set for **State B**)
State C (Neither **State A** nor **State B** is satisfied)	If neither **State A** Nor **State B** is satisfied, **State C** will indicate the final status. All the light functions belonging to the Group command function will be set when the Group command signal is toggled, according to the diagram shown below: 

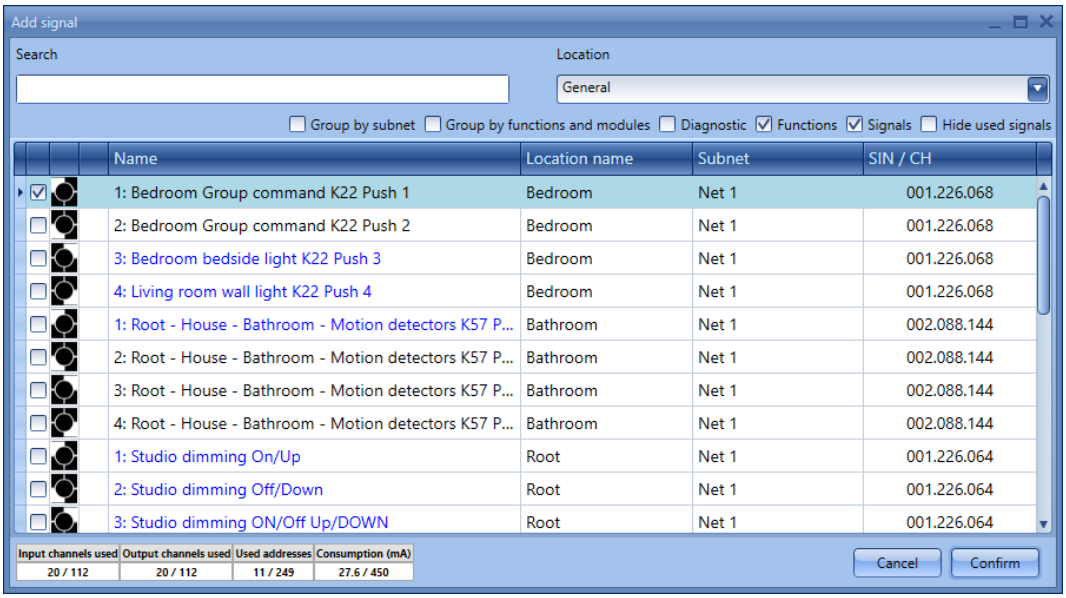
Any light function available in the current project can belong to several *Group light command* functions: as shown in the example below, there are three different *Group light command* functions: Light 3 and Light 4 are part of **Group1** and **Group2.** All the light functions belong to **Group3**.



10.6.2 How to switch a Group of light command functions with input commands

First, the user has to add the Group command signals to control the light functions present in the *Light function list* section.

By double clicking in the *Signals* area, a window will appear with all the available signals.



Select the signal/signals you want to use as *Group command*. If more than one signal is selected in this window (you can choose up to 20 signals), the function will be activated/deactivated if at least one signal is active (the system performs the logical OR).
The different colours have the following meanings:

- black signal → not used in any function
- blue signal → the signal is already used in another function
- blue signal + warning signal (yellow triangle)→ the signal is already used in this function

The available input signals are classified as:

- Push button
- Switches
- Function

10.6.2.1 Push button:

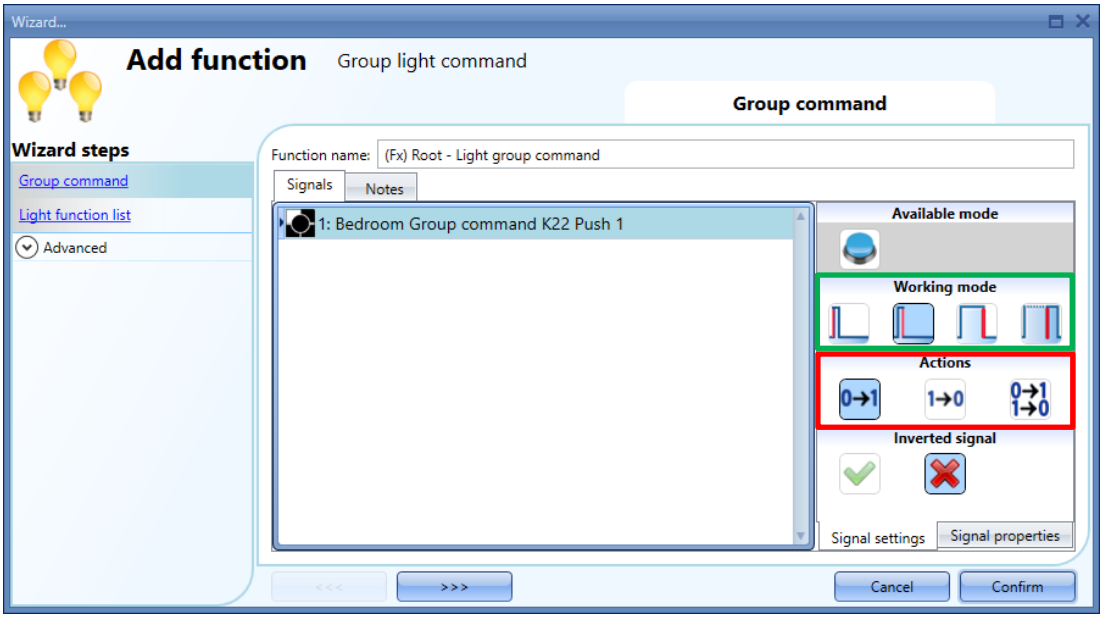
This is a signal active for a short time following a human action (for example, a push button pressed or kept pressed). Four different actions (*Working mode)* can be chosen, as shown in the figure below, green rectangle:

- When the button is pressed (*action on rising*);
- When the button is released (*action on falling*);
- When the push button is kept pressed from 1 to 5 seconds (*long pressure*);
- When the push button is kept pressed for a time longer than the long pressure, from 0.5 to 15 seconds (*very long pressure*)

The long and very long activation times can be set directly from this wizard, in the *Signal properties* tab window, without needing to re-open the wizard of the module.

If one of the four working modes is selected, three different types of *Actions* are available to perform it, according to the duration of the activation of the push button: (See figure below, red rectangle)

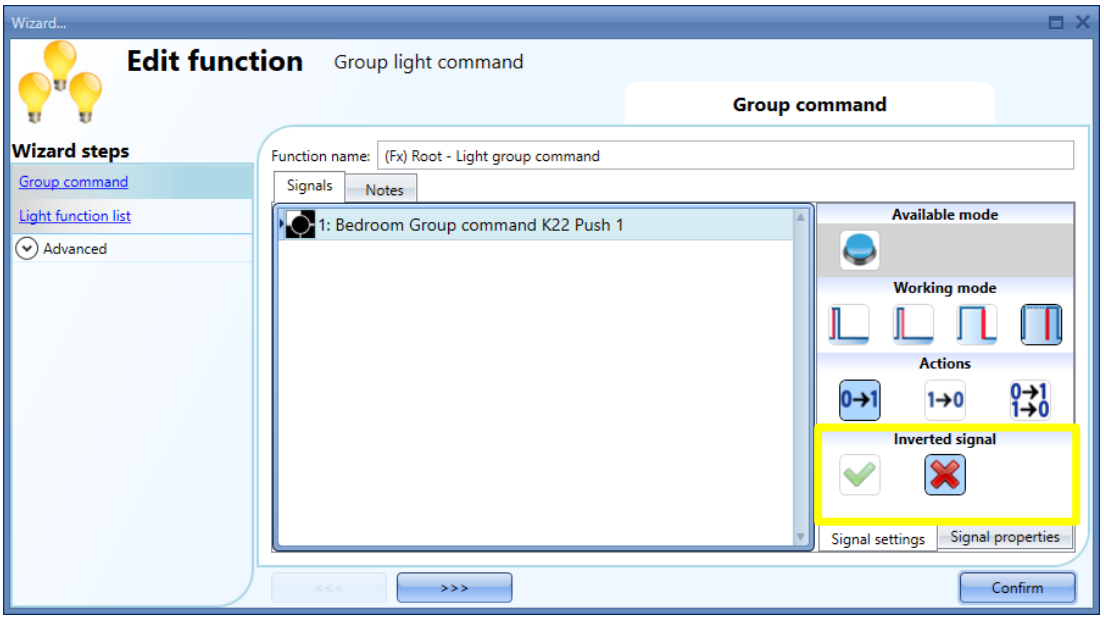
- Go to State A 0→1
- Go to State B 1→0
- Toggle A-B-C 0→1 1→0 (toggle among the three statuses according to the diagram)



Here are some examples:

Working mode	Event type			
0→1	As soon as the push button is pressed, the function will be activated.	As soon as the push button is released, the function will be deactivated.	After a long press, the function will be activated at the release of the push button.	After a very long press, the function will be activated at the release of the push button.
1→0	As soon as the push button is pressed, the function will be deactivated.	As soon as the push button is released, the function will be deactivated.	After a long press, the function will be deactivated at the release of the push button.	After a very long press, the function will be deactivated at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the function will be toggled.	As soon as the push button is released, the function will be toggled.	After a long press, the function will be toggled at the release of the push button.	After a very long press, the function will be toggled at the release of the push button.

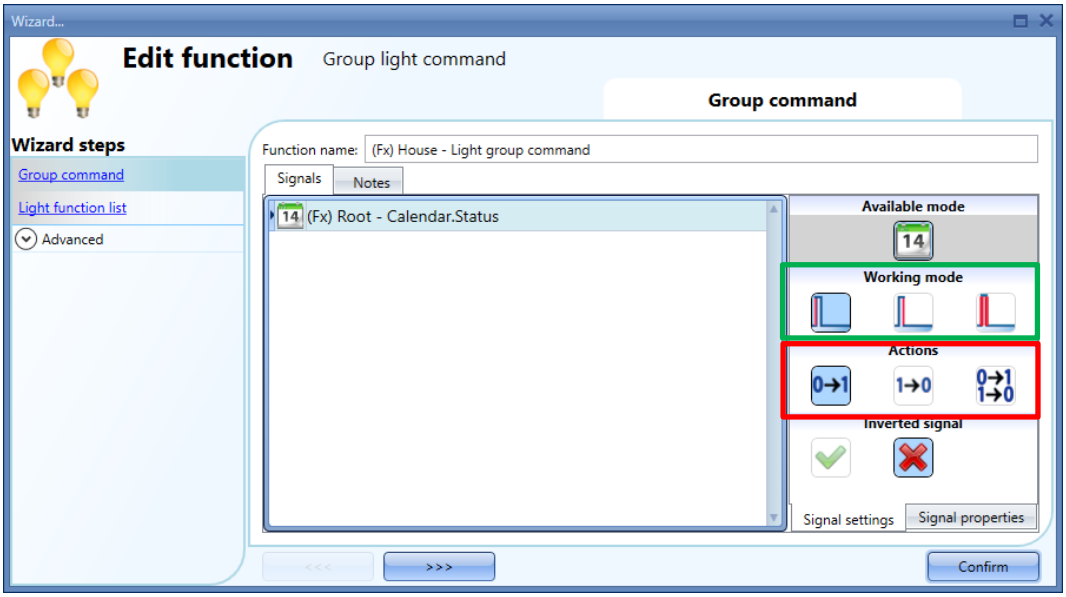
In the *Signals setting* window, the user can also enable the reversing of the signal. See the yellow rectangle in the figure below.



10.6.2.2 Switch and Function:

If a level signal is selected, such as a switch or a function (e.g. Global Calendar set as "*a calendar for level signal*"), the available working modes (see the green rectangle in the picture below) are rising edge (i.e. when the switch is closed or the function is activated), falling edge (i.e. when the switch is open or the function is deactivated), or both (i.e. each change of level status). For each of them, the user can set the Action conditions: there are three different types, as shown in the red rectangle in the picture below:

- Go to State A 0→1
- Go to State B 1→0
- Toggle A-B-C 0→1 1→0 (toggle among the three statuses according to the diagram)

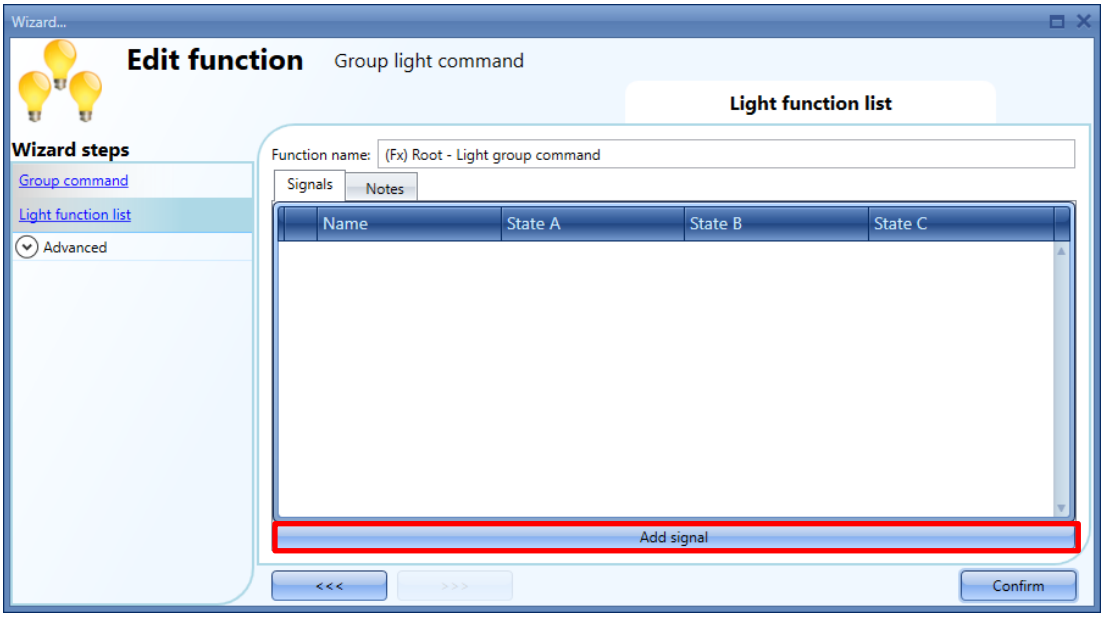


The signals of this type perform the selected action when they change level, according to the table below:

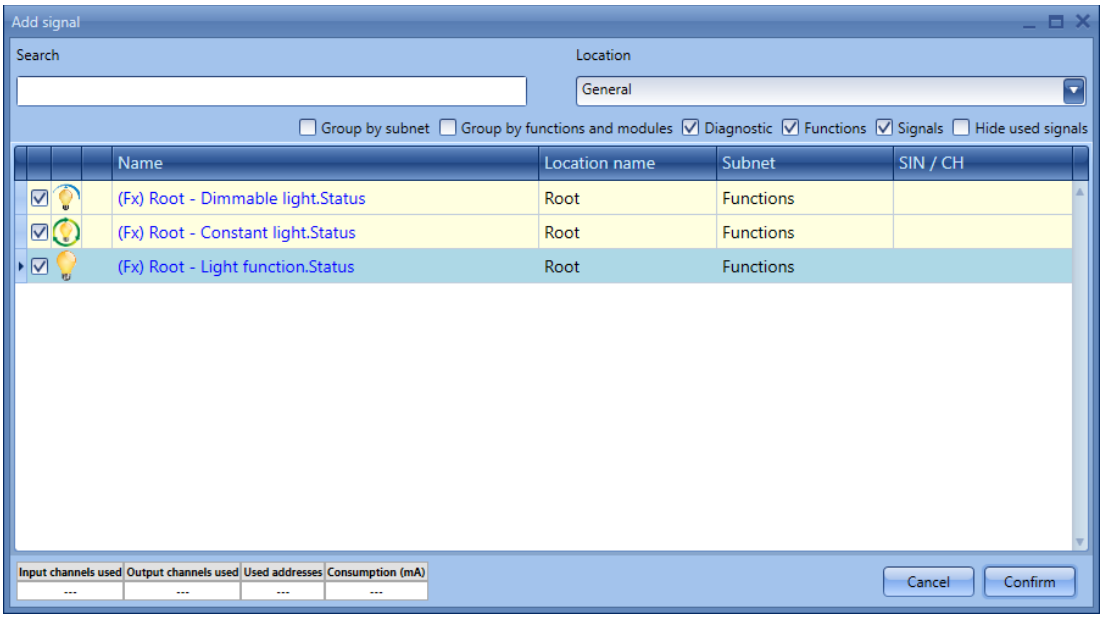
	Event type		
Working mode			
0→1	As soon as the switch is closed /function goes ON, the *Group function* will be activated.	As soon as the switch is opened /function goes OFF, the *Group function* will be activated.	Every time the switch/function is toggle, the Group function will be activated
1→0	As soon as the switch is closed /function goes ON, the *Group function* will be deactivated.	As soon as the switch is opened /function goes OFF, the *Group function* will be deactivated.	Every time the switch/function is toggle, the Group function will be deactivated
0→1 1→0	As soon as the switch is closed /function goes ON, the *Group function* will be toggled.	As soon as the switch is opened /function goes OFF the *Group function* will be toggled.	Every time the switch/function is toggle, the Group function will be toggled.

10.6.3 How to add the Light functions

To select the light functions that will react to the *Group command* event, click on *Light function list* and then click on the *Add signal* button:

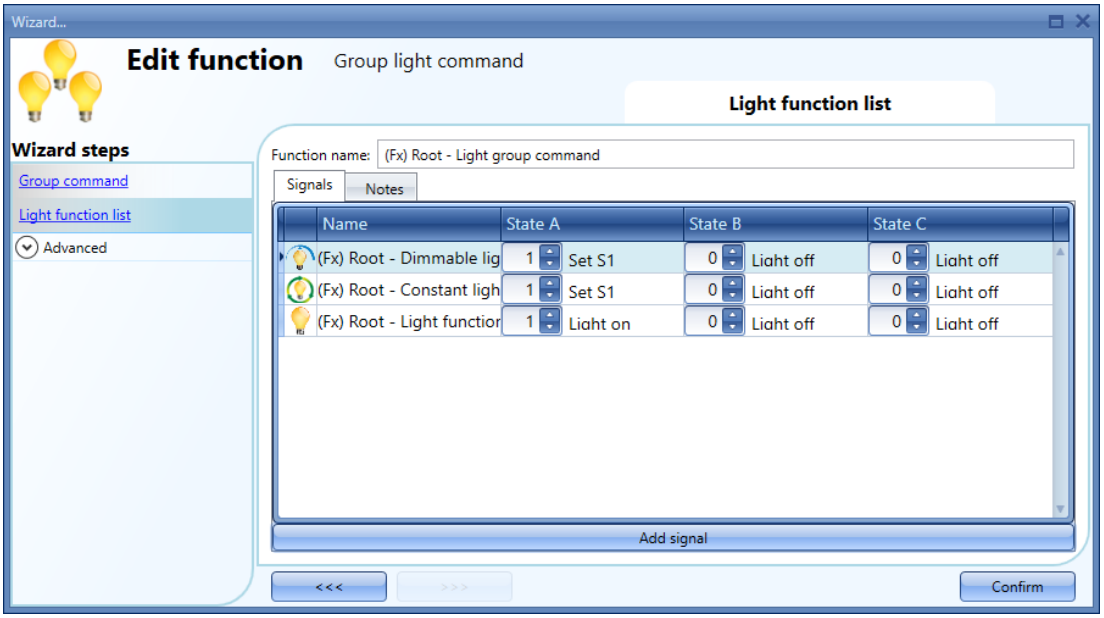


The user may choose which functions they want to include in the list by selecting them from the *Signals* window, and then pressing *Confirm* to add them to the function.



The functions available in this window are Light On/Off, Dimmable light and Constant light functions. Up to 100 functions can be chosen and they are managed in parallel.

As shown in the picture below, in the *Light function list* window, the user has to define the value the light function has to have when the signal is activated for each state, A, B and C:



The available values for the *Dimmable light* function are shown in the table below:

Value	Description of the value	Note
0	OFF	Light OFF
1	S1	Light set @ scenario S1
2	S2	Light set @ scenario S2
3	S3	Light set @ scenario S3
4	S4	Light set @ scenario S4
5	S5	Light set @ scenario S5
6..9	No action	
10..100	Manual @ setpoint (%)	The dimmable light function control is switched to Manual mode and the dimmer output is set to the selected value (%).

The available values for the *Constant light* function are shown in the table below:

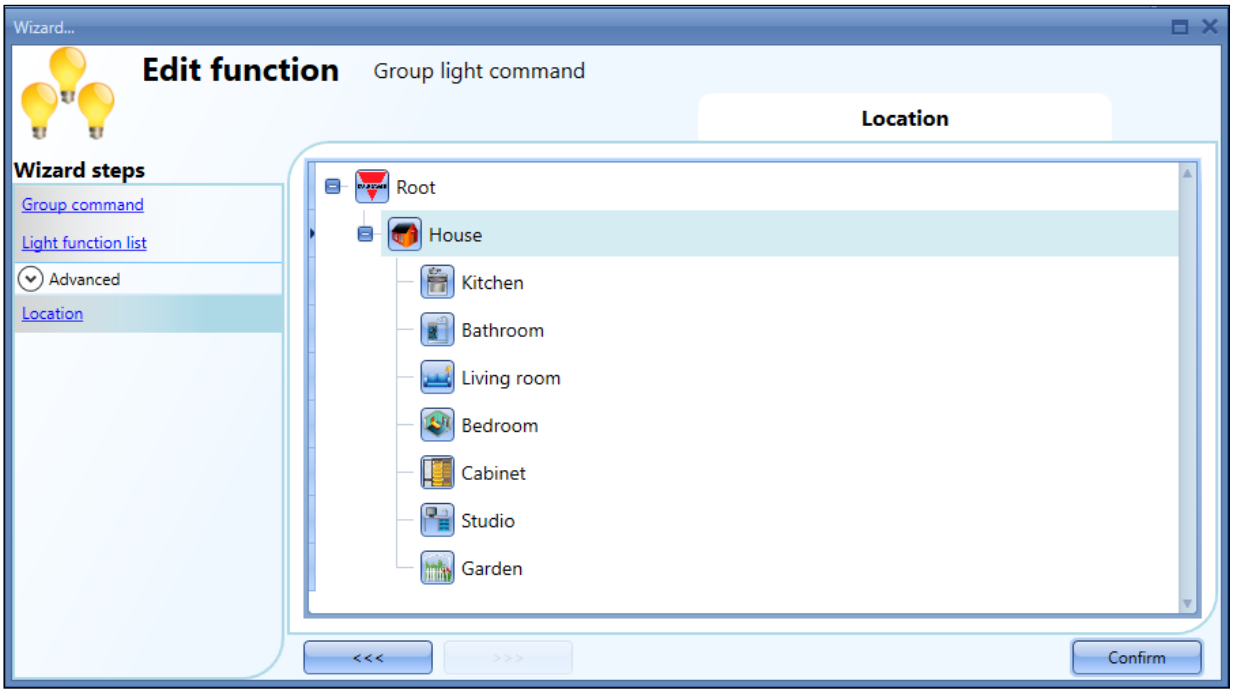
Value	Description of the value	Note
0	OFF	Light OFF
1	S1	Light set @ scenario S1: the last ON can be *Manual* (value in the range 10 - 99) or *Auto* (value in the range 100 - 30000)
2	S2	Light set @ scenario S2
3	S3	Light set @ scenario S3
4	S4	Light set @ scenario S4
5	S5	Light set @ scenario S5
6	Manual 100%	The dimmer output is set at 99%. The timeout to go back to automatic mode is activated
7	Automatic / Manual (set % / Lux)	The light function control is toggled Auto / Manual. When it is switched to Auto the last lux level is applied; when it is switched to manual the current dimmer output is maintained. The command is executed only when the function status is ON. The timeout to go back to automatic mode is **not** activated
8	Automatic / Manual (set % / Lux) + timeout	The light function control is toggled Auto / Manual. When the function is switched to Auto the last Lux level set is applied; when the function is switched to manual the current dimmer output is maintained. The command is executed only when the function status is ON. The timeout to go back to automatic mode is activated
9	Manual mode + timeout @ present % +10%	The light function control is switched to Manual and the dimmer is increased by 10% each time up to 99% and then decreased by 10% each time down to 10%. If this command is executed when the light is OFF, the first output value will be 10%. The timeout to go back to automatic mode is activated
10 - 99	Manual @ setpoint (%)	The light function control is switched to Manual mode and the dimmer output is set to the selected value (%). The timeout to go back to automatic mode is activated
100 - 30000	Automatic @ setpoint (lux)	The light function control is switched to Automatic mode and the algorithm is regulated at the selected lux value

Note1: The group light command function does not manage any delay between the light functions that belong to the group: all the light functions will be triggered at the same time and the functions cannot be sorted by a priority list.

Note2: If the light functions managed by the Group command functions have a valid energy save timer, it will be triggered by the On command.

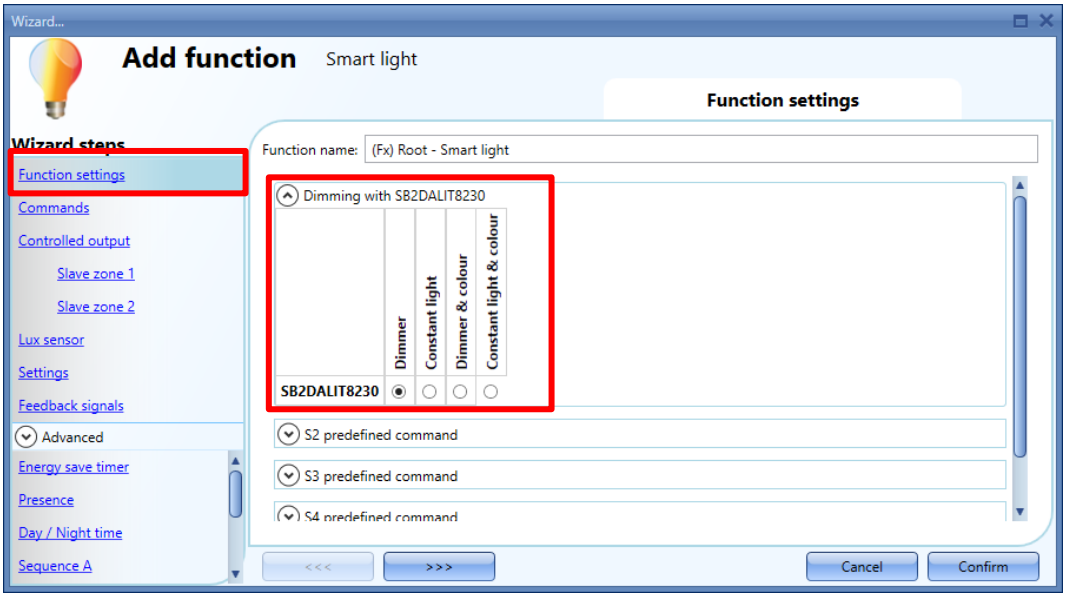
10.6.4 How to change the location of a function

To access the *Location* menu, the user should enable it in the advanced functionalities.
From this window the location where the function is placed can be changed by simply selecting and clicking on that required.



10.7 How to set a Smart light function

To set a *Smart light* function the user has to select *Smart light* function from the *Add* menu (see picture below). The following window appears:



In *the Function settings* tab of the window, the user has to select the type of lighting control he needs to set. The available control types are the following ones:

- **Dimmer**

The DALI output (suitable for any DALI ballast type) is managed as *Dimmable light: t*he light can be dimmed manually or automatically.

- **Constant light**

The DALI output (suitable for any DALI ballast type) is managed as a *Constant light*: the light is changed automatically by the system according to a Luxmeter threshold (101 lux to 5000 Lux) set by the user. The colour temperature is not managed.

- **Dimmer & Colour and Constant light & colour**

The DALI output (suitable <u>ONLY</u> for DALI ballasts type8) is managed as a standard Dimmable light /Constant light with the additional control of the temperature colour. The light intensity is managed according to the standard Dimmable/Constant light control, whilst, the **tuneable white** control can be set manually by the user or can be dynamically changed creating a relationship between day time and Table colour (*Colour table*).

These are the main new features and enhancements of the *Smart light* function:

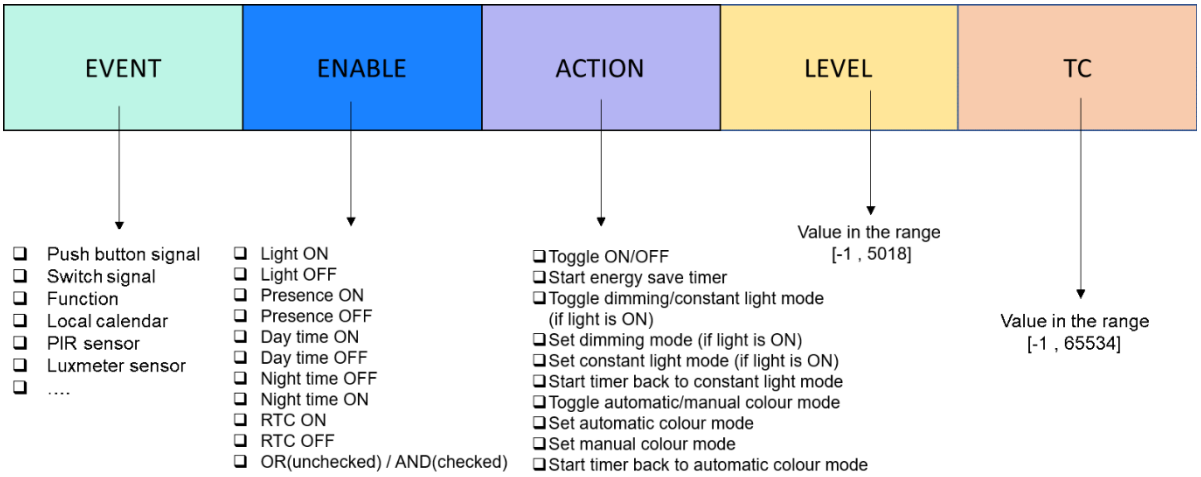
- New command format and new functionalities completely customizable
- Colour table for automatic temperature colour change
- New signal *Room presence condition*
- New comparators for *Day* and *Night* condition
- Energy save timer has two independent actions
- External functions command can be managed in two different ways (digital and analogue)
- Local calendar (RTC) with editable actions and midifiable via Modbus and BACnet

10.7.1 The command structure

The Smart light function has a new command format and new functionalities completely customizable:

The command stucture consists of blocks, which can be defined according to the automation type the user needs.

The diagram below shows the command structure:



The Smart light function, whatever type of control has been chosen (Dimmable light, Constant light), is an **Event controlled** function: as soon as an **Event** occurs, such as a *Push button click* or an *RTC ON* signal, the DALI ballast output changes its status (the **Level intensity** + the **Colour temperature** value), according to the settings.
The action is executed **only if** the Enable condition is verified.

Note: **CT** means **Colour temperature**

According to the command structure shown above, the following tables list the values the user can set for each block of the command.

Table 1 – Available values fot **Event** command

Action type	Description
Event selection	Push button signals: - Click event - Very long press event - Short press event Switch signals Level functions RTC events Presence (PIR) events Luxmeter thresholds

Table 2 – Available value for **Enable** field

Action type	Description
Enable field	The enable lists the automation status that can be selected in order to accept the commands. Standard commands (the ones not generated by automation status) applies the enable in "event mode", only when the command is triggered. Automation commands (the ones generated by automation status) applies the enable in "logic mode". The command is generated every time the logic state rises. • Light ON • Light OFF • Room presence ON • Room presence OFF • Day time ON • Day time OFF • Night time ON • Night time OFF • RTC ON • RTC OFF • OR (unchecked) / AND (checked)

Note:

- **OR (unchecked) / AND (checked)**

Two or more Enable conditions can be logically combine according to the selection on the last checkbox: the logical OR is performed between the Enable condition when it is unchecked, when it is checked the logical AND is performed.

*Table 3 – Available values for **Action** field*

Action type	Description
Action field	The action lists special commands that can be selected, such as: • Toggle ON/OFF • Start energy save timer • Toggle Dimmer / Constant mode • Set Dimmer mode • Set Constant mode • Start timer to return in constant light mode • Toggle between Auto / Manual TC mode • Set Auto TC mode • Set Manual TC mode • Start timer to return in Auto TC mode

Table 4 – Available values for **Level** field

Action type	Description
Level field (-1 to 5018)	-1 = null (the actual level is not changed) 0 = switch OFF 1= switches to last ON level 2-3-4-5 loads predefined Scenarios 6 = switches to 100%, 7 = switch to 10%, 8 = switches to 50%, 9 = switches to 75% 10..100 = switches to 10-100% 101..5000 activate Lux control *Special commands:* 5001 = Start A sequence 5002 = Stop A sequence 5003 = Start / Stop A sequence 5004 = Start B sequence 5005 = Stop B sequence 5006 = Start / Stop B sequence 5007 = Activate disable automation 5008 = Deactivate disable automation 5009 = Activate / deactivate disable automation 5010 = Activate ovveride 100% 5011 = Deactivate ovveride 100% 5012 = Activate / deactivate ovveride 100% 5013 = Activate ovveride OFF 5014 = Deactivate ovveride OFF 5015 = Activate / deactivate ovveride OFF 5016 = Activate calibration mode 5017 = Dectivate calibration mode 5018 = Activate / deactivate calibration mode

Table 5 – Available values for **Colour temperature** field

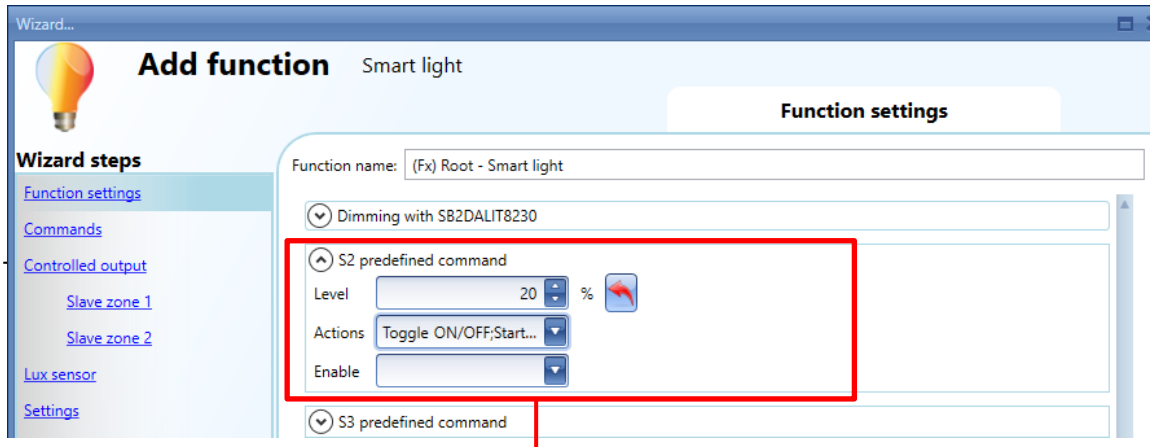
Action type	Description
Colour temperature field (-1 to 65534)	-1 = null (the actual CT is not changed) 0 = it activates auto colour mode. If light is on, every 60sec it applies new CT value calculated using the CT table. 1..65534: applies new CT value and switches to manual CT mode. If the light is off, it loads the value set in the TC field (it doesn't turns the light ON). A timeout to return in auto colour mode can be managed.

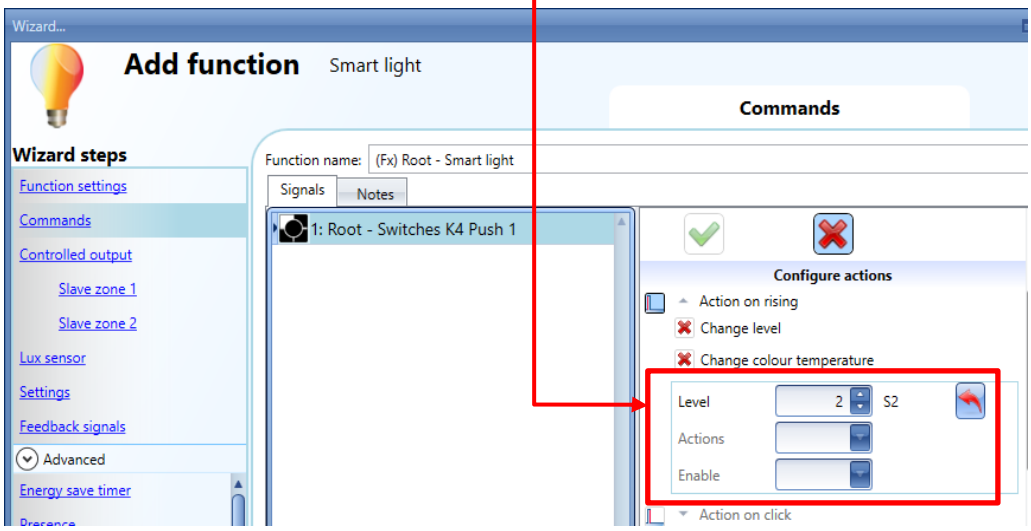
Note: The Temperature colour control is managed ONLY by the Dimmer& colour and Constant light & colour. ***The Colour temperature field is not shown in the Smart light function sets as Dimmer or Constant light.***

10.7.2 Configure predefined scenarios S2, S3, S4 and S5

The scenario S1 is a special scenario that is used to turn a light ON to the last valid output (for example 10% for Dimmable light and 500 lux for a Constant light function)

There are 4 predefined scenarios S2, S3, S4 and S5 that can be configured by the user in the *Function settings* tab of the Smart light function.





10.8 Use cases

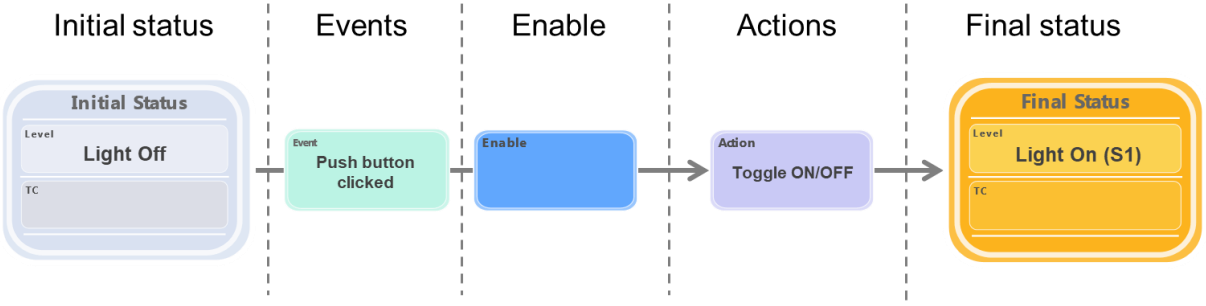
In the next pages a collection of the most used cases is listed together with the relevant configuration for the Smart light function. For each example is shown a flown diagram, followed by the settings the user has to do.

Note: The following use cases are suitable for all the types of control the Smart light is capable of.

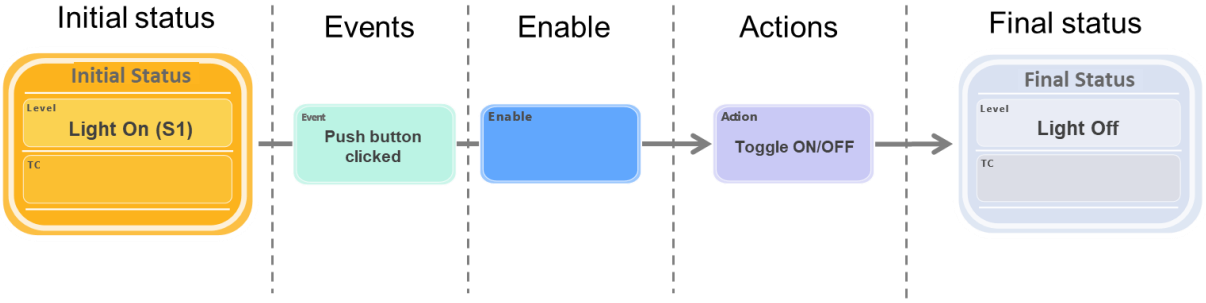
10.8.1 Toggle the light ON/OFF with a push button

Diagram overview

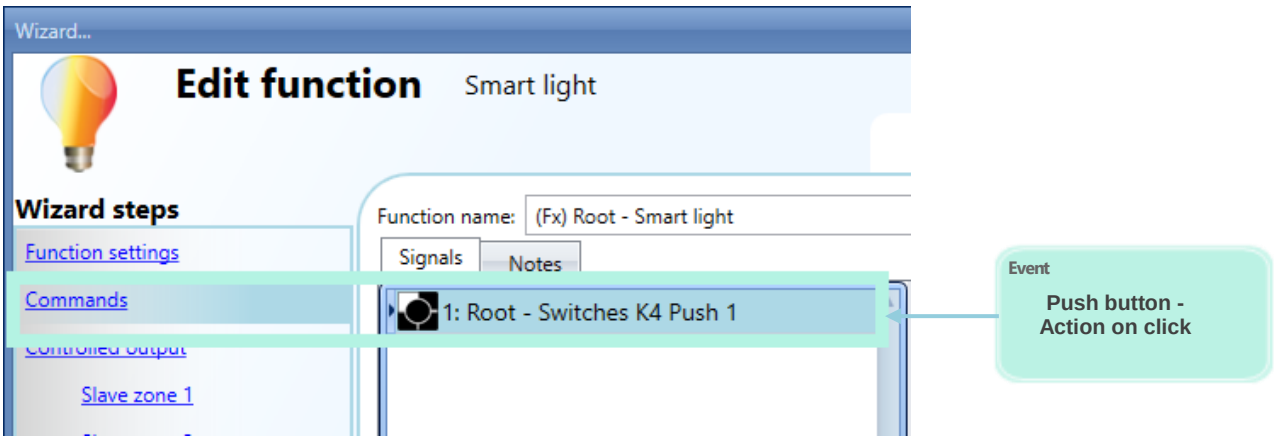
The **light** turns ON to scenario S1 whenever the ***Push-button*** is clicked.



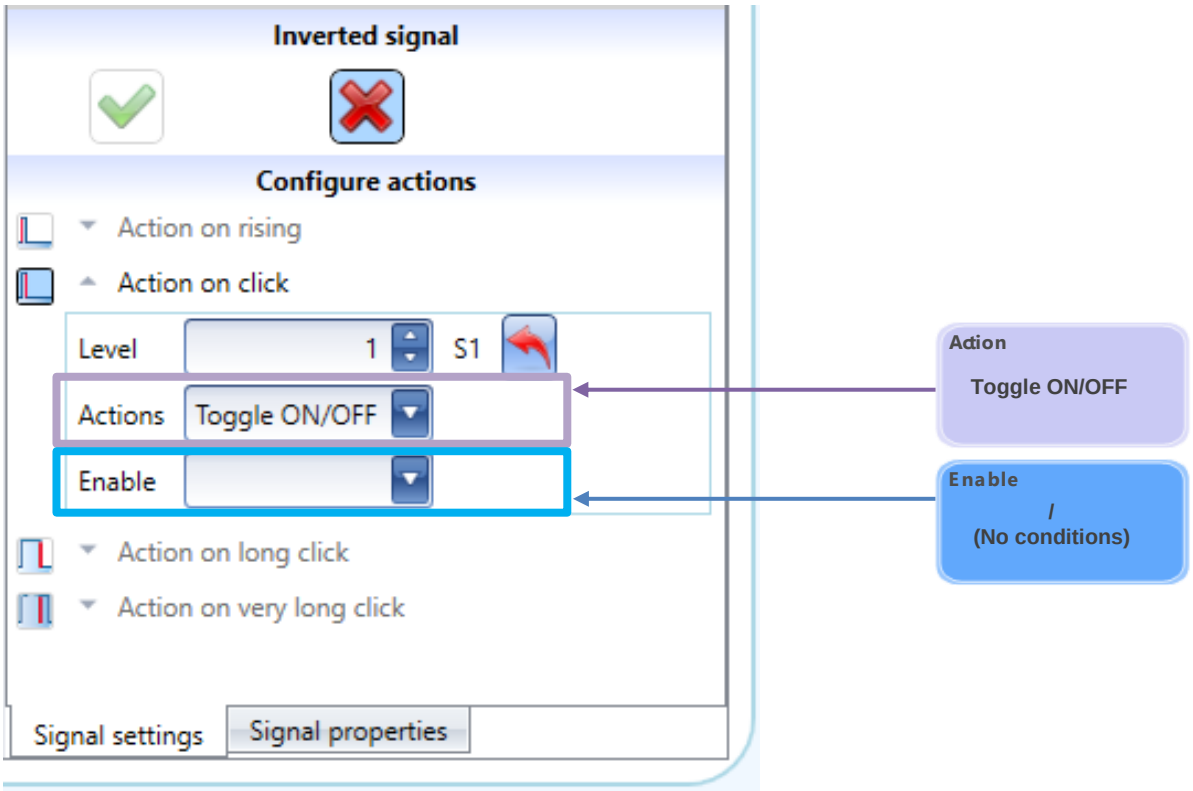
The next time the push button is clicked, the **light** function will turn OFF, thus the Action is set to perform a *Toggle ON/OFF*, as shown below:



Push Button settings

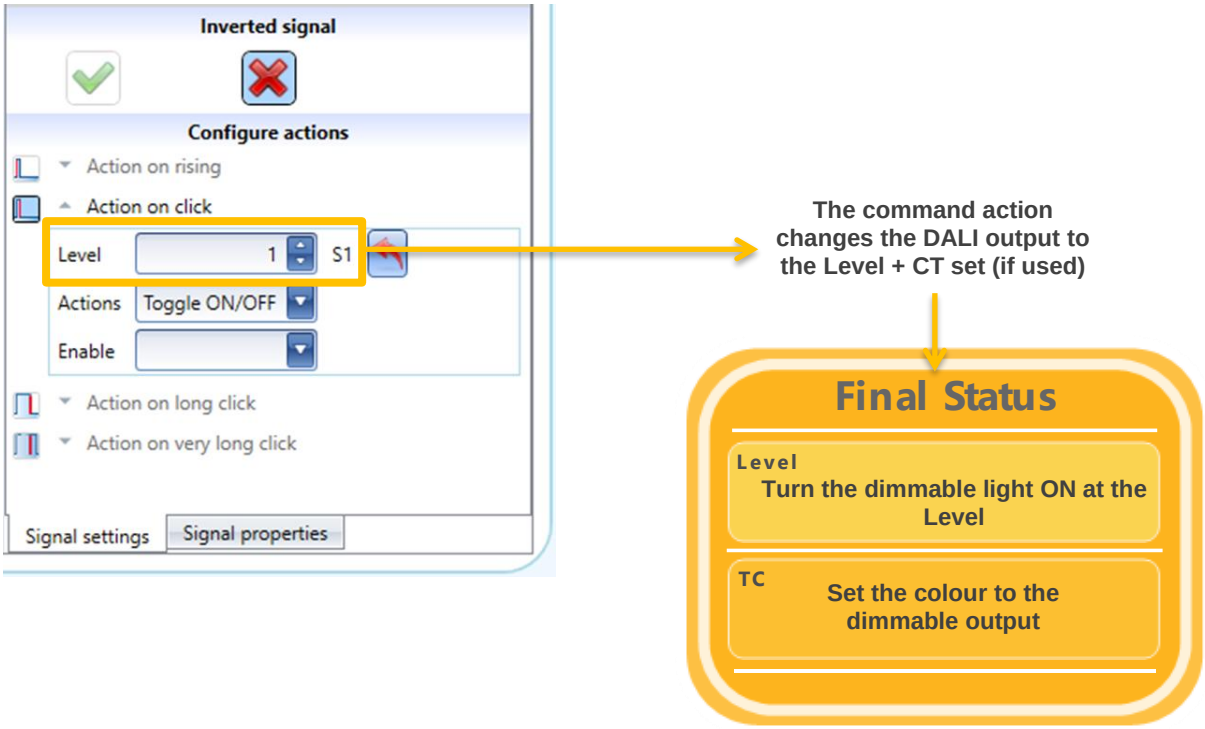


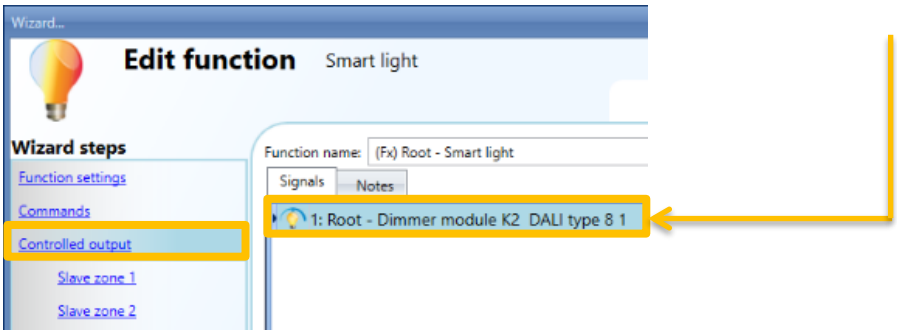
Command settings



Controlled output settings

The DALI output will be set to the selected level (+ **CT** if the temperature color functionality is used)





To select the output signal that is controlled by the Smart light function, click on *Controlled output* and then double click on the Signal window. Once the output window is opened, select the relevant outputs from the list.

The signals available in the output window are the following ones:

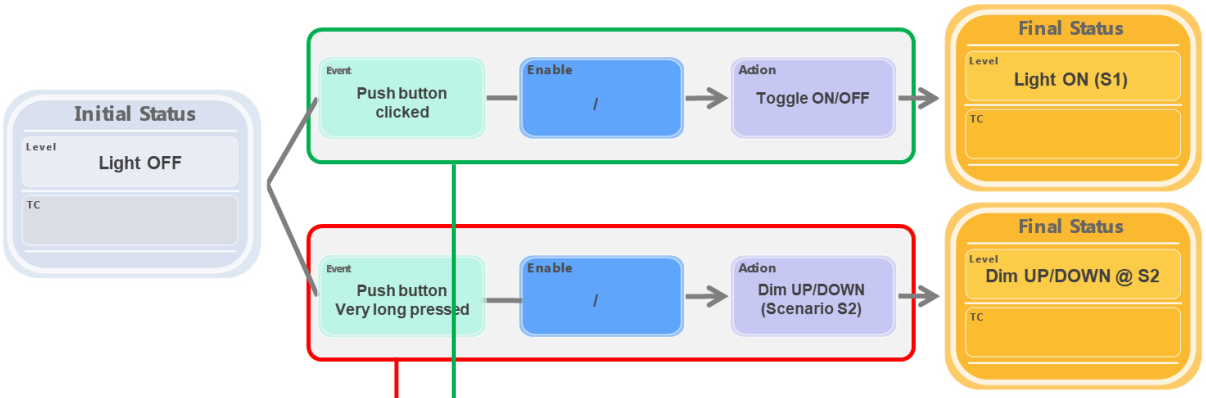
Max 1 signal can be chosen, as follows:

- 1 DALI Ballast
- 1 DALI Group (to manage in parallel multiple DALI ballasts)
- 1 DALI Master (It manages all the DALI ballasts connected to a SB2DALIT8230 master DALI)

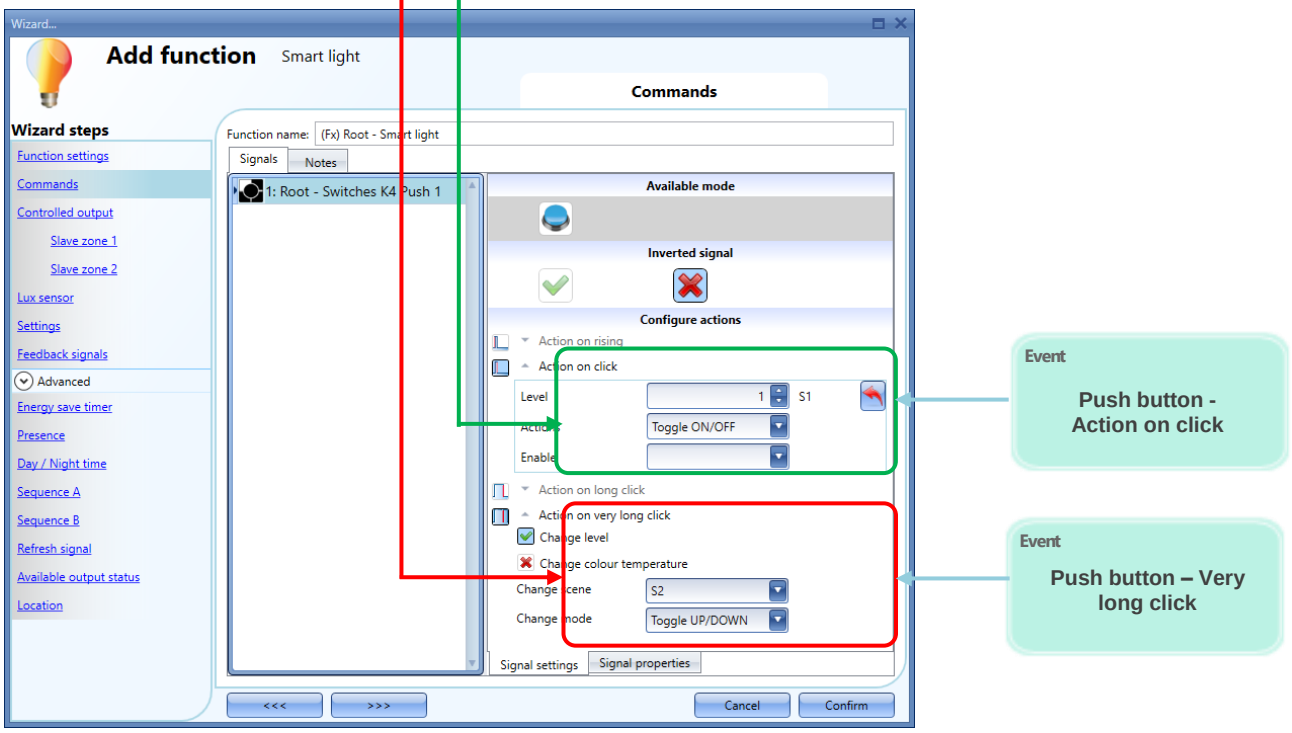
See the DALI management chapter on this manual to configure the DALI master.

10.8.2 Push button - Toggle ON/OFF + Dim UP/DOWN

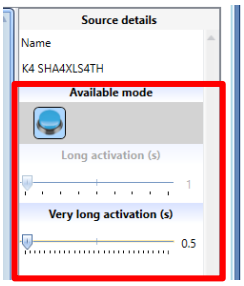
Toggle the light ON/OFF with a push button and dim the light UP/DOWN with the same push button
In the example shown below, the light function will toggle ON(S1)/OFF when the push button is clicked, the same button held downmakes the light dim UP/DOWN from Scenario S2.



Push Button settings



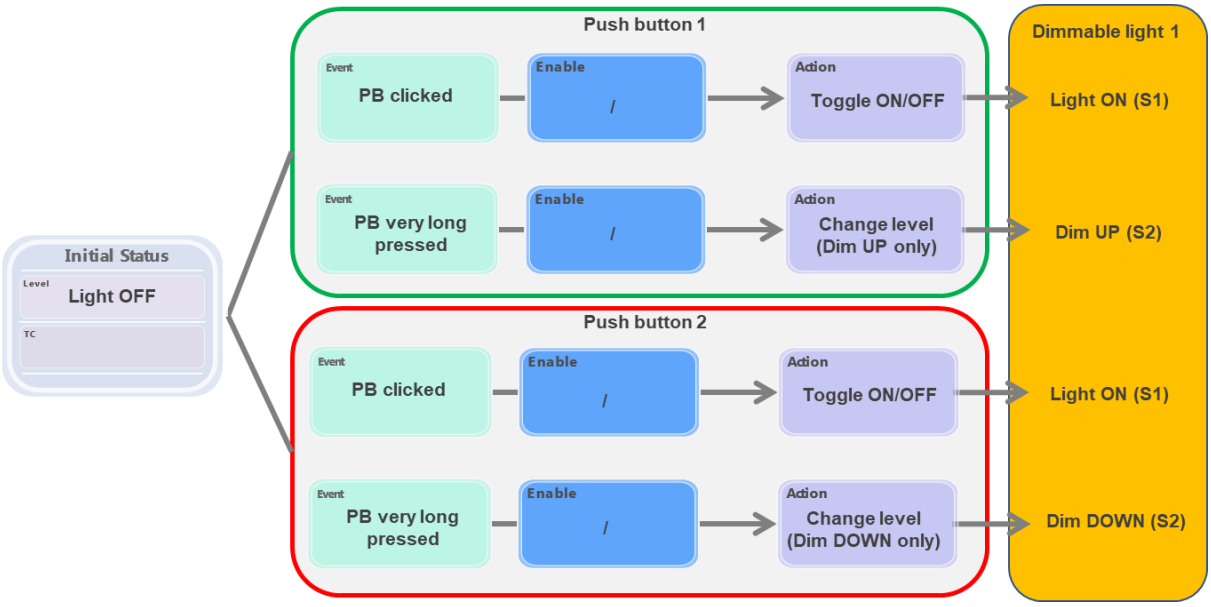
Note: you cannot select to toggle ON(S1)/OFF using Rising Edge because dimming will not work.
The very long time has to be adjusted in order to speed up the dimming.



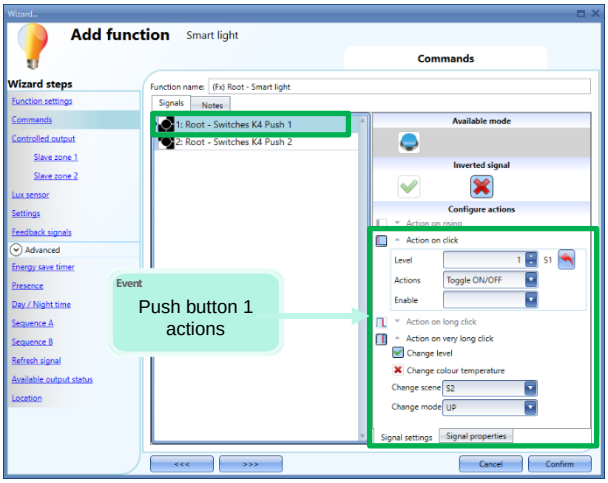
10.8.3 Push button 1,2 - Toggle ON/OFF and Dim UP , Dim DOWN

Toggle the light ON/OFF with a push button and dim the light UP with push button 1 and dim the light DOWN with Push button 2

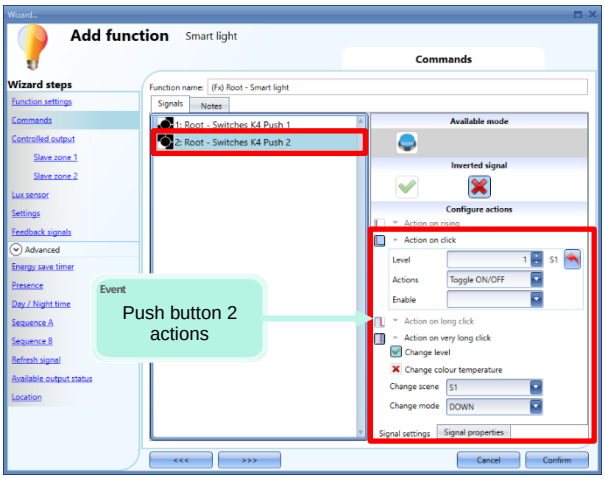
In the example shown below, the light function will toggle ON/OFF when the Push button 1 or Push button 2 is clicked. If the Push button 1 is held down (see Very long press timing) the light will be dimmed UP (starting from scenario S2); the same actions will be performed in the opposite direction whenever the Push button 2 is held down.



Push Button 1 Settings

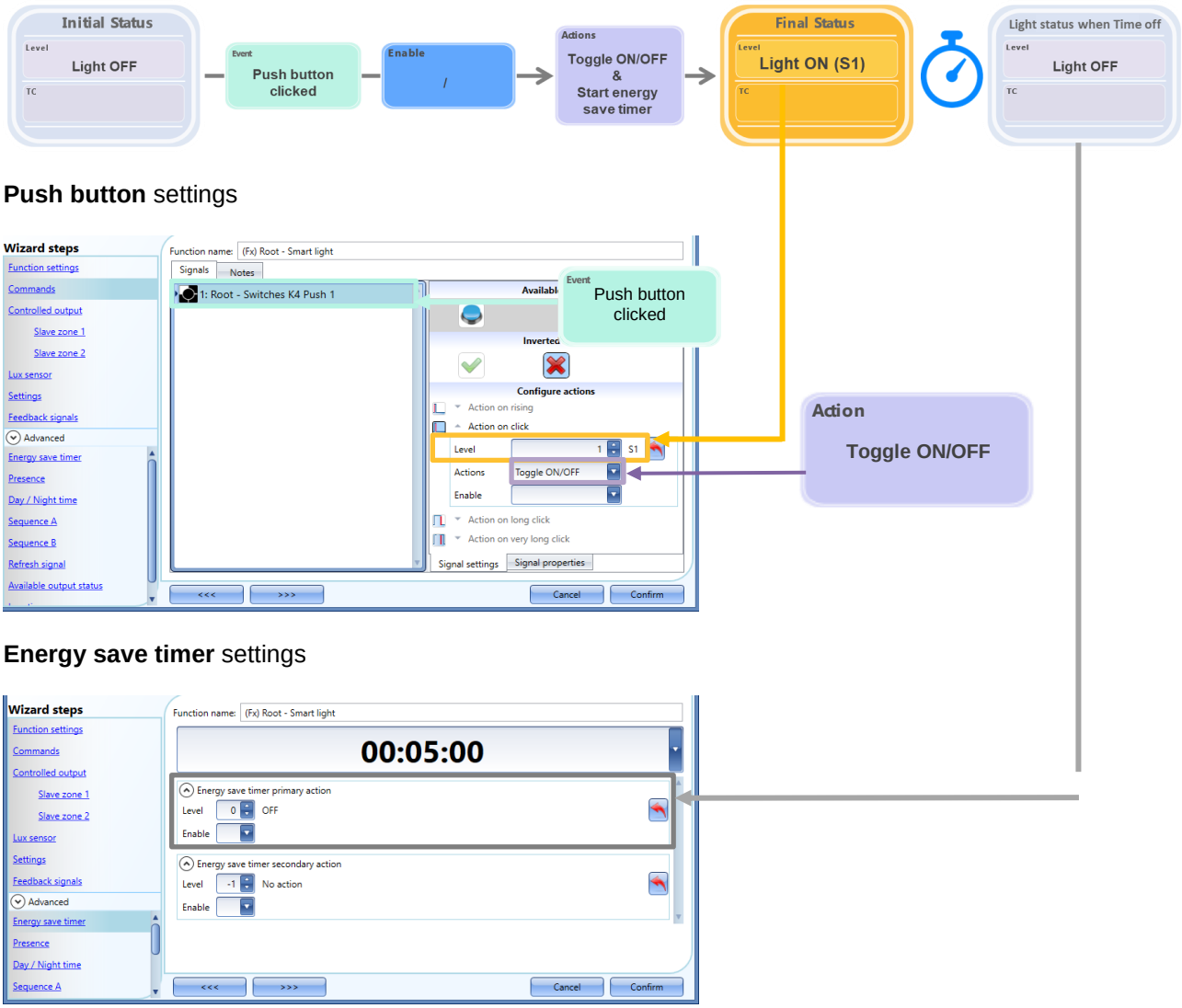


Push Button 1 Settings



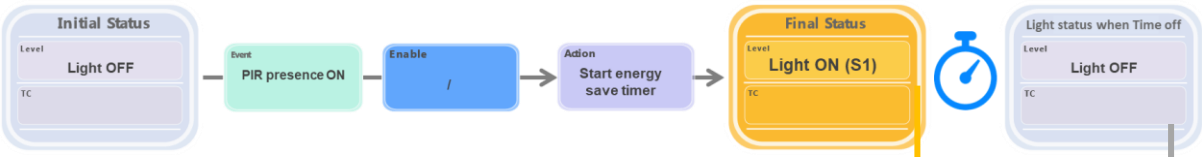
10.8.4 Push Button + ES timer

Turn the light ON with a push button and start the energy save timer to switch the light OFF when it expires. In the example shown below the light function will toggle ON/OFF if the button is clicked and starts the energy save timer. The energy save timer is set to 5 minutes, and it will turn the light OFF when expired



10.8.5 PIR sensor + ES timer

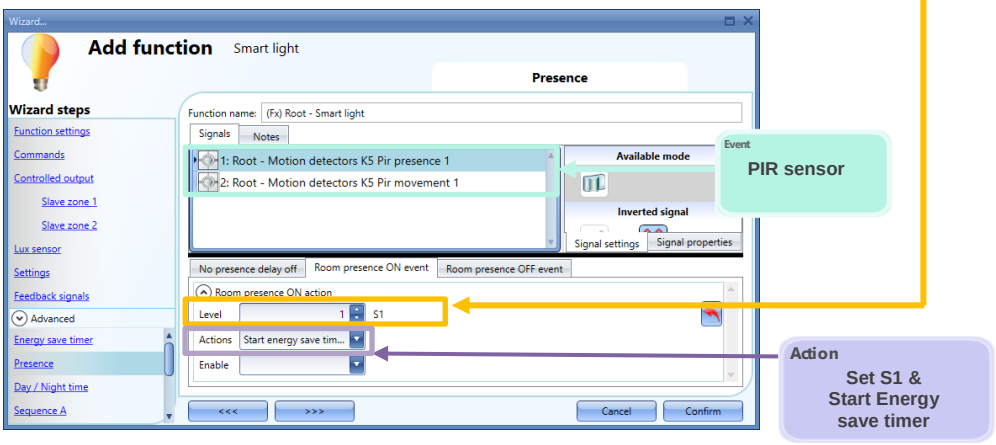
Turn the light ON with PIR sensor and start the energy save timer to switch the light OFF when time expires.



Presence settings

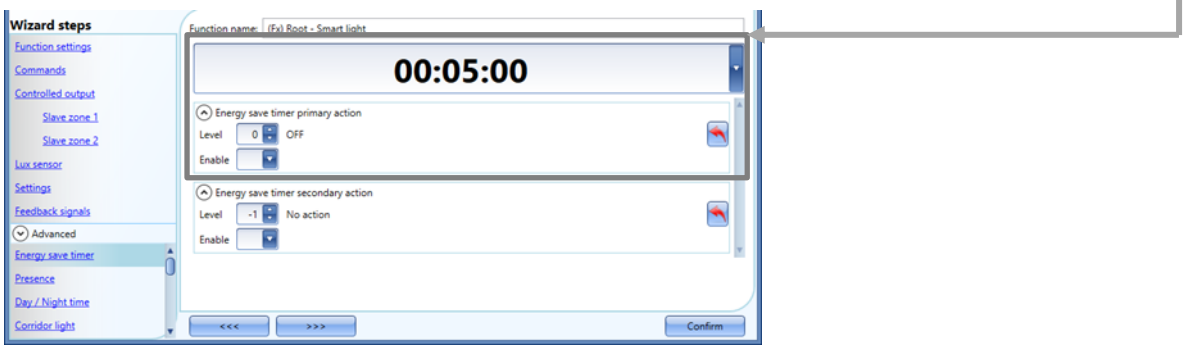
PIR movement ON turns the light ON at scenario S1

PIR presence ON reloads the energy save timer

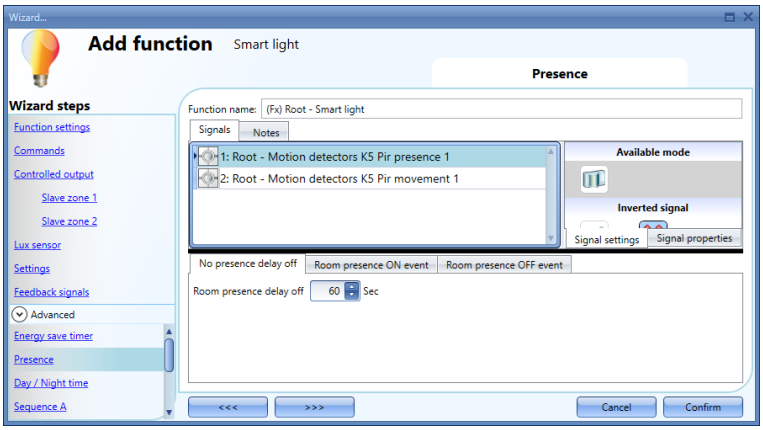


Energy save timer settings

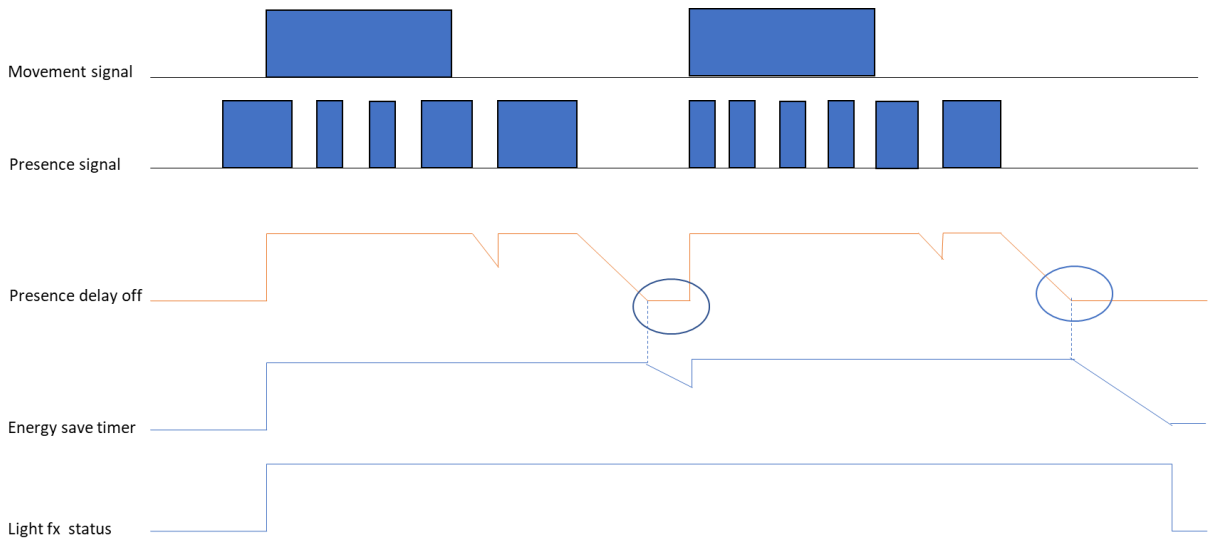
Energy save timer turns the light OFF after 5 minutes



No presence delay off settings

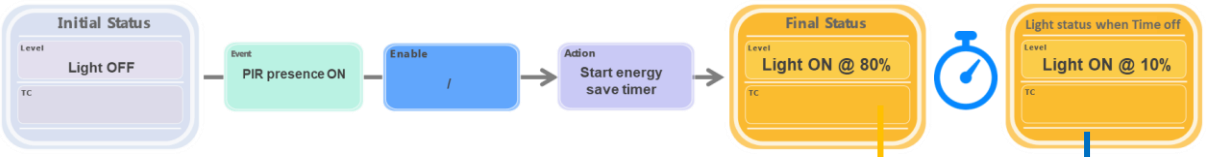


The graph below shows how the *Room presence delay off* works versus the Energy save timer:



10.8.6 PIR sensor + ES timer

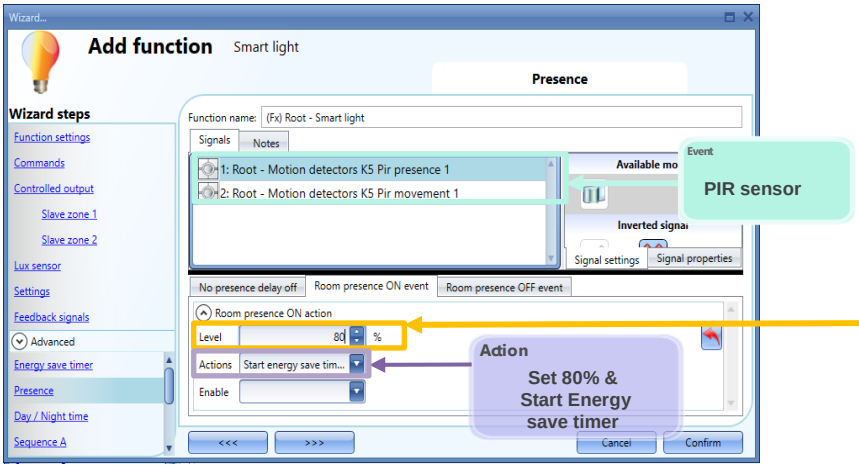
Turn the light ON to 80% by means of a PIR sensor, PIR sensor starts the Energy save timer. When the energy save timer expires, the light is set to 10%.



Presence Settings

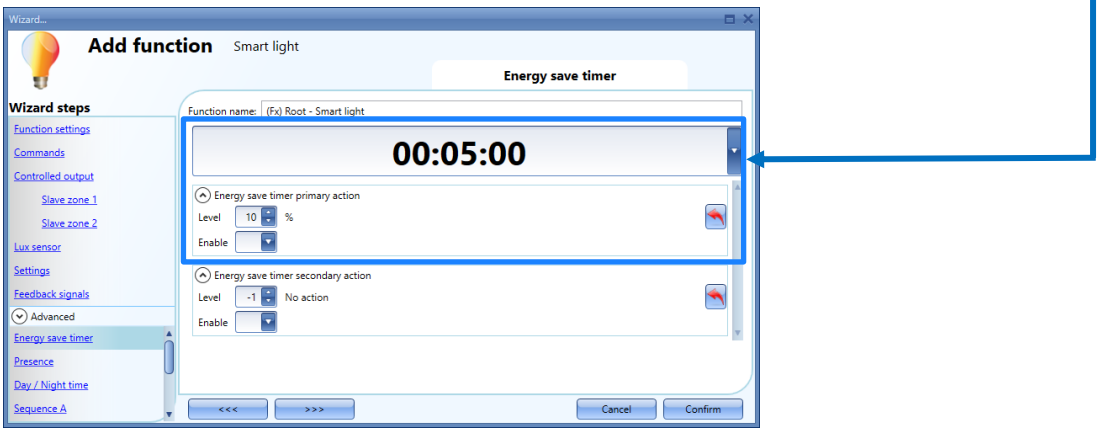
PIR Movement ON turns the light ON at 80%

PIR Presence ON reloads the energy save timer



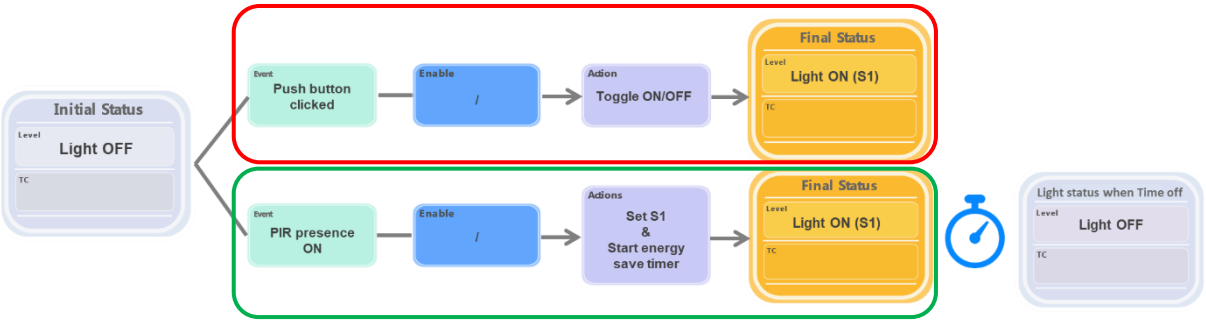
Energy save timer settings

Energy save timer turns the light to 10% after 5 minutes

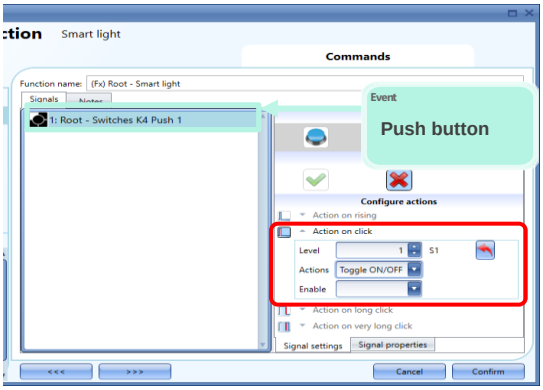


10.8.7 Push button + PIR sensor + ES timer

Toggle the light ON/OFF by means of Push button and PIR sensor. The PIR starts the Energy save timer to switch the light OFF when it expires.



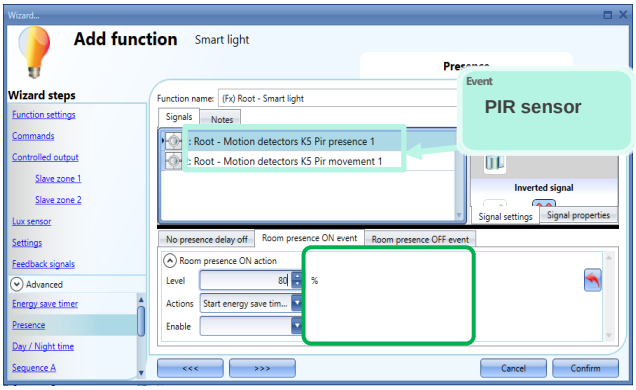
Push button settings



Presence settings

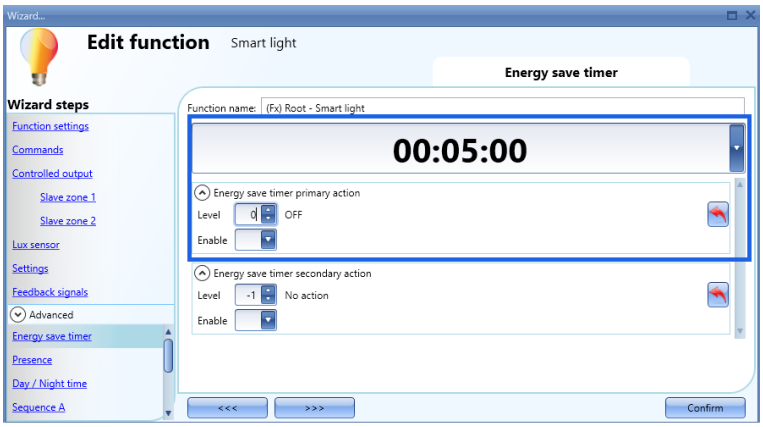
PIR Movement ON turns the light ON at 80%

PIR Presence ON reload the ES timer



Energy save timer settings

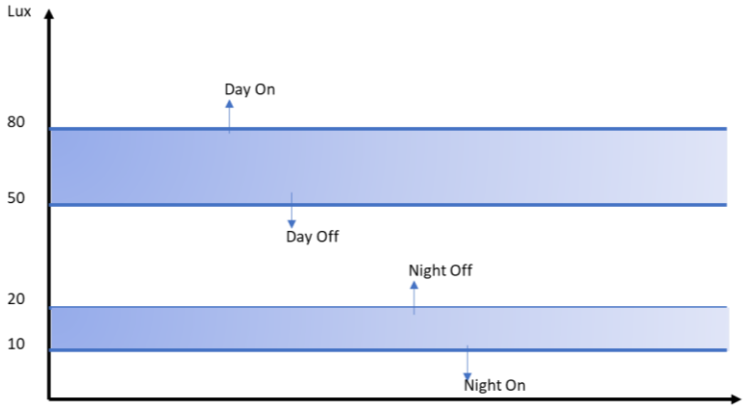
Energy save timer turns the light OFF after 5 minutes

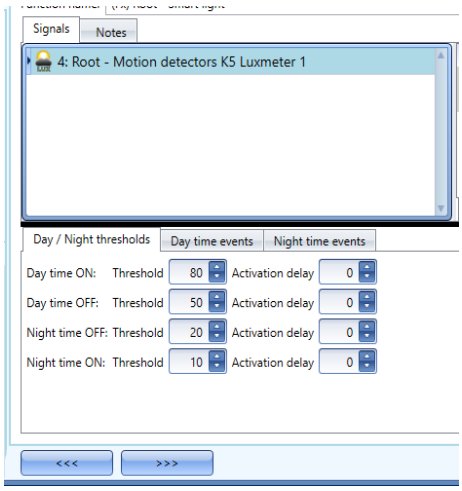


Day/Night time settings

The Smart light function has a new Day Night comparator where the user can set up to 4 different light thresholds. These are available as Enable conditions, such as

Day time ON
Day time OFF
Night time ON
Night time OFF



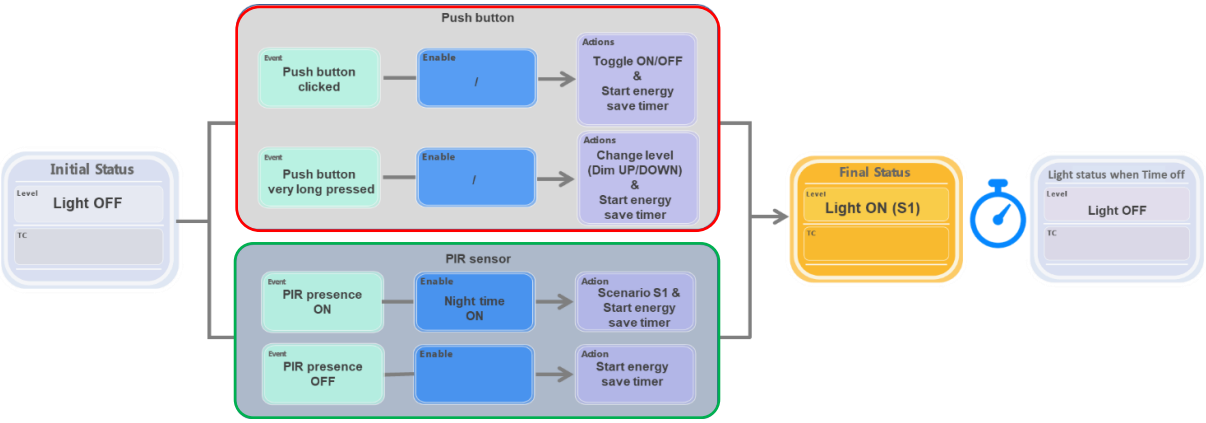


10.8.8 Push Button + PIR sensor enabled by the Luxmeter sensor

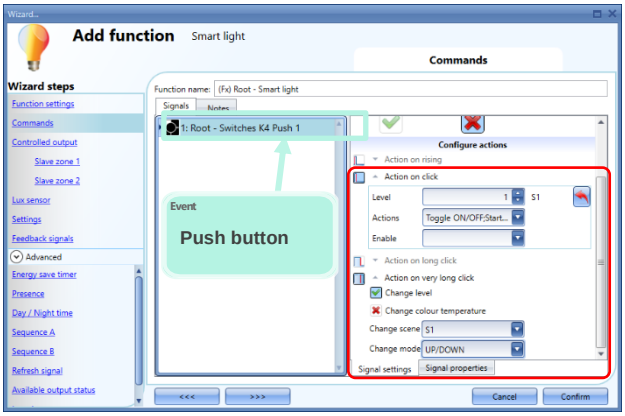
Toggle the light ON/OFF with push button and dim the light UP/DOWN with the same push button. The push button click will start also the energy save timer.

The energy save timer will be reloaded at any click or every time the PIR sensor detects presence.

The PIR sensor will turn the light ON automatically only if the Night time condition is ON.

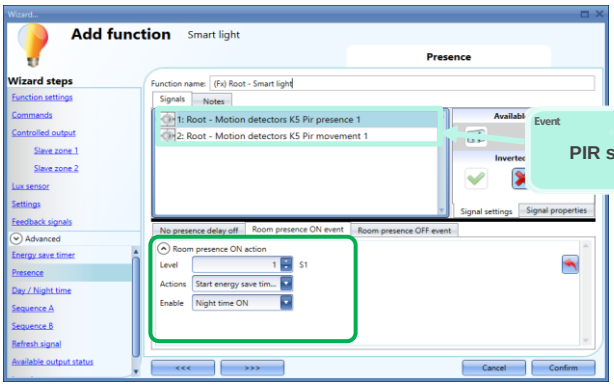


Push button settings

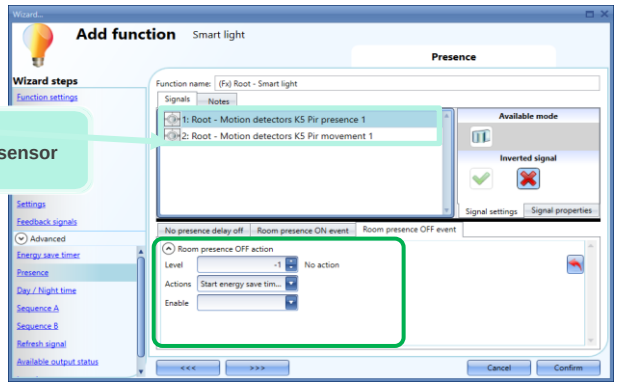


PIR sensor

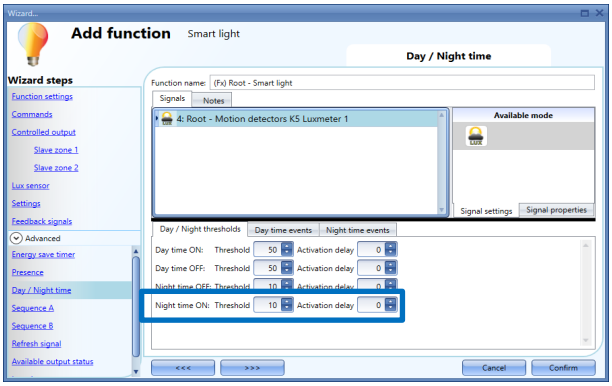
Presence ON settings



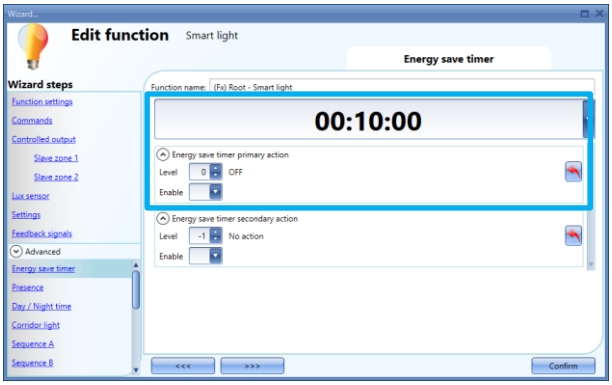
Presence OFF settings



Night time ON threshold settings (Luxmeter sensor)

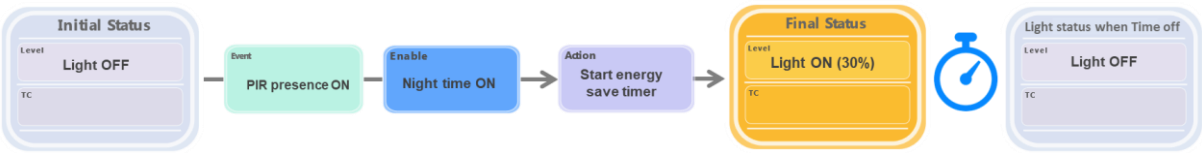


Energy save timer settings

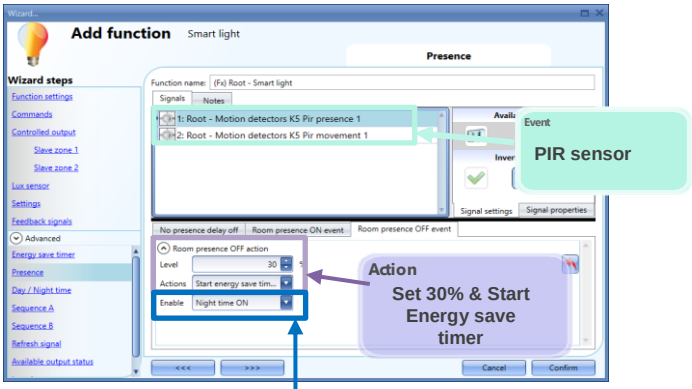


10.8.9 PIR sensor + ES timer and Night time condition

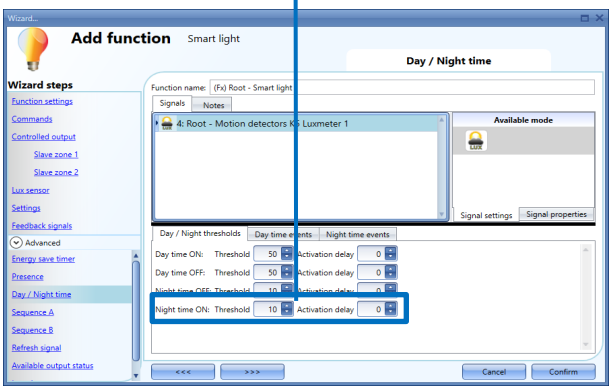
Turn the light ON to 30% by means of the PIR sensor only in Night time ON condition. PIR sensor starts the Energy save timer to switch the light OFF



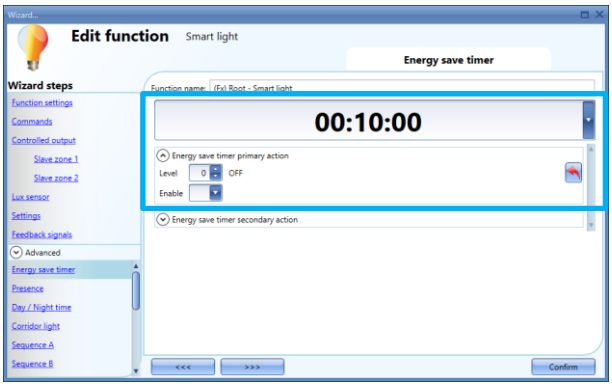
Presence settings



Luxmeter settings

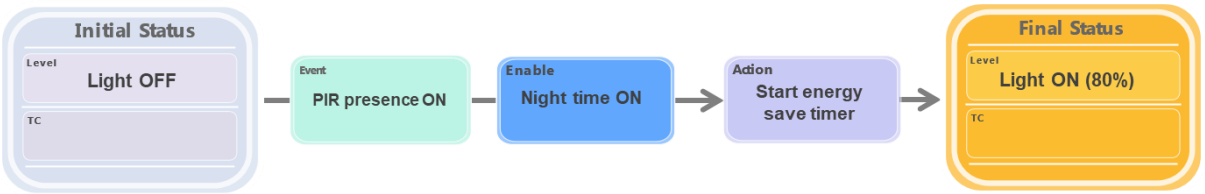


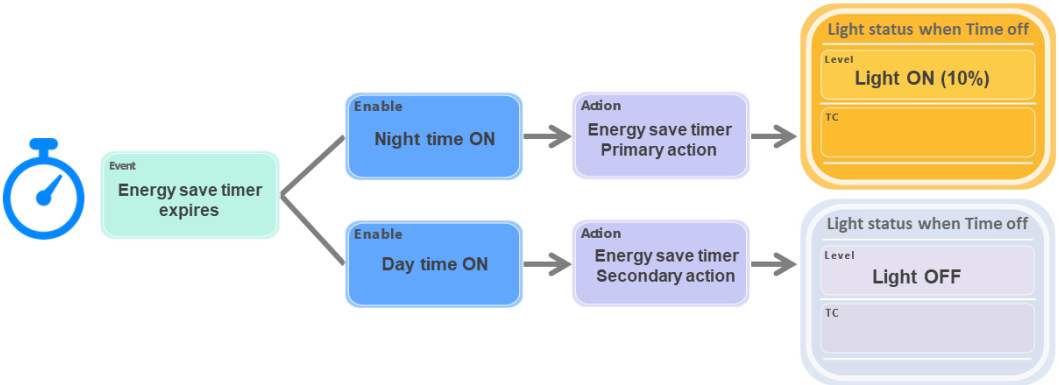
Energy save timer settings



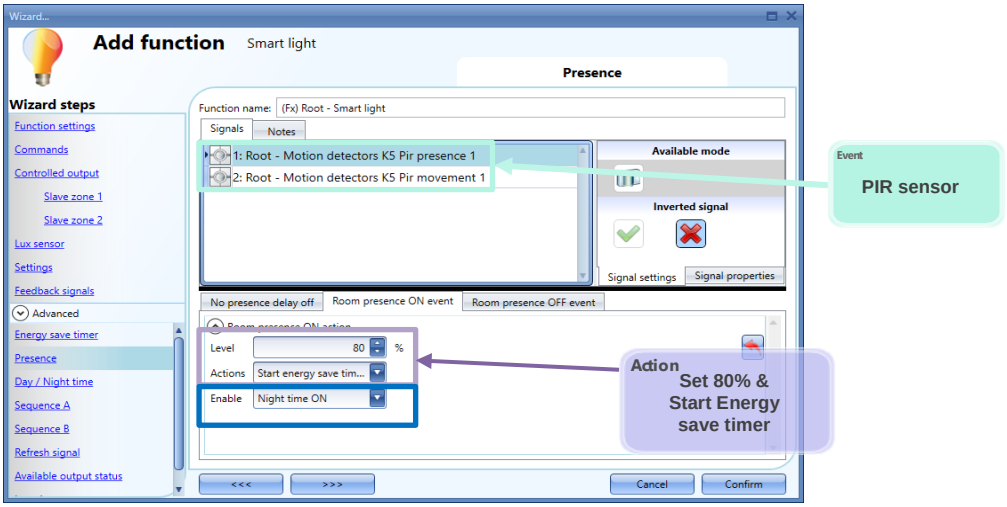
10.8.10 PIR sensor + ES timer + Day and Night time conditions

Turn the light ON to 80% by means of the PIR sensor in Night time ON condition, PIR sensor starts the Energy save timer. When the timer expires, if Night time ON the light will be set to 10%, if Day time ON the light is turned OFF

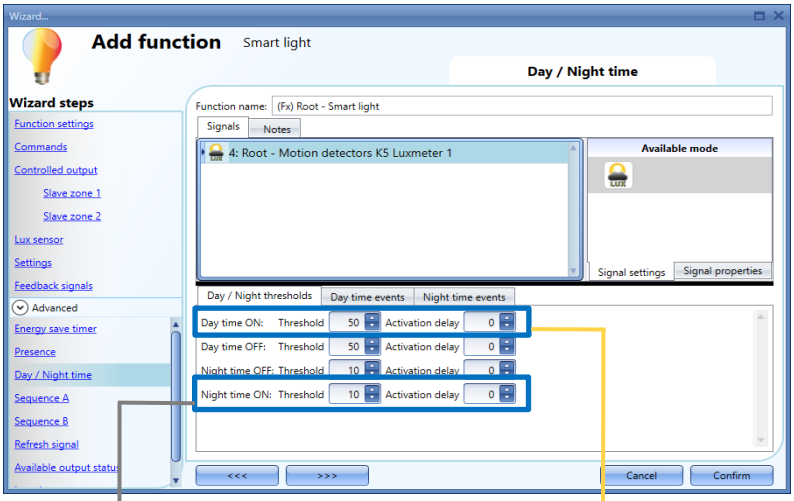




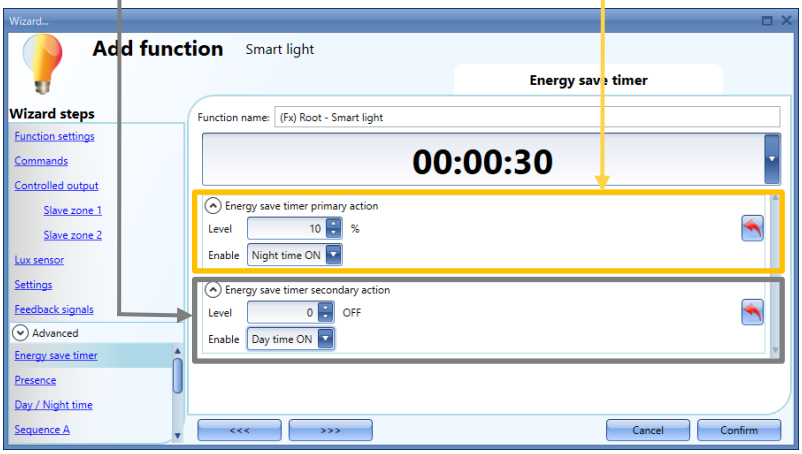
Presence settings



Luxmeter settings

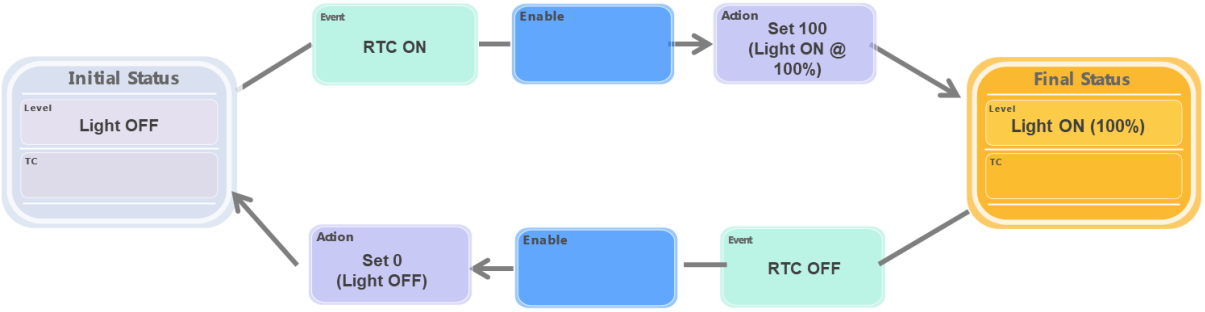


Energy save timer settings



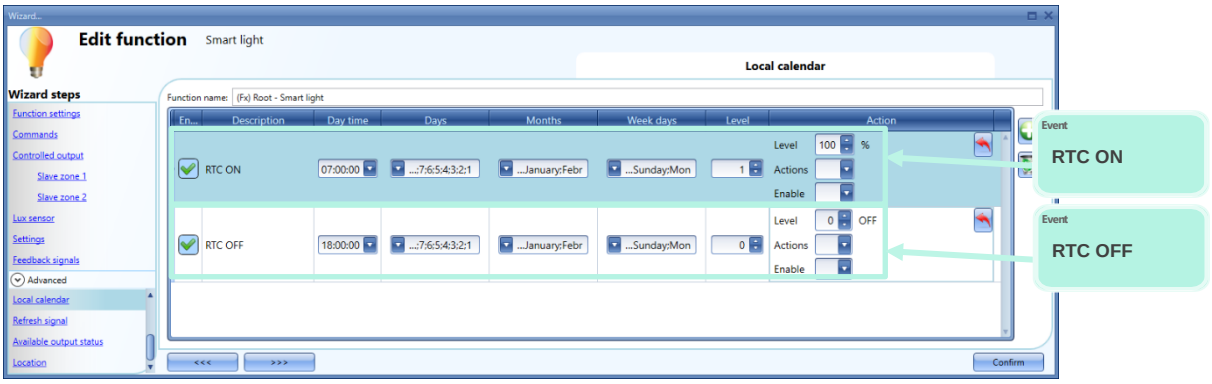
10.8.11 RTC (Local calendar)

Local calendar turns the light ON at 100% when RTC is between 7:00 and 18:00 (RTC ON), meanwhile outside this period (RTC OFF) the light is turned OFF



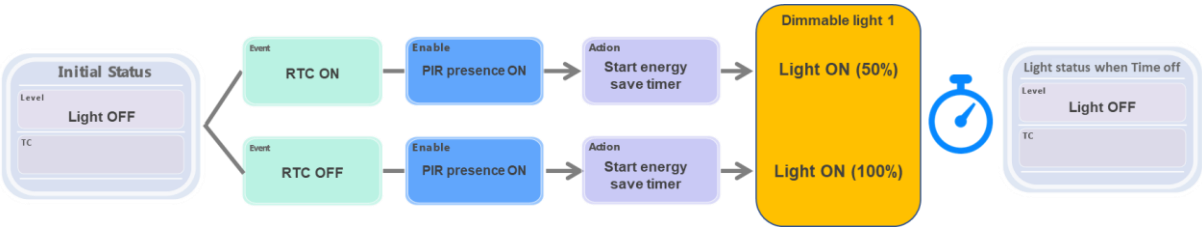
Local calendar (RTC) settings

PIR turns the light on at 100 % when time clock is between 07:00 and 18:00;
PIR turns the light OFF when time clock is before 07:00 or after 18:00;



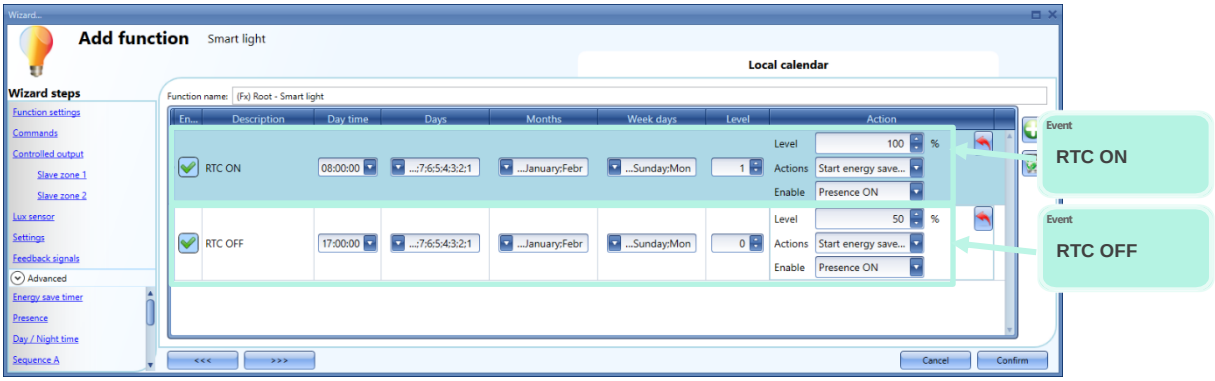
10.8.12 PIR + RTC + ES timer

PIR sensor turns the light ON at 50% and starts the Energy save timer when RTC is between 8:00 and 17:00 (RTC ON), meanwhile outside this period (RTC OFF), PIR sensor turns the light ON to 100% and starts the Energy save timer

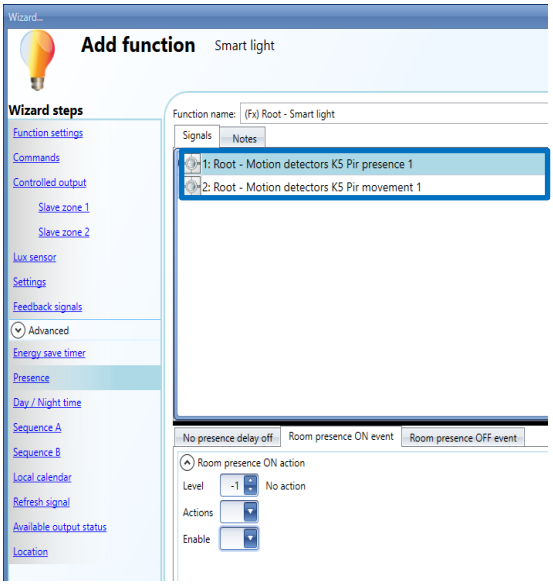


Local calendar (RTC) settings

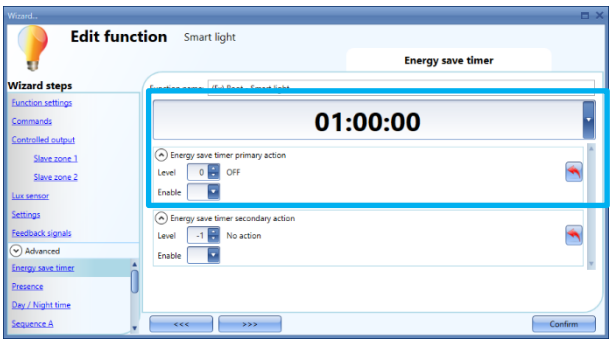
PIR turns the light on at 50 % when time clock is between 08:00 and 17:00;
PIR turns the light on at 100 % when time clock is before 08:00 or after 17:00;



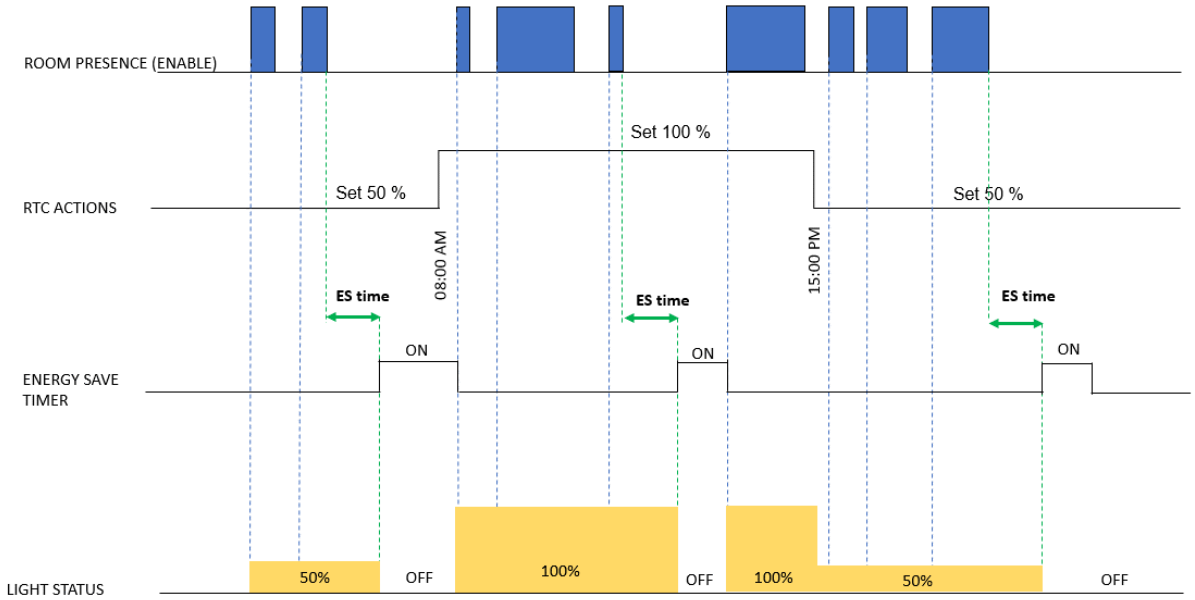
PIR presence settings



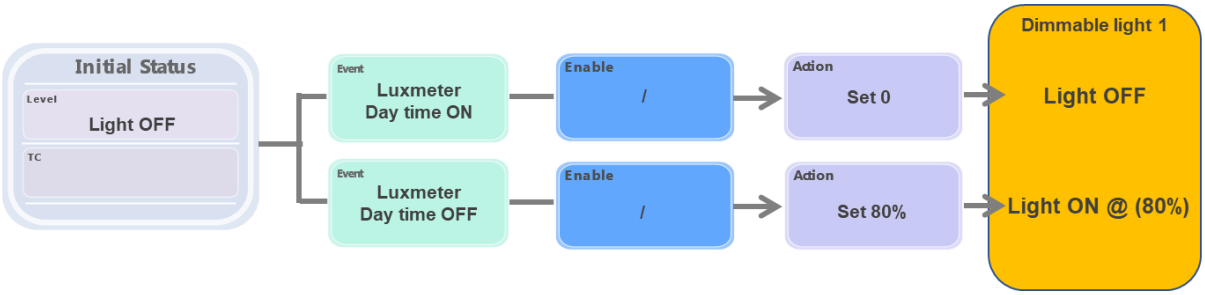
Energy save timer settings



The graph below shows when the light is turned ON by PIR sensor according to the RTC conditions and the ES timer:



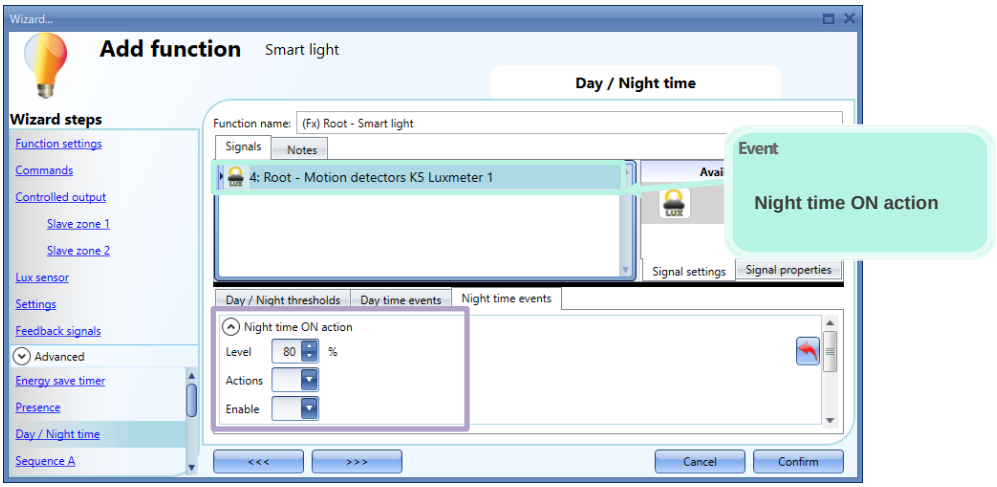
10.8.13 Luxmeter sensor turns the light ON to 80% when Night time ON

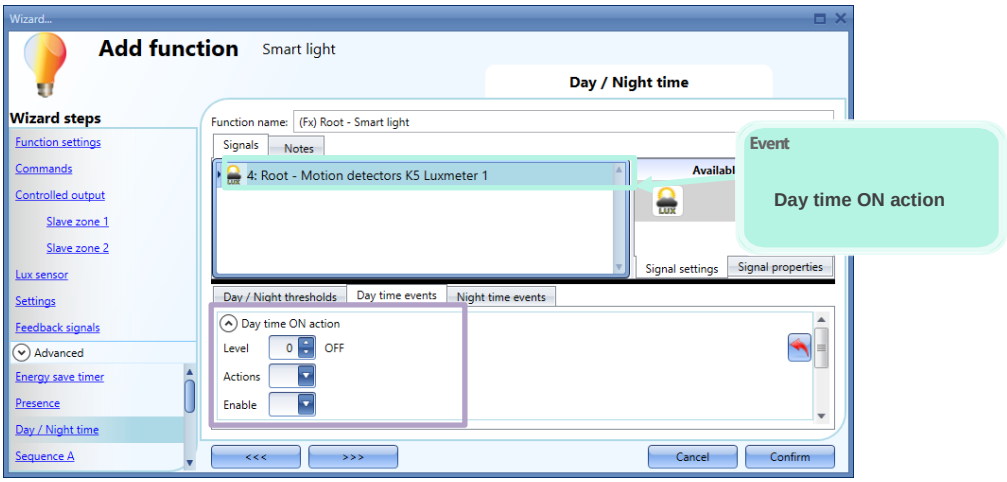


Luxmeter settings

See examples above

Day / Night time settings

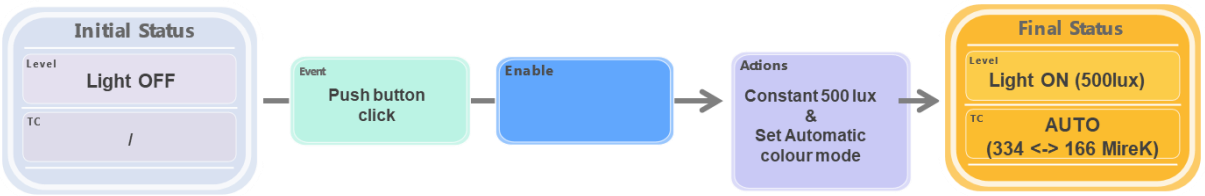




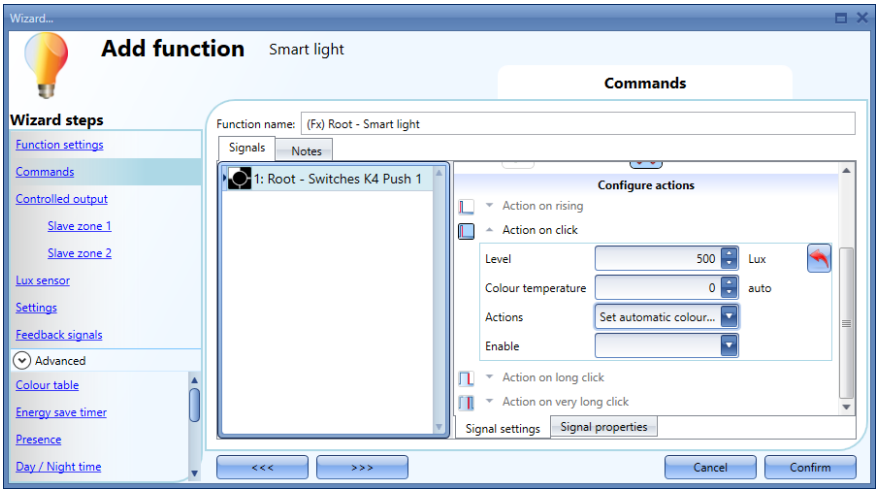
10.8.14 Change the colour temperature automatically according to the colour table

This use case is suitable only for DALI ballasts TYPE 8

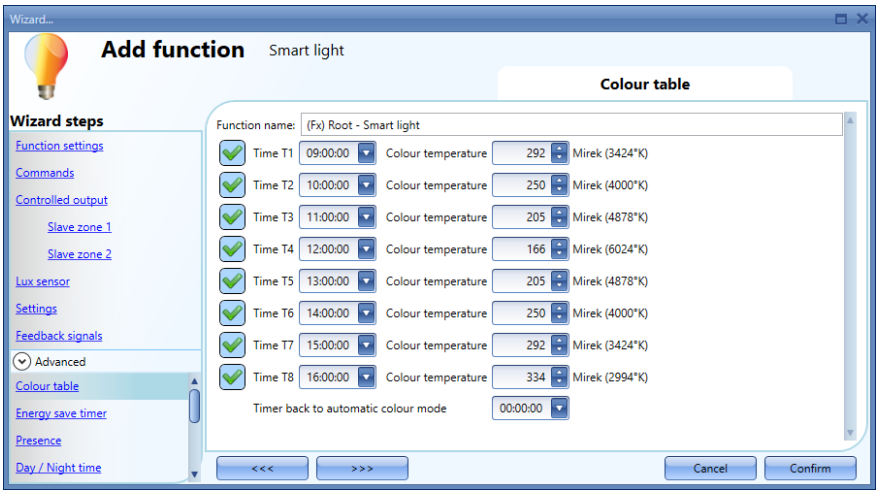
The colour temperature is automatically changed according to the colour table that is synchronized by the RTC time. The tonality of the light is set warmer in the morning, then changed to colder levels at noon and then slowly set again to warm light in the afternoon.
A push button turns ON/OFF the Constant light and activate the automatic management of the colour temperature.



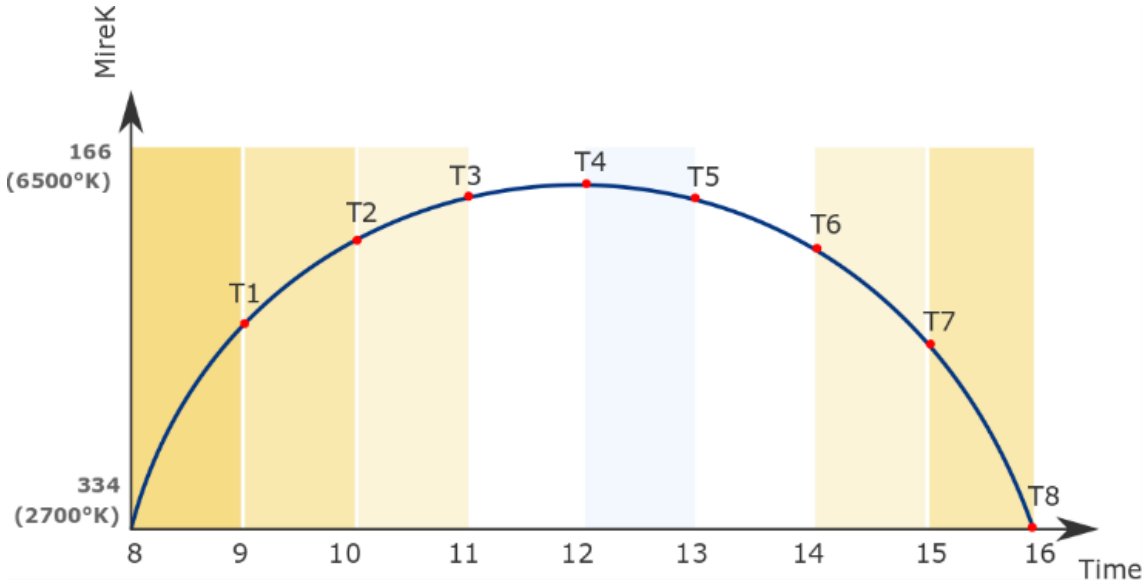
Push button settings



Colour table settings
The light intensity changes according to the Constant light algorithm while the colour temperature is changed according to the *Colour* table setpoints (T1 to T8).

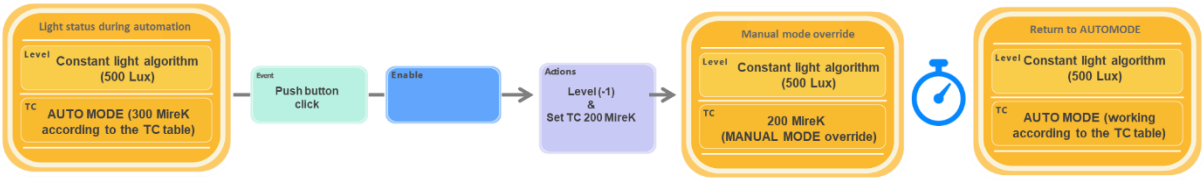


The light tonality is automatically changed by the Smart light function according to the diagram shown below:



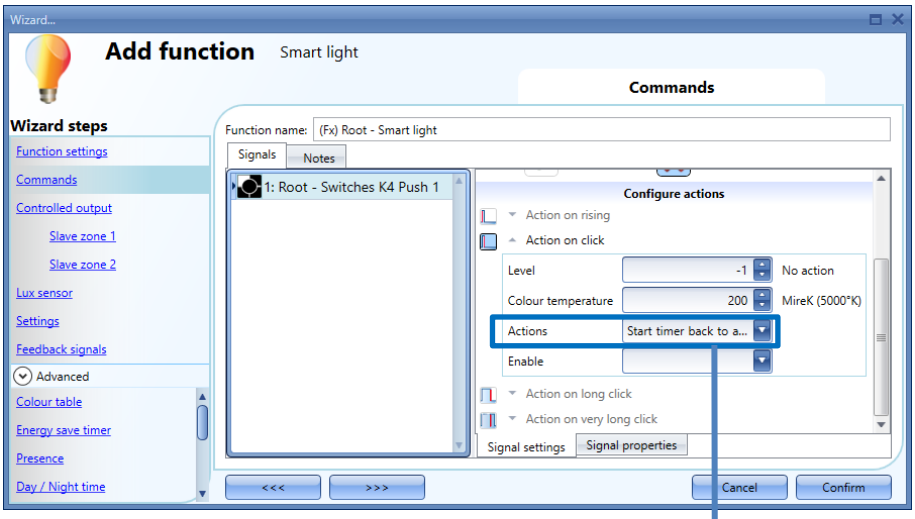
10.8.15 Change manually the colour temperature overriding the colour table managed by the RTC.

This use case is suitable only for DALI ballasts TYPE 8
When the colour temperature is automatically managed by the Colour table, it can be changed manually by a push button click. The push button starts a timer back to the automatic colour control.



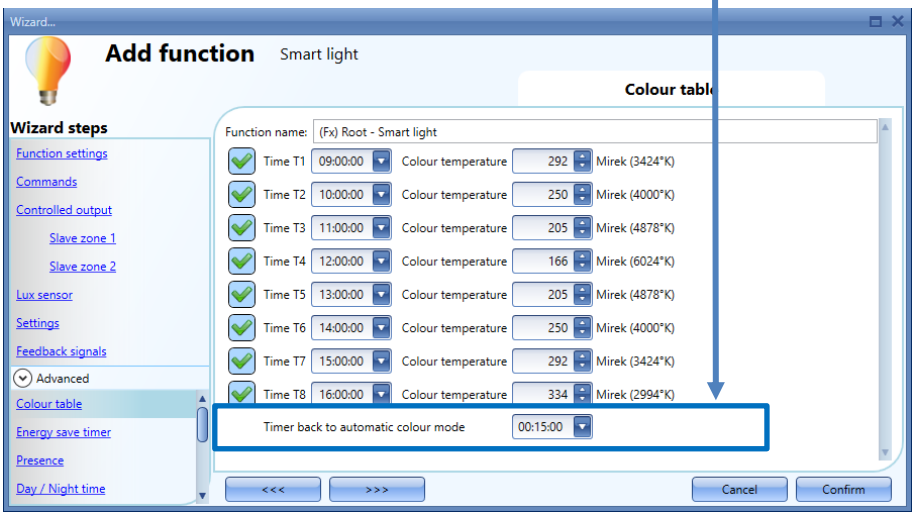
Push button settings

Start timer back to automatic colour mode



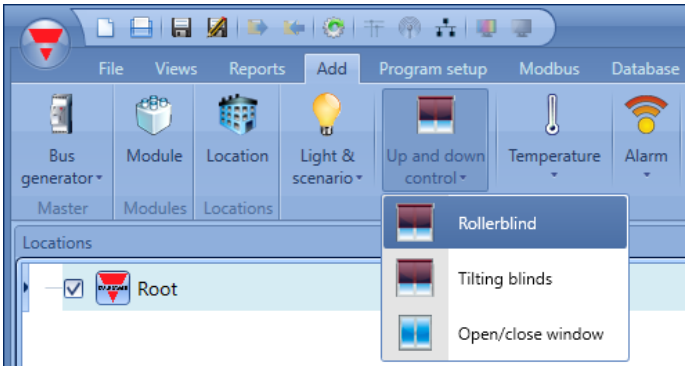
Colour table settings

The Intensity changes according to the Constant light algorithm and the colour temperature is changed according to the Colour table setpoints (T1 to T8).

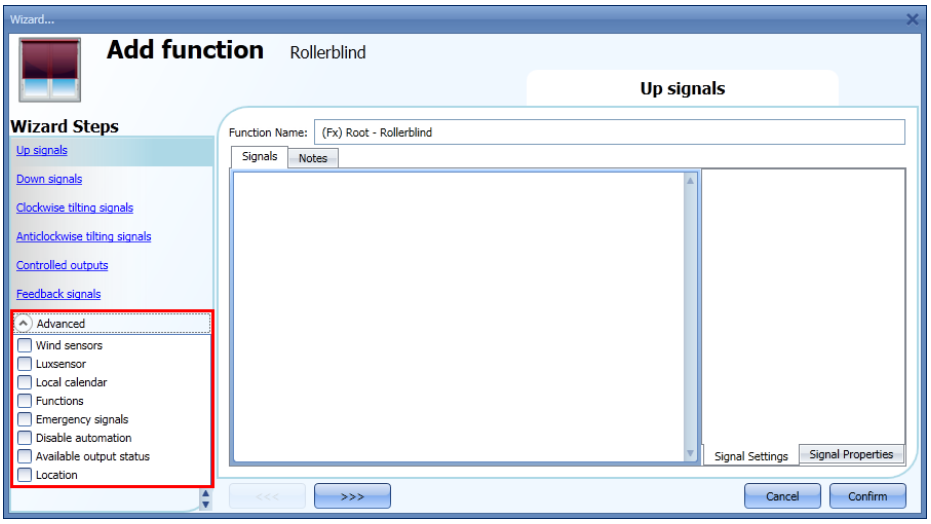


10.9 How to set a Rollerblind function

This function allows the operator to manage the motor to control roller blinds.
The user can either configure a basic function to move blinds up and down or implement an automated system by programming the relevant objects of the function.
To set up this kind of function, the user should select *Up and down control* from the Add menu, then select *Rollerblind* (see picture below). The new function will be added into the selected location.



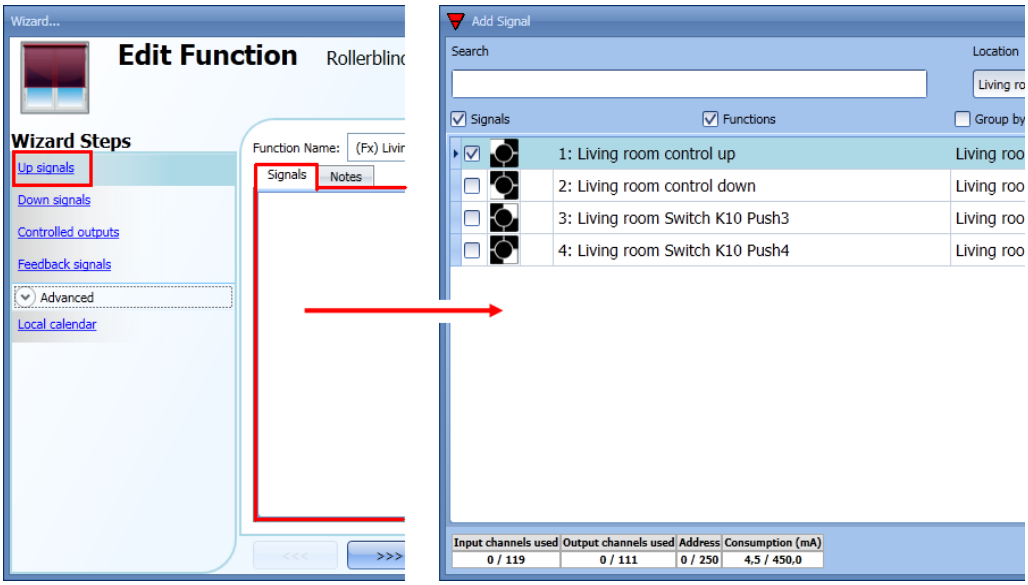
This function manages one or more motor outputs controlled by one or more input commands. The command might be a physical signal like a pushbutton, a switch, a function or a remote command (Webserver, SMS, Modbus TCP/IP).
The automation of the tilting slats can be managed by accessing the *Advanced* section, where the user can select different kinds of automation: wind sensors, rain sensors, lux sensors, calendar.



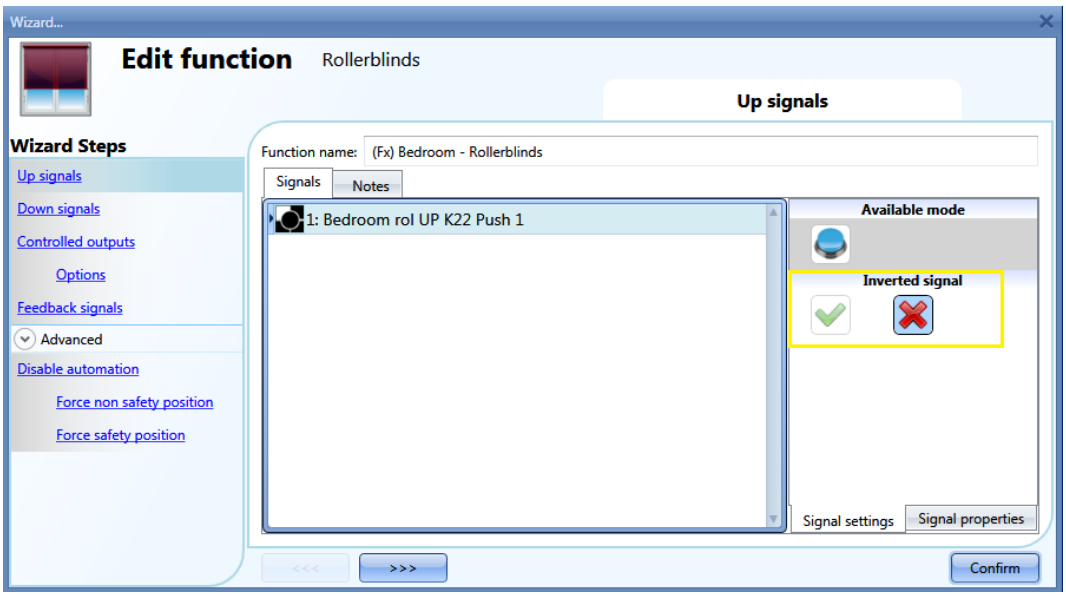
N.B. For the function to behave correctly, we suggest giving an up/down reset command every time the system is started.
The reset command can be given by a pushbutton, via the webserver with administrator access or via the UWP 3.0 Tool using the live signals mode.
When the reset command is sent and the motor is running, the user should wait for the running time to be completed (the running time is that set in the rollerblind module wizard).
For safety reasons, the reset command can be stopped anytime by another command; should this happen, another reset command should be sent to ensure the correct behaviour of the automation starting from a known position.

10.9.1 How to move Up/Down blinds using a manual input

The user should add the input signals to control the Up/Down movement of the blinds.
To add the Up/down signal, the user should click on the relevant section, then double click on the *Signal* window and select the input signal from the list (see picture below).
The input signal may be a pushbutton or a switch.

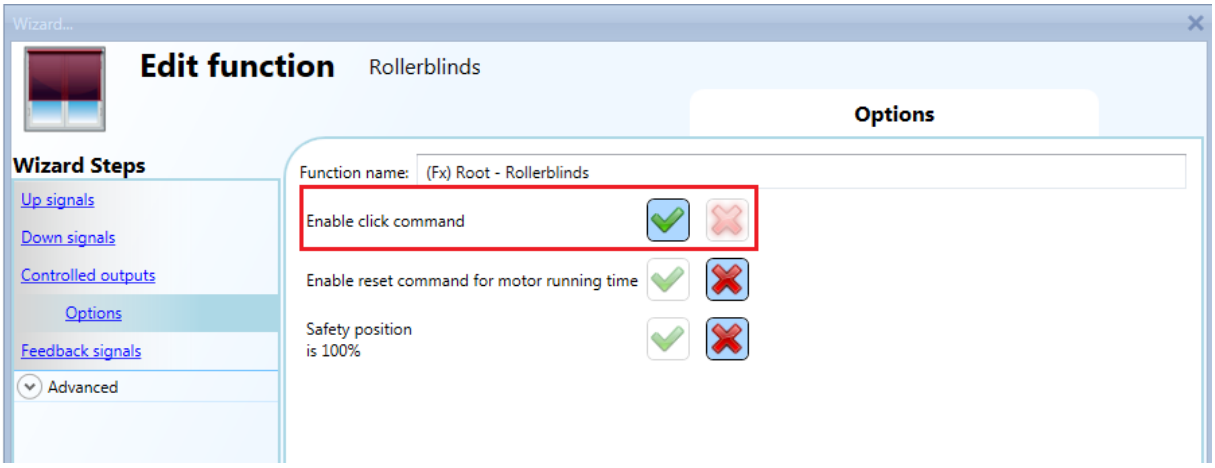


Once the Up/Down signal is entered, the user can also select to use the inverted logic of the signal (see yellow rectangle in the picture below).



All signals selected in the Up/Down window work in parallel (OR level). According to the release time of the signal, the output can behave in two ways:

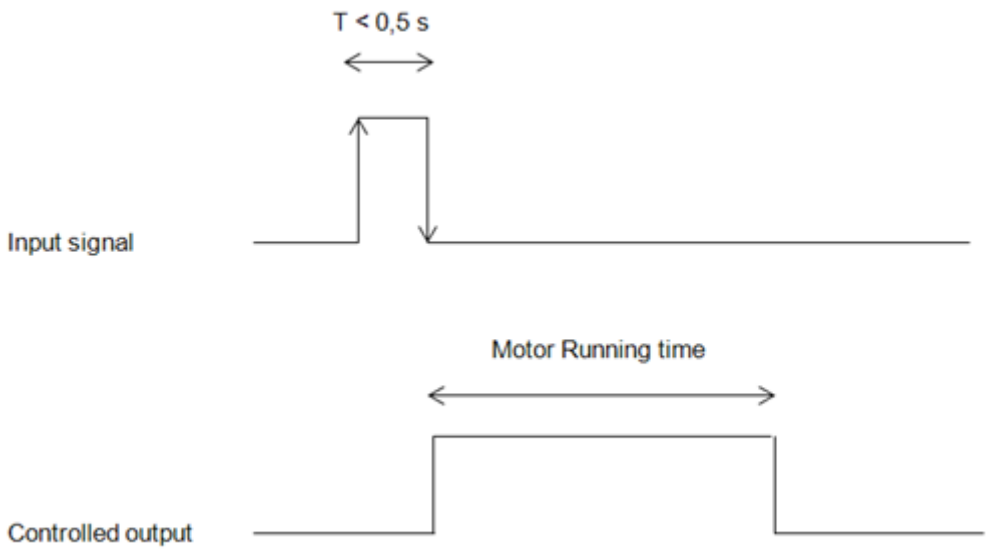
1) A short click action (<0,5sec) on the pushbutton, if enabled, generates a “fully Up/Down” command. The click action is enabled in the *Controlled output options* (see picture below).



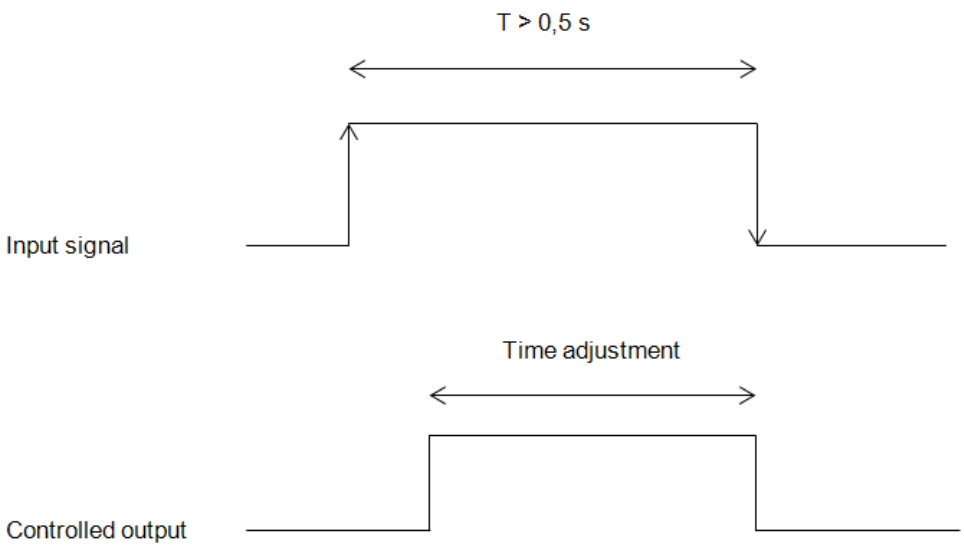
2) A long press (> 0,5 sec) activates the motor output until the input signal is ON (the blind goes up/down until the user keeps the pushbutton pressed or the selected signal is active). This functionality doesn’t need to be enabled.

In the figure below, two examples of activation of the input signal are shown.

Example 1: short press with the “click command” option enabled

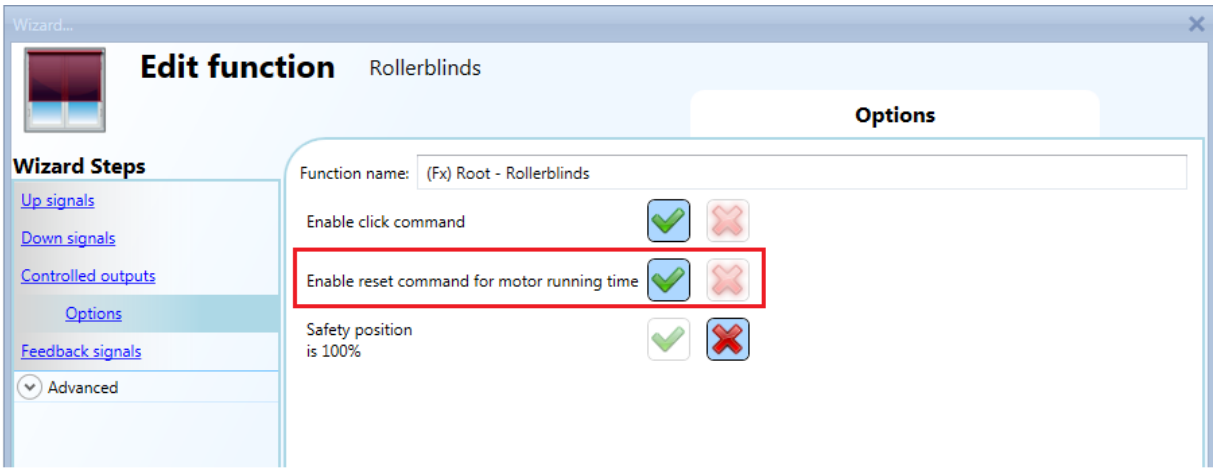


Example 2: long press for the “Up/Down” adjustment

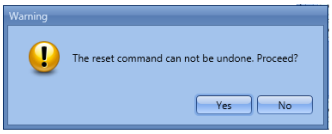


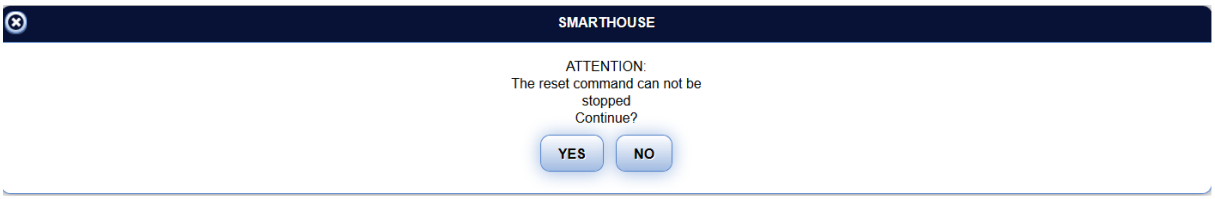
10.9.2 How to set the reset command

The reset command is used to reset the position of the roller blinds after a system start-up or after the downloading of a new project on the UWP 3.0.
This command is activated by a short click that makes the rollerblind go fully up/down (according to the selected direction) and resets the position to 0/100%.



The reset command is also available via the webserver (when accessed as administrator) and via the UWP 3.0 Tool in the live signals mode. When the command is sent by using the webserver or the Tool, a window with a warning message is shown to advise the user not to stop it.

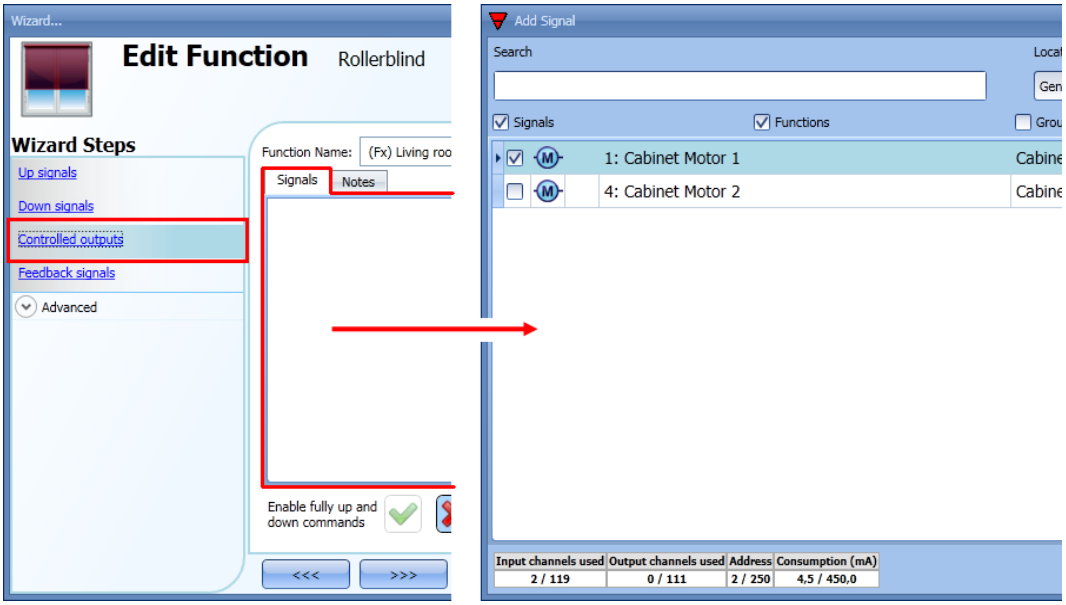




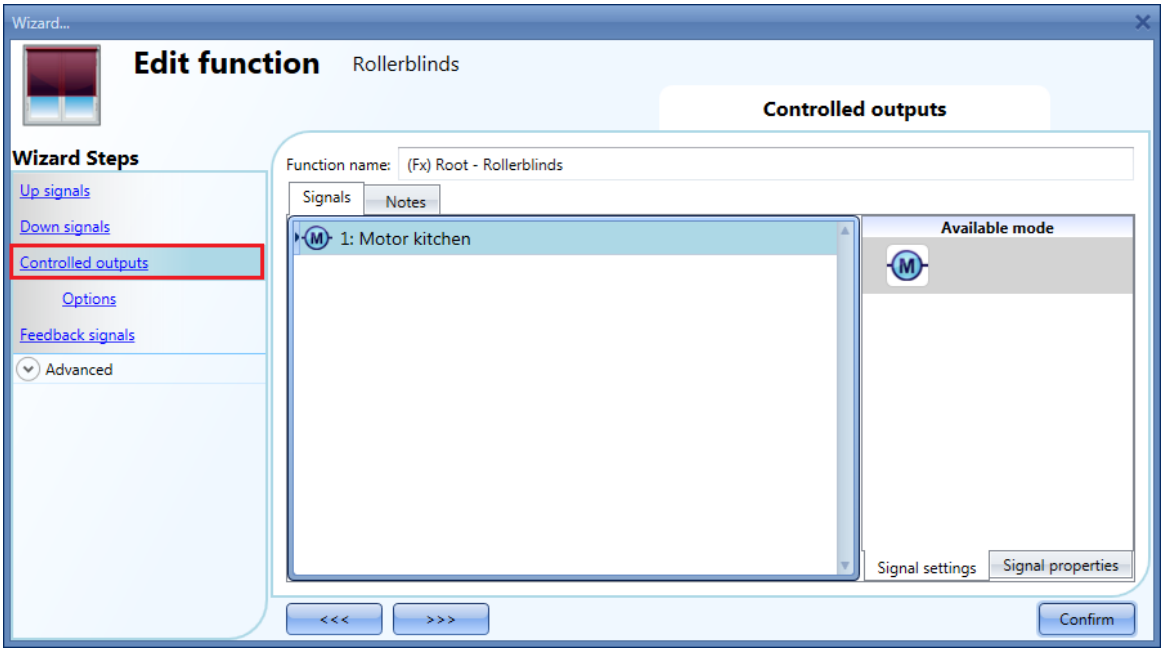
10.9.3 How to select and configure the output

To select the output signal that is controlled by the *Rollerblinds* function, the user has to click on *Controlled output* and then double click on the Signal window. Once the output window is opened, select the relevant motor output from the list.

.

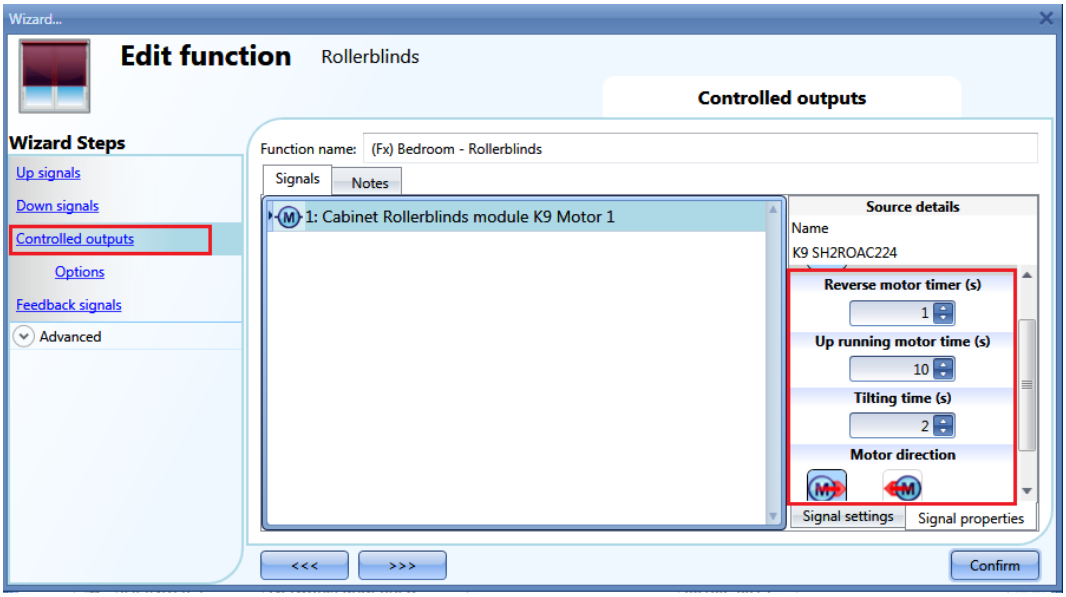


The signals available in this window are only the motor output type.
Up to 10 motor outputs can be chosen and they will be managed in parallel but they must be in the same Dupline network.



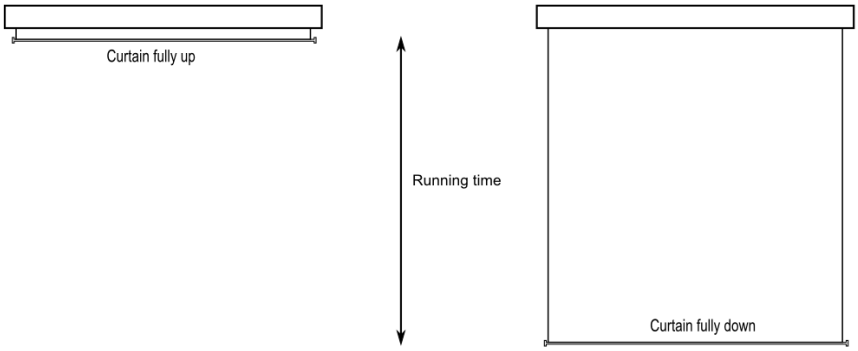
10.9.4 How to set the running time for Motor output

The running and tilting time of the motor have to be set when the rollerblind module is added.
These settings are needed for the correct operation of the function and they should be defined in the wizard relevant to the rollerblind module.
To access this, the user should open the Rollerblind module wizard by clicking on the relevant icons, and then select *Controlled outputs* (see picture below).

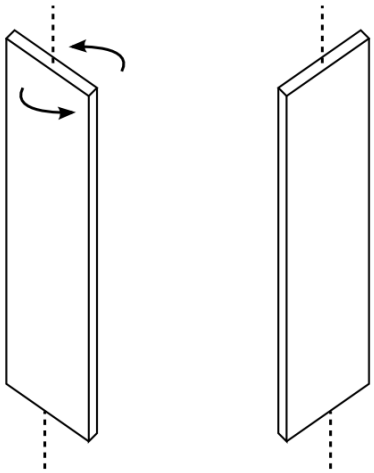


Reverse motor time (seconds):
This is the time in seconds to change the motor direction (this time should be set to 0.5 seconds at least, to avoid damaging the motor).

Running motor time (seconds):
This is the time for the blinds to go from a fully up position to a fully down position.
Starting from a fully wrapped position, the user must use the manual command to completely open the blind. The entire running time must be measured: the accuracy of the time is 0,1 second.



The motor tilt time (seconds):
This is the time for a complete tilting movement (a slats rotation of 180°). Starting from a fully tilted position, the user must use the manual command to tilt the blind into the opposite position.
The whole tilting time must be measured: the accuracy of the time is 0,1 second.



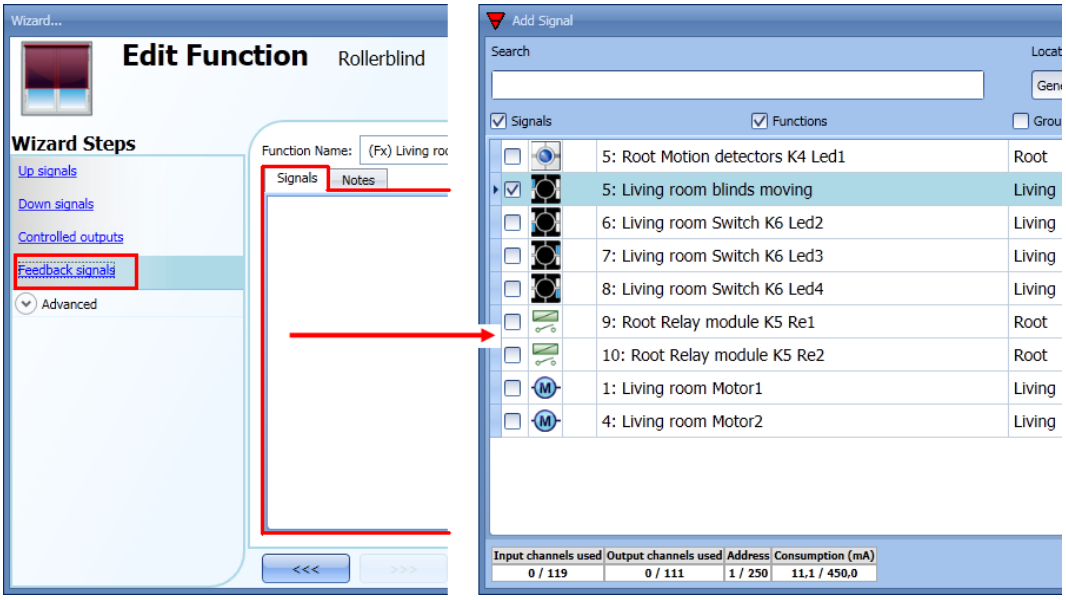
The motor direction:
The user can change the direction of the motor by clicking on the relevant icons.

	Clockwise movement
	Anti clockwise movement

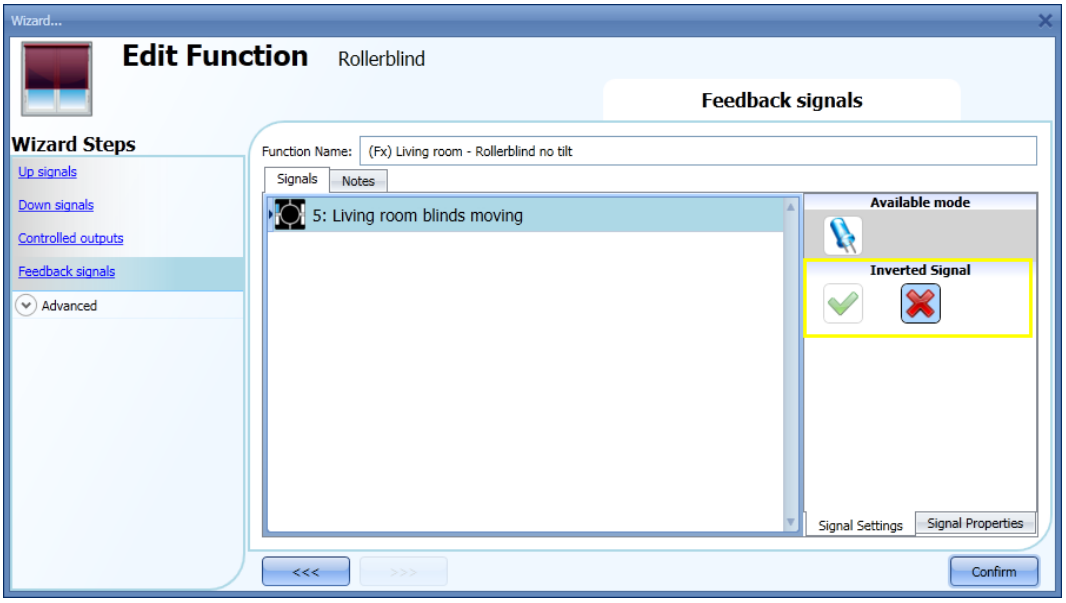
NB. For all those modules with serial numbers higher than MN46 (rev4), the tool manages two different running times: one time for motor up and another one for motor down. Please note that all the modules produced with a lower serial number (that means an earlier firmware revision) do not manage this new feature and an error will be shown by the tool.

10.9.5 How to set a signal to be a feedback of the status of the function

To select the feedback signal that indicates the status of the function, click on *Feedback signals* and then double click on the *Signal window*.
When the motor is moving, the feedback signal will be activated.



The signals available in this window are LEDs and relays.
Up to 50 signals can be added and they will be managed in parallel.
The logic of each feedback signal can be set as standard or inverted (see the yellow rectangle).



10.9.6 How to manage rollerblinds with automations

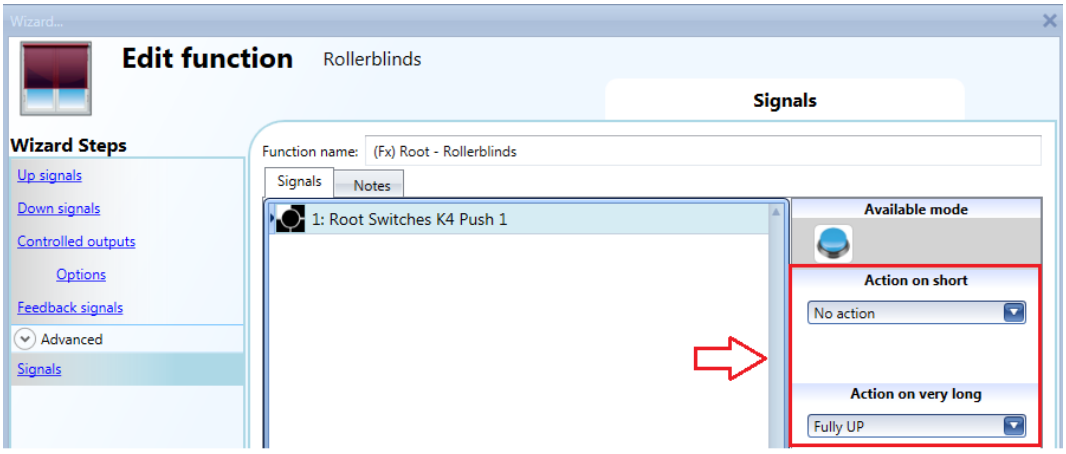
The automatic adjustment up/down of the curtains can be managed by wind sensors (the speed value measured by the sensor can be used to adjust up/down the blinds), by calendar functions (calendar to adjust up/down at pre-defined time intervals), by rain sensors (the curtains can be adjusted according to the weather) or by luxmeters moving blinds up/down according to the levels of ambient light.
All those automation have to be enabled in the Advanced section.

10.9.7 How to customize the push button behaviour

In the *Advanced* settings, the user can customize the behaviour of each push button used to control the blinds.

To add the signal to control the Up/Down movement, the user should click on the relevant section, then double click on the *Signals* window and select the input signal from the list.

By clicking on each added signal, it is possible to select the action to be carried out for the “short press” and “very long press” on the pushbutton.



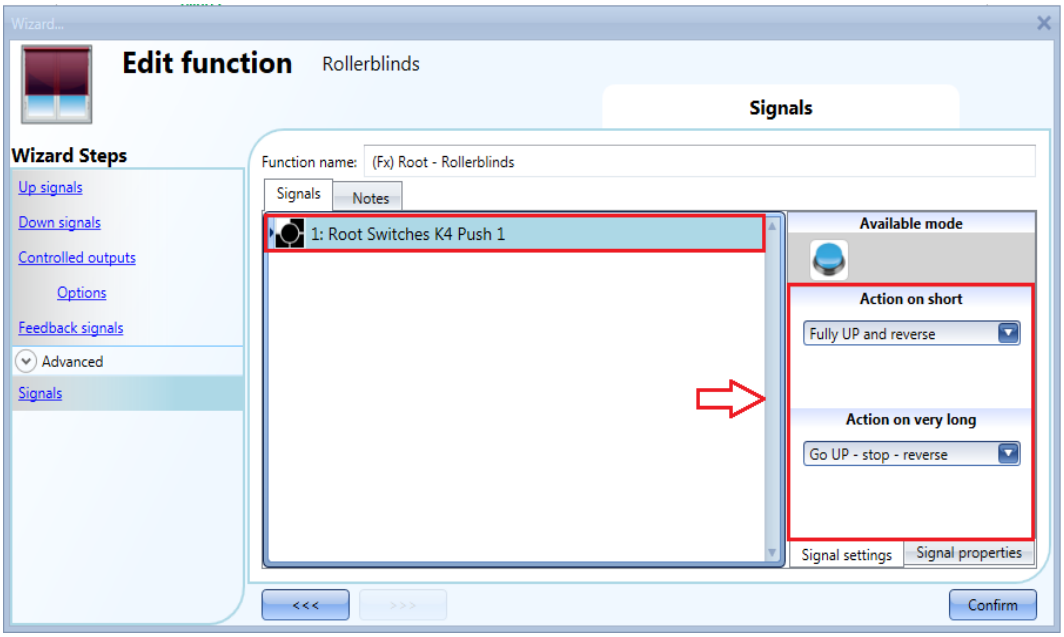
The available actions are listed in the tables below.

Short pressure of the push button	
No action	No action will be carried out if the push button is pressed briefly
Stop	The motor is stopped
Reset UP position	The motor is activated for the whole up running time
Reset DOWN position	The motor is activated for the whole down running time
Fully UP	The motor is activated for the time it needs to go fully up from the current position. E.g.: if the current position is 40%, the motor will be activated for 60% of the up running time.
Fully DOWN	The motor is activated for the time it needs to go fully down from the current position. E.g.: if the current position is 20%, the motor will be activated for 80% of the up running time.
Fully UP and reverse	The motor is activated for the time it needs to go fully up from the current position. When the button is pressed again, the motor is stopped. The next time the push button is pressed, the motor reverses its direction and will go down for the whole down running time.
Move UP step by step	The motor is activated to go up for the selected running percentage each time the pushbutton is pressed. E.g.: if 20% is selected, each time the push button is pressed, the motor will be activated for 20% of the up running time.
Move DOWN step by step	The motor is activated to go down for the selected running percentage each time the pushbutton is pressed. E.g.: if 20% is selected, each time the push button is pressed, the motor will be activated for 20% of the down running time.
Move to xx%	Each time the push button is pressed, the roller blind will be brought to the set (xx%) position.

Long pressure of the push button	
No action	No action will be carried out if the push button is pressed continuously
Go UP - stop	The rollerblind is moved up while the pushbutton is kept pressed. When it is released, the motor is stopped.
Go DOWN - stop	The rollerblind is moved down while the pushbutton is kept pressed. When it is released, the motor is stopped.
Go UP – stop - reverse	The rollerblind is moved up while the pushbutton is kept pressed. When it is released, the motor is stopped. The next time the push button is pressed, the motor reverses its direction.
Fully UP	The motor is activated for the time it needs to go fully up from the current position. E.g.: if the current position is 40%, the motor will be activated for 60% of the up running time.
Fully DOWN	The motor is activated for the time it needs to go fully down from the current position. E.g.: if the current position is 20%, the motor will be activated for 80% of the up running time.
Move to xx%	Each time the push button is kept pressed, the roller blind will be brought to the set (xx%) position.

When the motor is running, any action on the pushbuttons will stop it.

In the picture below, an example is shown where a push button is used to send a fully up/down command with a short click and to adjust the position up/down using the long activation. When the motor is running, a long pressure will stop the motor.

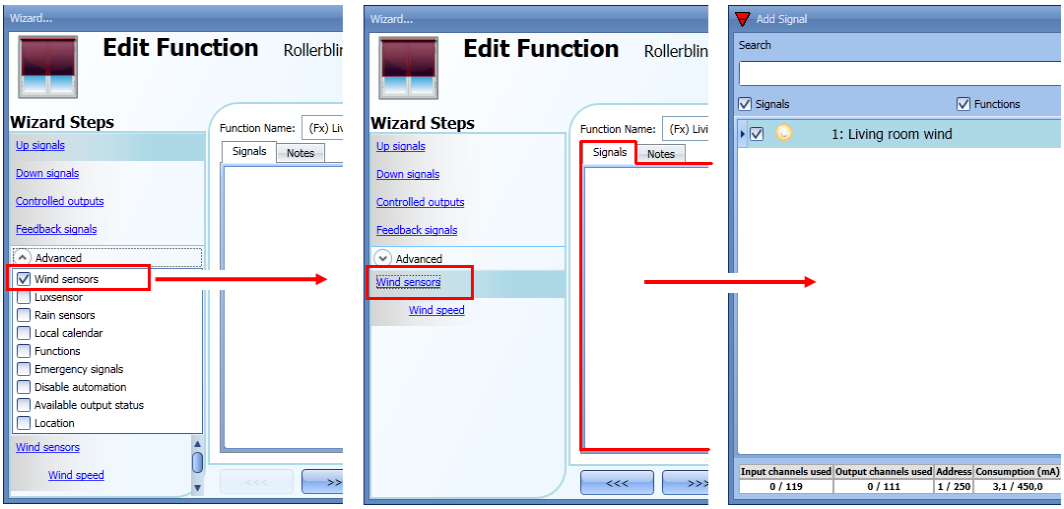


10.9.8 How to use the wind sensor to control blinds

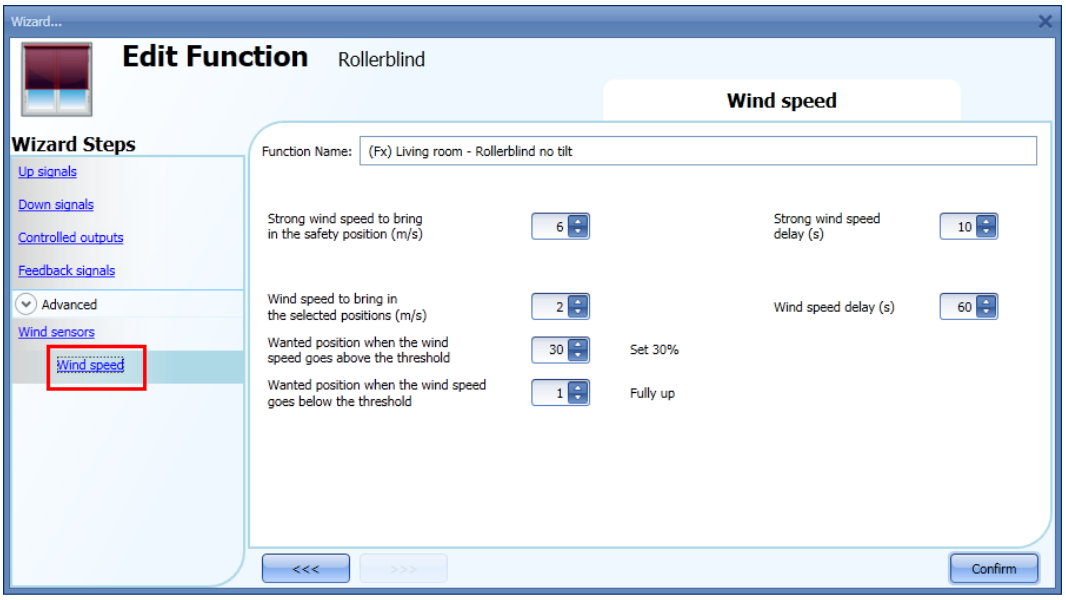
The wind sensor can be used to perform different actions:

1. Adjust the position of the blinds according to the measured wind speed
2. Bring blinds into a safety position if the measured wind speed is very high

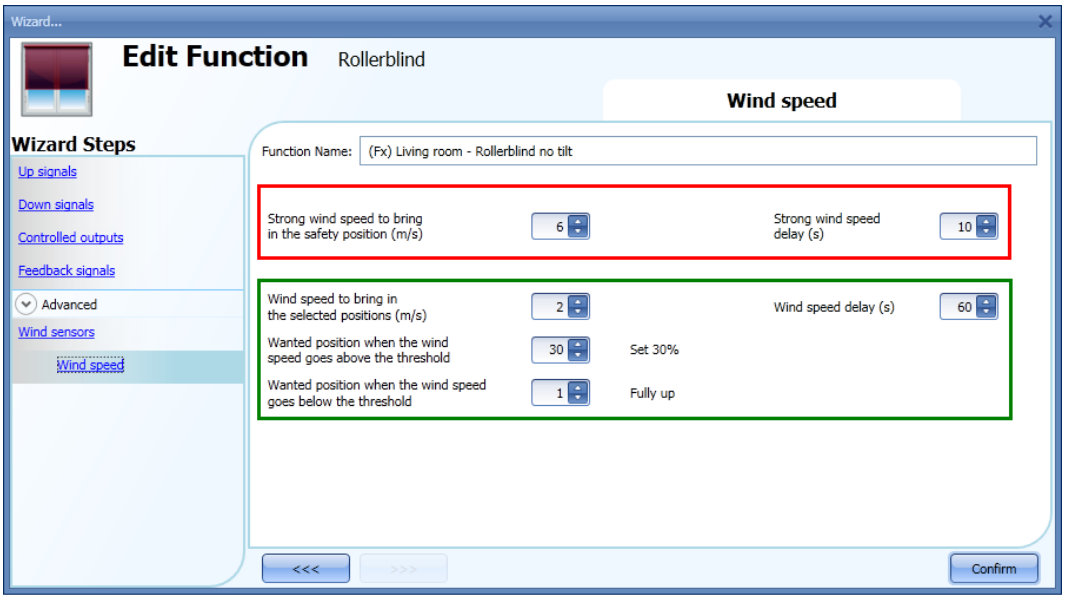
When a wind sensor needs to be used, it must first be enabled in the *Advanced* section.
The *Wind sensors* menu will appear. After selecting it, with a double click on the Signal window, the list of available signals will appear.
Select the required signal/signals and click on *Confirm*. Only one signal can be selected.



Once the wind sensor is added, the user has to set the wind speed limits in the *Wind speed* menu:



The *Wind speed* menu is split into two main fields:



Highlighted in red: the user can select a strong wind threshold value - when the wind speed goes over this threshold, the blinds are automatically brought into the safety position (the safety position is fully up/down according to the settings in the output of the function).
While the strong wind condition is active, the blind stays in the safety position and all other automations are not accepted. The only way to control the blind is to use manual commands.
The user can select a delay time to move the blind to the safety position: the blind will be moved only when the strong wind condition is active for a time longer than the set value. If the delay time is set to 0 seconds, the delay time is disabled.
.

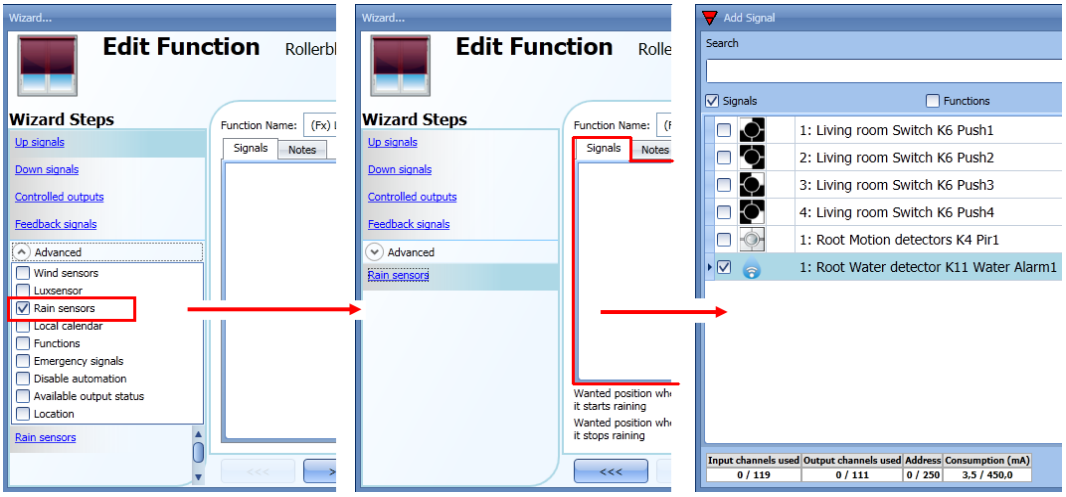
Example 1
In the picture above, when the wind speed goes over 6 m/s for more than 10 seconds, the blind is moved to the safety position. In order to avoid damage to the blinds, we suggest setting a small delay value (max 30 seconds).

Highlighted in green: the user can set another wind threshold. When the wind speed goes over/under this value, it is possible to select different actions for the blind. The user can also select a delay time for each action

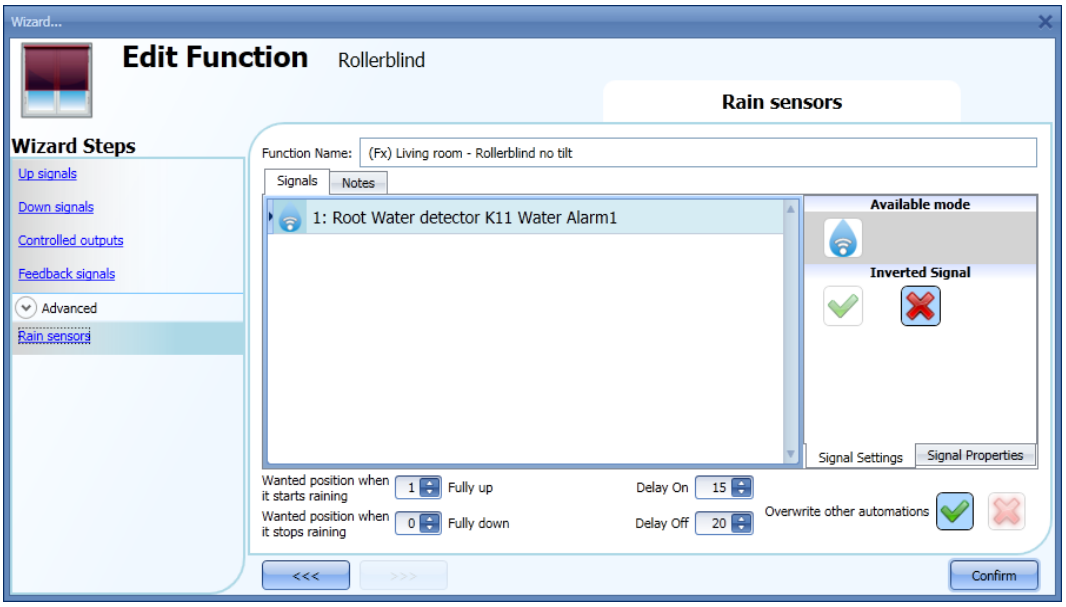
Example 2
In the picture above, when the wind speed goes over 2 m/s for more than 60 seconds, the blind is moved to 30% of the running time. When the wind speed goes below 1 m/s for more than 60 seconds, the blind is moved to the fully UP position
In order to avoid continuous movement of the blind, we suggest setting a minimum delay value of 60 seconds.

10.9.9 How to use the rain sensor to control blinds

If the user needs to control the curtains when it is raining, the rain sensor has to be selected in the *Advanced* section. To add the rain sensor signal, the user has to select the relevant field, then double click on the *Signal* window and select the input signal from the list (see picture below).



The input signal can be a rain sensor or a generic pushbutton/switch. Up to 50 signals can be added. Once the rain sensor is added, the user has to select the action to be carried out when one of the sensors added detects water:



Wanted position when it starts raining:
The user can select a value for the position of the blind when the sensor detects water.

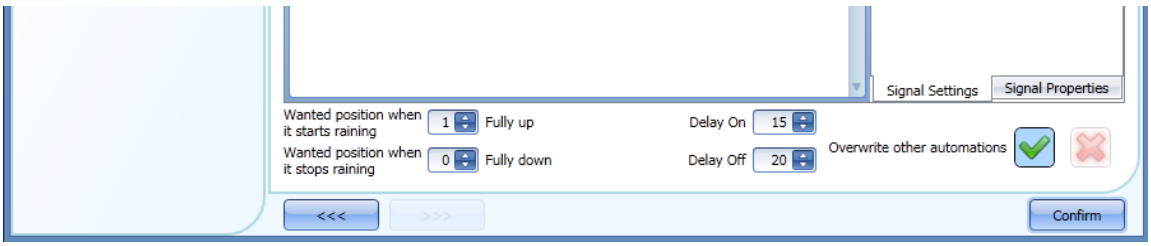
Delay On:
A delay time for the activation of the rain condition can be set (if the value is set to 0 the delay timer is not enabled).

Wanted position when it stops raining:
The user can select a value for the position of the blind when the sensor stops detecting water.

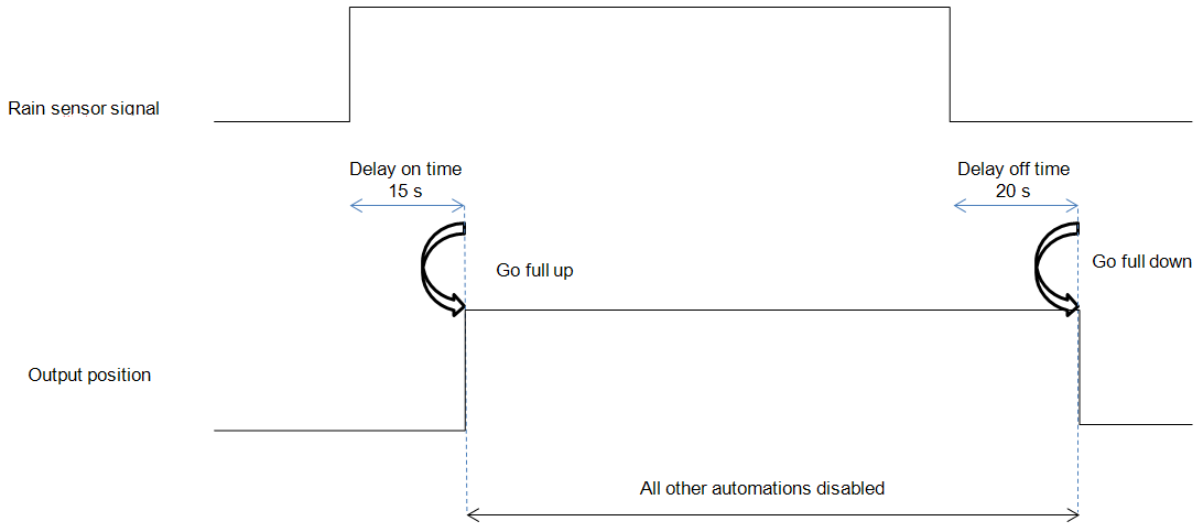
Delay off:
A delay time for the deactivation of the rain condition can be set (if the value is set to 0, the delay timer is not enabled).

Overwrite other automations:
When the green V is checked, until the rain sensor detects water, all the other events coming from other automations, are not considered by the function (the rain condition overwrite other automations). If the red cross is checked, the rain condition doesn't overwrite other automations.

In the next pictures a simple example of how to configure a rain sensor is shown:
If the rain sensor has been detecting water for 15 seconds, the blind is moved to a fully up position. When the rain signal disappears and the delay off timer expires, the blind is moved to a fully down position.

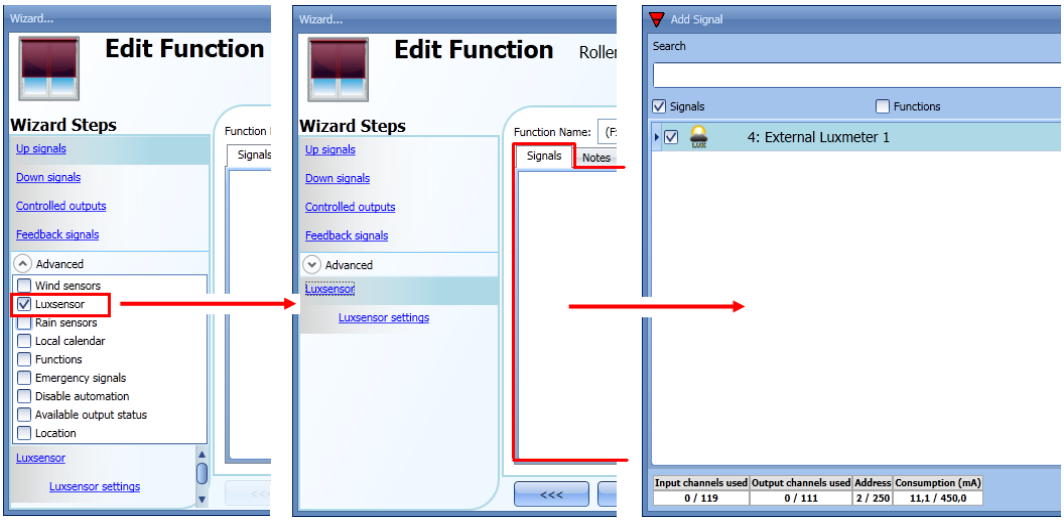


If the option *Overwrite other automations* is selected, until the rain signal is high, the other automations are disabled.

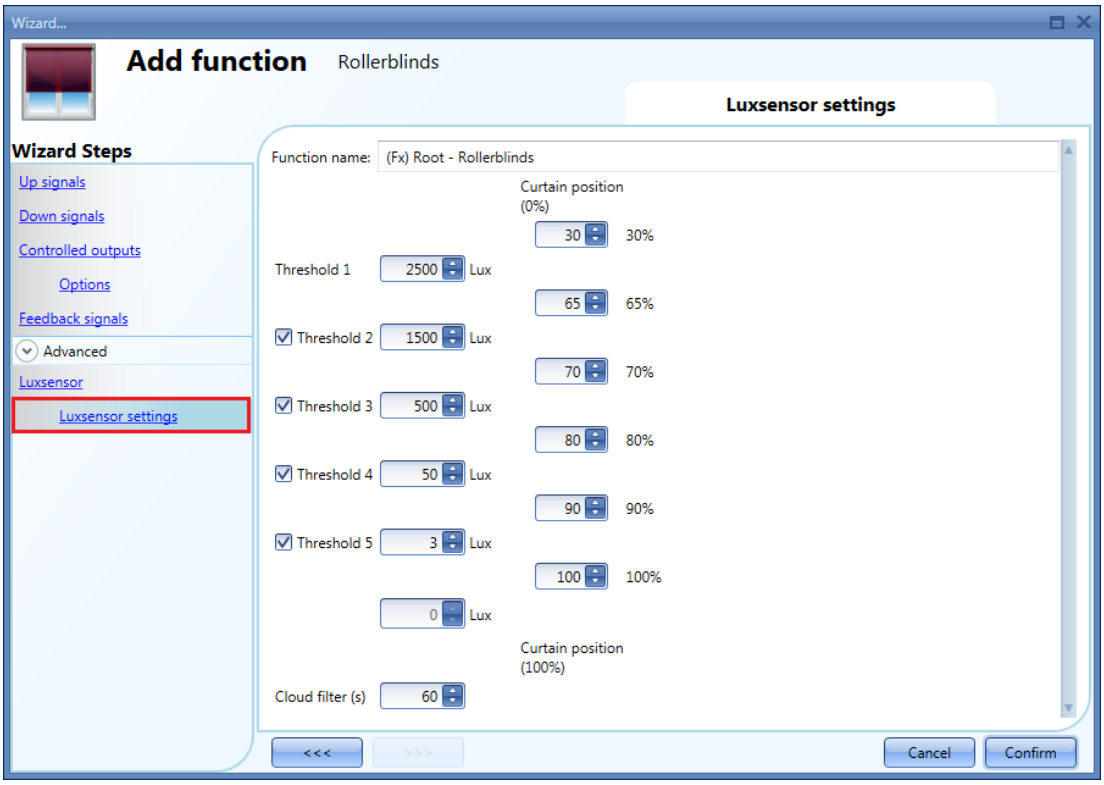


10.9.10 How to control the blind according to the daylight

The blind position can be managed in an automatic way by using lux meters: the user can define up to 5 different light levels to make the smart-house system move the blinds up/down.
To select the lux sensor, the user should click on *Luxsensor* and then double click on the *Signal window*. Once the output window is opened, select the relevant signal from the list.



The user can add up to 10 lux sensors: if more lux signals are linked to the function, the average value is calculated and used.
Once the lux sensor is added, the user can change the threshold values and the action that the blind has to perform when these values are reached.

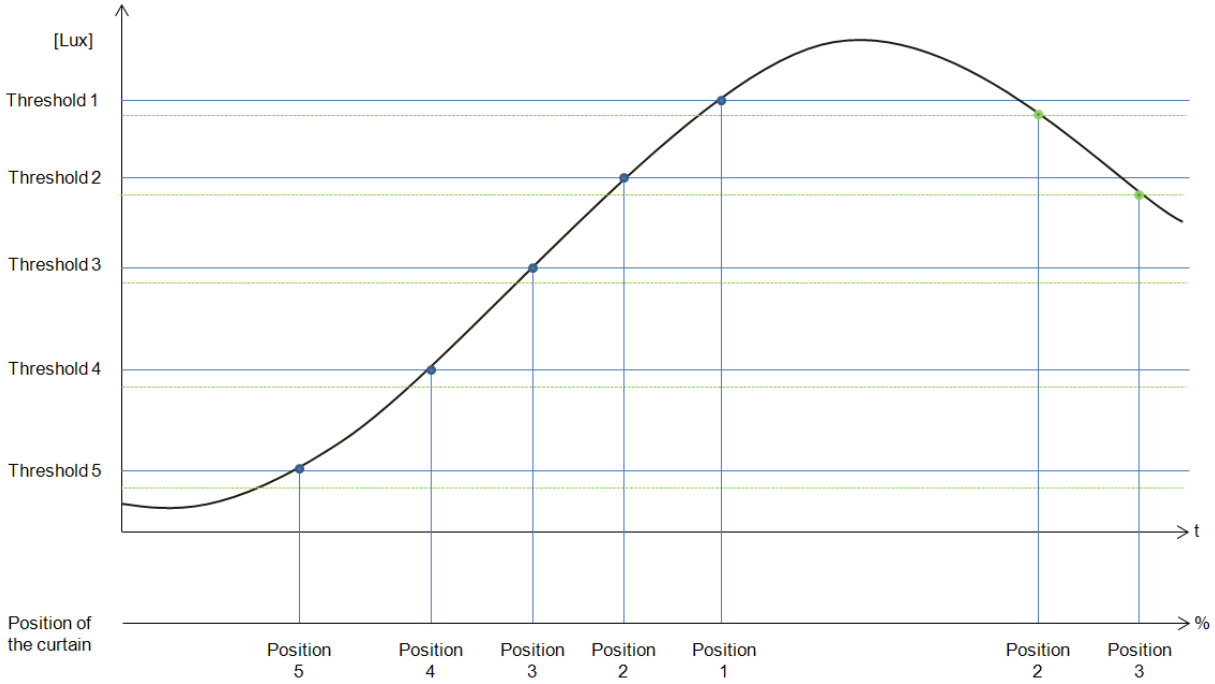


The up/down movements and the tilting start when the thresholds are surpassed, so if the blind position is changed manually by means of a light switch or automatically e.g. by the scheduler, the automation of

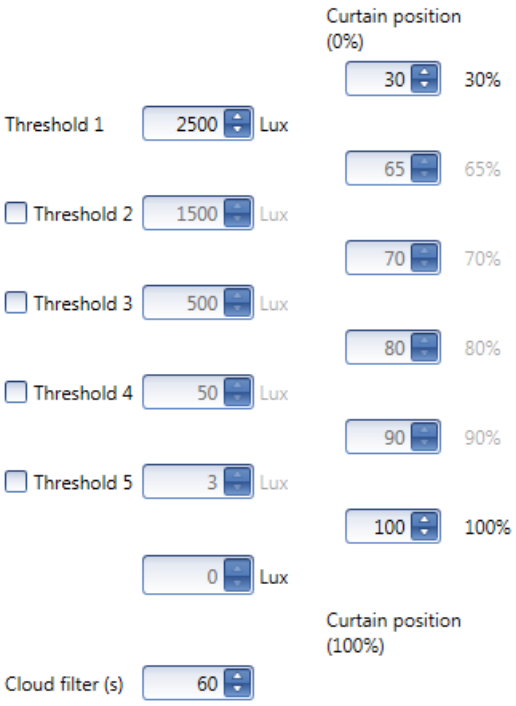
the lux sensor will not change it unless the thresholds are surpassed.
The hysteresis value is fixed at 10% and it is used for all the thresholds.
When the light level is increasing, the threshold values are used: when the light level is decreasing the following value is used:

value= threshold value – hysteresis.

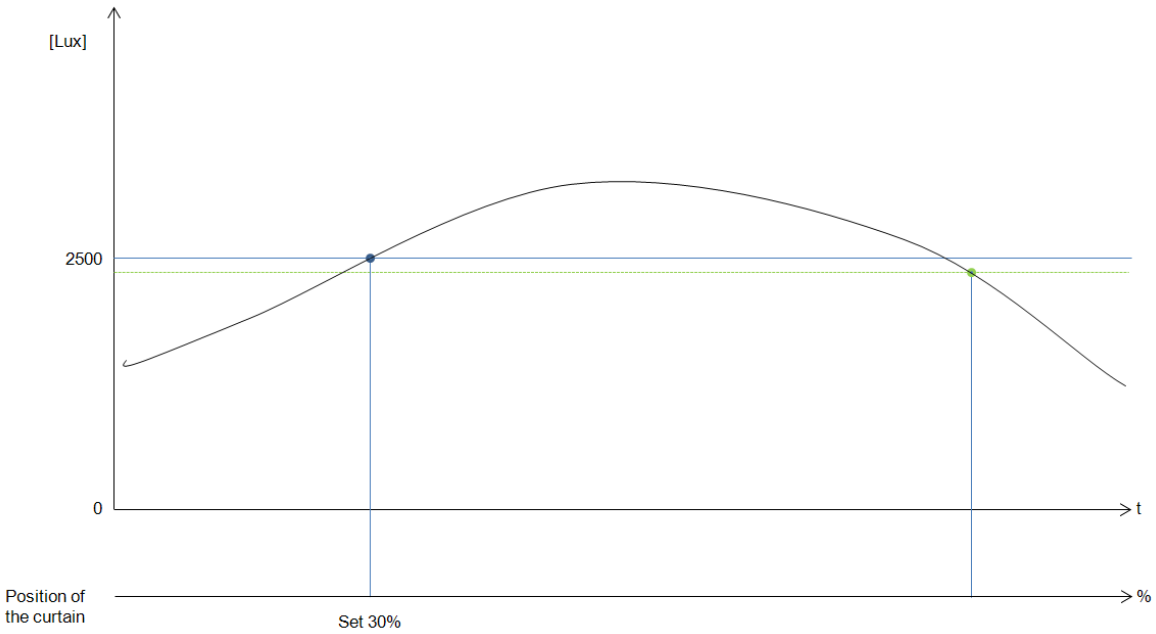
The user can also set a cloud filter to avoid the up/down movements if a cloud shields the sun for a short time. We suggest setting the time for at least 60 seconds; if the time is set to 0 the filter is disabled.
The picture below shows how the blind position is changed according to the light level. At least one threshold must be used with the lux sensor.



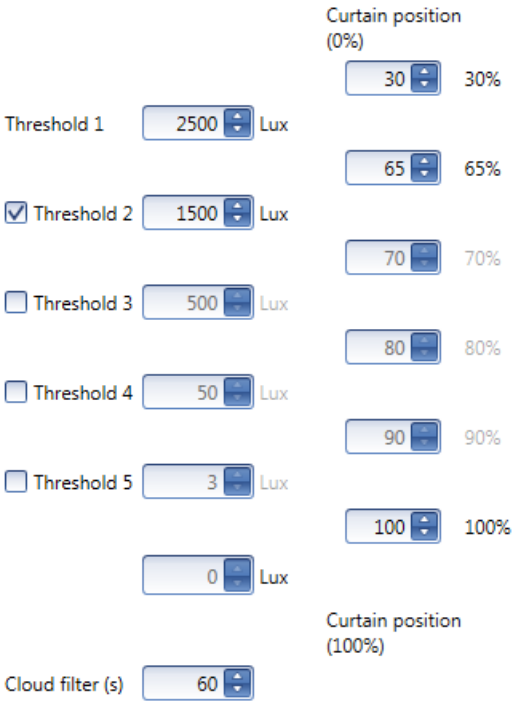
Example 1:
In the first example only one threshold is used, the cloud filter set is 60 seconds. When the light reaches threshold, the blind is moved to 30% of the running.



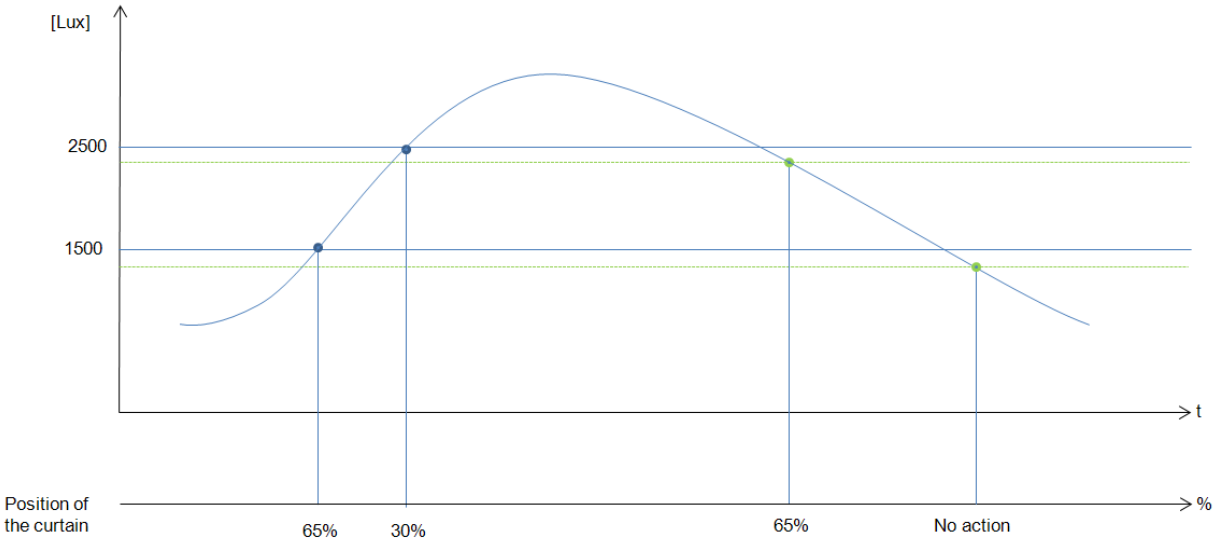
In the picture below, the behaviour of the output is shown according to the light level.



Example 2:
In the second example, threshold 1 is set to 2500 Lux and threshold 2 is set to 1500 Lux: the cloud filter is 60 seconds. When the light goes over 1500 Lux, the blind moves for the 65% of the running time, when the light goes over 2500 Lux, it moves for the 30% of the running time.



In the picture below, the behaviour of the output is shown according to the light level.

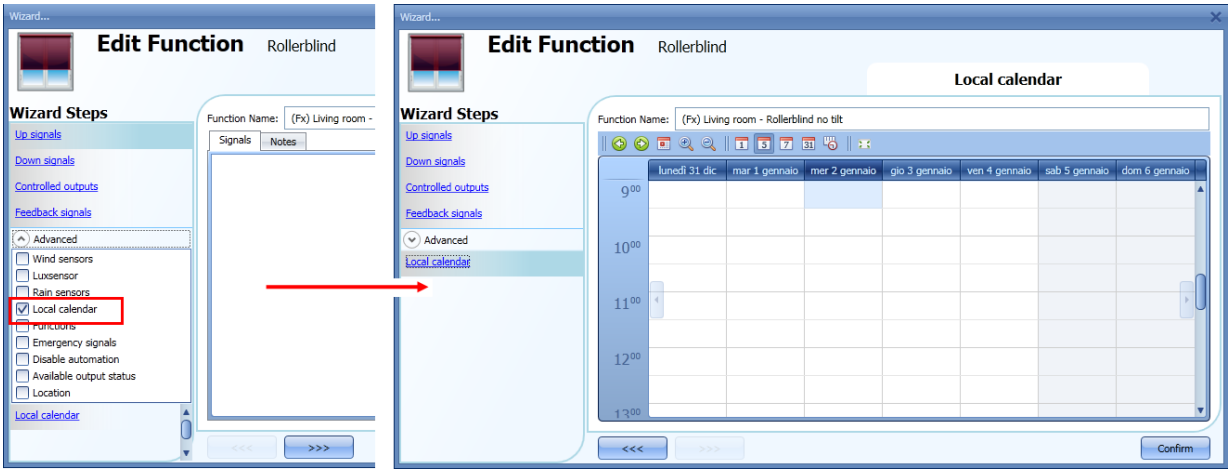


10.9.11 How to move the blinds up/down and tilt them with the calendar

There are two ways of managing the *Rollerblinds* function with the calendar: by setting the “local” calendar inside the function or using a “global” calendar function.

10.9.11.1 The local calendar function

To enable this, the relevant menu has to be enabled in the Advanced section..



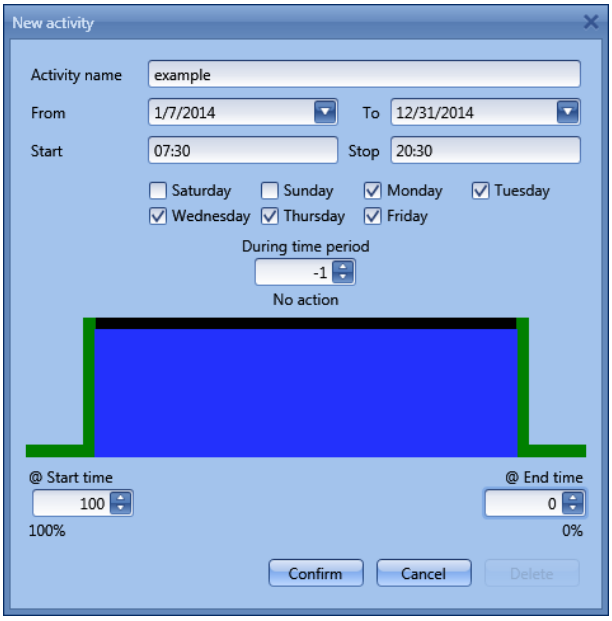
The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day.
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally.
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

Calendar activities

Once the user has selected his preferred visualization, to enter a time band he has to double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From*:* the start date for the calendar activity.

To*:* the end date of the calendar activity.

Start*:* The time of the start of the activity.

Stop: The time of the end of the activity.

@ ***start time up/down movement***: in this field the user can select the action of the function at the start time.
The user can select the following actions:

- No action (-1)
- Set the position of the blind at a fixed value between 1% and 100% (1-100)

@ ***end time***: in this field the user can select the action of the function at the stop time.
The user can select the following actions:

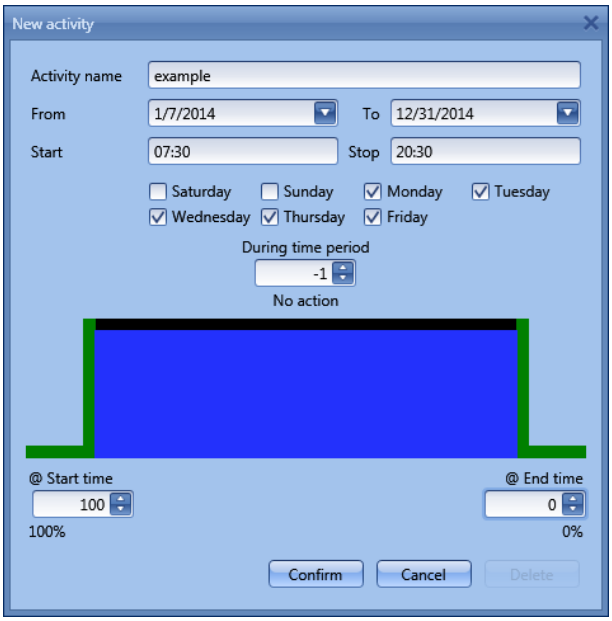
- No action (-1)
- Set the position of the blind at a fixed value between 1% and 100% (1-100)

Days: The user has to select the days of the week for the activity.

If ***Level activity*** is selected, the user can disable all the automations of the function (rain sensors, wind sensors, luxmeters and schedulers).

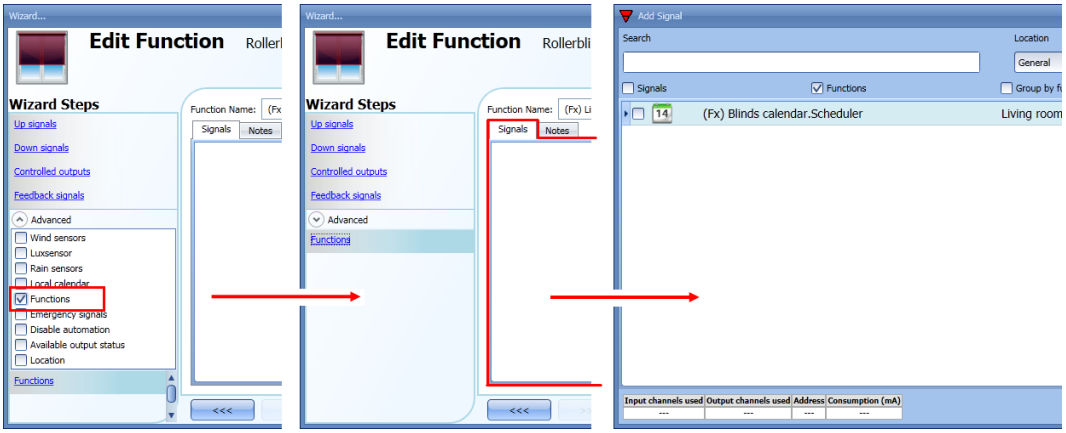
- No action (-1)
- Disable automations (1)

In the example below, the calendar is set to work all the year (from 1st January to 31st December) . The working days are: Monday, Tuesday, Wednesday, Thursday and Friday (the calendar does not work on Saturday and Sunday).
The start time is 7:30 and the stop time is 20:30.
At 7:30 the blind starts moving up. At 20:30 the blind starts moving down.



10.9.11.2 The global calendar function

Before using a global calendar it has to be defined as a function (See how to set a global calendar).
The field function has to be selected in the *Advanced* menu.
By clicking on *Functions* and then double clicking in the *Signal* window, the *Add signal* window will appear. Select the required calendar function.

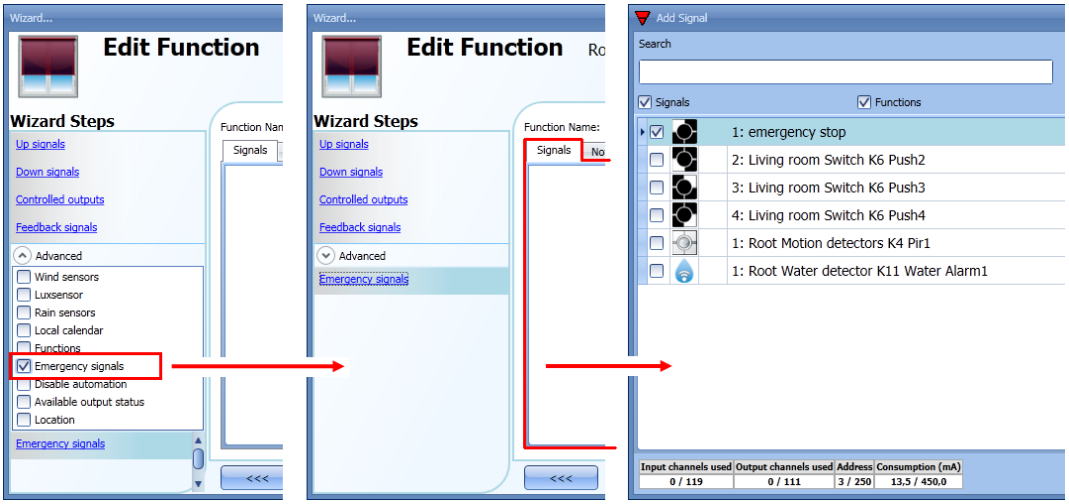


The behaviour of the calendar will be the same as that of the local calendar.

An example of this automation is a global calendar to move all the blinds down at ten o'clock in the evening.

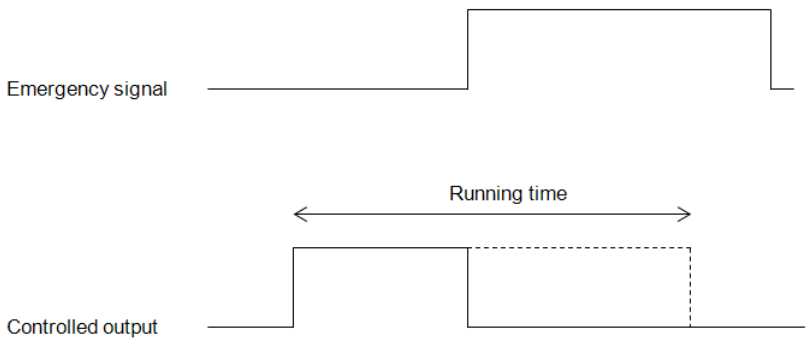
10.9.12 How to use the emergency signals

In some applications, for the safety of the user, a pushbutton is required to stop the motor at any time.
The *Emergency signal* field allows the user to stop the motor when the signal is active. The roller blind module does not accept any manual or automatic command until the emergency signal is active.
To add an emergency signal, the user should select the relevant section in the advanced menu, then double click on the *Signal* window and select the input signal from the list (see picture below).



When the signal is added, the inverted logic can be used (see the yellow rectangle in the picture below).

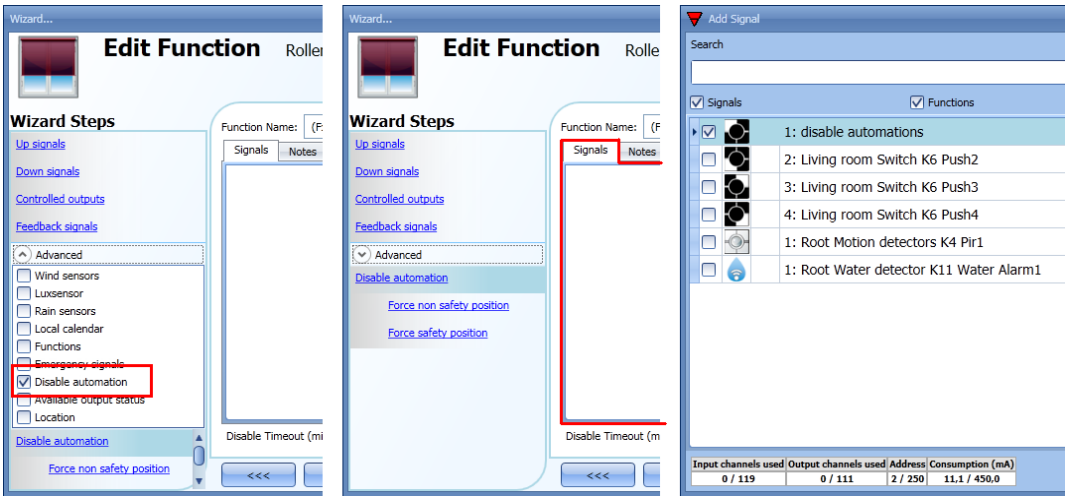
In the example below the motor is stopped immediately when the emergency signal is activated.



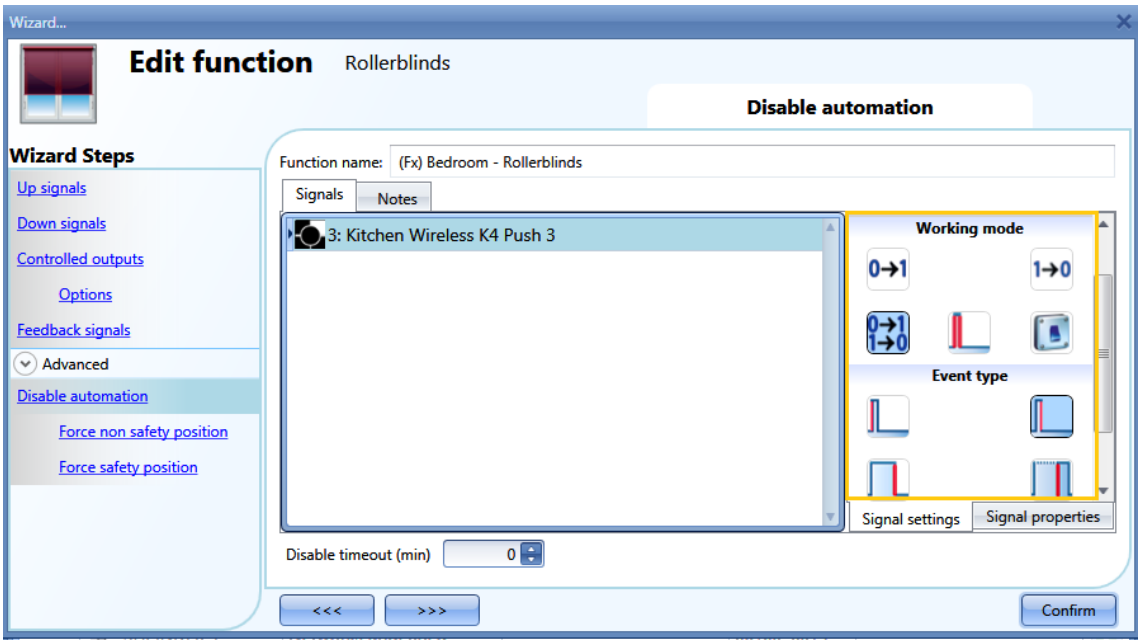
10.9.13 How to stop the automation

Sometimes it is necessary to stop all the automations to service the blinds or simply to clean the windows.

To add a disable automation signal, the user has to select the relevant section in the advanced menu, then double click on the *Signals* window and select the input signal from the list (see picture below).



When the signal is added, the user should select the working mode and the type of the event:



If a push button type is selected, the way of working marked with the red rectangle, has to be done following the table below.

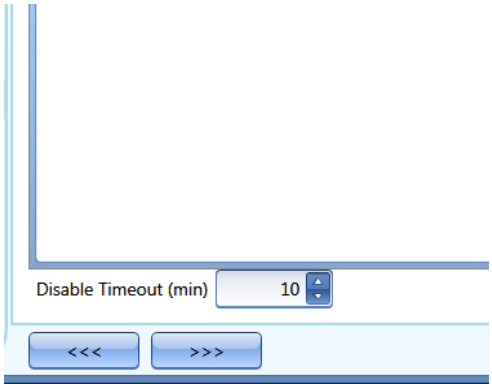
	Event type			
Working mode				
0→1	As soon as the push button is pressed, the automation is disabled.	After a short press (less than 1 second), the automation is disabled at the release of the push button.	After a long press, the automation is disabled at the release of the push button.	After a very long press, the automation is disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automation is enabled again.	After a short press (less than 1 second), the automation is enabled again at the release of the push button.	After a long press, the automation is enabled again at the release of the push button.	After a very long press, the automation is enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the automation will be enabled/disabled in toggle mode.	After a short press (less than 1 second), the automation will be enabled/disabled in toggle mode at the release of the push button.	After a long press, the automation will be enabled/disabled in toggle mode at the release of the push button.	After a very long press, the automation will be enabled/disabled in toggle mode at the release of the push button.
	The automation will be enabled/disabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

	Event type	
Working mode	**Signal activated**	**Signal deactivated**
1→0	The automation is enabled again	No action
0→1	The automation is disabled	No action
0→1 1→0	The automation is disabled/enabled in toggle mode	No action
	The automation is disabled/enabled in toggle mode	The automation is disabled/enabled in toggle mode
	The automation is disabled	The automation is enabled

The user might need to set a time after which the automation is enabled again even if the selected signal is still active: to achieve this, the Disable timeout field has to be set.

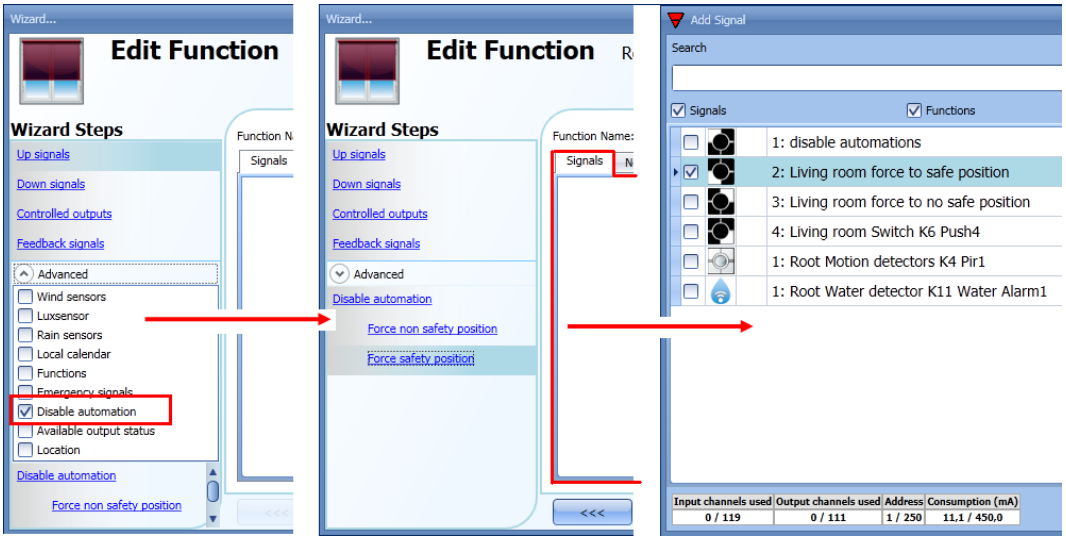
To access the stop automation functionalities, they must be enabled in the Advanced section field.
If the user needs to be sure that the disabling of the automation is automatically removed, the *disable automatism timer* has to be set.



The timer starts each time the disable status becomes activated: the disable status is automatically deactivated as soon as it has elapsed.
In the figure above a disable timeout of 10 minutes is set. The maximum delay is 59minutes.

10.9.14 How to force the safety condition

If the user wants to force the safety position regardless of all other signals used in the function, the *Force safety position* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force safety position*, then double click on the signal window and select the right signal to use.

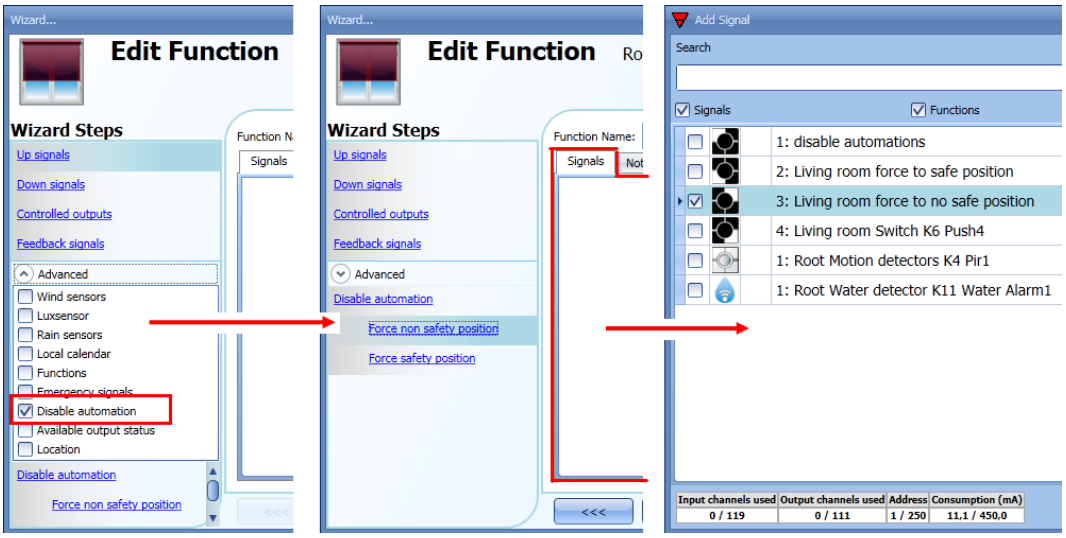


Each signal used in the *Force safety position* window works as a level signal: until the signal is active, the blind is forced into the safety position.
When the *force to safety position* signal is activated and the *force to no safety position* signal is activated, the force to safety position signal has the priority.

10.9.15 How to force the non-safety condition

If the user wants to force the non-safety position regardless of all other signals used in the function, the *Force non safety position* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force non safety position*, then double click on the signal window and select the right signal to use.

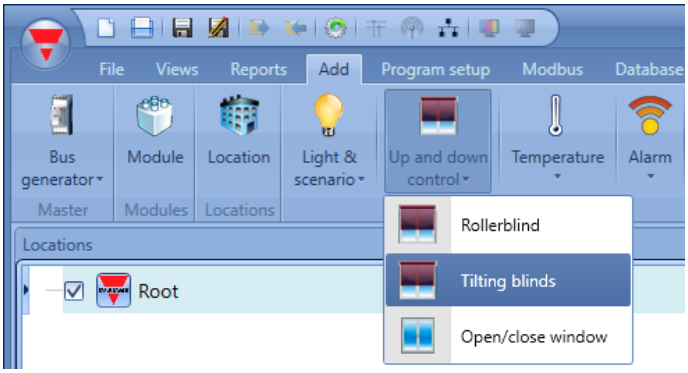
.



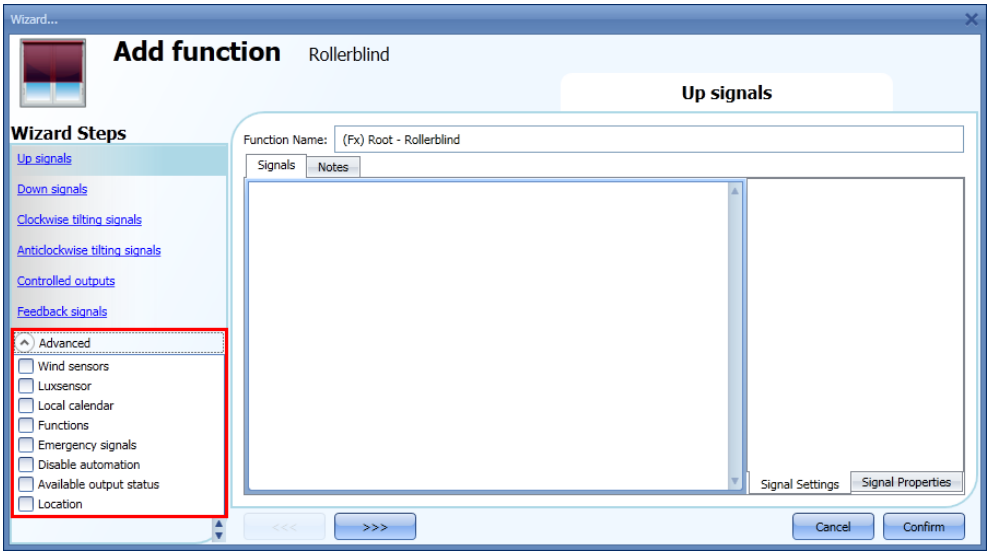
Each signal used in the *Force non-safety position* window works as a level signal. Until the signal is active, the blind is forced to the non-safety position.
When the *Force to safety position* signal is activated and the force to no safety position signal is activated, the *force to safety position* signal has the priority.

10.10 How to set a Rollerblind function with tilting slats

This function allows the operator to manage the motor to control tilting slats.
The user can either configure a basic function to move blinds up and down and tilt the slats clockwise/anticlockwise or implement an automated system by programming the relevant objects of the function.
To set up this kind of function, the user should select *Up and down control* from the Add menu, then select *Tilting slats* (see picture below). The new function will be added into the selected location.



This function manages one or more motor outputs controlled by one or more input commands. The command might be a physical signal like a pushbutton, a switch, a function or a remote command (Webserver, sms, Modbus TCP/IP).
The automation of the tilting slats can be managed by accessing the *Advanced* section, where the user can select different kinds of automation: wind sensors, rain sensors, lux sensors, calendar.

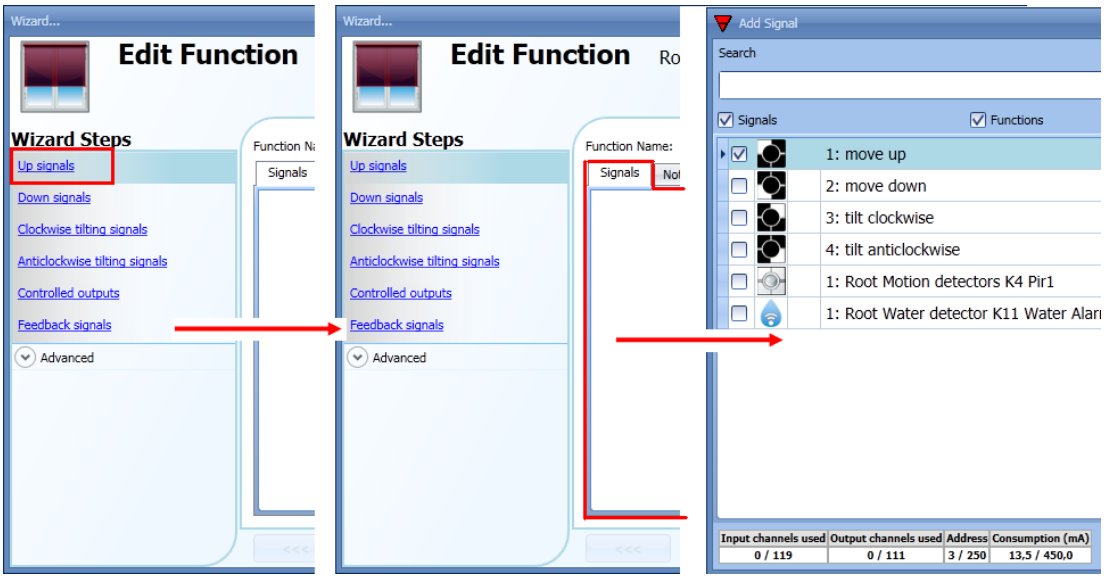


N.B. For the function to operate correctly, we suggest giving an up/down reset command every time the system is started.
The reset command can be given by a pushbutton, via the webserver with administrator access or via the UWP 3.0 Tool by using the live signals mode.
When the reset command is sent and the motor is running, the user should wait for the running time to be completed (the running time is that set in the rollerblind module wizard).
For safety reasons, the reset command can be stopped anytime by another command; should this happen, another reset command should be sent to ensure the correct behaviour of the automation starting from a known position.

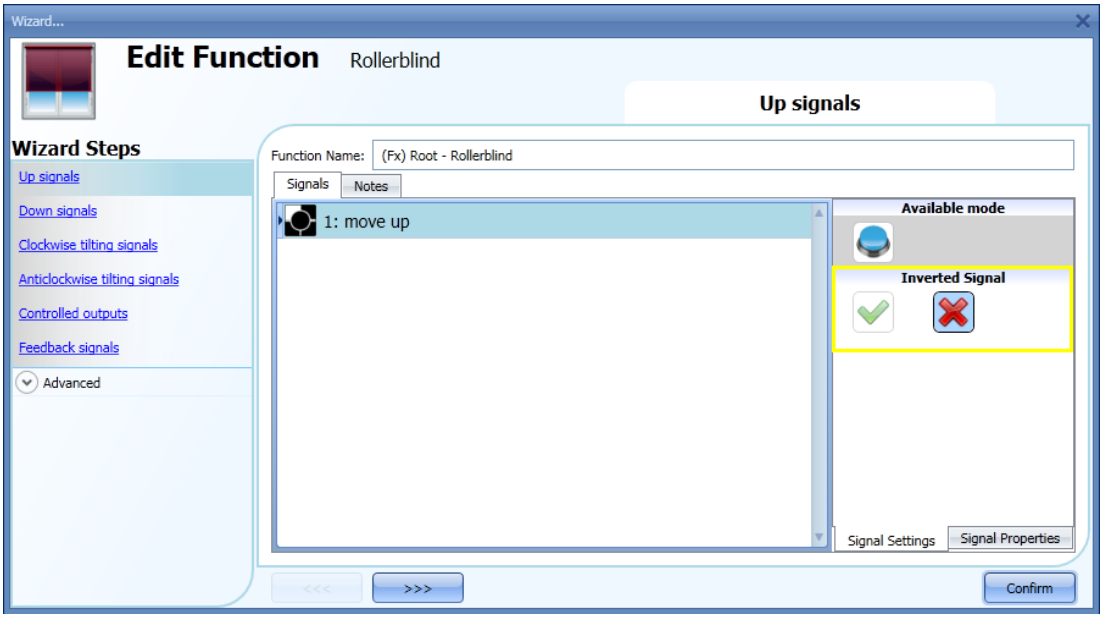
10.10.1 How to move blinds UP/DOWN using a manual input

The user should add the input signals to control the Up/Down movement of the blinds.
To add the Up/down signal, the user should click on the relevant section, then double click on the *Signal* window and select the input signal from the list (see picture below).
The input signal may be a pushbutton or a switch.

.

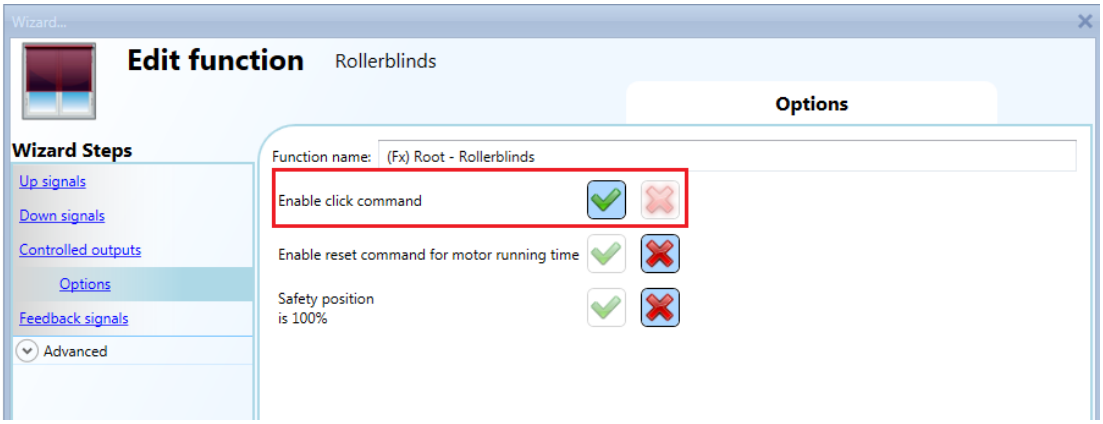


Once the UP/DOWN signal is entered, the user can also decide to use the inverted logic of the signal (see yellow rectangle in the picture below).



All the selected signals in the Up window work in parallel (they are managed in an OR logic). According to how long the pushbutton is pressed, the system will do different things:

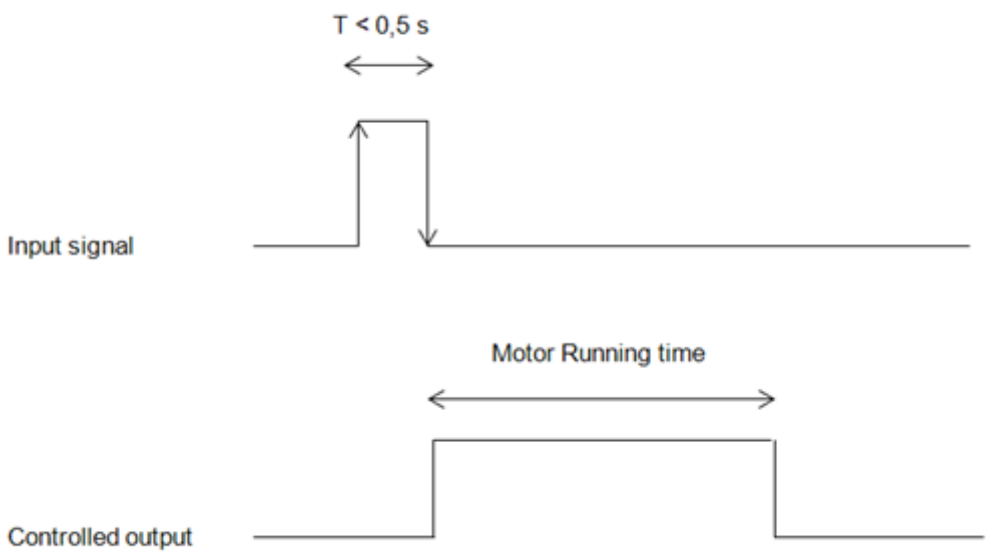
1) A short click action (<0,5sec) on the pushbutton, if enabled, generates a “fully Up/Down” command. The click action is enabled in the *Controlled output options* (see picture below).



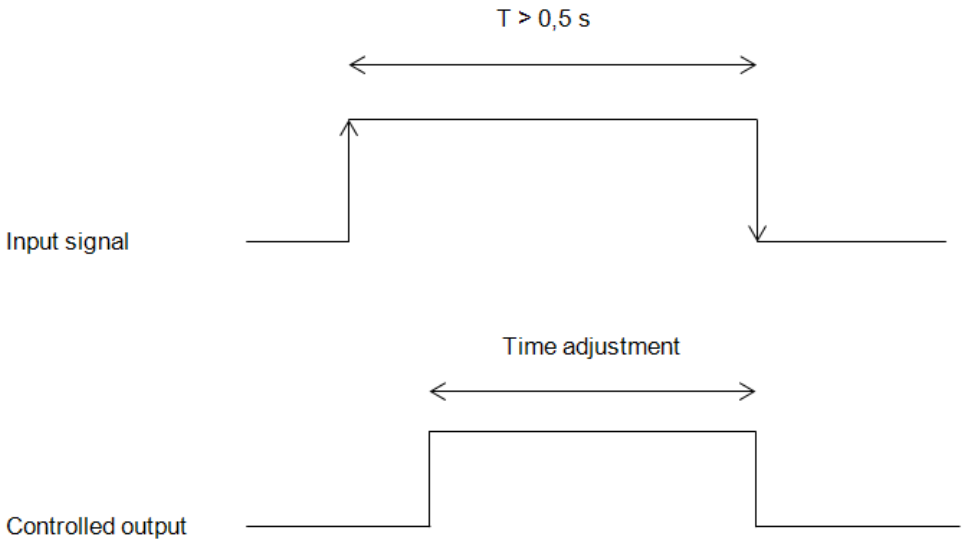
2) A long press (> 0,5 sec) activates the motor output until the input signal is ON (the blind goes up/down until the user keeps the pushbutton pressed or the selected signal is active).
This functionality doesn’t need to be enabled.

In the figure below, two examples of activation of the input signal are shown.

Example 1: short press with the “click command” option enabled

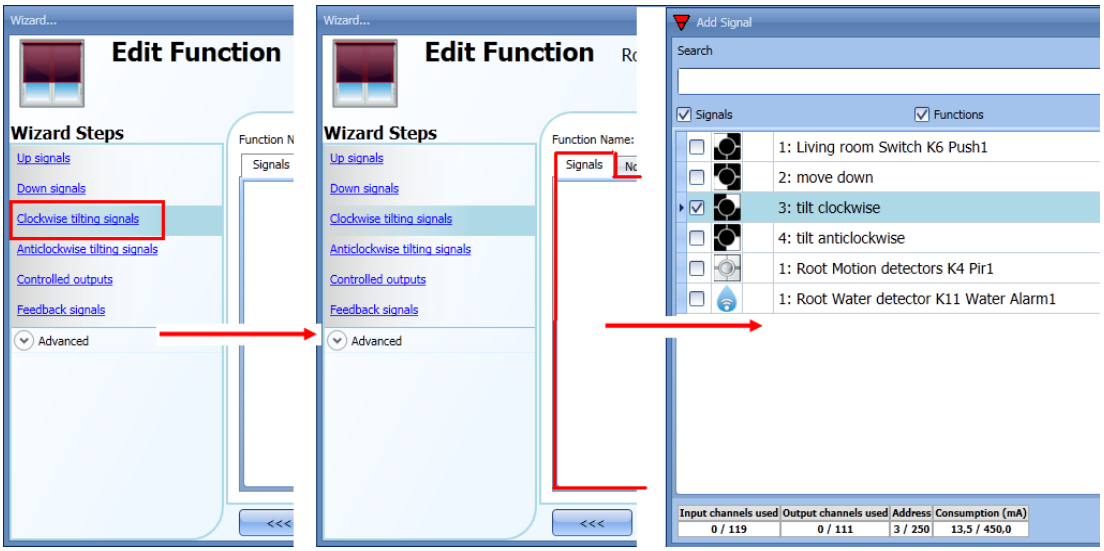


Example 2: very long press for the “Up/Down” adjustment

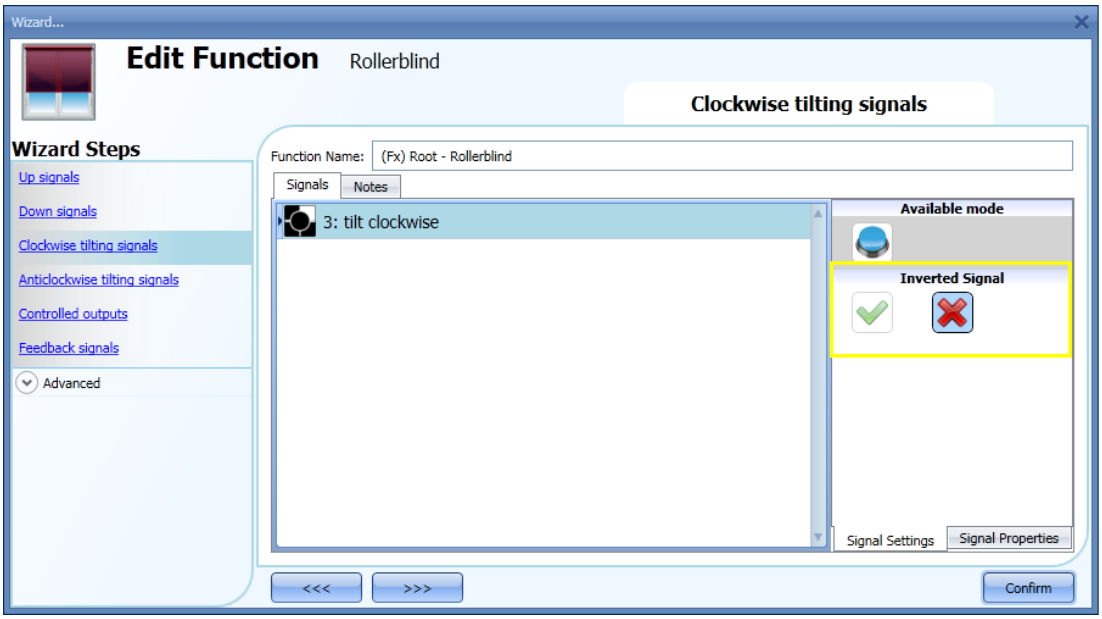


10.10.2 How to tilt blinds clockwise/anticlockwise using a manual input

First the user has to add the input signals to tilt the blinds clockwise/anticlockwise.
To do this, the user has to select the relevant section, then double click on the *Signal* window and select the input signal from the list (see picture below).
The input signal can be a pushbutton or a switch.



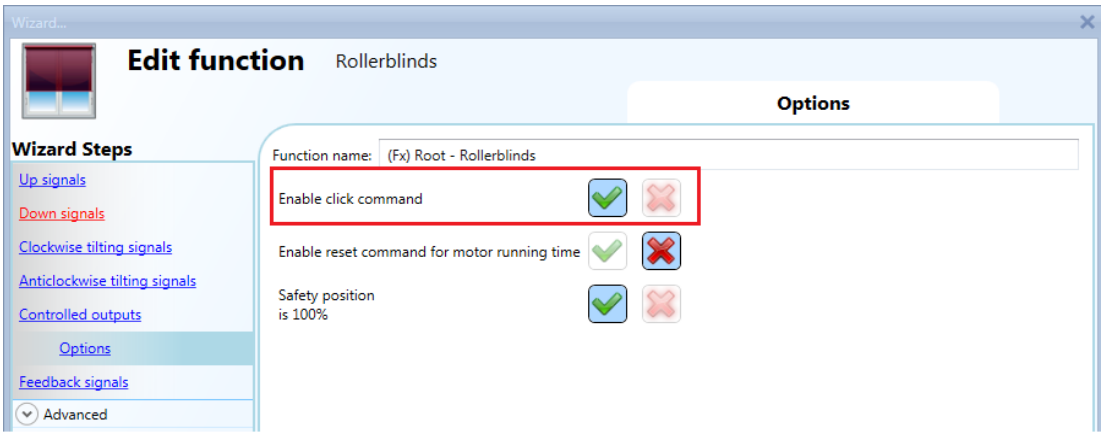
Once the signal is entered, the user can also choose to use the inverted logic of the signal (see yellow rectangle in the picture below).



All the selected signals in the *Clockwise/Anticlockwise tilting signals* window work in parallel (they are managed in an OR logic).
According to how long the pushbutton is pressed, the system will do different things:

1) A short click action (<0,5sec) on the pushbutton, if enabled, generates a “Fully tilted” command.
The click action is enabled in the *Controlled output options* (the reset command options must be

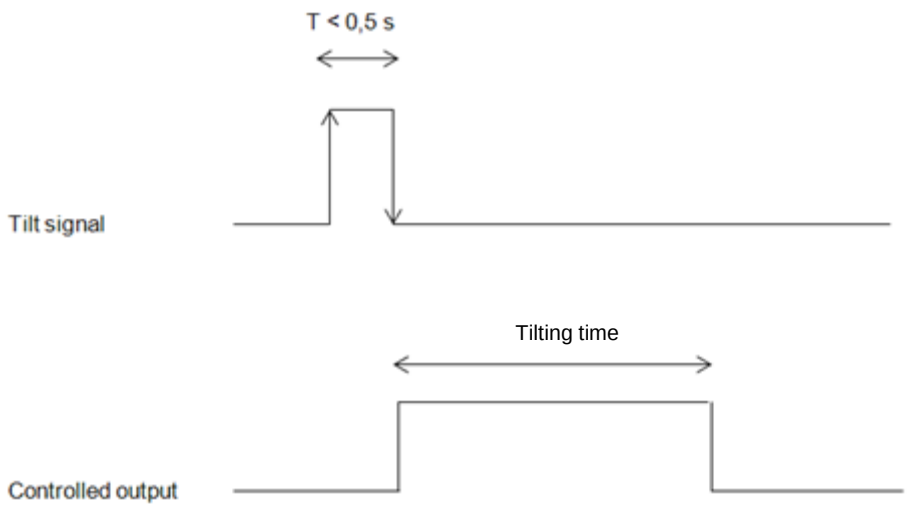
disabled, otherwise the click action will reset the blind up/down…see picture below).



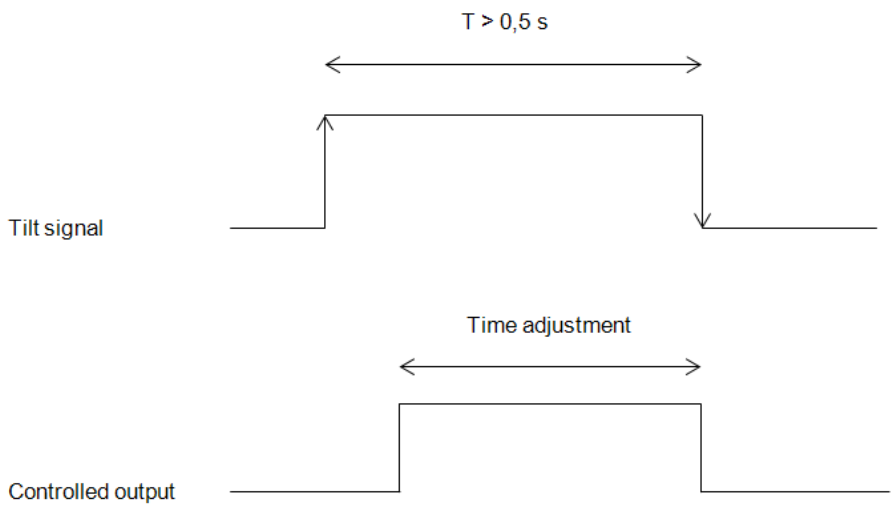
2) A **very** long press (> 0,5 sec) activates the motor output until the input signal is ON (the blind is tilted clockwise/anticlockwise until the user keeps the pushbutton pressed or the selected signal is active). This functionality doesn't need to be enabled.

In the figure below, two examples of activation of the input signal are shown.

Example 1: click action for the full tilting

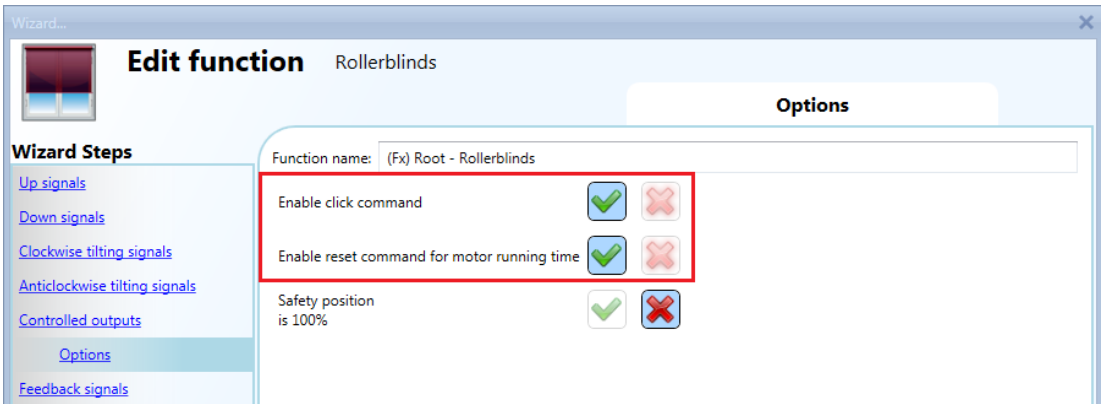


Example 2: very long action for the tilting adjustment



10.10.3 How to set the reset command

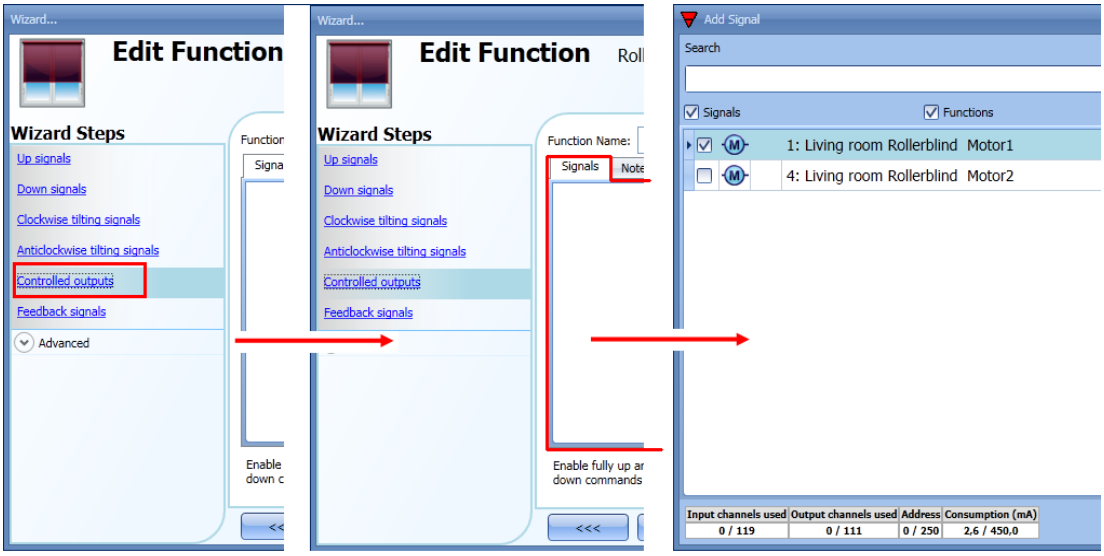
The reset command is used to reset the position of the roller blinds after a system start-up or after the downloading of a new project on the UWP 3.0.
This command is activated by a short click that makes the rollerblind go fully up/down (according to the selected direction) and resets the position to 0/100%.



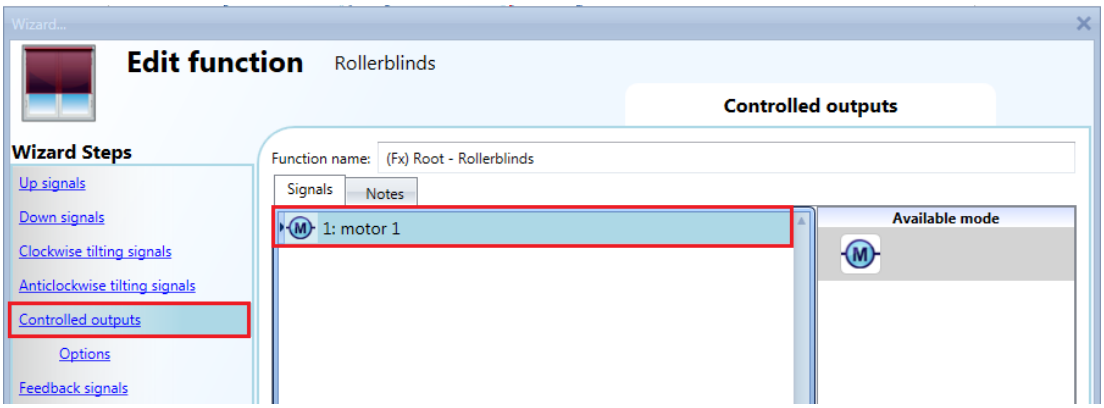
The reset command is also available via the webserver (when accessed as administrator) and via the UWP 3.0 Tool in the live signals mode. When the command is sent by using the webserver or the Tool, a window with a warning message is shown to advise the user not to stop it.

10.10.4 How to select and configure the output

To select the output signal that is controlled by the *Tilting blinds* function, the user has to click on *Controlled output* and then double click on the Signal window. Once the output window is opened, select the relevant motor output from the list.

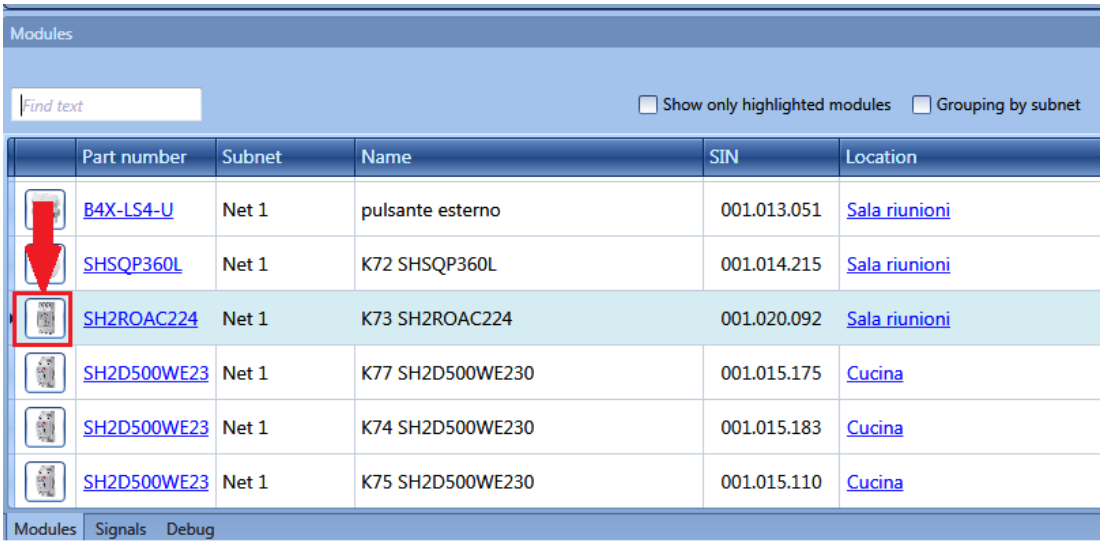


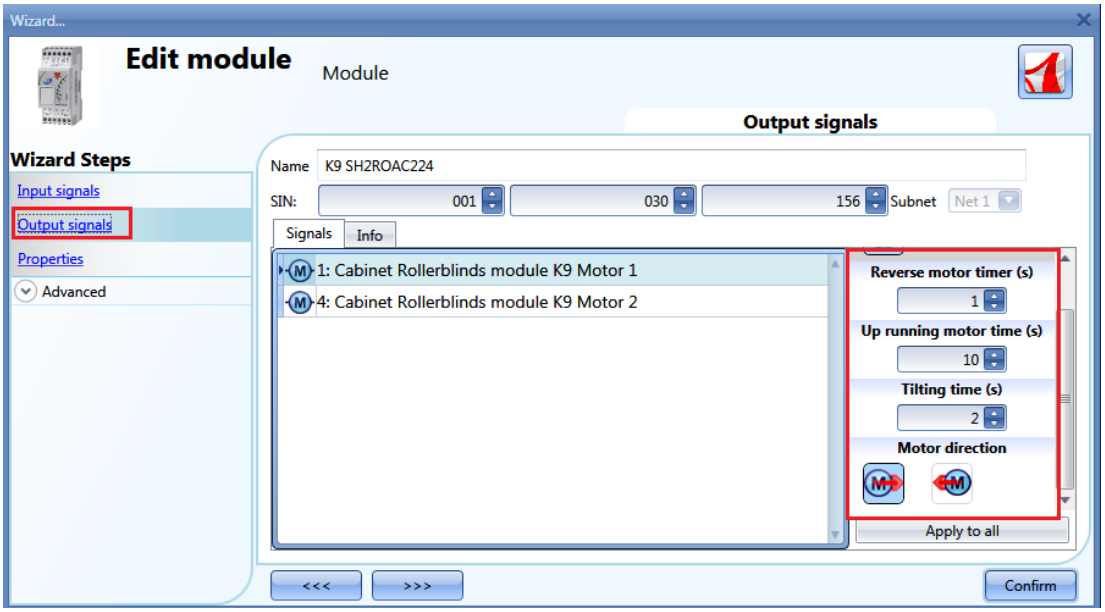
The signals available in this window are only the motor output type.
Up to 10 motor outputs can be chosen and they will be managed in parallel.



10.10.5 How to set the running and tilting time

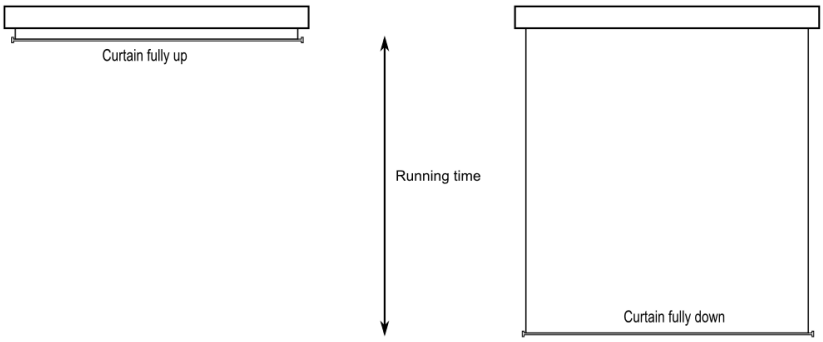
The running and tilting time of the motor have to be set when the rollerblind module is added.
These settings are needed for the correct operation of the function and they should be defined in the wizard relevant to the rollerblind module.
To access this, the user should open the Rollerblind module wizard by clicking on the relevant icons, and then select *Output signal* (see picture below).



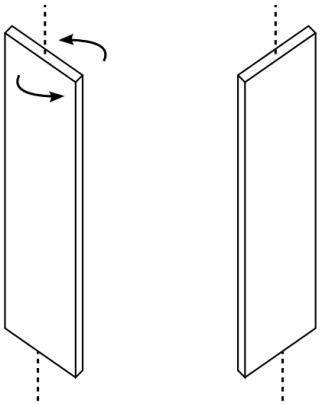


Reverse motor time (seconds):
This is the time in seconds to change the motor direction (this time should be set to 1 second at least, to avoid damaging the motor).

Running motor time (seconds):
This is the time for the blinds to go from a fully up position to a fully down position.
Starting from a fully wrapped position, the user must use the manual command to completely open the blind. The entire running time must be measured: the accuracy of the time is 0,1 second.



The motor tilting time (seconds):
This is the time for a complete tilting movement (a slats rotation of 180°). Starting from a fully tilted position, the user must use the manual command to tilt the blind into the opposite position.
The whole tilting time must be measured: the accuracy of the time is 0,1 second.



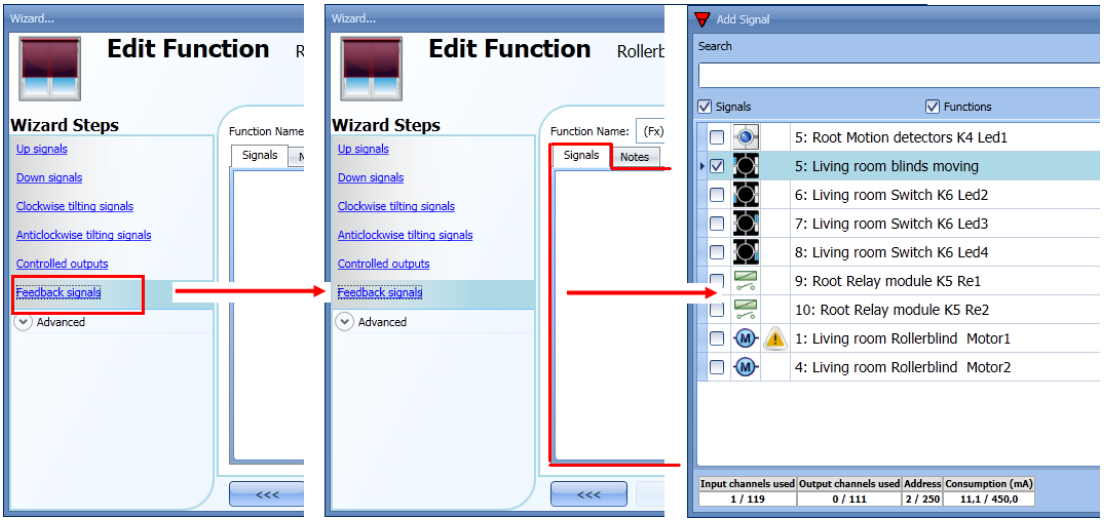
The motor direction:
The user can change the direction of the motor by clicking on the relevant icons.

	Clockwise movement
	Anti clockwise movement

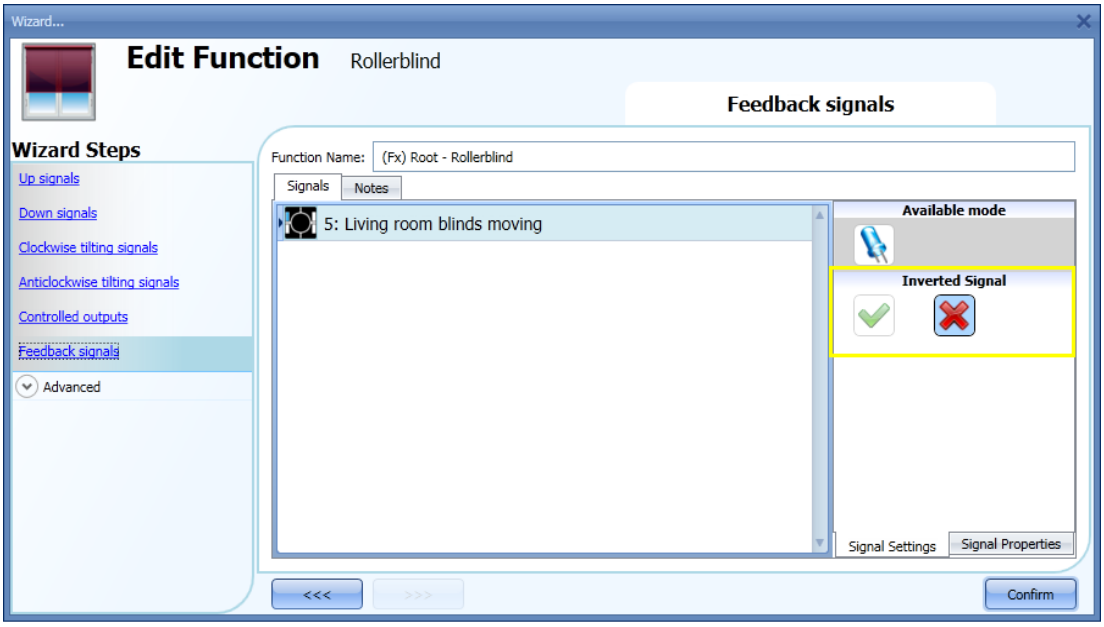
NB. For all those modules with serial numbers higher than MN46 (rev4), the tool manages two different running times: one time for motor up and another one for motor down. Please note that all the modules produced with a lower serial number (that means an earlier firmware revision) do not manage this new feature and an error will be shown by the tool.

10.10.6 How to set a feedback signal

To select the feedback signal that indicates the status of the function, click on *Feedback signals* and then double click on the *Signal window*.
When the motor is moving, the feedback signal will be activated.



The signals available in this window are LEDs and relays.
Up to 50 signals can be added and they will be managed in parallel.
The logic of each feedback signal can be set as standard or inverted (see the yellow rectangle).

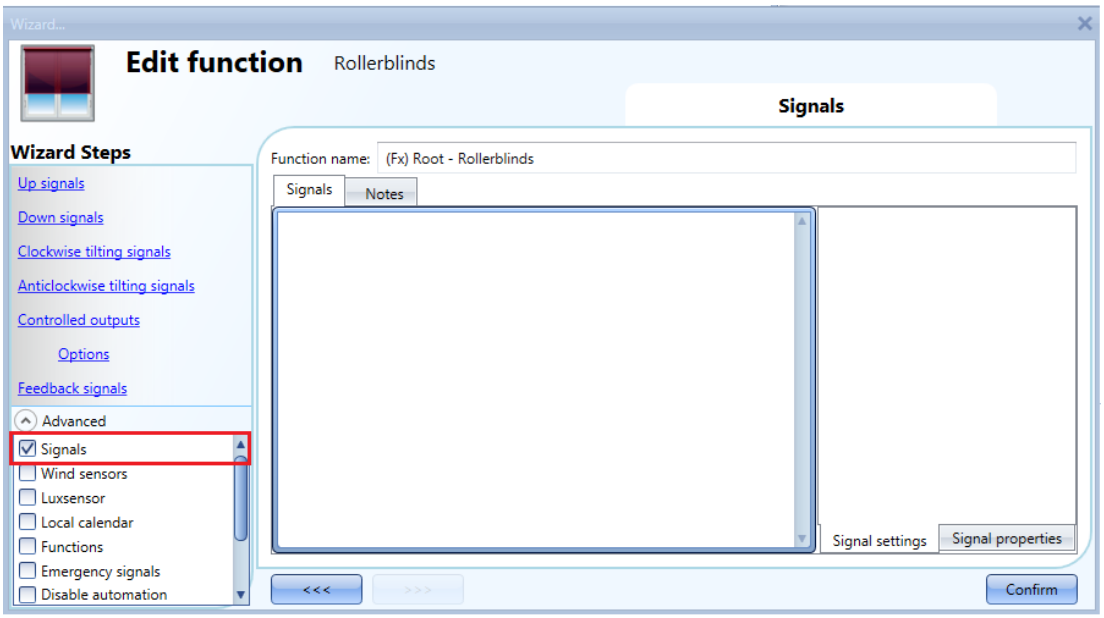


10.10.7 How to manage blinds with automations

The automatic up/down positioning and the tilting of the blinds can be managed by wind sensors (the speed value measured by the sensor can be used to move blinds up/down), by calendar functions (calendar to move up/down at pre-defined time intervals), by rain sensors (the blinds can be moved according to the weather condition) and by lux meters moving blinds up/down according to the levels of ambient light.
All these automations have to be enabled in the Advanced section.

10.10.8 How to customize the push button behaviour

In the *Advanced* settings, the user can customize the behaviour of each push button used to control the blinds.



To add the signal to control the Up/Down movement and the tilting, the user should click on the relevant section, then double click on the *Signal* window and select the input signal from the list.

By clicking on each added signal, it is possible to select the action to be carried out for the “short press” and “very long press” on the pushbutton.

In the picture below, an example is shown where two push buttons are used to send a fully up/down command with a short click and to tilt the slats clockwise/anticlockwise by using the very long activation.

The available actions are listed in the tables below.

Short pressure of the push button	
No action	No action will be carried out if the push button is pressed briefly
Stop	The motor is stopped
Reset UP position	The motor is activated for the whole up running time
Reset DOWN position	The motor is activated for the whole down running time
Fully UP	The motor is activated for the time it needs to go fully up from the current position. E.g.: if the current position is 40%, the motor will be activated for 60% of the up running time.
Fully DOWN	The motor is activated for the time it needs to go fully down from the current position. E.g.: if the current position is 20%, the motor will be activated for 80% of the up running time.
Fully UP and reverse	The motor is activated for the time it needs to go fully up from the current position. When the button is pressed again, the motor is stopped. The next time the push button is pressed, the motor reverses its direction and will go down for the whole down running time.
Move UP step by step	The motor is activated to go up for the selected running percentage each time the pushbutton is pressed. E.g.: if 20% is selected, each time the push button is pressed, the motor will be activated for 20% of the up running time.
Move DOWN step by step	The motor is activated to go down for the selected running percentage each time the pushbutton is pressed. E.g.: if 20% is selected, each time the push button is pressed, the motor will be activated for 20% of the down running time.
Fully clockwise tilting	The slats are fully tilted clockwise
Fully anticlockwise tilting	The slats are fully tilted anticlockwise
Fully anticlockwise tilting and reverse	The slats are fully tilted anticlockwise and at the next press of the button they are fully tilted clockwise.
Tilt clockwise step by step	The slats are tilted clockwise for the selected tilting percentage each time the pushbutton is pressed. E.g.: if 20% is selected, each time the push button is pressed, the motor will be activated for 20% of the tilting time.
Tilt anticlockwise step by step	The slats are tilted anticlockwise for the selected tilting percentage each time the pushbutton is pressed. E.g.: if 20% is selected, each time the push button is pressed, the motor will be activated for 20% of the tilting time.
Move to xx% and tilt to yy%	Each time the push button is pressed, the roller blind will be brought to the set (xx%) position.

Very long pressure of the push button	
No action	No action will be carried out if the push button is pressed continuously
Go UP - stop	The blind is moved up while the pushbutton is kept pressed. When it is released, the motor is stopped.
Go DOWN - stop	The blind is moved down while the pushbutton is kept pressed. When it is released, the motor is stopped.
Go UP – stop - reverse	The blind is moved up while the pushbutton is kept pressed. When it is released, the motor is stopped. The next time the push button is pressed, the motor reverses the direction.
Fully UP	The motor is activated for the time it needs to go fully up from the current position. E.g.: if the current position is 40%, the motor will be activated for 60% of the up running time.
Fully DOWN	The motor is activated for the time it needs to go fully down from the current position. E.g.: if the current position is 20%, the motor will be activated for 80% of the up running time.
Clockwise tilting - stop	The slats are tilted clockwise while the push button is kept pressed. When it is released, the motor is stopped.
Anticlockwise tilting -.stop	The slats are tilted anticlockwise while the push button is kept pressed. When it is released, the motor is stopped.
Fully clockwise tilting	The slats are fully tilted clockwise
Fully anticlockwise tilting	The slats are fully tilted anticlockwise
Tilt anticlockwise - stop-reverse	The slats are tilted anticlockwise while the push button is kept pressed. When it is released, the motor is stopped. The next time the button is pressed, the tilting direction is changed.
Move to xx% and tilt to yy%	Each time the push button is kept pressed, the roller blind will be brought to the set (xx%) position.

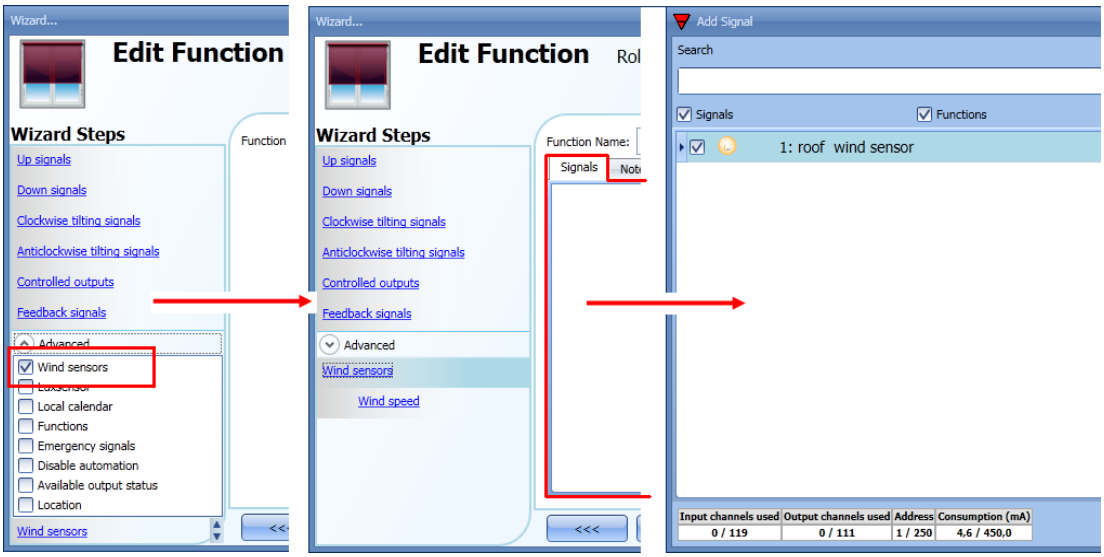
When the motor is running up or down, any action on the push button will stop it.

10.10.9 How to use the wind sensor to control blinds

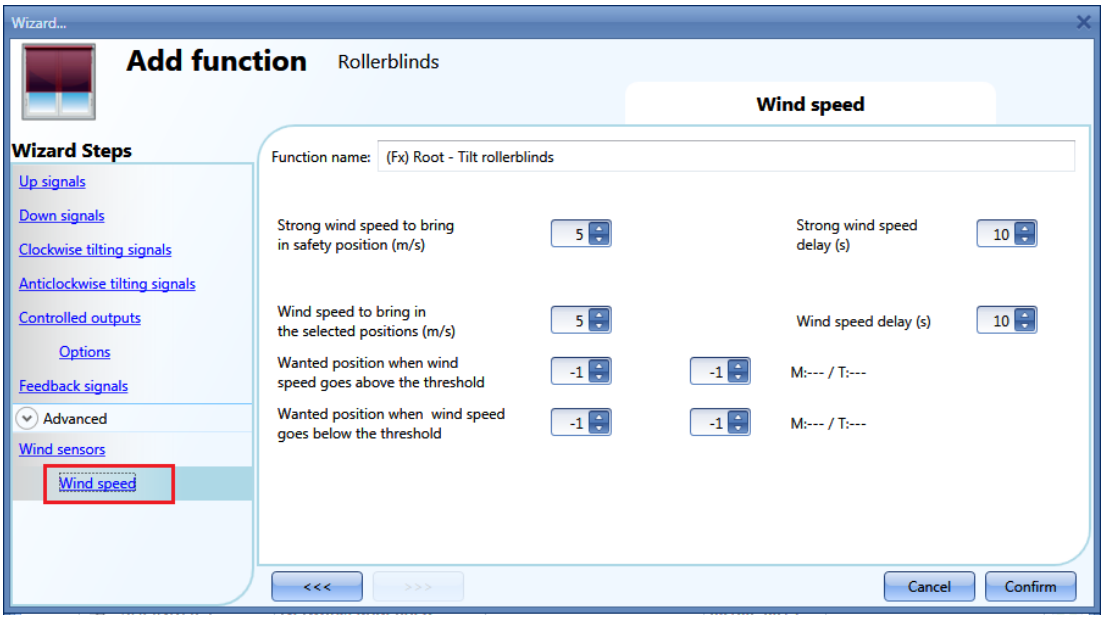
The wind sensor can be used to perform different actions:

3. Adjust the position of the blinds according to the measured wind speed
4. Adjust the tilting position of the slats
5. Bring blinds into a safety position if the measured wind speed is very high

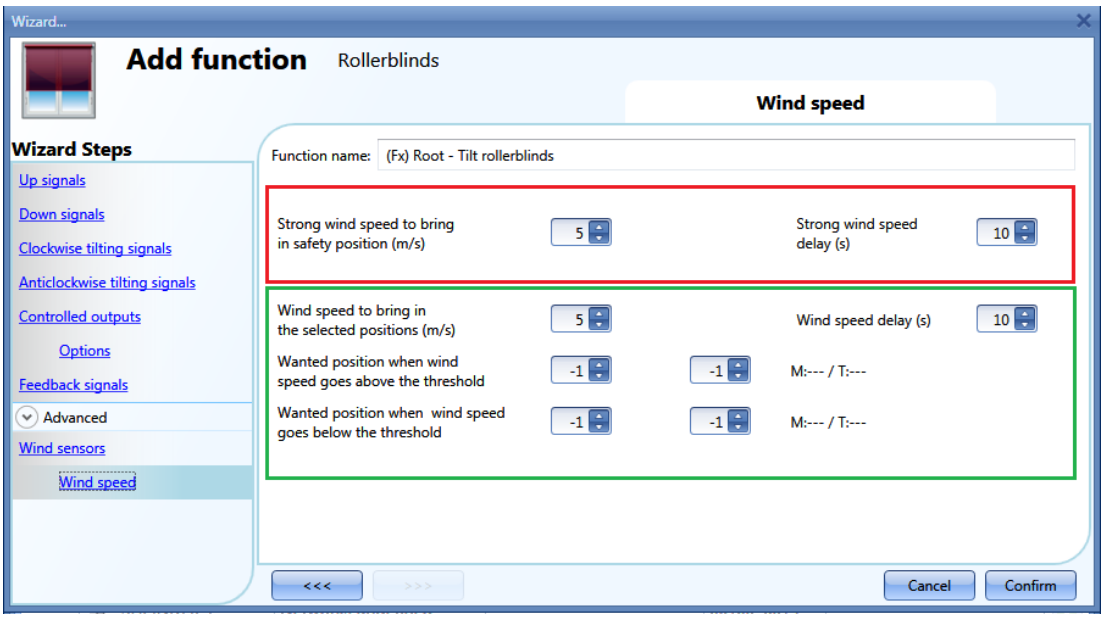
When a wind sensor needs to be used, it must first be enabled in the *Advanced* section.
The *Wind sensors* menu will appear. After selecting it, with a double click on the Signal window, the list of available signals will appear.
Select the required signal/signals and click on *Confirm*. Only one signal can be selected.



Once the wind sensor is added, the user has to set the wind speed limits in the *Wind speed* menu:



The *Wind speed* menu is split into two main fields:



Highlighted in red: the user can select a strong wind threshold value - when the wind speed goes over this threshold, the blinds are automatically brought into the safety position (the safety position is fully up/down according to the settings in the output of the function).
While the strong wind condition is active, the blind stays in the safety position and all other automations are not accepted. The only way to control the curtain is to use manual commands.
The user can select a delay time to move the blind to the safety position: the blind will be moved only when the strong wind condition is active for a time longer than the set value. If the delay time is set to 0 seconds, the delay time is disabled.

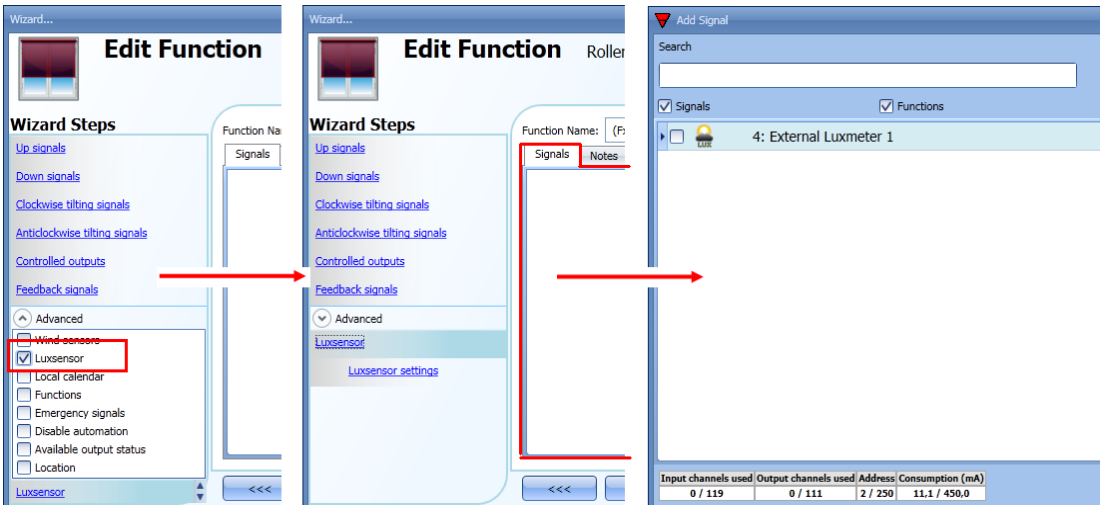
Example 1
In the picture above, when the wind speed goes over 6 m/s for more than 10 seconds, the blind is moved to the safety position. In order to avoid damage to the blinds, we suggest setting a small delay value (max 30 seconds).

Highlighted in green: the user can set another wind threshold. When the wind speed goes over/under this value, it is possible to select different actions for the blind. The user can also select a delay time for each action

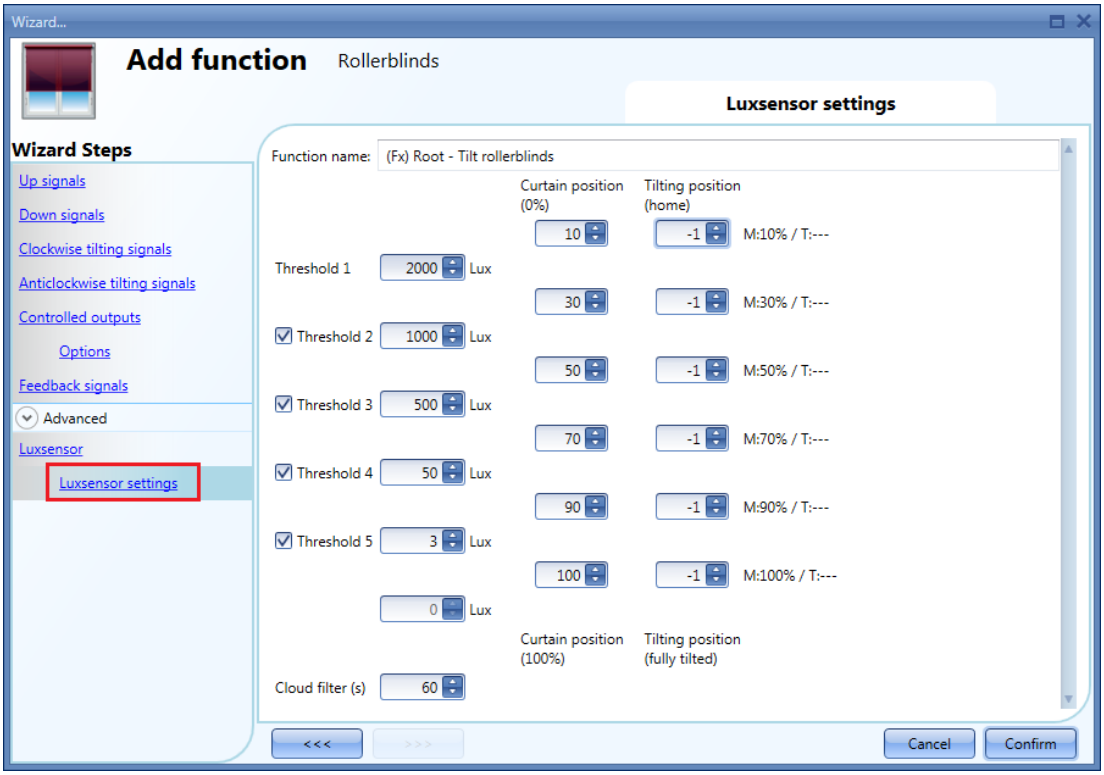
Example 2
In the picture above, when the wind speed goes over 3 m/s for more than 60 seconds, the blind is moved to 50% of the running time; when the blind reaches 50% the motor stops for a few seconds and then it is fully tilted.
When the wind speed goes below 3 m/s for more than 60 seconds, the blind is moved to 15% of the running time; when the blind reaches 15% the motor stops for a few seconds and then it is tilted for 50% of the tilting time.
In order to avoid continuous movement of the blind, we suggest setting a minimum delay value of 60 seconds.

10.10.10 How to control the blind according to the daylight

The blind position can be managed in an automatic way by using lux meters: the user can define up to 5 different light levels to make the smart-house system move the blinds up/down and tilt them.
To select the lux sensor, the user should click on *Luxsensors* and then double click on the *Signal window*. Once the output window is opened, select the relevant signal from the list.



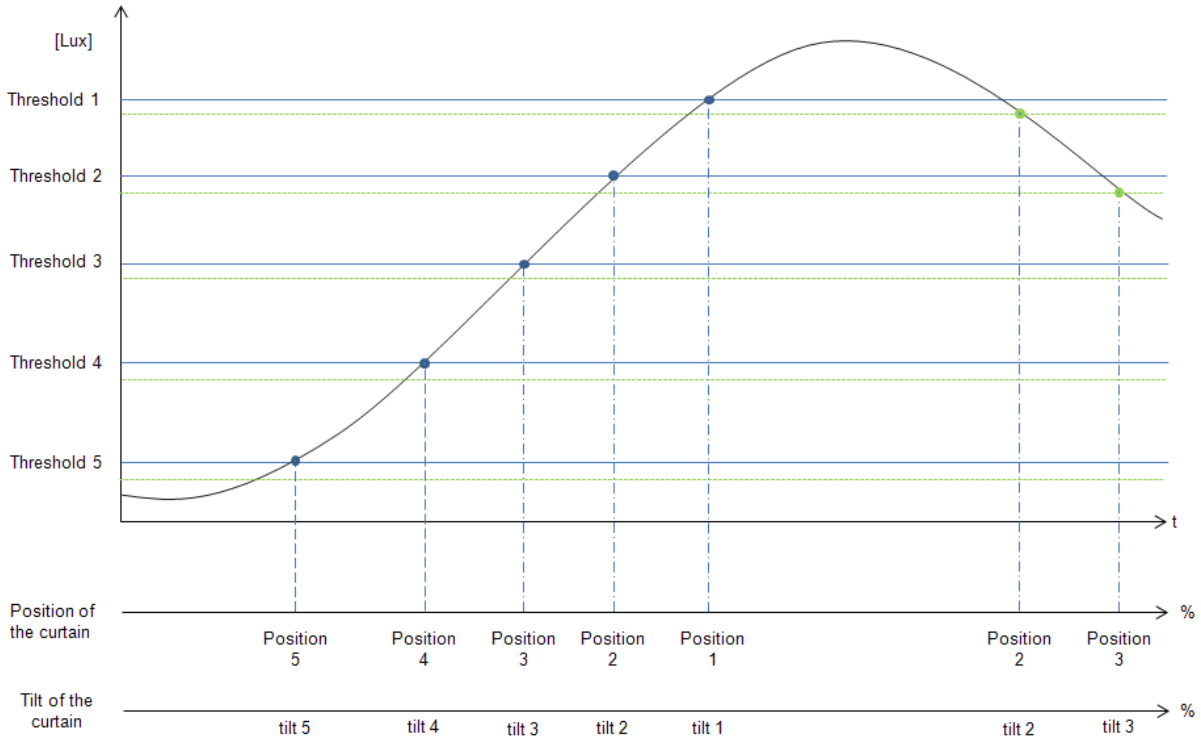
The user can add up to 10 lux sensors: if more lux signals are linked to the function, the average value is calculated and used.
Once the lux sensor is added, the user can change the threshold values and the action that the blind has to perform when these values are reached.



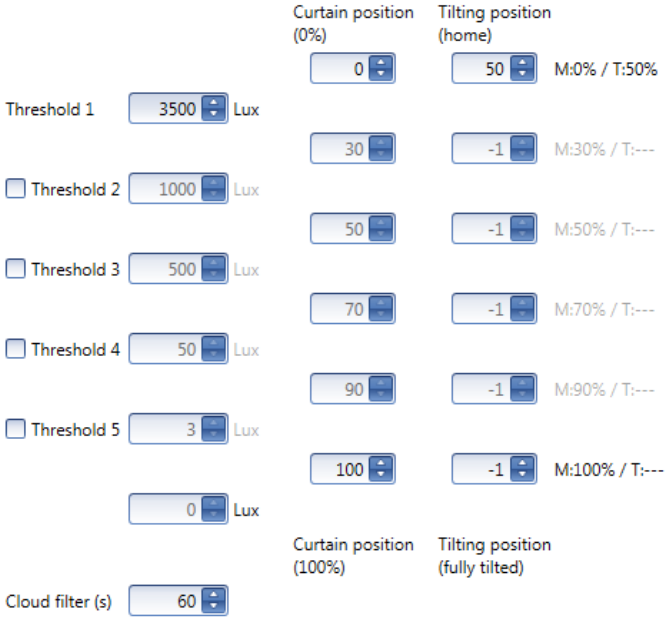
The up/down movements and the tilting start when the thresholds are surpassed, so if the blind position is changed manually by means of a light switch or automatically e.g. by the scheduler, the automation of the lux sensor will not change it unless the thresholds are surpassed.
The hysteresis value is fixed at 10% and it is used for all the thresholds.
When the light level is increasing, the threshold values are used: when the light level is decreasing the following value is used:

value= threshold value – hysteresis.

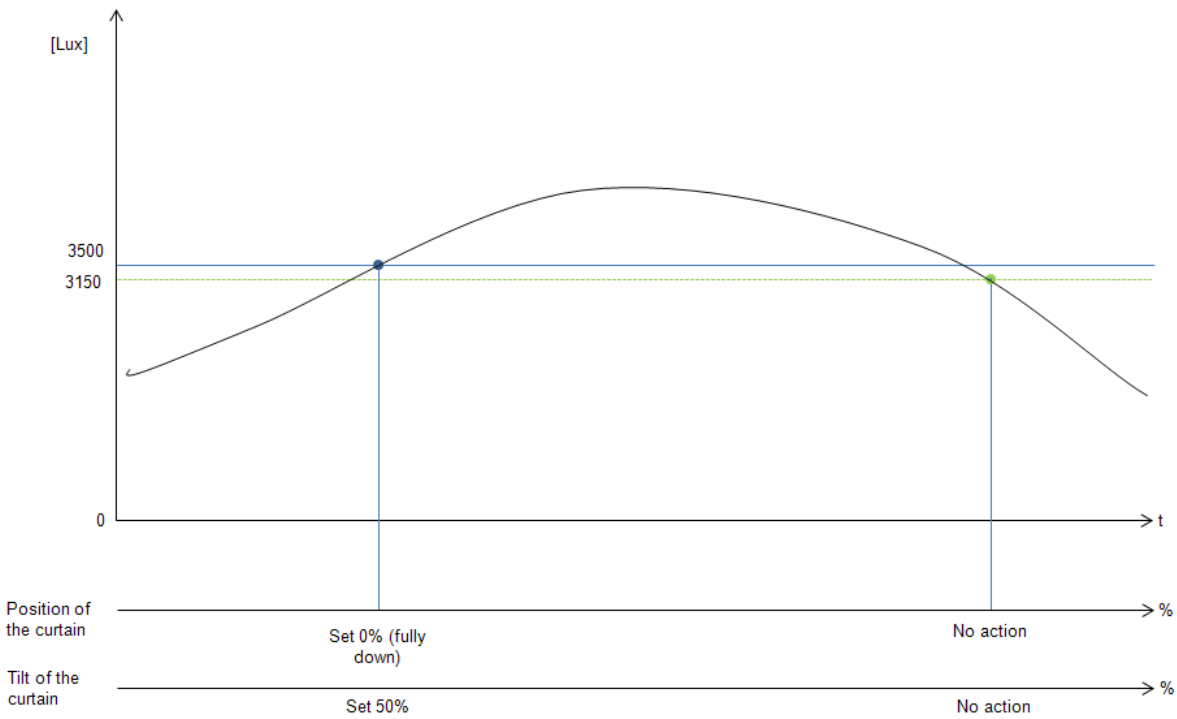
The user can also set a cloud filter to avoid the up/down movements if a cloud shields the sun for a short time. We suggest setting the time for at least 60 seconds; if the time is set to 0 the filter is disabled.
The picture below shows how the blind position is changed according to the light level. At least one threshold must be used with the lux sensor.



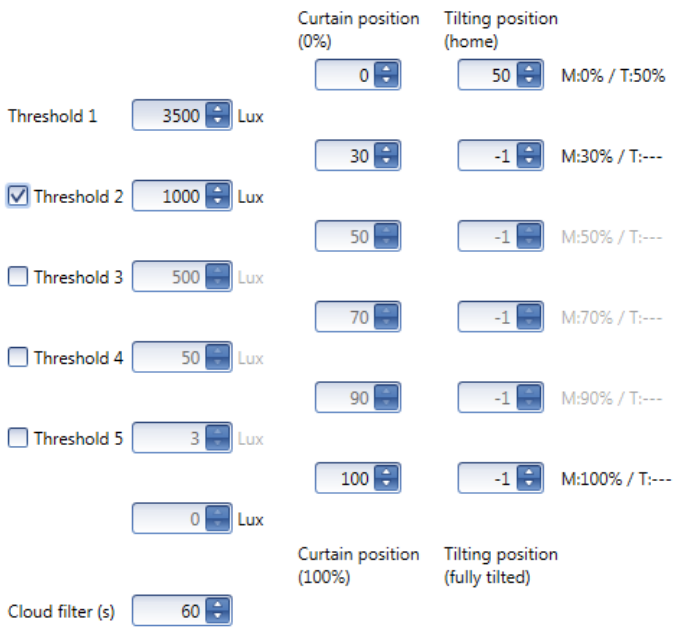
Example 1:
In the first example only one threshold is used and it is set at 3500 Lux: the cloud filter is 60 seconds. When the light goes over 3500 Lux, the blind starts moving until it reaches the fully down position. When the blind is down it is tilted 50% (90°).



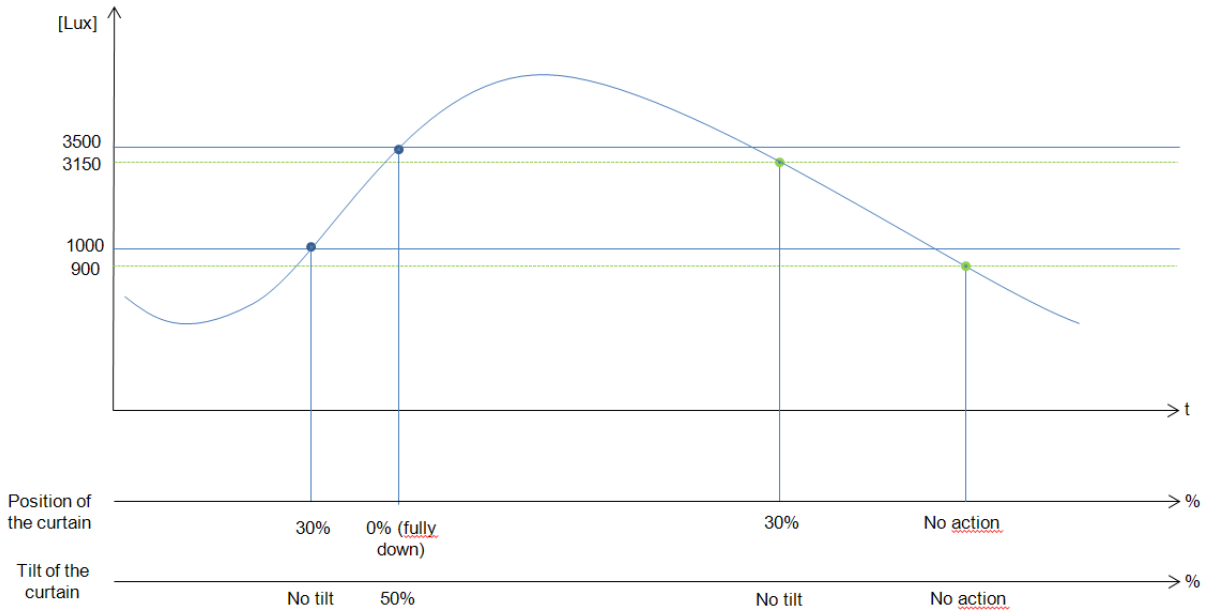
In the picture below, the behaviour of the output is shown according to the light level.



Example 2:
In the second example, threshold 1 is set to 3500 Lux and threshold 2 is set to 1000 Lux: the cloud filter is 60 seconds. When the light goes over 3500 Lux, the blind starts moving until it reaches the fully down position. When the blind is fully down, it is tilted 50% (90°).



In the picture below, the behaviour of the output is shown according to the light level.

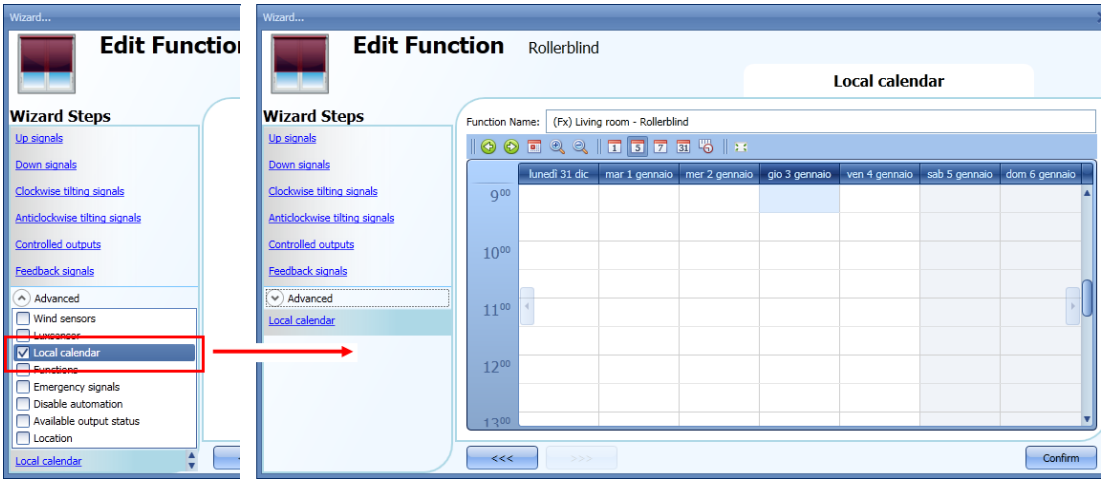


10.10.11 How to move the blinds up/down and tilt them with the calendar

There are two ways of managing the *Tilting blinds* function with the calendar: by setting the "local" calendar inside the function or using a "global" calendar function.

The local calendar function

To enable this, the relevant menu has to be enabled in the Advanced section.

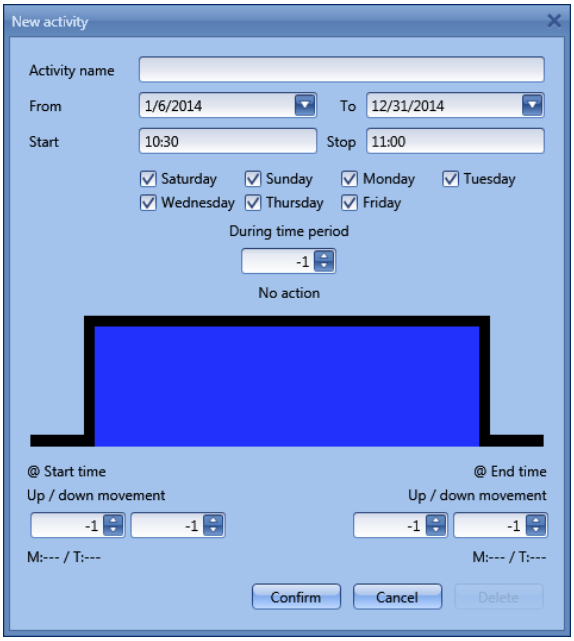


The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons :

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day.
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally.
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

Calendar activities
Once the user has selected his preferred visualization, to enter a time band he has to double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From*:* the start date for the calendar activity.

To*:* the end date of the calendar activity.

Start*:* The time of the start of the activity.

Stop: The time of the end of the activity.

@ start time up/down movement: in this field the user can select the action of the function at the start time.
The user can select the following actions:

- No action (-1)
- Set the position of the blind at a fixed value between 0% and 100% (0-100)

@ start time tilting: in this field the user can select the action of the function at start time.
The user can select the following actions:

- No action (-1)
- Set the tilting position at a fixed value between 0% and 100% (0-100)

@ ***end time***: in this field the user can select the action of the function at the stop time.
The user can select the following actions:

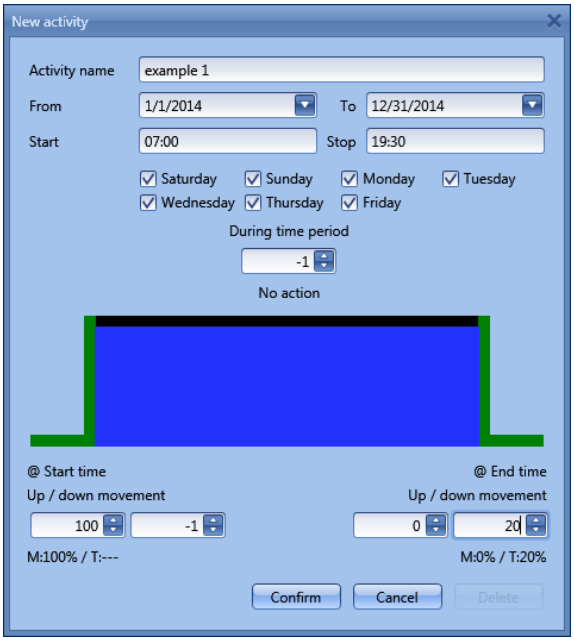
- No action (-1)
- Set the position of the blind at a fixed value between 0% and 100% (0-100)

@ ***end time tilting***: in this field the user can select the action of the function at end time.
The user can select the following actions:

- No action (-1)
- Set the tilting position at a fixed value between 0% and 100% (0-100)

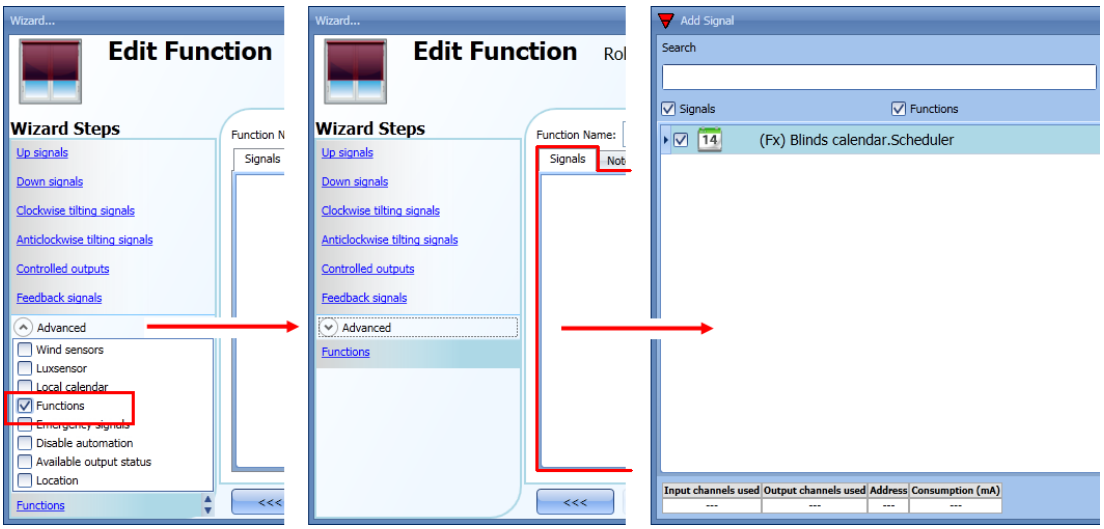
Days: The user has to select the days of the week for the activity.

In the example below, the calendar is set to work all the year (from 1st January to 31st December) . The working days are: Monday, Tuesday, Wednesday, Thursday and Friday (the calendar does not work on Saturday and Sunday).
The start time is 7:00 and the stop time is 19:30.
At 7:00 the blind starts moving up: when the curtain is fully up there is no tilt. At 19:30 the blind starts moving down: when the blind is fully down it is tilted at 20%.



The global calendar function

Before using a global calendar it has to be defined as a function (See how to set a global calendar). The field function has to be selected in the *Advanced* menu.
By clicking on *Functions* and then double clicking in the *Signal* window, the *Add signal* window will appear. Select the required calendar function.

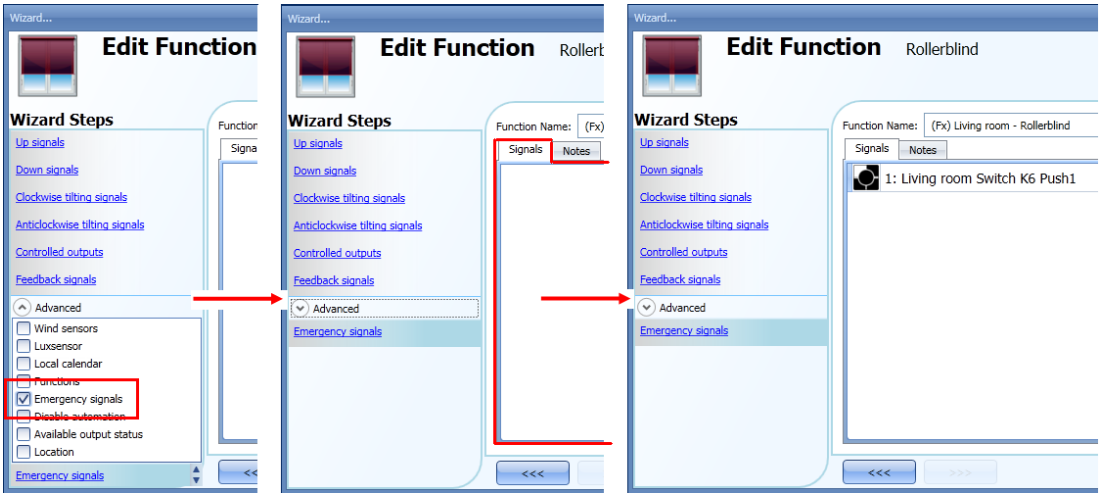


The behaviour of the calendar will be the same as that of the local calendar.

An example of this automation is a global calendar to move all the blinds down at ten o'clock in the evening.

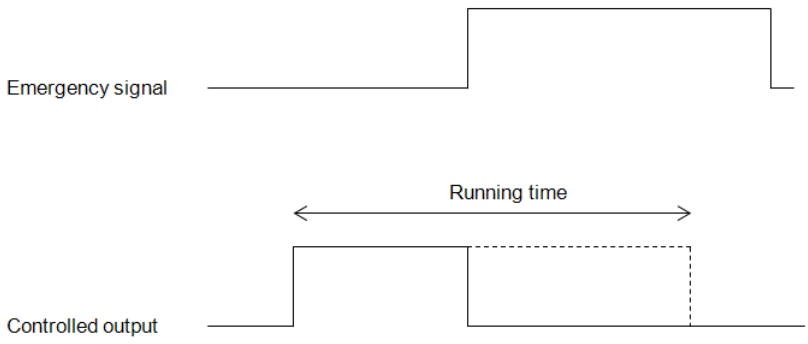
10.10.12 How to use the emergency signals

In some applications, for the safety of the user, a pushbutton is required to stop the motor at any time. The *Emergency signal* field allows the user to stop the motor when the signal is active. The roller blind module does not accept any manual or automatic command until the emergency signal is active.
To add an emergency signal, the user should select the relevant section in the advanced menu, then double click on the *Signal* window and select the input signal from the list (see picture below).



When the signal is added, the inverted logic can be used (see the yellow rectangle in the picture below).

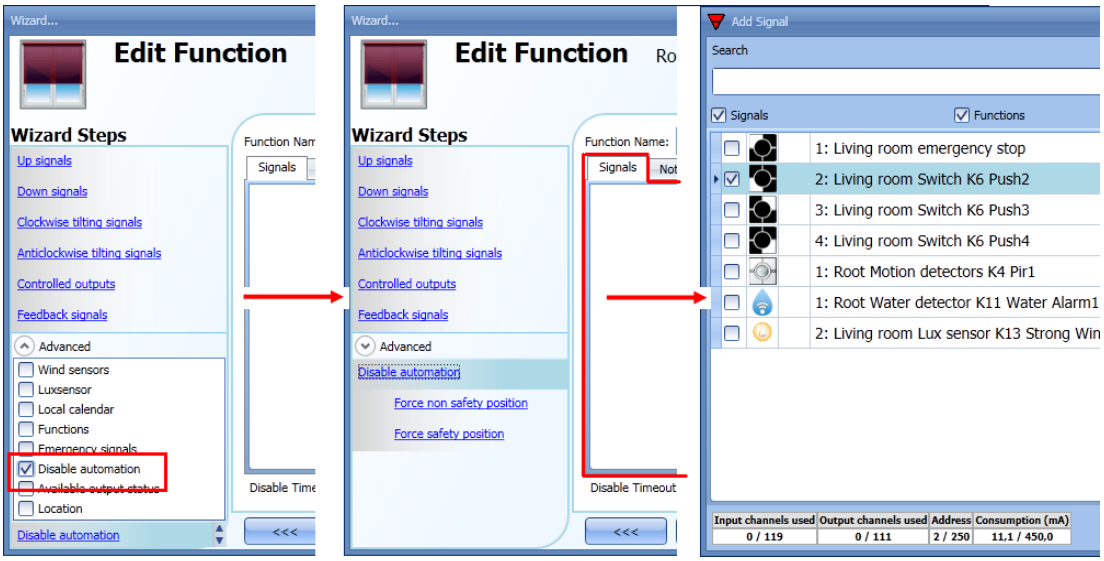
In the example below, the motor is stopped immediately when the emergency signal is activated.



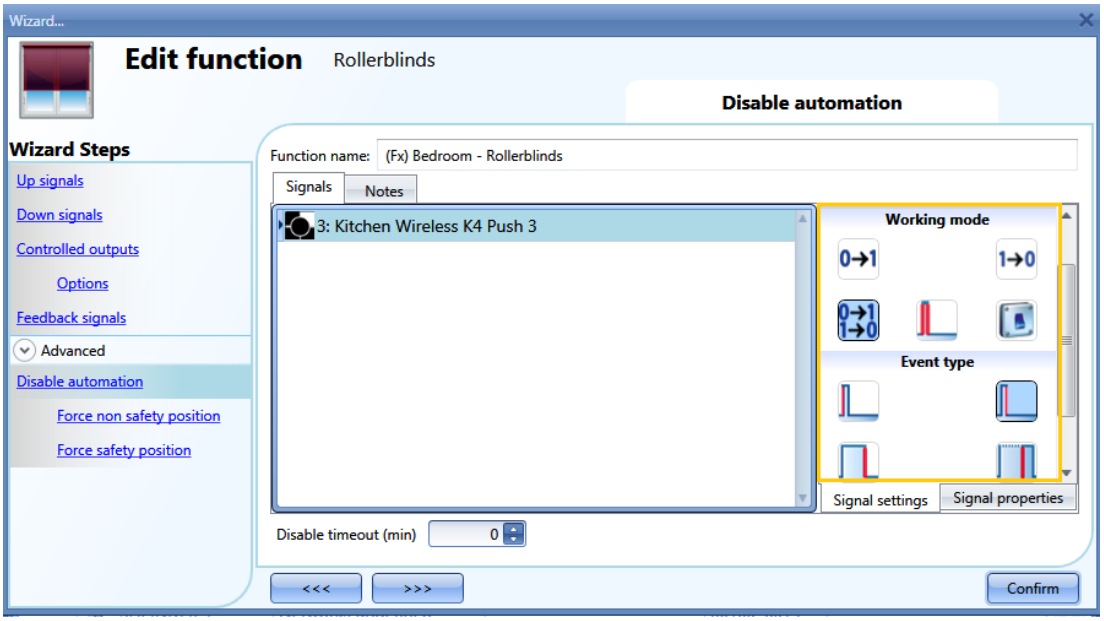
10.10.13 How to stop the automations

Sometimes it is necessary to stop all the automations to service the blinds or simply to clean the windows.

To add a disable automation signal, the user has to select the relevant section in the advanced menu, then double click on the *Signals* window and select the input signal from the list (see picture below).



When the signal is added, the user should select the working mode and the type of the event:



If a push button type is added, the working mode, highlighted with the yellow rectangle, has to be selected according to the table below:

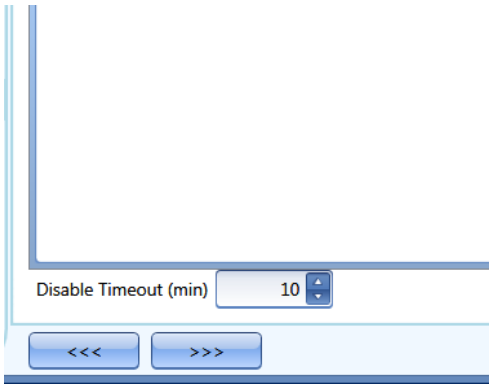
	Event type			
Working mode				
0→1	As soon as the push button is pressed, the automation is disabled.	After a short press (less than 1 second), the automation is disabled at the release of the push button.	After a long press, the automation is disabled at the release of the push button.	After a very long press, the automation is disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automation is enabled again.	After a short press (less than 1 second), the automation is enabled again at the release of the push button.	After a long press, the automation is enabled again at the release of the push button.	After a very long press, the automation is enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the automation will be enabled/disabled in toggle mode.	After a short press (less than 1 second), the automation will be enabled/disabled in toggle mode at the release of the push button.	After a long press, the automation will be enabled/disabled in toggle mode at the release of the push button.	After a very long press, the automation will be enabled/disabled in toggle mode at the release of the push button.
	The automation will be enabled/disabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

	Event type	
Working mode	**Signal activated**	**Signal deactivated**
1→0	The automation is enabled again	No action
0→1	The automation is disabled	No action
0→1 1→0	The automation is disabled/enabled in toggle mode	No action
	The automation is disabled/enabled in toggle mode	The automation is disabled/enabled in toggle mode
	The automation is disabled	The automation is enabled

The user might need to set a time after which the automation is enabled again even if the selected signal is still active: to achieve this, the Disable timeout field has to be set.

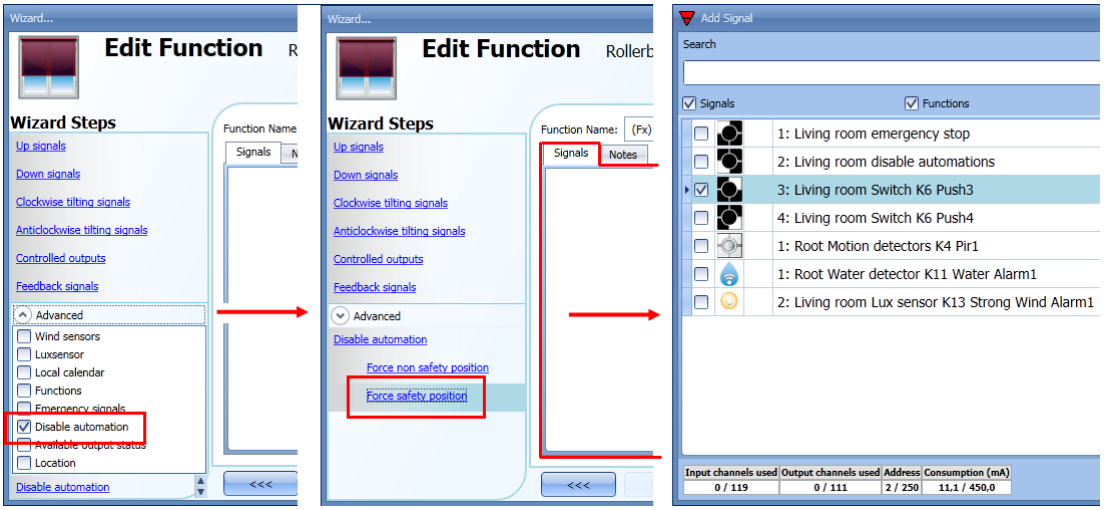
To access the stop automation functionalities, they must be enabled in the Advanced section field.
If the user needs to be sure that the disabling of the automation is automatically removed, the *disable automatism timer* has to be set.



The timer starts each time the disable status becomes activated: the disable status is automatically deactivated as soon as it has elapsed.
In the figure above a disable timeout of 10 minutes is set. The maximum delay is 59minutes.

10.10.14 How to force the safety condition

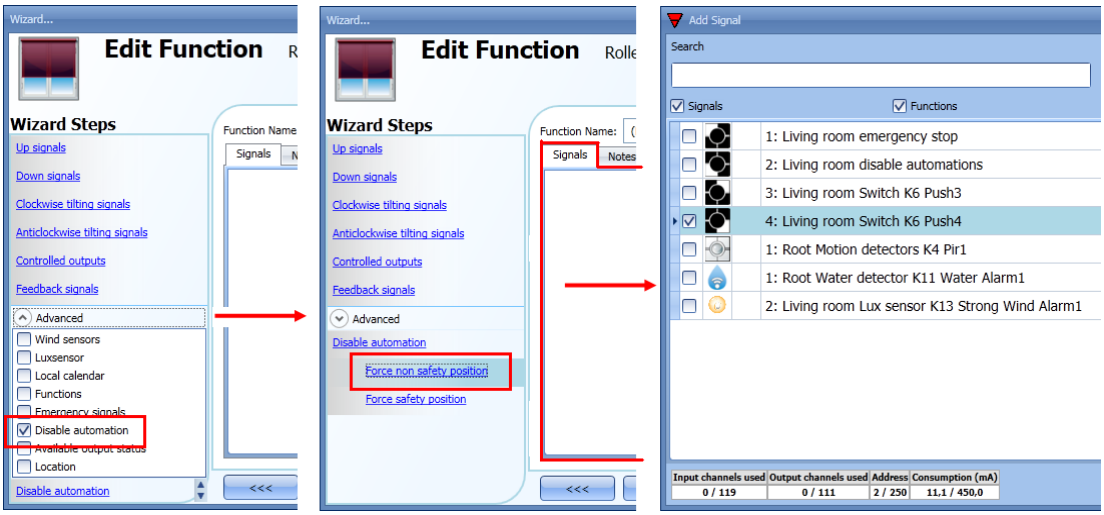
If the user wants to force the safety position regardless of all other signals used in the function, the *Force safety position* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force safety position*, then double click on the signal window and select the right signal to use.



Each signal used in the *Force safety position* window works as a level signal: until the signal is active, the blind is forced into the safety position.
When the *force to safety position* signal is activated and the *force to no safety position* signal is activated, the force to safety position signal has the priority.

10.10.15 How to force the non-safety condition

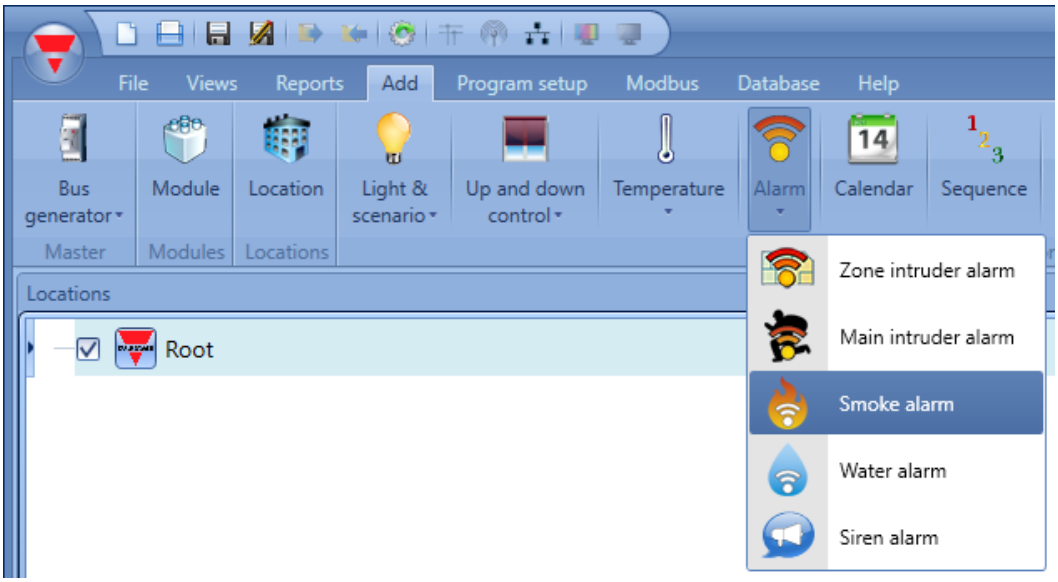
If the user wants to force the non-safety position regardless of all other signals used in the function, the *Force non safety position* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force non safety position*, then double click on the signal window and select the right signal to use.



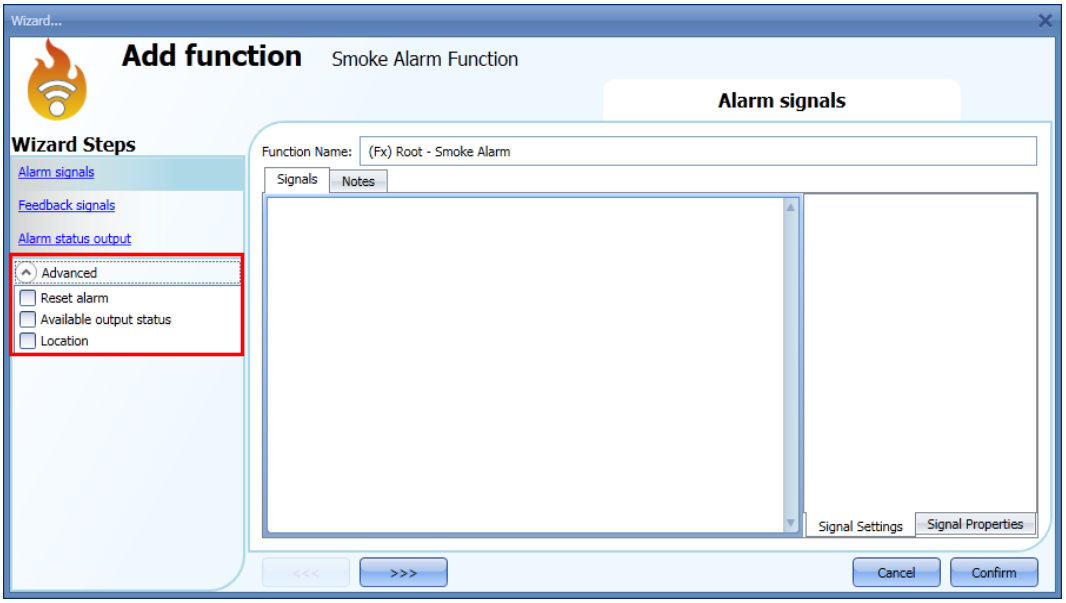
When the *Force output on* signal is activated and at the same time the *Force output off* signal is activated, the *Force output on* signal has priority.

10.11 How to add a smoke alarm function

The user can configure a basic smoke alarm function in order to be warned about smoke in the house. To set up a smoke alarm function the user should select *Alarm* from the Add menu, then select *Smoke Alarm* (see picture below). The new function will be added into the selected location.
The wizard relevant to a Window control function can also be opened by typing Alt +A +A +S.



This function manages one or more outputs controlled by one or more input commands. The command might be a signal from a smoke detector, a pushbutton, a switch, a function or a remote command (Webserver, sms, Modbus TCP/IP)
The automation of the *Smoke alarm* function can be managed by accessing the *Advanced* section.
In the advanced section the user can select the reset signal: this is used to stop the alarm signal in order to silence the system when the alarm has been acknowledged, and also to reset the alarm output. The alarm will be inhibited for a time period, allowing the user to remove the smoke from around the sensor in case of a non-fire situation.
In the *Advanced* section the user can also configure two signals in order to force the status of the function on or off.

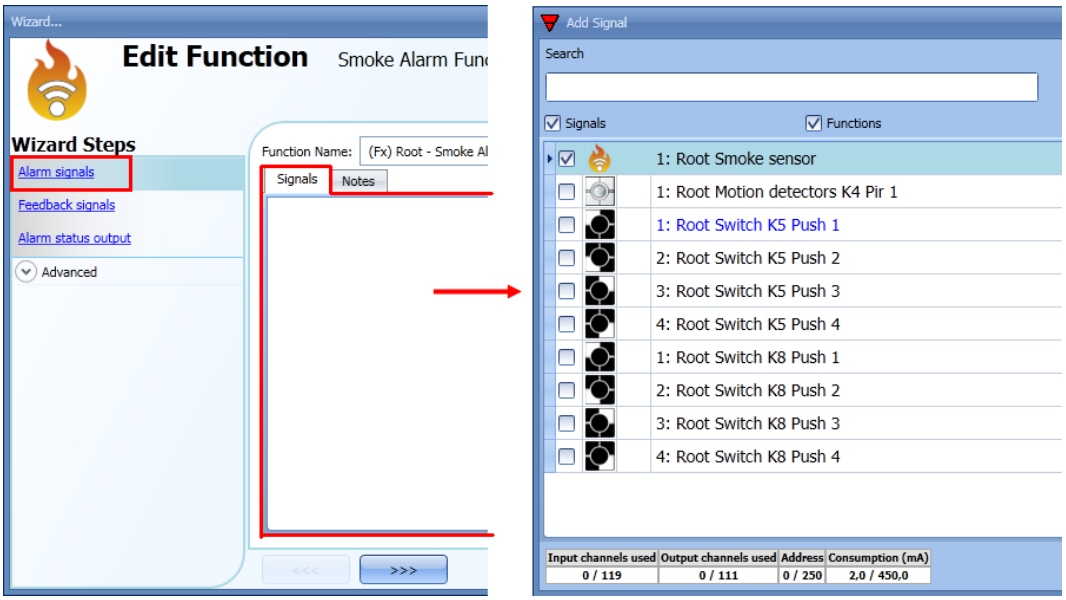


10.11.1 How to set a simple smoke alarm system

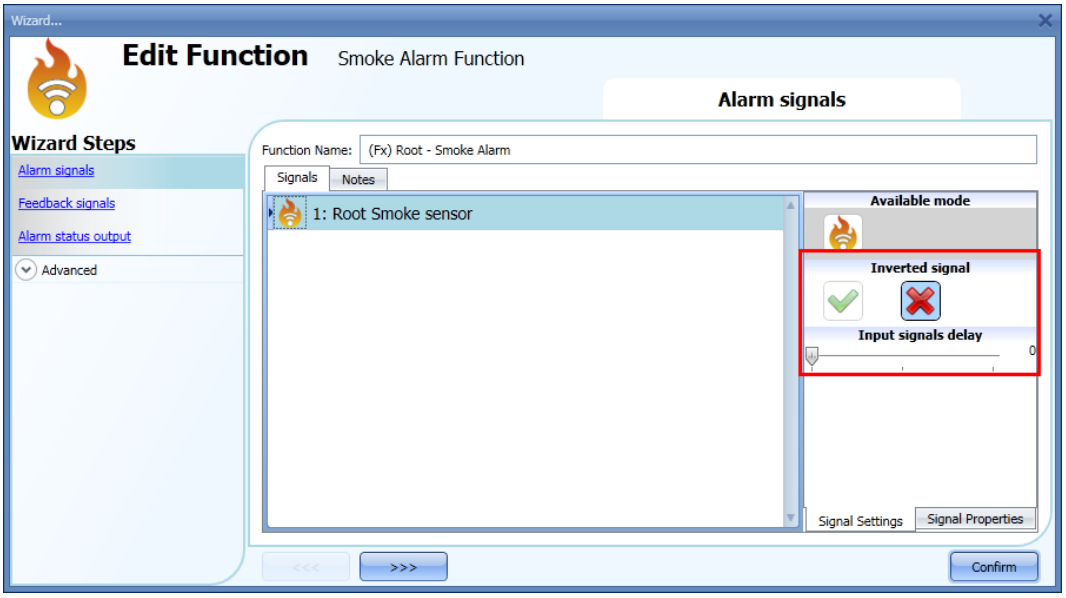
If the user wants to create a simple smoke alarm system, he needs at least two signals to have a fully functional smoke alarm system:

3) A smoke sensor as input signal
4) A general output that is activated when the alarm is on

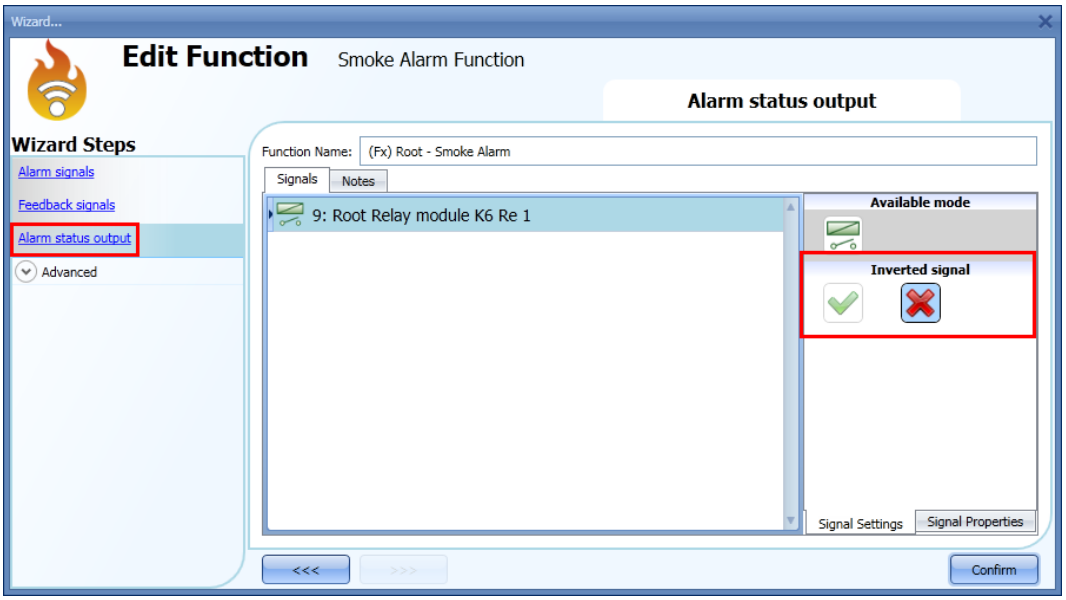
Once the modules are added to the configuration (smoke sensor and output module), the user should select the alarm input signal in the relevant section of the *Smoke alarm* wizard, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



Once the alarm signal is entered, the user can also decide to use the inverted logic of the signal (see the red rectangle in the picture below). The inverted logic is enabled by selecting the green V.
Each alarm signal added to the function has a delay time that can be set in order to filter the activation of the alarm when the smoke is detected. This might be used to avoid false alarm triggering.



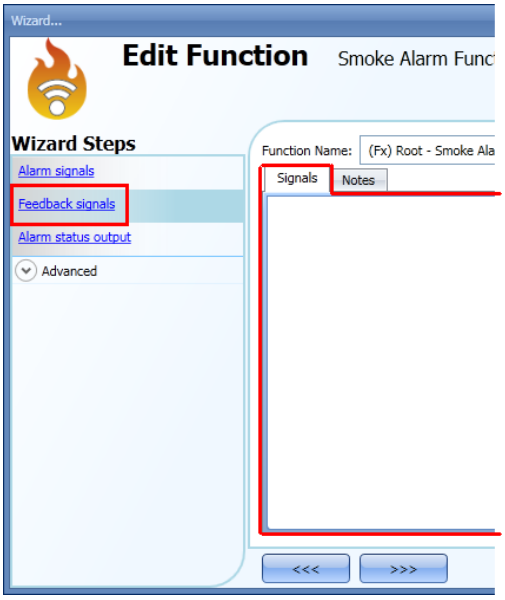
To add the alarm output, the user should select the relevant section in the wizard of the function, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).
The output signal can be a relay, a LED or a buzzer. The output signal will stay on until smoke is detected by the sensor.

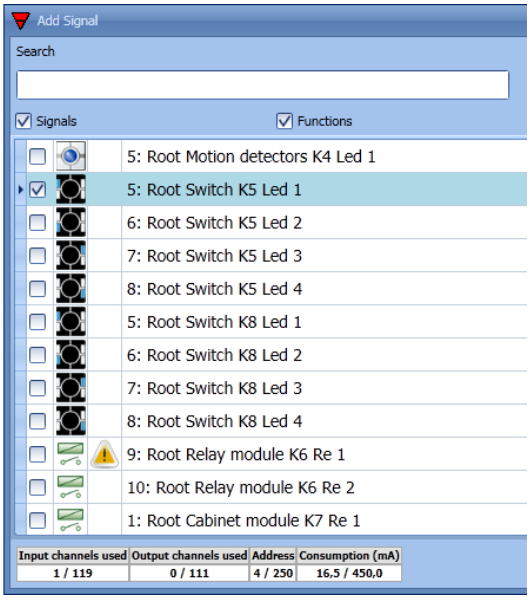


Example 1:
Using this easy setting, the function works in this way: the smoke function is ready to react as soon as the smart-house system is started. While the sensor doesn't detect smoke, the output is off. When the sensor detects smoke, the function is activated and the output goes on and stays on until the smoke signal disappears.

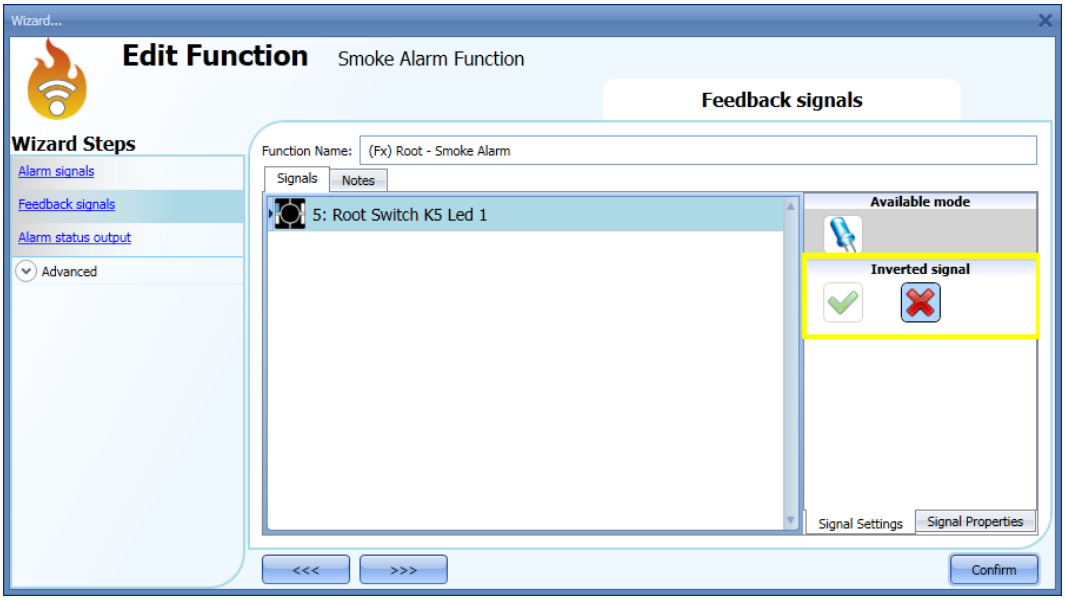
10.11.2 How to add a feedback signal

If the user wants to use a feedback signal to check the status of the function, he should select the relevant section, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).
The feedback signal can be a relay, a LED or a buzzer, and it behaves according to the status of the function: when smoke is detected, the feedback signal starts blinking 1 second ON 1 second OFF. When no smoke is detected the feedback is off. When the reset signal is active the feedback is on.





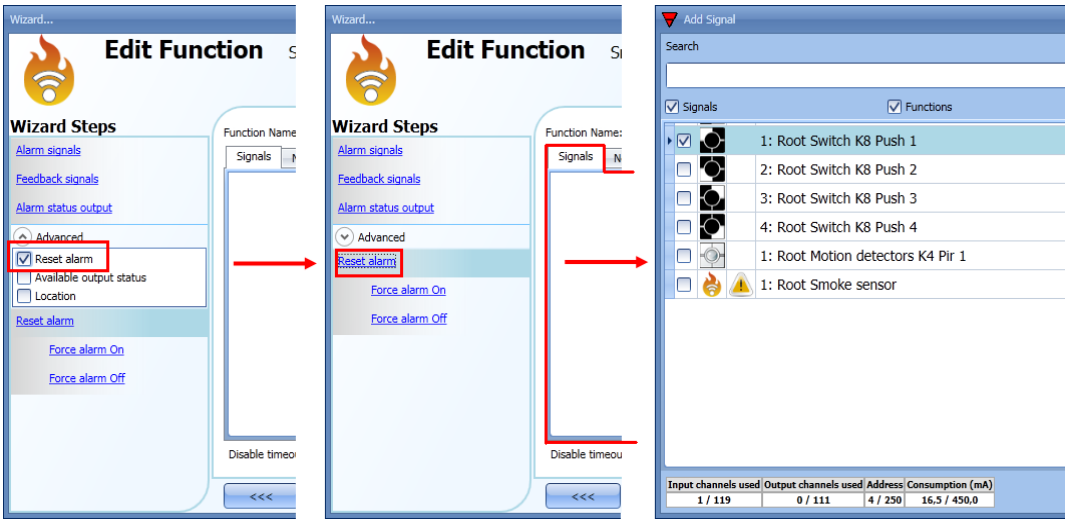
Once the alarm signal is entered, the user can also decide to use the inverted logic of the signal (see the red rectangle in the picture below). The inverted logic is enabled by selecting the green V.



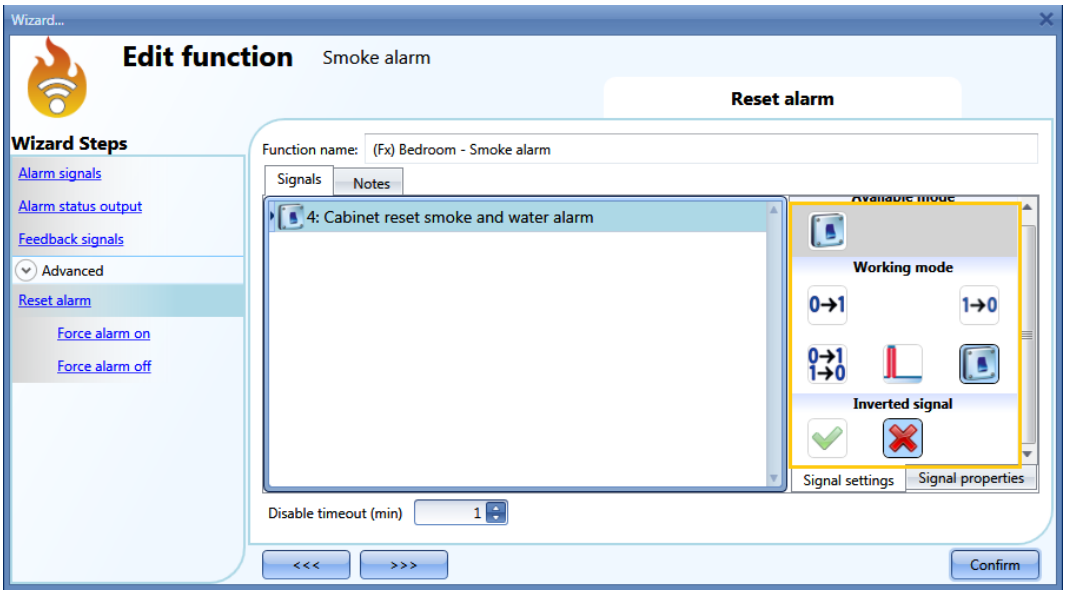
10.11.3 How to add the reset signal to inhibit the output status

When the reset signal is activated, the smoke alarm is inhibited.
If a smoke condition is detected by a sensor during the time period that the reset is active, the sensor will react locally, but the alarm will not be activated.
If the signal is still present when the reset signal timeout has expired, the alarm will be activated.

To select the reset signal, the user should enable it in the *Advanced* section, then click on *Reset alarm* and then double click on the Signals window: once the window is open, select the relevant reset signal from the list (see picture below).



When the reset signal is added, the user should select the working mode and the type of the event:



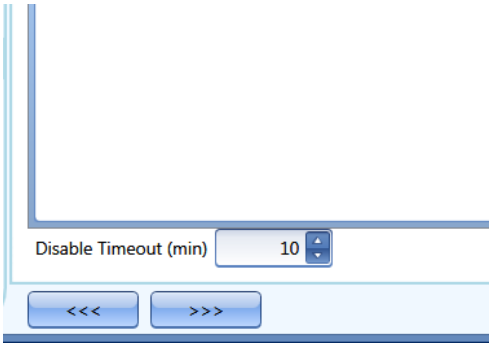
If a push button type is selected, the working mode can be defined following the table below:

Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
1→0	As soon as the push button is pressed, the function is disabled.	After a short press (less than 1 second), at the release of the push button the function is disabled.	After a long press, the function is disabled at the release of the push button.	After a very long press, the function is disabled at the release of the push button.
0→1	As soon as the push button is pressed, the function is enabled again.	After a short press (less than 1 second), the function is enabled again at the release of the push button.	After a long press, the function is enabled again at the release of the push button.	After a very long press, the function is enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the function will be enabled/disabled in toggle mode.	After a short press (less than 1 second), the function will be enabled/disabled in toggle mode at the release of the push button.	After a long press, the function will be enabled/disabled in toggle mode at the release of the push button.	After a very long press, the function will be enabled/disabled in toggle mode at the release of the push button.
	The function will be enabled/disabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should be done following the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The function is disabled	No action
1→0	The function is enabled again	No action
0→1 1→0	The function is disabled/enabled in toggle mode	No action
	The function is disabled/enabled in toggle mode	The function is disabled/enabled in toggle mode
	The function is disabled	The function is enabled

The user might need to set a time after which the smoke function is enabled again even if the selected signal is still active. To achieve this the *Disable timeout* field has to be set.



The timer starts every time the reset status is activated. The disable status is automatically deactivated as soon as it has expired.
In the figure below a disable timeout of 10 minutes is set. The maximum delay is 59 minutes.
If the timer is set to 0 minutes, the timer is not enabled.

10.11.4 How to use a siren function for the alarm output

The output of the smoke alarm function follows the status of the function: if the alarm is active, the output is on, if the alarm is not active, the output is off.
If a pulsating output is needed, such as a siren, the *Siren alarm* must be used.

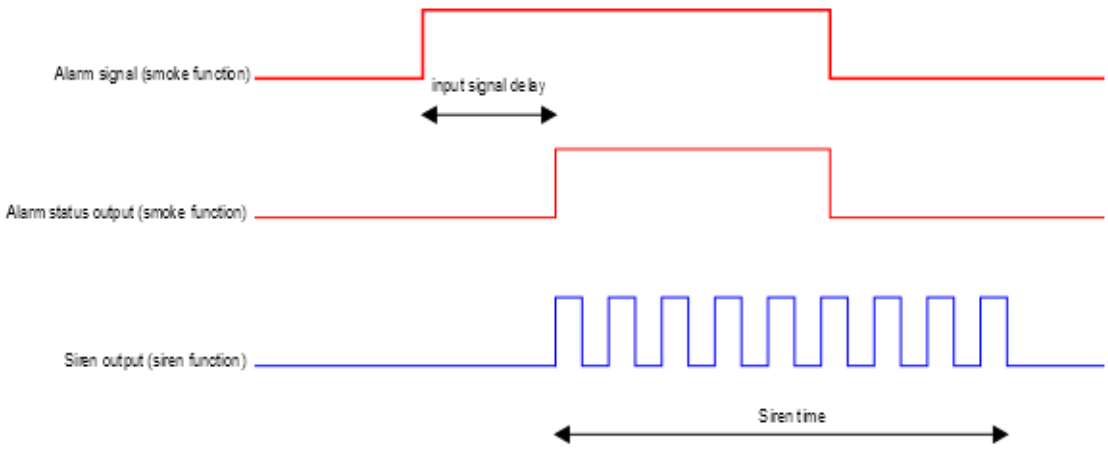
The *Siren alarm* function allows the user to change the activation time of the output, set a pulsating output and use only one output signal shared between more than one alarm functions.

The steps to add an *Alarm siren* function are as follows:

5) Create and configure the *smoke alarm* with its alarm signals
6) From the *Add* menu, select Alarm and click on *Siren alarm*
7) Configure the *Siren alarm* function (see also paragraph *How to set a Siren alarm function* for more details)

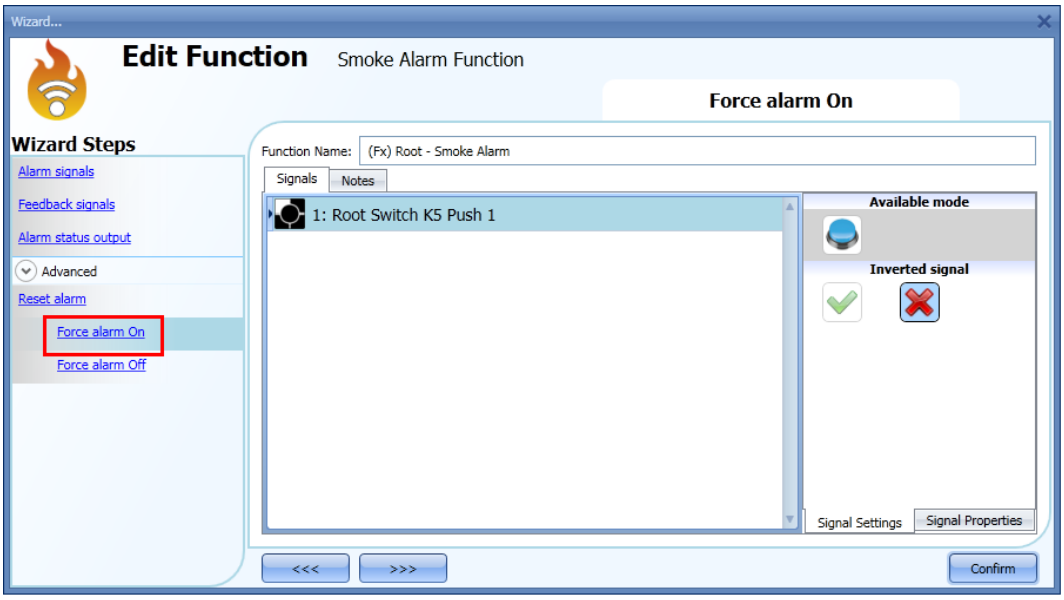
Once the siren alarm is configured, the output of the siren will be triggered when the alarm condition is activated. Activation of the reset signal in the smoke alarm function will reset the timer and the siren output.

Example1:
In the first example, the alarm siren function is triggered by the alarm signal. The siren output is not deactivated when the alarm signal goes OFF and it stays on until its timer has expired.



10.11.5 How to force the alarm status ON

If the user wants to force the alarm status ON regardless of all the other signals used in the function, the Force alarm On field has to be used: to enable it, select Reset in the Advanced section, select *Force alarm On*, then double click on the signal window and select the appropriate signal to use (see picture below).



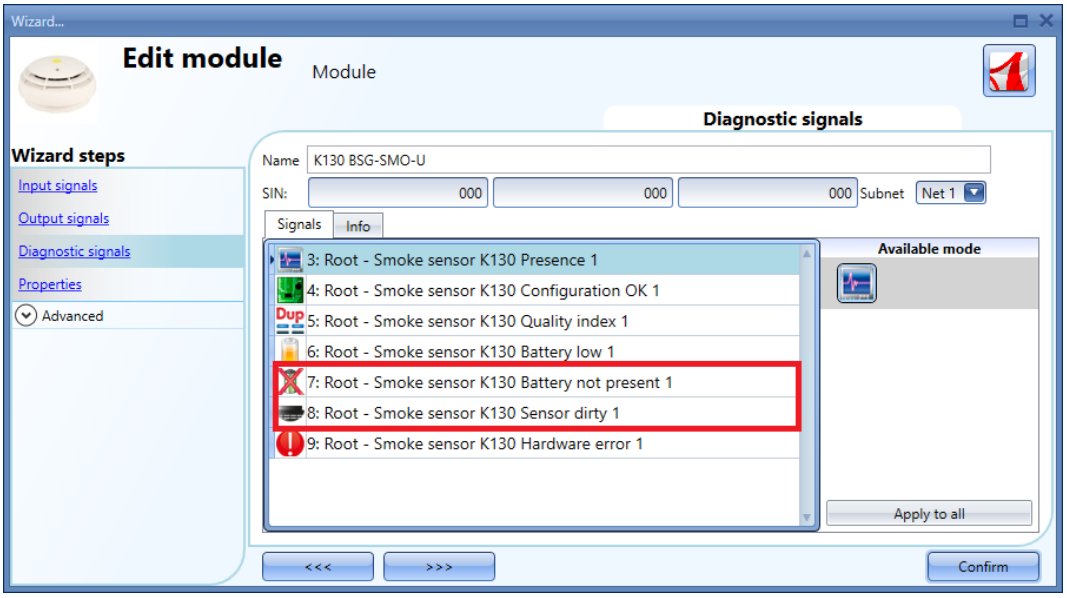
The signal used in the *Force alarm On* window works as a level signal: while the signal is active, the function is forced to the ON condition (the output stays active).
When the *Force alarm On* signal is activated and at the same time the *Force alarm Off* signal is activated, the *Force alarm On* signal has priority.

10.11.6 How to force the alarm status OFF

If the user wants to force the alarm status OFF regardless of all the other signals used in the function, the *Force alarm Off* field has to be used: to enable it, select *Reset* in the *Advanced* section, select *Force alarm Off*, then double click on the signal window and select the appropriate signal to use (see picture below).

The signal used in the *Force alarm Off* window works as a level signal: while the signal is active, the function is forced to the Off condition (the output stays off).
When the *Force alarm On* signal is activated and at the same time the *Force alarm Off* signal is activated, the *Force alarm On* signal has priority.

N.B. In order to avoid a false activation of the diagnostic signals *Battery not present* and *Sensor dirty*, the BSG-SMOx-U module has an internal delay filter set to 5 minutes (this time delay cannot be changed by the user): if a smoke detector module is disconnected from the Dupline bus, the status of these diagnostic signals will be shown after 5 minutes the connection has been re-established.

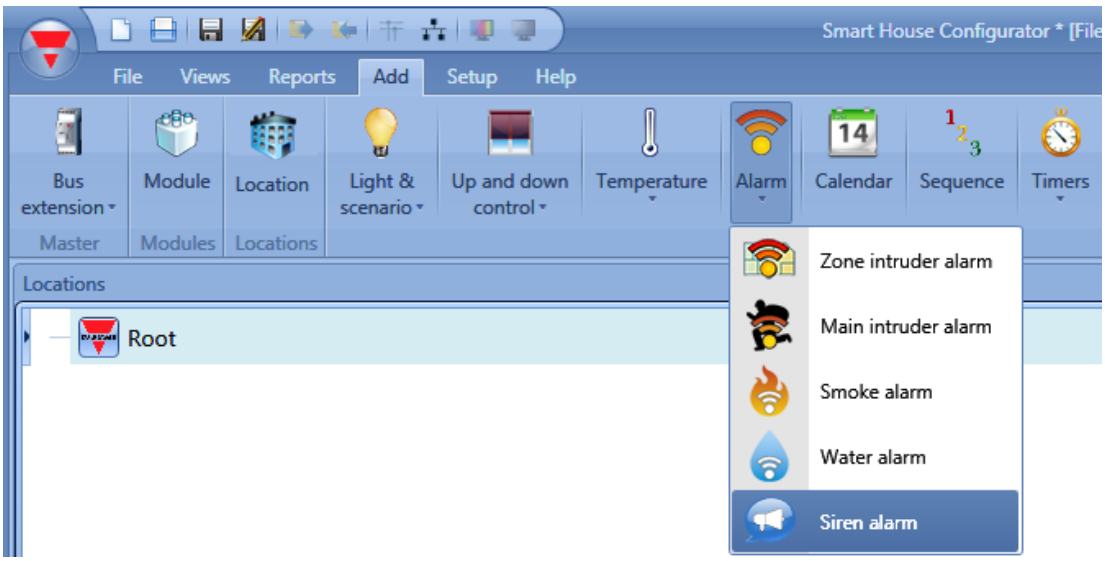


10.12 How to set a Siren alarm function

The *Siren alarm* function allows the user to manage an output when an alarm is detected. It has two purposes:

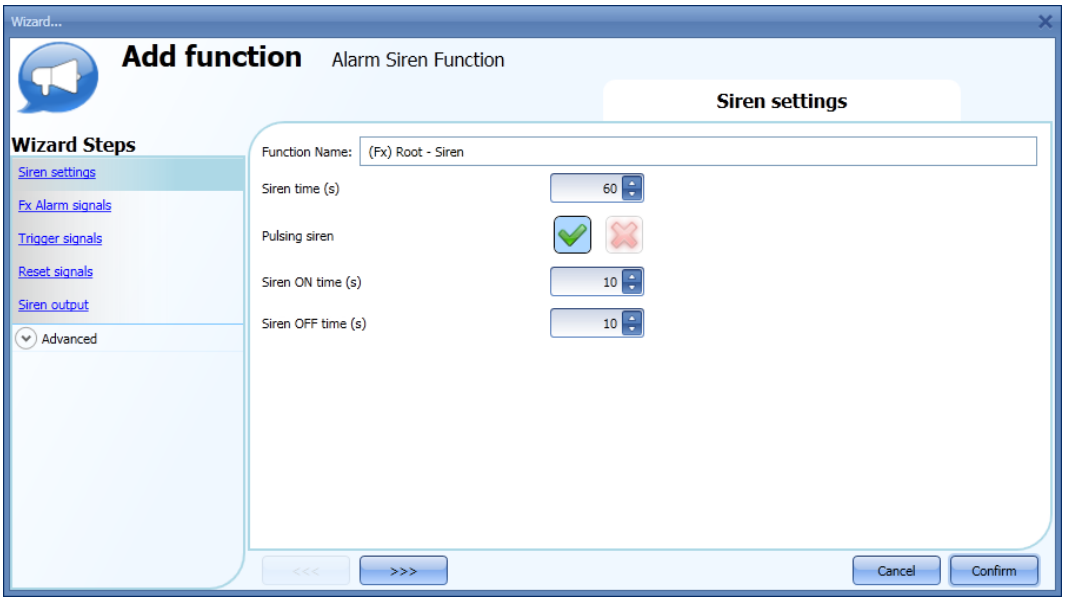
8) To allow the user to have the maximum flexibility for the activation of the output.
9) To allow the user to use a single output signal as a common output for more alarms.

To set up an Alarm siren function the user should select *Alarm* from the Add menu, then select *Siren alarm* (see picture below). The new function will be added into the selected location.



10.12.1 How to change settings for the siren output

The siren settings can be changed in the first step of the wizard (see picture below)

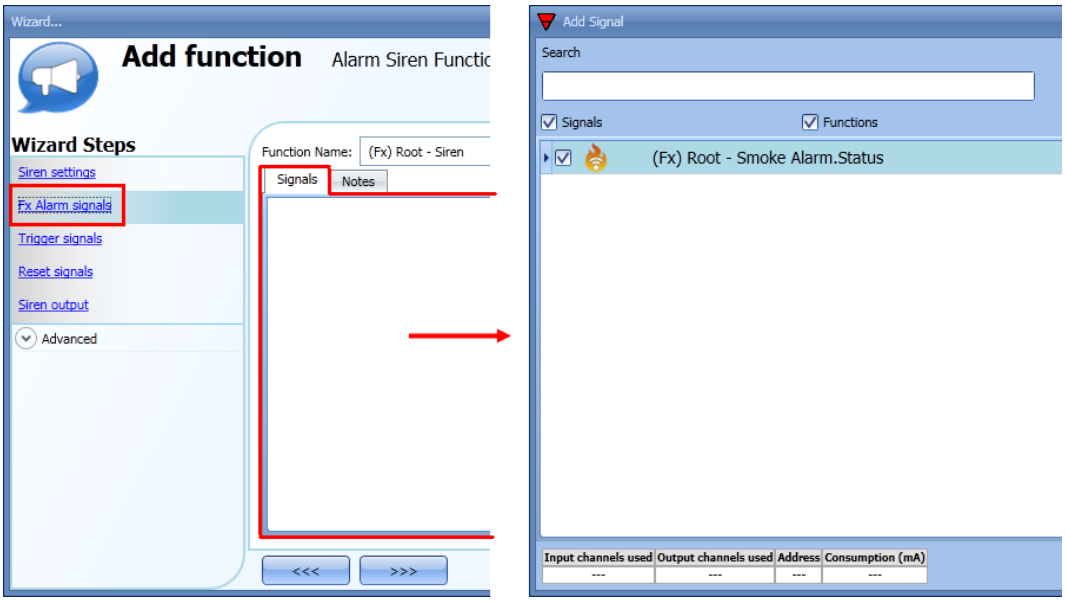


With the *Siren time* the user can define the activation time of the siren output when it is triggered by an

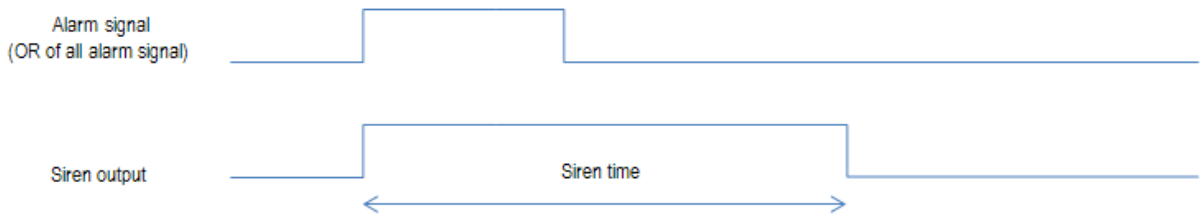
alarm function.
The user can also use a pulsing siren: if the green V is selected, the siren output will pulsate with the intervals selected in the *Siren ON time* and *Siren OFF time* fields.
If the red cross is selected, the output of the siren will stay ON until the siren time runs out.

10.12.2 How to link an alarm function to the siren

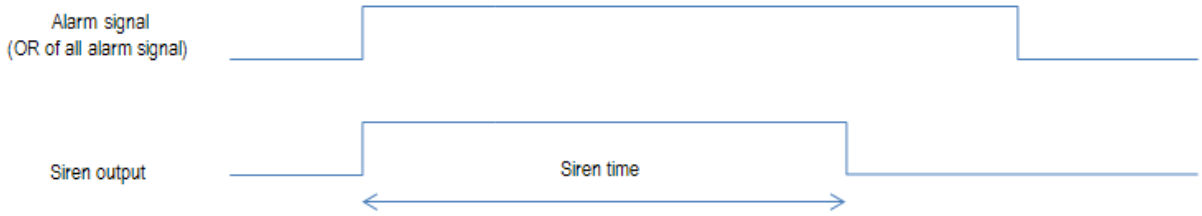
In the *Fx Alarm signals* field, the user can select the Alarm function that will trigger the Siren.
The functions used as the trigger work as event signals: as soon as they are ON, the siren also goes ON and stays On regardless of the status of the alarm functions.
The picture below shows how to add a function that triggers the siren function.



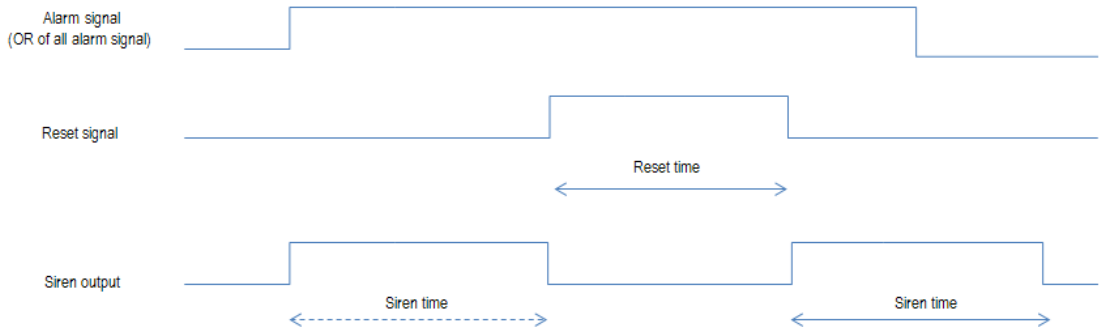
Example 1:
The siren is triggered when the alarm is activated, and stays active even if the alarm goes off.



Example2:
If the alarm signal is still active when the siren time runs out, the siren goes off.

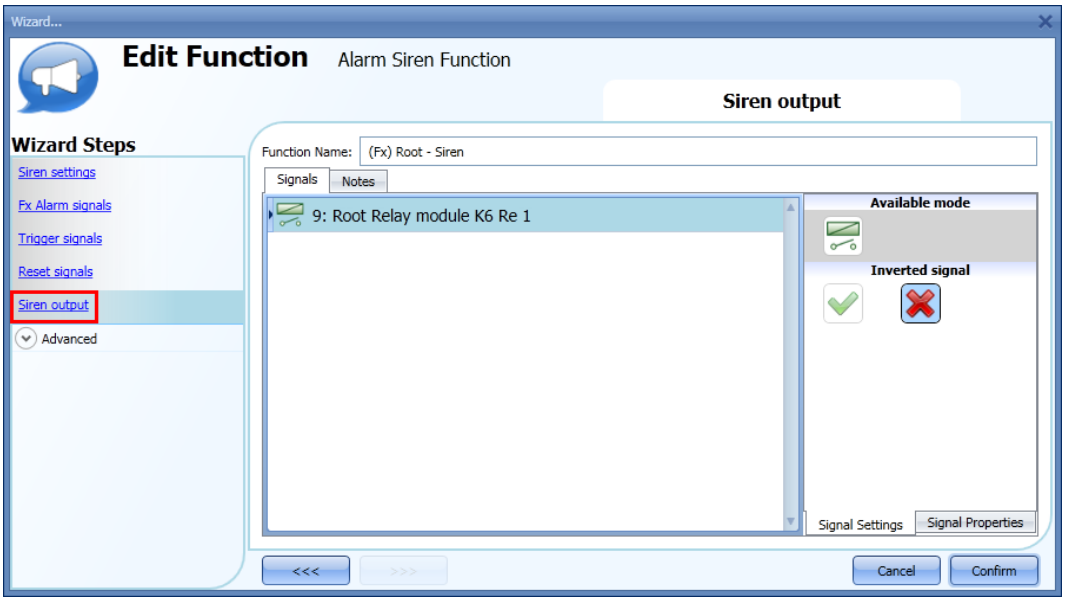


Example3:
If a reset signal is activated, the siren is inhibited during the reset time. Once it has expired, if the alarm signal is active, the siren is triggered again.



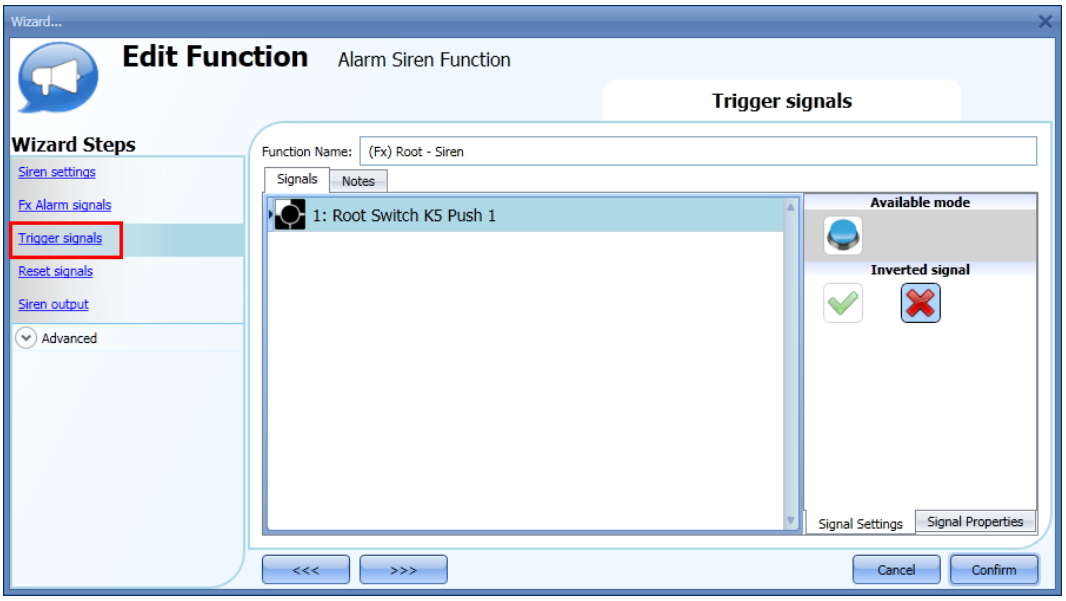
10.12.3 How to add the siren output

To add the output signal, the user has to select the relevant section in the wizard of the function, then double click on the *Signals* window and select the signal from the list of those available.
The output will follow the status of the function: when the siren is on, the output is activated according to the output settings; when the reset signal occurs or the siren function goes off, the output is deactivated.



10.12.4 How to use a manual signal to activate the siren

The siren function can also be activated with a manual signal instead of using alarm functions.
To add the triggering signal, the user should select the relevant section in the wizard of the function, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



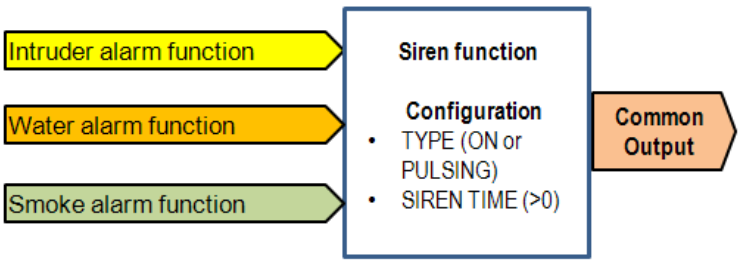
The behaviour of the signal used as the trigger works in the same way as described for the alarm functions: as soon as the trigger signal goes on, the siren is activated and stays on until the siren time has expired.

10.12.5 How to use more than one alarm with a common siren

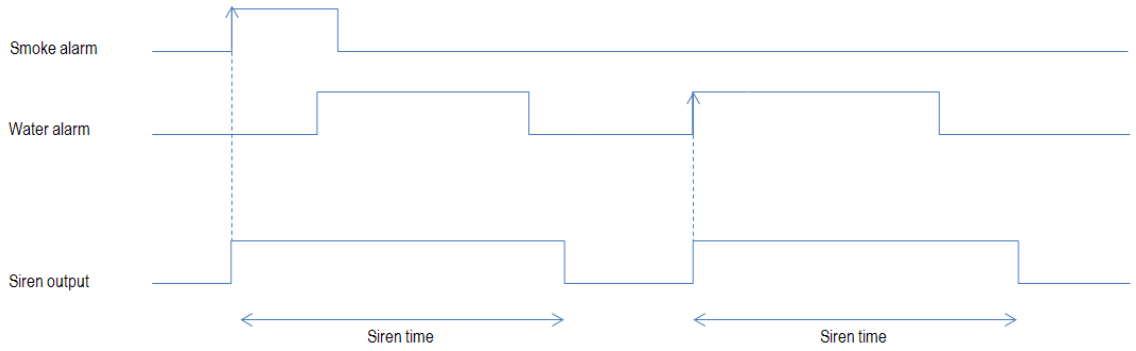
The siren alarm output can be used by different alarm functions.
The steps to create a common siren are the following:

1. Configure the alarm functions (intruder alarm, smoke alarm and water alarm)
2. When the alarm functions are configured, add the *Siren alarm* function
3. Configure the *Siren alarm* function
4. In the *Fx alarm signals* field enter all the created alarm functions
5. Add the common output in the siren function

The siren function is triggered when one of the configured alarm functions detects an alarm.
When an alarm is detected, the siren goes on and the output is turned on according to the configured parameters, regardless of the status of the alarm signal that triggered the siren (see picture below).

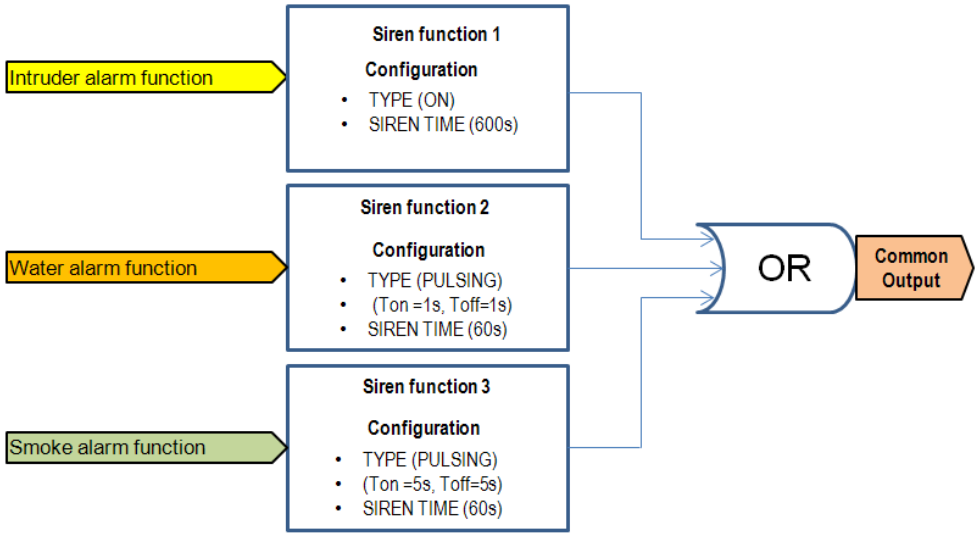


Example 4:
The first alarm event triggers the siren: all the other alarm events do not affect it if it is already on.



If the user needs to use different activations of the siren, in order to recognize the kind of the alarm triggering it, he must use different siren functions linked together with a multigate function.

Example 5:
In the picture below, three siren functions are created and linked together with a multigate function.
Using these settings, the output will be activated in three different ways according to the detected alarm.

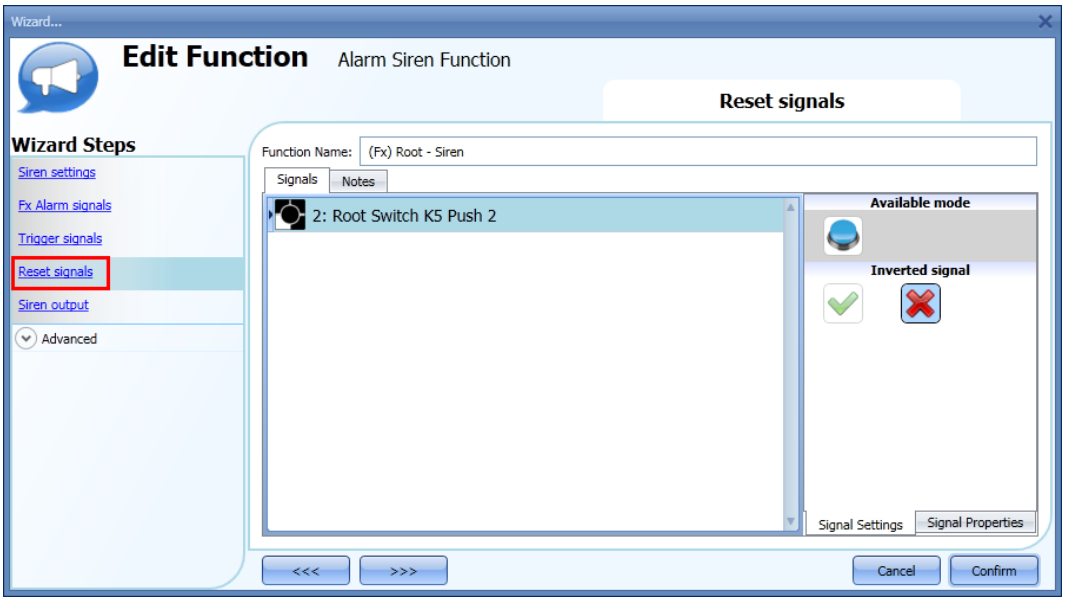


10.12.6 How to reset the siren

There are two possible ways to reset the siren function:

1. The reset is done automatically when the user activates the reset signal of the alarm function linked to the siren: the reset signal is automatically transmitted to the siren function. The siren stays off while the reset signal is active. When the reset time expires, the siren goes on as soon as a new alarm event is detected.

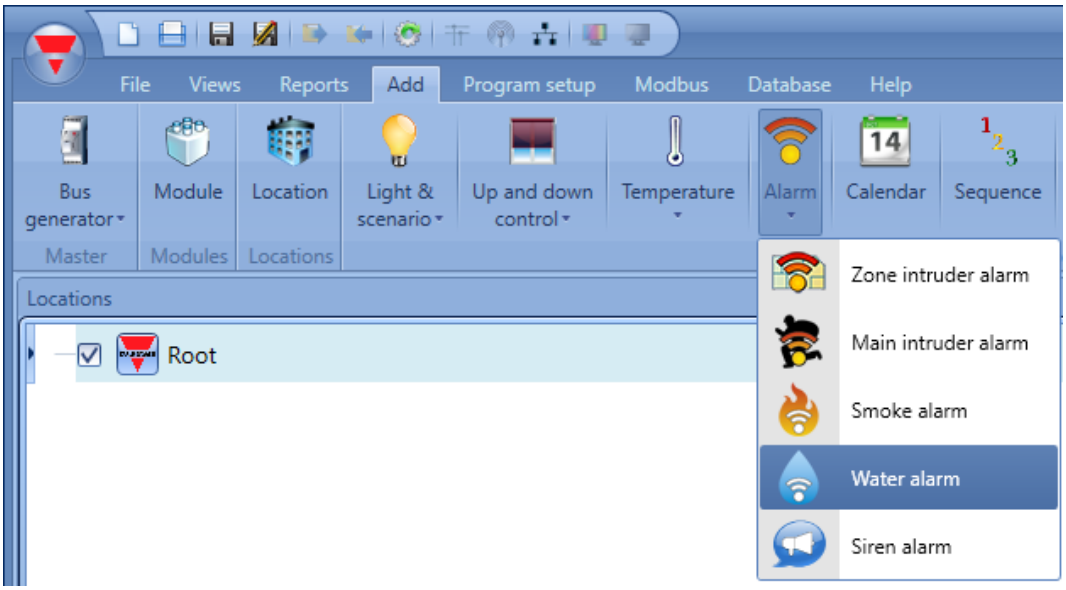
2. The reset is done manually. To add a manual reset the user has to select the relevant section in the wizard of the function, then double click on the *Signals* window and select the reset signal from the list of those available (see picture below).



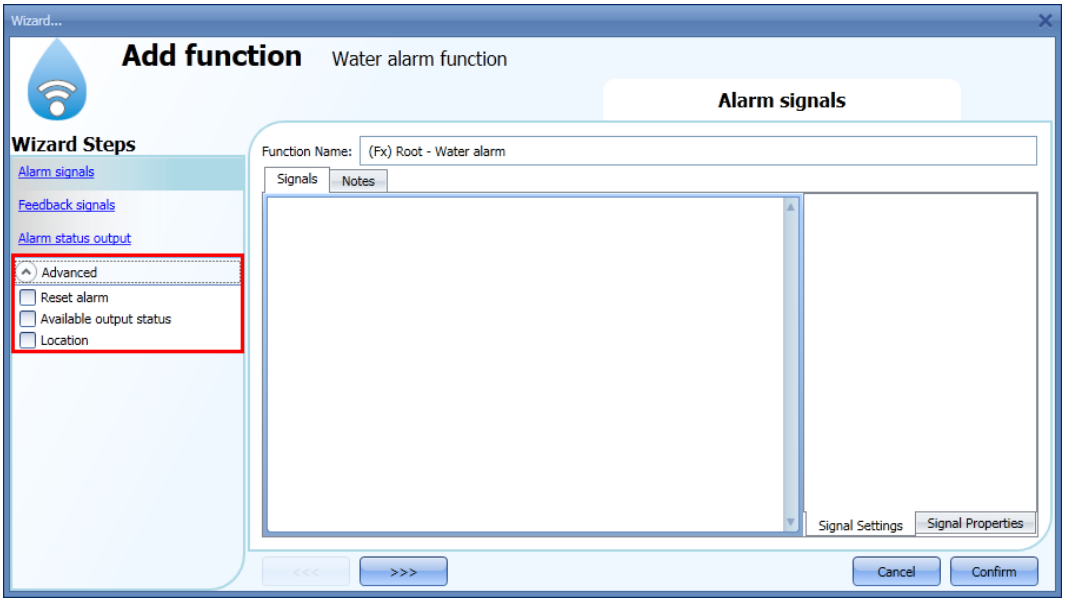
When the reset is activated, the siren is inhibited and stays in this condition until another alarm signal triggers the siren.

10.13 How to add a water alarm function

The user can configure a basic water alarm function in order to monitor water flood on the floor.
To set up a water alarm function the user should select *Alarm* from the Add menu, then select *Water Alarm* (see picture below). The new function will be added into the selected location.
The wizard relevant to a *Water alarm* function can also be opened by typing Alt +A +A +W.



This function manages one or more outputs controlled by one or more input commands. The command might be a signal from a water detector, a pushbutton, a switch, a function or a remote command (Webserver, sms, Modbus TCP/IP)
In the *Advanced* section, the user can select the reset signal: this is used to turn the output signal off to silence the alarm when it has been acknowledged. The alarm will be inhibited for a time period, allowing the user to remove the water from around the sensor in case of a false triggering.
In the *Advanced* section the user can also configure two signals in order to force the status of the function on or off.

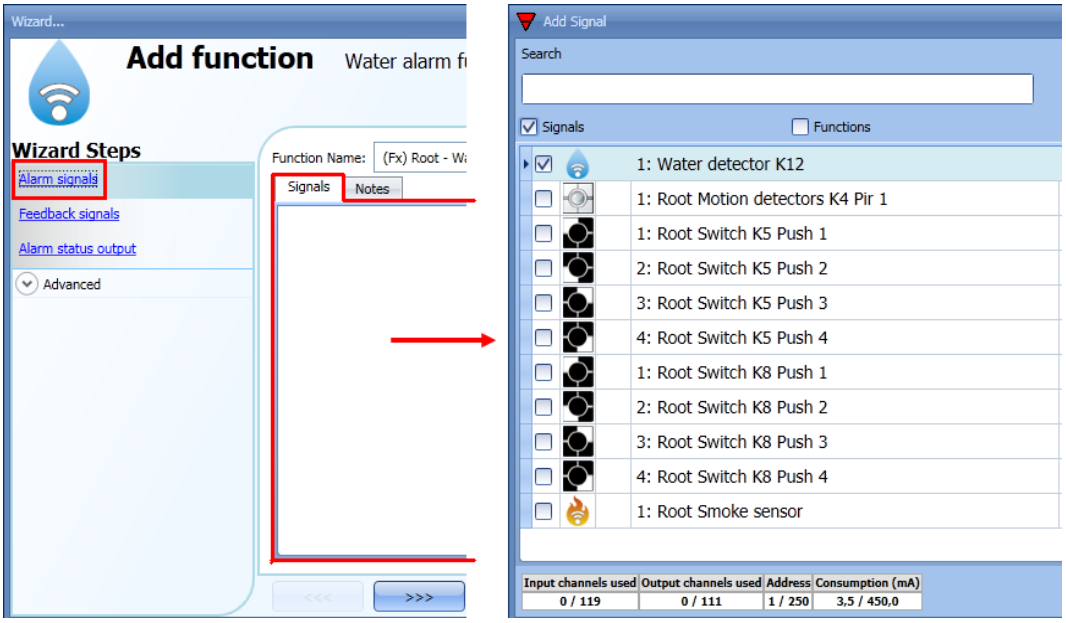


10.13.1 How to set a simple water alarm system

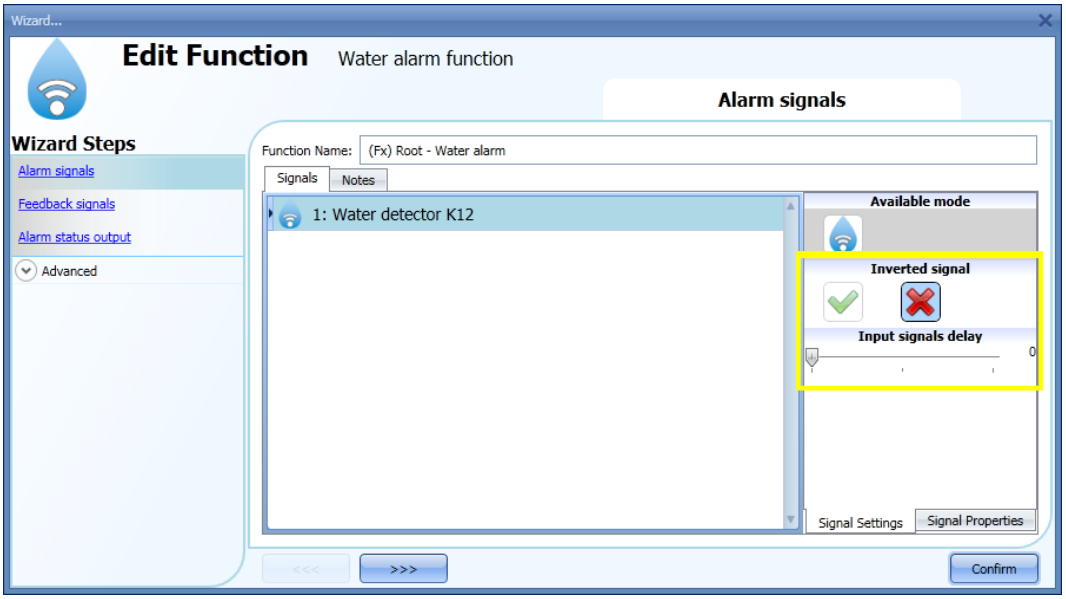
If the user wants to create a simple water alarm system, he needs at least two signals to have a fully functional water alarm system:

10) A water sensor as input signal
11) A general output that is activated when the alarm is on

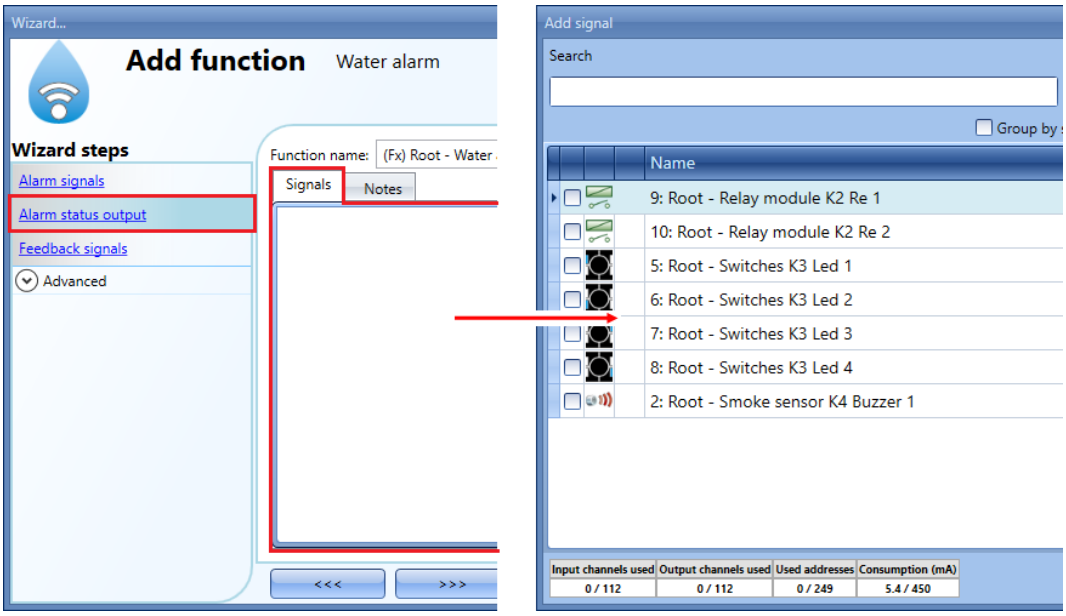
Once the modules are added to the configuration (water sensor and output module), the user has to select the alarm input signal in the relevant section of the *Water alarm* wizard, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



Once the alarm signal is entered, the user can also decide to use the inverted logic of the signal (see the red rectangle in the picture below). The inverted logic is enabled by selecting the green V.
Each alarm signal added to the function has a delay time that can be set in order to filter the activation of the alarm when the water is detected. This may be used to avoid false alarm triggering.



To add the *Alarm status output*, the user has to select the relevant section in the wizard of the function, then double click on the *Signals* window and select the output signal from the list of those available (see picture below).

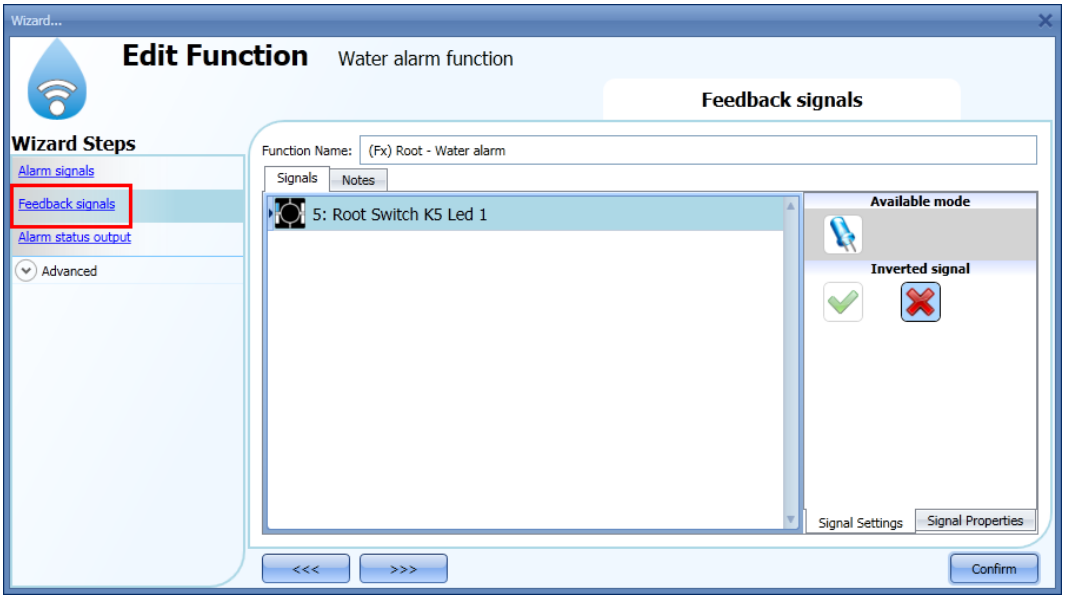


The output signal can be a relay, a LED or a buzzer. The output signal will stay on until water is detected by the sensor.

Example 1:
Using these easy settings, the function works in this way: the water function is ready to react as soon as the smart-house system is started. Until the sensor does not detect water, the output is off. When the sensor detects water, the function is activated and the output goes on and stays on until the water signal disappears.

10.13.2 How to add a feedback signal

If the user wants to use a feedback signal to check the status of the function, he has to select the relevant section, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).
The feedback signal can be a relay, a LED or a buzzer, and it behaves according to the status of the function: when the water is detected the feedback signal starts blinking 1 second ON 1 second OFF. When no water is detected the feedback is off: when the reset signal is active the feedback is on.

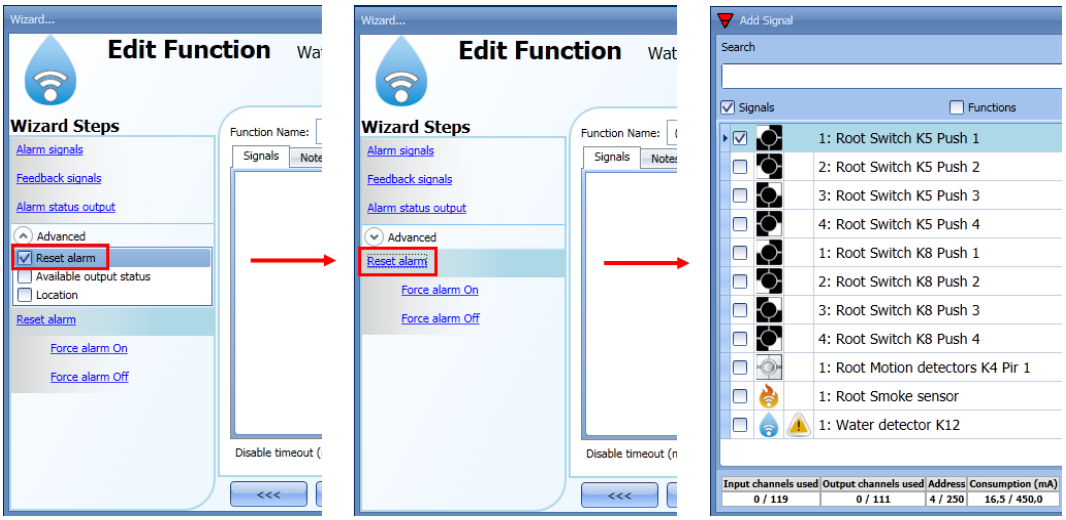


Once the alarm signal is entered, the user can also decide to use the inverted logic of the signal (see the red rectangle in the picture below). The inverted logic is enabled by selecting the green V.

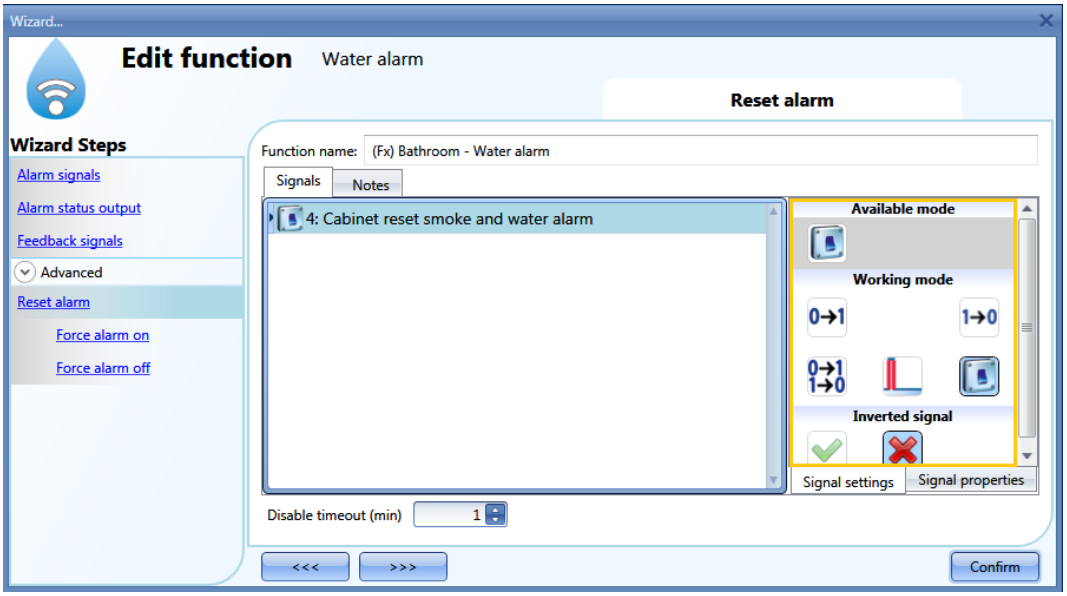
10.13.3 How to add the reset signal to inhibit the function status

When the reset signal is activated, the water alarm is inhibited.
If a water leakage event is detected by a sensor during the time period when the reset is active, the sensor will react locally, but the alarm will not be activated.
If the signal is still present when the reset signal timeout expires, the alarm will be activated.

To select the reset signal, the user has to enable it in the *Advanced* section, then click on *Reset alarm* and then double click on the Signals window: once the window is open, select the relevant reset signal from the list (see picture below).



When the reset signal is added, the user has to select the working mode and the type of the event:



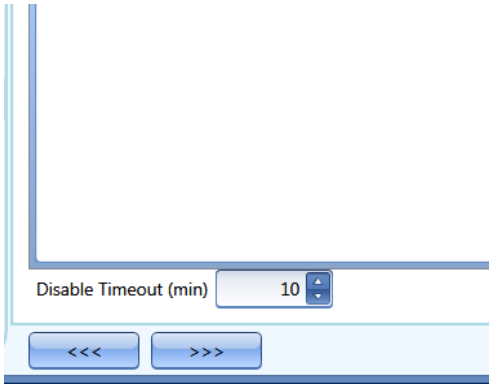
If a push button type is selected, the working mode can be defined following the table below:

Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, the function is disabled.	After a short press (less than 1 second), at the release of the push button. The function is disabled.	After a long press, the function is disabled at the release of the push button.	After a very long press, the function is disabled at the release of the push button.
1→0	As soon as the push button is pressed, the function is enabled again.	After a short press (less than 1 second), the function is enabled again at the release of the push button.	After a long press, the function is enabled again at the release of the push button.	After a very long press, the function is enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the function will be enabled/disabled in toggle mode.	After a short press (less than 1 second), the function will be enabled/disabled in toggle mode at the release of the push button.	After a long press, the function will be enabled/disabled in toggle mode at the release of the push button.	After a very long press, the function will be enabled/disabled in toggle mode at the release of the push button.
	The function will be enabled/disabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the set of the working mode has to be carried out following the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The function is disabled	No action
1→0	The function is enabled again	No action
0→1 1→0	The function is disabled/enabled in toggle mode	No action
	The function is disabled/enabled in toggle mode	The function is disabled/enabled in toggle mode
	The function is disabled	The function is enabled

The user might need to set a time after which the water alarm function is enabled again, even if the selected signal is still active. To do this, the *Disable timeout* field has to be set.



The timer starts each time the reset status is activated; the disable status is automatically deactivated as soon as it has expired.
In the figure below, a disable timeout of 10 minutes is set. The maxim delay is 59 minutes.
If the timer is set to 0 minutes, the timer is not enabled.

10.13.4 How to use a siren function for the alarm output

The output of the water alarm function follows the status of the function: if the alarm is active, the output is on; if the alarm isn't active, the output is off.
If a pulsating output is needed, such as a siren, the *Siren alarm* must be used.

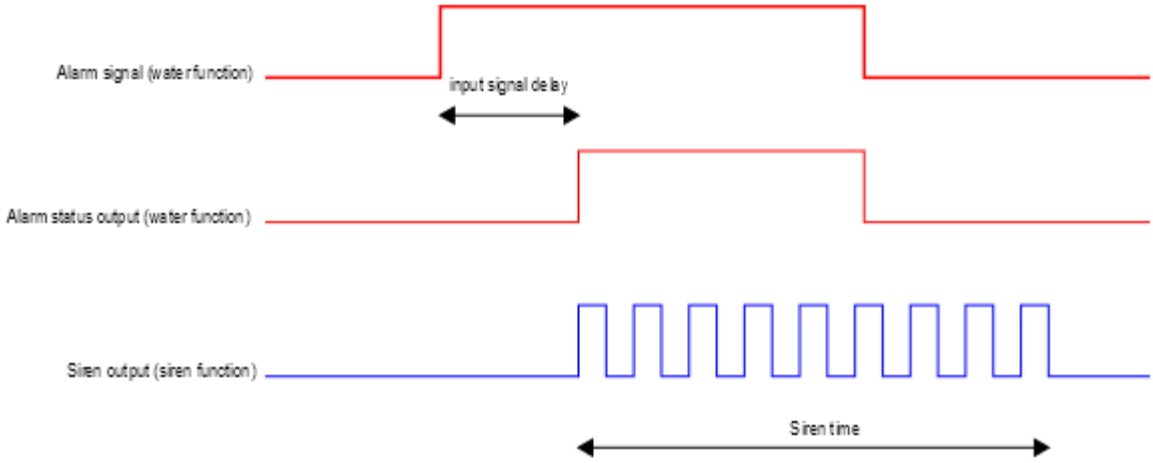
The *Siren alarm* function allows the user to change the activation time of the output, set a pulsating output and use only one output signal shared between more than one alarm functions.

The steps to add an *Alarm siren* function are as follows:

1) Create and configure the *water alarm* with its alarm signals
2) From the *Add* menu, select Alarm and click on *Siren alarm*
3) Configure the *Siren alarm* function (see also paragraph *How to set a Siren alarm function* for more details)

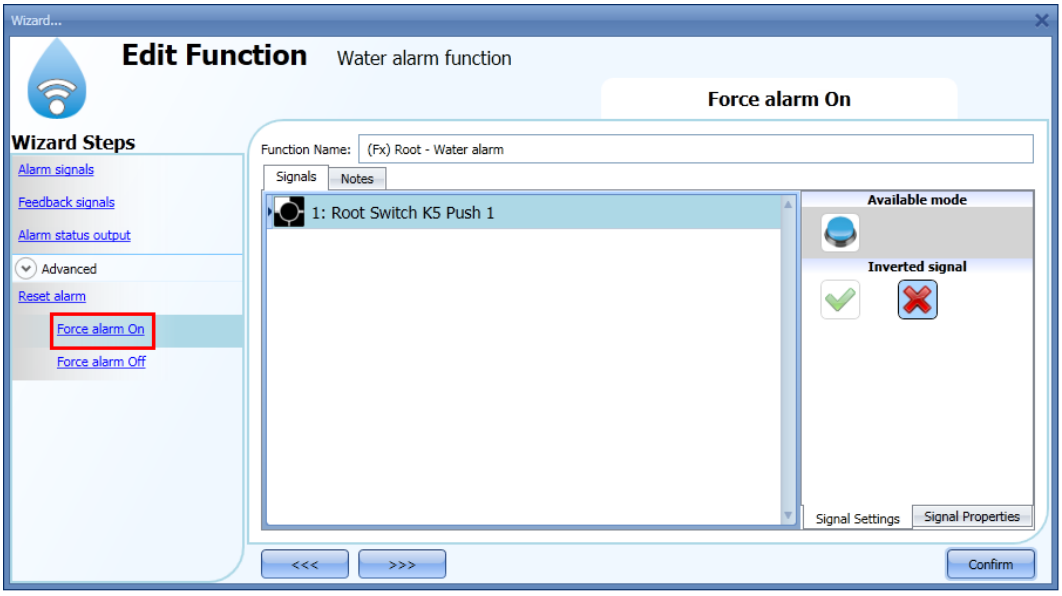
Once the siren alarm is configured, the output of the siren will be triggered when the alarm condition is activated. An activation of the reset signal in the water alarm function will reset the timer and the siren output.

Example1:
In the first example, the alarm siren function is triggered by the alarm signal. The siren output is not deactivated when the alarm signals goes OFF and it stays on until its timer has expired.



10.13.5 How to force the alarm status ON

If the user wants to force the alarm status ON regardless of all the other signals used in the function, the *Force alarm On* field has to be used: to enable it, select *Reset* in the *Advanced* section, select *Force alarm On*, then double click on the signal window and select the signal to use (see picture below).

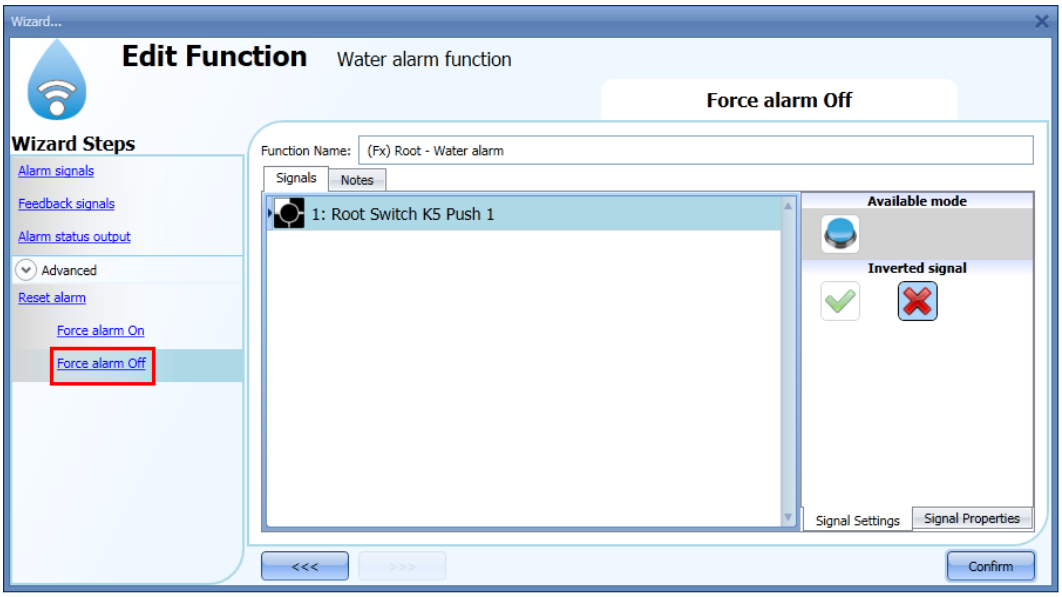


The signal used in the *Force alarm On* window works as a level signal: while the signal is active, the function is forced to the ON condition (the output stays active).
When the *Force alarm On* signal is activated and at the same time *Force alarm Off* signal is activated, the *Force alarm On* signal has priority.

10.13.6 How to force the alarm status OFF

If the user wants to force the alarm status OFF regardless of all the other signals used in the function, the *Force alarm Off* field has to be used: to enable it, select *Reset* in the *Advanced* section, select *Force alarm Off*, then double click on the signal window and then select the signal to use (see picture below).

The signal used in the *Force alarm Off* window works as a level signal: while the signal is active, the function is forced to the Off condition (the output stays off).
When the *Force alarm On* signal is activated and at the same time *Force alarm Off* signal is activated, the *Force alarm On* signal has priority.

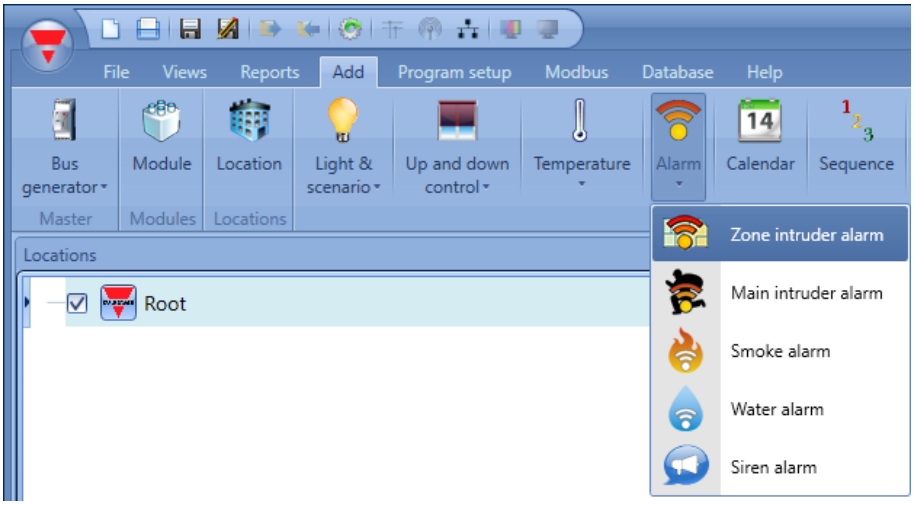


10.14 How to set an intruder alarm function

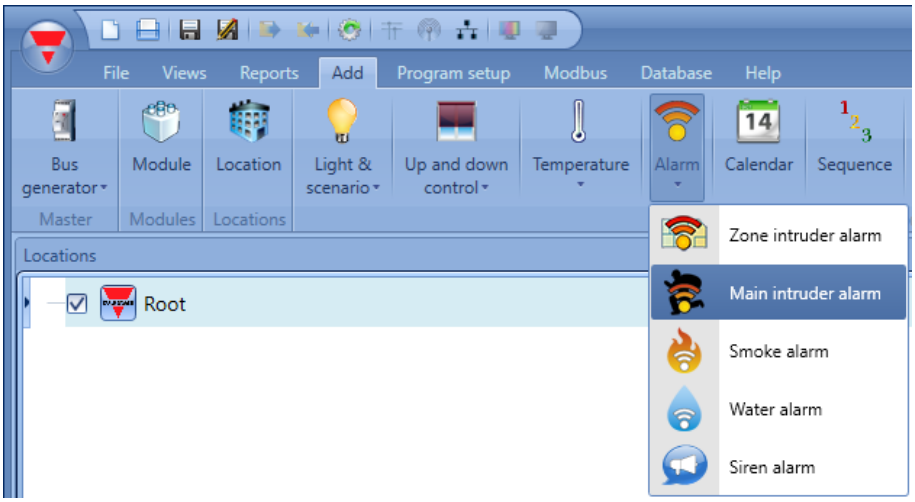
The intruder alarm function is used to protect the house against burglars and undesired intruders.
Two main steps have to be configured to create an intruder alarm function:

4) In the first step the user has to create at least one zone alarm function; each zone function might correspond to a part of the house that has to be monitored or just a single sensor.
5) In the second step the user has to create a main alarm function; the main alarm function is used to manage all the zone functions (it is used for arming/disarming and collecting all the zone statuses).

To set up an intruder alarm function, the user has to select *Alarm* from the Add menu, then select *Zone intruder alarm* (see picture below).
The wizard relevant to the zone alarm function can also be opened by typing Alt +A +A +Z (See table of short cuts).



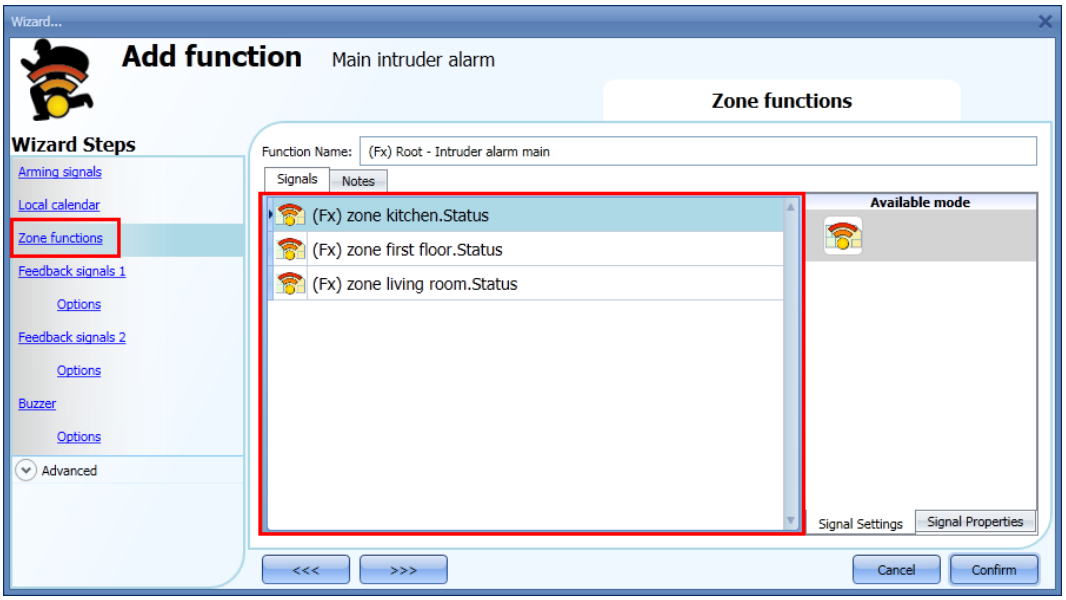
Once the zone alarm is added, the user has to select *Alarm* from the Add menu, then select *Main intruder alarm* (see picture below).
The wizard relevant to a main alarm function can also be opened by typing Alt +A +A +M .



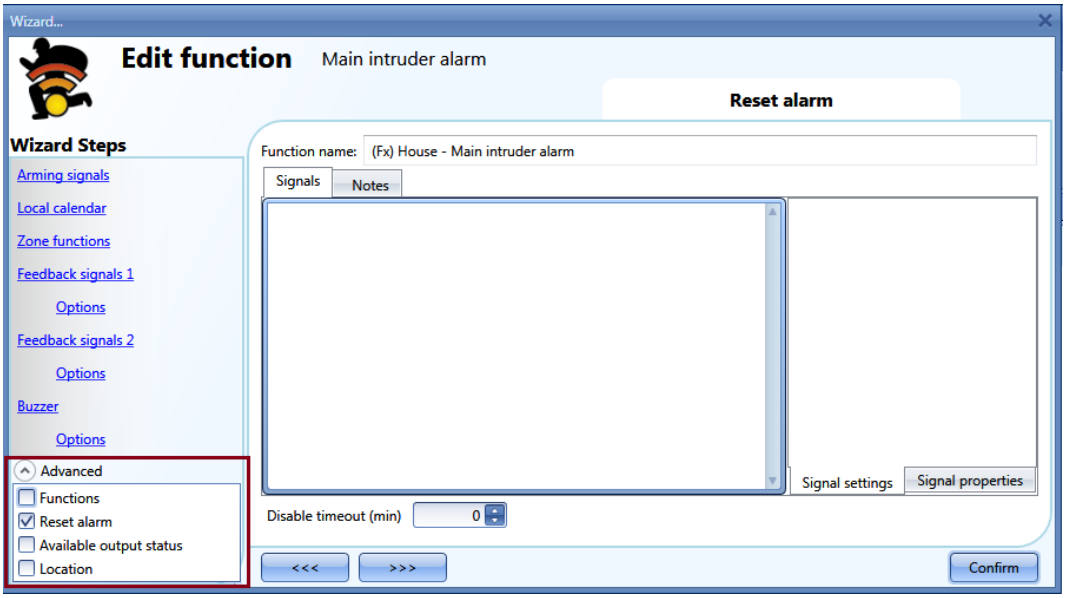
The first step to do is to add the arming signals to arm the alarm and this can be done in three different ways:

1) *Arming signals*: signals, functions and global calendar function programmed as level signals can be used.
2) *Local calendar*: a local calendar can be set
3) In the *Advanced area, Functions:* global calendars and function can be added

The last step is to link the zone alarm to the main alarm function. To do this the user has to open the Main alarm function, select the field "*Zone functions*", double click on the signals window, and select the required zone function from the list of the available zones (see picture below). More than one zone function can be linked to the same *main alarm* function.



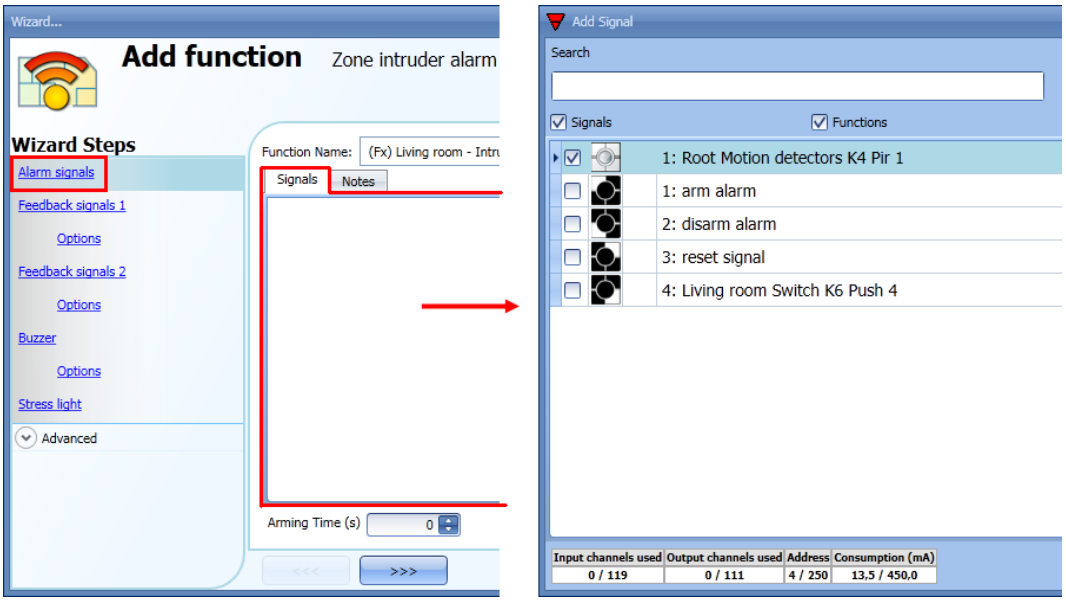
In the *Advanced* section, the user can select the reset signal: this is used to turn the output signal off to silence the alarm when it has been acknowledged. The alarm will be inhibited for a time period, allowing the user to verify the alarm situation.
In the *Advanced* section the user can also configure two signals in order to force the status of the function on or disarm it.


Wizard...
Edit function
Main intruder alarm
Reset alarm
Wizard Steps
Arming signals
Local calendar
Zone functions
Feedback signals 1
Options
Feedback signals 2
Options
Buzzer
Options
Advanced
Functions
Reset alarm
Available output status
Location
Function name:
(Fx) House - Main intruder alarm
Signals
Notes
Signal settings
Signal properties
Disable timeout (min)
0
<<<
>>>
Confirm

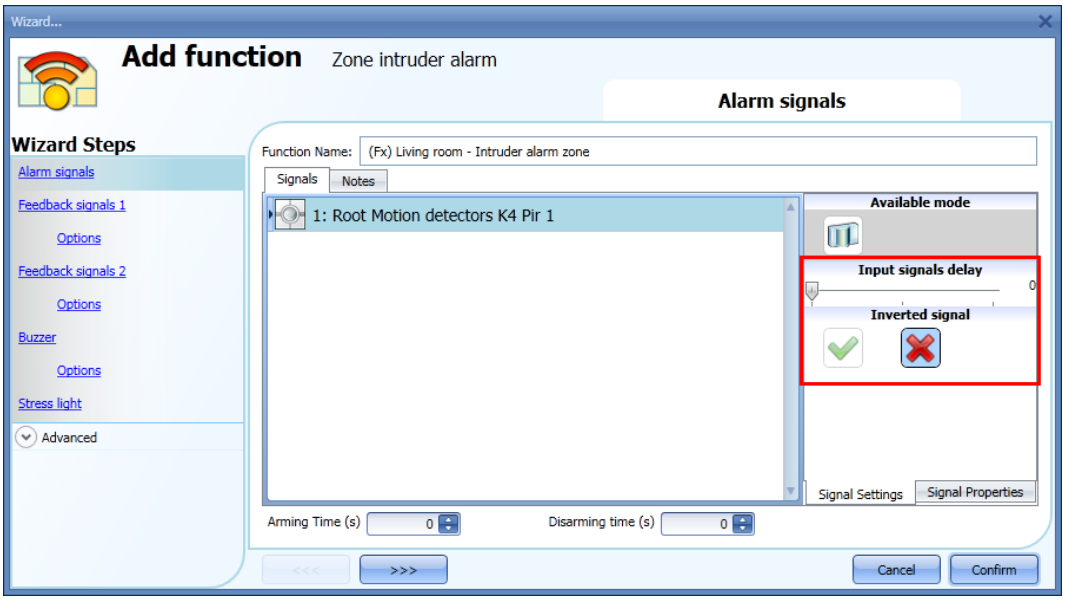
10.14.1 How to add the alarm signals

All the alarm signals have to be entered in the *zone intruder alarm* function.

In the *Alarm signals* field, the user can add all the signals that monitor that zone.
These signals may be window sensors, PIR sensors, push-buttons or other input devices.
To add a new alarm signal, the user has to select the relevant section in the wizard of the zone intruder alarm function, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



Once the alarm signal is entered, the user can also decide to use the inverted logic of the signal (see the red rectangle in the picture below). By selecting the green V, the inverted logic is enabled.
Each alarm signal added to the function has a delay time that can be set in order to filter the activation of the alarm when a burglar is detected; this might be used to avoid false alarm triggering.



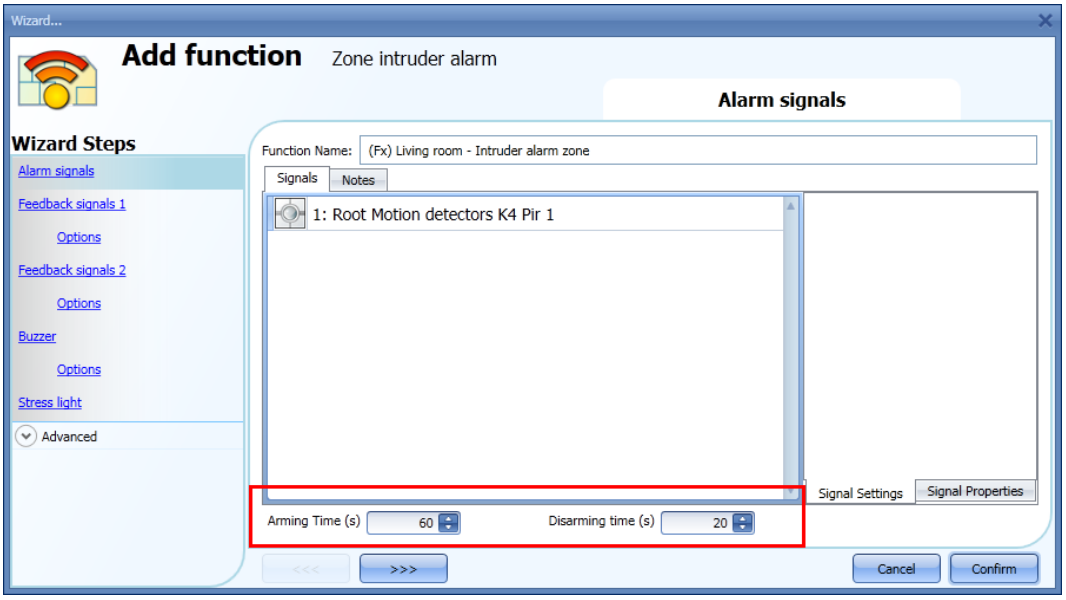
The number of detected pulses needed to give the alarm condition has to be set in the module properties wizard.
The delay set in the field *Input signal delay* is a further delay that can be added.

10.14.2 How to set the arming and disarming time

The arming time is the time needed for the home owner to leave the house: it is the time period from when the owner arms the alarm to when it reacts on an alarm signal.

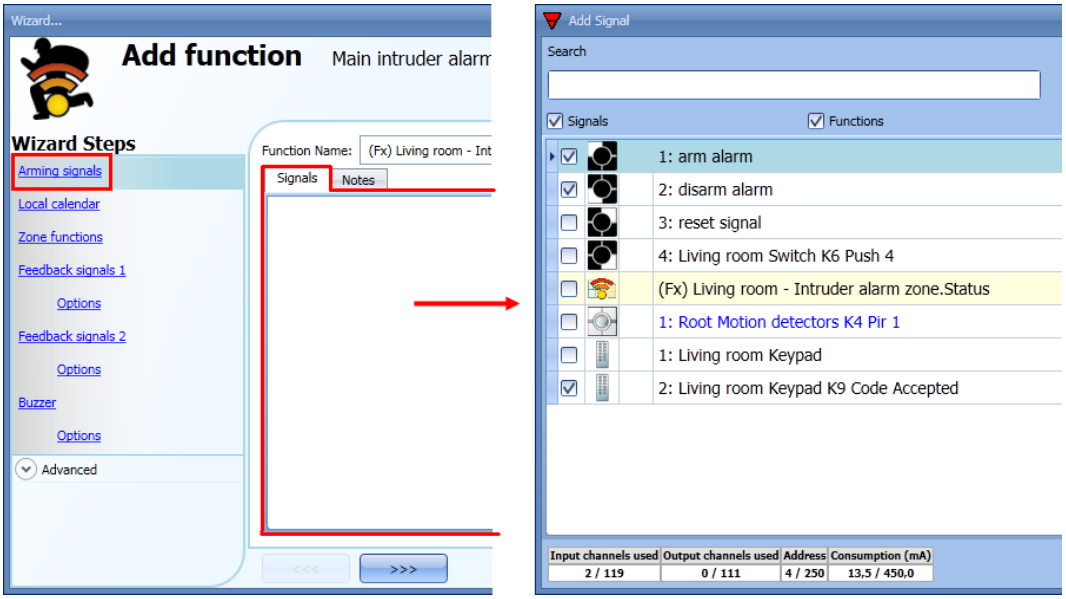
The disarming time is the time period that runs from when the alarm signal is activated to the alarm sounds. This can be used to delay the alarm when the user enters the room where he has the PIR and the alarm on/off switch, giving him time to deactivate the alarm when he comes home.

The arming and disarming time can be set in the alarm signals window in the *zone intruder alarm* function (in the picture below, the zone in the living room has an arming time of 60 seconds and the disarming time is set to 20 seconds).
These times are defined in each zone function. If they are set to 0 seconds, the reaction of all the alarm signals are immediate (all signals are armed immediately and the alarm sounds as soon as the alarm signal is activated).

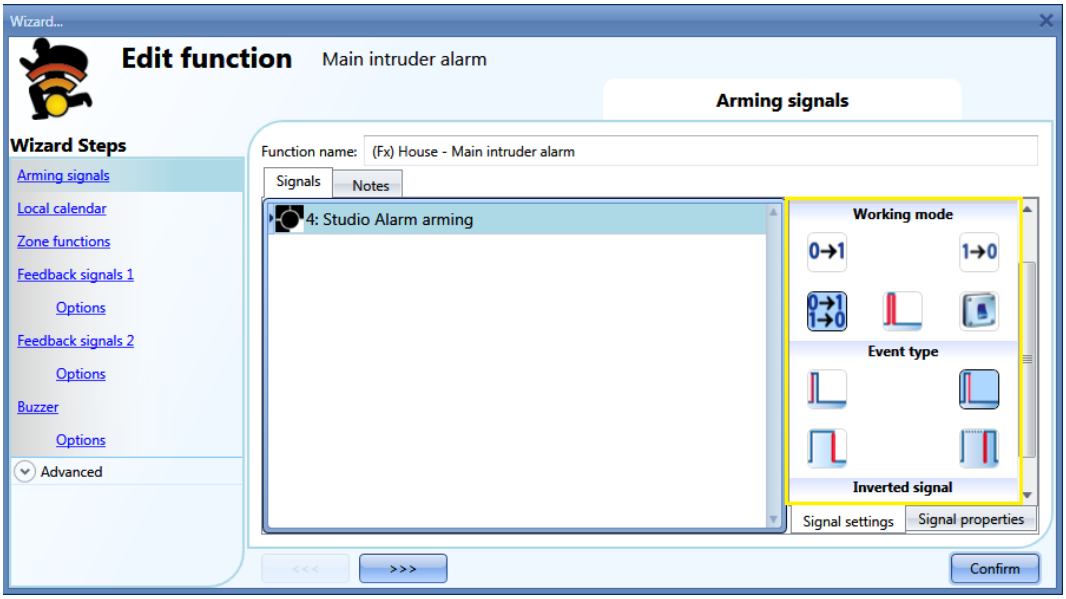


10.14.3 How to arm and disarm the intruder alarm

The arming/disarming signals should be entered in the *main intruder alarm* function and they are automatically used in each zone function linked to the main intruder alarm function. When the arming/disarming signal is activated, all the linked zones are automatically armed/disarmed.
To add the arming/disarming signal, the user should select the relevant section in the wizard of the *main intruder alarm* function, then double click on the *Signals* window and select the input signal from the list of available signals (see picture below).



When the arming signal is activated, all the linked zone functions are automatically armed; each zone is armed according to the arming time set by the user.



If a push button type is selected, the working mode can be defined following the table below:

Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, the function is armed.	After a short press (less than 1 second), at the release of the push button the function is armed.	After a long press, the function is armed at the release of the push button.	After a very long press, the function is armed at the release of the push button.
1→0	As soon as the push button is pressed, the function is disarmed.	After a short press (less than 1 second), the function is disarmed at the release of the push button.	After a long press, the function is disarmed at the release of the push button.	After a very long press, the function is disarmed at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the function will be armed/disarmed in toggle mode.	After a short press (less than 1 second), the function will be armed/disarmed in toggle mode at the release of the push button.	After a long press, the function will be armed/disarmed in toggle mode at the release of the push button.	After a very long press, the function will be armed/disarmed in toggle mode at the release of the push button.
	The function will be armed/disarmed in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation will be armed when the signal goes ON, and it is disarmed again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should be carried out following the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The function is armed	No action
1→0	The function is disarmed	No action
0→1 1→0	The function is Armed/disarmed in toggle mode	No action
	The function is armed/disarmed in toggle mode	The function is armed/disarmed in toggle mode
	The function is armed	The function is disarmed

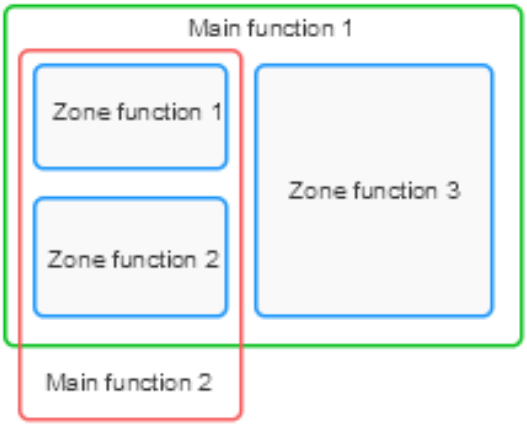
10.14.4 How to manage zones using different arming/disarming signals

The user can have a part of the house secured with the intruder alarm when he is at home.
An example could be a house with two floors, where the owner wants to have the alarm active on the ground floor during the night when he is sleeping. This means that the user can walk around on the 1st floor without triggering the alarm, while the ground floor is secured.

To create this, the user must enter two main alarm functions:

6) One function is used to arm/disarm all zones
7) The other function is used to arm/disarm the zones on the ground floor

In the picture below, the scheme of the example is shown.



The “main alarm function 1” is used to arm/disarm all the zones (all zones must be linked in the function). For example, the alarm signal could be the keypad at the entrance of the house, used for arming the entire system when the user leaves the house.
The “main function 2” is used to arm/disarm zone 1 and zone 2 (zone 3 is not linked to this function). For example, the alarm signal could be a pushbutton on the first floor used to arm the zones on the ground floor. This could also be done automatically by using a calendar.
The arming time of each zone is always the same, regardless of the signal that activates the arming condition.

10.14.5 Output status of the intruder alarm

The intruder alarm is designed to provide a complete overview of what is happening in the house, allowing the owner to monitor both main and zone functions.

The status of each zone function is the result of the logical *OR* of all its alarm signals.

Table of the zone intruder alarm function statuses:

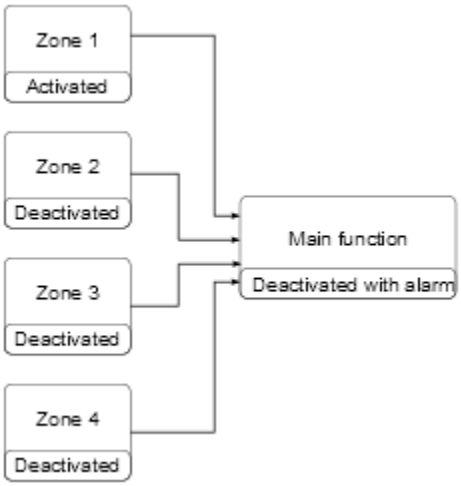
Status (This is the result of the logical OR of all the alarm signals in the zone function)	Description
Deactivated with no sensor active	All the alarm signals present in the zone function are not active
Deactivated with sensor active	At least one alarm signal in the zone is on, but the alarm is not armed
Arming	The zone is arming and all its alarm signals are off
Arming with one sensor active	The zone is arming and at least one alarm signal is on
Armed	The zone is armed and all its alarm signals are off
Disarming	The zone is disarming and at least one alarm signal is on
Alarm	The zone is armed and at least one alarm signal is on

The main intruder alarm function status is the result of the combination of the statuses of its linked zone functions.

Table of the main intruder alarm function statuses:

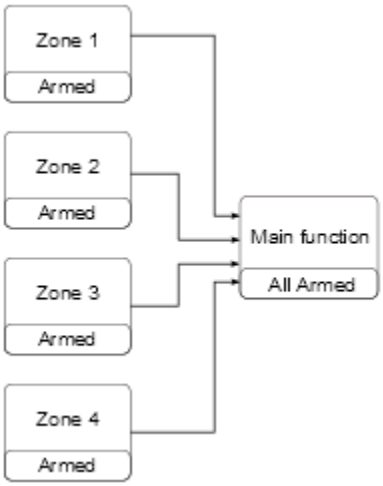
Status (This is the result of the logical OR of all alarm signals in the zone function)	Description
Deactivated with no sensor active	All the alarm signals used in all the linked zone functions are off
Deactivated with sensor active	At least one alarm signal used in a linked zone is on
Arming in at least one zone	At least one linked zone is arming, no alarm signals are on
Armed in all zones	All the linked zones are armed with all alarm signals off
Armed in at least one zone	Not all the linked zones armed, no alarm signals are on
Arming with one sensor active (warning)	At least one linked zone is arming with at least one alarm signal on
Disarming	At least one linked zone is disarming with at least one alarm signal on
Alarm in at least one zone	At least one linked zone is armed with an activated alarm signal
Waiting for a reset and previously in alarm	When one of the linked zones changes its status from an alarm condition (alarm on) to a sensor no longer activated, the status is latched until the reset signal occurs.

Example 1:
The alarm is off because the user is in the house.
In zone 1 one PIR sensor detects the presence of the owner, while all other zone statuses are deactivated because there are no people in the rooms.
The alarm status of the main function is deactivated because in zone 1 there is an alarm signal active.

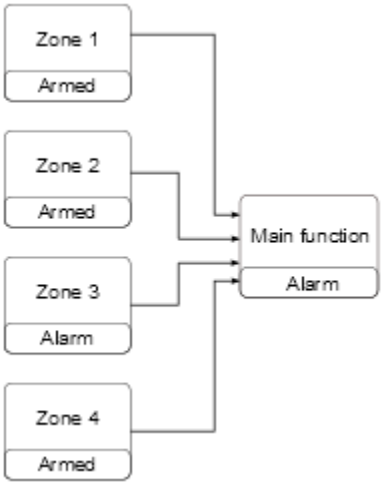


The home owner leaves the house and uses the keypad placed in zone 1 to arm the alarm.
All the zones are arming: zone 4 is immediately armed because the arming time is set to 0 s, zones 2-3 are arming with no alarm signals active, zone 1 status is warning because the owner is detected by the PIR sensor.
The status of the function is Warning. If the owner leaves zone 1 within the arming time, zone 1 status becomes “arming”.

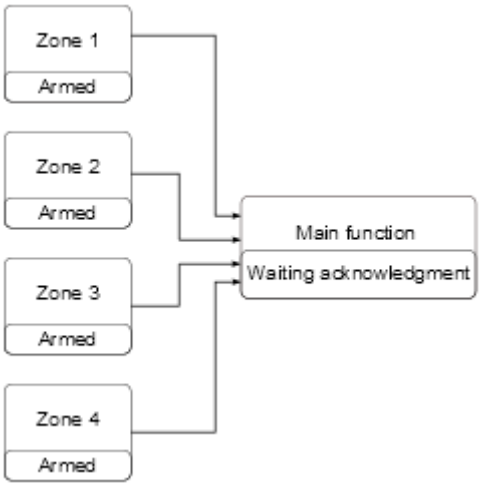
When all the zones enter the armed status, the status of the main intruder alarm function becomes “All armed”.



When all zones are armed, if an intrusion is detected in zone 3, the status changes to “pre-alarm” and then when the disarming time expires the zone goes into “alarm”. The alarm status is transmitted to the main function and the main status becomes “alarm”.



If the alarm input signal is no longer active, the previous alarm condition is still saved and the status of the main function is “Waiting for acknowledgment” to advise the home owner that an intrusion has been detected. This status is reset when the main alarm function is disarmed.

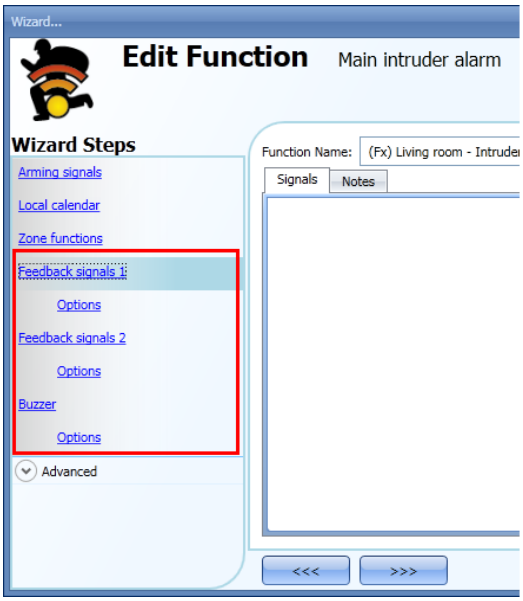


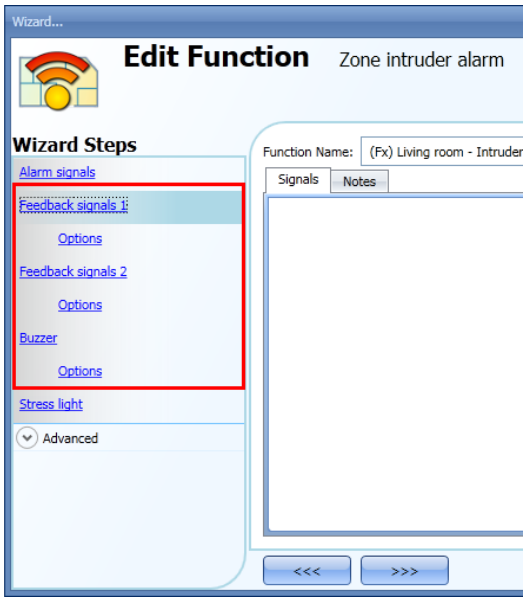
10.14.6 How to set a feedback signal

The feedback signal can be used to warn the user about the status of the intruder alarm.
The feedback signals are managed from both the zones and the main functions.

When the user wants to monitor the status of a specific zone, he should use the feedback signals in the zone function. If the user wants to monitor the status of the main intruder alarm, he should configure the feedback signals in the main alarm function.

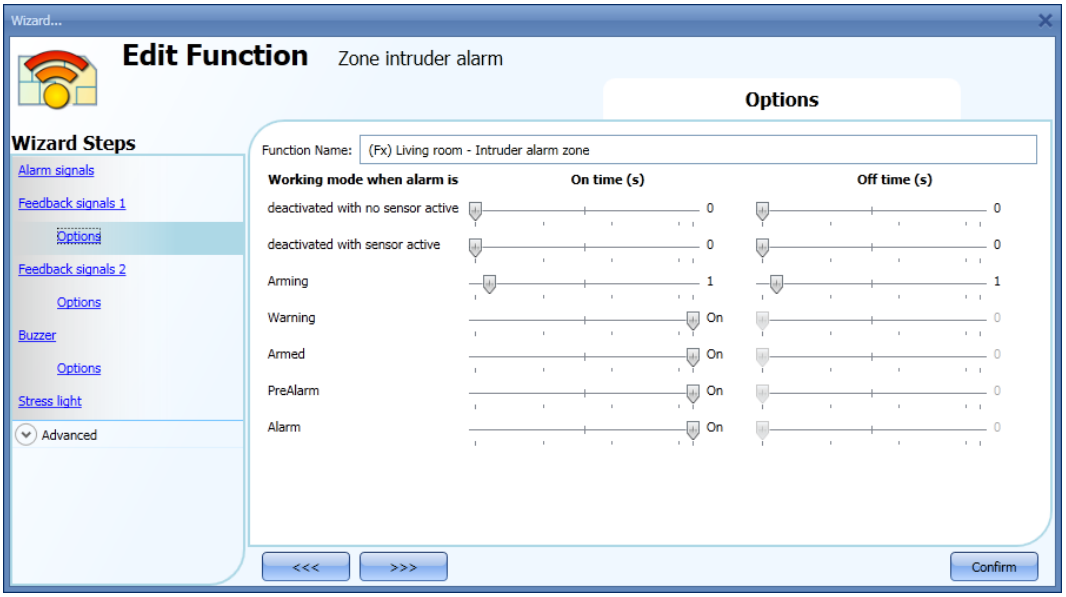
To add the feedback signals, the user has to select the relevant section in the wizard of the function, then double click on the *Signals* window and check the feedback signals from the list of those available. There are three different sections for the feedback signals, to allow the user to set up three different settings (in the picture below the feedback signals for the main function and the zone function are shown).

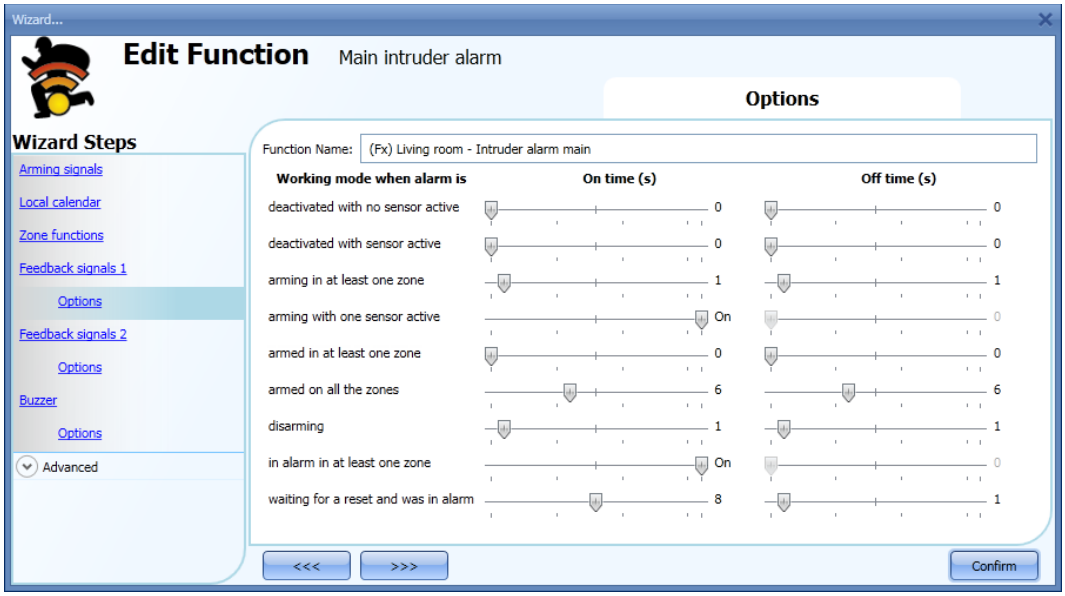




Regardless of the selected section (Feedback signal 1, Feedback signal 2 and Buzzer), the user can add LEDs, Relays and buzzers.
Once the signals are added, the user has to set up their behaviour.

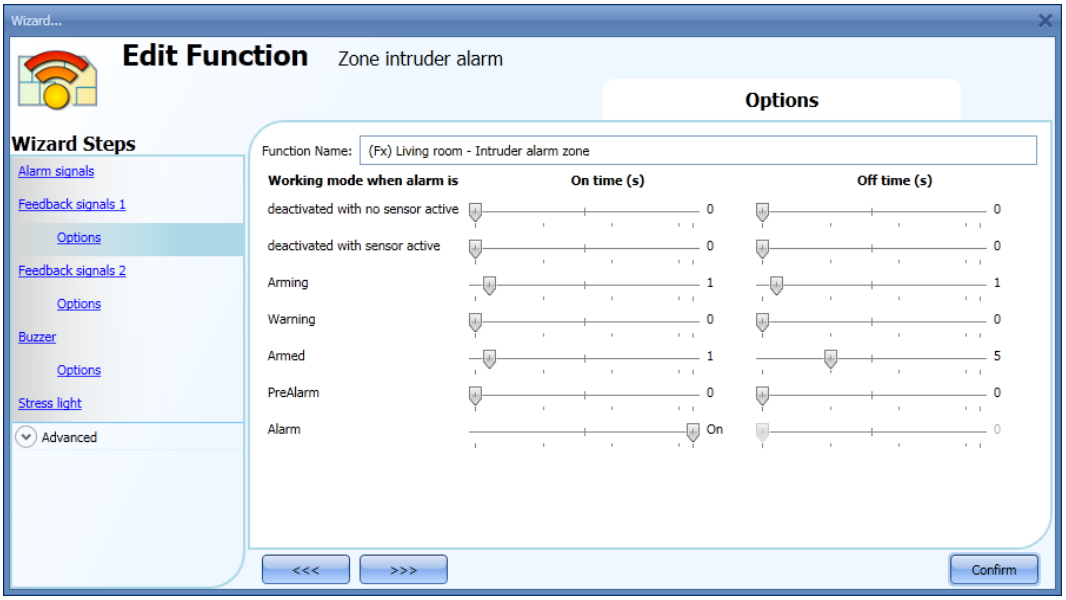
In the first picture the configuration window for the feedback signal in the zone function is shown, while in the second picture the window to configure the behaviour of the feedback signals in the main alarm function is shown.



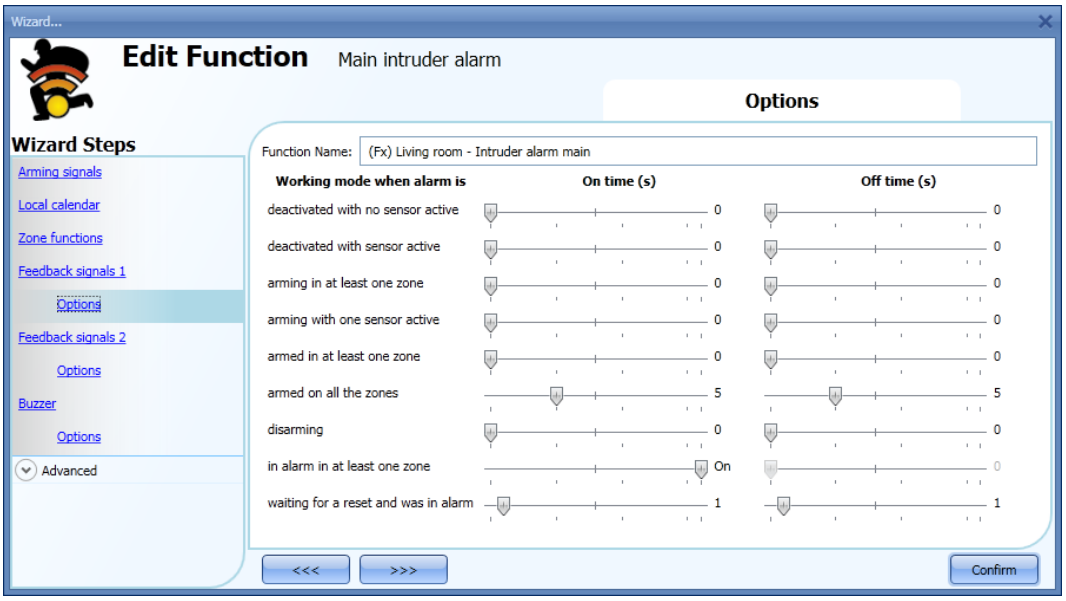


For all the statuses, the user can set the On/Off activation time. If both the on/off times are set to 0 seconds, the status will not be shown by the feedback signal.

Example 1:
The first example shows how to set a feedback signal to warn the user when the zone is arming, when the zone is armed and when the zone is in alarm.
When the function is arming, the feedback signal will blink 1 second on and 1 second off. When the function is armed, the feedback signal will blink 5 seconds off and 1 second on. When the function is in alarm, the feedback signal will stay steady on.



Example 2:
The second example shows how to customize the feedback activation for the main alarm. When all zones are armed the feedback signal will blink 5 seconds on and 5 seconds off. When the alarm is detected, the feedback will stay steady on. When the function is waiting for acknowledgment, the feedback signal will blink 1 second on and 1 second off.

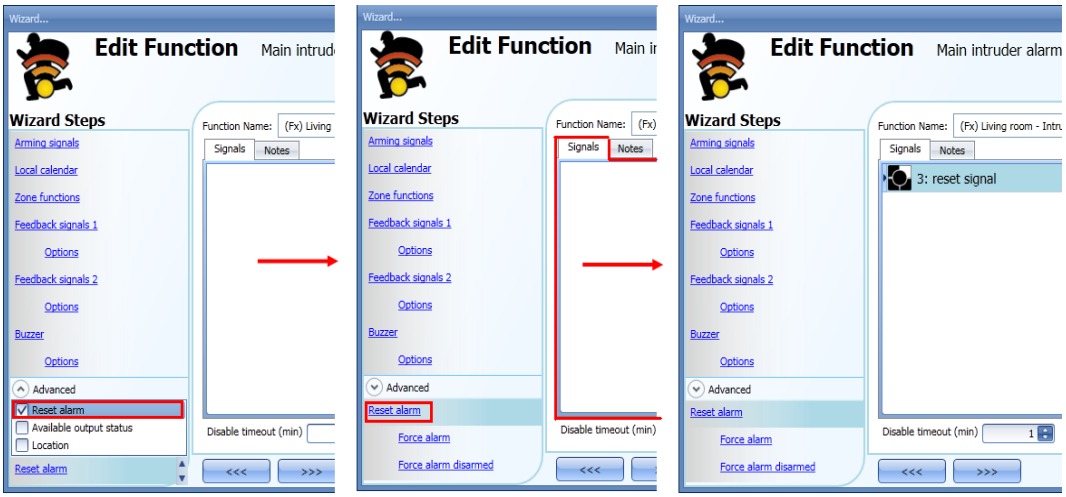


The user can customize up to three different types of feedback signals: feedback signal1, feedback signal 2, buzzer.

10.14.7 How to reset the alarm

The reset signal is used to inhibit the alarm status in the main function and in all the linked zone functions. The reset signal must be entered in the main alarm function, but it is directly linked to its zone alarm function.
If a siren function is used in combination with the intruder alarm, the reset status is also automatically linked to the siren function.

To add the reset signal, the user has to select the relevant section in the wizard of the main intruder alarm function, then double click on the *Signals* window and select the feedback signal from the list of those available (see picture below).



The reset signal can be managed as an event signal or a level signal: the event signal triggers the *Disable timeout*. When the alarm is reset, at the end of the Disable timeout, all the zones will be armed again.

If an event type is selected, the working mode has to be carried out following the table below.

Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, the reset condition is activated.	After a short press (less than 1 second), at the release of the push button, the reset condition is activated.	After a long press, the reset condition is activated at the release of the push button.	After a very long press, the reset condition is activated at the release of the push button.
1→0	As soon as the push button is pressed, the reset condition is deactivated.	After a short press (less than 1 second), the reset condition is deactivated at the release of the push button.	After a long press, the reset condition is deactivated at the release of the push button.	After a very long press, the reset condition is deactivated at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the reset is activated/deactivated in toggle mode.	After a short press (less than 1 second), the reset is activated/deactivated in toggle mode at the release of the push button.	After a long press, the reset is activated/deactivated in toggle mode at the release of the push button.	After a very long press, the reset is activated/deactivated in toggle mode at the release of the push button.
	The reset is activated/deactivated in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The reset is activated when the signal goes ON, and it is deactivated again when the signal goes OFF.			

If a level signal is used, the setting of the working mode has to be carried out following the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The reset is activated	No action
1→0	The reset is deactivated	No action
0→1 1→0	The reset is activated/deactivated in toggle mode	No action
	The reset is activated/deactivated in toggle mode	The reset is activated/deactivated in toggle mode
	The reset is activated	The reset is deactivated

10.14.8 How to use the alarm with a siren

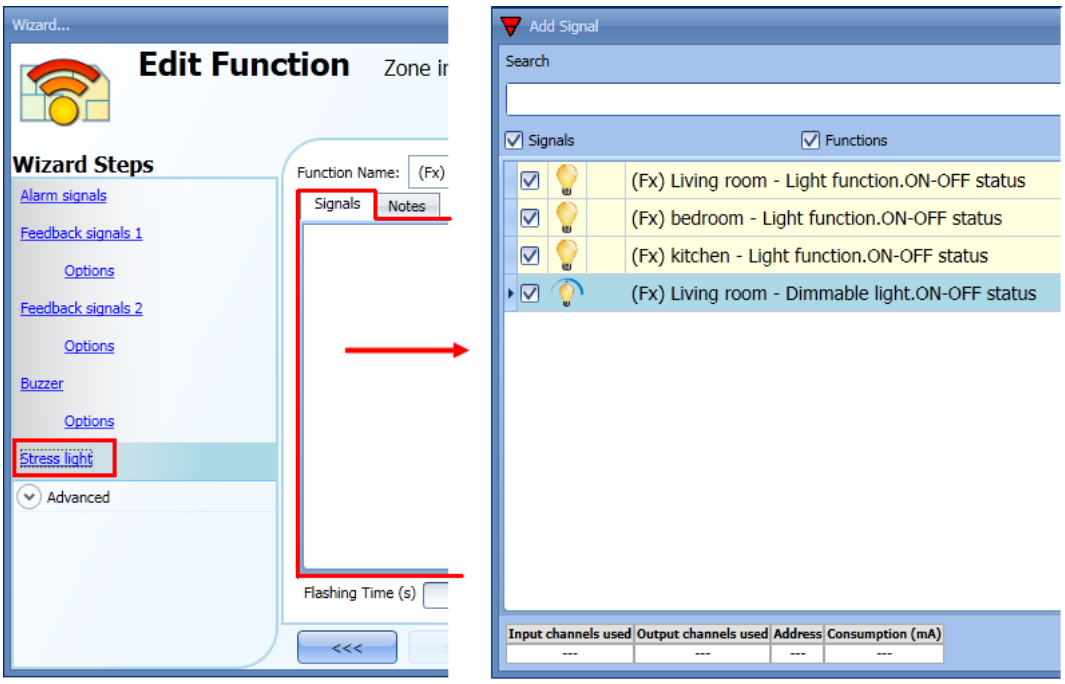
If the user wants to use a siren output when the alarm is on, a siren function has to be used. The siren function is not included in the intruder alarm function and must be created and linked to the main intruder alarm function.

The siren function will be triggered by the main alarm function as soon as an alarm condition occurs. The reset signal of the main alarm function is automatically connected to the siren: when the siren is on and the reset signal is activated, the siren goes off (see also the *How to set a Siren alarm function*).

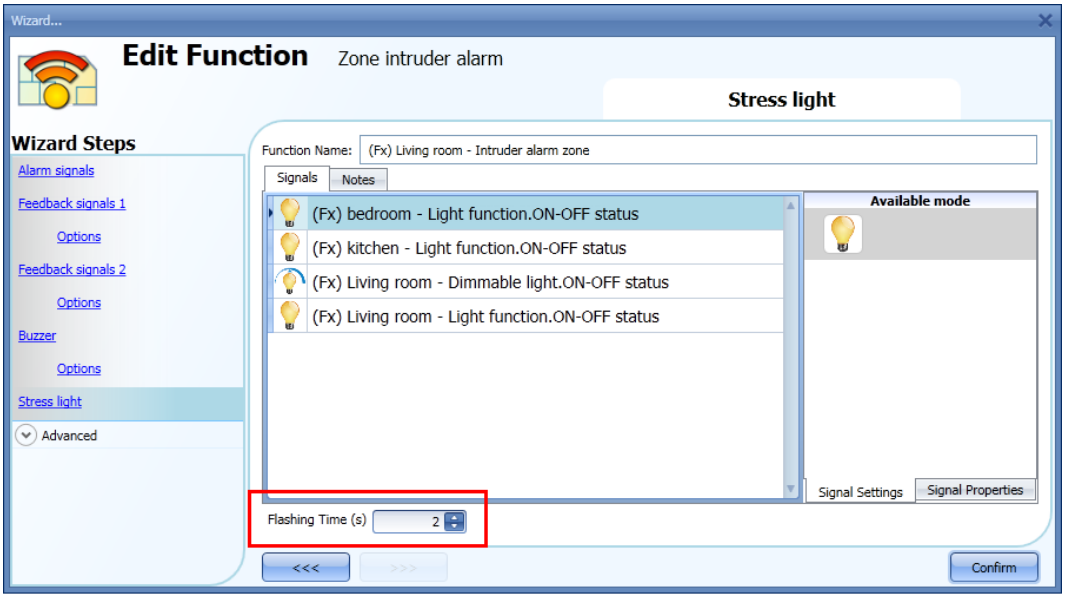
10.14.9 How to set the stress lights when the alarm is active

The stress light field allows the user to turn on/off one or more light functions when the zone is in alarm. When the zone function is in alarm, all the lights present in the list “Stress lights” are activated according to the flashing time settings. When the alarm condition in the zone ends, all the stress lights stop flashing, even if the siren and the main alarm function are still activated.
The user can add on/off lights or dimmable lights.
When a dimmable light is activated by the stress light, the output is managed from 10 to 100%: the output is never switched off because the complete switching off could damage ballast.

To add a function to the list of *Stress lights*, the user should select the relevant section in the wizard of the zone intruder alarm function, then double click on the *Signals* window and select the light function from the list of those available (see picture below).



The *Flashing time* of all the lights added to the list has to be set at the bottom of the stress lights window (see picture below).

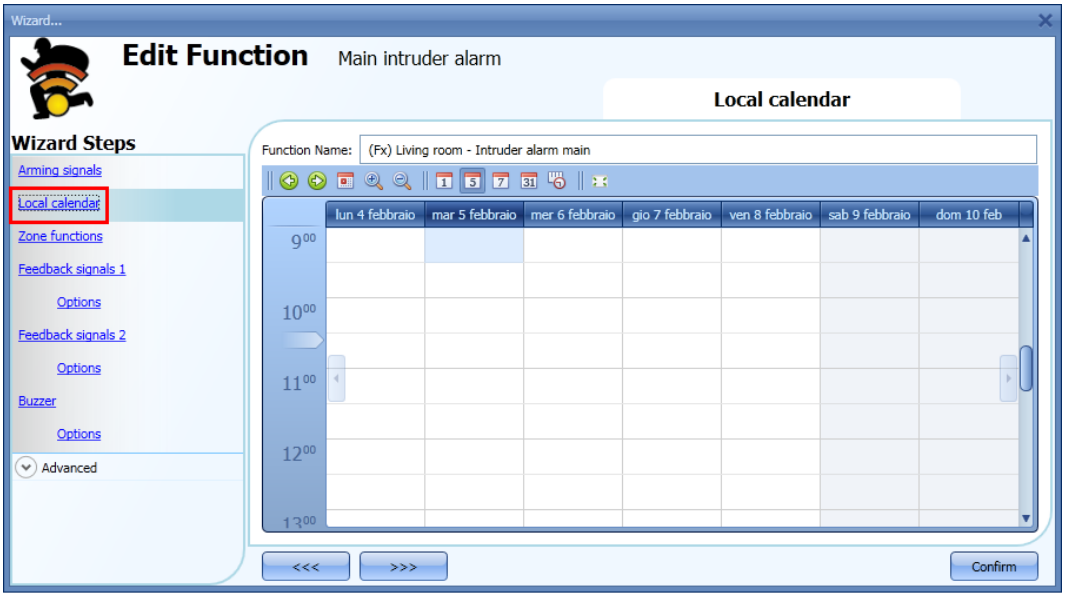


10.14.10 How to arm and disarm the alarm with the calendar

The user can program a calendar function in order to arm/disarm the intruder alarm.
There are two ways of managing the intruder alarm with calendars: by configuring the local calendar inside the main intruder alarm function or by using a global calendar function.

The local calendar function

To enable this, the relevant menu has to be enabled in the Advanced section (see picture below).



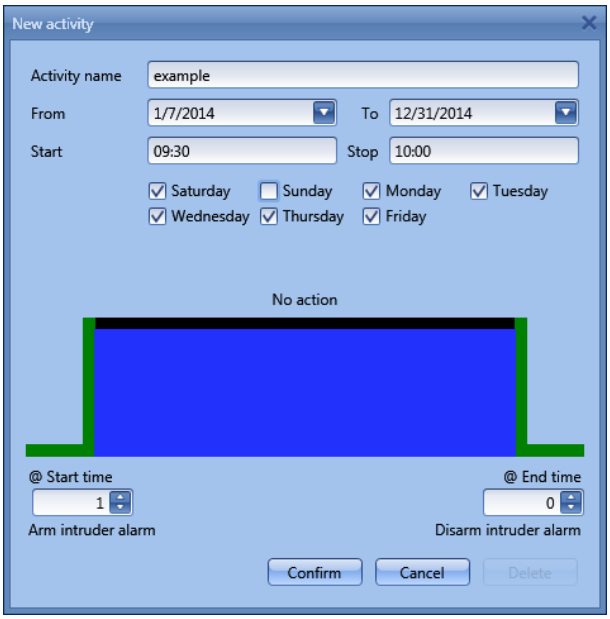
The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week
	One step forward in the calendar. E.g: if a week is visualized, this button allows the user to go to the next week
	Go to the current day
	Magnifying lens to see more/fewer time bands
	The calendar visualizes one day horizontally
	The calendar visualizes 5 days horizontally
	The calendar visualizes 7 days horizontally
	The calendar visualizes 31 days horizontally
	The calendar visualizes 7 days vertically

Calendar activity

Once the user has selected his preferred type of visualization, to enter a new activity he has to double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that is to appear on the calendar. This is a mandatory field.

***From**:* This is the starting date for the activity.

***To**:* This is the end date of the activity.

***Start**:* This is the start time of the activity.

Stop: This is the end time of the activity.

The activity will be repeated in the following years, using the same dates and times.

The action managed by the calendar can be of two types:

3) Event activity: the system will perform the selected actions only at the start time and at the end time and they are managed as events
4) Level activity: during the time band, all the automation can be disabled.

If ***Event activity*** is selected, the user can decide what action he wants to be performed when the time band starts and finishes.

@ ***start time***: in this field the user can select the action to be performed at the start time.
The available actions are:

- (-1) No action: nothing will be done at the start time
- (0) Disarm the intruder alarm: the intruder alarm is disarmed at the start time
- (1) Arm the intruder alarm: the intruder alarm is armed at the start time

@ ***end time***: in this field the user can select the action to be performed at the end time.
The available actions are:

- (-1) No action: nothing will be done at the end time
- (0) Disarm the intruder alarm: the intruder alarm is disarmed at the end time
- (1) Arm the intruder alarm: the intruder alarm is armed at the end time

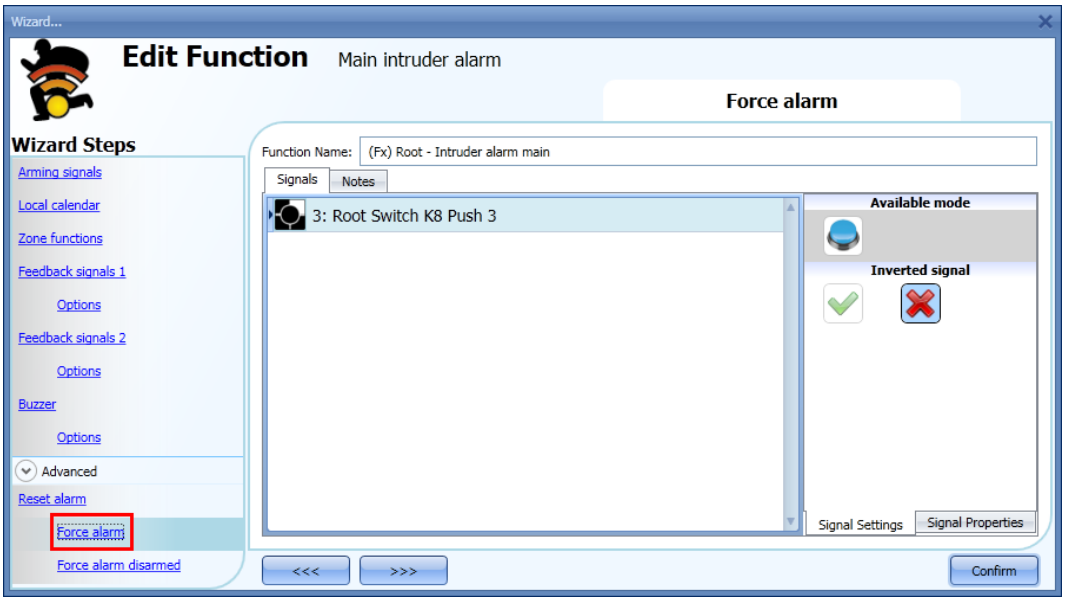
If ***Level activity*** is selected, the user can disable the automation of the calendar.

During time period: in this field the user selects the action he wants to perform during the time band.

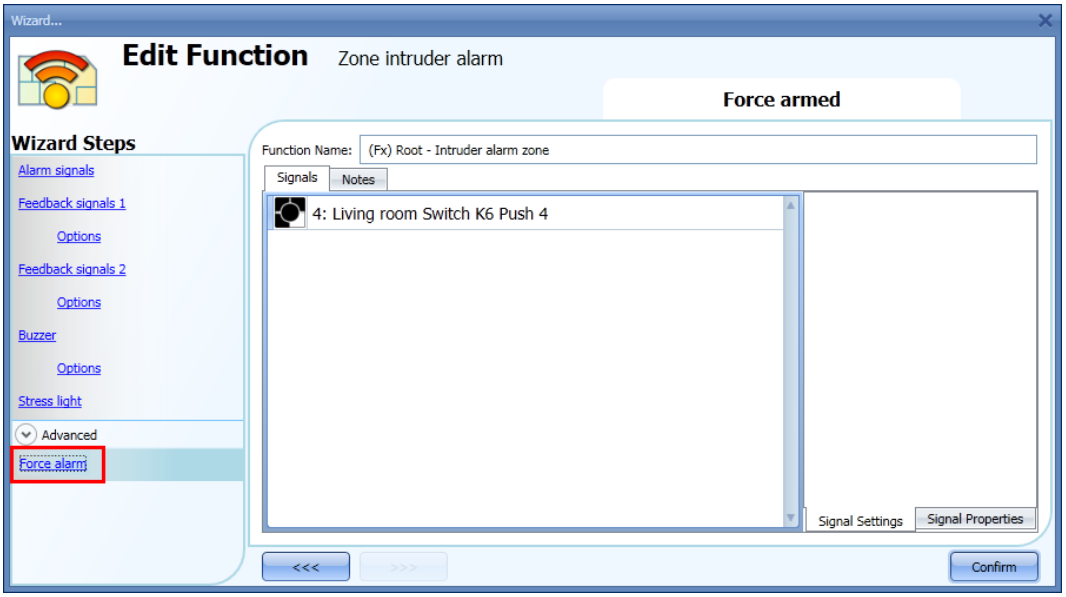
- No action
- Disable automations

10.14.11 How to force the alarm condition

If the user wants to force the alarm condition regardless of all the other signals used in the function, the *Force alarm* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force alarm*, then double click on the signals window and select the appropriate signal to use.
The alarm can be forced in all the zones by using the main intruder alarm (see picture below)



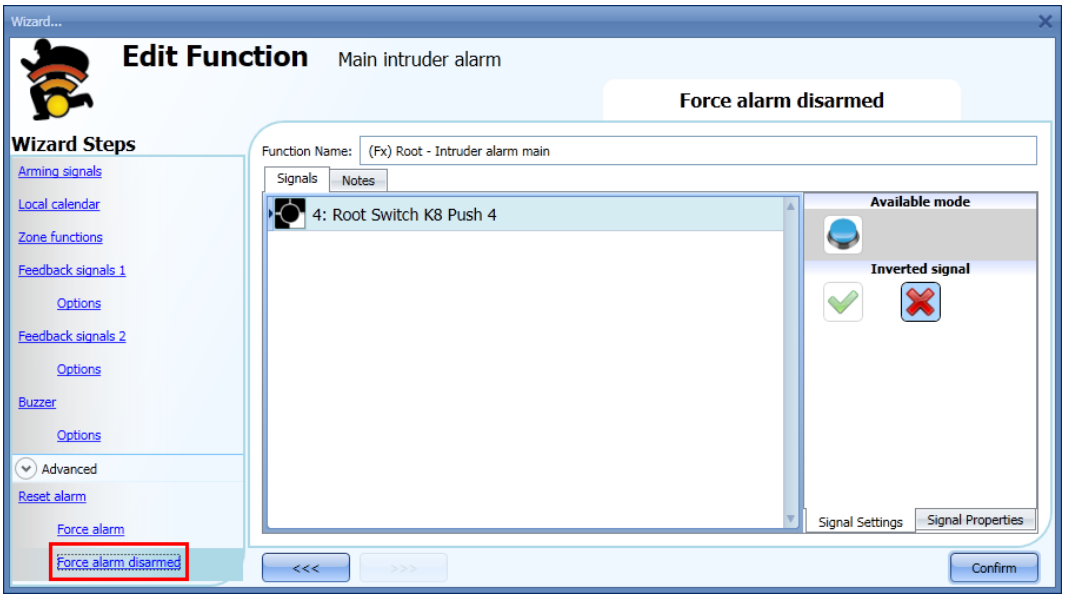
The alarm can be forced in only a single zone by adding the force alarm signal in that specific zone function (see below).



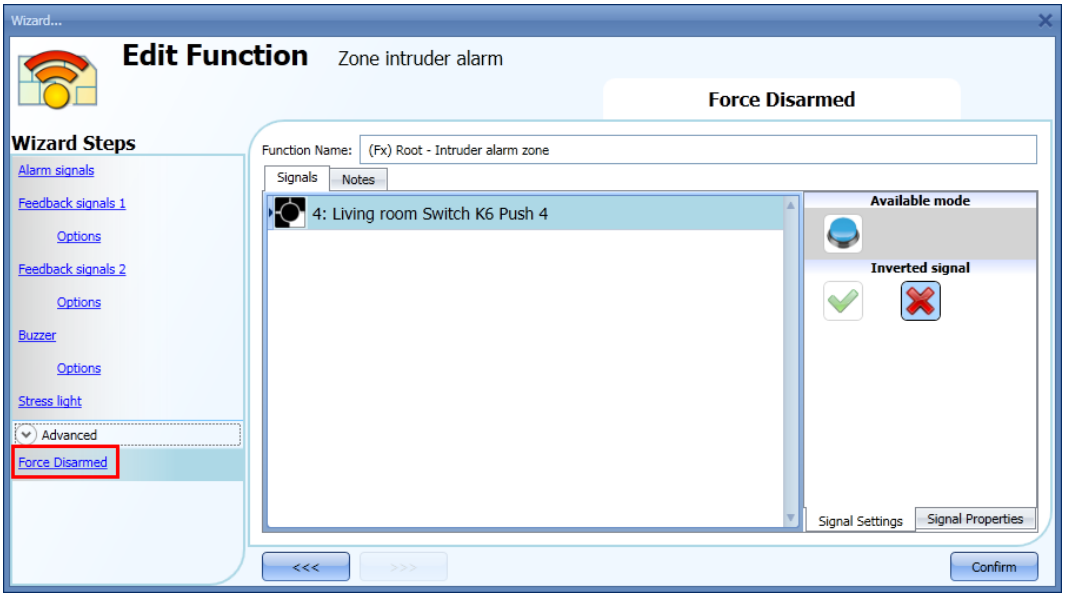
Each signal used in the *Force alarm* window works as a level signal. While the signal is active, the main alarm function, and all the linked zones, are forced into the alarm condition.
When the *Force alarm* signal is activated and at the same time the *Force alarm disarmed* signal is activated, the *Force alarm* signal has priority.

10.14.12 How to force the alarm into disarmed

If the user wants to force the alarm into disarmed regardless of all other signals used in the function, the *Force alarm disarmed* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force alarm disarmed*, then double click on the signals window and select the appropriate signal to use. The alarm can be forced into disarmed in all the zones by using the main intruder alarm (see picture below).



The alarm can be forced into disarmed in only a single zone by adding the *Force alarm disarmed* signal in the zone function (see picture below). An example could be a room of the house where the intruder alarm has to be disarmed even if all the other zones are armed (e.g. the room has some pets that could trigger the alarm).



Each signal used in the *Force alarm disarmed* window works as a level signal. Until the signal is active, the main alarm function, and all linked zones, are forced into disarmed.

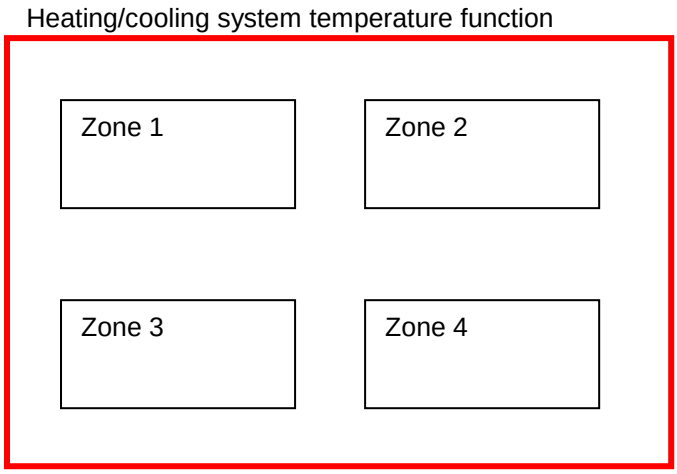
10.15 How to set a temperature function

The temperature function is used to manage the heating and ventilation of the house. The user can manage the climate inside the house, creating different zones depending on the family's requirements.

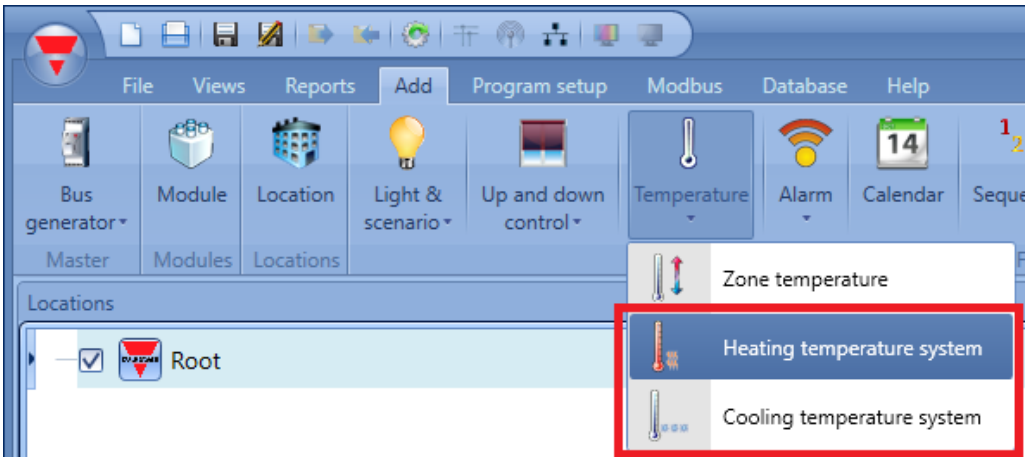
There are two main steps to configure the temperature control:

- In the first step the user has to add a heating/cooling temperature system: this is the main core of the temperature control and it is used to manage all the zone functions (it collects all the zone statuses when a request for heating/cooling is received from a zone and it activates the system, providing heating/cooling in the zone). It manages the pump output with the relevant delay via the valve activation and the on/off switching according to the outdoor temperature.

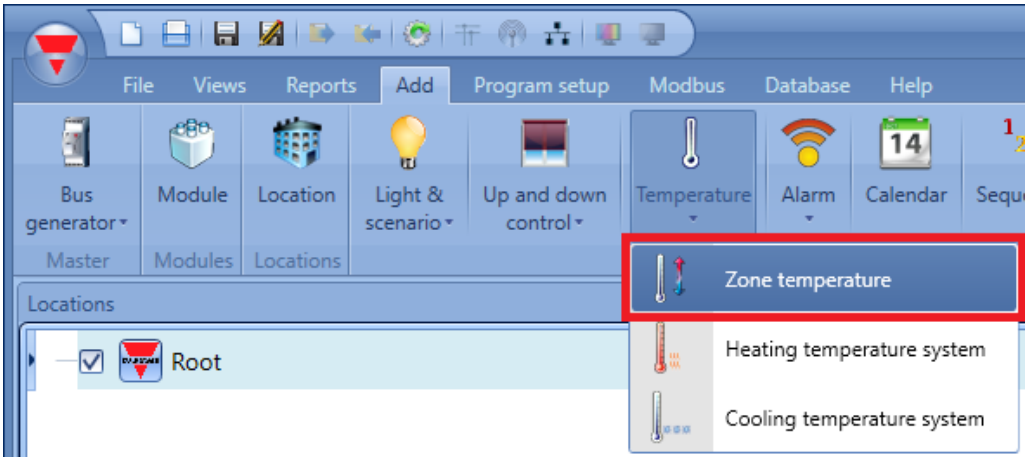
- In the second step the user must enter at least one zone temperature function; each zone function can correspond to a part of the house where the user wants to control the heating/cooling.



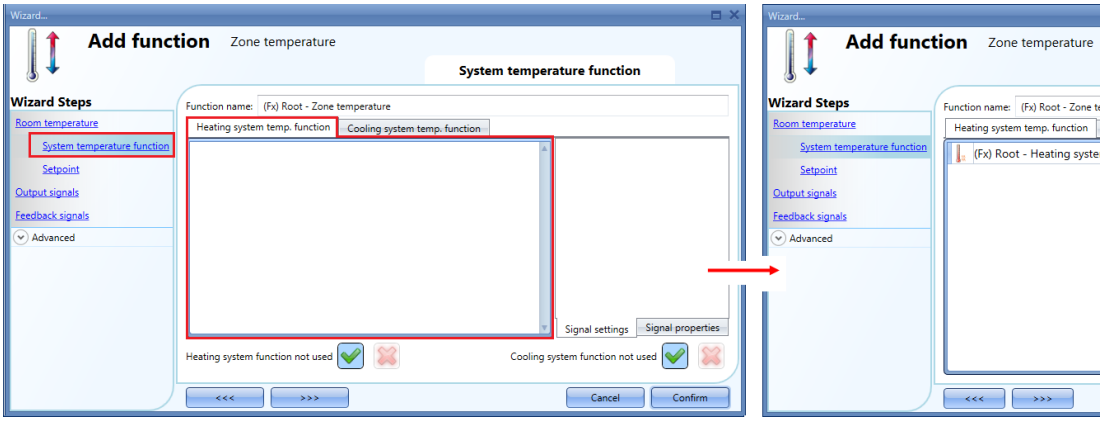
To set up a temperature function, the user has to select *Temperature* from the Add menu, then select at least one *System temperature* function (Heating or Cooling).



Once the system temperature function is added, the user should click on *Temperature* from the Add menu and select *Zone temperature* (see picture below).



The last step is to link the temperature system function to the zone temperature functions: to do this the user has to open the zone function, click on *System temperature function*, double click on the correct window (according to the heating or cooling mode), and then select the required system function from the list of available functions (see picture below). More than one zone function can be controlled by the same system function.



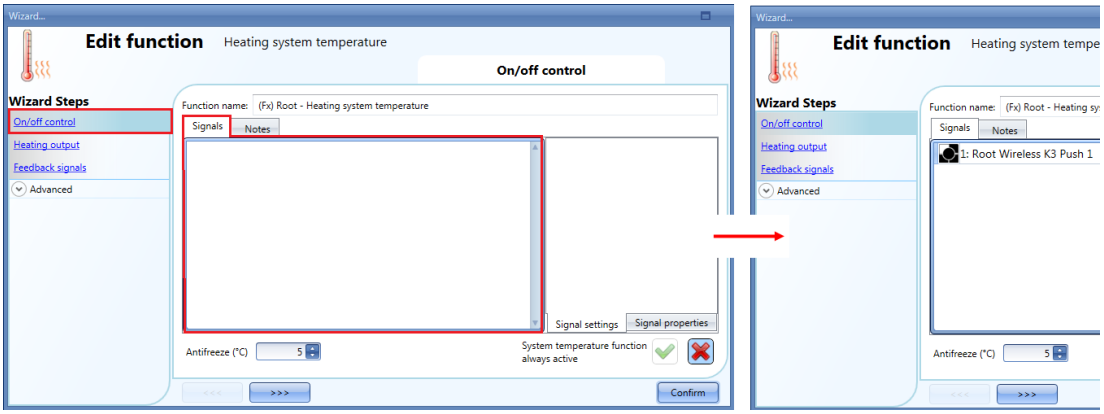
According to the added system temperature, the zone function can manage either only heating, only cooling, or both.

10.15.1 Heating/Cooling control system configuration

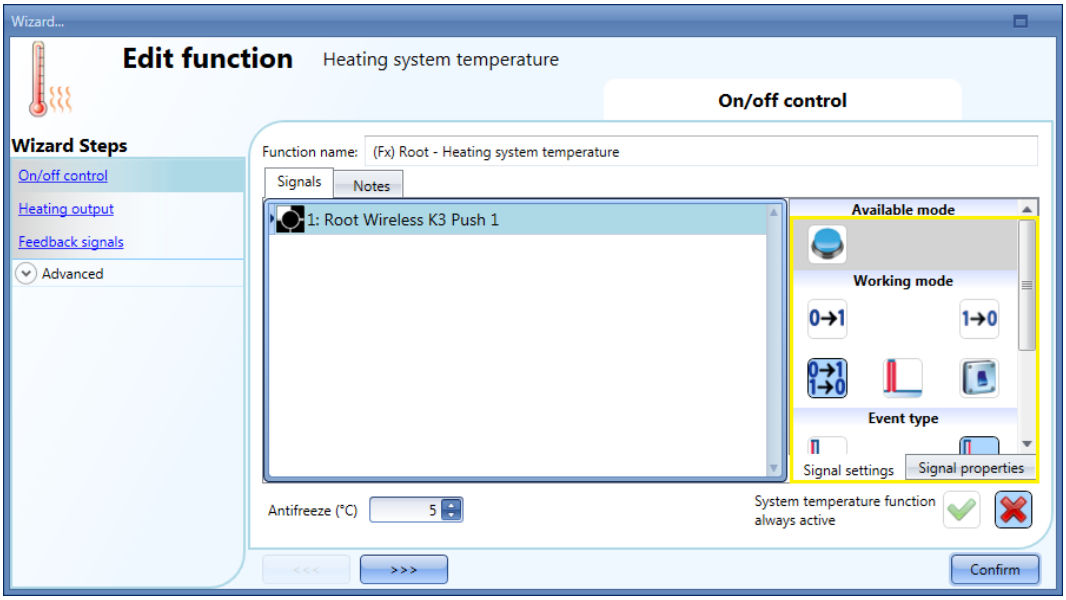
The heating/cooling control system function is the core function and it is used to control the activation/deactivation of the pump of the system. When it is on, the heating or cooling mode is on: when it is off, the temperature control is off even if there is a request from a zone function.

10.15.2 How to activate/deactivate the control system function

To add the on/off signal, the user has to select the relevant section in the wizard of the heating/cooling control system function, then double click on the *Signals* window and select the input signal from the list of available signals (see picture below).



Once the on/off signal is entered, the user can also decide to use the inverted logic of the signal (see yellow rectangle in the picture below).



If a push button type is added, the working mode, highlighted with the yellow rectangle, has to be selected according to the table below:

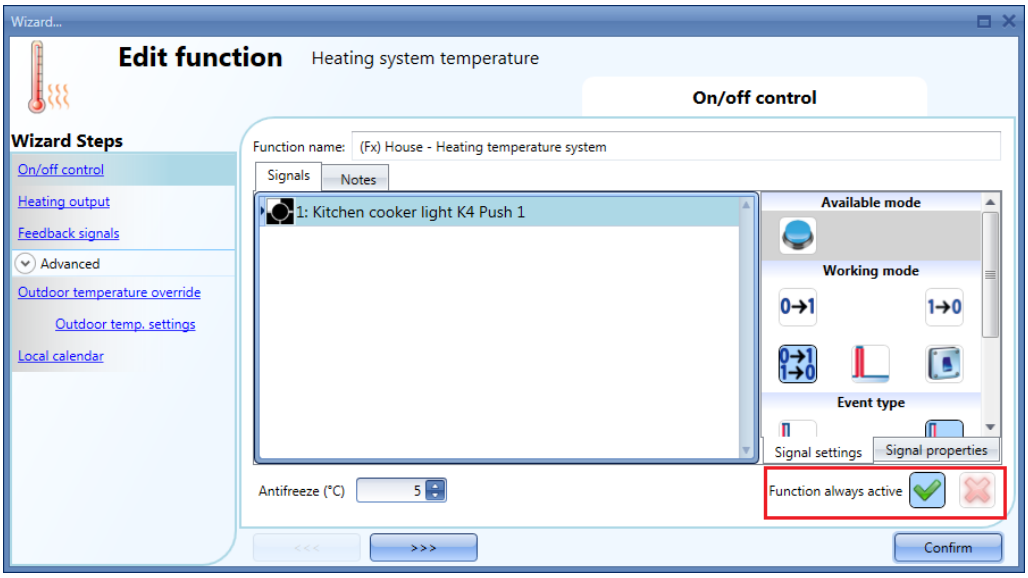
Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, the function is activated.	After a short press (less than 1 second), at the release of the push button the function is activated.	After a long press, the function is activated at the release of the push button.	After a very long press, the function is activated at the release of the push button.
1→0	As soon as the push button is pressed, the function is deactivated.	After a short press (less than 1 second), the function is deactivated at the release of the push button.	After a long press, the function is deactivated at the release of the push button.	After a very long press, the function is deactivated at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the function will be activated/deactivated in toggle mode.	After a short press (less than 1 second), the function will be activated/deactivated in toggle mode at the release of the push button.	After a long press, the function will be activated/deactivated in toggle mode at the release of the push button.	After a very long press, the function will be activated/deactivated in toggle mode at the release of the push button.
	The function will be activated/deactivated in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The function is switched on when the signal goes ON, and it is switched off when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The function is activated	No action
1→0	The function is deactivated	No action
0→1 1→0	The function is activated/deactivated in toggle mode	No action
	The function is activated/deactivated in toggle mode	The function is activated/deactivated in toggle mode
	The function is activated	The function is deactivated

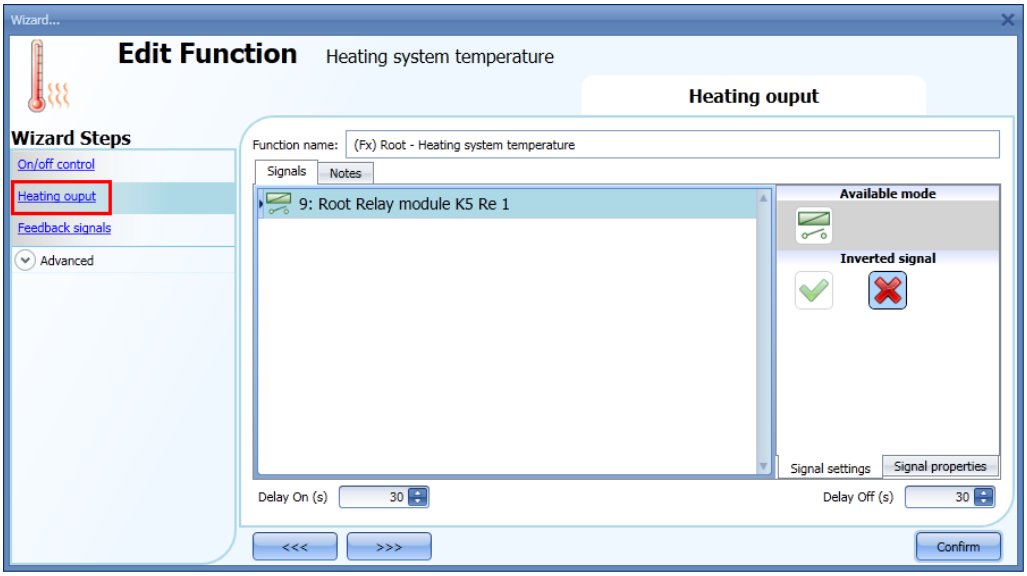
10.15.3 How to set the Heating/Cooling control *system* function always active

By enabling *System temperature function always active* (see below), this function will always be activated and the On/Off signals and the calendar will be ignored.

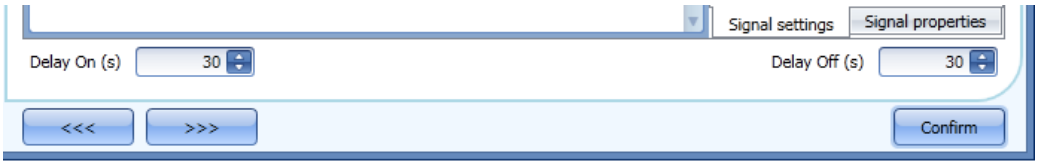


10.15.4 How to add the heating output

The heating output must be added in the relevant field of the wizard function (see picture below).



Once the output signal is added, the user can set two delays (delay on/off) to control the activation/deactivation of the system output (pump) and the zone output (valve) when a request for heating/cooling is received from a zone.



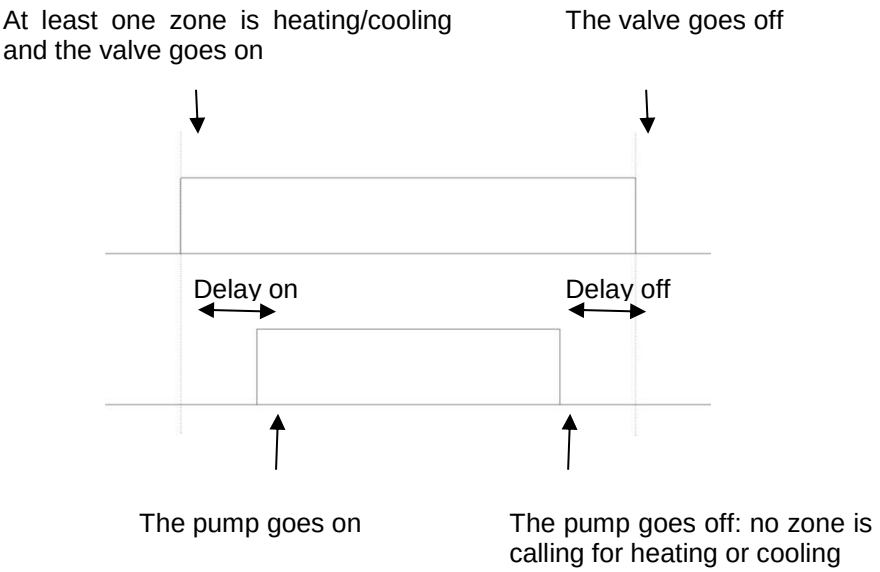
When the zone temperature status goes on:

- The valve output in the zone goes on immediately.
- The pump output in the heating control system starts when the *Delay on* time expires. This is to make sure that the pump is not activated before a valve in the zone is completely opened.

When the zone temperature status goes off:

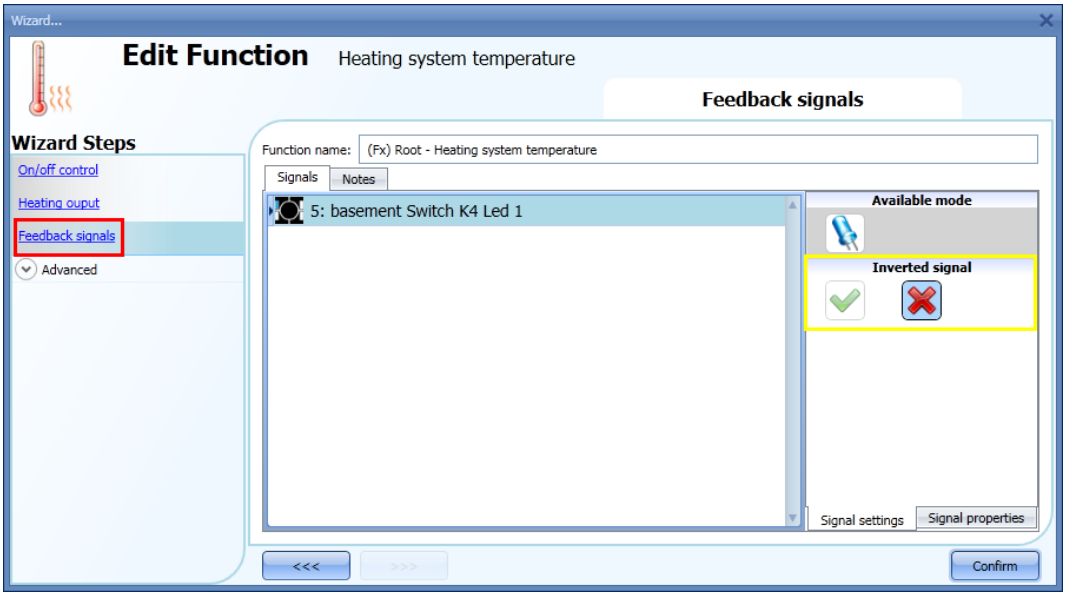
- The pump of the heating control system goes off immediately (only if there are no more requests for heating/cooling from other zones).
- If only one zone is used, the valve goes off immediately; otherwise, if more zones are used, the valve in the last activated zone is switched off when the Delay Off time has elapsed.

See picture below:



10.15.5 How to set a feedback signal

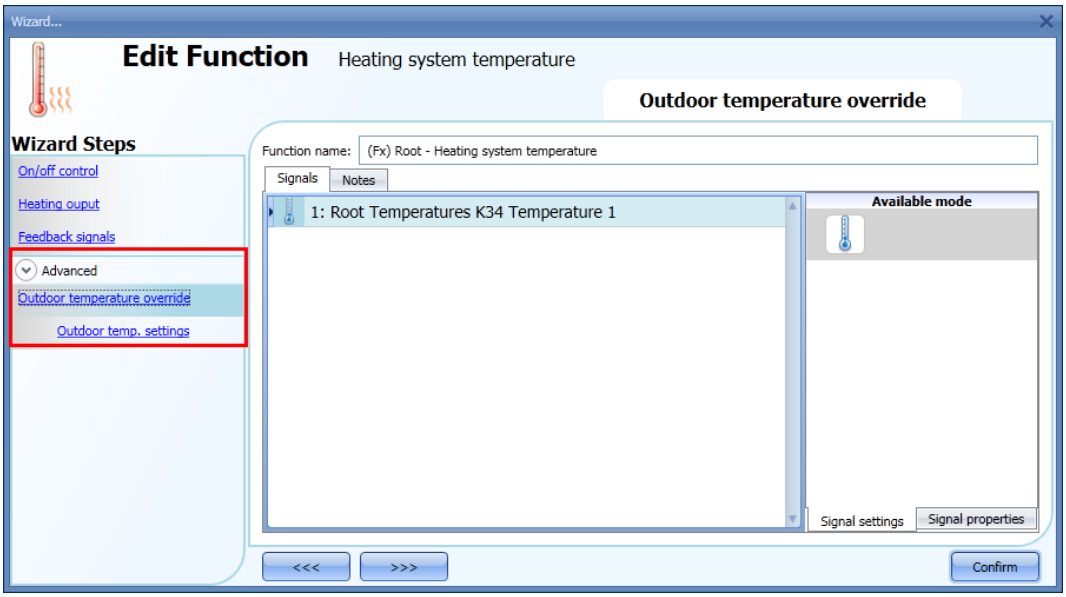
The feedback signal can be used to give information about the status of the heating control system function.
It follows the status of the function: it is on if the function is on, and off if the function is off.
To add the feedback signal, the user should click on the relevant field in the wizard of the function, then double click on the *Signals* window and select the feedback signal from the list of those available.



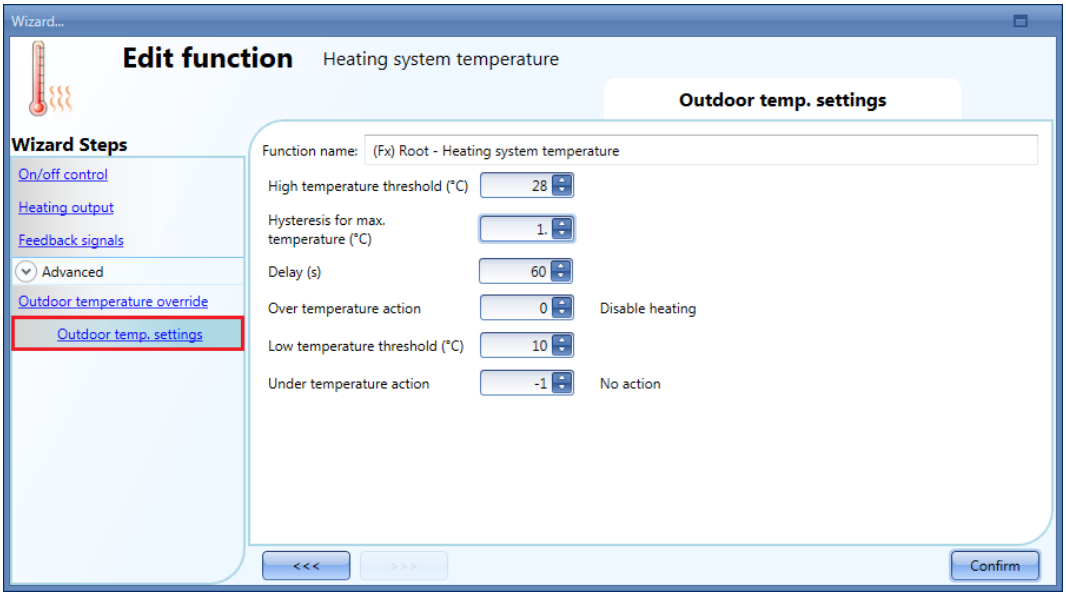
The inverted logic for the feedback can be selected as shown in the yellow rectangle in the picture above.

10.15.6 How to switch off heating/cooling according to the outdoor temperature

By accessing the *Advanced* section and enabling *Outdoor temperature override*, the user can configure the control system function to switch on/off according to the outdoor temperature.



Firstly, the user should enter one or more temperature values coming from the temperature sensors. The user can add up to ten different temperature signals; if more values are linked, the average value is used. If a temperature sensor is faulty, the average value is calculated by using the other signals.
In the field *Outdoor temperature settings,* the user can configure all the settings (see picture below).

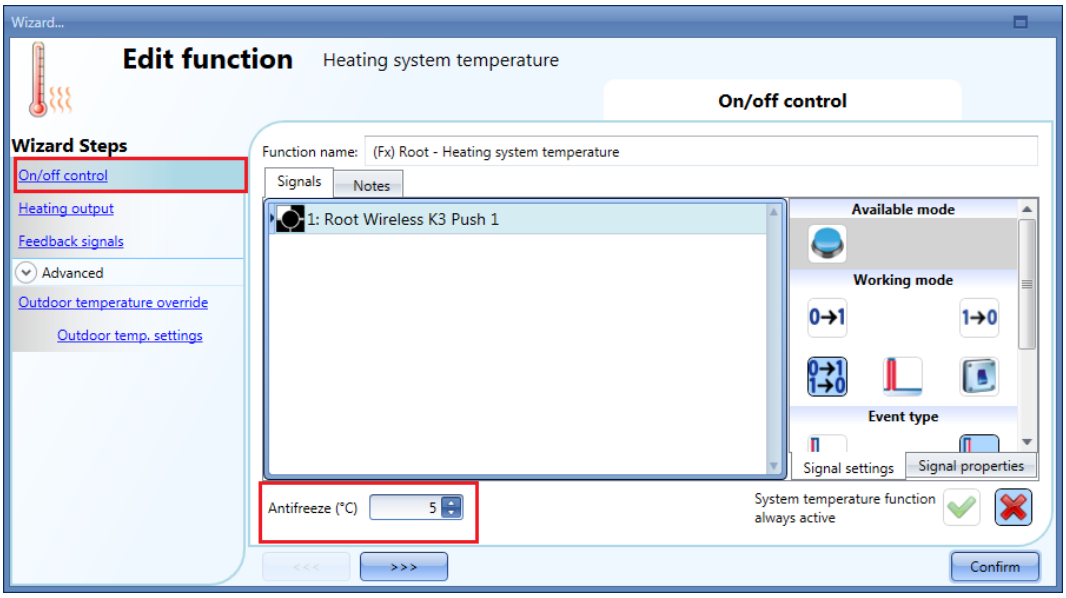


The user can set two different thresholds: the *High temperature threshold* and the *Low temperature threshold*.
Usually the *low temperature threshold* is used to disable cooling, while the *high threshold* is used to disable heating.
The Hysteresis and the delay on time are applied to both thresholds.

In the *Over temperature action* field, the user can set the action to be performed when the measured temperature goes over the *High temperature threshold*. In the same way, in the *Under temperature action* field, the user can set the action to be performed when the measured temperature goes below the *Low temperature threshold*.
It is possible to use both thresholds to manage the deactivation of the output.

10.15.7 How to set the antifreeze temperature

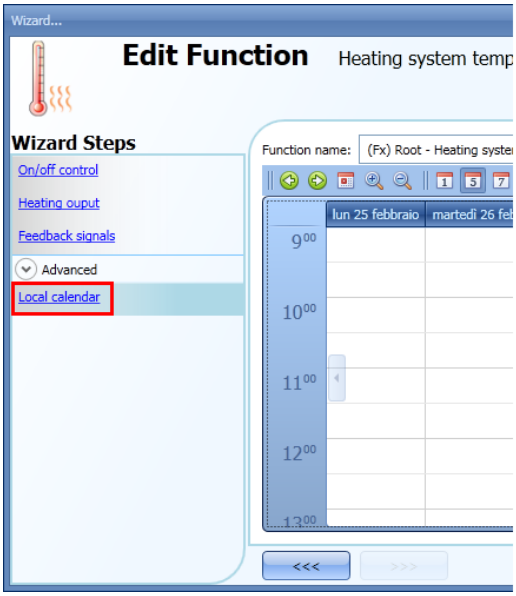
In the heating system temperature, the user can set the antifreeze value. This is a common value that is automatically used in all the zone functions that are linked to the system function.
If the zone status is set to OFF (set point OFF selected), and the measured temperature in the zone falls below the antifreeze value, the output in the zone is activated until the temperature is once again above the antifreeze limit.
The antifreeze value can be selected in the ON/OFF control field of the heating temperature .

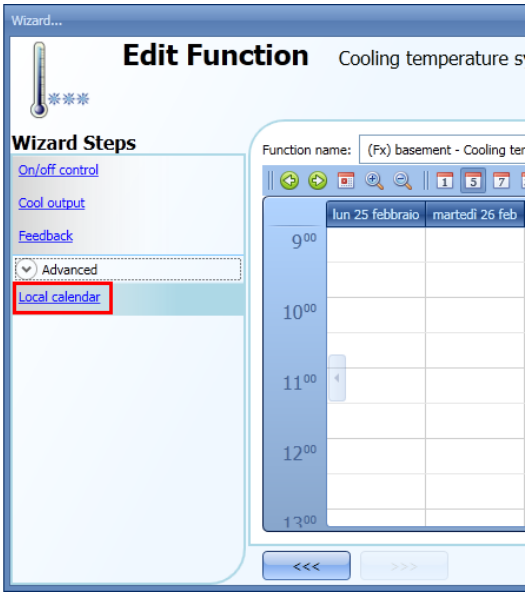


The antifreeze regulation works with a hysteresis of one degree:
Heating ON ➔ temperature < antifreeze temperature
Heating OFF ➔ temperature ≥ antifreeze temperature

10.15.8 How to control heating/cooling with the calendar

The user can program a local calendar to activate/deactivate the heating/cooling system temperature function, using daily activities.
To enable this, the relevant menu has to be selected in the *Advanced* section (see picture below).





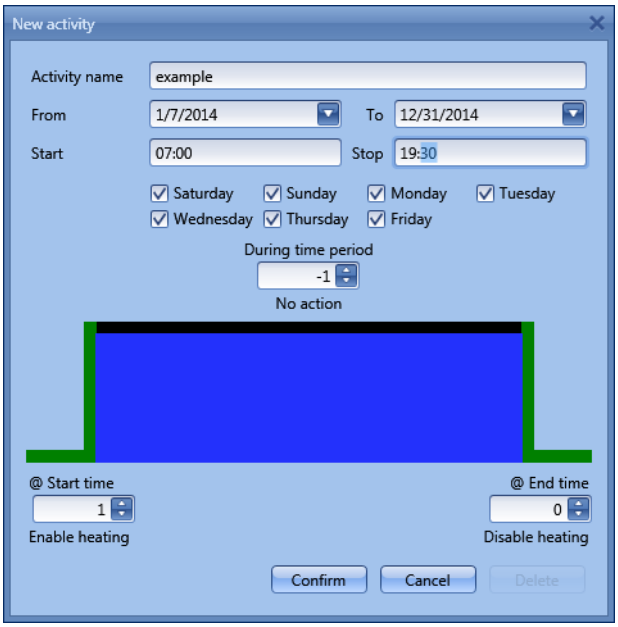
The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day.
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally.
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

Calendar activity

Once the user has selected his preferred visualization, to enter a time band he has to double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From*:* the start date for the calendar activity.

To*:* the end date of the calendar activity.

Start*:* The time of the start of the activity.

Stop: The time of the end of the activity.

The activity will be repeated in the following years, using the same dates and times.

The action managed by the calendar can be of two types:

5) Event activity: the system will perform the selected actions only at start time and end time and they are managed as events
6) Level activity: during the time band, all the automations are disabled.

The user can decide what action he wants to do when the time band starts and finishes.

@ ***start time***: in this field the user can select the action to be performed at the start time.
The available actions are:
- (-1) No action: nothing will be done at start time
- (0) Disable heating/cooling: heating/cooling is disabled
- (1) Enable heating/cooling: heating/cooling is enabled

@ ***end time***: in this field the user can select the action to be performed at the end time.
The available actions are:
- (-1) No action: nothing will be done at end time
- (0) Disable heating/cooling: heating/cooling is disabled
- (1) Enable heating/cooling: heating/cooling is enabled

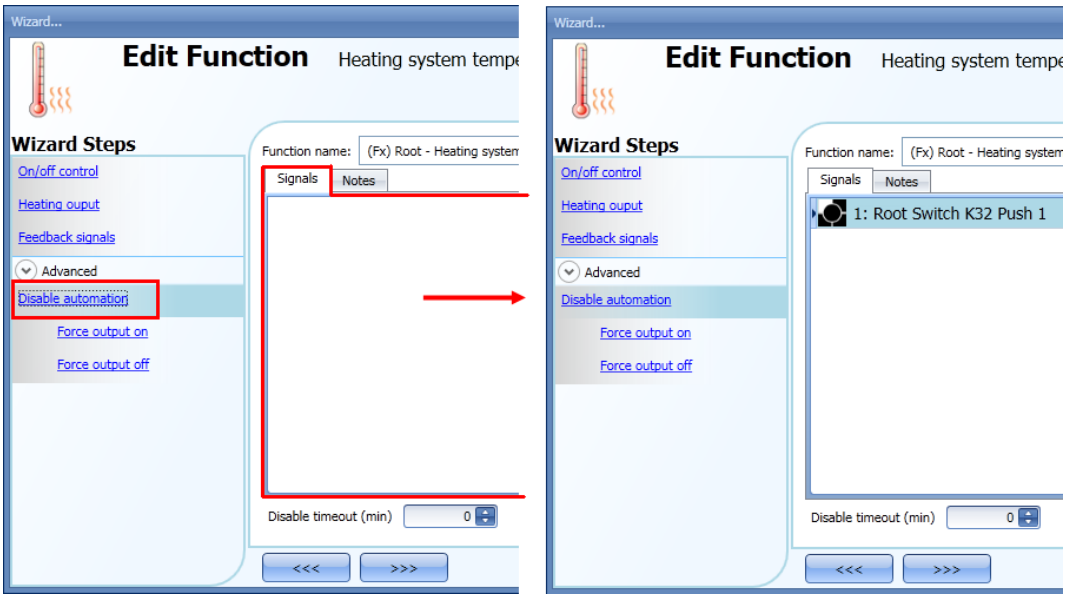
If ***Level activity*** is selected, the user can disable the automation of the calendar.

During time period: in this field the user selects the action he wants to perform during the time band.

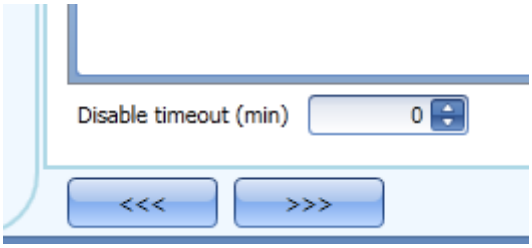
- No action
- Disable automations

10.15.9 How to disable the automations

The user can disable the calendar automation by using a physical signal such as a pushbutton or by using the calendar level activity.
To add the disable signal, the user should enable the relevant field in the wizard of the system temperature function, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



The timer *Disable timeout* starts every time the disable status is activated: the disable status is automatically deactivated as soon as the time expires.



If the disable timeout is set to 0, the timer is disabled and the disable condition stays active.

If a push button is selected as the disable automation signal, the way of working has to be carried out following the table below.

Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, the disable condition is activated.	After a short press (less than 1 second), at the release of the push button, the disable condition is activated.	After a long press, the disable condition is activated at the release of the push button.	After a very long press, the disable condition is activated at the release of the push button.
1→0	As soon as the push button is pressed, the disable condition is deactivated.	After a short press (less than 1 second), the disable condition is deactivated at the release of the push button.	After a long press, the disable condition is deactivated at the release of the push button.	After a very long press, the disable condition is deactivated at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the disable condition is activated/deactivated in toggle mode.	After a short press (less than 1 second), the disable condition is activated/deactivated in toggle mode at the release of the push button.	After a long press, the disable condition is activated/deactivated in toggle mode at the release of the push button.	After a very long press, the disable condition is activated/deactivated in toggle mode at the release of the push button.
	The disable condition is activated/deactivated in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

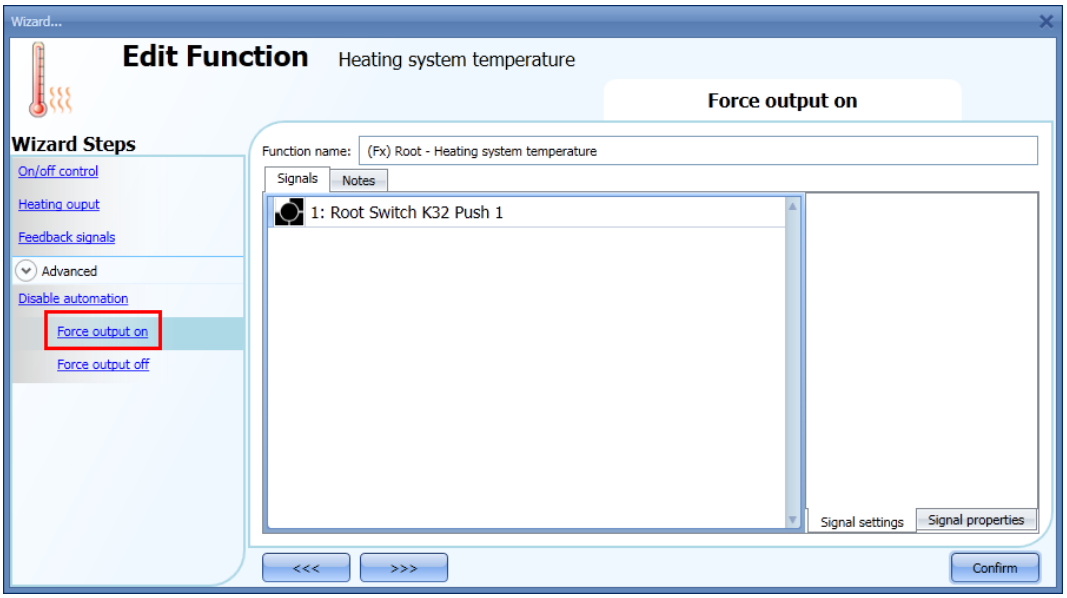
If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The disable condition is activated	No action
1→0	The disable condition is deactivated	No action
0→1 1→0	The disable condition is activated/deactivated in toggle mode	No action
	The disable condition is activated/deactivated in toggle mode	The disable condition is activated/deactivated in toggle mode
	The automation is disabled	The automation is enabled

If the user wants to disable the automations using the calendar, see also the previous paragraph *Level activity.*

10.15.10 How to force the control system function to the on status

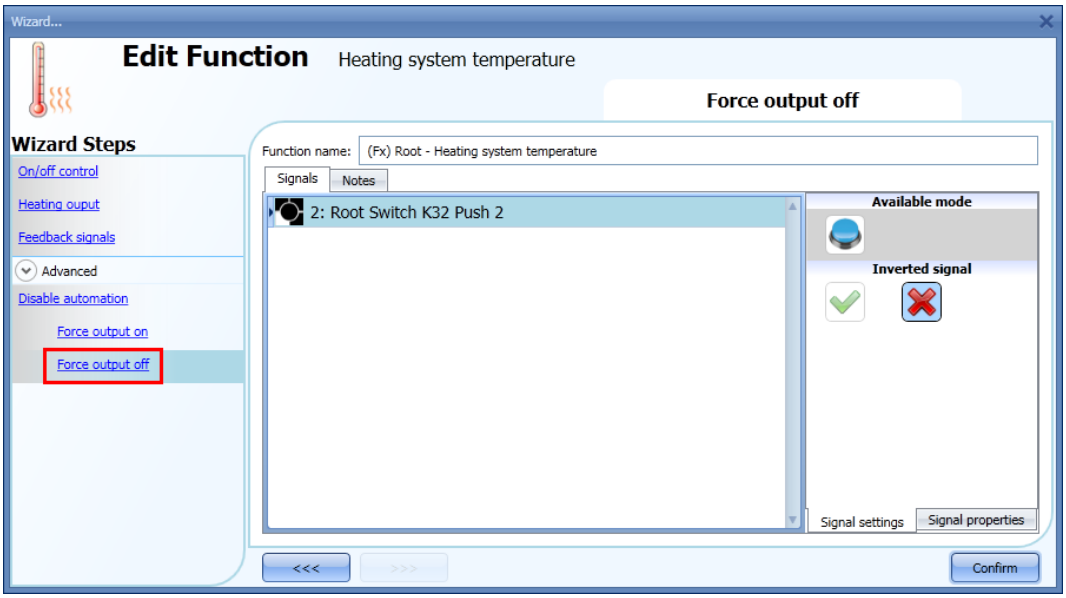
If the user wants to force the activation of the output regardless of all the other signals used in the function, the *Force output on* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output on*, then double click on the signals window and select the appropriate signal to use (see picture below).



Each signal used in the *Force output on* window works in level mode: until the signal is active, the output stays on.
When the *Force output on* signal is activated and at the same time the *Force output off* signal is activated, the *Force output on* signal has priority.

10.15.11 How to force the control system function to the off status

If the user wants to force the deactivation of the output regardless of all the other signals used in the function, the *Force output off* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output off*, then double click on the signals window and select the appropriate signal to use (see picture below).



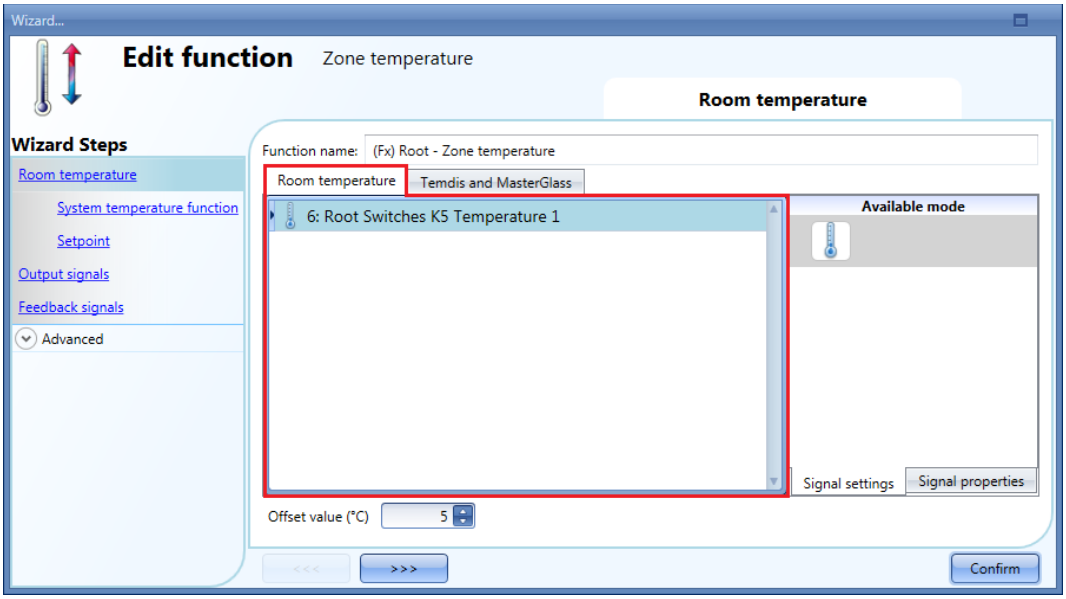
Each signal used in the *Force output off* window works in level mode: until the signal is active, the output stays off.
When the *Force output on* signal is activated and at the same time the *Force output off* signal is activated, the Force output on signal has priority.

10.16 Zone temperature configuration

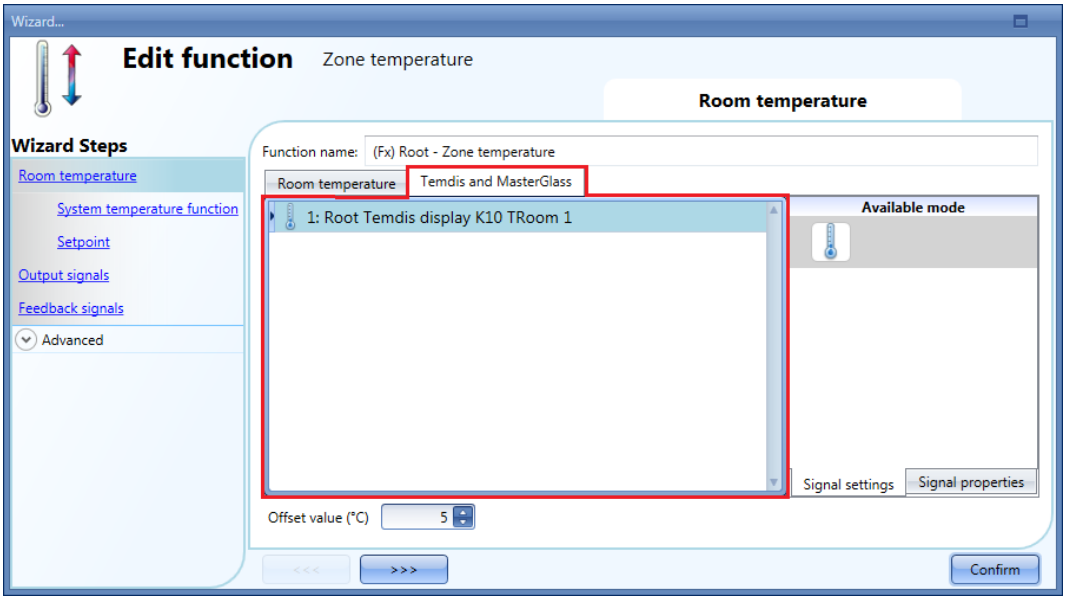
10.16.1 How to add the temperature signal

In the first step of the function, the user must enter the temperature signal that has to be used for the control of heating/cooling.

In the *Room temperature signals* window, the user should add one or more temperature signals that come from all the temperature sensors apart from the TEMDIS display and MasterGlass Temdis, which has a dedicated tab window.

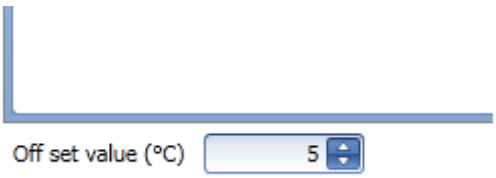


In the *Temdis and MasterGlass* signals window, the user can add the temperature signals (*TRoom and TAuxiliary*) of the Temdis modules (SHxxXTEMDIS) and the *TRoom* of MasterGlass Temdis modules (SHGxxxW-BSLT)



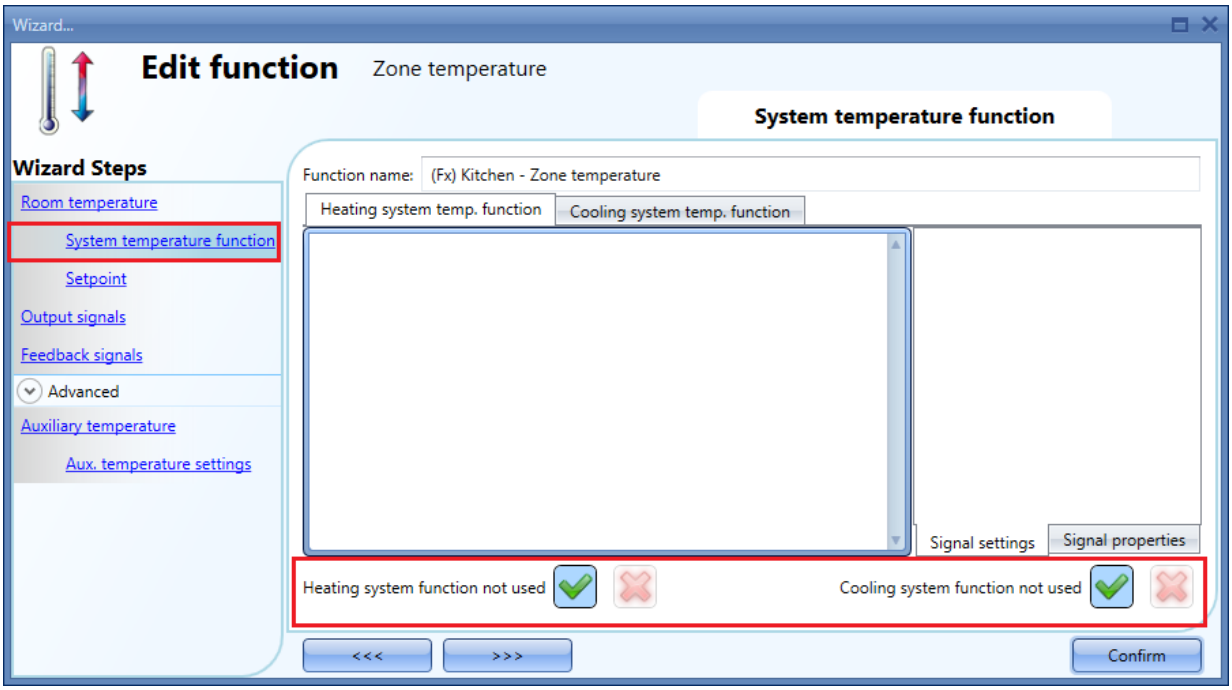
When more temperature values are used, the temperature used for the control is calculated as the average value of all the added signals (room temperature signals + Temdis and/or Masterglass temperature signals).
This average value is also the one displayed by the TEMDIS module plus the configured offset value.

This offset value is used in the calculation of the regulating temperature for heating/cooling. It can be set in the range [-5, +5] and it is added to the average temperature value measured.



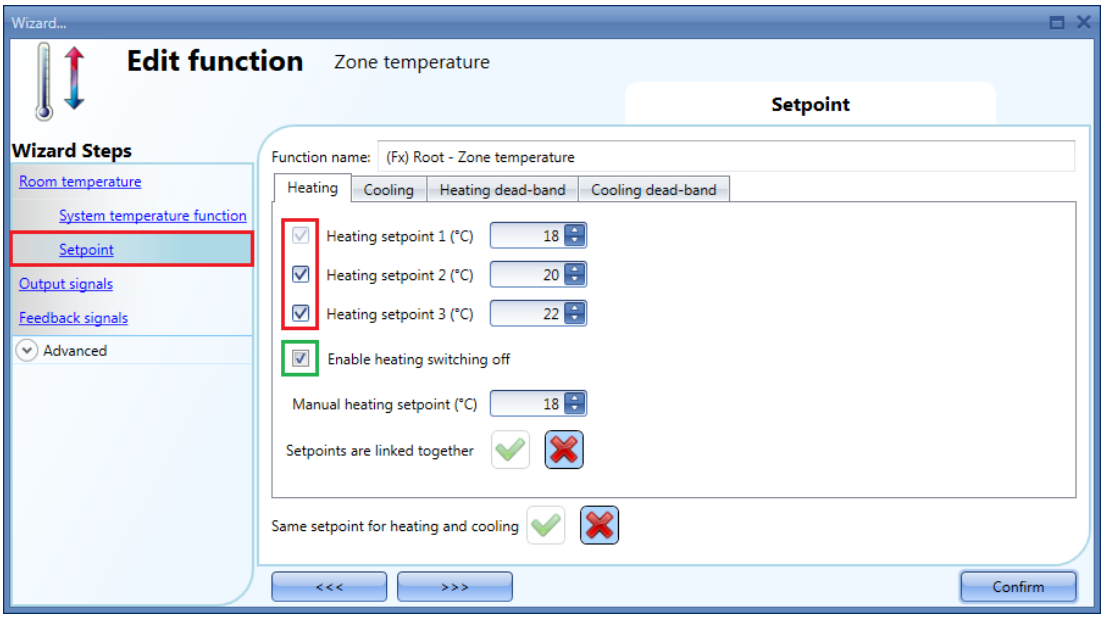
10.16.2 How to use a zone temperature function without a heating/cooling control system function

The user can create a *Zone temperature* function for Heating and/or Cooling control without needing to link it to the heating/cooling control function.
In the option *System temperature function* of the wizard, if there is no main function selected, the green "V" icon is selectable and the user can manage either only heating or only cooling, or both.
If at least one heating system temperature or cooling system temperature is present, this option will be unavailable.



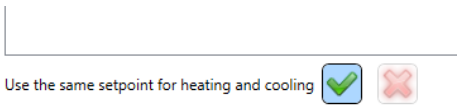
10.16.3 How to configure the setpoints

Up to three different set points can be used for the control of the zone temperature: T1 (comfort), T2 (activity) and T3 (economy).
To change the predefined set point values, the user should click on the relevant field in the wizard of the zone temperature function. There are two steps to configure the set point values used for heating and cooling.



The user can select one, two or three setpoints (T1, T2 ,T2) by clicking on the relevant box/es.
When the option *Enable heating switching off* is selected, this allows the user to switch the heating off using a TEMDIS display or remotely via the webserver (see green rectangle in the picture above). If this feature is not enabled, this possibility will be hidden in the TEMDIS display and webserver.

It is possible to select the green "V" icon in the option *Use the same setpoint for heating and cooling*: by doing this, the same set points used for heating are also used for cooling. With this option enabled, in the Temdis display, even if cooling is active, the home owner needs to manage only the heating set points.

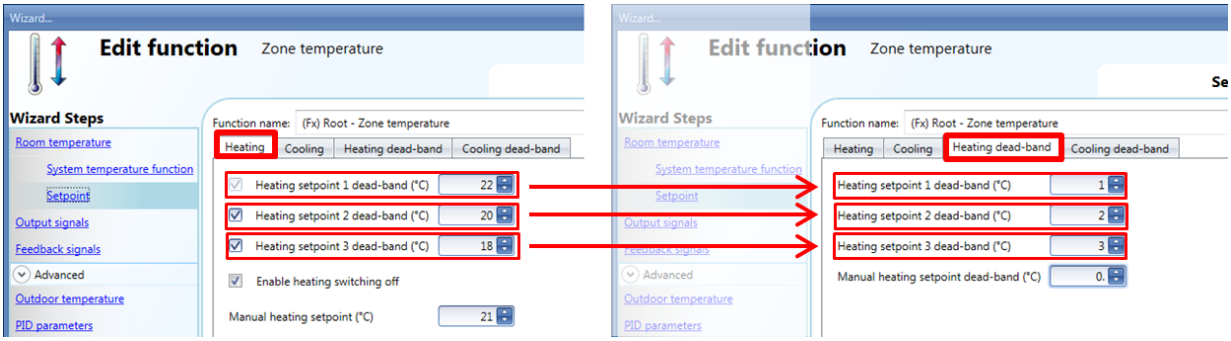


10.16.4 How to use the heating and cooling dead-bands

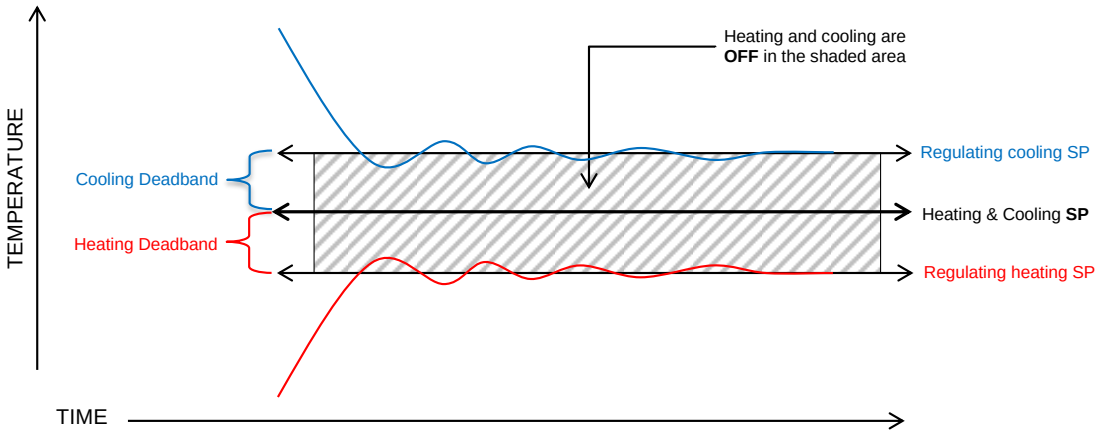
Advanced temperature features, such as dead-bands, allow for a more comfortable environment with more control over heating and cooling when they are needed at the same time, mainly in building applications.
In an application where two outputs are used to control heating and cooling of the same process, this has the effect of defining a band around the setpoint in which neither the heating nor the cooling output is on. This avoids having the heating and cooling devices working against one another.
However, if the application sometimes requires use of heating and cooling simultaneously, a negative dead-band value can be specified.

To allow you to optimize the control of your process, the heating dead-band and the cooling dead-band can be different values. When the dead-band value is applied to an output, it has the effect of shifting the setpoint (see example below).

The user can set up three dead-bands for heating and cooling, each working with the relevant setpoint T1 (setpoint 1), T2 (setpoint 2) and T3 (setpoint 3).



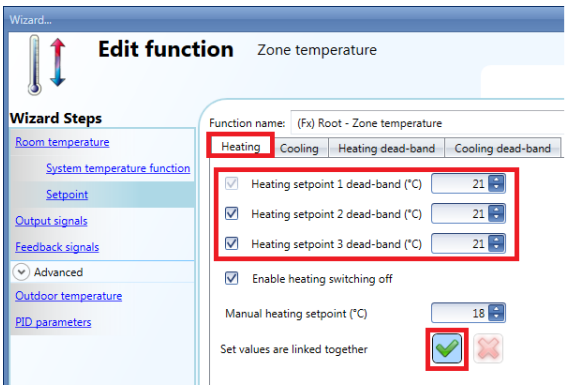
In the example shown above, setpoint1 is 21° (22-1) so heating will be switched off at 21° and switched on according to the set hysteresis; setpoint2 is 18° (20-2) so heating will be switched off at 18° and switched on according to the set hysteresis; setpoint3 is 15° (18-3) so heating will be switched off at 15° and switched on according to the set hysteresis.

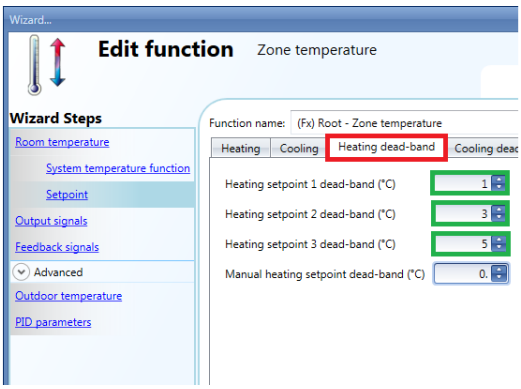


There are two ways of working with dead-bands:

1) The three setpoints can be independent: the option *Setpoints are linked together* is not enabled. In this situation the user can change one setpoint without affecting the others and the system will show three different values when selecting T1, T2 or T3. If dead-bands are used, the three setpoints are shifted as explained above.

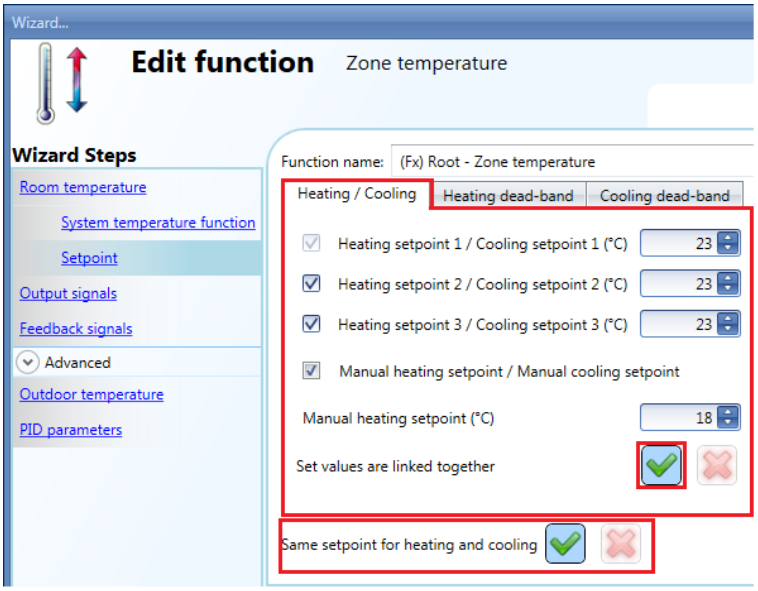
2) The three setpoints can be linked together: the option *Setpoints are linked together* is enabled. In this case the setpoint is only one (setpoint1, setpoint2 and setpoint3 have the same value) and the three regulating levels are managed using the dead-bands.



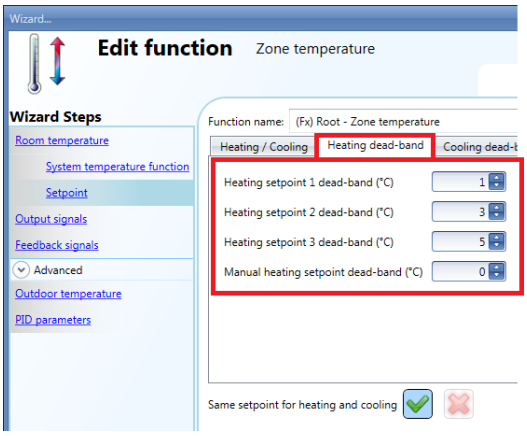


In this situation, if setpoint1 is selected, the regulating level is 20° (i.e 21-1), if setpoint2 is selected, the regulating level is 18° (i.e 21-3), if setpoint3 is selected, the regulating level is 16° (i.e 21-5). In the TEMDIS and webserver all the three setpoints will be 21°, and the changing of any one of them will also change the other two.

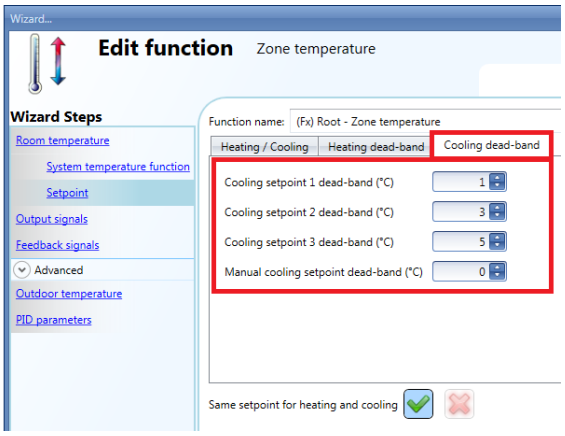
Example:
The following example shows how the system behaves with the following settings, where we can have one setpoint if an office is occupied (comfort level), a second one if the office is not occupied (economy level), and a third one during the night (night setback).



Heating dead-band

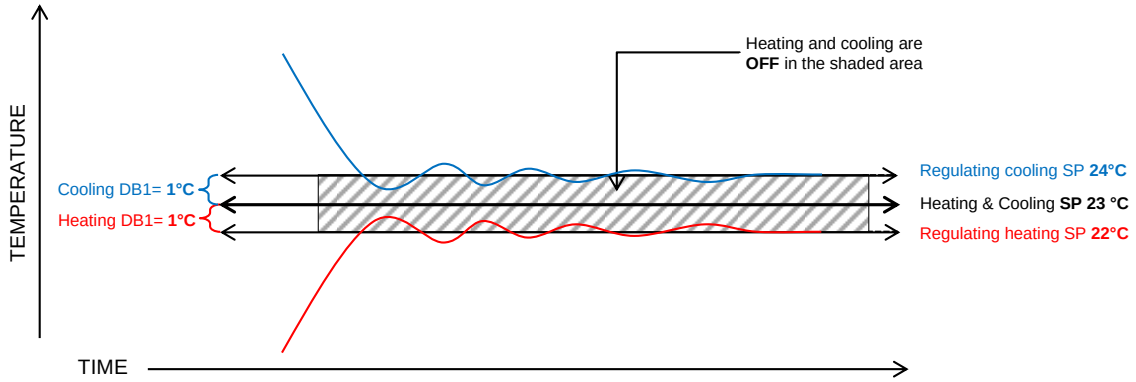


Cooling dead-band



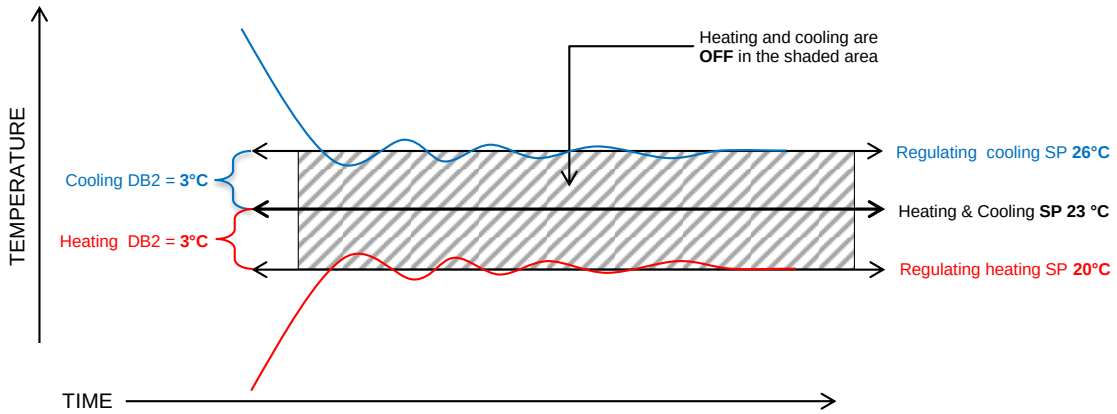
Office occupied- comfort setpoint
Heating and cooling setpoint=23°C, Dead-band 1°C

Heating will be regulated at 22°C
Cooling will be regulated at 24°C



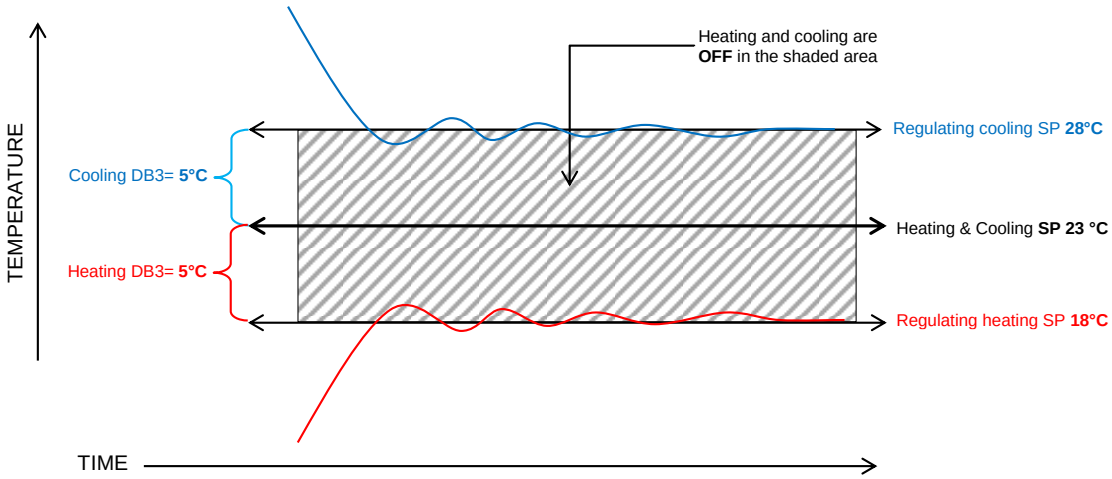
Office not occupied- economy setpoint
Heating and cooling setpoint=23°C, Dead-band 2= 3°C

Heating will be regulated at 20°C
Cooling will be regulated at 26°C



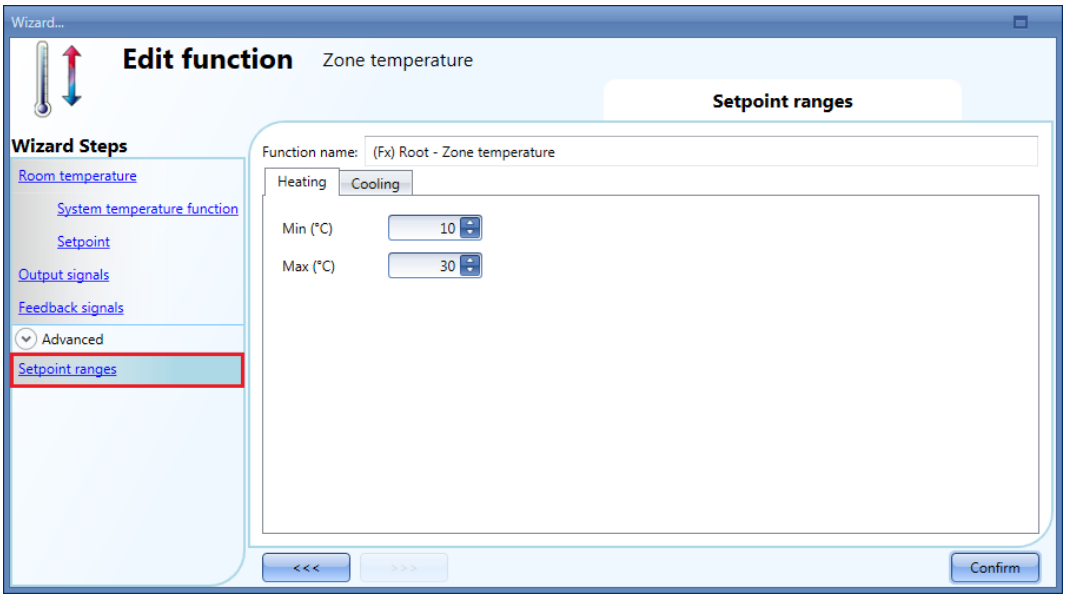
Night time- night setback
Heating and cooling setpoint=23°, Dead-band 3= 5°C

Heating will be regulated at 18°
Cooling will be regulated at 28°



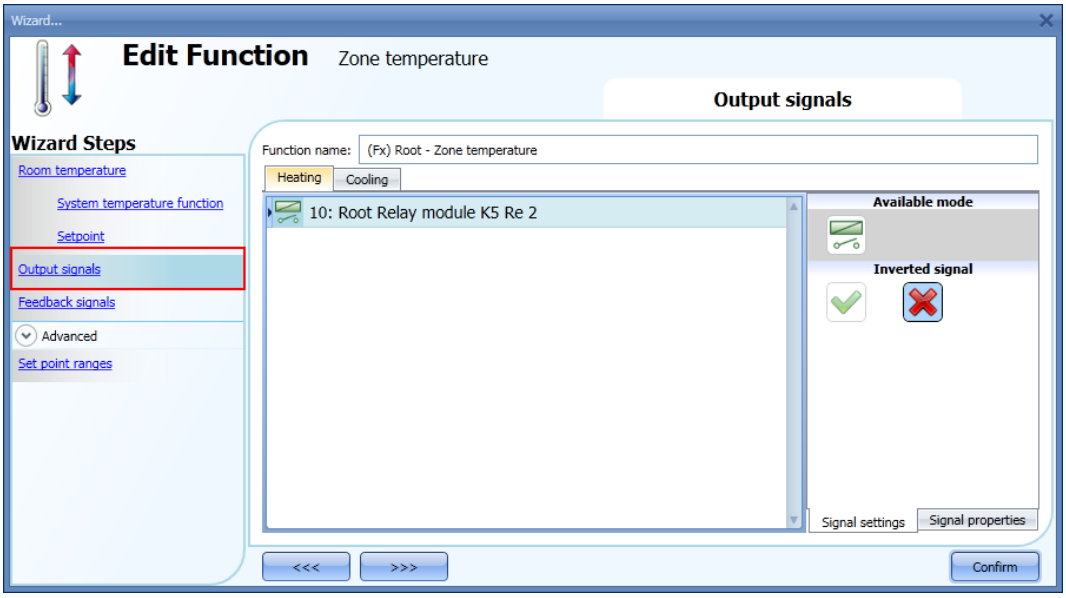
10.16.5 How to set the maximum and minimum values for the setpoints

In the Advanced section of the wizard, in *Set point ranges*, the user can select the operating range for the set points. This range will be used by the TEMDIS, Masterglass Temdis modules and all remote commands such as webserver.



10.16.6 How to add the heating output

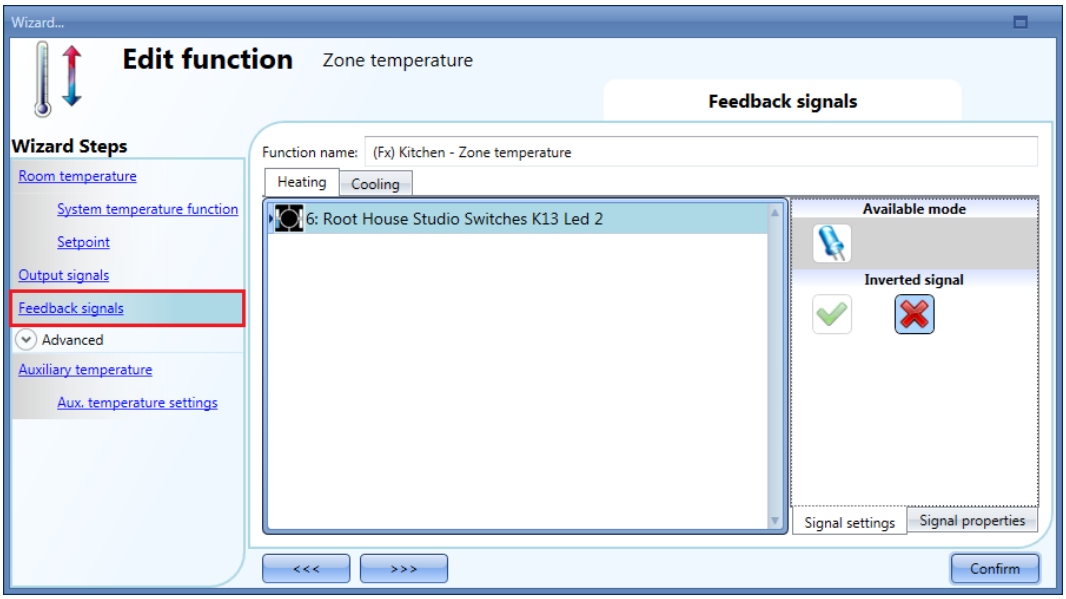
The heating output must be added in the relevant field of the function (see picture below)



The on/off switching of the output is managed by the system temperature function according to the timing described in the paragraph *How to add the heating output*.

10.16.7 How to set a feedback signal

The feedback signal can be used to advise the user about the status of the zone temperature function. When the signal is on, the system function is active and the output is on. When the feedback signal is off, the output is off.
To add the feedback signal, the user should select the relevant section in the wizard of the function, then double click on the *Signals* window and select the feedback signal from the list of those available.

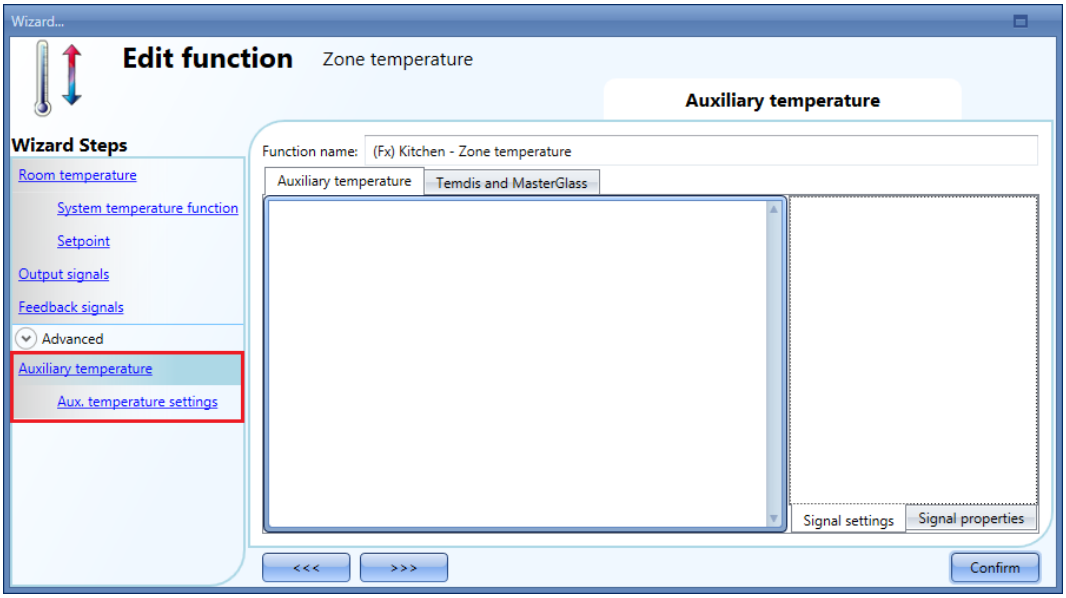


The inverted logic for the feedback can be selected in the *Signal settings* tab window.

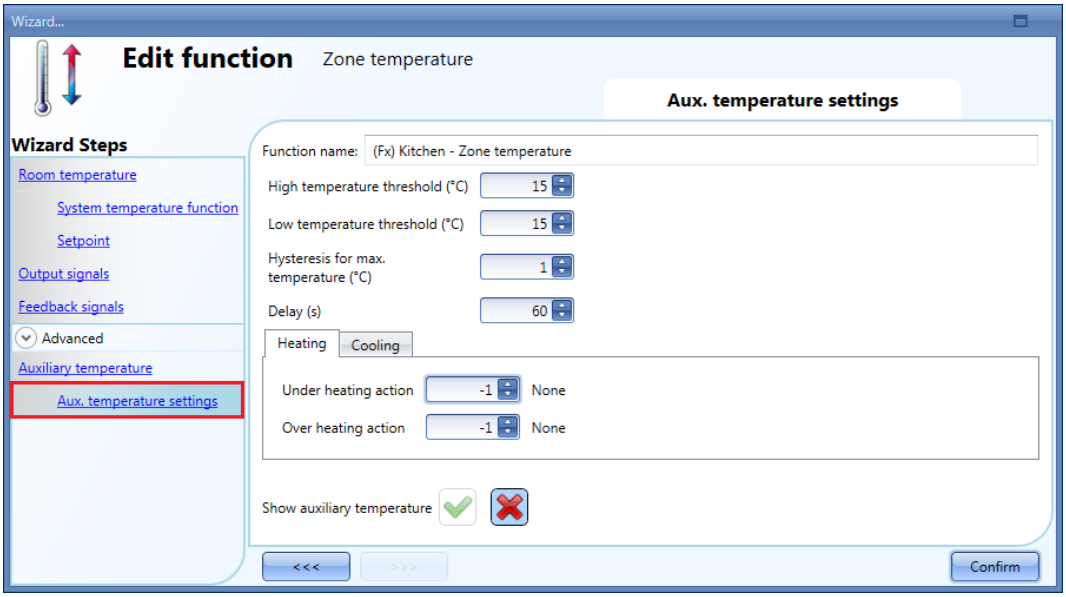
10.16.8 How to use an auxiliary temperature to monitor the temperature in the zone

The user can use an extra temperature signal to keep the zone “safe”. In the *Advanced* section, the user should select the relevant *Auxiliary temperature*.
Two different types of signals can be used: in the *Auxiliary temperature* signals window, the user should add one or more temperature signals read from different temperature modules (e.g. BSI-TEMANA-U, SHA4XLS4TH, SHE5XLS4TH, SHGxxxW-BLSx). In the *Temdis and MasterGlass* signals window, the user should add the TEMDIS “T Auxiliary” signal (usually this is the probe that is often placed in the floor to monitor the temperature in the heating tubes).

When more auxiliary temperatures are used, the auxiliary temperature value is calculated as the average value of all the added ones. Should one of them be faulty, the average value is calculated from the other ones.



In the *Aux. temperature settings* of the function, the user can set two different thresholds: the *High temperature threshold* (over value) and the *Low temperature threshold* (under value).
The *Hysteresis* and the *Delay on* time are applied to both the thresholds.

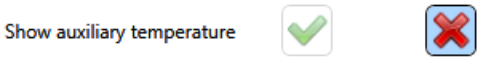


The user can then select the action to be performed when the temperature goes over the *high threshold limit* and the action when the temperature goes below the *low threshold limit*. In the table below the possible actions are shown.

Under/over heating action	
-1	No action
0	Force the heating OFF
1	Force the heating ON
Under/over cooling action	
-1	No action
0	Force the cooling OFF
1	Force the cooling ON

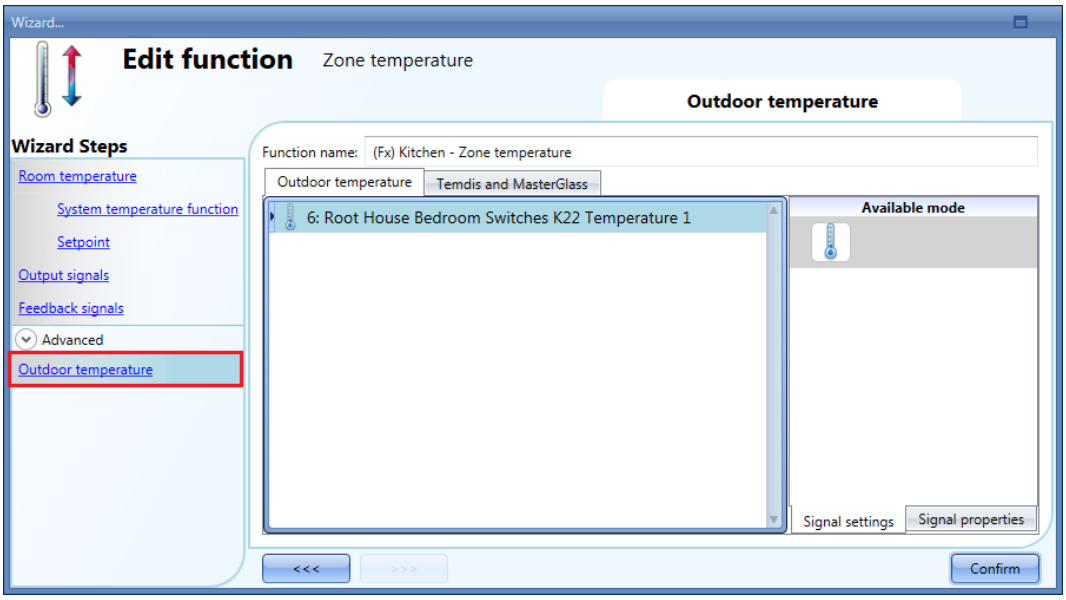
If an ON/OFF action is enabled, it has priority over the regulation according to the room temperature.

By selecting the green “V” icon at the bottom of the window *Show auxiliary temperature*, the auxiliary temperature is shown in the Temdis Display.



10.16.9 How to enable the display of the external temperature in the TEMDIS display

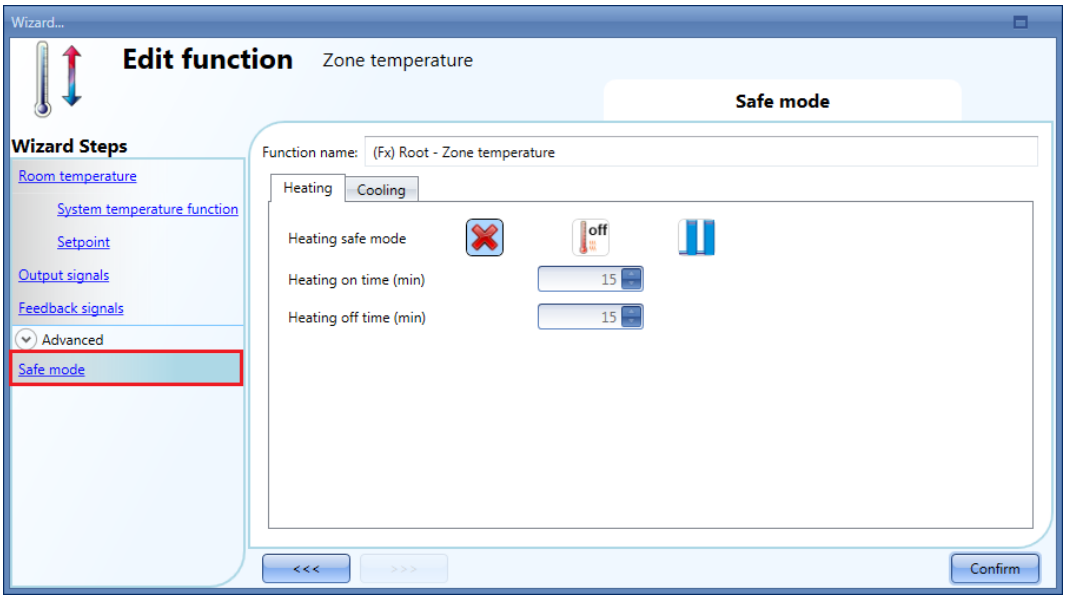
To add the external temperature signal, the user should select the relevant section in the wizard of the zone function, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



Two different types of signals can be used: in the *Outdoor temperature* signals window, the user can add one or more temperature signals from the sensor modules (e.g. BSI-TEMANA-U, SHA4XLS4TH, SHE5XLS4TH, SHGxxxW-BLSx). In the *Temdis and MasterGlass* signals window, the user can select to use the TEMDIS “T Auxiliary” signal.

10.16.10 How to set the safe mode

The safe condition can be set in order to keep the system in a safe working mode in case of a fault in the regulating temperature. The user should choose different actions to be performed for heating and cooling. To configure the safe mode settings, the user has to click on the relevant field *Safe mode* of the zone function wizard (see picture below).



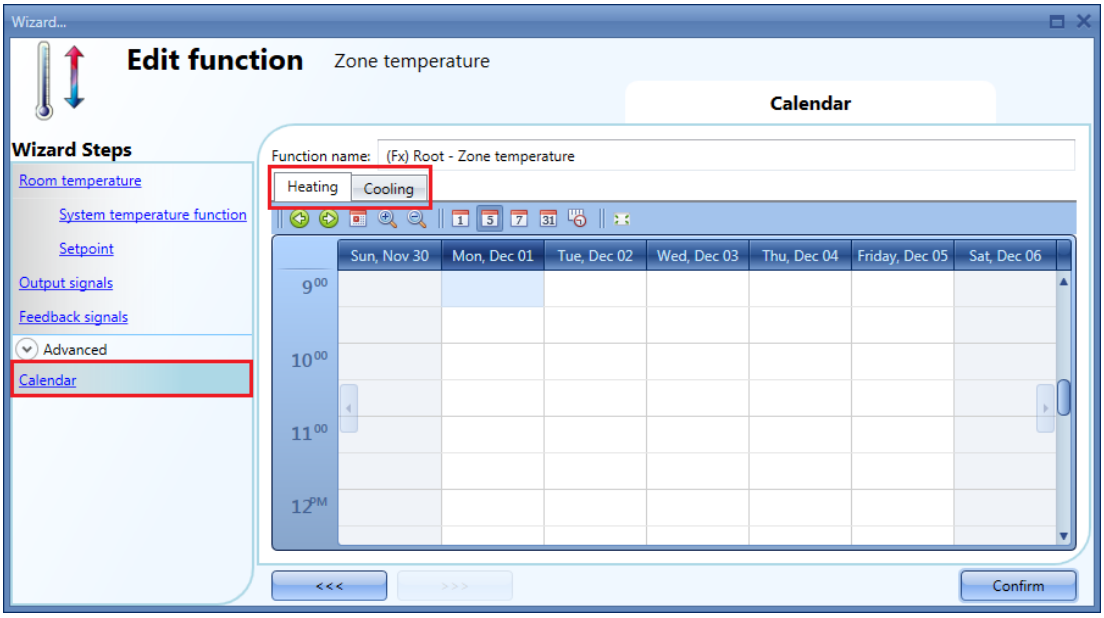
The user can then select the action to be performed when the safe condition is needed (faulty temperature sensor). In the table below the list of available actions is shown.

Safe action (heating)	
	No action
off	Force the heating output OFF
	Force heating output to recycle (the output goes on/off according to the time settings). When this icon is selected, the on/off time settings are automatically enabled.
Safe action (cooling)	
	No action
off	Force the cooling output OFF
	Force cooling output to recycle (the output goes on/off according the time settings). When this icon is selected, the on/off time settings are automatically enabled.

10.16.11 How to manage heating/cooling using the calendar automation

The user can program a calendar automation in order to activate/deactivate the heating/cooling in a defined period of time. There are two ways of managing the zone function with the calendar: by setting the “local” calendar inside the function or by using a “global” calendar function.

Local calendar
To enable this, the relevant menu has to be selected in the *Advanced* section (see picture below).



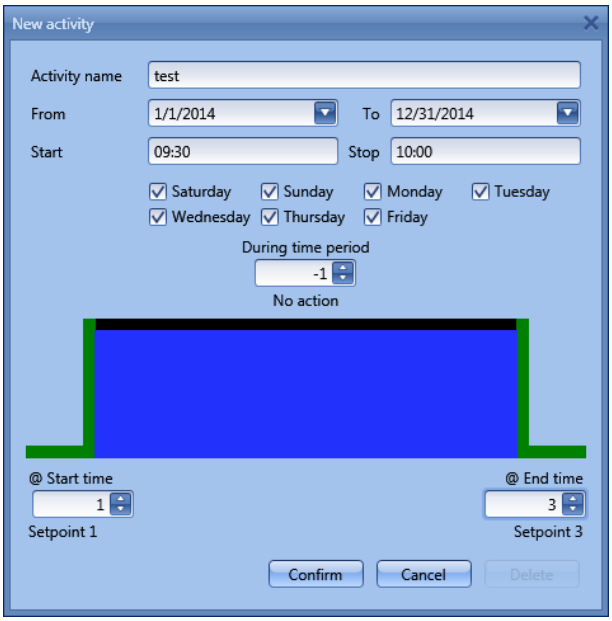
The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day.
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

Calendar activity

Once the user has selected his preferred visualization, to enter a time band he should double click on the required day and the following window will appear.



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From*:* the start date for the calendar activity.

To*:* the end date of the calendar activity.

Start*:* The time of the start of the activity..

End: The time of the end of the activity.

The activity will be repeated in the following years, using the same dates and times.

The action managed by the calendar can be of two types:

1) Event activity: the system will perform the selected action only at start time and end time and they are managed as events (when an event occurs the set point is overwritten)
2) Level activity: during the time band, all the automation can be disabled.

The user can decide what action he wants to be performed when the time band starts and ends.

@ ***start time***: in this field the user can select the action to be performed at the start time.
The available actions are:

- (-1) No action: nothing will be done at start time
- (0) Switch OFF: heating/cooling is switched Off
- (1) Setpoint 1: heating/cooling setpoint 1 is selected
- (2) Setpoint 2: heating/cooling setpoint 2 is selected
- (3) Setpoint 3: heating/cooling setpoint 3 is selected

@ ***end time***: in this field the user can select the action to be performed at the end time.
The available actions are:

- (-1) No action: nothing will be done at end time

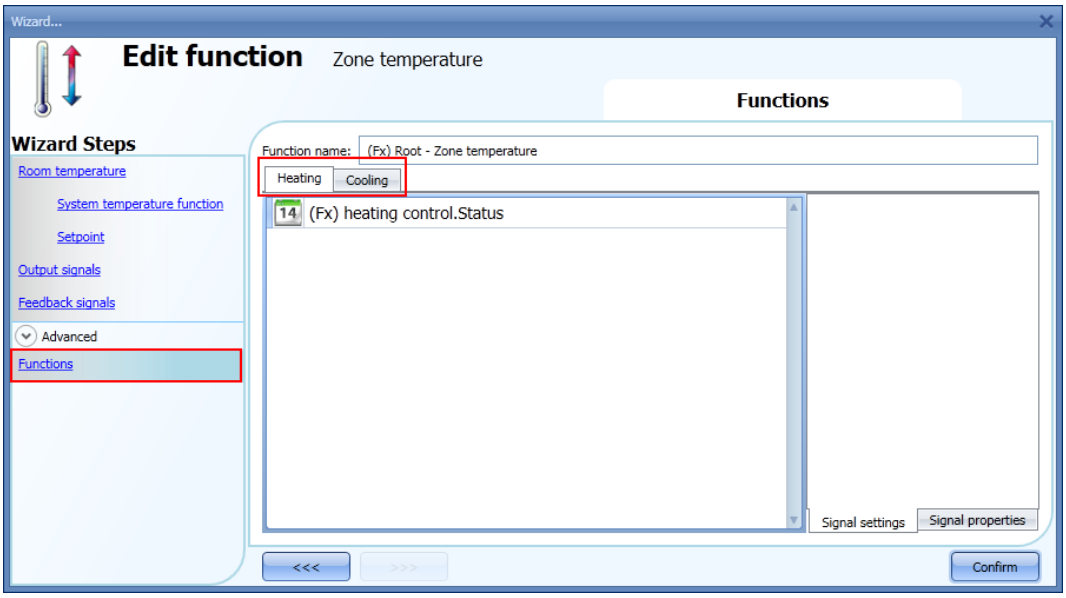
- (0) Switch OFF: the heating/cooling is switched Off
- (1) Setpoint 1: heating/cooling setpoint 1 is selected
- (2) Setpoint 2: heating/cooling setpoint 2 is selected
- (3) Setpoint 3: heating/cooling setpoint 3 is selected

If ***Level activity*** is selected, the user can disable the automation of the calendar.

During time period: in this field the user selects the action he wants to perform during the time band.
- No action
- Disable automations

Global calendar

The global calendar must be created before using the calendar function.
Once the global calendar is created, it must be added in the *Functions* field (for more details see also the paragraph *Setpoint management using a function status*).
The behavior of the calendar is the same as that described for the local calendar.

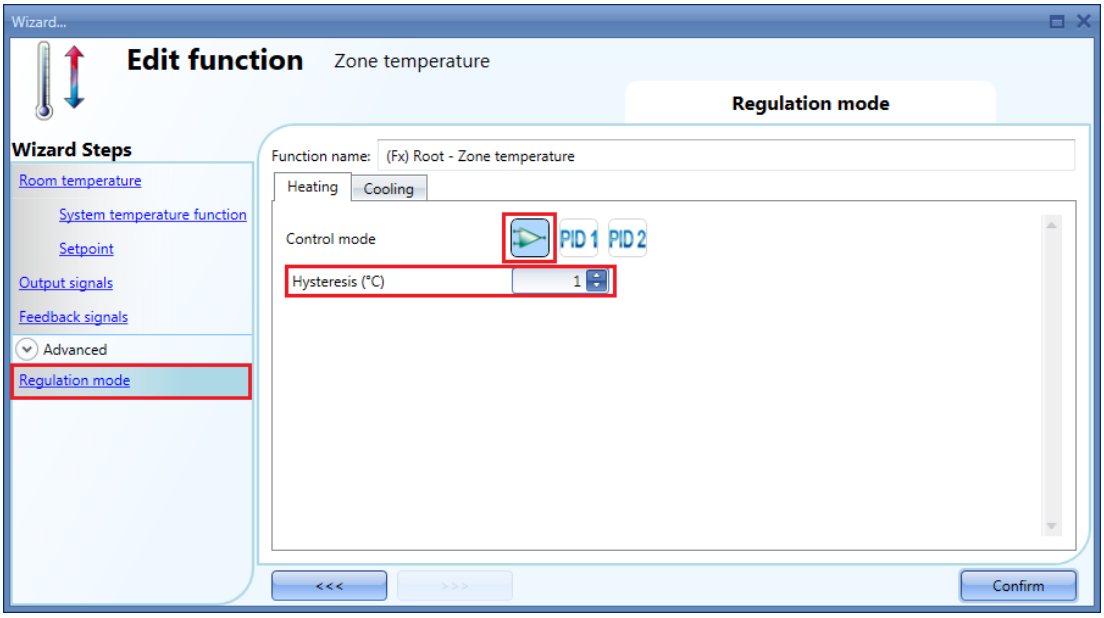


10.16.12 How to change the set point using the function status

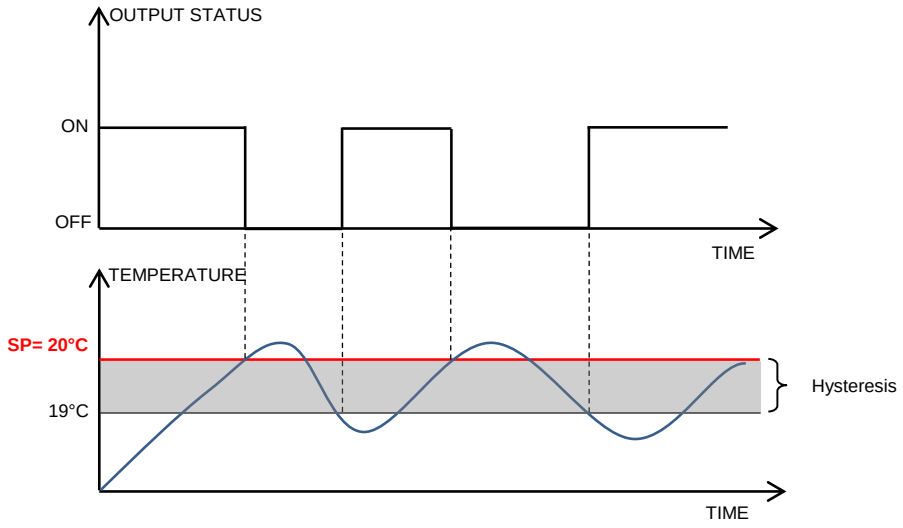
A generic function can be linked to the zone in order to change the setpoint according to its status: when the generic function status is one, setpoint 1 is selected, when it is two, setpoint 2 is selected, when it is three, setpoint3 is selected. When the status of the linked function is zero, the heating/cooling is switched off.
An example of the application could be an external calendar used to switch off all the temperature functions.

10.16.13 How to set the ON/OFF control

If the first icon is selected, the temperature function works as an on/off function, regulating according to the setpoint and the hysteresis.



If the setpoint is 20° and the hysteresis is 1°, the output is switched off at 20° and activated again at 19°. (see the graph below)



10.16.14 How to set the PID

If the user wants to use the PID regulation algorithm, the *Regulation mode* field must be selected in the *Advanced* sections.

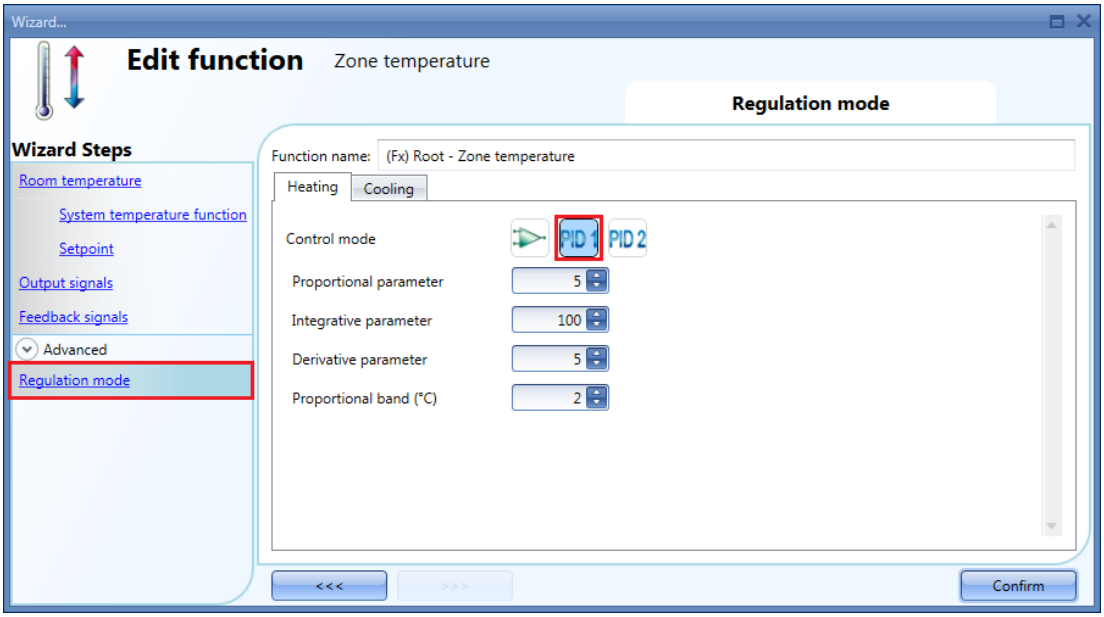
The PID output is a percentage from 0% to 100%, which can be applied on a cycle time relay output or on an analogue output module.

Example 1: relay output and cycle time of 10 seconds.
If the PID output is 0%, the relay will be off for all of the 10 seconds.
If the PID output is 30%, the relay will be off for 7 seconds and on for 3 seconds.
If the PID output is 100%, the relay will be on for all of the 10 seconds.

Example 1: 0-10 analogue output SHPOUTV224 ***(Only PID2 algorithm)***
If the PID output is 0%, the output of the SHPOUTV224 is 0V.
If the PID output is 30%, the output of the SHPOUTV224 is 3V.
If the PID output is 100%, the output of the SHPOUTV224 is 10V.

10.16.15 How to set the PID 1 mode

By default, the Regulating mode is set as a *PID 1* mode (see the red rectangle in the picture below).



Proportional parameter (default value 5)
The proportional action gives a change to the input (temperature value measured) directly proportional to the control error (error=setpoint-temperature).
This adds an amount to the output based on the current error.

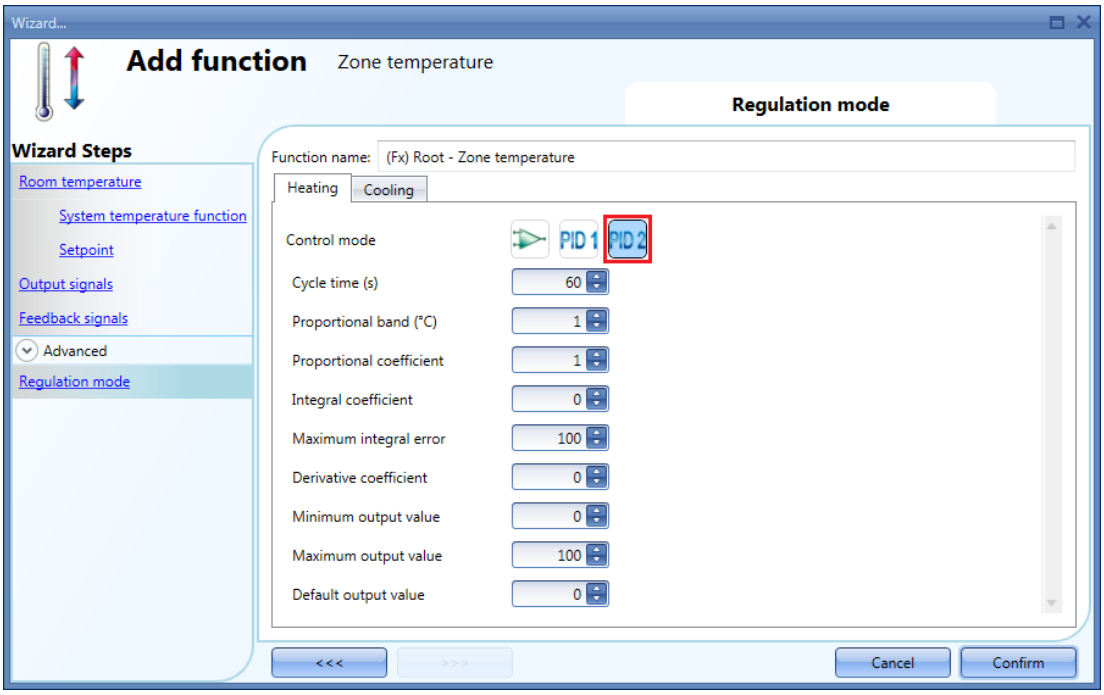
Integrative parameter (default value 100)
This adds an amount to the output based on the sum of the error.
It gives a change to the output proportional to the integrated error, and its main purpose is to reduce the error by increasing or decreasing the output, until the temperature value reaches the desired setpoint value.
By default the Integral term is disabled.

Derivative parameter (default value 5)
This subtracts an amount from the output based on the rate of change of the error.
The less commonly used derivative action is used in some cases to speed up the response or to stabilize the system, and it gives a change to the output inversely proportional to the rate of change of the controlled variable. By default the Derivative term is disabled (value 0).

Proportional band (°C) (default value 2)
When Proportional-Integral-Derivative (PID) control is used, the controller modulates output power by adjusting the output power percentage within a proportional band.
Power is proportionally reduced as the process temperature gets closer to the setpoint temperature.
It is the width of the band above (cooling: direct-acting outputs) or below (heating: reverse-acting outputs) the setpoint within which the controller modulates the output as the process value approaches the setpoint.

10.16.16 How to set the PID 2 (PID advanced)

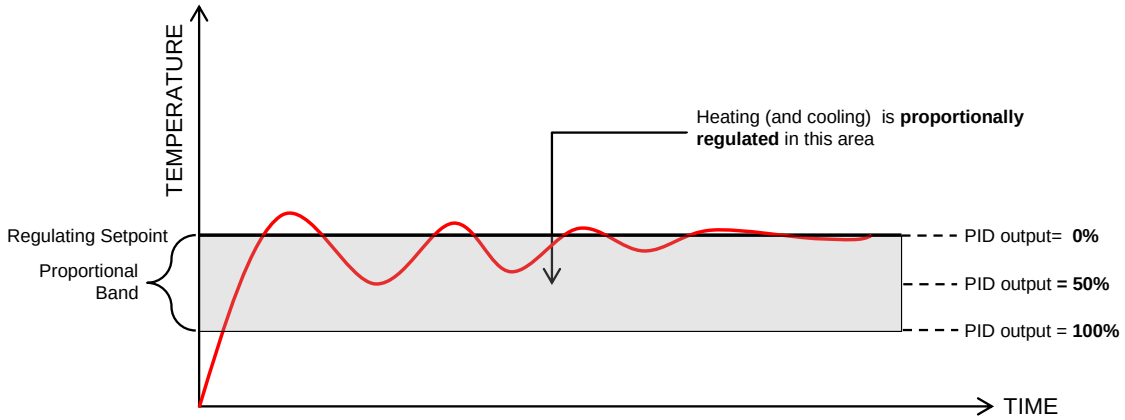
If the user wants to use the PID 2 regulator and be free to define all the parameters, the Regulation mode field must be selected in the Advanced section and the third icon should be checked (PID 2). The available settings are shown in the picture below:



Cycle time (s)
When an output is used for PID control, a cycle time that is appropriate for the application must be specified. The cycle time is the period of time in which the PID makes its calculation and updates the output.
For example, let's suppose that the cycle time is set to 10 seconds and that the control algorithm calculates the output percentage equal to 50%. A normally open relay will be closed for 5 seconds and open for 5 seconds in each ten second cycle. If the output percentage changes to 70%, the relay will be closed for 7 seconds and open for 3 seconds in each ten second cycle.

Proportional band (°C) (default value 1)
When Proportional-Integral-Derivative (PID) control is used, the controller modulates output power by adjusting the output power percentage within a proportional band.

Power is proportionally reduced as the process temperature gets closer to the setpoint temperature.
It is the width of the band above (cooling: direct-acting outputs) or below (heating: reverse-acting outputs) the setpoint, within which the controller modulates the output as the process value approaches the setpoint.



Proportional coefficient (default value 1)
The proportional action gives a change to the input (temperature value measured) directly proportional to the control error (error=setpoint-temperature).
It adds an amount to the output based on the current error.

Integral coefficient (default value 0)
This adds an amount to the output based on the sum of the error.
It gives a change to the output proportional to the integrated error, and its main purpose is to reduce the error by increasing or decreasing the output, until the temperature value reaches the desired setpoint value.
By default the Integral term is disabled.

Maximum integral error (default value 100)
This is the maximum value the integral term can have.

Derivative coefficient (default value 0)
This subtracts an amount from the output based on the rate of change of the error.
The less commonly used derivative action is used in some cases to speed up the response or to stabilize the system, and it gives a change to the output inversely proportional to the rate of change of the controlled variable. By default the Derivative term is disabled (value 0).

Minimum output value
This is the minimum value the output of the PID algorithm can have

Maximum output value
This is the maximum value the output of the PID algorithm can have

Default output value
This is the default value applied to the PID output when the input is faulty or not available, or the disable status is active (default 0).

To have some hints on how to set these parameters, please look at **How to set PID parameters.pdf.**

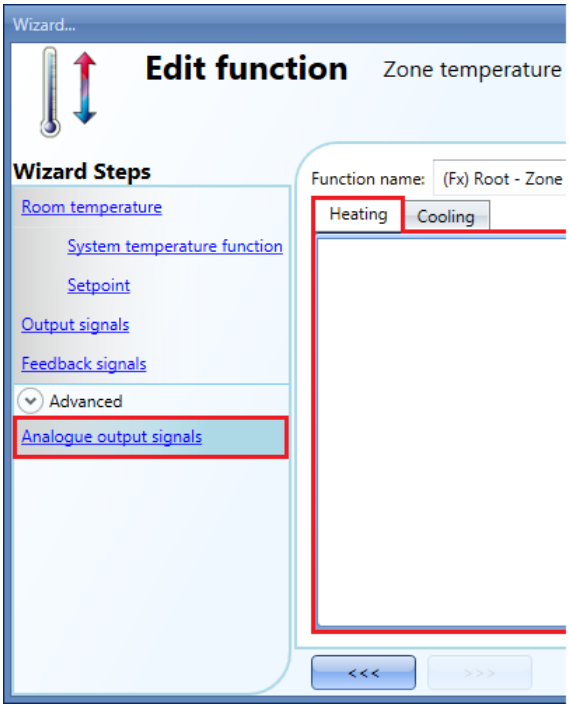
10.16.17 How to add Analogue output signals

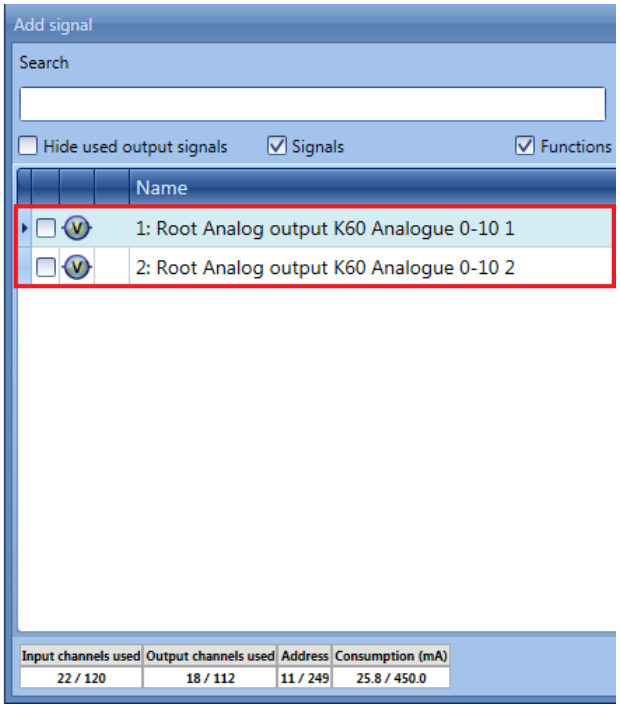
This field is enabled only if PID2 is checked in the field *Regulation mode*.
The analogue output signals permit an actuator to be managed like a motorized valve driven by 0…10 V DC or by a 3-position signal: the output is set according to the PID calculation.

If a SHPOUTV224 module is used:

- If the PID output is 10%, the SHPOUTV224 output is 1V
- If the PID output is 60%, the SHPOUTV224 output is 6V

To add the *Analogue output signals* the user should click on the relevant field in the *Advanced* sections then double click on the *Signals* window and select the signal from the list of those available (see picture below):





The output types available for this field are those from the module SHPOUTV224 (to control actuators with 0…10 V inputs).

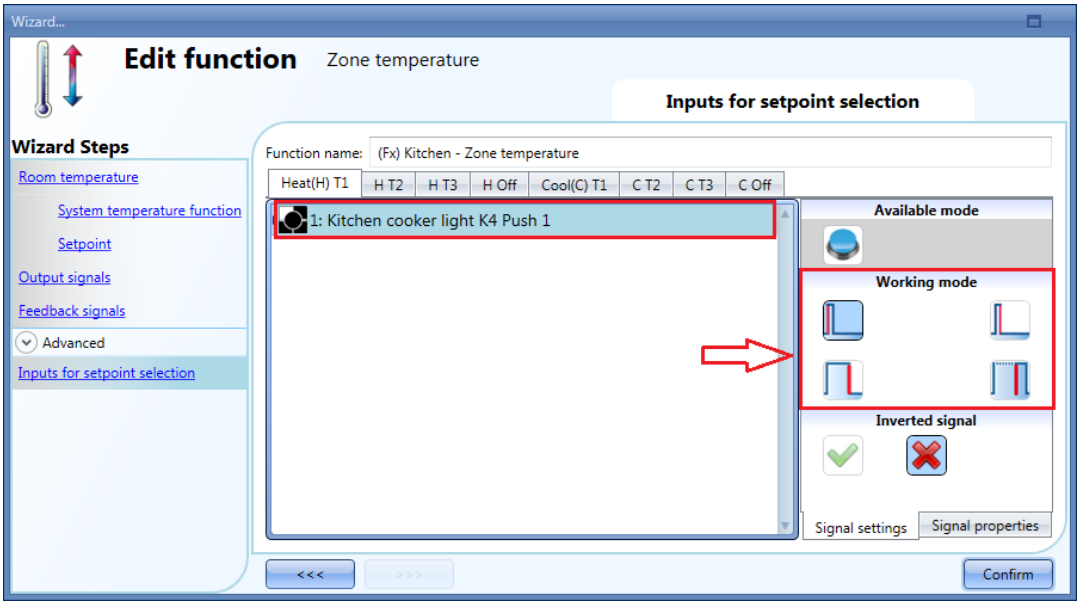
10.16.18 How to activate the set point with pushbuttons and functions

The user can manage the setpoint activation by using signals such as pushbuttons, switches and functions.
The activation of the setpoint works on events: when the pushbutton is pressed, the setpoint is changed. The same input can also be used to activate two different setpoints: for example, with a short click setpoint 1 is selected, while with a long pressure the heating is switched OFF.

The user can select different working modes to activate the required setpoint1 (see table below).

Working mode	Action
	Setpoint is activated when the button is pressed (rising edge)
	Setpoint is activated when the button is released (falling edge)
	Setpoint is activated after a long pressure
	Setpoint is activated after a very long pressure

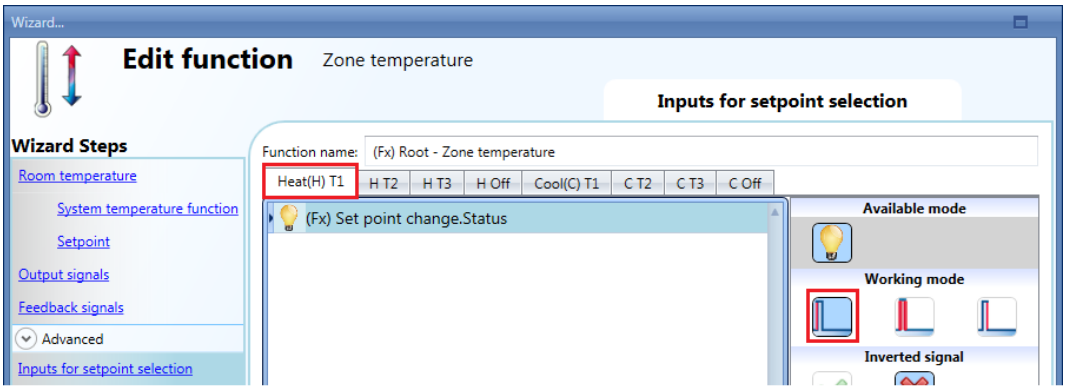
In the next picture, the push button is used to activate the heating setpoint1 as soon as it is pressed.

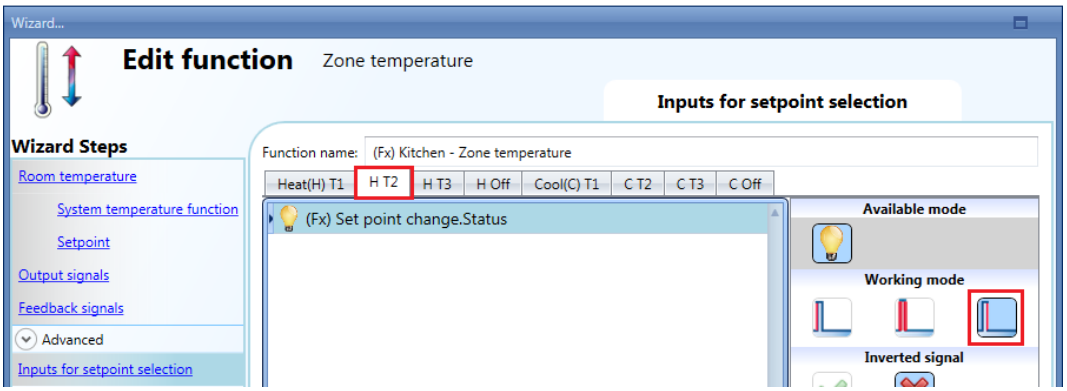


If the signal used is a function, the user can select different working modes to activate the required setpoint (see table below).

Working mode	Action
	Set point is activated when the function is activated (function goes On).
	Set point is activated when the function is deactivated (function goes Off).
	Set point is activated at each change of the status of the function (function goes on or off).

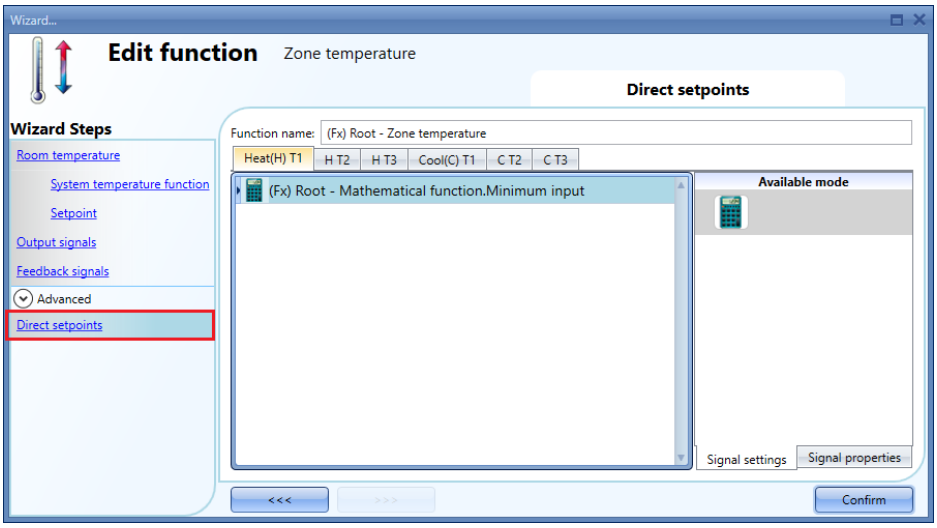
In the picture below, an example of configuration is shown, where the function "(FX) Set Point change" is used to set two different setpoints: when the light goes On, setpoint 1 is activated: when the light goes Off, setpoint 2 is activated.





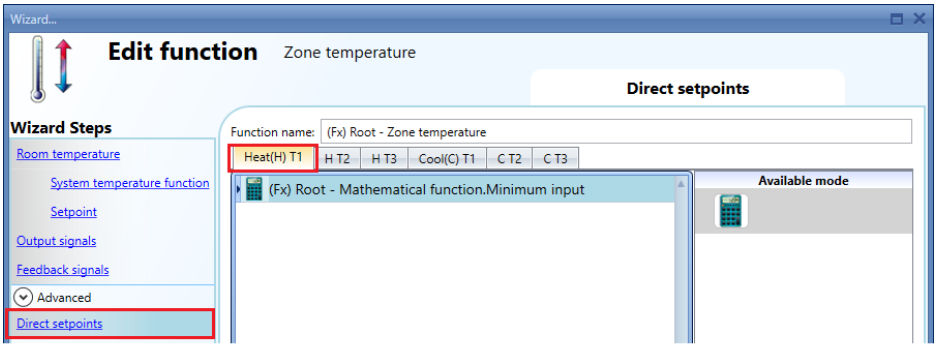
10.16.19 How to use external signals as Direct setpoints

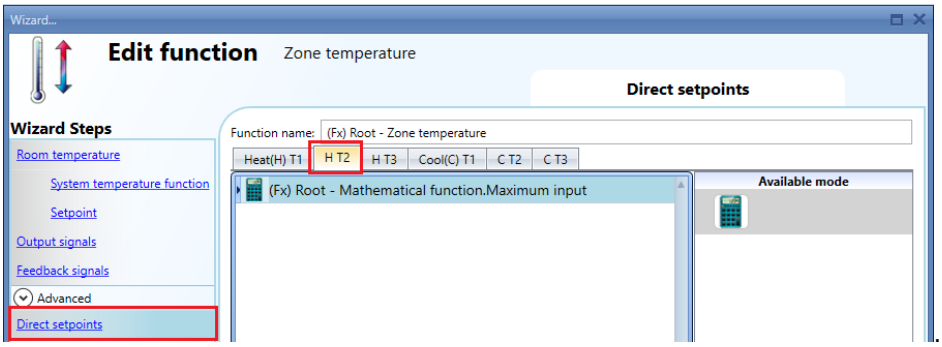
An external signal can be used as a direct setpoint for a zone temperature function, for example the output status from a *Mathematical* function.
To add the *Direct setpoints* the user should click on the relevant field in the *Advanced* sections, then double click on the *Direct setpoints* window and select the signal from the list of those available.



As shown in the picture above, a *Mathematical* function can be used as Heating Setpoint 1(T1) regulation, setpoint2 or setpoint3.

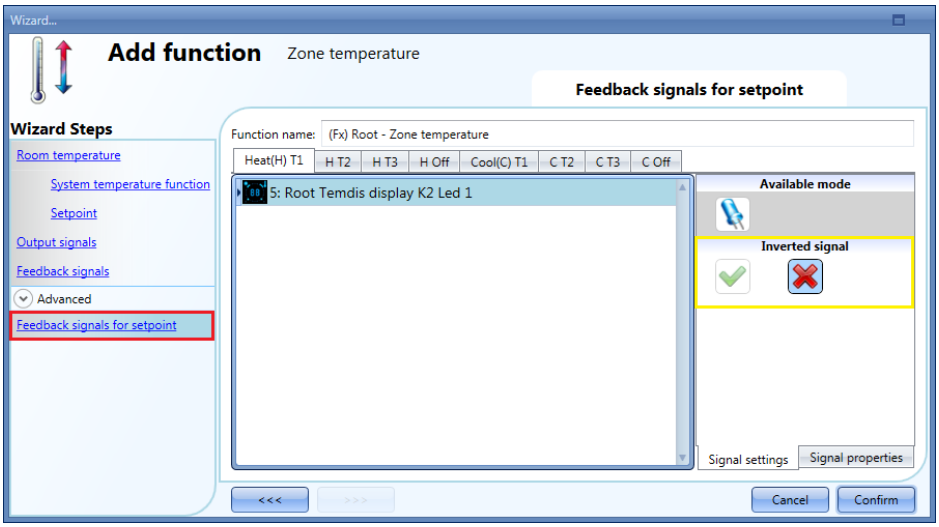
In the next picture, an example of configuration is shown: the *Minimum input* status of a *Mathematical* function is used to manage the heating setpoint1, while the *Maximum* input status of a *Mathematical* function is used to manage the heating setpoint 2.





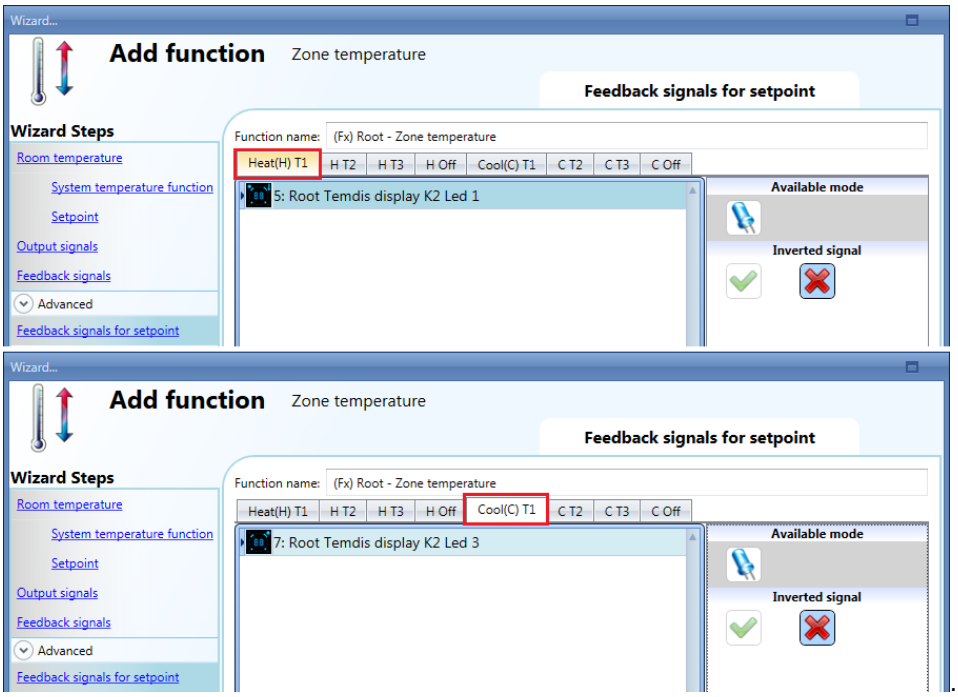
10.16.20 How to set the feedback signals for the selected setpoint

A feedback signal can be used to give information about the regulating setpoint, so that a LED switches on if the associated setpoint is active at that moment.
To add the *Feedback signals for setpoint* the user should click on the relevant field in the *Advanced* sections, then double click on the *Signals* window and select the feedback signal from the list of those available.



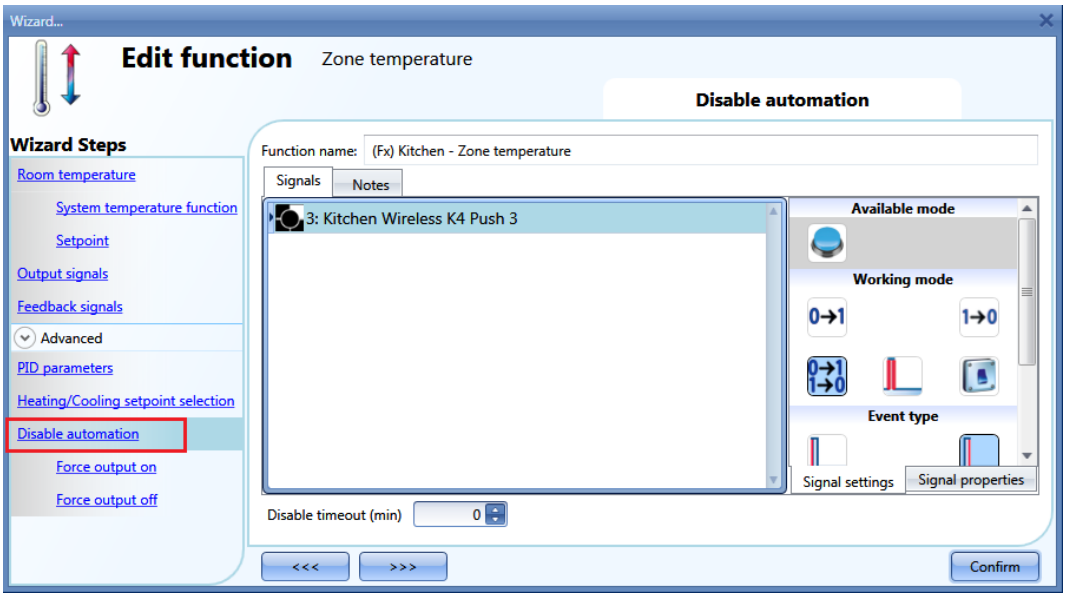
As shown in the picture above, LEDs can be programed to be on or off, indicating if the temperature function is regulating according to setpoint1, setpoint2 or setpoint3.
The inverted logic for the feedback can be selected as shown in the yellow rectangle in the picture above.

In the next picture, an example of configuration is shown: feedback Led1 turns on when heating setpoint1 is activated, while feedback LED3 goes on when cooling setpoint 1 is activated.

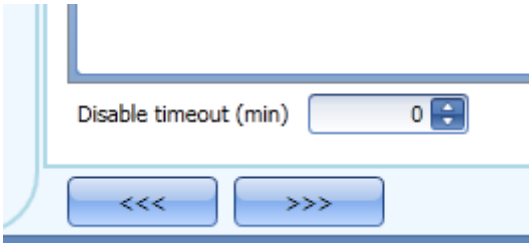


10.16.21 How to disable the automations

The user can disable the calendar automation by using a physical signal such as a pushbutton or by using the calendar level activity (for more info see the calendar paragraph).
To add the disable signal, the user should select *Disable automation* in the Advanced section, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



If the user needs to be sure that the disabling of the automation is automatically removed, the *disable automation timer* has to be set.
The timer starts each time the disable status is activated: the disable status is automatically deactivated as soon as its time has elapsed.
The maximum delay is 59minutes.



If the disable timeout is set to 0, the timer is disabled and the disable condition stays activated until it is removed manually.

If a push button is selected as the disable automation signal, the working mode, marked with the red rectangle, has to be carried out following the table below.

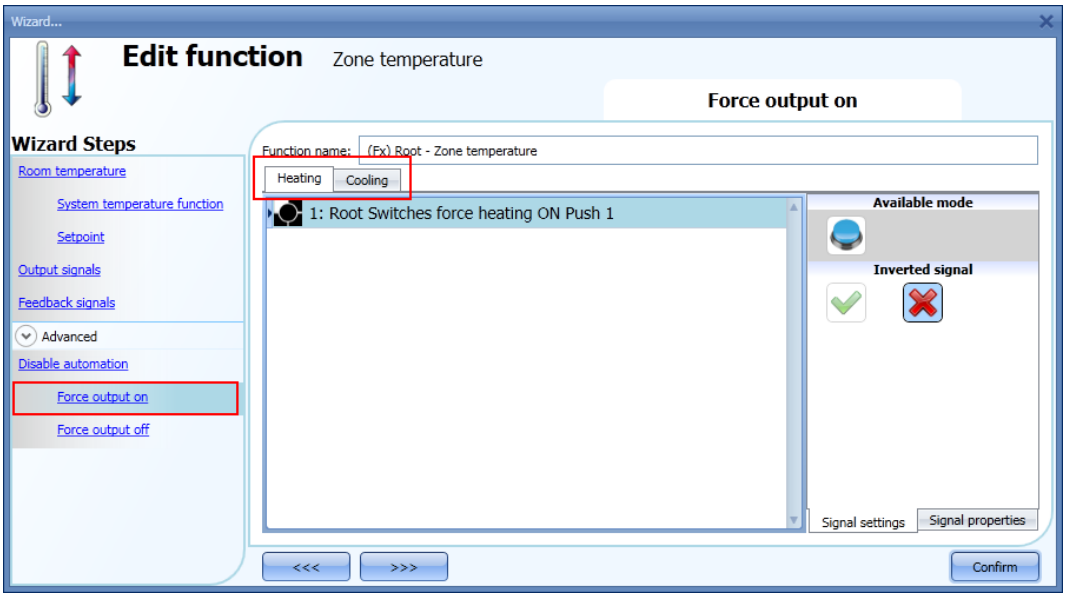
Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, the disable condition is activated.	After a short press (less than 1 second), at the release of the push button, the disable condition is activated.	After a long press, the disable condition is activated at the release of the push button.	After a very long press, the disable condition is activated at the release of the push button.
1→0	As soon as the push button is pressed, the disable condition is deactivated.	After a short press (less than 1 second), the disable condition is deactivated at the release of the push button.	After a long press, the disable condition is deactivated at the release of the push button.	After a very long press, the disable condition is deactivated at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the disable condition is activated/deactivated in toggle mode.	After a short press (less than 1 second), the disable condition is activated/deactivated in toggle mode at the release of the push button.	After a long press, the disable condition is activated/deactivated in toggle mode at the release of the push button.	After a very long press, the disable condition is activated/deactivated in toggle mode at the release of the push button.
	The disable condition is activated/deactivated in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The disable condition is activated	No action
1→0	The disable condition is deactivated	No action
0→1 1→0	The disable condition is activated/deactivated in toggle mode	No action
	The disable condition is activated/deactivated in toggle mode	The disable condition is activated/deactivated in toggle mode
	The automation is disabled	The automation is enabled

10.16.22 How to force heating/cooling ON

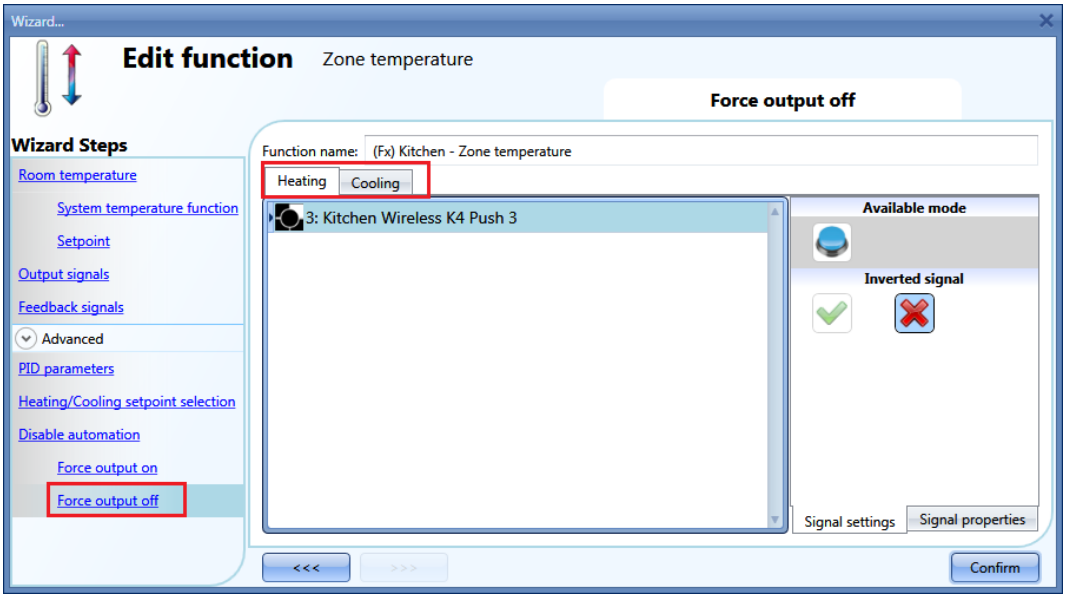
If the user wants to force the activation of the heating/cooling output regardless of all other signals used in the function, the *Force output on* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output on*, then double click on the signal window (there are two available windows: one for heating and one for cooling) and select the appropriate signal to use (see picture below).



Each signal used in the *Force output on* window works as a level signal. Until the signal is active, the heating/cooling output is forced on.
When the *Force output on* signal is activated and at the same time the *Force output off* signal is activated, the *Force output on* signal has priority.

10.16.23 How to force heating/cooling OFF

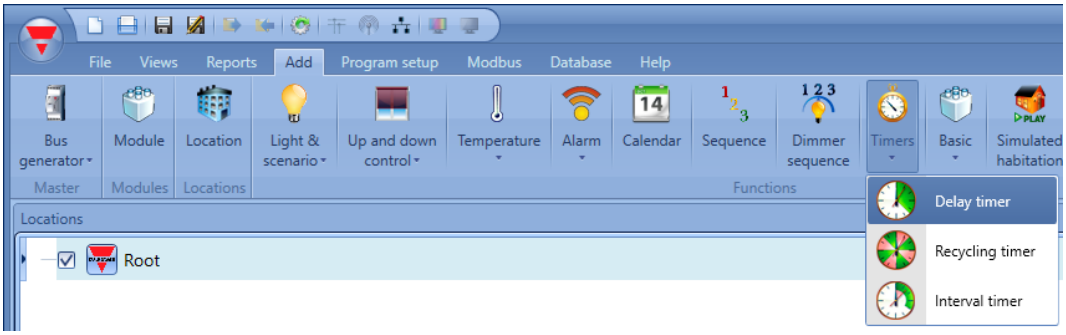
If the user wants to force the deactivation of the heating/cooling output regardless of all other signals used in the function, the *Force output off* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force output off*, then double click on the signal window (there are two available windows: one for heating and one for cooling) and select the appropriately signal to use (see picture below).



Each signal used in the *Force output off* window works as a level signal. Until the signal is active, the heating/cooling output is forced off.
When the *Force output on* signal is activated and at the same time the *Force output off* signal is activated, the *Force output on* signal has priority.

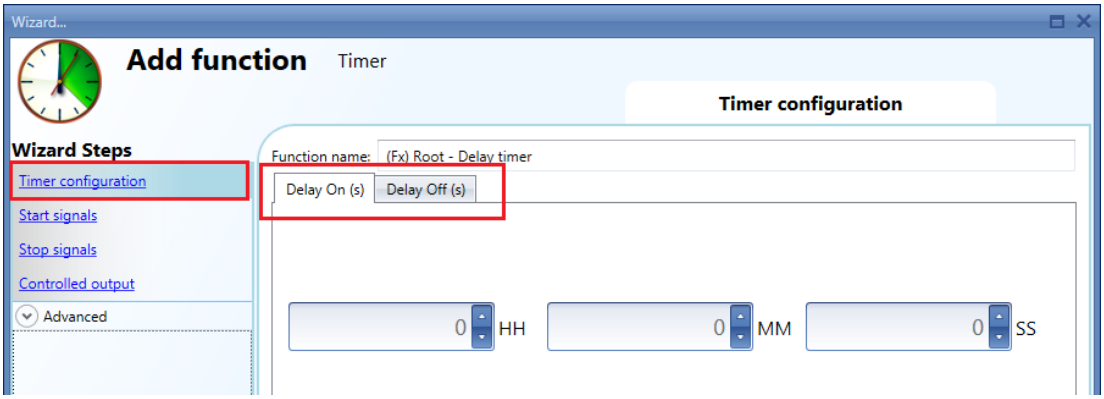
10.17 How to set a delay timer function

To set up a delay timer function the user has to select *Timer* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to a Timer function can also be opened by typing Alt+A+E.

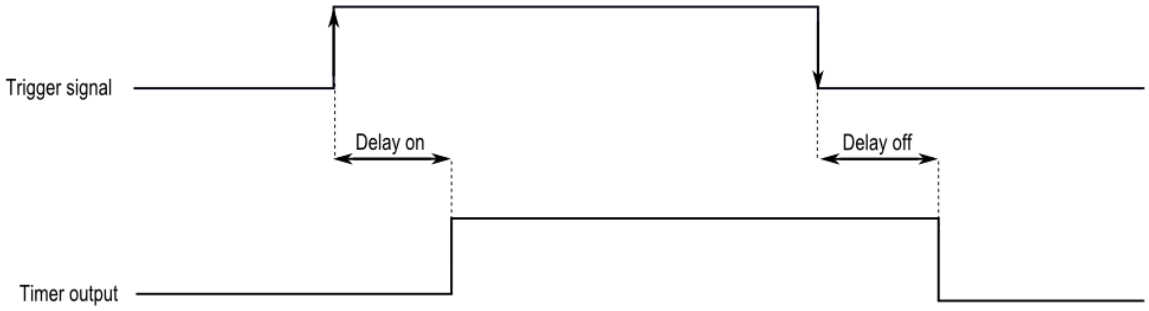


Using the delay timer, the output replicates the status of the input, applying a delay On and/or a delay Off time.
This timer function is driven by the level of the input signal: the start condition is the activation of the signal present in *Trigger signals*. The function will activate the output at the end of *Delay on*, and it will maintain this condition until the trigger signal is active.
When the input goes off, the function will deactivate the output after a *Delay off* time.

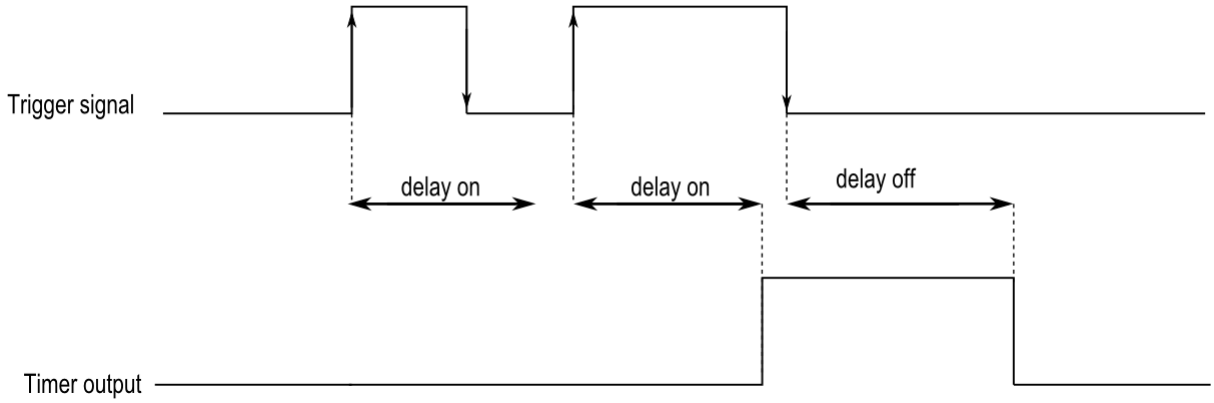
In the first step of the function, the user can manage the settings of the timer; in particular, the *delay on* and *off* can be set (see picture below).



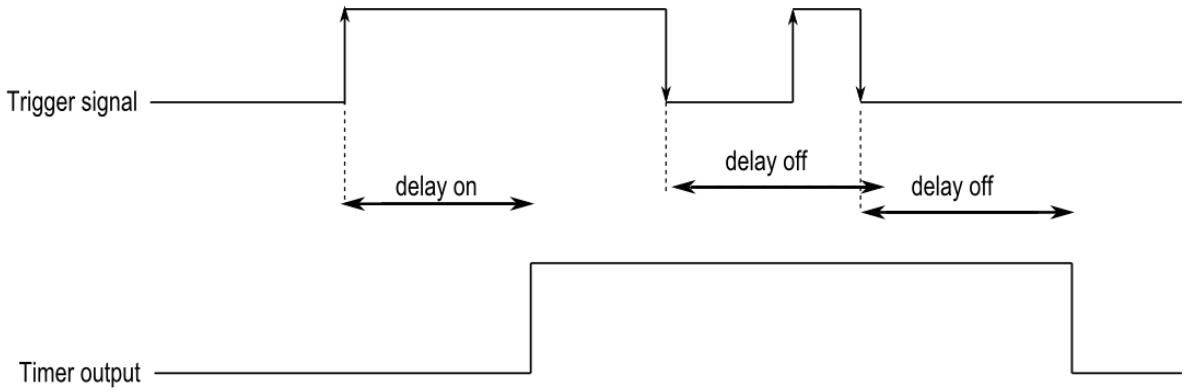
Example 1:
The output is turned on at the end of delay on and stays on until the trigger signal is active. When the trigger signal is deactivated, at the end of delay off the output is turned off.



Example 2:
If the trigger signal goes off before the end of the delay on time period, the delay on timer is reset and the function will not be activated.

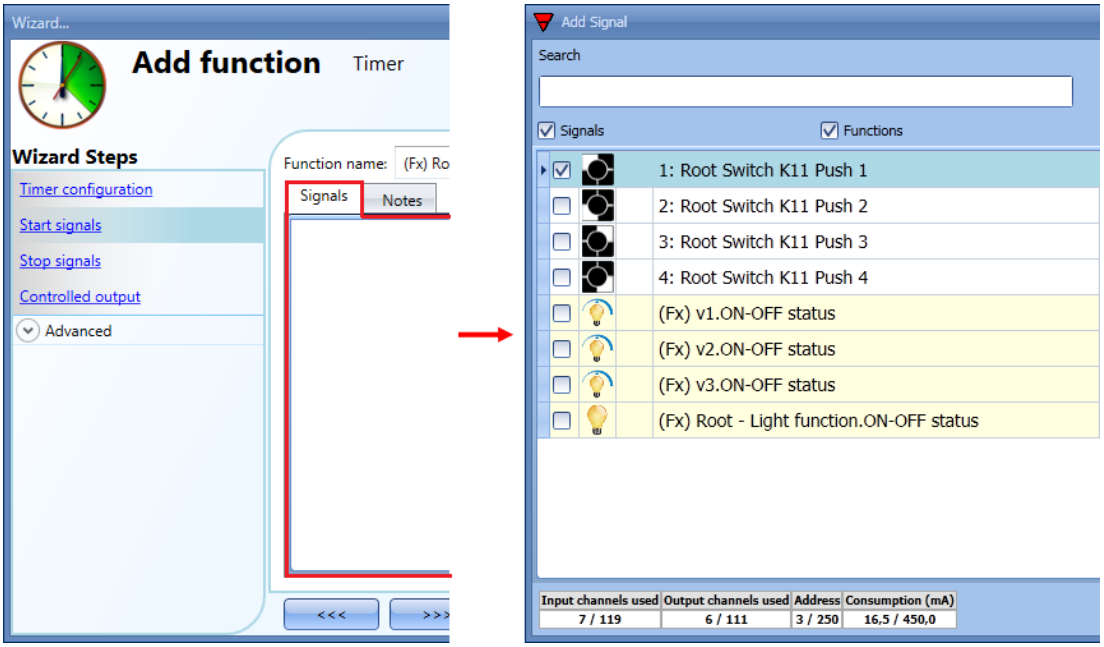


Example 3:
If the trigger signal goes on before the end of the delay off time period, the delay off timer is reset and the function will stay on.

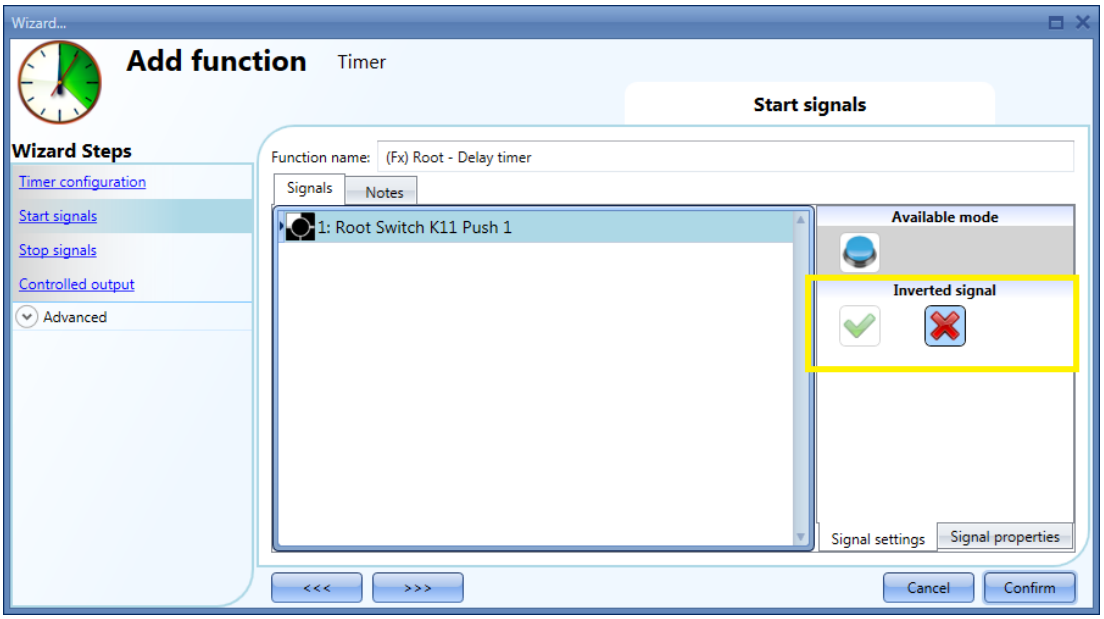


10.18 How to add the start signal

The timer needs to be started by a signal: this can be done directly by an input, or controlled by a function. The reaction of the start signal is based on its level and only one input signal can be added to the delay timer function.
To add the input signal of the timer, the user has to select *Start signals* from the wizard menu (see picture below).

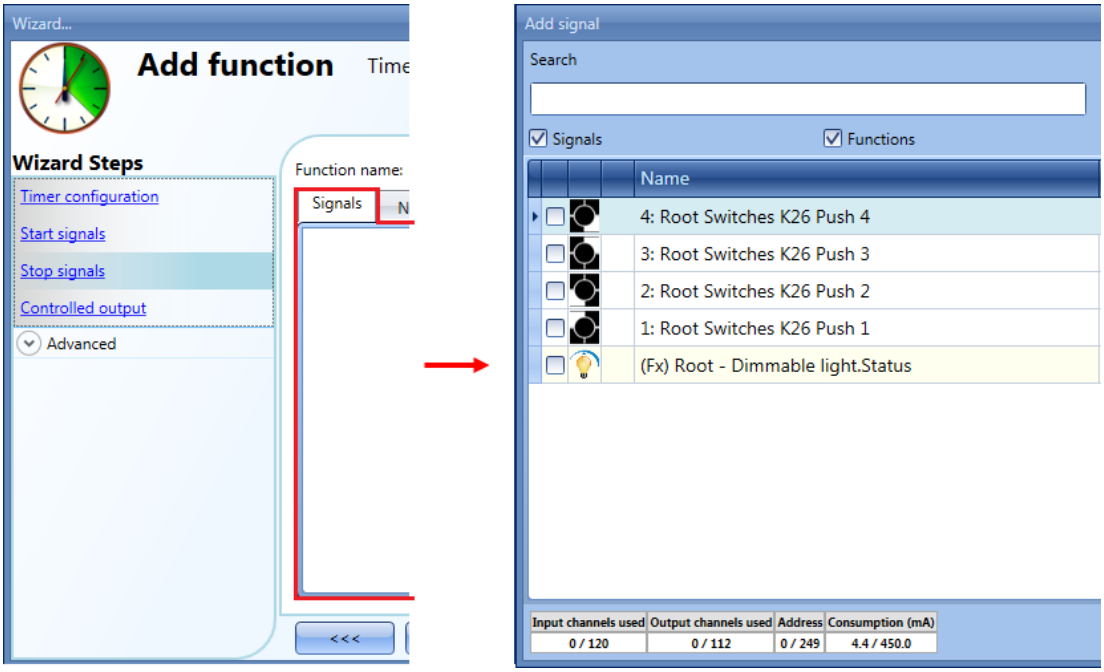


The inverted logic for the input signal can be enabled by selecting the green V in *Inverted signal* (see picture below).

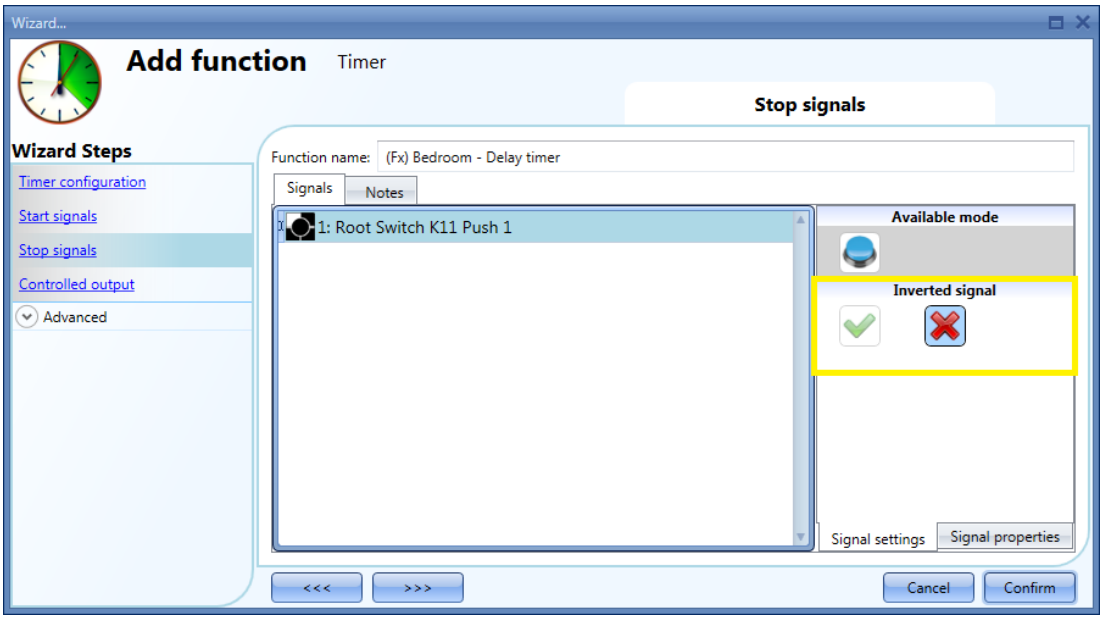


10.19 How to add the stop signal

If the timer needs to be stopped by a signal, a signal or a function can be used.
To add the stop signal of the timer, the user has to select *Stop signals* from the wizard menu (see picture below).
Only one signal can be added.
The timer is stopped when the signal is active.

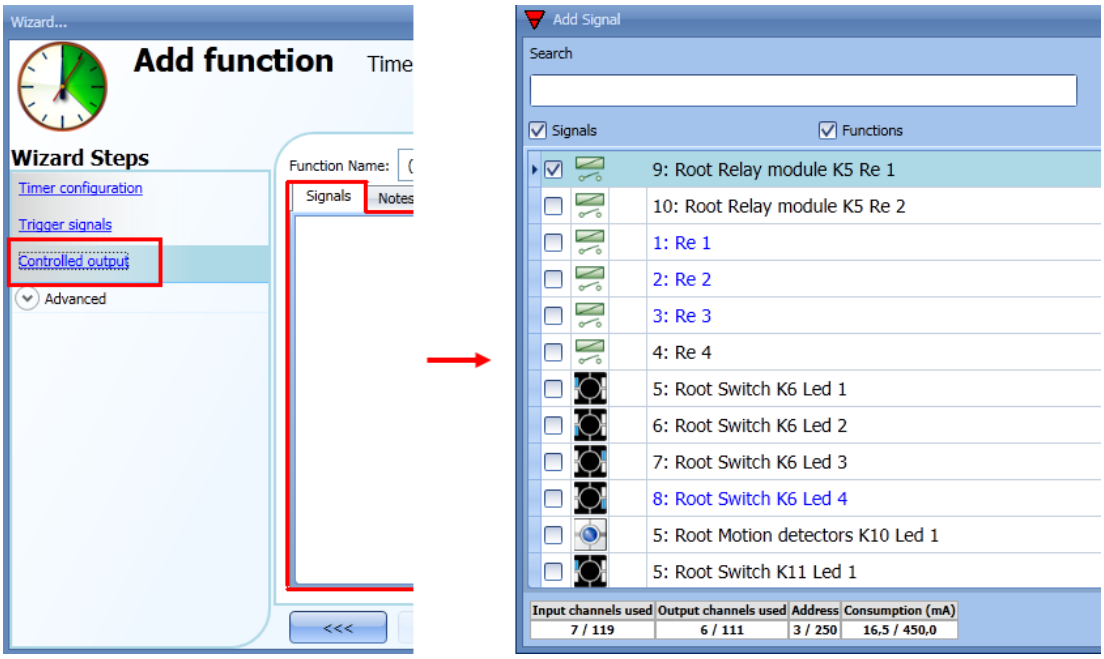


The inverted logic for the stop signal can be enabled by selecting the green V in *Inverted signal* (see picture below).

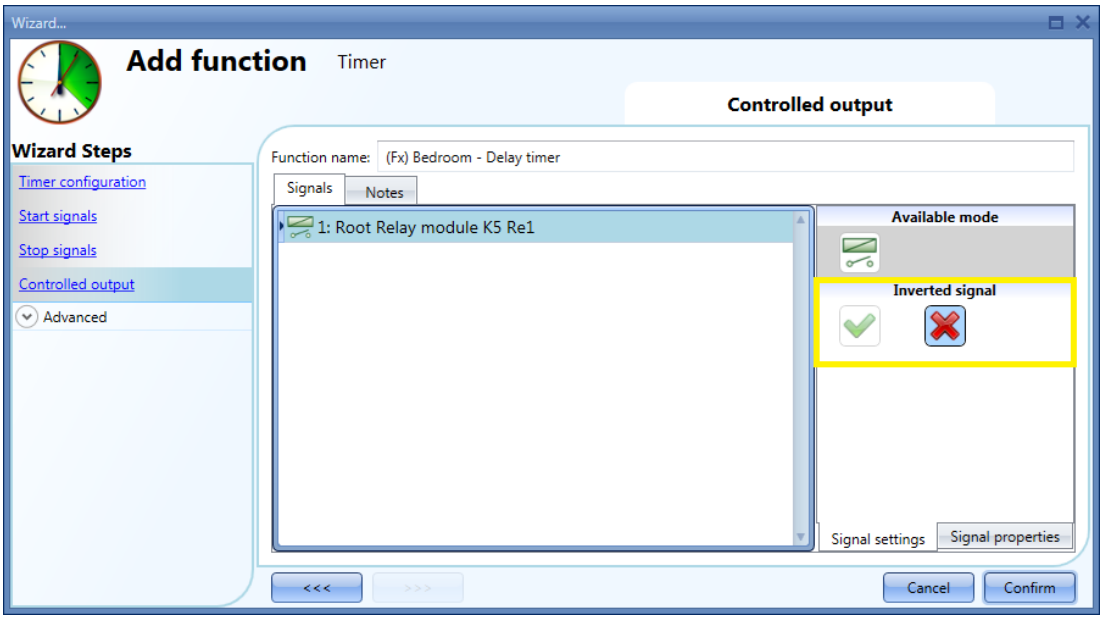


10.19.1 How to add the *output* signal

The timer can control an output directly by adding the signal to the *Controlled output* field (see picture below). If the user wants to use the function without any direct outputs, this field should be empty.

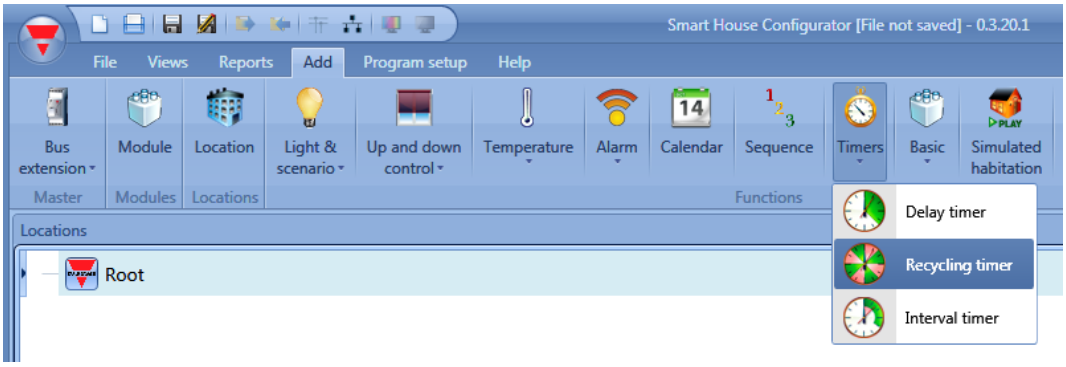


The inverted logic for the input signal can be enabled by selecting the green V in *Inverted signal* (see picture below).

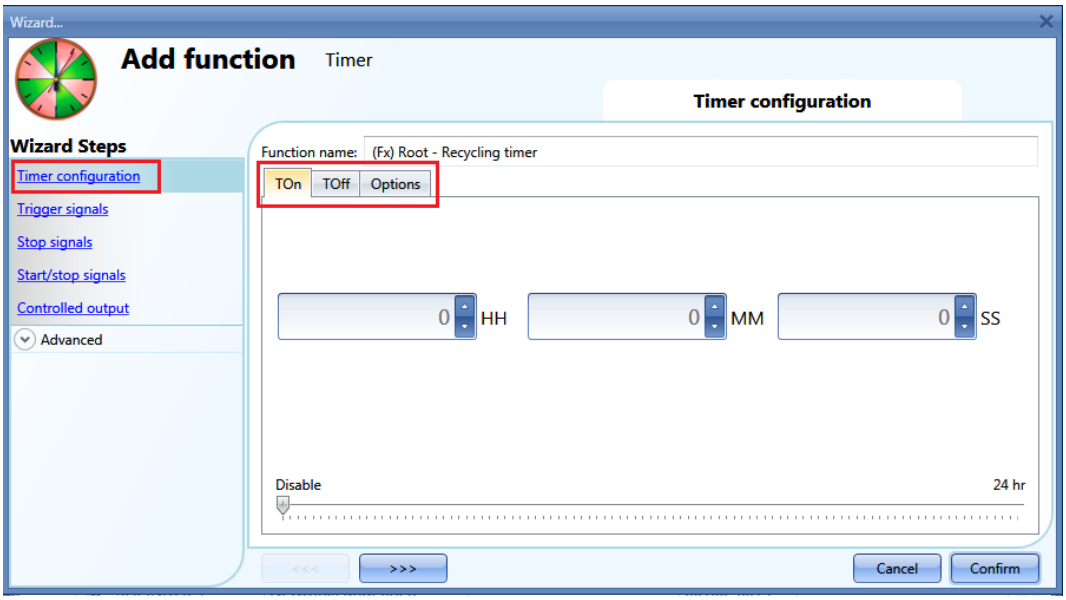


10.20 How to set a recycling timer function

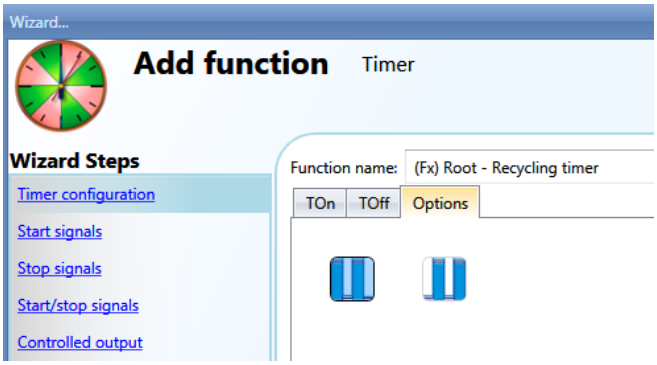
To set up a recycling timer function the user has to select *Timer* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to a Timer function can also be opened by typing Alt+A+E.



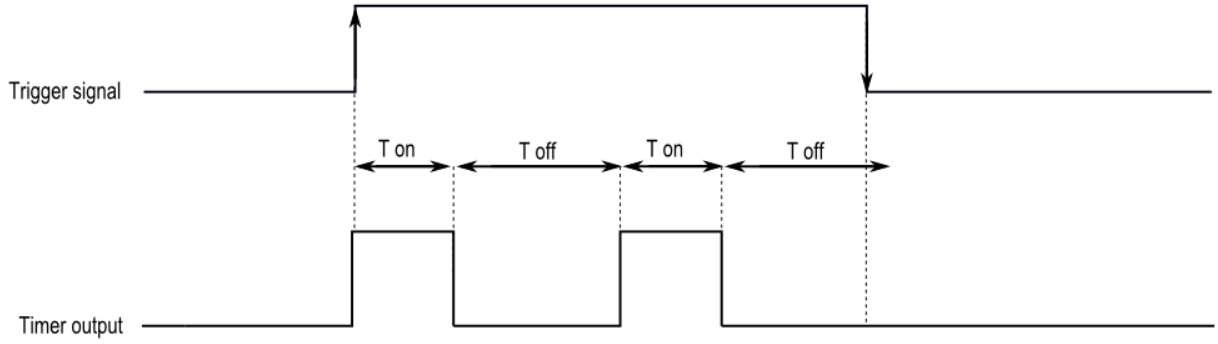
In the recycling timer function, until the trigger input is on, the output goes on and off with fixed timing. As soon as the trigger signal is activated, the output starts going on/off according to the Ton and Toff times; when the stop signal is activated, the output goes off.
To configure the recycling timer, the user has to select the relevant field when the timer function is added to the project (see picture below).



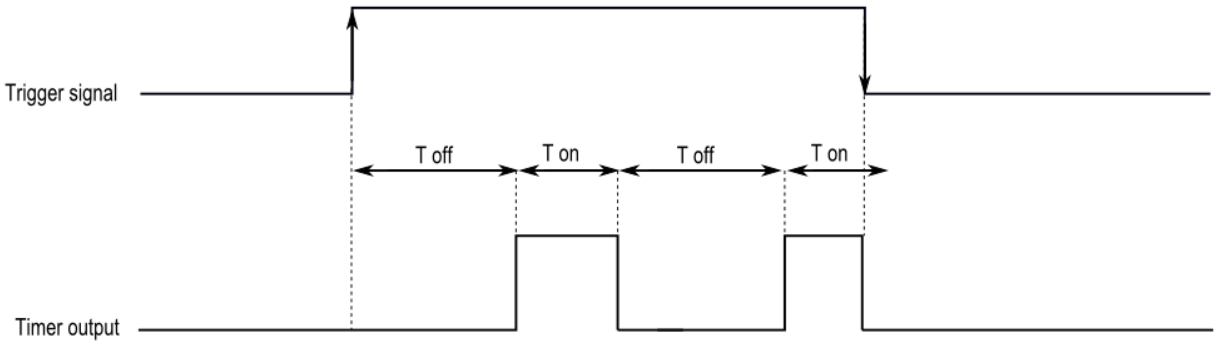
The Ton and Toff values can be entered in the relevant windows. In the *Options* field the user can set the first action for the output when the timer is activated (to start with the on or to start with the off condition).



Example 1:
In *Options* the recycling timer is selected to start with an on event: when the trigger signal is activated, the output starts with the on period.

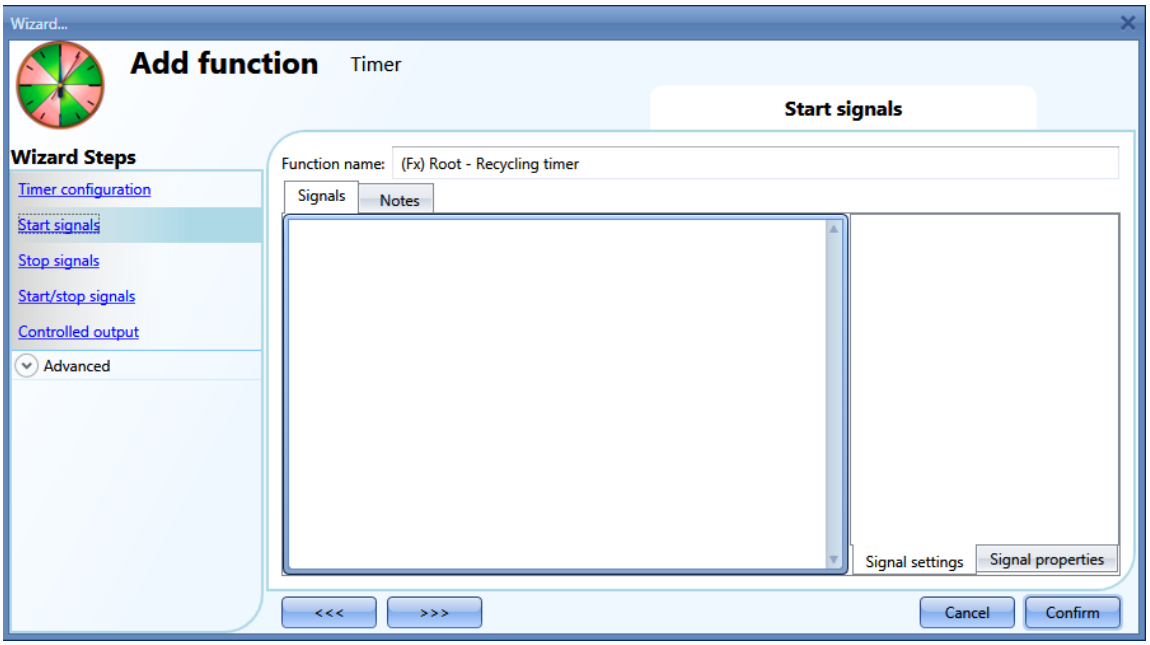


Example 2:
In *Options* the recycling timer is selected to start with an off event: when the trigger signal is activated, the output starts with the off period. As soon as the trigger signal is deactivated, the output goes off.

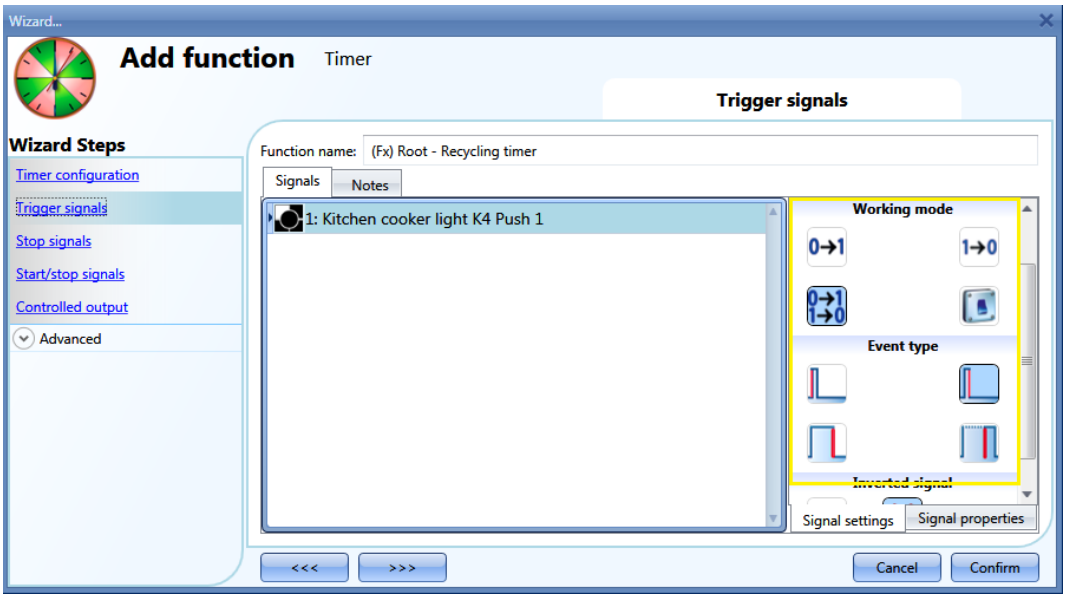


10.20.1 How to add the start signal

The timer needs to be started by a signal: this can be done directly by an input, or by means of a function. The reaction to this signal is programmable according to what is written below.
To add the trigger signal of the timer the user has to select *Start signals* from the wizard menu (see picture below).



When the trigger signal is entered, the user can set the working mode, the event type and the logic for it (shown in the picture below).



The activation of the output is according to the working mode of the trigger signal.
For example, if the trigger signal is set as “Switch on event”, the timer starts as soon as the trigger signal is activated (rising edge); if the trigger signal is set as “Switch off event”, the timer starts at the falling edge of the input.

If a push button type is selected, the way of working (yellow rectangle), has to be selected following the table below.

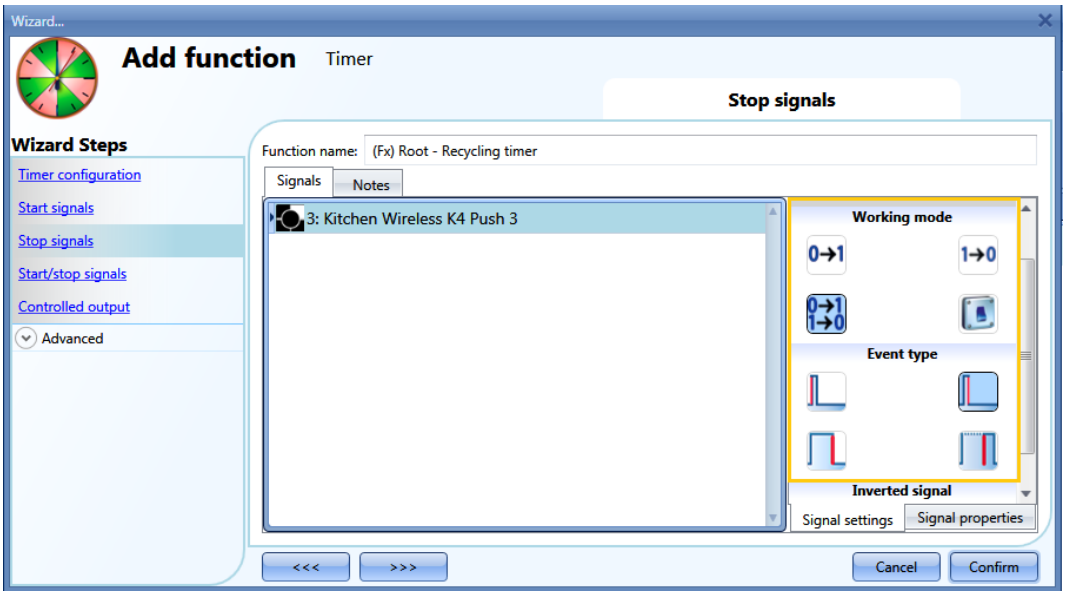
Working mode	Event type			
0→1	As soon as the push button is pressed, the timer is started on the rising edge.	*With this combination the system does nothing.*	After a long activation, the timer is started on the rising edge.	After a very long activation, the timer is started on the rising edge.
1→0	*With this combination the system does nothing.*	The interval timer is started on the falling edge.	After a long activation, the timer is started on the falling edge.	After a very long activation, the timer is started on the falling edge.
0→1 1→0	As soon as the push button is activated, the timer is activated on the rising and falling edge.	After a short activation (less than 1 second), the timer is activated on the rising and falling edge.	After a long activation, the timer is activated on the rising and falling edge.	After a very long activation, the timer is activated on the rising and falling edge.
	The timer will be started when the push button is pressed and toggled again when the push button is released.			

If a switch signal is used, the set of the working mode has to be carried out following the table below:

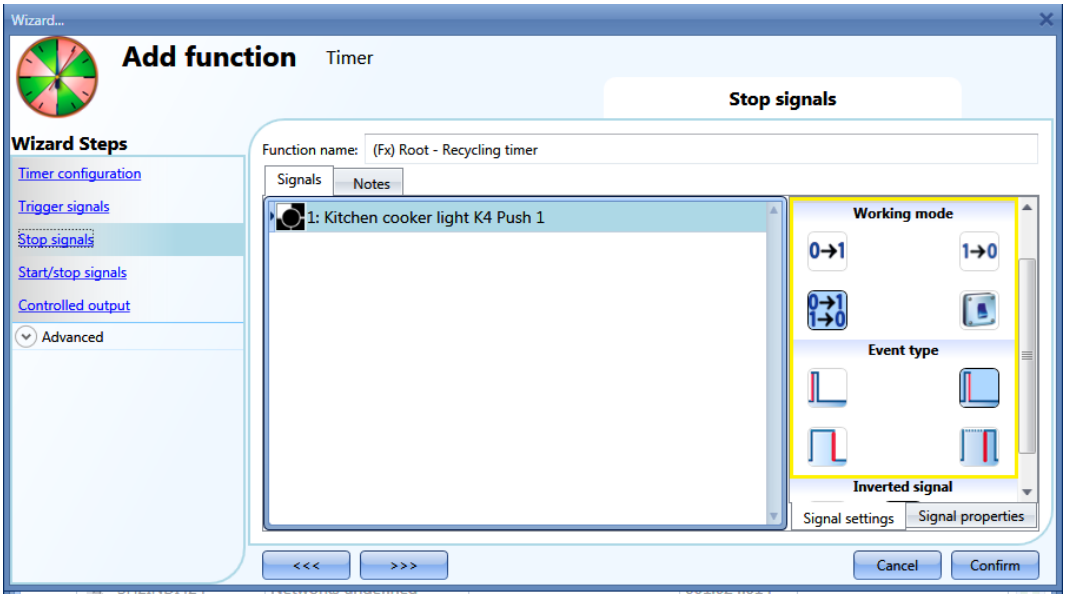
Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The timer is started	No action
1→0	No action	The timer is started
0→1 1→0	The timer is started	The timer is started
	The timer is started	The timer is started

10.20.2 How to add the stop signal

The timer needs to be stopped by a signal: this can be done directly by an input, or by means of a function. The reaction to this signal can be programed.
To add the stop signal of the timer, the user has to select *Stop signals* from the wizard menu (see picture below).



When the trigger signal is entered, the user can set the working mode, the event type and the logic for the trigger signal (see the picture below).



In the table below, all the possible activations of the stop signal are shown.

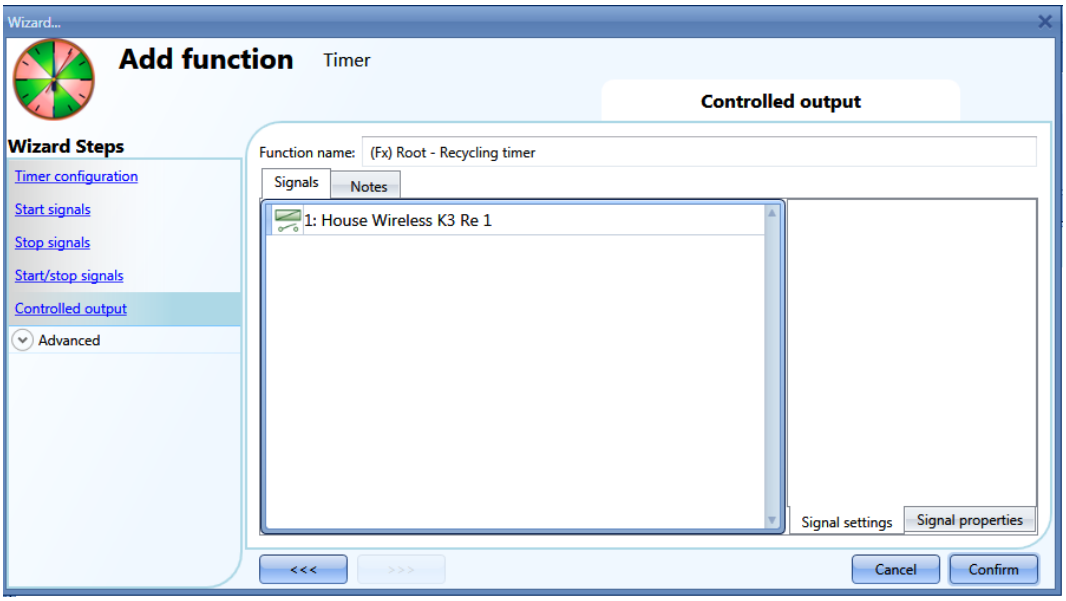
Working mode	Event type			
0→1	As soon as the push button is pressed, the timer is stopped on the rising edge.	*With this combination the system does nothing.*	After a long activation, the timer is stopped on the rising edge.	After a very long activation, the timer is stopped on the rising edge.
1→0	*With this combination the system does nothing.*	The timer is stopped on the falling edge	After a long activation, the timer is stopped on the falling edge	After a very long activation, the timer is stopped on the falling edge
0→1 1→0	As soon as the push button is activated, the timer is stopped on the rising and falling edge.	After a short activation (less than 1 second), the timer is stopped on the rising and falling edge.	After a long activation, the timer is stopped on the rising and falling edge.	After a very long activation, the timer is stopped on the rising and falling edge.
	The timer is stopped when the push button is pressed and toggled again when the push button is released.			

If a switch signal is used, the setting of the working mode has to be done following the table below:

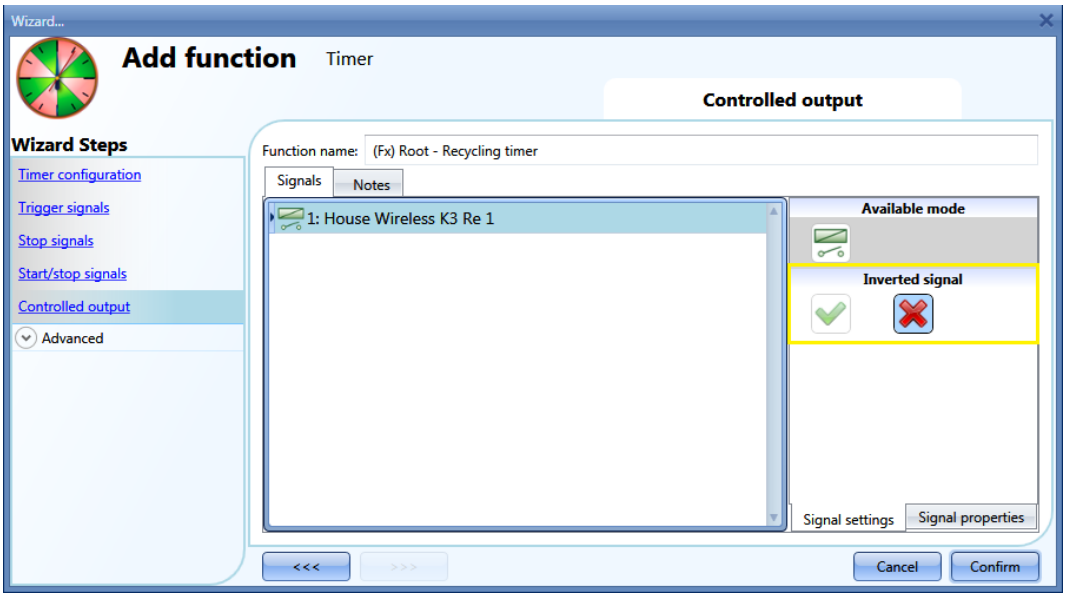
Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The timer is stopped	No action
1→0	No action	The timer is stopped
0→1 1→0	The timer is stopped	The timer is stopped
	The timer is stopped	The timer is stopped

10.20.3 How to add the *output* signal

The timer can control an output directly by adding the signal to the *Controlled output* field (see picture below). If the user wants to use the function without any direct outputs, this field must be empty.

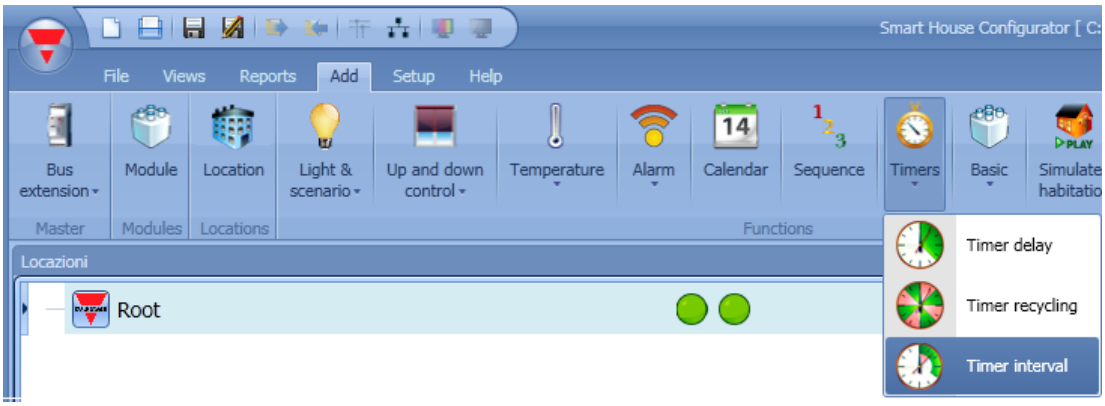


The inverted logic for the input signal can be enabled by selecting the green V in *Inverted signal* (see picture below).



10.21 How to set an interval timer function

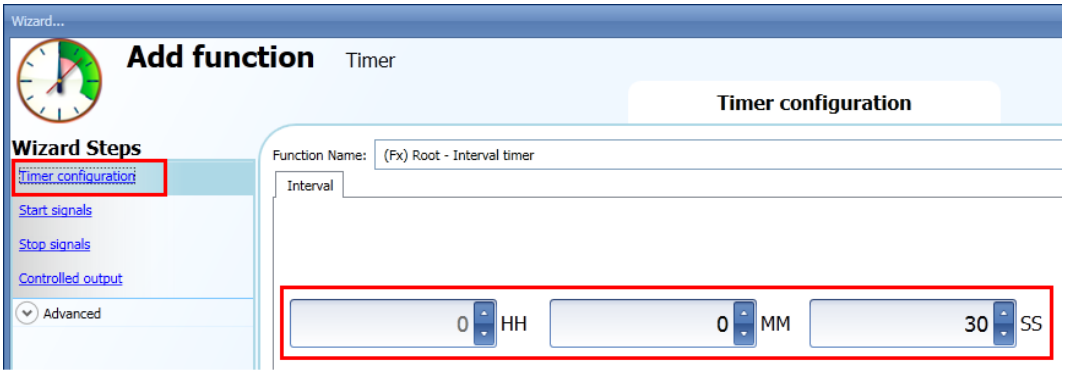
The timer function can be used to control an output where an automated temporization is required.
To set up a timer function, the user should select *Timer* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to a Timer function can also be opened by typing Alt+A+E.



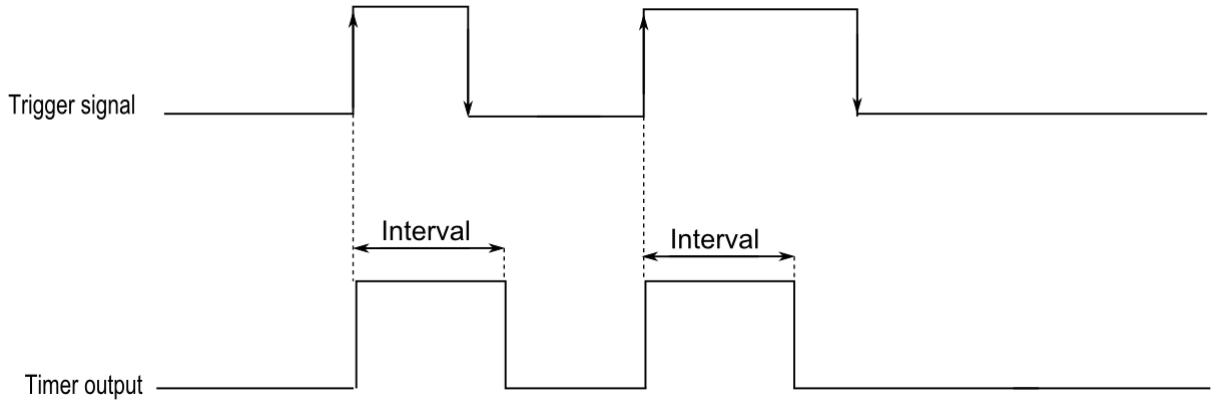
Using the interval timer, the output is activated by the trigger signal for a configurable interval time.
The output of the timer is activated when one of the following events happens:

- A rising edge of the trigger signal (eOn)
- A falling edge of the trigger signal (eOff)
- On each change of the trigger signal (each change)

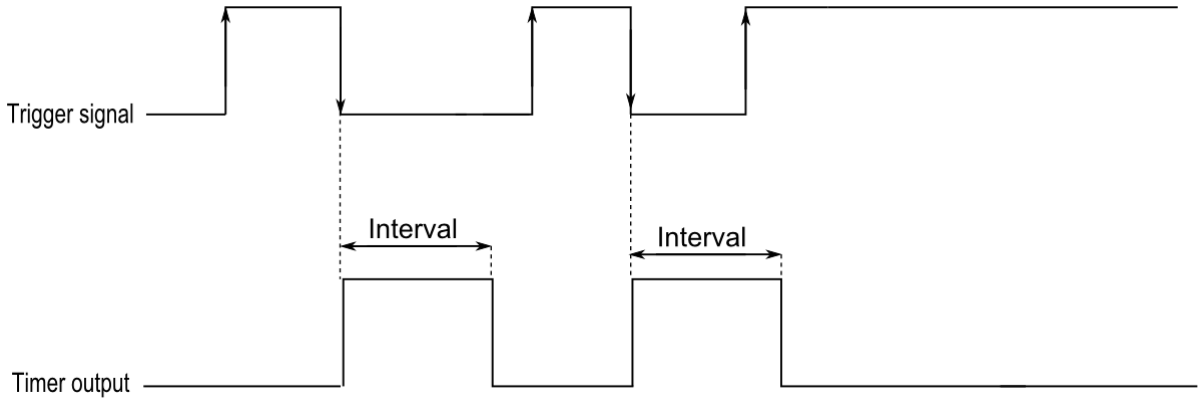
In the first step of the wizard the user can set the interval time for the output.



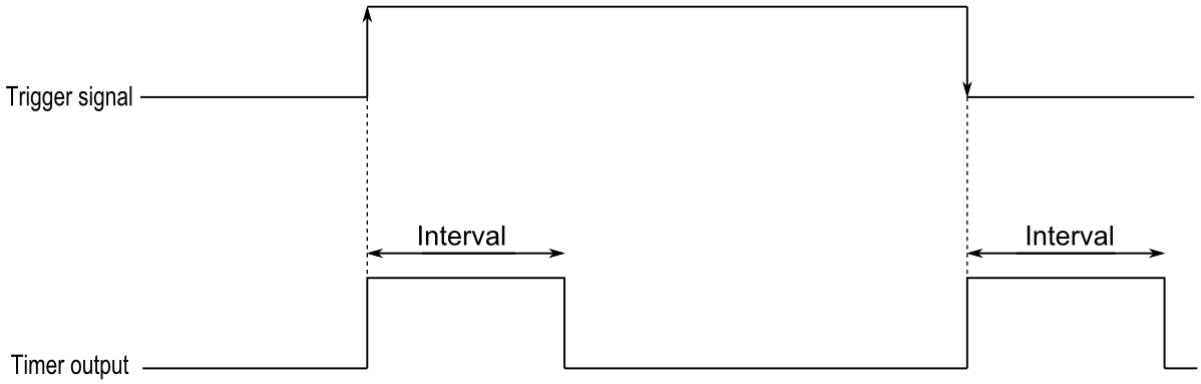
Example 1:
The timer is started on the rising edge of the input signal.



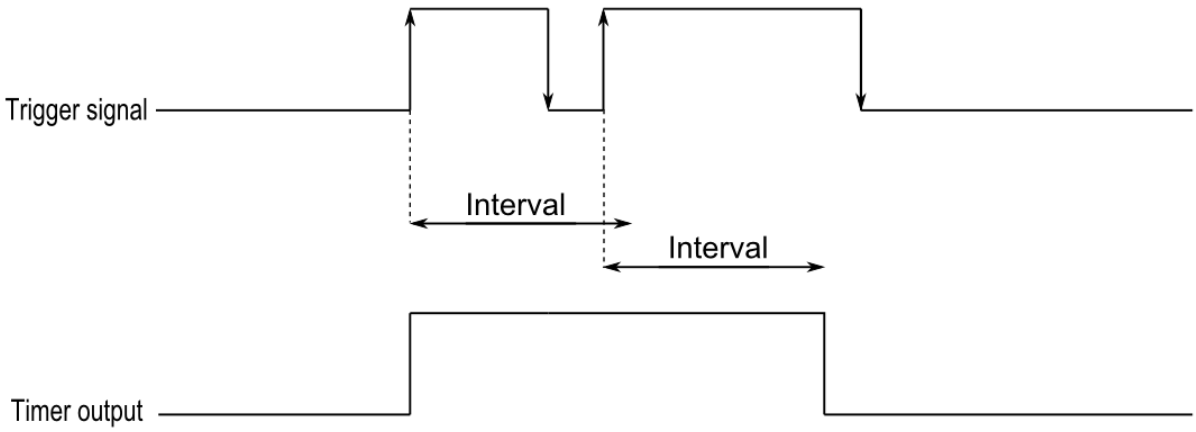
Example 2:
The timer is started on the falling edge of the input signal.



Example 3:
The timer is started on each change of the input signal.

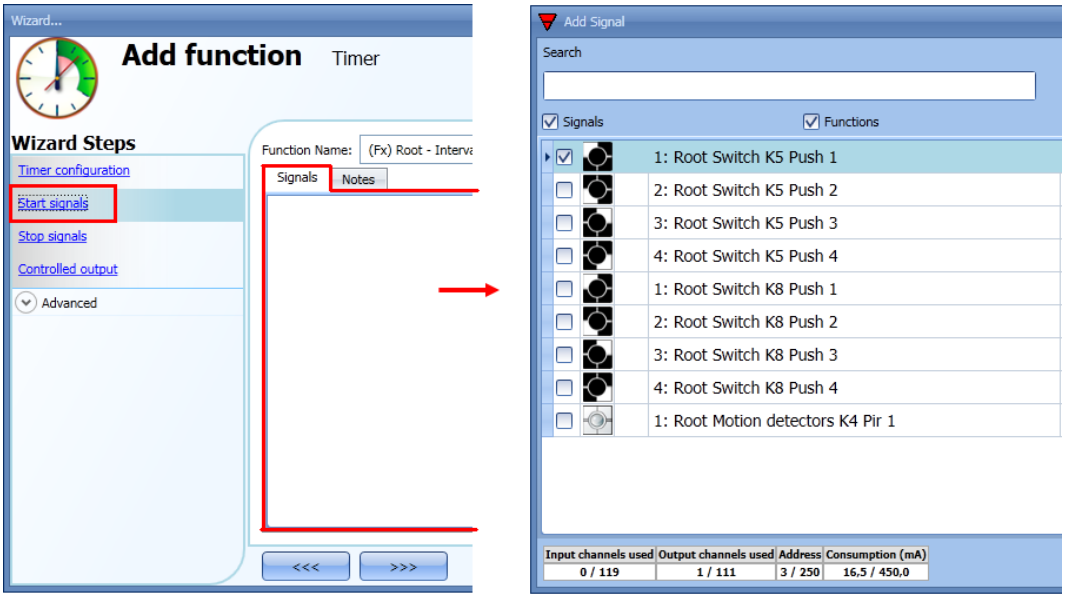


Example 4:
If a trigger event is received before the timer has expired, the timer is reloaded.

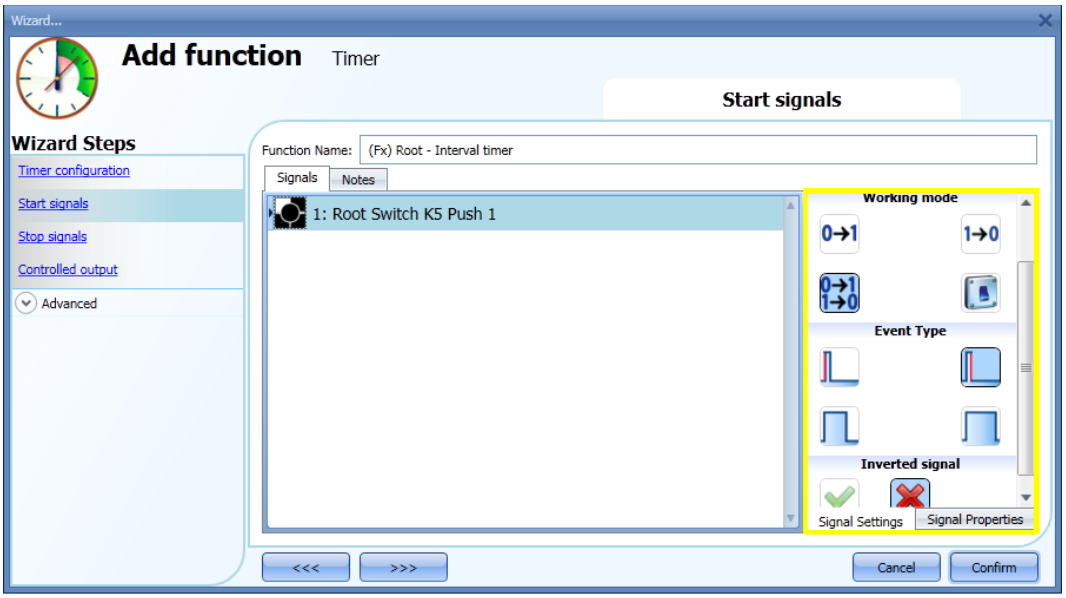


10.21.1 How to add the start signal

The timer needs to be started by a signal: this can be done directly by an input, or by means of a function. The reaction to this signal is event based.
To add the trigger signal, the user should select *Trigger signals* from the wizard menu (see picture below).



When the trigger signal is entered, the user can set the working mode, the event type and the logic for the trigger signal (shown in the picture below).



The activation of the output is according to the working mode of the trigger signal.
For example, if the trigger signal is set as “Switch on event”, the timer starts as soon as the trigger signal is activated (rising edge); if the trigger signal is set as “Switch off event”, the timer starts at the falling edge of the input.

If a push button type is selected, the way of working has to be selected following the table below.

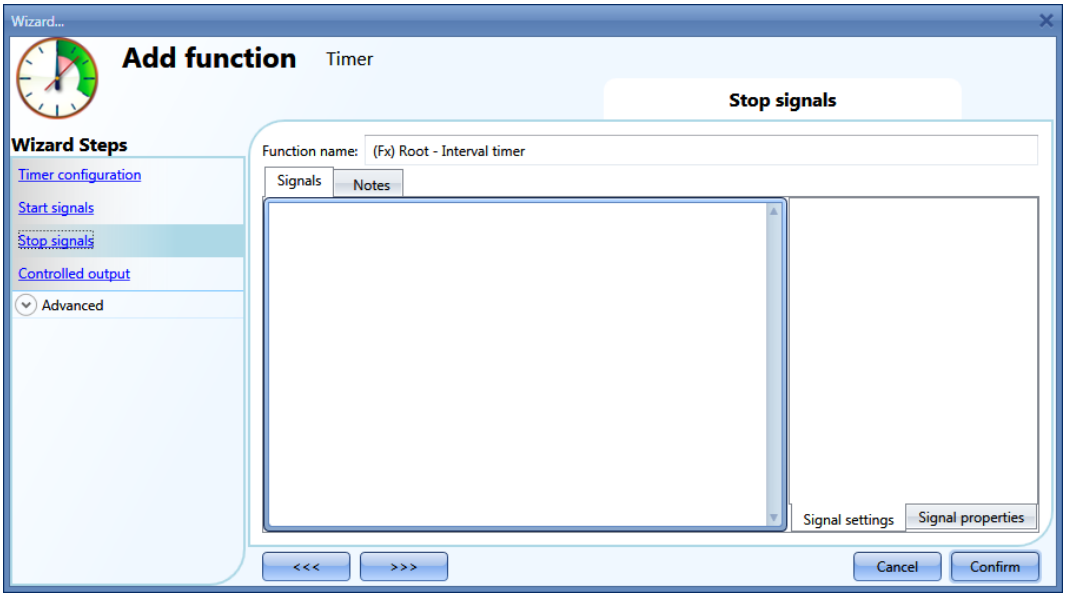
Working mode	Event type			
0→1	As soon as the push button is pressed, the interval timer is started on the rising edge.	*With this combination the system does nothing.*	After a long activation, the interval timer is started on the rising edge.	After a very long activation, the interval timer is started on the rising edge.
1→0	*With this combination the system does nothing.*	The interval timer is started on the falling edge.	After a long activation, the interval timer is started on the falling edge	After a very long activation, the interval timer is started on the falling edge
0→1 1→0	As soon as the push button is activated, the interval timer is activated on the rising and falling edge.	After a short activation (less than 1 second), the interval timer is activated on the rising and falling edge.	After a long activation, the interval timer is activated on the rising and falling edge.	After a very long activation, the interval timer is activated on the rising and falling edge.
	The interval timer will be started when the push button is pressed and toggled again when the push button is released.			

If a switch signal is used, the setting of the working mode has to be done following the table below:

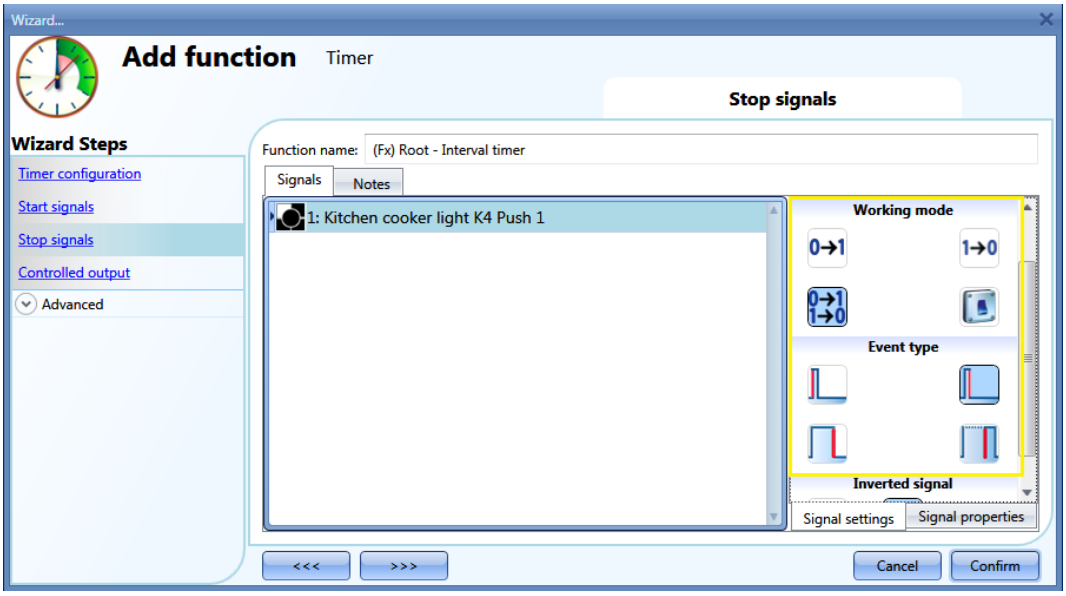
Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The timer is started	No action
1→0	No action	The timer is started
0→1 1→0	The timer is started	The timer is started
	The timer is started	The timer is started

10.21.2 How to add the stop signal

The timer needs to be stopped by a signal: this can be done directly by an input, or by means of a function. The reaction to this signal is event based.
To add the stop signal, the user has to select *Stop signals* from the wizard menu (see picture below).



The stop signal works only if the timer *(the delay off)* is in progress.
When the trigger signal is entered, the user can set the working mode, the event type and the logic for the trigger signal (shown in the picture below).



In the table below, all the possible activations of the stop signal are shown.

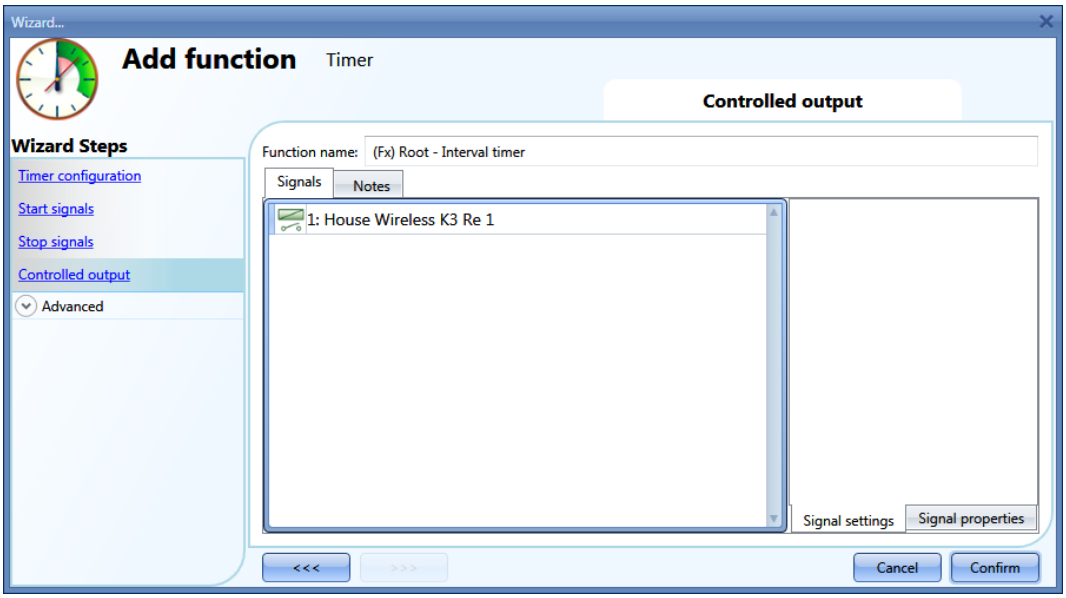
Working mode	Event type			
0→1	As soon as the push button is pressed, the interval timer is stopped on the rising edge.	*With this combination the system does nothing.*	After a long activation, the interval timer is stopped on the rising edge.	After a very long activation, the interval timer is stopped on the rising edge.
1→0	*With this combination the system does nothing.*	The interval timer is stopped on the falling edge	After a long activation, the interval timer is stopped on the falling edge	After a very long activation, the interval timer is stopped on the falling edge
0→1 1→0	As soon as the push button is activated, the interval timer is stopped on the rising and falling edge.	After a short activation (less than 1 second), the interval timer is stopped on the rising and falling edge.	After a long activation, the interval timer is stopped on the rising and falling edge.	After a very long activation, the interval timer is stopped on the rising and falling edge.
	The interval timer is stopped when the push button is pressed and toggled again when the push button is released.			

If a switch signal is used, the setting of the working mode has to be done following the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The timer is stopped	No action
1→0	No action	The timer is stopped
0→1 1→0	The timer is stopped	The timer is stopped
	The timer is stopped	The timer is stopped

10.21.3 How to add the output signal

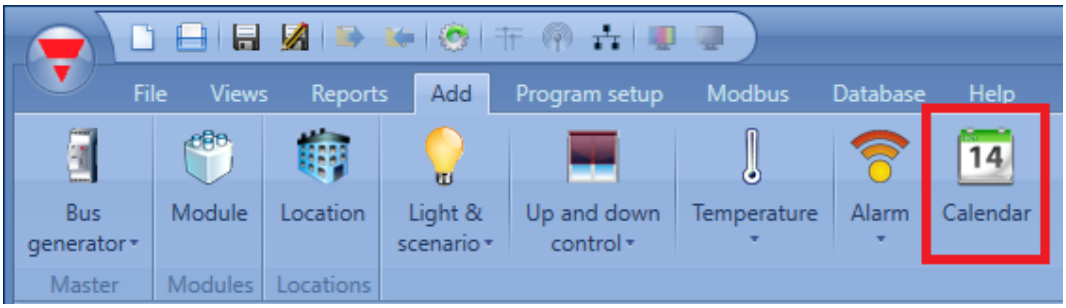
The timer can control an output directly by adding the signal to the *Controlled output* step (see picture below). If the user wants to use the function without any direct outputs, this field must be empty.



The inverted logic for the input signal can be enabled by selecting the green V in *Inverted signal.*

10.22 How to set a global calendar function

The calendar function can be used to automatically activate other functions during the year creating special automations.
To set up a calendar function the user has to select *Calendar* from the Add menu (see picture below).
The new function will be added into the selected location.
The wizard relevant to a Calendar function can also be opened by typing Alt+A+C.

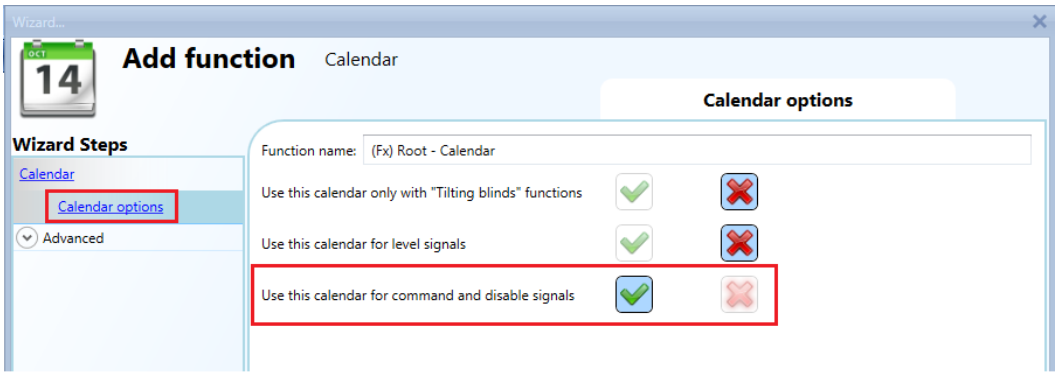


The calendar function can be used to perform different actions: for example, an external calendar could be created in order to switch off all the lights in an office at the end of the day, or move down all blinds at a certain hour in the evening.
The calendar can also be used to disable the automation of a function: for example, the switching on/off of the light with the PIR sensor can be disabled during the day.

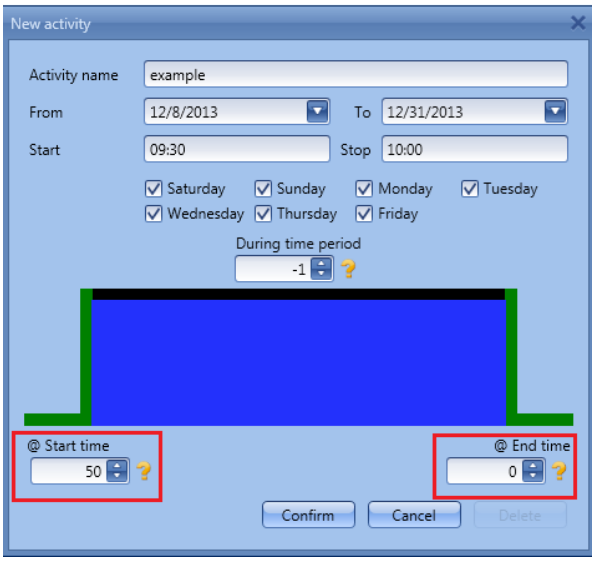
10.22.1 How to use a global calendar in different functions

As described in the previous paragraph, the calendar activities have different meanings according to the function where the calendar is used.

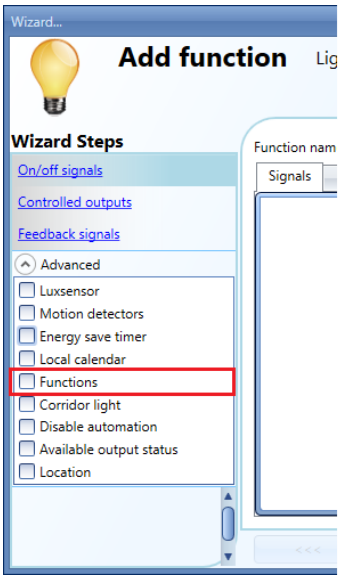
A simple example is where the user uses the calendar to control two functions: an on/off light and a dimmable light.
In the *Calendar options*, the user should select the green V *Use this calendar for command and disable signals.*

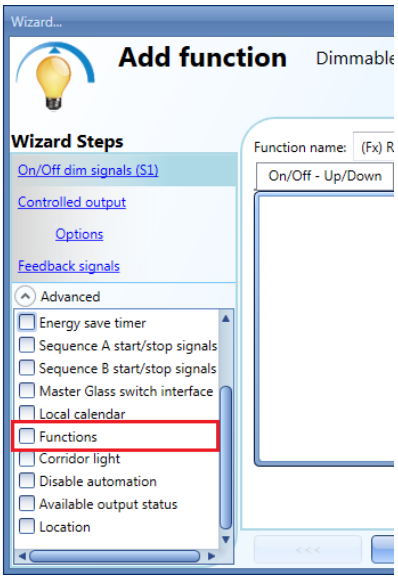


The calendar settings are the ones shown in the picture below. The event at start time is 50 and the event at stop time is 0.



Once the external calendar function is created, the calendar must be linked to the other functions by using the advanced field *Functions*.

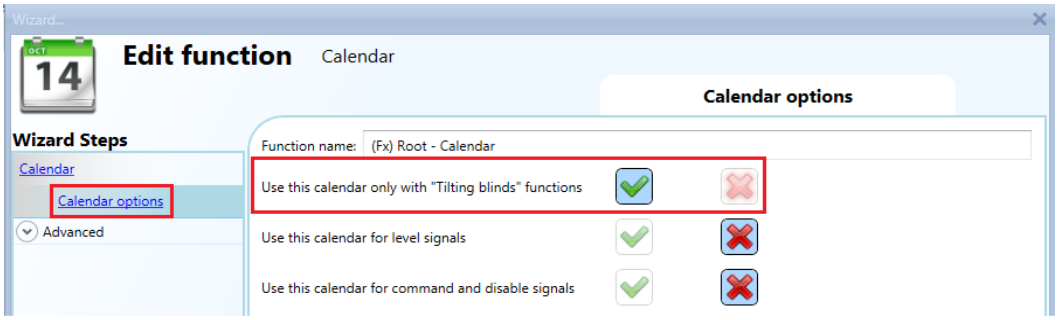


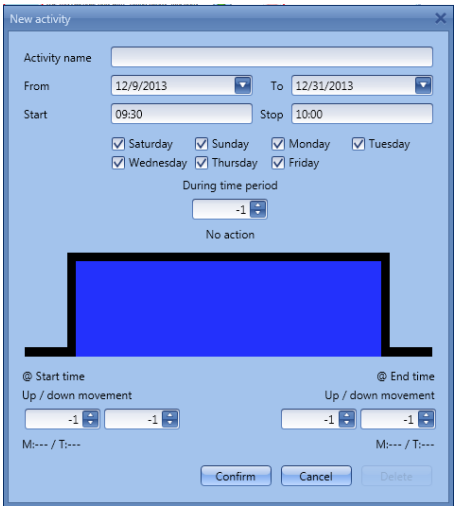


At the start time, the dimmable light is switched on at 50%, the light on/off is switched on. At the end time, both the on/off light and dimmer are switched off.

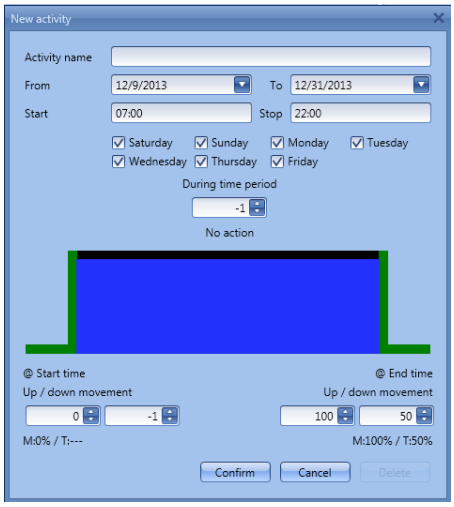
10.22.2 How to use a global calendar in roller blind functions

When the user wants to control the automation of *Tilting blinds* functions, a special global calendar has to be created.
In the *Calendar options*, the user should select the green V *Only for blinds with tilting.*





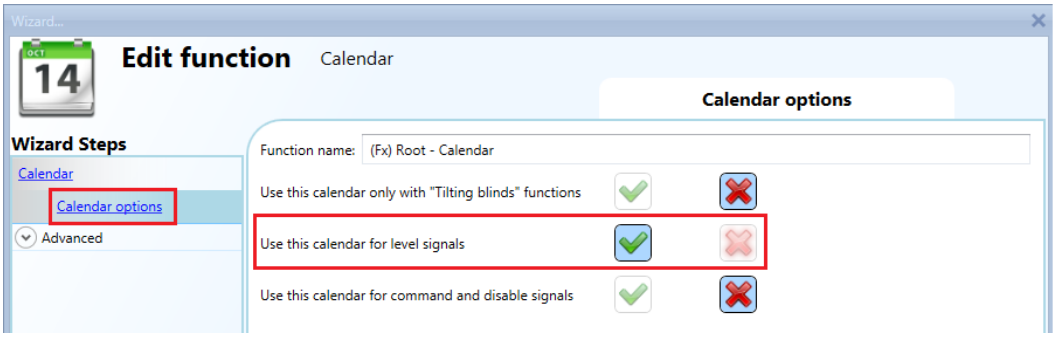
When the green V is selected, the activity menu of the calendar is changed, showing two actions at start time and at end time: the action for the up/down movement and the action for the tilting movement.



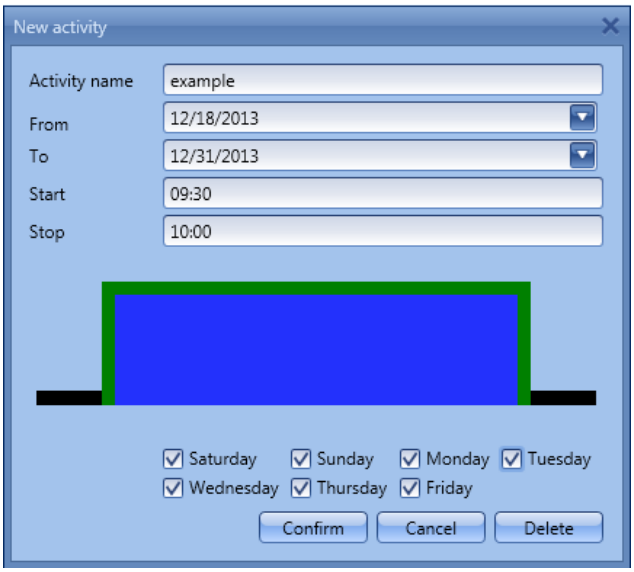
All the functions where the calendar is added will go up/down and tilt their slats according to the calendar settings.
In the next picture an external function used in all the *Tilting blind* functions is shown. At 7:00 in the morning, the blinds go up and at 22:00 in the evening they all go down and are half tilted.

10.22.3 How to use a global calendar in Multigate functions or as a level input signal in any type of function

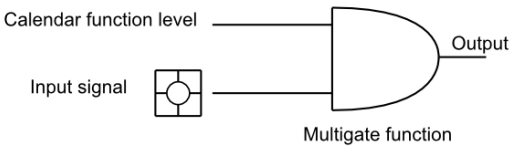
When the user wants to control the input signal in a Multigate function or use the calendar as a level input signal, a global calendar *Use this calendar for level signals* has to be created.
In the *Calendar options*, the user should select the green V *Use this calendar for Multigate function.*
By selecting this option, the user can use the calendar as an input signal in any type of function according to the calendar settings.

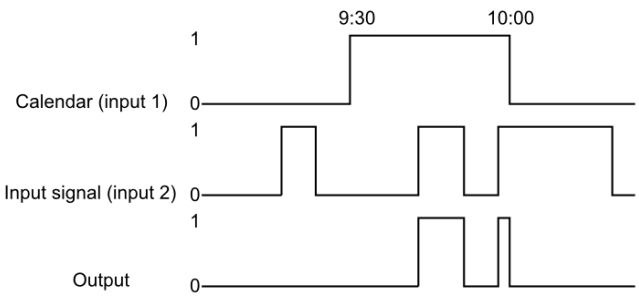


When the green V is selected, the activity menu of the calendar is changed, showing the level action during the time.
The user must enter the time interval during which the calendar is active. The green border indicates that the calendar is active during the entire selected period.



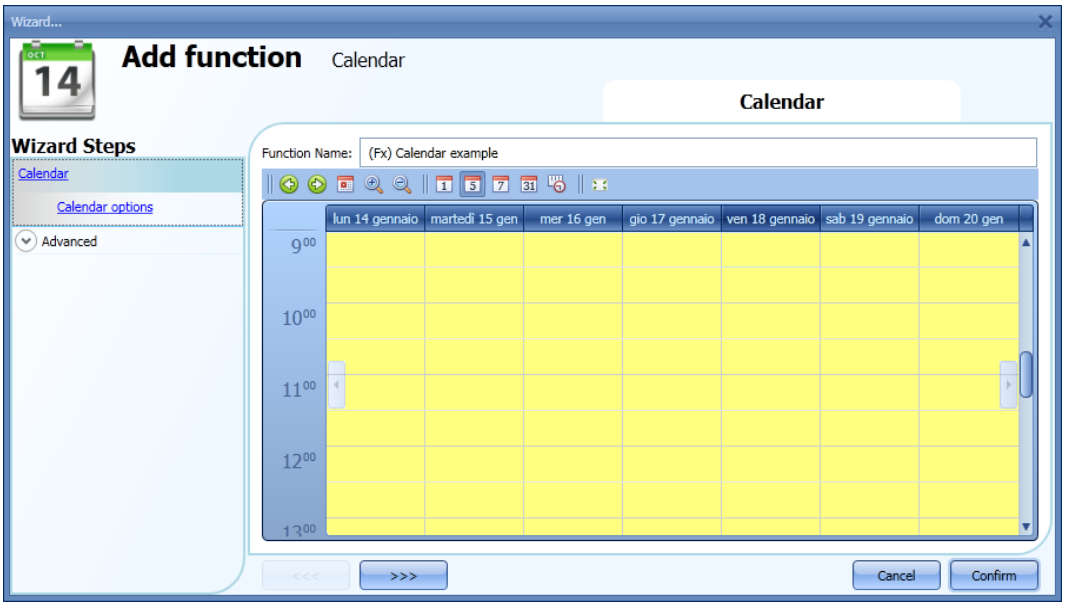
In the next picture a global calendar function is used to enable the input in a multigate function: the output signal is activated when the enable input is on and the pushbutton is activated. At 9:30 in the morning, the calendar signal goes On (multigate input 1 enabled) at 10:00 in the morning the calendar signal goes off (multigate input 1 disabled) and the output is switched off.



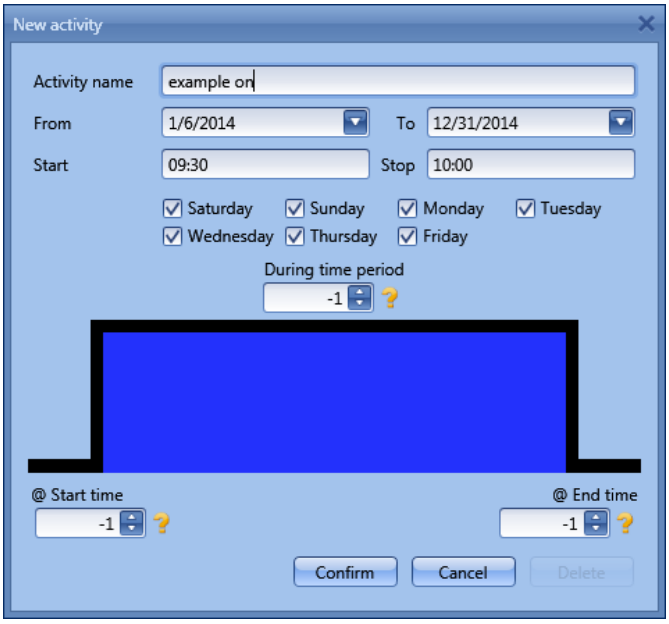


10.22.4 Add a new activity to the calendar

Once the calendar function is added, to configure a new activity the user can double click anywhere in the calendar (yellow area).



It is not necessary to go to a specific day of the calendar; the user can change the day in the activity window.



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From: the start date for the calendar activity.

To: the end date of the calendar activity.

Start: The time of the start of the activity.

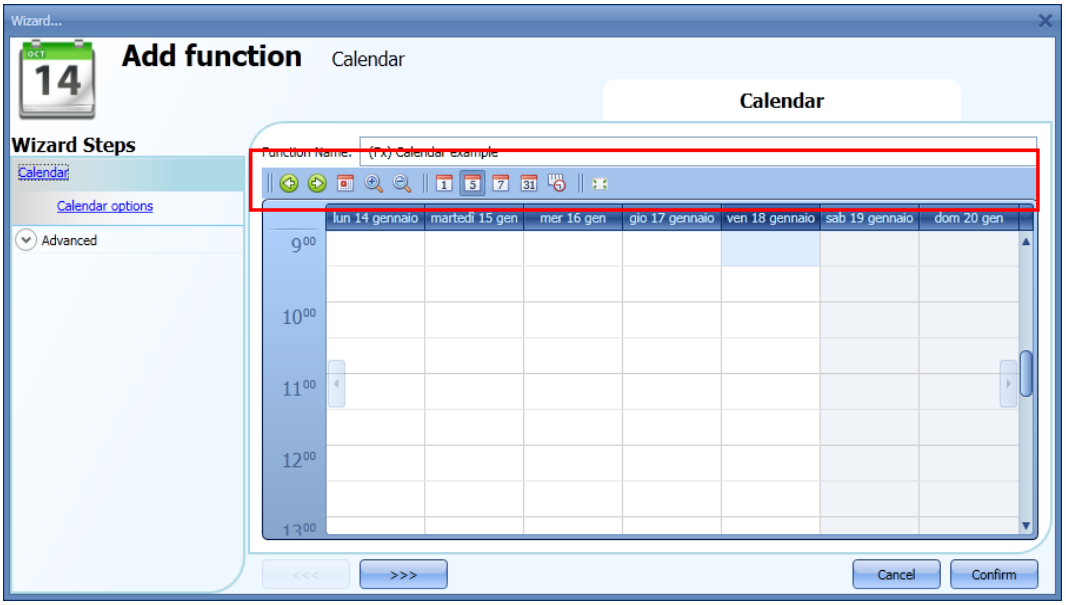
Stop: The time of the end of the activity.

@ start time: in this field the user can select the action of the function at the start time.
The action is identified by a number from -1 to 100. According to the function which the calendar is linked to, the number identifies a specific action (see paragraph *Calendar actions*).
The rising edge is marked in green indicating that an action is selected at the start time.

@ end time: in this field the user can select the action of the function at the end time.
The action is identified by a number from -1 to 100. According to the function which the calendar is linked to, the number identifies a specific action (see paragraph *Calendar actions*).
The falling edge is marked in green indicating that an action is selected at the end time.

Days: The user should select the days when the calendar activities must be applied.

At the top of the wizard of the calendar function, the user can access the bar to change the visualization of the calendar:



Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day.
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally.
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

10.22.5 Meaning of the actions in the different functions

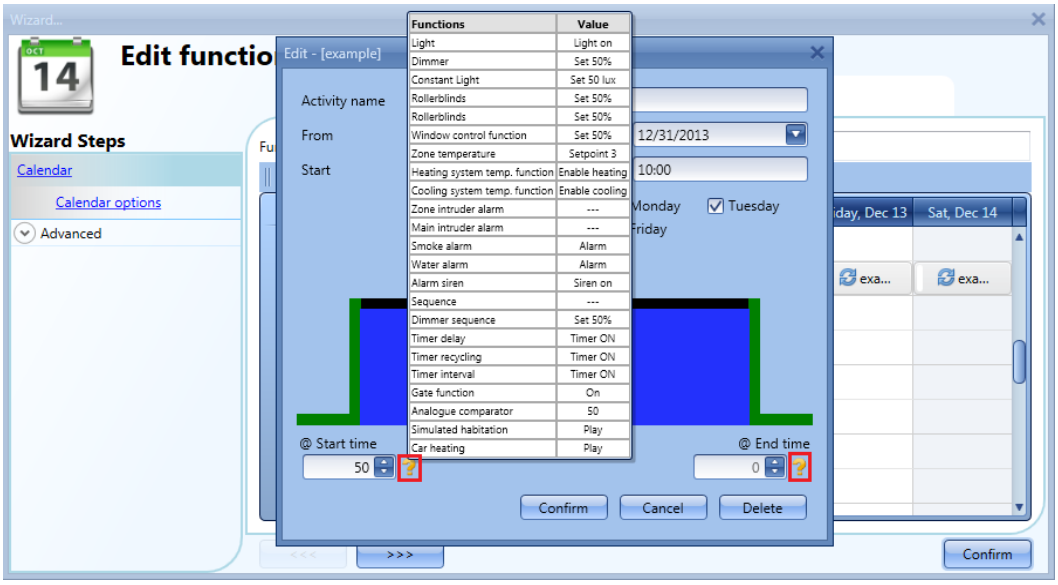
According to the function where the global calendar is used, the actions defined in the activity settings of the calendar can have different meanings.
In the table below, the different actions are shown according to the type of function.

n	Functions									
	ON/OFF light	**Dimmable light**	**Intruder alarm**	**Roller blind**	**Window**	**Sequence**	**Car-heating**	**Playback**	**System temp.**	**Zone temp.**
-1	No action	No action	No action	No action	No action	No action	No action	No action	No action	No action
0	OFF	OFF	Disarm alarm	0%	0%	Stop	OFF	Stop	OFF	OFF
1	ON	S1	Arm alarm	1%	1%	Start	Car ready	Play	ON	T1
2	ON	S2		2%	2%	Start		Play	ON	T2
3	ON	S3		3%	3%	Start		Play	ON	T3
4	ON	S4		4%	4%	Start		Play	ON	
5	ON	S5		5%	5%	Start		Play	ON	
6	ON	6%		6%	6%	Start		Play	ON	
7	ON	7%		7%	7%	Start		Play	ON	
8	ON	8%		8%	8%	Start		Play	ON	
9	ON	9%		9%	9%	Start		Play	ON	
10	ON	10%		10%	10%	Start		Play	ON	
11	ON	11%		11%	11%	Start		Play	ON	
12	ON	12%		12%	12%	Start		Play	ON	
13	ON	13%		13%	13%	Start		Play	ON	
14	ON	14%		14%	14%	Start		Play	ON	
15	ON	15%		15%	15%	Start		Play	ON	
16	ON	16%		16%	16%	Start		Play	ON	
17	ON	17%		17%	17%	Start		Play	ON	
18	ON	18%		18%	18%	Start		Play	ON	
19	ON	19%		19%	19%	Start		Play	ON	
20	ON	20%		20%	20%	Start		Play	ON	
21	ON	21%		21%	21%	Start		Play	ON	
22	ON	22%		22%	22%	Start		Play	ON	
23	ON	23%		23%	23%	Start		Play	ON	
24	ON	24%		24%	24%	Start		Play	ON	
25	ON	25%		25%	25%	Start		Play	ON	
26	ON	26%		26%	26%	Start		Play	ON	
27	ON	27%		27%	27%	Start		Play	ON	
28	ON	28%		28%	28%	Start		Play	ON	
29	ON	29%		29%	29%	Start		Play	ON	
30	ON	30%		30%	30%	Start		Play	ON	
31	ON	31%		31%	31%	Start		Play	ON	
32	ON	32%		32%	32%	Start		Play	ON	
33	ON	33%		33%	33%	Start		Play	ON	
34	ON	34%		34%	34%	Start		Play	ON	
35	ON	35%		35%	35%	Start		Play	ON	
36	ON	36%		36%	36%	Start		Play	ON	
37	ON	37%		37%	37%	Start		Play	ON	
38	ON	38%		38%	38%	Start		Play	ON	
39	ON	39%		39%	39%	Start		Play	ON	
40	ON	40%		40%	40%	Start		Play	ON	
41	ON	41%		41%	41%	Start		Play	ON	
42	ON	42%		42%	42%	Start		Play	ON	

43	ON	43%		43%	43%	Start		Play	ON	
44	ON	44%		44%	44%	Start		Play	ON	
45	ON	45%		45%	45%	Start		Play	ON	
46	ON	46%		46%	46%	Start		Play	ON	
47	ON	47%		47%	47%	Start		Play	ON	
48	ON	48%		48%	48%	Start		Play	ON	
49	ON	49%		49%	49%	Start		Play	ON	
50	ON	50%		50%	50%	Start		Play	ON	
51	ON	51%		51%	51%	Start		Play	ON	
52	ON	52%		52%	52%	Start		Play	ON	
53	ON	53%		53%	53%	Start		Play	ON	
54	ON	54%		54%	54%	Start		Play	ON	
55	ON	55%		55%	55%	Start		Play	ON	
56	ON	56%		56%	56%	Start		Play	ON	
57	ON	57%		57%	57%	Start		Play	ON	
58	ON	58%		58%	58%	Start		Play	ON	
59	ON	59%		59%	59%	Start		Play	ON	
60	ON	60%		60%	60%	Start		Play	ON	
61	ON	61%		61%	61%	Start		Play	ON	
62	ON	62%		62%	62%	Start		Play	ON	
63	ON	63%		63%	63%	Start		Play	ON	
64	ON	64%		64%	64%	Start		Play	ON	
65	ON	65%		65%	65%	Start		Play	ON	
66	ON	66%		66%	66%	Start		Play	ON	
67	ON	67%		67%	67%	Start		Play	ON	
68	ON	68%		68%	68%	Start		Play	ON	
69	ON	69%		69%	69%	Start		Play	ON	
70	ON	70%		70%	70%	Start		Play	ON	
71	ON	71%		71%	71%	Start		Play	ON	
72	ON	72%		72%	72%	Start		Play	ON	
73	ON	73%		73%	73%	Start		Play	ON	
74	ON	74%		74%	74%	Start		Play	ON	
75	ON	75%		75%	75%	Start		Play	ON	
76	ON	76%		76%	76%	Start		Play	ON	
77	ON	77%		77%	77%	Start		Play	ON	
78	ON	78%		78%	78%	Start		Play	ON	
79	ON	79%		79%	79%	Start		Play	ON	
80	ON	80%		80%	80%	Start		Play	ON	
81	ON	81%		81%	81%	Start		Play	ON	
82	ON	82%		82%	82%	Start		Play	ON	
83	ON	83%		83%	83%	Start		Play	ON	
84	ON	84%		84%	84%	Start		Play	ON	
85	ON	85%		85%	85%	Start		Play	ON	
86	ON	86%		86%	86%	Start		Play	ON	
87	ON	87%		87%	87%	Start		Play	ON	
88	ON	88%		88%	88%	Start		Play	ON	
89	ON	89%		89%	89%	Start		Play	ON	
90	ON	90%		90%	90%	Start		Play	ON	
91	ON	91%		91%	91%	Start		Play	ON	
92	ON	92%		92%	92%	Start		Play	ON	
93	ON	93%		93%	93%	Start		Play	ON	
94	ON	94%		94%	94%	Start		Play	ON	
95	ON	95%		95%	95%	Start		Play	ON	
96	ON	96%		96%	96%	Start		Play	ON	
97	ON	97%		97%	97%	Start		Play	ON	

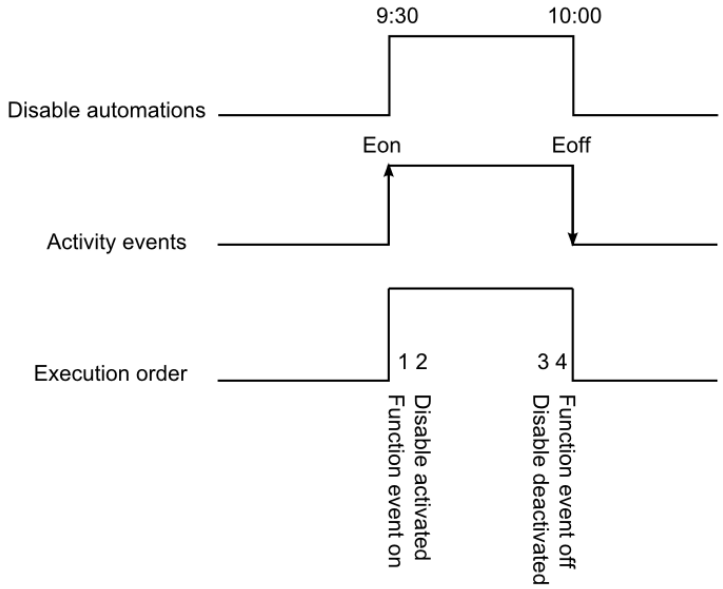
98	ON	98%		98%	98%	Start		Play	ON	
99	ON	99%		99%	99%	Start		Play	ON	
100	ON	100%		100%	100%	Start		Play	ON	

The user can look at the table in any time by moving the mouse cursor on the yellow question mark icon near the value (see picture below).



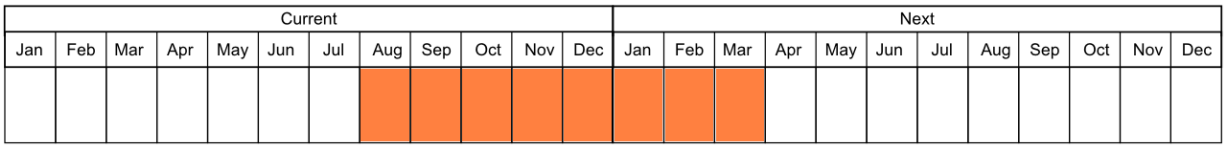
10.22.6 Simultaneous activities

When the user sets two activities in the same calendar (an event activity and a level activity) using the same start/stop times and dates, at start time the event activities are always executed before the level activity, at stop time the event activity is executed after the level activity (see picture below).

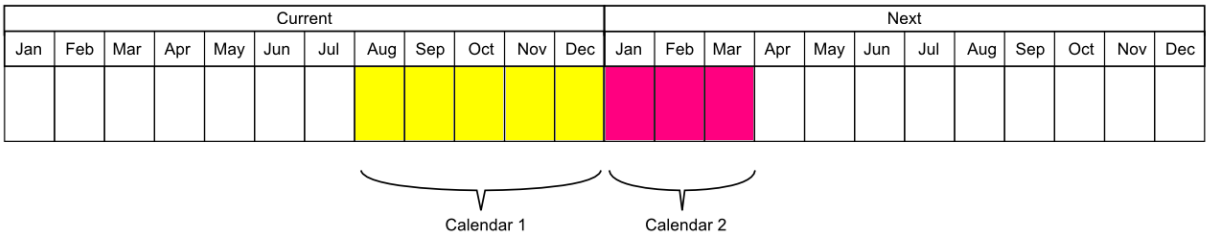


10.22.7 How to create a calendar in-between two years

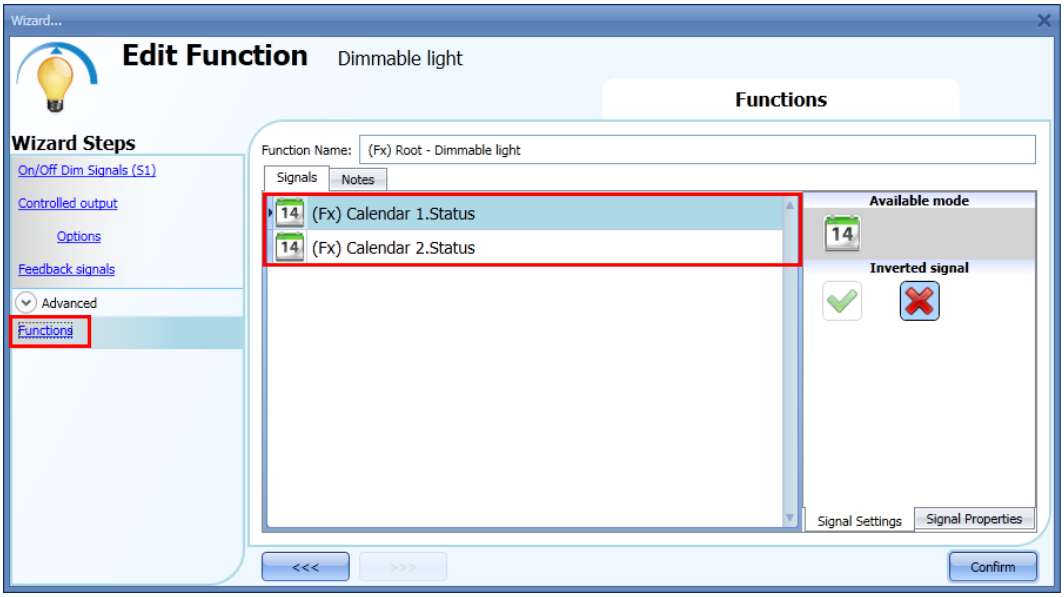
If the user wants to create an automation with a calendar where an activity starts in the current year and finishes in the next year, two separate activities must be created and linked to the function.
As an example, let us consider the creation of an automation that has to start on 1st August and finish on 1st March.



Both activities are set with the same actions at start/end time; the only difference is the date: the first activity will start on 1st August and end on 31st December, the second activity will start on 1st January and end on 1st March.



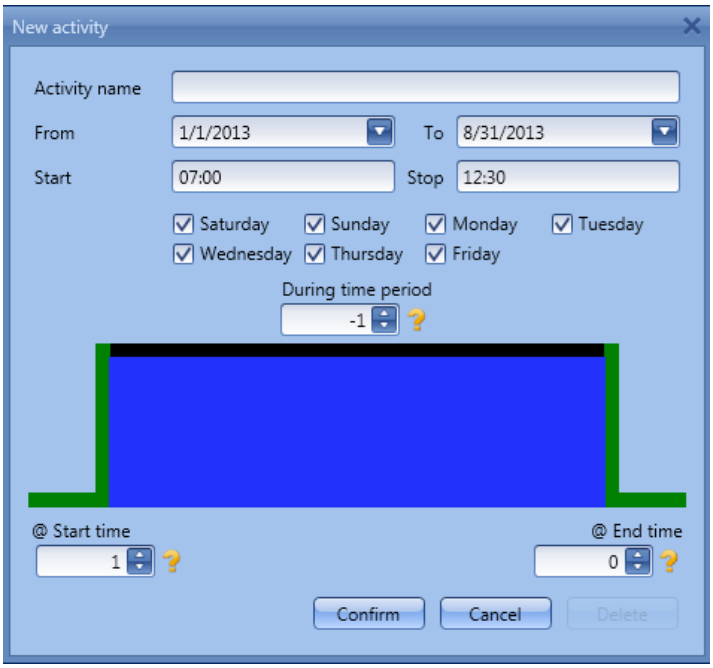
Once configured, the calendar must be added in the field *Function* of the function that has to be automated (see picture below).



10.22.8 How to create a recurring calendar

Once a calendar function is configured, in the following years it is not necessary to update the settings of the activities, since the calendar is automatically renewed.

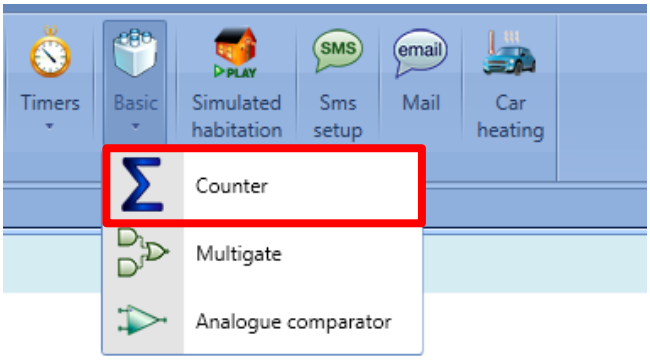
In the example below, a calendar is set to work from 1st January to 31st August 2013. The green edges indicate that an action is present both at the start and end time.



The calendar will automatically be updated to 2014 the following year and so on so forth.

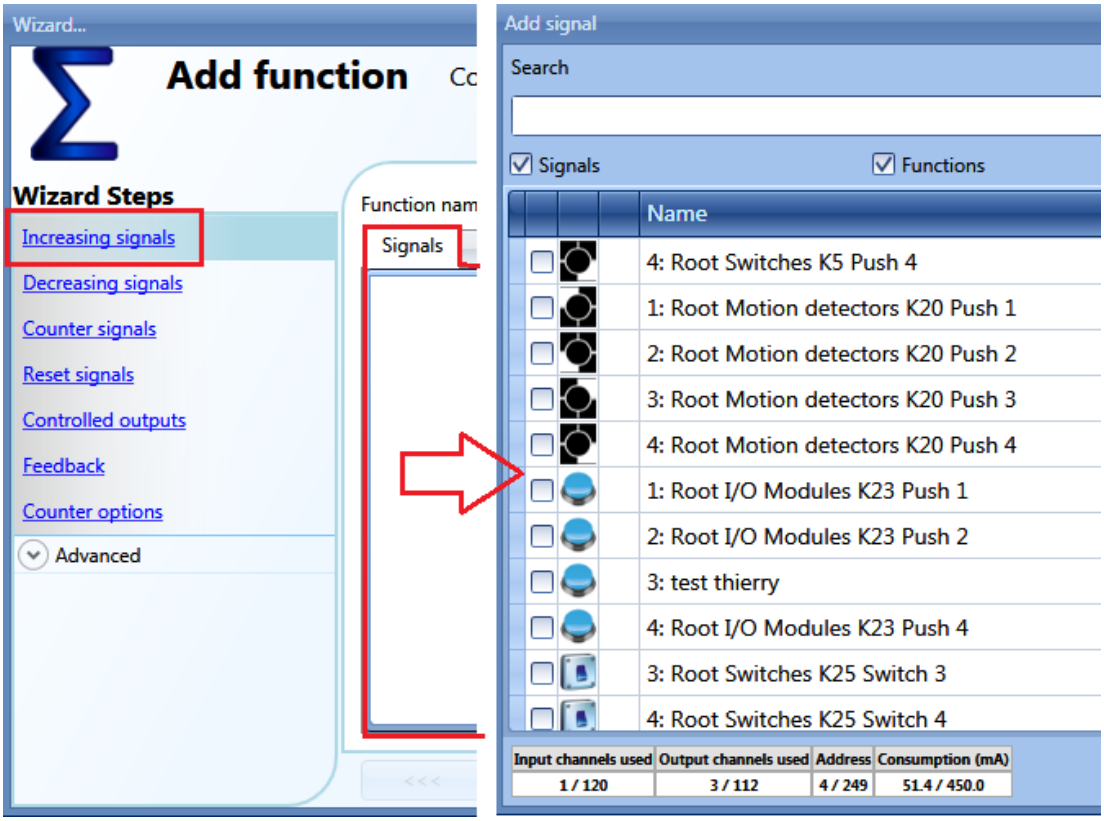
10.23 How to configure a counter function

The *Counter* function is a counter of input pulses with increments and decrements, which logs the total counter or just the partial one between the last two sampling times. The counted value can also be used to control an analogue output module.
To set up a counter function, the user has to select *Basic* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to a Counter function can also be opened by typing Alt+A+7+C (See table of short cuts).



10.23.1 How to add the increasing input

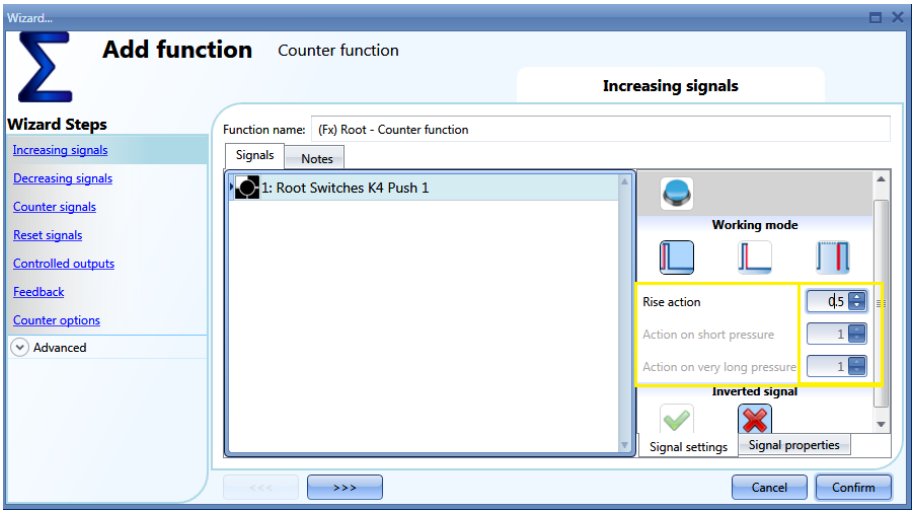
To add the input signals that will make the output value increase, the user has to click on *Increasing signals*, then double click on the *Signals* window and select the input signal from the list (see picture below).
The increasing signals may be actual signals such as pushbutton or switch, or functions, so that the number of activations can be counted.



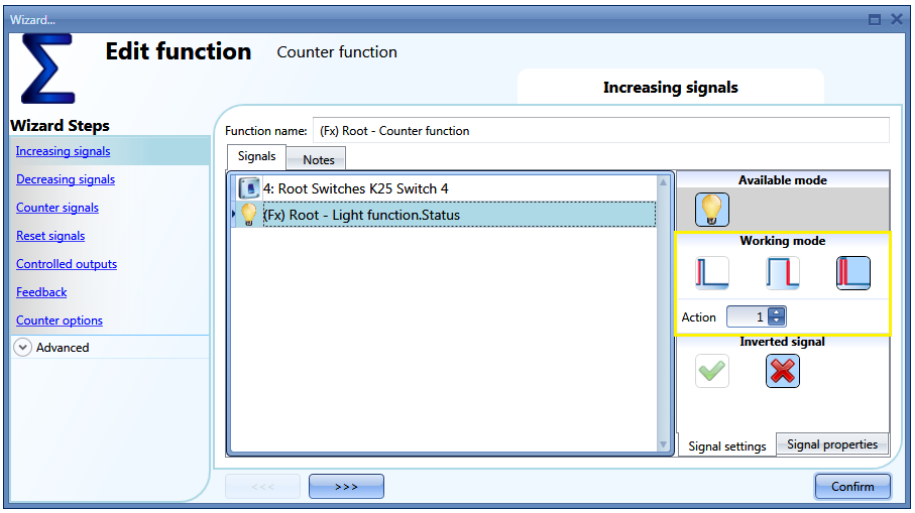
10.23.1.1 How to configure the selected increasing signals

As shown in the picture below, if a pushbutton is selected, the user can select the type of action that will make the counter increase: rising edge (i.e. as soon as the pushbutton is pressed), falling edge (i.e. as soon as the pushbutton is released), very long pressure (i.e. when the pushbutton is kept pressed for the set seconds). For each of these actions, the increasing value can be set (yellow rectangle).

For example, in the picture below, as soon as the pushbutton is pressed, the counter is increased by 0.5. This means that every two pulses, the counter is incremented by 1.

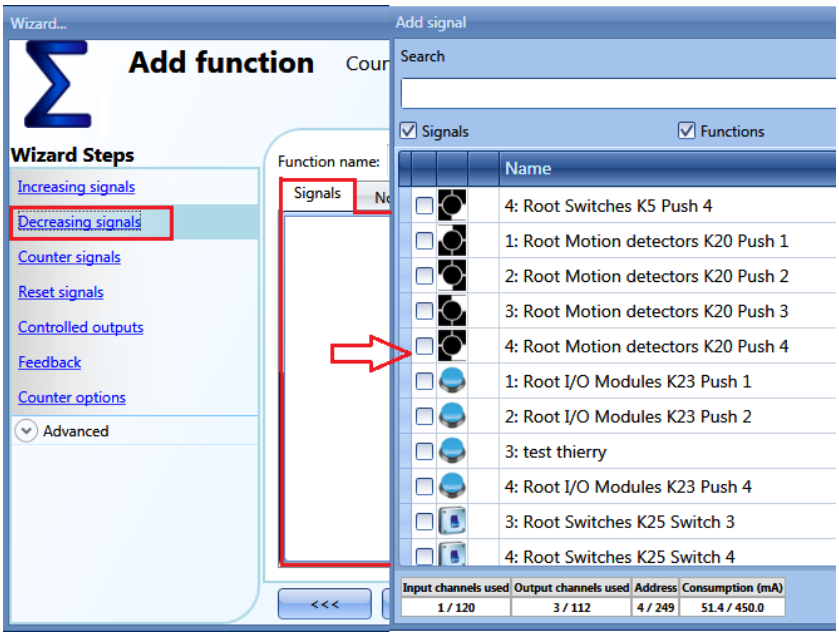


If a level signal is selected, such as a switch or a function, the available actions are rising edge (i.e. when the switch is closed or the function is activated), falling edge (i.e. when the switch is open or the function is deactivated), or both. For each of them, the user can set the increasing value (yellow rectangle).



10.23.2 How to add the decreasing input

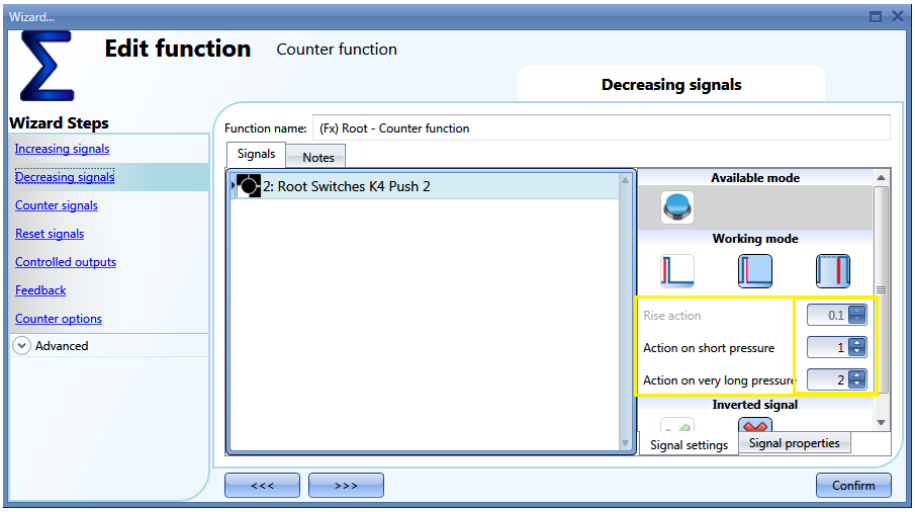
To add the input signals that will make the output value decrease, the user has to click on *Decreasing signals*, then double click on the *Signals* window and select the input signal from the list (see picture below).
The decreasing signals may be actual signals such as pushbutton or switch, or functions, so that the number of activations can be counted.



10.23.2.1 How to configure the selected decreasing signals

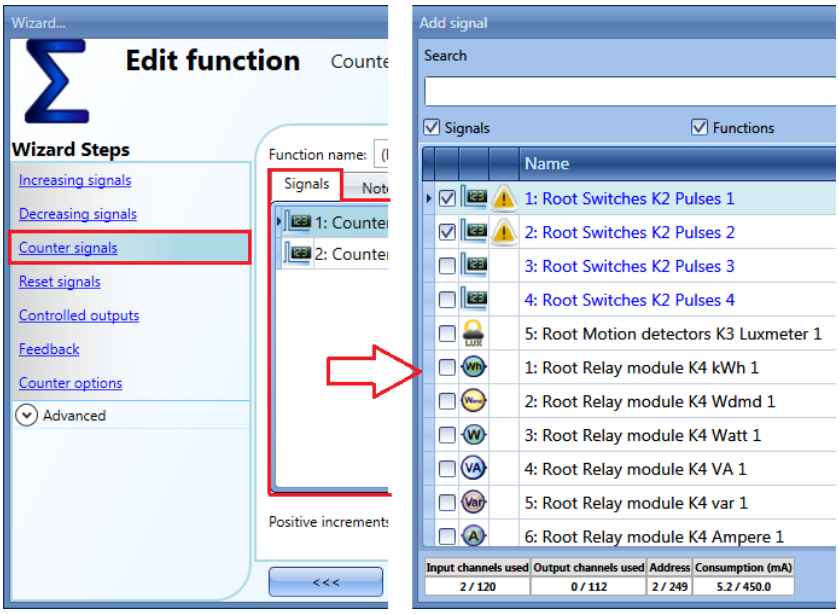
As shown in the picture below, if a pushbutton is selected, the user can select the type of action that will make the counter decrease: rising edge (i.e. as soon as the pushbutton is pressed), falling edge (i.e. as soon as the pushbutton is released), very long pressure (i.e. when the pushbutton is kept pressed for the set seconds). For each of these actions, the decreasing value can be set (yellow rectangle).

For example, in the picture below, as soon as the pushbutton is released, the counter is decreased by 1, while if the pushbutton is kept pressed, the counter is decreased by two.



10.23.3 How to sum up analogue values

To add analogue values or values coming from a counting module, the user has to select *Counter signals.*



In this field any analogue signals can be added and the counting function calculates the sum. It can be used as a virtual energy meter to sum up the values coming from different meters or it can be used to sum up the counting values coming from different input modules such as SH2INDI424, SHPINCNT4, SHPINCNTS04 and SHJWINS04xxx.

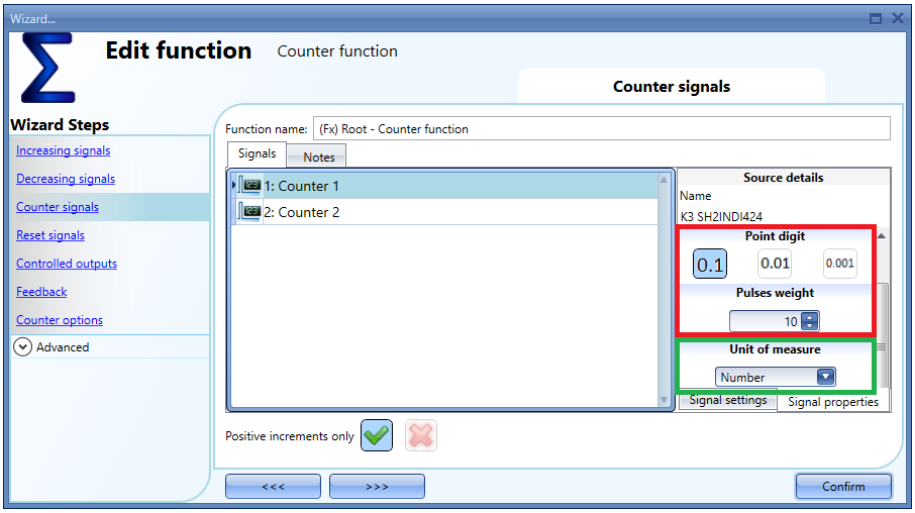
In the figure below, two counter values are added in the counter function, sourced from the module SH2INDI424: the result of the function is the sum of the two.

Counter function= Counter 1+ Counter 2

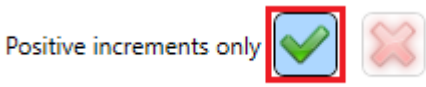
To set the increasing value of each pulse received on the counting input, in the *Signal properties* tab window the *Point digit* and *Pulses weight* field have to be correctly set.
In the figure below, each pulse received will increase the function by one, since the increasing factor is calculated in the following way:

Increasing value = Number of pulses x Point digit x Pulses weight

Thanks to this calculation, the user can define any increasing value.



Positive increments only
When the green *V* is selected, only the positive values are considered for the counters added in the *Counters signals* window. Negative values will not be considered for the calculation.



10.23.4 How to configure the counting function to count visitors to retail outlets

The counting systems can be divided into 2 groups: one-way systems or two-way systems.

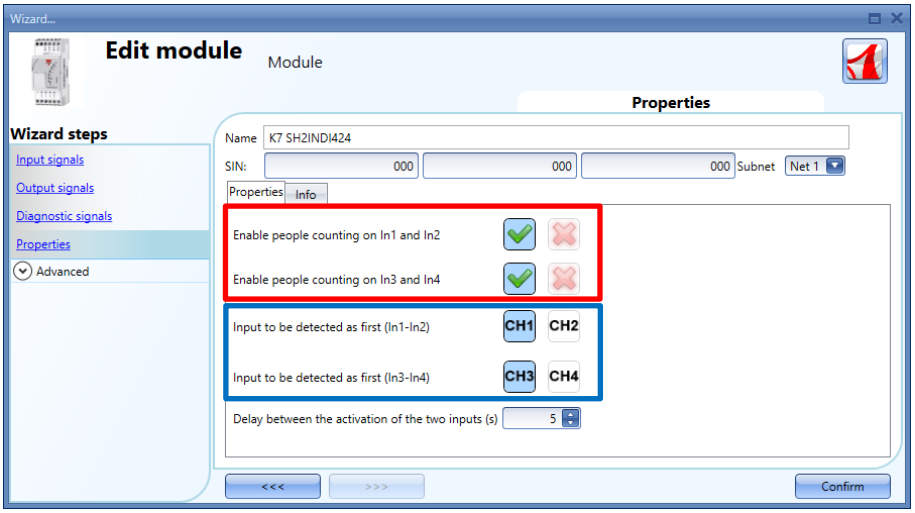
10.23.4.1 One-Way System

One-way systems count the number of passages in front of the detector, regardless of whether the passage is incoming or outgoing. The total number of visitors is obtained thanks to the division by 2 performed by our system.
To set the UWP 3.0 system for this kind of detection, the paragraph *How to configure the selected increasing signals* has to be followed, setting the *action* on the signal activation equal to *0.5* to obtain the division by 2.

10.23.4.2 Two-Way System

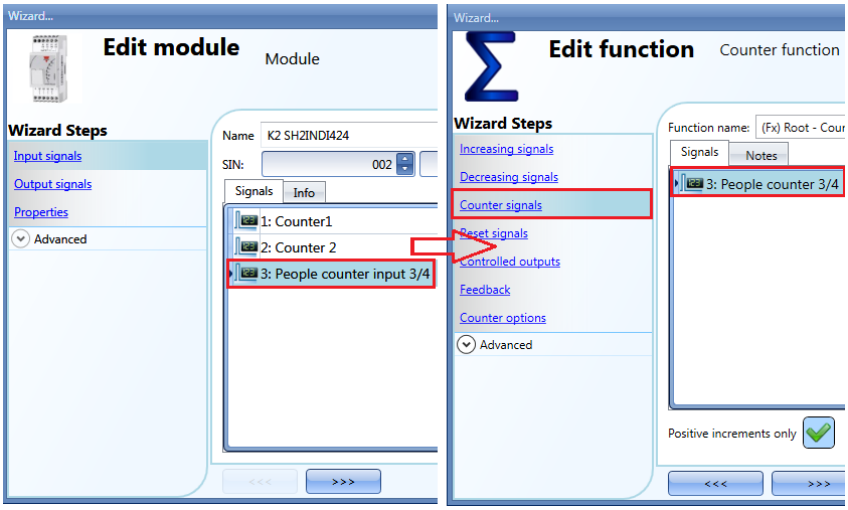
Two-way systems count the number of passages in front of the detectors, taking into account the direction of movement. The distinction is thus made between entrances and exits. The exact number of people present within the premises is available at all times.
The direction of movement is detected by using two sensors: if sensor **1** detects movement *before* sensor **2**, people are entering and the counter is incremented; if sensor **1** detects movement *after* sensor **2**, people are exiting and the counter is decremented.
This operation is carried out directly by the module such as SH2INDI424, SHPINCNT4, SHPINCNTS04 and SHJWINS04xxx which has to be configured as described below.

In the *Properties* field in the wizard of the input modules, if two inputs have to work in a pair as described above, the check boxes marked in red have to be enabled: by doing this, input 1 is coupled with input 2 and input 3 is coupled with input 4. It is not mandatory to couple all the four inputs.
Once this check box is enabled, the module increments/decrements the internal counter, according to which sensor is detected first (fields marked in blue).



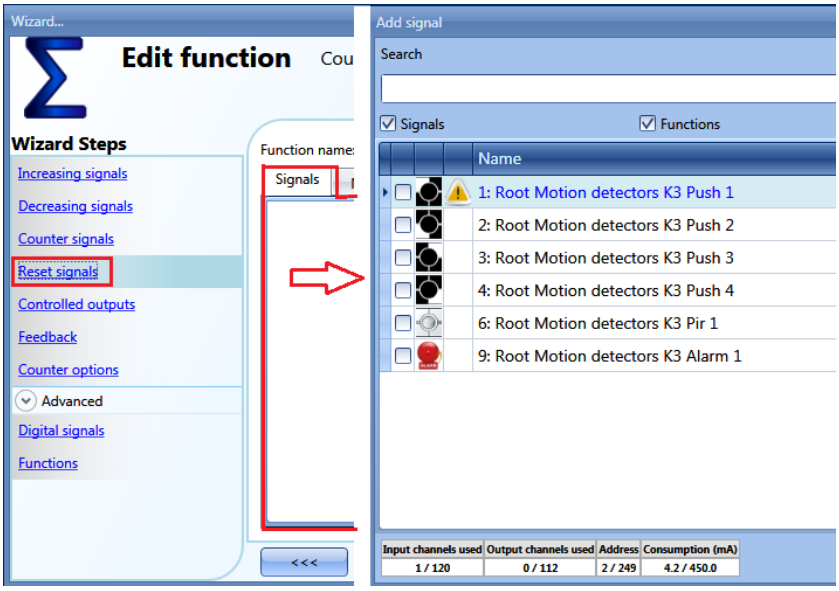
The *Delay between the activation of the two inputs(s)* field is used to make no increment or decrement if the paired sensors are detected with an interval time longer than the number of seconds set in this field.

In the picture below, an example is shown where input 1 and input 2 are configured as “standalone” pulse counter 2, while inputs 3 and 4 are configured to count people by detecting the direction: since they are working together, only one counter is present. This counter then has to be added in the field *Counter signals* of the Counting function.



10.23.5 How to reset the counting value

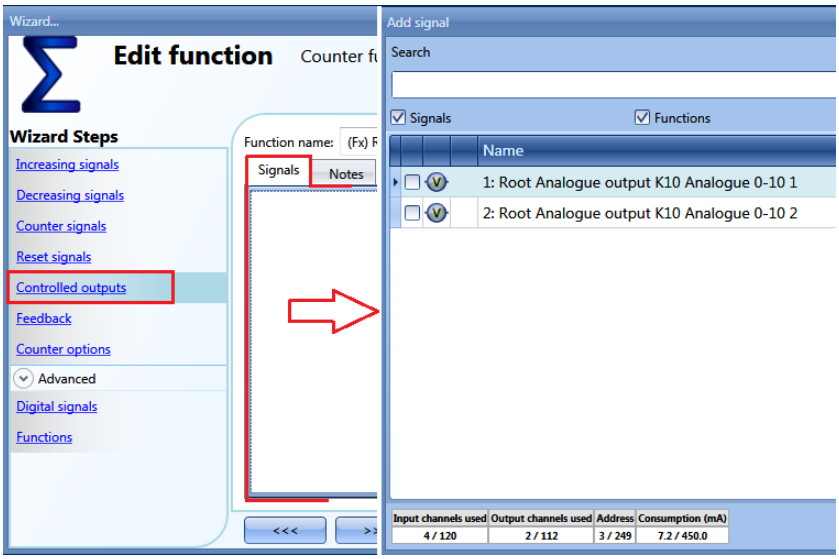
To reset the counting value, the user has to click on *Reset signal*, then double click on the *Signals* window and select the input signal from the list (see picture below).
The reset signals may be actual signals such as pushbutton or switch, or functions.



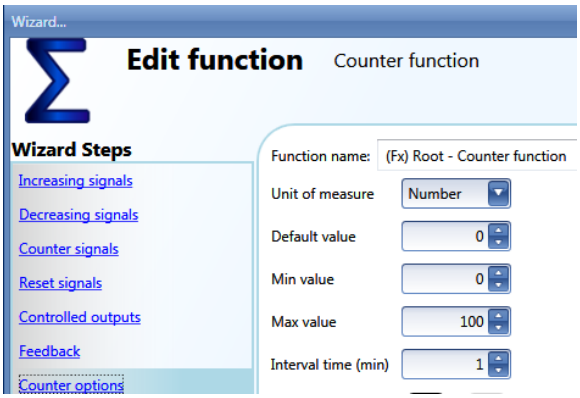
Each time the reset signal is activated, the counting value is reset.

10.23.6 How to add an analogue output signal

To add an analogue output signal, the user has to click on *Controlled outputs*, then double click on the *Signals* window and select the output signal from the list (see picture below).
The outputs are the signals from the SHPOUTV224. The analogue output value of the SHPOUTV224 is set at the value calculated by the counter.



The 0-10 V output value is calculated based on a linear relation between the minimum and maximum values set in the counter function, as explained in the example below.

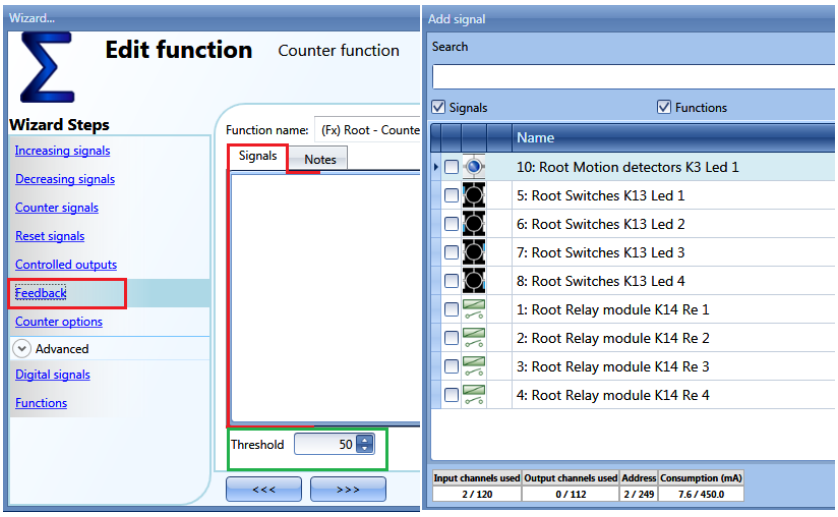


If the max and min values of the counting function are 100 and 0, and the virtual output value is 30, the voltage output of the SHPOUTV224 will be 3V.
If the output of the counting function is 50, the voltage output of the SHPOUTV224 will be 5V.

The output could also be a virtual output, not linked to any actual signal.

10.23.7 How to add a feedback signal or a relay output

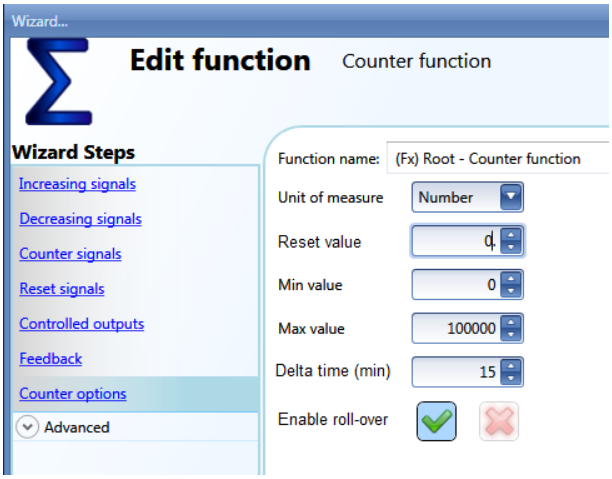
To add a feedback signal or a relay output, the user has to click on *Feedback signal*, then double click on the *Signals* window and select the input signal from the list (see picture below).
The feedback signals may be LEDs or relay outputs.



The selected signals will be on if the counter is greater than a configurable threshold, otherwise it will be off.
The threshold can be set in the field marked with the green rectangle.

10.23.8 How to set the counting options

To configure the counting options, the user has to click on *Counter options*: a window with the different configuration options will appear (see picture below).



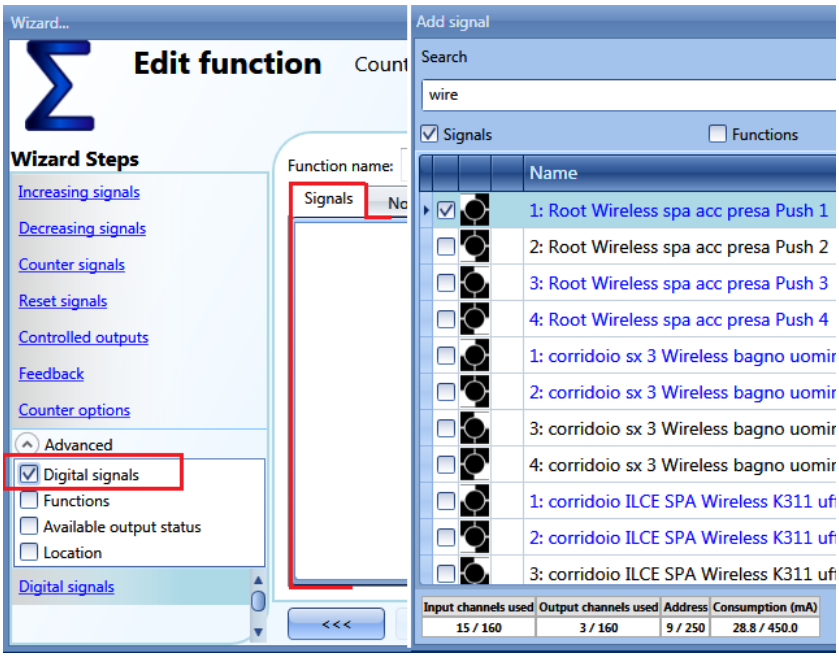
In this window, the user has to set the *Reset value* of the counter at start-up, the *maximum* and the *minimum* values the counter is to have and above or below which the counter will not go, and the *Delta time,* which is the interval time used to calculate how much the counter has been increased.

Delta= Counter $_{\text{Delta time}}$ - Counter $_{\text{Delta time-1}}$

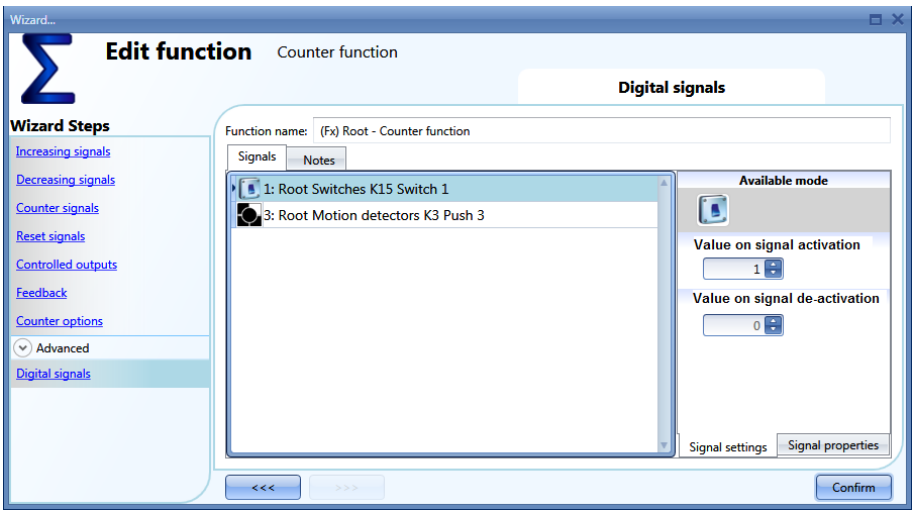
This is also the value logged if the *Delta time* is not zero. If *Delta time* is set to 0, no delta calculation is made.
If the field *Enable roll-over* is selected, the function starts again from the minimum value once the maximum value is reached. If the roll-over is not enabled, once the maximum is reached, the function stops at this value.

10.23.9 How to set a predefined value using signals

If the user needs to set a predefined value activating a signal (e.g. the output is set to 10 if a push button is pressed), in the *Advanced* settings *Digital signals* has to be selected.

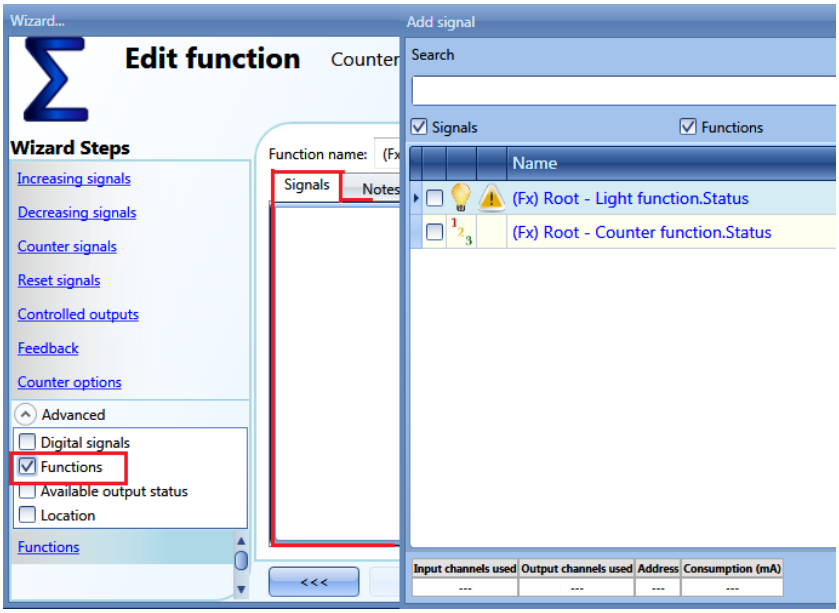


As shown in the picture below, in the *Signal settings* tab window, the user can define the value the counter function is to have when the signal is activated (*Value on signal activation*) and the value the function is to have when the signal is deactivated (*Value on signal de-activation*).
Different settings can be configured for each output.



10.23.10 How to set the counter function equal to another function

If the user needs to set the counter function equal to another function, *Function* has to be selected in the *Advanced* settings.



10.23.11 How to log the outputs of the counting function

To be logged, the counter function has to be added to the database in the usual way.
In the logged file, two columns will be present: the first one (*COUNTER 1*) indicates the total value calculated; the second one (*COUNTER 2*) the difference in the Delta time:

Delta= Counter $_{\text{Delta time}}$ - Counter $_{\text{Delta time-1}}$

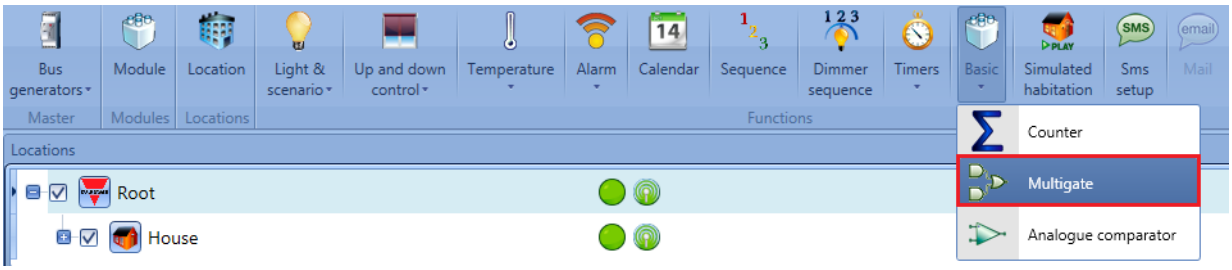
Since the *Delta time* can be different from the sampling time, the difference calculated from two consecutive rows in column *COUNTER 1* might not correspond to the relevant line in column *COUNTER 2*. If *Delta time* is set to 0, the column *COUNTER 2* will not be present.

A	B	C	D	F	G	L	M
Position	Timestam	Date	Hour	Object Name	Location Name	COUNTER 1	COUNTER 2
3	1,4E+09	07/07/2014	15:08:30	(Fx) Root - Counter function1	Root	10	0
4	1,4E+09	07/07/2014	15:09:30	(Fx) Root - Counter function1	Root	31	21
5	1,4E+09	07/07/2014	15:10:30	(Fx) Root - Counter function1	Root	31	0
6	1,4E+09	07/07/2014	15:11:30	(Fx) Root - Counter function1	Root	39	0
7	1,4E+09	07/07/2014	15:12:30	(Fx) Root - Counter function1	Root	42	11
8	1,4E+09	07/07/2014	15:13:31	(Fx) Root - Counter function1	Root	47	5
9	1,4E+09	07/07/2014	15:14:30	(Fx) Root - Counter function1	Root	47	0
10	1,4E+09	07/07/2014	15:15:30	(Fx) Root - Counter function1	Root	47	0
11	1,4E+09	07/07/2014	15:16:30	(Fx) Root - Counter function1	Root	47	0
12	1,4E+09	07/07/2014	15:17:30	(Fx) Root - Counter function1	Root	47	0
13	1,4E+09	07/07/2014	15:18:30	(Fx) Root - Counter function1	Root	47	0
14	1,4E+09	07/07/2014	15:19:30	(Fx) Root - Counter function1	Root	47	0
15	1,4E+09	07/07/2014	15:20:30	(Fx) Root - Counter function1	Root	47	0
16	1,4E+09	07/07/2014	15:21:30	(Fx) Root - Counter function1	Root	47	0
17	1,4E+09	07/07/2014	15:22:30	(Fx) Root - Counter function1	Root	47	0
18	1,4E+09	07/07/2014	15:23:30	(Fx) Root - Counter function1	Root	47	0
19	1,4E+09	07/07/2014	15:24:30	(Fx) Root - Counter function1	Root	47	0
20	1,4E+09	07/07/2014	15:25:30	(Fx) Root - Counter function1	Root	47	0

N.B: If a BACnet map with at least a Counter function has been created from an older version of SB Tool (older than *SB Tool 6.2.3 – R594*), the BACnet map must be **regenerated again**.

10.24 How to set a Multigate function

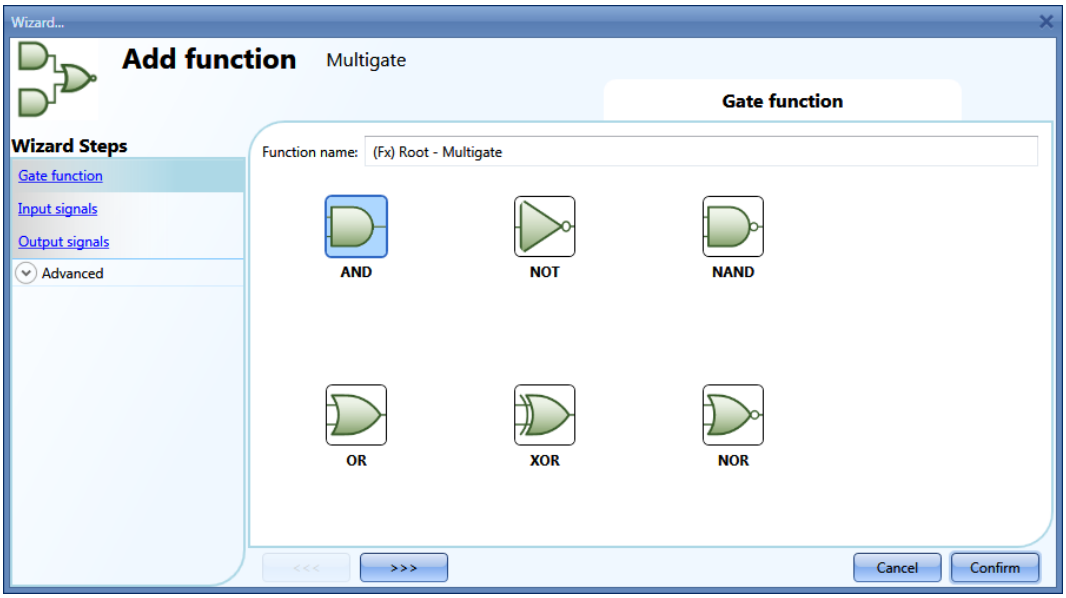
The multigate function can be used to perform a logical operation with one or more inputs to have a single logic output status.
To set up a multigate function the user has to select *Basic* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to a Multigate function can also be opened by typing Alt+A+7+M.



The multigate function can be used to perform different actions: for example to collect signals from more than one point to monitor, for example, door contacts or window switches in an alarm function, or to monitor more signals and/or functions to get a quick overview of a status.
The user can link together more multigate functions to achieve customized logical functions.

10.24.1 How to select the logic operation

In the first step of the wizard, the user can select the logic operation of the multigate function by clicking on the required icon (in the picture below the AND logic is selected).

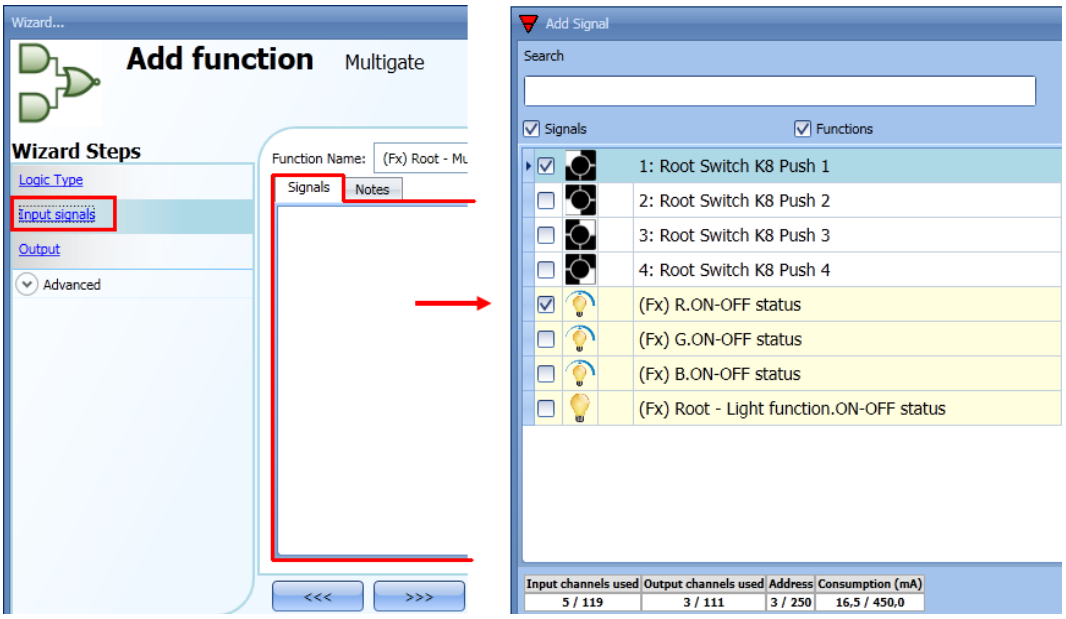


The selectable logic operators are:

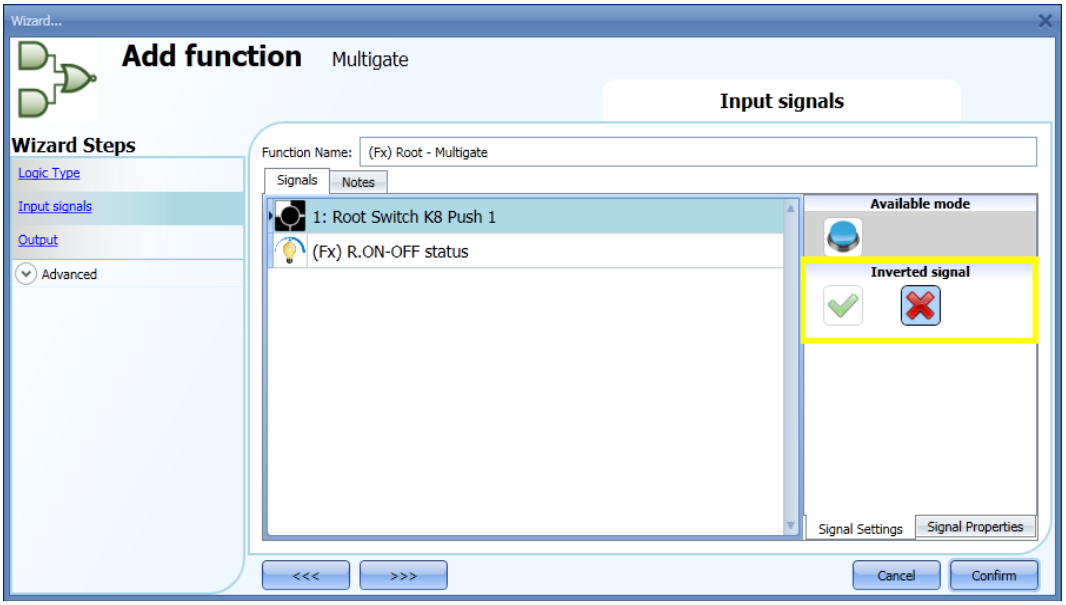
- And
- Not
- Nand
- Or
- Xor
- Nor

10.24.2 How to add the input signals

In the *Input signals* step of the wizard, the user should enter all the signals used for the logic operation of the multigate function.
The user can use physical signals such as pushbutton or switch, or status signals from the other functions configured in the project. Up to 100 input signals can be added.
When the NOT logic is selected, only the first added input signal will be inverted.

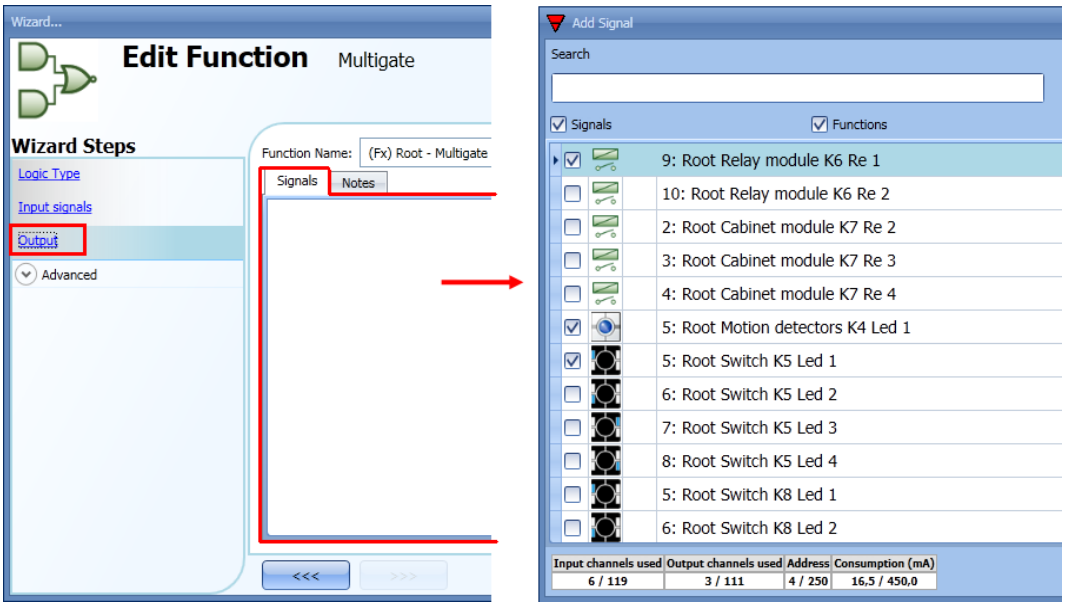


Once the input signals are entered, the user can also choose to use the inverted logic (see yellow rectangle in the picture below).
The inverted logic is enabled by selecting the green V.

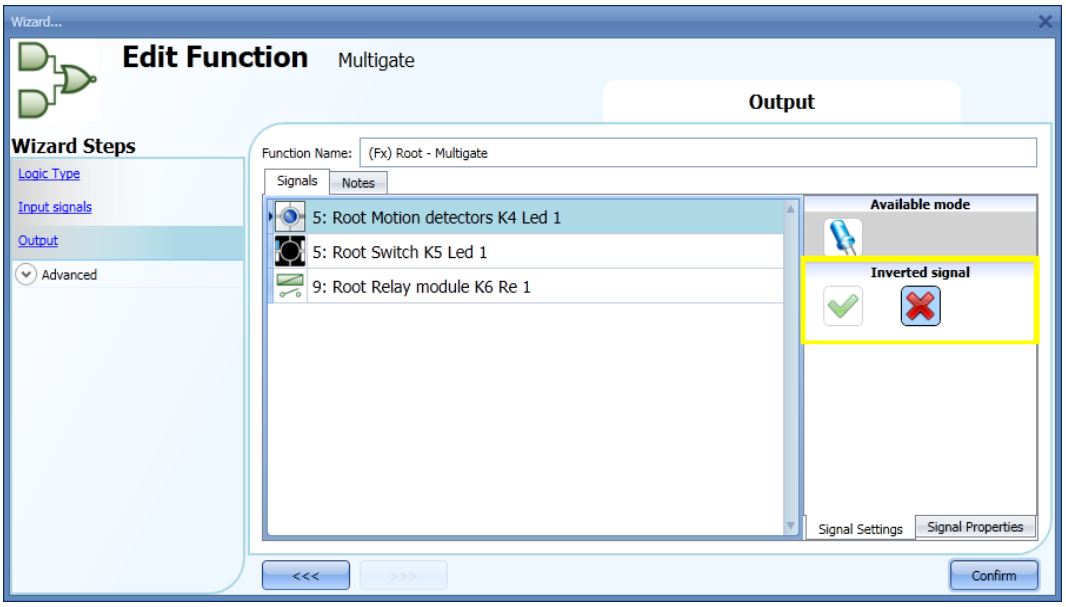


10.24.3 How to add the output signal

The multigate can be used as an input signal for other functions or an output signal can be directly activated by it.



Once the output signals are entered, the user can also choose to use the inverted logic of the signal (see yellow rectangle in the picture below).
The inverted logic is enabled by selecting the green V (when the multigate function is activated, the output stays deactivated).

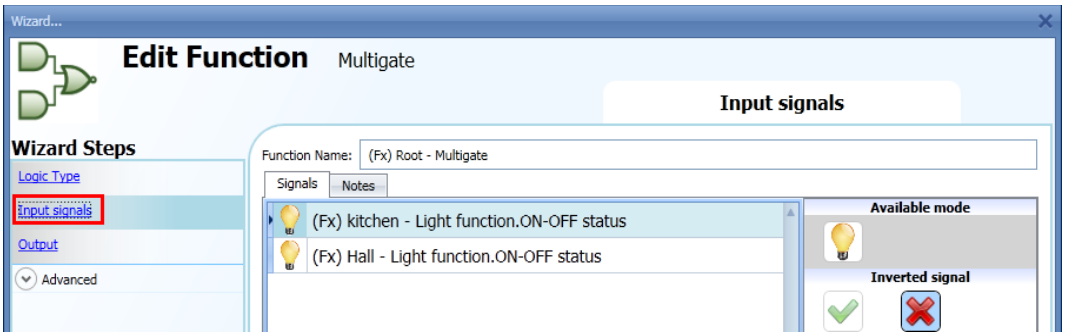


10.24.4 How to check the status of more functions

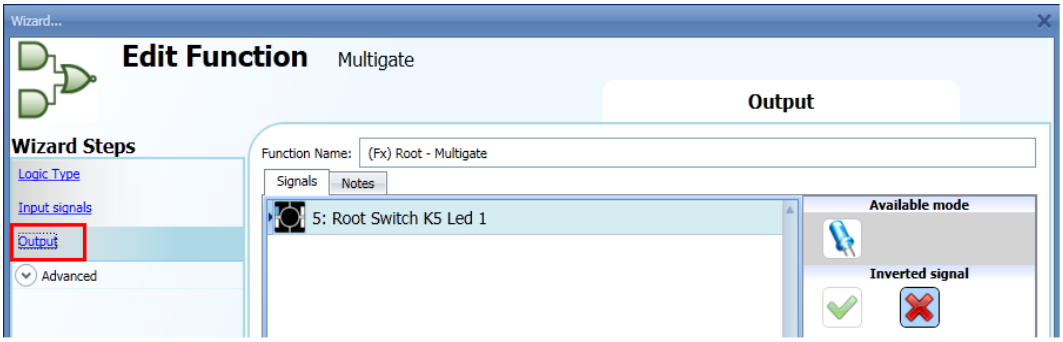
In the first example, the multigate function is used to check the status of two light functions: when both the lights are on, the multigate is activated. The status of the multigate is indicated by the LED output.

The *Logic type* used is the AND gate.

In the *Input signals* step, the user has to add the two light functions.



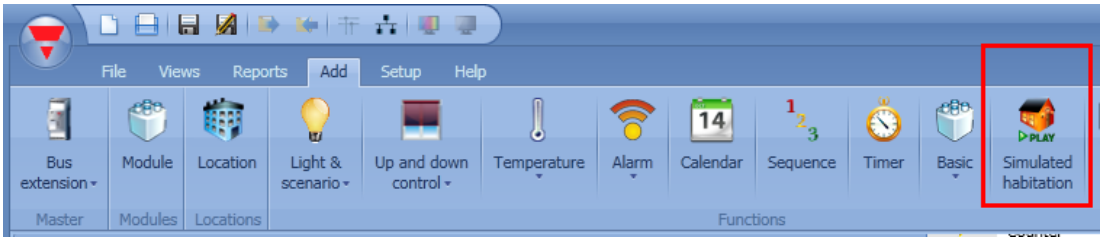
In the *Output signals* step, the user has to add the LED signal.



When both the two lights are ON, the multigate function is activated and the LED output is on.

10.25 How to set the simulated habitation function

The simulated habitation function can be used to give the impression that the house is inhabited even if the user is out.
To set up a simulated habitation function the user has to select *Simulated habitation* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to a Simulated habitation function can also be opened by typing Alt+A+D. (See table of short cuts).



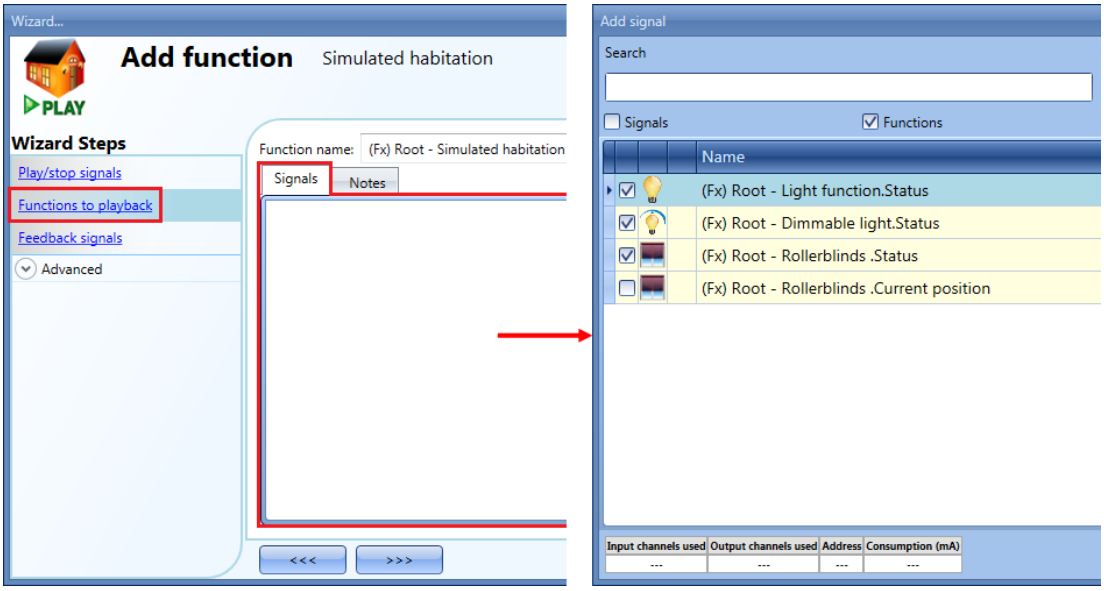
In the Simulated habitation function, the user can add all the functions that he wants to reproduce and that have been recorded during the previous week.
The system always records every change in the output status of the selected function when it is not in playback, and then, when the signal used to start the playback is active, the pattern that has been recorded during the past week is played back.
The system does not record any event when the function is in playback mode.

10.25.1 How to add a function to be recorded

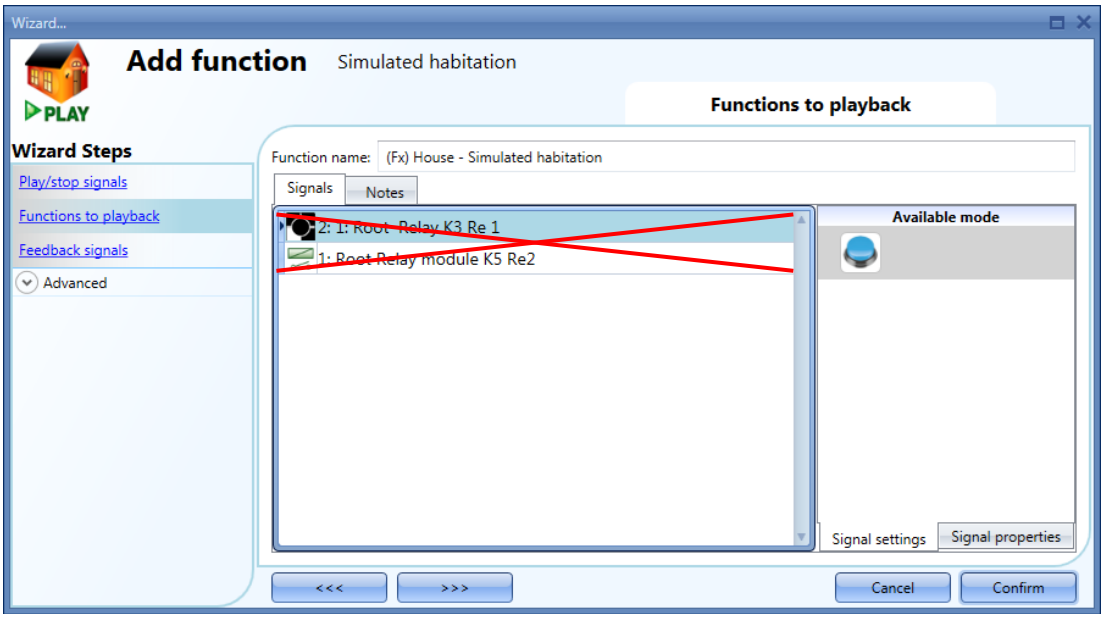
The functions that can be recorded are:

- Light on/off
- Dimmable light
- Rollerblind
- Siren
- Timer (only interval type)

To select the output signals that have to be recorded and played back by the simulated habitation function, click on *Functions to playback* and then double click on the Signals window. Once the output window is opened, select the required functions from the list (see picture below).
Only Functions can be recorded.



Other types of signals (e.g. relays, push buttons), will not be recorded.



Each added function is automatically recorded. The logging starts as soon as the configuration file is written into the UWP 3.0. The recording period is a week (the week starting from the writing of the configuration) and when it ends the new data overwrites the previous data.

Example:
In this simple example, the user has set up a simulated habitation function in order to record a light function. The recording starts as soon as the configuration is written (Monday at 12:00) and ends the following week at the same time (highlighted in yellow in the picture below).
Starting from Monday in week 2, all the actions recorded the previous week are overwritten with the new light activations.

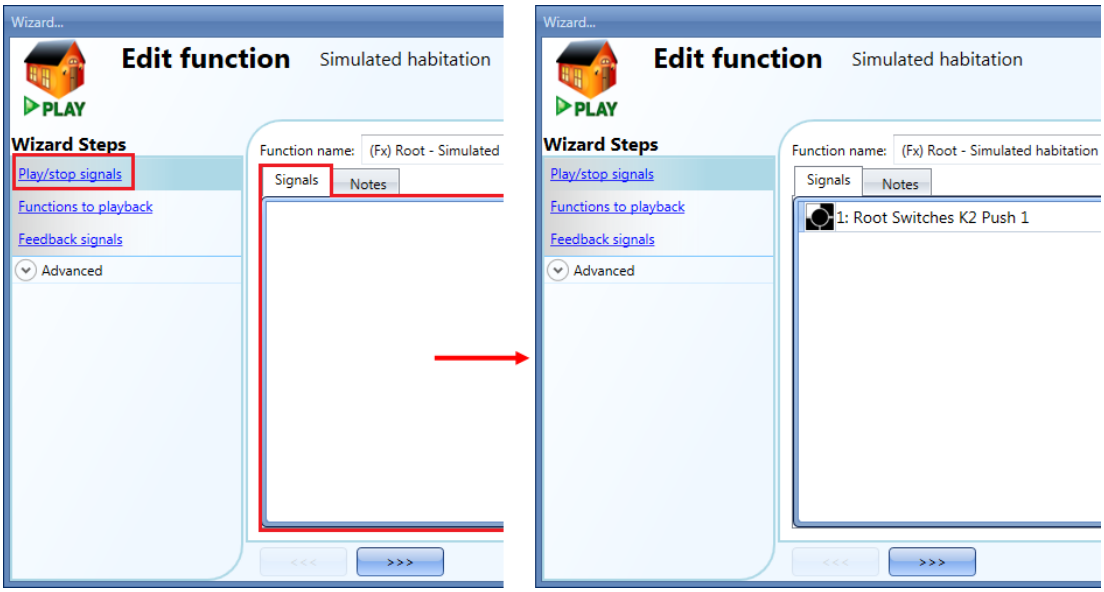
Week 1							Week 2		
Monday	Tuesday	Wednesday	Thursday	Friday	Saturday	Sunday	Monday	Tuesday	Wednesday
12:00 Write configuration	10:00 light on 11:30 light off	14:00 light on 19:00 light off	17:00 light on 21:00 light off		18:00 light on 23:00 light off	13:00 light on 19:00 light off	12:00 End of the recording week		

Let us assume that the controller has been running for some time, and at 10:00 on Wednesday week 2 the user leaves the house for a holiday and activates the playback of the simulated habitation.
All the patterns recorded the previous week are played back (highlighted in blue), giving the impression that somebody is using the light in the house.

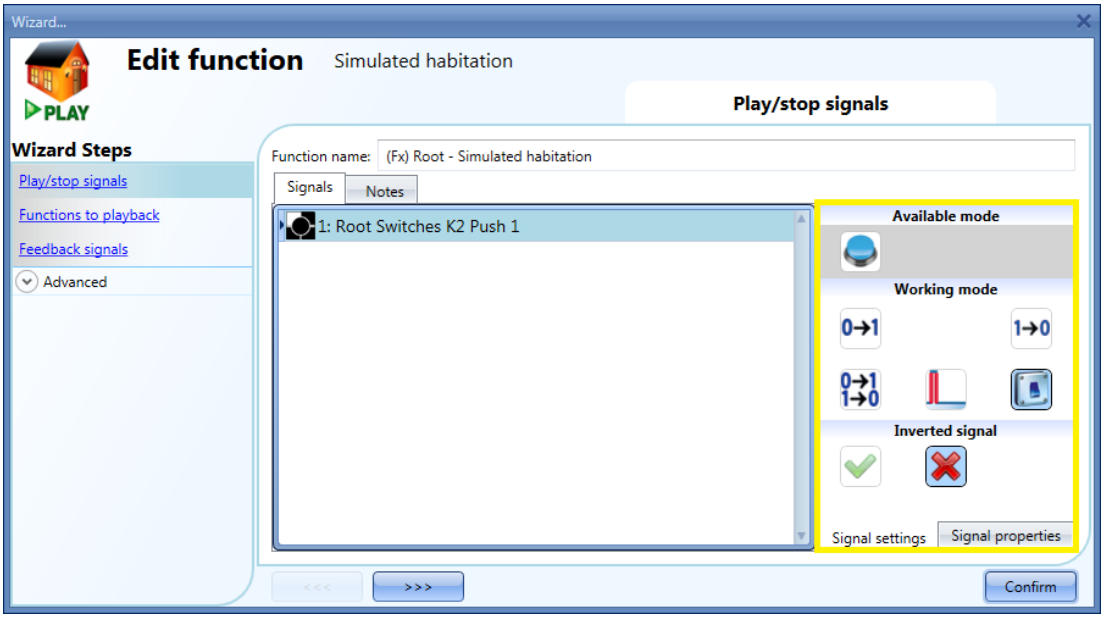
Week 1							Week 2						
Monday	Tuesday	Wednesday	Thursday	Friday	Saturday	Sunday	Monday	Tuesday	Wednesday	Thursday	Friday	Saturday	Sunday
12:00 Write configuration	10:00 light on 11:30 light off	14:00 light on 19:00 light off	17:00 light on 21:00 light off		18:00 light on 23:00 light off	13:00 light on 19:00 light off			10:00 start playback 14:00 light on 19:00 light off	17:00 light on 21:00 light off		18:00 light on 23:00 light off	13:00 light on 19:00 light off

10.25.2 How to play/stop the simulated habitation function

To add a play/stop signal, the user has to select the relevant section *Play/stop signals* in the menu, then double click on the *Signals* window and select the input signal from the list (see picture below).



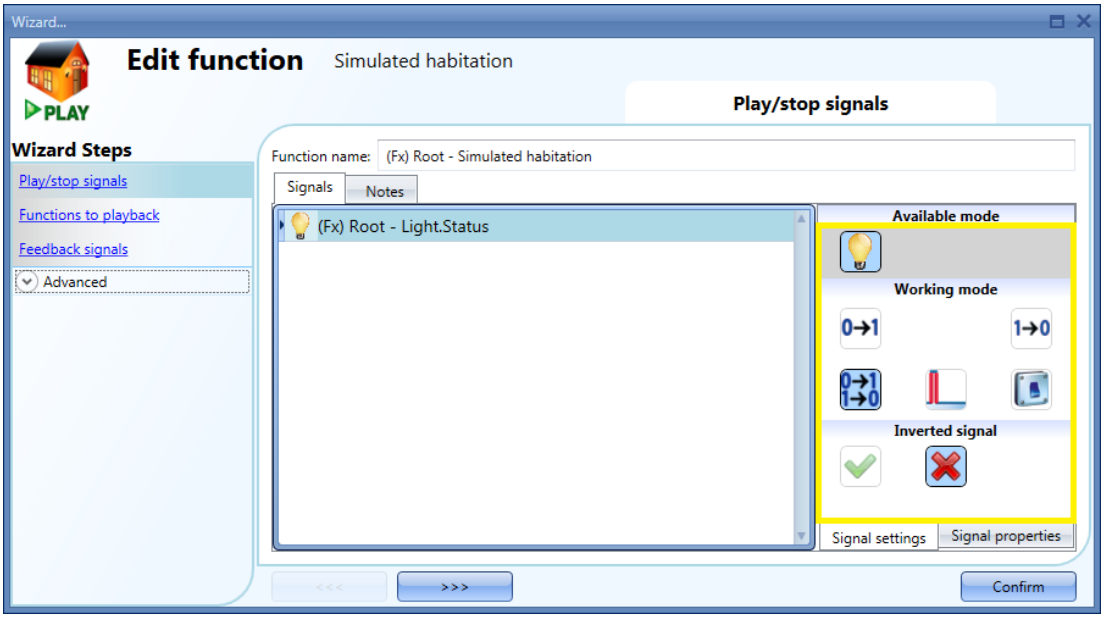
When the signal is added, the user should select the working mode and the type of the event required:



If a push button type is selected, the way of working has to be selected according to the table below.

	Event type			
Working mode				
1→0	As soon as the push button is pressed, the function is played.	After a short press (less than 1 second), the function is played at the release of the push button.	After a long press, the function is played at the release of the push button.	After a very long press, the function is played at the release of the push button.
0→1	As soon as the push button is pressed, the function is stopped.	After a short press (less than 1 second), the function is stopped at the release of the push button.	After a long press, the function is stopped at the release of the push button.	After a very long press, the function is stopped at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the function will be played/stopped in toggle mode.	After a short press (less than 1 second), the function will be played/stopped in toggle mode at the release of the push button.	After a long press, the function will be played/stopped in toggle mode at the release of the push button.	After a very long press, the function will be played/stopped in toggle mode at the release of the push button.
	The function will be played/stopped in toggle mode when the push button is pressed and toggled again when the push button is released.			

In the Signals window the user may select a function status as a input signal from the list (see picture below). When the function status is added, the user should select the working mode as shown in the picture below:



If a function status or a switch signal is used, the setting of the working mode has to be carried out following the table below:

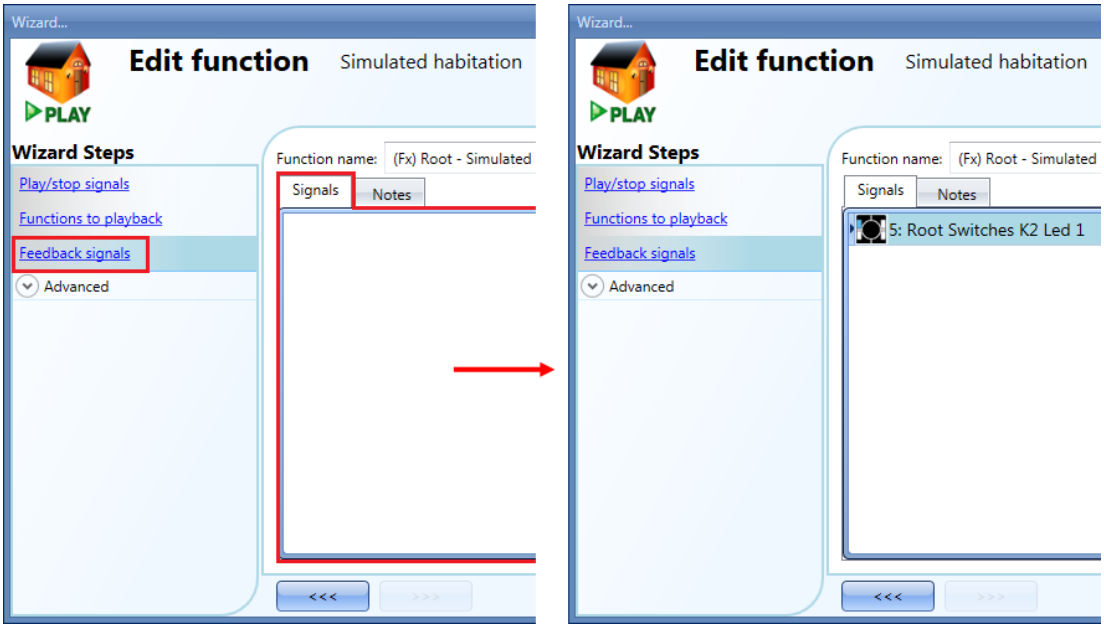
	Event type	
Working mode	Rising Edge	Falling Edge
0→1	The simulated habitation is played on rising edge of the input signal	No action
1→0	No action	The simulated habitation is stopped on falling edge of the input signal
0→1 1→0	The simulated habitation is played on the first rising edge of the input signal, the simulated habitation is stopped on the second rising edge of the input signal	No action
	The simulated habitation is played on rising edge of the input signal. The simulated habitation is stopped on falling edge of the input signal	
	The status of the simulated habitation (play/stop) is changed at every change of the input signal	

The user can also choose to invert the signal by selecting the green *V* below *Inverted signal*, in the Signal Setting tab window. If *Inverted* is enabled, the following table applies:

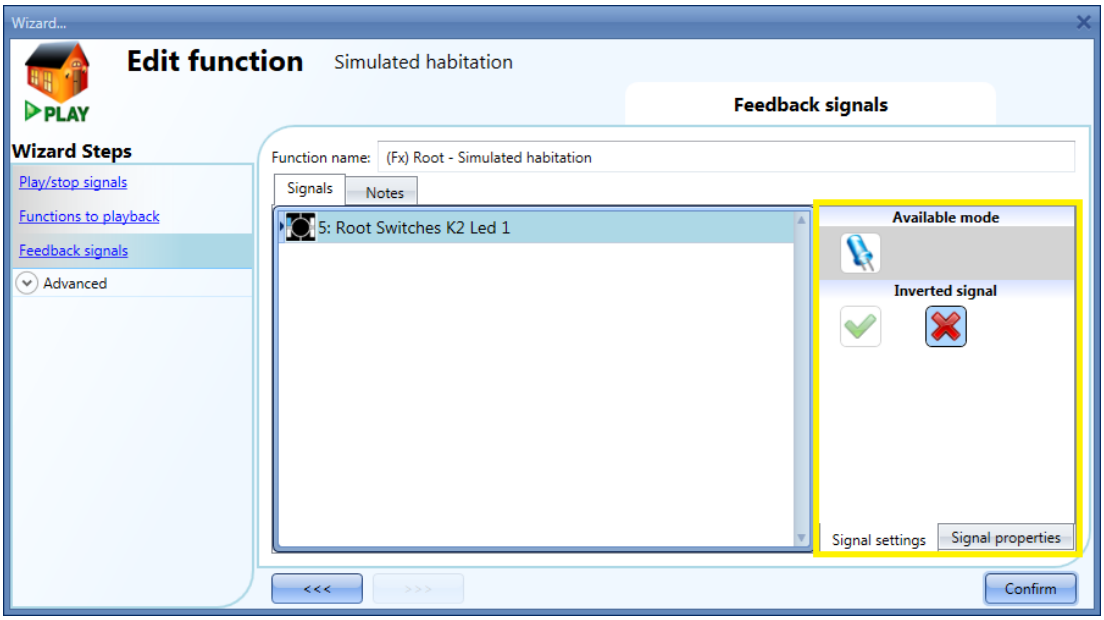
	Event type	
Working mode	Rising Edge	Falling Edge
0→1	No action	The simulated habitation is played on falling edge of the input signal
1→0	The simulated habitation is stopped on rising edge of the input signal	No action
0→1 1→0	No action	The simulated habitation is played on the first falling edge of the input signal, the simulated habitation is stopped on the second falling edge of the input signal
	The simulated habitation is stopped on rising edge of the input signal. The simulated habitation is played on falling edge of the input signal	
	The status of the simulated habitation (play/stop) is changed at every change of the input signal	

10.25.3 How to add the feedback signals

If the user wants a feedback signal to check the status of the simulated habitation function, he has to select the relevant section, then double click on the *Signals* window and then add the input signal from the list of those available (see picture below).
The feedback signal can be a relay, a LED or a buzzer, and it behaves according to the status of the function: when the function is in *play* mode, the feedback is on; when the function is in *stop* mode, the feedback is off.

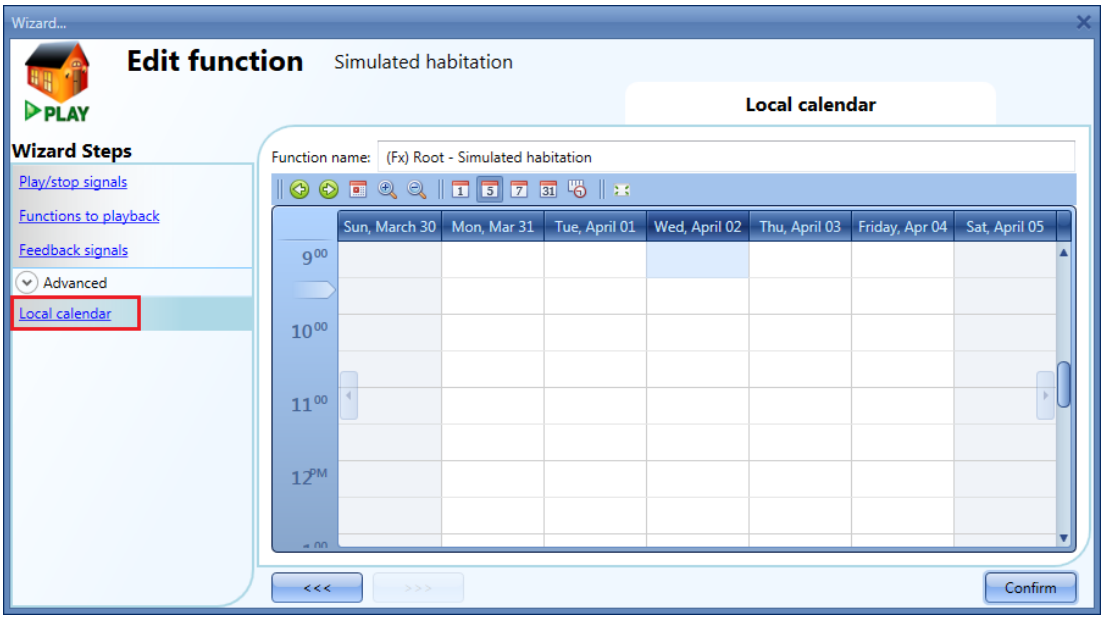


Once the feedback signal is entered, the user can also choose to use the inverted logic (see yellow rectangle in the picture below). The inverted logic is enabled by selecting the green V.



10.25.4 How to play/stop the function using a calendar

The user can play/stop the simulated function by setting the calendar inside the function.
To enable it, select *Local calendar* in the *Advanced* section (see picture below).

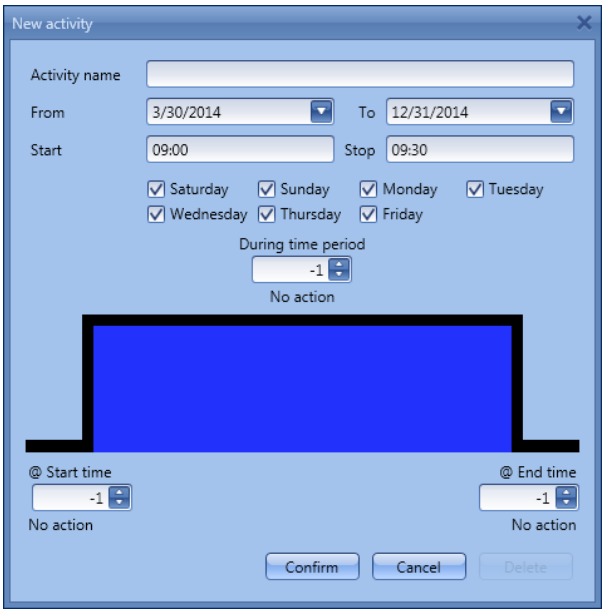


The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as seen below:

Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day.
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

Calendar activities
Once the user has selected his preferred visualization, to enter a time band he should double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From*:* the start date for the calendar activity.

To*:* the end date of the calendar activity.

Start*:* The time of the start of the activity.

Stop: The time of the end of the activity.

@ ***start time***: in this field the user can select the action of the function at start time.
At the start time the actions that the user can choose are:

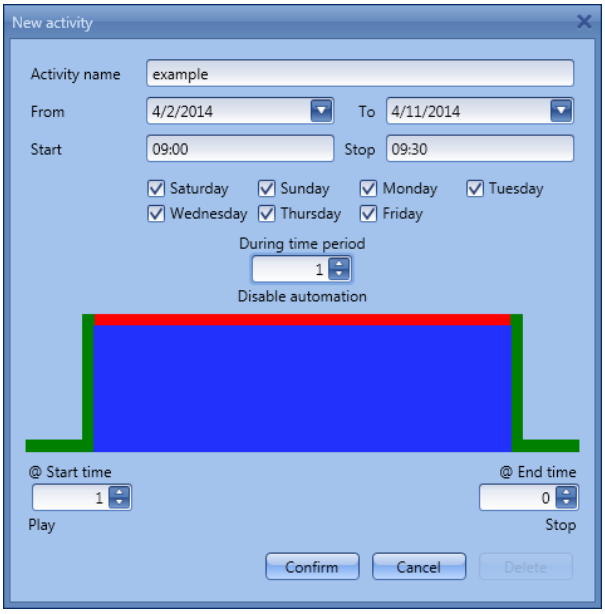
- No action (-1)
- Stop the simulation (0)
- Play the simulation (1)

@ ***end time***: in this field the user can select the action of the function at end time.
At the stop time the actions that the user can choose are:

- No action (-1)
- Stop the simulation (0)
- Play the simulation (1)

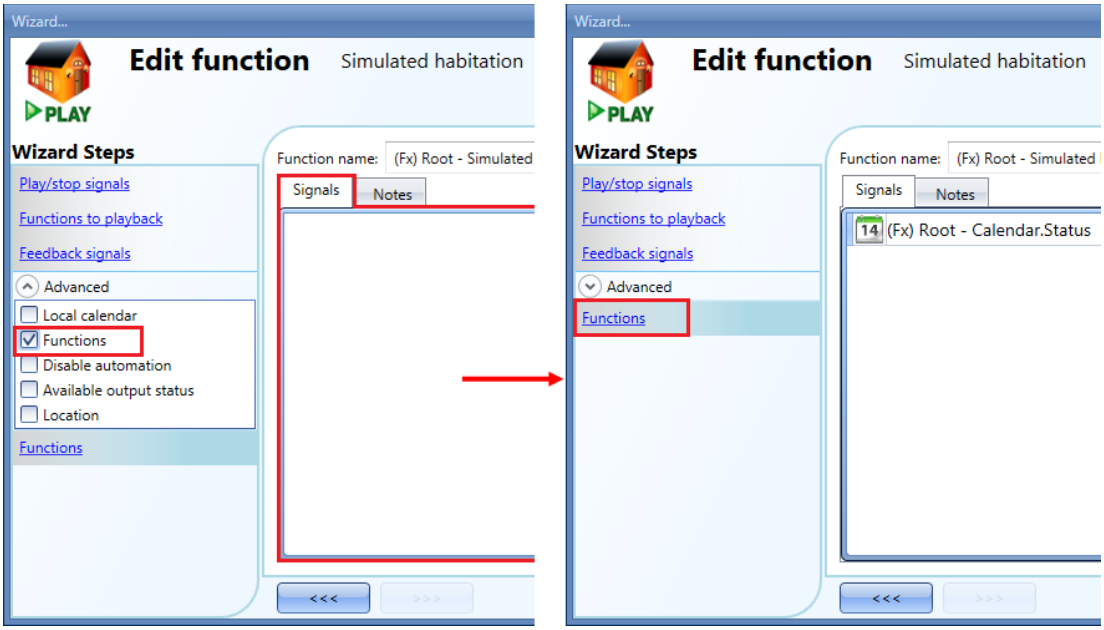
Days: The user should select the days when the calendar has to work.

In the example below, the calendar is set to play the simulation at 09:00 on 2nd April.



10.25.5 How to play/stop Simulated habitation using a Function

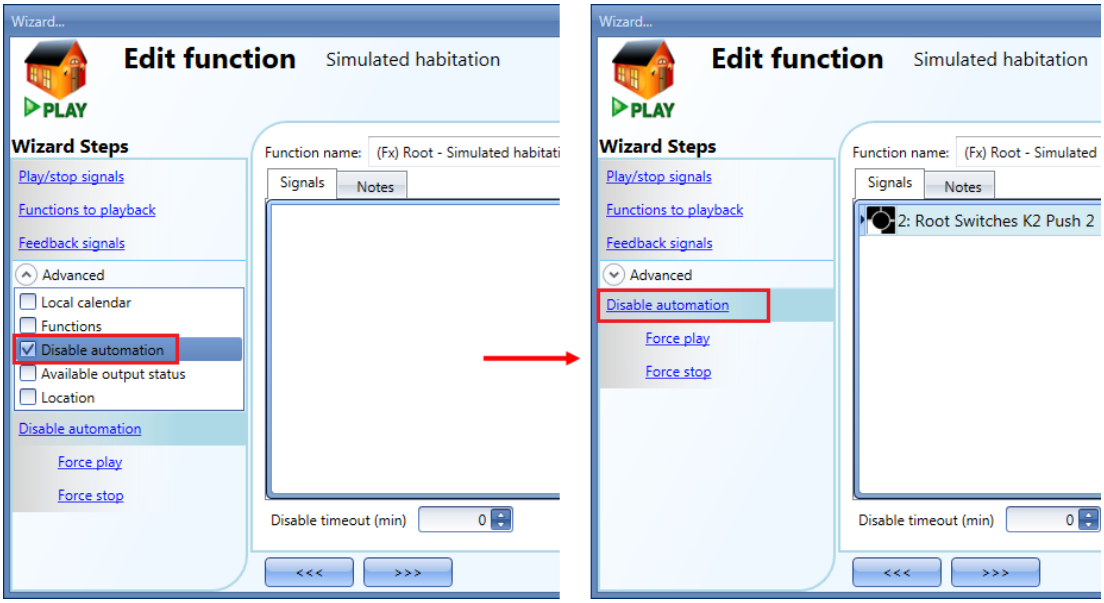
The user can play/stop the simulated habitation by means of a Function such as the *Global calendar*, the *Light function*, etc…To use it, select *Functions* in the *Advanced* section; then double click on the *Signals* window and select the appropriate function to be used (see picture below):



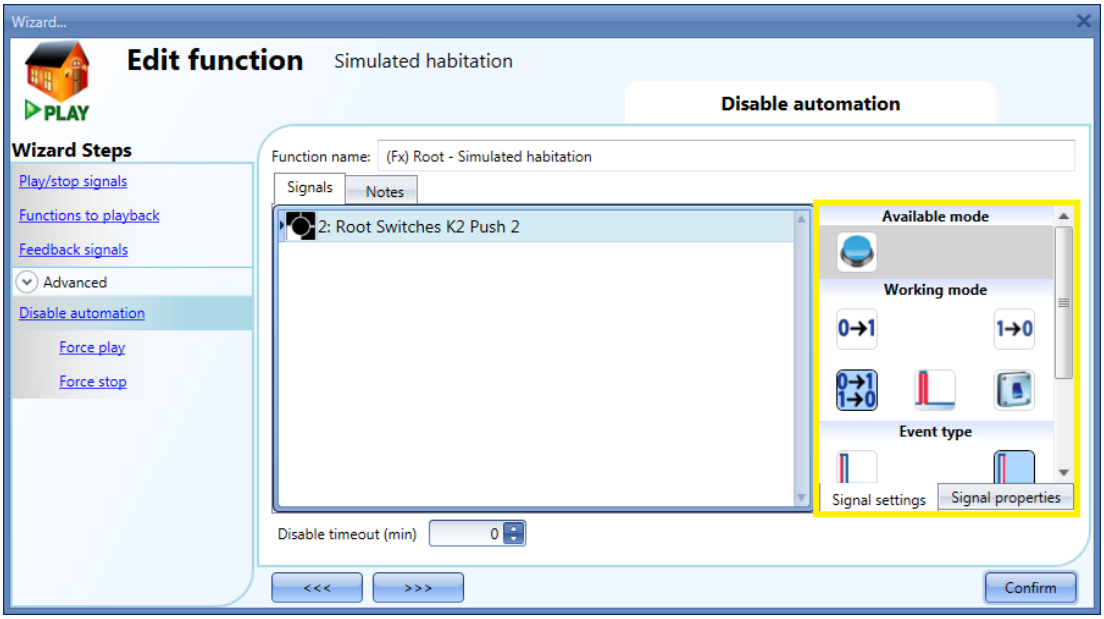
The playback of the simulated habitation acts according to the function status: when the function is active, the simulation is played; when the function is not active, the simulation is stopped.

10.25.6 How to disable the calendar automation

If the user wants to disable the automation from the internal calendar, the *Disable automation* field has to be used: to enable it, select *Disable automation* in the *Advanced* section, then double click on the signal window and select the appropriate signal to use (see picture below).



When the signal is added, the user should select the working mode and the type of the event required:



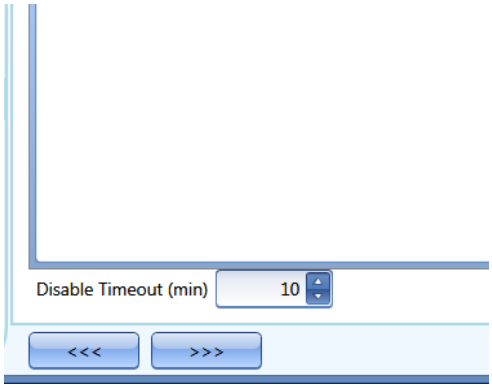
If a push button type is added, the working mode, highlighted with the yellow rectangle, has to be selected according to the table below:

Working mode	Event type			
0→1 Switch on event	As soon as the push button is pressed, all automations are disabled.	After a short press (less than 1 second), at the release of the push button all automations are disabled.	After a long press, all automations are disabled at the release of the push button.	After a very long press, all automations are disabled at the release of the push button.
1→0 Switch off event	As soon as the push button is pressed, the automations are enabled again.	After a short press (less than 1 second), the automations are enabled again at the release of the push button.	After a long press, the automations are enabled again at the release of the push button.	After a very long press, the automations are enabled again at the release of the push button.
0→1 1→0 Event toggle	As soon as the push button is pressed, the automations will be disabled/enabled in toggle mode.	After a short press (less than 1 second), the automations will be disabled/enabled in toggle mode at the release of the push button.	After a long press, the automations will be disabled/enabled in toggle mode at the release of the push button.	After a very long press, the automations will be disabled/enabled in toggle mode at the release of the push button.
Level signal	The automations will be disabled/enabled in toggle mode when the push button is pressed and toggled again when the push button is released.			

If a switch signal is used, the setting of the working mode has to be carried out following the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1 Switch on event	The automations are disabled	No action
1→0 Switch off event	The automations are enabled	No action
0→1 1→0 Event toggle	The automations are disabled/enabled in toggle mode	No action
Level signal	The automations are disabled/enabled in toggle mode	The automations are disabled/enabled in toggle mode

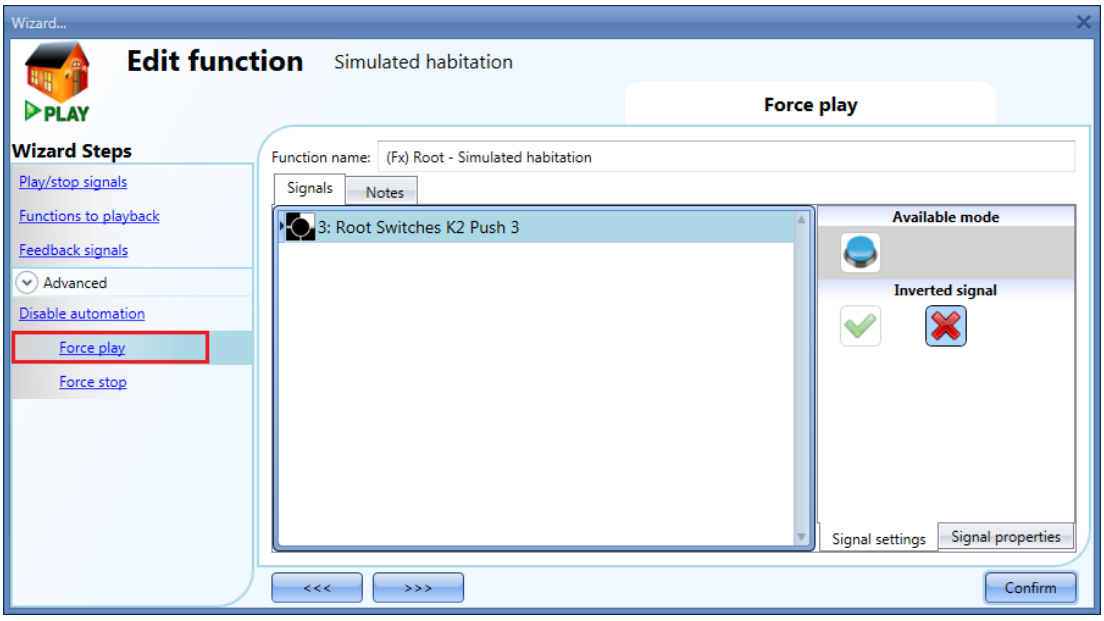
If the user needs to be sure that the disabling of the automation is automatically removed, the *disable automation timer* has to be set.



The timer starts every time the reset status becomes activated. The disable status is automatically deactivated as soon as the time has elapsed.
The timer starts each time the disable status becomes activated: the disable status is automatically deactivated as soon as the time has elapsed.
In the figure above, a disable timeout of 10 minutes is set. The maximum delay is 59minutes.

10.25.7 How to force the function to play

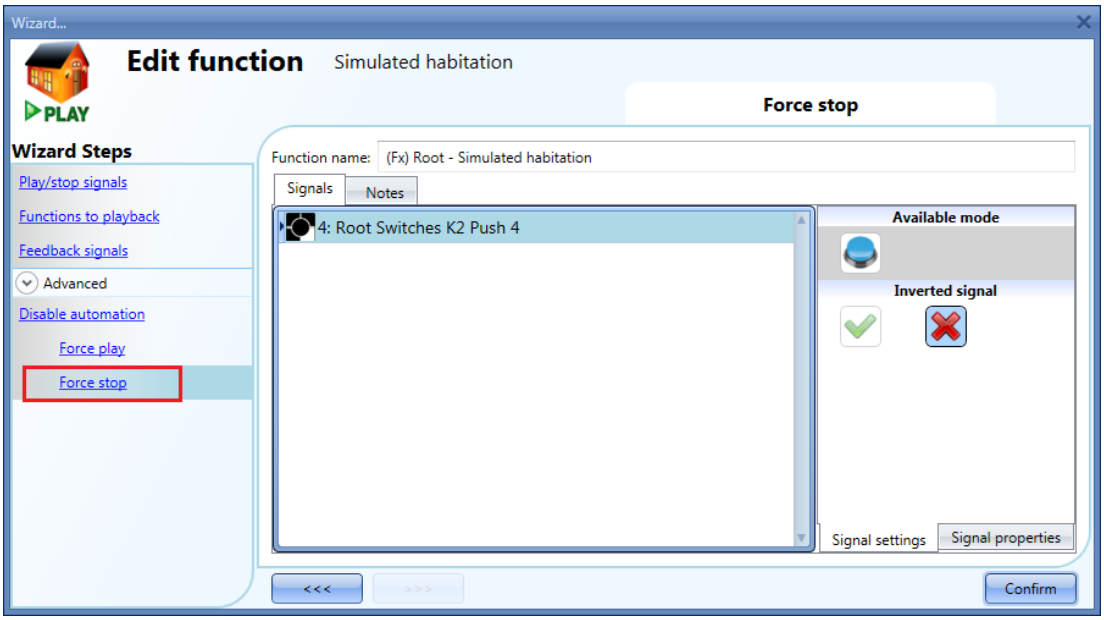
If the user wants to force the playback of the simulated habitation regardless of all other signals used in the function, the *Force play* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force play*, then double click on the signal window and select the appropriate signal to use (see picture below).



Each signal used in the *Force play* window works as a level signal. Until the signal is active, the function is forced to play.
When the *Force play* signal is activated and at the same time the *Force stop* signal is activated, the *Force play* signal has priority.

10.25.8 How to force the function stop

If the user wants to force the stop of the simulated habitation regardless of all other signals used in the function, the *Force stop* step has to be used: to enable it, select *Disable automation* in the *Advanced* section, select *Force stop*, then double click on the signal window and select the appropriate signal to use (see picture below).



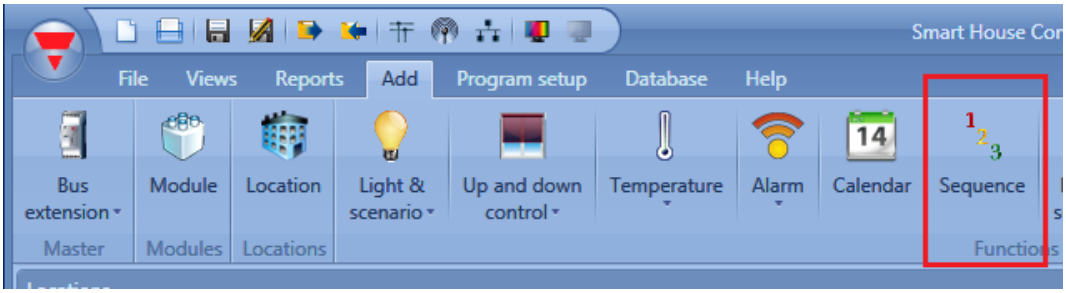
Each signal used in the *Force stop* window works as a level signal. Until the signal is active, the function is forced to stop (no simulation active).
When the *Force play* signal is activated and at the same time the *Force stop* signal is activated, the *Force play* signal has priority.

10.26 How to set the sequence function

The sequence function allows the user to put together the functions already created and activate/deactivate them with just one click.
All the selected functions are activated according to a certain time and order. The sequence starts activating the first function in the list and goes on to activate the others following the predefined order, until the last function in the list is executed.

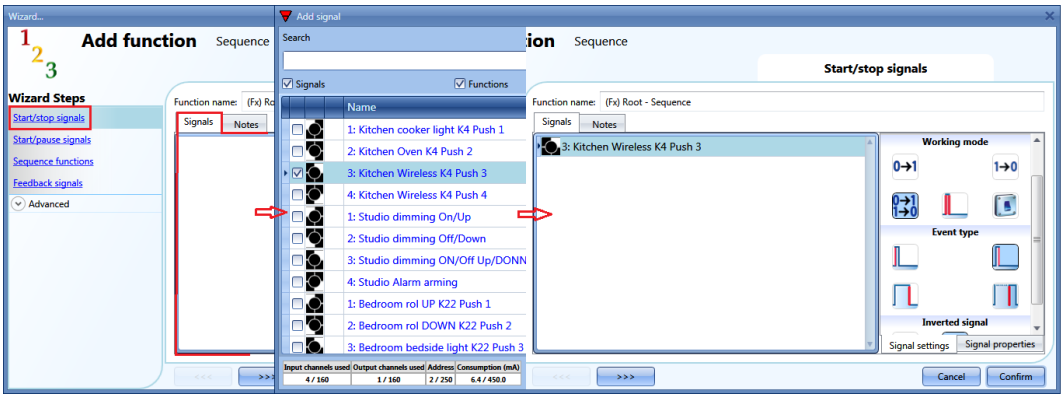
The functions that can be controlled are: lights, roller blinds and windows, intruder alarm, sirens, timers and zone temperature functions.

To add a sequence function the user should select *Sequence* from the Add menu (see picture below).
The new function will be added in the selected location.
The wizard relevant to a Sequence function can also be opened by typing Alt+A+S



10.26.1 How to set the start/stop signals

If the user wants to add the start/stop signals, he has to select the relevant field, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



The input signal can be used either to start or to stop the sequence. Every time the sequence is started, it restarts from the beginning.

If a push button type is added, the working mode should be selected according to the table below:

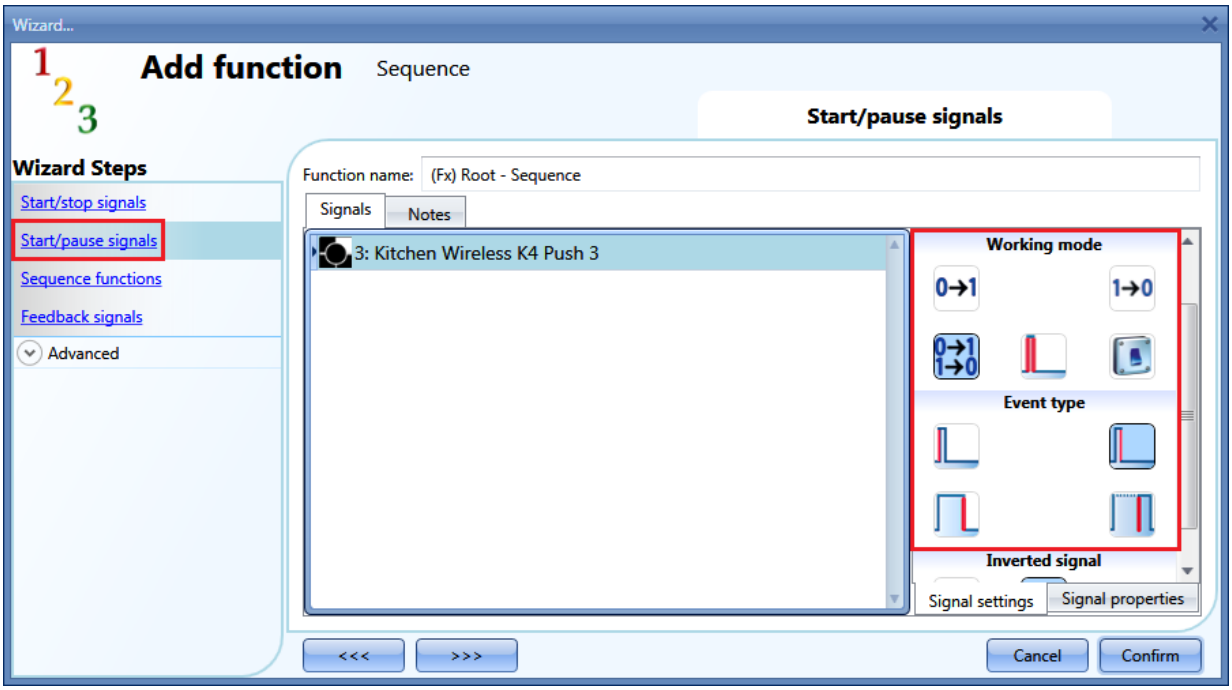
Working mode	Event type			
	Event when push button pressed	Event when push button released	Event with long pressure	Event with very long pressure
0→1	As soon as the push button is pressed, the sequence is started.	After a short press (less than 1 second), the sequence is started at the release of the push button.	After a long press, the sequence is started at the release of the push button.	After a very long press, the sequence is started at the release of the push button.
1→0	As soon as the push button is pressed, the sequence is stopped.	After a short press (less than 1 second), the sequence is stopped at the release of the push button.	After a long press, the sequence is stopped at the release of the push button.	After a very long press, the sequence is stopped at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the sequence will be started/stopped in toggle mode.	After a short press (less than 1 second), the sequence will be started/stopped in toggle mode at the release of the push button.	After a long press, the sequence will be started/stopped in toggle mode at the release of the push button.	After a very long press, the sequence will be started/stopped in toggle mode at the release of the push button.
	The sequence will be started/stopped in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The sequence will be started when the signal goes ON, and it will be stopped when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The sequence is started	No action
1→0	The sequence is stopped	No action
0→1 1→0	The sequence is started/stopped in toggle mode	No action
	The sequence is started/stopped in toggle mode	The sequence is started/stopped in toggle mode
	The sequence is started	The sequence is stopped

10.26.2 How to set the start/pause signals

If the user wants to add the start/pause signals, he has to select the relevant field, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



The input signal can be used either to start or to pause the sequence. Every time the sequence is started, it starts from where it was paused.

If a push button type is added, the working mode should be selected according to the table below:

Working mode	Event type			
	Event when push button pressed	Event when push button released	Event with long pressure	Event with very long pressure
0→1	As soon as the push button is pressed, the sequence is started.	After a short press (less than 1 second), the sequence is started at the release of the push button.	After a long press, the sequence is started at the release of the push button.	After a very long press, the sequence is started at the release of the push button.
1→0	As soon as the push button is pressed, the sequence is paused.	After a short press (less than 1 second), the sequence is paused at the release of the push button.	After a long press, the sequence is paused at the release of the push button.	After a very long press, the sequence is paused at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the sequence will be started/paused in toggle mode.	After a short press (less than 1 second), the sequence will be started/ paused in toggle mode at the release of the push button.	After a long press, the sequence will be started/ paused in toggle mode at the release of the push button.	After a very long press, the sequence will be started/ paused in toggle mode at the release of the push button.

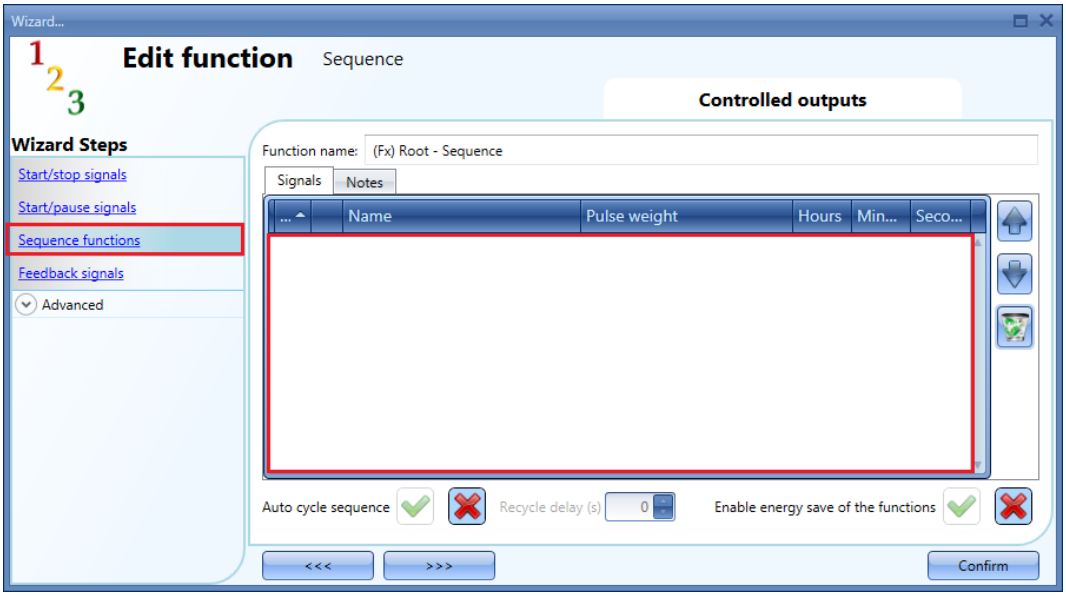
	The sequence will be started/ paused in toggle mode when the push button is pressed and toggled again when the push button is released.
	The sequence will be started when the signal goes ON, and it will be paused when the signal goes OFF.

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The sequence is started	No action
1→0	The sequence is paused	No action
0→1 1→0	The sequence is started/paused in toggle mode	No action
	The sequence is started/paused in toggle mode	The sequence is started/paused in toggle mode
	The sequence is started	The sequence is paused

10.26.3 How to add a function to the sequence

If the user wants to add a function to the sequence, he should select the relevant field, then double click on the *Signals* window to enter the list of available functions (see picture below).



Auto cycle sequence: if this option is enabled, the sequence is repeated continuously: each time the last function is executed, the sequence starts again from the beginning. All the functions are executed in a loop and the sequence can be stopped by using the stop signal(s).

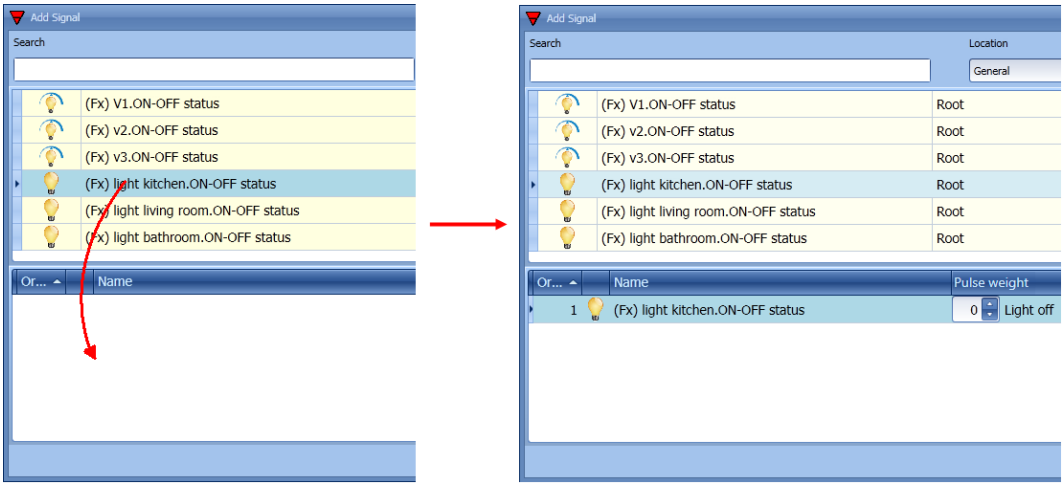
Recycle delay (s): It is the delay (in seconds) between two repetitions of the sequence if the recycling option is enabled.

Enable energy save of the functions: if this option is enabled, if a light function with energy save timer is started by the sequence, it will be switched off as soon as the relevant timer is expired. If this option is not enabled, the energy save timer is not used in any function.

Once the functions list is opened, the user should select the functions he wants to add.

The functions can be added using *drag-and-drop* according to the following steps:

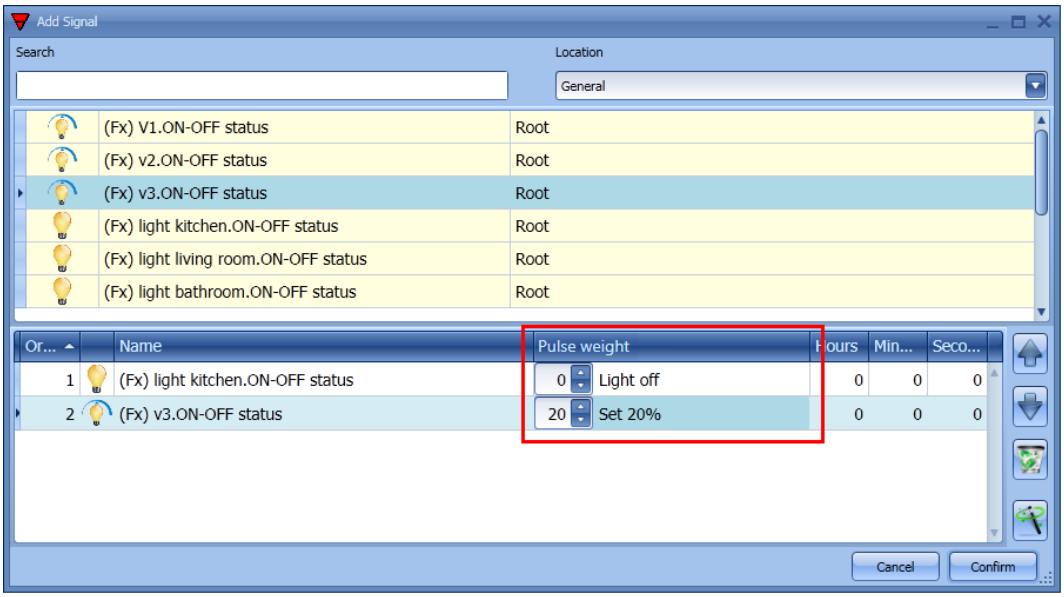
1. Move the mouse pointer until it is directly over the function you want to add.
2. Hold down the left mouse button.
3. With the left mouse button held down, move the pointer to where you want to drop the object (in the bottom window).
4. Drop the object by releasing the left mouse button.
5. The function is now added to the sequence (see picture below).



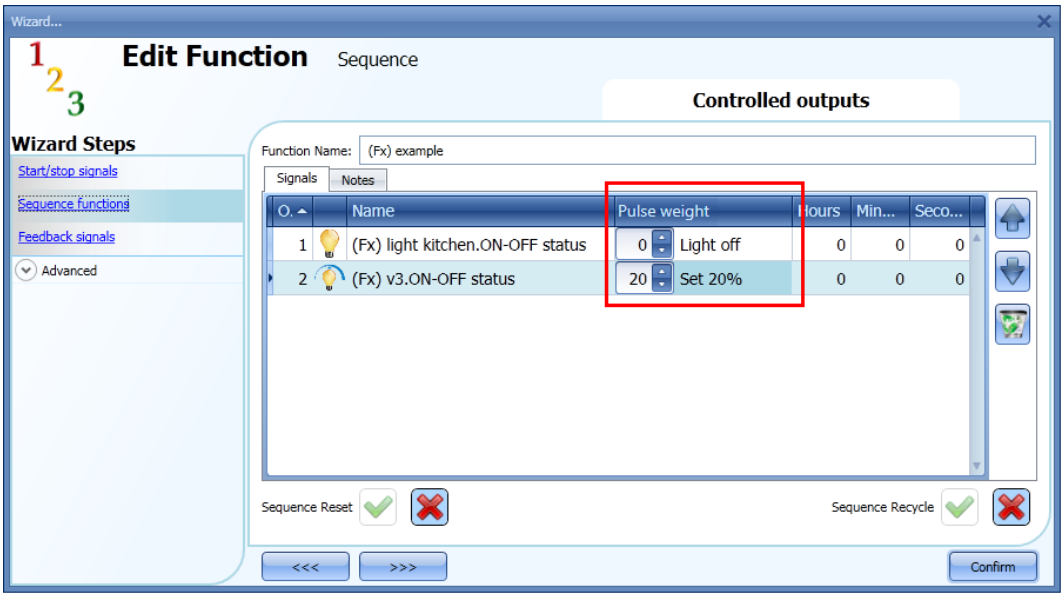
Another way to add a function to the sequence is by double clicking the line of the function.

10.26.4 How to set the action for the functions in the sequence

When a function is added to the sequence, the user can modify the action to be performed. The action of each function can be changed using the spin-box (see picture below).



The user can also change the action of the function in the sequence functions window.

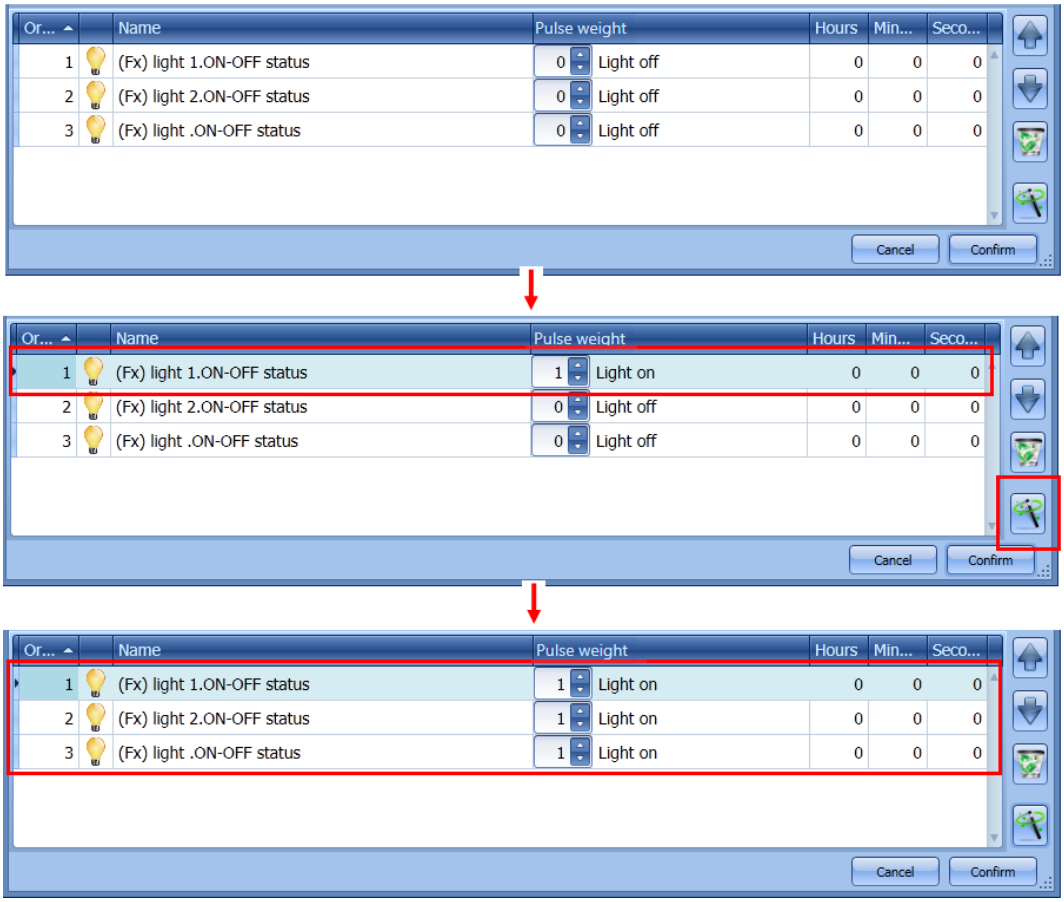


In the table below, all the possible actions are shown according to the type of function:

Function	Available actions
Light on/off	- no action - light ON - light OFF
Dimmable light	- no action - light OFF - set Scenario 1 - set Scenario 2 - set Scenario 3 - set Scenario 4 - set Scenario 5 - set % (from 6 to 100)
Main intruder alarm	- no action - arm intruder alarm - disarm intruder alarm
Rollerblind	- no action - set position (from 1 to 100) - set tilting position (from 1 to 100)
Window	- no action - set position (from 1 to 100)
Zone temperature	- no action - set T1 - set T2 - set T3 - set off
Siren	- no action - trigger ON - trigger OFF
Interval timer	- no action - trigger ON - trigger OFF

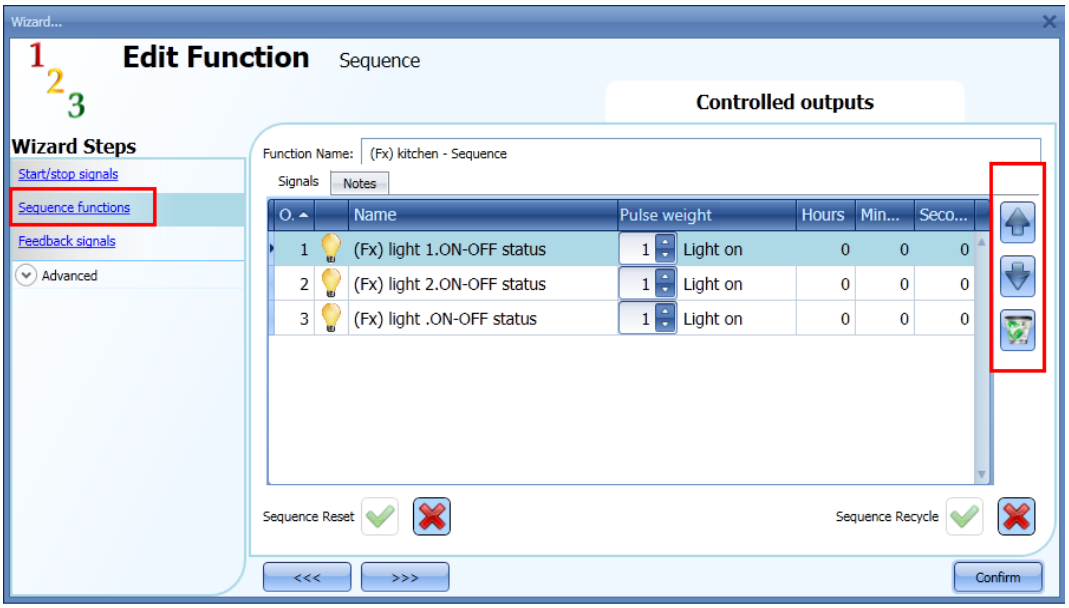
The *Apply to all* icon allows the user to set the same actions for all the functions of the same type with a simple click. This feature is useful to configure a sequence with a number of functions that perform the same action; an example could be a “welcome” sequence, which is used to switch all lights on when the user arrives home.
In the picture below an example of how to use this option is shown.

First, all the required functions have to be added; secondly, the required action has to be selected in one function; thirdly, by clicking on *Apply to all,* the action will be configured in all the functions of the same type.

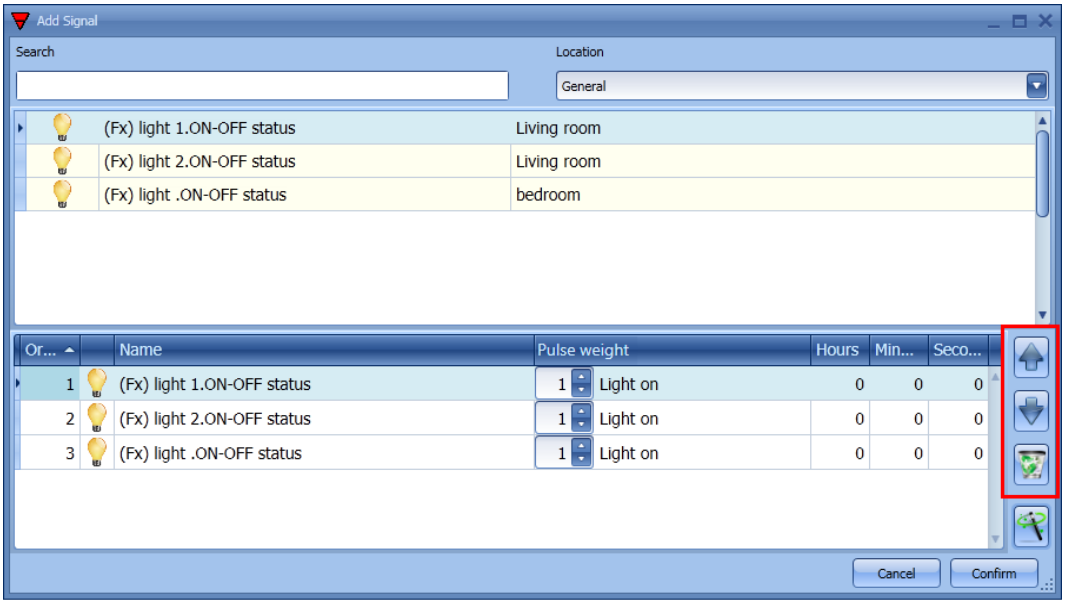


10.26.5 How to modify the sequence

All the functions added to the sequence can be changed at any time. The user can do this in the main window of the sequence function (as shown below).



Or in the window where the functions are added to the sequence.



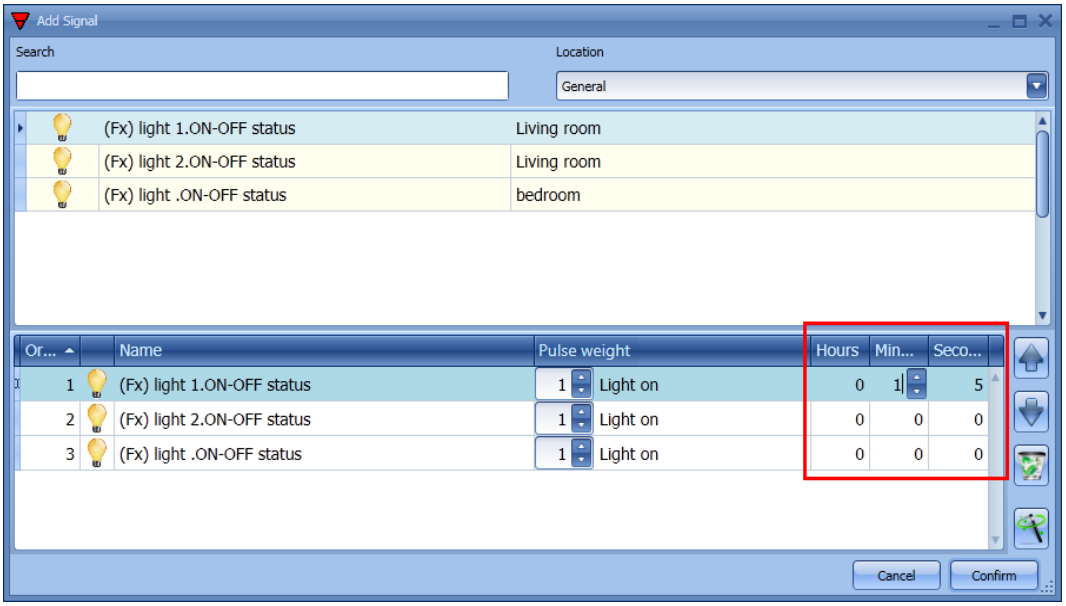
Using the icons on the right side of the window, the user can move up/down or delete a function from the list.

Table of actions:

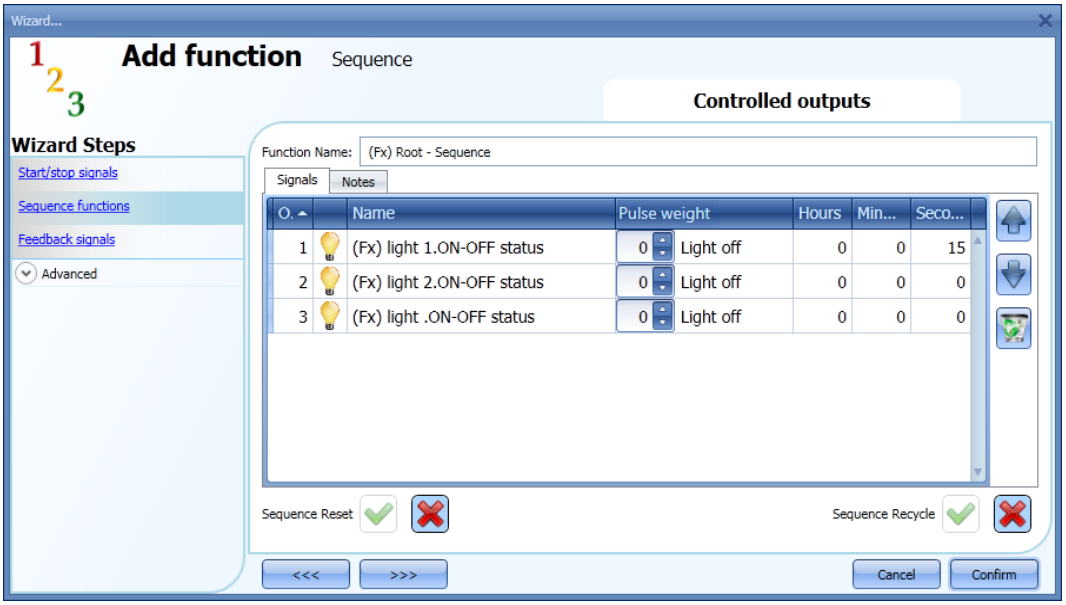
	By clicking on this icon the selected function will be moved upwards
	By clicking on this icon the selected function will be moved downwards
	By clicking on this icon the selected function will be deleted

10.26.6 How set the delay time between the activation of two functions

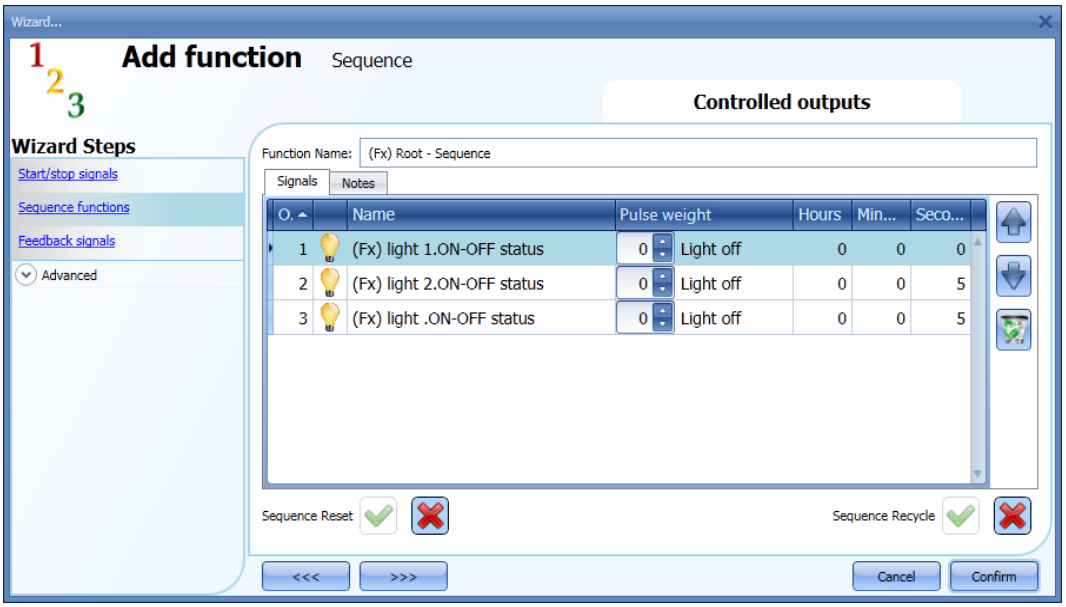
The delay between the activation of two functions can be set with a double click in the field *Hours, Minute, Seconds,* as highlighted by the red rectangle.
If the time value is set to 0, the function is activated without any delay in respect to the previous function. This means that the two functions will be managed at the same time.
In the first function of the list, this time is the delay between the activation of the start command and the activation of the first action.



Example 1:
After 15 seconds from the activation of the start signal all the lights (lights 1-2-3) are switched off at the same time and the sequence ends.

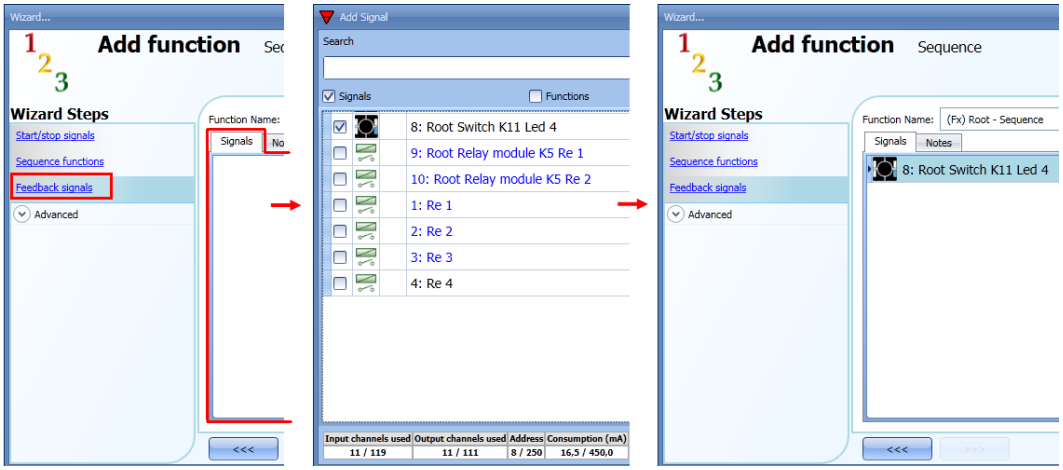


Example 2:
The first light function is switched on as soon as the start signal is activated. After 5 seconds light two is switched on and after another 5 seconds light 3 is switched on, then the sequence ends.

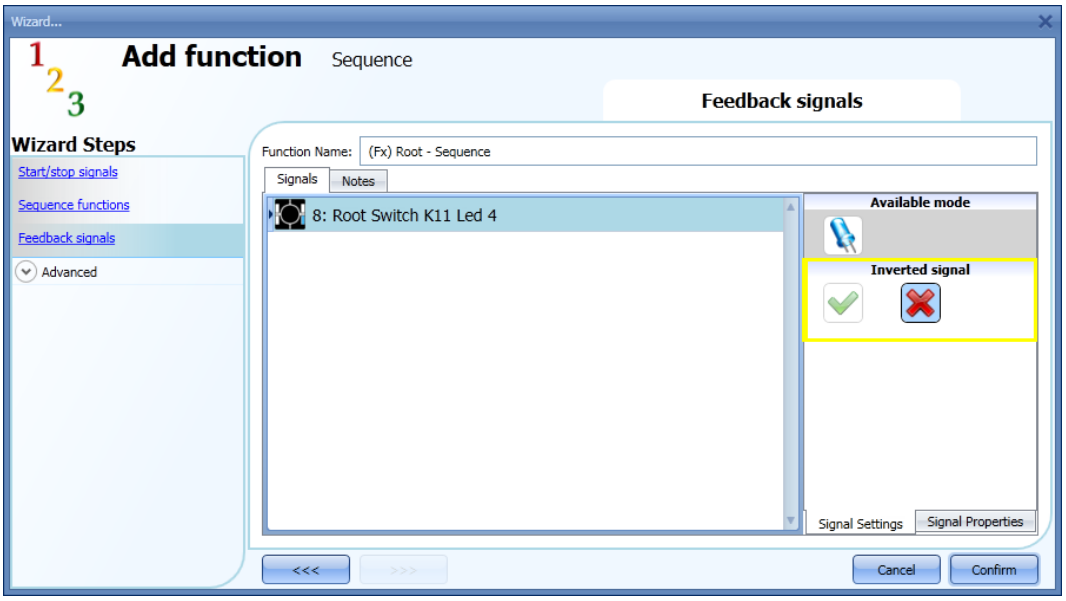


10.26.7 How to add the feedback signals

If the user wants to use a feedback signal to check the status of the sequence function, he has to select the relevant section, then double click on the *Signals* window and select the feedback signal from the list of those available (see picture below).
The feedback signal can be a relay, a LED or a buzzer, and it behaves according to the status of the function: when the sequence is in progress the feedback signal is on; when the sequence is not running the feedback is off.

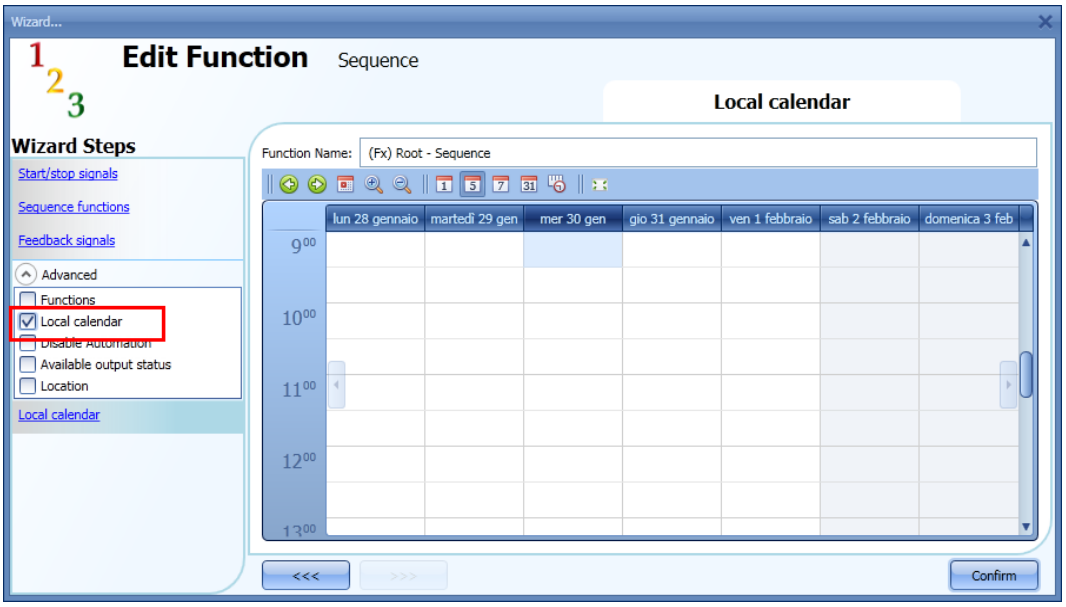


Once the feedback signal is entered, the user can also select to use the inverted logic.



10.26.8 How to start/stop the sequence using a calendar

If the user wants to start/stop the sequence using an automation with a calendar, the relevant menu has to be enabled in the *Local calendar* field (see picture below).

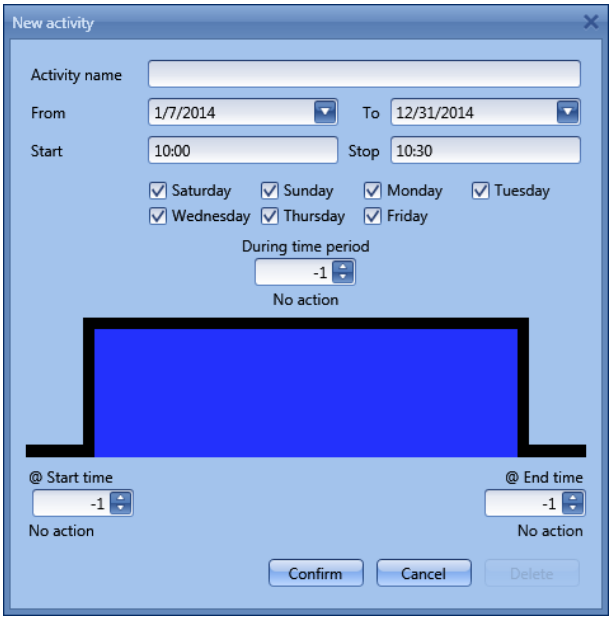


The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally.
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

Calendar activities
Once the user has selected his preferred visualization, to enter a time band he should double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From*:* the start date for the calendar activity.

To*:* the end date of the calendar activity.

Start*:* The time of the start of the activity.

Stop: The time of the end of the activity.

@ ***start time***: in this field the user can select the action of the function at start time.
At the start time the actions that the user can choose are:

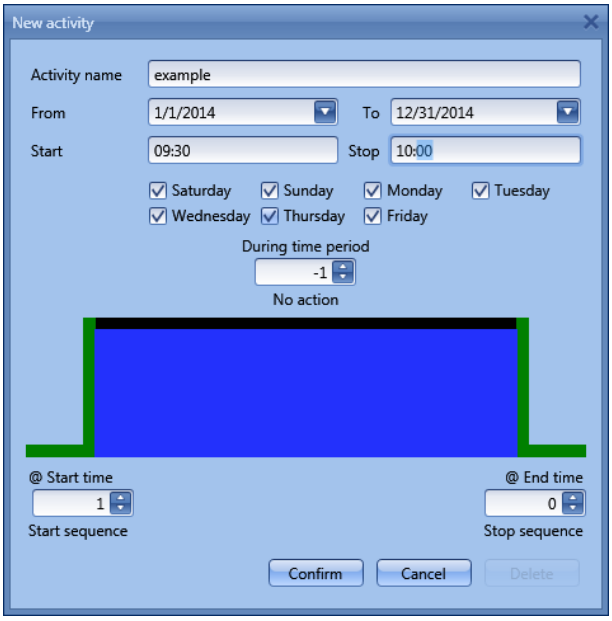
- No action (-1)
- Stop (0): the sequence is halted
- Start (1): the sequence is started

@ ***end time***: in this field the user can select the action of the function at stop time.
At the stop time the actions that the user can choose are:

- No action (-1)
- Stop (0): the sequence is halted
- Start (1): the sequence is started

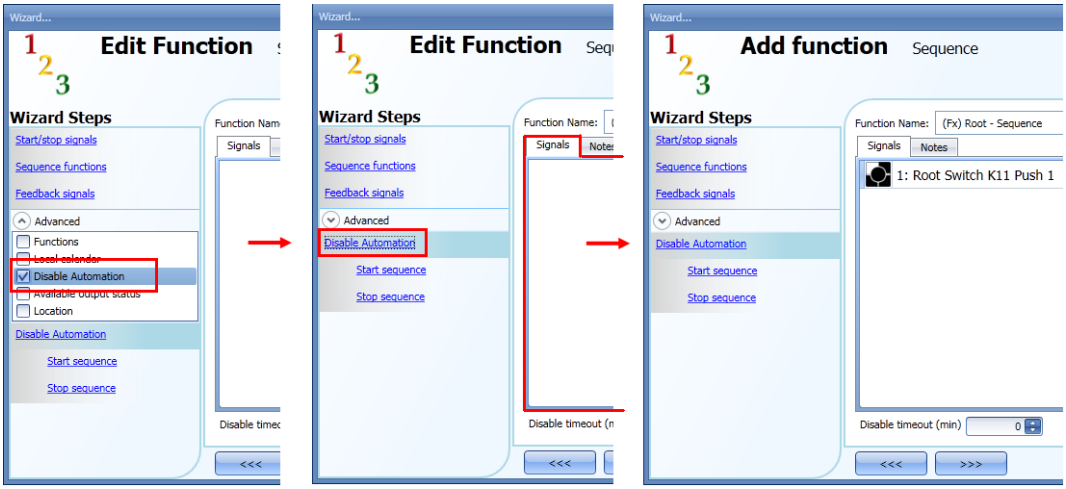
Days: The user should select the days when the calendar has to operate.

In the example below the calendar is set to start a sequence at 9: 30 and stop it at 10 o'clock.

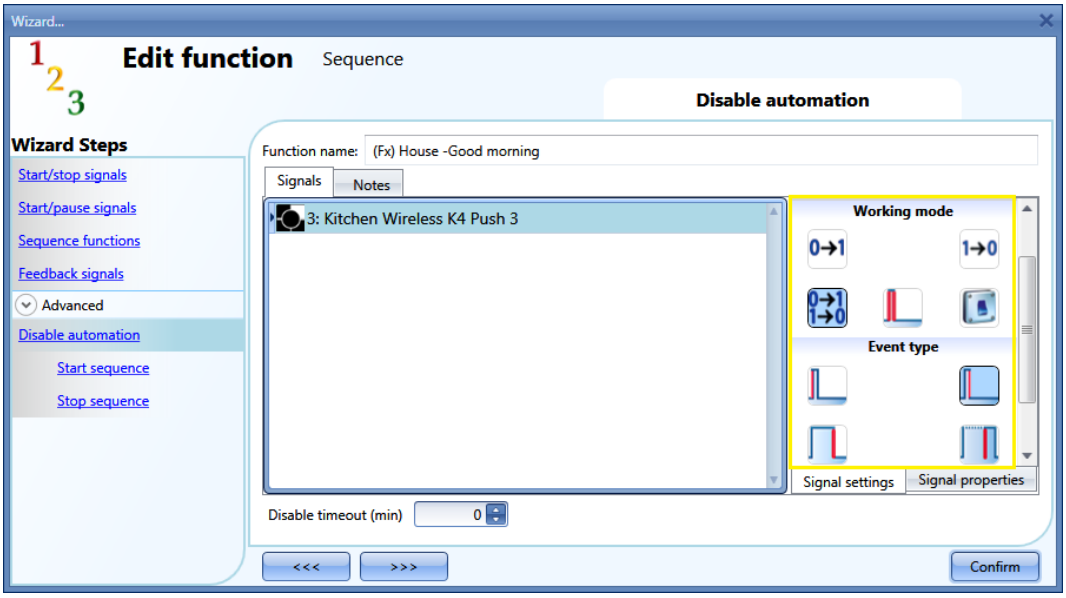


10.26.9 How to disable the calendar automation

If the user wants to disable the automation managed by the internal calendar, the *Disable automation* field has to be used: when the disabling is active, the sequence function can only be activated/deactivated by using manual start/stop commands.
To enable it, select *Disable automations* in the *Advanced* section, then double click on the signal window and select the signal to use (see picture below).



When the signal is added, the user has to select the working mode and the type of the event:



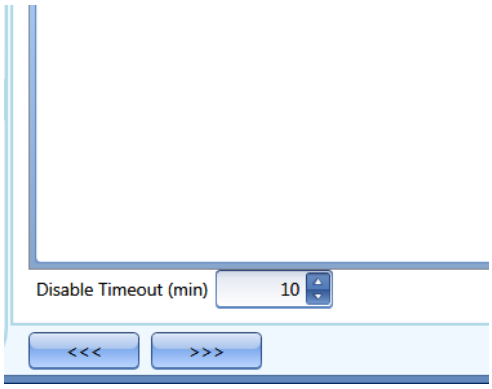
If a push button type is added, the working mode, highlighted by the yellow rectangle, has to be selected according to the table below:

Working mode	Event type			
	Event when push button pressed	Event when push button released	Event with long pressure	Event with very long pressure
0→1	As soon as the push button is pressed, all automations are disabled.	After a short press (less than 1 second), at the release of the push button all automations are disabled.	After a long press, all automations are disabled at the release of the push button.	After a very long press, all automations are disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automations are enabled again.	After a short press (less than 1 second), the automations are enabled again at the release of the push button.	After a long press, the automations are enabled again at the release of the push button.	After a very long press, the automations are enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the automations will be disabled/enabled in toggle mode.	After a short press (less than 1 second), the automations will be disabled/enabled in toggle mode at the release of the push button.	After a long press, the automations will be disabled/enabled in toggle mode at the release of the push button.	After a very long press, the automations will be disabled/enabled in toggle mode at the release of the push button.
	The automations will be disabled/enabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	Signal deactivated
0→1	The automations are disabled	No action
1→0	The automations are enabled	No action
0→1 1→0	The automations are disabled/enabled in toggle mode	No action
	The automations are disabled/enabled in toggle mode	The automations are disabled/enabled in toggle mode
	The automation is disabled	The automation is enabled

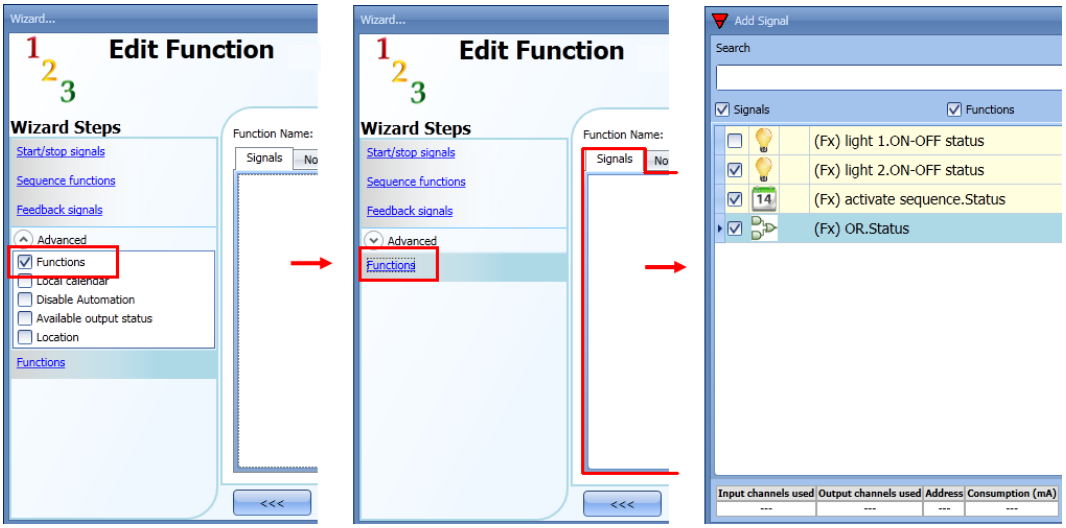
The user might need to set a time after which the calendar automation is enabled again, even if the selected signal is still active: to achieve this the *Disable timeout* field at the bottom of the window should be set.



The timer starts each time the disable status is activated: the disable status is automatically deactivated as soon as the time has elapsed.
In the figure above a disable timeout of 10 minutes is set. The maximum delay is 59 minutes.
If the timer is set to 0 minutes, the timer is disabled and the disable status will stay activated until a manual command is received.

10.26.10 How to use an external function to start/stop a sequence

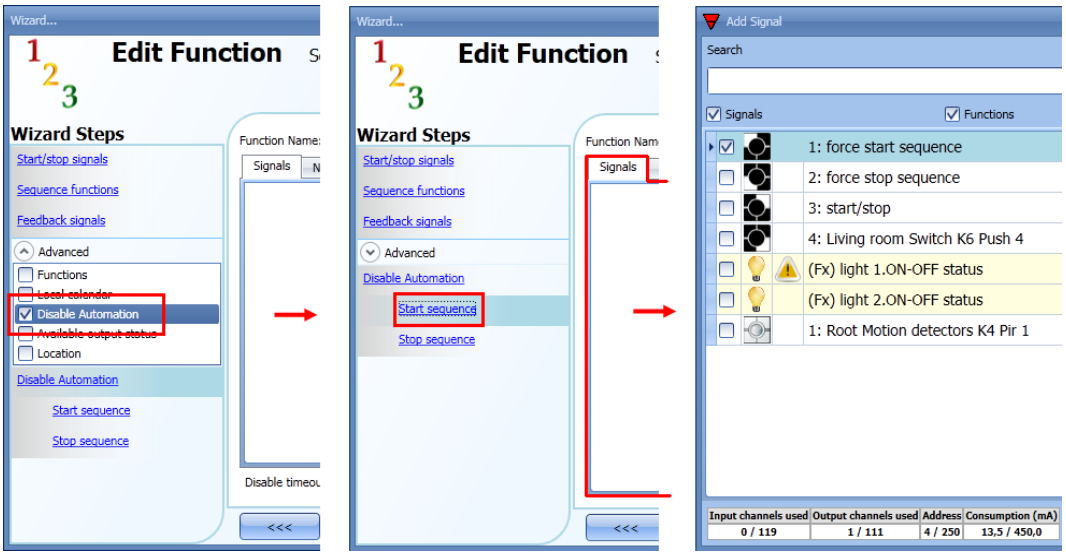
If the user wants to use an external function to start/stop a sequence, the *Function* field has to be used: to enable it, select *Functions* in the *Advanced* section, then double click on the signals window and select the relevant function from the list (see picture below).



All the added functions will trigger the sequence according to the status they have: for example, a light that is switched on will make the sequence start; the same function that is switched off will make the sequence stop.
The global calendar has to be added in this field and it works in the same way as the internal calendar.

10.26.11 How to force the sequence active

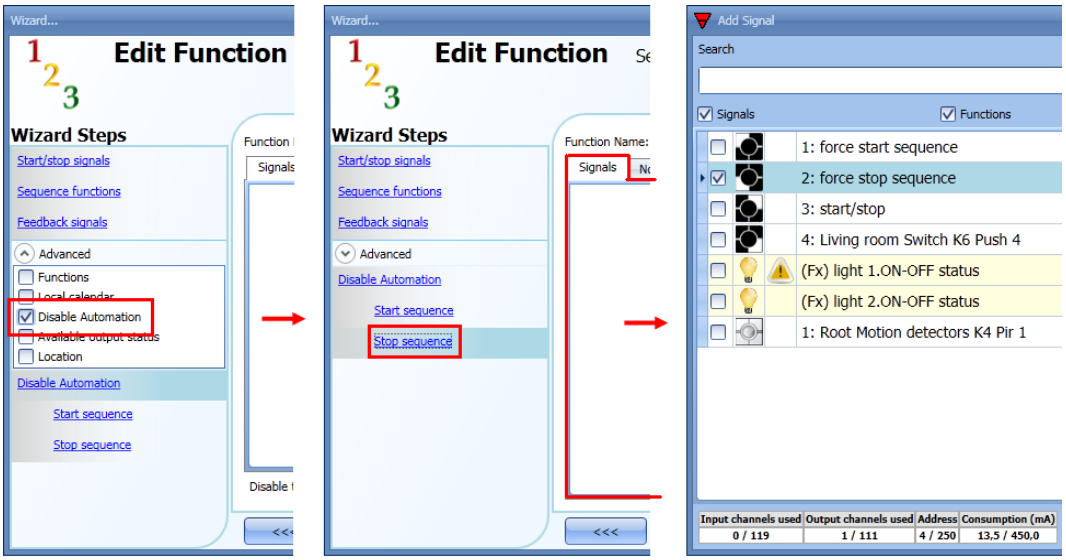
If the user wants to force the sequence active regardless of all the other signals used in the function, the *Start sequence* field has to be used: to enable it, select *Disable automations* in the *Advanced* section, select *Start sequence*, then double click on the signals window and select the signal to use (see picture below).



Each signal used in the *Start sequence* window works in level mode. Until the start signal is active, the function is executed and cannot be deactivated until the start signal disappears.
When the *Start sequence* signal is activated and at the same time the *Stop sequence* signal is activated, the start sequence signal has priority and the function stays activated.

10.26.12 How to force the sequence stop

If the user wants to force the sequence stop regardless of all the other signals used in the function, the *Stop sequence* field has to be used: to enable it, select *Disable automations* in the *Advanced* section, select *Stop sequence*, then double click on the signals window and select the signal to use (see picture below).



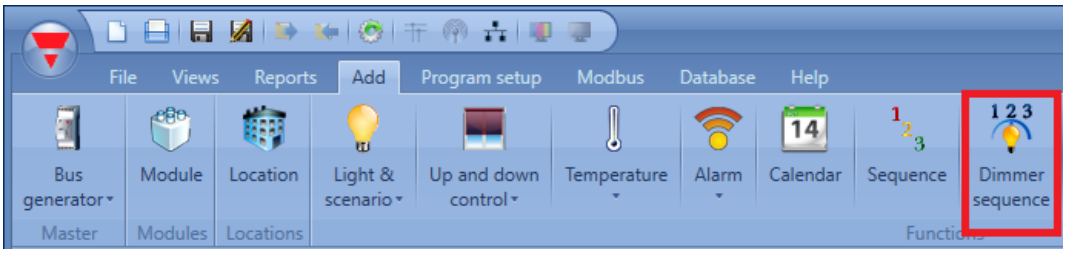
Each signal used in the *Stop sequence* window works in level mode. While the *Stop signal* is active, the function is stopped and cannot be activated until the stop signal disappears.
When the *Start sequence* signal is activated and at the same time the *Stop sequence* signal is activated, the *Start sequence* signal has priority and the function stays activated.

10.27 How to set the dimmer sequence function

The *Dimmer sequence* function allows the user to manage, together, the dimmable light functions already created.
The light level of all the added functions is set according to those defined in each step of the sequence, with the aim of making all the dimmers reach the final level at the same time.

This function can be used to create different scenarios, such as switching all the lights off at the same time regardless of the starting level of each single light.

To add a *Dimmer sequence* function the user should select *Dimmer sequence* from the Add menu (see picture below). The new function will be added in the selected location.

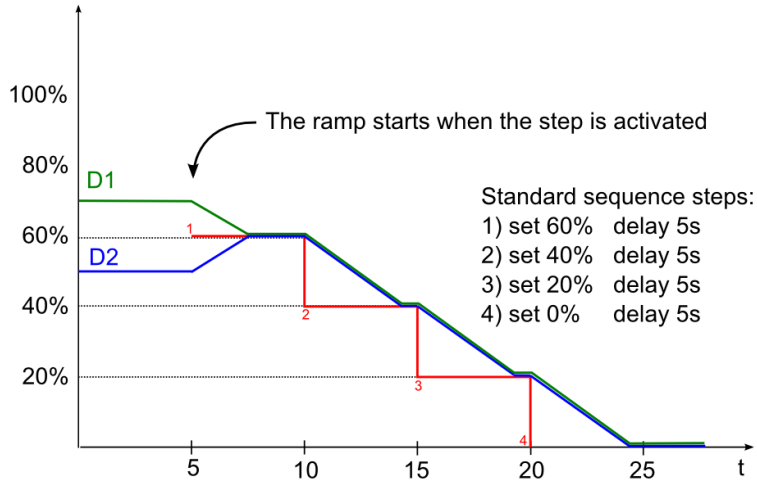


10.27.1 How the dimmer sequence works

The main difference between the standard sequence function and the dimmer sequence function is that the standard sequence executes all the steps, while the dimmer sequence skips the unnecessary steps and executes only the ones required if the *smart option* is enabled.

In the following figure, a standard sequence function is shown, where all the steps are executed regardless of the actual level (the starting level of dimmer 1 is 70%, the starting level of dimmer 2 is 50%). At instant 0 the sequence is activated and the first step is executed with a delay of 5 seconds.
When the sequence is started, both the dimmers are set to 60%, so dimmer 2 is brought from 50% to 60% and then down to 0%.

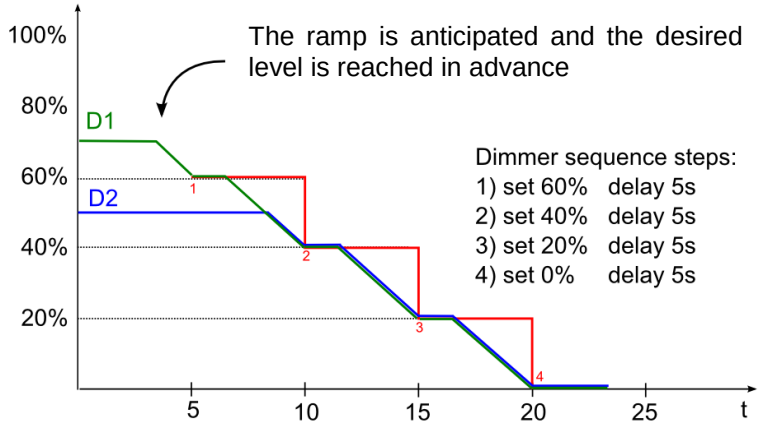
Standard sequence



The following figure shows the behaviour of the dimmer sequence using the same steps.

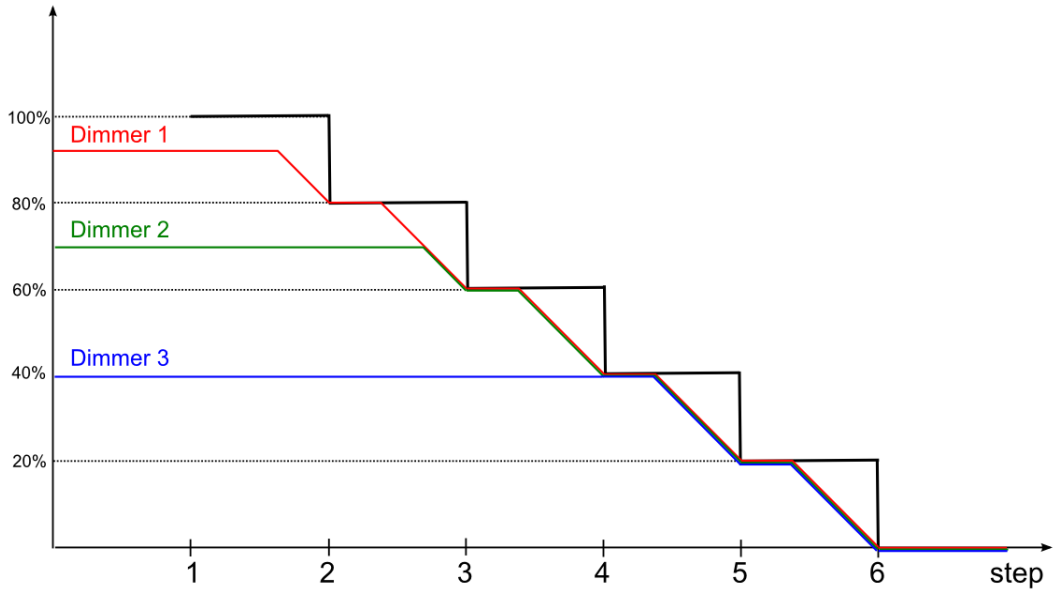
At instant 0 the sequence is activated and even if the first step is executed with a delay of 5 seconds, dimmer 1 starts going down immediately.
Dimmer 2 skips the first unnecessary step of the sequence, waits for dimmer 1 to reach 50% and then goes directly to the second step together with dimmer 1. In the dimmer sequence function, each dimmer starts dimming the output before the step activation in order to reach the required level at the beginning of each step.

Dimmer sequence

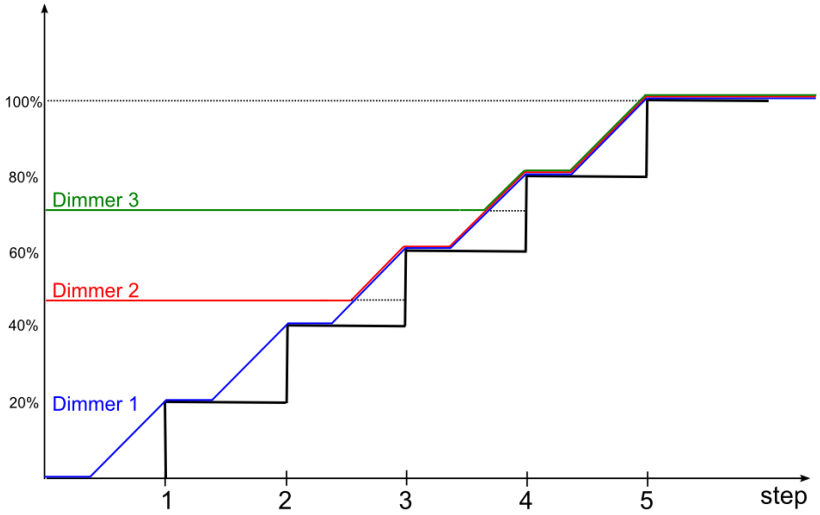


We suggest using the dimmer sequence with steps that are always increased or always decreased.

In the figure below, a “Switching off scenario” is shown, where all the steps are created in order to guarantee that all the dimmers of the sequence are managed at the same time. In this example, three dimmers are added to the sequence with three different starting light levels: each function starts dimming down only when needed.

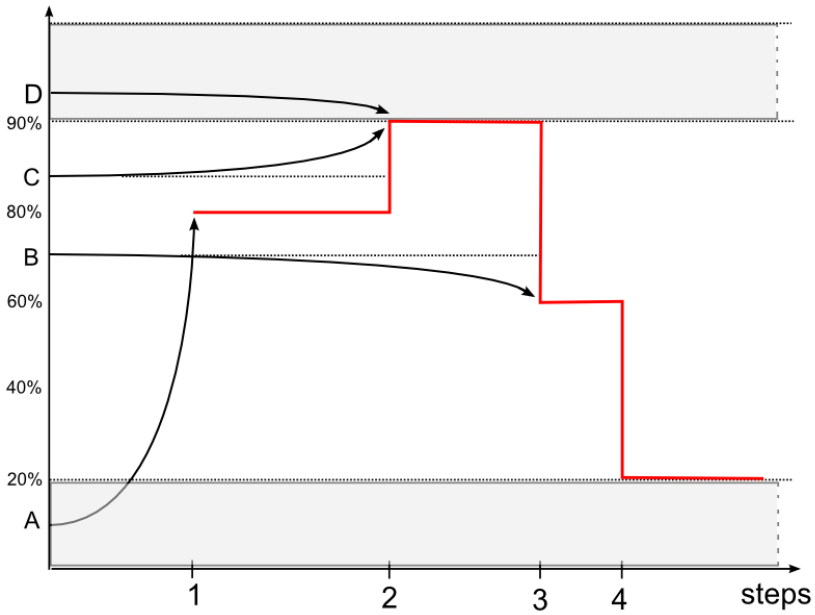


In the figure below, a “Switching on scenario” is shown where all the steps are created in order to reach the maximum level at the same time.



There are some special cases where the steps of the sequence are not designed for creating a linear sequence; the figure below shows the behavior of the different sequence steps according to the starting light level (the steps are not added in a linear way). There could be three different situations:

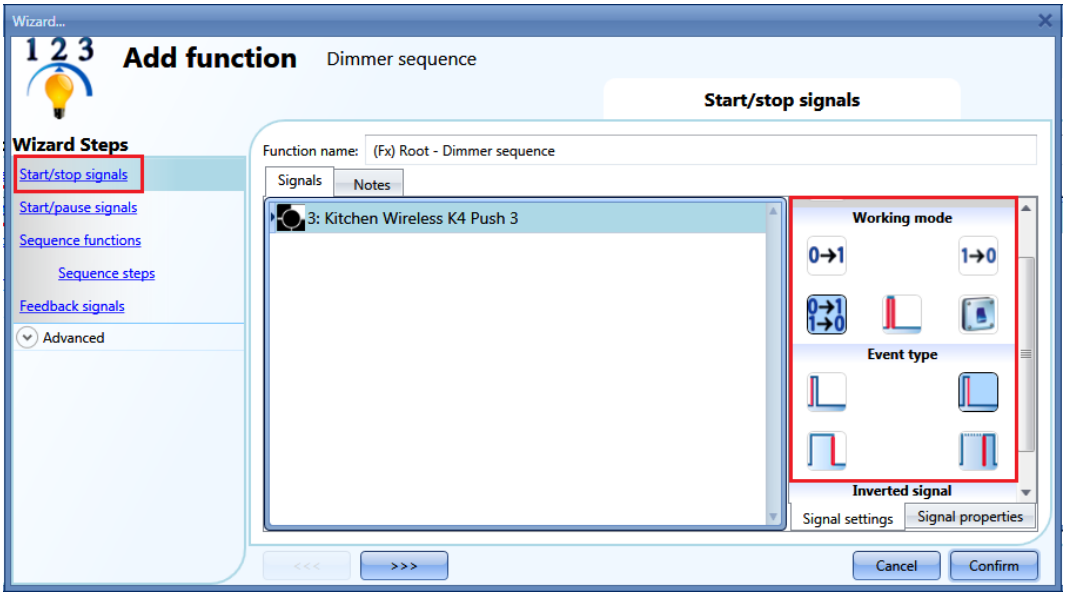
If the dimmer level is lower than the minimum step of the sequence, the dimmer is brought to the first step of the sequence (see dimmer A).
If the dimmer level is between the minimum and maximum steps of the sequence, the dimmer is brought to the closest step (see dimmers B and C).
If the dimmer status is higher than the maximum step of the sequence, the dimmer is brought to the highest sequence step (see dimmer D).



If a dimmer is switched off while a sequence is running, it will stay off and the sequence will not modify its status.

10.27.2 How to set the start/stop signals

If the user wants to add the start/stop signals, he has to select the relevant field, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



The input signal can be used either to start and/or to stop the sequence.

If a push button type is added, the working mode should be selected according to the table below:

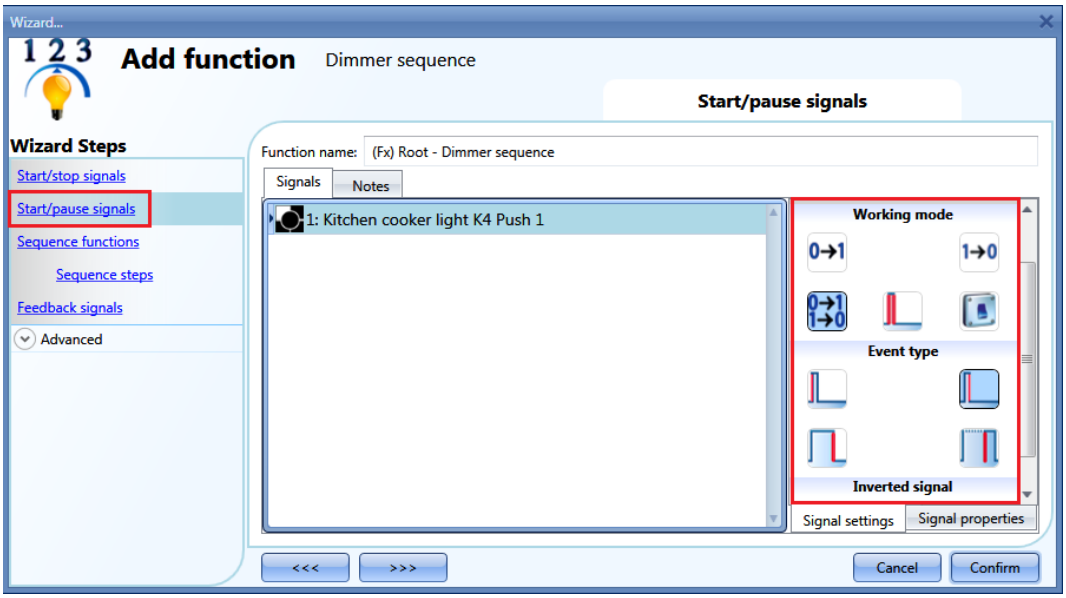
Working mode	Event type			
	Event when push button pressed	Event when push button released	Event with long pressure	Event with very long pressure
0→1	As soon as the push button is pressed, the sequence is started.	After a short press (less than 1 second), the sequence is started at the release of the push button.	After a long press, the sequence is started at the release of the push button.	After a very long press, the sequence is started at the release of the push button.
1→0	As soon as the push button is pressed, the sequence is stopped.	After a short press (less than 1 second), the sequence is stopped at the release of the push button.	After a long press, the sequence is stopped at the release of the push button.	After a very long press, the sequence is stopped at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the sequence will be started/stopped in toggle mode.	After a short press (less than 1 second), the sequence will be started/stopped in toggle mode at the release of the push button.	After a long press, the sequence will be started/stopped in toggle mode at the release of the push button.	After a very long press, the sequence will be started/stopped in toggle mode at the release of the push button.
	The sequence will be started/stopped in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The sequence will be started when the signal goes ON, and it will be stopped when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The sequence is started	No action
1→0	The sequence is stopped	No action
0→1 1→0	The sequence is started/stopped in toggle mode	No action
	The sequence is started/stopped in toggle mode	The sequence is started/stopped in toggle mode
	The sequence is started	The sequence is stopped

10.27.3 How to set the start/pause signals

If the user wants to add the start/pause signals, he has to select the relevant field, then double click on the *Signals* window and select the input signal from the list of those available (see picture below).



The input signal can be used either to start and/or to pause the sequence. When the sequence is paused, a start event will activate the sequence from the step where it was paused.

If a push button type is added, the working mode should be selected according to the table below:

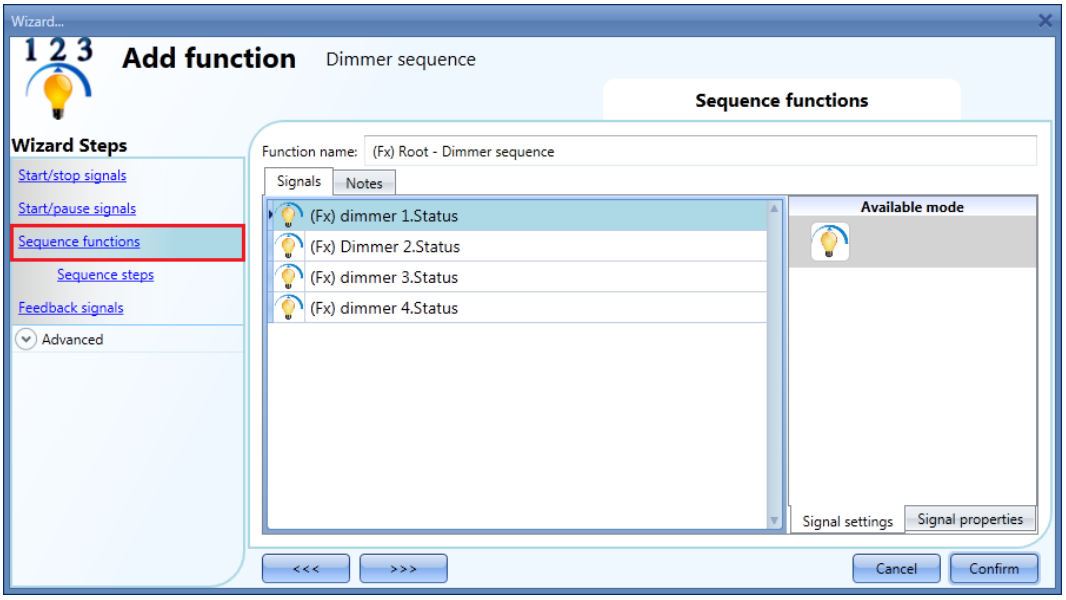
Working mode	Event type			
	Event when push button pressed	Event when push button released	Event with long pressure	Event with very long pressure
0→1	As soon as the push button is pressed, the sequence is started.	After a short press (less than 1 second), the sequence is started at the release of the push button.	After a long press, the sequence is started at the release of the push button.	After a very long press, the sequence is started at the release of the push button.
1→0	As soon as the push button is pressed, the sequence is paused.	After a short press (less than 1 second), the sequence is paused at the release of the push button.	After a long press, the sequence is paused at the release of the push button.	After a very long press, the sequence is paused at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the sequence will be started/paused in toggle mode.	After a short press (less than 1 second), the sequence will be started/paused in toggle mode at the release of the push button.	After a long press, the sequence will be started/paused in toggle mode at the release of the push button.	After a very long press, the sequence will be started/paused in toggle mode at the release of the push button.
	The sequence will be started/paused in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The sequence will be started when the signal goes ON, and it will be paused when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The sequence is started	No action
1→0	The sequence is paused	No action
0→1 1→0	The sequence is started/paused in toggle mode	No action
	The sequence is started/paused in toggle mode	The sequence is started/paused in toggle mode
	The sequence is started	The sequence is paused

10.27.4 How to add a function to the sequence

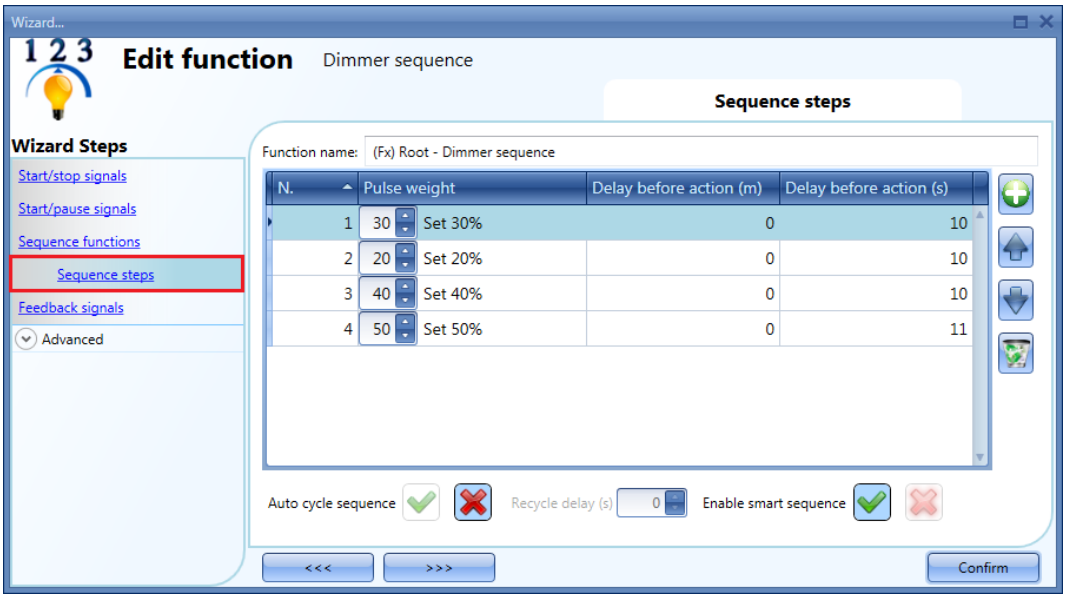
If the user wants to add a function to the dimmer sequence, he should select the relevant field, then double click on the *Signals* window to enter the list of available functions: once the functions list is opened, the user should select the functions he wants to add (see picture below).



Only *Dimmable light functions* can be added to the sequence.

10.27.5 How to set the sequence steps

When all the functions are added, the user can add and/or modify the actions to be performed. The set action will be carried out for all the functions added.
The action can be added/changed by selecting *Sequence steps* (see picture below).



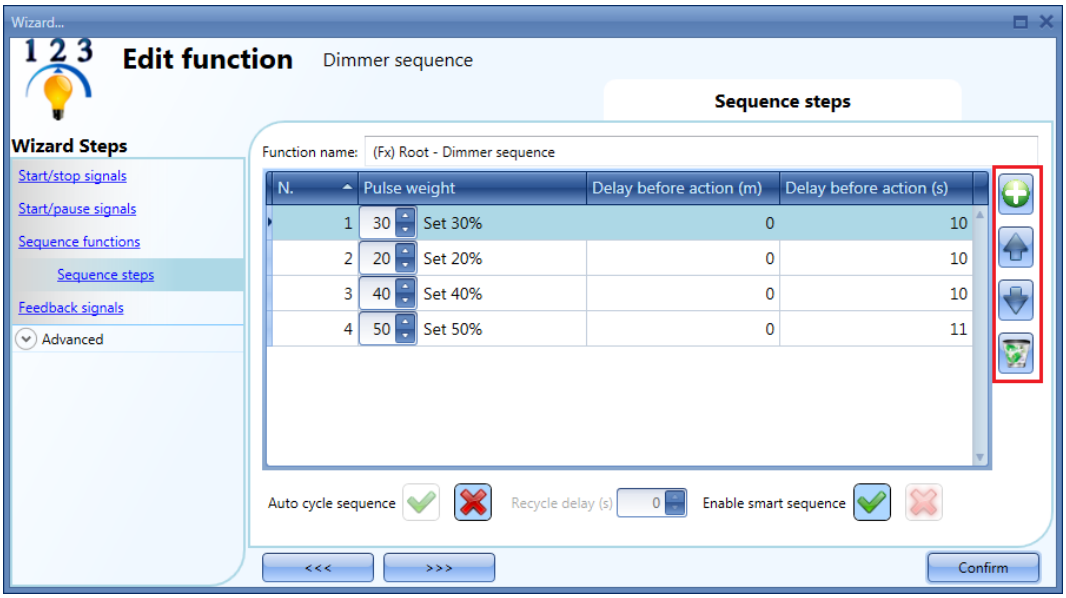
Auto cycle sequence: if this option is enabled, the sequence is repeated continuously: each time the last function is executed, the sequence starts again from the beginning. All the functions are

executed in a loop and the sequence can be stopped by using the stop signal(s).
Recycle delay (s): It is the delay (in seconds) between two repetitions of the sequence if the recycling option is enabled.

Enable smart sequence: if this option is enabled, the sequence skips the unnecessary steps.

10.27.6 How to modify the sequence steps

All the steps added to the sequence can be changed at any time by clicking on *Sequence steps,* double clicking on the field to be changed..
The user can do this in the main window of the dimmer sequence function (as shown below).



Using the icons on the right side of the window, the user can add, move up/down or delete a step from the list.

Table of actions:

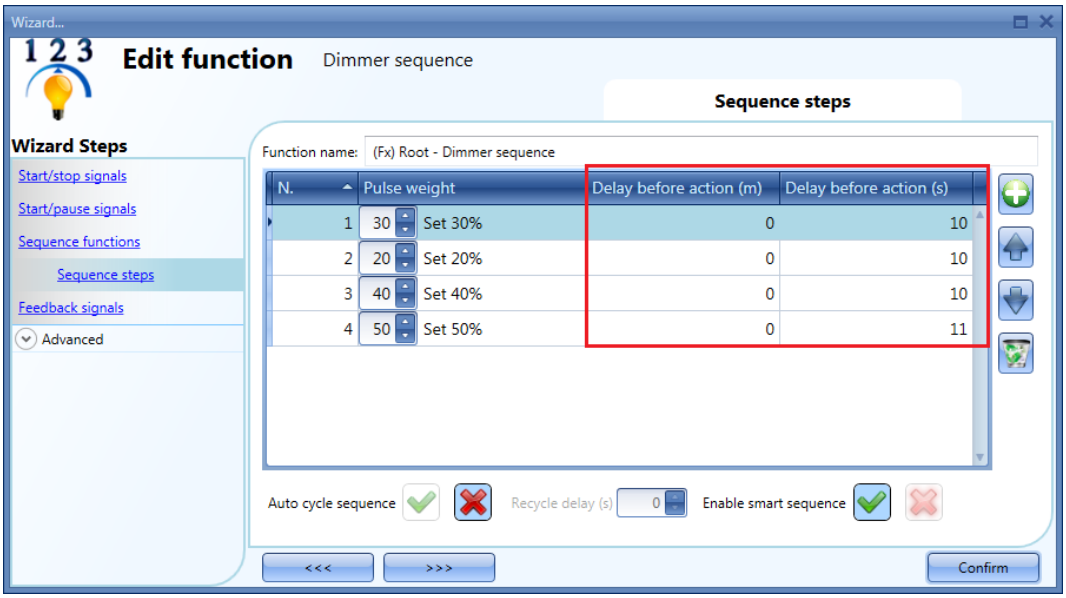
	By clicking on this icon a new step is added
	By clicking on this icon the selected step will be moved upwards
	By clicking on this icon the selected step will be moved downwards
	By clicking on this icon the selected step will be deleted

10.27.7 How to set the delay time between the activation of two steps

The delay between the activation of two steps can be set with a double click in the field *Delay before action,* as highlighted by the red rectangle.

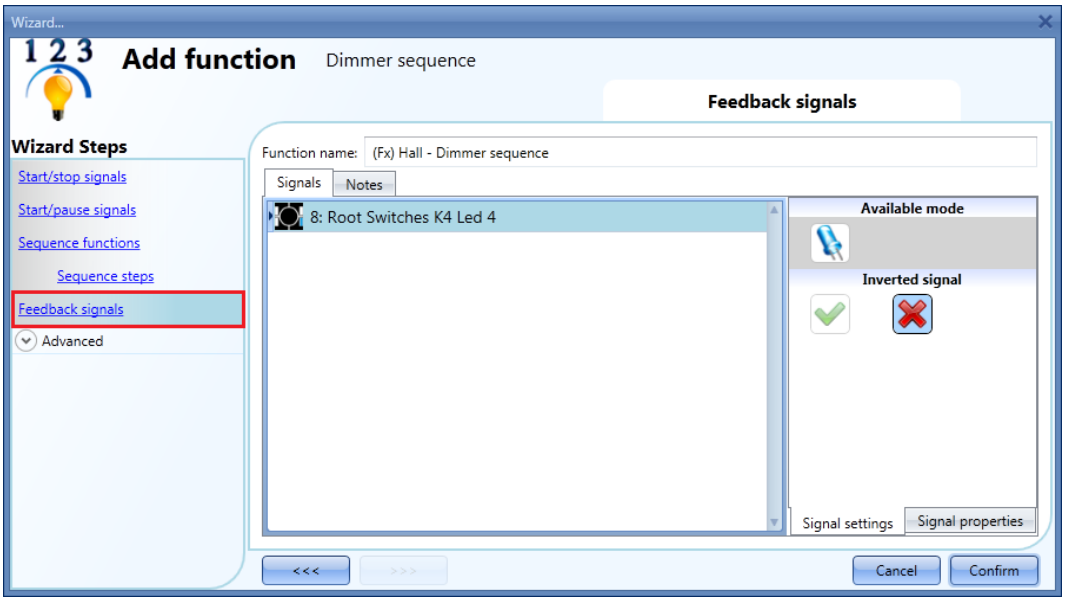
Note: If the pause time value is set to 0, the step is activated without any delay in respect to the previous one. This means that the two steps will be managed at the same time and this kind of setting is not recommended.

In the first step of the list, the *Delay before action* time is the delay between the start command and the activation of the first action.



10.27.8 How to add the feedback

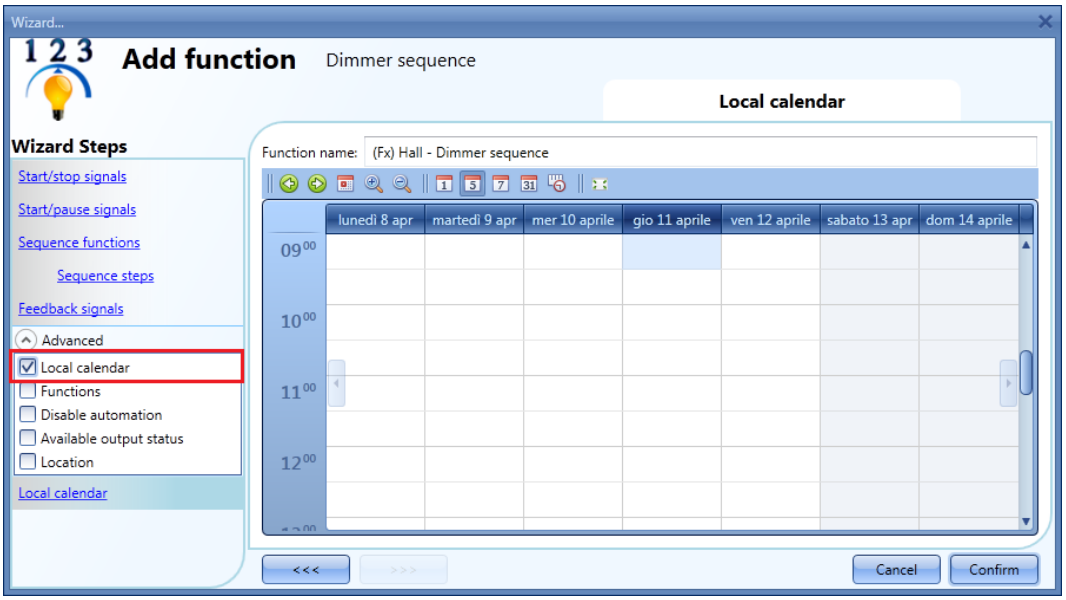
If the user wants to use a feedback signal to check the status of the sequence, he has to select the relevant section, then double click on the *Signals* window and select the feedback signal from the list of those available (see picture below).
The feedback signal can be a relay, a LED or a buzzer, and it behaves according to the status of the function: when the sequence is in progress the feedback signal is on; when the sequence is not running the feedback is off.



Once the feedback signal is entered, the user can also select the inverted logic.

10.27.9 How to start/stop the sequence using a calendar

If the user wants to start/stop the sequence using an automation with a calendar, the relevant menu has to be enabled in the *Local calendar* field (see picture below).



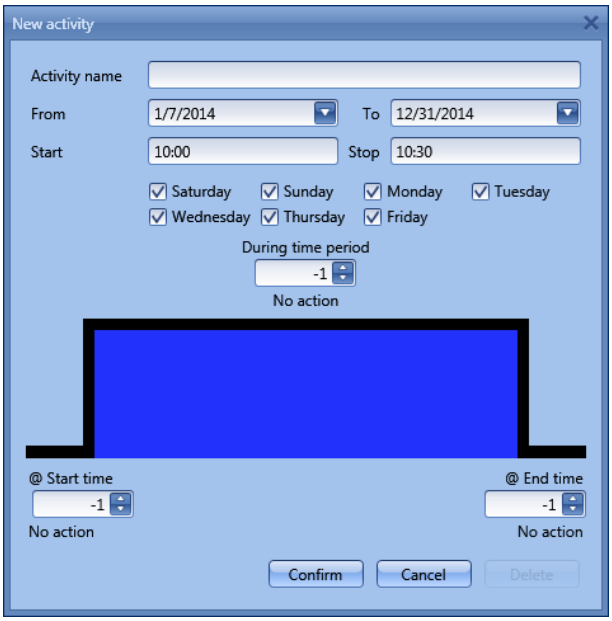
The user can select the type of visualization he prefers by clicking on the icons of the *Tool bar* as shown below:

Tool bar icons:

	One step back in the Calendar. E.g.: if a week is visualized, this button allows the user to go back to the previous week.
	One step forward in the calendar. E.g. if a week is visualized, this button allows the user to go to the next week.
	Go to the current day
	Magnifying lens to see more/fewer time bands.
	The calendar visualizes one day horizontally.
	The calendar visualizes 5 days horizontally.
	The calendar visualizes 7 days horizontally.
	The calendar visualizes 31 days horizontally.
	The calendar visualizes 7 days vertically.
	Full screen view

Calendar activities

Once the user has selected his preferred visualization, to enter a time band he should double click on the required day and the following window will appear:



Subject: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

From*:* the start date for the calendar activity.

To*:* the end date of the calendar activity.

Start*:* The time of the start of the activity.

Stop: The time of the end of the activity.

@ ***start time***: in this field the user can select the action of the function at start time.
At the start time the actions that the user can choose are:

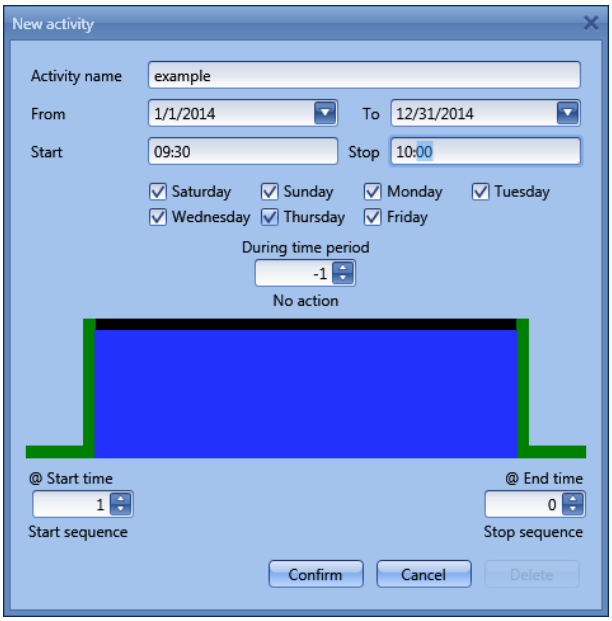
- No action (-1)
- Stop (0): the sequence is halted
- Start (1): the sequence is started

@ ***end time***: in this field the user can select the action of the function at stop time.
At the stop time the actions that the user can choose are:

- No action (-1)
- Stop (0): the sequence is halted
- Start (1): the sequence is started

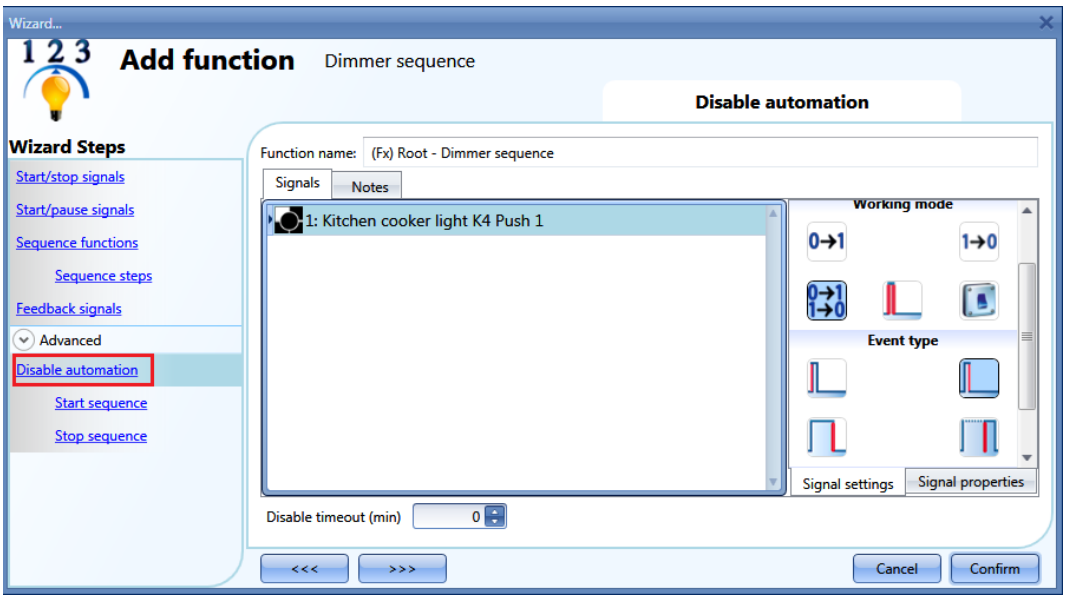
Days: The user should select the days when the calendar has to operate.

In the example below the calendar is set to start a sequence at 9:30 and stop it at 10 o'clock.



10.27.10 How to disable the calendar automation

If the user wants to disable the automation managed by the internal calendar, the *Disable automation* field has to be used: when the disabling is active, the sequence function can only be activated/deactivated by using manual start/stop commands.
To enable it, select *Disable automations* in the *Advanced* section, then double click on the signals window and select the signal to use (see picture below).



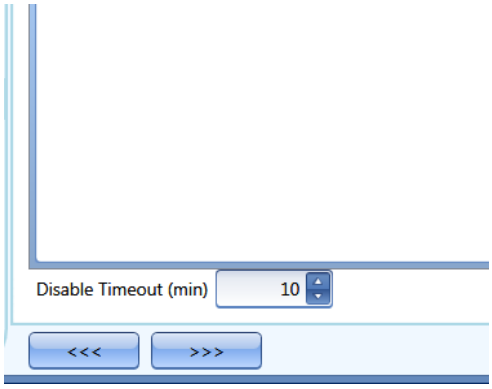
When the signal is added, the user has to select the working mode and the type of the event.
If a push button type is added, the working mode has to be selected according to the table below:

Working mode	Event type			
	Event when push button pressed	Event when push button released	Event with long pressure	Event with very long pressure
0→1	As soon as the push button is pressed, all automations are disabled.	After a short press (less than 1 second), at the release of the push button all automations are disabled.	After a long press, all automations are disabled at the release of the push button.	After a very long press, all automations are disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automations are enabled again.	After a short press (less than 1 second), the automations are enabled again at the release of the push button.	After a long press, the automations are enabled again at the release of the push button.	After a very long press, the automations are enabled again at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the automations will be disabled/enabled in toggle mode.	After a short press (less than 1 second), the automations will be disabled/enabled in toggle mode at the release of the push button.	After a long press, the automations will be disabled/enabled in toggle mode at the release of the push button.	After a very long press, the automations will be disabled/enabled in toggle mode at the release of the push button.
	The automations will be disabled/enabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automation is disabled when the signal goes ON, and it is enabled again when the signal goes OFF.			

If a switch signal is used, the setting of the working mode should follow the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The automations are disabled	No action
1→0	The automations are enabled	No action
0→1 1→0	The automations are disabled/enabled in toggle mode	No action
	The automations are disabled/enabled in toggle mode	The automations are disabled/enabled in toggle mode
	The automation is disabled	The automation is enabled

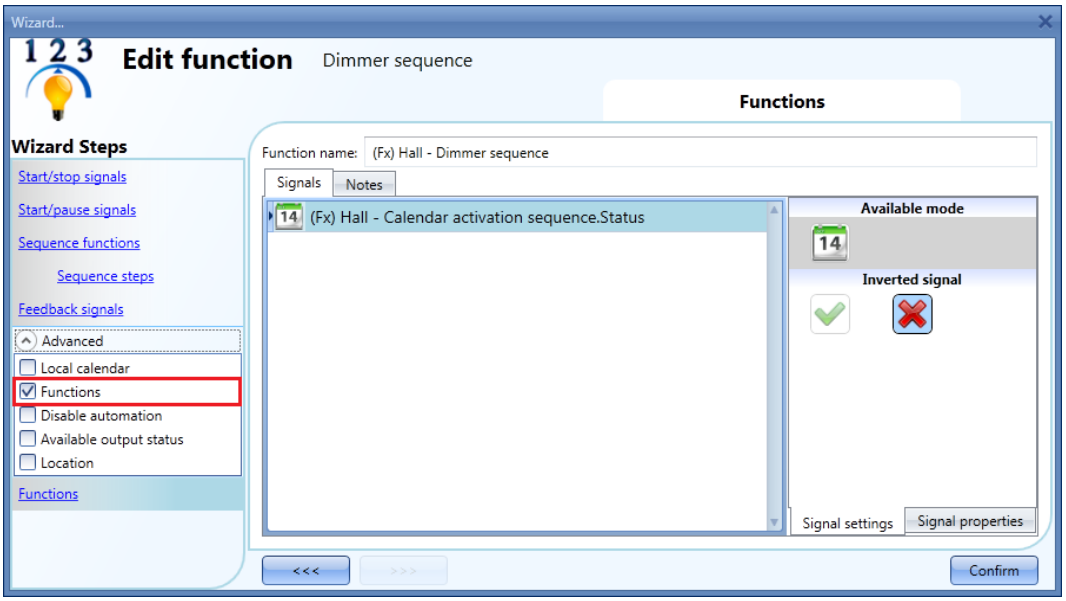
The user might need to set a time after which the calendar automation is enabled again, even if the selected signal is still active: to do this the *Disable timeout* field at the bottom of the window should be set.



The timer starts each time the disable status is activated: the disable status is automatically deactivated as soon as the time expires.
In the figure above a disable timeout of 10 minutes is set. The maximum delay is 59 minutes.
If the timer is set to 0 minutes, the timer is disabled and the disable status will stay activated until there is a manual command.

10.27.11 How to use an external function to start/stop a sequence

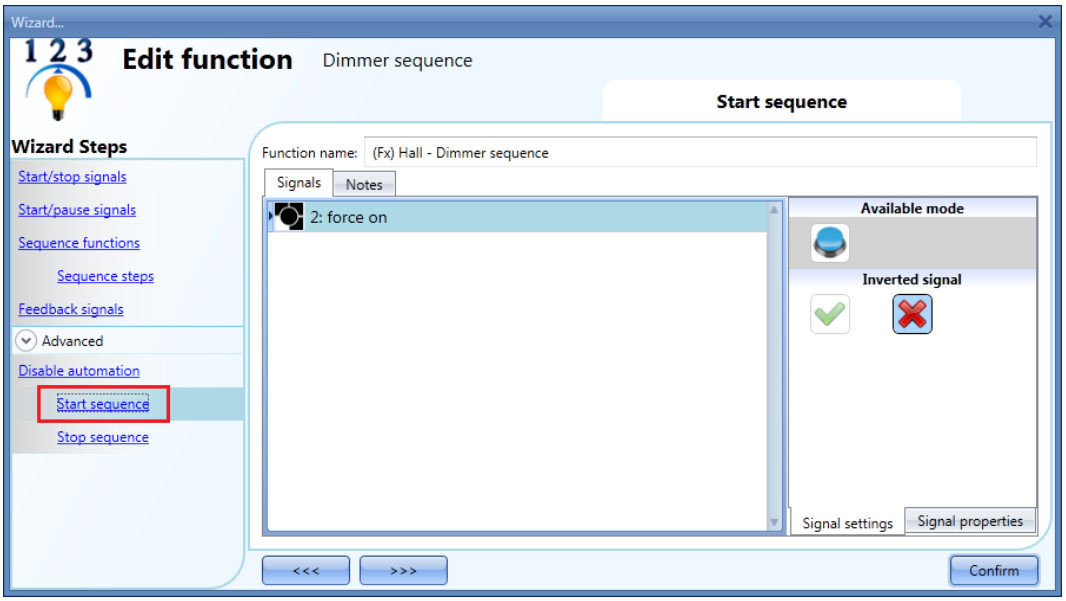
If the user wants to use an external function to start/stop a sequence, the *Function* field has to be used: to enable it, select *Functions* in the *Advanced* section, then double click on the signals window and select the relevant function from the list (see picture below).



All the added functions will trigger the sequence according to the status they have: for example, a light that is switched on will make the sequence start; the same function that is switched off will make the sequence stop.
The global calendar has to be added in this field and it works in the same way as the internal calendar.

10.27.12 How to force the sequence active

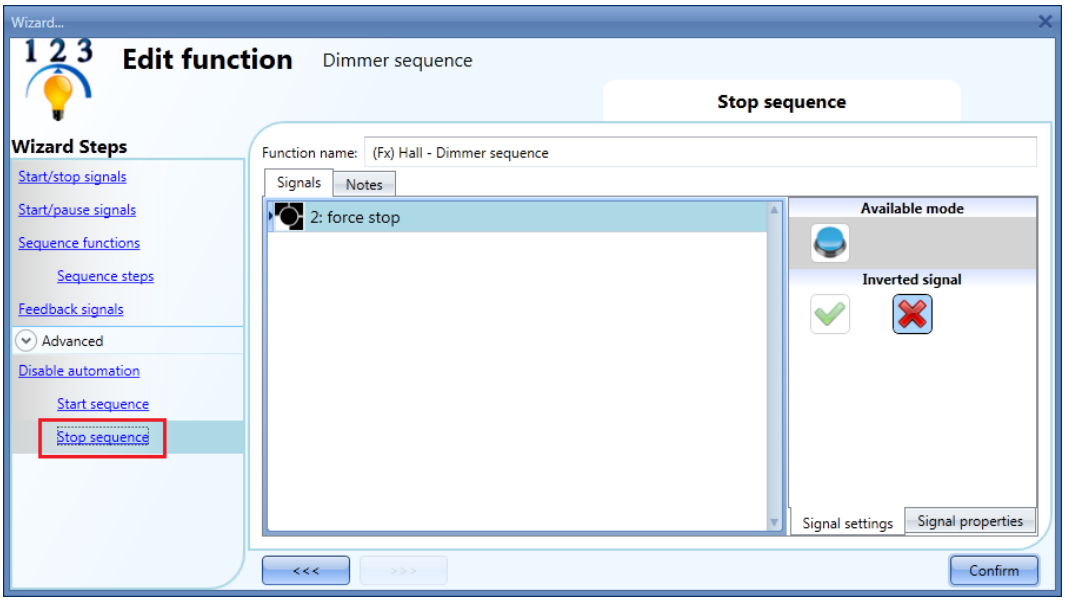
If the user wants to force the sequence active regardless of all the other signals used in the function, the *Start sequence* field has to be used: to enable it, select *Disable automations* in the *Advanced* section, select *Start sequence*, then double click on the signals window and select the signal to use (see picture below).



Each signal used in the *Start sequence* window works in level mode. While the start signal is active, the function is operative and cannot be deactivated until the start signal disappears.
When the *Start sequence* signal is activated and at the same time the *Stop sequence* signal is activated, the start sequence signal has priority and the function stays activated.

10.27.13 How to force the sequence stop

If the user wants to force the sequence stop regardless of all the other signals used in the function, the *Stop sequence* field has to be used: to enable it, select *Disable automations* in the *Advanced* section, select *Stop sequence*, then double click on the signals window and select the signal to use (see picture below).

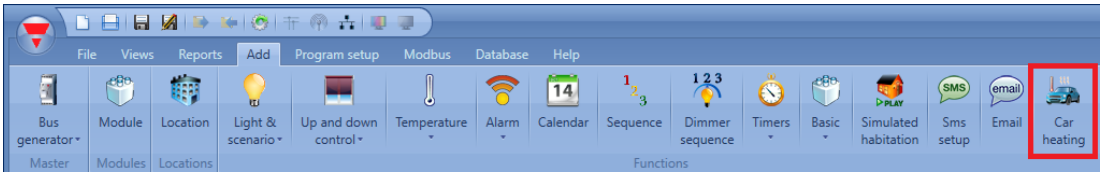


Each signal used in the *Stop sequence* window works in level mode. While the *Stop signal* is active, the function is stopped and cannot be activated until the stop signal disappears.
When the *Start sequence* signal is activated and at the same time the *Stop sequence* signal is activated, the *Start sequence* signal has priority and the function stays activated.

10.28 How to set the car heating function

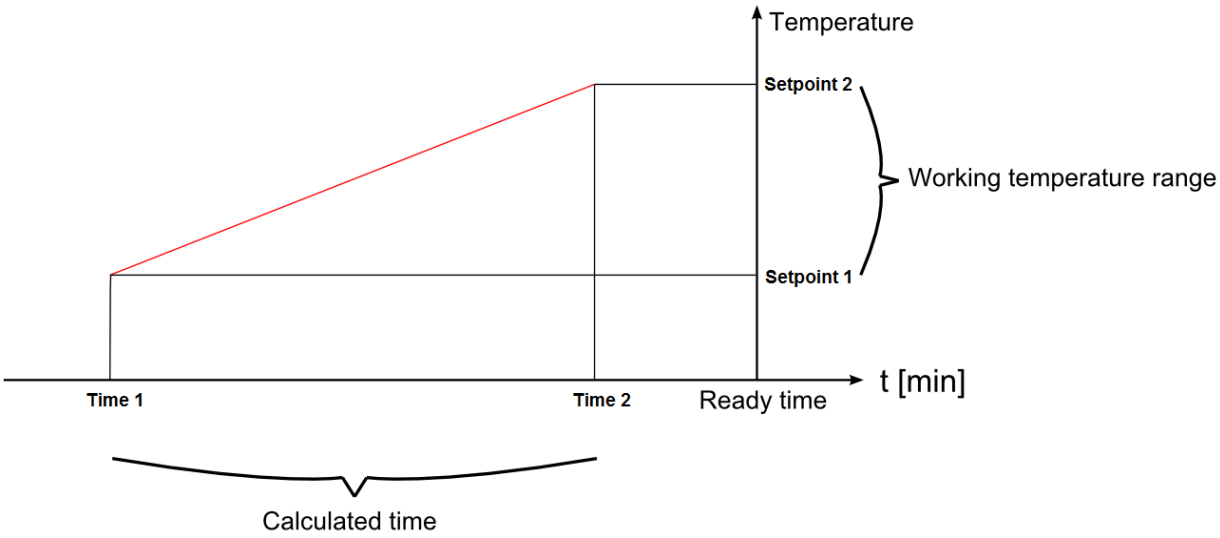
The car heating function allows the user to heat the car so that it is ready at a predefined set time. The user must set a time, two external temperatures limits (SP 1 and SP 2) and two timers (T1 and T2) so as to define the extreme points of a straight line. The straight line is used in the algorithm to define when the output should be on to heat the car.

To add a car heating function the user has to select *Car heating* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to a Car heating function can also be opened by typing Alt+Y.



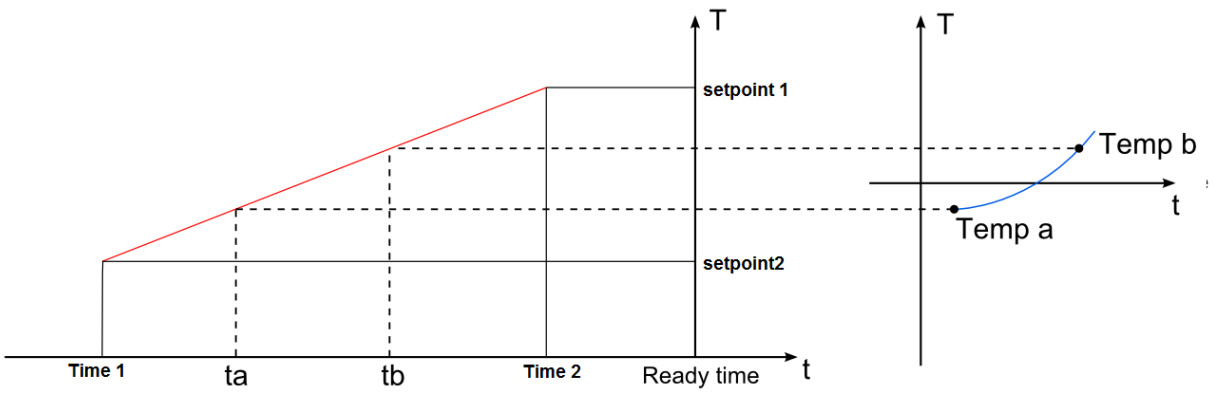
10.28.1 How the heating algorithm works

Referring to the picture below, the *Ready time* is the time at which the car must be ready and warmed. If the external temperature is inside the working limits (*Setpoint1* and *Setpoint2*), the output is activated before the *Ready time* according to a calculation on the measured temperature.
The activation time of the output is calculated on the basis of the defined parameters, using the straight line highlighted in red in the picture below.

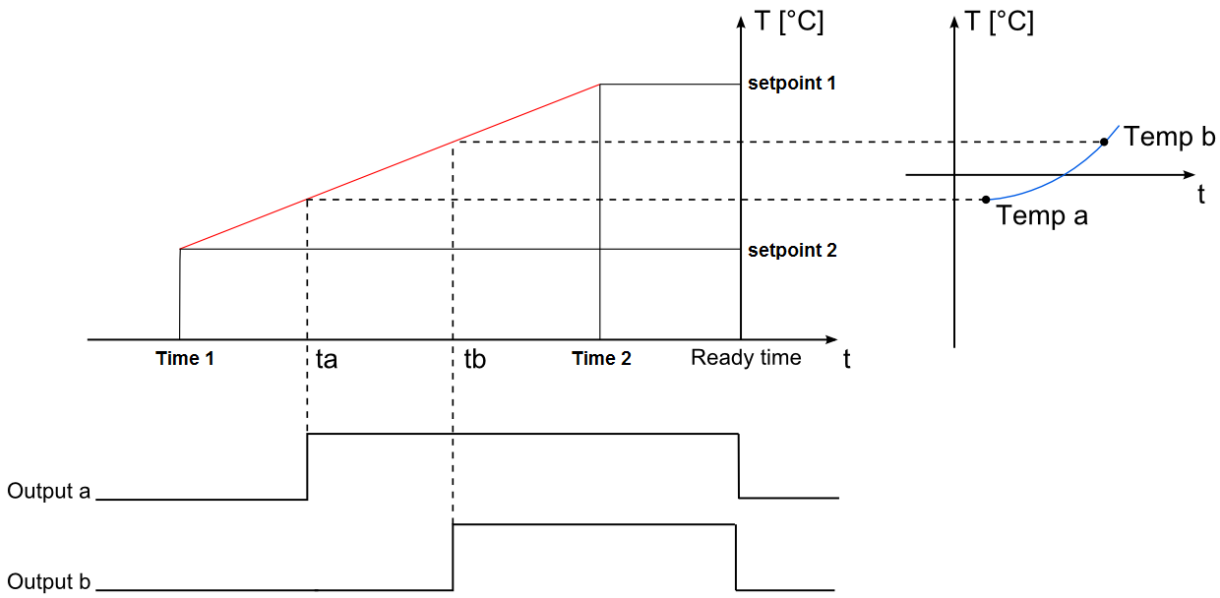


If the outdoor temperature is lower than or equal to *Setpoint1*, the heating is activated a *Time 1* before the *Ready time*. If the temperature is above *Setpoint2*, the heating is never activated. If the measured temperature is between *Setpoint1* and *Setpoint2*, the activation time of the heating output is calculated using the straight red line.

In the picture below it is shown how the algorithm works according to the outside measured temperature. If the temperature has a value “Temp a” inside the range [*Setpoint1*, *Setpoint2*], the output will be activated at the time “ta”.

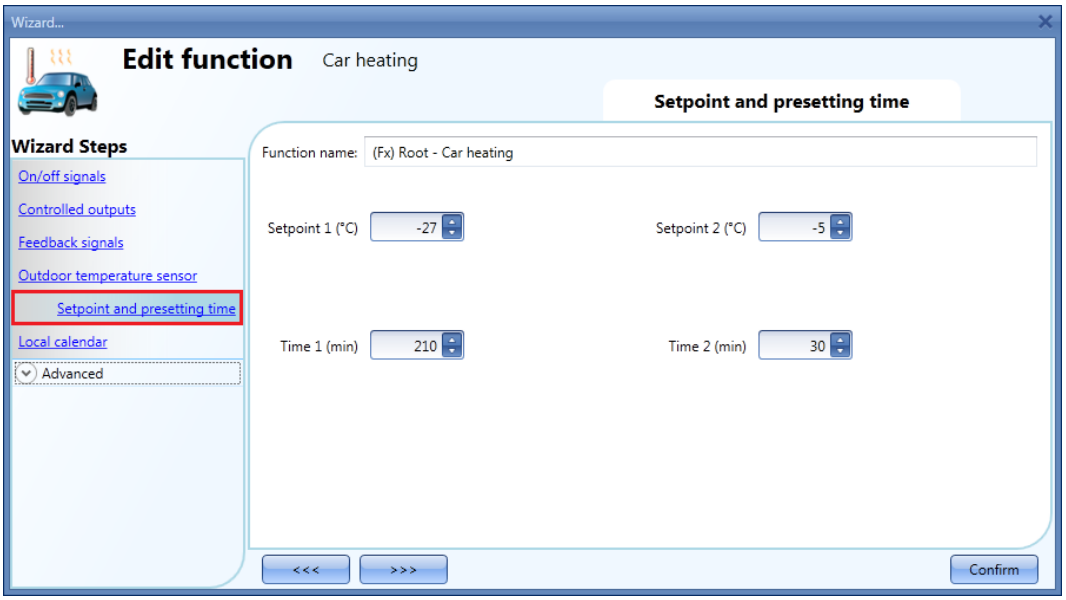


If the measured temperature has a value Temp b (higher than Temp a), the output will be activated later (at time tb), because the temperature is higher and the output requires less heating time to warm the car.



10.28.2 How to set the setpoints and times

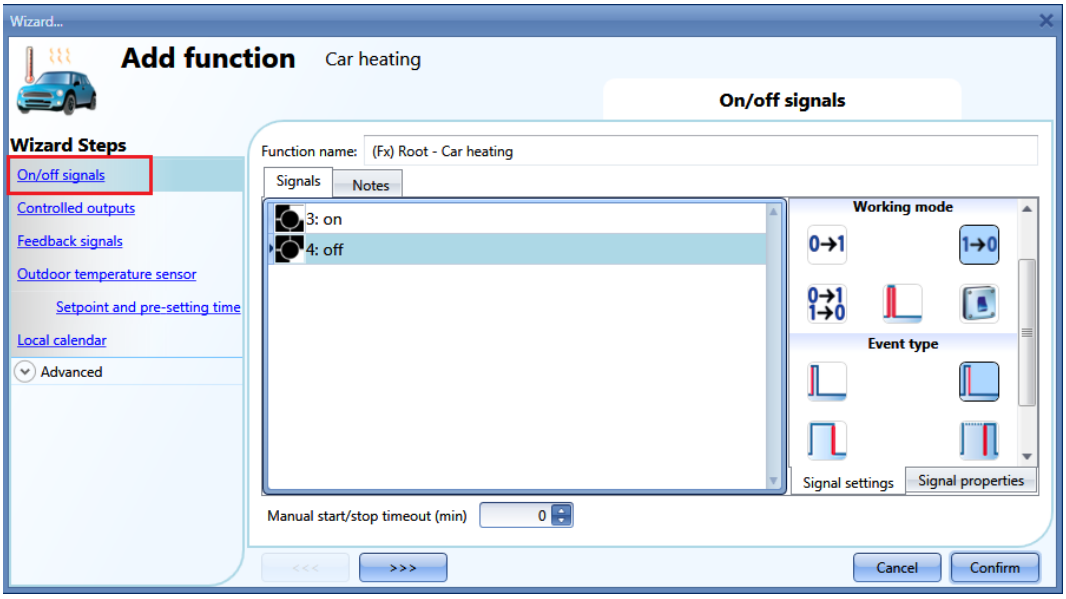
The pre-setting time and the setpoints have to be entered in the relevant field as shown in the picture below.



To set the time at which the car has to be ready, please read paragraph *How to set the ready time*.

10.28.3 How to set the manual signals

Even if the function is not started on the basis of the algorithm described above, it is possible to activate it manually with manual inputs. Likewise, if the function is active, it is possible to switch it off manually with the stop inputs.
If the user wants to use a manual on/off signal, he has to select the relevant section, then double click on the *Signal* window and select the input signal from the list of available signals (see picture below).



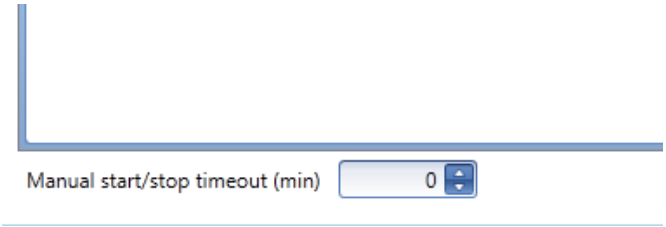
The input signal can be used to start and stop the car heating: the start signal activates the function, the stop signal deactivates it.
If a push button type is selected, the way of working marked with the red rectangle, has to be done following the table below.

Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, the output is activated.	After a short press (less than 1 second), the output is activated at the release of the push button.	After a long press, the output is activated at the release of the push button.	After a very long press, the output is activated at the release of the push button.
1→0	As soon as the push button is pressed, the output is deactivated.	After a short press (less than 1 second), the output is deactivated at the release of the push button.	After a long press, the output is deactivated at the release of the push button.	After a very long press, the output is deactivated at the release of the push button.
0→1 1→0	As soon as the push button is pressed, the output will be activated/deactivated in toggle mode.	After a short press (less than 1 second), the output will be activated/deactivated in toggle mode at the release of the push button.	After a long press, the output will be activated/deactivated in toggle mode at the release of the push button.	After a very long press, the output will be activated/deactivated in toggle mode at the release of the push button.
	The output will be activated/deactivated in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The function is switched on when the signal goes ON, and it is switched off when the signal goes OFF.			

If a switch signal is used, the set of the working mode has to be done following the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The output is activated	No action
1→0	The output is deactivated	No action
0→1 1→0	The output is activated/deactivated in toggle mode	No action
	The output is activated/deactivated in toggle mode	The output is activated/deactivated in toggle mode
	The function is activated	The function is deactivated

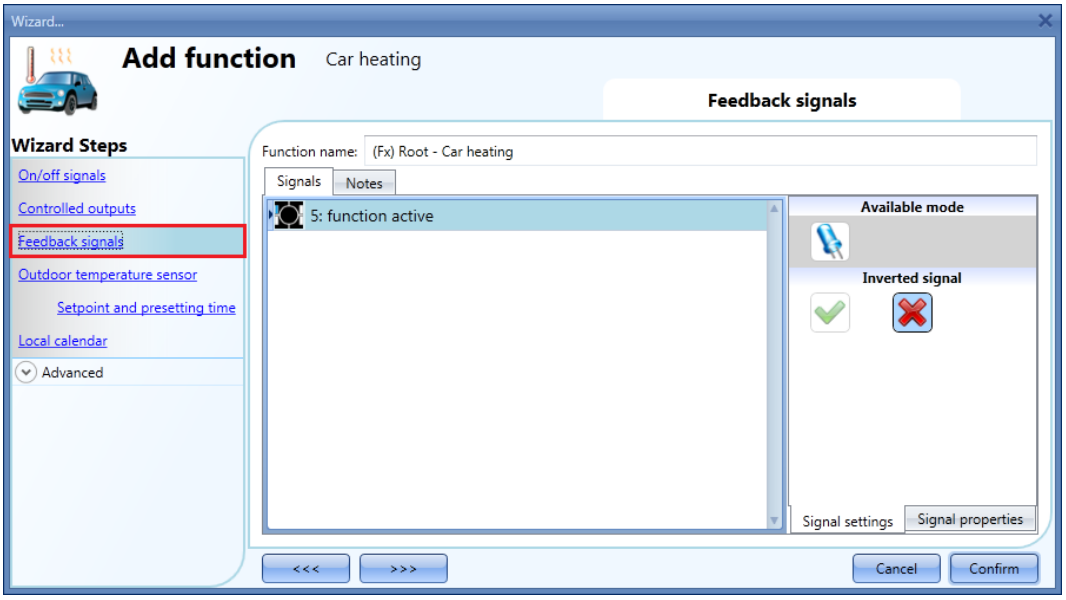
The user could set a time after which the output is switched off even if the selected start signal is still active, to achieve this the “manual start/stop timeout” field has to be set.



The timer starts counting every time an input signal is activated, as soon as this timer expires the output goes back to the previous status.
The maximum delay is 59 minutes, if the timer is set to 0 minutes, the timer is deactivated.

10.28.4 How to add the feedback signal

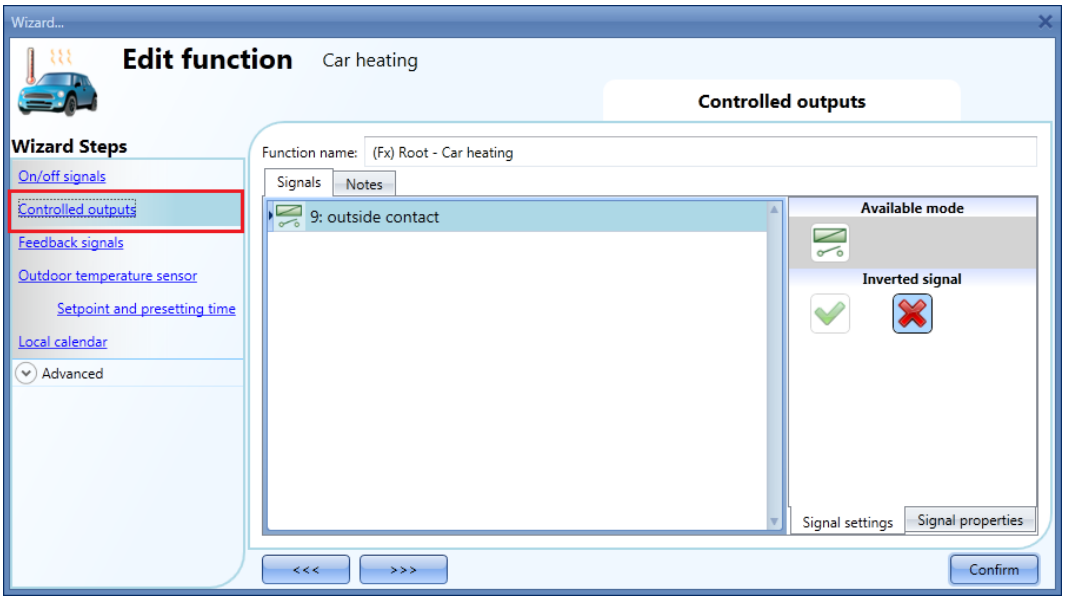
If the user wants to use a feedback signal to check the status of the car heating function, he has to select the relevant section, then double click on the *Signal* window and select the input signal from the list of available ones (see picture below).



The feedback signal can be a relay, a LED or a buzzer, and it follows the status of the function: when the function is ON the feedback is on, when the function is OFF the feedback is off.

10.28.5 How to add the output

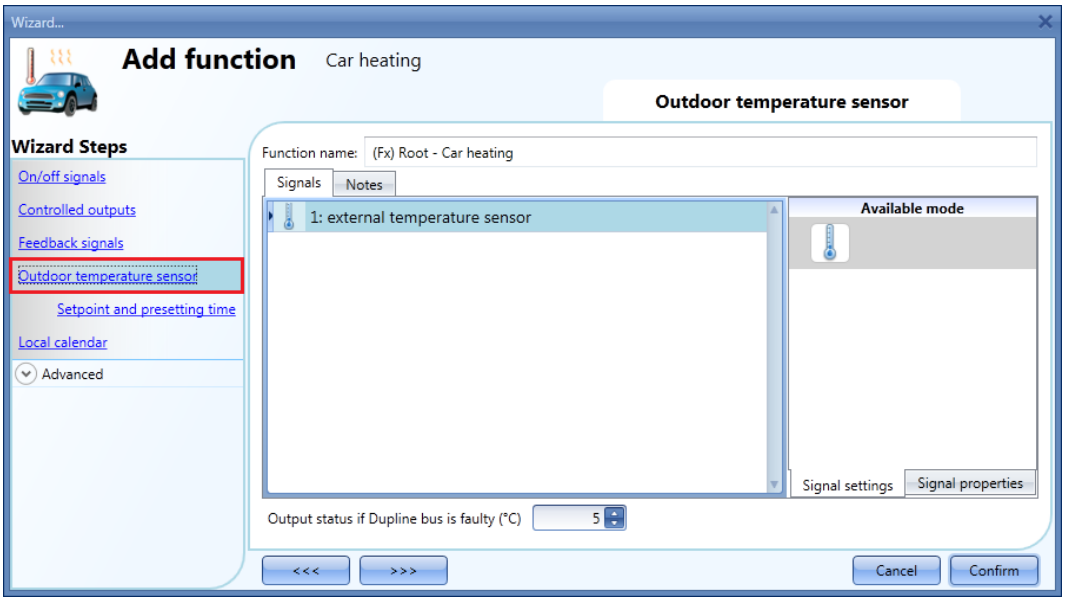
To add an output signal to the function, the user has to select the relevant section, then double click on the *Signal* window and select the output signal from the list of available ones (see picture below).



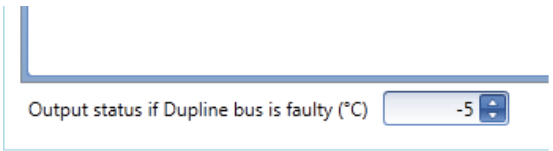
The output signal can be a relay, a LED or a buzzer, and it follows the status of the function: when the function is ON the output is activated, when the function is OFF the output is deactivated.

10.28.6 How add the temperature sensor

To add the temperature sensor to the car heating function, the user has to select the relevant section, then double click on the *Signal* window and select the temperature signal from the list of available ones (see picture below).
The user can add up to 10 different temperature signals: if more signal are present, the average value is used as reference temperature.
If one temperature sensor is faulty, the invalid temperature value is discarded from the calculation of the average temperature.

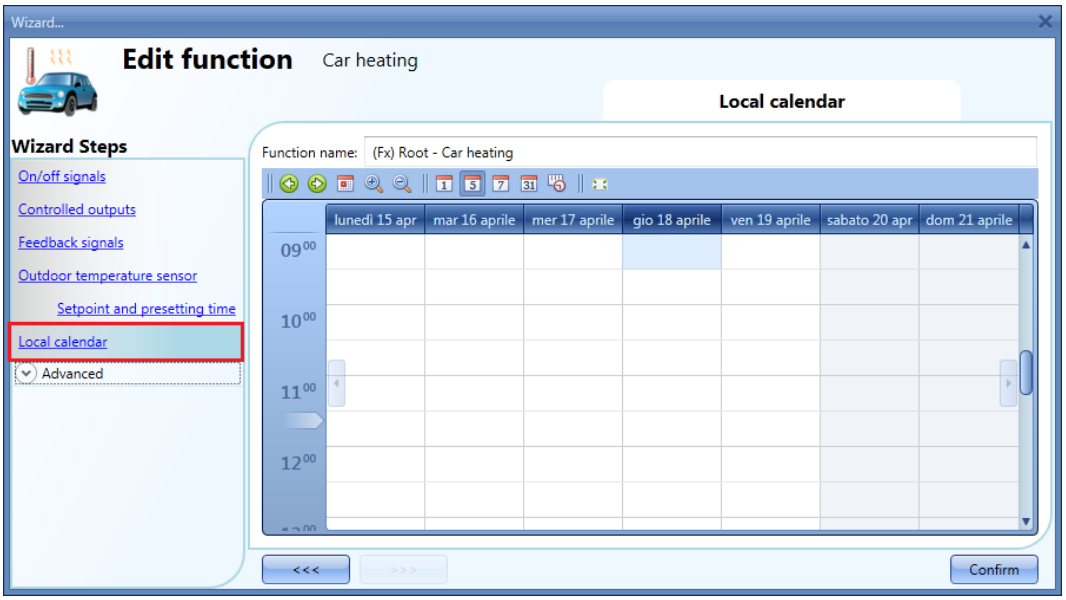


Should the temperature sensor be faulty, the system will use a default value that the user can set from -50°C to +50°C.



10.28.7 How to set the ready time

The *Ready time* (the time at which the car has to be ready) must be set in the Local calendar field (see picture below).



The user can select the kind of visualization he prefers clicking on the icons of the *Tool bar* reported below:

Tool bar icons description:

	One step back in the Calendar. Ex: if a week is visualized, this button let's the user go back to the previous week.
	One step forward in the calendar. Ex: if a week is visualized, this button let's the user go to the next week.
	Go immediately to the current day.
	Magnifying lens to see more/less time bands.
	The calendar visualizes horizontally one day.
	The calendar visualizes horizontally 5 days.
	The calendar visualizes horizontally 7 days.
	The calendar visualizes horizontally 31 days
	The calendar visualizes vertically 7 days
	Full screen view

Calendar activities
Once the user has selected his preferred kind of visualization, to enter a time band he has to double click on the wanted day and the following window will appear:

Activity name: In this field the user defines the name of the event that will appear on the calendar, this is a mandatory field.

From*:* the beginning date for the calendar automation.

To*:* the end date of the calendar automation.

Ready at: in this field the user can select the time when the car has to be ready.

Days: The user has to check the days when the calendar have to work.

In the example above the calendar is set to get the car ready at 7:30 during the week except on Saturday and Sunday.

10.28.8 How to disable the calendar automation

If the user wants to disable the heating managed by the calendar, the disable automations step has to be used: when the heating is disabled, the car heating output can only be activated using manual input signals.
To enable it, check *Disable automations* in the *Advanced* section, then double click on the signal window and select the real signal to use (see picture below).

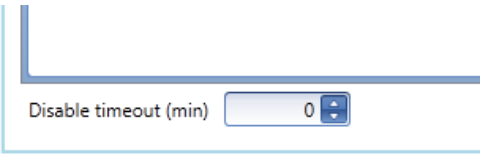
If a push button type is selected, the working mode can be selected according to the following table:

Working mode	Event type			
	Event on push button pressed	Event on push button released	Event on long pressure	Event on very long pressure
0→1	As soon as the push button is pressed, all automations are disabled.	After a short press (less than 1 second), at the release of the push button all automations are disabled.	After a long press, all automations are disabled at the release of the push button.	After a very long press, all automations are disabled at the release of the push button.
1→0	As soon as the push button is pressed, the automations are enabled again.	After a short press (less than 1 second), the automations are enabled again at the release of the push button.	After a long press, the automations are enabled again at the release of the push button.	After a very long press, the automations are enabled again at the release of the push button.
0→1 1→0 Event toggle	As soon as the push button is pressed, the automations will be disabled/enabled in toggle mode.	After a short press (less than 1 second), the automations will be disabled/enabled in toggle mode at the release of the push button.	After a long press, the automations will be disabled/enabled in toggle mode at the release of the push button.	After a very long press, the automations will be disabled/enabled in toggle mode at the release of the push button.
	The automations will be disabled/enabled in toggle mode when the push button is pressed and toggled again when the push button is released.			
	The automations are disabled when the signal goes ON, and are enabled again when the signal goes OFF.			

If a switch signal is used, the set of the working mode has to be done following the table below:

Working mode	Event type	
	Signal activated	**Signal deactivated**
0→1	The automations are disabled	No action
1→0	The automations are enabled	No action
0→1 1→0	The automations are disabled/enabled in toggle mode	No action
	The automations are disabled/enabled in toggle mode	The automations are disabled/enabled in toggle mode
	The automations are disabled	The automations are enabled

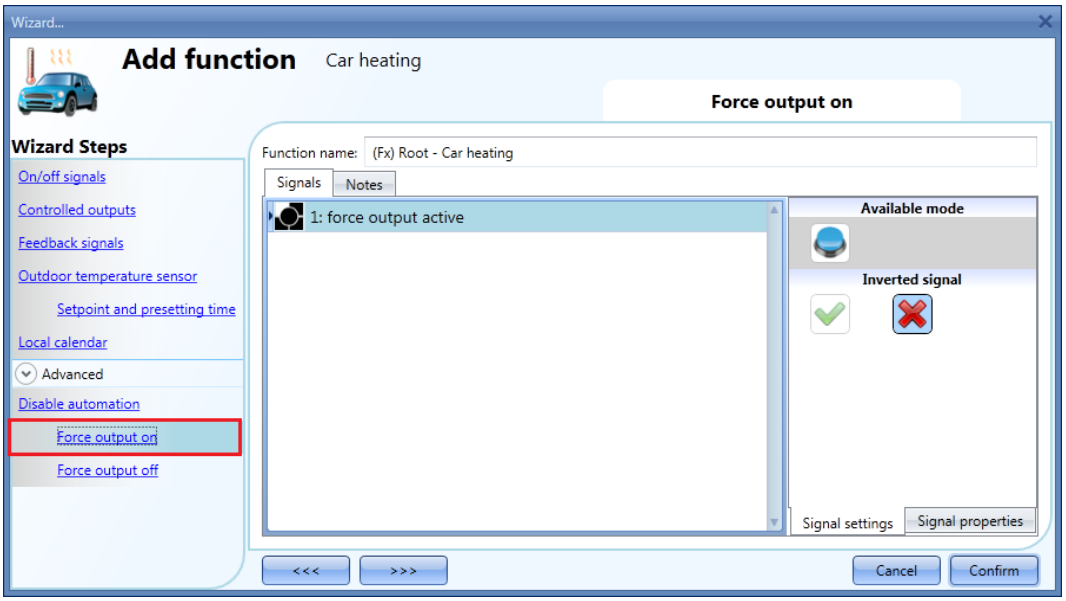
The user might need to set a time after which the calendar automation is enabled again even if the selected signal is still active: to achieve this the “Disable timeout” field at the bottom of the window has to be set.



The timer starts every time the disable inputs is activated: as soon as the timer expires, the disable status is automatically deactivated.

10.28.9 How to force the output on

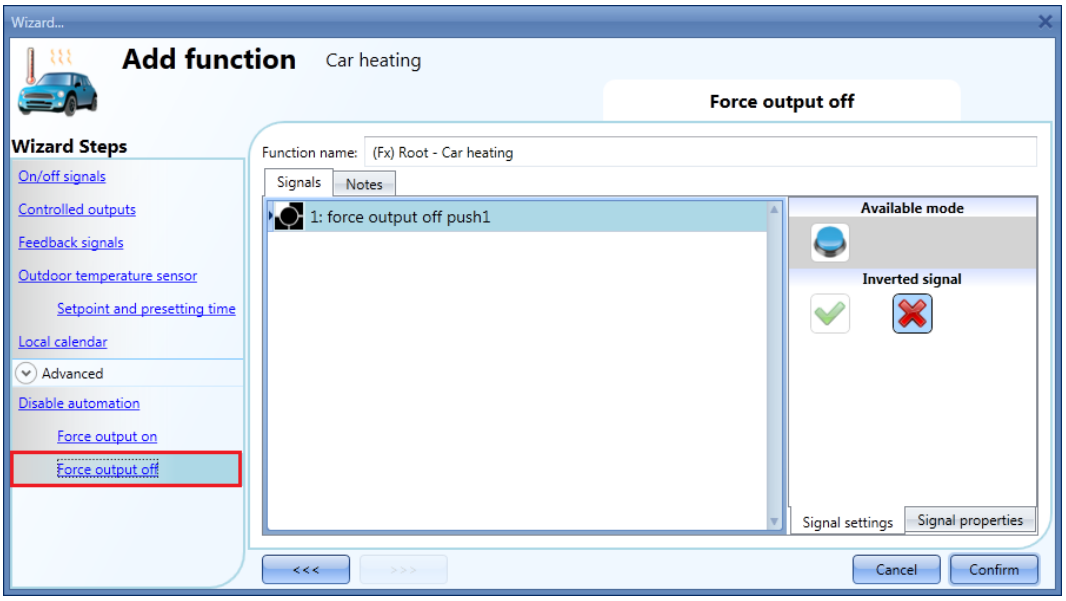
If the user wants to force the output ON regardless of all the other signals used in the function, the *Force output on* step has to be used: to enable it check *Disable automations* in the *Advanced* section, select *Force output on*, then double click on the signal window and select the real signal to use (see picture below).



Each signal used in the *Force output on* window works as a level signal: until the signal is active, the car heating function is forced on.
When the *force output on* signal is activated and the *force output off* signal is activated, the force to output on signal has the priority.

10.28.10 How to force the output off

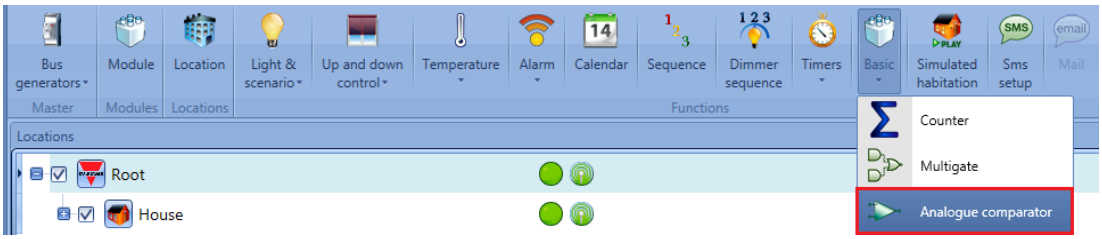
If the user wants to force the output OFF regardless of all the other signals used in the function, the *Force output off* step has to be used: to enable it check *Disable automations* in the *Advanced* section, select *Force output off*, then double click on the signal window and select the real signal to use (see picture below).



Each signal used in the *Force output off* window works as a level signal: until the signal is active, the car heating function is forced off.
When the *force output on* signal is activated and the *force output off* signal is activated, the force to output on signal has the priority.

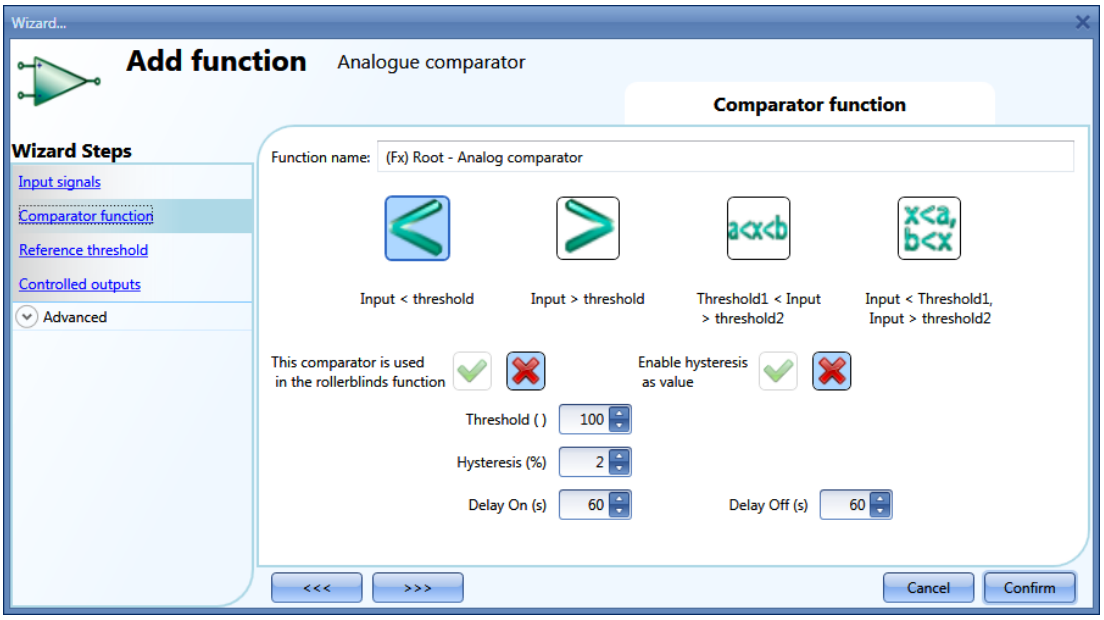
10.29 How to set an analogue comparator function

The analogue comparator function can be used to compare two values.
To add the analogue comparator function the user has to select *Basic* from the Add menu (see picture below). The new function will be added into the selected location.
The wizard relevant to the analogue comparator function can also be opened by typing Alt+A+7+A.



10.29.1 Comparator settings

In the field *Comparator function*, the user can choose the type of comparator by selecting the desired icon (in the picture below the inside limits comparator is selected).

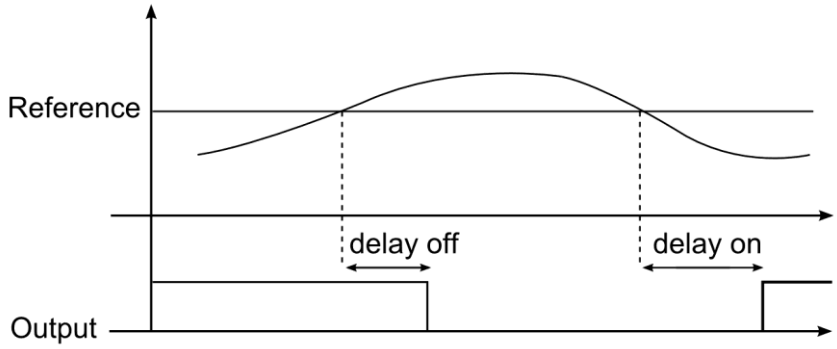


If this function is used into a roller blind function, the relevant check box has to be selected.

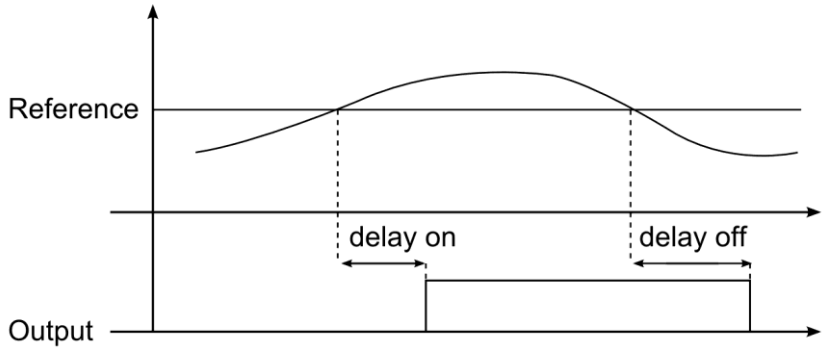
The user can also select the type of hysteresis: as a percentage of the full scale or as a fixed value (*Enable hysteresis as value* has to be checked).

The available types of comparison are the following:

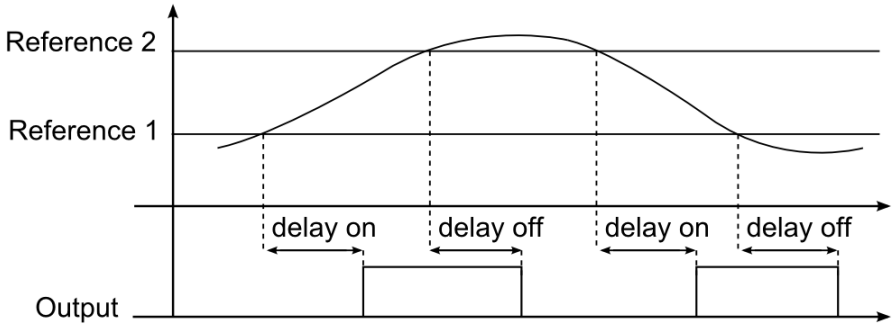
1) ***Lower than***:



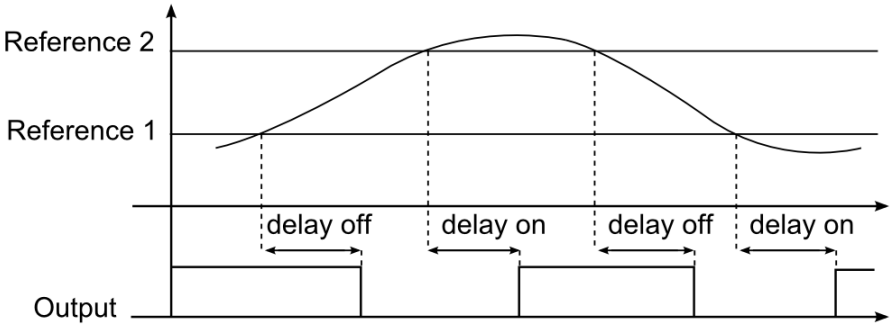
2) **Higher than**:



3) ***Inside limits***:



4) ***Outside limits***:

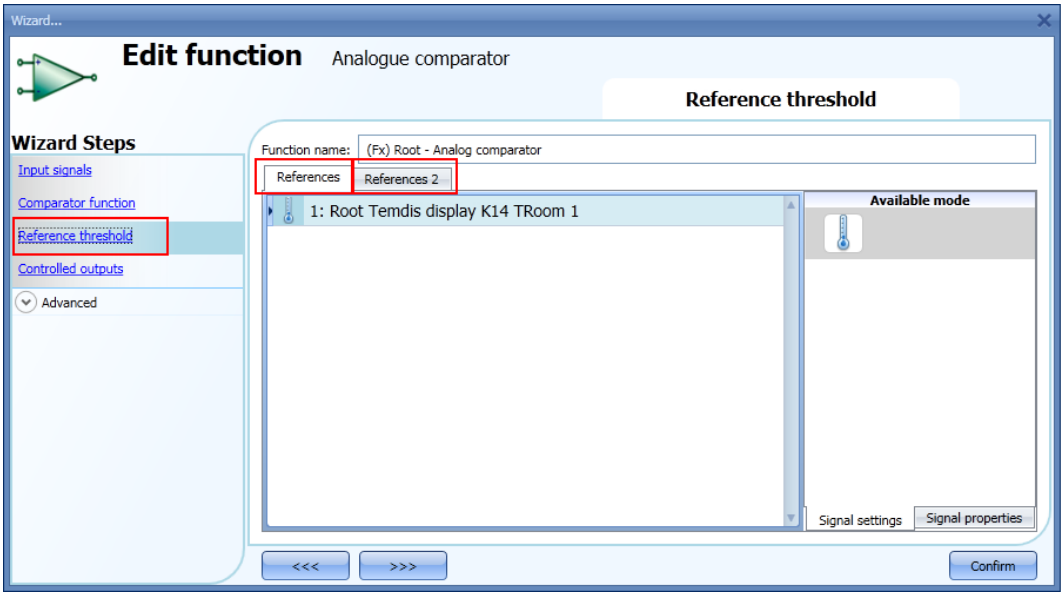


According to the comparator type selected, the user should then set the hysteresis and the threshold limits (when a comparator with two thresholds is used, the hysteresis is the same in both limits).
The *delay on* is the delay before the activation of the output when the comparison is true, the *delay off* is the delay before the deactivation of the output when the comparison is false.

10.29.2 How to set the reference values

There are two types of reference signals available in the comparator function: the first is a numeric value set as threshold in the field *Comparator function*, the second is an actual reference coming from a module (such as temperatures, light levels, energy measurement and so on) and configured in *Reference threshold*.

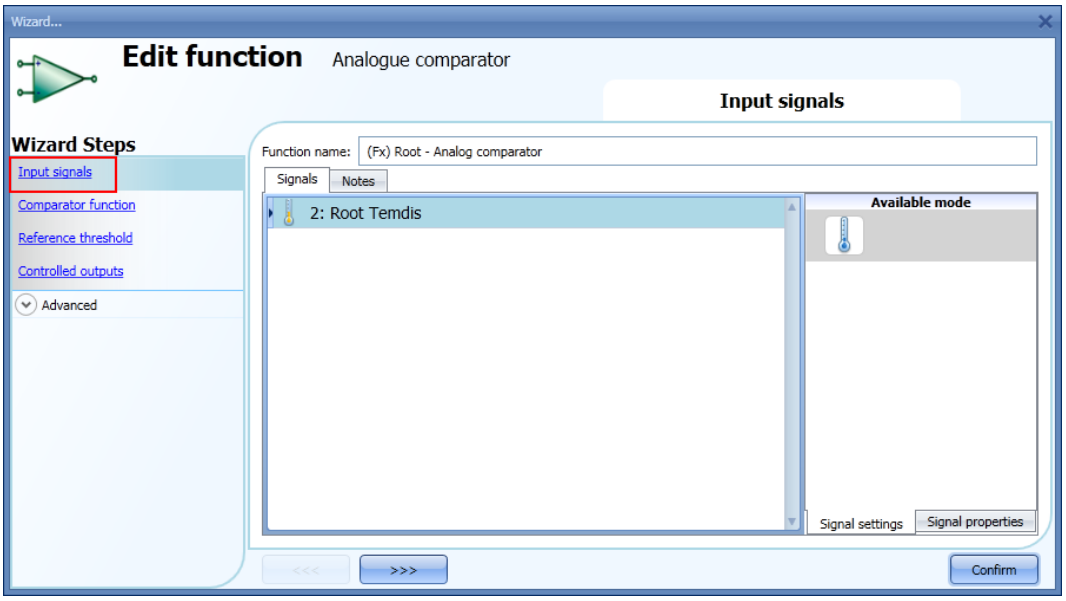
If the user does not add any actual signal in the *Reference threshold* field, the comparator works according to the threshold values configured in the field *Comparator function*.
The real reference signal has priority over the threshold limit and the latter is used only if a fault should occur to the real reference signal.



In the case of a comparator function with two reference signals (*inside limit* or *outside limit* type), *Reference 2* is always used as the high threshold limit; *Reference 1* is always used as the low threshold limit.
If the signal added as *Reference 1* (lower limit) has a higher value than the signal added as *Reference 2* (higher limit), the comparator function works as a comparator with a single reference (higher than or lower than type).

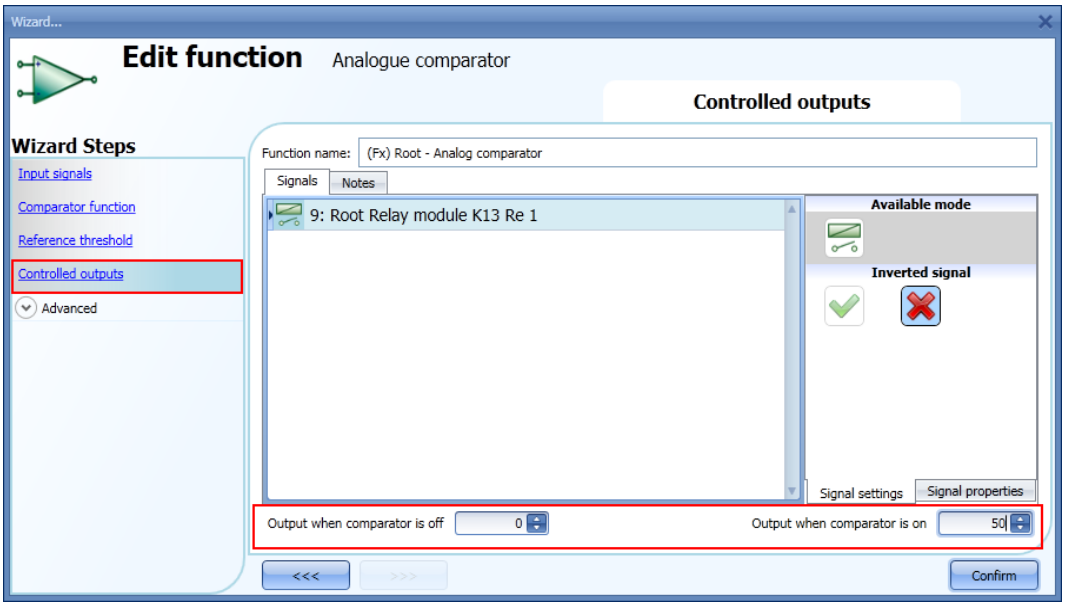
10.29.3 How to add the input signals to be compared

In the Input signals field of the wizard the user has to enter the signals to compare.
According to the signal added, the function automatically changes the units of measurement of the threshold limits.
In the picture below, a temperature signal is used as input and the unit of measurement displayed is °C/°F .



10.29.4 How to add the output signal

In the *Controlled output* field, the user can add one or more output signals which are activated according to the status of the comparator. When the function is on, the output is switched on; when the function is off the output is switched off. Once the output signals are entered, the user can also choose to use the reverse logic of the signal.



At the bottom of the window, the user can also set two analogue values for the output status of the function status when the comparator is respectively on and off.
This functionality is used when the analogue comparator status is used in other functions.
For example, the user can set an analogue comparator as shown in the picture above, which is then linked to a roller blind function. When the comparator status is true, the function output is 50, otherwise it is 0: the roller blind function will be set at 50% when the comparator is true, and then back to 0% (fully up or down) when the comparator is false.

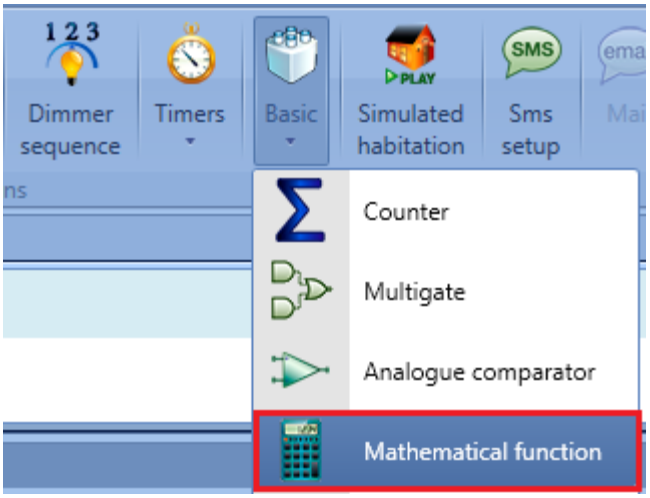
The digital output like relay, are only managed is the output status is set to 0 or 1, other values are considered as “don’t care”.

10.30 How to configure a mathematical function

The *Mathematical function* provides a set of mathematical operations on two or more analogue input signals. The output value can also be used to control an analogue output module such as the SHPOUTV224 or motor control modules (SH2RODC224, SH2ROAC224, SHDRODC230).
The available operations are listed below:

- **Average**: This operation returns the *average* value of all signals present in the *Analogue input* field*;*
- **Maximum**: This operation returns the maximum value among the input signals present in the *Analogue input* field*;*
- **Minimum**: This operation returns the minimum value among the input signals present in the *Analogue input* field*;*
- **Difference**: This function returns the difference between the first and second input added in the *Analogue input;*
- **Linearization**: It is used to calculate an analogue output by means of a user defined curve. An example of it is the calculation of a temperature setpoint that is changing according to the outdoor temperature.

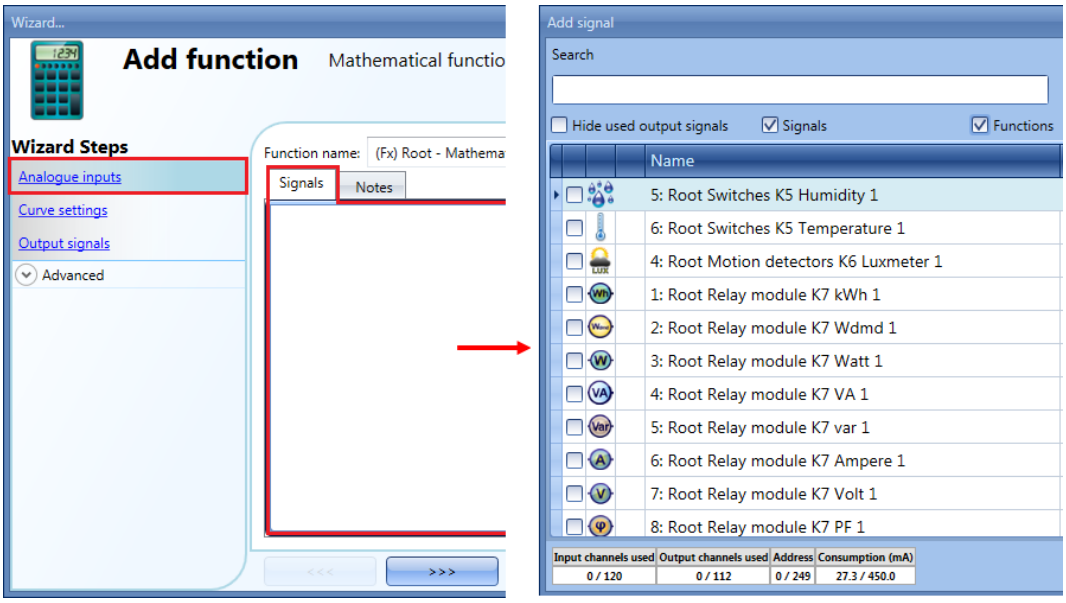
To set up a mathematical function, the user has to select *Basic* from the Add menu (see picture below). The new function will be added into the selected location.



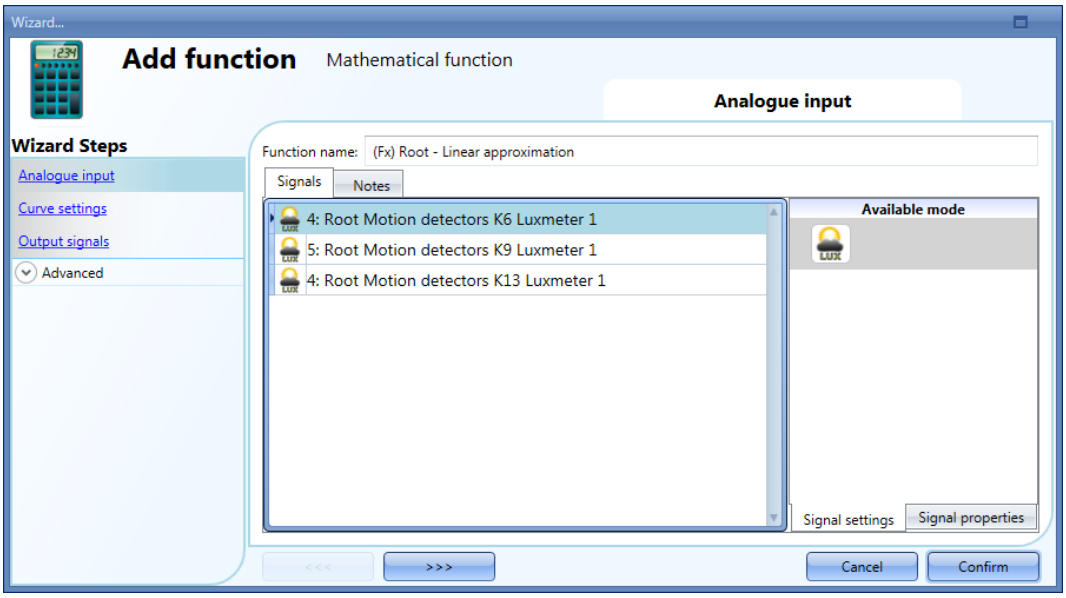
10.30.1 How to add the input signals

To add the input signals for the *Mathematical function*, the user has to click on *Analogue input*, then double click on the *Signals* window and select the input signal from the list (see picture below).

The input signal may be a physical signal such as a temperature, humidity, lux sensor, etc.. or an analogue function output (e.g. *Analogue output* function etc..) configured in the project. The signals added in this field must be all the same type (e.g. all temperature or humidity ones). Up to 50 input signals can be added.

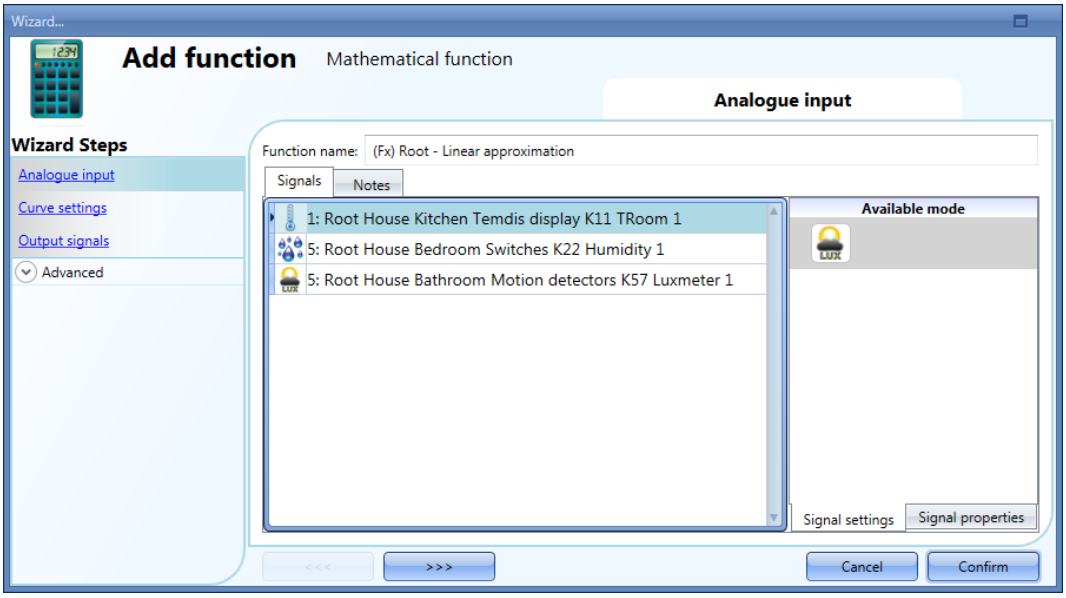


A correct selection of the input signals is shown in the picture below:



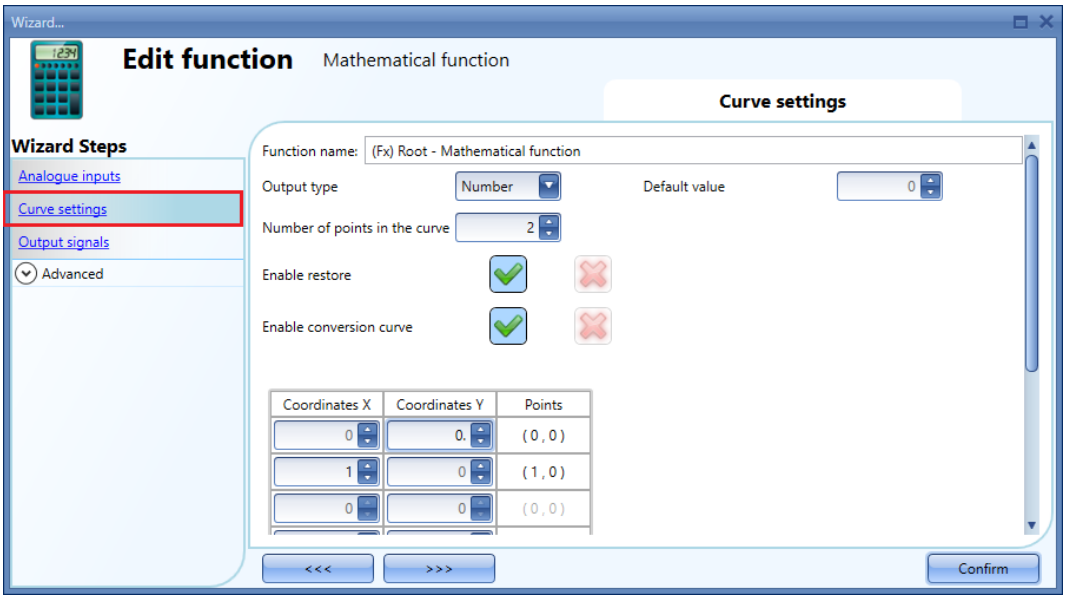
All the added signals are of the same type (three lux-meters).

In the example shown below, there are different types of analogue signals in the same *Mathematical function*, therefore the generated output signal is not a valid one.



10.30.2 How to set the curve

In the *Curve settings* field of the wizard, the user can configure the parameters of the conversion curve and manage the output status.



Output type: In this field the user can select the unit of measurement and the icon type applied to the output of the *Mathematical function*;

Default value: In this field the user can set up the default value applied to the output when all the input signals value are not (or yet) available;

Number of point in the curve: In this field the user can select the number of points used to build the curve. By default, the minimum value is 2 points (to build a straight line), while the maximum value is 10 points. These settings are available if *Enable conversion curve* is enabled;

Enable restore: When this option is enabled (the green "V" is selected), the last value of the output is maintained after a system start-up. Otherwise (when the red "cross" is chosen), the default value is applied at start-up;

Enable conversion curve: When this option is enabled (the green "V" is selected), the conversion curve is enabled for the *Mathematical function.* Otherwise (when the red "cross" is chosen), the conversion curve is not enabled.

10.30.3 How to generate an output according to the *conversion curve*

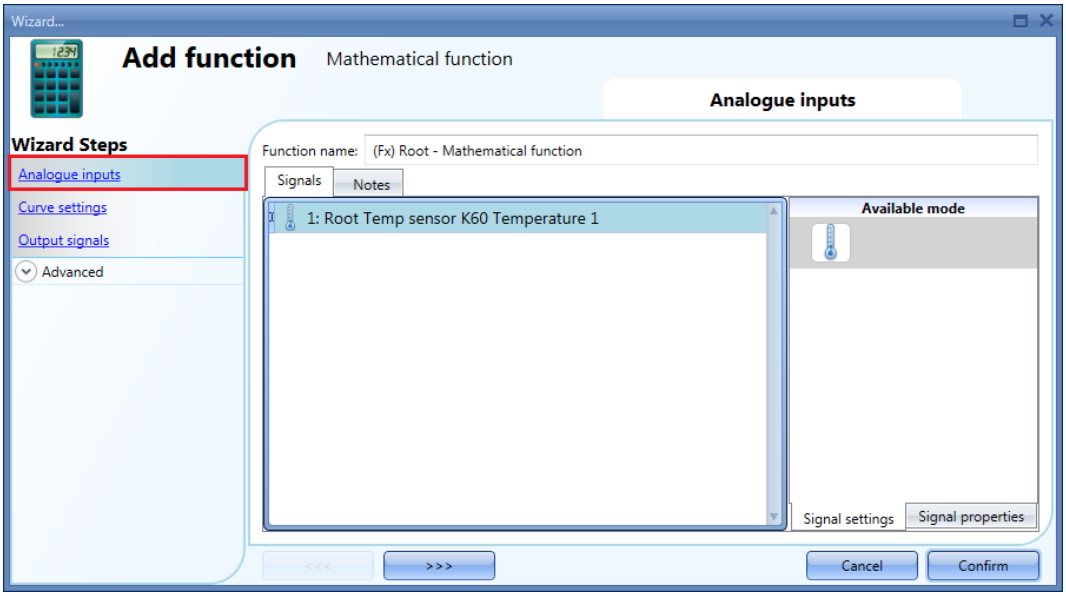
By means of the linear conversion, the user can obtain an analogue output signal, according to a predefined curve. In the *Curve settings* field, the user can set the output value according to the input value. When the input signal value is equal to the value indicated in the *Coordinates X* field, the output of the function will be the value indicated in the *Coordinates Y* field. When the value of the input signal is in-between two points, the output value will be calculated according to the linear conversion (straight line). The user can add up to 10 points and the minimum number is 2 points.

N.B. From the UWP 3.0 Tool 5.3.3 version or higher, if *Signed number* unit is used in *Output type* field, the negative outputs value can be managed.

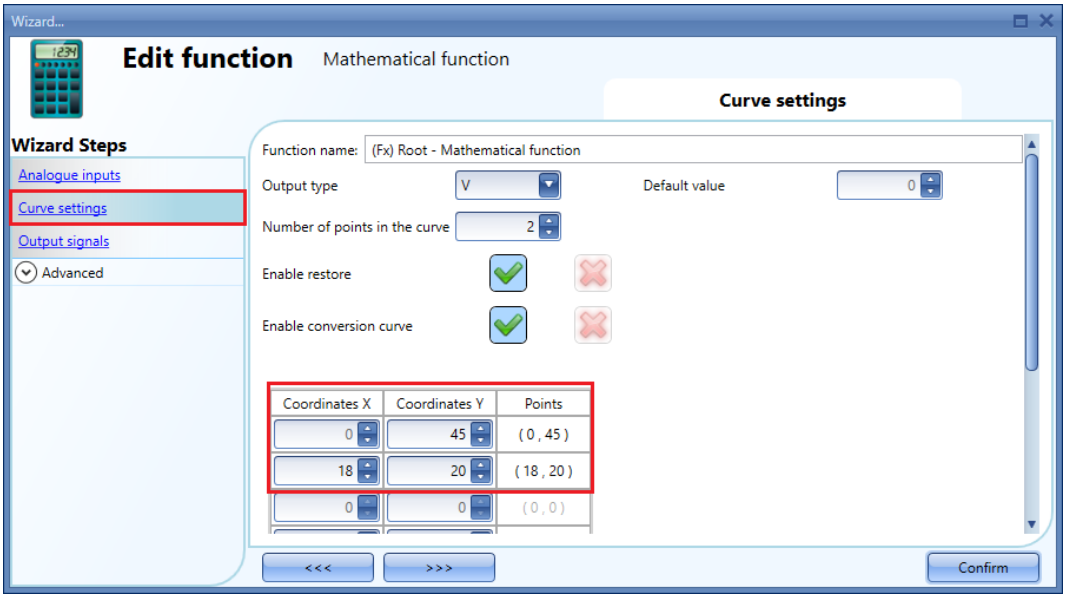
10.30.3.1 Example: heating setpoint compensation

In the example shown below, the *Mathematical function* is used to compensate the water temperature of a boiler according to the outdoor one. With a lower external temperature, the hot water temperature will be increased to obtain a better response of the heating system.

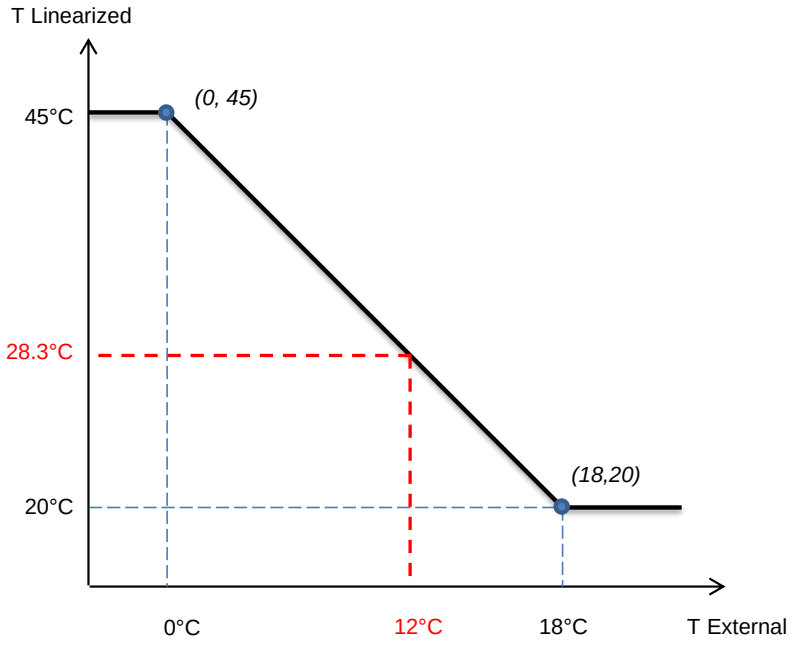
In the *Analogue input* field, the user has to add an outdoor temperature value read from a temperature module (e.g. BSI-TEMANA-U).



When the external temperature is equal or lower than 0° C, the hot water temperature will be maintained at 45° C. If the external temperature is between 0°C and 18° C, the hot water temperature will be adjusted according to a straight line as shown in the graph below. If the external temperature is higher than 18° C, the hot water temperature will be maintained at 20° C.



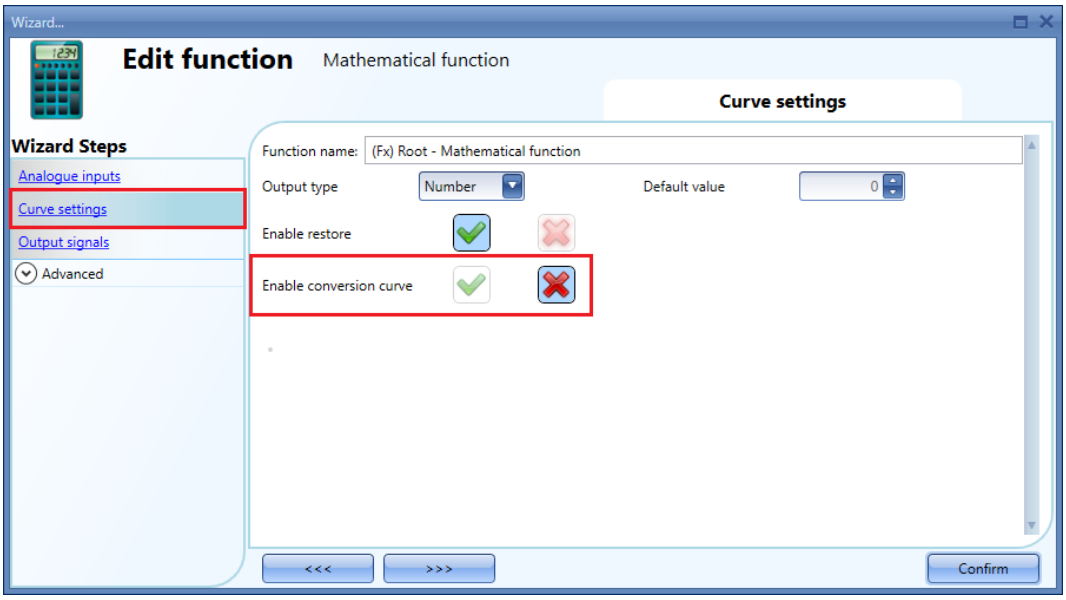
If the external temperature value is 12°C, the linearized value of the *Mathematical function* will be 28.3°C, as shown in the graph below:



10.30.4 How to use a Mathematical function without a conversion curve

If the user disables the conversion curve, the output signals such as *Average input values linearized + offset*, *Minimum linearized value*, *Maximum linearized value* are equals to *Average input values + offset*, *Minimum input value* and *Maximum input value* output signals.

The user can disable the conversion curve by clicking on the red cross icon on the *Enable conversion curve* field:

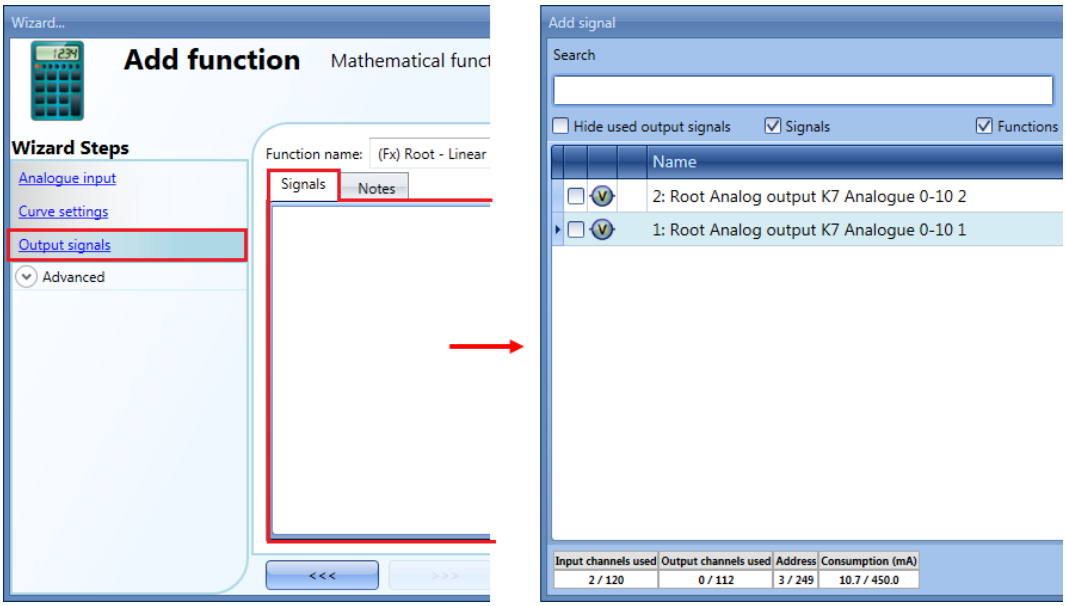


The user can still change the value of the fileds *Output type*, *Default value, and Enable restore.*

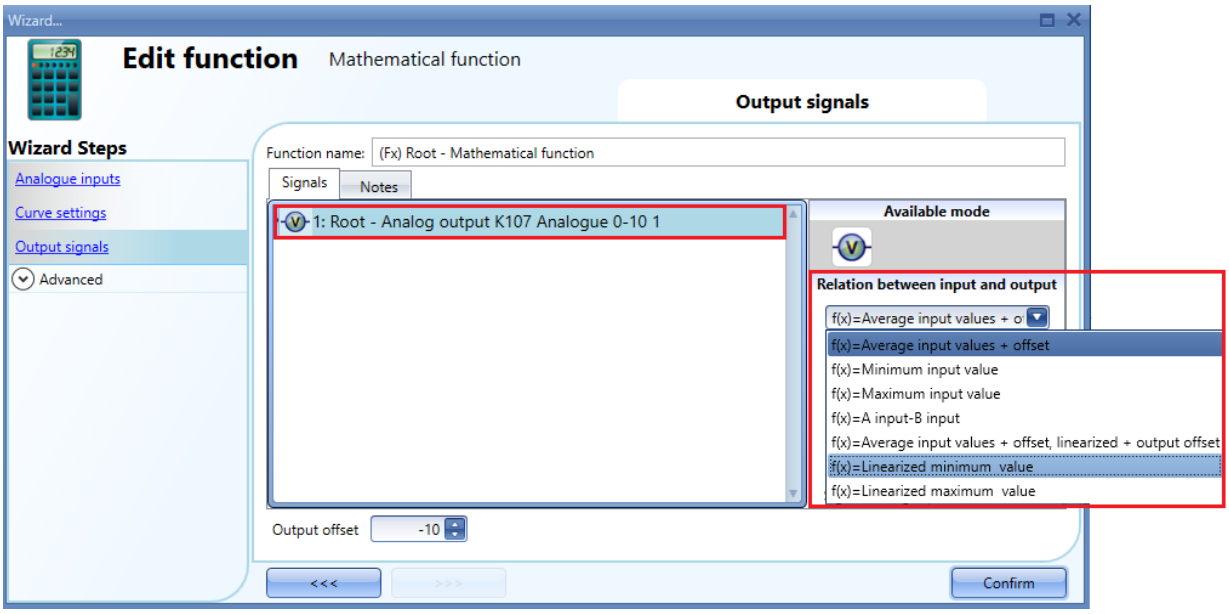
10.30.5 How to add an analogue *output signals*

To add an analogue output signal, the user has to click on *Output signals*, then double click on the *Signals* window and select the output signal from the list (see picture below).

The available outputs are the signals managed by the modules: SHPOUTV224, SH2RODC224, SH2ROAC224, SHDRODC230. For example, the analogue output value of the SHPOUTV224 is calculated using the output of the *Mathematical function* as the virtual range in the *Signal properties* of the module.

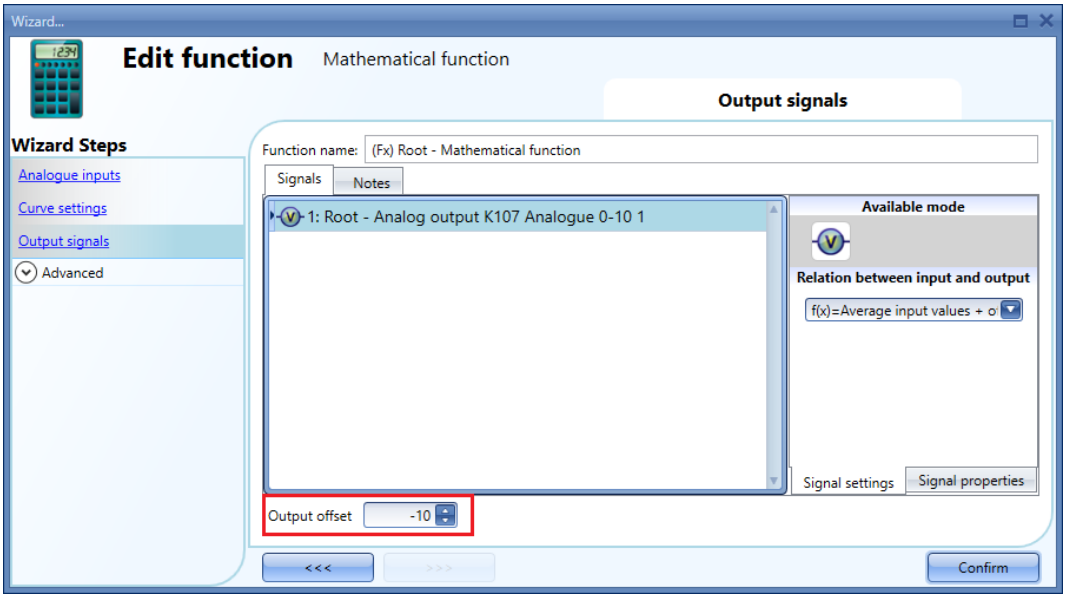


The user can then select in the *Relation between input and output* field, which calculation has to be used to set the output.



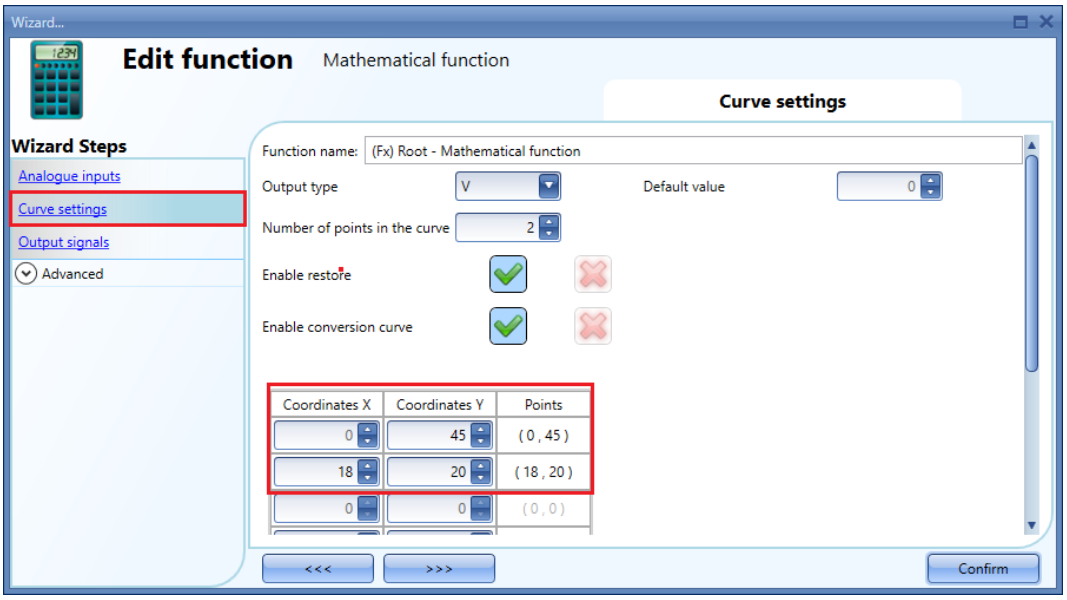
The 0-10 V output value is calculated based on a linear relation between the input and output values of the *Mathematical function*.

Output offset:
In this field the user can set the value that is added to the output signal value. The offset value can be positive or negative.

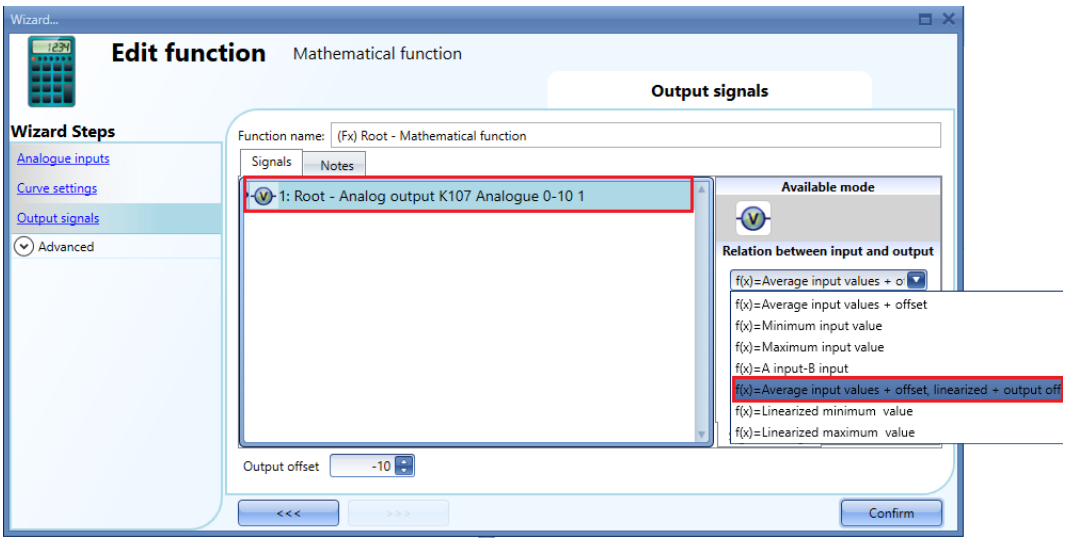


10.30.6 How to use an analogue output signal to regulate the valve position

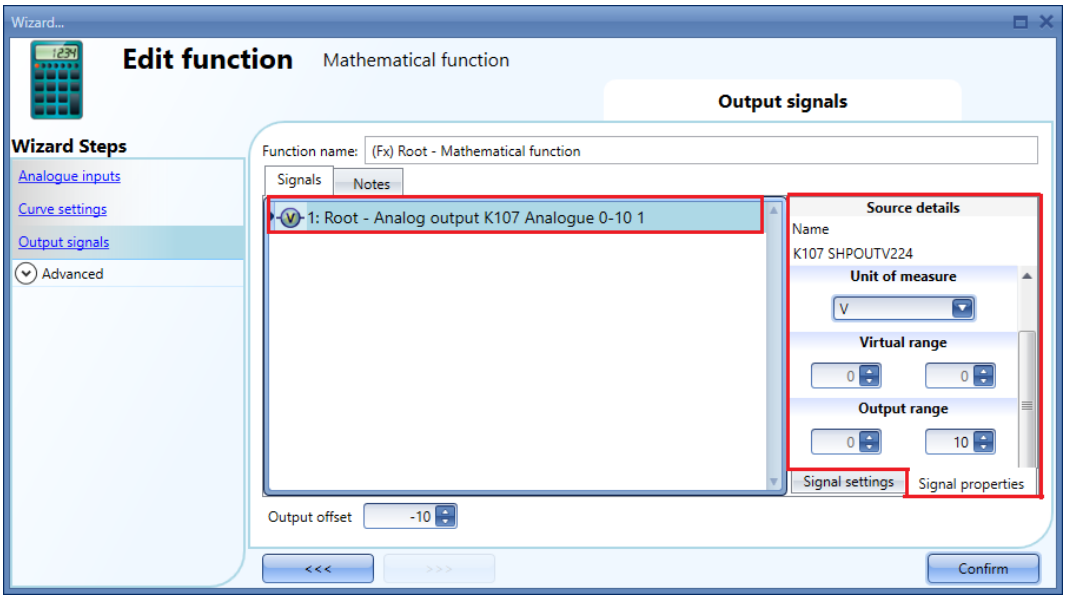
In the example shown below, an external temperature sensor is used as a input signal.
The external temperature is linearized with a conversion curve and the Volt unit is selected as Output type: the icon will also be changed accordingly. See the picture shown below for the curve settings:



In the *output signals* window, the first output of SHPOUTV224 module is added, then, in the *Signal settings* window is assigned the *Average input values linearized + offset* as the *Relation between input and output*. See the picture shown below:



The 0-10 V output value is calculated based on a linear relation between the minimum and maximum values of the function: in the Signal properties window of the output, the user can set the Virtual range (0 - 100) and the Output range (0–10 V) according to the project, as explained in the picture below:

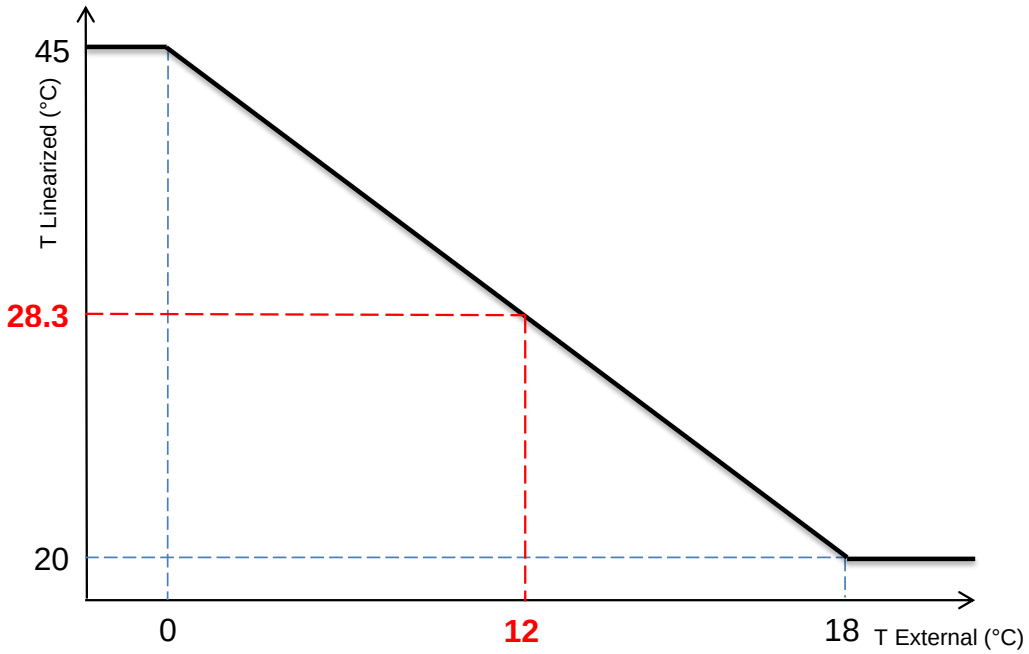


If the minimum and maximum virtual values of the function are 0 and 100, and the virtual output value is 30, the voltage output of the SHPOUTV224 will be 3V
In the example shown below, the linearized output values are between 20°C (min value) and 45°C (max value), so the voltage output is from 2.0V(min) up to 4.5 V(max).

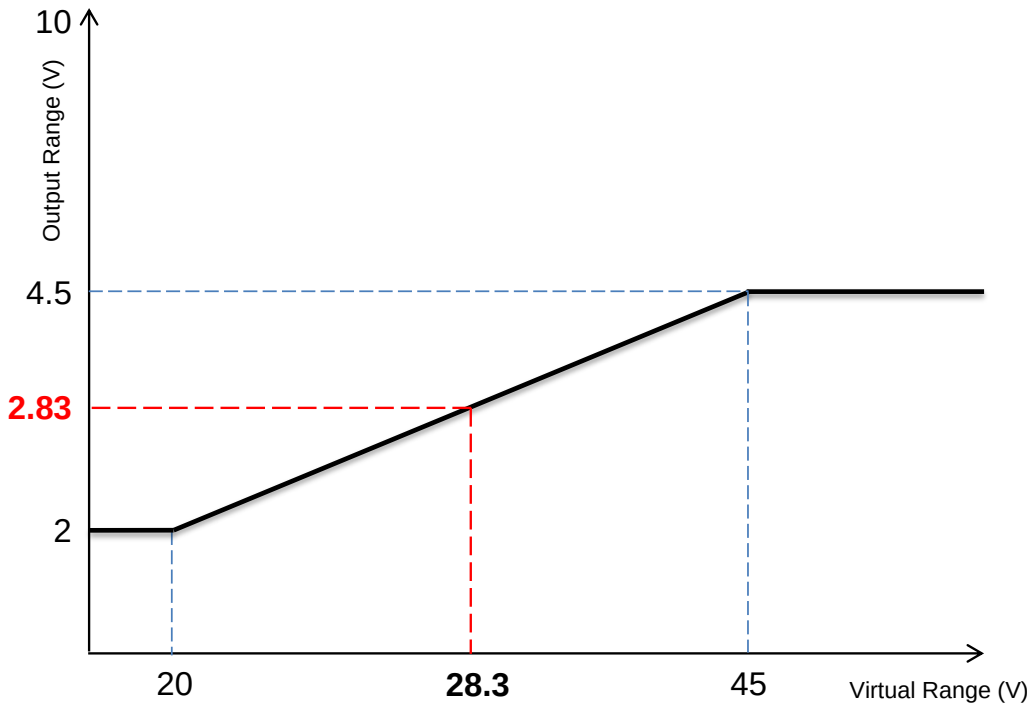
If the output of the *Mathematical function* is 28.3°C (equal to 28.3V virtual value), the voltage output of the SHPOUTV224 will be 2.83 V.

In the graph below it is shown the relation between the output of *Mathematical function* and the Voltage output of the SHPOUTV224 module.

The linear conversion from the external temperature to the linearized temperature

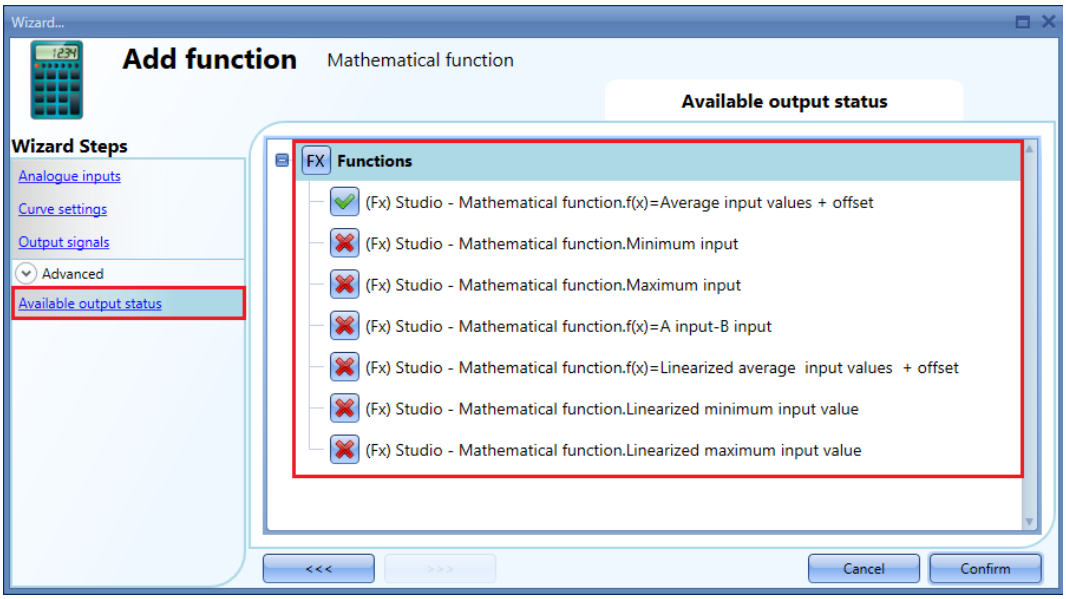


The linearized values are used as *Virtual Range* for the *Output Range* (in Volts).



10.30.7 How to use the calculated values

In the *Available output status* field, the user can decide which value out from this function he needs to use in the configuration of his system, for example as an input in another function or in a sequence. Like all the other advanced functionalities, this one also has to be enabled.



Function

f(x)=Average input values + offset
It is the result of the instantaneous average calculation of the input signals plus the offset value;

The other statuses that can be enabled and used in other functions are:

f(x)=Minimum input value
It is the instantaneous minimum value among the input signals (without offset)

f(x)=Maximum input value
It is the instantaneous maximum value among the input signals (without offset)

f(x)=A input – B input
It is the difference between the signals A and B: the first two signals of the input list without offset. When one of the two inputs is not valid or not present, the value is set to 0

f(x)=Average input values linearized + offset
It is the result of the linear conversion of the *Average input value + offset* value

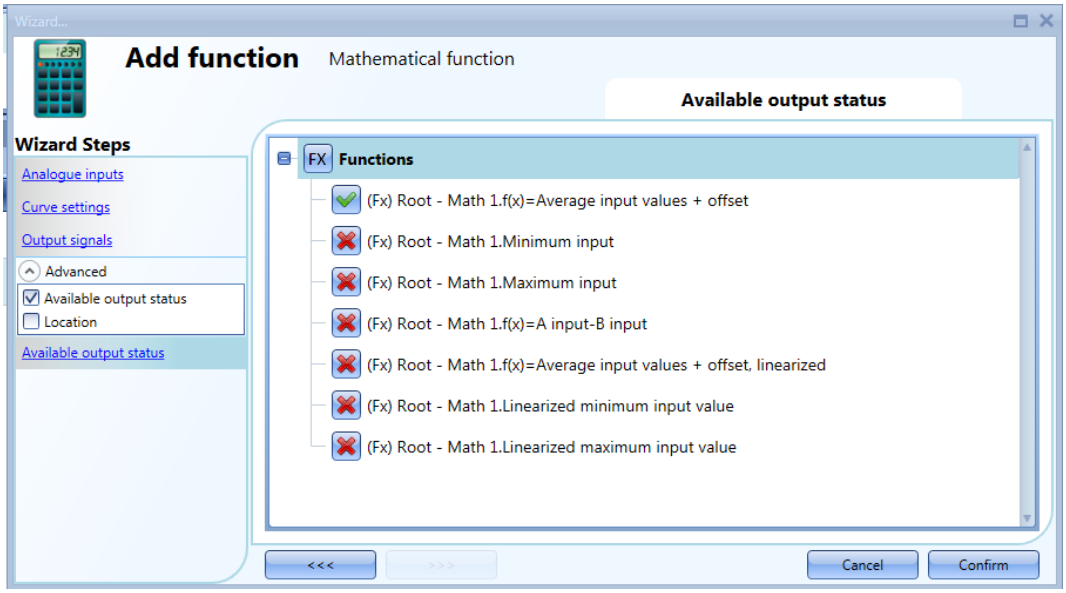
f(x)=Minimum linearized value
It is the result of the linear conversion of the *Minimum input value*

f(x)=Maximum linearized value
It is the result of the linear conversion of the *Maximum input value*

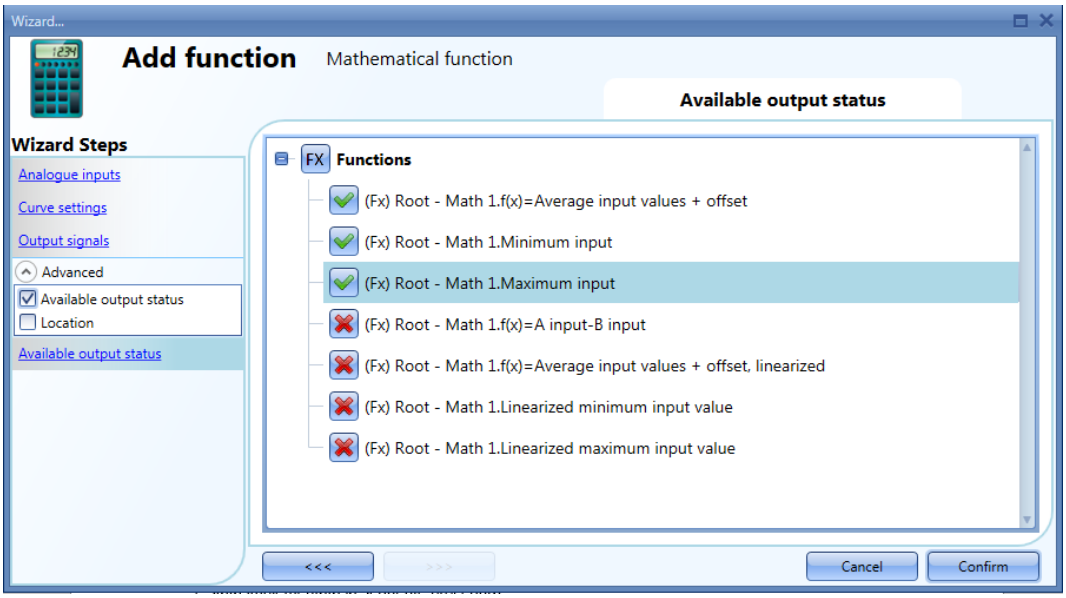
10.30.8 How to log additional function status in the Database

This procedure has to be followed in order to log in the Database the addition status of the functions (for example the additional status of math function)

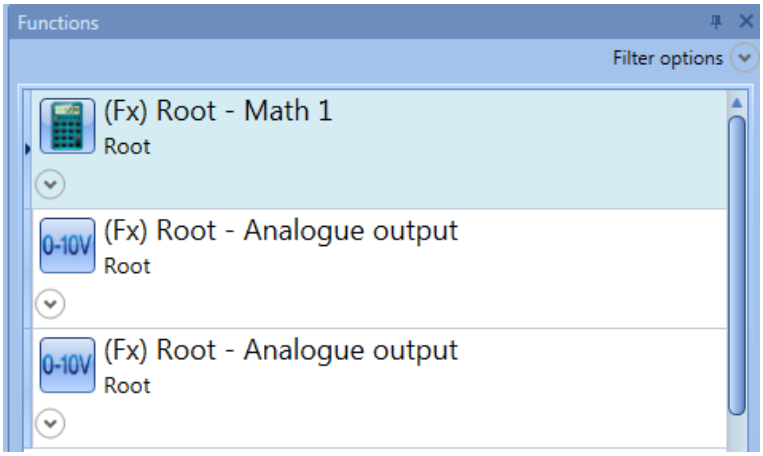
1) Add a function (es. "Math1")
   a. Select the advanced menu and enable the “available output status”



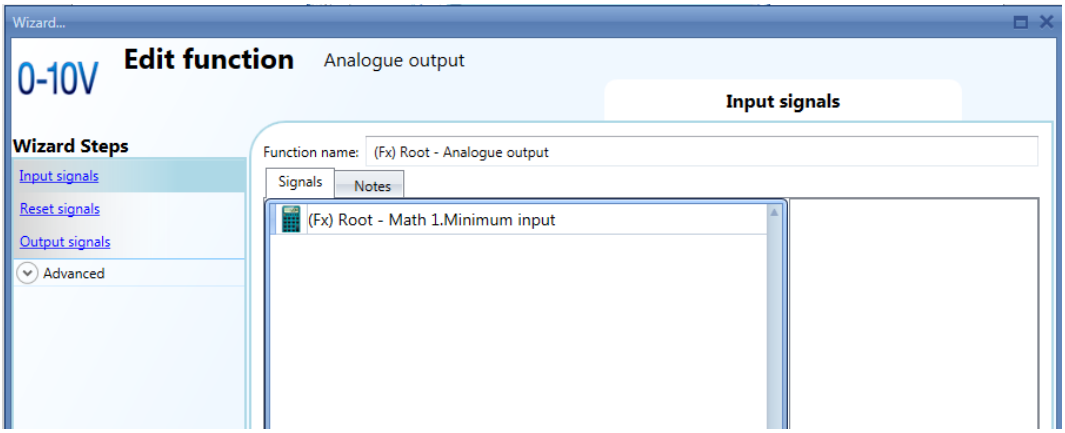
   b. Select the additional function status (es. “Minimum input” and “maximum input”)



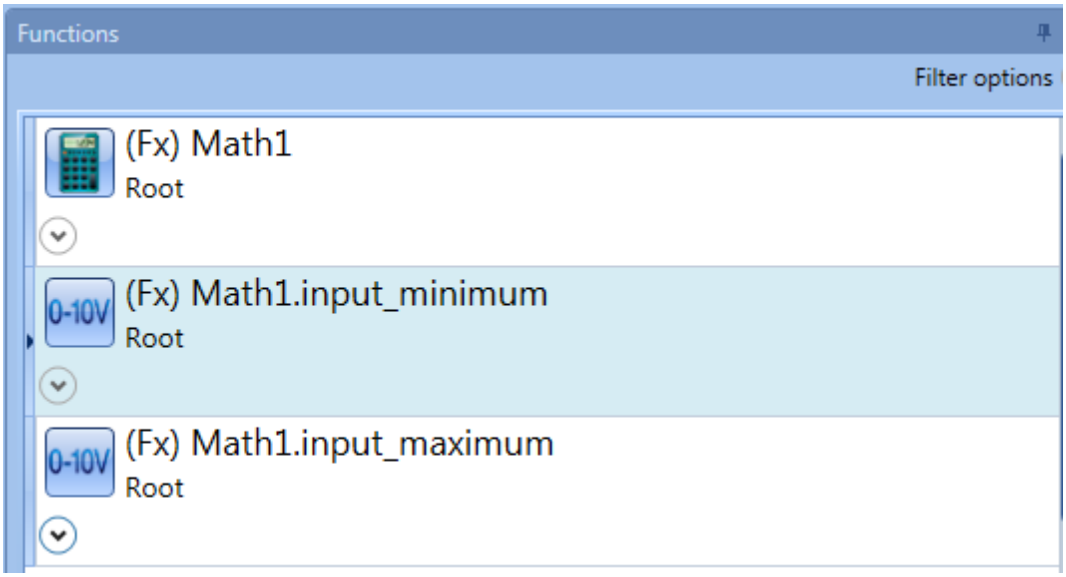
2) Add one AnalogOutput function for each additional status selected at step 1 b.



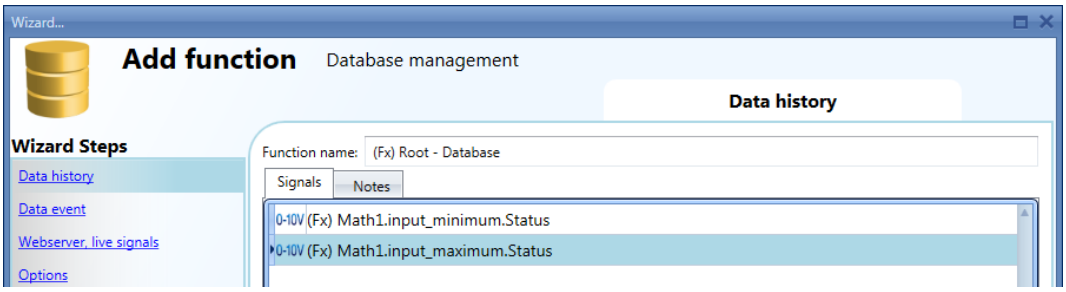
a. For each AnalogOutput function, add as input signal one of the additional function status selected at step 1



b. Change the name of the AnalogOutput functions in order to have a clear description of the additional status (es. function_name" + "additional_status")



3) In the DB select the AnalogOutput functions added at step 2

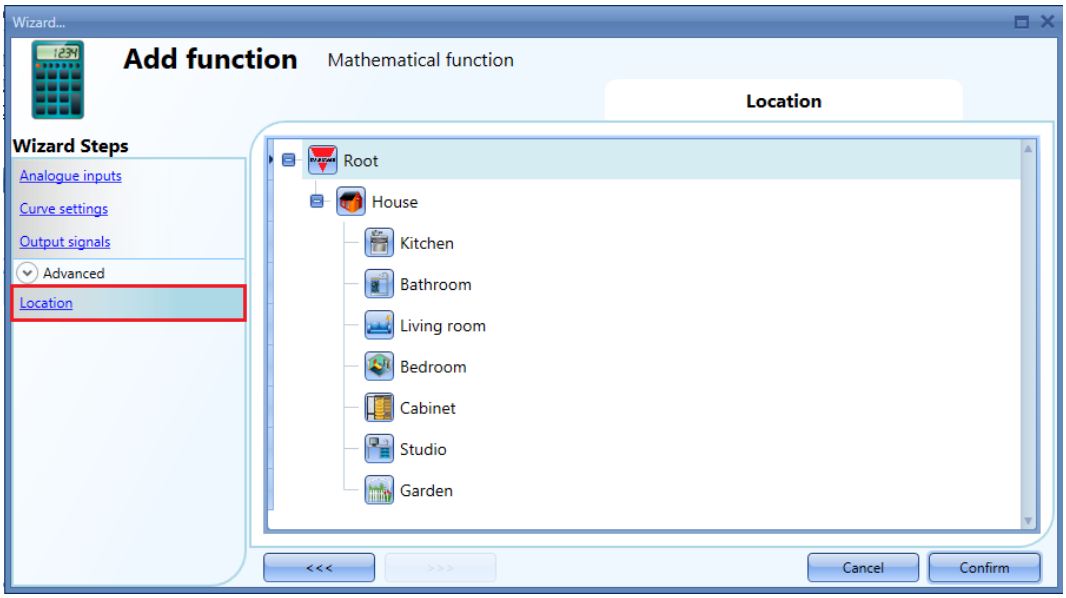


4) Save and write the configuration file in the UWP 3.0

5) The report file will show the logged value in the field “counter 1” of the AnalogOutput function

10.30.9 How to change the location of a function

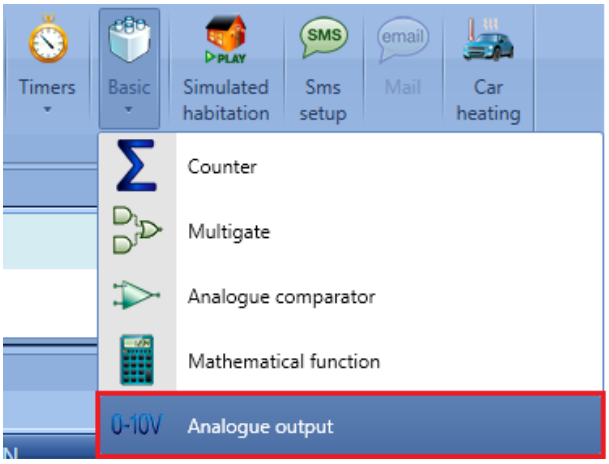
To access the *Location* menu, the user has to enable it in the *Advanced* section.
From this window, the location where the function is placed can be changed by simply selecting and clicking on the required one.



10.31 How to configure an analogue output function

The *Analogue output* function provides a driver interface and reset feature which can be used to control an analogue output module. It is mainly used for motor driving where it is periodically necessary to reset the position (fully open or fully closed).

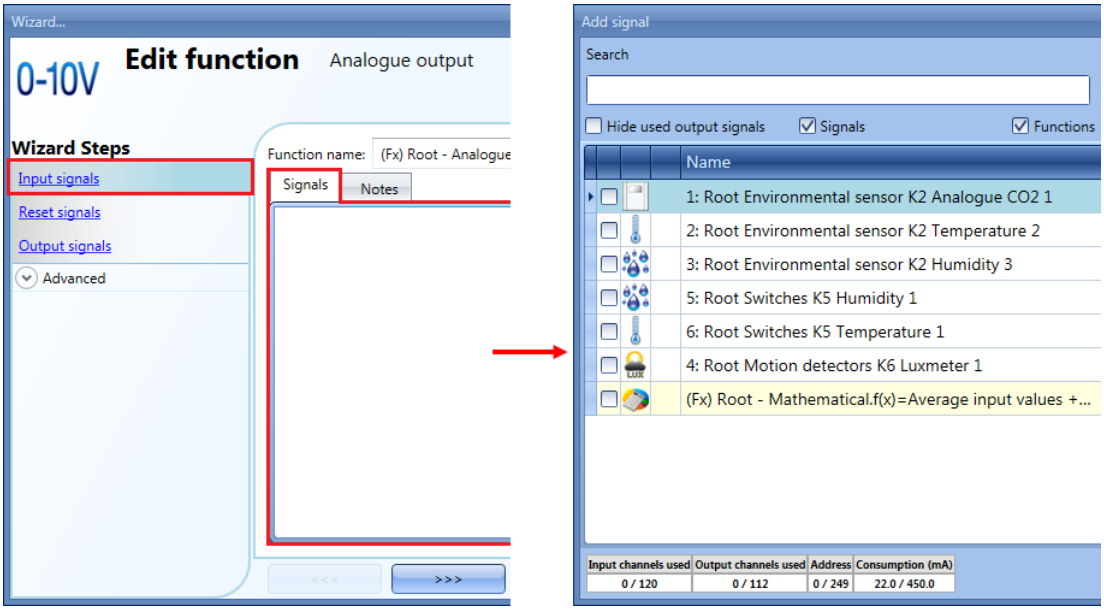
To set up an *Analogue output* function, the user has to select *Basic* from the Add menu (see picture below). The new function will be added into the selected location.



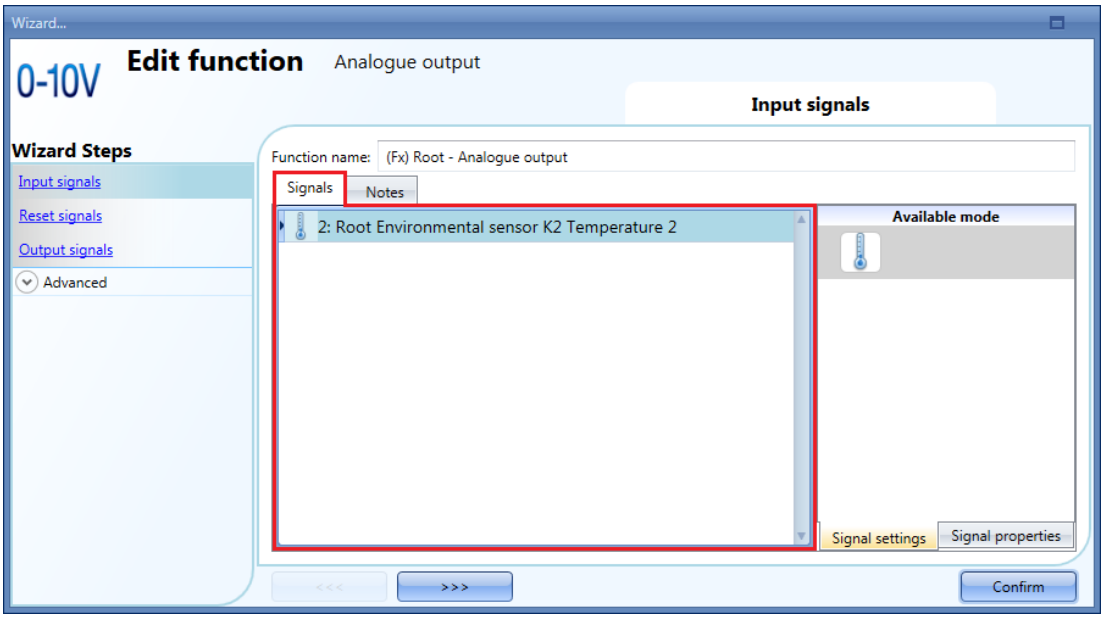
10.31.1 How to add the input signal

To add the input signal for the *Analogue output* function, the user has to click on *Input signals*, then double click on the *Signals* window and select the input signal from the list (see picture below).

The input signal may be a physical signal such as temperature sensors, humidity sensors, lux sensors, etc.. or an analogue function status (e.g. *PID Control* function, *Zone temperature* function etc..) configured in the project.



Only one input signal can be added to the *Analogue output* function



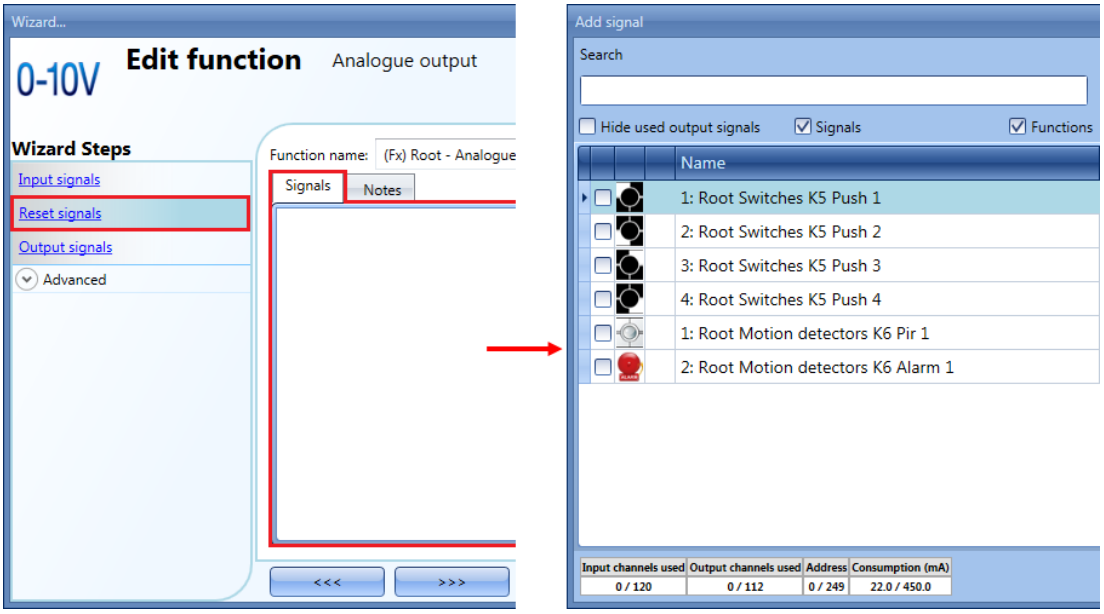
10.31.2 How to reset the Analogue output value

The Analogue output function is mainly used for motor driving where it is periodically necessary to reset the position. When a rollerblind module such as SH2ROAC224, SH2RODC230 and SHDRODC230 is used in the *Analogue output* function, the user needs to manage the Reset.

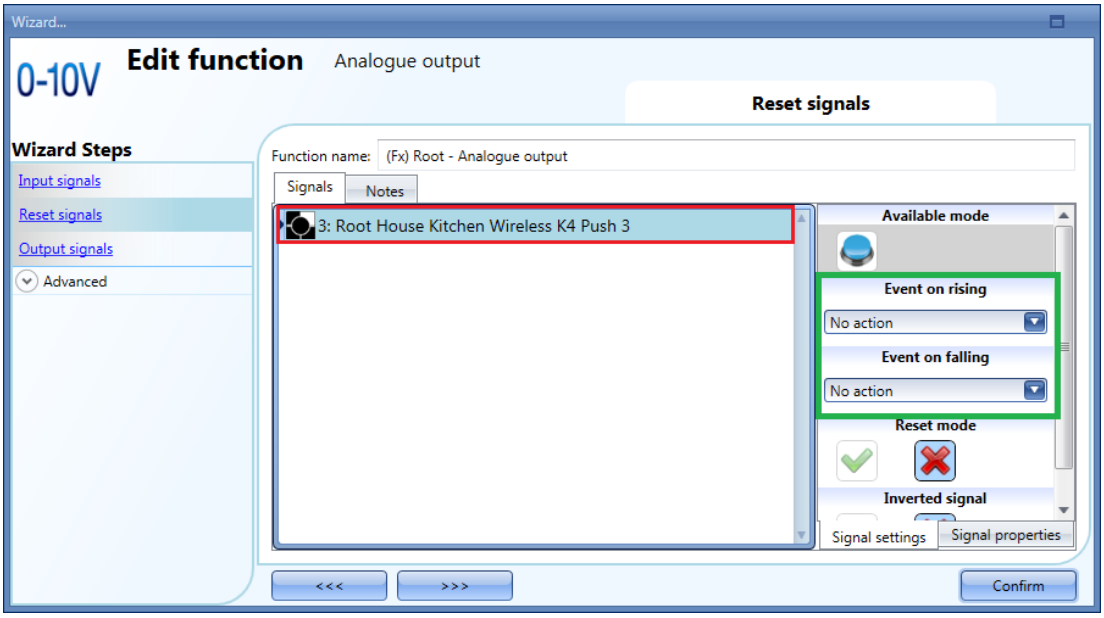
Let's say, for example, that the *Analogue output* function is used to regulate the heating and/or cooling of a zone with a *PID Control* function as an input signal. The output value of the *Analogue output* function will vary depending on the output of the *PID Control* function, but the continuous fluctuations will generate an error on the motor position.

N.B. Reset to 0/100% works only if the position of the output coincides with the 0/100%:

To reset the output value, the user has to click on *Reset signals*, then double click on the *Signals* window and select the input signal from the list (see picture below).
The reset signals may be physical signals such as pushbutton or switch, or functions like a calendar.
Up to 10 *Reset signals* can be added.



If a level signal is selected, such as a switch or a function, the available actions are **Event on rising** (i.e. when the switch is closed or the function is activated), **Event on falling** (i.e. when the switch is open or the function is deactivated), or both (see the green box in the picture below).

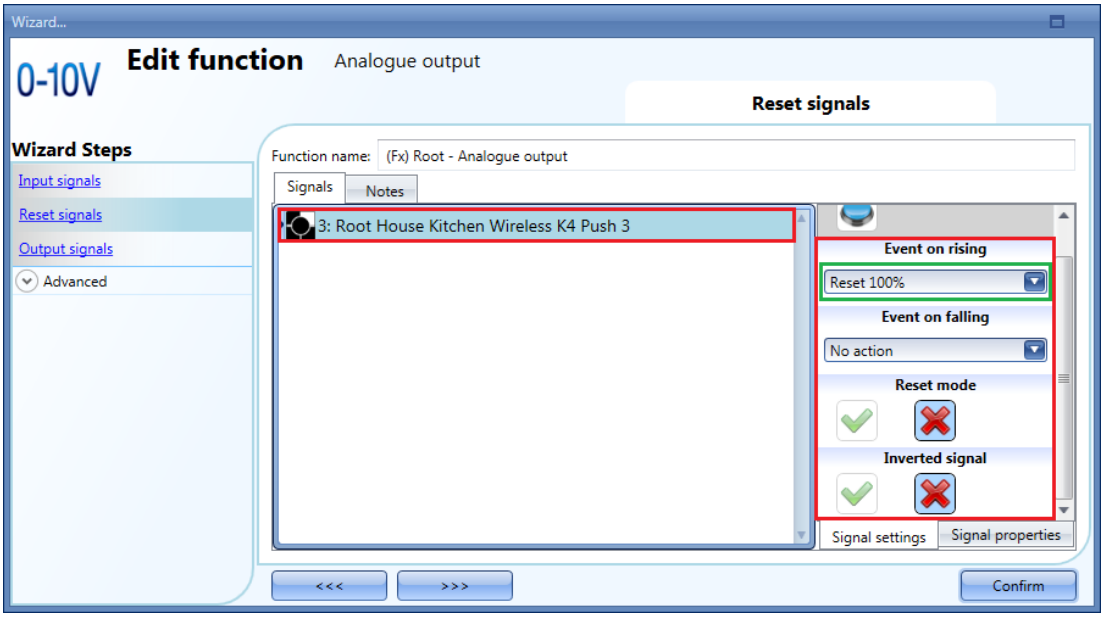


The available actions for *Event on rising* or *Event on falling* are listed in the tables below:

Reset action	
No action	No action will be carried out if the push button is pressed/released or the function is activated/deactivated
Reset 100%	The reset signal (when the push button is pressed/released or the function is activated/deactivated) will set the output value to 100%
Reset 0%	The reset signal (when the push button is pressed/released or the function is activated/deactivated) will set the output value to 0%

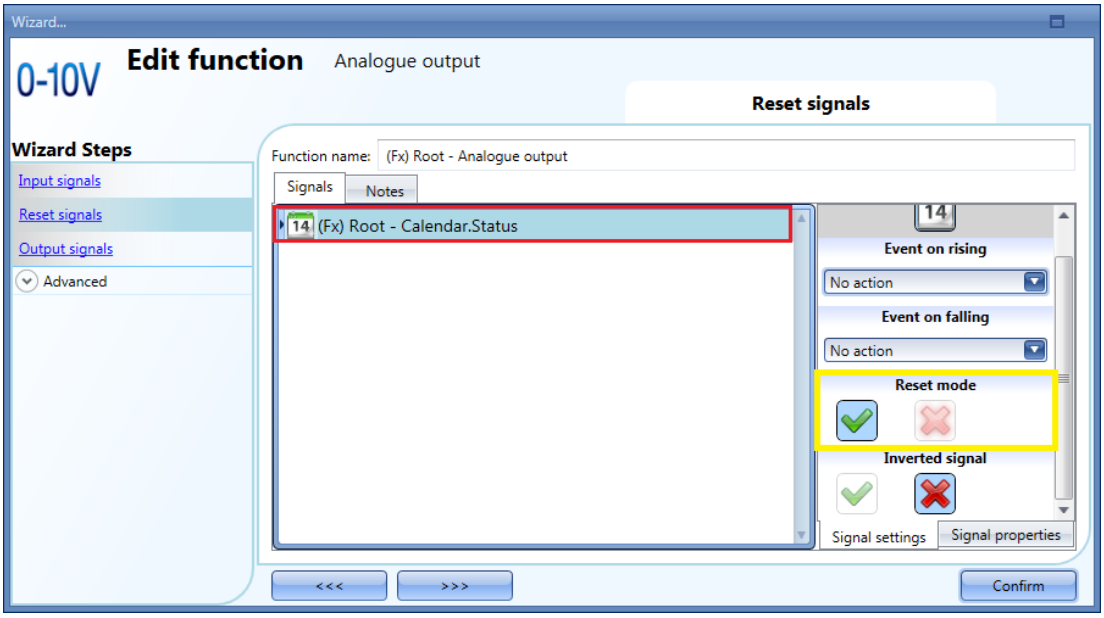
The inverted logic for the input signal can be enabled by selecting the green V in *Inverted signal.*

For example, in the picture below, as soon as the push button 3 is pressed, (*Event on rising is set to* Reset 100%), the reset signal is activated and the output value is reset to 100%.



10.31.3 How to force the Reset output

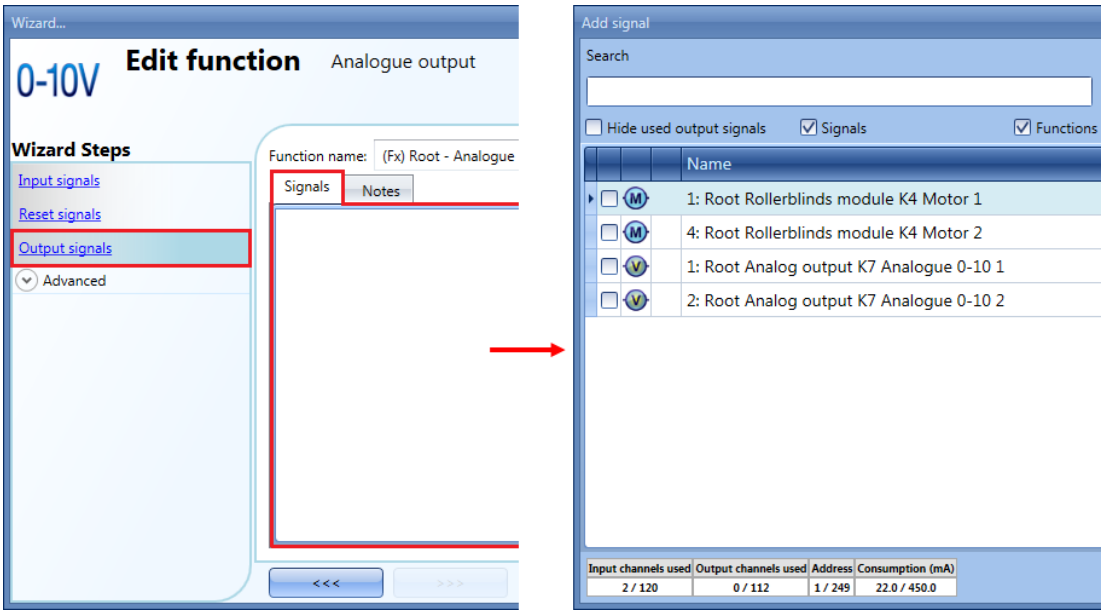
If the *Reset mode* is not enabled, Reset to 0/100% works only if the position of the output coincides with the 0/100%: if the user wants to force the Reset output, regardless of all the other signals used in the function, he must click on the signals window and select the real signal to use (see yellow box in the picture below), then in *Reset mode* select the green “V” icon to force the output with this signal.



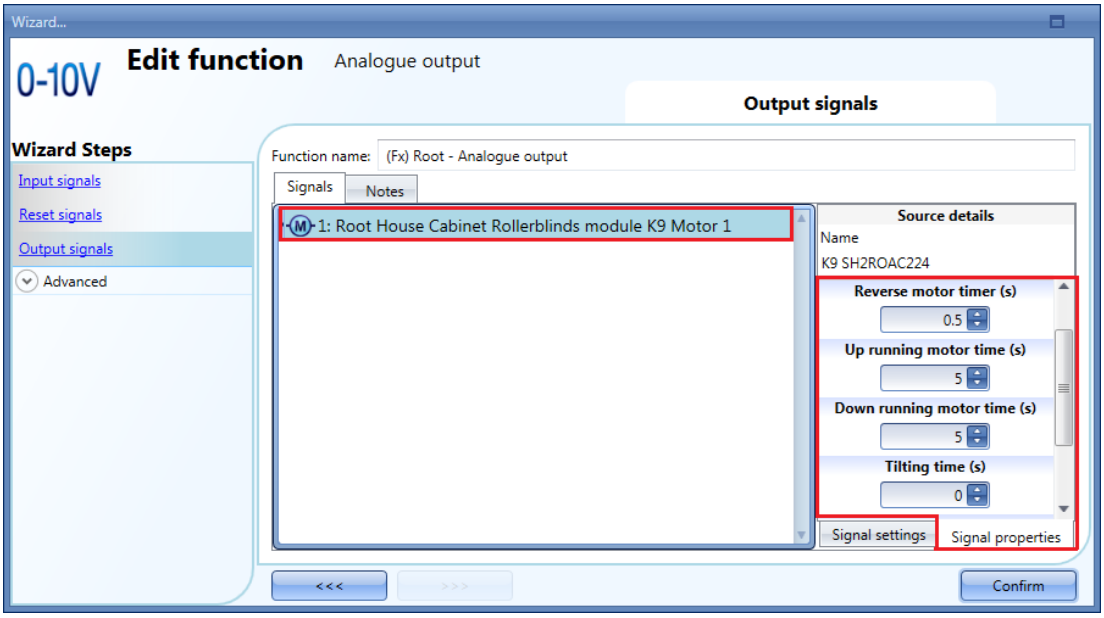
10.31.4 How to add an analogue output signal

The outputs are the signals from the rollerblind modules such as SH2ROAC224, SH2RODC230 and SHDRODC230. The Analogue output function may also manage all the Smart Dupline analogue output modules such as SHPOUTV224 and the dimmer modules, but the reset control cannot be managed for the SHPOUTV224 and dimmer modules.

To add an analogue output signal, the user has to click on *Controlled outputs*, then double click on the *Signals* window and select the output signal from the list (see picture below).



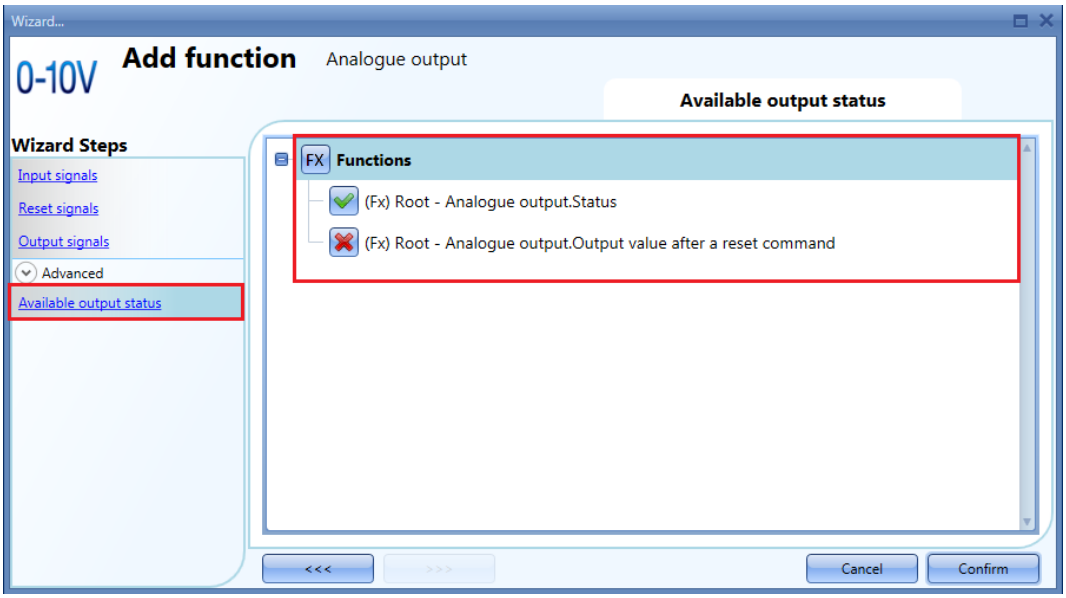
The running parameters of the rollerblind modules can be set directly from this wizard, in the *Signal properties* tab window, without the need to re-open the wizard of the module.



N.B The user must set the *Up running motor time* and *Down running motor time* for the Reset command: any value set in *Tilting time* will not be used.

10.31.5 How to enable the function diagnostic

In the *Available output status* the user can decide which status from the *Analogue output* function he needs to use in the configuration of his system, for example as an input in another function or in a sequence. Like all the other advanced functionalities, this one also has to be enabled.



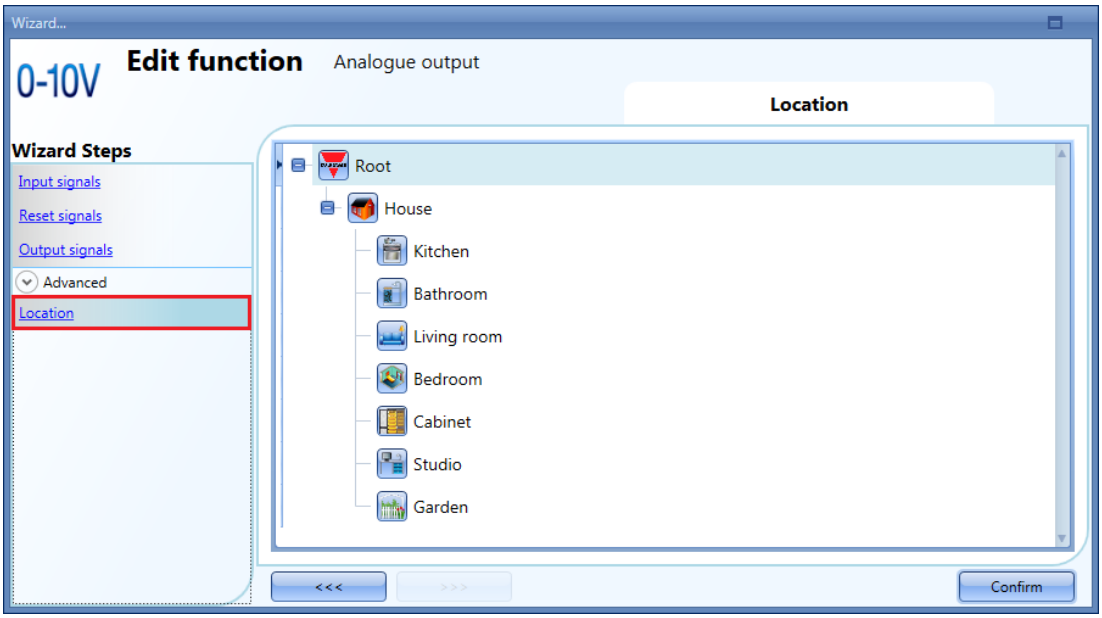
The statuses that can be enabled and used in other functions are:

Analogue output.Status*: This output status shows the analogue output value of the function*

Analogue output.Output value after a reset command: *Analogue output value after a reset command*

10.31.6 How to change the location of a function

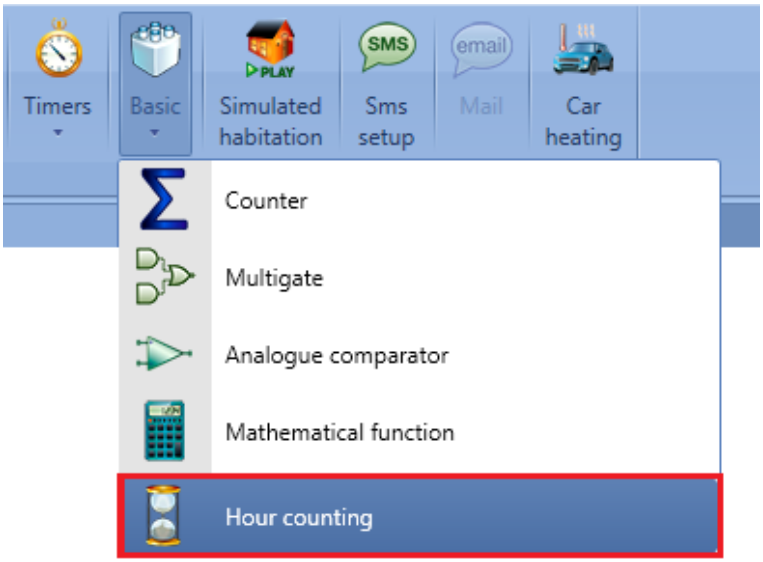
To access the *Location* menu, the user should enable it in the advanced functionalities.
From this window the location where the function is placed can be changed by simply selecting and clicking on the one required.



10.32 How to configure an hour counting function

The purpose of the *Hour counting* function is to count the hours a function output has been ON, since the last reset. This is typically used in light control for preventive replacement of fluorescent light tubes before they burn out. It is also used for keeping track of HVAC pump running hours for early replacement and for planning maintenance.

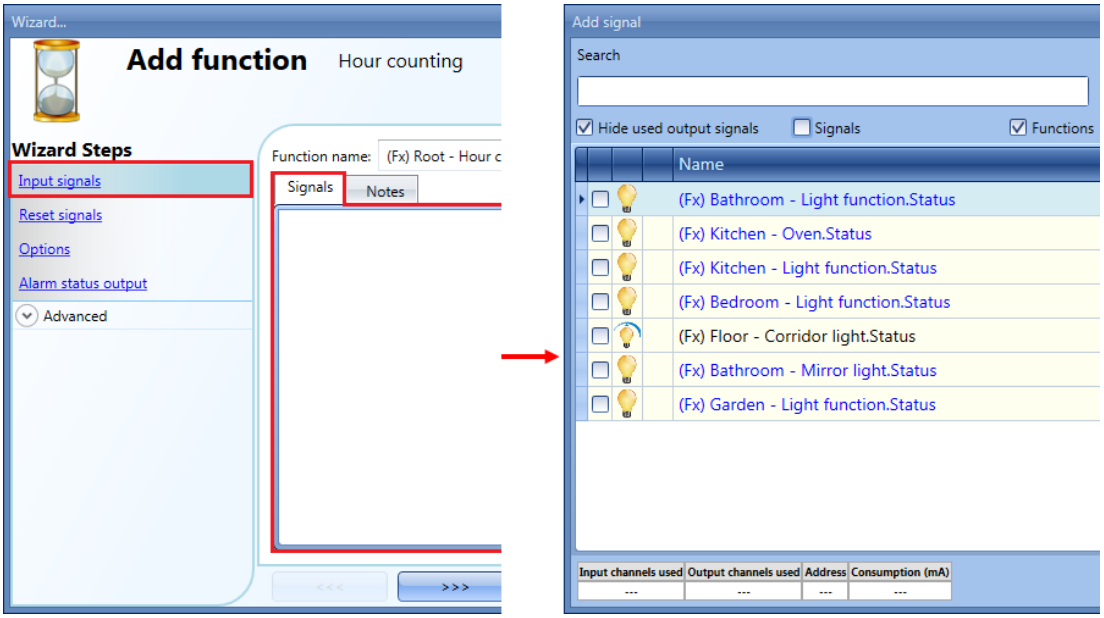
To set up an *Hour counting* function, the user has to select *Basic* from the Add menu (see picture below). The new function will be added into the selected location.



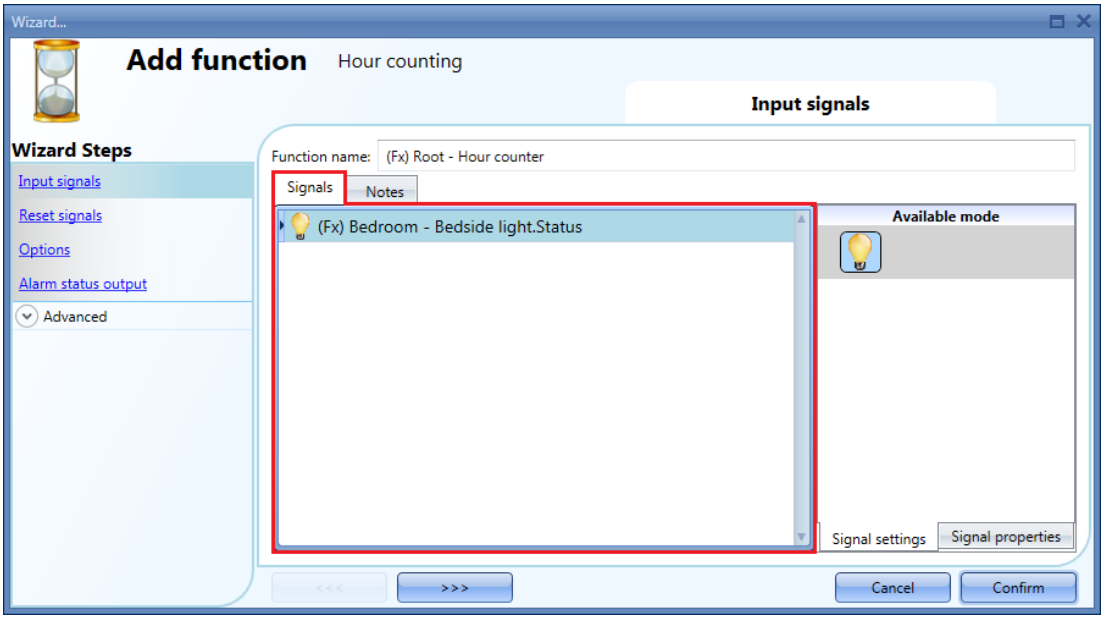
10.32.1 How to add the input signal

To add the input signal (the function to be monitored for “ON” hours status) for the *Hour counting* function, the user has to click on *Input signals*, then double click on the *Signals* window and select the input signal from the list (see picture below).

The input signal may be a physical signal or a function status (e.g. *Light function, Zone temperature* function etc..) configured in the project.

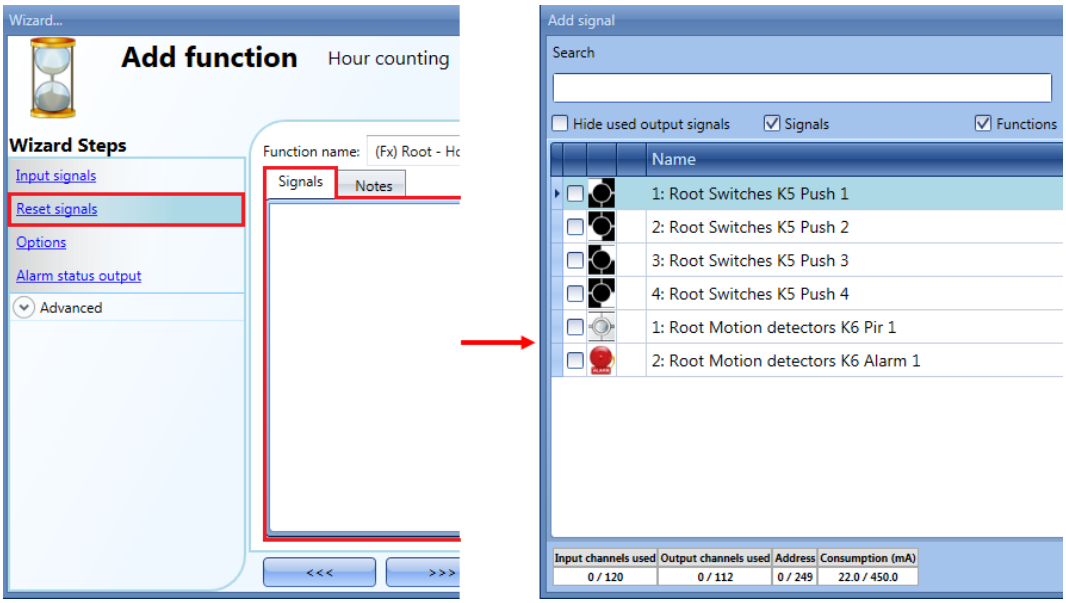


Only one input signal can be added to the *Hour counting* function.

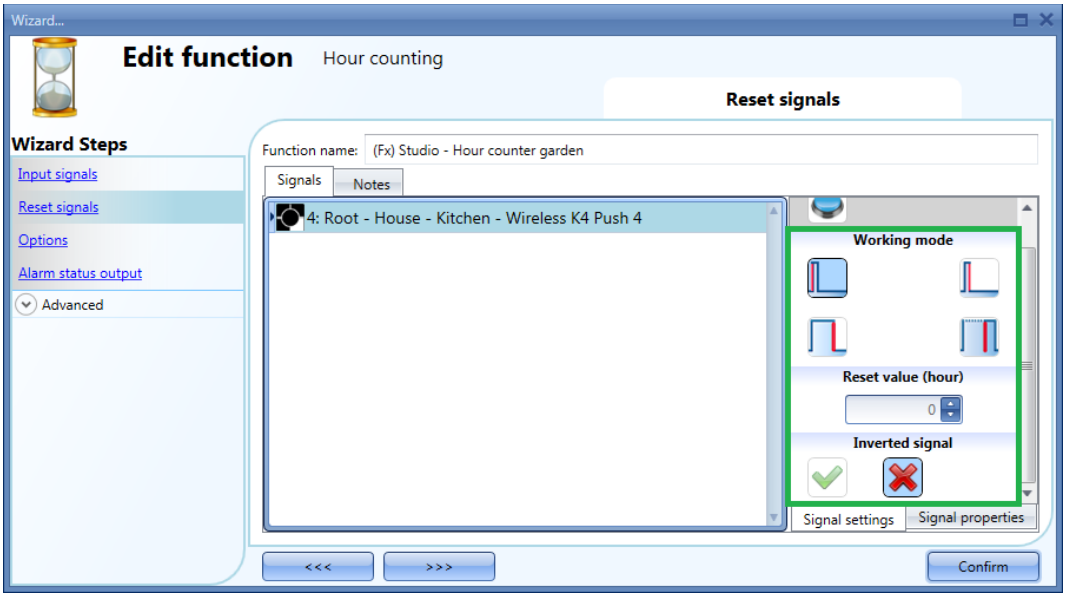


10.32.2 How to reset the Hour counting value

To reset the *Hour counting* value, the user has to click on *Reset signals*, then double click on the *Signals* window and select the input signal from the list (see picture below).
The reset signals may be physical signals such as pushbutton or switch, or functions such as a calendar.
Up to 10 Reset signals can be added.

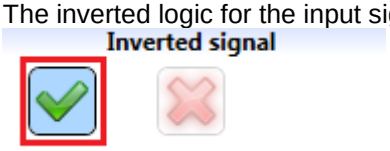


Each time the reset signal is activated, the counting value is reset.



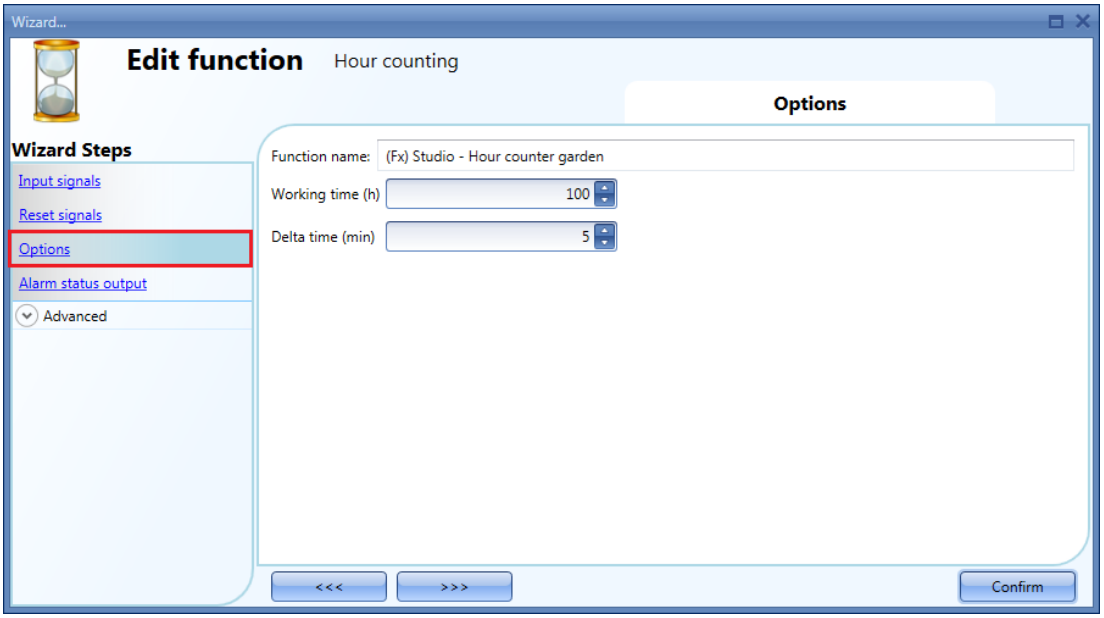
For each signal it is possible to set a certain reset value when the reset signal is triggered (e.g. when the push button is pressed): the user may set the *Reset value (hour)* field.

The inverted logic for the input signal can be enabled by selecting the green V in *Inverted signal.*



10.32.3 How to set the Hour counting options

To configure the *Hour counting* options, the user has to click on O*ptions*: a window with the different configuration options will appear (see picture below).



In this window, the threshold can be set in the field *Working time*: if the counted value is greater than the *Hour counting* value, the Alarm status will be on, otherwise it will be off.

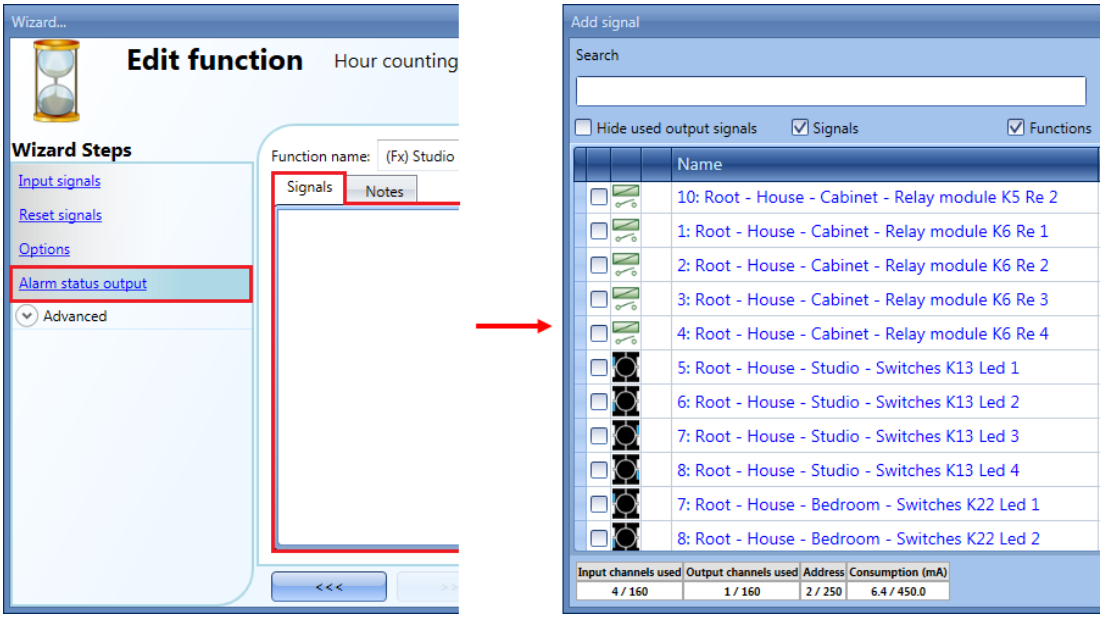
Delta time is the interval time used to calculate how much the counter has been increased during the period *delta time*.

Delta counter= Counter $_{\text{Delta time}}$ - Counter $_{\text{Delta time-1}}$

This is also the value logged if the *Delta time* is not zero. If *Delta time* is set to 0, no delta calculation is made.

10.32.4 How to add an Alarm status output

The user can configure an alarm output to be warned when the *Working hours limit* is reached. To add the alarm output, the user should select the relevant section in the wizard of the function, then double click on the *Alarm status output* window and select the output signals from the list of those available (see picture below).
The output signal can be a relay, a LED or a buzzer. The output signal will stay on until the reset signal appears.



The selected signals will be on if the counted value is greater than the configurable threshold *Working time (h)*, otherwise they will be off.

10.32.5 How to log the outputs of the Hour counting function

It is possible to log the *Hour counting* values with a selectable time interval, just like the standard Counter function.

To be logged, the Hour counting function has to be added to the database in the usual way.
In the logged file, two columns are present: the first one (*COUNTER 1*) indicates the total value calculated; the second one (*COUNTER 2*) shows the difference calculated during the Delta time:

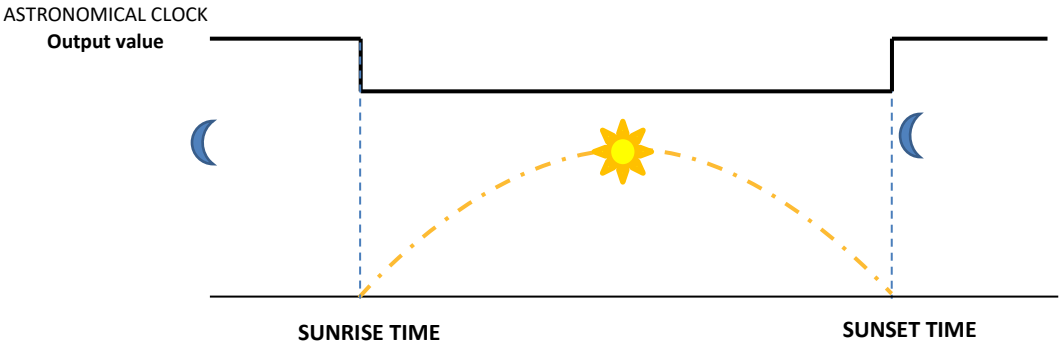
Delta= Counter $_{\text{Delta time}}$ - Counter $_{\text{Delta time-1}}$

Since the *Delta time* can be different from the sampling time, the difference calculated from two consecutive rows in column *COUNTER 1* might not correspond to the relevant line in column *COUNTER 2*. If *Delta time* is set to 0, the column *COUNTER 2* will not be present.

A	B	C	D	F	G	L	M
Position	Timestam	Date	Hour	Object Name	Location Name	COUNTER 1	COUNTER 2
3	1,4E+09	07/07/2014	15:08:30	(Fx) Root - Counter function1	Root	10	0
4	1,4E+09	07/07/2014	15:09:30	(Fx) Root - Counter function1	Root	31	21
5	1,4E+09	07/07/2014	15:10:30	(Fx) Root - Counter function1	Root	31	0
6	1,4E+09	07/07/2014	15:11:30	(Fx) Root - Counter function1	Root	39	0
7	1,4E+09	07/07/2014	15:12:30	(Fx) Root - Counter function1	Root	42	11
8	1,4E+09	07/07/2014	15:13:31	(Fx) Root - Counter function1	Root	47	5
9	1,4E+09	07/07/2014	15:14:30	(Fx) Root - Counter function1	Root	47	0
10	1,4E+09	07/07/2014	15:15:30	(Fx) Root - Counter function1	Root	47	0
11	1,4E+09	07/07/2014	15:16:30	(Fx) Root - Counter function1	Root	47	0
12	1,4E+09	07/07/2014	15:17:30	(Fx) Root - Counter function1	Root	47	0
13	1,4E+09	07/07/2014	15:18:30	(Fx) Root - Counter function1	Root	47	0
14	1,4E+09	07/07/2014	15:19:30	(Fx) Root - Counter function1	Root	47	0
15	1,4E+09	07/07/2014	15:20:30	(Fx) Root - Counter function1	Root	47	0
16	1,4E+09	07/07/2014	15:21:30	(Fx) Root - Counter function1	Root	47	0
17	1,4E+09	07/07/2014	15:22:30	(Fx) Root - Counter function1	Root	47	0
18	1,4E+09	07/07/2014	15:23:30	(Fx) Root - Counter function1	Root	47	0
19	1,4E+09	07/07/2014	15:24:30	(Fx) Root - Counter function1	Root	47	0
20	1,4E+09	07/07/2014	15:25:30	(Fx) Root - Counter function1	Root	47	0

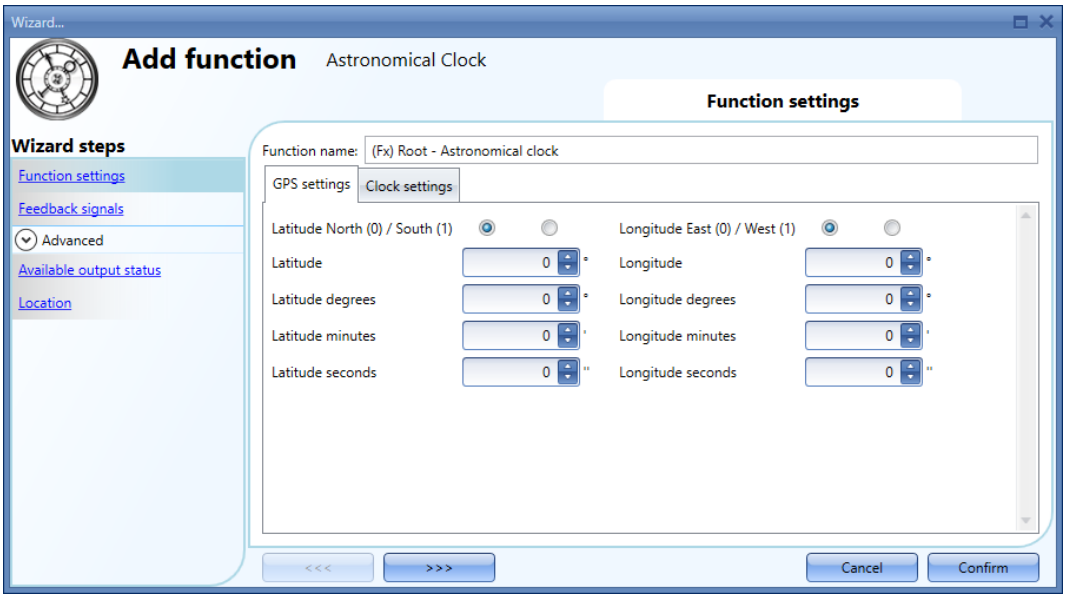
10.33 Astronomical clock function

The *Astronomical clock* function automatically calculates the sunset and sunrise times according to the geographical location: the calculation is performed by the latitude and longitude coordinates provided by the user. The output value of the function is a level signal: when the time is between the sunset and sunrise times, the output value is ON (Night condition), otherwise the output value is OFF (Day condition). The function is an ideal solution for turning the lights ON at sunset and OFF at sunrise, typical for street lights application.



10.33.1 How to add the Astronomical clock function

To add the *Astronomical clock* function the user has to select *Astronomical clock* function from the *Add* menu (see below). The following window will appear:

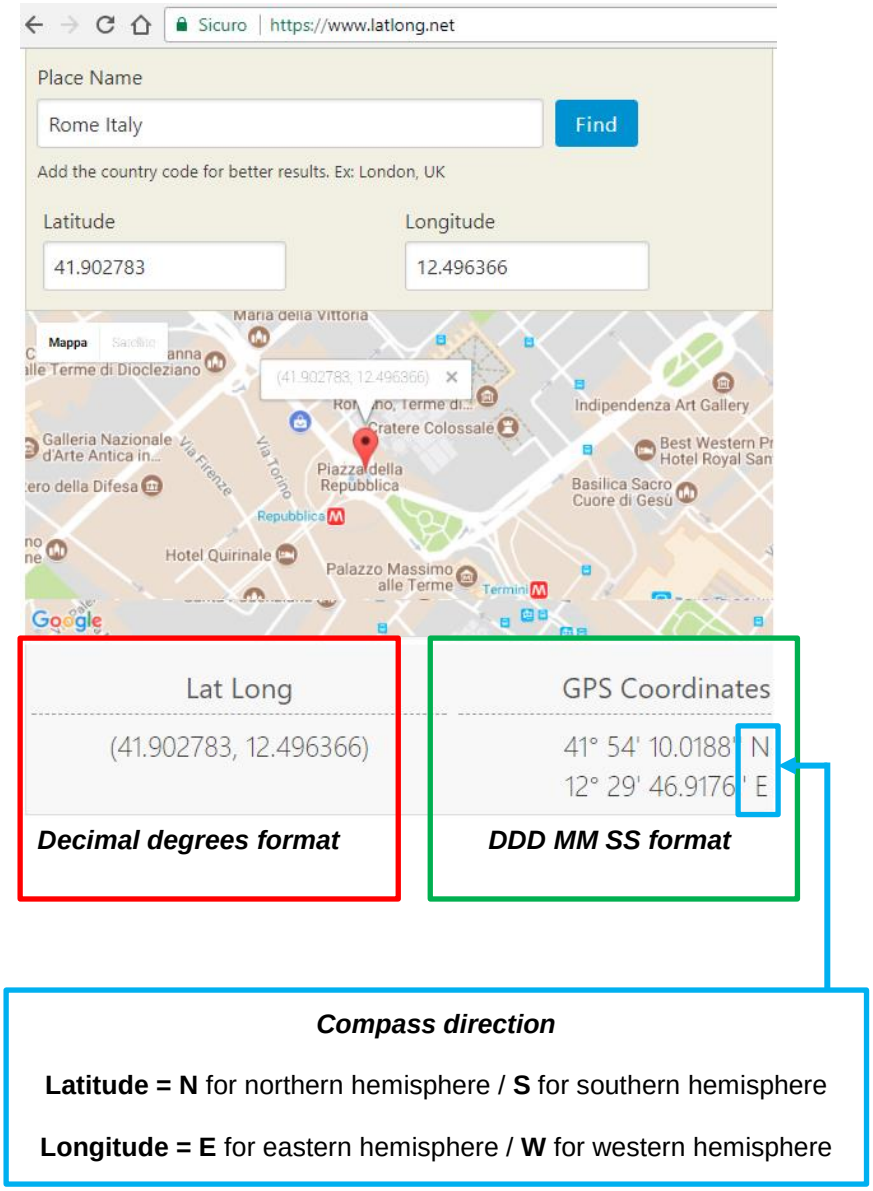


To calculate the sunset and sunrise times, the user has to enter the *Latitude* and *Longitude* coordinates of the geographical location of the installation. This information can be provided in two different formats:

- Decimal degrees format + compass direction;
- Degrees minutes seconds (DDD MM SS) format + compass direction;

It may be necessary to consult the internet or a GPS service to find the Latitude and Longitude coordinates of the location. The user can refer to the Web Site https://www.latlong.net/ to find this information.

For instance, the city of Rome (IT) has the following coordinates:



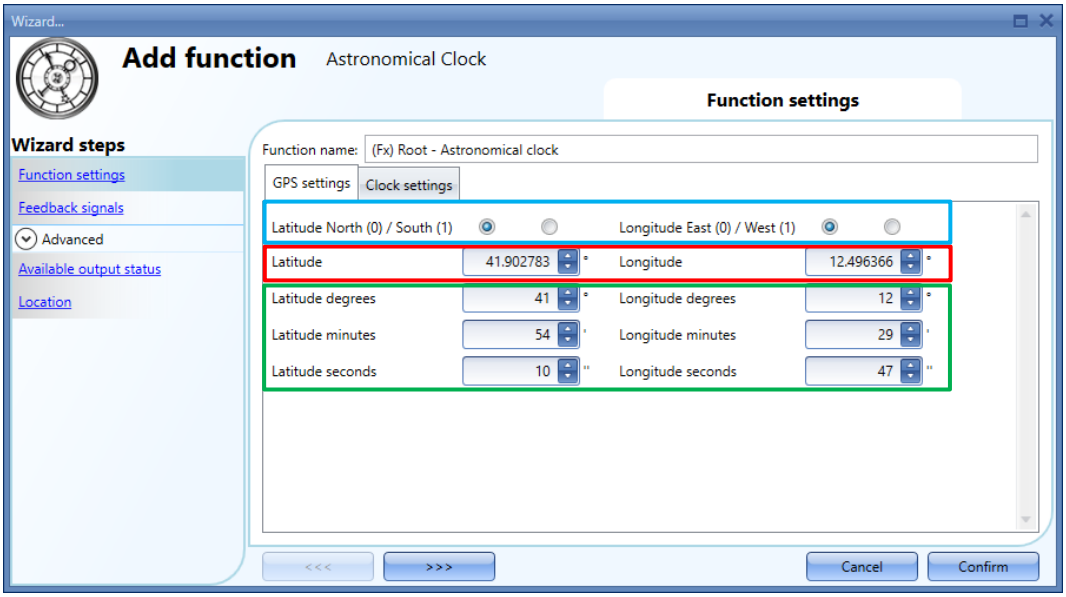
10.33.2 How to configure the Astronomical clock function

10.33.2.1 Set the Latitude and Longitude coordinates

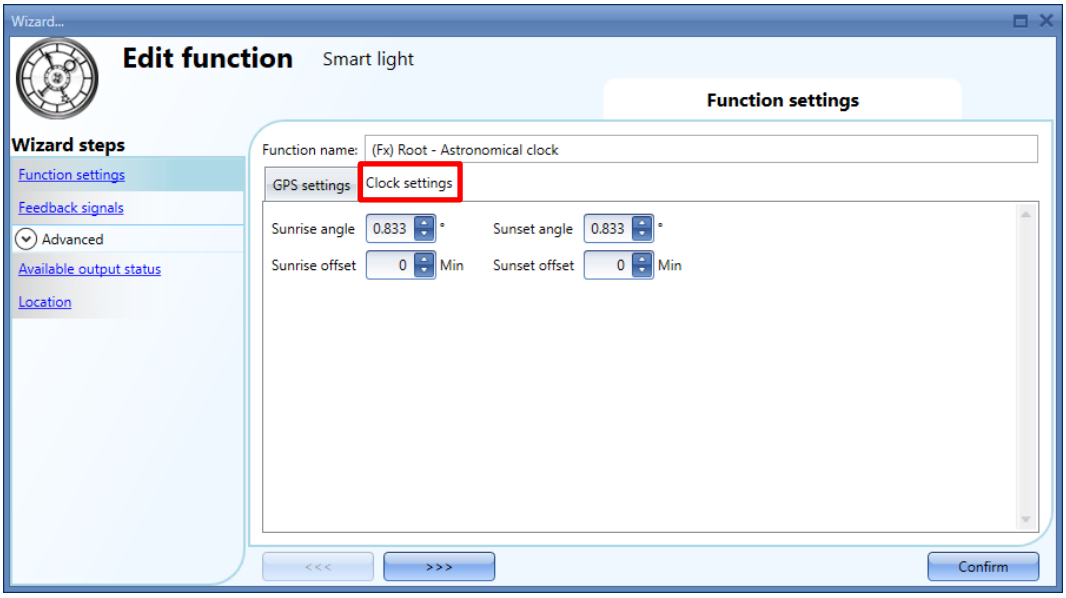
In the *GPS settings* window the latitude and longitude coordinates must be defined. There are two ways:

a) ***Decimal degrees format:***
Set the *Latitude*, *Longitude* and *compass direction* fields as shown in the red and blue boxes below (the degrees minutes seconds fields will automatically be filled in);

b) ***DDD MM SS format:***
Set the related fields, both for *Latitude, Longitude* and *compass direction* as shown in the green and blue boxes below (the decimal degrees fields will automatically be filled in);



10.33.2.2 Programming sunrise and sunset angles



In the *Clock settings* tab, for sunrise and sunset times calculation, the user can set the angle of the sun below the horizon. The default value is 0.833 degrees for both *Sunrise angle* and *Sunset angle.* These values can be changed by the user considering the following indication:

- The higher the value set in the *Sunrise angle* field, the earlier the sunrise condition will occur;
- The higher the value set in the *Sunset angle* field, the later the sunset condition will occur;

10.33.2.3 Programming sunrise and sunset offsets

The user can specify an offset to be applied to sunrise and sunset times: the output value of the function is delayed or advanced in according to the calculated sunrise and sunset times.

Sunrise offset (default value is 0)
The offset value can be set in minutes in the range [-120 minutes to + 120 minutes]: this value will be added to or removed from the calculated sunrise time.

Sunset offset (default value is 0)
The offset value can be set in minutes in the range [-120 minutes to + 120 minutes]: this value will be added to or removed from the calculated sunset time.

10.33.3 Use cases

In the following pages some of the most used cases are listed together with the relevant configuration for the *Astronomical clock* function and the linked functions.

10.33.3.1 Turn the light ON/OFF according to sunrise/sunset times

The light is turned ON at sunset time (*Astronomical clock* function status goes ON) and the light is turned OFF at sunrise time (*Astronomical clock* function status goes OFF). A typical application is street lights or outdoor lights.

Functions settings

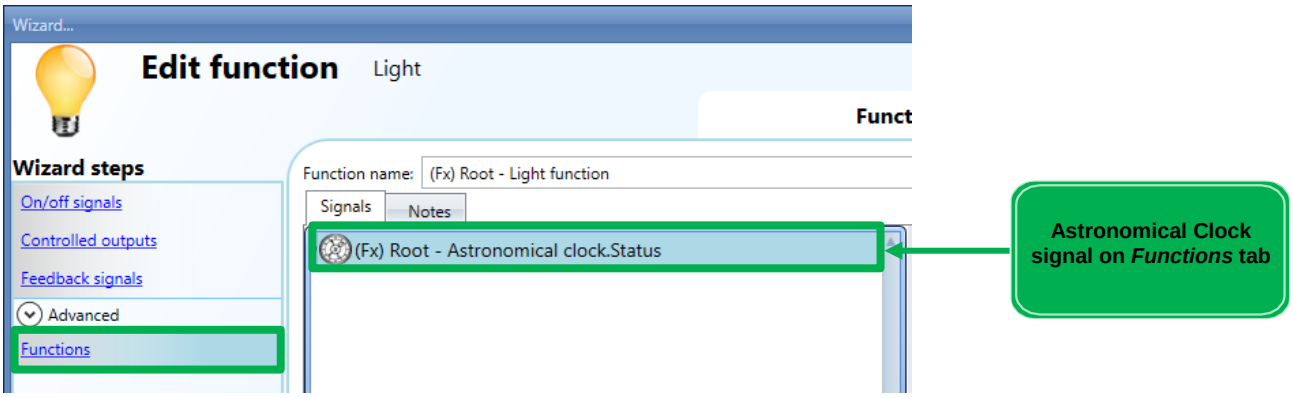
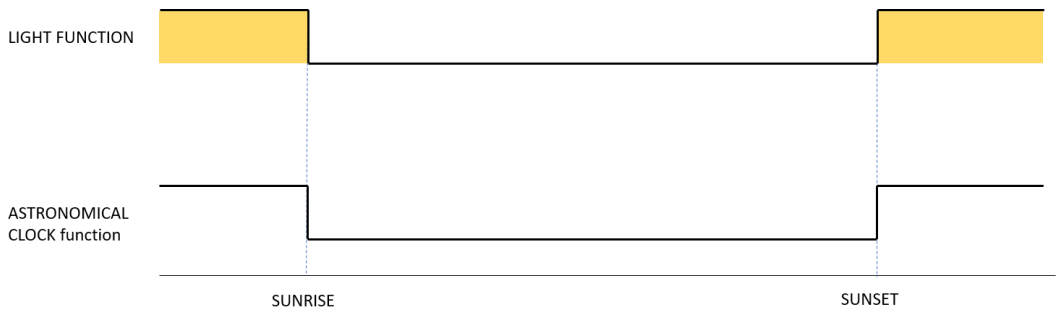


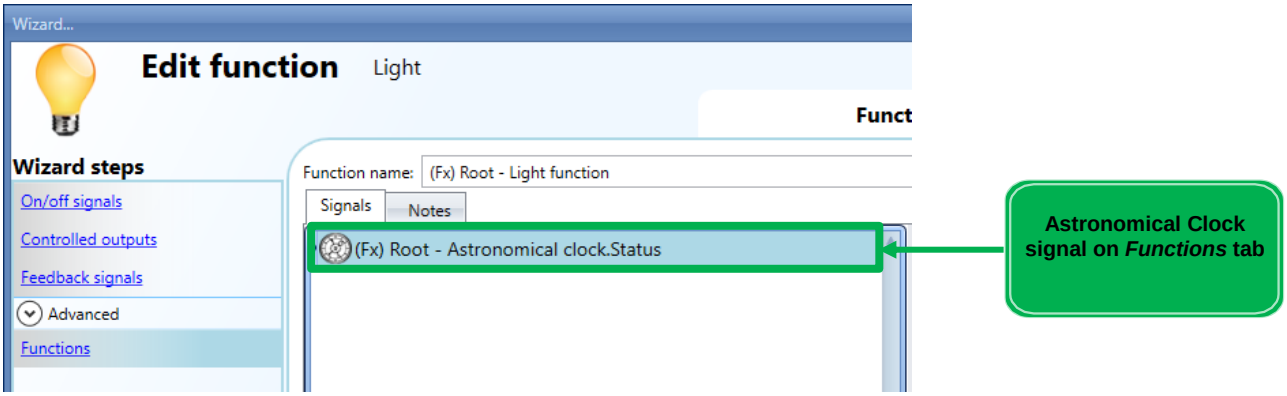
Diagram example



10.33.3.2 Turn the light ON/OFF according to sunrise/sunset times + offset value

The light is turned ON at *sunset offset* time (*Sunset time +/- Sunset offset* field value), then it is turned OFF at *sunrise offset* time (*Sunrise time +/- Sunrise offset* field value).

Functions settings



Sunrise and sunset times offset

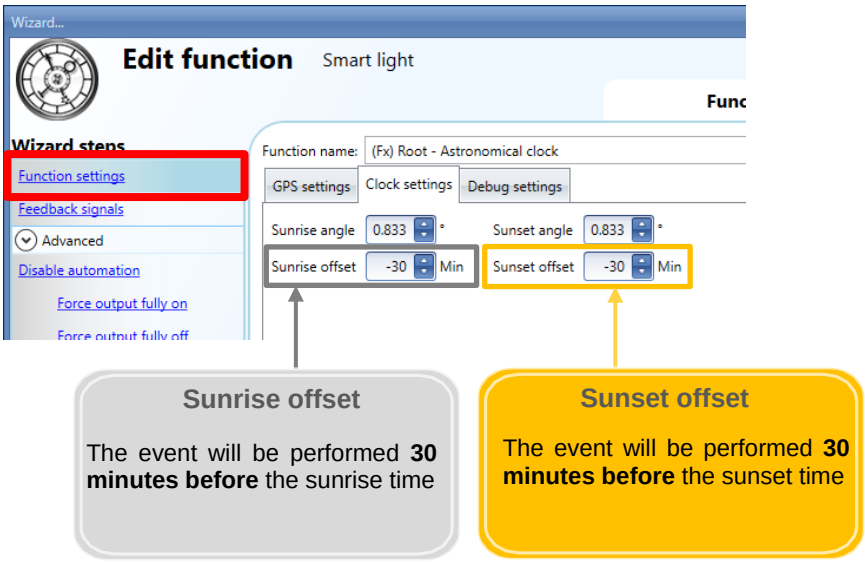
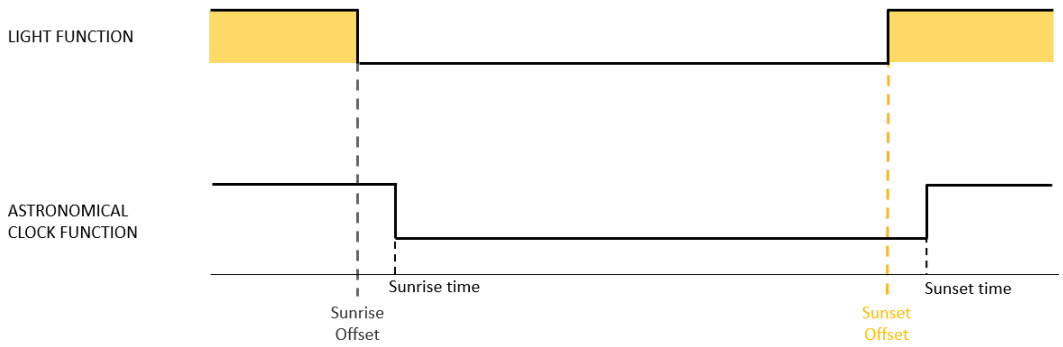


Diagram example



10.33.3.3 Move Rollerblind's position according to sunset / sunrise times

The blind is moved to 30% at sunrise time and it is moved to the *Fully DOWN* position at sunset time.

Signals settings

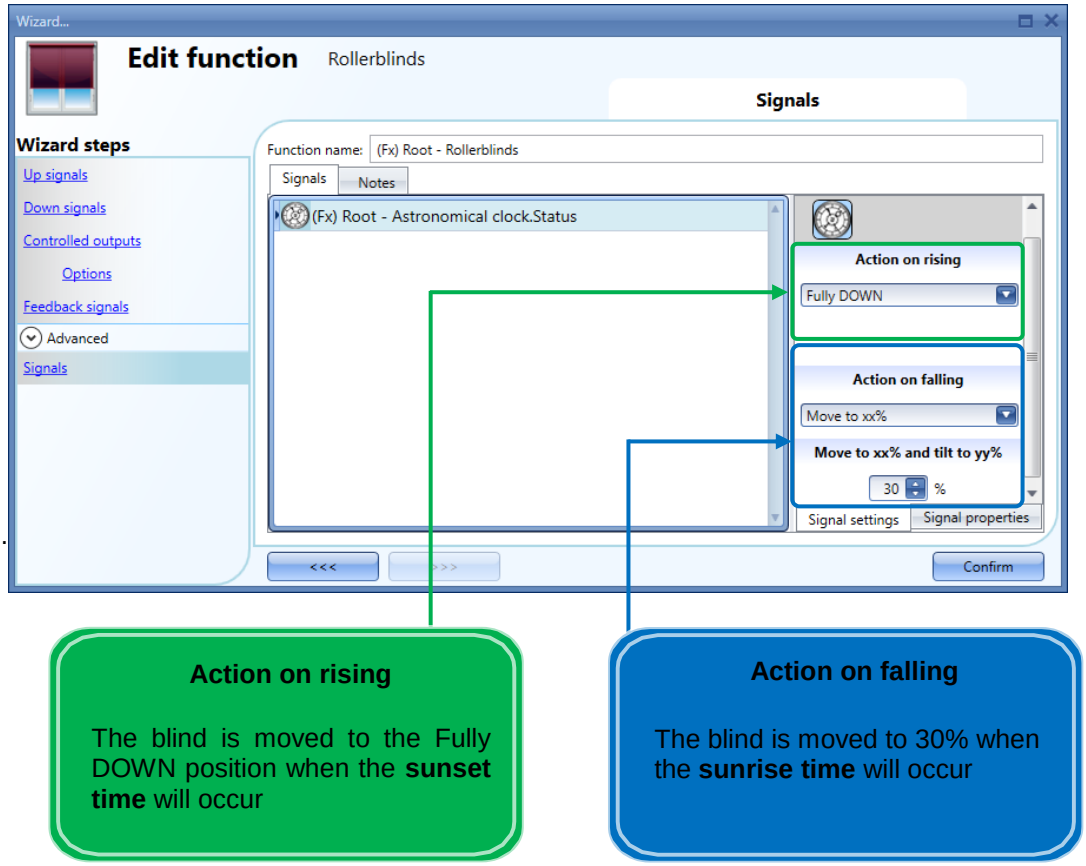
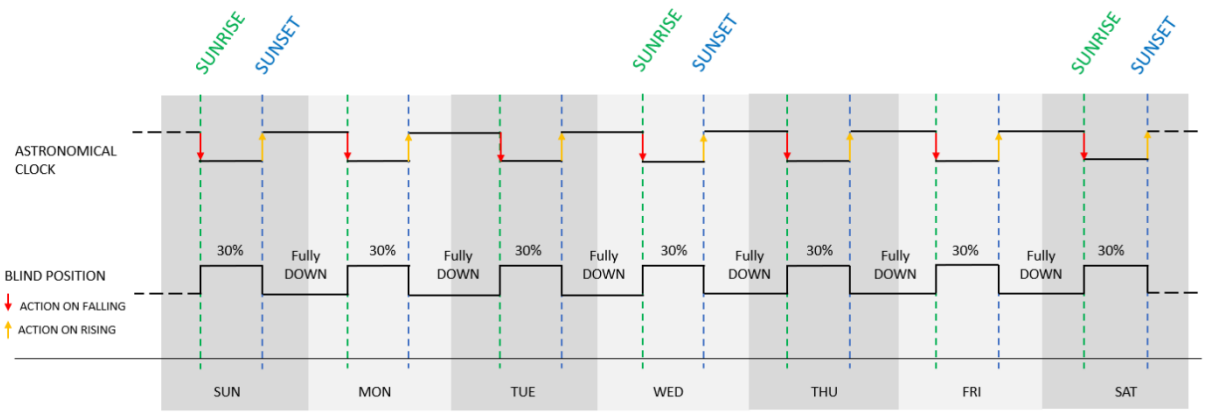


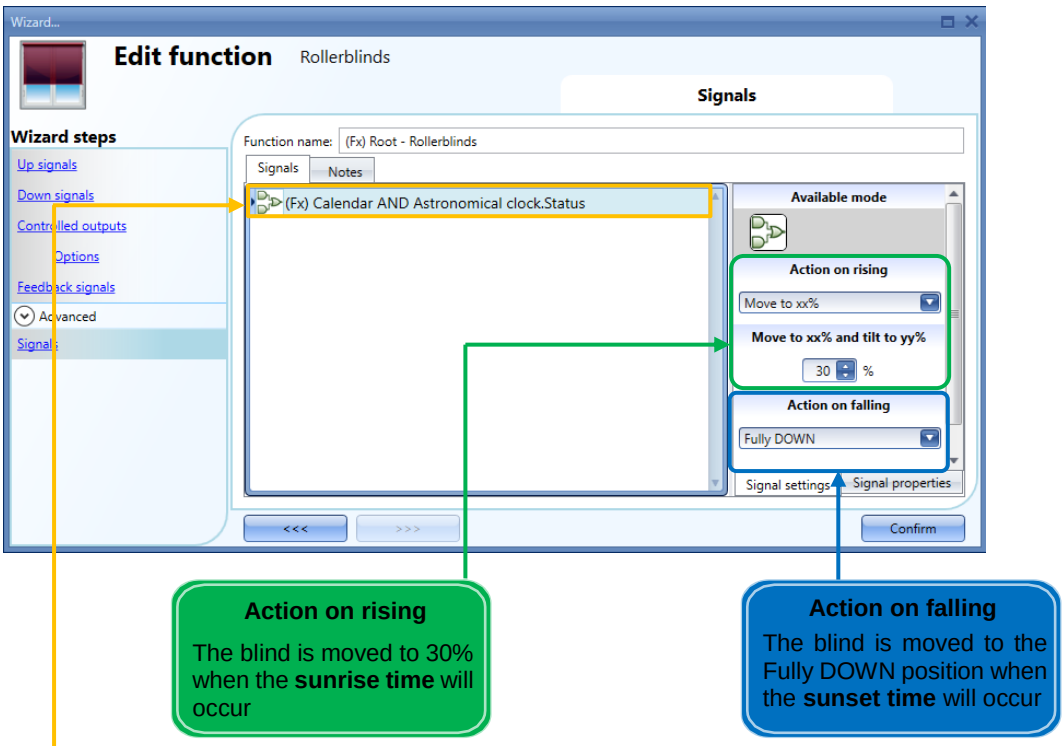
Diagram example



10.33.3.4 Move blind according to Astronomical clock and Level calendar on working days

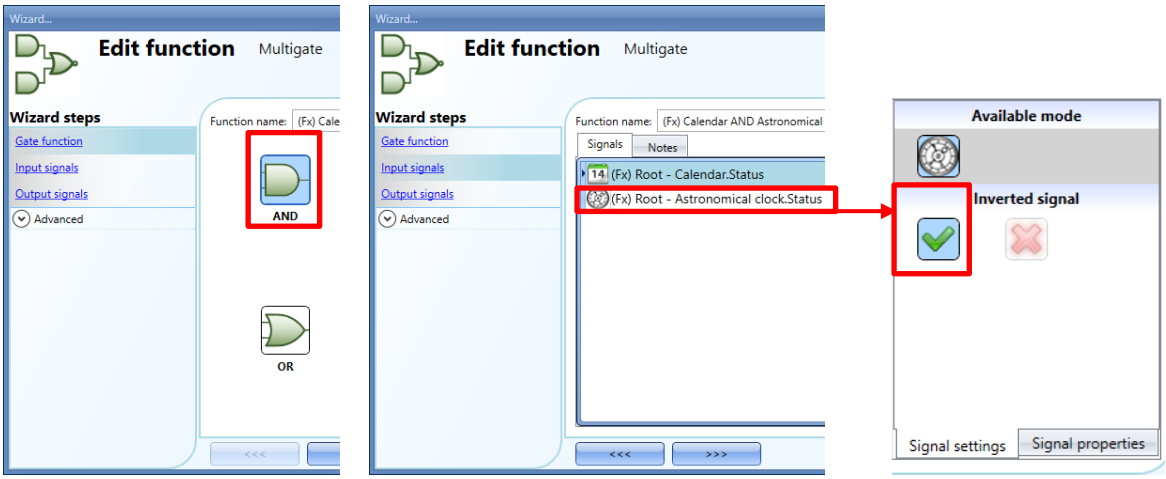
The blind is moved to 30% at sunrise time and it is moved to the *Fully DOWN* position at sunset time, from Monday to Friday. On Saturday and Sunday no action is performed.

Rollerblind function settings



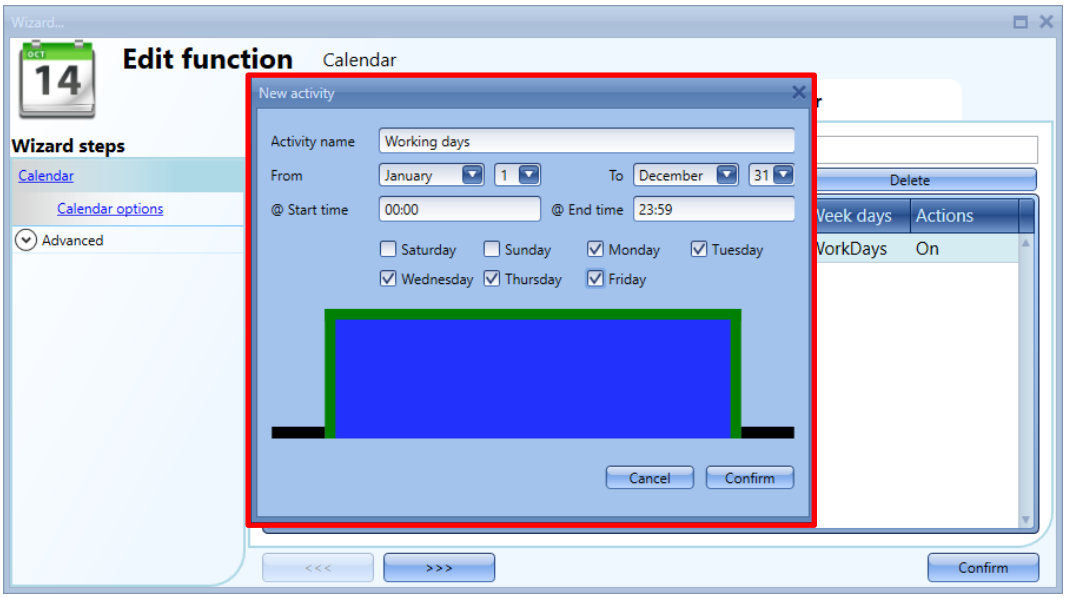
Multigate settings

The *Multigate* function is used to check the status of the *Calendar* and *Astronomical clock* functions:
The **Inverted signal** mode is selected for the *Astronomical clock* fx (Night value is OFF instead of ON)



Calendar settings

The *Local calendar* function is set as a level signal, the level is ON from Monday to Friday, while the level signal is OFF on Saturday and Sunday.



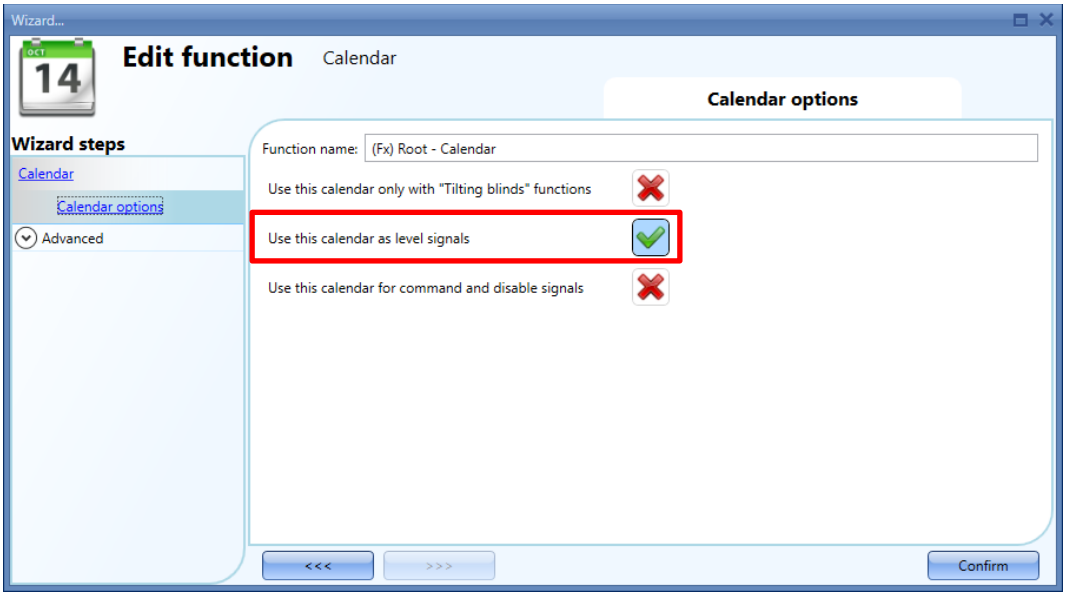
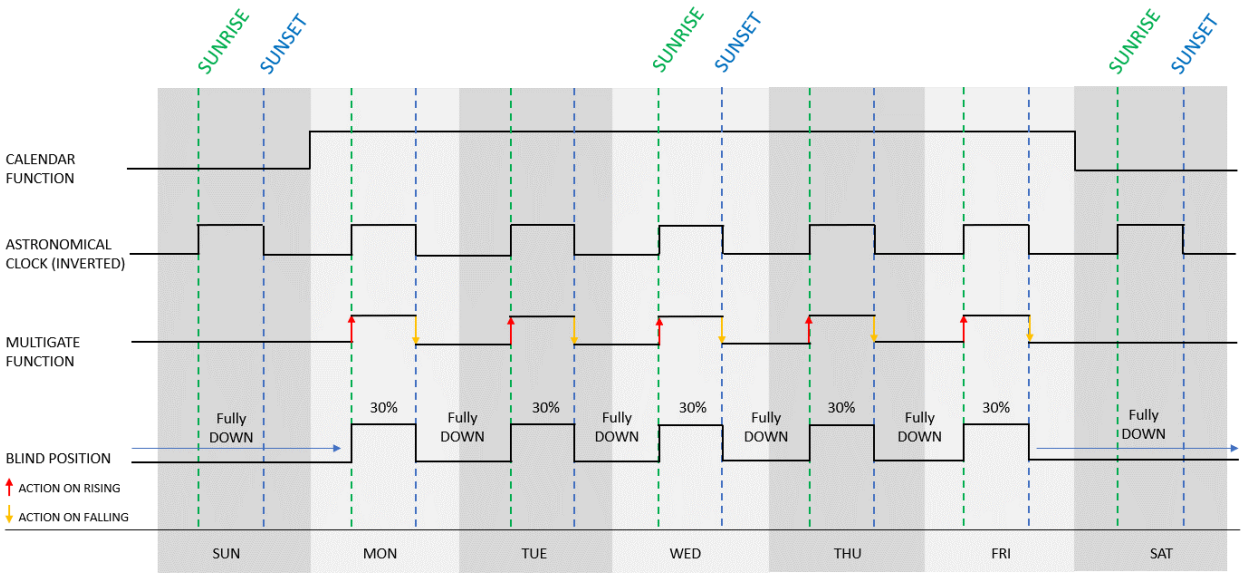


Diagram example

The *Multigate* function is used to check the status of the *Calendar* and *Astronomical clock* functions and it is used to change the blind position:

- The blind is moved to 30% at sunrise time: the *Multigate* function goes ON when both inputs go ON (the Calendar is ON and the Astronomical clock is in Day condition);
- The blind is moved to the fully DOWN position at sunset time: the *Multigate* function goes OFF when at least one of its inputs goes OFF (the Calendar is ON and the Astronomical clock is in Night condition);

Note: The Calendar is ON from Monday to Friday



10.34 Twin pumps function

How to automatically control two or more loads, balancing the working hours

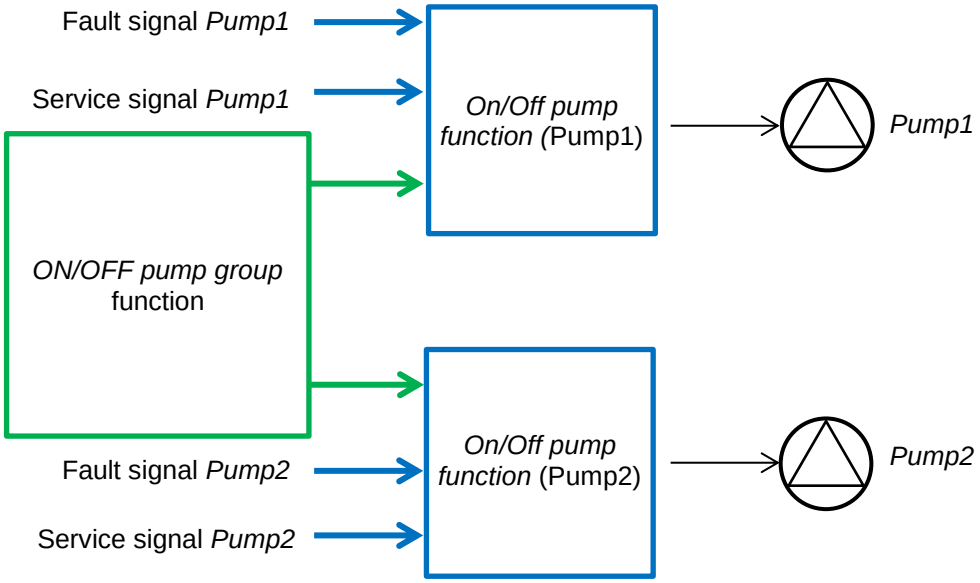
The *On/Off pump group function* and the *On/Off pump function* can be used to alternate the activation of a group of loads to balance the working time or to activate a different number of loads at the same time according to a changing request.

Usually this regulation is used to manage groups of pumps or twin pumps.

There are two main steps to configure this automation:

- In the first step the user has to create at least one *On/Off pump* function for each output to be managed in parallel. Each function offers specific input signals to detect fault and service conditions and an internal *working hour counter,* which is updated when the output is active.
- In the second step the user has to create a group function: the *On/Off pump group* function is designed to control two or more *On/Off pump* functions linked together.

The example below shows a system configuration with two *On/Off pump* functions (blue rectangles) managed by a group function, the *On/Off pump group* function (green rectangle). The digital outputs are connected to two pumps, *Pump1* and *Pump2.*

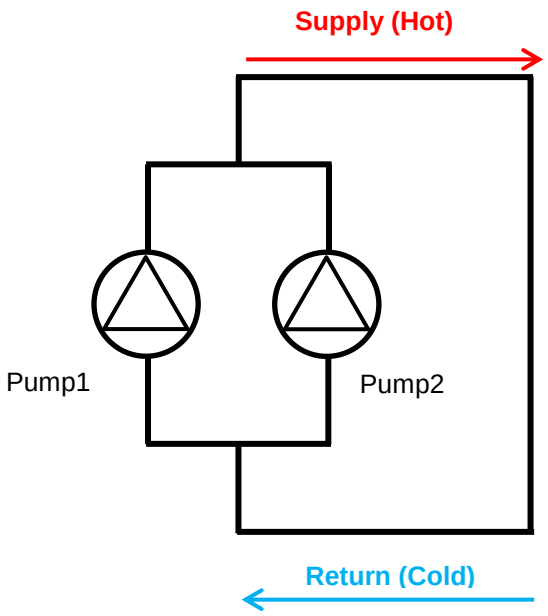


This is the best solution for building and commercial offices where redundancy is required in heating and cooling water installations. If a pump should fail and a standby pump is not installed, it could cause major disruption to the occupants. In these installations, any lack of services, as a result of a pump failure, could lead to a building shutdown. The parallel working mode offered by the twin pump function prevents the problem described above.

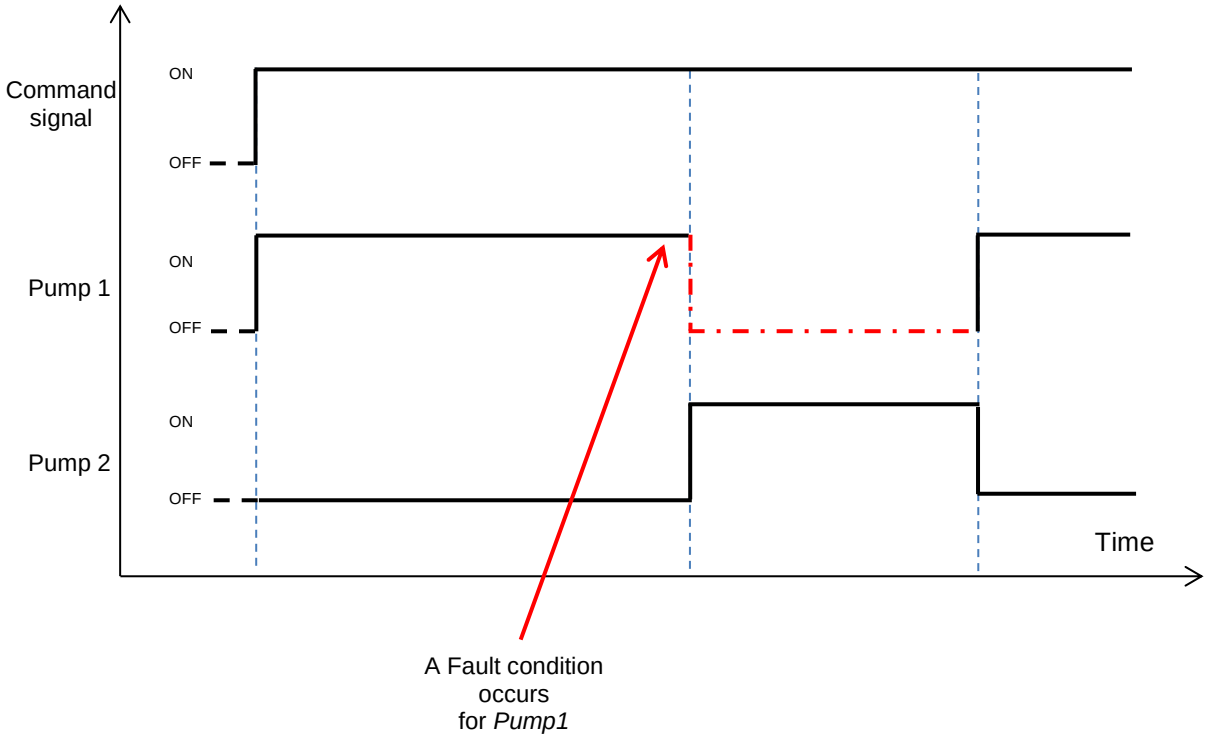
These examples represent the common scenarios where the twin pump functions are applied.

- Automatically alternates operation of the outputs
- If the operating output enters into a lockout condition, it will be automatically replaced by another available output.
- If the output is not available because in a lockout condition or out of service, it is excluded.
- Parallel working mode: a predefined number of outputs is activated in consecutive mode. The number of outputs can be activated by an analogue input value.

This function is primarily designed to provide a load redundancy and better load efficiency in a system. In the event of a lockout condition of the pump in operation (e.g. *Pump1*), the *On/Off pump group* function automatically switches ON the other one connected in parallel (i.e. *Pump2*). When the pump in fault condition is restored, it will return as available in its group. The diagram below shows two pumps connected in parallel in the system: **this configuration is typically used in a water circuit.**

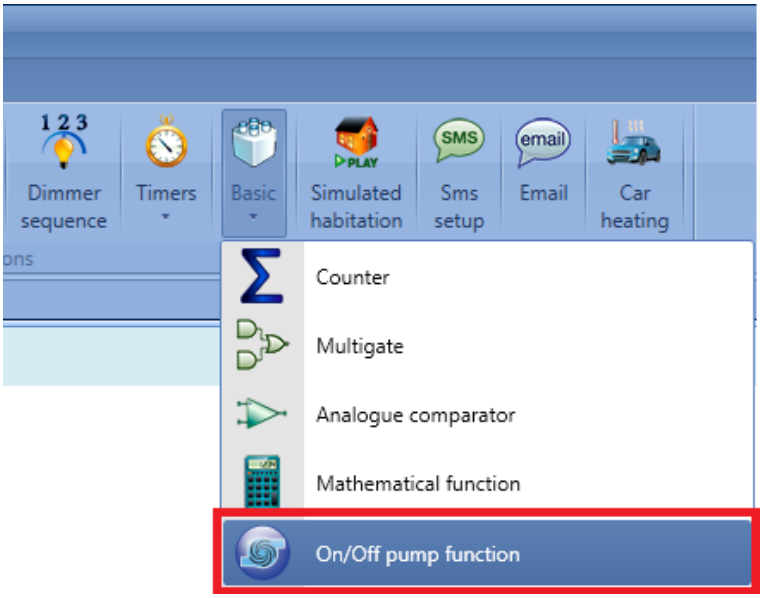


As shown in the graph below, after the system start-up, the *On/Off pump* functions (*Pump1, Pump2*) are activated by the *On/Off pump group* functions according to the priority list: let us assume that *Pump1* goes ON and *Pump2* remains OFF and is available as a stand-by pump. When a fault condition for *Pump1* occurs, it will be automatically replaced by *Pump2* and this will remain active until the fault condition is restored; after that, the *On/Off pump group* function compares the working hour counters of the available outputs and updates the priority list, which is used to select the output that has to be activated first: the one with fewer working hours will have the highest priority.
In the example below, Pump1 will be activated.



10.35 How to configure an On/Off pump function

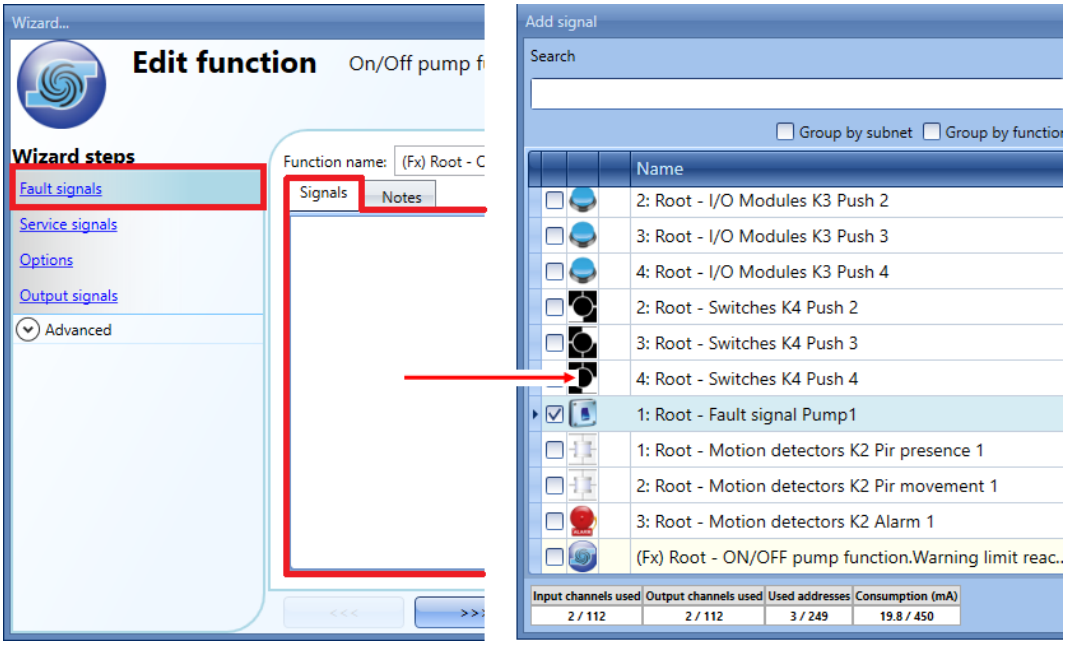
To set up an On/Off pump function, the user has to select *On/Off pump function* from the *Basic* menu, as shown below:



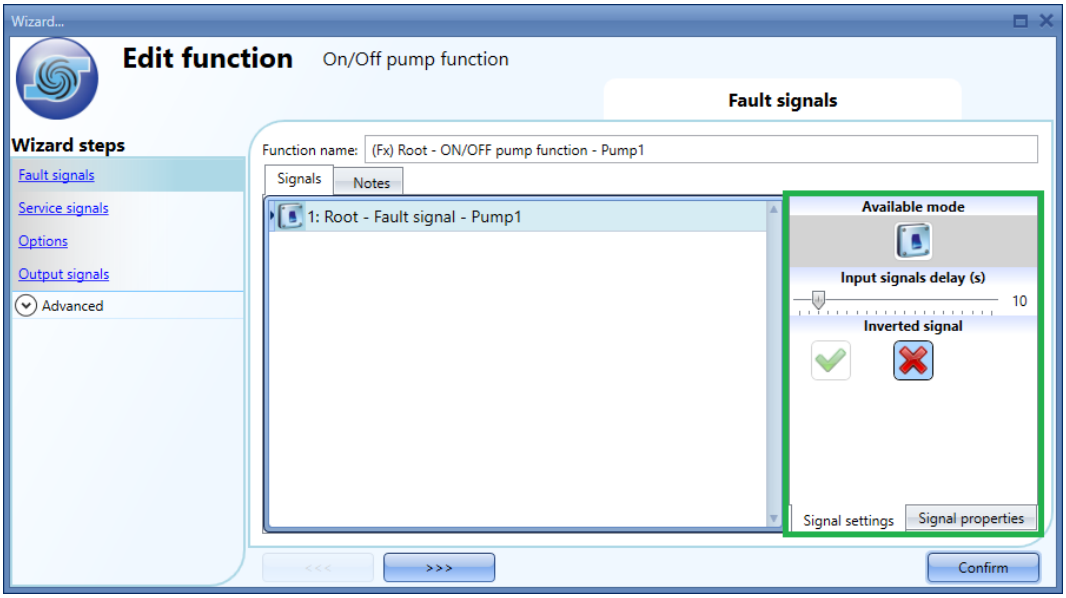
10.35.1 How to add the fault signals

The fault signals are used to detect when a fault condition occurs: when at least one of the fault signals goes ON (the input value is 1), the relevant output (i.e. the pump connected to the function) goes OFF and the *On/Off pump group* function automatically replaces the faulty pump with another one available in its group.

To add the fault signals, the user has to click on the *Fault signals* window, then double click on the *Signals* window and select the input signals from the list of those available (see below):
up to 10 signals can be selected and the system will perform a logical OR of all of them.

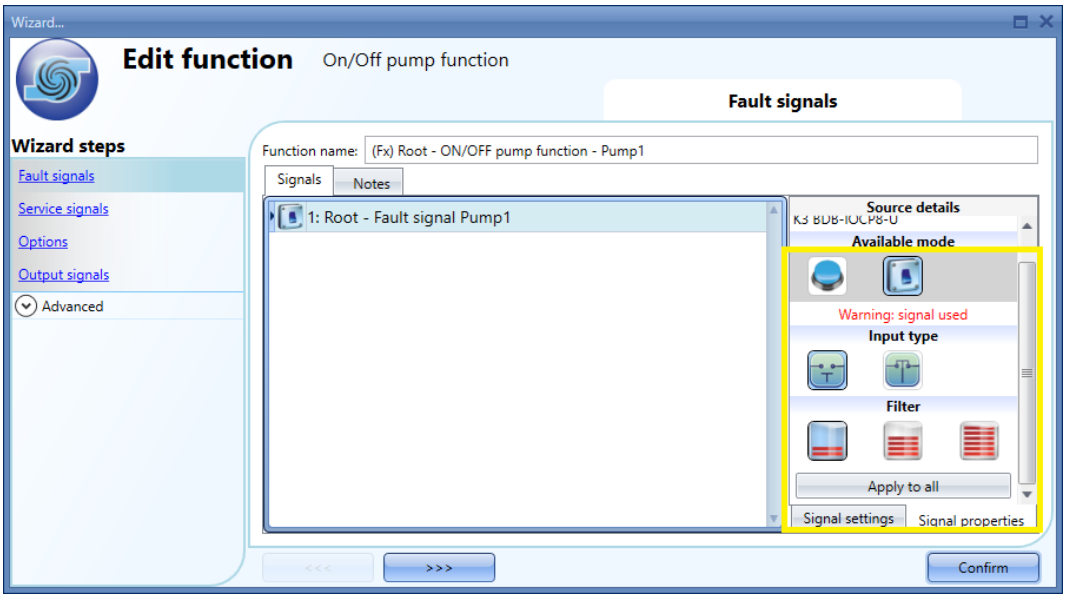


Once a fault signal is entered, the user can also decide to use the inverted logic of the signal. By selecting the green *V*, the inverted logic is enabled.
Each fault signal added to the function has a delay time that can be set in the field *Input signal delay(s):* this time is a further delay that can be added to detect the fault condition (see the green rectangle below).



In the picture displayed above, the user has selected one input signal of a BDB-INCON4-U module that is connected to the on-board contact on the pump device: when a fault condition is detected, the output signal of this function is excluded immediately from its group function and the output will be disabled while the fault condition is present.

In the *Signal properties* window the user can set the *Fault signals* properties according to the physical device.

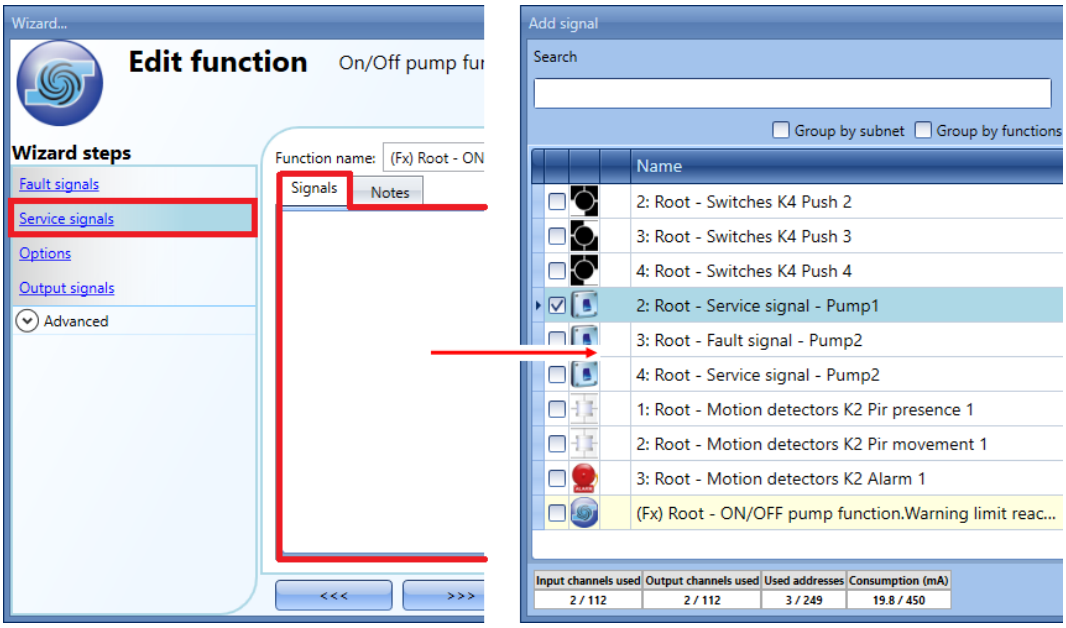


N.B. The user must refer to the electrical specifications (or wiring diagram) of the physical device in order to set the input signal correctly.

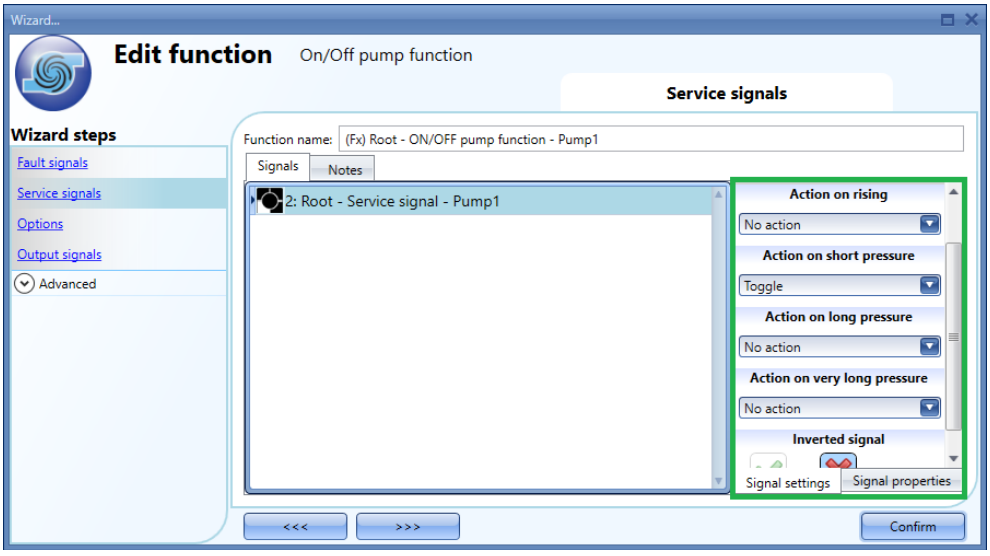
10.35.2 How to add the service signals

The service signals are used to put the digital output managed by the *On/Off pump* function into maintenance mode to let the user operate safely, for example to replace the motor. Until the service condition is activated, another pump in the group will continue operating until the service condition for the pump under maintenance is deactivated.

To add the service signals, the user has to click on the *Service signals* window, then double click on the *Signals* window and select the input signals from the list of those available (see below):
up to 10 signals can be selected and the system will perform a logical OR of all of them.



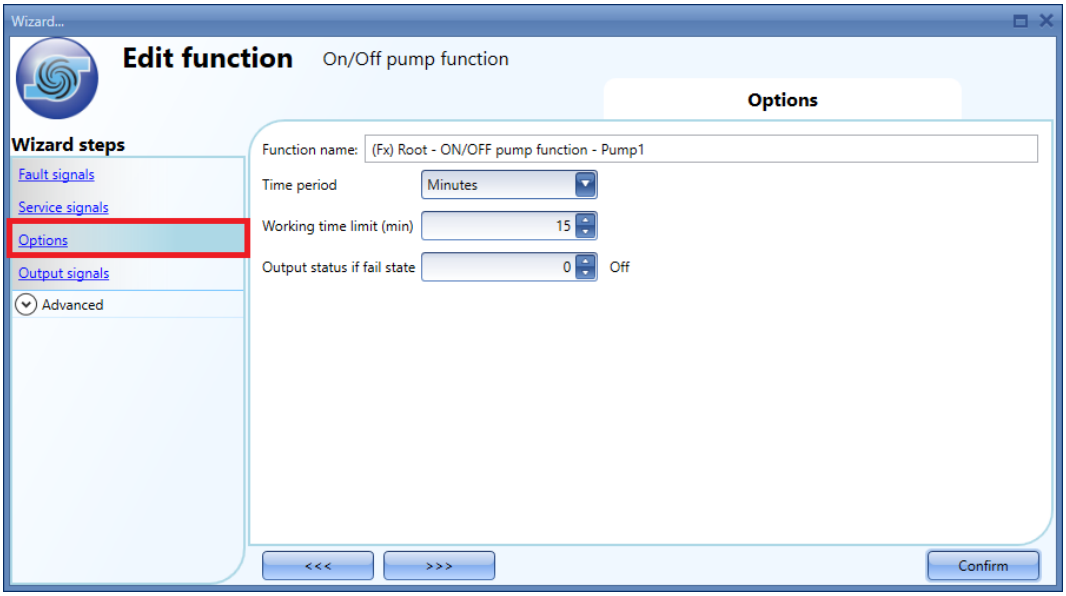
Each signal used in the *Service signals* window works as a level signal: until the signal is active, the function is forced into the OFF condition. In the *Signal settings* window the user can set the actions that have to be performed according to the different actions allowed by the system, as shown in the green rectangle below:



The Service condition overrides the fault condition: when the fault condition is verified concurrently, the service condition has the priority.

10.35.3 How to set the *function* options

To configure the *On/Off pump* function options, the user has to click on the *Options* field of the wizard: a window with different options will appear (see below).



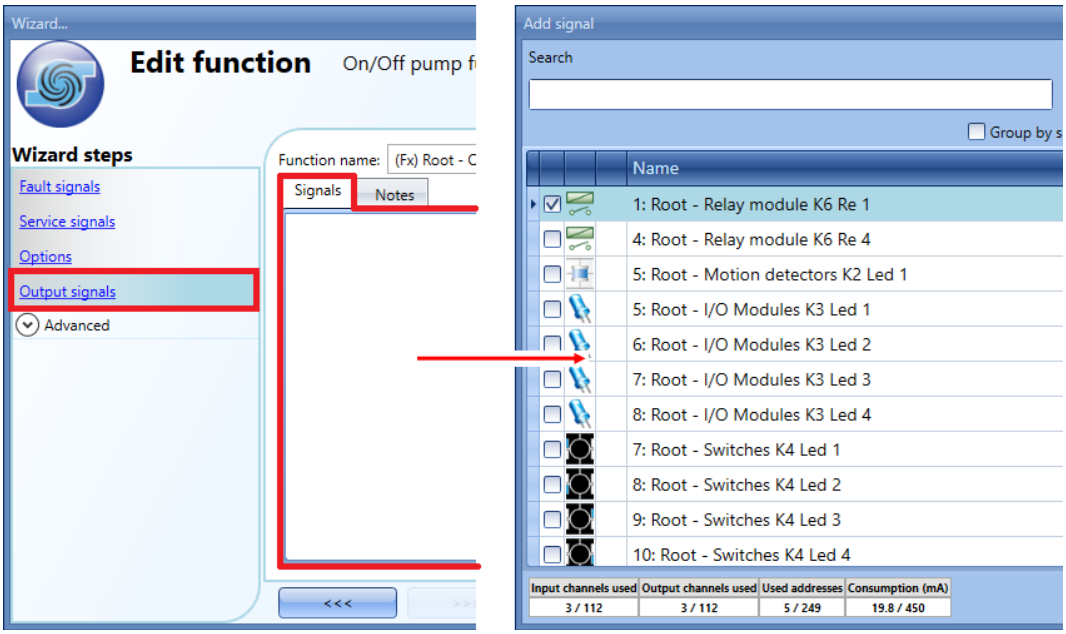
Time period
This defines the unit of time used in the *Working time limit(min)* field: the user can select *Minutes* or *Hours;*

Working time limit (min)
When the counted value is greater than the value set in this field, the diagnostic signal *Warning limit reached* will be turned ON.

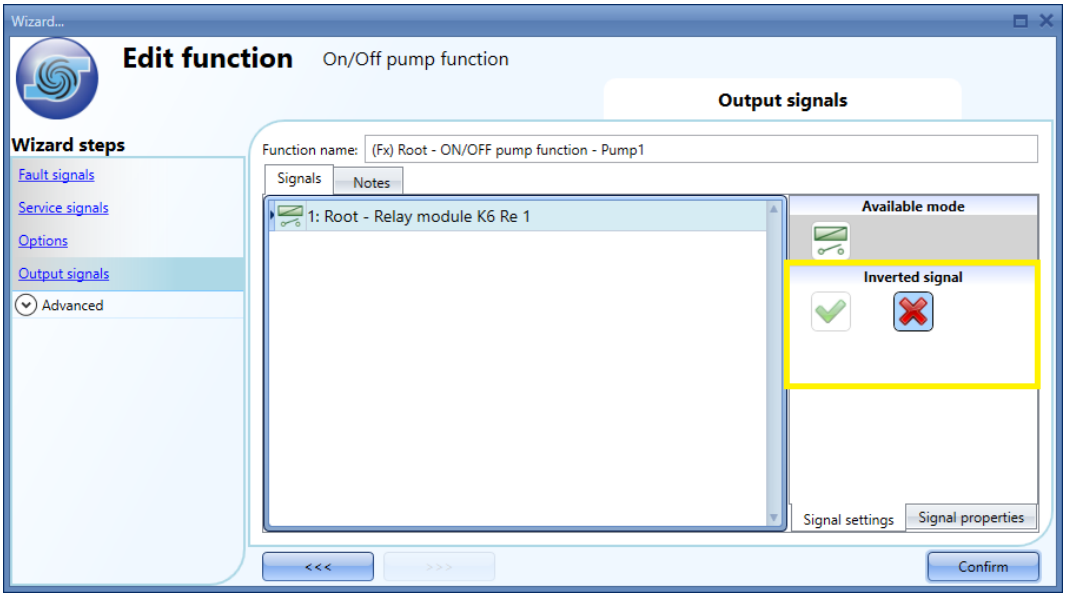
Output status if fail state
In this field the user can set the default value applied to the *output* when it is in a fault condition. If this option is set to 1 (On), and a fault condition is present, the On/Off pump function will be maintained ON but the group function will activate the next available *On/Off pump function* with the lower working hour value.

10.35.4 How to add the output signals

To add the digital output managed by the pump function, the user has to select the relevant section in the wizard of the function, then double click on the *Output signals* window and select from the *Signals* window the one that is directly connected to the load to be managed, such as a pump or a motor .



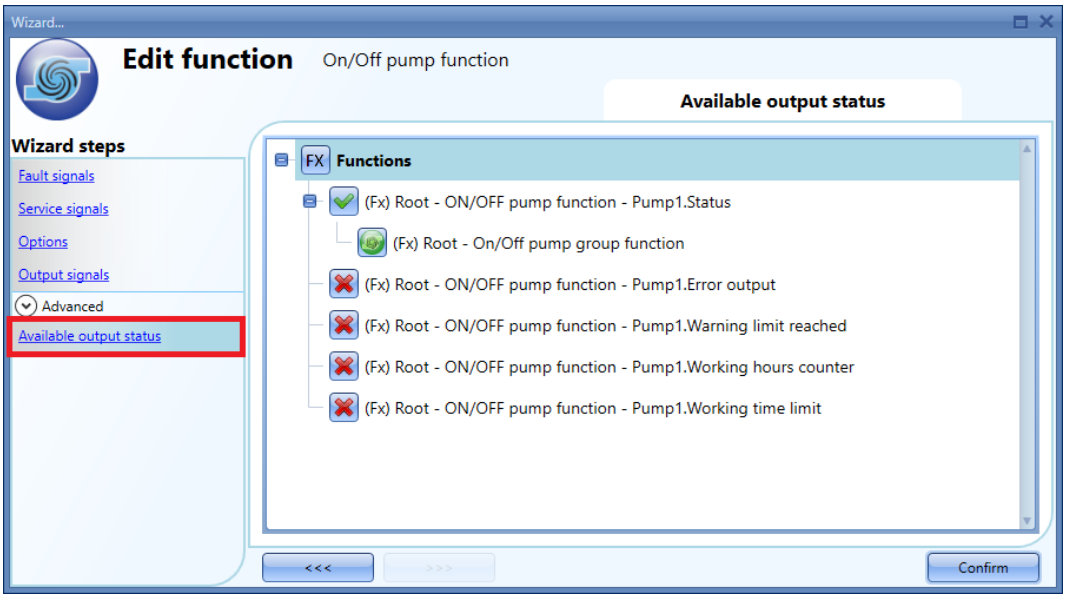
The output signal can be a relay or a remote control switch, and it follows the status of the function: when the function is ON, the output will be activated, when the function is OFF, the output will be deactivated.



The inverted logic for the output signals can be selected in the *Signal settings* tab window, as shown in the yellow rectangle above.

10.35.5 How to enable the function diagnostic

In the *Available output status* field, the user can decide which status from the *On/Off pump* function needs to be used in the configuration of the system, for example as an input in another function or in a sequence. Like all the other advanced functionalities, this one also has to be enabled.



The statuses that can be enabled and used in other functions are:

(Fx)=ON/OFF pump function.Status
This is the function status: the value will be ON when the function is activated and it will be OFF when the function is deactivated.

(Fx)= ON/OFF pump function.Error output
This output status shows the status condition of the function:
Ok = when the function is working well
Service= when the function is in maintenance mode
Fault= when a fault condition has been detected

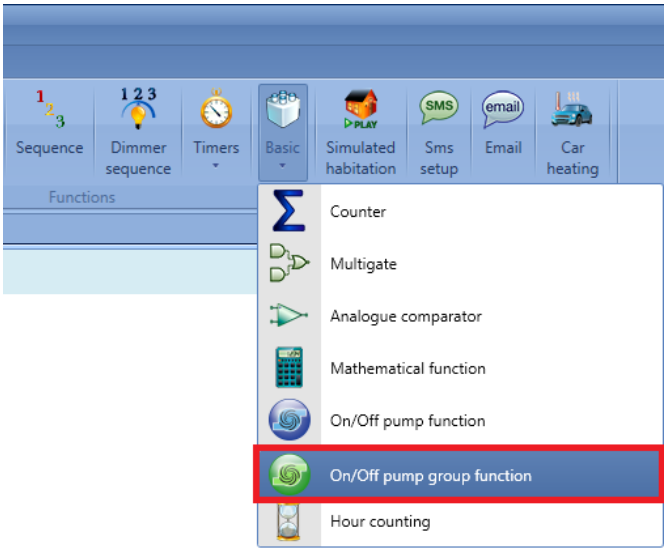
(Fx)= On/Off pump function.Warning limit reached
This shows a warning message when the *working time limit* has been reached

(Fx)= On/Off pump function.Working hours counter
This shows the working hour value (minutes or hours) of the function

(Fx)= On/Off pump function.Working time limit
This shows the working time limit (setpoint expressed in minutes or hours) of the function

10.36 How to configure an On/Off pump group function

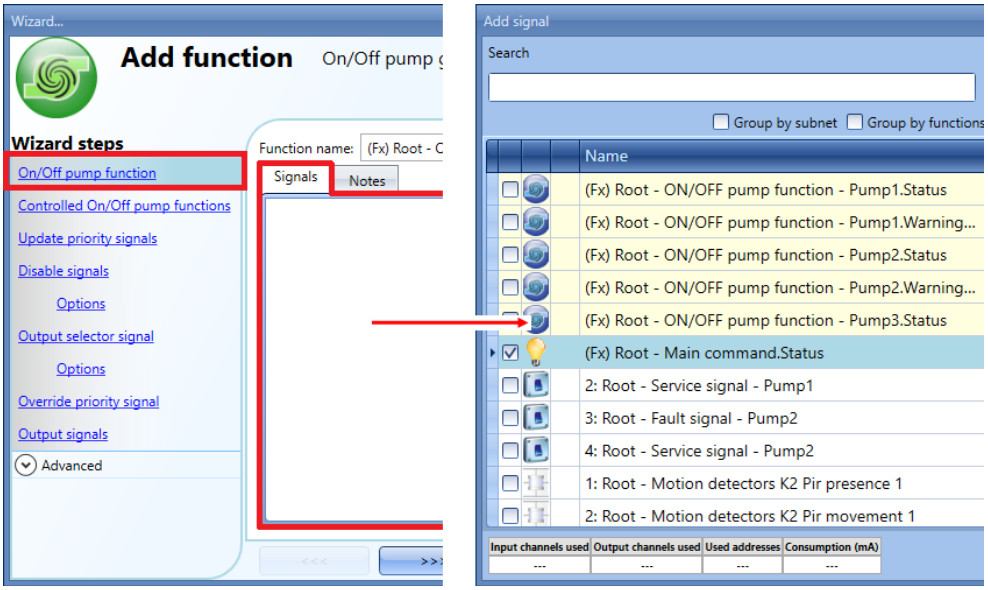
To add the *On/Off pump group* function, the user has to select *Basic* from the *Add* menu (see below) and then *On/Off pump group*. The new function will be added into the selected location.



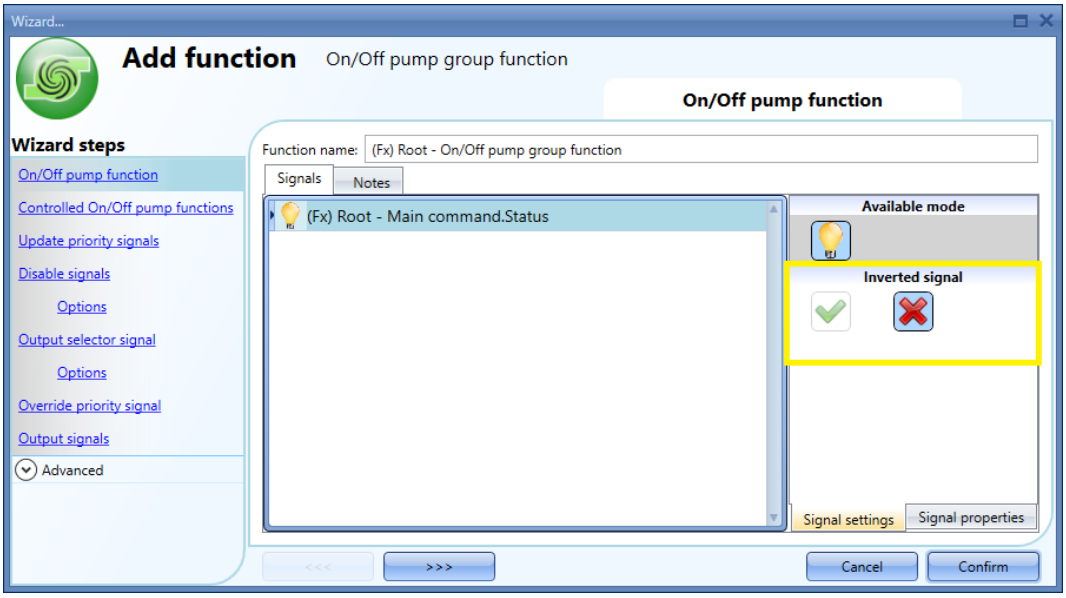
10.36.1 How to activate the regulation for the group of pumps

In the *On/Off pump function* field, the user has to select one digital input signal, used to activate/deactivate the *On/Off pump group* function. At the activation of the command signal, the controlled *On/Off pump* functions are activated according to the priority list and the number of outputs. The outputs that are not activated are set to 0 (OFF). Each time the command signal is turned OFF, the priority list is updated. The priority is defined according to the working time of each load in the group.

By double clicking in the *Signals* area, a window will appear with all the available signals (see below):



The signal used in the *On/Off pump function* field works as a level signal: while the signal is active, the *On/Off pump group function* will remain active.

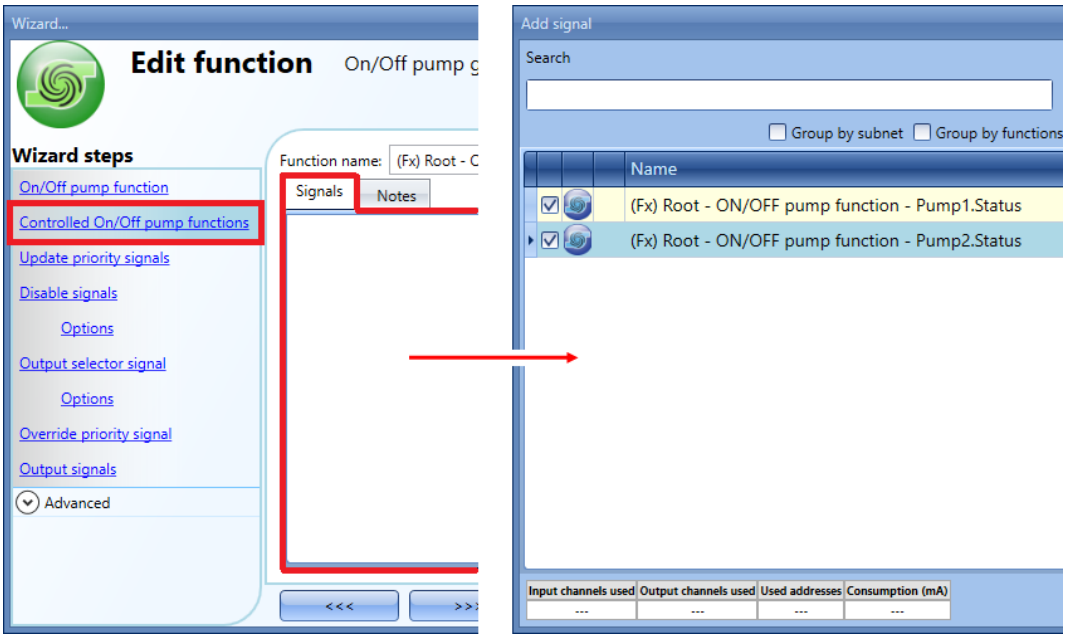


The inverted logic can be selected in the *Signal settings* tab window, as shown in the yellow rectangle above.

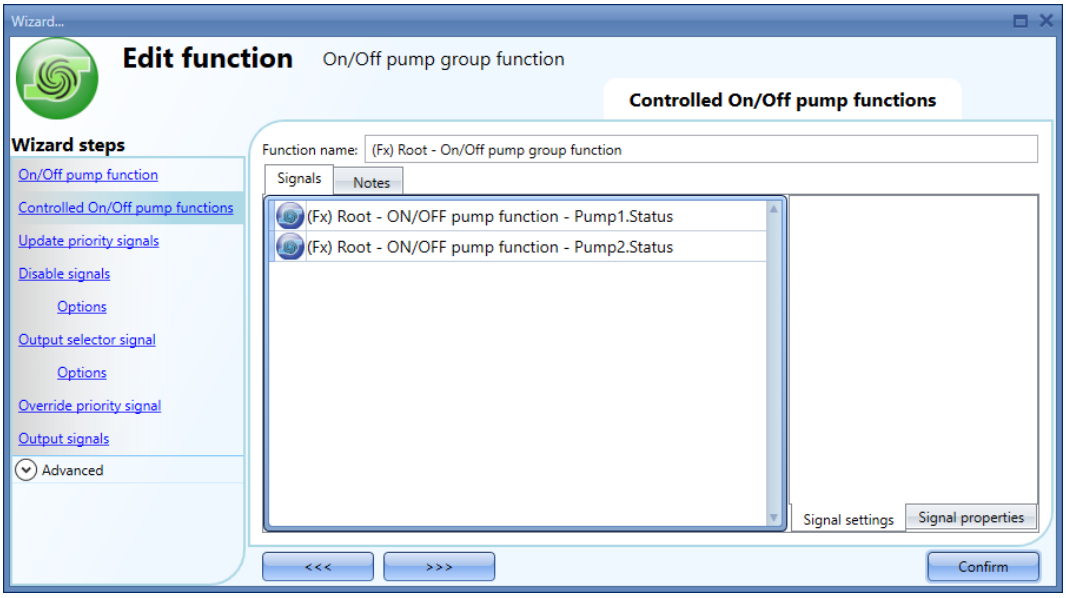
10.36.2 How to select the controlled On/Off pump functions

In the *Controlled On/Off pump functions* field of the wizard, the user has to select the *On/Off pump* functions (as described above), to be managed by the *On/Off pump group* function.

By double clicking in the *Signals* area, a window will appear with all the available signals: two or more *On/Off pump* functions can be linked to the same *group* function (see below):



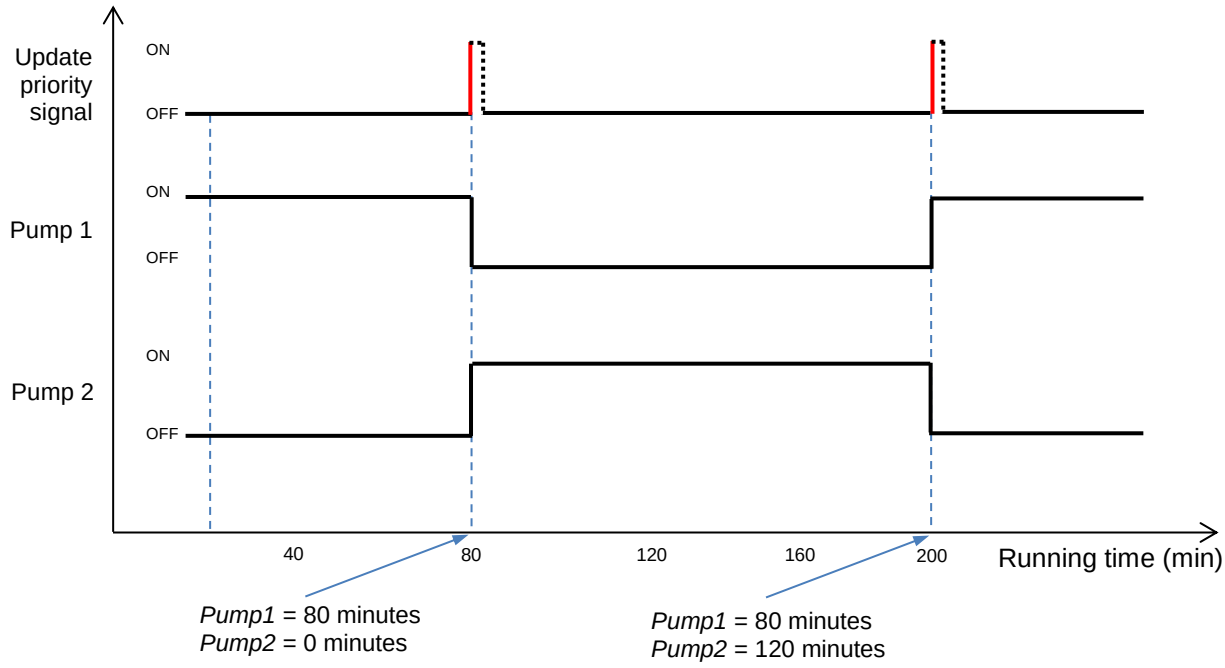
Up to 50 *On/Off pump* functions can be selected for each *On/Off pump group* function.



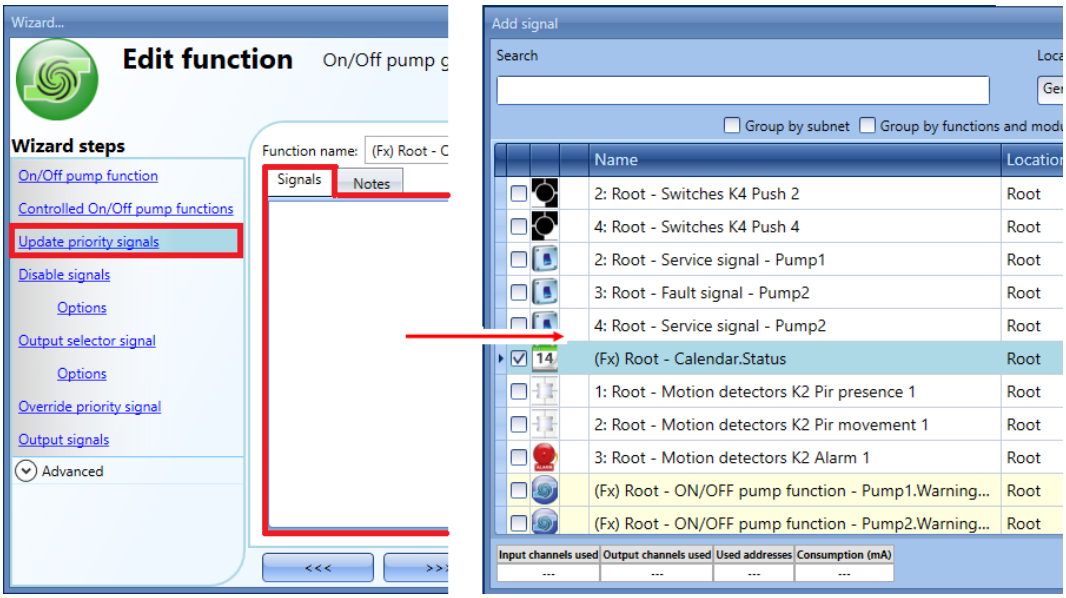
10.36.3 How to manually update the priority list based on the working hour counters

When at least one of the update priority signals is triggered, the *On/Off pump group* function compares the working hour counters of the controlled functions and updates the priority list, which is used to select which *On/Off pump* function has to be activated first: the loads with fewer working hours will have the highest priority.

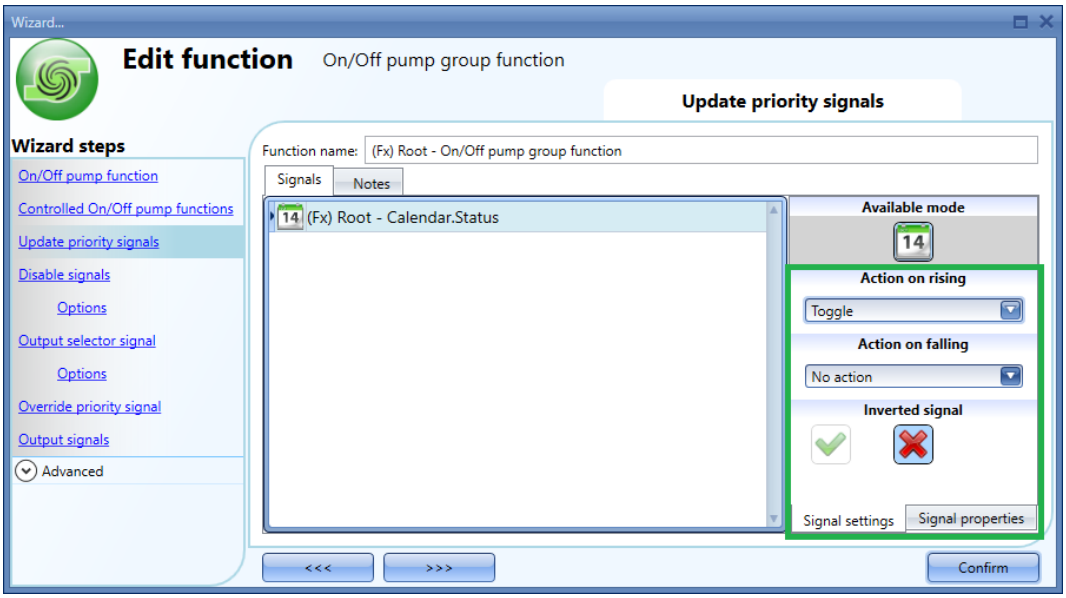
As shown in the example below, after the system start-up, the *On/Off pump* functions (*Pump1*, *Pump2*) are activated by the *On/Off pump group* function according to the priority list: Pump1 goes ON and Pump2 remains OFF. When the *Update priority signal* is triggered 80 minutes later, since the working hour counter of *Pump1* is 80 minutes, while the one of *Pump2* is 0 minutes, ***Pump1* will be immediately replaced by *Pump2***. Next time the *Update priority signal* is triggered, for example 120 minutes later than the first trigger, the working hour counter is 80 minutes for *Pump1* and 120 minutes for *Pump2*: ***Pump2* will be immediately replaced by *Pump1*.**



N.B. *The On/Off pump* functions in fault or service conditions will not be used.

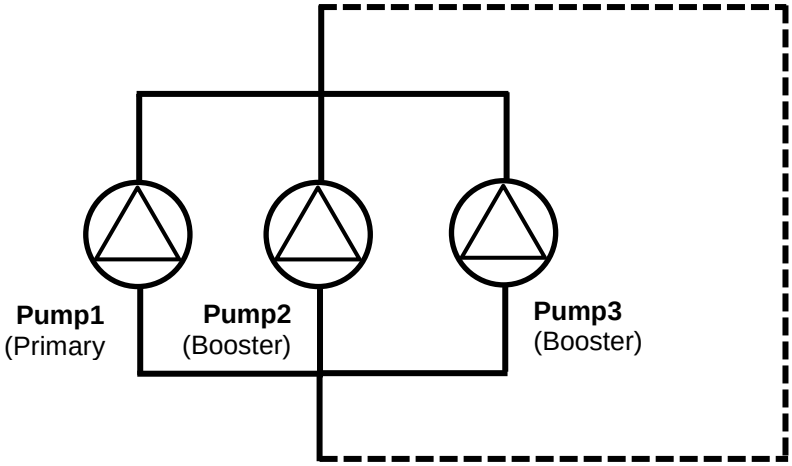


If a level signal is selected, such as a switch or a function such as a *Calendar*, the available actions are **Action on rising** (i.e. when the switch is closed or the function is activated), **Action on falling** (i.e. when the switch is open or the function is deactivated), or both (see the green box below).



10.36.4 How to activate more than one load at the same time

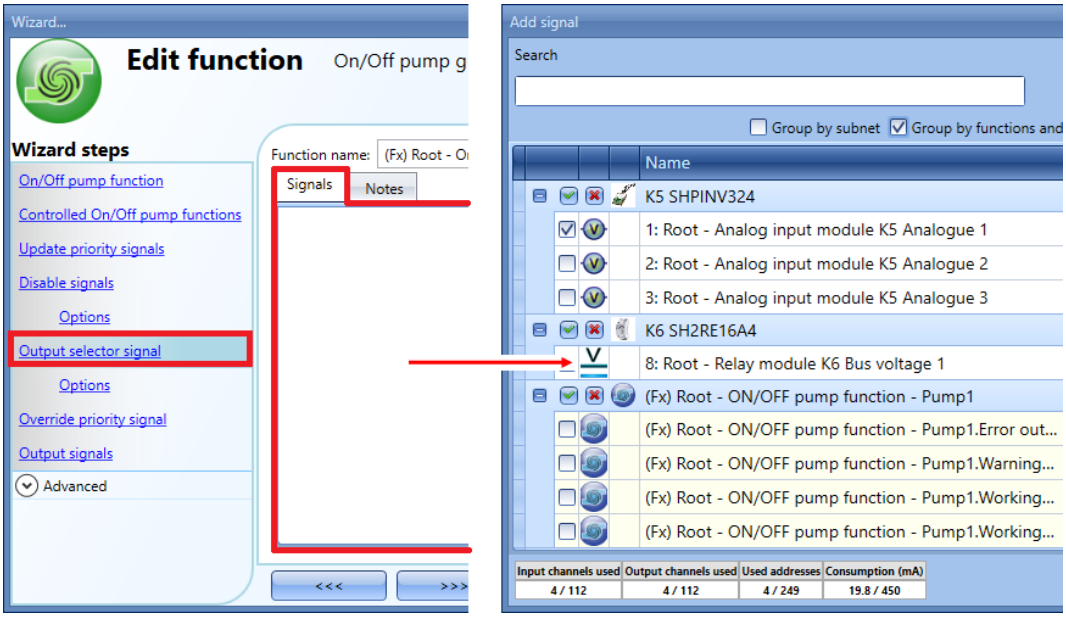
The *On/Off pump group* function can manage more than two *On/Off pump* functions in parallel at the same time. The parallel working mode can help when (e.g. in a water circuit) the flow has to be increased by adding additional pumps, instead of the operation of just one.
As shown in the example below, *Pump1* works in normal conditions. When further boost is required by the system, *Pump2* and *Pump3,* connected in parallel together with *Pump1,* are activated.



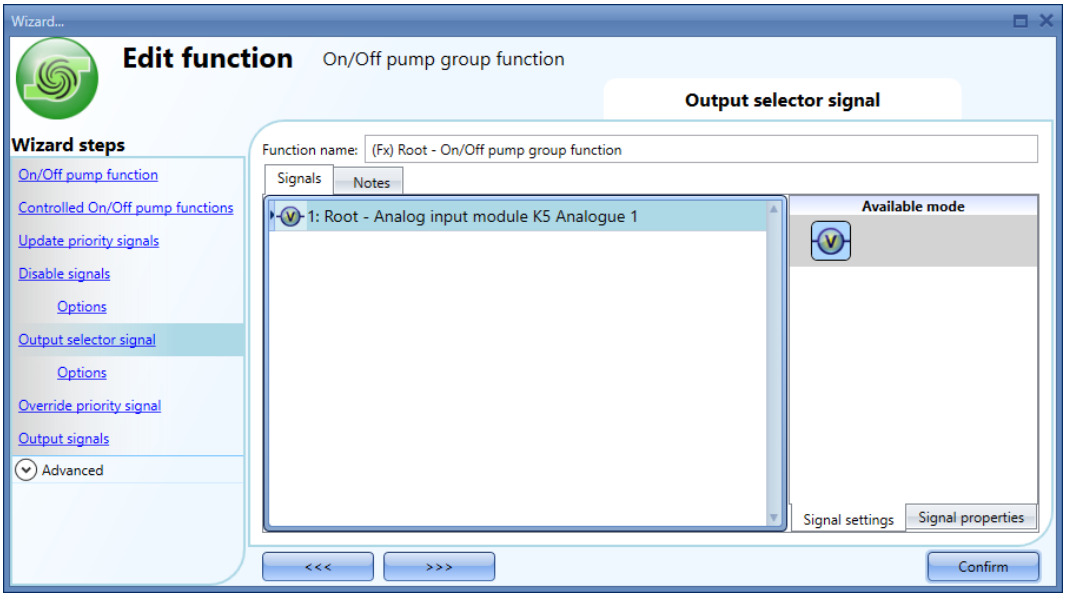
The number of outputs (the number of pumps that have to be activated in parallel) can be dynamically defined by an analogue input signal in the range of 0..10V, called the *Output selector signal*.

10.36.4.1 How to add the output selector signal

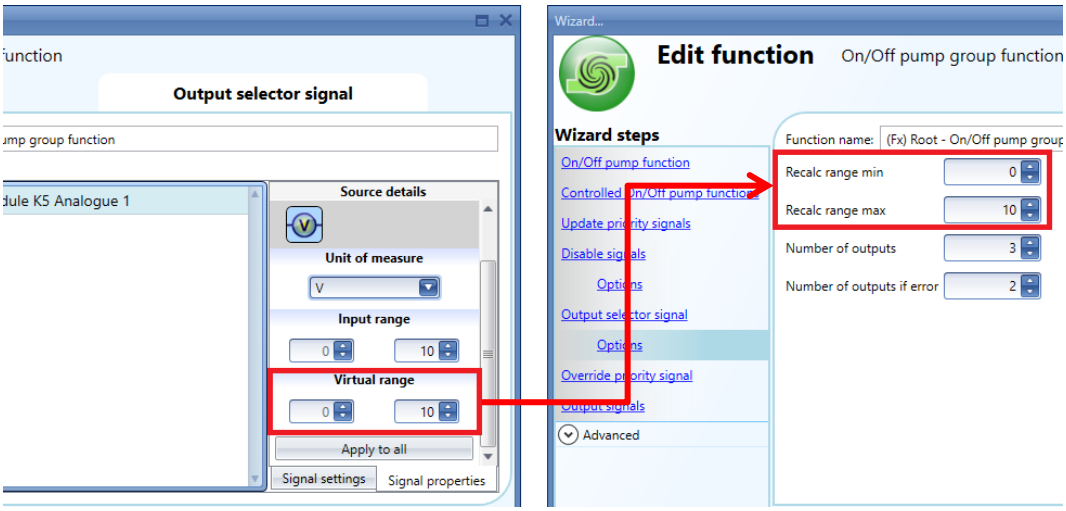
To add the output selector signal, the user should click on the *Output selector signal* field of the wizard, then double click on the *Signals* window and select the signal from the list of those available (see below):



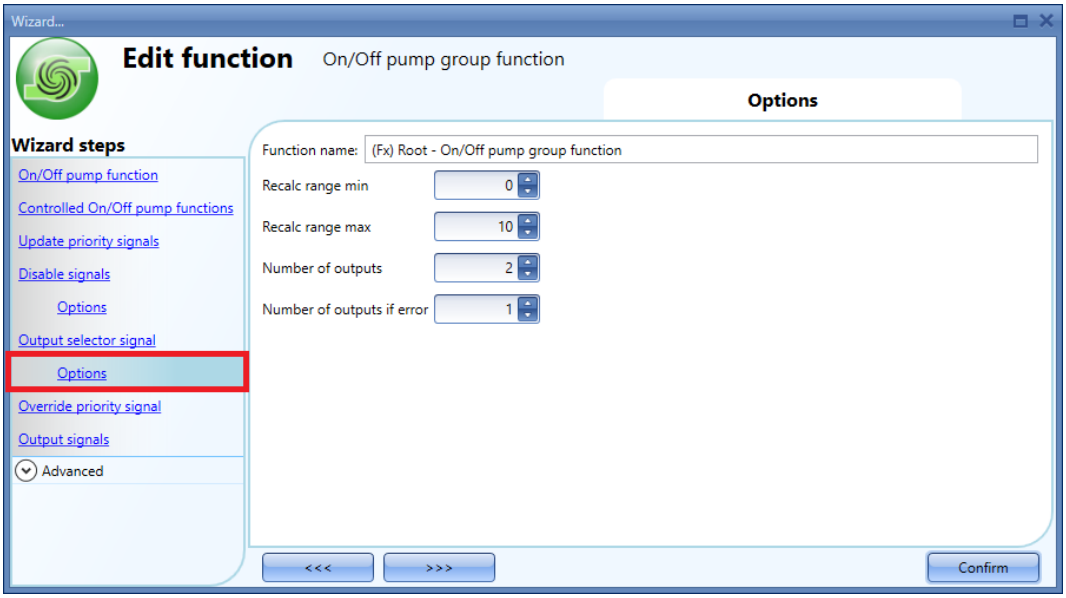
Only one analogue input signal in the range of 0..10V can be selected. The analogue input modules that are available in this field are the SHPINV2T1P124 and the SHPINV324.



When the analogue input signal is present in the *Output selector signal* field, in the *Signal settings* area the "virtual range" of the selected unit of measurement has to be set in the *Virtual range* field, while the relevant input value has to be set in *Input range (*it could be a 0-10V value or a percentage, according to the selected input*)*. The *Recalc range min* and *Recalc range max* fields in the *Options* window are automatically updated with the values inserted in the *Virtual range,* as shown in the red rectangles below.



To configure the *Output selector signal* options, the user has to click on the *Options* field of the wizard: a window with different options will appear (see below):



Recalc range min
This defines the minimum value of the Output selector signal. It is used to scale the Output selector signal. When the Output selector signal is present, it will be the min value of the *Virtual range*.

Recalc range max
This defines the maximum value of the Output selector signal. It is used to scale the Output selector signal. When the Output selector signal is present, it will be the max value of the *Virtual range*.

Number of output
When the Output selector is not present, this defines the number of outputs (e.g. pump) that have to be activated in parallel. The outputs with fewer working hours will be selected.
When the Output selector signal is present, the parameter defines the maximum number of virtual outputs that have to be activated at the same time.

Number of output if error
When the Output selector signal is present but its value is not available (or outside the permitted range for analogue inputs 0..10V), the parameter defines the number of outputs that have to be activated.

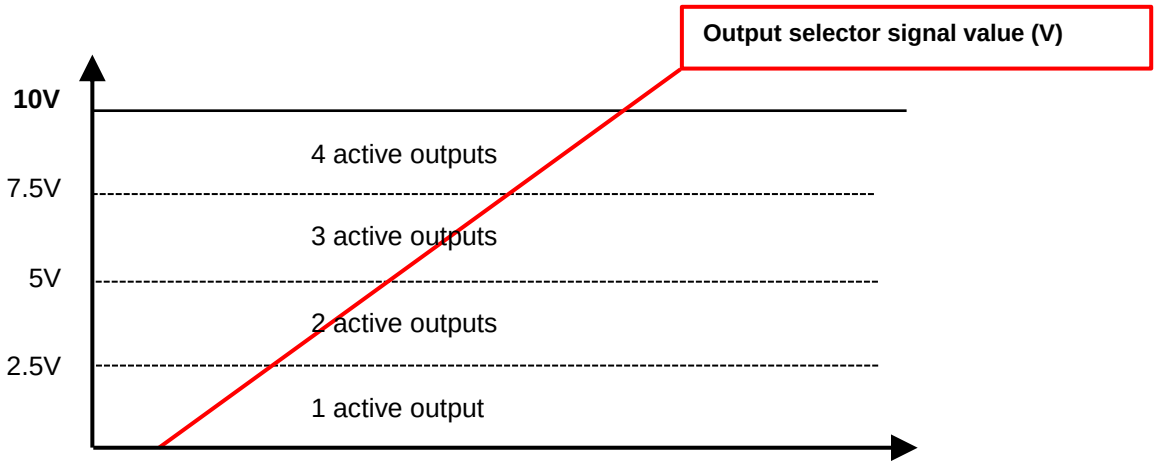
Example

The output selector signal value is used to obtain the number of outputs that have to be activated in parallel *(Number of outputs)*. On the basis of *Number of outputs,* the analogue input signal range (0 to 10 V) is divided into steps according to the following formula:

Steps = [Number of outputs -1]

Thresholds are then derived to decide how many outputs have to be activated at the same time.

The sub-ranges in which the input value lies will determine how many outputs will be activated, as shown in the graph below, where the user needs to manage up to 4 outputs (i.e. pumps) in parallel at the same time, therefore the signal range is divided into three steps and three thresholds.



The number of outputs that will be activated depends on the sub-range where the analogue input value lies:

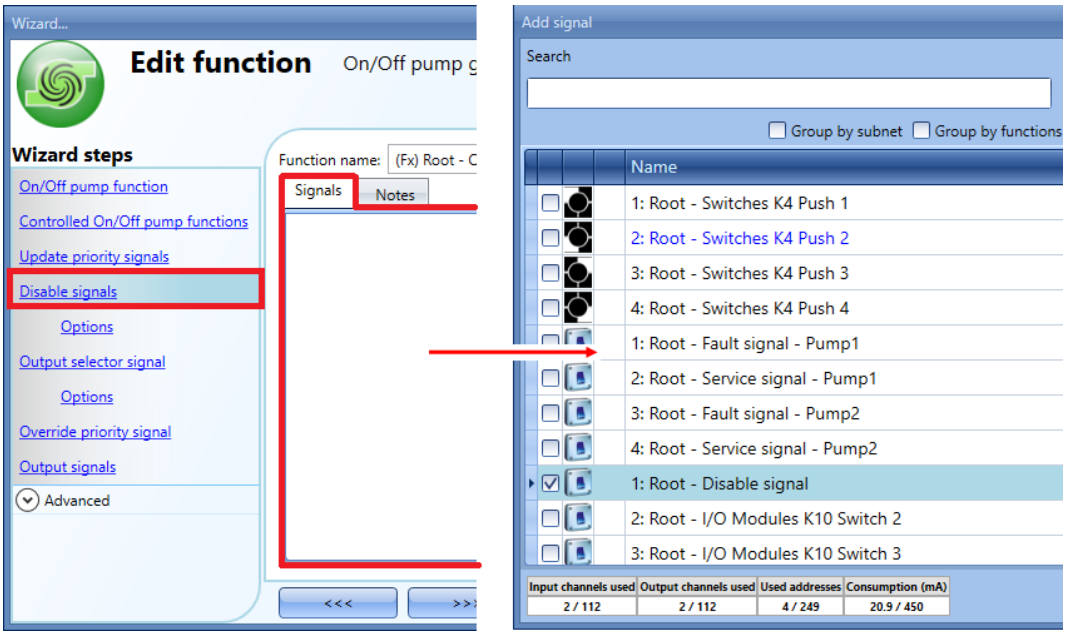
0 – 2.5V	=	1 output will be activated
2.6 – 5 V	=	2 outputs will be activated
5.1 – 7.5V	=	3 outputs will be activated
7.6 – 10V	=	4 outputs will be activated

Each time the analogue input value goes above or below a threshold, the number of outputs will be changed immediately.

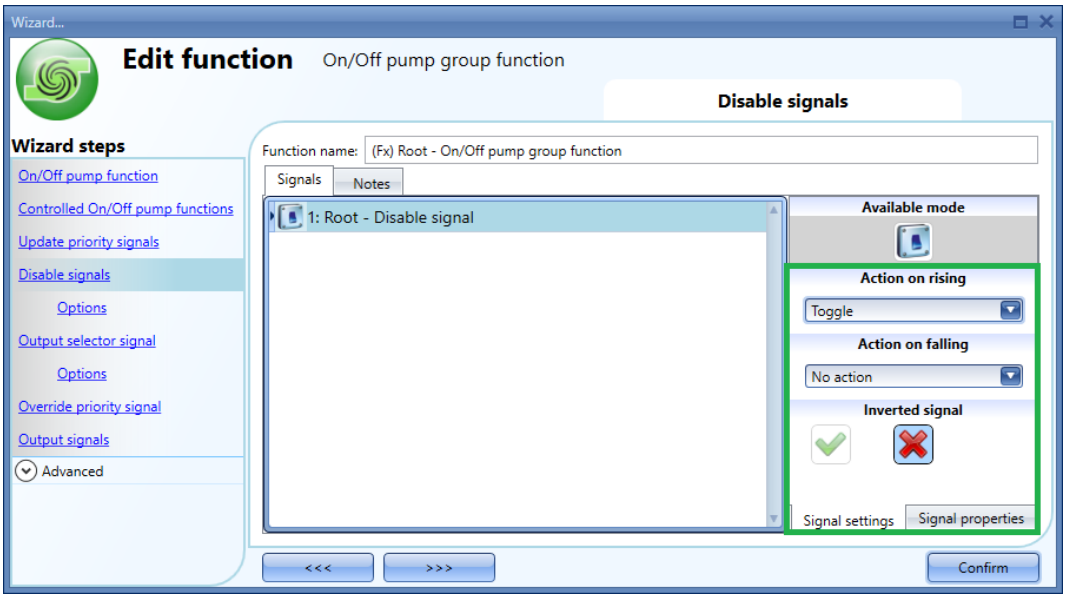
10.36.5 How to disable the automation

The user can disable the activation of a group of pumps by means of a disable signal. When a disable signal is active, all the *On/Off pump* functions are set to disable status (the output value will be set to 0) and the command signal value will be ignored.

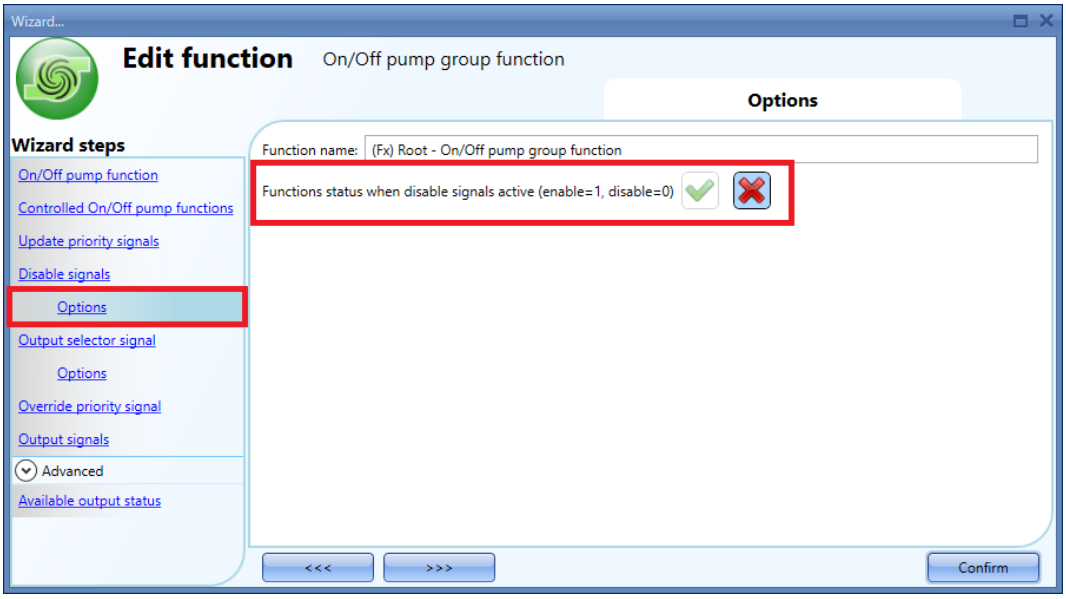
To add the disable signals, the user has to click on *Disable signals*, then double click on the *Signals* window and select the input signals from the list: up to 50 *Disable signals* can be added (see below).



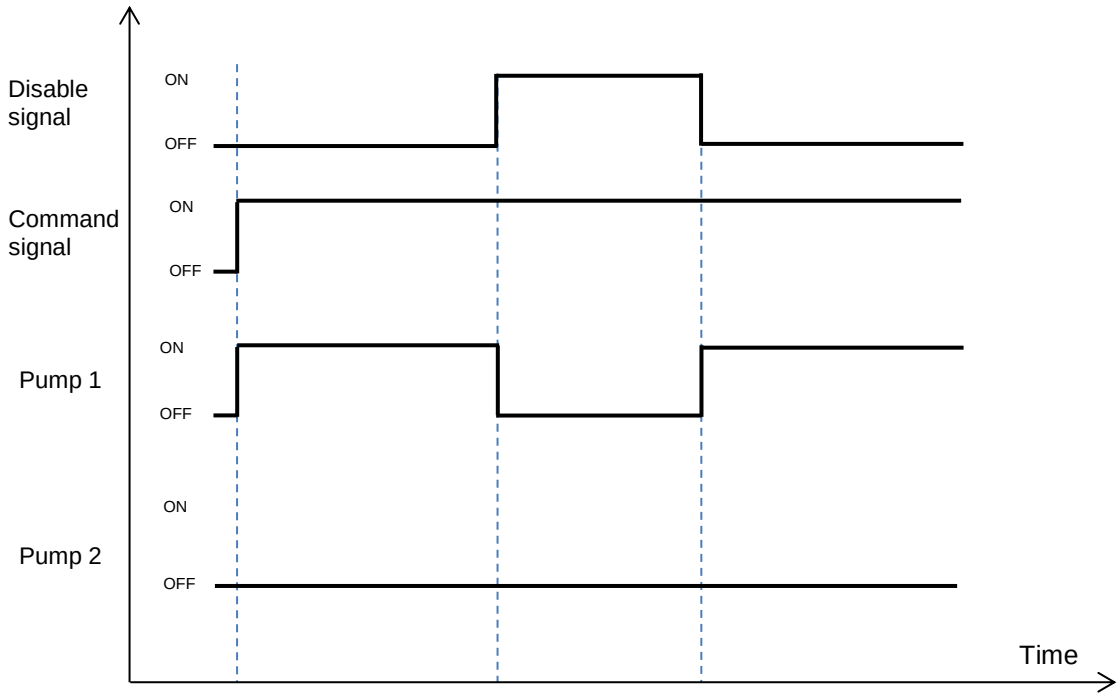
If a level signal is selected, such as a switch, or a function, the available actions are ***Action on rising*** (i.e. when the switch is closed or the function is activated), ***Action on falling*** (i.e. when the switch is open or the function is deactivated), or both. The inverted logic for the *Disable signal* can be enabled by selecting the green *V* in *Inverted signal* (see the green box below).



In the *Options* field *of Disable signals*, if the green *V* icon is selected for *Functions status when disable signals active*, the function status of the connected *On/Off pump* functions will be set as **active**: otherwise, if the red *X* is selected, the function status will be set as **non-active.**

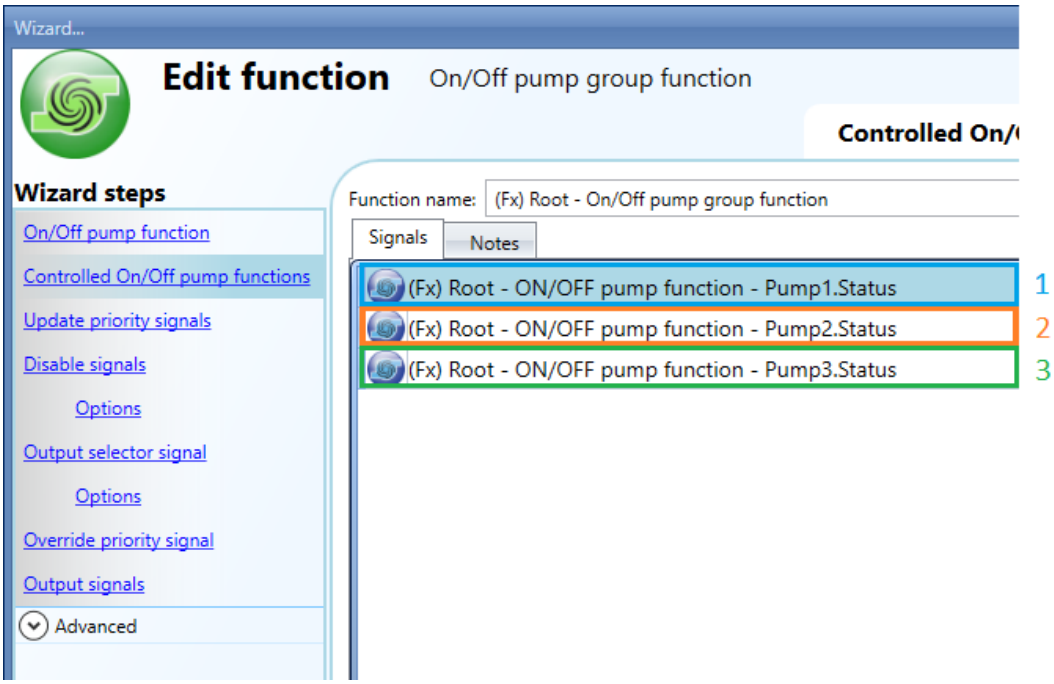


Example
As shown in the graph below, as soon as the *Disable signal* is activated, the outputs of all the *On/Off pump* functions will be turned OFF (the function status will be set to Disable).
Even if the Command signal is set to ON, it will not be considered while the Disable signal is active.

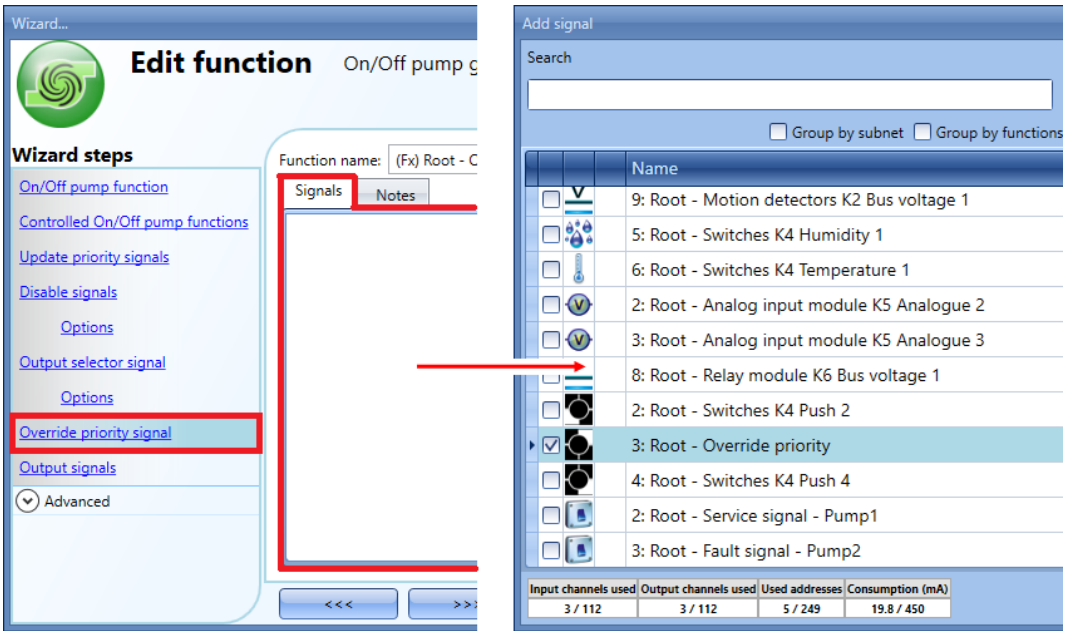


10.36.6 How to manually alternate the activation of the loads in a group

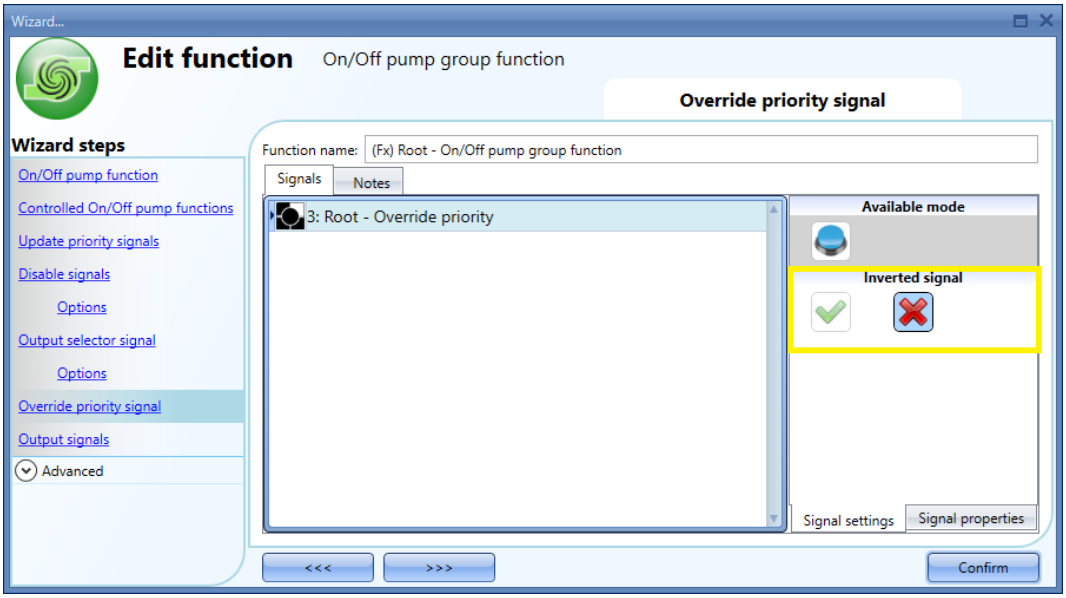
When the *On/Off pump group* function is active, the override priority signal allows the immediate selection of a specific *On/Off pump* function regardless of the priority list based on the working hour counters. When the override priority signal is triggered, the selected *On/Off pump* functions will be activated according to the order in which the functions have been added in the *Controlled On/Off pump functions* window, as presented in the picture below.



To add the override priority signal, the user has to select the *Override priority signal* field, then double click on the *Signals* window and select the input signal from the list of those available.
Only one signal can be selected (see below).



The override signal can be a *digital input* or an *analogue input* signal. The inverted logic can be enabled by selecting the green *V* in *Inverted signal,* as shown in the yellow rectangle below.

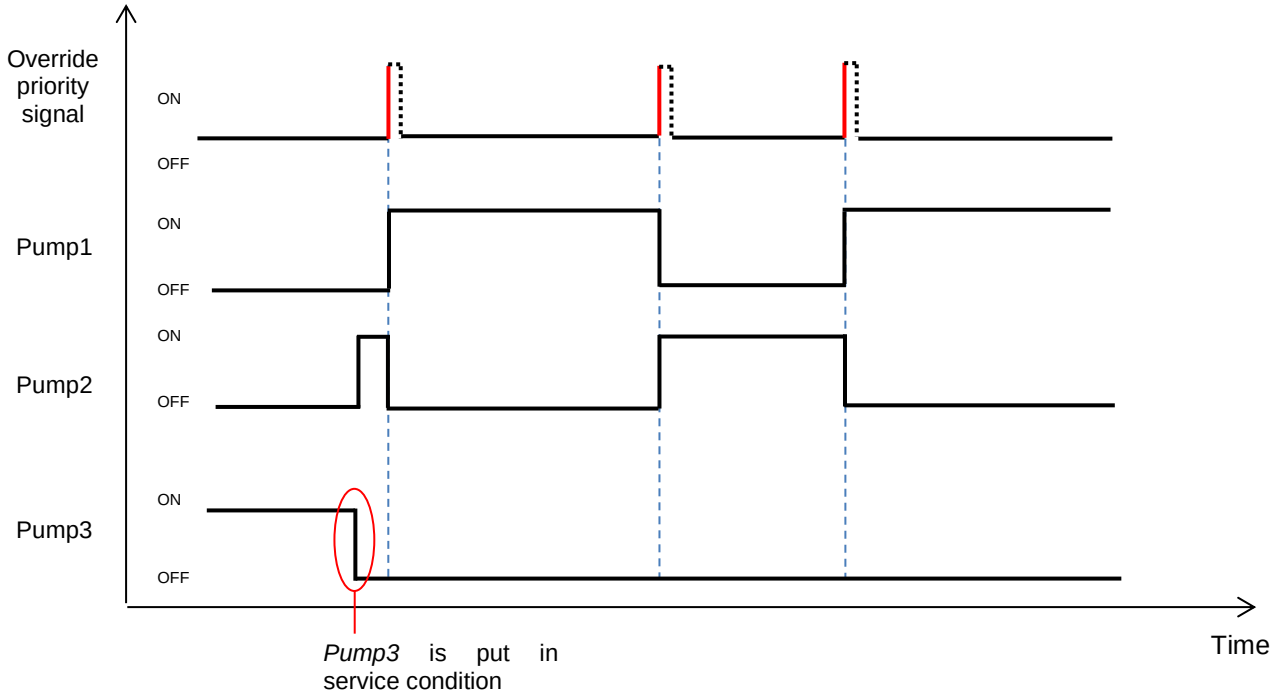


10.36.6.1 How to manually alternate the activation by means of a digital signal

For a *digital input* signal, such as a level function, a switch or a pushbutton, the action will be triggered on the activation only. Each time the override priority signal is triggered, the next available *On/Off pump* function in the list will be activated. When the last *On/Off pump* function in the list is reached, the next time the override priority signal is triggered, the first *On/Off pump* function in the list will be re-activated.

This is useful when the user needs to manually alternate the *On/Off pump* functions controlled by the group function.

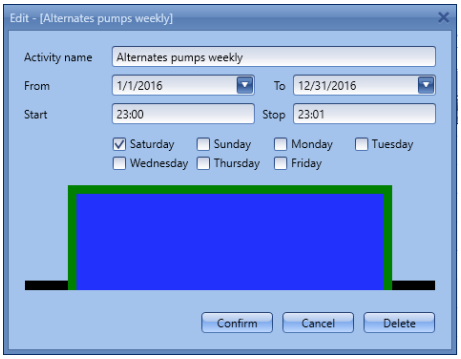
Example
As shown in the graph below, the user has to replace an old pump with a new one in an array of pumps [*Pump1, Pump2, Pump3*]. Let us assume that *Pump3* is replaced with a new one: the newer pump would be always selected instead of *Pump1or Pump2*, so the user needs a way to activate *Pump1* or *Pump2* manually.



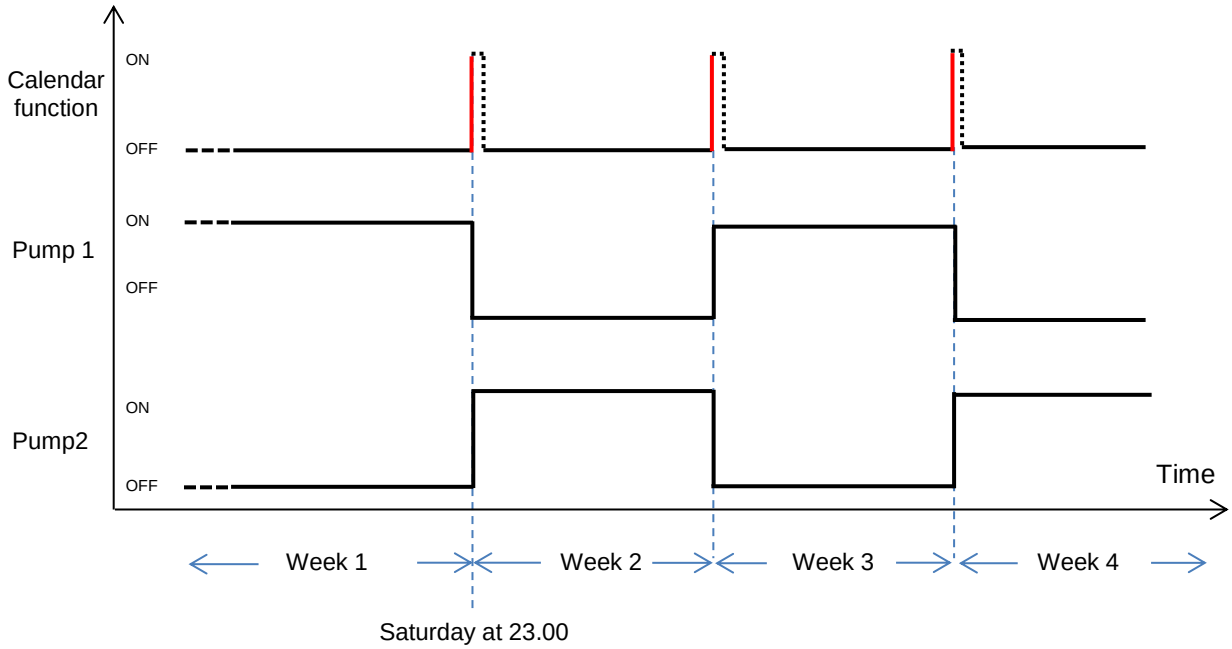
10.36.6.2 How to automatically alternate the activation by a Calendar function

This feature is useful in a scenario where the user wants to automatically alternate the operation of the *On/Off pump* functions according to a *Calendar* function.

Example
As shown below, a scheduler activity of a *Calendar* function is configured as a level signal to override the priority list based on the working hour counters of the *On/Off pump group* function. The calendar triggers the change at exactly 23.00 on Saturdays.
Since this action has to be done every week of the year, the dates 1st January and 31stDecember have to be inserted in the fields **From** and **To**.



As shown in the graph below, after the *On/Off pump group* function start-up, the *On/Off pump* functions are activated according to the priority list: *Pump1* goes ON and *Pump2* remains OFF.
According to the settings in the figure above, the calendar action is triggered ON at 23.00 on Saturdays: as a result of the automation, the next available *On/Off pump* function in the list will be activated, in this case *Pump2 will be turned ON and Pump1* will be turned OFF. *Pump2* will remain active until the next time the override priority signal is triggered by the *Calendar* function: after that, the *On/Off pump group* function will switch the operations between *Pump1* and *Pump2*.



10.36.6.3 How to manually alternate the activation by means of an analogue signal

If an analogue signal, such as a *Counter* function, is used to manually alternate the loads, the action will be triggered at every input variation. The user has to consider these rules:

If the analogue input value is <=0 - no override action will be performed;

If the analogue input value is >0 - the analogue input signal value will be rounded to an integer number and it is used as an "index of the pump to activate": the output will be activated according to the order in which the functions have been added in the *Controlled On/Off pump functions* window.

In the example below, the analogue value of a *Counter* function is used as *Override priority signal* on a group function with 3 pumps :

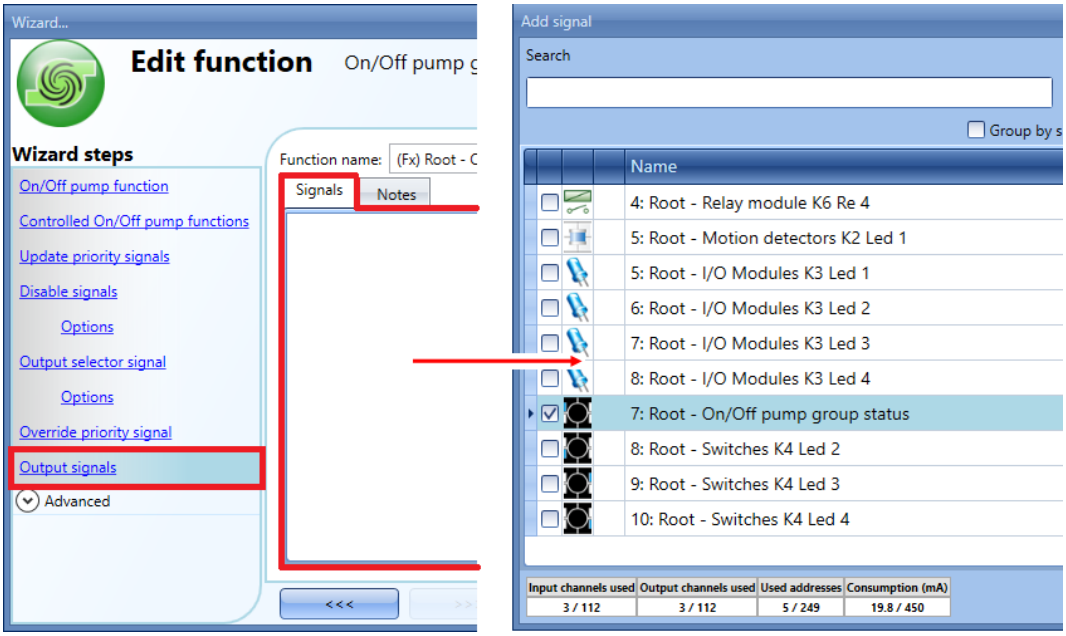
- *Counter* value **1.2** is rounded to **1** = the first On/Off pump function (Pump1) will be activated;
- *Counter* value **2.7** is rounded to **2**= the second On/Off pump function (Pump2) will be activated;
- *Counter* value **3.5** is rounded to **3**= the third On/Off pump function (Pump3) will be activated;

If the selected *On/Off pump* function is in fault state or in service condition, the override command is ignored and the status will remain unchanged for all pumps.
Otherwise, the selected *On/Off pump* function will remain activated until either:

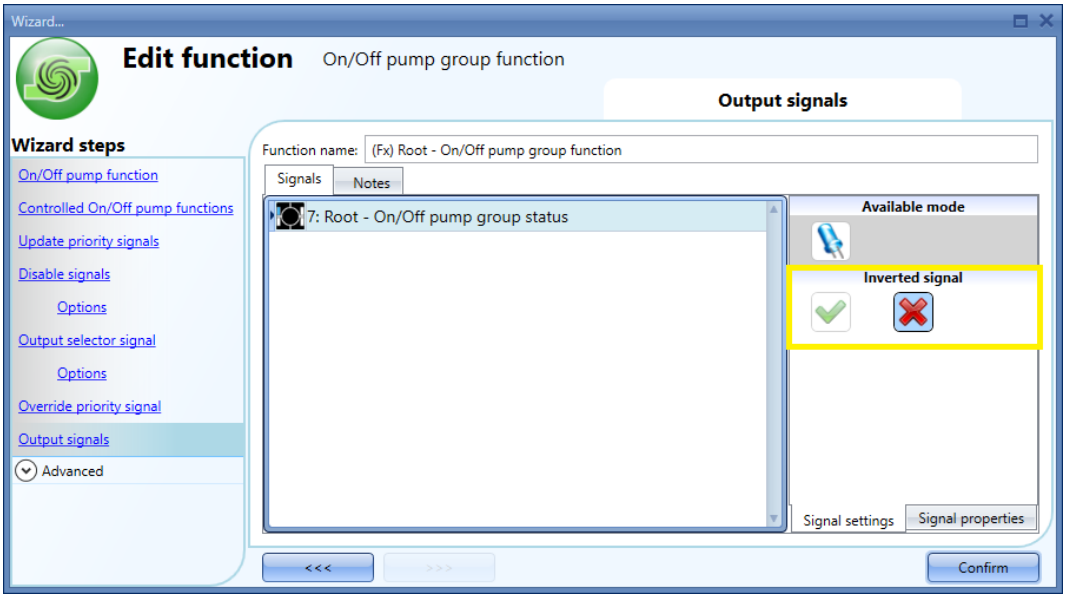
- it is changed to fault status or service condition; or
- the *On/Off pump group* function is turned OFF. If the override priority signal is a *digital input*, the standard priority list based on the working hour counters value will be applied when the *On/Off pump group* function turns ON again.

10.36.7 How to add the output signals

To select the feedback signal that indicates the status of the *On/Off pump group* function, click on *Output signals* and then double click on the *Signals* window: up to 50 signals can be chosen and they are managed in parallel (see below):



The signals available in this window are LEDs and relays.
The logic of each feedback signal can be set as normal or inverted (see yellow rectangle).



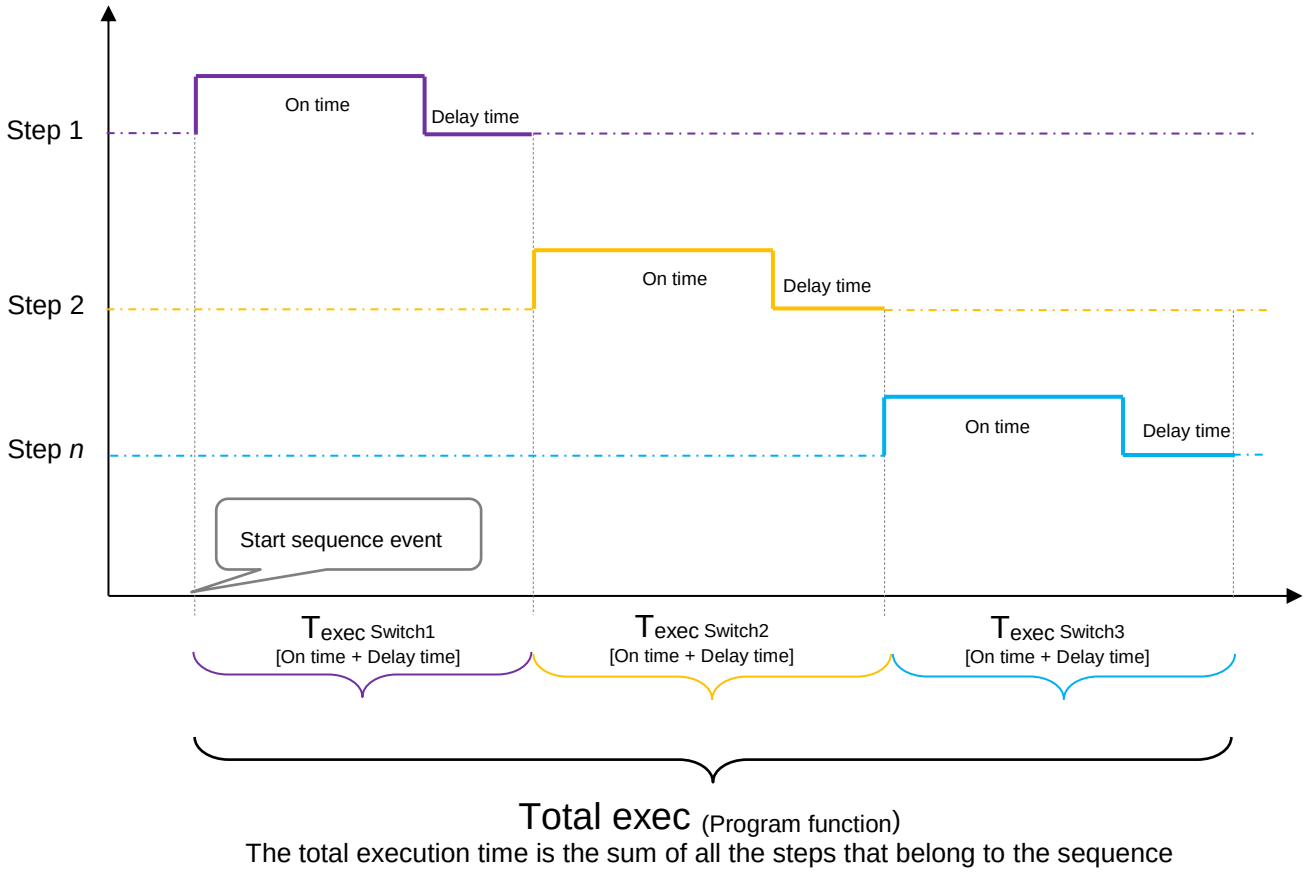
10.37 How to set the Program function

A *Program function* is a sequence that is just an ordered list of steps: each step in a sequence is identified by an index number that represents the sequential order in which the steps will be executed when the sequence is started. A *Program function* allows the user to define, for the selected *Switch* functions, the activation time and the sequential order in which the steps are executed.

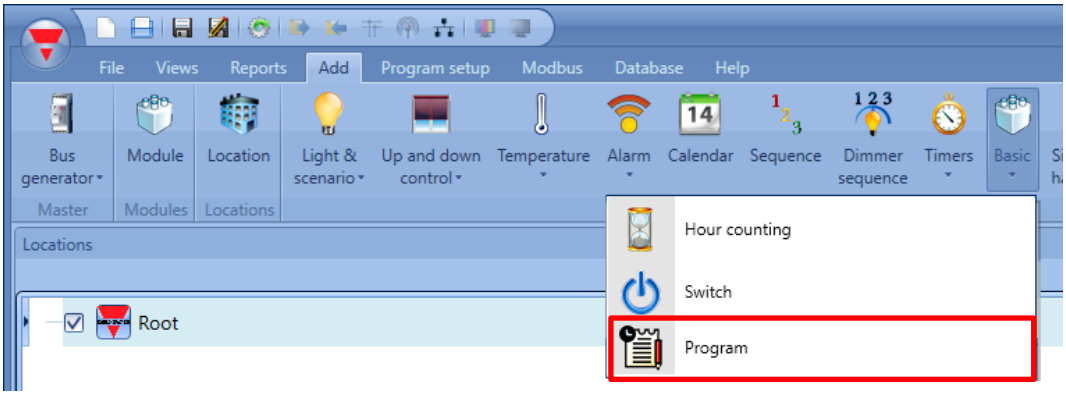
The functions that can be controlled are *Switch functions* only. *See the procedure How to add a Switch function.*

A *Switch* function can be used in more than one Program function. The Switch function always executes the last command received.

The *Program* function starts activating the first step in the list: all the *Switch* functions that belong to the step will be switched ON for the *On time* period. When this period expires, the step is deactivated and all the related *Switch* functions will be switched OFF. Between one step and the next, the system waits for the *Delay time* period, then the Program function goes ON to activate the other steps, following the predefined order, until the last step in the list is executed.

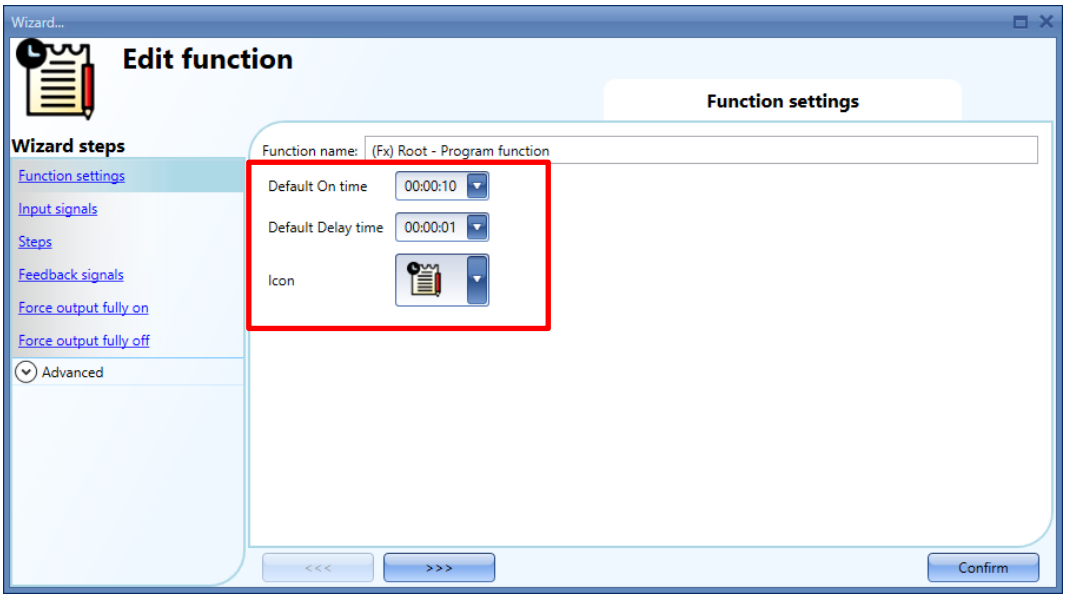


To add a *Program* function the user should select *Program* from the *Add* menu (see below): the new function will be added in the selected location.



10.37.1 How to set the Function settings

As shown below, in the *Function settings* tab of the *Program* function, the following parameters can be set by the user:



Default On time (default is 00:00:00)
In this field the user can set the *On time* that will be applied to any step which is added to the Steps list. The user can change the default value by clicking the arrow in the related time field. The format is *HH:MM:SS*.

Default Delay time (default time is 00:00:05)
In this field the user can define the default Delay time that will be applied to any step that is added to the *Program* function. The user can change the default value by clicking the small arrow in the related time field. The format is *HH:MM:SS*.

Icon
The user can set an icon to the *Program* function, selecting from the list, as shown below. **The selected icon will be shown both in the UWP 3.0 Tool and in the WEB-App interface**:

Icon

10.37.2 How to command the Program function with Input signals

The *Program function* can be commanded by using input signals, such as push buttons, switches and functions. The following actions can be performed*:*

Start action (all steps)
As soon as a *Start action* is performed, all the configured steps of the sequence will be started (from the first step to the last step in the *Steps list*) according to the configuration.

Start action (selected steps)
As soon as a *Start steps action* is performed, only the selected steps, as defined in the two numeric fields will be started according to the configuration. **The Start steps action will not be performed if a Start action is already running**: the user has to stop the *Program function* first and when the Program function is inactive, start it again.

Pause action
As soon as a *Pause action* is performed, the *Program function* will be paused immediately (the *Pause action* can be performed only if the *Program* function is running).

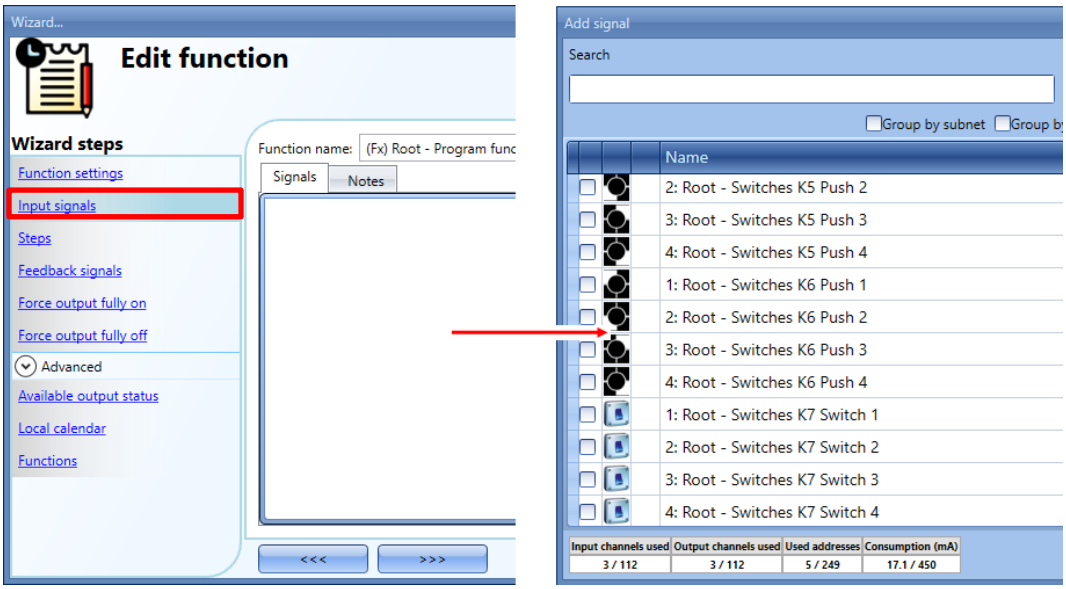
Note: If a Step is active, the behaviour of the *Pause action* is as shown in the table below:

Step condition	Behaviour of pause action
On time	When the sequence is paused, the *On time* timer of the current step is frozen; As soon as a Start action is performed again, the sequence starts again from the point at which the step was paused before.
Delay time	When the sequence is paused, the Delay time countdown will continue. As soon as a Start action is performed again the next step is started.

Stop action
When a *Stop action* is performed while the *Program* function is running, the current step is stopped immediately. In order to bring all the steps to disabled status, the system will send a *Stop action* starting from the first step to the last according to the *Delay time* values.
This means a sequence is completely ended after all the Delay time values of the sequence.

1.1.1.1 How to set the action to an Input signal

The user has to select the Input signals tab of the *Program* function, then double click on the *Signals* window and select the input signal from the list of those available (see below):



The available signals can be push buttons, switches and functions. The following table shows the actions available for push buttons:

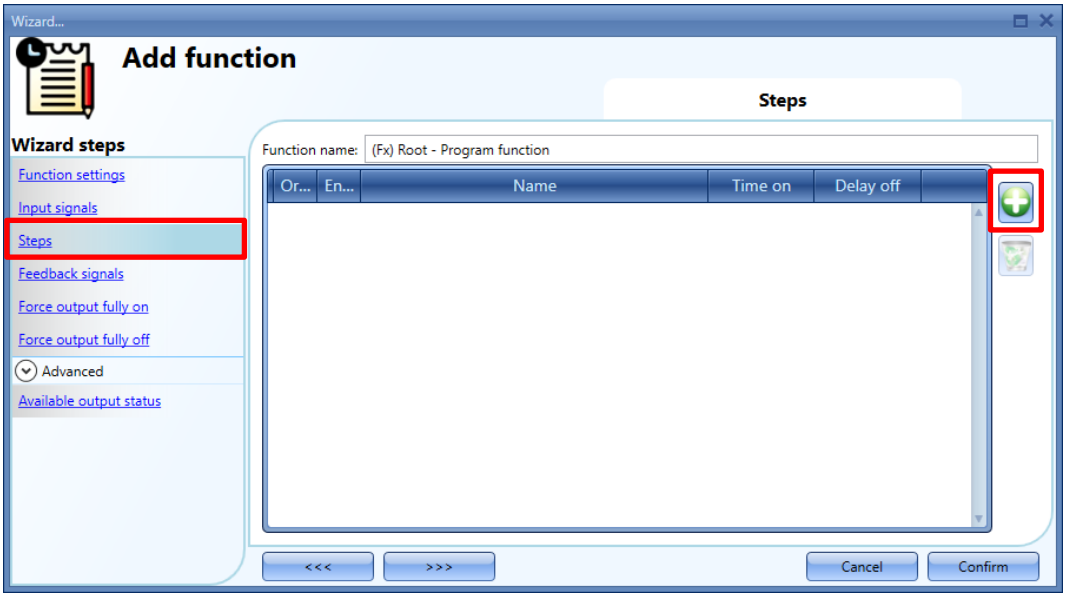
Event type	Action			
	Start (All steps)	**Start steps** (ONLY the selected range) 1 1	**Pause**	**Stop**
Action on rising	As soon as the push button is pressed, the Program function is started	As soon as the push button is pressed, only the selected steps are started	As soon as the push button is pressed, the Program function is paused	As soon as the push button is pressed, the Program function is stopped
Action on click	As soon as the push button is clicked, the Program function is started	As soon as the push button is clicked, only the selected steps are started	As soon as the push button is clicked, the Program function is paused	As soon as the push button is clicked, the Program function is stopped
Action on long click	After a long press, the Program function is started	After a long press, only the selected steps are started	After a long press, the Program function is paused	After a long press, the Program function is stopped
Action on very long click	After a very long press, the Program function is started	After a very long press, only the selected steps are started	After a very long press, the Program function is paused	After a very long press, the Program function is stopped

The following table shows the actions available for switches and functions:

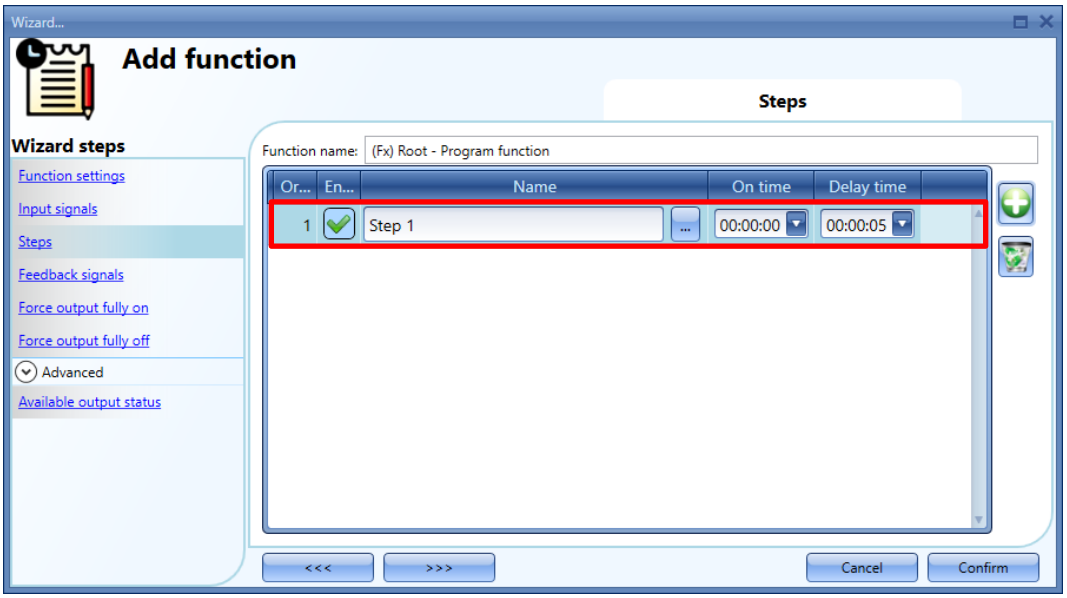
	Action			
Event type	**Start** (All steps)	**Start steps** (ONLY the selected range) 1 1	**Pause**	**Stop**
Action on rising	As soon as the switch is closed/, function goes ON, the *Program* function is started	As soon as the switch is closed/function goes ON, the selected steps are started	As soon as the switch is closed/function goes ON, the Program function is paused	As soon as the switch is closed/ function goes ON, the Program function is stopped
Action on falling	As soon as the switch is opened/ function goes OFF, the Program function is started	As soon as the switch is opened function goes OFF, the selected steps are started	As soon as the switch is opened/function goes OFF, the Program function is paused	As soon as the switch is opened/ function goes OFF, the Program function is stopped

10.37.3 How to add steps to the Program function

To add a step to the *Program* function, the user has to select the *Steps* tab, then click on the *icon* as shown below:



A new *Step* item will be added to the *Steps* window, as indicated in the red rectangle below:



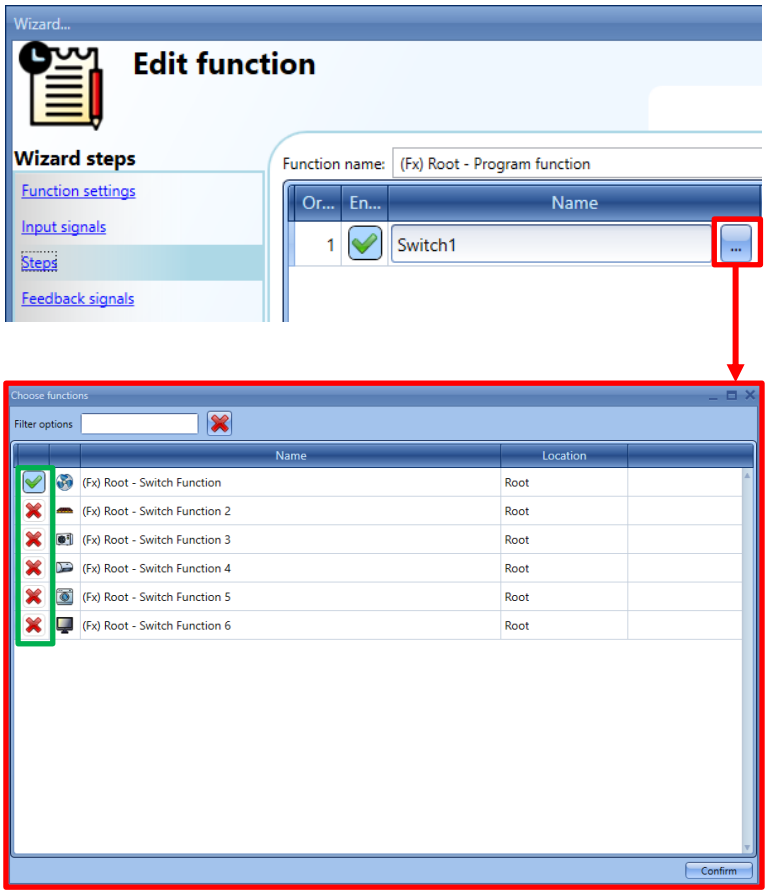
In the Step window the user can add up to 100 steps.

2.1.1.1 How to configure a step in a sequence

Once a step is added, to configure it in a sequence the user can follow the procedure below:

1) In the *Name* field the user can change the default name (Step *n*) giving it its proper name.

2) By clicking the ... *Choose function* button, the following window will appear:



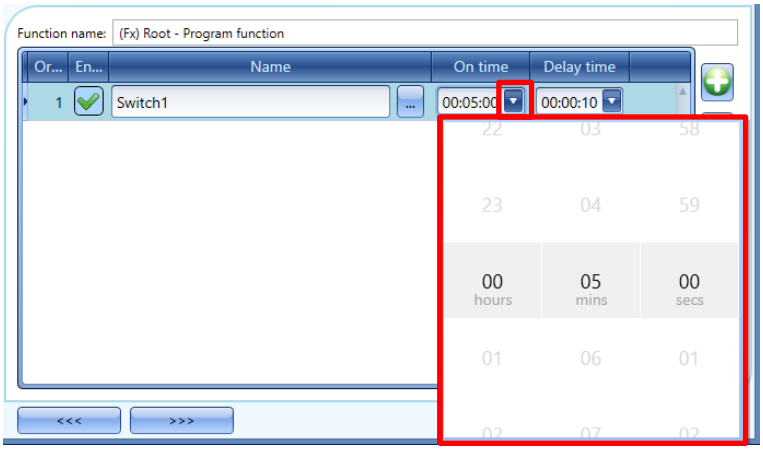
In the *Choose functions* window the user can select which *Switch* functions have to be added to the related step by clicking the small red cross, the function will be selected (a small *V* icon appears).

All the Switch *functions* enabled in the same step are managed in parallel mode.

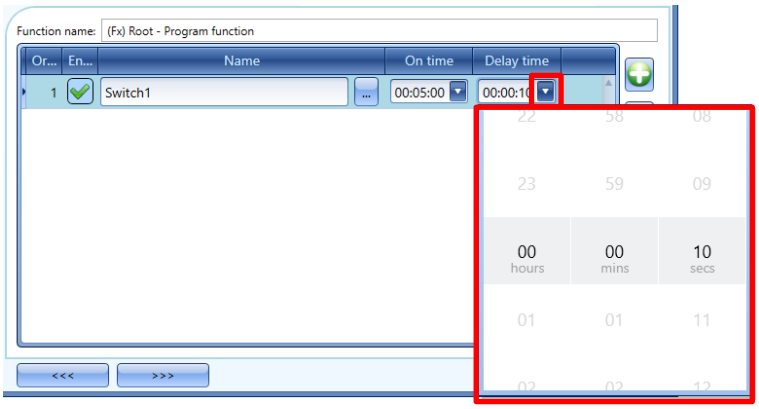
Note: A Switch function can be added to a step once.

A maximum of 50 *Switch* functions can be added to each step. Click the *Confirm* button to close the *Choose function* window.

3) In the *On time* field, the user can set the period in which the step has to be activated. To change the default value, click the small arrow icon and select the *Hours, Minutes, Seconds* as highlighted by the red rectangle below:



4) In the *Delay time* field, the user can set the delay between the activation of two steps (the maximum time which can be set is 5 minutes), as shown in the red rectangle below:

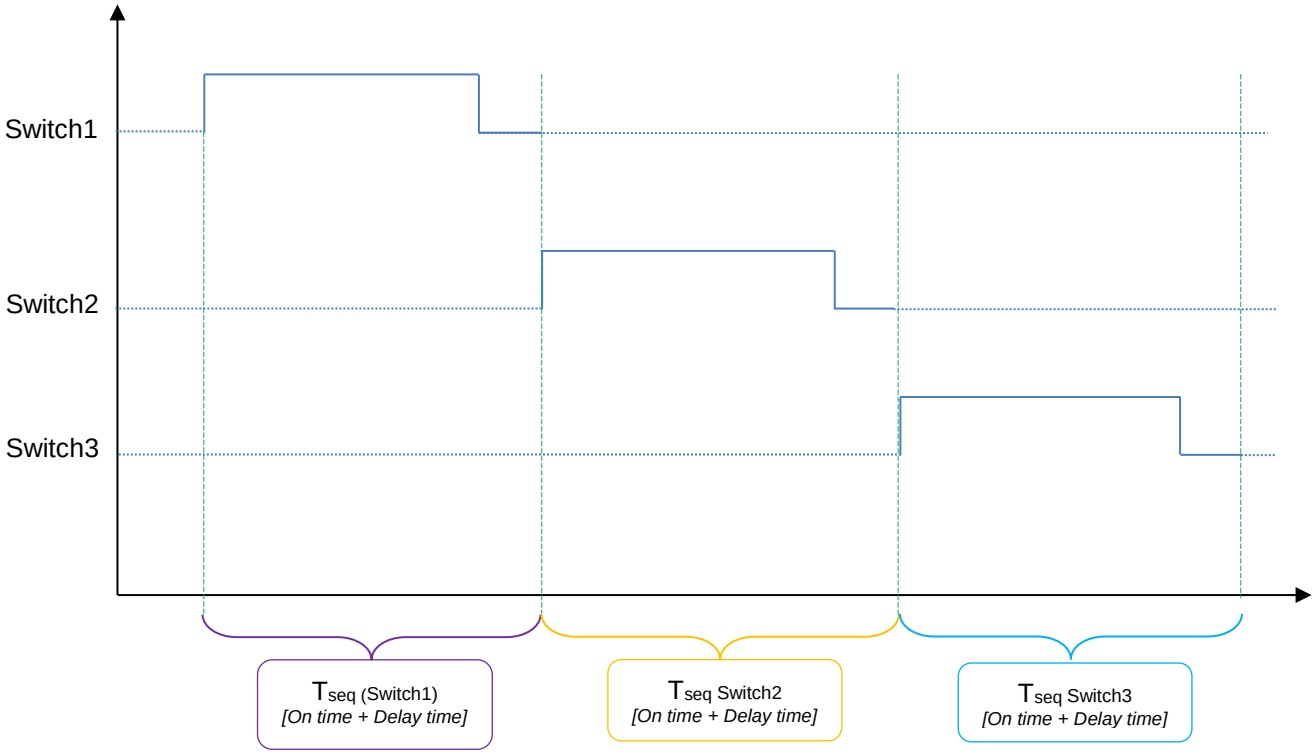


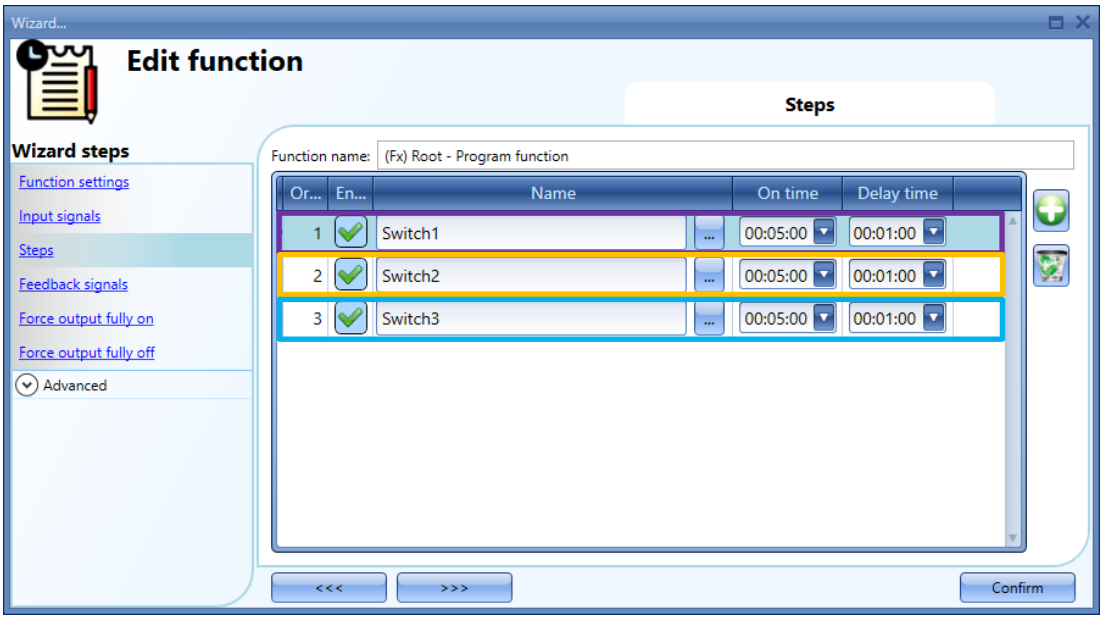
N.B: If the *Delay time* value is set to 00:00:00, the next step is activated with no delay from the previous step.

The user can follow this procedure to add additional steps to the sequence.

Example

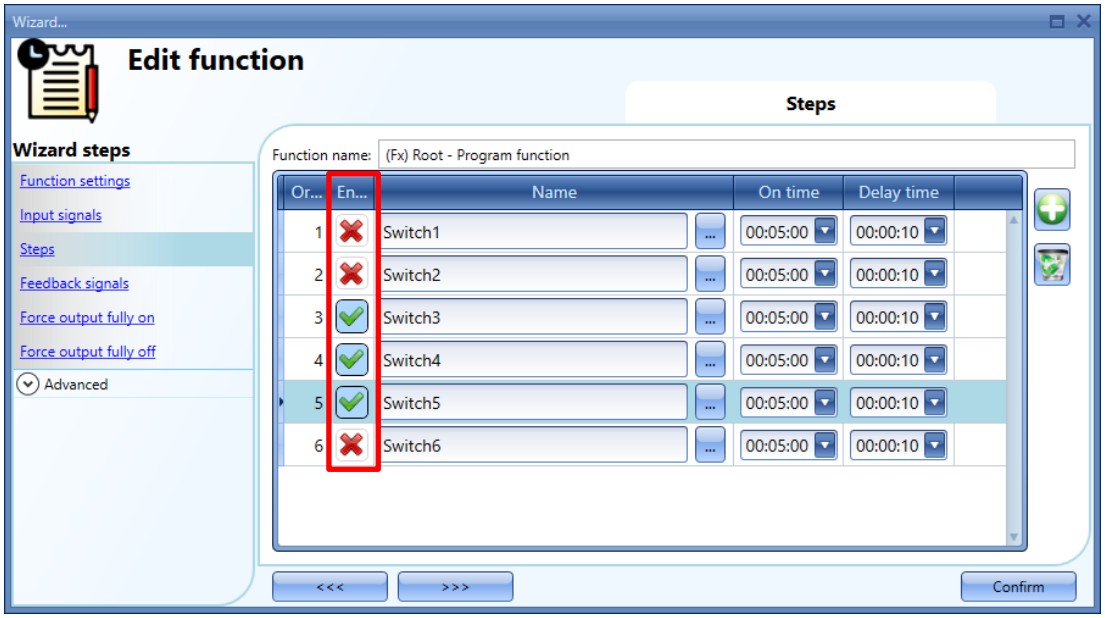
In the graph shown below, the first step (*Switch1*) is activated when the sequence is started: all the *Switch* functions related to *Switch1 step* will be switched ON for *On time* value (5 minutes) When this expires, the system waits for the *Delay time* timer (1 minute) then the next step *(Switch2)* is activated for *On time* value. When this expires the system waits for the Delay time value (30 seconds) and continues with the third step *(Switch3)*. The system waits 30 seconds and the sequence ends.





1.1.1.1 How to execute a specific set of steps

The user can enable/disable at any time which steps are to be executed when the Program function starts. The user can do this in the *Steps window*. They can enable the steps by clicking on the icon related to each step, in the *Enable* column as shown in the red rectangle below:

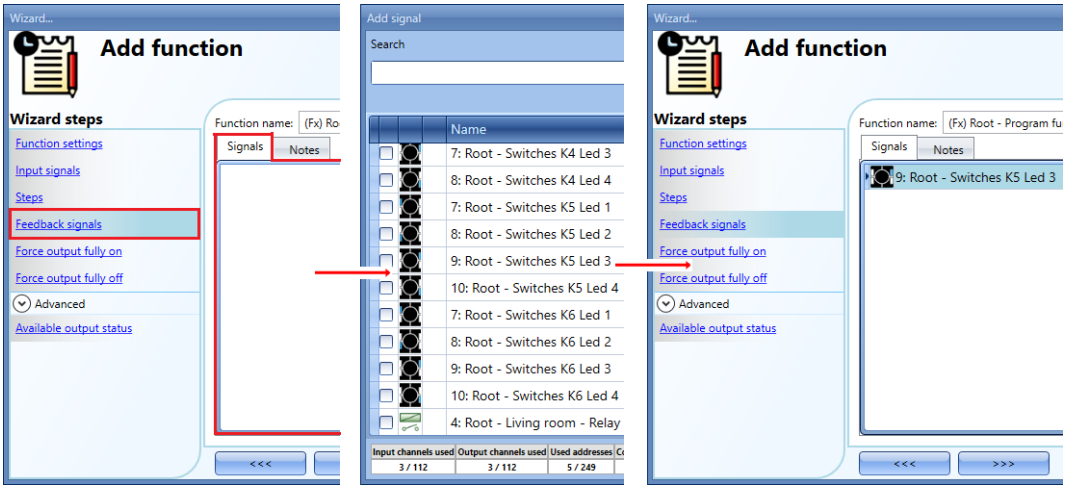


Click on the *Confirm* button, then in order to save the changes the next time the *Program* function is started, the user has to download the configuration to the UWP 3.0 controller by clicking the *Send to controller* button.

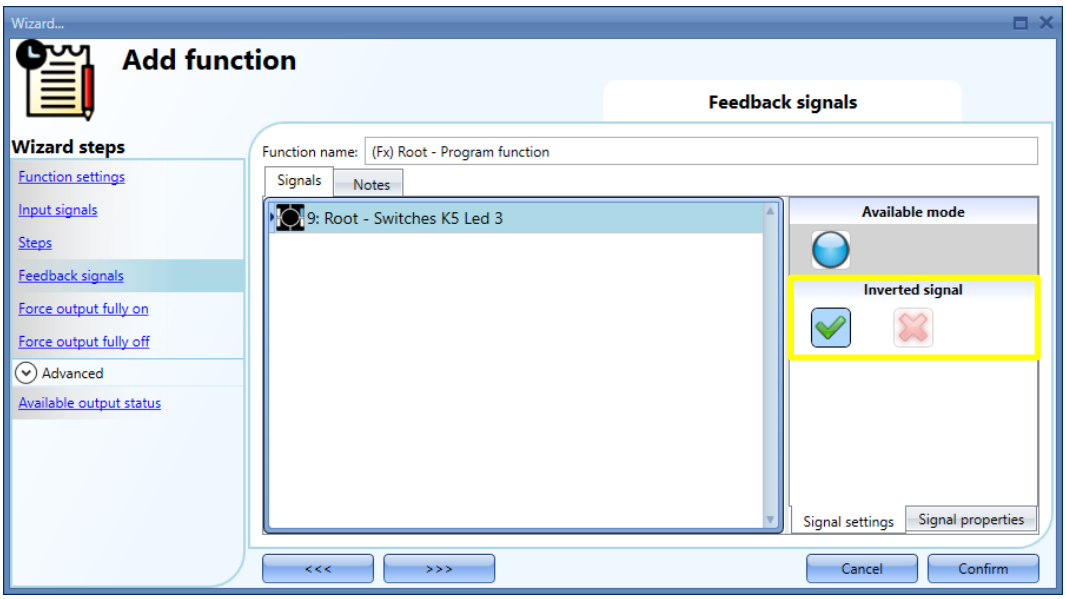
In the example above, the next time the *Program* function is started, only steps **Switch3, Switch4, Switch5** will be executed.

10.37.4 How to add the feedback signals

If the user wants to use a feedback signal to check the status of the *Program* function, he has to select the relevant section, then double click on the *Signals* window and select the feedback signal from the list of those available (see below).
The feedback signal can be a relay, a LED or a buzzer, and it behaves according to the status of the function: when the sequence is in progress the feedback signal is on; when the sequence is not running the feedback is off.

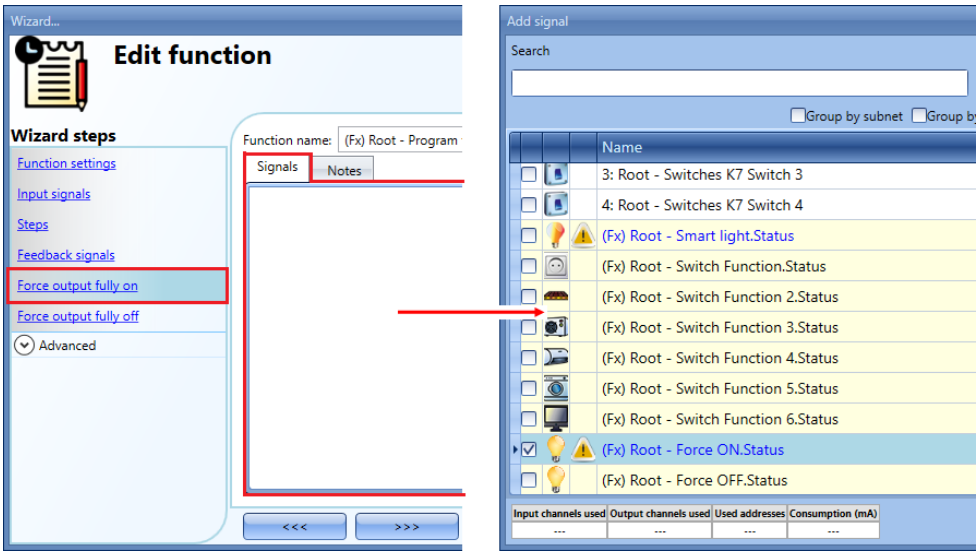


Once the feedback signal is entered, the user can also select the inverted logic.

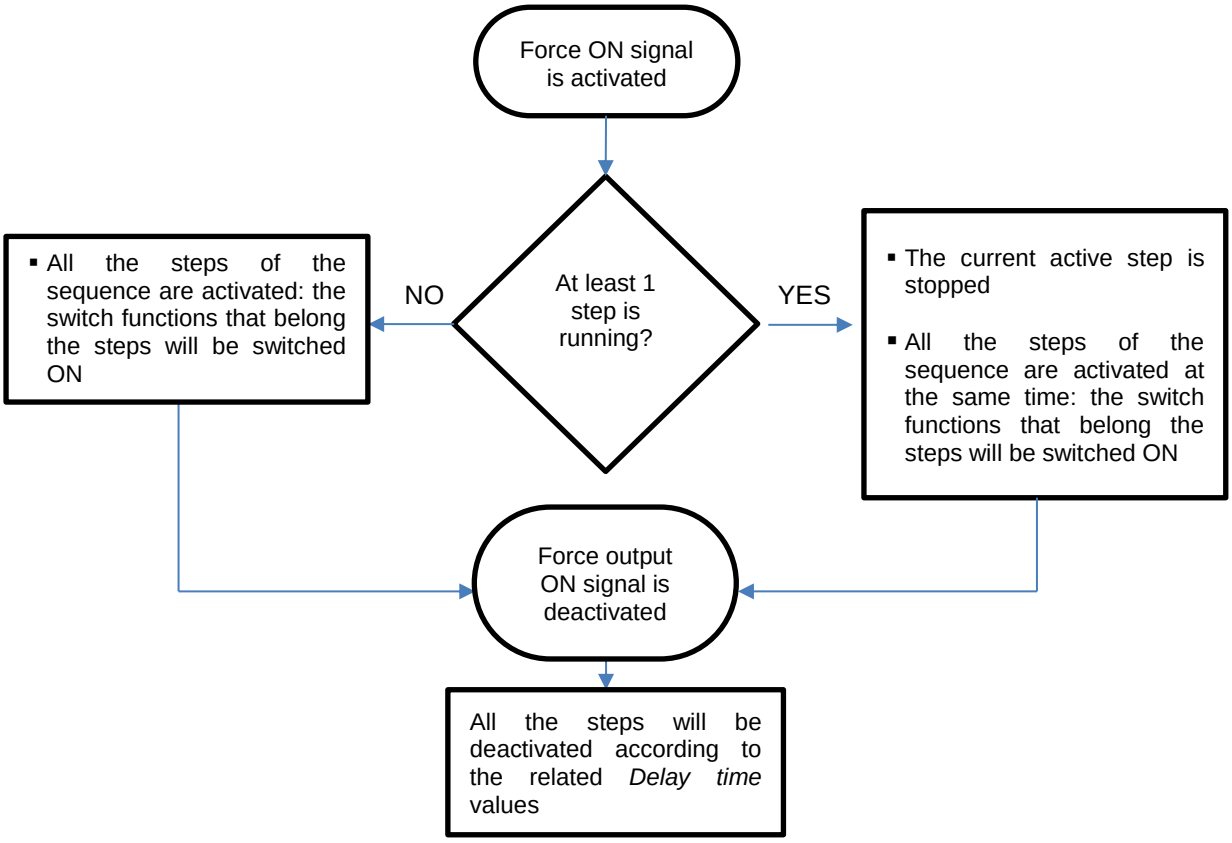


10.37.5 How to force the sequence active

If the user wants to force the sequence active (all the steps set to ON), select *Force output fully on* of the function: to enable it, double click on the *Signals* window and select the level signal to use.



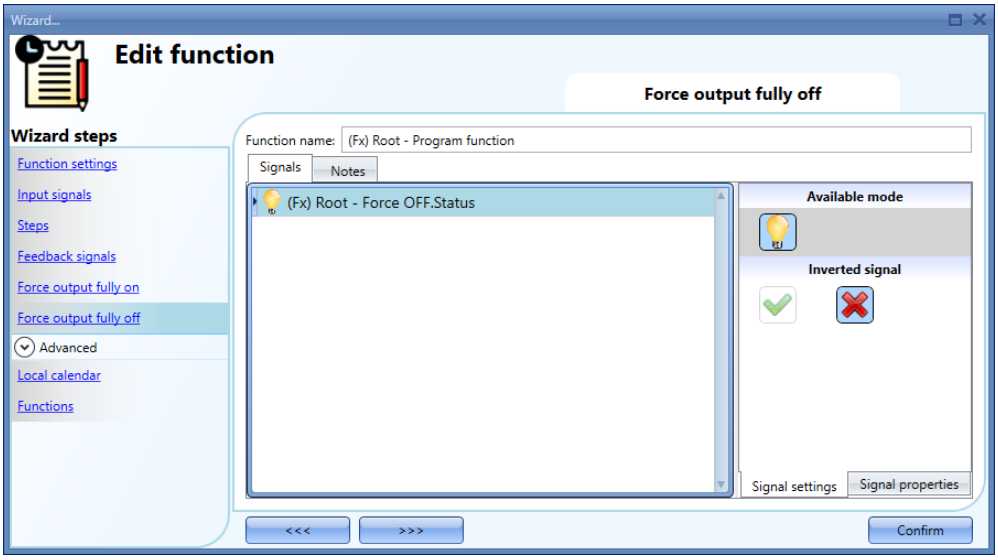
Each signal used in the *Force output fully on* window works in level mode. The behaviour when the signal is activated is as follows:



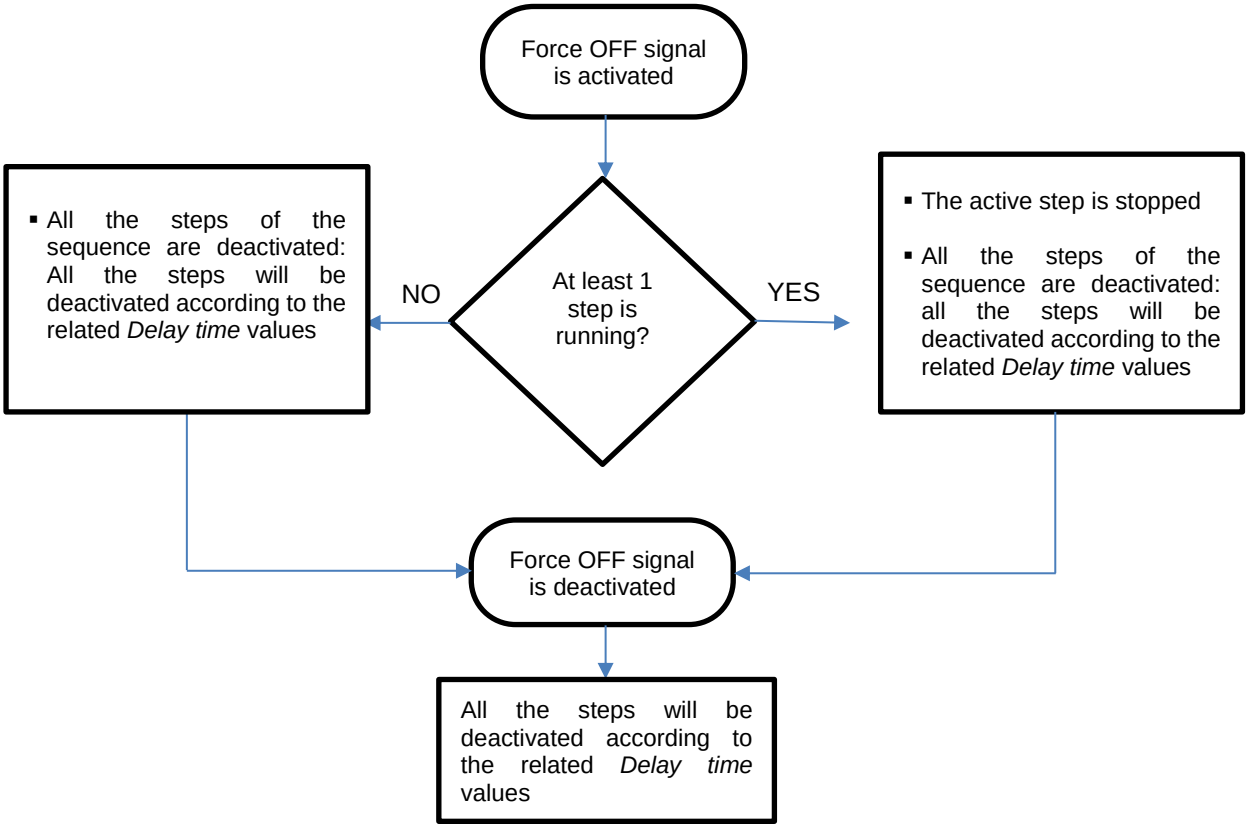
NOTE: When the *Force output fully on* signal is activated and at the same time the *Force output fully off* signal is activated, the Signal Off has priority and the Force output fully off command is executed.

10.37.6 How to force the sequence inactive

If the user wants to force the sequence inactive regardless of all the other signals used in the function, a level signal has to be used in the *Force output fully off* window: to enable it, double click on the *Signals* window and select the level signal to use (see below):

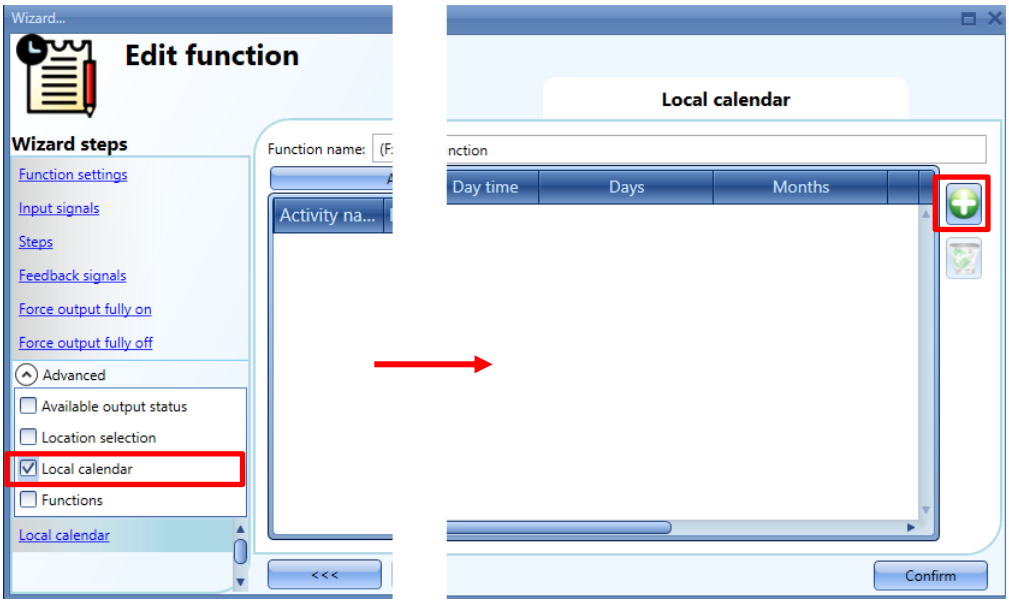


Each signal used in the *Force output fully off* window works in level mode. While the *signal* is active, the sequence is maintained inactive: all the Switch functions that belong to the steps will be maintained OFF and cannot be activated until the stop signal disappears.

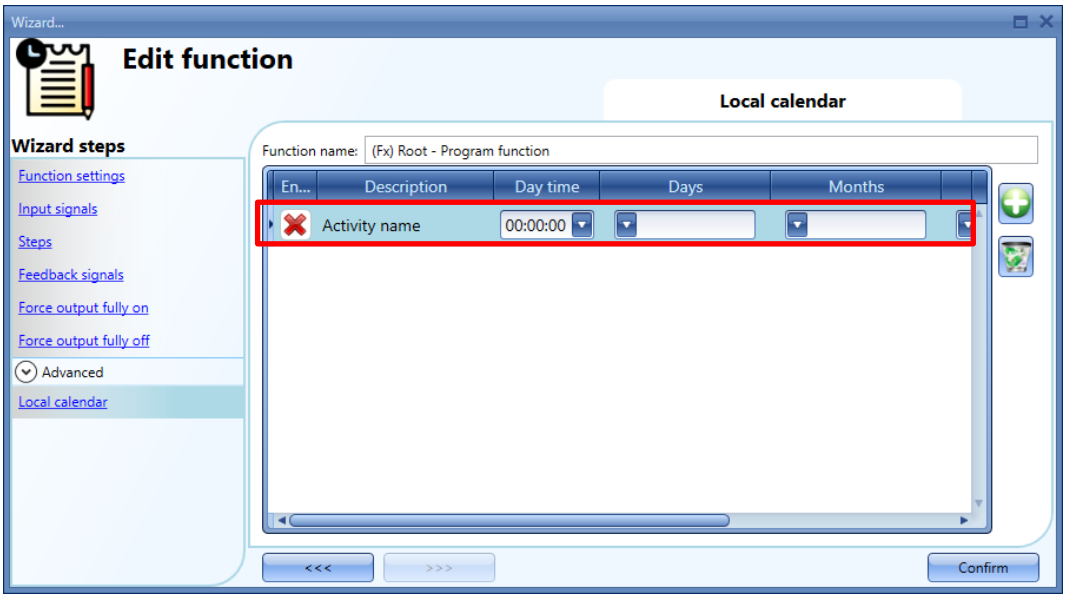


10.37.7 How to start/stop the sequence using a calendar

If the user wants to start/stop the sequence using a Local calendar, the relevant menu has to be enabled in the *Local calendar* field (see below).



To add a new *Calendar activity*, the user has to click the **+** button and the following window will appear:



Enable: The user can enable/disable the calendar activity.

Activity name: In this field the user defines the name of the event that will appear on the calendar. This is a mandatory field.

Day time*:* The user can select the date (*Hours, Minutes, Seconds)* when the calendar activity has to operate.

Days: The user can select the days of the month when the calendar has to operate.

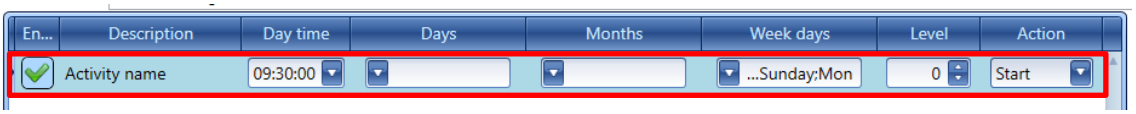
Months: The user can select the months of the year when the calendar has to operate.

Week days: The user can select the days of the week when the calendar has to operate.

Action: in this field the user can select the action which has to be performed. The actions that the user can choose are:

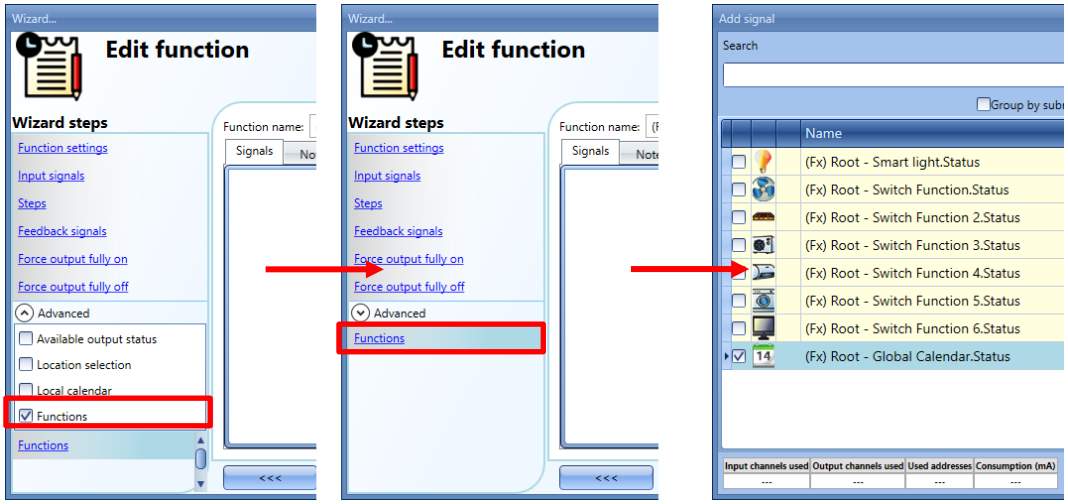
- No action (-1)
- Stop (0): the sequence is stopped
- Start (1): the sequence is started
- Pause (2): the sequence is paused

In the example below the calendar is set to start a sequence at 9: 30 every day.



10.37.8 How to use an external function to start/stop a sequence

If the user wants to use an external function to start/stop a sequence, the *Function* field has to be used: to enable this select *Functions* in the *Advanced* section, then double click on the signals window and select the relevant function from the list (see below).



All the added functions will trigger the sequence according to their status: for example, if a *Global calendar* is added in this field it works in the same way as the internal calendar.

10.38 How to manage the Program function from the WEB-App

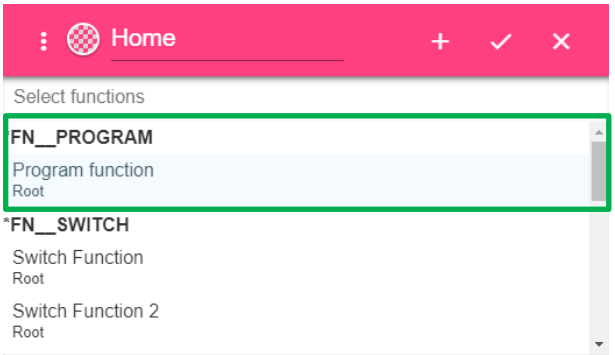
The Program function is available via WEB-App as a function widget: the user has to first configure the Program function from the **UWP 3.0 Tool.** From the Web-App the user can:

- Send actions (Start – Pause -Stop the sequence)
- Change the *On time* value of one or more steps
- Change the *On time* of all steps by a percentage value
- Enable/disable the steps that have to be executed

10.38.1 How to add a Program function widget

To add the *Program* function widget to a WEB-App page, click on to access the *editing mode.*

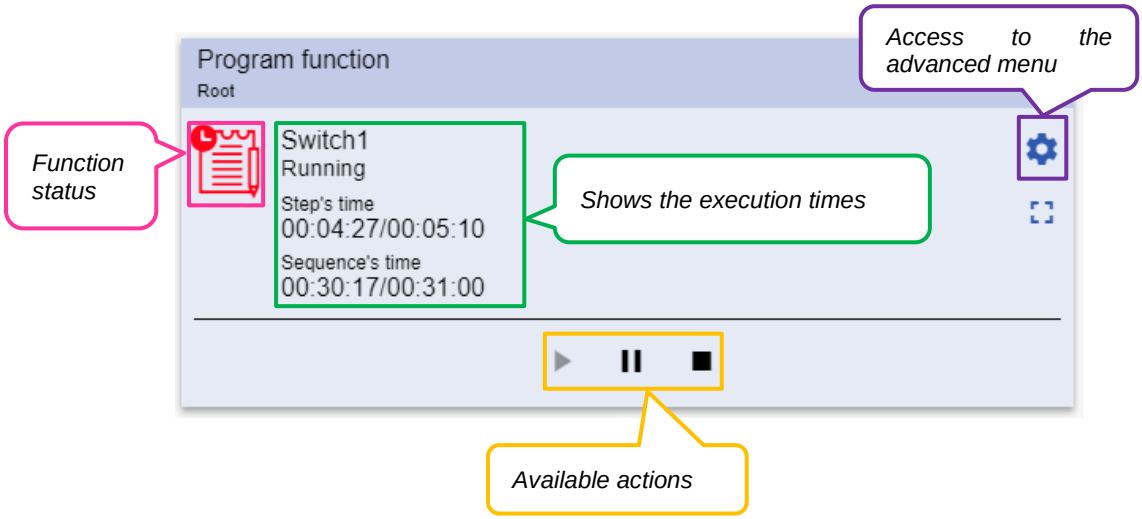
From the **editing toolbar**, click on + and select the **Functions** field, then select the Program function from the list, as shown below:



From the **editing toolbar,** click on ✓ to add it to the page, as shown below:

2.1.1.1 Main widget overview

The picture below shows an example of a *Program* function widget in running mode



The actions available in the *Program* function widget are as follows:

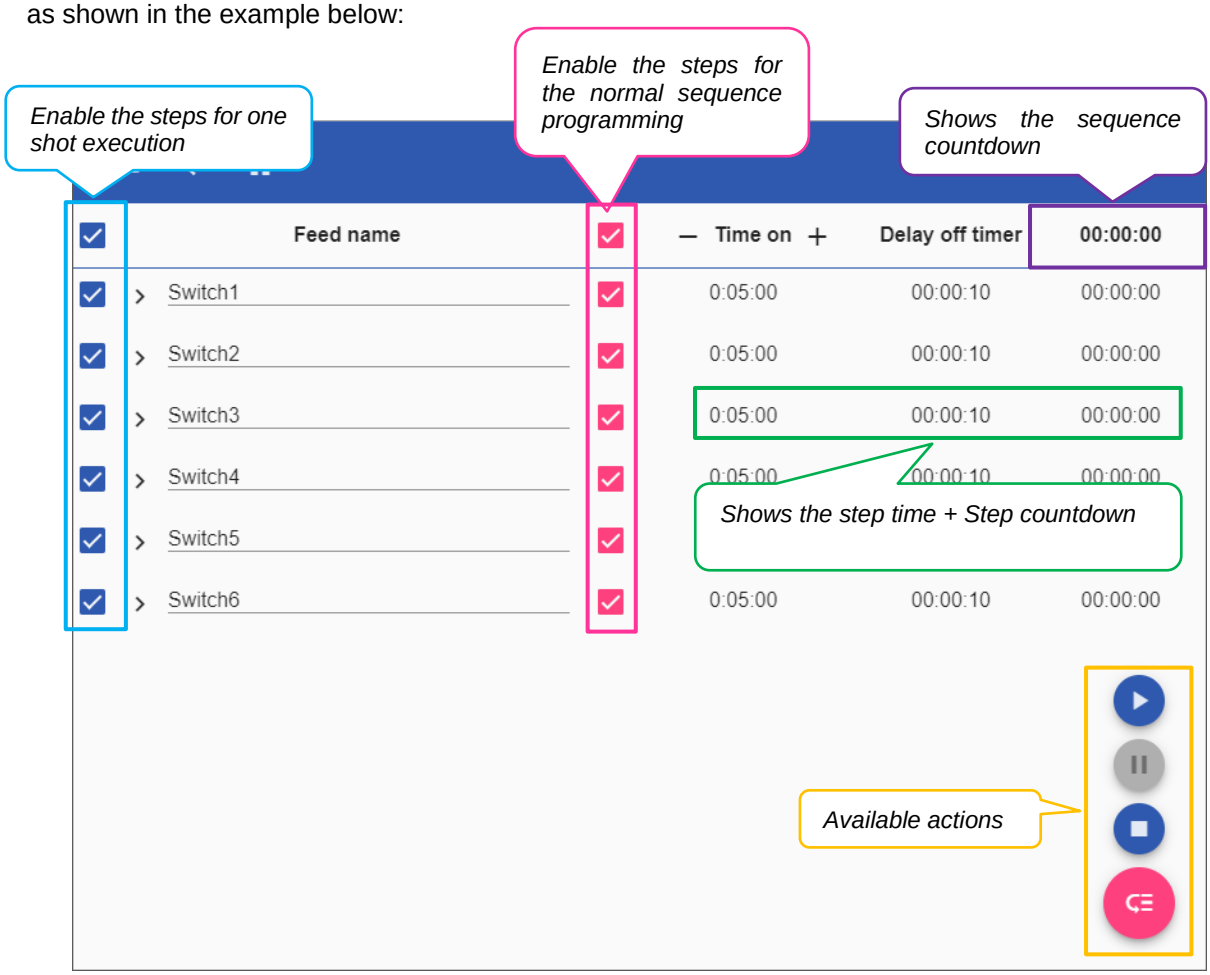
Item	Description
	By clicking the button, the drawer is expanded. The actions available are as follows: ▪ **Start**: The *Program* function is started ▪ **Pause:** The *Program* function is paused ▪ **Stop:** The *Program* function is stopped
	Shows the Program function status Click the icon to toggle the function **Start/Stop**

The information available in the *Program* function widget is as follows:

Item	Description
Switch1 Running	Shows the name of the current active step
Step's time 00:04:27/00:05:10	Shows the countdown of the current active s ***[Step time] / [Step countdown]***
Sequence's time 00:30:17/00:31:00	Shows the total execution time of the entire seque ***[sequence time] / [Sequence countdown]***

10.38.2 How to manage the Program function from the advanced menu

The user can access the advanced menu by clicking on the button. The following window will appear as shown in the example below:

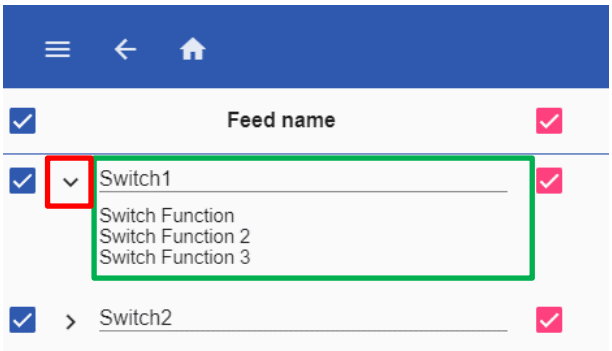


The table below shows the procedures the user can follow via the advanced menu:

Type	Procedure
Sequence programming	▪ How to set a steps sequence once ▪ How to change the sequence programming
Set parameters	▪ How to change the *On time* (Individually) ▪ How to change the *On time* (Multi-change)

3.1.1.1 How to check which Switch functions belong to a step

The user can check which *Switch* functions belong to a step by clicking the ***V*** button:
a list is shown, as displayed in the example below:



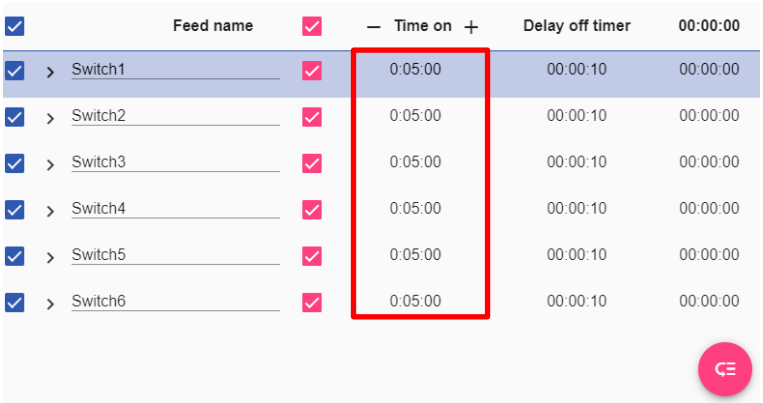
The relation between the step and the *Switch* functions can not be changed by the WEB-App widget

4.1.1.1 How to change the On time value

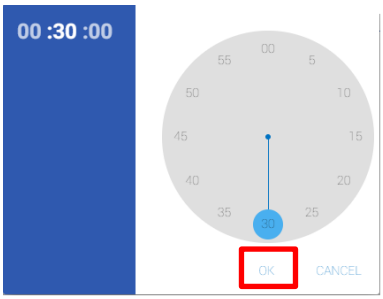
The user can change the *On time* value for each step in the sequence: there are two different ways to change the *On time* value:

10.38.2.1.1 Individually

In the *On time* column (red rectangle below) click on the *time* field:



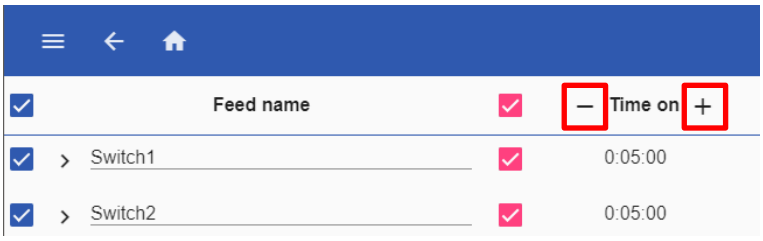
In the popup that appears, the user can change the *Hours, Minutes, Seconds* values.



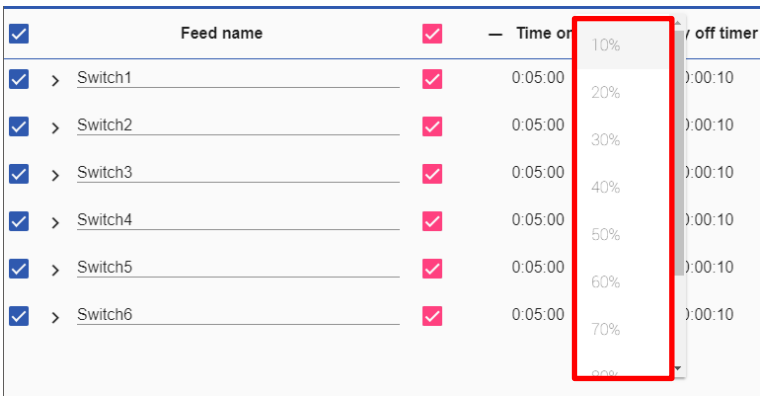
Click OK to save the change

10.38.2.1.2 Multi-change

To change the *On time* value of all the steps at the same time, the user has to click on the + or – button in the On time column (red rectangle below):



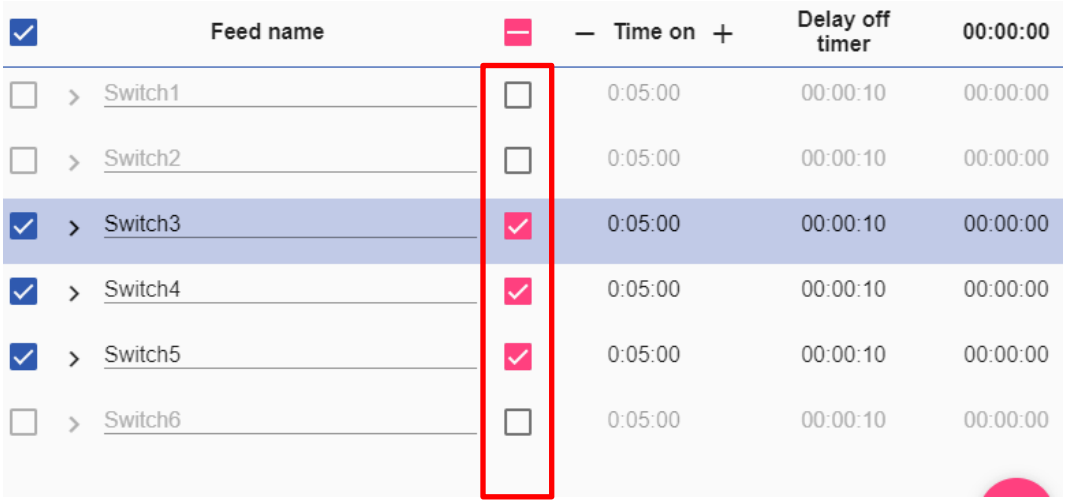
The user can select the percentage value that will be applied to all the steps of the sequence



5.1.1.1 How to change the sequence programming

In the *advanced menu*, the user can select which steps have to be executed each time the sequence is started. To do this, they can follow the procedure shown below:

1) The check-box next to each step has to be selected and unchecked to remove a step from the sequence execution, as shown in the example below:



Note: To enable/disable all the steps at the same time, the user can click the in the column header.

2) Click and select to save the changes; otherwise by clicking on the button, the last valid set of steps will be restored.

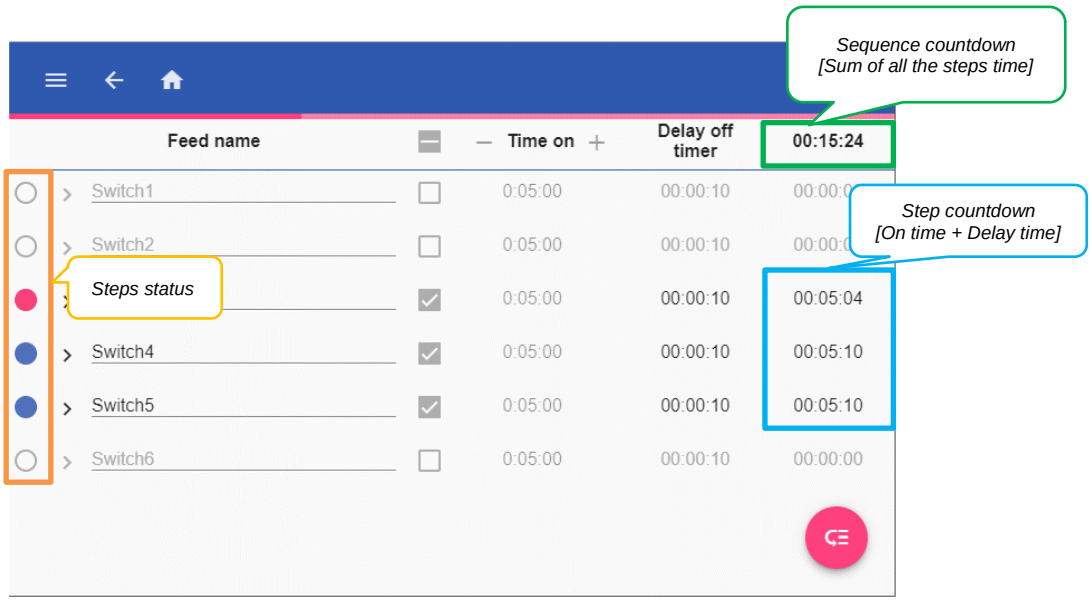
As shown below, the steps *Switch2, Switch3 and Switch4* have been selected. Each time the sequence is started, only the flagged steps will be executed.

3) Click and select the button to play the sequence.

The user can check the status of each step of the sequence by the dot indicator next to each step, as shown in the table below:

Indicator	Behaviour
	Current active step
	Enabled steps
	Disabled steps

The last column shows the countdown of the total sequence (green rectangle) and for each step the countdown is shown (blue rectangle).



The sequence can be paused or stopped at any moment by the user, by clicking the relevant buttons.

When the sequence is running the user is not able to change the sequence set: in order to change it, the sequence must first be stopped.

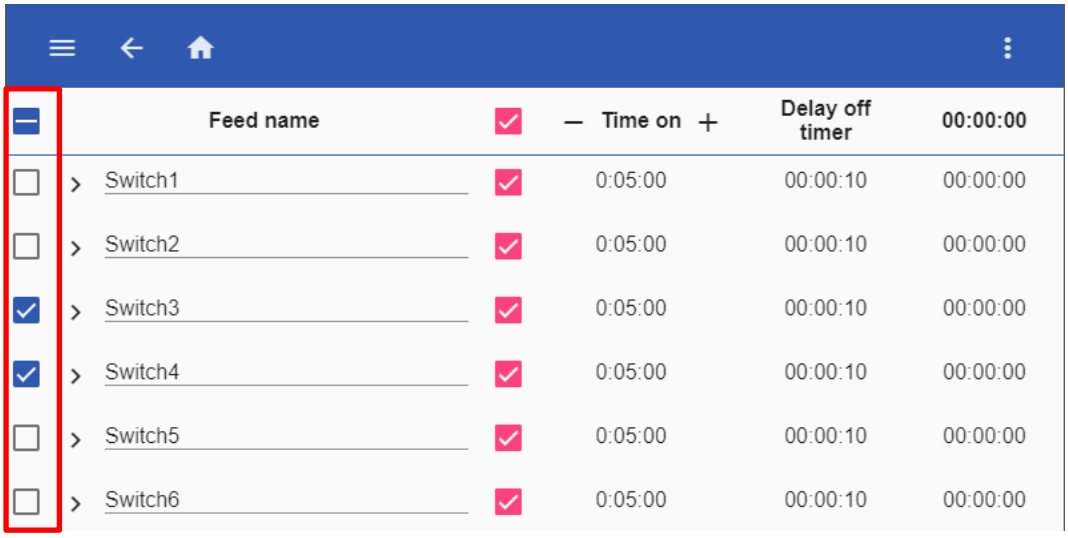
10.38.3 How to execute a set of steps one time only

In the *advanced menu*, the user can select which steps have to be executed one time only: this configuration overwrites the behaviour of the sequence, allowing the execution of a specific set of steps. When the sequence ends the previous programming will be restored. This procedure can be followed only if the sequence is not running.

To do this, the user can follow the procedure shown below:

4) The check-box next to each step has to be selected; to disable a step, the user has to deselect the related check-box, as shown in the example below:

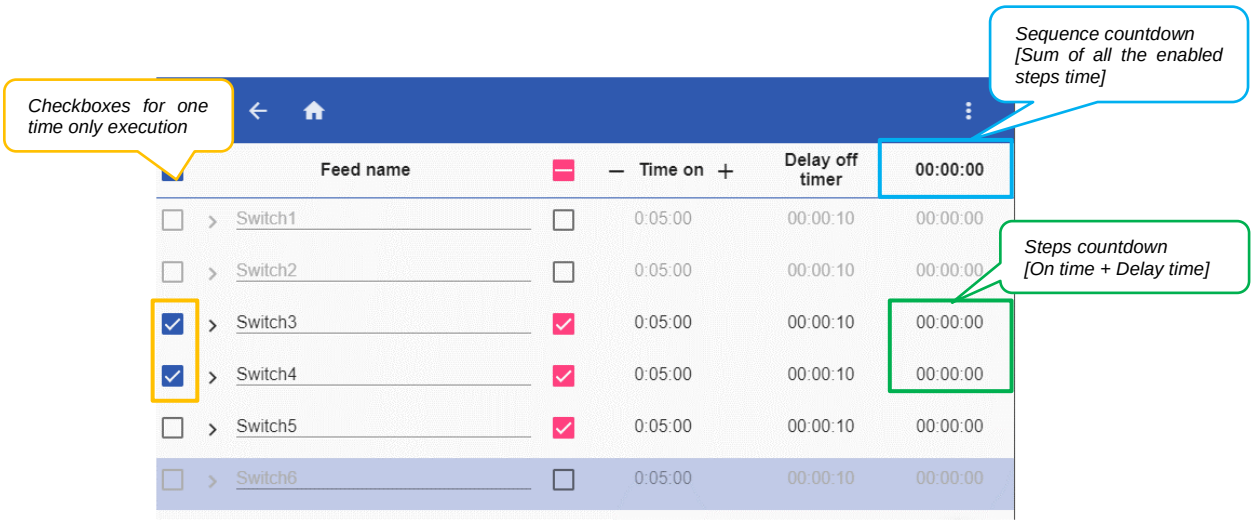
N.B: The steps that are not enabled will not be selectable and playable



Note: To enable/disable all the steps at the same time, the user can click on the in the column header.

5) Click and select the button to play the sequence.

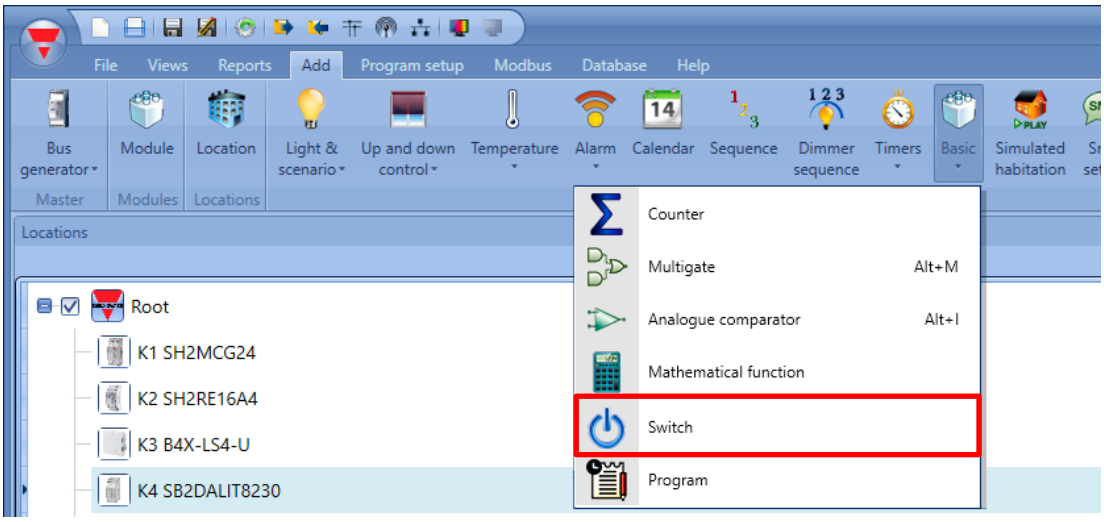
The sequence can be paused or stopped at any moment by the user, by clicking the relevant buttons.


Checkboxes for one time only execution
Sequence countdown [Sum of all the enabled steps time]
Steps countdown [On time + Delay time]
Feed name
Time on
Delay off timer
00:00:00
Switch1
Switch2
Switch3
Switch4
Switch5
Switch6
0:05:00
00:00:10
00:00:00

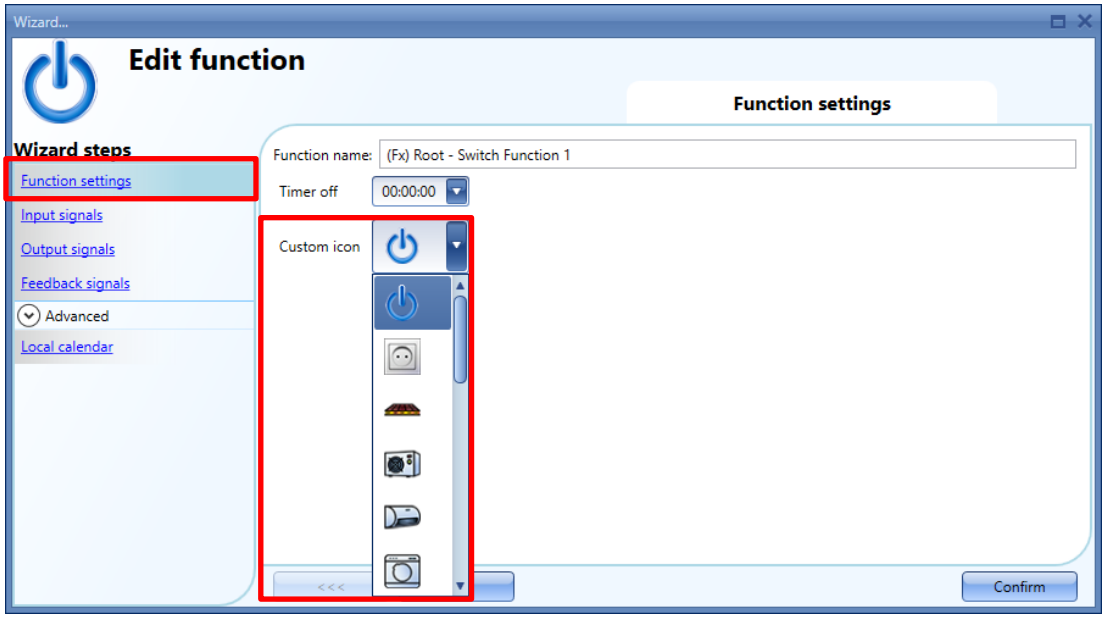
10.39 How to set a Switch function

This function allows the user to manage one or more *digital outputs*, such as relays, at the same time. The user can either automate the ON/OFF switching of the function outputs with the *Program* function (the *Switch* functions connected to the *Program* function will be turned ON and OFF according to the programmed sequential order), or with the Local calendar function (calendar to turn the outputs ON/OFF at pre-defined time intervals).

To set a *Switch* function the user has to select the *Switch* function from the *Add* menu (see figure below). The new function will be added into the selected location.



In the *Function settings* tab of the *Program* function, the user can choose a custom icon for the *Switch* function, selecting from the *Custom icon* list, as shown below:



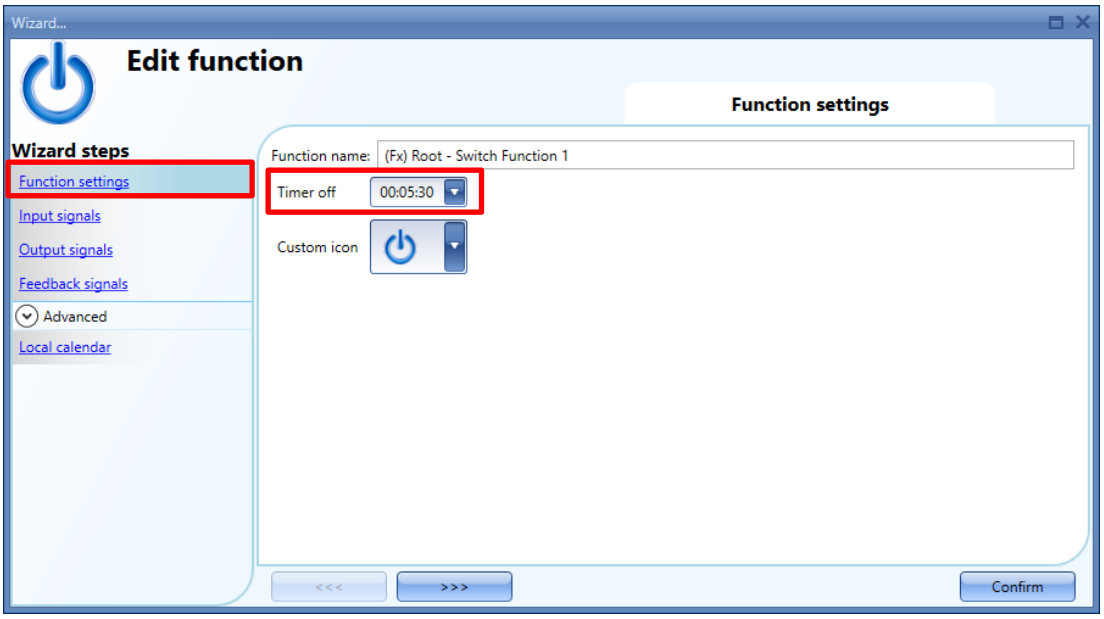
The selected icon will be shown both in the UWP 3.0 Tool and in the WEB-App interface.

10.39.1 How to turn the function OFF after a predefined time period

If the user needs to be sure that the Switch function is turned OFF after a predefined time period, the *Timer off* should be used in the *Function settings* window.

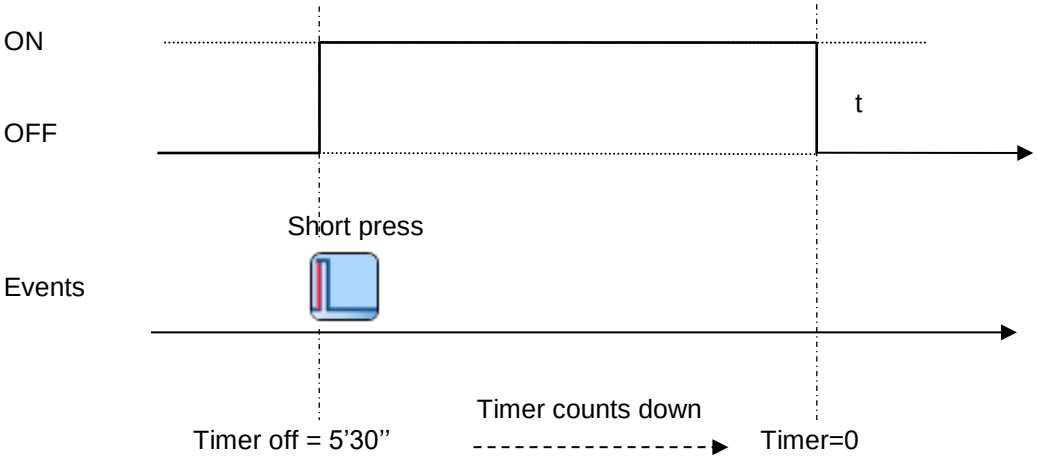
The *Timer off* starts every time the *Switch* function is turned ON, causing the output to go OFF automatically when the set time has expired. The *Timer off* can be set up to a maximum value of 23:59:59 (*HH:MM:SS*) while when it is set to 0 it does nothing: The shortest time that can be set is 1 second.

The *Timer off* can be set and changed by means of the WEB-App.



To understand better the behaviour of the *Timer off* timer, if the *Switch* function is set according to the previous picture, the graph below is valid:

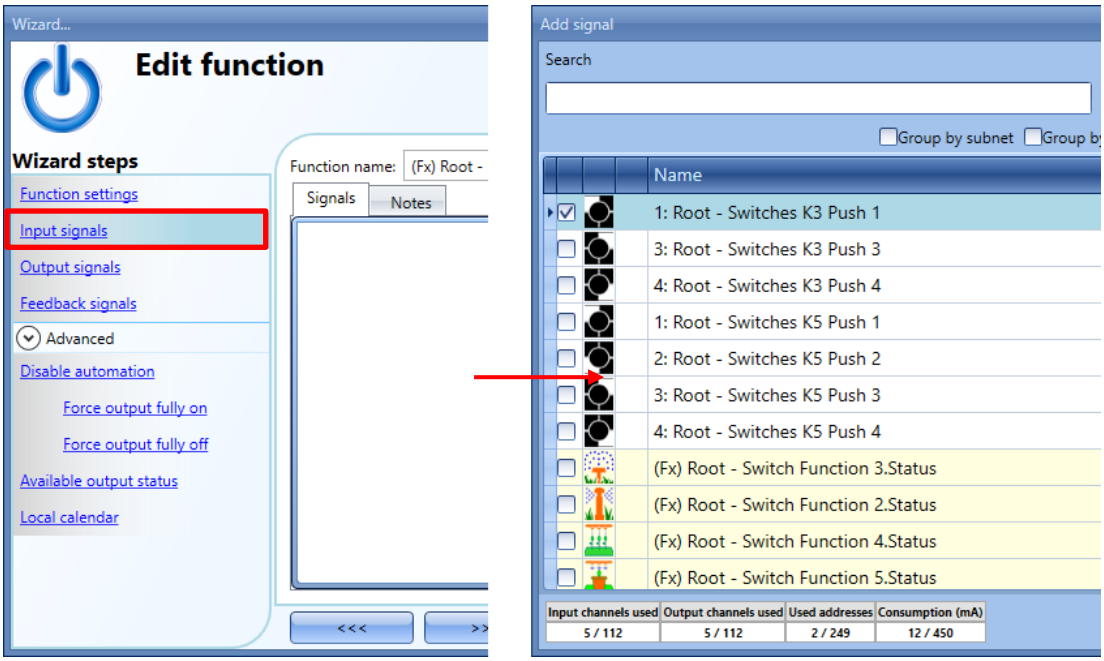
Switch function status



10.39.2 How to turn the output ON/OFF with input commands

This function manages the ON/OFF switching of one or more outputs by means of one or more input commands. The input command might be a real signal, a function or a remote command (UWP 3.0 Tool Live Signals, WEB-App, Modbus TCP/IP or BACnet).

The user has to select the *Input signals* tab of the *Program* function, then double click on the *Signals* window and select the input signal from the list of those available (see below):

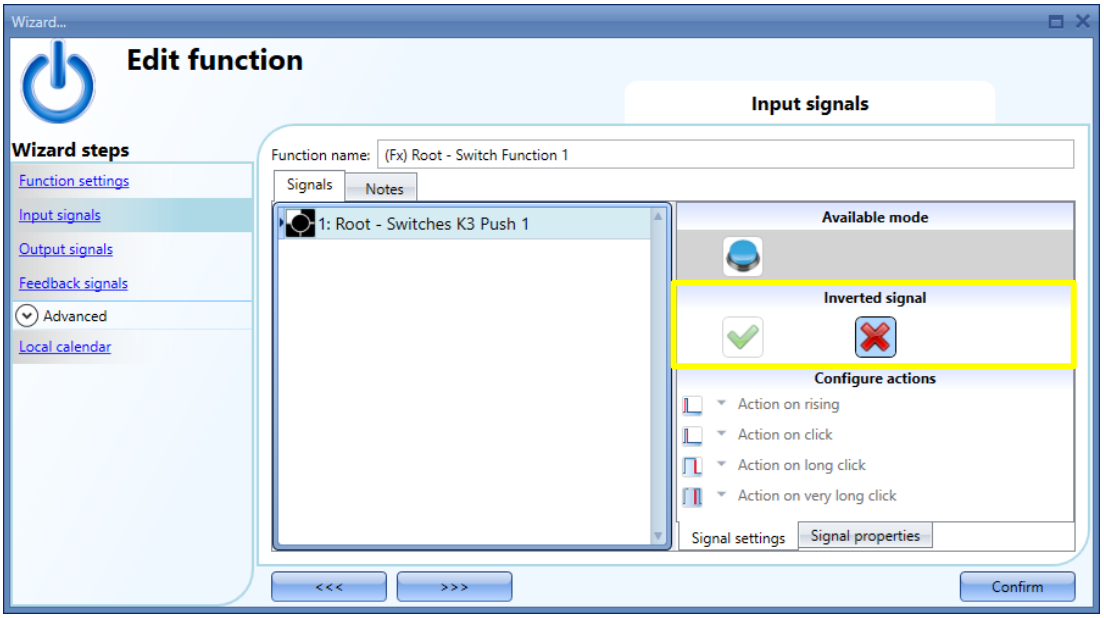


If more than one signal is selected in this window (up to 100 signals can be used) the function will be activated/deactivated if at least one signal is active (the system does the logical OR).

The different colours have the following meanings:

- Black signal → not used in any function
- Blue signal → the signal is already used in another function
- Blue signal + warning signal (yellow triangle) → the signal is already used in this function

In the *Signal settings* window, the user can also enable the reversing of the signal. See the yellow rectangle below:



The *long* and *very long* activations can be set directly in the *Signal properties* tab window, without the need to re-open the Wizard of the module, as shown in the red rectangle below:

The available input signals are classified as:

- Push button
- Switch
- Function
- Remote

The following table shows the actions available for push buttons:

Event type	ON	OFF	TOGGLE
Action on rising	As soon as the push button is pressed, the *Switch* function is turned ON	As soon as the push button is pressed, the *Switch* function is turned OFF	As soon as the push button is pressed, the *Switch* function is toggled
Action on click	As soon as the push button is clicked, the *Switch* function is turned ON	As soon as the push button is clicked, the *Switch* function is turned OFF	As soon as the push button is clicked, the *Switch* function is toggled
Action on long click	After a long press, the *Switch* function is turned ON	After a long press, the *Switch* function is turned OFF	After a long press, the *Switch* function is toggled
Action on very long click	After a very long press, the *Switch* function is turned ON	After a very long press, the *Switch* function is turned OFF	After a very long press, the *Switch* function is toggled

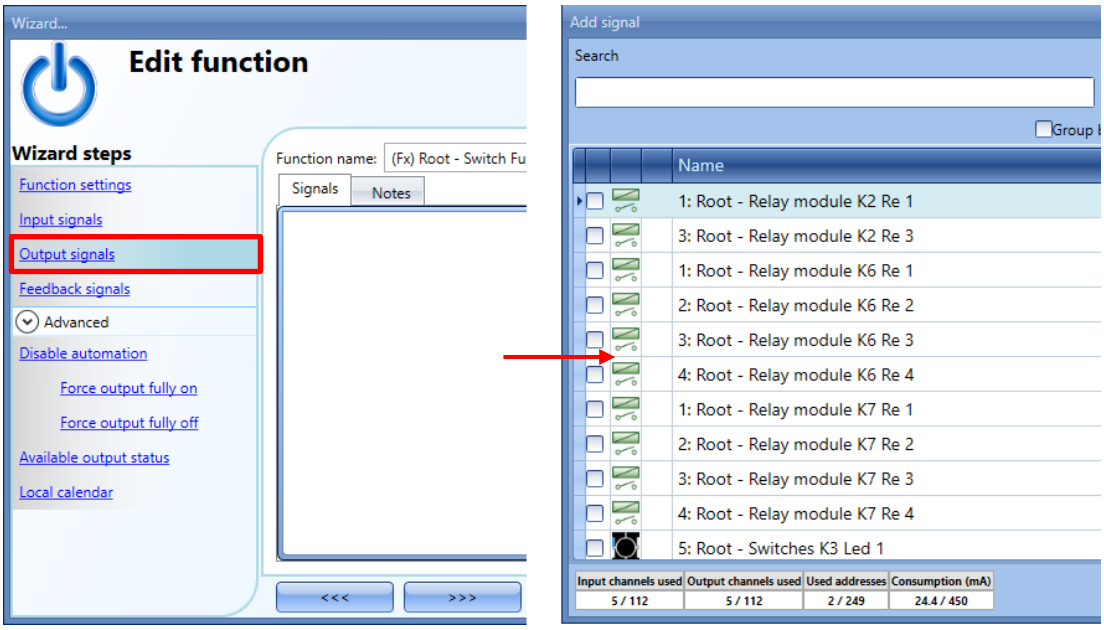
The following table shows the actions available for switches and functions:

Event type	ON	OFF	TOGGLE
Action on rising	As soon as the switch is closed/function goes ON, the *Switch* function is switched ON	As soon as the switch is closed/function goes ON, the selected steps are started	As soon as the switch is closed/function goes ON, the Program function is paused
Action on falling	As soon as the switch is opened/function goes OFF, the *Switch* function is switched ON	As soon as the switch is opened/function goes OFF, the *Switch* function is switched OFF	As soon as the switch is opened/function goes OFF, the *Switch* function is toggled

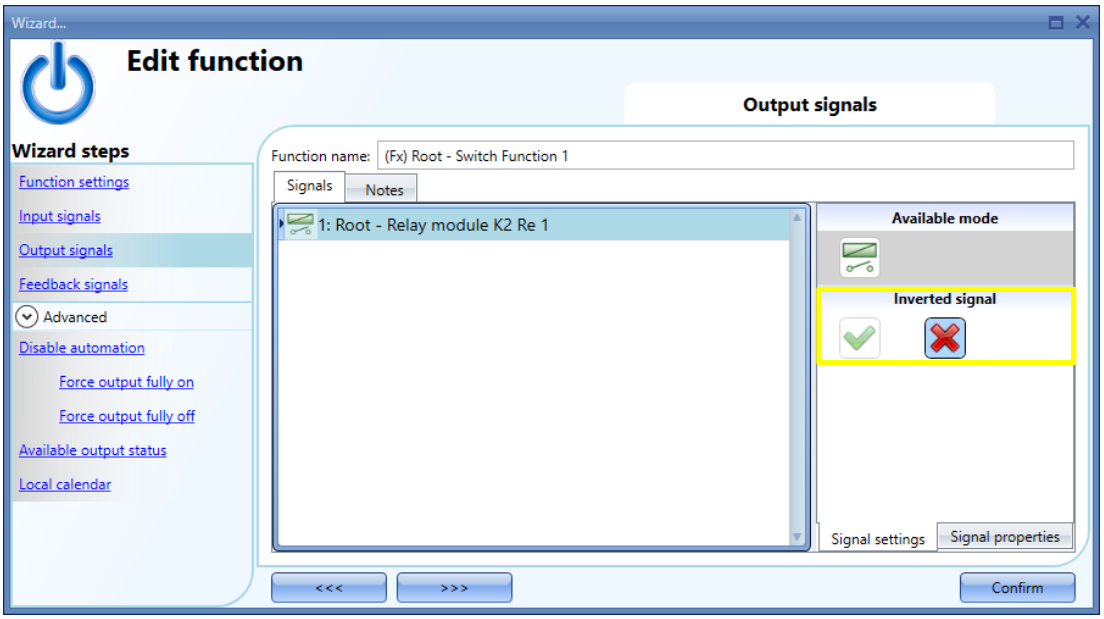
As soon as the *Switch* function is turned ON and the set *Timer off* value is higher than 0, the *Time off* timer will start counting down. When the timer expires, the *Switch* function goes OFF.

10.39.3 How to select the output of the Switch function

To select the output signal that is connected to the *Switch* function, the user has to click on *Output signals* and then double click on the *Signals* window:

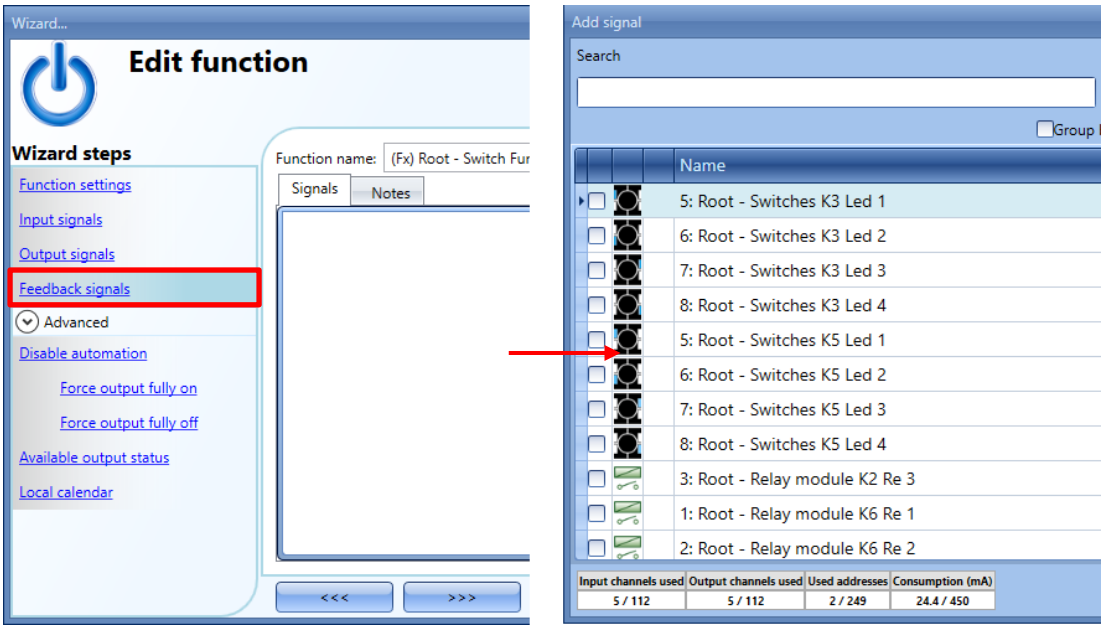


The signals available in this window are relays and LEDs.
Up to 50 signals can be chosen and they are managed in parallel.
The logic of each output signal can be set as normal or inverted (see yellow rectangle).



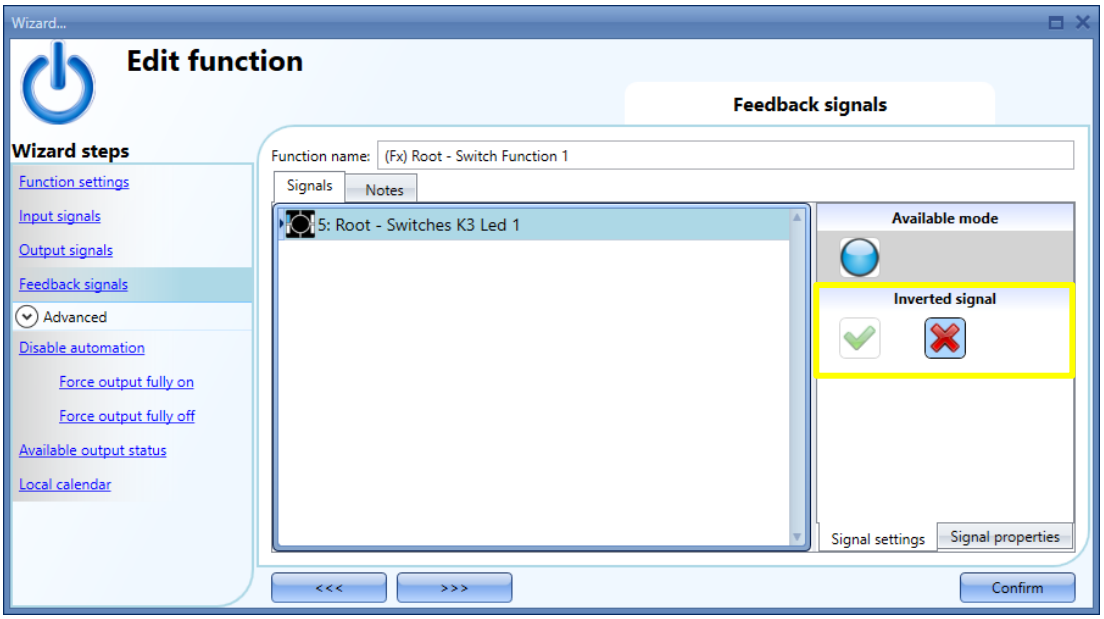
10.39.4 How to set a signal as feedback of the status of the function

To select the feedback signal that indicates the status of the *Switch* function, click on the *Feedback signals* tab and then double click on the *Signals* window.



The signals available in this window are LEDs and relays.
Up to 50 signals can be chosen and they are managed in parallel.

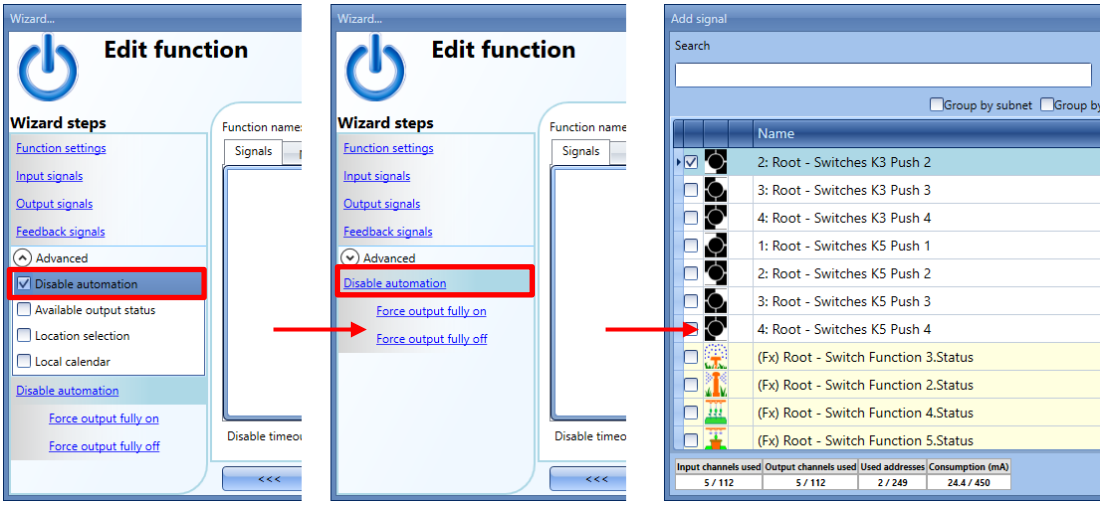
The logic of each output signal can be set as normal or inverted (see yellow rectangle).



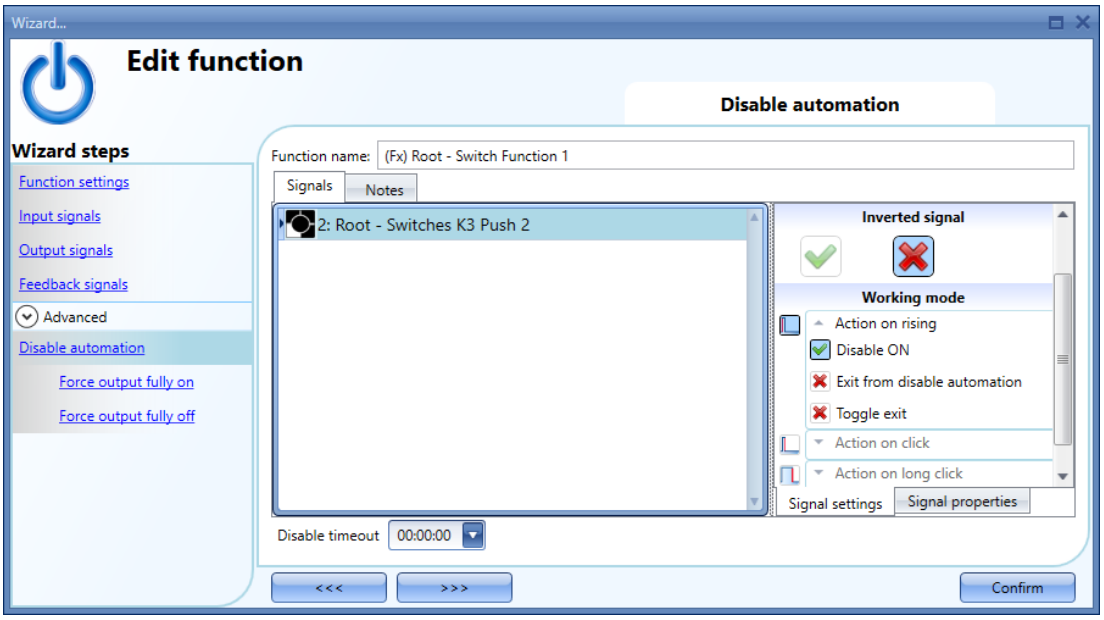
10.39.4.1 Disabling automation using signals

The user might need to disable the automation related to the *Switch* function: for example, if the *Timer off* is not to be started when the function is activated: As soon as the *Switch* function is turned ON, the *Timer off* timer will remain disabled.

To disable the automation, the user has to select the *Disable automation* field in the *Advanced* section, then in the *Signals* window the signals have to be added, as shown below:



The required signal has to be added by double clicking in the *Signals* window.



If a push button type is selected, the following table shows the actions available:

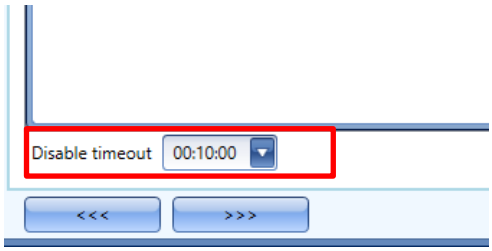
Event type	Disable ON	Exit from disable automation	Toggle exit (Only if *Disable ON* is selected)
Action on rising	As soon as the push button is pressed, the *Disable automation* is activated.	As soon as the push button is pressed, the *Disable automation* is deactivated.	As soon as the push button is pressed, the *Disable automation* is activated/deactivated in toggle mode.
Action on click	As soon as the push button is clicked, the *Disable automation* is activated.	As soon as the push button is clicked, the *Disable automation* is deactivated.	As soon as the push button is clicked, the *Disable automation* is activated/deactivated in toggle mode.
Action on long click	After a long press, the *Disable automation* is activated.	After a long press, the *Disable automation* is deactivated.	After a long press, the *Disable automation* is activated/deactivate d in toggle mode.
Action on very long click	After a very long press, the *Disable automation* is activated.	After a very long press, the *Disable automation* is deactivated.	After a very long press, the *Disable automation* is activated/deactivated in toggle mode.

The following table shows the actions available for switches and functions:

Event type	Disable ON	Exit from disable automation	Toggle exit (Only if *Disable ON* is selected)
Action on rising	As soon as the switch is closed/function goes ON, the *Disable automation* is enabled	As soon as the switch is closed/function goes ON the *Disable automation* is disabled	As soon as the switch is closed/function goes ON, the *Disable automation* is disabled/enabled in toggle mode
Action on falling	As soon as the switch is opened/function goes OFF, the *Disable automation* is enabled	As soon as the switch is opened/function goes OFF, the *Disable automation* is disabled	As soon as the switch is opened/function goes OFF, the *Disable automation* is disabled/enabled in toggle mode

The user might need to set a time after which the automation is enabled again even if the selected signal is still active: to do this the *Disable timeout* field should be set.

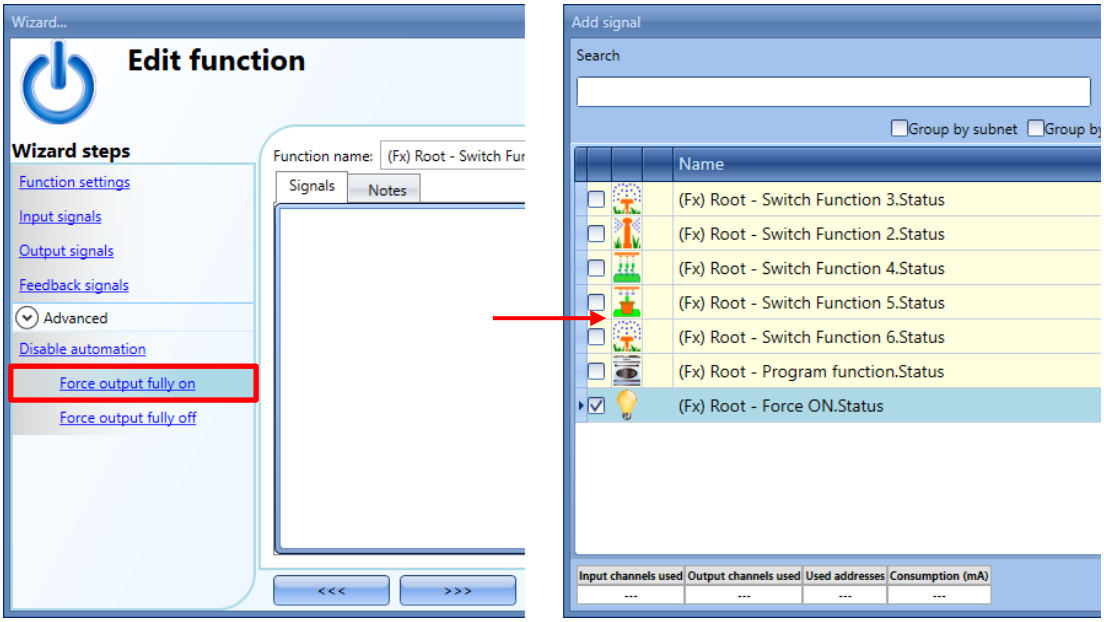
To access the stop automation, this functionality must be enabled in the *Advanced* section field.
If the user needs to be sure that 'disable' is automatically removed, the *Disable timeout* field should be set.



The timer starts each time the disable status becomes activated: the disable status is automatically deactivated as soon as the time has elapsed.
In the figure above a disable timeout of 10 minutes is set. The maximum *Disable timeout* value that can be set is 23:59:59 (*HH:MM:SS*).

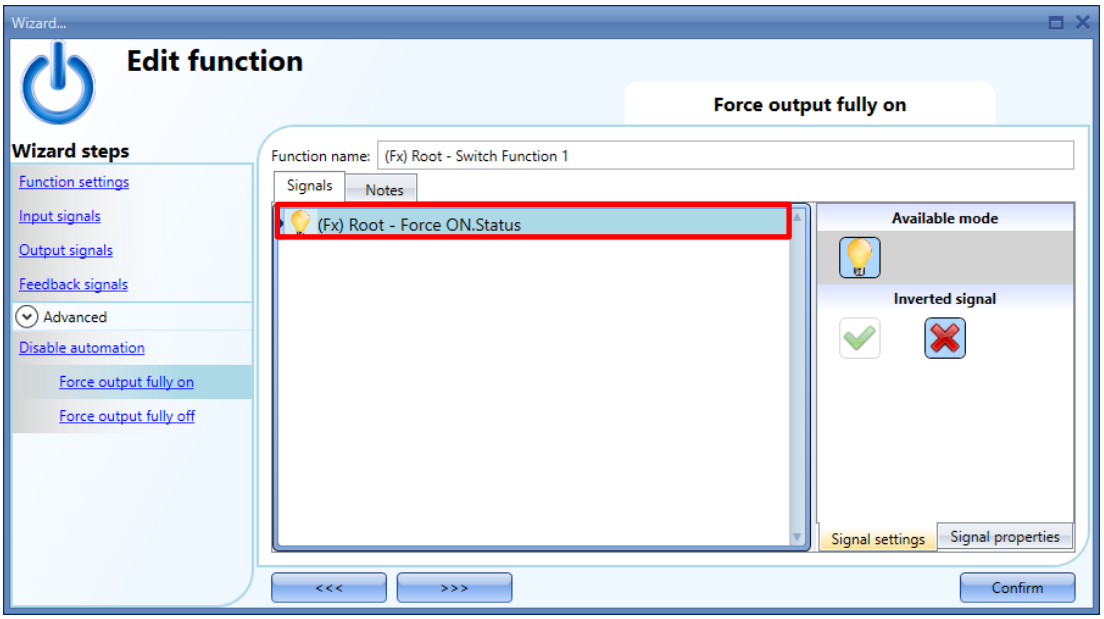
10.39.5 How to force the output ON

If the user wants to force the output on regardless of all other signals used in the function, the *Force output fully on* step has to be used: to enable this, select *Disable automation* in the *Advanced* section, select *Force output fully on*, then double click on the *Signals* window and select the right signal to use.



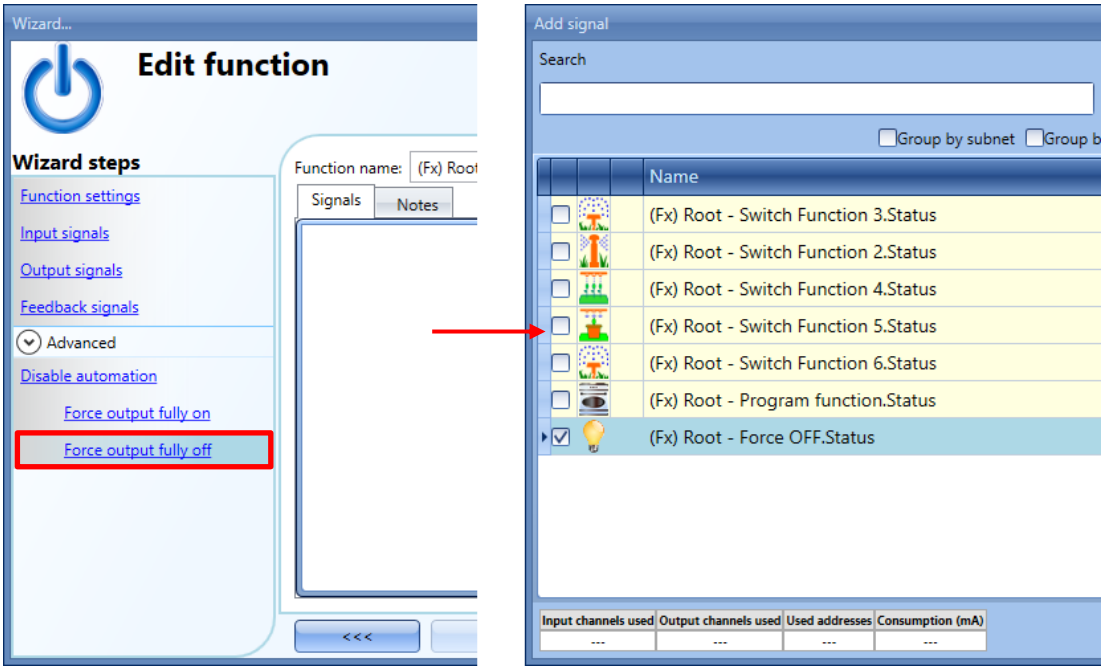
Each signal used in the *Force output fully on* window works as a level signal: Until the signal is active, the *Switch* function is forced on.

When the *Force output fully on* signal is activated and the *Force output fully off* signal is activated, the *Force output fully on* signal has the priority.

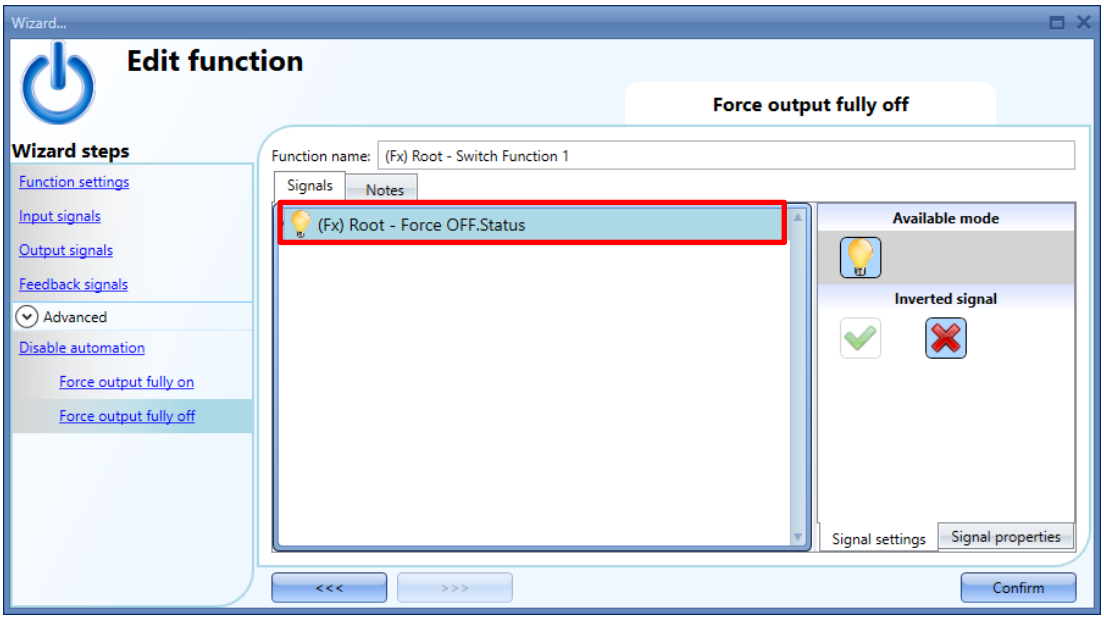


10.39.6 How to force the output OFF

If the user wants to force the output off regardless of all other signals used in the function, the *Force output fully off* step has to be used: to enable this, select *Disable automation* in the *Advanced* section, select *Force output fully off*, then double click on the *Signals* window and select the right signal to use:

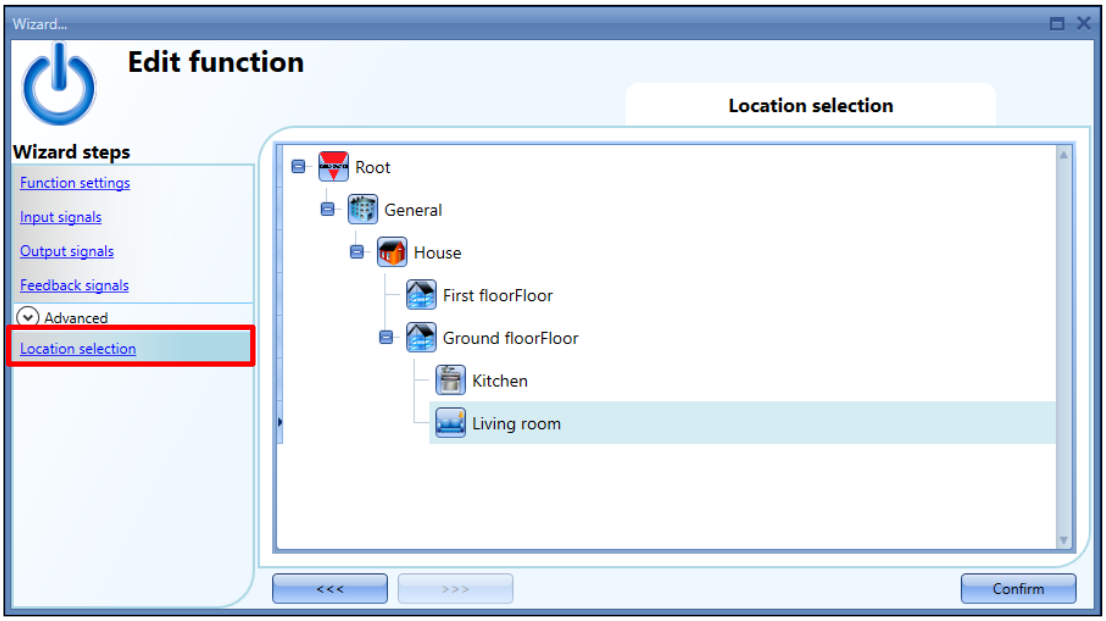


Each signal used in the *Force output fully off* window works as a level signal: Until the signal is active, the light function is forced off. When the *force output fully on* signal is activated and the *force output fully off* signal is activated, the *Force output fully on* signal has the priority.



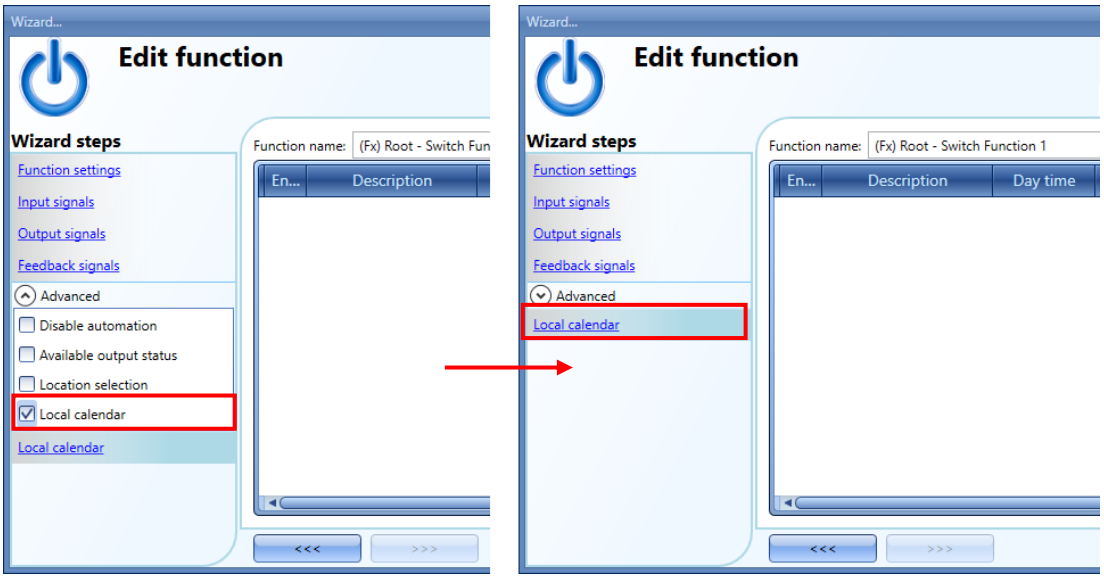
10.39.7 How to change the location of a function

To access the *Location* menu, the user should enable this in the *Advanced* functionalities.
From this window the location where the function is placed can be changed by simply selecting and clicking on the one required.



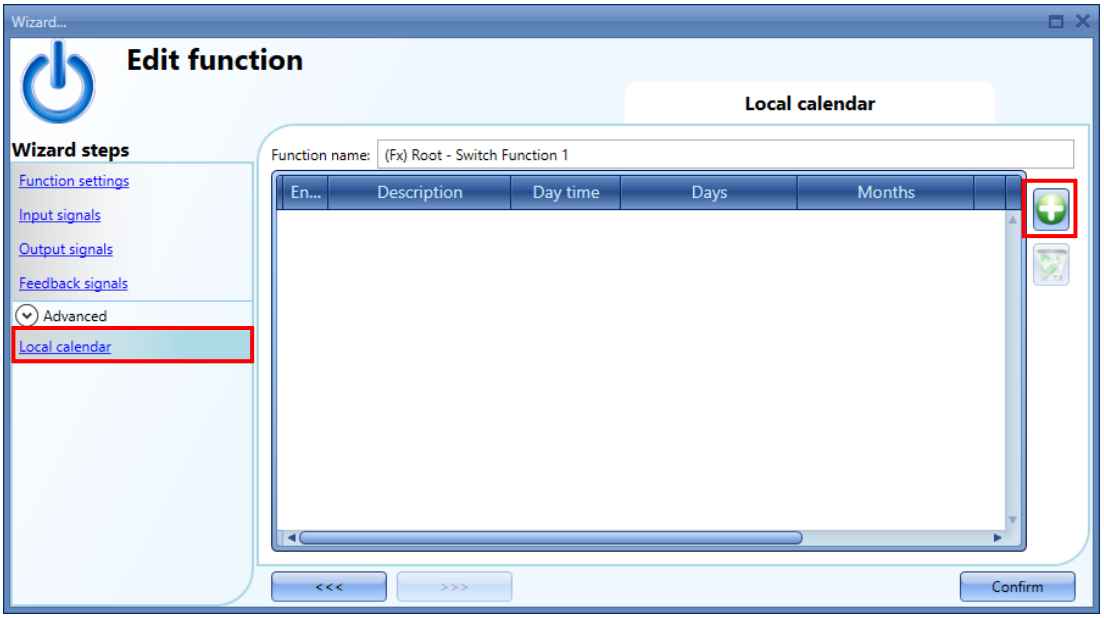
10.39.8 How to turn the Switch function ON/OFF using the Local calendar

If the user wants to turn the *Switch* function ON/OFF using an automation with the *Local calendar*, the relevant menu has to be enabled in the *Advanced* section.



10.39.8.1 Calendar activity

Once the user has selected the *Local calendar* field, to add a new *Calendar activity* they have to click on the **+** button, where the following window will appear:



A new *activity* will be added to the *Local calendar* list, as shown in the example below:

The available fields are as follows:

Enable: The user can enable/disable the calendar activity;

Activity name: In this field the user defines the name of the activity that will appear on the calendar list. This is a mandatory field;

Day time*:* The user can select the date (*Hours, Minutes, Seconds)* when the calendar has to operate;

Days: The user can select the days of the month when the calendar has to operate;

Months: The user can select the months of the year when the calendar has to operate;

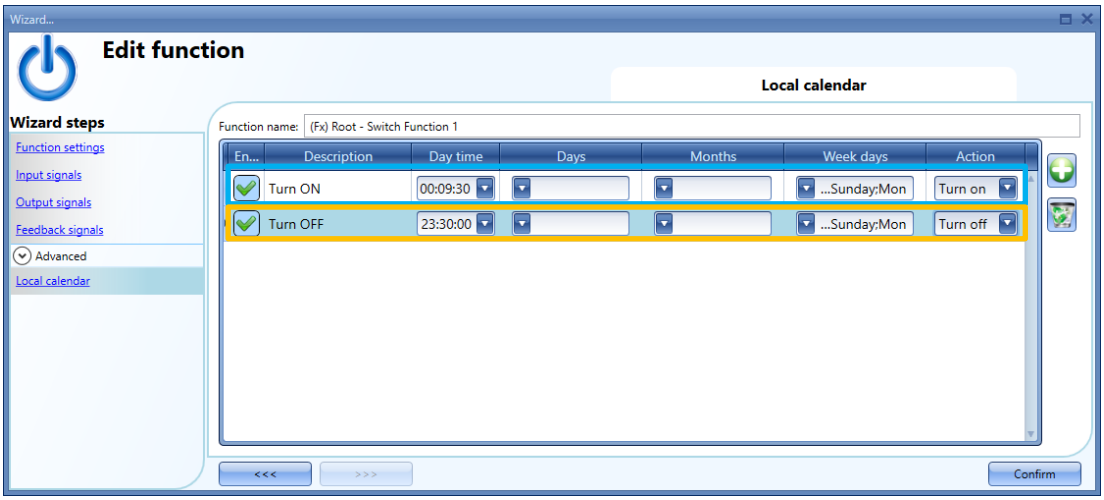
Week days: The user can select the days of the week when the calendar has to operate;

Action: in this field the user can select the action which has to be performed when the period starts. The actions that the user can choose are:

- Toggle: Switch function will be toggled at start time
- Turn ON: Switch function will be turned ON at start time
- Turn OFF: Switch function will be turned OFF at start time
- Automation toggle: Disable automation will be toggled at start time

Example

In the example below the user has configured two calendar activities: the first is set to turn the *Switch* function ON at 9:30 every day (Turn ON – blue rectangle below). The Switch function will be turned OFF at 23:30 every day with the second activity (Turn OFF – orange rectangle below):



11 How to configure BACnet objects in UWP 3.0

11.1 How BACnet is implemented in UWP 3.0

The UWP 3.0 operates as a server on BACnet/IP. This allows BACnet clients like DDC controllers and building management systems to read and control the UWP 3.0 I/O's and functions. Two different groups of BACnet objects are created:

- BACnet objects related to modules
- BACnet objects related to functions

All the BACnet objects in the UWP 3.0 supports COV, allowing the BACnet clients to be automatically updated by UWP 3.0 every time an object event occurs (e.g. change of present value).

11.2 BACnet objects related to modules

Each I/O-module, sensor or actuator that can be connected to the UWP 3.0 smart building system has a defined set of BACnet objects, reflecting the signals of the module. For example the bus-powered Co2/Temp sensor SHSUCOT has two BACnet objects of type "analog input", one for the CO2 value and the other for the temperature value. The PIR/Lux sensor SHSQP360L has two "binary input" objects for "presence" and "movement", and an "analog input" object for the lux value. The light switch B5X-LS4-U has 4 "binary input" objects (one for each button) and 4 "binary outputs" (one for each LED).

<u>In general:</u>

- For UWP 3.0 digital input and digital sensor signals, BACnet "binary input" objects are created
- For UWP 3.0 digital output signals, BACnet "binary output" objects are created
- For UWP 3.0 analog input, analog sensor signals and energy meter values, BACnet "analog input" objects are created
- For UWP 3.0 analog output signals, BACnet "analog output" objects are created

Since the scaling and engineering units of the analog signals have already been defined during the UWP 3.0 setup, the tool has the information it needs to create the analog BACnet objects correctly.

The object names (property: Object Name) are identical to the signal names in the UWP 3.0 tool, so they are very easy to identify when scanning the BACnet/IP network. If desired the object name can be changed.

This group of BACnet objects related to modules is typically used in those cases, where the controls strategy is performed by an external controller, and the UWP 3.0 is fully or partly acting as remote I/O system.

In App A of this manual is defined the complete list of modules with associated BACnet objects.

11.3 BACnet objects related to functions

Each UWP 3.0 function has a defined set of BACnet objects, reflecting the inputs, outputs and operating parameters of the function. For example the simple Light ON/OFF function has a "multi-state input" object to read the status of the light output, a "bitstringvalue" object with diagnostics information, a "multi-state output" object for switching the light ON or OFF, an "analog value" object to read/change the energy save time, and an "analog value" object to read/change the lux set-point defining day/night mode.
This group of BACnet objects related to functions is typically used in those cases, where the controls

strategy is performed by UWP 3.0 (e.g. constant light control). The BACnet objects related to functions allows other controllers or the building management system to read the function status, change operating parameters and trigger the function to go to a specific output state.

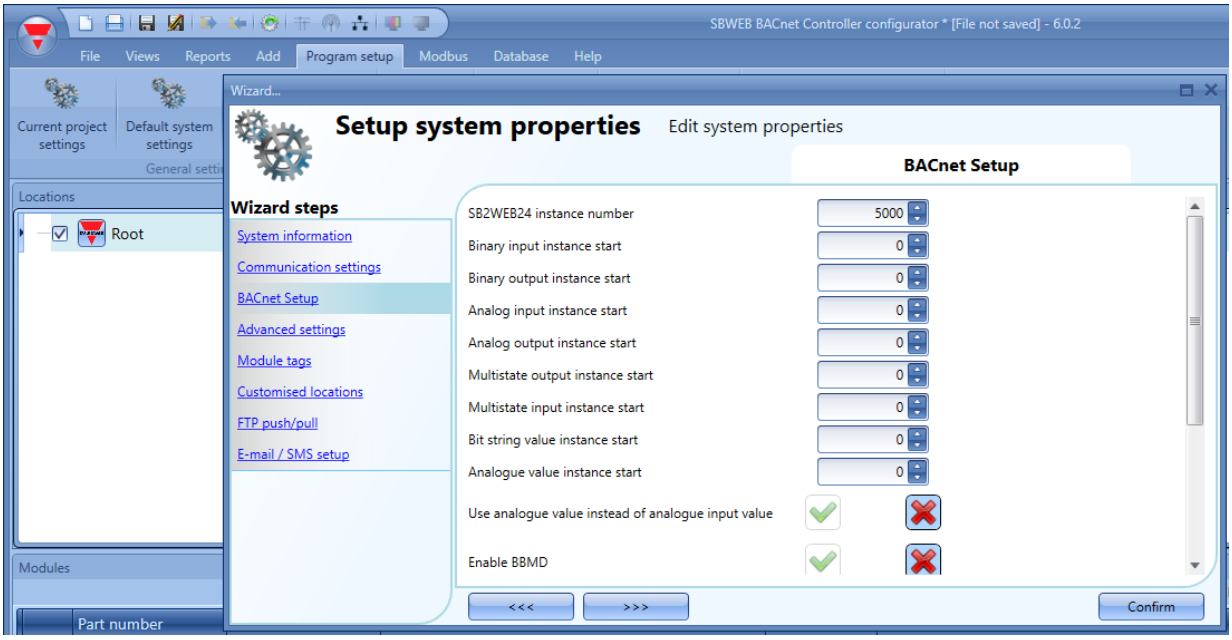
The UWP 3.0 can be used in a mixed mode where it is partly acting as controller (typically lighting functions) and partly as remote I/O for a DDC controller for HVAC systems.

In App B of this manual is defined the complete list of functions with associated BACnet objects.

11.4 Basic settings for instance numbers

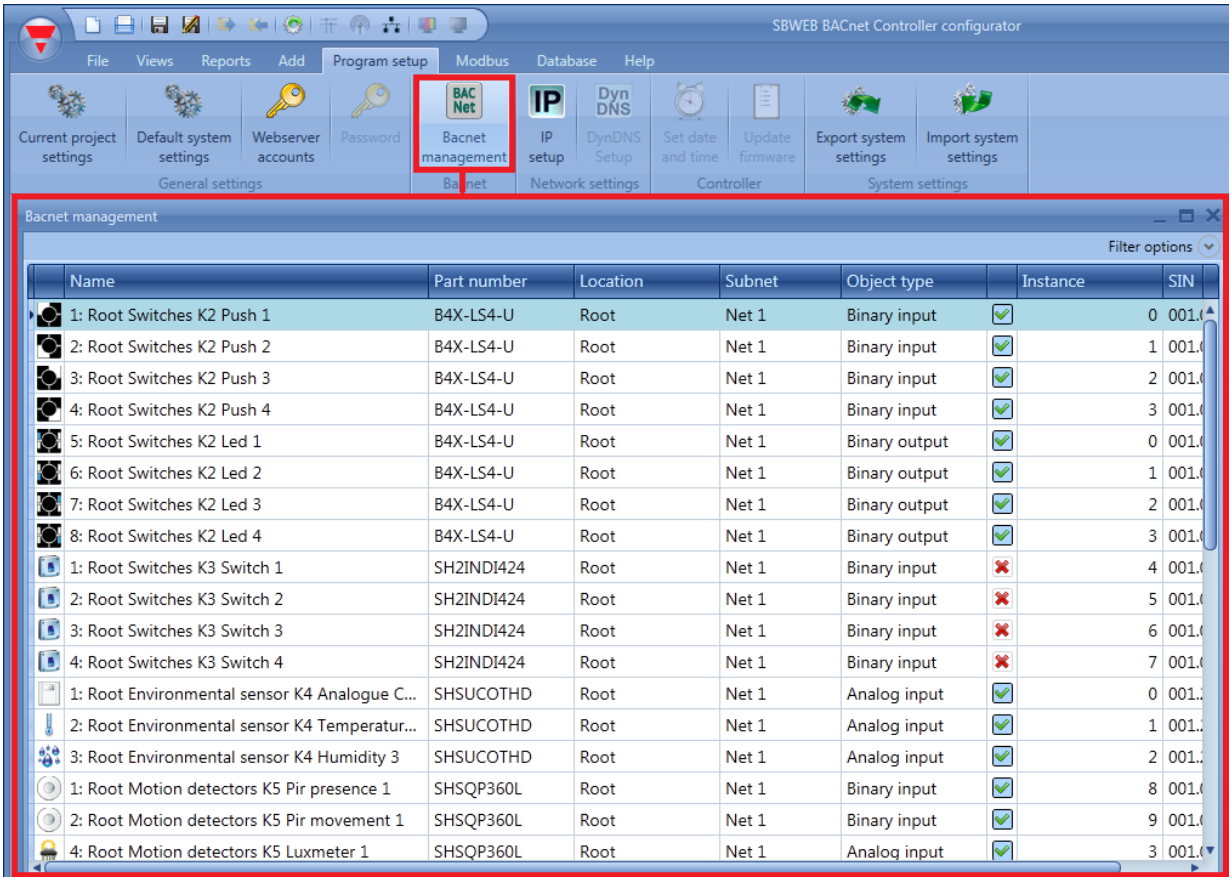
In the *Default system settings* menu, select BACnet setup as shown below. Here the instance number for the
Device object can be defined (default:5000).

For the other BACnet object types that appear in the UWP 3.0 (Binary Input, Binary Output, Analog Input, Analog Output, Analog Value, Multi-state Input, Multi-state output, Bitstringvalue) there is the possibility to define the start instance number for each object type. The tool will then assign the first created object of this type with the entered instance number, and the rest objects of this type will be assigned the following numbers consecutively. If desired, the individual instance numbers can still be changed manually in the signal view menu if desired.



11.5 Selection of BACnet objects to be created

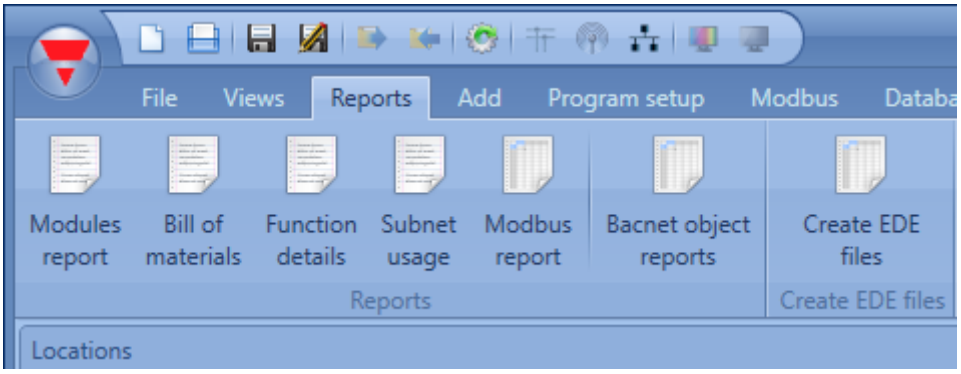
In the menu “BACnet management” under “Program setup” the user can define which BACnet objects shall be created. As can it can be seen in below screen shot, a list with all possible BACnet objects (as described in 11.2 and 11.3) appears, each with a tick box. The selection is very simple: Simply tick the objects to be used. Please note the column with the object instance numbers. These can be changed, simply by clicking the box and changing the number.
Also the live signal view contains the column with BACnet instance numbers for each signal, making it easy to identify the BACnet reference.



Name	Part number	Location	Subnet	Object type		Instance	SIN
1: Root Switches K2 Push 1	B4X-LS4-U	Root	Net 1	Binary input	✓	0	001.
2: Root Switches K2 Push 2	B4X-LS4-U	Root	Net 1	Binary input	✓	1	001.
3: Root Switches K2 Push 3	B4X-LS4-U	Root	Net 1	Binary input	✓	2	001.
4: Root Switches K2 Push 4	B4X-LS4-U	Root	Net 1	Binary input	✓	3	001.
5: Root Switches K2 Led 1	B4X-LS4-U	Root	Net 1	Binary output	✓	0	001.
6: Root Switches K2 Led 2	B4X-LS4-U	Root	Net 1	Binary output	✓	1	001.
7: Root Switches K2 Led 3	B4X-LS4-U	Root	Net 1	Binary output	✓	2	001.
8: Root Switches K2 Led 4	B4X-LS4-U	Root	Net 1	Binary output	✓	3	001.
1: Root Switches K3 Switch 1	SH2INDI424	Root	Net 1	Binary input	✗	4	001.
2: Root Switches K3 Switch 2	SH2INDI424	Root	Net 1	Binary input	✗	5	001.
3: Root Switches K3 Switch 3	SH2INDI424	Root	Net 1	Binary input	✗	6	001.
4: Root Switches K3 Switch 4	SH2INDI424	Root	Net 1	Binary input	✗	7	001.
1: Root Environmental sensor K4 Analogue C...	SHSUCOTHD	Root	Net 1	Analog input	✓	0	001.
2: Root Environmental sensor K4 Temperatur...	SHSUCOTHD	Root	Net 1	Analog input	✓	1	001.
3: Root Environmental sensor K4 Humidity 3	SHSUCOTHD	Root	Net 1	Analog input	✓	2	001.
1: Root Motion detectors K5 Pir presence 1	SHSQP360L	Root	Net 1	Binary input	✓	8	001.
2: Root Motion detectors K5 Pir movement 1	SHSQP360L	Root	Net 1	Binary input	✓	9	001.
4: Root Motion detectors K5 Luxmeter 1	SHSQP360L	Root	Net 1	Analog input	✓	3	001.

11.6 BACnet report

In the menu “BACnet object reports” documentation for the selected BACnet objects with object types, instance numbers and object names can be obtained.

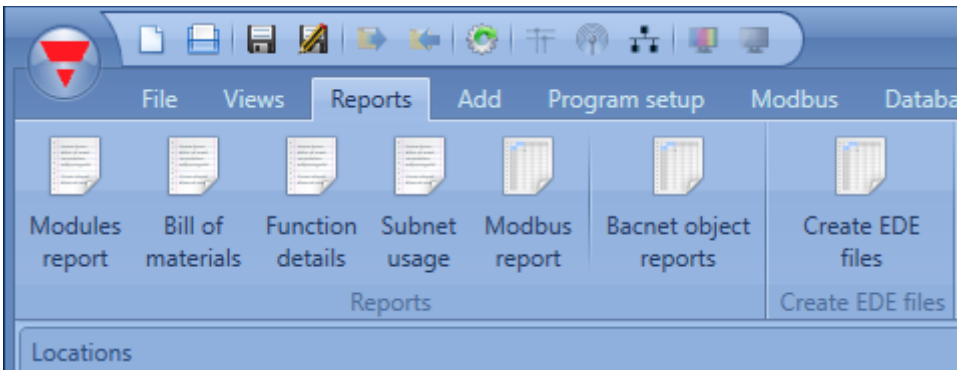


N.	Part number	Name	SIN	Subnet	Root	Object type	Instance number	Object identifier
1	B4X-LS4-U	1: Root Switches K2 Push 1	001.013.102	Net 1	Root	Binary input	0	12582912
2	B4X-LS4-U	2: Root Switches K2 Push 2	001.013.102	Net 1	Root	Binary input	1	12582913
3	B4X-LS4-U	3: Root Switches K2 Push 3	001.013.102	Net 1	Root	Binary input	2	12582914
4	B4X-LS4-U	4: Root Switches K2 Push 4	001.013.102	Net 1	Root	Binary input	3	12582915
5	B4X-LS4-U	5: Root Switches K2 Led 1	001.013.102	Net 1	Root	Binary output	0	16777216
6	B4X-LS4-U	6: Root Switches K2 Led 2	001.013.102	Net 1	Root	Binary output	1	16777217
7	B4X-LS4-U	7: Root Switches K2 Led 3	001.013.102	Net 1	Root	Binary output	2	16777218
8	B4X-LS4-U	8: Root Switches K2 Led 4	001.013.102	Net 1	Root	Binary output	3	16777219
9	SH2INDI424	1: Root Switches K3 Switch 1	001.054.189	Net 1	Root	Binary input	4	12582916

11.7 How to create an EDE-file

In some cases, if the BACnet client does not have the option to scan the network or if offline configuration is needed, EDE-files are needed to set up the BACnet client to communicate with the UWP 3.0. The EDE-files define all the objects, object types, instance numbers etc. in the UWP 3.0 in a standardized that can be read by the client.

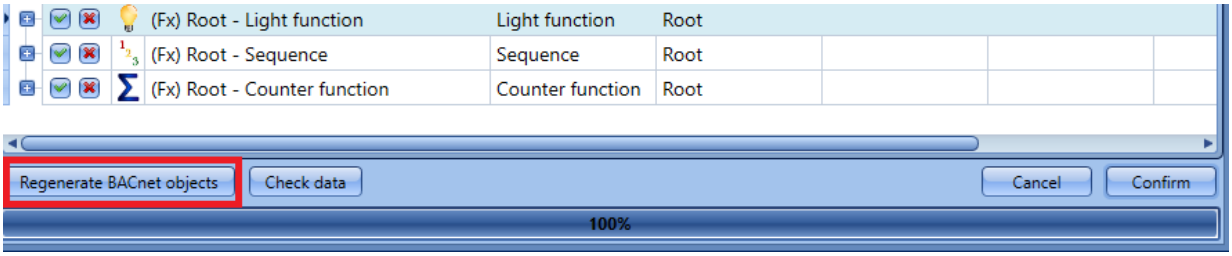
To create the EDE-files, select the menu “Create EDE files”



The UWP 3.0 Tool will ask for the name and path of the files to be saved, and subsequently they will be ready for use in the setup of the BACnet client.

11.8 How to Regenerate BACnet map

If all the addresses have to be reset and a new BACnet map is needed, the user has to click on the *Bacnet management* icon and press *Regenerate BACnet objects* button, as shown in the picture below.



N.B: If a BACnet map with at least a Counter function has been created from an older version of SB Tool (older than *SB Tool 6.2.3 – R594*), the BACnet map must be **regenerated again**.

12 How to configure a DALI network with a SB2DALIT8230 module

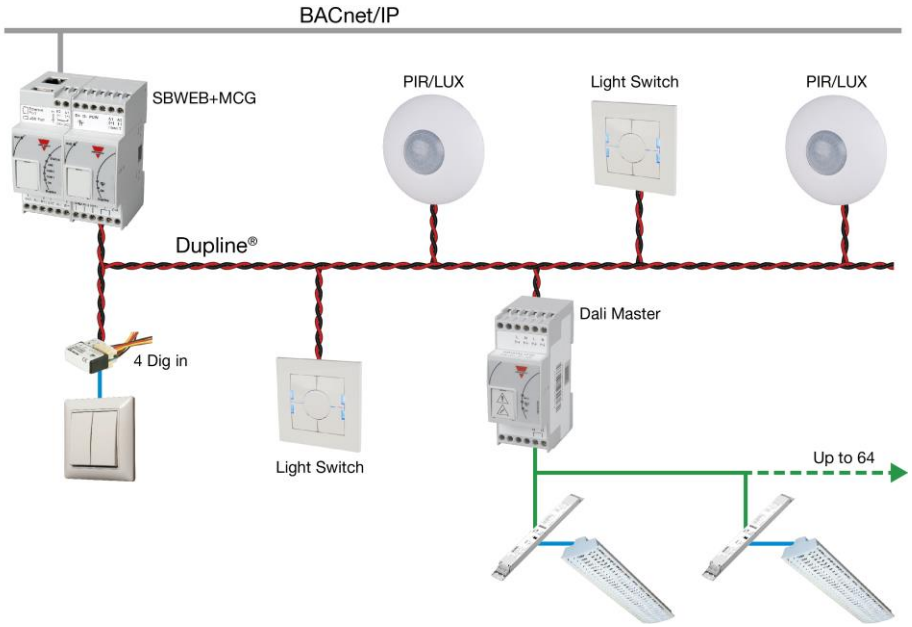
12.1 How to interface Dupline to DALI

The DALI bus is the standard for interfacing to lighting actuators such as ballasts for dimming of fluorescent lighting, LED drivers, RGB lighting etc.

The SB2DALIT8230 module is a gateway between the Dupline bus and the DALI (Digital Addressable Lighting Interface) protocol, it operates as DALI master and has a built-in DALI power supply. As shown below, the SB2DALIT8230 connects directly to the Dupline bus, which means it can be installed de-centrally.

Up to 7 SB2DALIT8230 modules can be connected to one Dupline bus, and each SB2DALIT8230 can manage up to 64 DALI ballasts. The SB2DALIT8230 supports DALI ballast, but not DALI sensors or DALI input modules. This is because the concept is to use the bus-powered Dupline devices for this purpose, as shown below.

The grouping of the lighting outputs is also managed by the UWP 3.0 tool, so the configuration can all be done in the same programming environment by using a single tool.



12.2 How many SB2DALIT8230s can be managed

Each SH2MCG24 can control up to 7 SB2DALIT8230s and each DALI ballast is managed as a dimmer output from a Dupline point of view. Dimmer outputs from different Dupline networks **cannot** be mixed in the same DALI group.

Each smart-Dupline network can manage up to 256 dimmer functions:

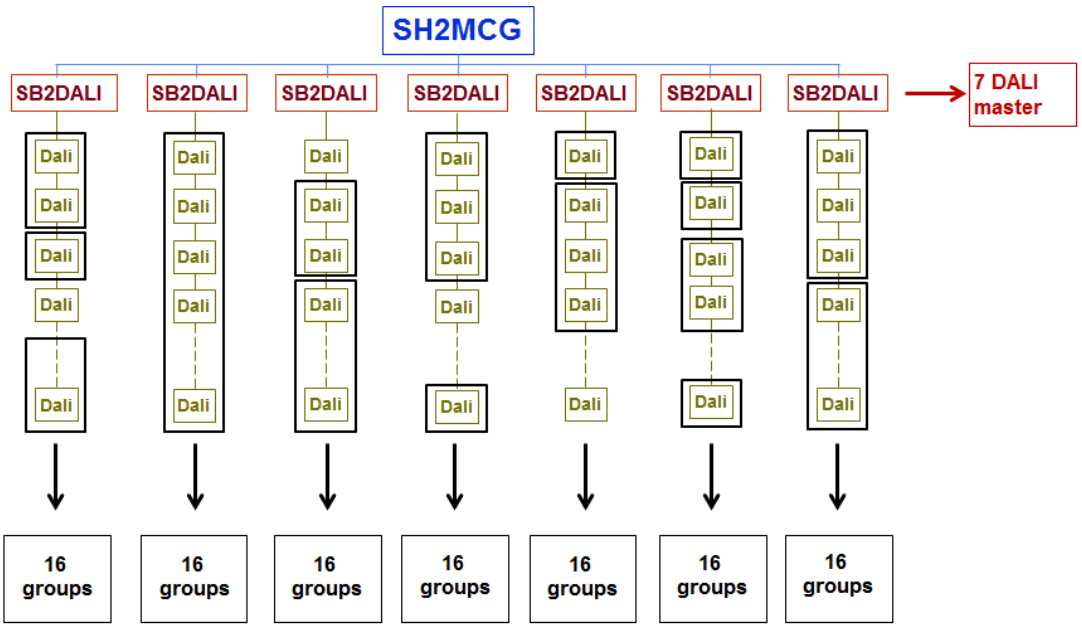
- 128 functions with 230V and 1-10V dimmers
- 128 functions with DALI devices

12.3 DALI groups

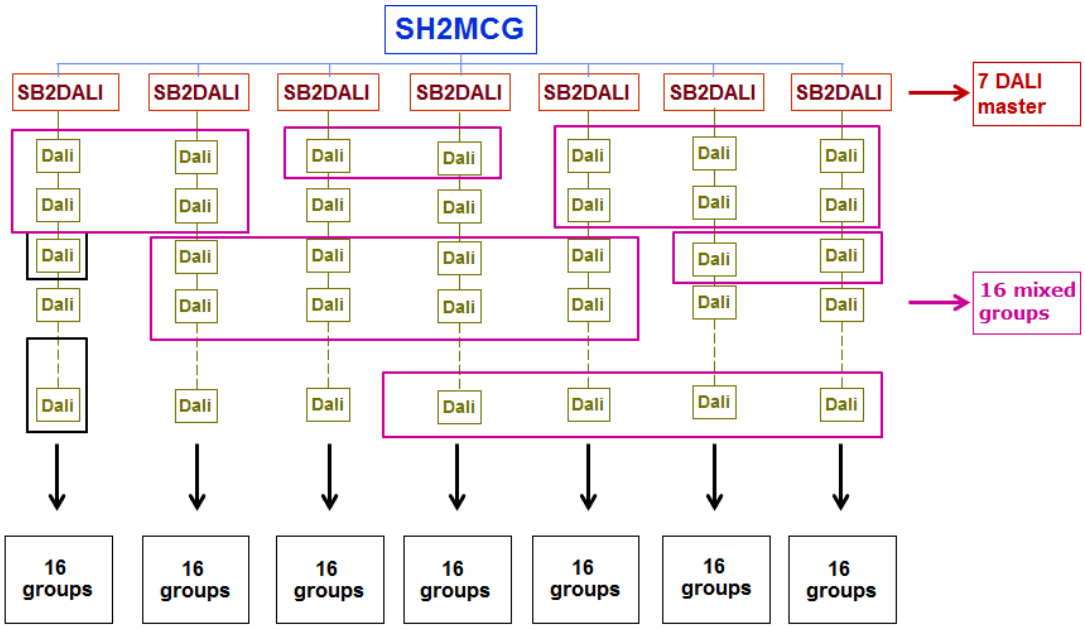
In the UWP 3.0 Tool the DALI groups are managed by the *DALI network manager* window. This collects all the functionalities and options to manage a Dali network. The maximum number of groups for each DALI network is 16. The SB2DALIT8230, the DALI Master module, assigns an address (0 to 63) to each connected DALI ballast. The advanced functionalities available in the DALI network manager window help the user to manage the DALI ballast, e.g. the randomly assigned address may be changed at any time or rearranged in the Device list according to the project being commissioned.

The examples below show how the DALI group can be managed in an installation:

A. Up to 16 DALI groups for each DALI master SB2DALIT8230



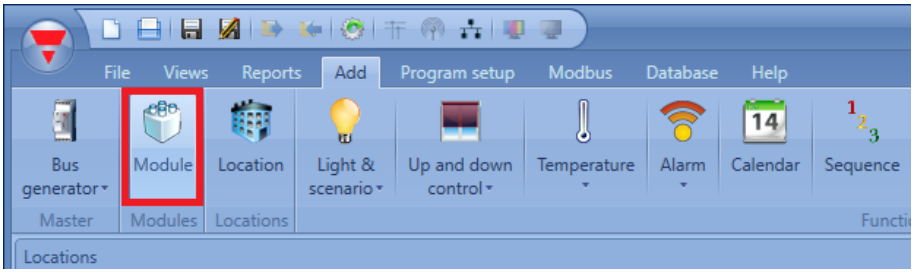
B. Up to 16 DALI groups belonging to different DALI masters



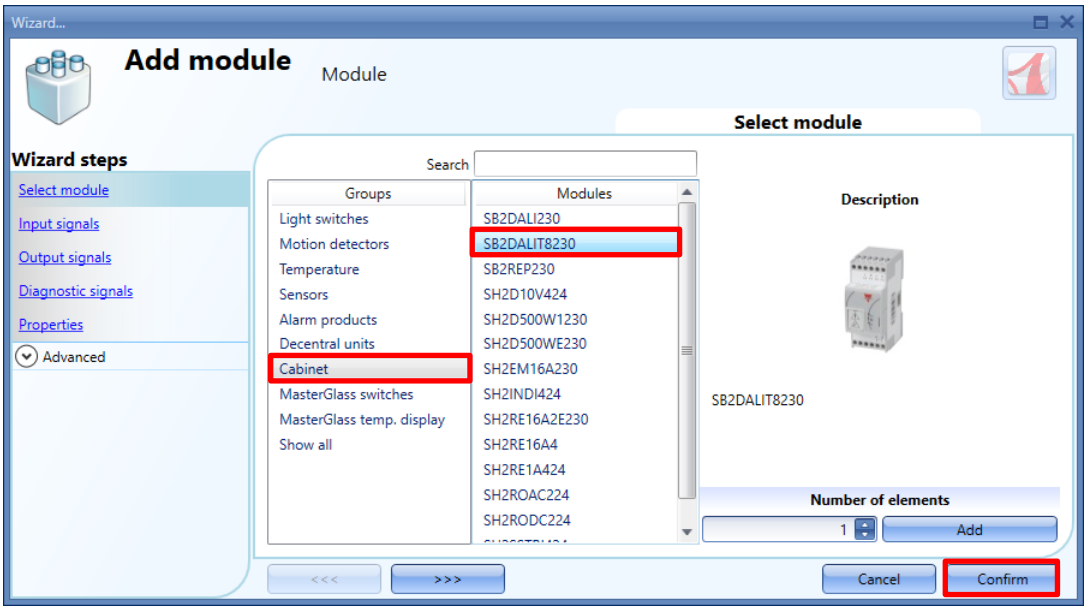
12.4 How to add the SB2DALIT8230 DALI master

The SB2DALIT8230 is a gateway between the Dupline bus and the DALI (Digital Addressable Lighting Interface) protocol and is a DALI master. It manages one DALI bus.The SB2DALIT8230 manages all the different types of DALI ballast, e.g. type 6, type8, etc..

To add it into the UWP 3.0 tool, click on *Add module, Cabinet, SB2DALIT8230.*



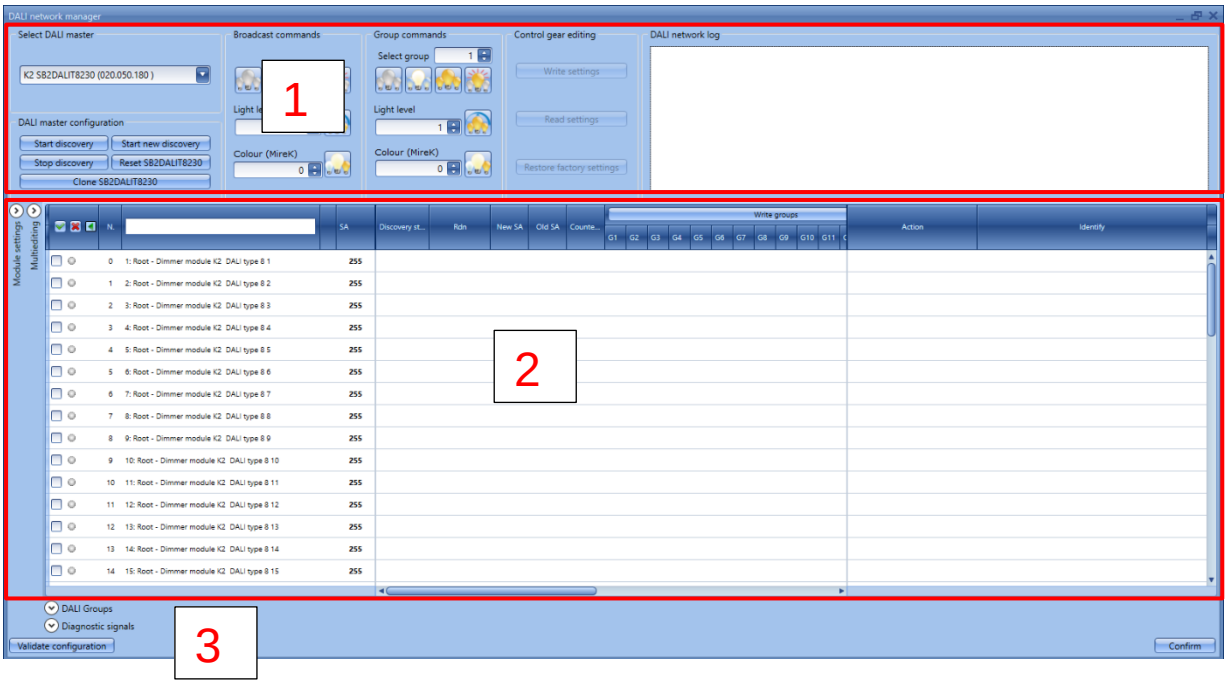
When the *Confirm* button is pressed, the module is inserted.



12.5 DALI network manager window

To manage and program a DALI network **connected** or not to an SB2DALIT8230 DALI master, click on the relevant image once it is added to the project:

The ***DALI network manager*** window will appear, as shown in the picture below:



The DALI network manager window helps the user to configure, test and set parameters for DALI ballasts. It can also be used for grouping DALI ballasts and performing broadcast commands and group commands. The window is divided into three areas:

12.5.1 Area 1 – Discovery and testing tools

Area1 is divided into two sections: The one on the left contains the tools to connect to the DALI masters and the discovery options. In the central section, the broadcast commands are present for the whole DALI network and for the DALI groups. The Debug window on the right shows any event related to discovery and program activities performed by the user.

12.5.2 Area 2 – Programming tools

This is the area where the available DALI ballasts are shown and where the properties of each one can be changed. This area is divided into three sections: the one on the left shows the signals information about the DALI ballast. In the central section the user can manage the settings, such as DALI groups, fade time and temperature colour. The section on the right offers the actions and commands with which to manage them individually.

12.5.3 Area 3 - Diagnostic tools

In the lower section the Diagnostic functionalities are collected in a hidden panel: the user can check this information if needed.

12.6 How to connect to a DALI network

The discovery and configuration procedure of a DALI network via the UWP 3.0 Tool can change slightly. There are two ways of doing this; please follow the instructions below:

Procedure	Description
Online configuration The SB2DALIT8230 DALI master is already connected to the DALI network	The user can follow this procedure to automatically find and program the DALI ballasts connected to the DALI bus. *This is the most common scenario*
Offline configuration The SB2DALIT8230 DALI master is not connected to the DALI network Initial settings are given to units before they are connected to the system	This configuration can be useful when the user needs to prepare an "off-line" configuration of the installation. Follow this procedure to manually add and program the DALI ballasts. *This is valid when the DALI master is not connected to an SBP2WEB24 or to any Dupline network*

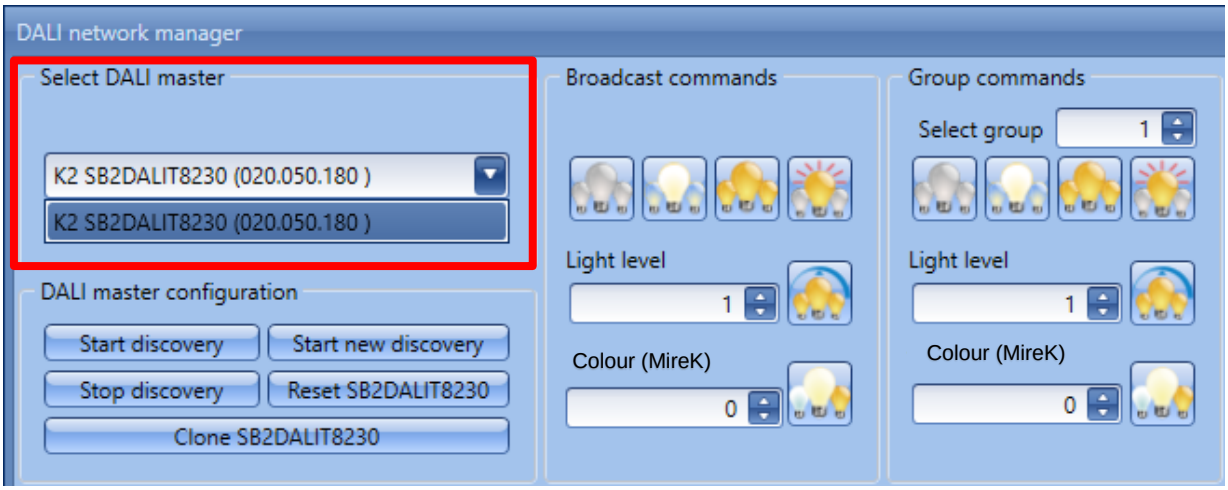
12.6.1 How to automatically find and address the ballasts – The network is connected to the master

The online procedure is valid when the DALI network is already connected to a SB2DALIT8230 and the DALI ballast information is given to units which are connected to the system.

Initial settings are given to units which are connected to the system

Step1. Check the wiring connection

Once the DALI network is properly connected to a SB2DALIT8230, this step checks that the wiring is correct for the DALI network installation.
To proceed, in the available DALI modules of the DALI network manager, the user has to select the SB2DALIT8230 module which is connected to the DALI network they want to test, as shown in the rectangle in the picture below:



Once the DALI master has been selected, the broadcast commands will be available, as shown in the picture below:

The broadcast commands can be executed to determine whether communication is working properly in the DALI circuit and whether all the devices in the DALI circuit are connected.
The broadcast commands are applied to all the DALI ballasts belonging to the connected network. The user can send the broadcast commands by clicking on the command buttons, as shown below:

Broadcast commands	Behaviour
All OFF	This command switches all the lights OFF
All MIN – Set all at min	This command switches all the lights ON to the minimum level
All MAX - Set all at max	This command switches all the lights ON to the maximum level
Flashing	This command initiates the blinking of all the lights
1 Set level	This command dims all the lights up to the values set in the related field
0 Set colour	This command sets all the lights to the temperature colour according to the value set in the related field. *Note: This command is available only for the ballast Type 8 devices*

Step2. Discover the DALI ballasts

The user, after having tested the DALI network connected to the DALI master, can start a network discovery in order to find and **address** all the DALI ballasts connected to the DALI bus.

12.7 DALI addressing

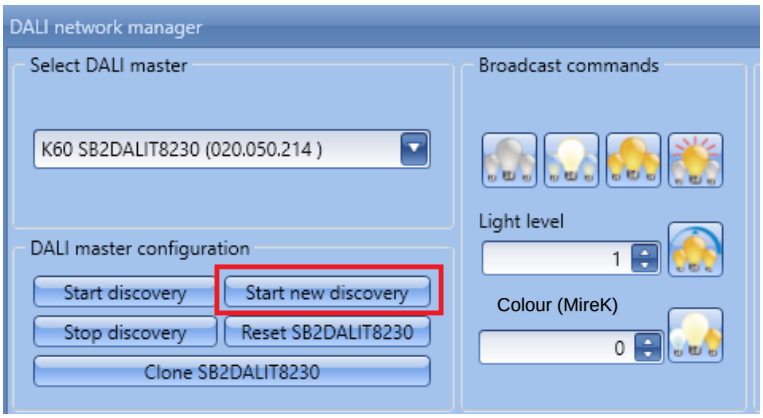
The DALI bus manages two types of addresses:

- A 6-bit SA Short Address (0-63) which is used in normal operations.
- A 24-bit random address (0 - FFFFFE hex) which is used when the ballast is first connected to a DALI network and also used to assign the short address. FFFFFF is the value usually present when the ballast comes out of the factory.

When a ballast is new**:**

- It does not have a short address and it cannot answer to individual addressing
- It does not belong to a group or scene and thus it cannot respond to any group or scene commands
- *It answers to broadcast commands* and can be tested.

When a new DALI network is installed, the first thing to do is to launch a new discovery to find the connected ballasts.



When the DALI network discovery starts, the master assigns a 24-bit random address to each ballast. This type of address can have values starting from 0 up to 16.777.214 (2^{24}-2).
In the unlikely event of a duplicated address, an error message is shown advising that there is a conflict and the user can solve this manually or by launching a *new discover.*

12.7.1 Start new discovery procedure

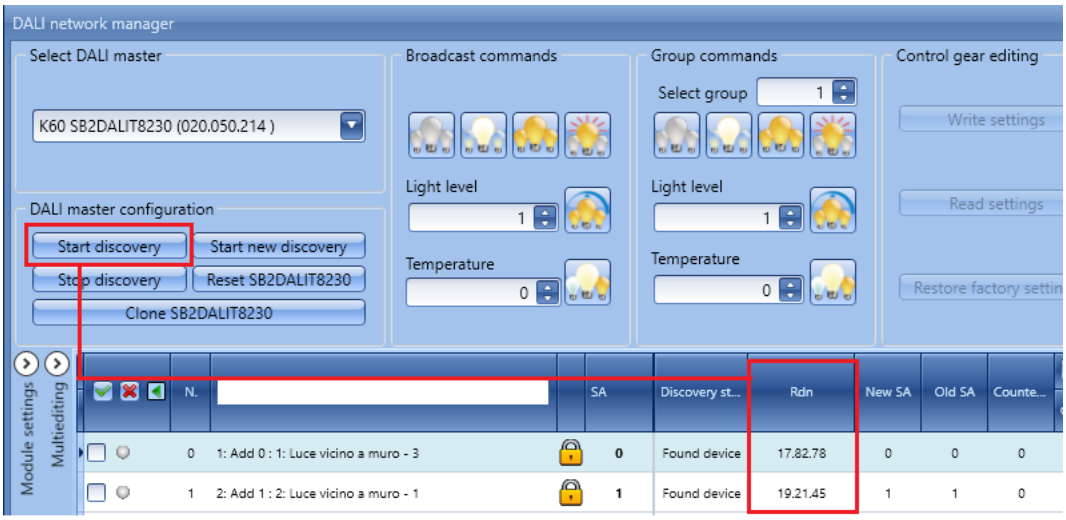
Each time the user selects the action *Start new discovery*, the SB2DALIT8230 sends an **INITIALISATION** broadcast **COMMAND** to all the devices in the network.

A **RANDOMISE** command is then sent to the ballasts and they generate a new 24-bit random address (Rdn) (this new address should be hold by the ballast until a new **INITIALISATION** command is received).

The DALI master then puts the ballasts in order according to their random addresses (Rdn), and assigns a short address (SA) to each of them. Short addresses start from 0 and go up to 63.

12.7.2 Start/stop discovery procedure

Each time the user clicks on *Start discovery*, the SB2DALIT8230 searches for the devices using the known RDNs (random addresses).



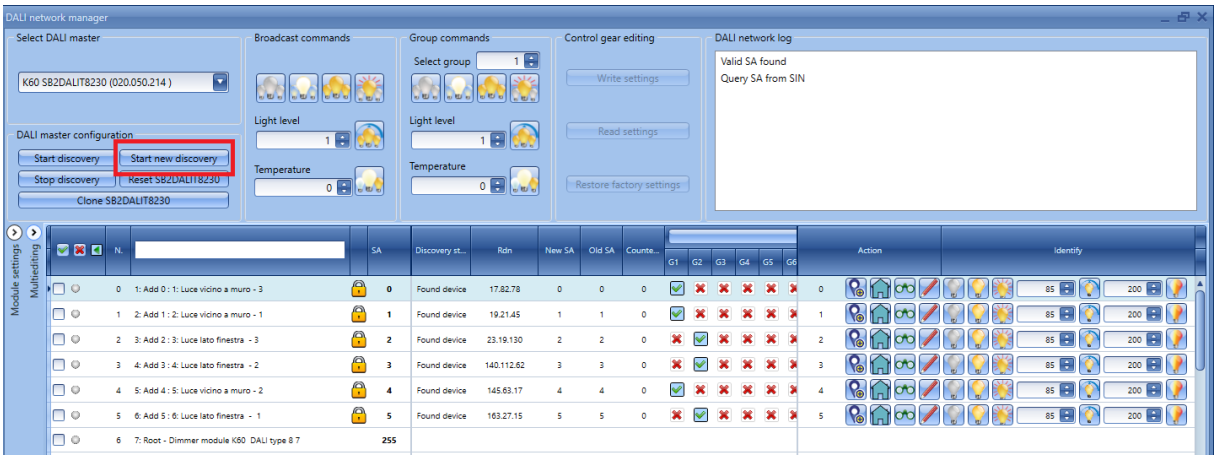
This discovery does not assign any new short addresses.

This command does not find any ballast with null Rdn (random address) and can be used to refresh the status of all the ballasts connected to the network.

IMPORTANT NOTE:

The first time a DALI network is programmed, the user must select the *Start new discovery* button because all the connected ballasts need to be initialised.

This action is mandatory in order to scan the DALI network for the first time and update the table of the connected ballasts (either it is a new network, or a network already programmed with another system).

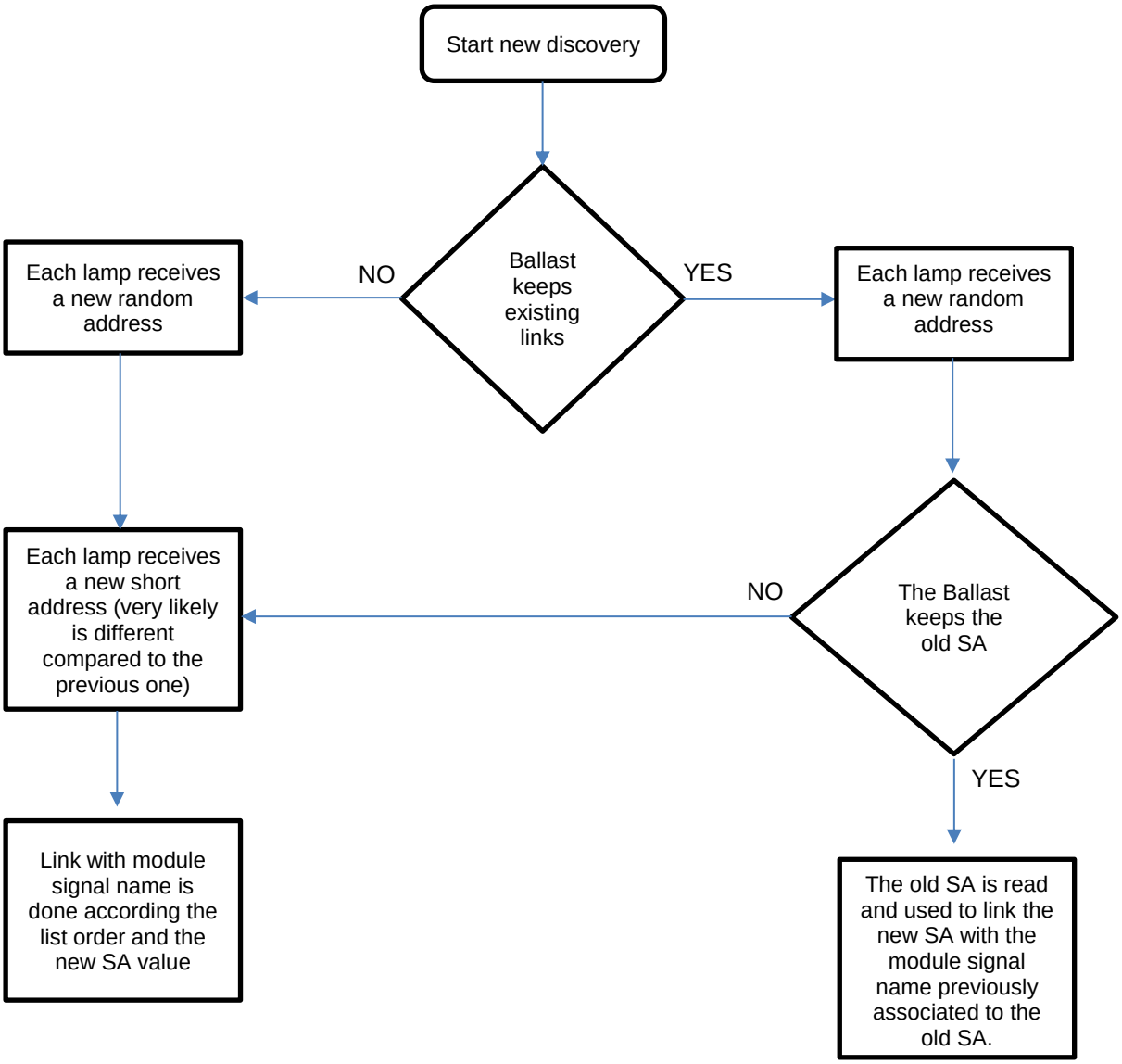


12.8 When *Start new discovery* is to be used

1. The first time an SB2DALIT8230 module is programmed
2. Every time an inexperienced installer wants to program the DALI network
3. Every time the installer wants to assign new short addresses
4. Every time a new ballast is added to the DALI network

12.8.1 New discovery maintaining existing short addresses

This procedure can be used when the user needs to extend an existing DALI network, or they need to replace a faulty ballast with a new one.
This procedure is based on the DALI ballast information: if the ballast maintains the old assigned SA (as it should), the system is able to maintain the present name even if a new SA is assigned.
This procedure may lead to a problem if the ballast reset its SA when an **INITIALIZE** command has been received (this should not be the case), this procedure may lead to a conflict if a lamp generates a new random address that is already in use (this very seldom happens).



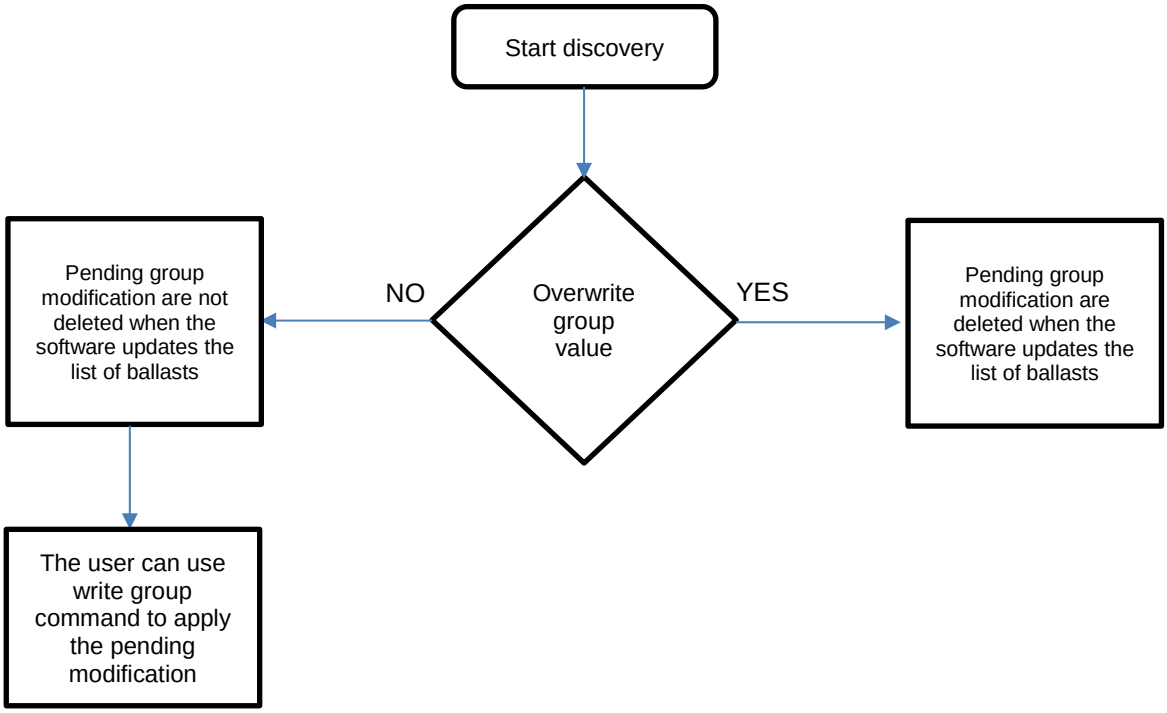
12.9 When *Start discovery* can be used

1. Each time the installer wants to refresh the parameters of all the ballasts in the list

This command does not change the existing SA.

12.9.1 Discovery overwriting groups value

This procedure can be used to scan the network while keeping the pending group changes, for example if the user is changing the groups of some ballasts and a discovery is needed, the pending modifications are not deleted when the software updates the list of connected ballasts.



12.10 Project design (Pre-programming – How to execute individual commands)

This section contains instructions about optional operations: the user can refer to this chapter if they need to identify each ballast individually and then put them into DALI groups.

Once the expected DALI ballasts have been discovered and shown in the Device list of the DALI network manager window, they can be individually identified by performing the individual commands and saved with a custom label (e.g. giving the name of the Location where it is installed). To complete this part, the ballasts can be grouped into up to 16 different DALI groups.

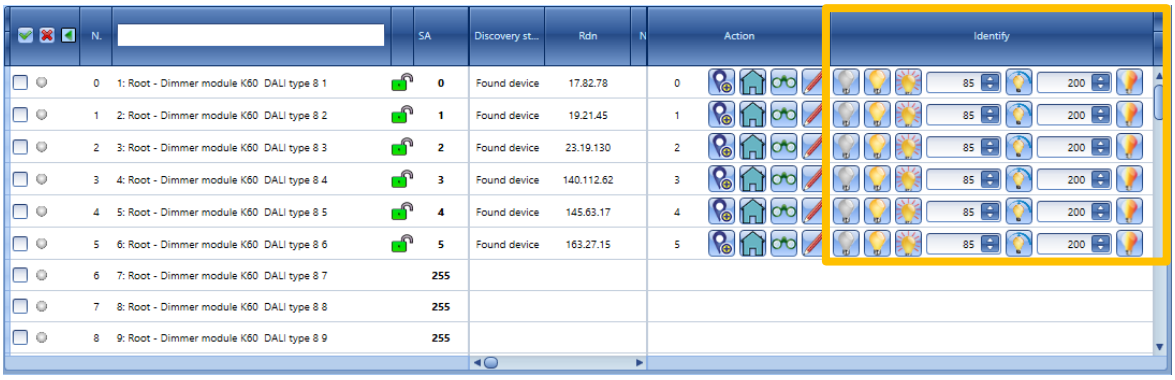
The **Device list** of the *DALI network manager* window is divided into three areas, as shown below:

Area 1 = Ballast signals;
Area 2 = Information and settings related to the ballasts;
Area 3 = Individual commands and operations;

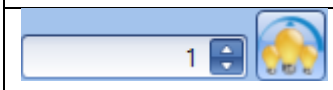
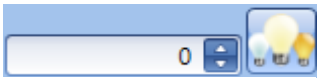
In the header row, as shown in the red rectangle below, the user can click on the *small green V* icon to select all the DALI ballasts: otherwise, if the user clicks on the small red cross, all the DALI ballasts will be de-selected. The can be used to show/hide the specific fields that have to be used to assign the DALI ballast to the DALI groups. These functionalities are available in order to manage the Multiediting operation (please refer to the relevant section in this manual).

12.10.1 Define the position where each ballast is located

To identify which DALI ballast in the *Device list* corresponds to which light in the building, the individual commands can be used. The individual commands are available for each ballast that has been found by the DALI bus scan, as shown in the orange rectangle below:

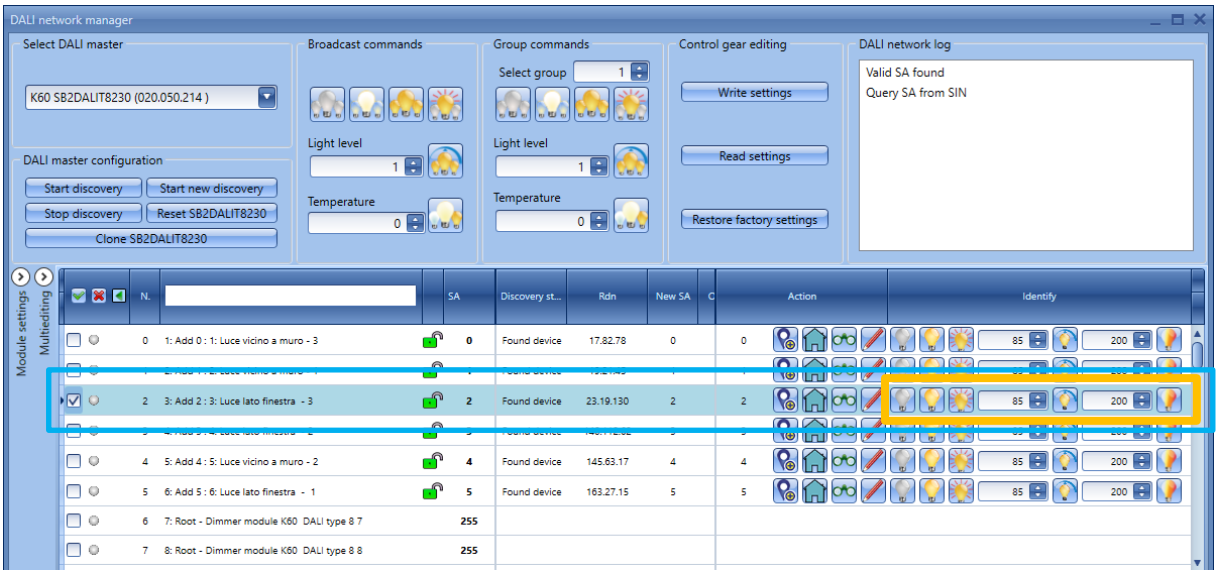


The behaviour of each command is shown in the table below:

Command	Behaviour
Switch off	This command switches the light OFF
Switch on	This command switches the light ON
Blink	This command initiates the blinking of the light
 Set level	This command dims the light up to the values set in the related numeric field
 Set colour	This command sets the light to the colour temperature (MireK) according to the value set in the related field *Note: This is available only for the ballast Type 8 devices*

12.10.1.1 How to identify the ballast individually

To identify the position of each ballast, the user simply needs to send a command and walk around the building area. The picture below shows an example:



The related ballast receives the command that changes its status according to the command that has been executed by the user.

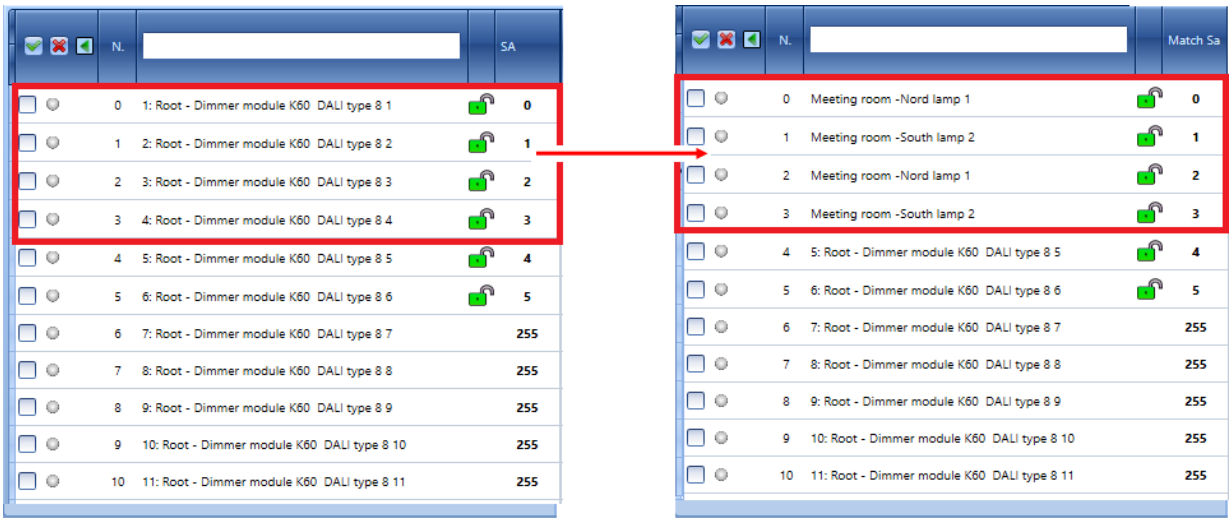
12.10.1.2 How to change the default signal name individually

Once it is located, the user can change the default label, giving it its proper name, e.g. the location where the device is installed. There are two different ways to change the default name:

1. Individually

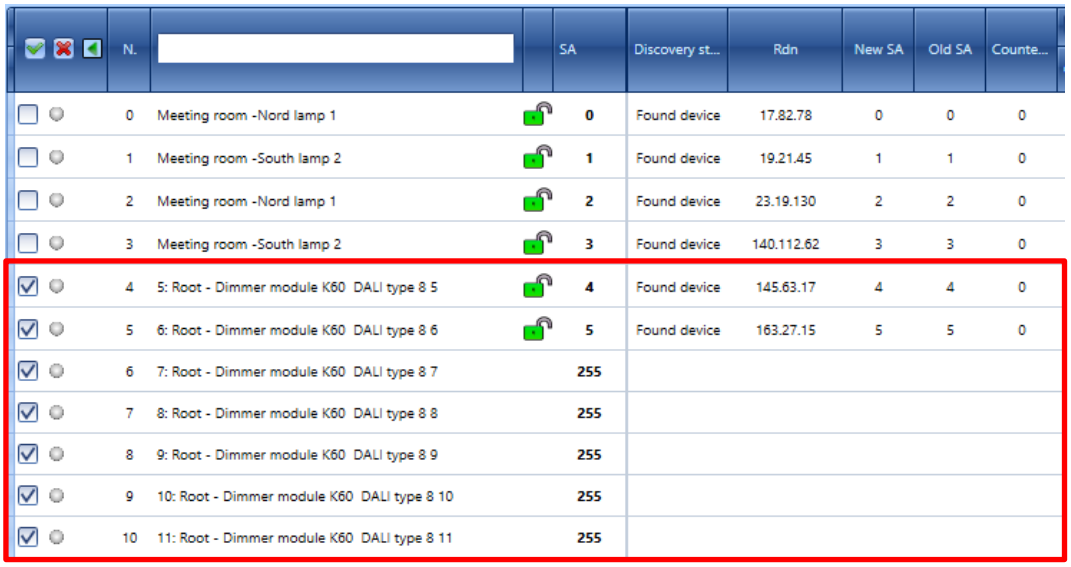
In the *Signal name* field of the selected ballast, type the new name and press Enter.

In the example below, the user has changed, one by one, the label name for the ballasts belonging to the *Meeting room* location:

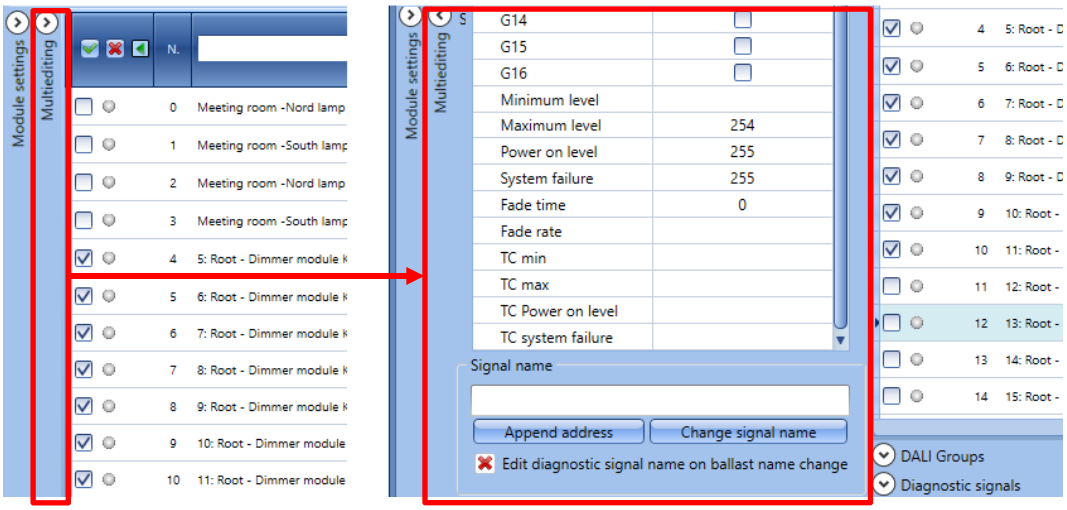


2. Multiediting

To change the default name of multiple ballasts at the same time, in the *Device list* section, the user can select which DALI ballasts have to be managed. To do this, the check-box next to each ballast has to be selected, as shown below:

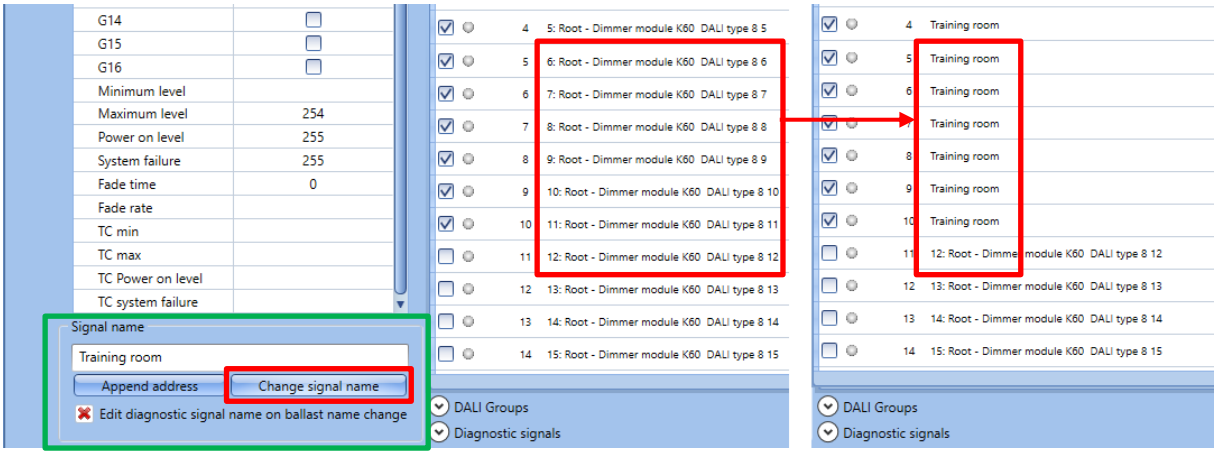


By clicking on the button, the *Multiediting* panel will then appear, as shown below:

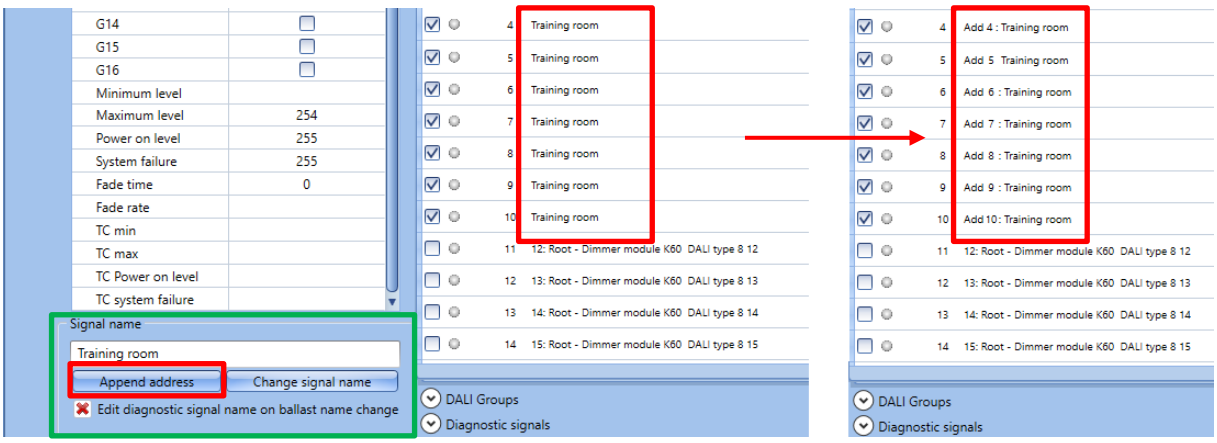


In the *Multiediting* panel, in the *Signal name* text-box, the user can type the name they want to assign to all the signals that have been selected.

By clicking on the *Change signal name* button, the text on the field is assigned to all the selected DALI ballasts:



By clicking on the *Append address* button, the *short address value* will also be added to the new signal name:

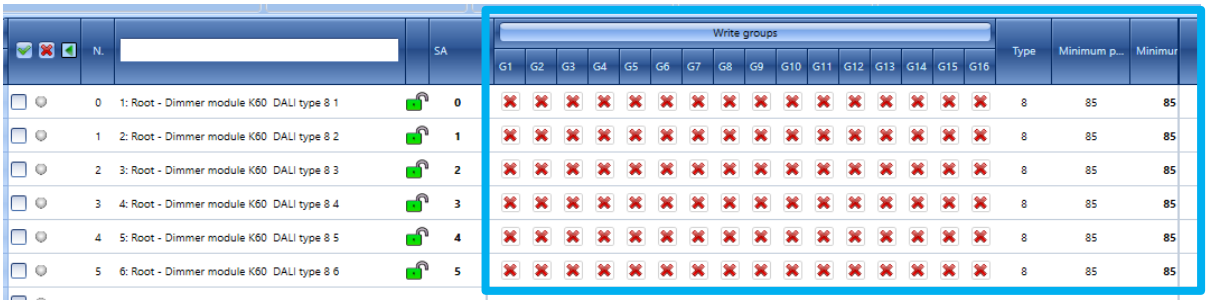


12.11 Programming – How to manage the ballast settings

As described in the previous sections of this manual, all the DALI ballasts already found and addressed are displayed in the *DALI list* section.

A DALI ballast is an intelligent device that can be configured to remember its power-on level, minimum level, maximum level, system failure level, fade rate and fade time, also which DALI groups it belongs to, up to 16. The ballasts *type 8* devices can also manage the tunable white control, such as Colour temperature (CT) minimum level, CT maximum level, Power-on level CT and system failure CT.

In the *Device list* section, as shown in the light -blue rectangle below, the user can read and change the value of the ballast parameters according to the project specification.



12.11.1 Group settings

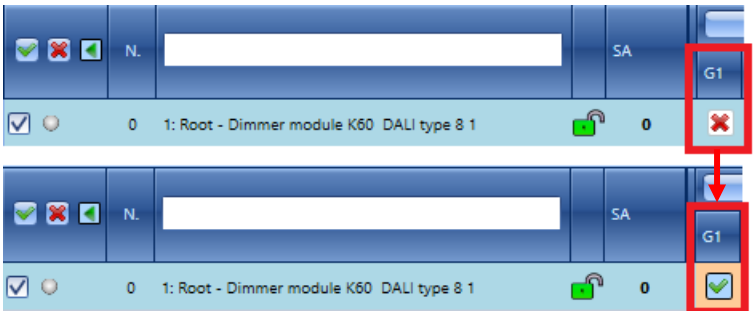
Each DALI ballast can be assigned to several groups, up to 16. The user can assign them individually or together at the same time by using the options available in the *Multiediting* panel.

NOTE: In the header row of the Device list, the user has to check that the Groups columns are visible, otherwise please click on the icon in order to show them.

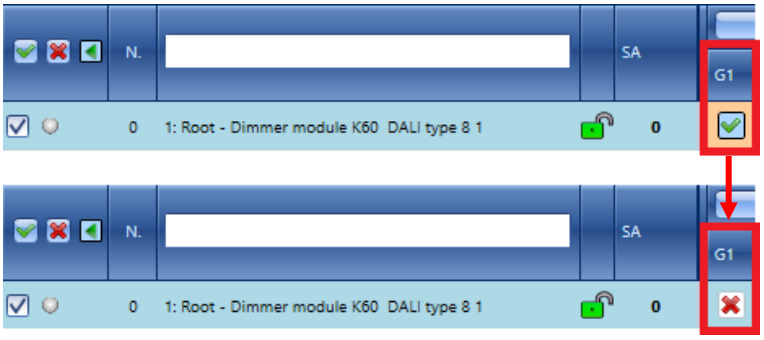
12.11.1.1 How to assign a single DALI ballast to one or more DALI Groups

The assigning of DALI ballasts to one of the 16 available groups is performed by activating the checkbox 1 to 16. In the *Device list* section, the user for each ballast can choose which groups they want to assign it to, as shown below:

- To assign the related ballast to a group (1 to 16), the *green V* icon must be checked



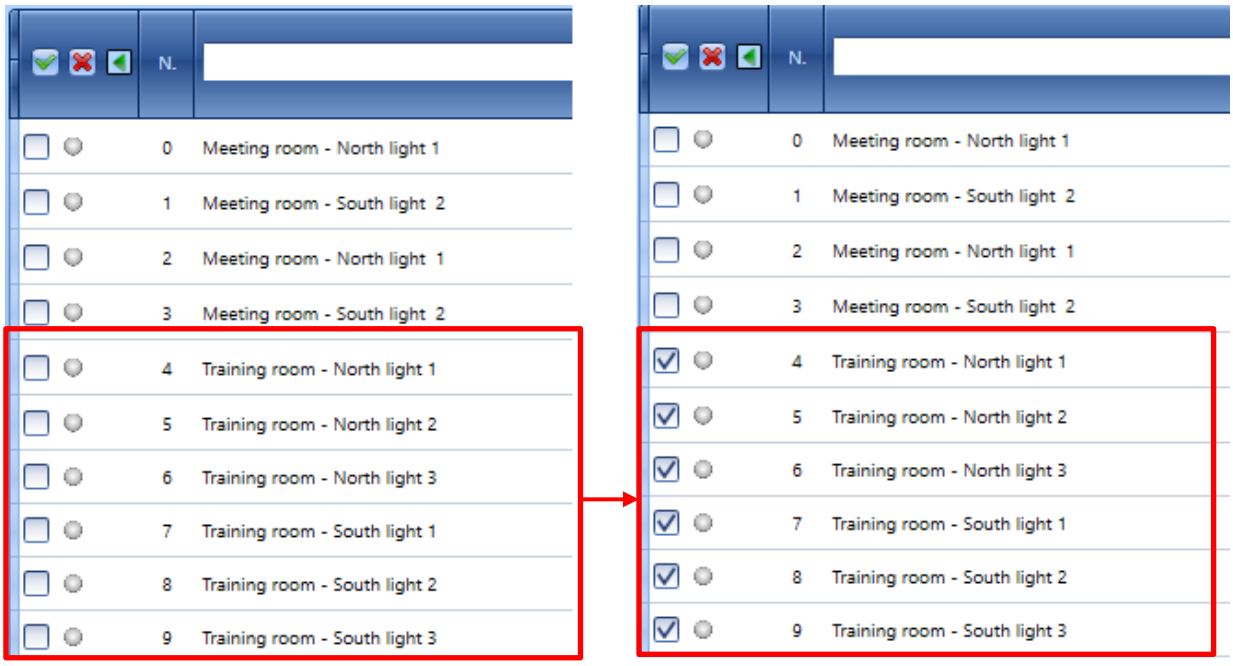
- To remove the ballast from a group (1 to 16), the red X icon must be checked



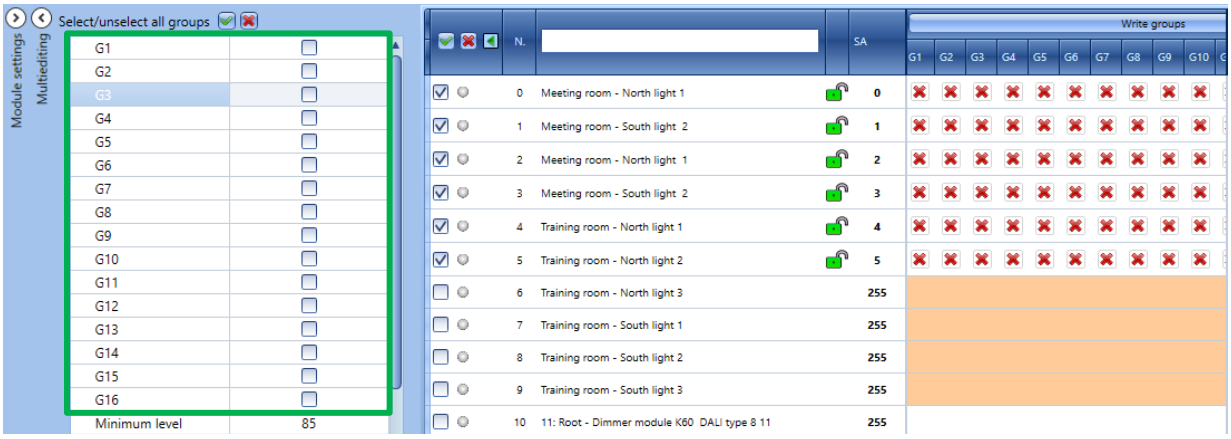
Any time a change is made for a ballast, the related row will be highlighted in orange. By clicking on the *Write groups* button, the group settings association are transferred to the DALI ballast, otherwise the changes will be lost.

12.11.1.2 Group settings – How to add ballast with Multiediting panel

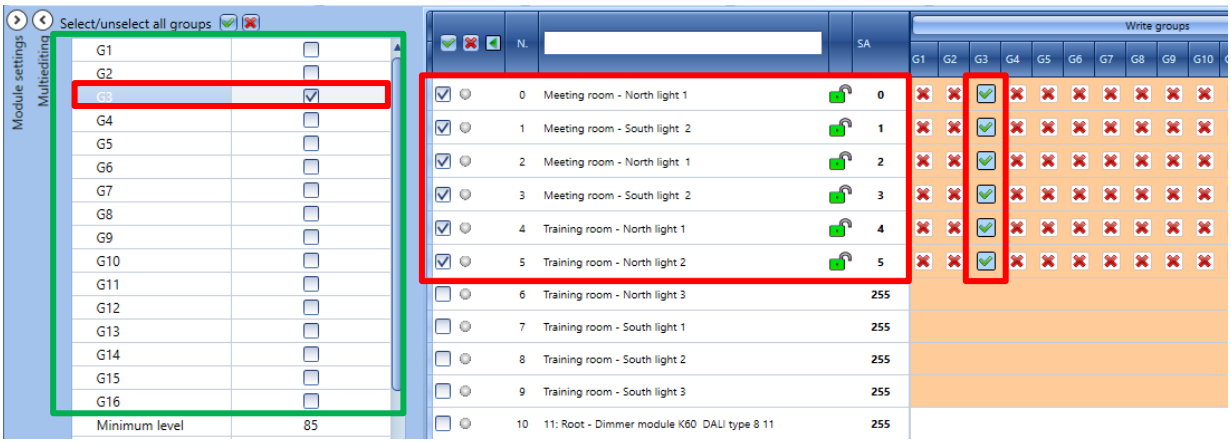
The assigning of DALI group to several DALI ballasts at the same time can be done via the fields available in the *Multiediting panel*. The user has to select the ballasts they want to manage by checking the check-box next to each device row, as shown in the example below:



Note: If the Multiediting panel should not be visible, click on the button to expand it: the fields shown in the green rectangle below are specific for the DALI groups, from 1 to 16.



When the user clicks on a checkbox, (G1…G16), that Group will be applied to all the DALI ballasts selected in the Device list. As shown in the example below, the DALI ballasts 1 to 7 have been added to DALI GROUP 3 (G3 check-box). The user can check the group assignment in the *Device list* section (see the *green V* icons).



The related rows will be highlighted in orange. By clicking on the *Write groups* button, the group settings association are transferred to the DALI ballasts.

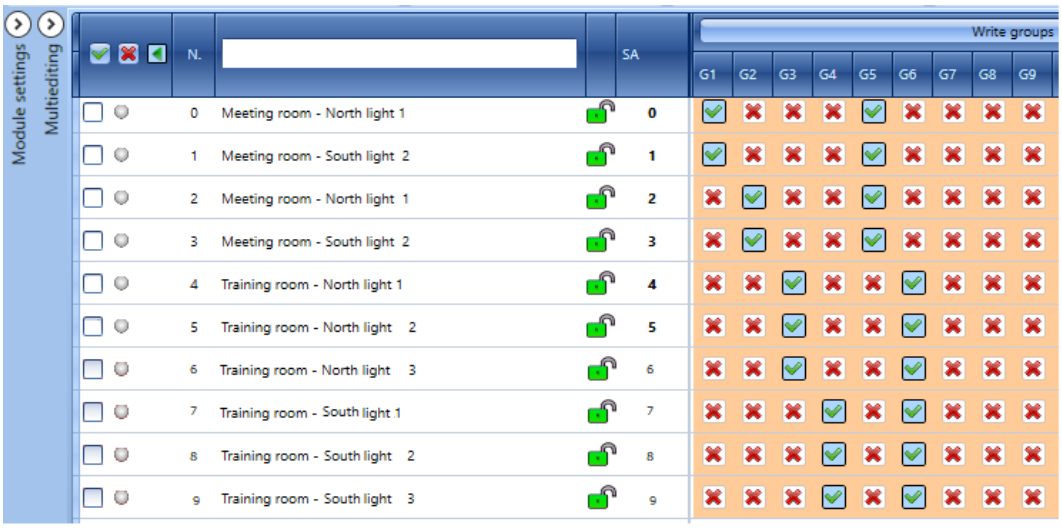
DALI groups example

As shown in the example below, The DALI ballasts are grouped by location, please remember that each ballast available in a DALI network can belong to **several DALI groups.**

As shown below, there are 10 DALI ballasts and they are grouped by their own *Location:*

The ballasts Meeting room – North light 1 and Meeting room - South light 2 are part of **Group1.**
The ballasts Meeting room – North light 1 and Meeting room – South light 2 are part of **Group2.** All of them belong to **Group5**.

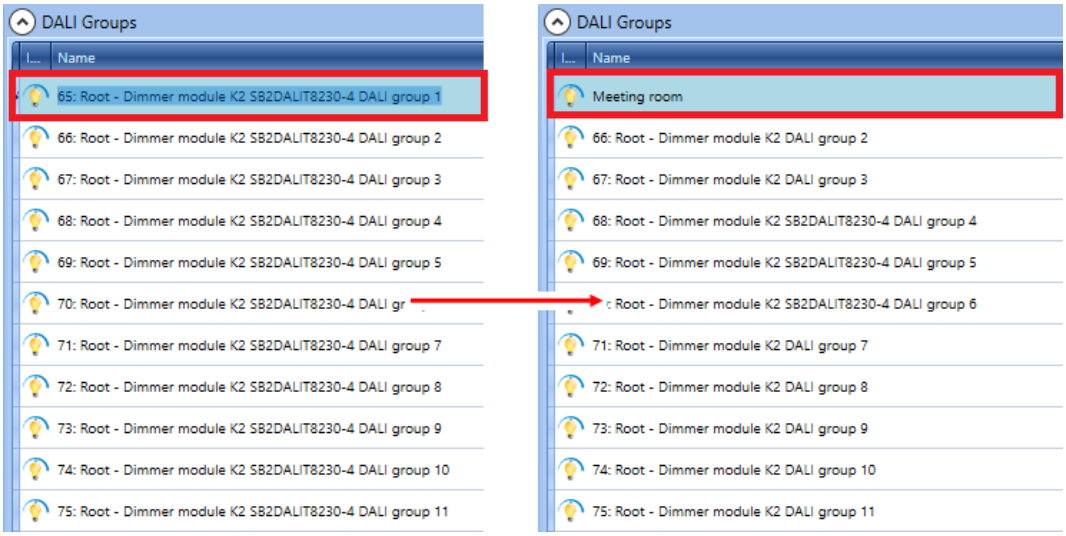
The ballasts Training room – North light 1 and Training room-North light 2 and Training room-North light 3 are part of **Group3.**
The ballasts Training room – South light 1 and Training room-South light 2 and Training room-South light 3 are part of **Group4 .** All of them belong to **Group6**.



12.11.2 How to rename the DALI groups

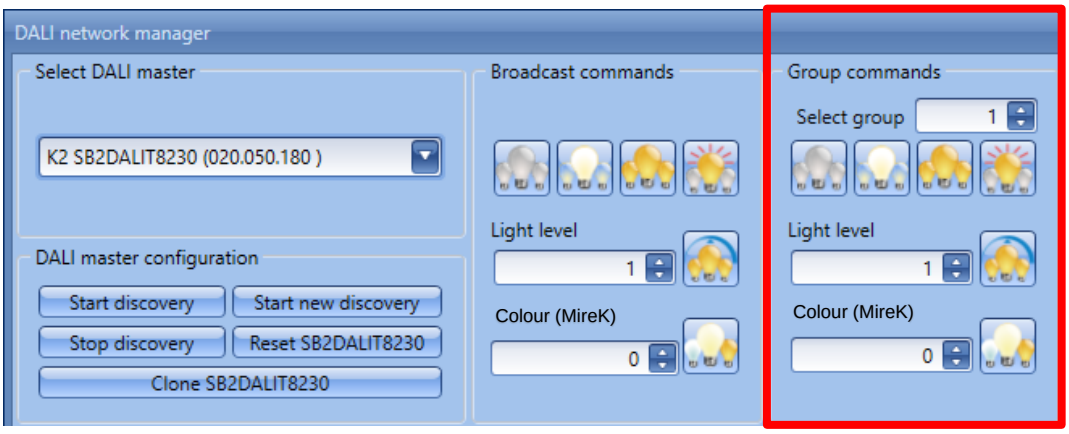
The *DALI Groups* panel allows the user to manually change the default label names of the DALI groups.

By clicking on the DALI Groups button, the *DALI groups* panel will be shown: the user can change the default label by entering the new name on each row, as shown in the example below, where the DALI group 1 has been renamed as *Meeting room*:



12.12 How to test a specific DALI group

For test purposes, the groups can be tested individually by Group commands: the user can send a specific command that will be applied only to the ballasts that belong to it.



To test the ballasts that belong to a specific DALI group, the user has to enter the group number (from 1 to 16) in the *Select group* text-box and press on one of the commands available.

The behaviour of the available commands is shown in the table below:

Group command	Behaviour
Switch Off	This command switches all the lights OFF
All MIN – Set all at min	This command switches all the lights ON to the minimum level
All MAX – Set all at max	This command switches all the lights ON to the maximum level
Blink	This command initiates the blinking of all the lights
1 Set level	This command dims all the lights up to the values set in the related field
0 Set colour	This command sets all the lights to the temperature colour according to the value set in the related field. *Note: This is available only for the DALI ballast type 8 devices*

12.13 Ballast settings

In the *Device list* section of the *DALI network manager* window the user can set ballast parameters to obtain the required intensity level of the DALI ballasts according to the *Smart light* functions where the ballasts are used. See *How to configure a Smart light function* in this manual.

As with the Groups settings procedures, there are two ways to manage the DALI ballast parameter settings: one by one or multiple DALI ballasts at the same time.

The descriptions of all the parameters are shown in the table below:

Field name	Description
Discovery status	This shows if the module is a new module or an existing module
Rnd	This contains the Random address assigned to the ballast
New SA	This contains the new Short Address given to the ballast
Old SA	This contains the old Short Address given to the ballast (if available)
Counter error	This field contains any error that occurred for the ballast
Type	This contains the description of the ballast type
Minimum physical level	This contains the minimum light level the ballast is able to manage
Minimum level	**Changes the minimum dimming value. The brightness value cannot go below this value**
Maximum level	**Changes the maximum dimming value. The brightness value cannot exceed this value**
Power on level	**The Power on level corresponds to the light value that the DALI ballast adopts when the lighting system is powered up**
System failure level	**The System Failure Level corresponds to the light value that the DALI ballast adopts when DALI bus fault occurs or when SB2DALIT8230 module is broken or not powered**
Fade time	**The fade time is the total time a DALI ballast takes to change brightness intensity from one level to another level. It is related to set % commands, i.e. set a dimming percentage, set a scenario.** *See details on the next page*
Fade rate	**The fade rate represents the speed of the dimming: it is related to manual dimming action.** *See details on the next page*
Min ballast CT	This contains the minimum colour value the ballast is able to manage
Max ballast CT	This contains the maximum colour value the ballast is able to manage
Min CT	**Sets the minimum colour value. The colour temperature value cannot go below this value**
Max CT	**Sets the maximum colour value. The colour temperature value cannot go above this value**
Power on level CT	**The Power on level corresponds to the colour temperature value that the DALI ballast adopts when the lighting system is powered up**
System failure CT	**The System Failure Level corresponds to the light colour value that the DALI ballast adopts when DALI bus fault occurs or when SB2DALIT8230 module is broken or not powered**

The fields **marked in bold** are editable, the others are read-only parameters.

N.B. The available parameters vary according to the type of DALI ballast device.
(the **CT fields** are shown ONLY for the DALI ballasts **Type 8**)

12.13.1 Tuning the fade time / fade rate parameters

The **fade time** and **fade rate** parameters are strictly related to the DALI ballast type. Each DALI ballast is delivered with its own factory settings and these values may differ due to the manufacturer. In the *DALI network manager* window, the user can adjust for each ballast these parameters in the proper fields, according to the project requirements.

To get the desired response time, all the DALI ballasts that belong to a *Smart light* function should have the same technical specifications and the parameters have to be set equal for all of them. If this is not possible, *i.e.* the DALI ballasts have different technical specifications, those parameters can be tuned by the user till the desired response time is achieved.

Fade time (min 0 - max 15)
The fade time parameter is the total time a DALI ballast takes to change light intensity from one level to another level. **The fade time is related to commands**, such as set percentage command, set a scenario (S2, S3, S4, S5), etc...
For each DALI ballast, the user can set the *fade time* value from a minimum of 0 to a maximum of 15: the lower is the value the faster the light intensity is changed.

Fade rate (min 1 - max 15)
The fade rate parameter is the speed of the manual dimming: For each DALI ballast, the user can set the *fade rate* value from a minimum of 1 to a maximum of 15: the lower is the value the faster is the speed of the dimming.

12.13.1.1 How to obtain the dimming time depending on the fade rate value

The formula shown below can be used to calculate the dimming time in seconds, according to the set fade rate value:

[(Fade rate + 1) * 10ms] * [Maximum level - Minimum level]

The maximum level, minimum level and fade rate parameters have to be set for each DALI ballast in the *DALI network manager* window.

Note: *Please always refer to the technical documentation of every kind of DALI ballast used in the project.*

The examples below show how to calculate the dimming time takes to the ballast to dim a light from the minimum light level to the maximum light level by a manual dim action.

Example1
In the example shown below, a DALI ballast is programmed with minimum level = 111 (on the logarithmic scale, which corresponds to 10%) and maximum level = 254 (on the logarithmic scale, which corresponds to 100%). The fade rate is set = 7.

	N.			SA	Minimum...	Minimum...	Maximum...	Power on...	System fa...	Fade time	Fade rate
	0	1: Root - Dimmer...		0	111	111	254	254	254	0	7

The manual dimming time from 10 to 100% (and vice versa) in seconds is:

[(7 + 1) * 10ms] * [254 - 111] = **11,44 seconds**

Example 2
In the example below, the fade rate is set = 15; the minimum level and maximum level parameters have not been changed.

N.		SA	Minimum...	Minimum...	Maximum...	Power on...	System fa...	Fade time	Fade rate
0	1: Root - Dimmer...	0	111	111	254	254	254	0	15

The manual dimming time from 10 to 100% (and vice versa) in seconds is:

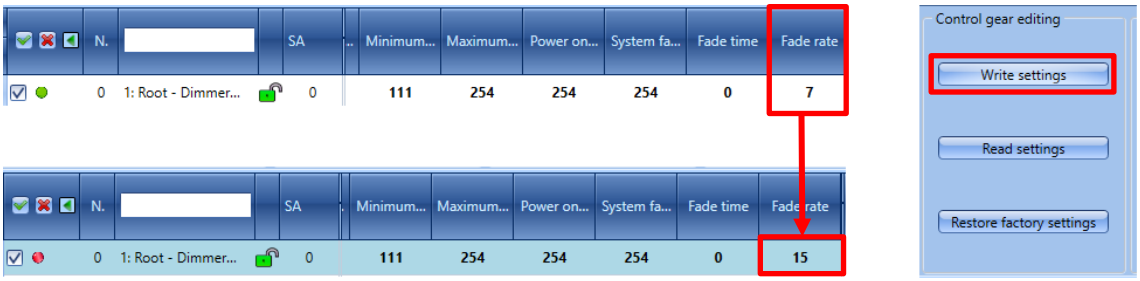
[(15 + 1) *10ms] * [254 - 111] = **22,88 seconds**

IMPORTANT NOTE: Smart light function has to be refreshed every fade rate changes

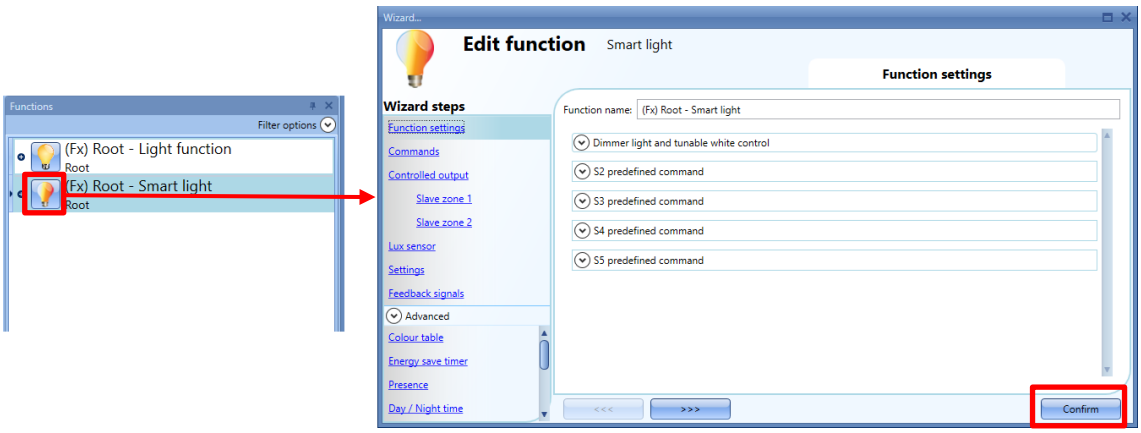
Every time the *fade rate* parameter is changed for at least a DALI ballast that belong to a *Smart light* function, the function has to be refreshed in order to manage the new value.

The procedure is reported as follows:

1) In the *DALI network manager* window, set the new value on the *Fade rate* field, then click on the *Write setting* button to store the changes onto DALI ballast:



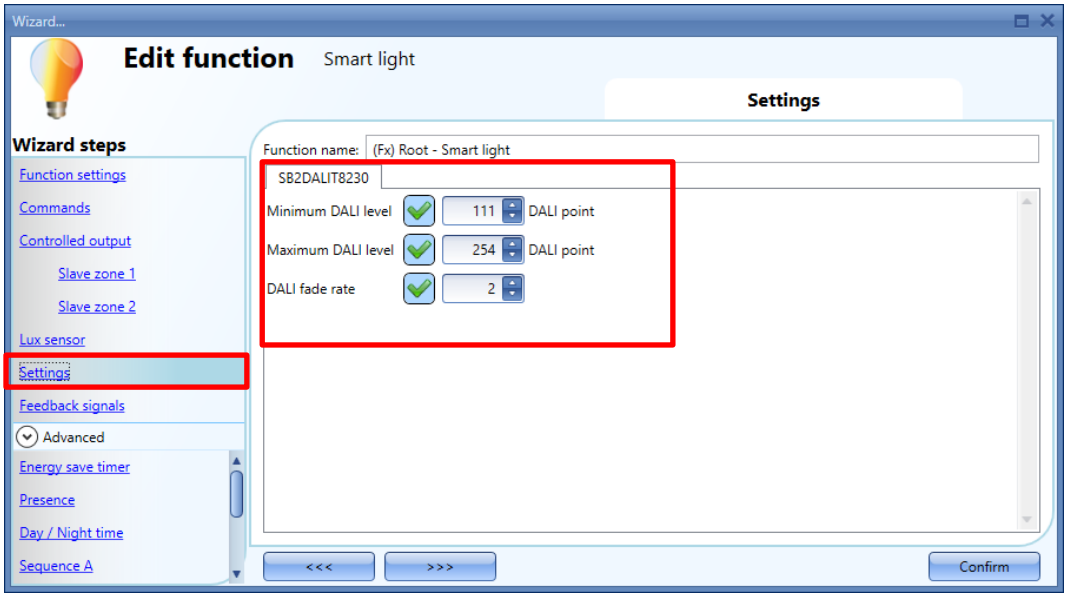
2) From the *Functions view* of the UWP 3.0 Tool, click on the relevant picture of the *Smart light* function (red rectangle in the picture below) where the DALI ballast(s) is used: the function window will be opened in edit mode;



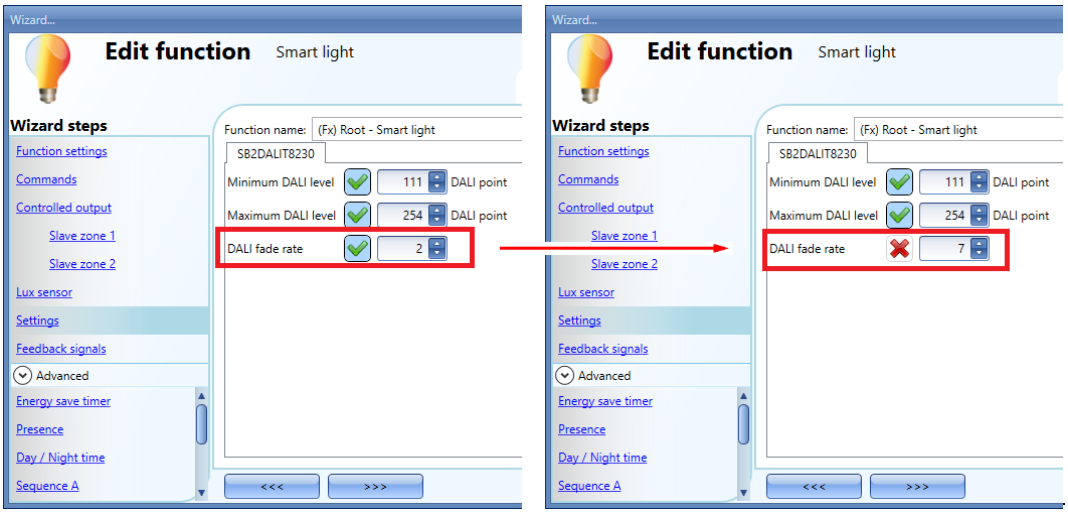
3. Click on the *Confirm* button to refresh the *Smart light* function, then click on the *Send to controller* button to download the configuration to the UWP 3.0 controller.

Using DALI ballasts with different fade rate values in the same *Smart light* function, the lowest fade rate value is applied to the Smart light function.

To set a specific fade rate value, the user can set it manually clicking on the Settings tab of a Smart light function, the following window will appear:

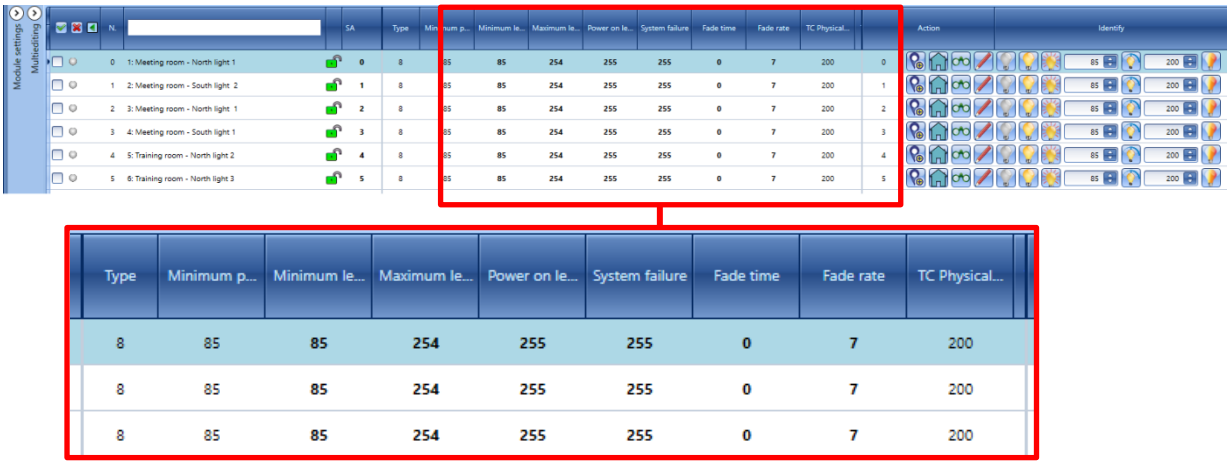


By clicking the green *V* icon, the user can set in the *DALI fade rate* field a specific value, as shown in the example below:



12.13.2 How to program the ballast individually

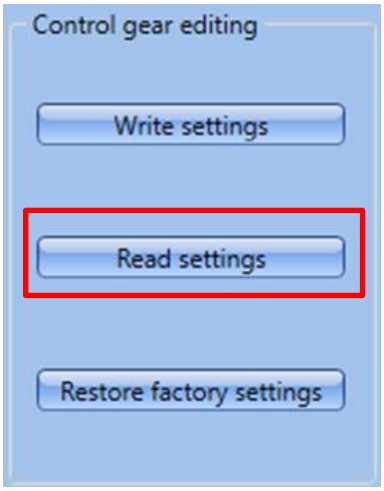
To change the current settings of a ballast, the user has to identify the ballast in the *Device list* section, then enter the new values in the required fields, as shown below:



Every time a change is made, the dot next to the DALI ballast will be coloured in red: this means the data is not synchronized between the settings in the UWP 3.0 tool and the data in the DALI ballast. See the example below:

Data needs to be synchronised

To synchronize the data, the user has to click on the *Read settings* button on the *DALI network manager* window: the updated data will be transferred to the ballasts.

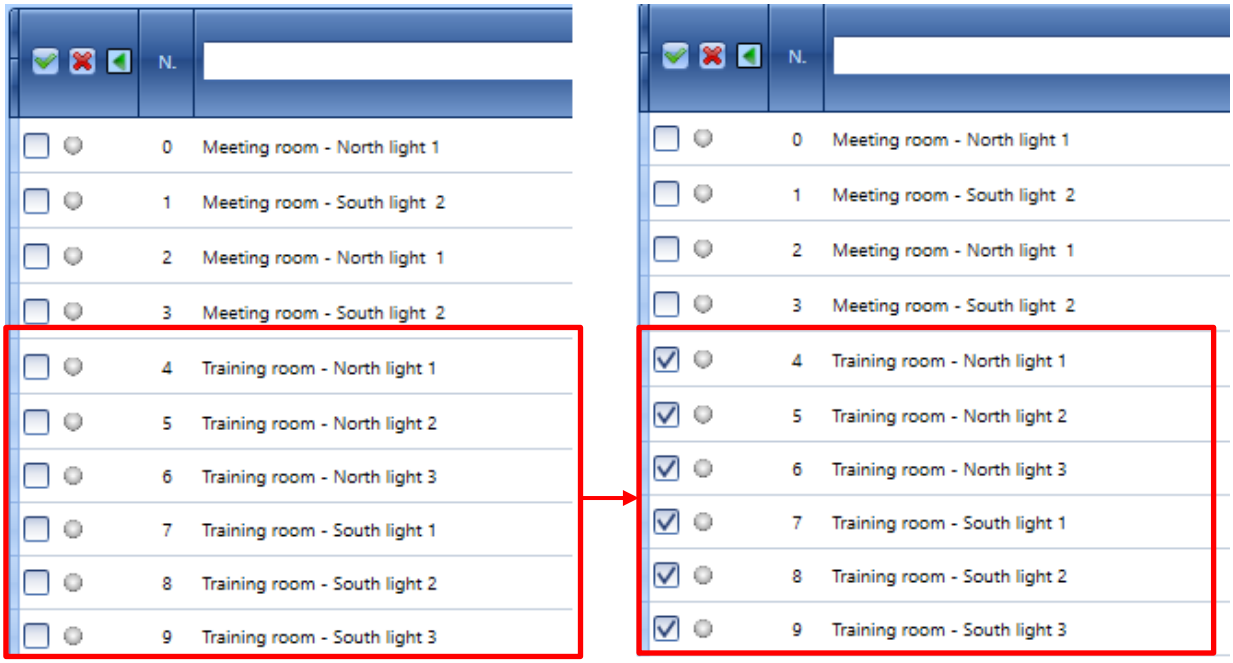


The dot next to the ballast will be coloured in green to indicate that the data has been synchronized properly, as shown below:

Data is synchronized

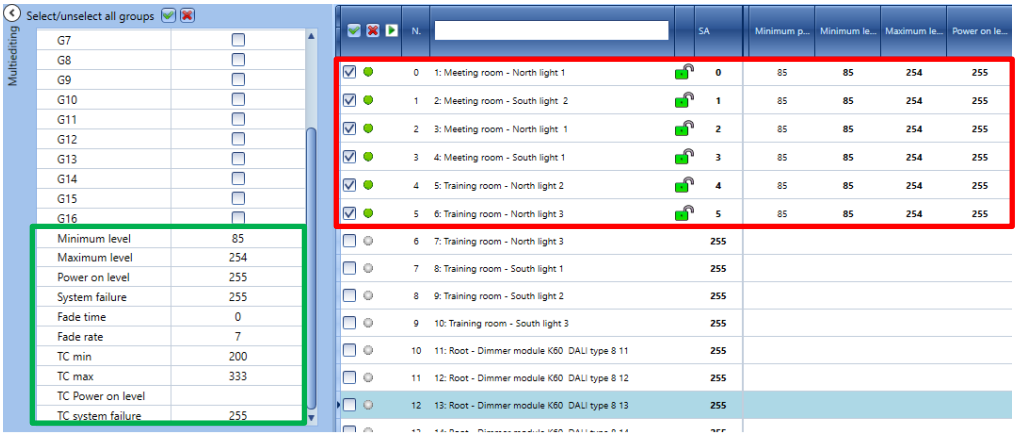
12.13.3 How to program the ballasts together via the Multiediting panel

The user can set and read parameters to multiple DALI ballasts at the same time by using the fields available in the Multiediting panel. The user has to select the ballasts they want to manage by checking the check-box next to each device row, as shown below:

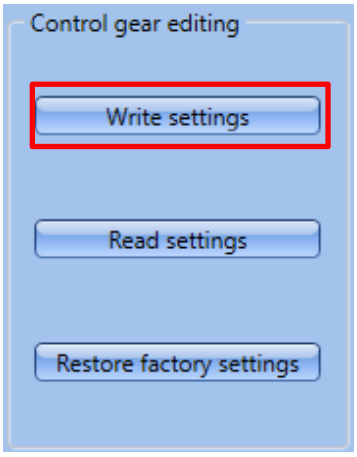


Note: If the Multiediting panel should not be visible, click on the button to expand it: the fields shown in the green rectangle below are specific for the settings of the DALI ballasts.

N.B. The available parameters vary according to the type of DALI ballast device.
*(the **CT fields** are shown ONLY for the DALI ballasts **type 8**).*



To send the new settings to the DALI ballasts, the user has to click on the *Write settings* button on the *DALI network manager* window: the updated data will be transferred to the DALI ballasts.

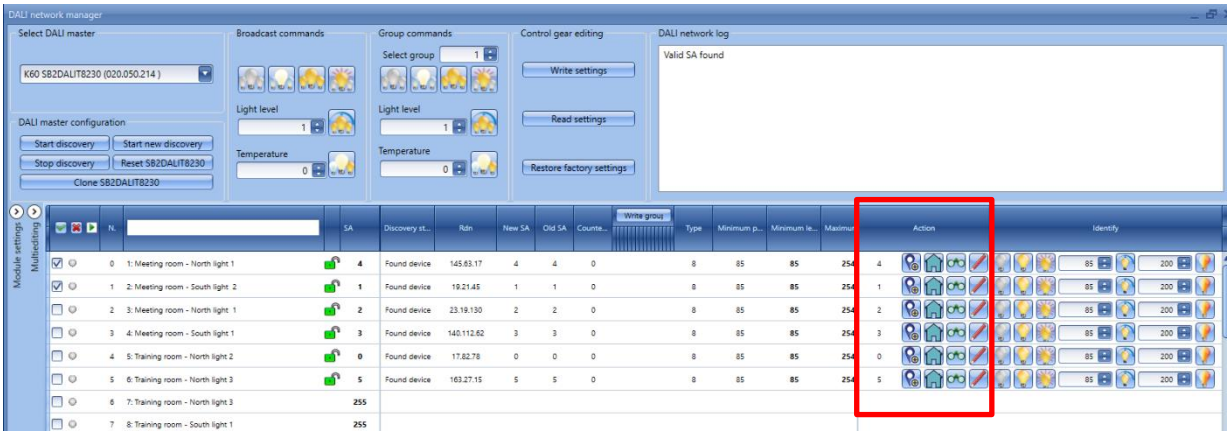


12.14 Advanced settings

By means of the *Advanced* settings, the user can perform the advanced features available on the *DALI network manager* window. Should an error occur during the addressing phase, the user can easily fix it without performing a new discovery. There are also additional features, in order to help the user to restore the entire commissioning project.

In the DALI network manager window of an SB2DALIT8230, for each DALI ballast that has been found, there are several advanced commands available. There are *single* and *multiple* advanced commands:

The single advanced commands for the ballast are shown in the *Action* section, as shown in the red rectangle below.



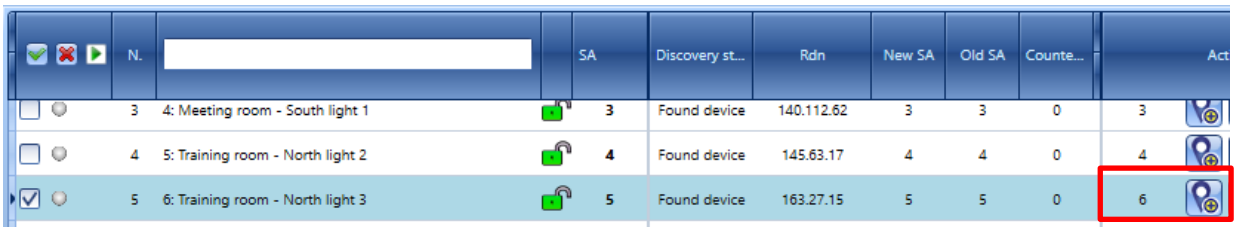
Description by column of the window marked in red:

Action	Description
0 Write SA	This sets a new SA (Short address) manually
Reset to factory reset	This resets the ballast device bringing it back to factory settings
Read data	This reads the data from the ballast
Write settings	This writes the new data to the ballast

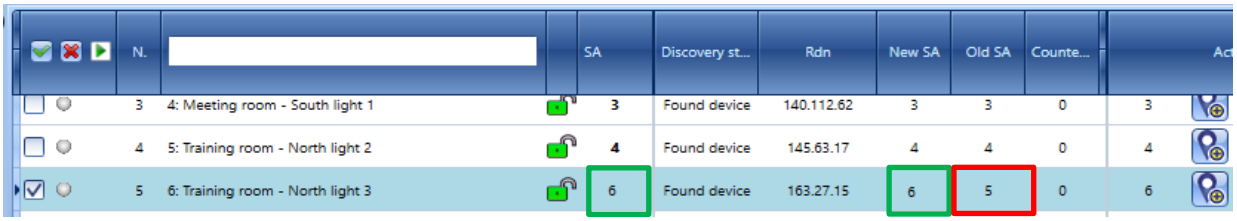
12.14.1 How to manually assign an address to a DALI ballast

The procedure shown below is valid whether the user has to manually change the SA (short address) of a DALI ballast or if they have to assign a new SA to an unconfigured device (*e.g.* to an out-of-box DALI ballast).

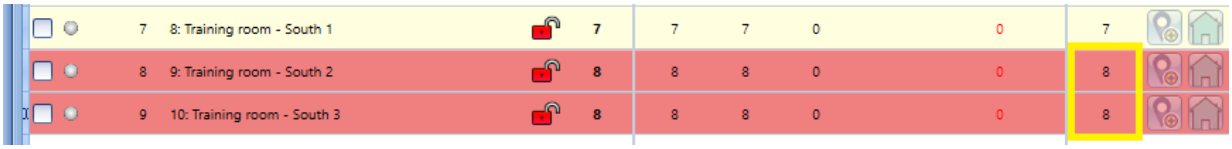
To change the SA of a DALI ballast, the user has to select the DALI ballast to be given a new SA. In the *Action* section of the *DALI list* window, the user has to enter the new value in the field next to the button, as shown in the red rectangle below:



To apply the change, click on the button. The new address will be set for the device immediately. The user can check the operation that has been carried out by looking at the two fields *New SA* and *Old SA*, as shown in the green box below:



Note: Should the user assigns the same address to two or more DALI ballasts, in the *Device list* the rows relevant to this duplicated address will be marked in red: the user must click on *Reset* to remove this error condition and must correctly re-address these ballasts.

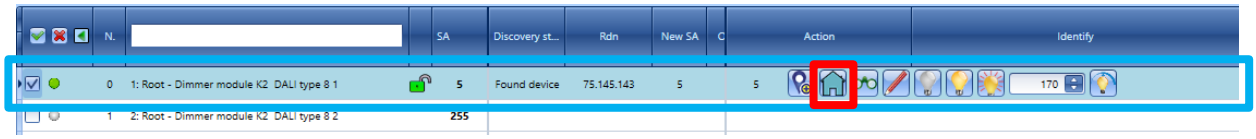


12.15 Restore ballasts to factory settings

In the *Device list* section, the user can restore the DALI ballasts to Factory settings: there are two ways to do this, one procedure to reset a ballast individually and another procedure to reset all the ballasts that are part of a DALI loop. This means that should this ballast be connected to a DALI bus, it will be recognized as a new one.

12.15.1 How to reset a DALI ballast

To bring the ballast back to the factory settings, the user has to select the DALI ballast in the *Device list*: the row will be highlighted in light-blue. In the *Action* section of the window, the user has to click the button, as shown in the red rectangle below:

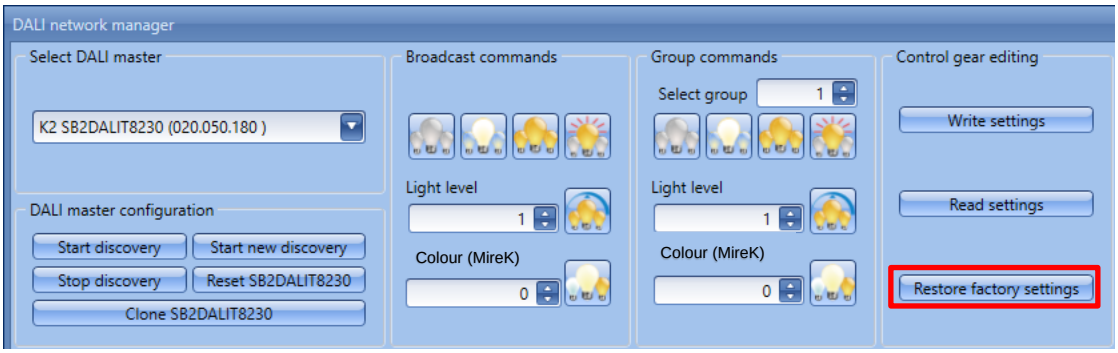


The ballast will be restored to the factory settings.

Note: The Rnd address will be changed with a new Rnd address

12.15.2 How to reset all the DALI ballasts of a DALI network

All the DALI ballasts that are part of a DALI network can be reset together: the user after having connected to the DALI network, has to click the *Restore factory settings* button in the *DALI network manager* window:



The ballast will be restored to Factory settings: this means all the settings saved in the device will be lost.

Next time the user performs a *Start new discovery* operation, only the DALI ballasts without an SA (Short address) will be listed in the *DALI list* at the bottom.

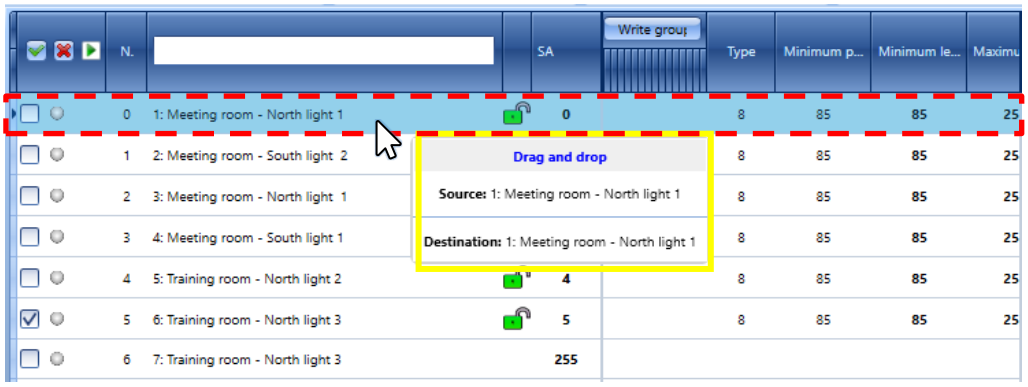
12.15.3 How to swap two ballasts

In the *Device list* section of the *DALI network manager* window, the DALI ballasts found by the Bus discovery are sorted by SA (Short address) as they are assigned randomly. Nevertheless, the user is able to change the order of the ballasts by changing the associations between the *Signals name* row and the DALI ballast settings.

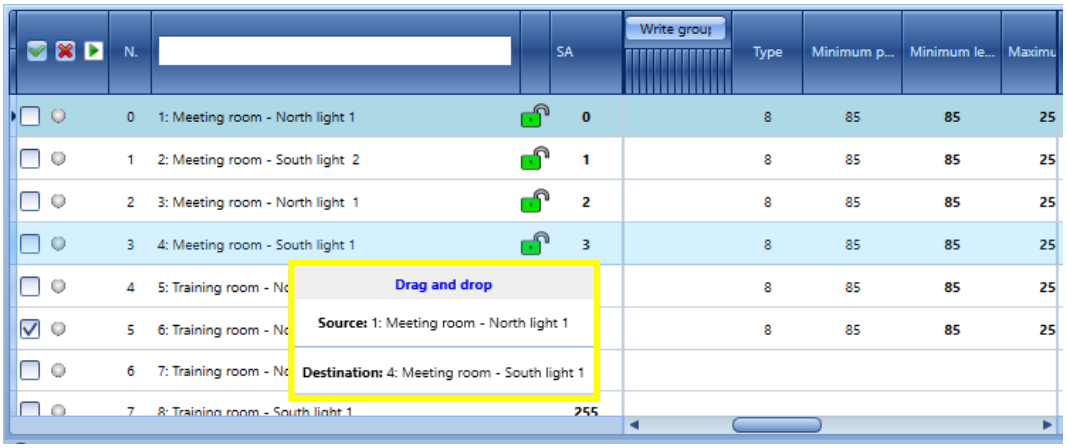
This functionality is useful when the user needs to rearrange the order of the rows after the DALI ballasts have been addressed. To move a Signal row to a new position, the user can swap two ballasts by using *drag & drop action,* according to the following steps:

3. Move the mouse pointer until it is directly over the DALI ballast (expressed in the related Signal row) you want to move;

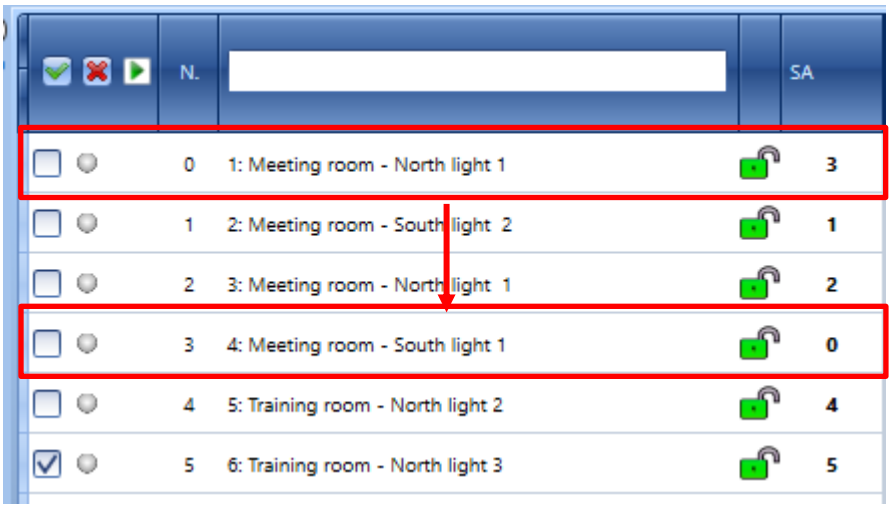
4. Press and hold down the left mouse button. When the user starts moving the module, a tool tip will appear (as shown in the yellow rectangle below):



5. With the left mouse button held down, drag the object to the desired destination row: in the ***Source*** field the name of the module to be moved is shown, while in the ***Destination*** field there is the name of the final position row;

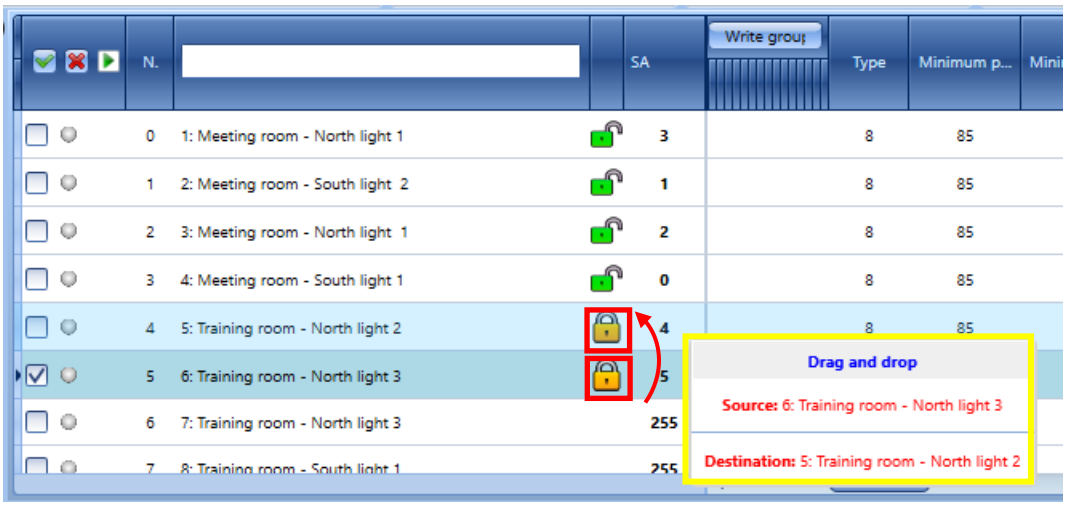


6. Drop the ballast by releasing the left mouse button: the DALI ballast with **SA 0** (*Meeting room – North light 1*) is now switched with the module that had before the **SA 3** (*Meeting room – South light 1)*, as shown below:



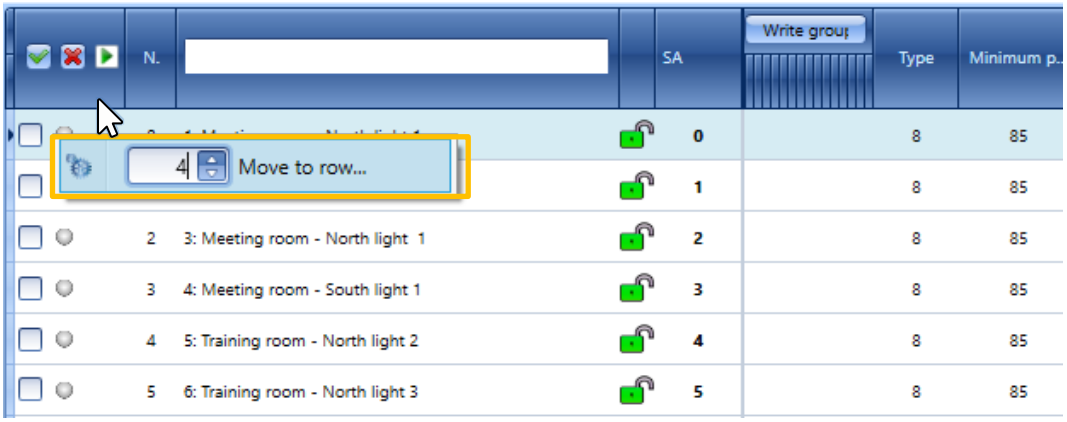
N.B. The *drag & drop can be done ONLY between unlocked Signal rows (the row has a* *icon)*

When a *drag & drop* action is performed on at least one locked signal row (the row has a icon) the ballasts shown by the tooltip are highlighted in red: in the example below, the ballast that has the short address 5 cannot be swapped with the ballast that has short address 4, since their position is locked.

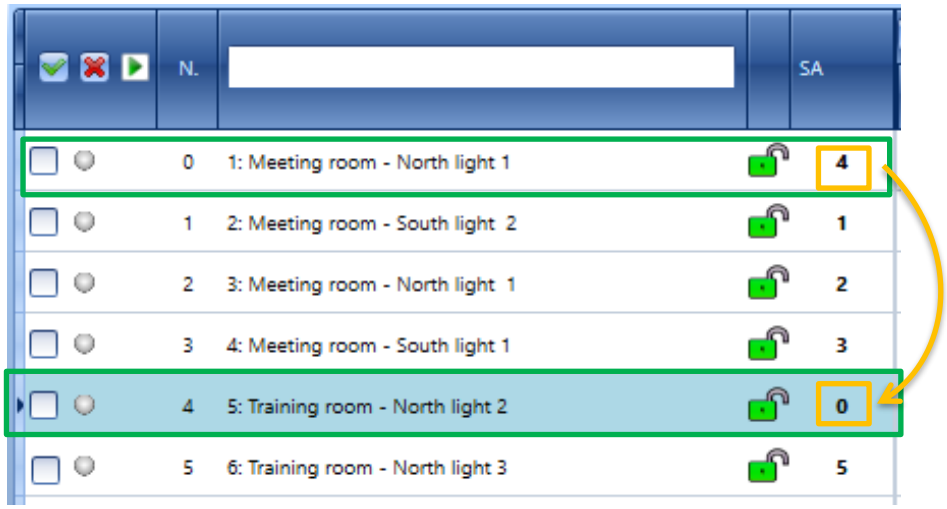


12.15.3.1 How to swap two ballasts by contextual menu

The same action can be performed by contextual menu. The user, by right-clicking on a signal row, opens a context menu, as shown in the orange rectangle in the picture below:

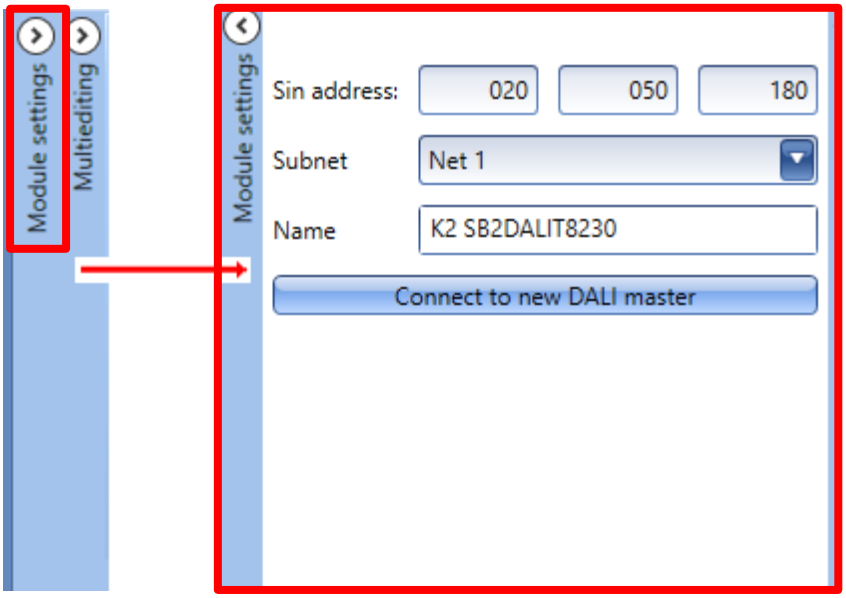


To swap two ballasts, enter the value of the destination row in the text box and press the *Move to row…* button. The selected row will be moved to the new row, as shown in the example below:



12.15.4 How to connect to a SB2DALIT8230 module from the DALI network manager window

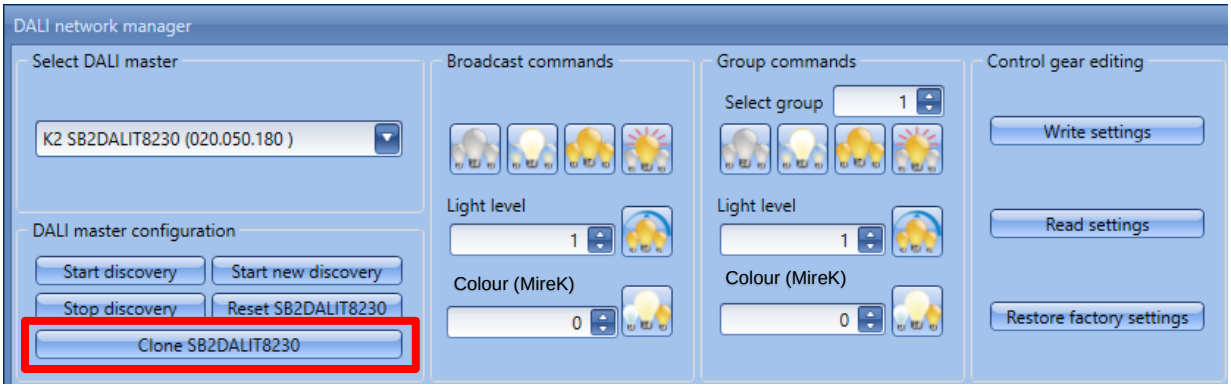
To connect to a new SB2DALIT8230 master DALI, directly the *DALI network manager* window, the user has to click on the *Module settings* button to expand the left panel: The *Module settings* menu will appear:



In the *Sin address* fields the user can insert the SIN address of the new SB2DALIT8230, then in the *Subnet* field set the Dupline network where the module is connected to. By clicking the *Connect to new module* button, the module will be added to the current project. The user can find the added module in the Available DALI masters.

12.15.5 How to restore the configuration of a new SB2DALIT8230 module

To copy a configuration that previously has been downloaded into a faulty SB2DALIT8230 module, select the new module in the upper left section of the DALI network manager window and click on the *Clone SB2DALIT8230* button, highlighted by the red rectangle in the picture below:

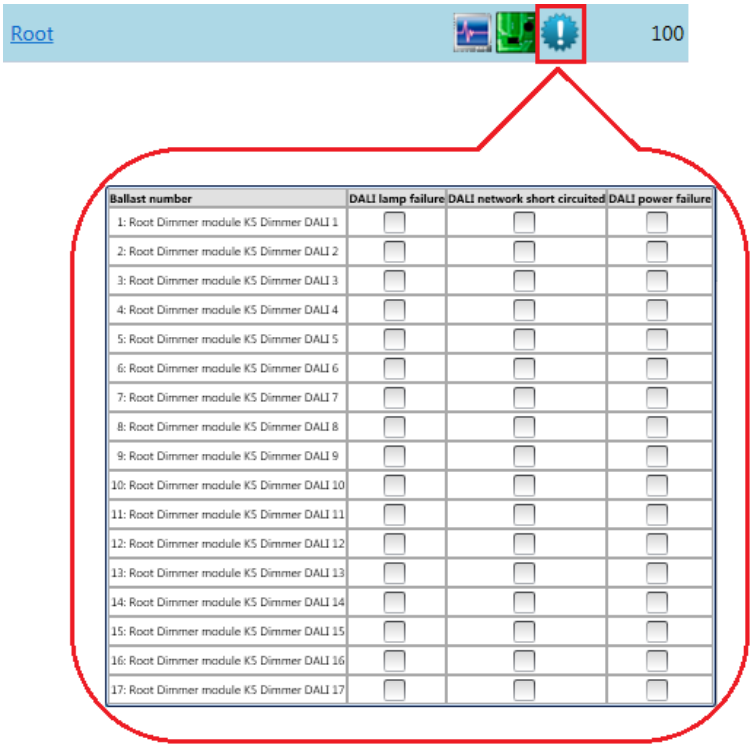


12.16 How to check the DALI bus status

Each SB2DALIT8230 has a dedicated diagnostic menu, accessible automatically when the *live signals* are enabled.

When *Live signals* are active, as below, the icons on the right appear and the one marked in red indicates the DALI bus status.

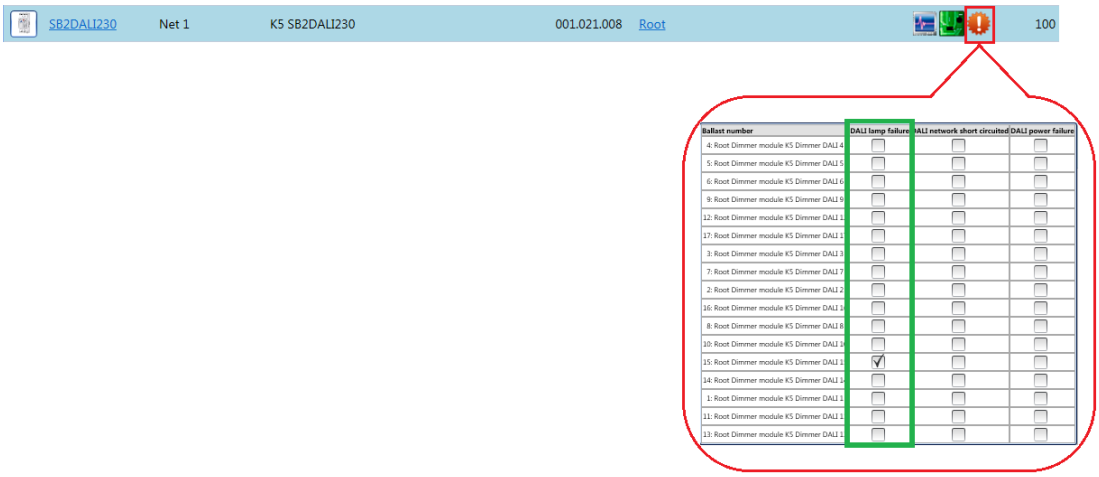
By means of this, the user can have real-time monitoring of the health status of the DALI bus.
If everything is working properly, the icon is blue and when the mouse hovers over it a tool tip appears, flagging no errors.



If a failure is present on the DALI bus, it will be shown by the icon turning orange and when the mouse hovers over the icon, a tool tip appears, flagging the relevant error. There is one line for each ballast.

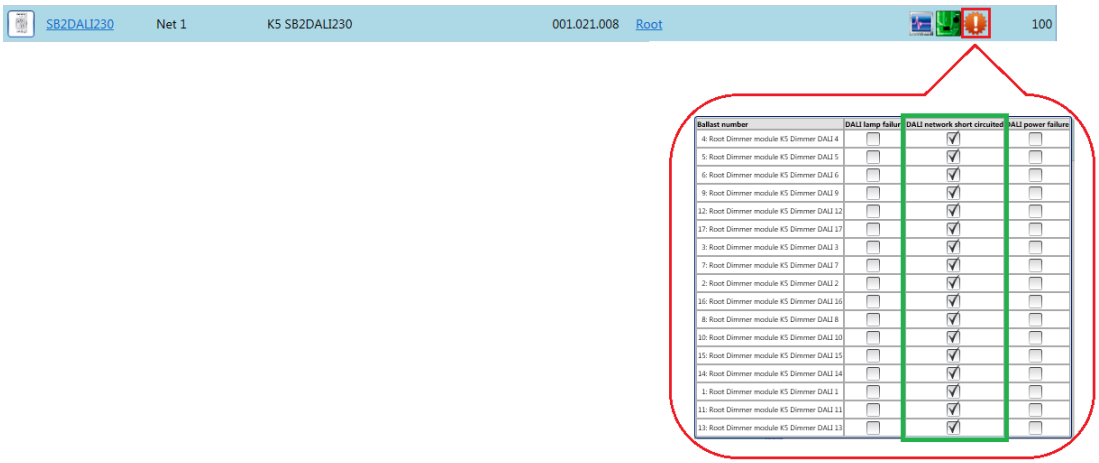
12.16.1 DALI lamp failure

If a lamp is damaged or missing, the icon relevant to the DALI bus status will turn orange and the relevant flag will be ticked (see the green box).



12.16.2 DALI network short-circuited

If a DALI network short-circuit is present, the icon relevant to the DALI bus status will turn orange and the relevant flag will be ticked (see the green box).

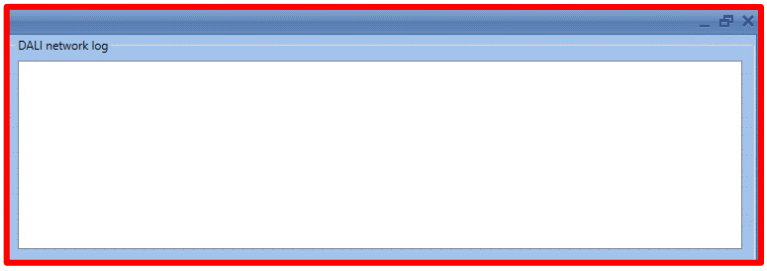


12.16.3 DALI power failure

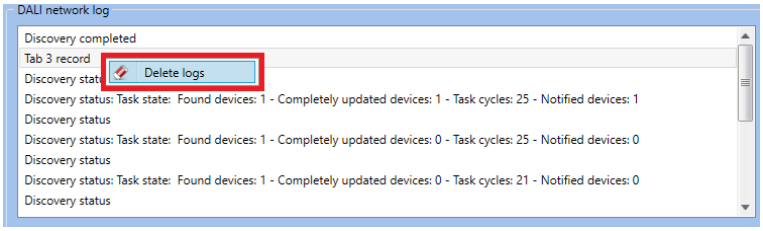
If a DALI ballast has a power failure, the icon relevant to the DALI bus status will turn orange and the relevant flag will be ticked.

12.16.4 How to see diagnostic info in the DALI network manager window

The *DALI network manager* window shows in the *DALI network log* box (red rectangle) all the events related to the DALI bus activity.



The user, by right-clicking on the *DALI network log* box, can delete the history list clicking on the *Delete logs* button, as shown in the red rectangle below:



13 How to configure a DALI network with a SB2DALI230 module

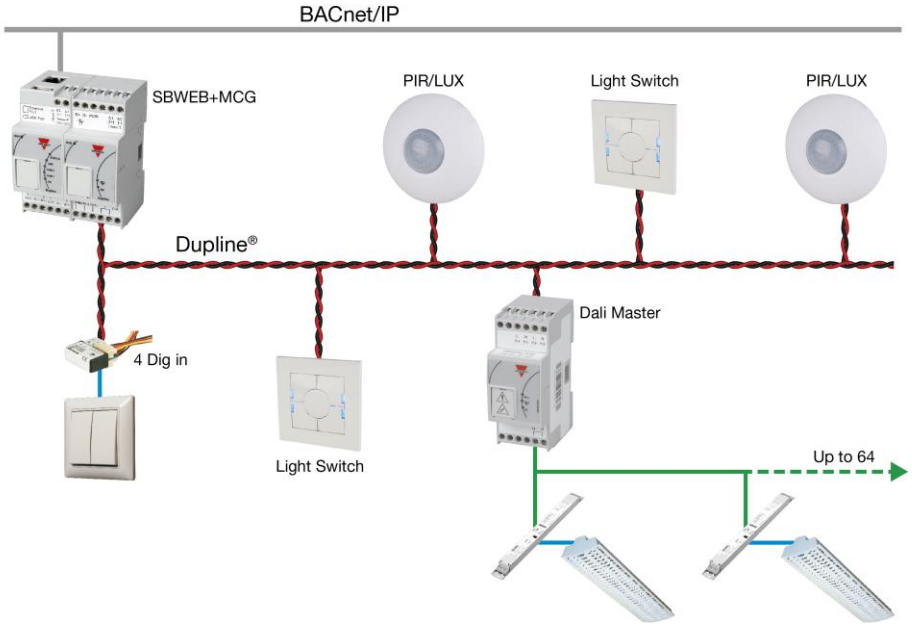
13.1 How to interface Dupline to DALI

The DALI bus is the standard for interfacing to lighting actuators such as ballasts for dimming of fluorescent lighting, LED drivers, RGB lighting etc.

The SB2DALI230 module is a gateway between the Dupline bus and the DALI (Digital Addressable Lighting Interface) protocol and it operates as DALI master and has a built-in DALI power supply. As shown below, the SB2DALI230 connects directly to the Dupline bus, hence it can be installed de-centrally.

Up to 7 SB2DALI230 modules can be connected to one Dupline bus, and each SB2DALI230 can manage up to 64 DALI actuators. The SB2DALI230 supports DALI actuators, but not DALI sensors and input modules. This is because the concept is to use the bus-powered Dupline devices for this purpose, as shown below.

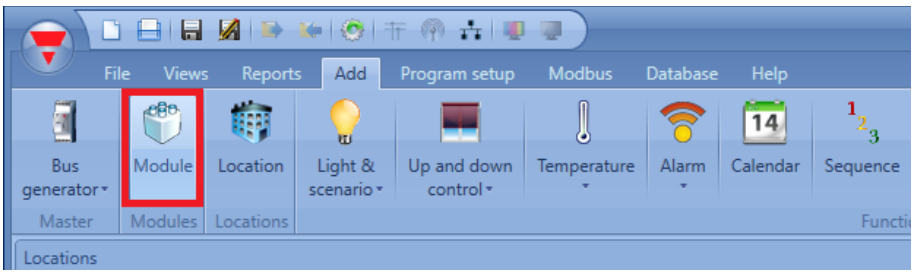
In the UWP 3.0 tool, DALI dimmers are managed in exactly the same way as Dupline dimmers. The grouping of the lighting outputs is also managed by the UWP 3.0 tool, hence all the configuration can be done in the same programming environment by using a single tool.



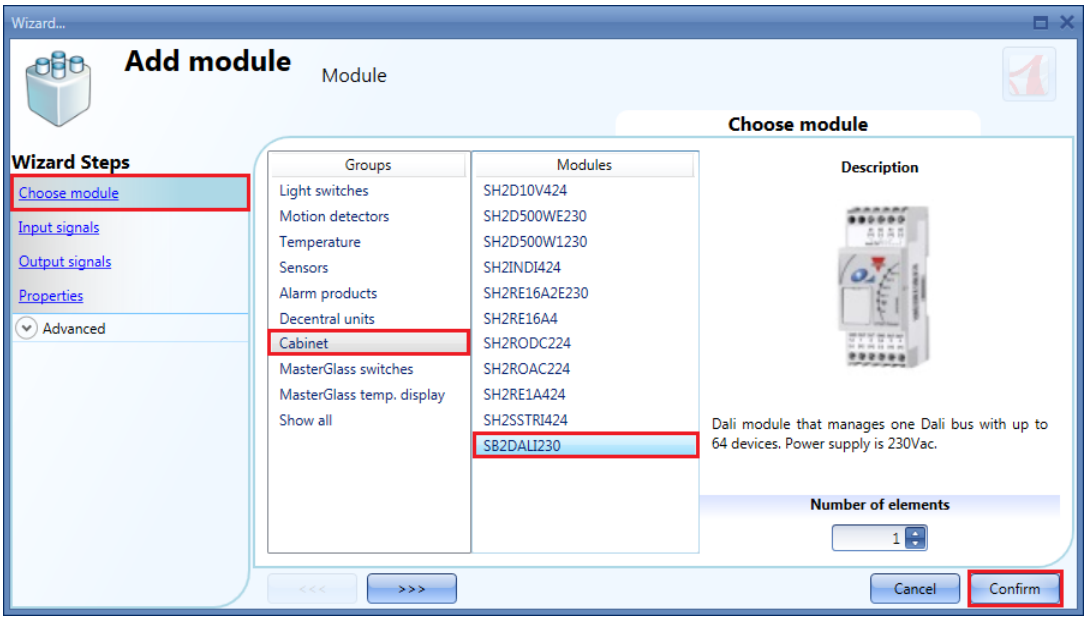
13.2 How to add the DALI master SB2DALI230

The SB2DALI230 is a gateway between the Dupline bus and the DALI (Digital Addressable Lighting Interface) protocol and is a DALI master. It manages one DALI bus.

To add it into the UWP 3.0 tool, click on *Add module, Cabinet, SB2DALI230.*



When the *Confirm* button is pressed, the module is inserted.



13.3 How many SB2DALI230 can be managed

Each SH2MCG24 can control up to 7 SB2DALI230s and each DALI ballast is managed as a dimmer output from a Dupline point of view. Dimmer outputs from different Dupline networks **cannot** be mixed in one dimmer function.

Each smart-Dupline network can manage up to 256 dimmer functions:

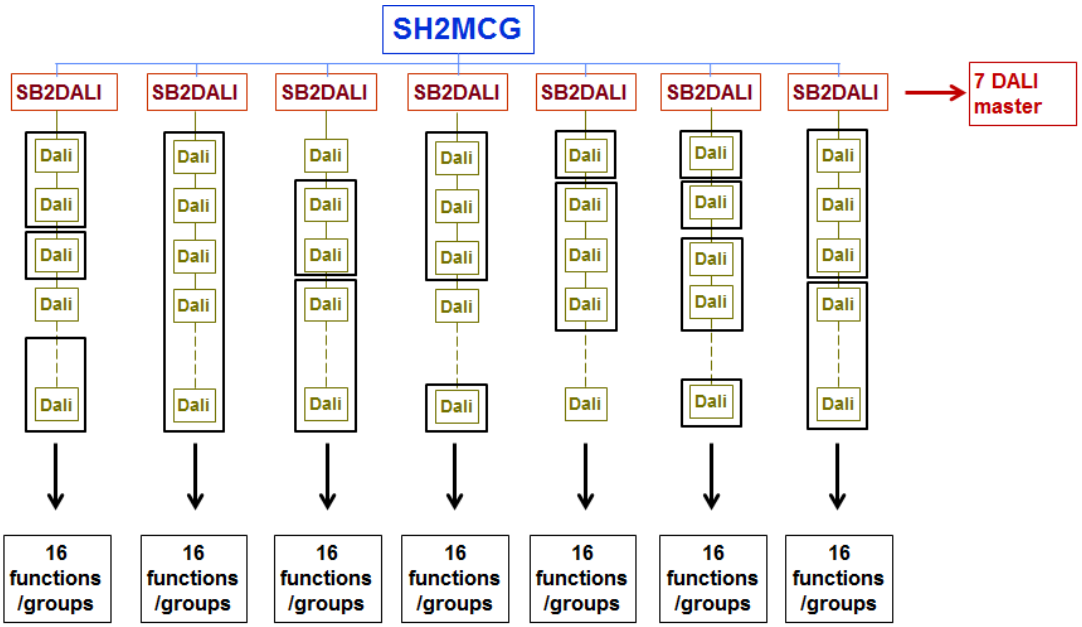
- 128 functions with 230V and 1-10V dimmers
- 128 functions with DALI devices

13.4 Functions and groups

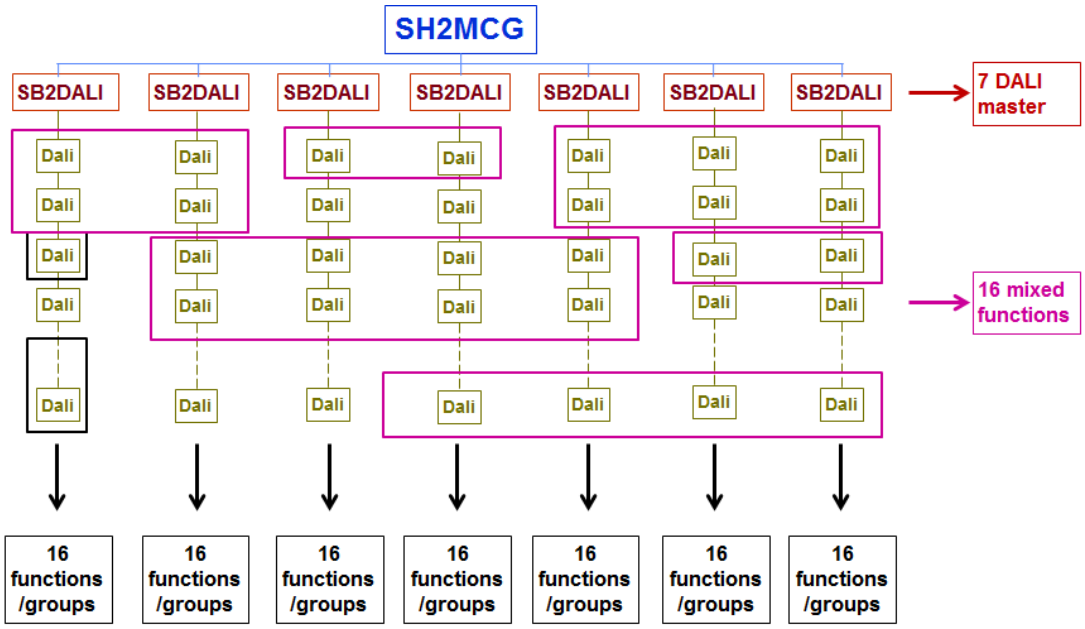
In the UWP 3.0 Tool the DALI groups are managed via the Dimmable or Constant light functions. The maximum number of functions (groups) for each SB2DALI230 is 16, and in each function the user can add up to 100 dimmer outputs (ballasts). The SB2DALI230 (DALI Master) assigns an address (0 to 63) to each connected ballast. This address may be changed at any time in the UWP 3.0 tool (*DALI advanced setup* menu).

A Dimmable or Constant light function uses an average of 3 input channels and 2 output channels (Input 3/120 - Output 2/112): this should be considered in the design of a DALI network integrated into the Dupline bus.

C. Up to 16 dimmer functions for each DALI master SB2DALI230



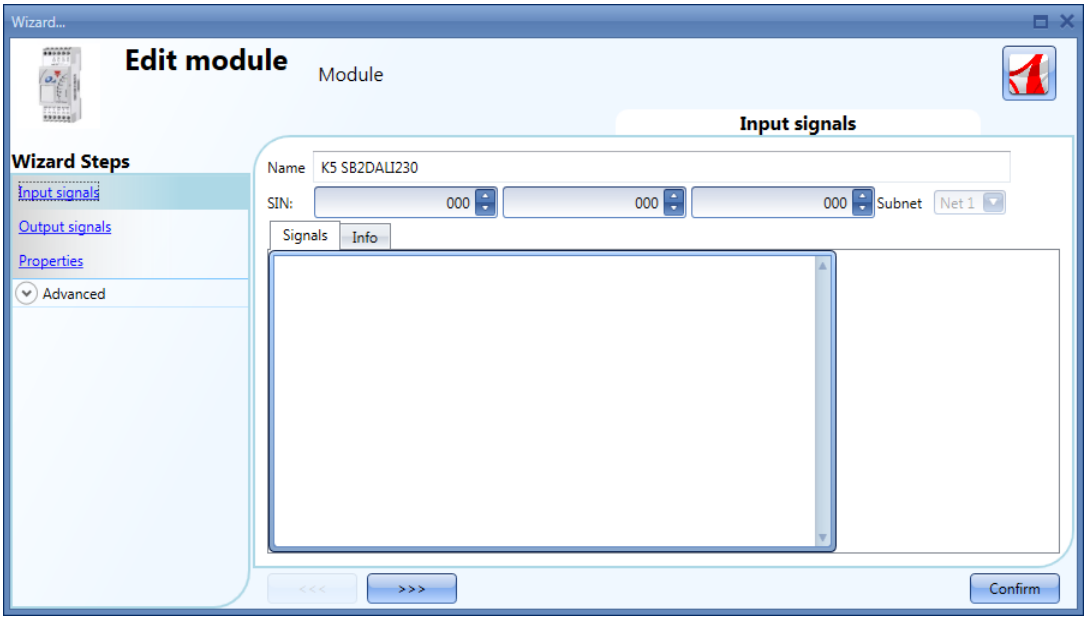
D. Up to 16 dimmer functions with DALI ballasts belonging to different DALI masters



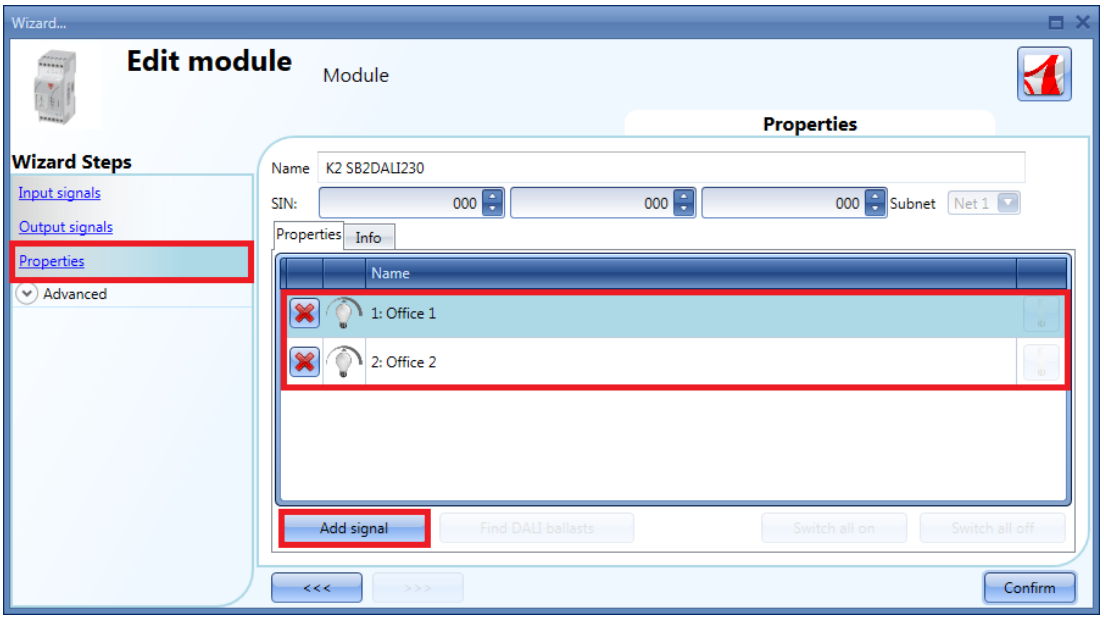
13.5 How to manually add a DALI ballast

To configure an SB2DALI230 module, click on the relevant picture once it is added to the project:

The configuration wizard will appear:

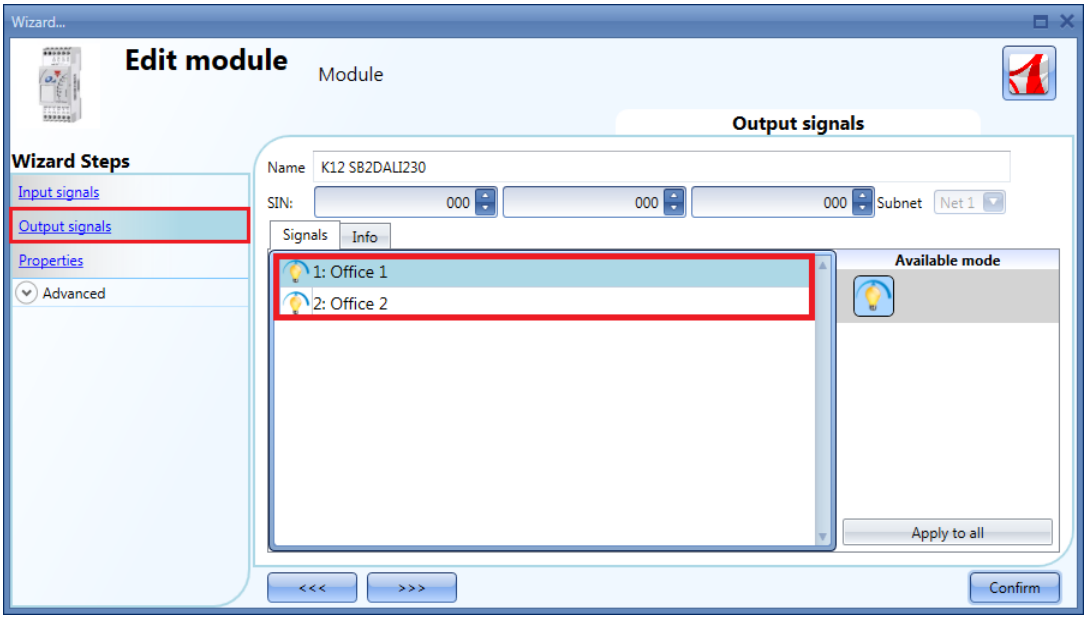


No input signals are present in this window.



In the *Properties* window the user can add up to 64 DALI outputs for each SB2DALI230.

Each time the *Add signal* button is pressed, a DALI dimmer output will be added. To save the configuration, press the *Confirm* button. If at least one SB2DALI230 is connected to the system, all the buttons next to *Add signal* will be active.

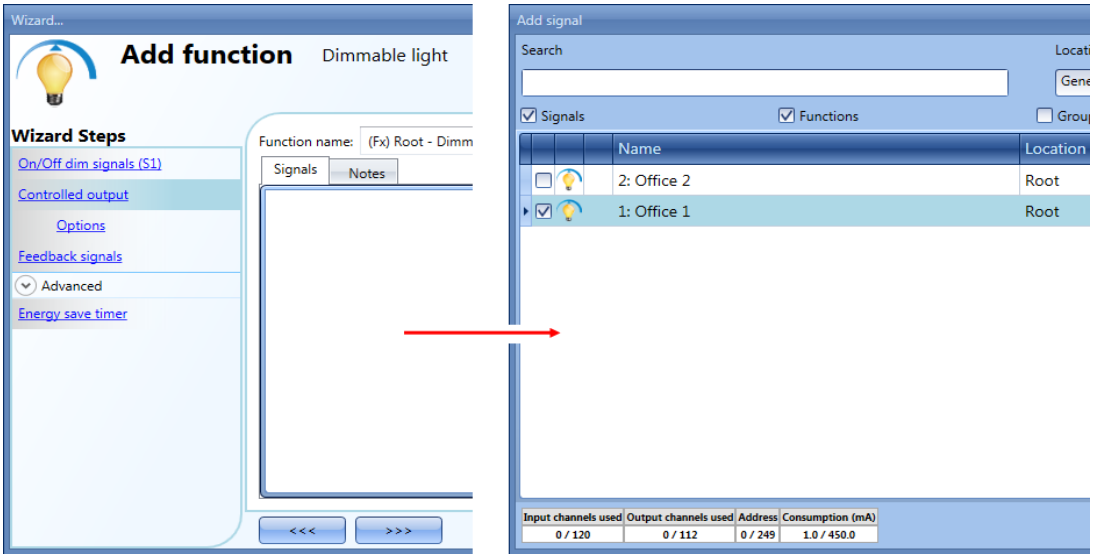


In the *Output Signals* window, all the DALI dimmers added in the *Properties* window will be shown.

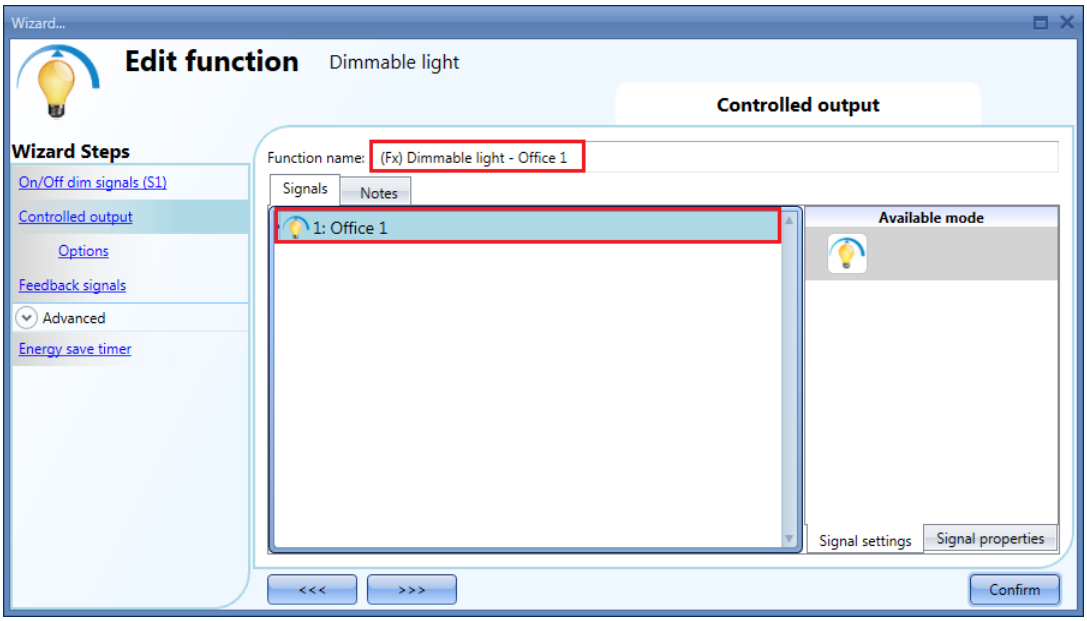
This configuration can be useful when the user needs to prepare an “off line” draft of the installation. Then, they will be able to load the project into a UWP 3.0 connected to a DALI bus and configure the ballasts.

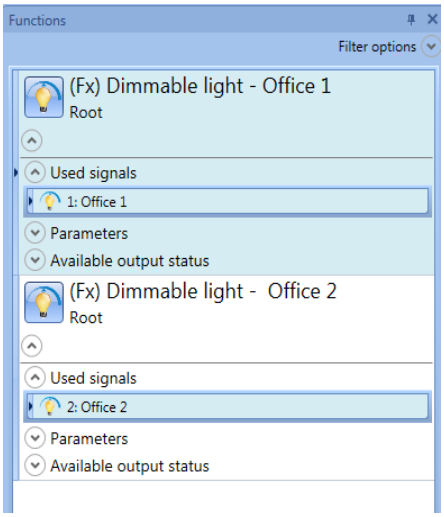
13.6 How to configure a dimmer function using DALI outputs

As shown in the example below, the user has to add the DALI outputs (ballast) in the *Controlled output* field as a standard dimmer output.



To select the output signal that is controlled by the dimmable light function, click on *Controlled output* and then double click on the *Signals* window. Once the output window is opened, select the relevant outputs from the list: up to 100 dimmer outputs can be added.





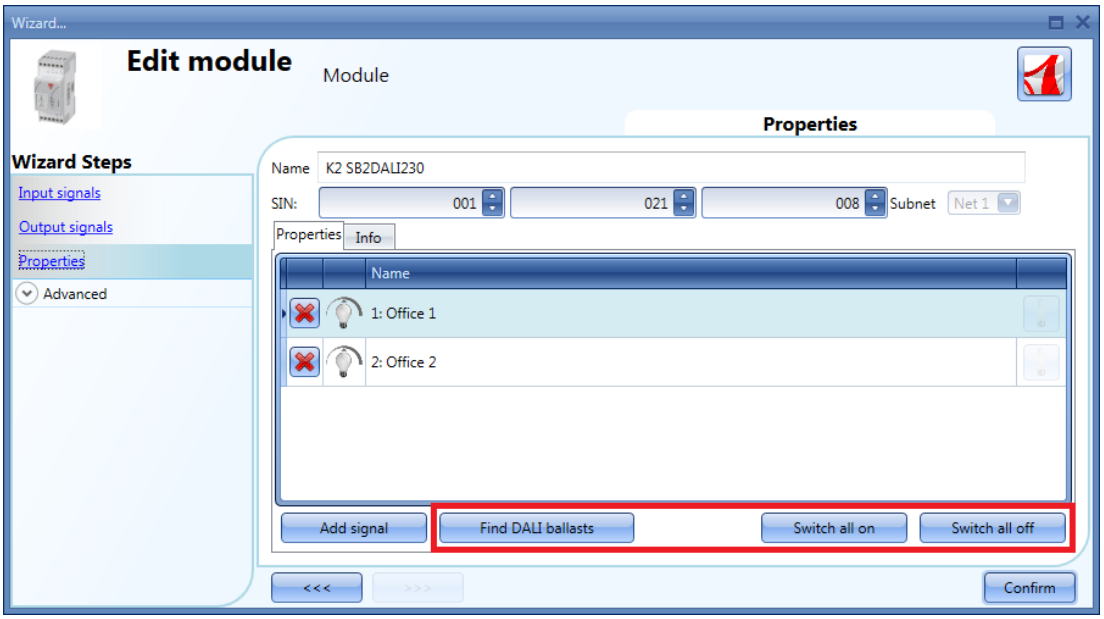
In the figure above, Office 1 and Office 2 Dimmable light functions are linked to the DALI outputs *Office 1* and *Office 2*.

13.7 How to automatically add a DALI ballast

The user connects an SB2DALI230 to the Smart House system and configures the module into the project.

13.8 How to automatically find the ballasts connected to the SB2DALI230

When an SB2DALI230 module is connected to the UWP 3.0 system and recognized by means of a network scan, the DALI buttons will be made active in the *Properties* window of the wizard of the module:

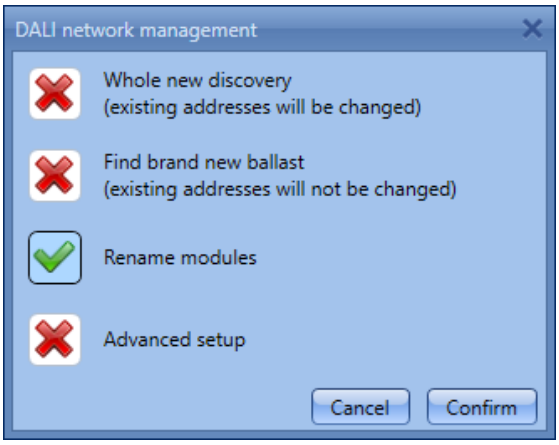


By means of the *Find DALI ballasts* button, the user can start a DALI network scan and the system will find the ballasts connected to the DALI bus.

By means of the *Switch all on* button, the user can switch ON all the lamps connected to the DALI bus.

By means of the *Switch all off* button, the user can switch OFF all the lamps connected to the DALI bus.

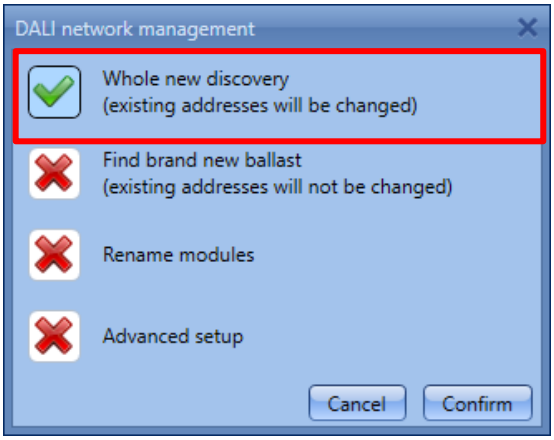
By clicking on *Find DALI ballasts,* the *DALI options* pop-up will appear (see picture below):



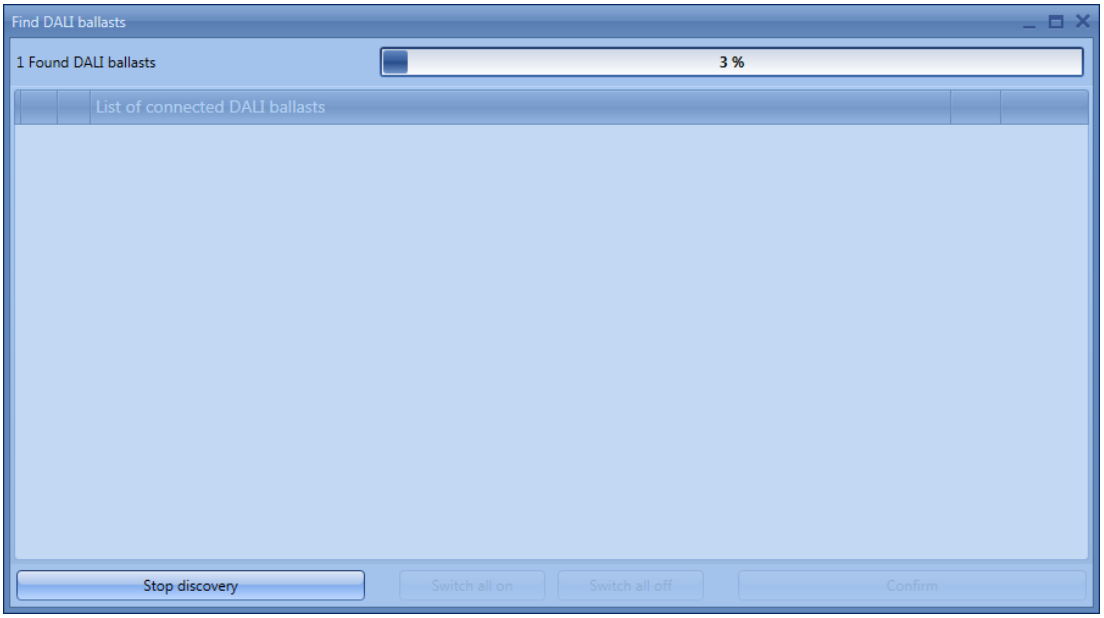
Field name	Description
Whole new discovery (existing addresses will be changed)	This provides a full scan and gives a new DALI address to all the ballasts found. This option is to be used the first time a network is configured or when completely new addresses have to be given to the DALI ballast.
Find brand new ballast (existing addresses will not be changed)	This gives a DALI address **only** to the new ballasts and does not affect the already configured ones. This is to be used when most of the network is already configured and new ballasts have been added.
Rename modules	This allows the user to manually change the addresses of the already configured modules.
Advanced setup	This allows the user to manage advanced setup features

13.8.1 How to launch a new network discovery

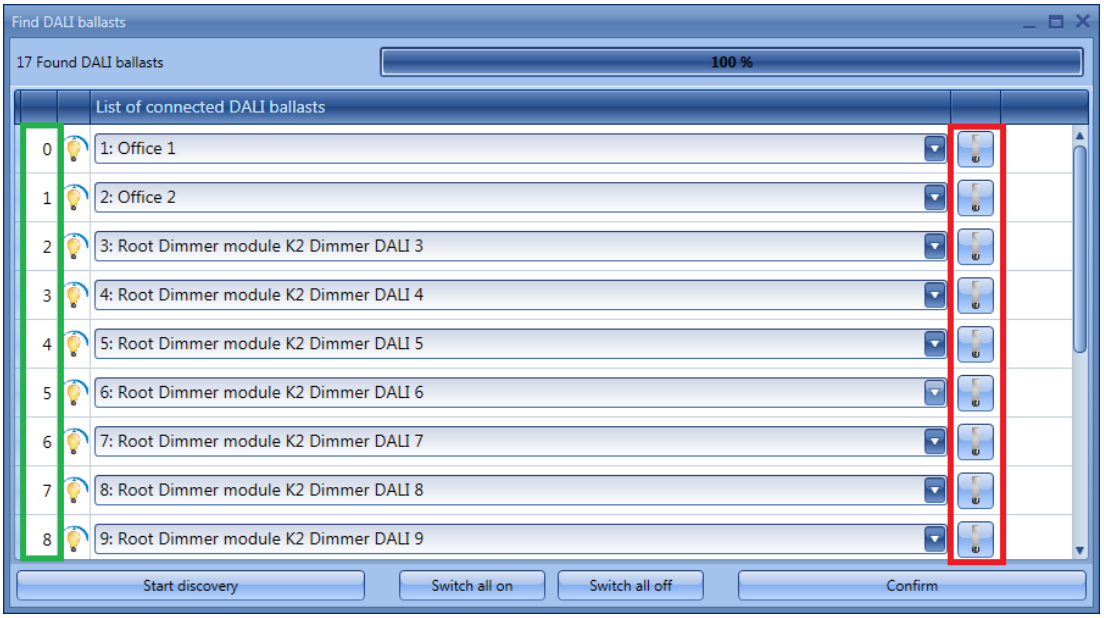
To perform a complete scan of the DALI bus, click on the *Find DALI ballasts* button and press on *New discovery*:



The following window will appear:



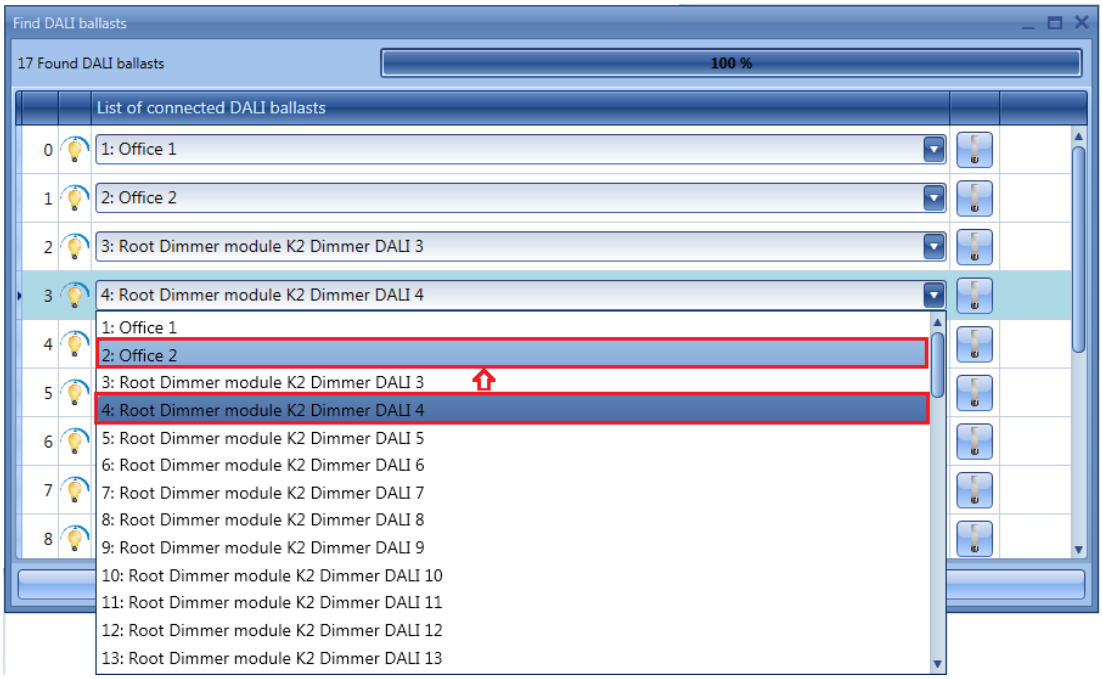
After a few seconds, when the bus scan is completed, all the ballasts found will be shown (see below).

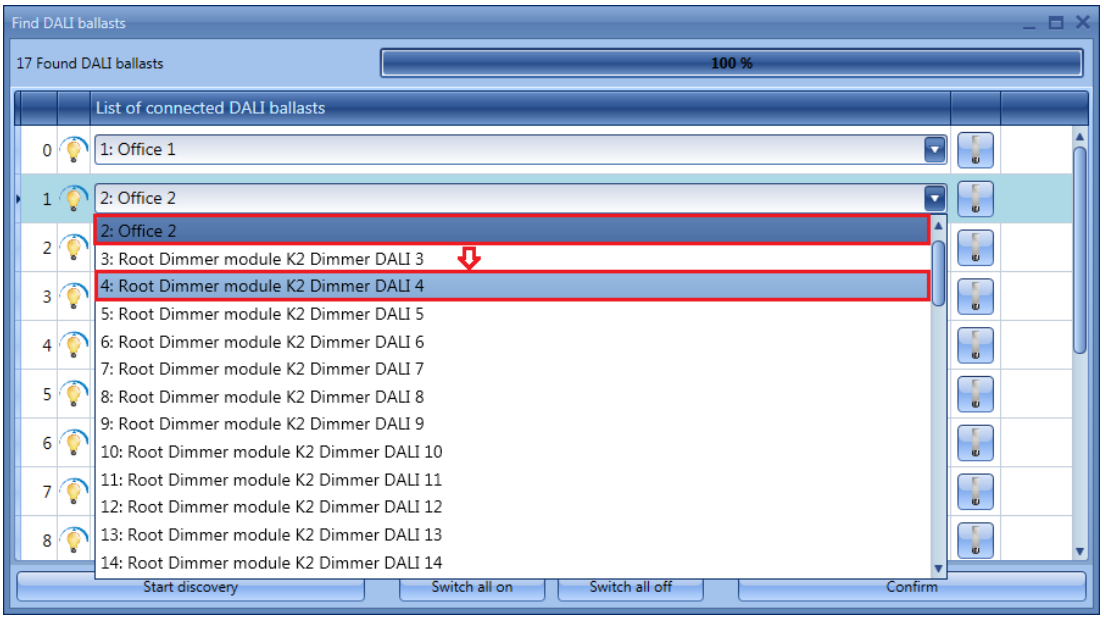


New discovery assigns a new address to each ballast, as shown in the first column (see the green rectangle). Each ballast can be switched on/off by means of the toggle button shown in the red rectangle. The installer can use this function to locate the ballast and give it its proper name.

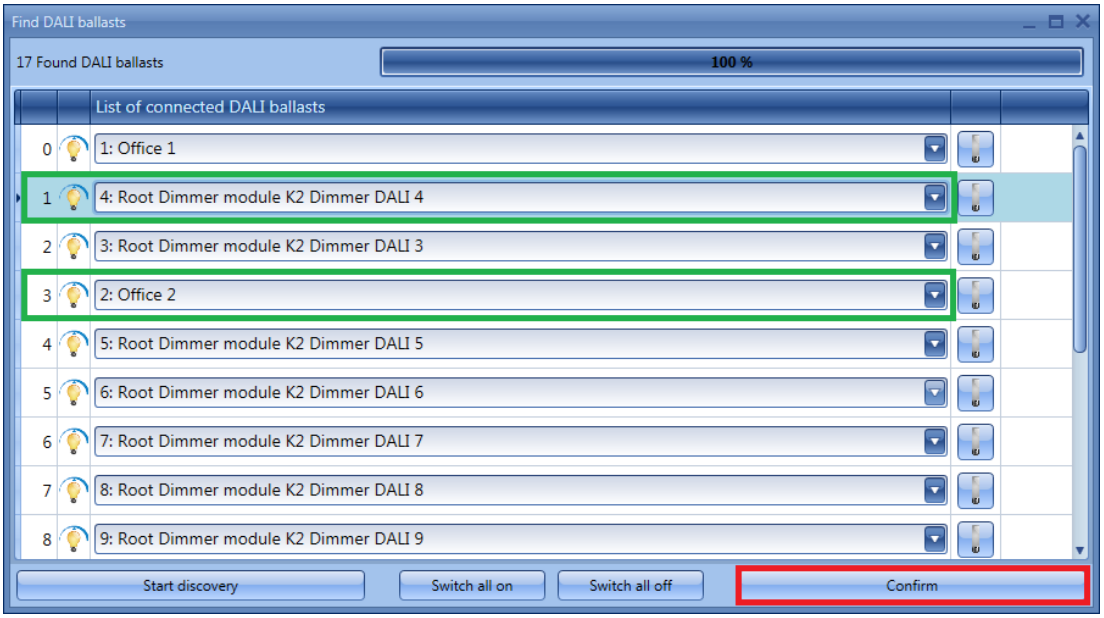
13.8.2 How to change a DALI address

In the example shown below, the lamp with address 3 (Dimmer output DALI 4) is given the address 1 (Office 2).



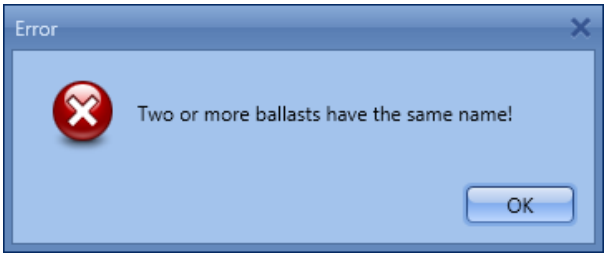


In the picture above, the ballast with address 1 (Office 2) has been addressed with address 3 (Dimmer output DALI 4)



The user may move all the desired devices to the new addresses. When all the devices have been addressed, press the *Confirm* button to save the configuration.

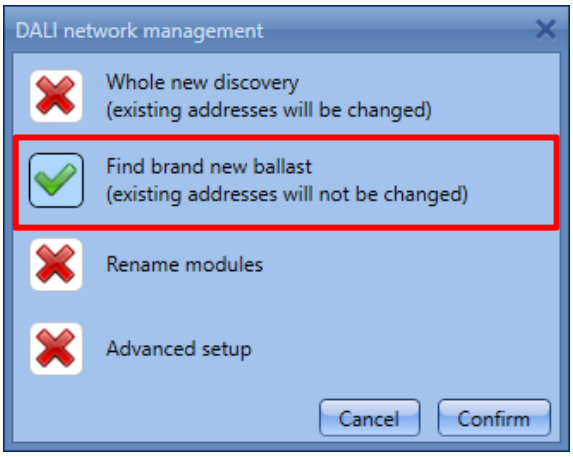
If two or more ballasts are given the same address, the following message will appear, preventing the user from saving the configuration until the correct mapping has been done.



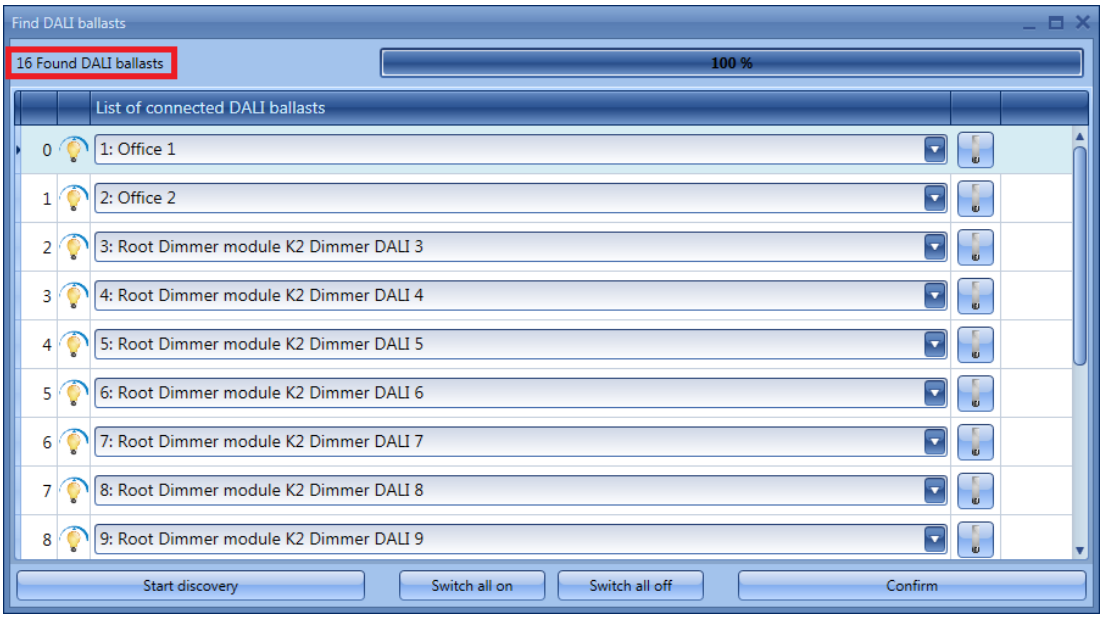
13.8.3 How to add new modules

At any time the user can add new DALI modules to the bus and configure them.

In the *Properties* window of the SB2DALI230 module press the *Find DALI ballasts* button, then select *Add new modules.*

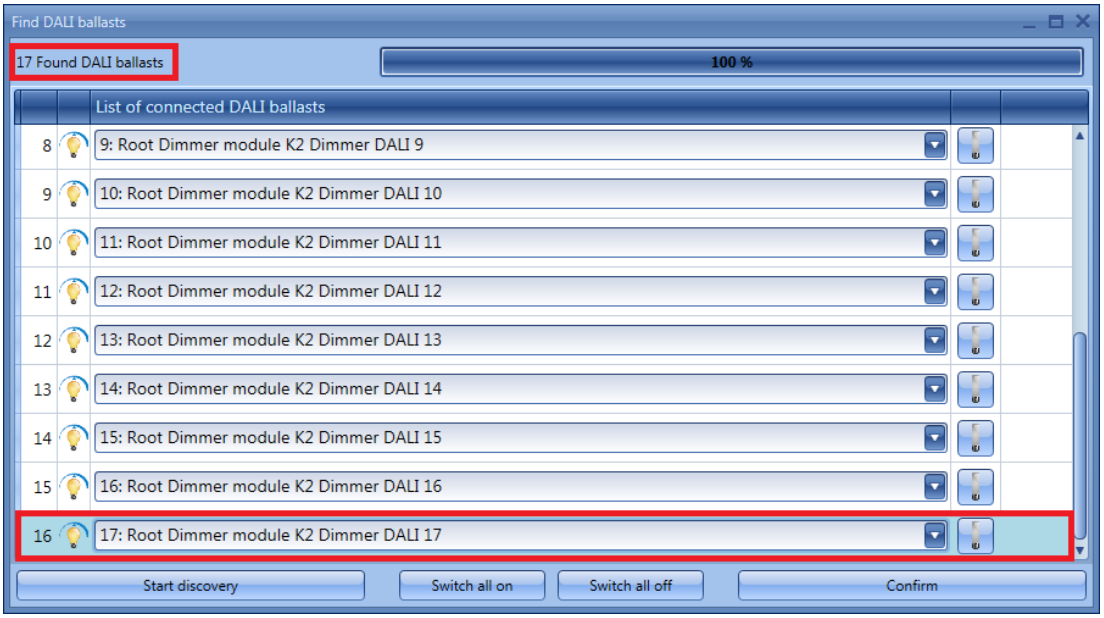


This function performs a new network scan, leaving unchanged the ballasts already configured.



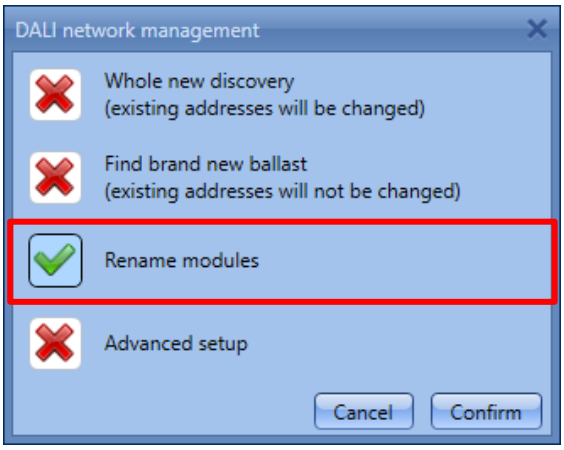
In the example shown below, one new DALI ballast has been found and added to the configuration with address 16.

All the newly discovered ballasts are shown at the bottom of the list.

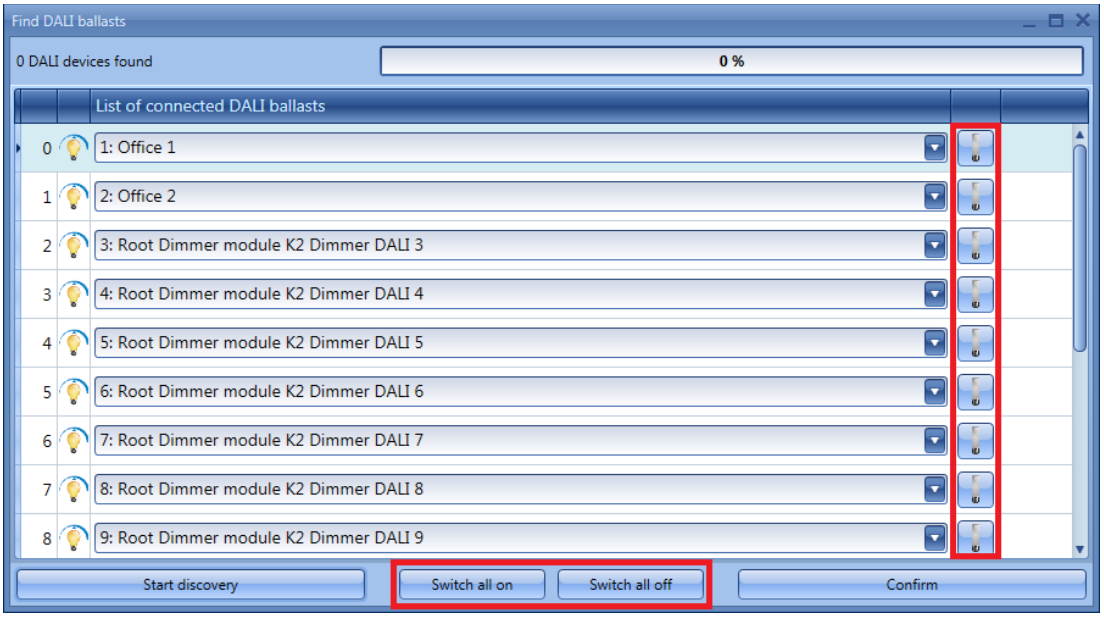


13.8.4 How to rename modules

By means of *Rename modules* the user can change the address of the already configured modules and rearrange the devices into a new order. In the *Properties* window of the SB2DALI230 module, press the *Find DALI ballasts* button and select *Rename modules*



The following window will appear:



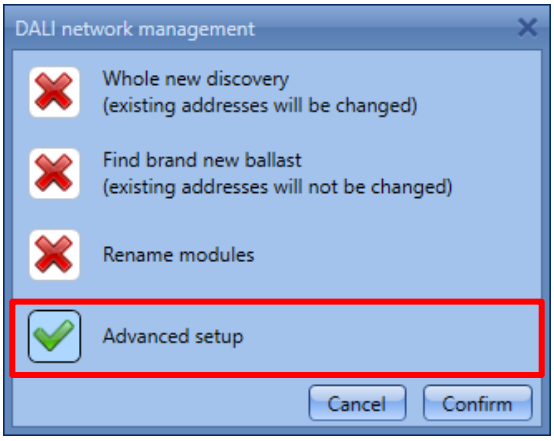
The *Start discovery* button will not perform any action as only renaming is allowed in this window.

Each ballast can be identified by means of the toggle button shown in the red rectangle and it is possible to switch them all on or all off by using the *Switch all on* or *Switch all off* buttons. Once the renaming is completed, press the *Confirm* button to save the new DALI modules configuration.

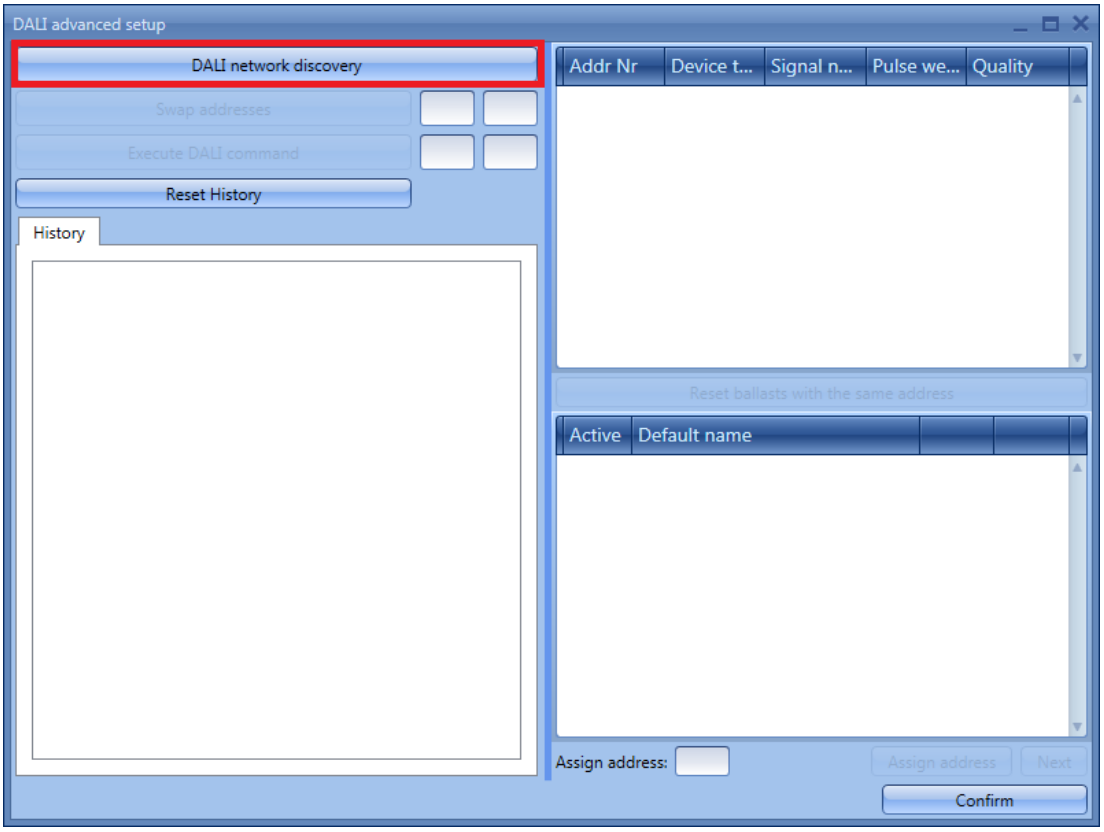
13.9 Advanced setup

By means of the *Advanced* setup, the user can access the advanced features available for the DALI bus. Should an error occur during the addressing phase, the user can easily fix it without performing a new discovery.

In the *Properties* window of the SB2DALI230, press *Find DALI ballasts* button, then select *Advanced setup.*

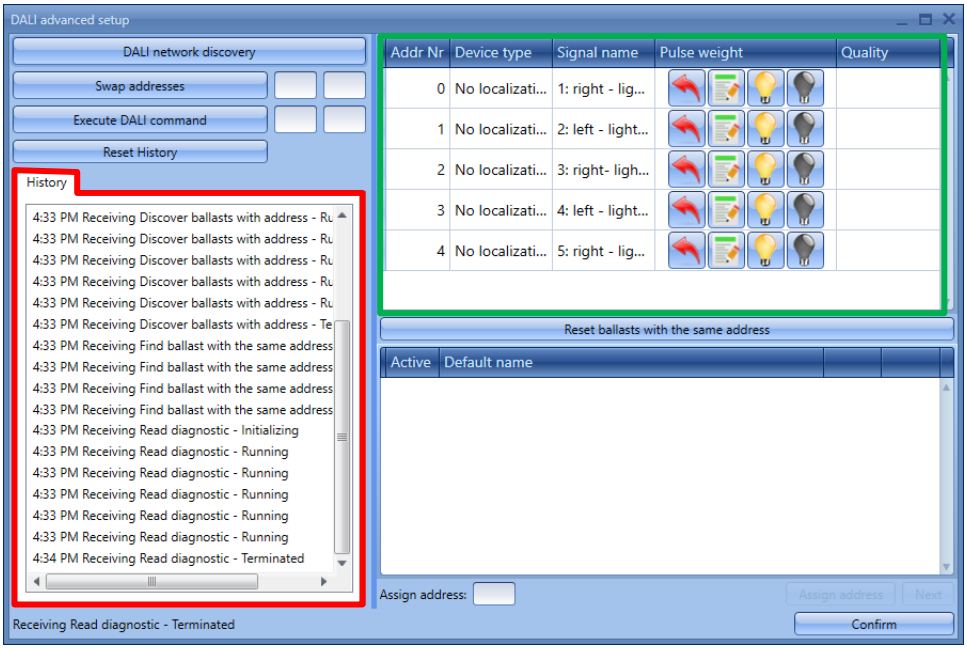


The following window will appear: to start the diagnostic, the user should first press the *DALI network discovery* button, which will start a DALI network scan.



Once the discovery has finished, the right part of the window shows the list of the connected DALI lamps (see green box). The system also shows, in the History box (red rectangle), details of the DALI bus activity.

By using the *Reset History* button it is possible to clear the history list.



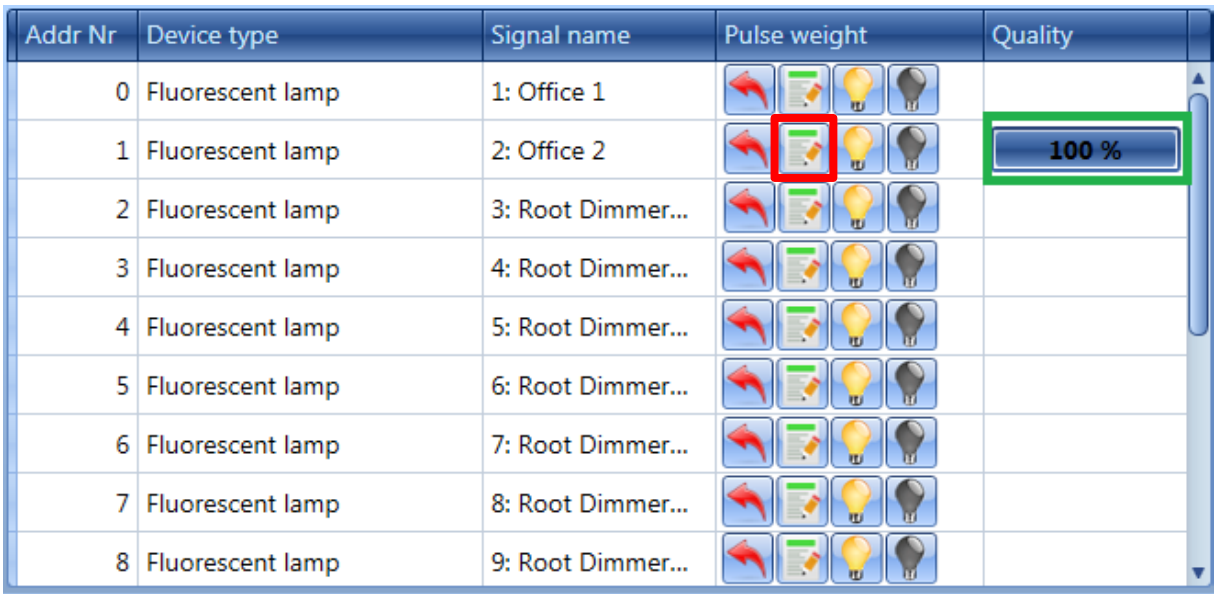
Description by column of the window marked in green:

Field name	Description
Addr. Nr.	This contains the address of the ballasts
Device Type	This contains the description of the ballast type
Signal name	This contains the name of the ballast
Pulse weight	Four actions are available: • Reset: this resets the ballast address, bringing it back to factory settings • Test: this sends an acknowledge command • On: Switch on the selected lamp • Off: Switch off the selected lamp
Quality	This gives an indication of the quality of the bus signal.

13.9.1 How to test a module

In the *Properties* window of the SB2DALI230 press the *Find DALI ballasts* button, then select *Advanced setup.*

The following window appears and the user should press the *DALI network discovery* button and wait for the bus data reading to finish. Once the discovery has finished, the right part of the window shows which DALI ballasts are connected to the bus.



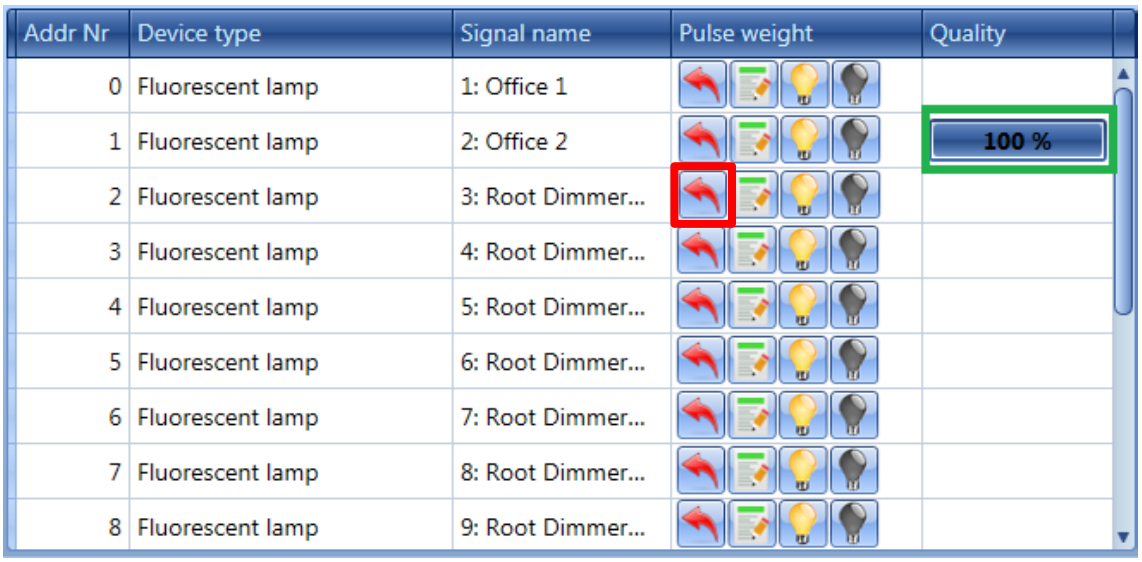
Addr Nr	Device type	Signal name	Pulse weight	Quality
0	Fluorescent lamp	1: Office 1		
1	Fluorescent lamp	2: Office 2		100 %
2	Fluorescent lamp	3: Root Dimmer...		
3	Fluorescent lamp	4: Root Dimmer...		
4	Fluorescent lamp	5: Root Dimmer...		
5	Fluorescent lamp	6: Root Dimmer...		
6	Fluorescent lamp	7: Root Dimmer...		
7	Fluorescent lamp	8: Root Dimmer...		
8	Fluorescent lamp	9: Root Dimmer...		

By means of the *Test* button, the system performs a full quality test of the bus signals. The result is reported in the *Quality* column.

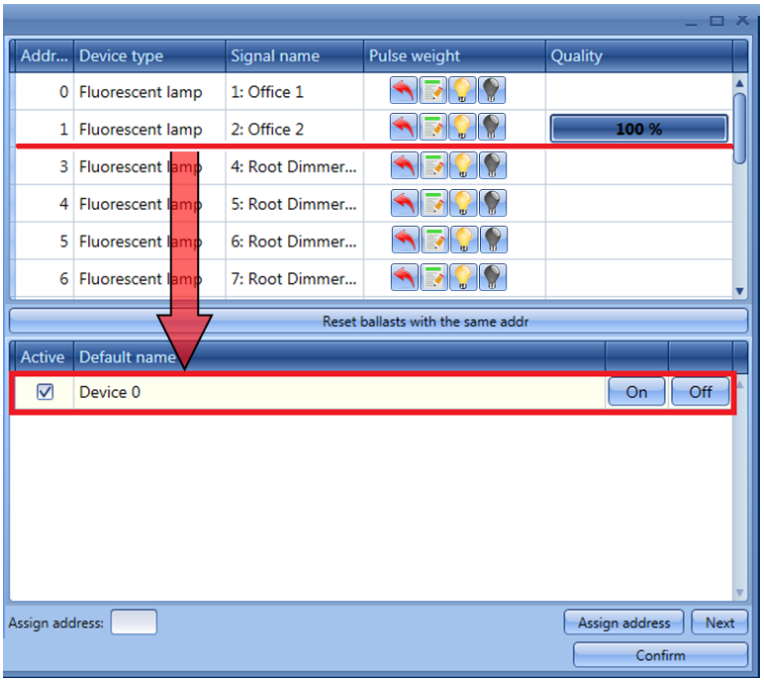
13.9.2 How to reset a DALI ballast

In the *Properties* window of the SB2DALI230 press the *Find DALI ballasts* button, then select *Advanced setup.*

The following window appears and the user should press the *DALI network discovery* button and wait for the bus data reading to finish. Once the discovery has finished, the right part of the window shows which DALI ballasts are connected to the bus.

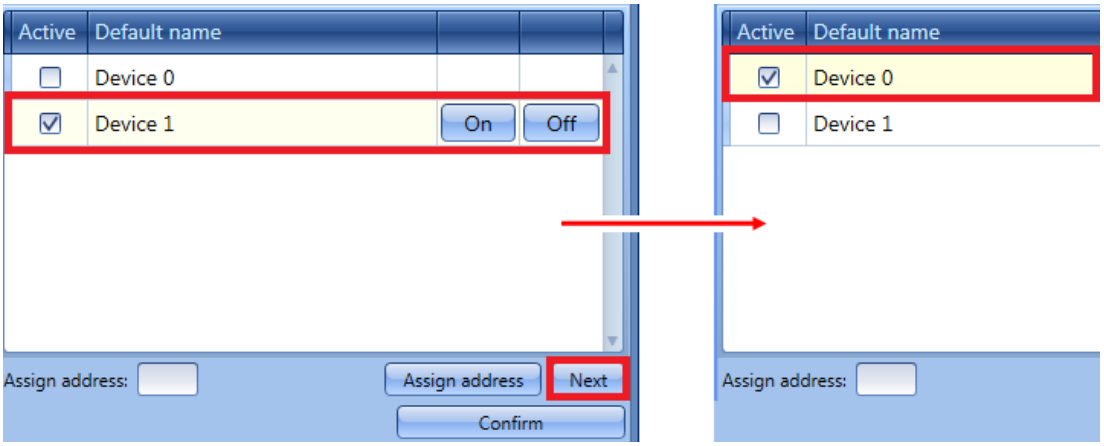


By means of the *Reset* button, the user can bring the ballast back to the factory settings. This means that should this ballast be connected to a DALI bus, it will be recognized as a new one.



All the modules can be reset and a temporary name will be assigned to each of them. As shown in the figure above, the device with address 2 has been reset: it has been moved from the window at the top to the window at the bottom, with the temporary name *Device 0*. It can be switched on or off with the relevant push buttons on the right.

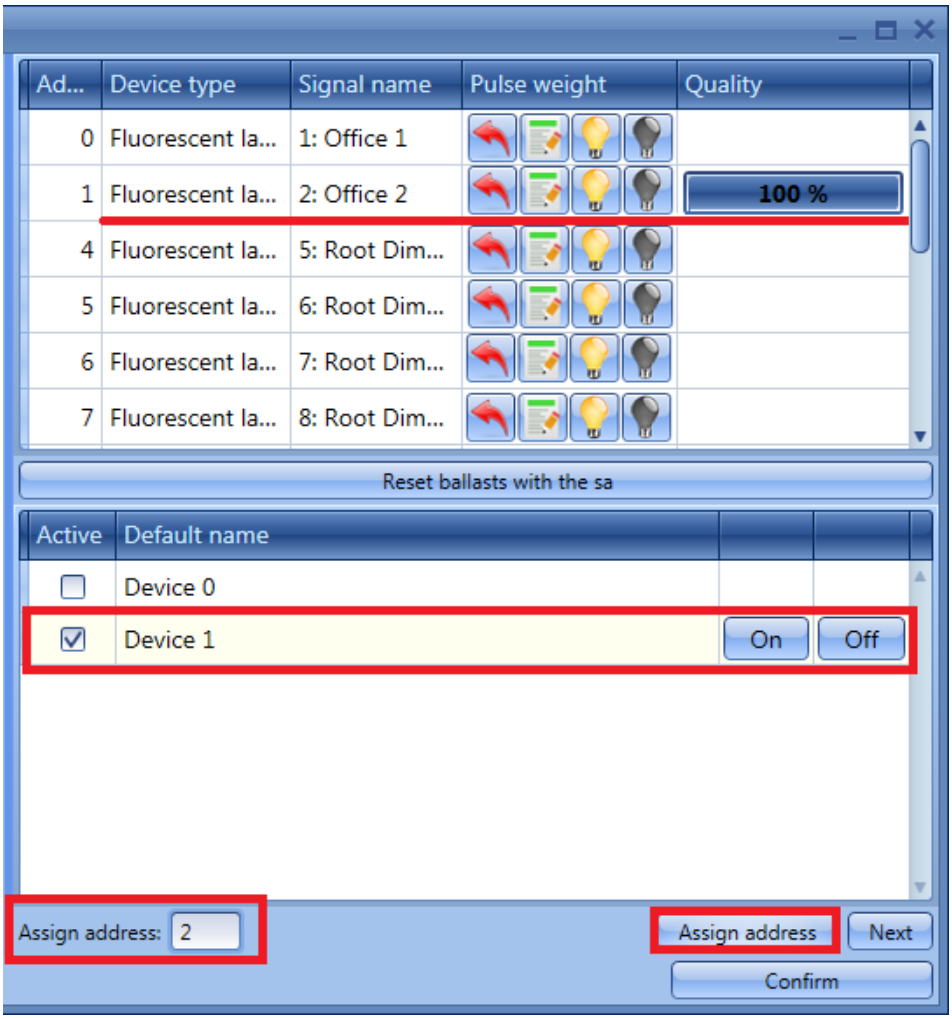
If more than one device has been reset, a list such as the one shown below will appear. According to the DALI protocol, only one device with no address can be switched on/off at any one time, so to go through the list, the *Next* button has to be used.



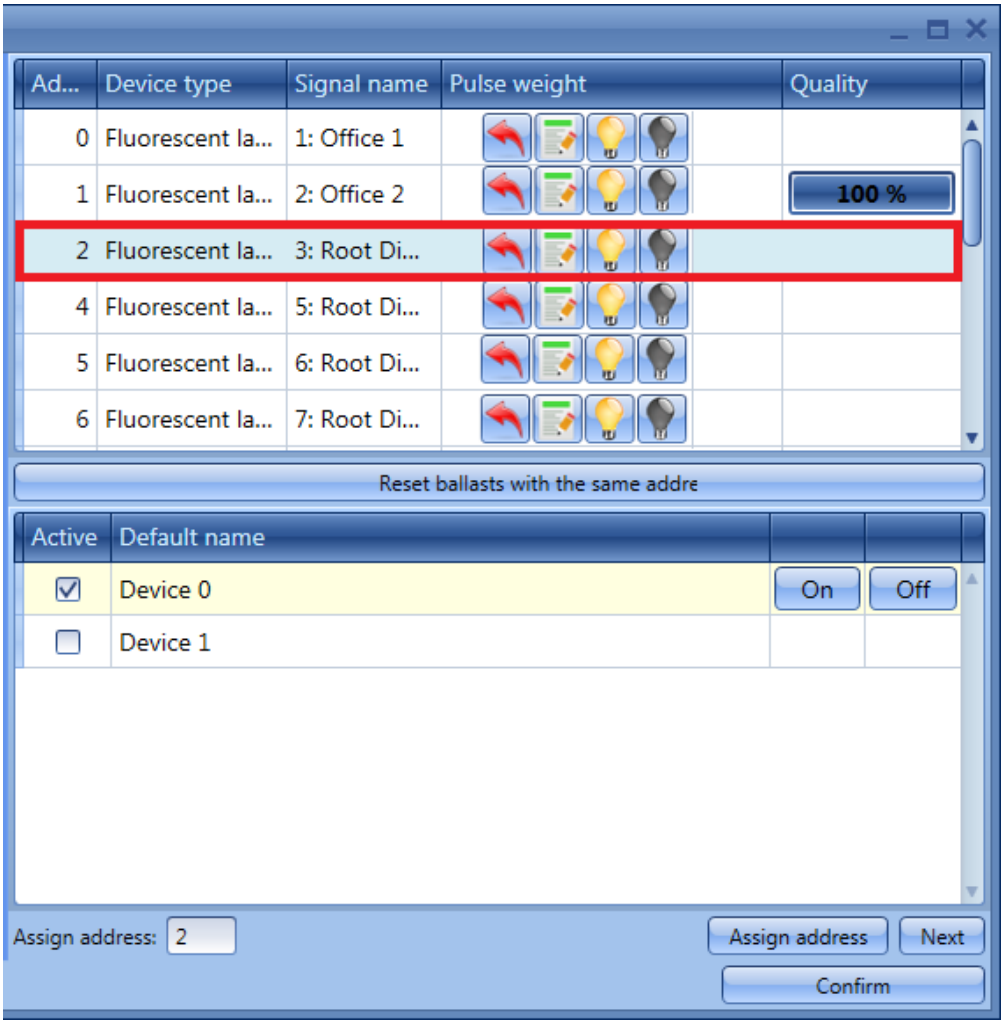
13.9.3 How to manually assign an address to a ballast

After having done what is described in *Advanced setup,* select the device in the window at the bottom and complete *Assign address to active* box with a free address in the range of 0 to 63 (in the example below the first free address is 2). Finally, press the *Assign address* button to assign the new address.
It is best to assign the lowest free address.

N.B. Even if the ballast is assigned with an address, it still remains in the window below until a new DALI scan is performed by clicking on *DALI network discovery.*



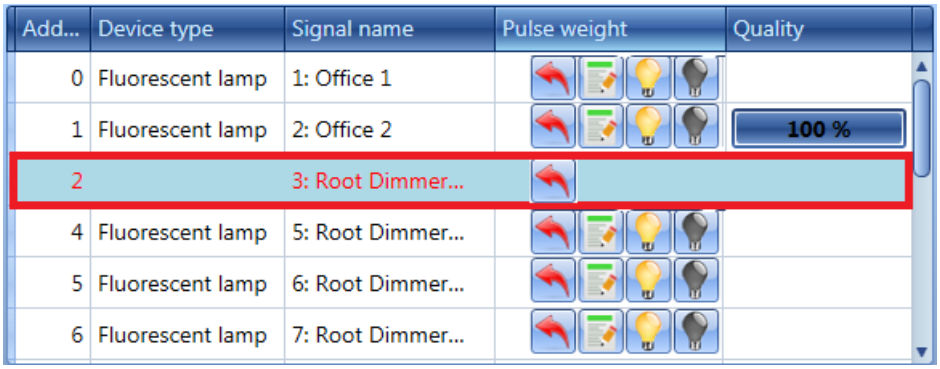
The ballast that was present in the window at the bottom (Device 1), is now present also in the window at the top with the newly assigned address as shown in the picture below (2 Fluorescent lamp).



The user can then move to the next device to be addressed by clicking on *Next*.
To save the new address click on *Confirm*.

The next time the user performs a new *DALI network discovery,* only the ballasts without an address will be listed in the window at the bottom.

Should the user assign the same address to two or more ballasts, in the window at the top the row relevant to this duplicated address will be marked in red: the user must click on *Reset* to remove this error condition and must correctly re-address these ballasts.

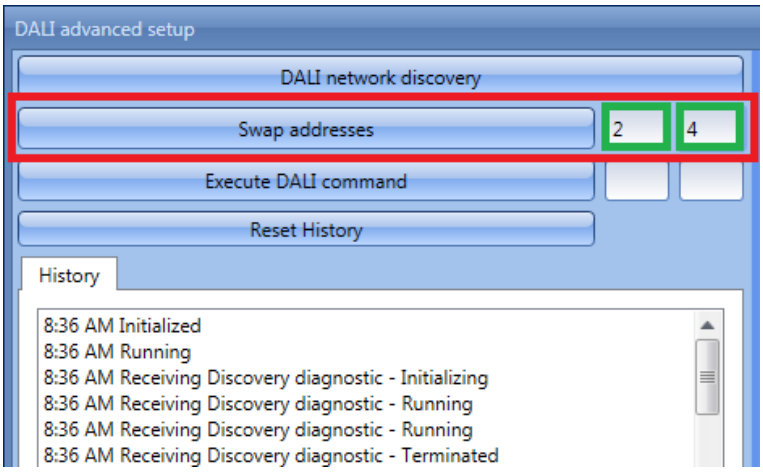


13.9.4 How to swap the addresses of two ballasts

In the *Properties* window of the SB2DALI230 press the *Find DALI ballasts* button, then select *Advanced setup.*

The following window appears and the user should press the *DALI network discovery* button and wait for the bus data reading to finish.

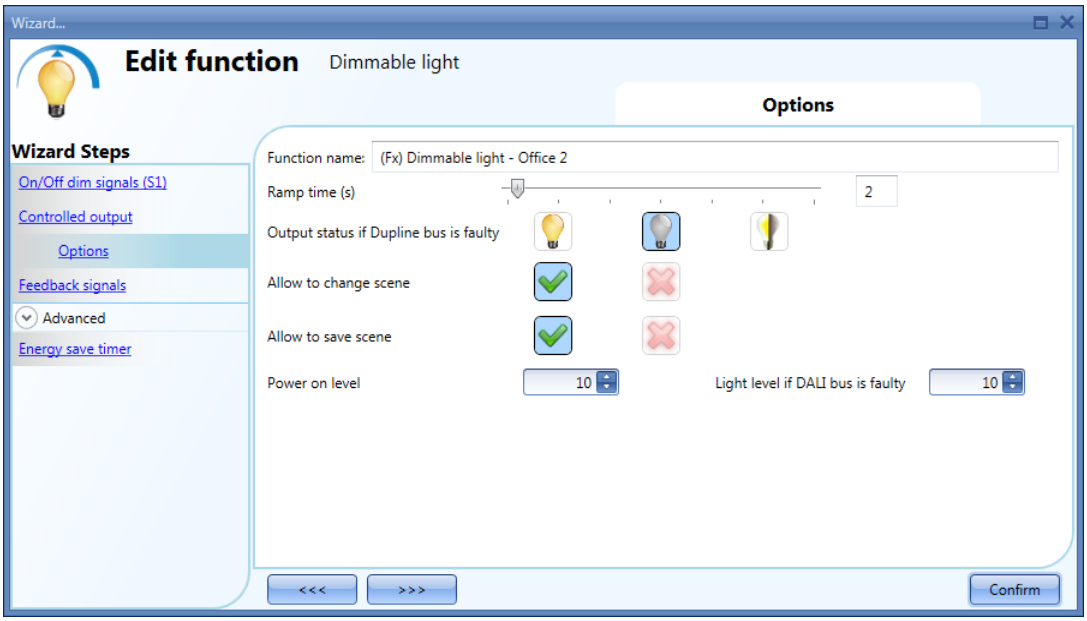
Swap addresses allows two ballast addresses to be exchanged. To swap two devices, enter the two addresses in the boxes (green rectangles), and press the *Swap addresses* button. It is mandatory to fill in both boxes.



13.10 How to change the settings of the DALI output in a function

To configure the output options in a Dimmable or Constant light function, the user has to click on O*ptions* in the wizard function: the following window will appear (see picture below).

If more outputs are present in the function, the settings are applied to all of them.



Ramp Time (s):
This is the required time to go from 0% to 100% when at least one input is active.
For example, if 10 seconds is set, once the light is on, this is the time the light will take to go from 0% to 100% and from 100% to 0% when the push button is kept pressed.
The ramp time is the same for all the dimmer outputs belonging to the same function.

Allow scene to be changed:
When this option is enabled (the green "*V*" is selected), the light level in all the scenarios will be modifiable when the input signal is activated. Otherwise (when the red "cross" is chosen), the user will not be able to change it (with the exception of scenario S1).

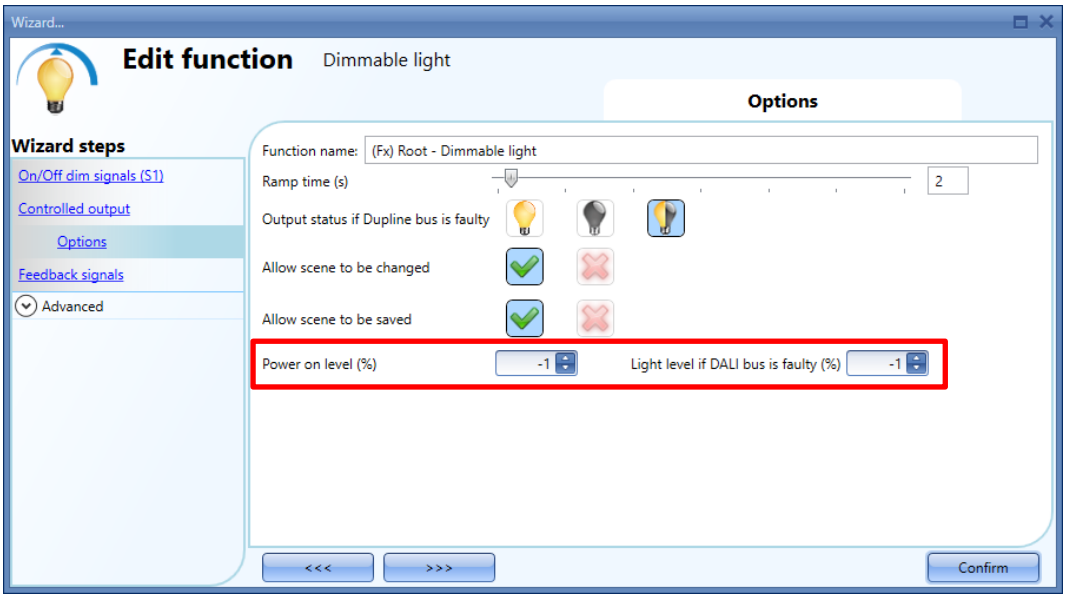
Allow scene to be saved:
When this option is enabled (the green "*V*" is selected), each time the dimmer is powered OFF, the current light level value is saved in the memory before the switching off of the module. Otherwise (when the red "cross" is chosen), the scenario is not saved and the power is switched OFF.

13.10.1 How to set / restore the DALI ballast output state

Note: To keep the last valid state in the DALI ballast after an error condition, the SB2DALI230 gateway must have the revision firmware 6 or higher and the UWP 3.0 Tool must be equal to or higher than version 6.5.3

The user means the Power on level (%), Light level if DALI bus is faulty (%) and Output status if Dupline bus is faulty fields are able to handle the DALI ballast output value.

Note2: The firmware equal to or older than revision firmware 5 can manage only the value from 10 to 100 for Power on level (%) and Light level if DALI bus is faulty (%).



13.10.2 How to set the Power on level (%) field

In the *Power on Level* field the user can select the value of the DALI ballast output when the DALI ballast is powered On: in this condition the lamp connected to the DALI ballast is turned On to the value set on this field and this value is kept until the DALI Ballast receives the first DALI command.

To set the *Power on level (%)* field, the user has to select the *Dimmable light* function or the *Constant light* function they want to manage, then click on the *Options* tab of the wizard.

The value can be set in the *Power on level(%)* field according to the table below:

Value	DALI ballast output state
-1	**Restore the last value:** the last ballast value present before the fault condition (if supported by ballast)
0	Output is kept OFF until the fault condition is restored
10 - 100	Output is set to the selected value (%) until the fault condition is restored

Note: Live Signals diagnostic info

Due to a power supply fault condition for a DALI ballast module, the UWP 3.0 controller detects the "ballast not present condition". After power up, the DALI ballast output is immediately set with the Power on level value set in the field until the first DALI command is received from the DALI bus.

13.10.3 How to set the Light level if the DALI bus is faulty

In the *Light level if DALI bus is faulty* field the user can select the value of the DALI ballast output when the DALI bus is in faulty condition. This condition occurs when the SB2DALI230 gateway is in one of the following conditions:

- The DALI bus cables are short-circuited;
- The DALI bus cables are not correctly connected to the DALI ballast;
- The SB2DALI230 module is not powered or faulty;

The lamp connected to the DALI ballast is turned On to the value set in this field and will be kept at the value set until the DALI bus signal returns and the DALI ballast refreshes its status with a valid DALI command.

To set the *Light level if DALI bus is faulty* field, the user has to select the *Dimmable light* function or the *Constant light* function they want to manage then click on the *Options* tab of the wizard.

The value can be set in the *Light level if DALI bus is faulty* field according to the table below:

Value	DALI ballast output state
-1	**Restore the last value: the** last ballast value present before the fault condition (if supported by ballast)
0	Output is kept OFF until the fault condition is restored
10 - 100	Output is set to the selected value (%) until the fault condition is restored

Note: Live signals diagnostic info

The "Dali bus fail state value" remains until the restore timeout event occurs or the light status changes.

The ballast output is updated according to the light function state (the restore timeout event is forced)

1) *The fault condition is activated when the Presence Diagnostic is not detected and is deactivated when the valid Smart Dupline response is received or Presence Diagnostic is detected.*

2) *The output value is sent to the module when the value changes. Because the module sets the start-up value at power up, the user can define a refresh timeout value that forces the output value to update if there are no changes.*

13.10.4 How to set the Output status if the Dupline bus is faulty

In *Output status if the Dupline bus is faulty* field the user can select the status of the output if the Dupline bus is faulty. This condition occurs when the Smart Dupline bus is not working properly or the SB2DALI230 gateway is in error. The most common scenarios are:

- The Smart Dupline bus is short-circuited;
- The SB2DALI gateway is not connected to the Smart Dupline bus;
- The UWP 3.0 controller and/or the SH2MCG24 generator are not powered or faulty;

The value can be set in the *Output status if the Dupline bus is faulty* field according to the table below:

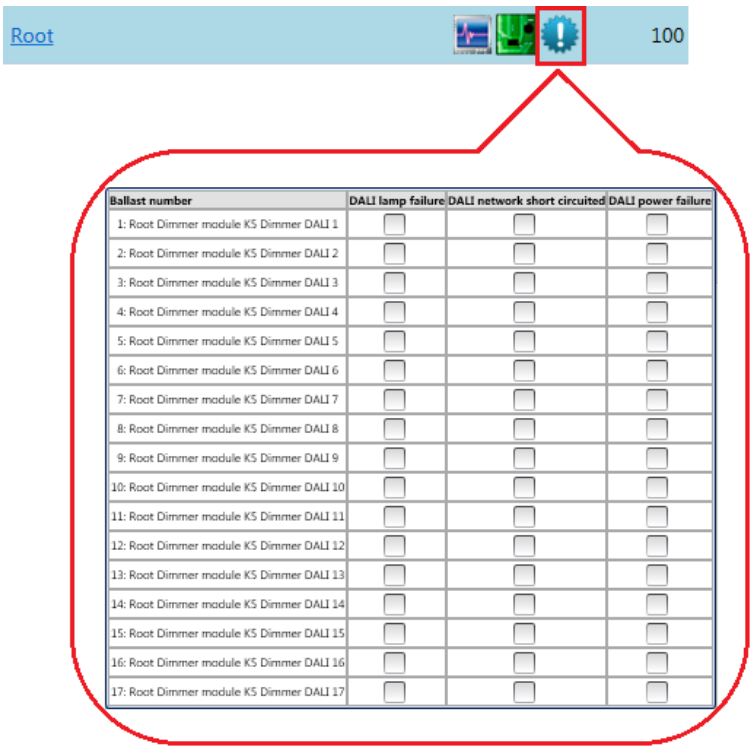
Value	DALI ballast output state
	The DALI ballast output is forced into the 'ON' status
	The DALI ballast output is forced into the 'OFF' status
	The DALI ballast output remains in the status it had before the loss of connection with the Dupline Bus. Each selected fail-state action will be kept active until the Dupline bus signal returns

13.11 How to check the DALI bus status

Each SB2DALI230 has a dedicated diagnostic menu, accessible automatically when the *live signals* are enabled.

When *Live signals* are active, referring to the picture below, the icons on the right appear and the one marked in red indicates the DALI bus status.

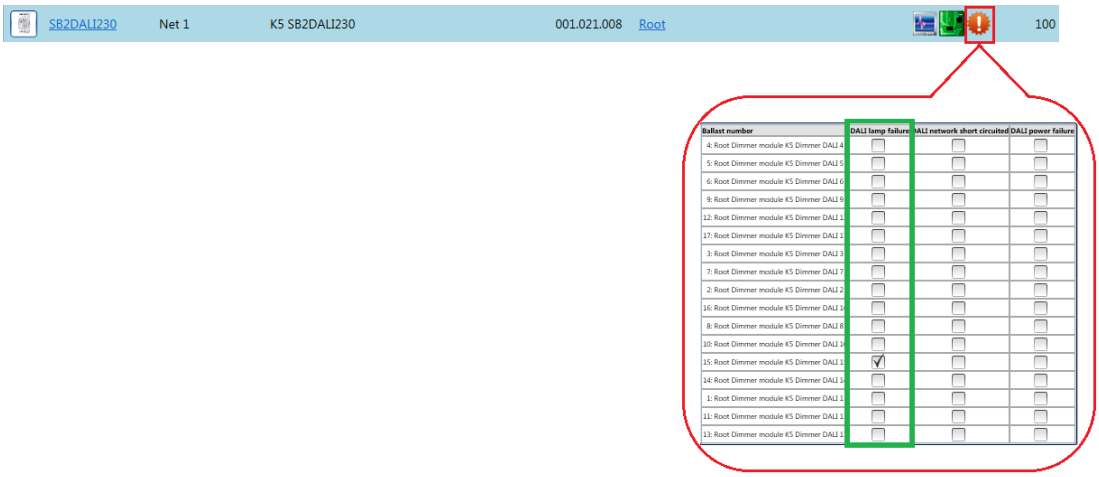
By means of this, the user can have a real time monitor of the health status of the DALI bus.
If everything is working properly, the icon is blue and when the mouse hovers over it a tool tip appears, flagging no errors.



If a failure is present on the DALI bus, it will be shown by the icon turning orange and when the mouse hovers over the icon, a tool tip appears, flagging the relevant error. There is one line for each ballast.

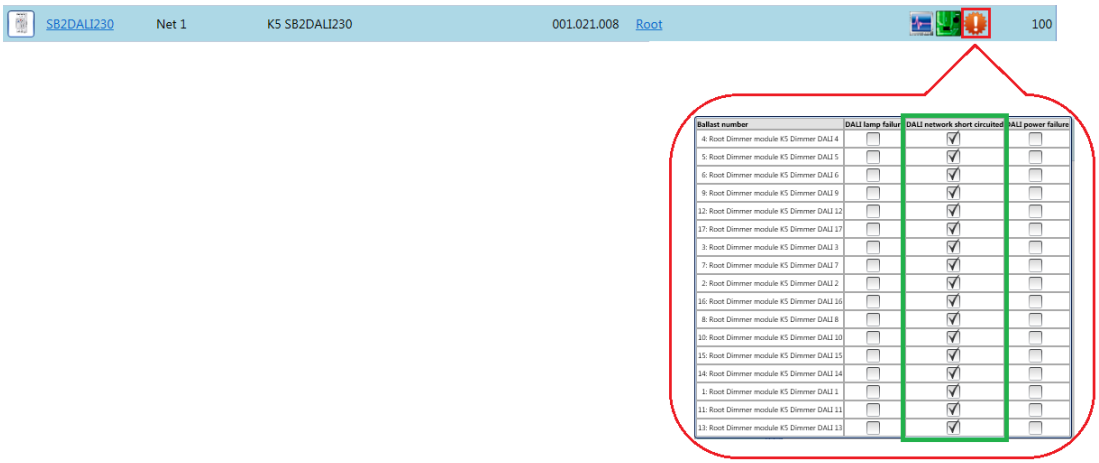
13.11.1 DALI lamp failure

If a lamp is damaged or missing, the icon relevant to the DALI bus status will turn orange and the relevant flag will be ticked (see the green box).



13.11.2 DALI network short-circuited

If a DALI network short-circuit is present, the icon relevant to the DALI bus status will turn orange and the relevant flag will be ticked (see the green box).



13.11.3 DALI power failure

If a DALI ballast has a power failure, the icon relevant to the DALI bus status will turn orange and the relevant flag will be ticked.

13.11.4 How to check if the DALI ballast supports the latest state

To check if the DALI ballast supports the latest restore state, the user has to set with -1 the Power On Level and Read System Failure fields and send the configuration to the UWP 3.0 controller and open the SB2DALI230 options. Click on *DALI Advanced menu* and once the discovery has been completed use the *Execute Dali command* to read the *Power On* and *Read System Failure* parameters as follows:

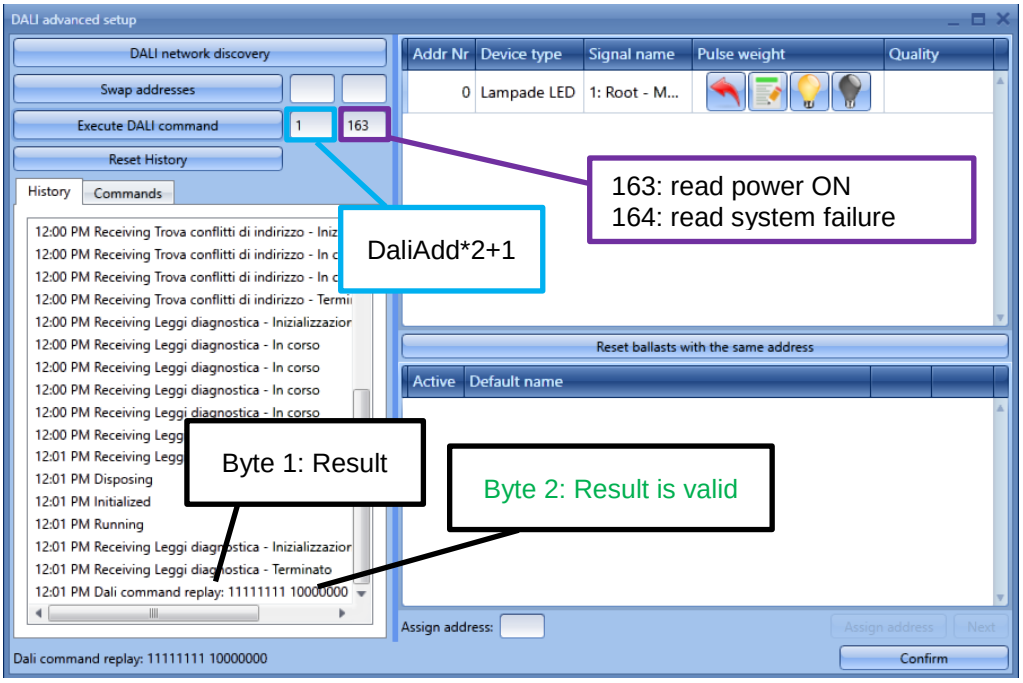
First byte: Dali Address*2+1

Second Byte: 163 (Power ON)

164 (Read System failure)

The response appears on the *History log* window: first byte is the value; second byte should be 10000000b (indicating positive command)

If the ballast manages correctly the restore state, the first byte should be 11111111, otherwise the ballast may not accept/manage these features

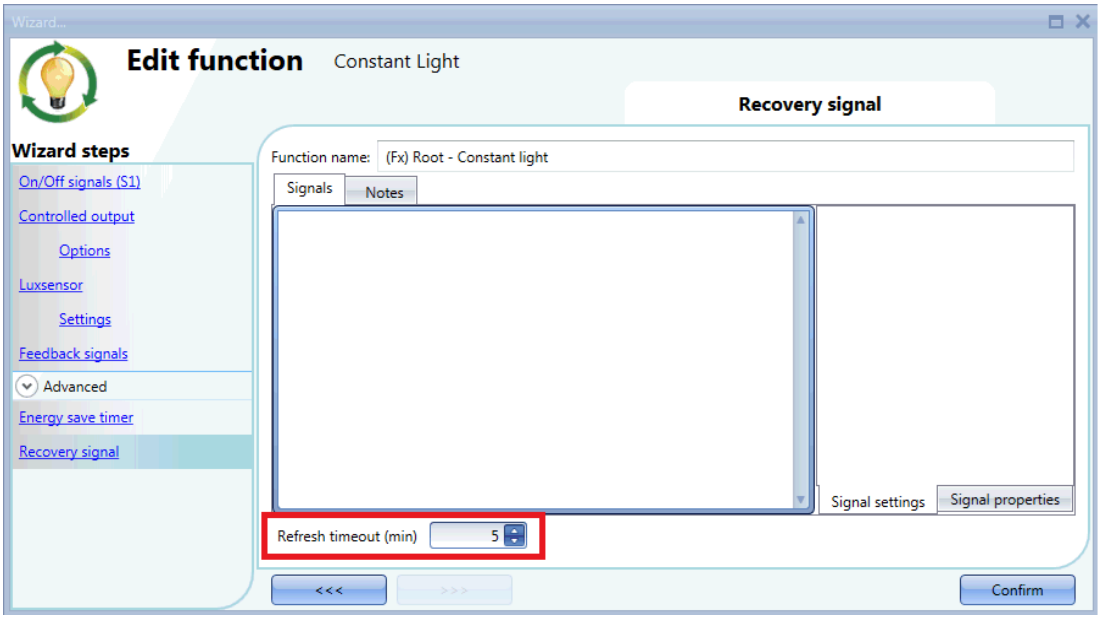


13.11.5 How to restore the latest value with the Recovery signal option

The *Restore the latest state* feature that can be set in *Power on level* and in *Light level if DALI bus is faulty (%) fields* may not be accepted/managed by some models of DALI ballast, so in order to manage the last DALI output state before a faulty condition, the *Recovery signal* option is available in the *Dimmable light / Constant light* function the *Recovery signal* option: this feature can be used to update the DALI output with the function status.

13.11.5.1 How to set auto-recovery using the (inside) timer

The user should click on a *Dimmable Light* or *Constant light* function where the DALI ballast is used, then click on the *Advanced menu* and select the *Recovery Signal* field: a window will appear, as shown below:



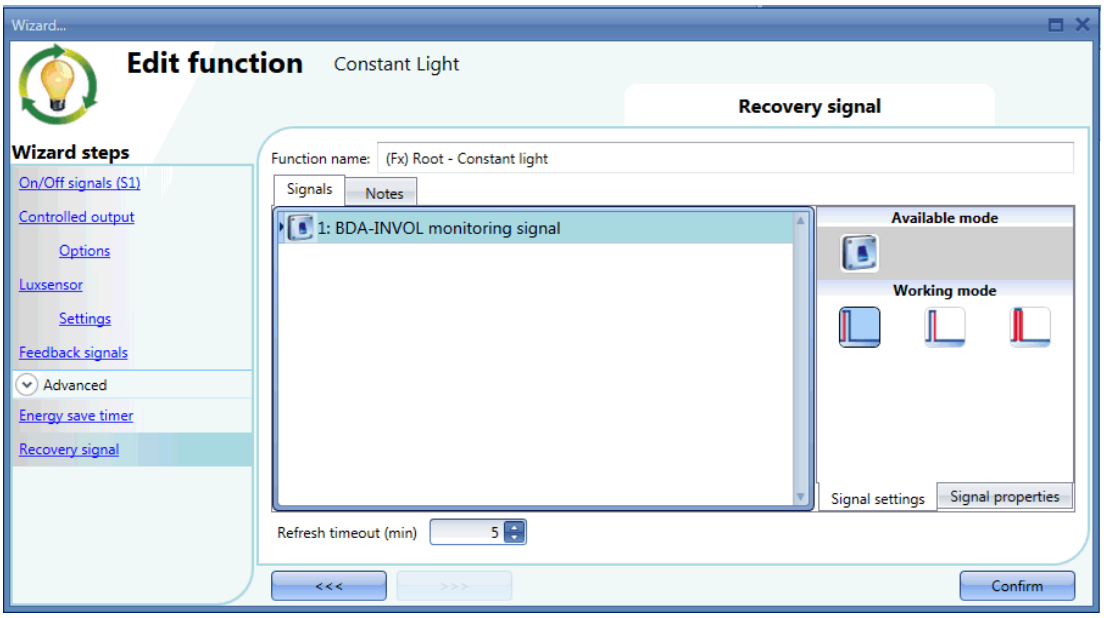
In the *Refresh timeout (min)* field the user can define a timeout value: when this timer expires the function automatically sends a command to the ballast and aligns the DALI outputs with the function status.
This timer is restarted every time there is a manual action; when the timer expires, it is automatically restarted and cycles continuously. The value of this field can also be changed via the Live signals.

In the example above the timeout is set to 5 minutes, this means that every 5 minutes the system checks the function value and the DALI ballast output value. If the ballast output value does not match the expected status, the system automatically sends a command to the DALI ballast in order to update its output value.

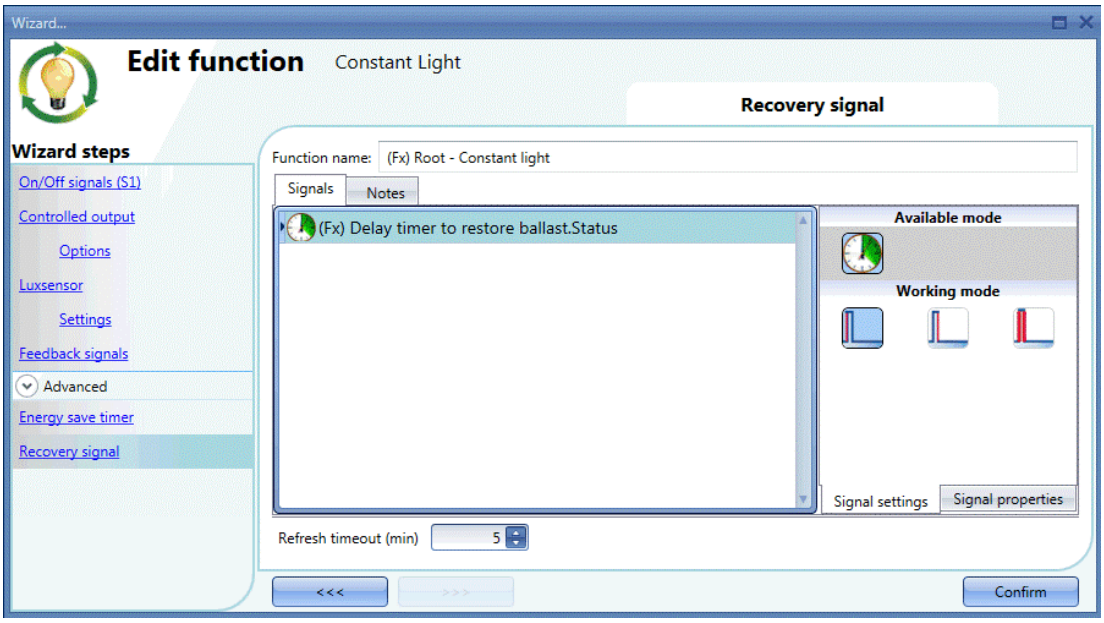
With this feature, if a power failure occurs during the night and the DALI ballast goes to *Power on level* (for example 50%) after 5 minutes the system will automatically turn Off the ballast, restoring the function value.

13.11.5.2 How to set manual recovery using a signal

The behaviour is the same as above, the only difference being that there is no need to wait for a timeout because it is possible to force the restore of the functions by using an input signal: a typical example is a BDA-INVOL-U module that monitors the power supply of the mains: when the Power line is lost the signal is deactivated, when the Power line is back again, the signal is activated and the DALI outputs are restored.

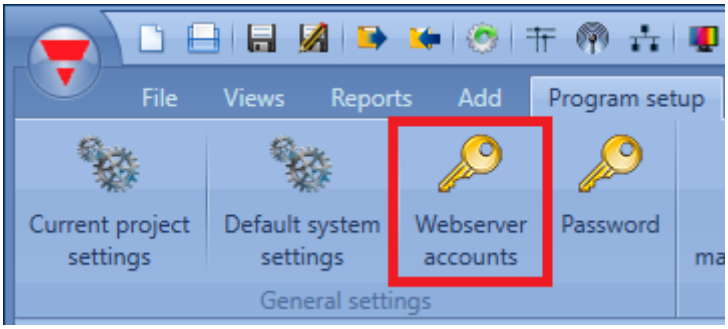


A typical application is using a BDA-INVOL-U signal inside a timer function in order to send the update command few seconds after the Power line is restored. In this way the DALI ballasts have time to start-up, as shown in the example in the example below:

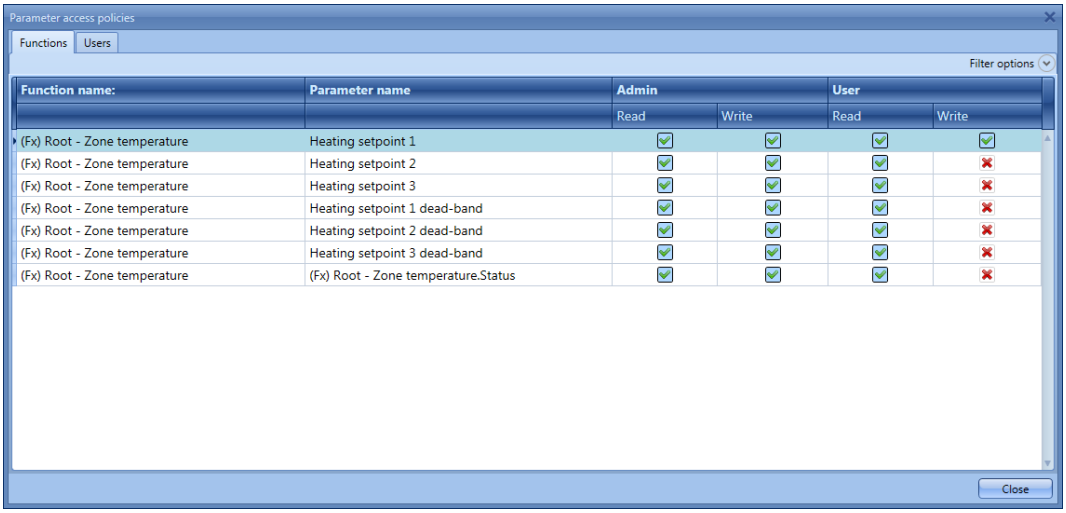


14 How to configure the webserver accounts

To set up the webserver, the user should select *Webserver accounts* from the *Setup* menu.
The relevant window of the webserver accounts settings can also be opened by typing Alt+S+W.



In the webserver window, the user can set read/write permission for two different accounts:

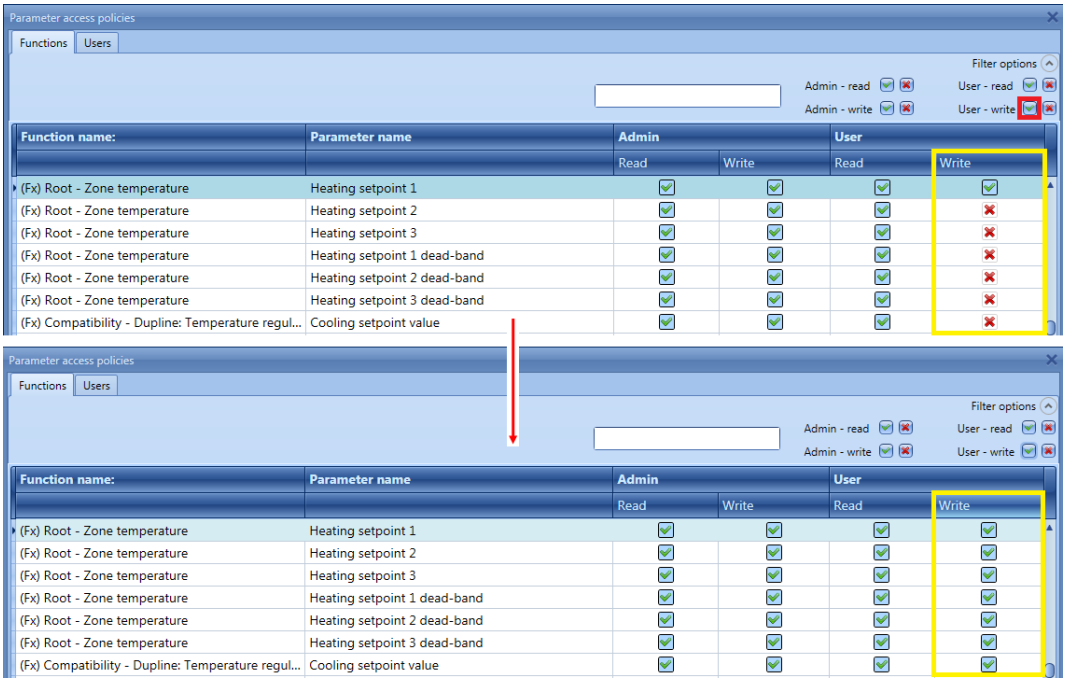


Function name:	Parameter name	Admin		User	
		Read	Write	Read	Write
(Fx) Root - Zone temperature	Heating setpoint 1	✓	✓	✓	✓
(Fx) Root - Zone temperature	Heating setpoint 2	✓	✓	✓	✗
(Fx) Root - Zone temperature	Heating setpoint 3	✓	✓	✓	✗
(Fx) Root - Zone temperature	Heating setpoint 1 dead-band	✓	✓	✓	✗
(Fx) Root - Zone temperature	Heating setpoint 2 dead-band	✓	✓	✓	✗
(Fx) Root - Zone temperature	Heating setpoint 3 dead-band	✓	✓	✓	✗
(Fx) Root - Zone temperature	(Fx) Root - Zone temperature.Status	✓	✓	✓	✗

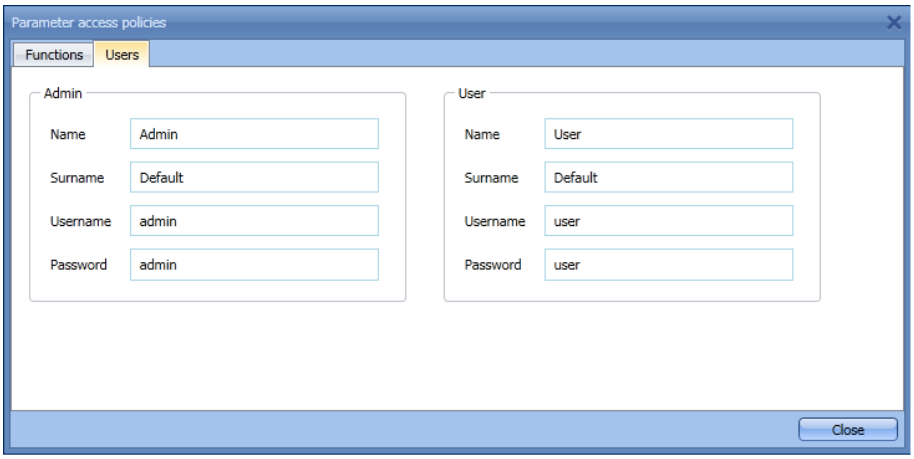
The **Admin** column is the admin account (by default read and write permissions are all ticked, indicating full access to all the functions). **User** is the column of the user account; here it is necessary to select the green "V" in order to allow the user account to modify (write) the function status using the webserver: in the picture above the User account can change the heating setpoint 1 value; other settings are not editable; however, the Admin account has full access to all the parameters.

By means of the *Filters options* menu, available in the top area of the *Functions* window, the user can enable or disable all the permissions for reading /writing functions.

In the picture below, all the Write permissions are enabled for the User account.



The access data for the two accounts can be configured by entering the *Users* window. The admin account (Admin) has a higher priority for the management of all the functions added: it has full access to all functions (all permissions ticked).
The user account (User) has a lower priority: it has access to and control of only the ticked functions.

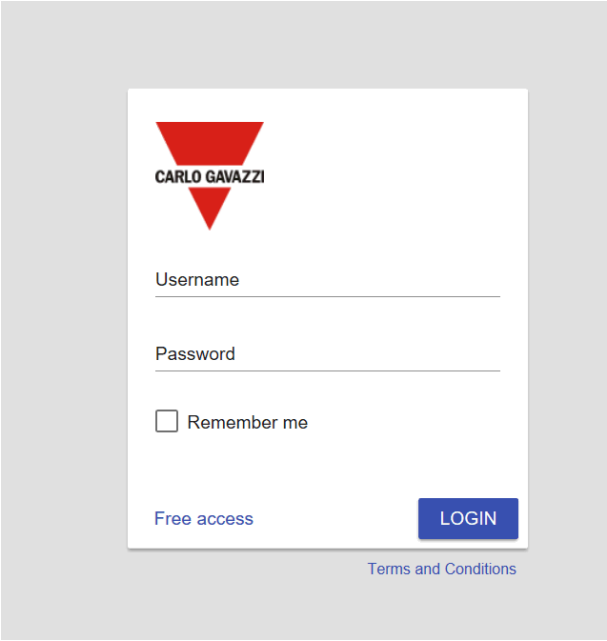


The username and password for the *Admin* account can only be changed by using the Tool, while those for the *User* account can be changed by means of the UWP 3.0 tool and the webserver.

N.B. If the webserver is accessed with an Internet Explorer release earlier than 10, the settings pages may not be visualised properly.

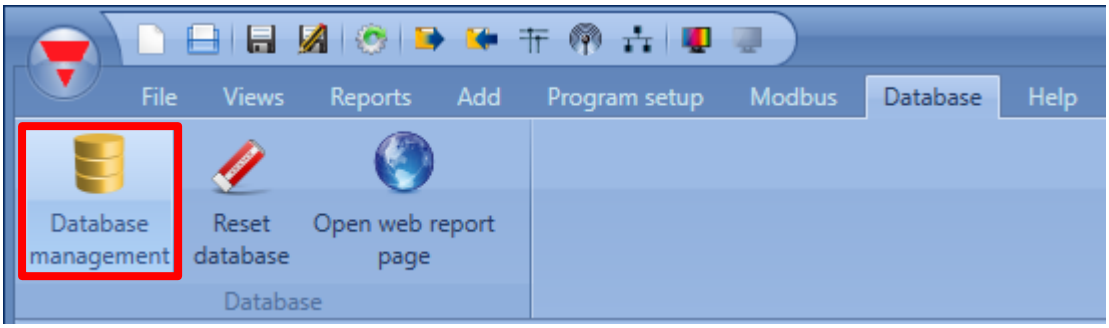
All values are shown in the graphs with the resolution of 0.1.

When the Web server account is created and the configuration downloaded into the UWP 3.0, the user can access it by typing the controller IP address as URL in a browser: the login page of the webserver is shown.

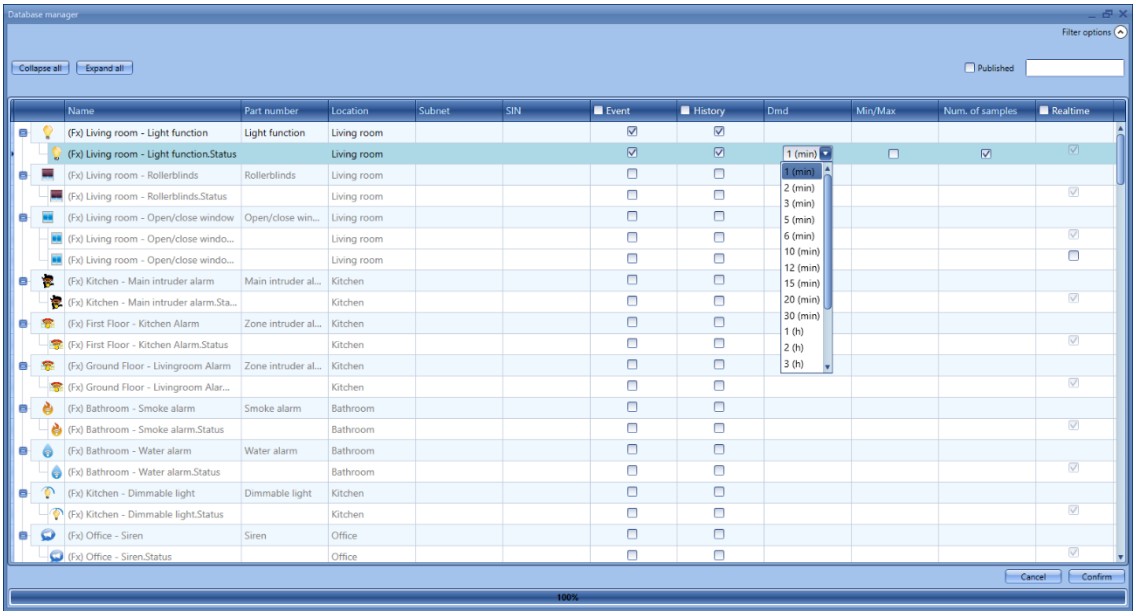


15 How to configure the database

The user can access the database by selecting *Database* and then *Database management.*



The *Database manager* window will appear:



The *Database manager* window allows the user to add the signals, such as analogue and digital signals and function statuses, to the Database of the UWP 3.0. For each signal there are different options to store its value to the Database, as described below:

Event
When the check-box is checked for a variable, each time it changes its status, the change is logged into the database. For example, the user can record a relay output and count the times it is switched On/Off. All logged values can be exported into a *Report* file (see also *Save events* data).

N.B: the analogue signals should be handled with care since the analogue signals usually change their value continuously: this means that the size of the database could increase excessively.

History
When the check-box is selected for a signal, the samples are logged into the database according to the *Dmd* value, defined individually in the *Dmd* column.

Dmd
For the selected signal, all logged variables are sampled and stored in the Database according to the time frame (from 1 minute to 24 hours).

Min/Max
When this option is enabled, the minimum and maximum values for the Dmd period are stored in the database.

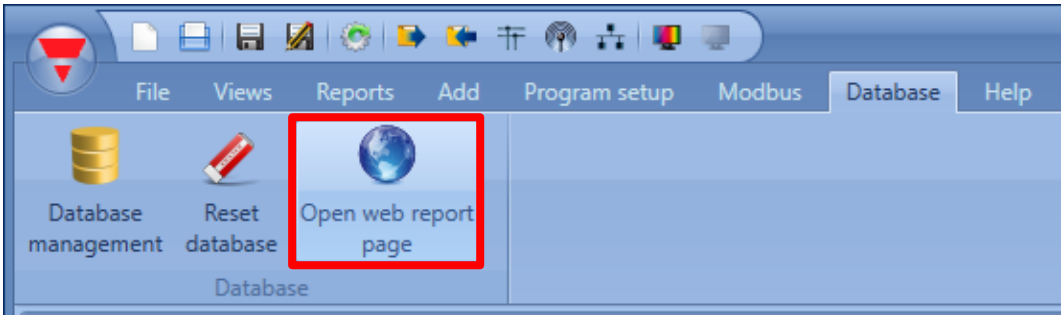
Num. of samples
When this option is enabled, the number of samples information is stored in the database.

Realtime
When this option is enabled, the selected signal is added in the webserver in the *Charts* and in the *Live signals* window in the UWP 3.0 Tool, even if it is not used in the functions.
If the user wants to log analogue values and show them in the webserver, they must add the value in the database, then once the variable is added it is possible to select the signal from the webserver and create the charts.

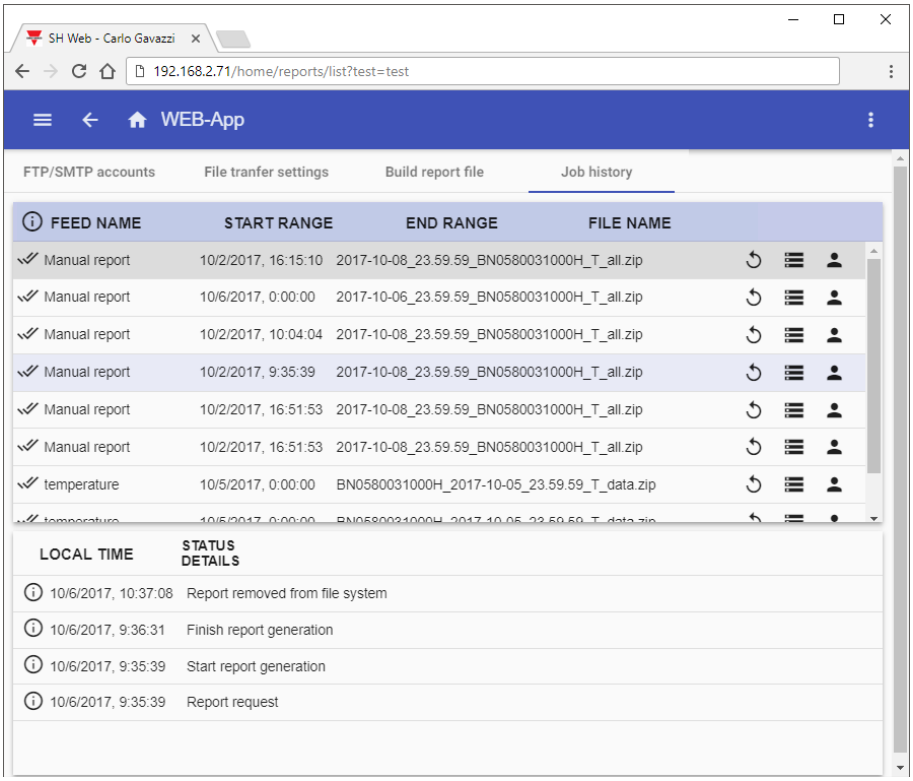
15.1 Database files

From the Database menu it is possible to reset the database or to access to the *Report* files.

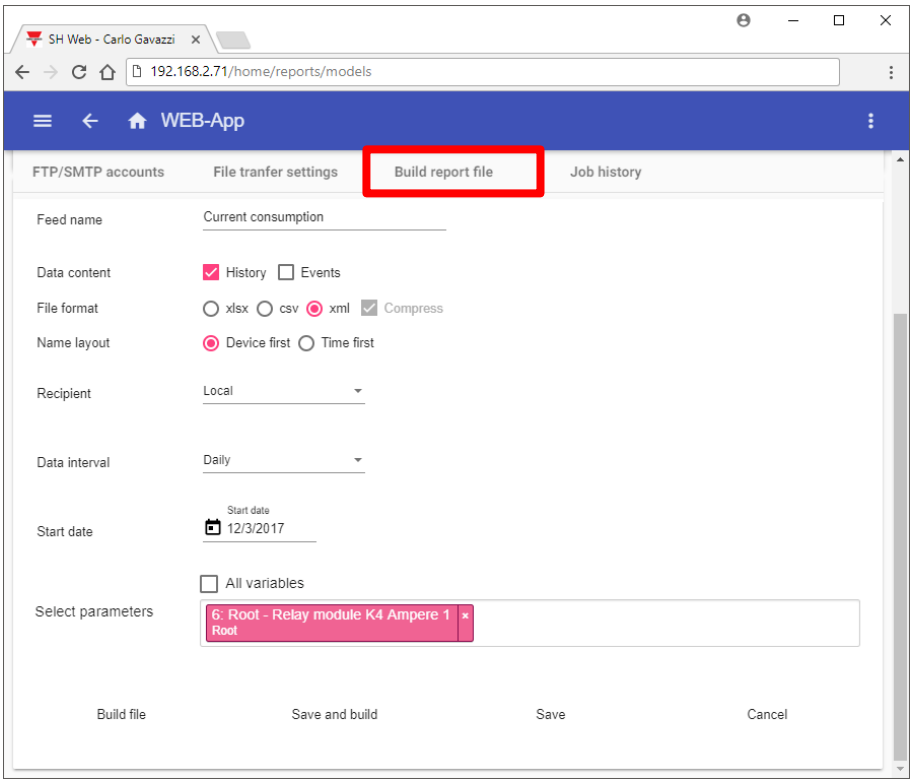
By clicking on *Open web report page*, as shown below, the user can get the access to the *Report* page on the integrated web-server.



The *Job history* page shows the list of the Reports which have already been generated: the user can download them in different formats, such as *.csv* file, *.xlsx* file or *.xml* file.

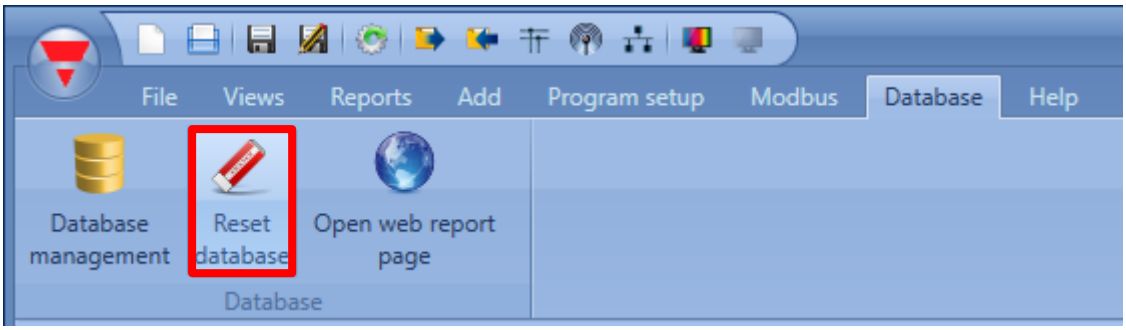


From the *Build report file* page of the *Reports* page, the user can generate new Reports manually, as shown in the example below:



Reset the database

If an error occurs in the database it can be solved by clicking on the *Reset database* button, as shown below:

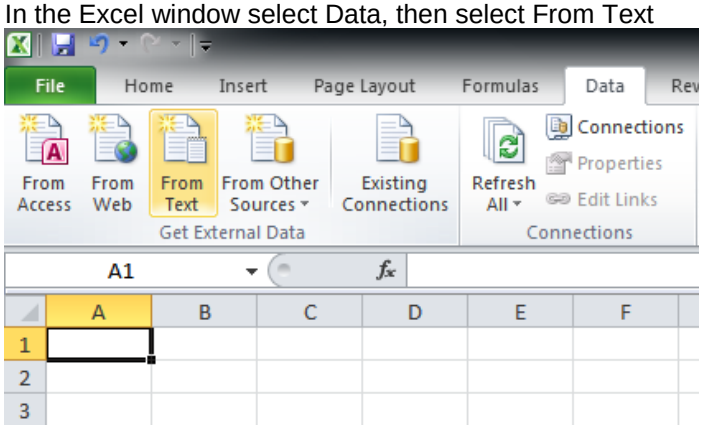


The database will be regenerated.

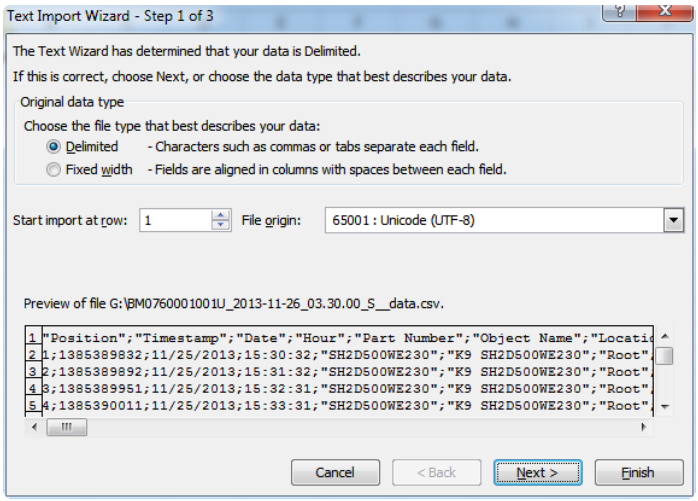
15.1.1 CSV file data format

To open a CSV file in EXCEL, please follow the steps below:

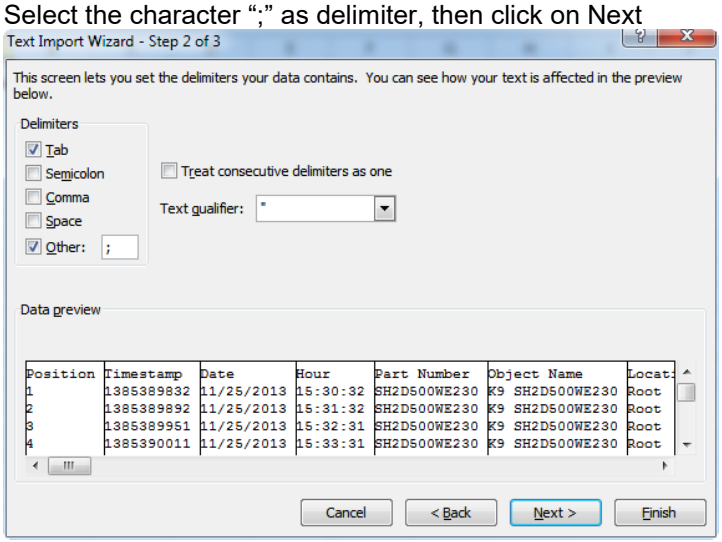
1. In the Excel window select Data, then select From Text



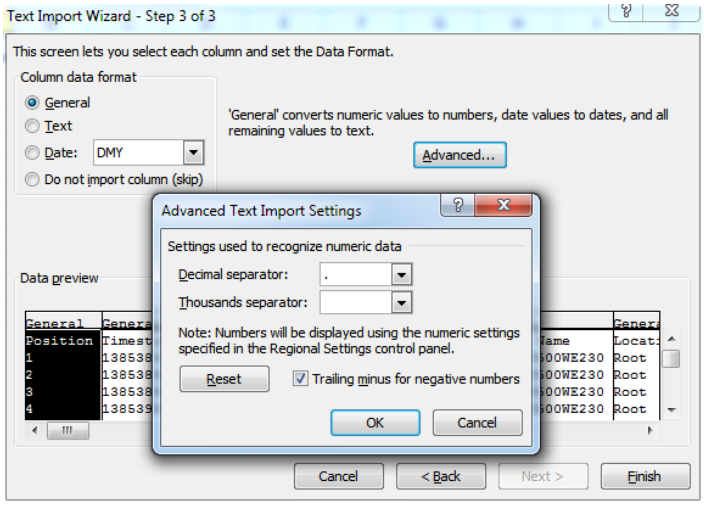
2. Select the CSV file and click on Next



3. Select the character ";" as delimiter, then click on Next



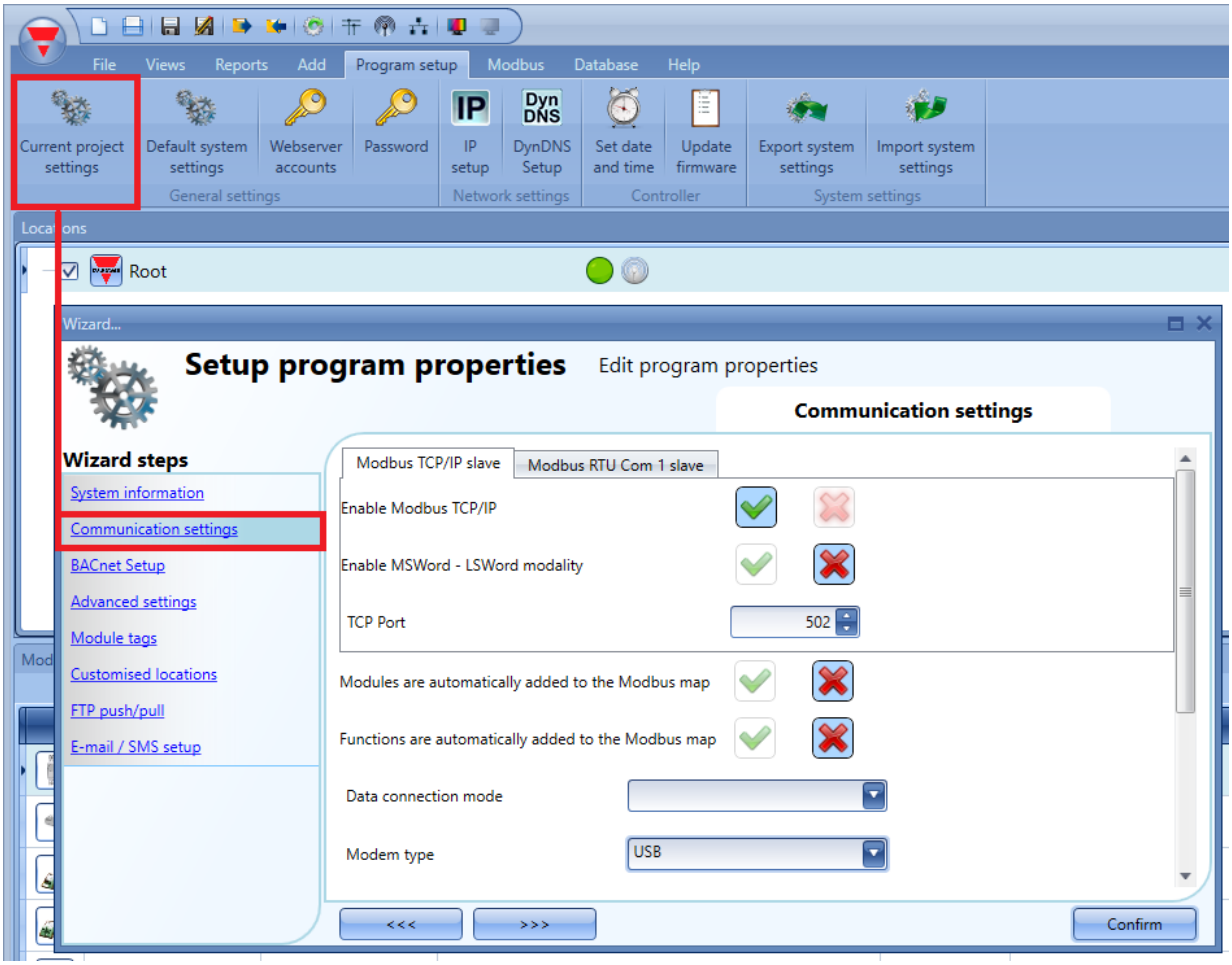
4. In the data click on Advanced and select dot "." as decimal separator, leaving empty the thousands separator



5. Click on OK and then on Finish

16 How to configure the UWP 3.0 as Modbus slave

Enable the Modbus protocol in the *Current project settings* menu in the field *Communication settings*.



Enable Modbus TCP/IP
When the green V is selected, the option is enabled and the system will answer the Modbus TCP/IP requests received on the selected TCP port.

Enable MSWord-LSWord modality
If this option is enabled, in the double word values, the most and least significant words are swapped. By default, the least significant word is sent.

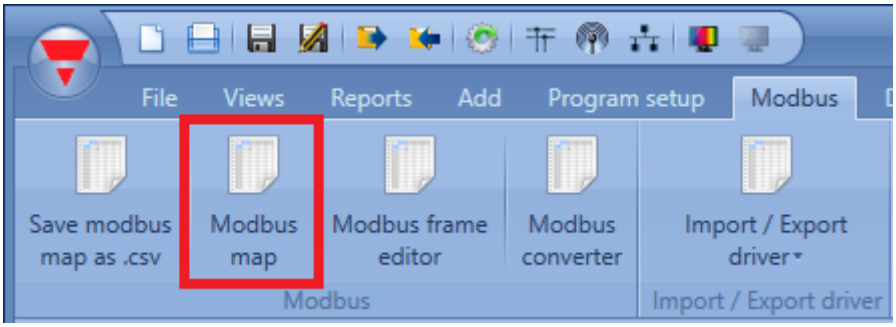
TCP port
The user can select the communication port for Ethernet connection (by default 502).

Modules are automatically added to the Modbus map
Each time a module is added into the project, the tool automatically adds the command descriptions and the corresponding tables to the Modbus map.

Functions are automatically added to the Modbus map
Each time a function is added into the project, the tool automatically adds the command descriptions and the corresponding tables to the Modbus map.

16.1 How to access the Modbus map

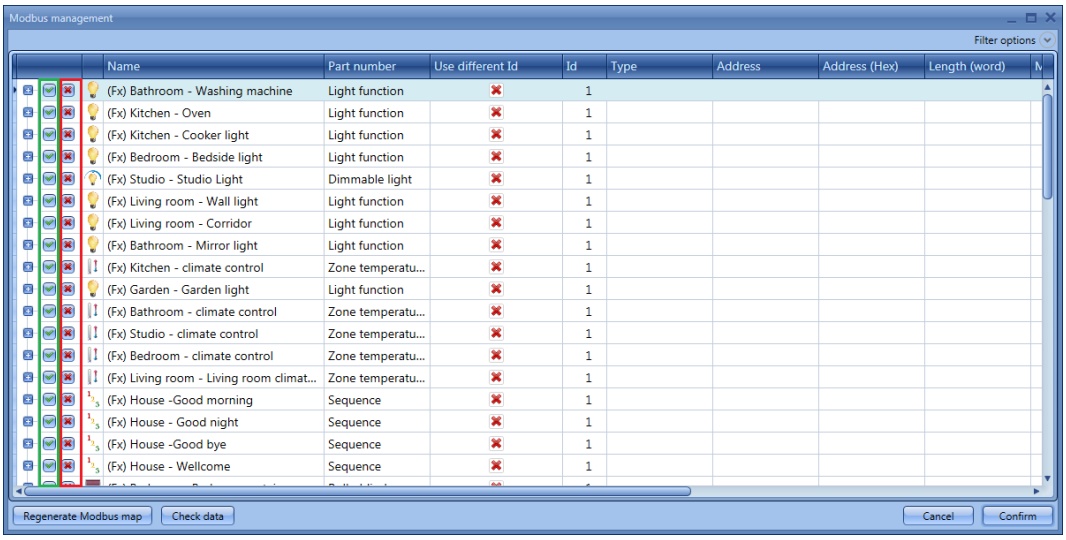
The user can access the *Modbus management* window by selecting *Modbus menu* and then clicking on the *Modbus map* icon.



The *Modbus management* window allows the user to configure the signals to read and write via Modbus. They are split into two groups:

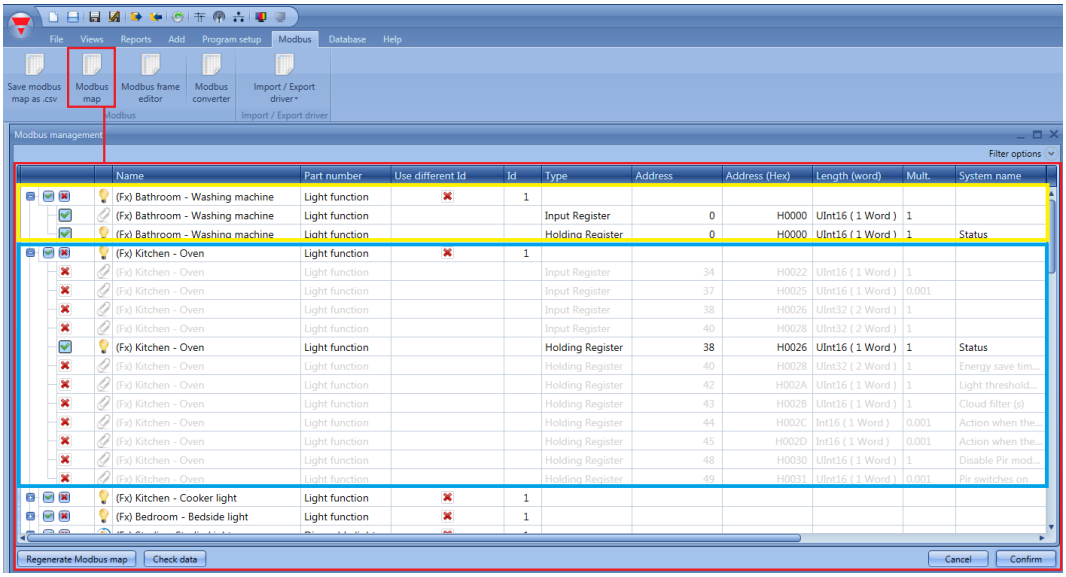
- Modbus signals related to modules;
- Modbus signals related to functions;

In the first column the user can select which functions/modules have to be added to the Modbus map: if the user clicks on the *small green V* icon, as shown in the green rectangle in the picture below, the selected modules/functions will be added to the Modbus map; otherwise if the user clicks on the small red cross, as shown in the red rectangle, the selected modules/functions will not be added to the Modbus map.



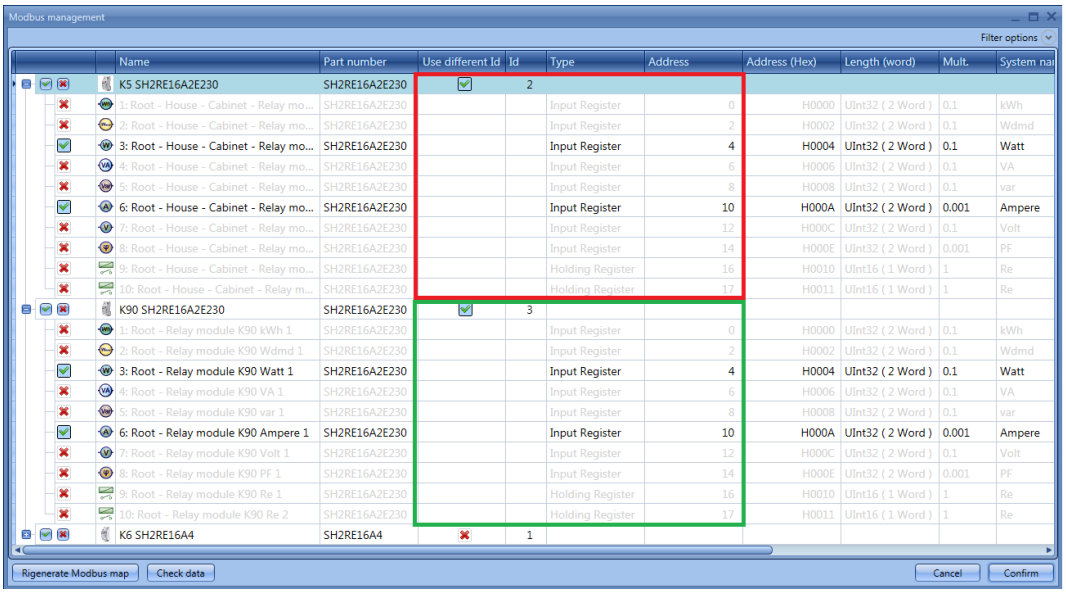
If the user clicks on the *expand/collapse* button, a selective selection of the signals (for modules and functions) can be done.

In the example shown below in the yellow rectangle, the function *(fx) Bathroom - Washing machine* is fully available in the Modbus map. While as shown in the blue rectangle, the function *(fx) Kitchen – Oven* has only the status signal available in the Modbus map.



16.1.1 How to change the default Modbus ID

If the user needs to have the values read from one type of module always at the same addresses, and he needs to change only the Modbus ID, the Modbus management menu has to be used.



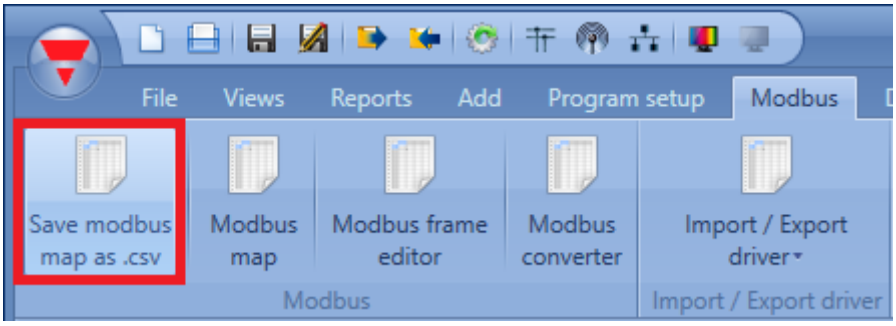
With the settings shown in the window above, the values read from the module SH2RE16A2E230 are always at the same addresses, and the user just needs to change the Modbus ID when he wants to read, for example, the power of the two modules.

The abstract of the Modbus map is shown below: the power is always at the address H0004, while the current can be read at the address H000A. The K5 module is addressed using Modbus ID 2, while the module K90 is addressed using Modbus ID 3.

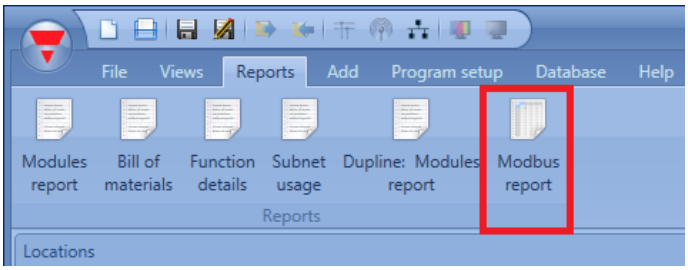
Type	Function name		Id	Address	Address (hex)	Type	Mult.	System name	Type
SH2RE16A2E230	3: Root - House - Cabinet - Relay module K5 Watt 1	IR	2	4	H0004	UInt32	0.1	Watt	W
SH2RE16A2E230	6: Root - House - Cabinet - Relay module K5 Ampere 1	IR	2	10	H000A	UInt32	0.001	Ampere	A
SH2RE16A2E230	3: Root - Relay module K90 Watt 1	IR	3	4	H0004	UInt32	0.1	Watt	W
SH2RE16A2E230	6: Root - Relay module K90 Ampere 1	IR	3	10	H000A	UInt32	0.001	Ampere	A

16.1.2 How to save the Modbus map

The Modbus map can be saved as a .csv file, that can be parsed very easily, by clicking on the icon *Save modbus map as .csv* in *Modbus* menu.
This Modbus map file can be directly read by the configuration software of the BTM-Tx-24 display.



All the Modbus addresses are also reported in *Modbus report* pdf file, in the *Reports* menu (it shows the same info as the .csv file but in pdf format).

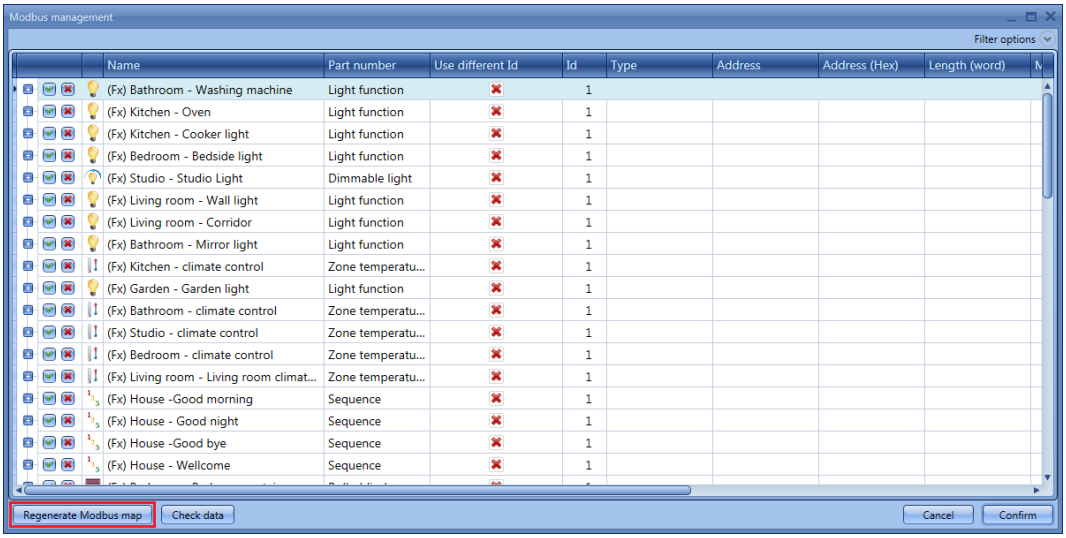


By clicking on the icon *Modbus report*, the Modbus table with all the mapped values is shown (see picture below).

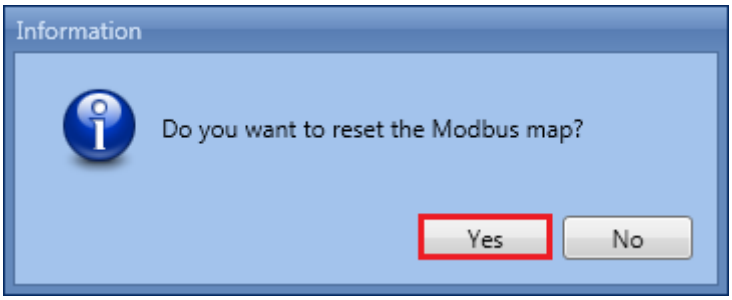
Type	Function name		Id	Address	Address (hex)	Type	Mult.	System name	Type
Dimmable light	(Fx) Studio - Studio Light	HR	1	152	H0098	UInt16	1	Status	--
Dimmable light	(Fx) Studio - Studio Light	HR	1	154	H009A	UInt32	1	Energy save delay (s)	s
Dimmable light	(Fx) Studio - Studio Light	HR	1	156	H009C	UInt16	1	Light threshold (Lux)	Lux
Dimmable light	(Fx) Studio - Studio Light	HR	1	157	H009D	UInt16	1	Cloud filter (s)	s
Dimmable light	(Fx) Studio - Studio Light	HR	1	160	H00A0	Int16	1	Scenario when the light gets belowthe threshold (%)	--
Dimmable light	(Fx) Studio - Studio Light	HR	1	161	H00A1	Int16	1	Scenario when the light gets abovethe threshold (%)	--
Dimmable light	(Fx) Studio - Studio Light	HR	1	163	H00A3	Int16	0.1	Action when the light gets below the threshold	%
Dimmable light	(Fx) Studio - Studio Light	HR	1	164	H00A4	Int16	0.1	Action when the light gets above the threshold	%
Dimmable light	(Fx) Studio - Studio Light	HR	1	170	H00AA	UInt16	0.1	S2(%)	%
Dimmable light	(Fx) Studio - Studio Light	HR	1	171	H00AB	UInt16	0.1	S3(%)	%
Dimmable light	(Fx) Studio - Studio Light	HR	1	172	H00AC	UInt16	0.1	S4(%)	%
Dimmable light	(Fx) Studio - Studio Light	HR	1	173	H00AD	UInt16	0.1	S5(%)	%
Light function	(Fx) Living room - Wall light	IR	1	196	H00C4	UInt16	1	Diagnostic flag	--
Light function	(Fx) Living room - Wall light	HR	1	229	H00E5	UInt16	1	Status	--
Light function	(Fx) Living room - Corridor	IR	1	230	H00E6	UInt16	1	Diagnostic flag	--
Light function	(Fx) Living room - Corridor	IR	1	233	H00E9	UInt16	0.001	Lux interval name	Number
Light function	(Fx) Living room - Corridor	IR	1	234	H00EA	UInt32	1	Energy save timer	s

If all the addresses have to be reset and a new map is needed, the user has to click on the *Modbus management* icon and press *Regenerate Modbus map*, as shown in the picture below.

N.B: Every time a new calendar activity is added to the Local calendar for SmartLight, Switch and Program functions, the Modbus map has to be regenerated.



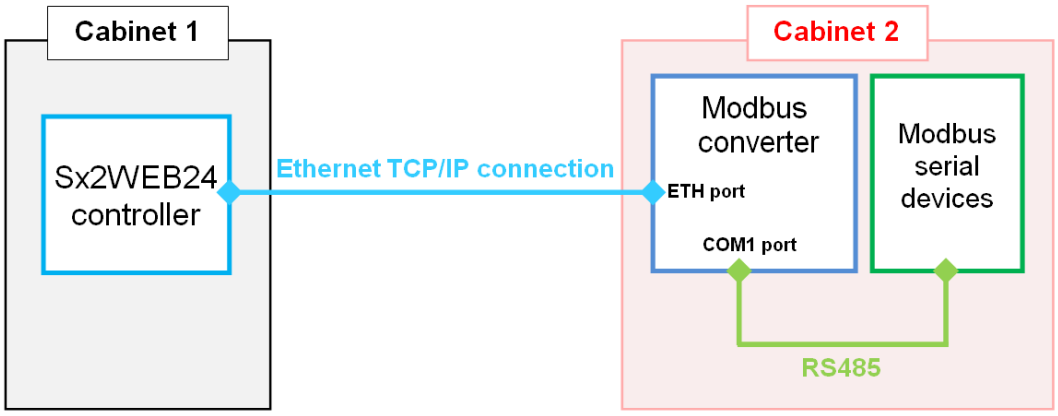
A window is shown and the user has to click on *Yes* to generate a new Modbus map.



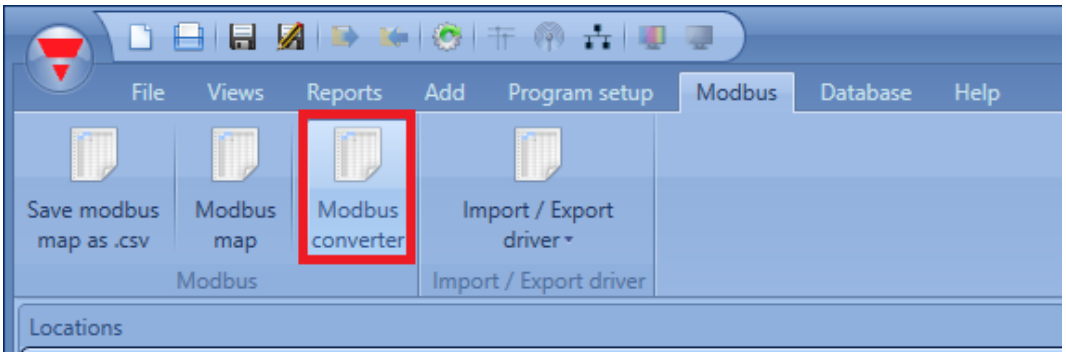
16.2 How to configure a Modbus converter device

The *Modbus converter* can be used to connect Modbus serial devices, such as energy meters, weather station, etc. to a UWP 3.0 controller via IP networks. In a building installation several Modbus serial devices can be located in cabinets that are far away from where the UWP 3.0 is placed. In this scenario a converter can be used as a gateway between the RS485 port and the Ethernet port.

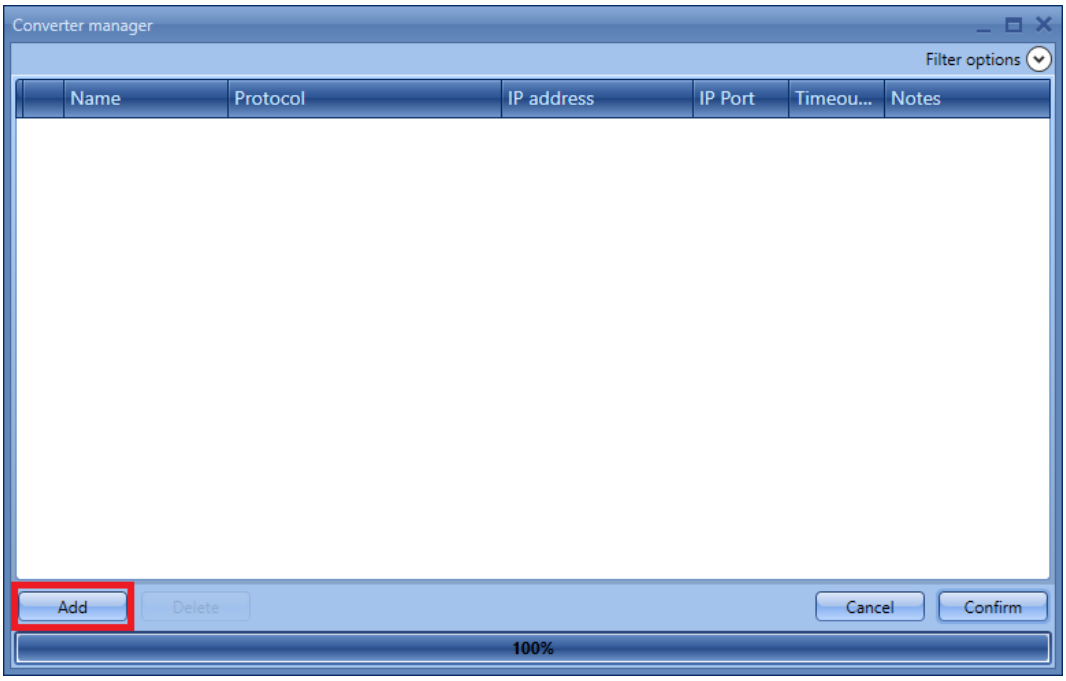
Example
In *Cabinet1* (black square), the installer has placed the UWP 3.0, while in Cabinet2 (red square), the installer has placed the serial devices connected via RS485 to the Modbus converter. The UWP 3.0 controller and the Modbus converter are connected to the same local IP network.



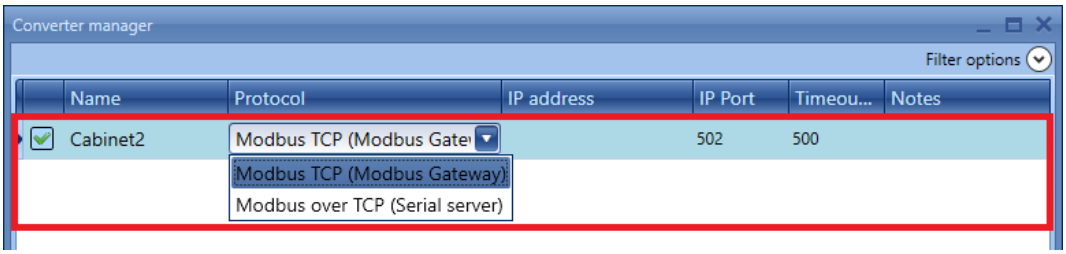
To configure the Modbus converter into the UWP 3.0 tool, the user has to click on *Modbus converter* in the *Modbus* menu, as displayed in the picture below:



The *Converter manager* window will appear, as presented in this picture:



To configure a Modbus converter device, the user has to click on *Add* and then fill in the necessary settings:



Name: in this field the user has to add the name of the converter that will be used in the UWP 3.0 tool to identify it

Protocol: in this field the user has to select the model of the Modbus converter used in the installation

Please refer to the table below for the device association:

Modbus converter	
Modbus TCP (Modbus Gateway)	CUTCP-3
Modbus over TCP (Serial server)	CUTCP-2

IP address, IP Port and Timeout: in these fields the user has to fill in the IP address, port and timeout of the Modbus converter.
N.B. Please refer to the user manual of the converter for the configuration instructions.

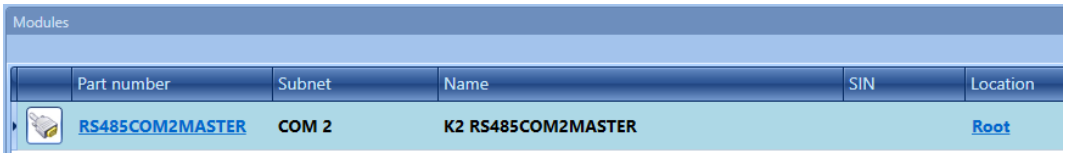
Notes: in this field the user can insert a text information about the Modbus converter, i.e. the place where it is installed.

By clicking on *Confirm*, the Modbus converter configuration will be saved.

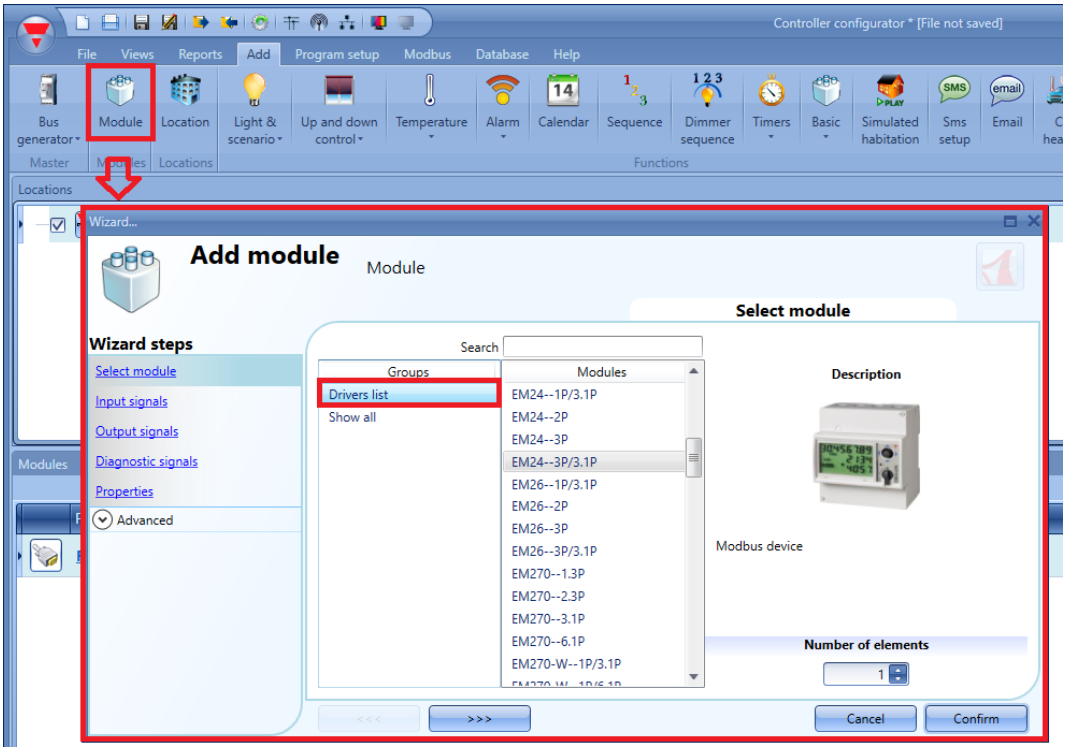
If the user selects one converter and clicks on *Delete*, the selected module will be removed from the *Converter manager* window.

16.2.1 How to configure a serial device connected to a Modbus converter

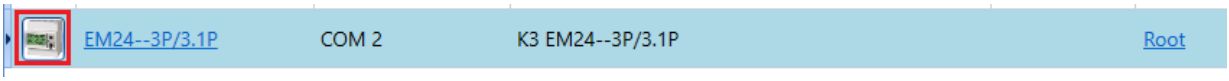
To configure a Modbus serial device, a bus generator RS485COMxMASTER has to be added in the *Modules* window.



Then, the user has to click on *Add* and then on *Module* and select the serial module from the *Driver list* menu, as shown in the picture below:

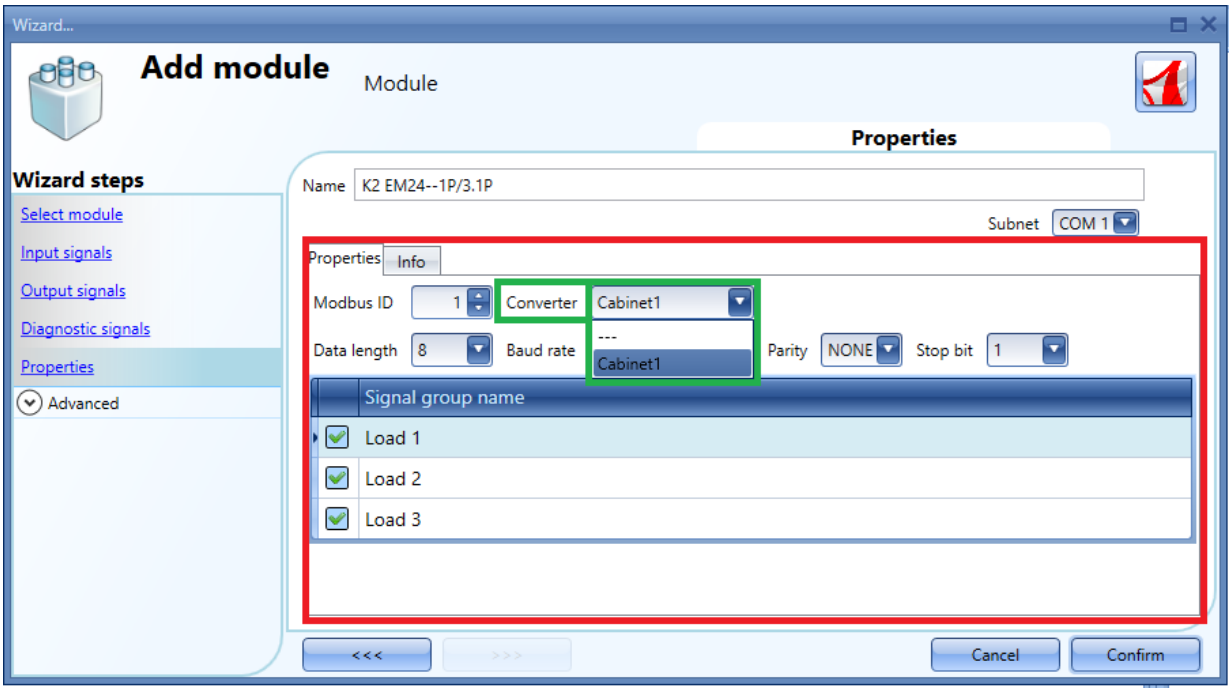


To select the Modbus converter connected to the Modbus serial device, click on the picture of the module:



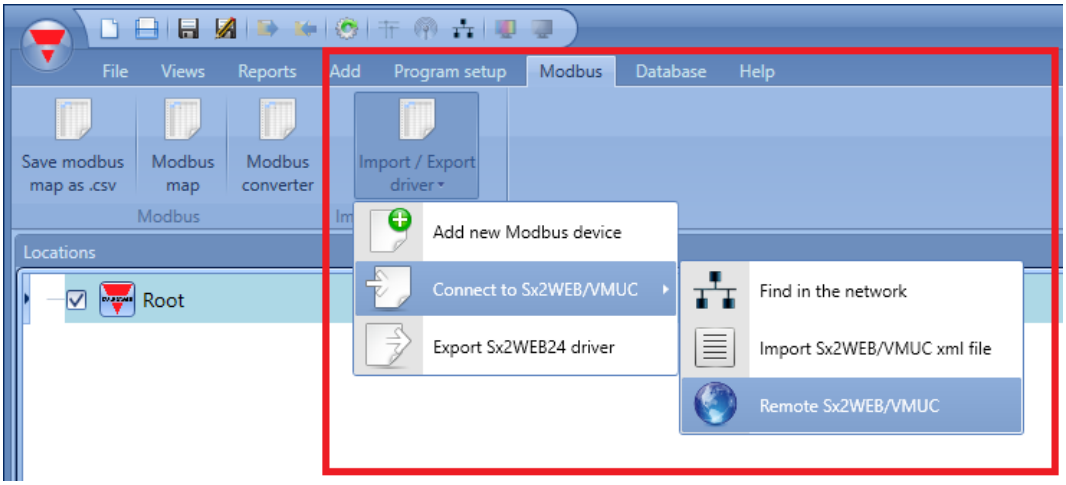
The configuration wizard will appear, where the serial parameters can be configured: Modbus ID, Data length, Baud rate, Parity, number of Stop bits.

In the *Converter* field of the *Properties* window, the user can choose the Modbus converter, as displayed in the green rectangle in picture below:



17 How to connect to VMUCs and UWP 3.0s

Any UWP 3.0 can be a Modbus master which reads and writes values from/to other UWP 3.0s or reads values from VMUCs.
The maps can be imported from the *Modbus* menu of the UWP 3.0 Tool, in the *Import / Export driver* field where the functions designed to extend the communication to VMUCs and other UWP 3.0 controllers connected to the Ethernet network are grouped. Please refer to the picture below.



There are three different scenarios according to the type of connection available:

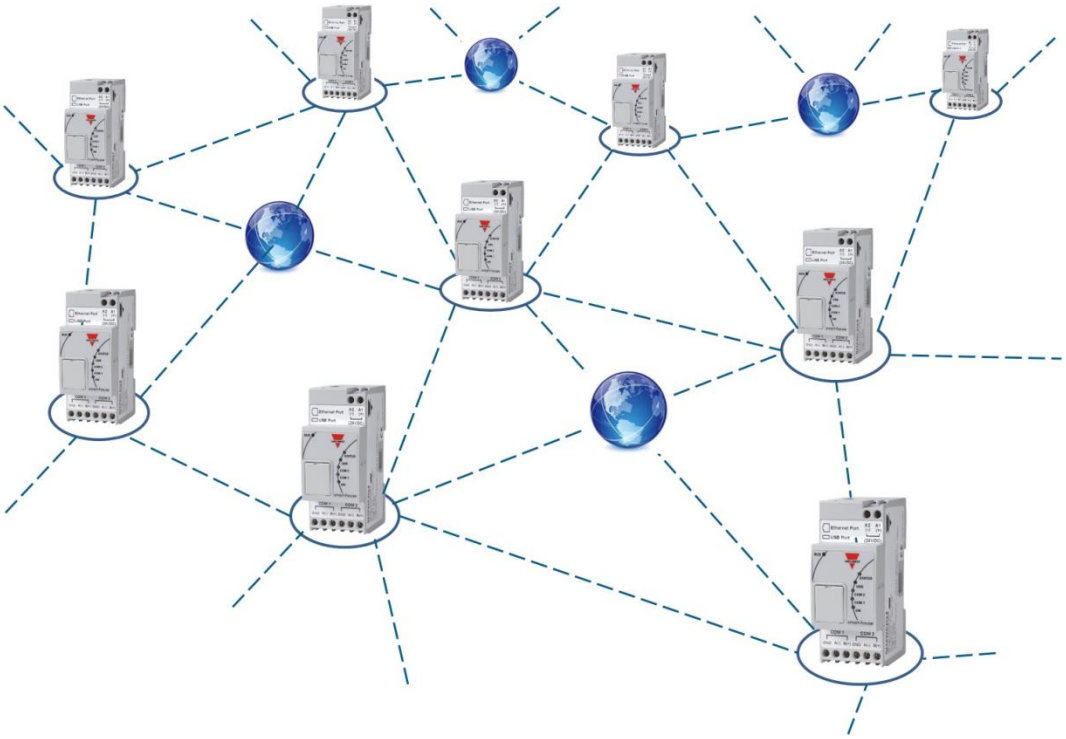
1) *The VMUCs and the UWP 3.0 controllers* are all connected to the same local network: in this situation the map of the variables (modules/functions) can be directly imported via the TCP/IP protocol;

2) The configuration is generated off-line: an xml file is imported containing the map of the variables of the slave device;

3) The master and the slaves are not connected to the same network: the map of the variables can be automatically read, by typing the IP address of the remote device;

However, the variables of the slave device are imported, they can be used by the master device in any logic, in the same way as local values.

Each UWP 3.0 can be connected to many VMUC / UWP 3.0 controllers, creating a kind of network, as shown in the example below.



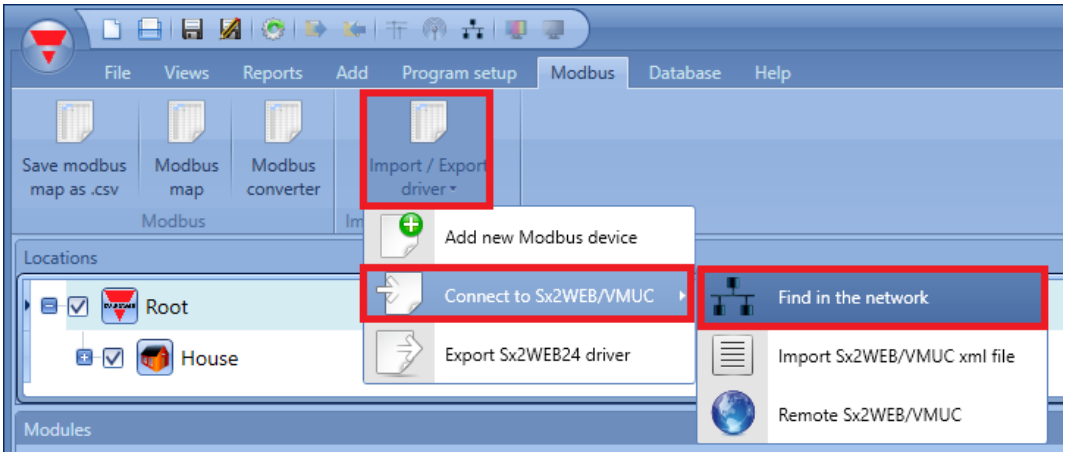
Each UWP 3.0 can be a Modbus master and a Modbus slave device at the same time.
An UWP 3.0 can read and write values from/to other Sx2WEB/UWP 3.0s, but can only read from VMUCs.

N.B. The TCP connection between an UWP 3.0 master and VMUC / Sx2WEB slave devices requires a firmware version equal to or higher than those shown in the table below:

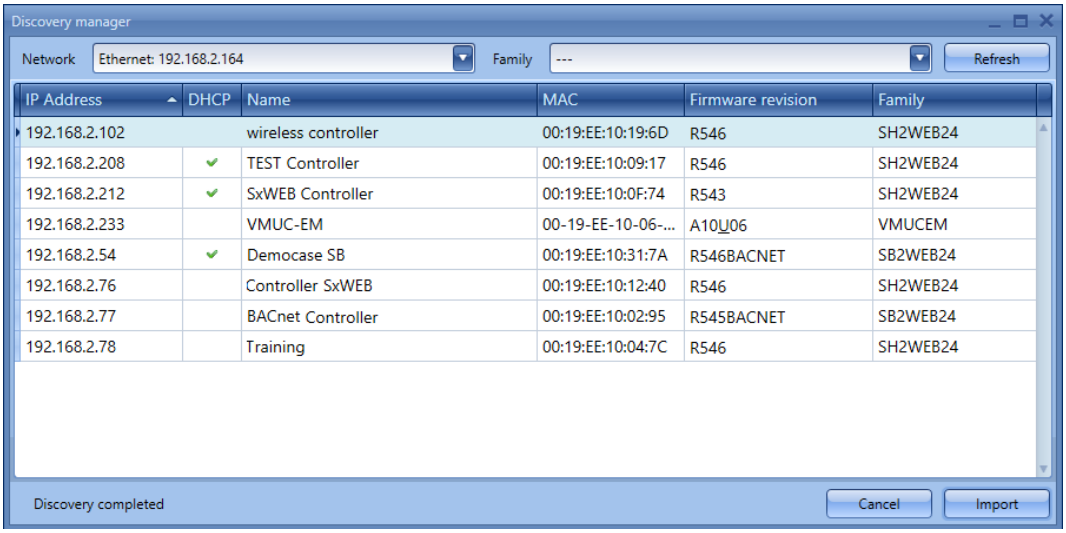
Firmware revision	
SH2WEB24	R54x
SB2WEB24	R54x
VMUCEM	A10
VMUCPV	A10

17.1 How to connect to devices in the same network

In the scenario where VMUC and/or UWP 3.0 controllers *are* all connected to the same network, the user has to select *Import / Export driver* from the *Modbus* menu, click on *Connect to UWP 3.0/VMU and* then on *Find in the network*. In this way the map of variables can be directly imported via the TCP/IP protocol. Please refer to the picture below.

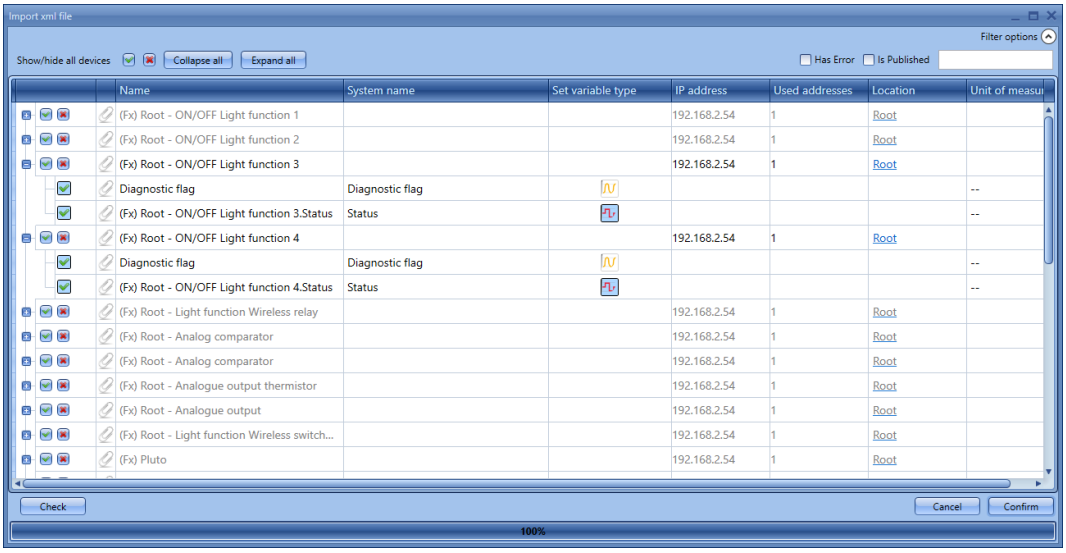


The *Discover manager* window will appear, showing all the VMUCs and UWP 3.0s available in the same network.
When the PC has more than one network card or has many IP addresses, it is possible to select the correct one from the list (it must be the same one as the UWP 3.0 master is connected to).
From the *Family* field it is possible to filter the controllers by the family they belong to (e.g. UWP 3.0, VMUCEM, etc…).



IP Address	DHCP	Name	MAC	Firmware revision	Family
192.168.2.102		wireless controller	00:19:EE:10:19:6D	R546	SH2WEB24
192.168.2.208	✔	TEST Controller	00:19:EE:10:09:17	R546	SH2WEB24
192.168.2.212	✔	SxWEB Controller	00:19:EE:10:0F:74	R543	SH2WEB24
192.168.2.233		VMUC-EM	00-19-EE-10-06-...	A10U06	VMUCEM
192.168.2.54	✔	Democase SB	00:19:EE:10:31:7A	R546BACNET	SB2WEB24
192.168.2.76		Controller SxWEB	00:19:EE:10:12:40	R546	SH2WEB24
192.168.2.77		BACnet Controller	00:19:EE:10:02:95	R545BACNET	SB2WEB24
192.168.2.78		Training	00:19:EE:10:04:7C	R546	SH2WEB24

By double clicking on the device, or selecting it and clicking on *Import*, the list of available variables for the selected controller (map of variables) will be loaded.

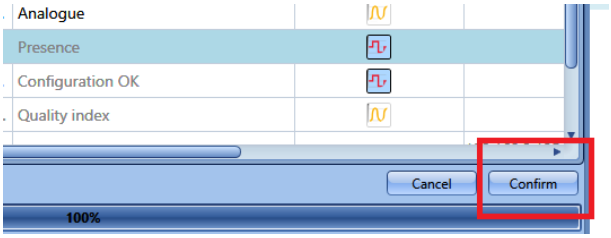


In the first column the user can select which functions and/or modules have to be added from the map of the Slave device: if the user clicks on the *small green V* icon, the selected modules/functions will be imported into the project; otherwise, if the user clicks on the small *X* cross, the selected modules/functions will not be imported. By clicking on the *Filter options* button, a selection of the variables is available.

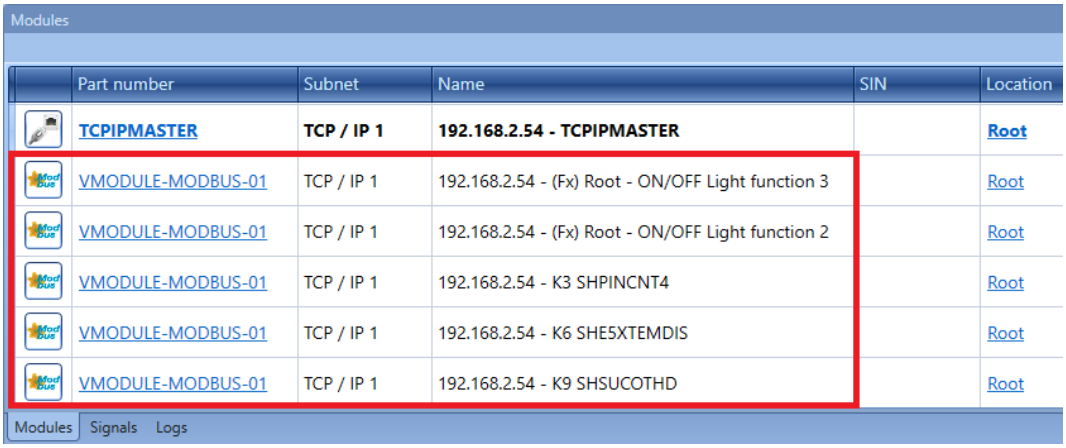
N.B: The Modbus TCP/IP slave capability has to be enabled in the slave device and the functions/modules to be shared also have to be enabled.

If the *Modbus TCP/IP Slave* protocol is not enabled in the slave controller, a warning messages is displayed in the *Discovery manager* window, as shown in the example on the left. (see above)

When the functions and/or modules to be shared have been selected, click on *Confirm*.

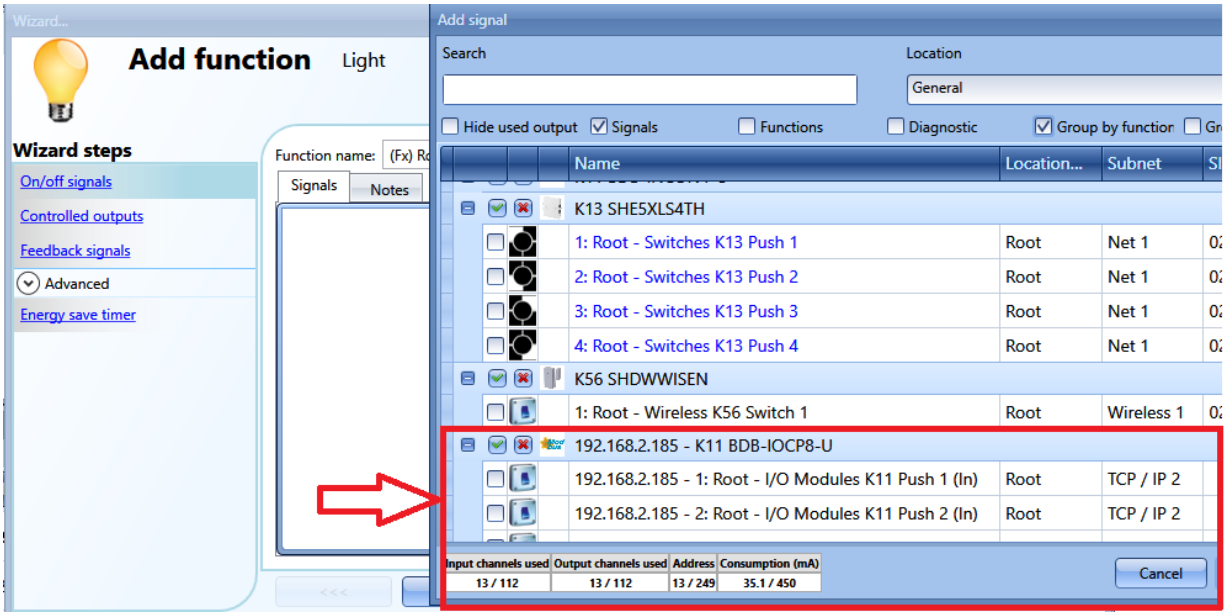


The selected variables will be added in the *Modules* window, indicating the IP address of the device they belong to, as shown in the red rectangle below.



All the imported modules/functions are treated in the same way as the modules/functions connected to the master device and can be used in any logic.

As an example, a *Light* function is shown below where inputs from the UWP 3.0 device with IP 192.168.2.185 are available.

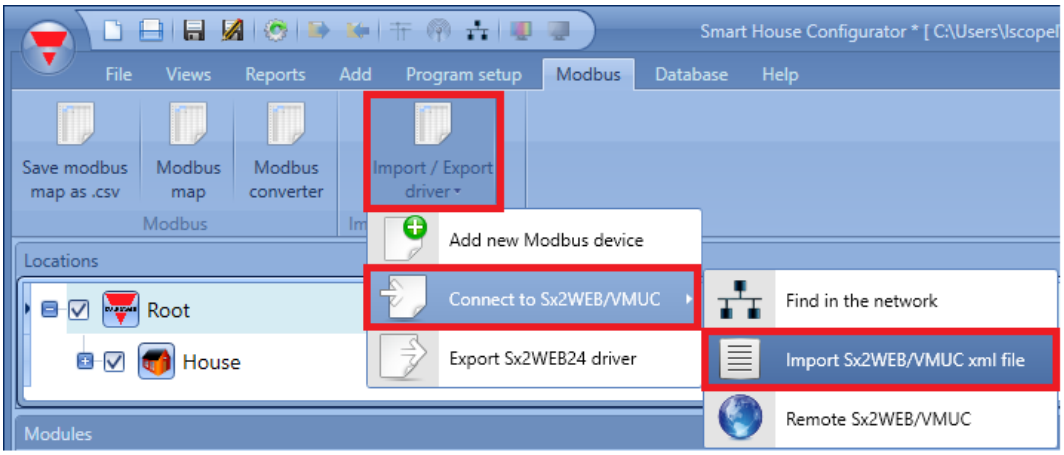


17.2 How to configure an offline device importing an xml file

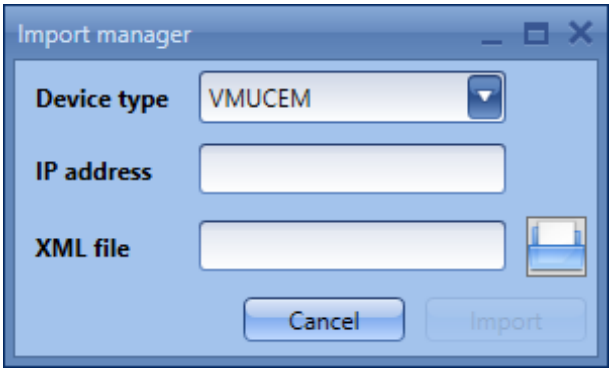
In the scenario where the user wants to connect an offline VMUC or UWP 3.0 which will be connected to the Ethernet at another time, the relevant xml file has to be imported.
This file has to be generated via the UWP 3.0 tool for the UWP 3.0 or via the VMUC webserver.
Please refer to the table below to generate the xml file of the slave device:

XML file creation	
Sx2WEB24	See *How to generate the xml file* of this manual
UWP 3.0	
VMUCEM	See the VMUC Instruction Manual *Web Server system for energy monitoring*

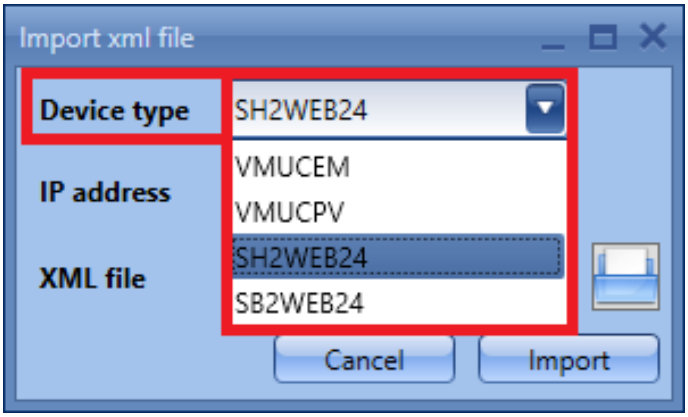
To import this, the user has to select *Import / Export driver* from the *Modbus* menu, click on *Connect to UWP 3.0/VMUC* and then on *Import UWP 3.0/VMUC xml file*.



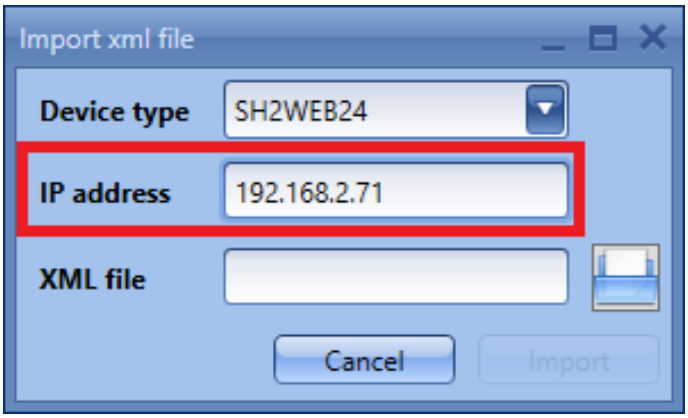
The *Import manager* window will appear, as shown in the picture below:



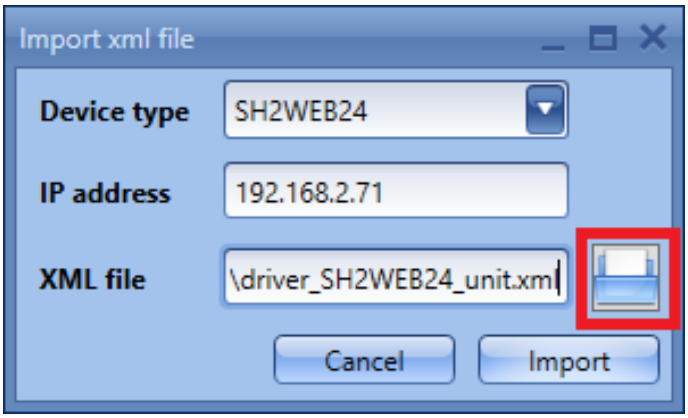
In the *Device type* field of the *Import manager* window the user has to select the device type to be imported.



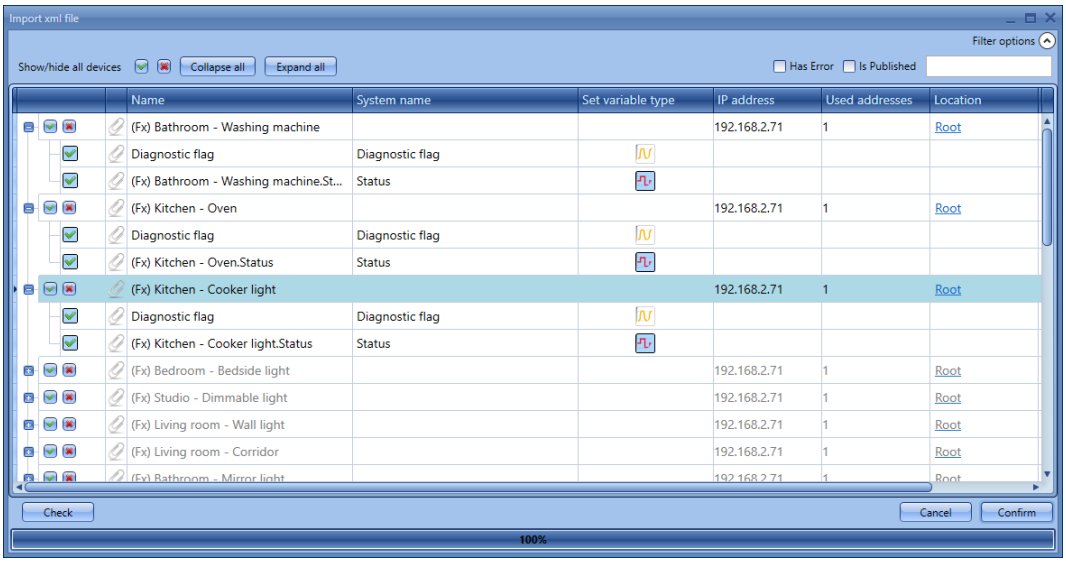
In the *IP address* field the user has to insert the IP address of the slave device.



In the *XML file* field the user has to locate the XML file previously generated by means of the VMUC or UWP 3.0 tool;

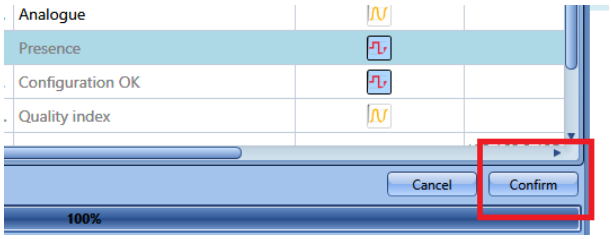


By clicking on *Import*, the list of available variables will be loaded, as shown below.

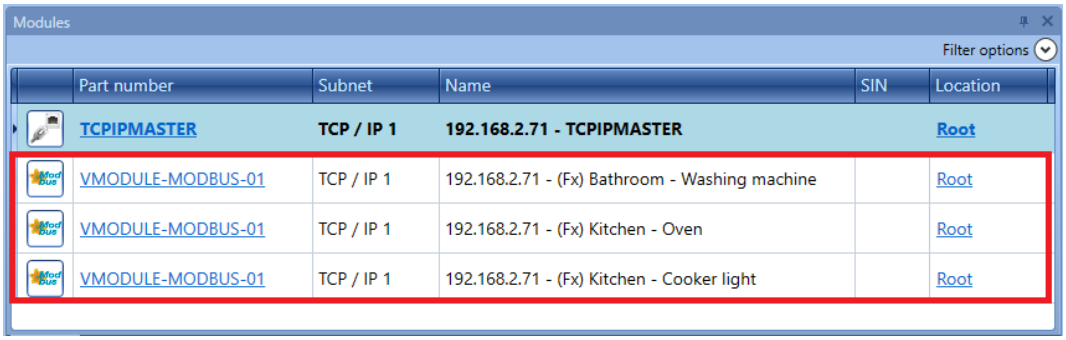


In the first column the user can select which functions and/or modules have to be added from the map of the Slave device: if the user clicks on the *small green V* icon, the selected modules/functions will be imported into the project; otherwise, if the user clicks on the small *X* cross, the selected modules/functions will not be imported. By clicking on the *Filter options* button, a selection of the variables is available.

Click on Confirm to import these.

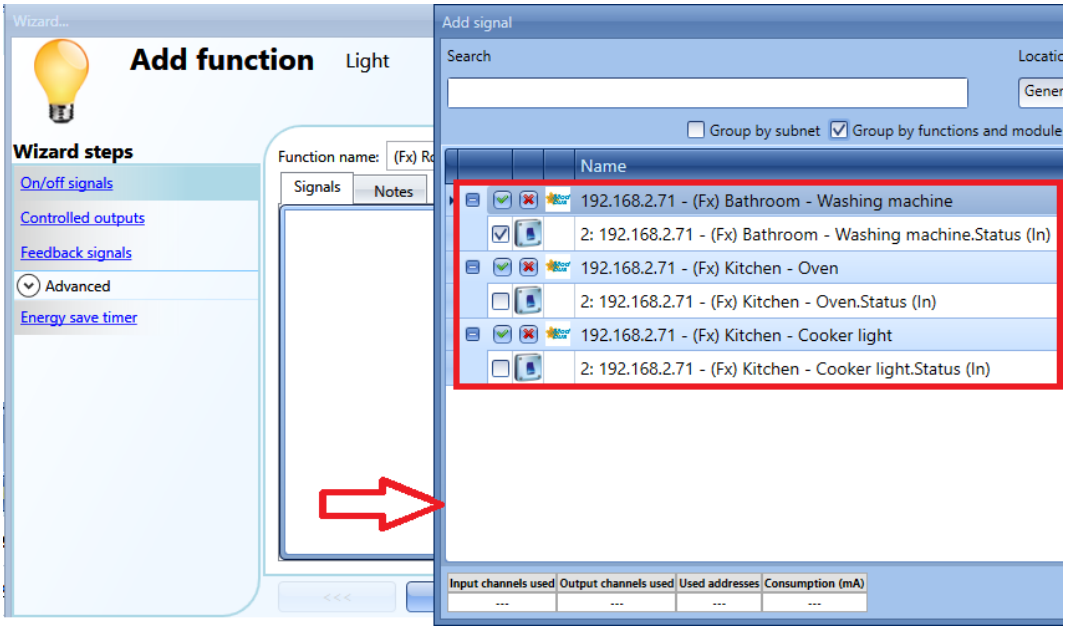


The selected variables will be added in the *Module*s window, indicating the IP address of the device they belong to, as shown in the red rectangle below.



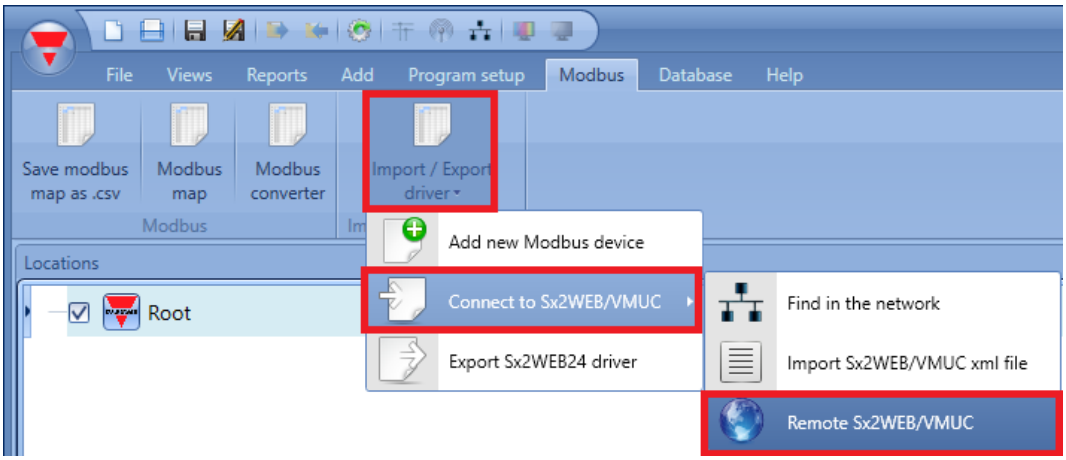
All the imported modules/functions are treated in the same way as the modules/functions connected to the master device and can be used in any logic.

As an example, a *Light* function is shown below where inputs from the UWP 3.0 device with IP 192.168.2.71 are available.

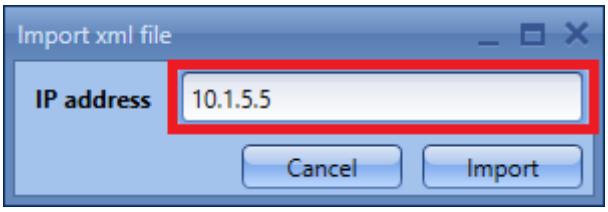


17.3 How to import the map of variables from a remote controller

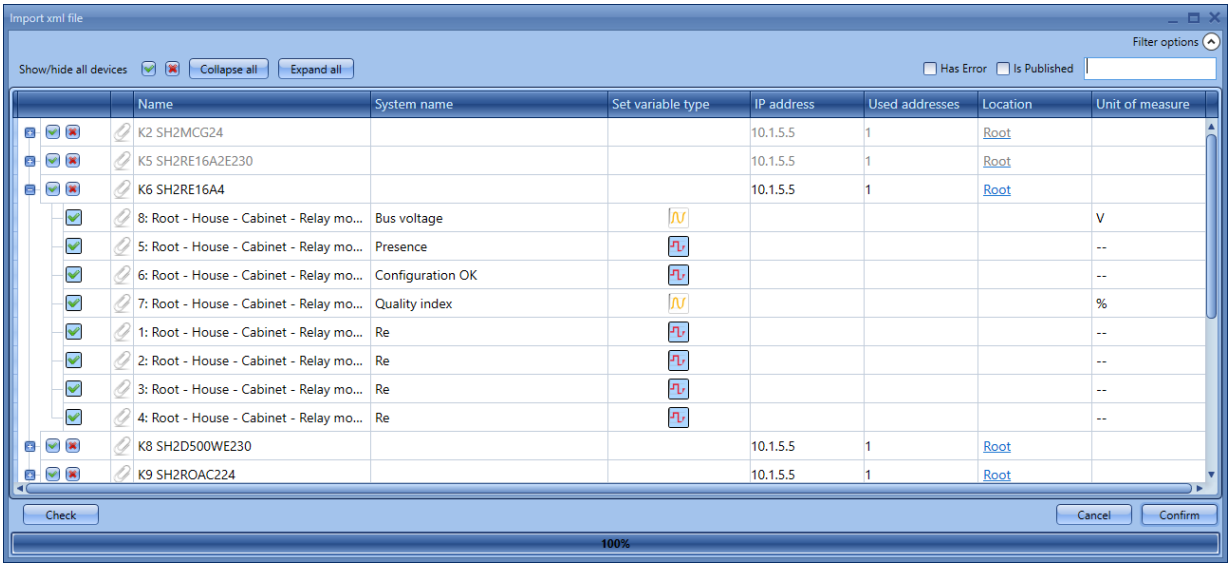
If the slave VMUC or UWP 3.0 is not connected to the same network as the master, select *Import / Export driver* from the *Modbus* menu, click on *Connect to UWP 3.0/VMUC* and then on *Remote UWP 3.0/VMUC*. In this scenario, the slave device cannot be discovered, but by adding the IP address the list of available variables can be imported automatically.



After clicking on *Remote UWP 3.0/VMUC*, the *Import manager* window will appear: insert the IP address of the remote slave device and click on *Import:* If the IP address of a remote device is not correct, the UWP 3.0 tool will prompt the following message in the *Log* window:
"Unable to import driver Modbus from *selected IP address*".
The user has to check the IP address and type it in correctly.



The list of available variables will be loaded, as shown below.



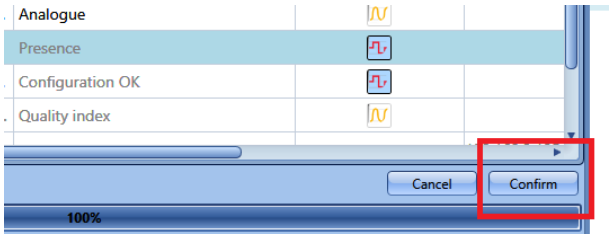
In the first column the user can select which functions and/or modules have to be added from the map of the Slave device: if the user clicks on the *small green V* icon, the selected modules/functions will be imported into the project; otherwise, if the user clicks on the small *X* cross, the selected modules/functions will not be imported. By clicking on the *Filter options* button, a selection of the variables is available.

Click on Confirm to import these.

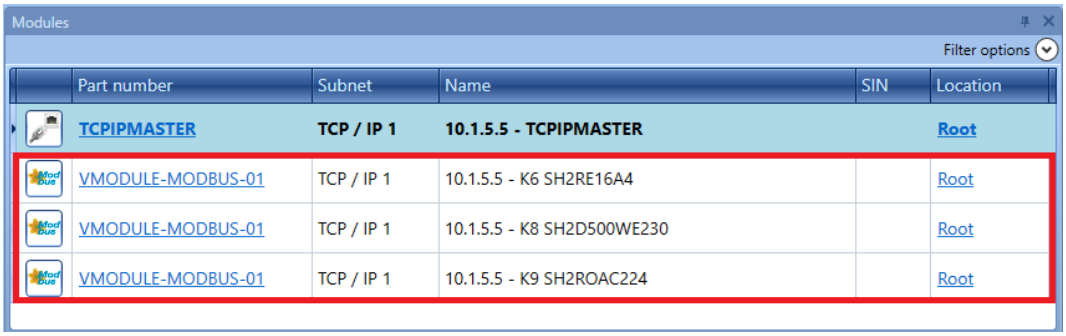
N.B: The Modbus TCP/IP slave capability has to be enabled in the slave device and the functions/modules to be shared also have to be enabled.

If the *Modbus TCP/IP Slave* protocol is not enabled in the slave controller, a warning messages is displayed in the *Discovery manager* window, as shown in the example on the left. (see above)

When the functions and/or modules to be shared have been selected, click on *Confirm*.

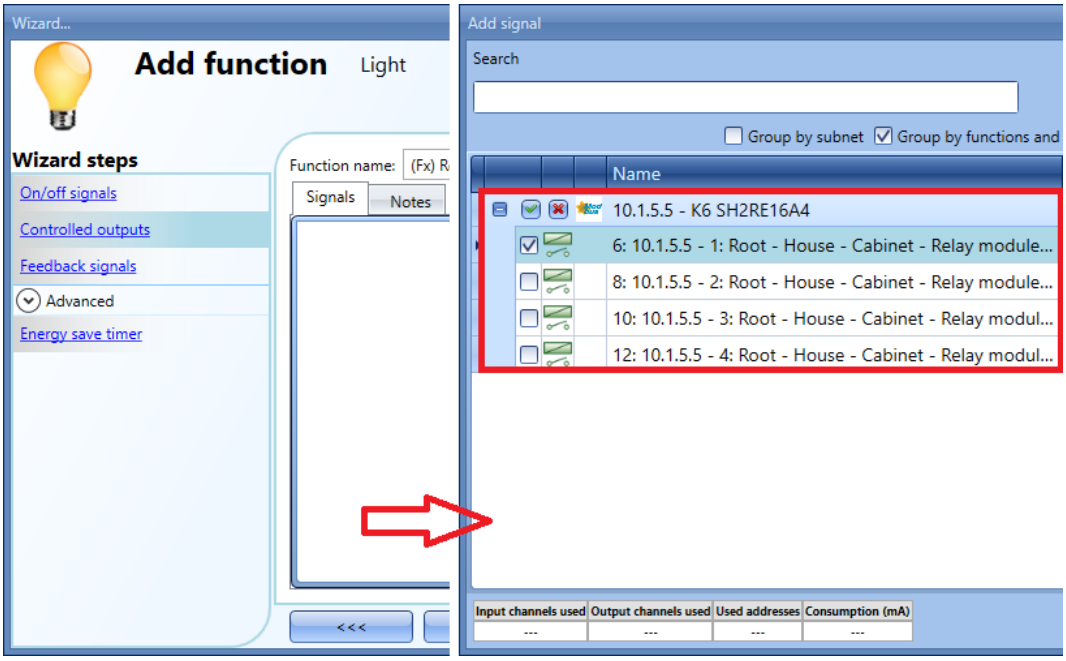


The selected variables will be added in the *Module*s window, indicating the IP address of the device they belong to, as shown in the red rectangle below.



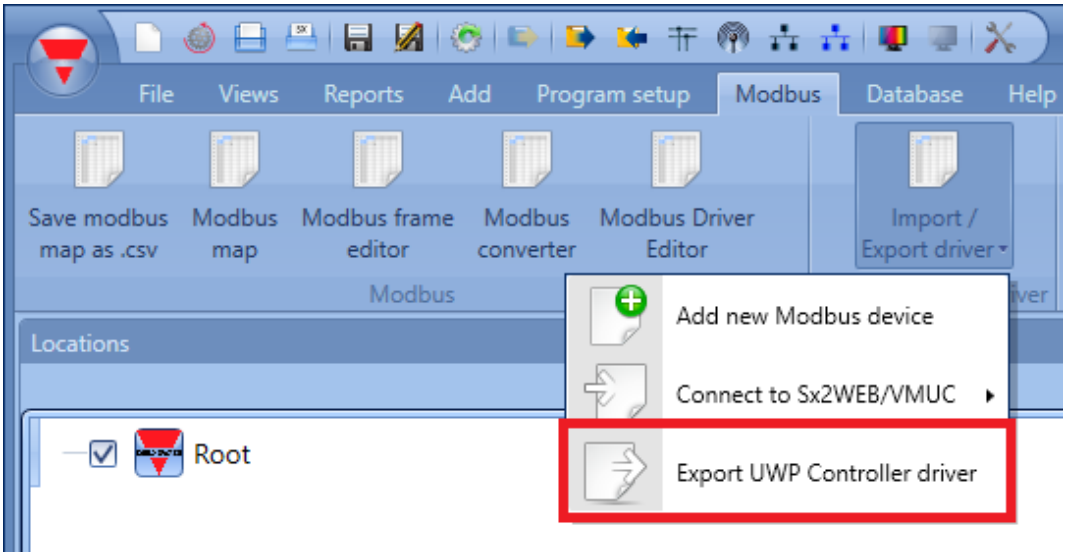
All the imported modules/functions are treated in the same way as the objects available (such as modules locally connected to the UWP 3.0 master and functions) and can be used in any kind of UWP 3.0 Tool logic (functions, database, etc..).

As an example, a *Light* function is shown below where inputs from the UWP 3.0 device with IP 10.1.5.5 are available.

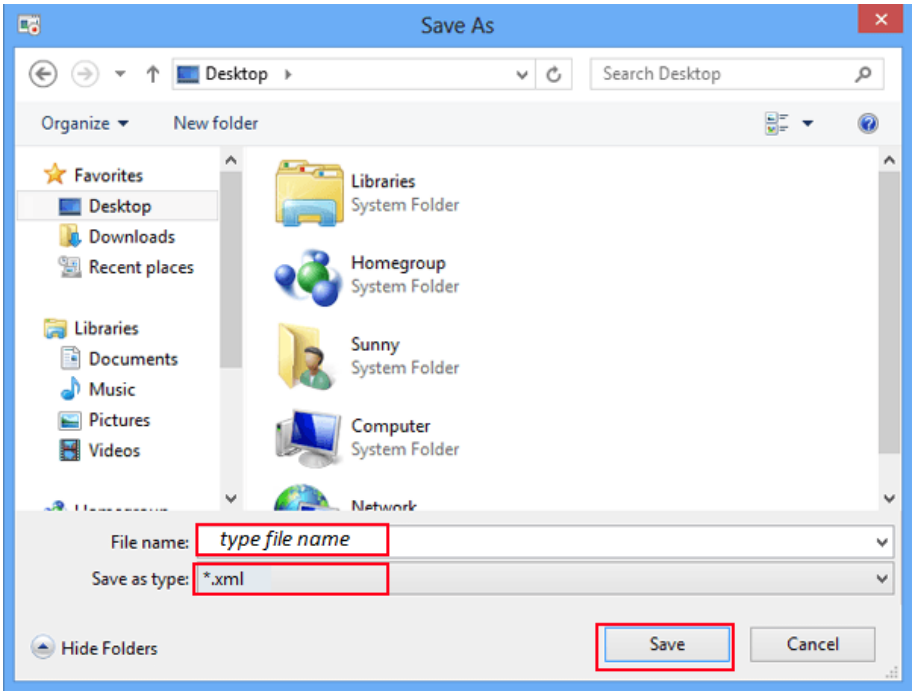


17.4 How to generate the xml file

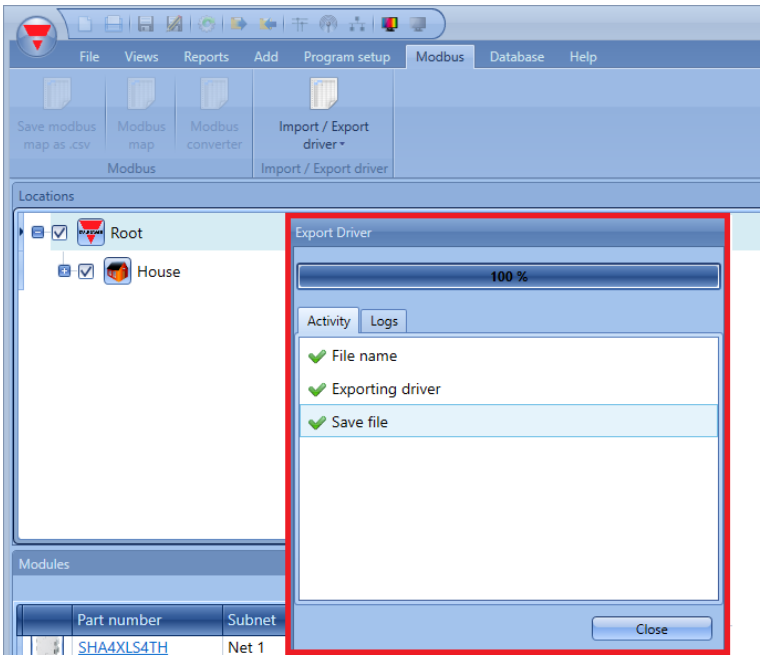
To export the map of the variables into an xml file, the user has to click on *Import / Export driver* in the *Modbus* menu and then on *Export UWP Controller driver*, as shown in the picture below.



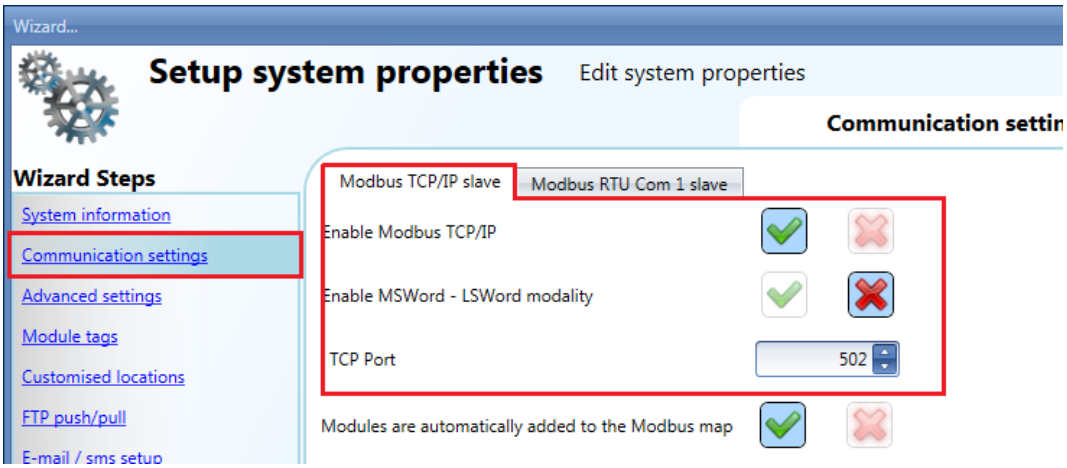
In the *Save as* window, select the folder where the file has to be saved.



The following window will confirm that the file has been saved correctly.

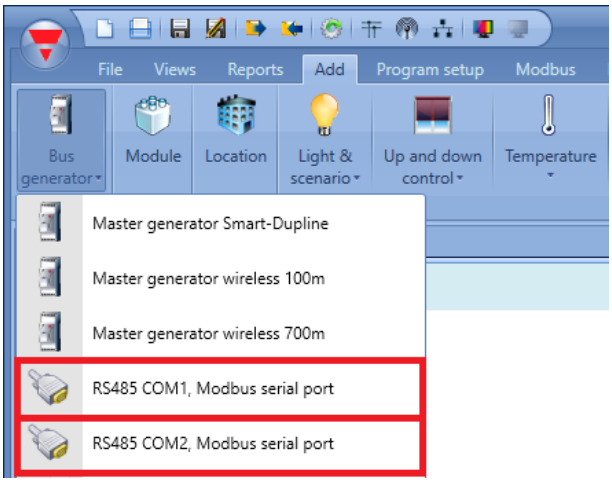


N.B. If a device has to be seen as a Modbus slave, this feature has to be enabled in the *Communication settings* of the project.

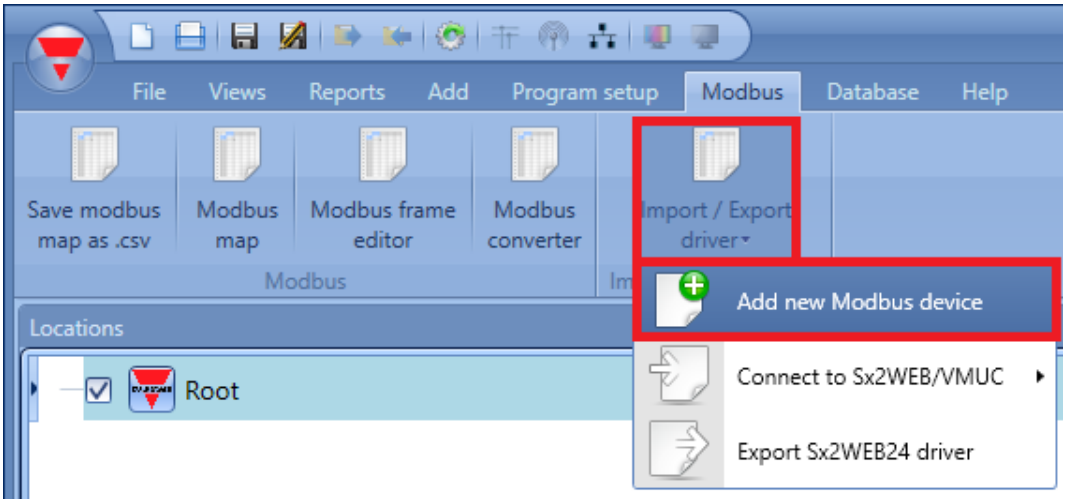


17.5 How to connect to generic Modbus devices

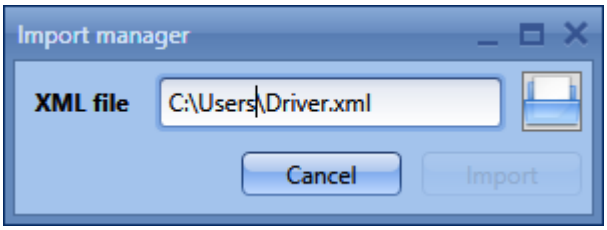
To connect a generic Modbus device to the project, a serial communication port has to be added to the project: click on *Add* and then on *Bus generator* and select the COM port which the generic Modbus devices are connected to.



To read data from any Modbus device, the user has to click on *Import / Export driver* in the *Modbus* menu, then on *Add new Modbus device*, as shown below.

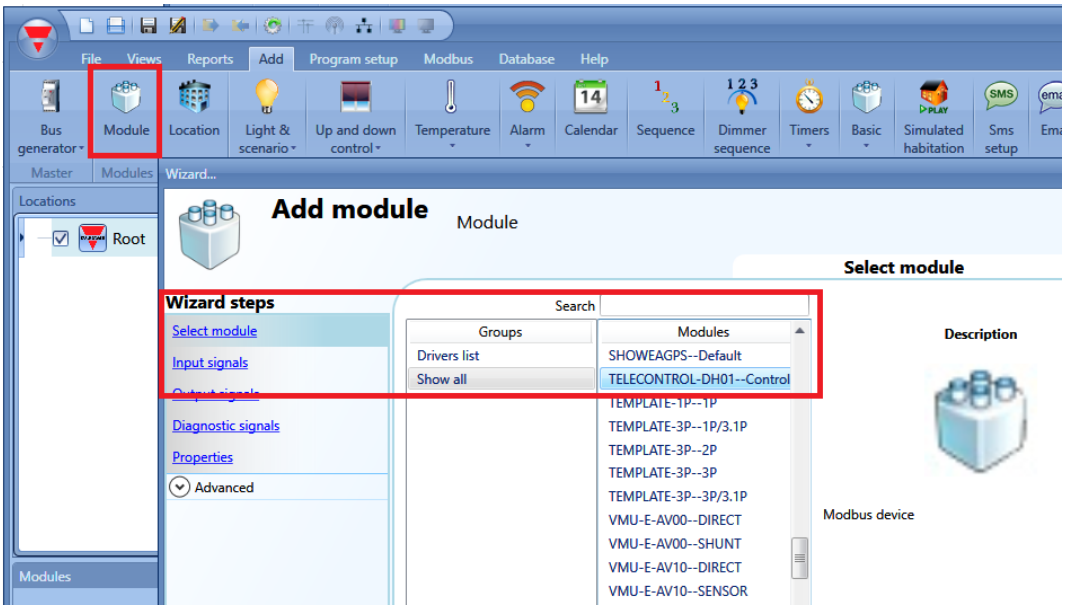


The *Import manager* window will appear: the user has to locate the driver previously generated as an xml file by means of VMUC:



The driver will be added to the UWP 3.0 tool module database and can easily be found in the *Add module* window.

A new line is added in the *Drivers list*, as shown below:



By clicking on *Confirm:* a new device will be added to the *Module* window.

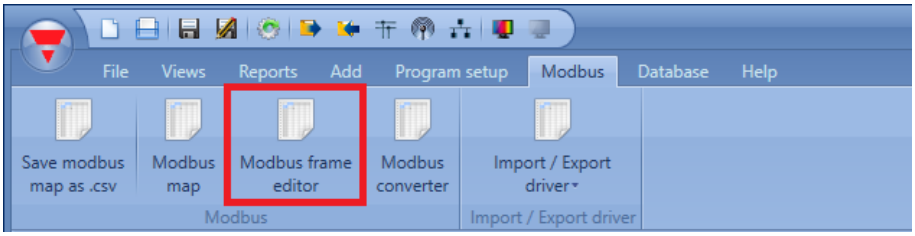
N.B. In case a Modbus device has been imported into a configuration with a version (e.g. v1) and during the project development the driver is updated to a newer version (e.g. v2), the former device with the older revision is not managed anymore, so the module must be added again to the project with the new driver version.

17.6 Modbus frame editor

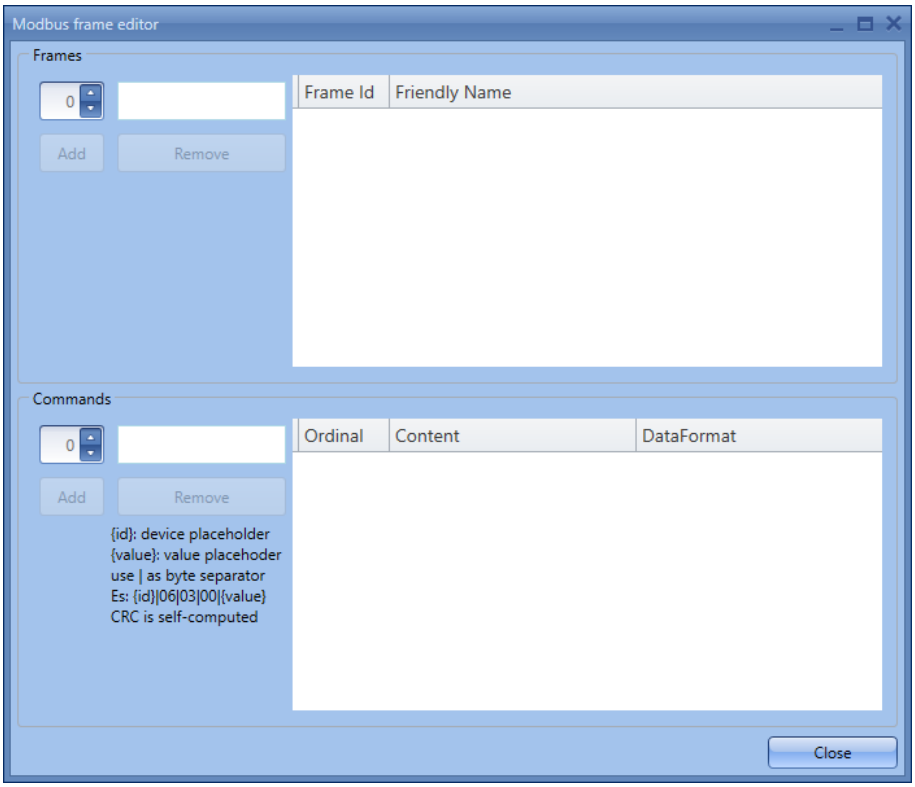
N.B: The user must be familiar with the Modbus protocol before using the *Modbus frame editor* and *Modbus output bridge* functions.

The *Modbus frame editor* function can be used to build a Modbus RTU command that can be sent to a specific Modbus serial device connected to a serial communication port (*COM1* or *COM2* master) of the UWP 3.0 controller. Then, the *Modbus output bridge* function can be used to link the Modbus RTU command to a selected Modbus device where the register will be written according to the value of the command. These two functions can be used, for example, to send a power ON command to a device or to control a fancoil unit by changing the setpoint.

To define a Modbus RTU command, the user has to click on *Modbus frame editor* in the *Modbus* menu, as shown below:



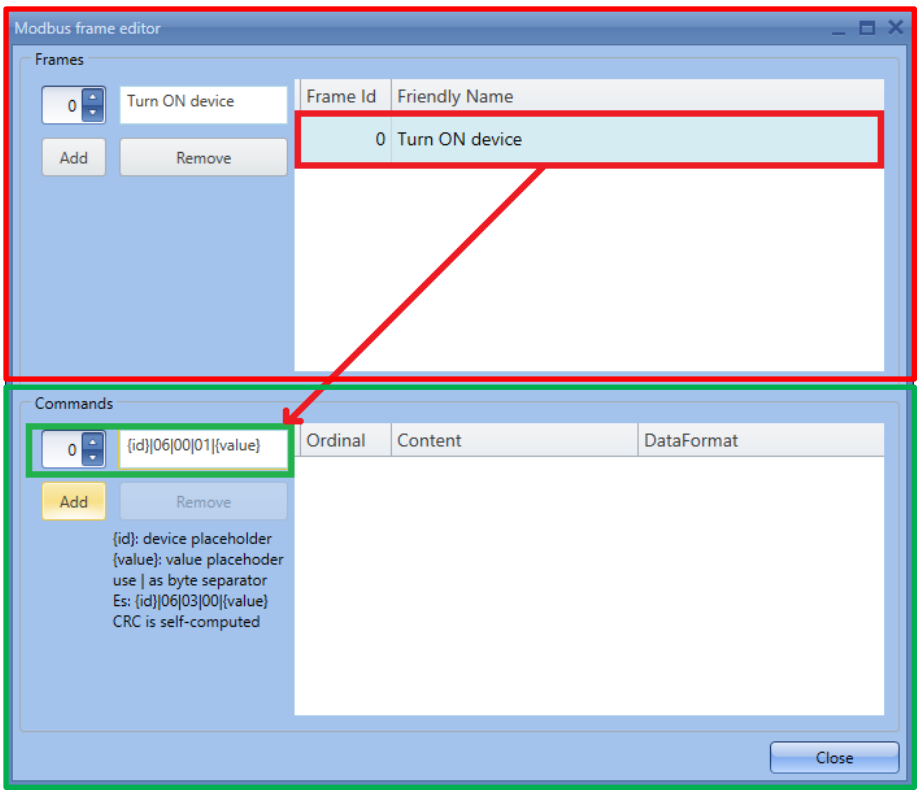
The *Modbus frame editor* window will appear, as shown here:



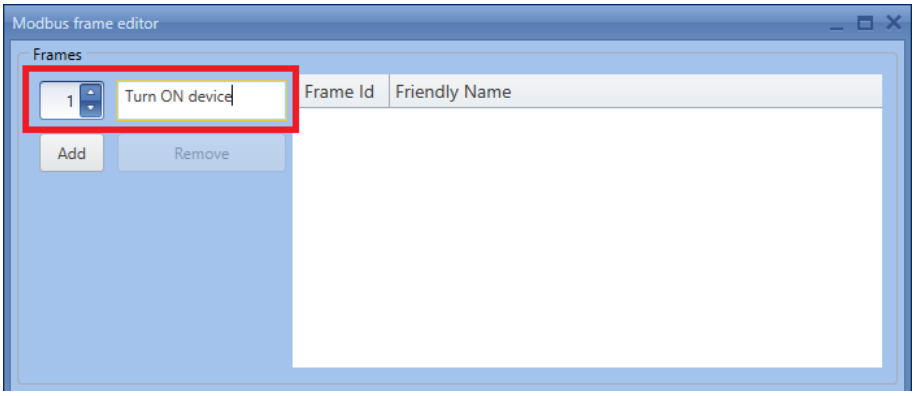
17.6.1 How to add a new Modbus RTU command

To create a new Modbus RTU command, the user has to fill in the necessary settings in the *Modbus frame editor* window. The *Modbus frame editor* window is divided into two main areas:

- In the first area the user defines the name of the Modbus frame (for example a *Turn ON* command as shown in the red rectangle below);
- In the second area, the user types in the Modbus frame, related to the previously defined name (as shown in the green rectangle below);



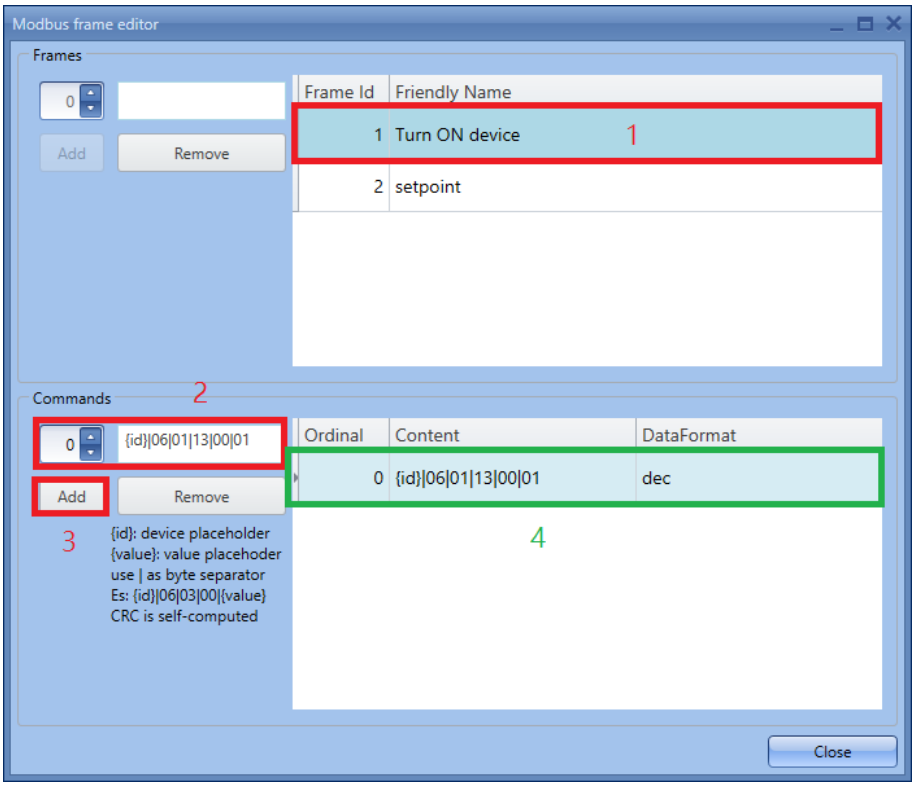
In the *name* field of the *Frames* area, the user has to enter the name of the Modbus RTU command, as shown below:



Each added frame has a specific identification number (*Frame id* field), which is used to identify the Modbus RTU command in the *Modbus output bridge* function (as illustrated in the next paragraph).

By clicking on the *Add* button, the Modbus command will be saved. If the user selects one Modbus RTU command and clicks on *Remove*, the selected command will be removed from the *Modbus frame editor* window.

In the *Commands* area of the *Modbus frame editor* window, after having selected at least one Modbus frame (1), the user has to manually insert the command (2) and press *Add* (3): the Modbus RTU command will be saved in the Commands list (4), as shown below.

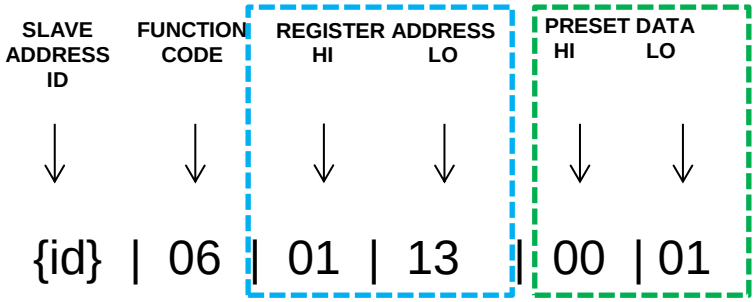


17.6.1.1 Modbus RTU command format

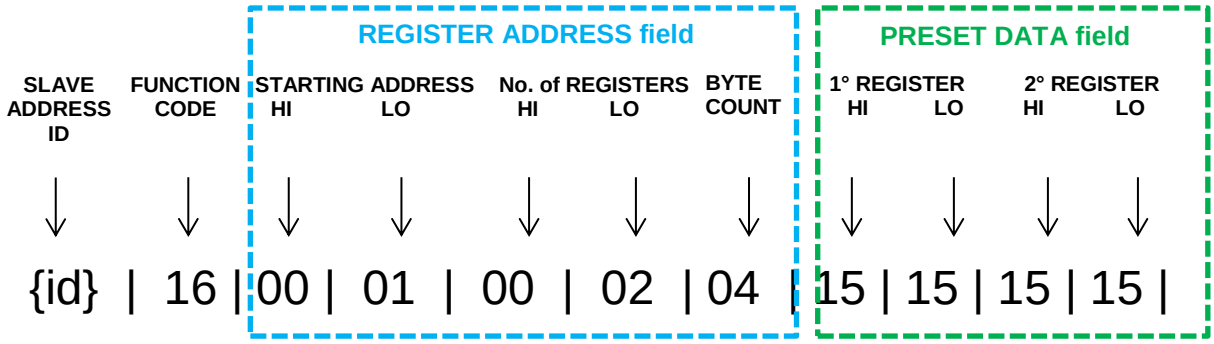
The structure of a Modbus RTU command changes according to the function code being used:

- The **function code 06** is used to write preset values into a single holding register;
- The **function code 16** is used to write preset values into a sequence of holding registers;

The example below shows a typical structure of a Modbus RTU command with function code **06**
(06: Preset Single Register as defined by the Modbus protocol specification)



The example below shows a typical structure of a Modbus command with function code **16**:
(16: Preset Multiple Register as defined by the Modbus protocol specification)



Following the two examples above, a Modbus command must be built using sets of characters separated by the "**|**" symbol (vertical bar). Each set of data has a specific position and must be edited according to the specifications below:

Slave address id
The ***Modbus id*** is self-computed by the system: the user has to enter the placeholder **{id}**;

Function code
The function code identifies the operation to be executed by the Modbus command: two codes are allowed:

- **06:** Preset Single Register;
- **16:** Preset Multiple Registers;

Register address *(Starting address, No. of register, Byte Count* fields if function code is 16)
The register address field defines the holding register of the serial device which has to be written.
It must be expressed in two sets of decimal characters that represent the first (HI) byte and the second (LO) byte of the holding register. If the function code 16 is being used, the Register field specifies the *Starting register*, *No. of Registers* and the *Byte Count* fields.

Example

The example below shows how to write the codes of the address **269_{dec}** into two decimal sets:
269_{dec} = $010D_{hex}$ -> (**01** is the first byte, **0D** is the second byte in hexadecimal format).
Then the two bytes must be coded into a decimal format again: |**01|13|** (**01_{dec}** = 01_{hex}, **13_{dec}** = $0D_{hex}$)

***Number of Registers*:** this field is mandatory only for function code 16. Two bytes in decimal format have to be added, indicating the number of registers to be written.

Example

The example below shows how to write the two bytes if the user has to preset 2 holding registers, thus **2_{dec}** into two decimal sets:

2_{dec} = 0002_{hex} -> (**00** is the first byte, **02** is the second byte in hexadecimal format).
Then the two bytes must be coded into a decimal format again: |**00|02|** (**00_{dec}** = 00_{hex}, **02_{dec}** = 02_{hex}).

***Byte Count*:** this field is mandatory only for function code 16. It indicates the number of bytes to be written. It is calculated using the formula: *Byte Count = No. of Registers * 2.*

Example

3 holding registers * **2** = **06** is the value of the *Byte Count* field that has to be entered.

Modbus RTU command example for function code 16

The example below shows a request to preset 2 holding registers (red fields) at address 1 (blue fields) to 10_{dec} (light green fields) and 514_{dec} (green fields). The *Byte Count* value is 4 (orange field).

Field Name	Value
Slave Address	{id}
Function	16
Starting Address Hi	00
Starting Address Lo	01
No. of Registers Hi	00
No. of Registers Lo	02
Byte Count	04
Data Hi	00
Data Lo	10
Data Hi	02
Data Lo	02

Data field
The data field is constructed using sets of two decimal digits. The user can define this field with a fixed or dynamic value, and the data length depends on the function code (**06** or **16**) being selected:

- ***Fixed value***:
    a. if the function code 06 is used, two bytes (HI and LO) of data have to be entered.
    b. If the function code 16 is used, two sets of decimal characters must be entered for each holding register of data to be written.

    For example, the value 1_{dec} must be entered with two sets of decimal characters ***|00|01|;***

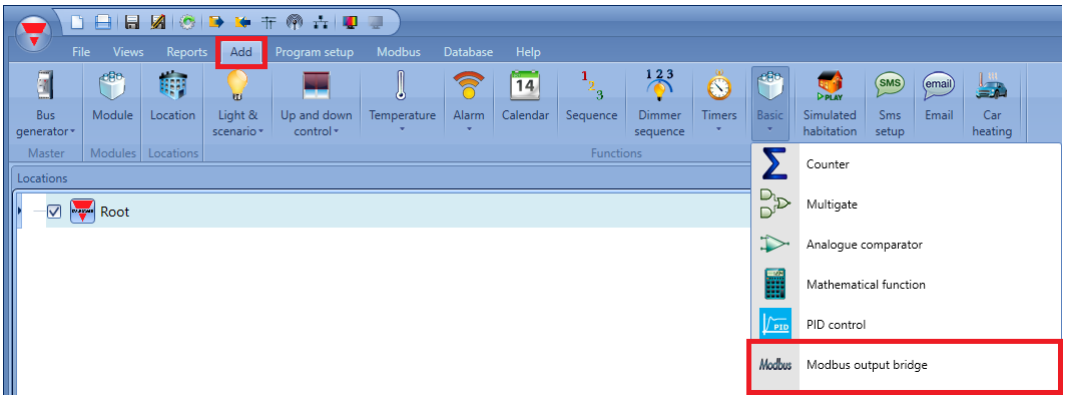
- ***Dynamic value****:*
    a. if the user enters the placeholder **{value}** in the Data field, the value to be written into the holding register is self-computed by the system according to the value of the input signal selected in the *Modbus output bridge* function: the data length for function code 06 is 2 bytes, whilst the data length for function code 16 is 4 bytes.

    For example, the Modbus RTU command used to write two holding registers with the self-computed value is ***{id}|16|00|01|00|02|04|{value};***

17.6.2 How to configure a Modbus output bridge function

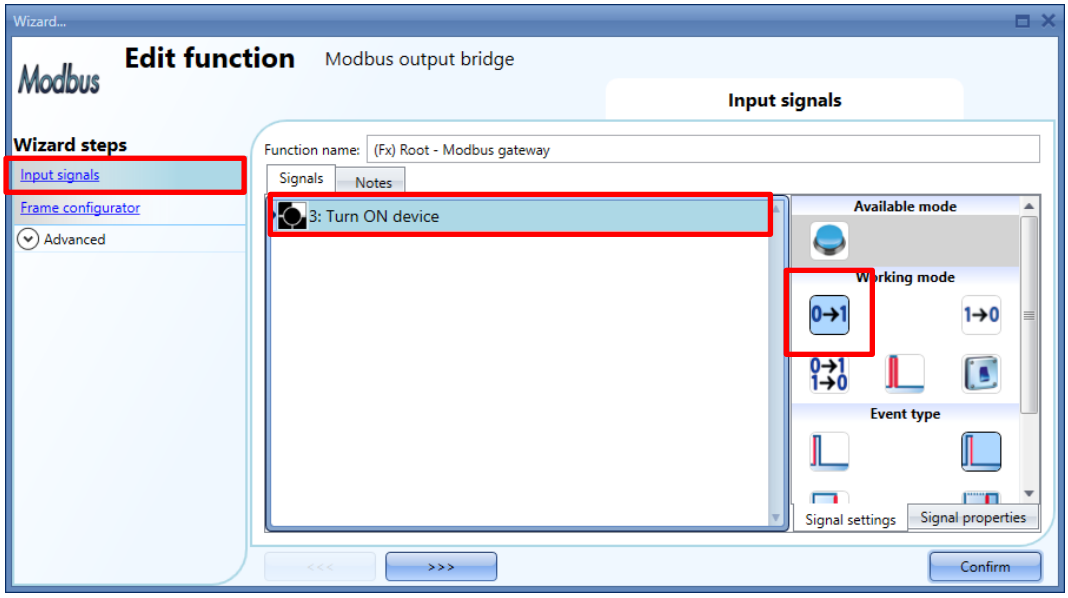
The *Modbus output bridge* function is used to link a Modbus RTU command together with the Modbus slave device and with an input signal to trigger the delivery of the Modbus command to the specified slave device.

To set up a Modbus output bridge function, the user has to select *Modbus output bridge* function from the *Basic* menu, as shown below:



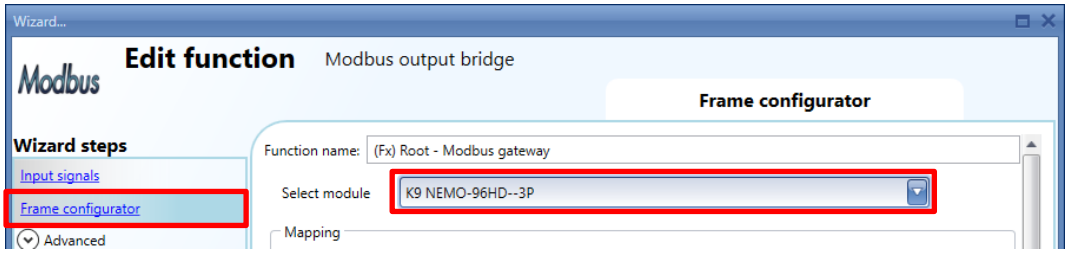
17.6.3 How to send the Modbus command manually

To add the input signal, the user has to select the *Input signal* field, then double click on the *Signals* window and select the input signal from the list of those available.
Only one signal can be selected (see below).

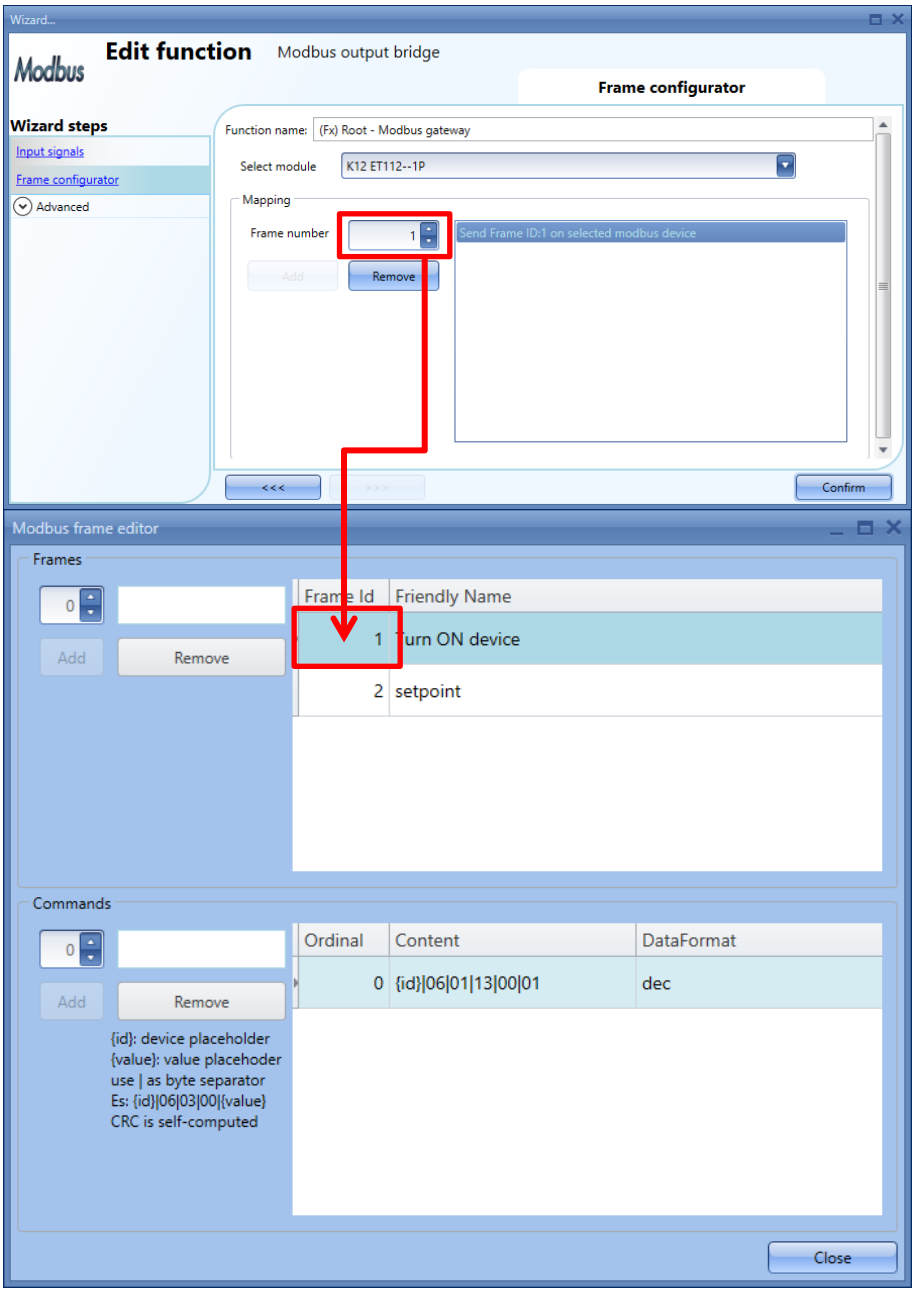


The Modbus RTU command will be sent when the push button is pressed, as shown in the red rectangle above.

In the *Select module* field of the *Frame configurator* window, the user has to select the Modbus serial device, as shown below:



In the *Frame number* field, the user has to select the Modbus command according to the *Frame Id* field, and click on the *Confirm* button to save the configuration, (see below).

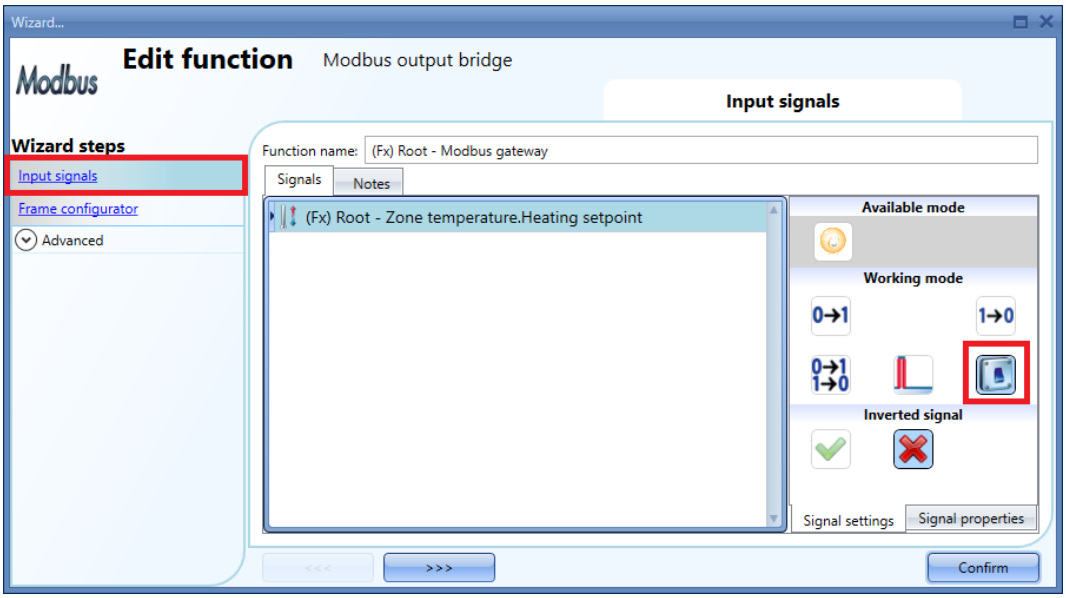


As a result of this configuration, each time the pushbutton is pressed, the Modbus RTU command *Turn ON device* will be sent to the Modbus device selected in the *Modbus output bridge* function.

17.6.4 How to send the Modbus RTU command automatically

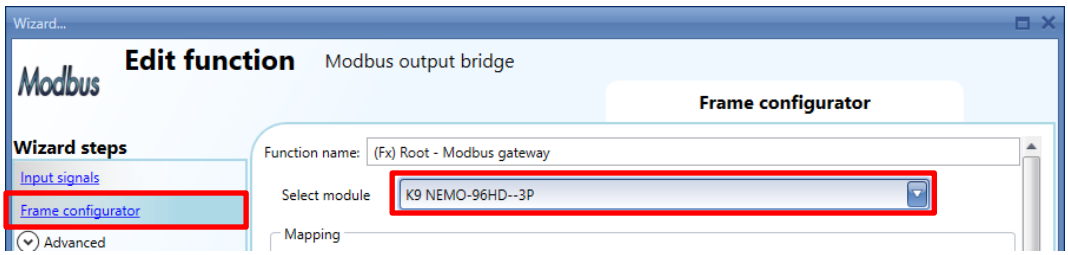
A Modbus RTU command can be sent automatically to a selected Modbus device, whenever the Modbus RTU command is edited with the placeholder *{value}* in the Data field of the Modbus RTU command.

Example
In the example below, the *Zone temperature.Heating setpoint* signal is added to the *Modbus output bridge* function in the Input signal window, as shown below:

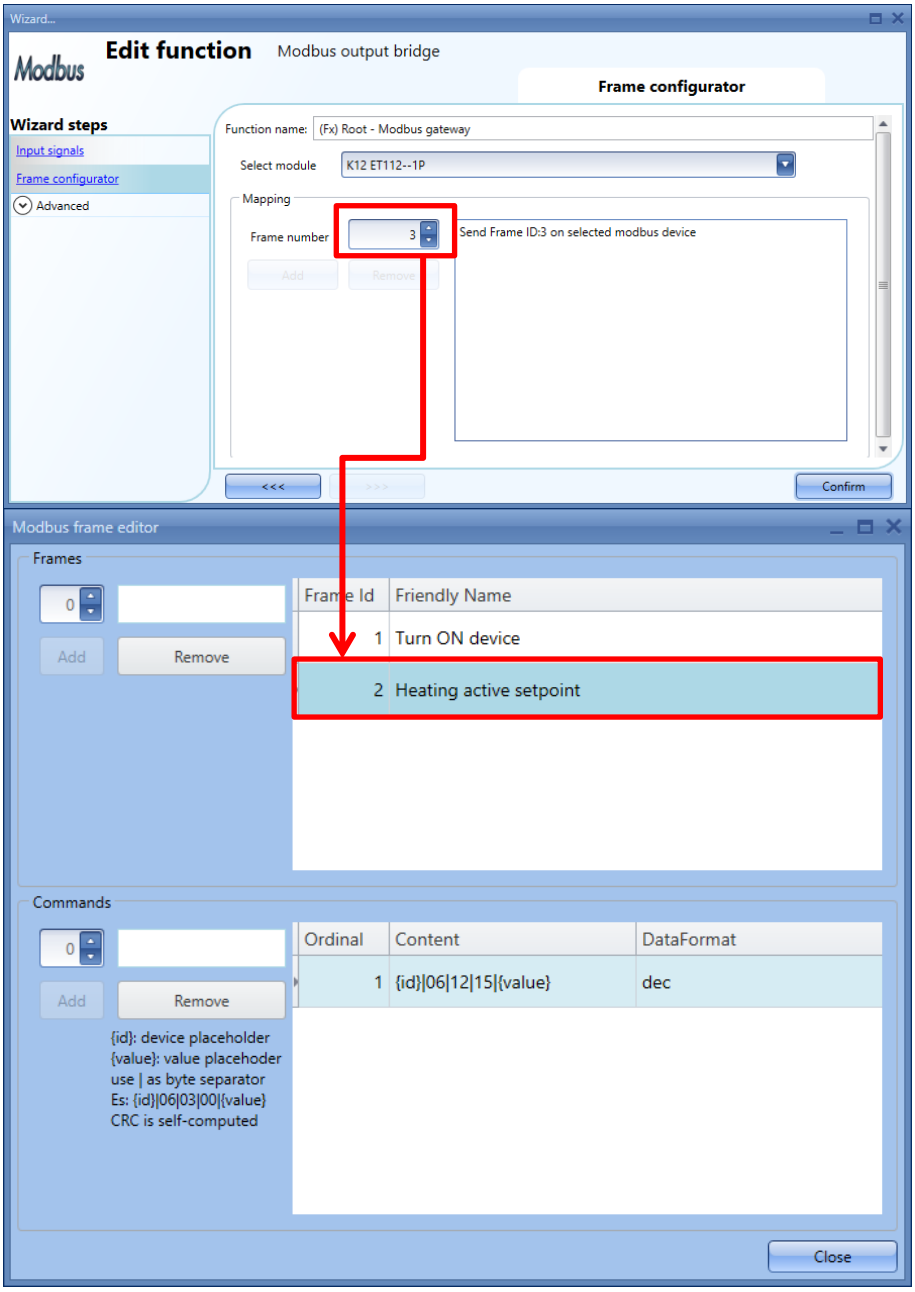


In the *Working mode* field of the Signals setting window, the *Every change* mode is selected (see the red rectangle above): whenever the *Zone temperature.Heating setpoint* signal changes, the Modbus RTU command will be sent to the Modbus device.

In the *Select module* field of the *Frame configurator* window, the user has to select the Modbus serial device, as shown below:



In the *Frame number* field, the user has to select the Modbus RTU command according to the *Frame Id* field, as shown in the *Modbus frame editor* function.

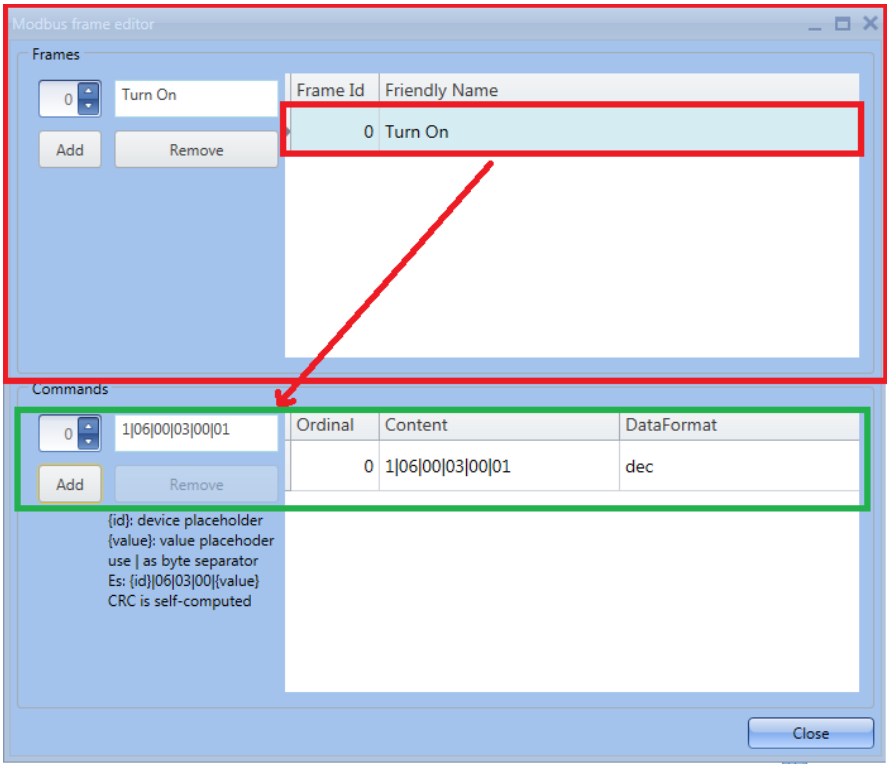


As shown above, the *Heating active setpoint command* ***{id}|06|12|15|{value}*** will be sent to the Modbus serial device every time its values change: as a result of this automation, the holding register indicated in the Modbus RTU command will be written with the new value.

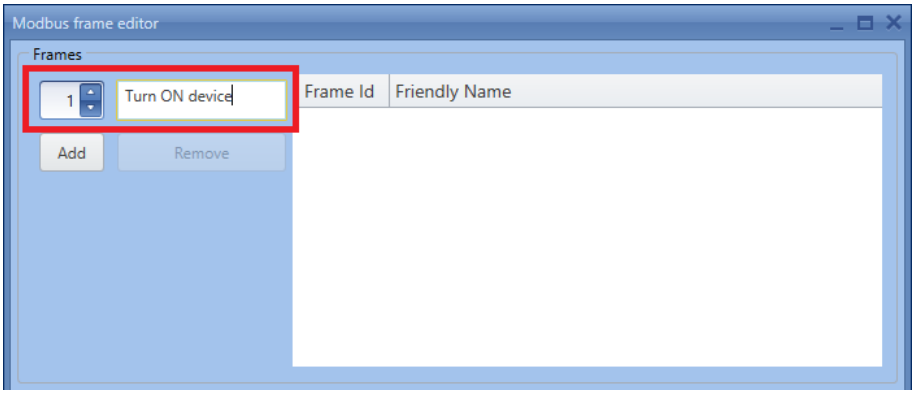
17.6.5 How to add a new Modbus TCP command

To create a new Modbus TCP command, the user has to fill in the necessary settings in the *Modbus frame editor* window. The *Modbus frame editor* window is divided into two main areas:

- In the first area the user defines the name of the Modbus TCP frame (for example a *Turn ON* command as shown in the red rectangle below);
- In the second area, the user types in the Modbus frame, related to the previously defined name (as shown in the green rectangle below);



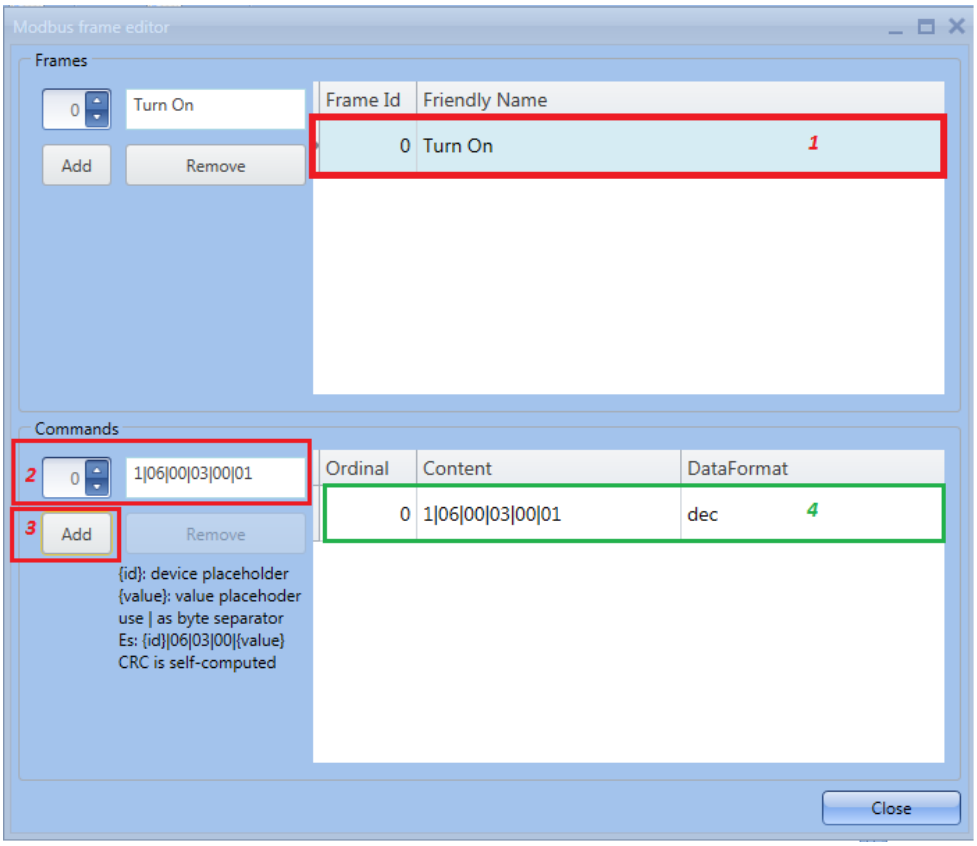
In the *name* field of the *Frames* area, the user has to enter the name of the Modbus TCP command, as shown below:



Each added frame has a specific identification number (*Frame id* field), which is used to identify the Modbus TCP command in the *Modbus TCP output bridge* function (as illustrated in the next paragraph).

By clicking on the *Add* button, the Modbus command will be saved. If the user selects one Modbus TCP command and clicks on *Remove*, the selected command will be removed from the *Modbus frame editor* window.

In the *Commands* area of the *Modbus frame editor* window, after having selected at least one Modbus frame (1), the user has to manually insert the command (2) and press *Add* (3): the Modbus TCP command will be saved in the Commands list (4), as shown below.



17.6.6 Modbus TCP command format

The command format is the same as that explained in the paragraph “Modbus RTU command format”, the only difference is the slave id address, which must be added without leaving it generic (see examples below)

The example below shows a typical structure of a Modbus TCP command with function code **06**
(06: Preset Single Register as defined by the Modbus protocol specification)

SLAVE ADDRESS ID	FUNCTION CODE	REGISTER ADDRESS HI	REGISTER ADDRESS LO	PRESET DATA HI	PRESET DATA LO
5	06	01	13	00	01

The example below shows a typical structure of a Modbus command with function code **16**:
(16: Preset Multiple Register as defined by the Modbus protocol specification)

		REGISTER ADDRESS field					PRESET DATA field			
SLAVE ADDRESS ID	FUNCTION CODE	STARTING ADDRESS HI	STARTING ADDRESS LO	No. of REGISTERS HI	No. of REGISTERS LO	BYTE COUNT	1° REGISTER HI	1° REGISTER LO	2° REGISTER HI	2° REGISTER LO
5	16	00	01	00	02	04	15	15	15	15

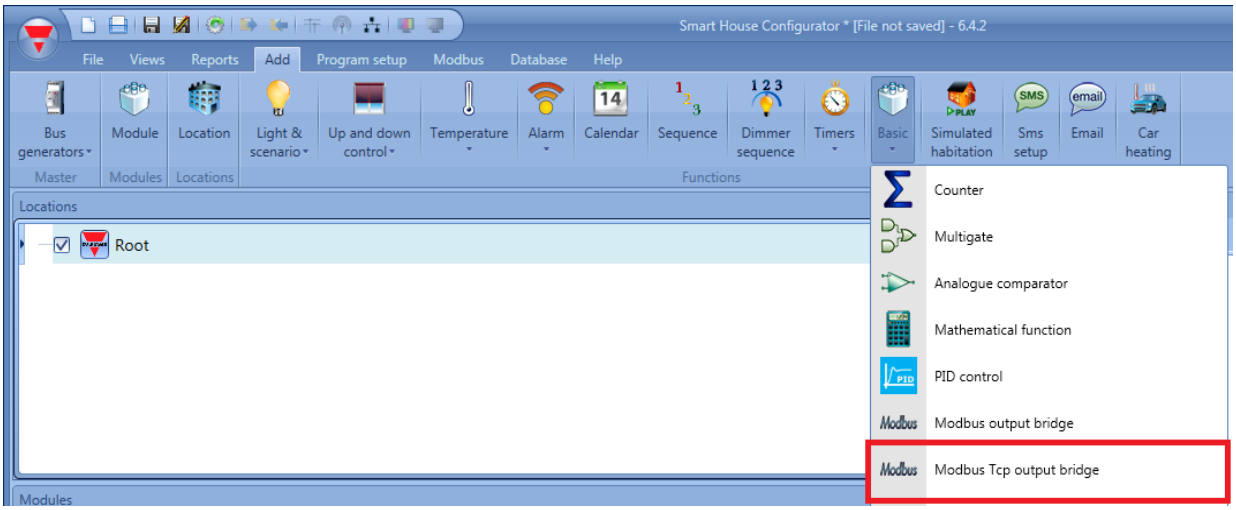
Following the two examples above, a Modbus command must be built using sets of characters separated by the “**|**” symbol (vertical bar). Each set of data has a specific position and must be edited according to the specifications below:

Slave address id
The user has to enter the specific Modbus ID (do not leave this field generic)

17.6.7 How to configure a Modbus TCP output bridge function

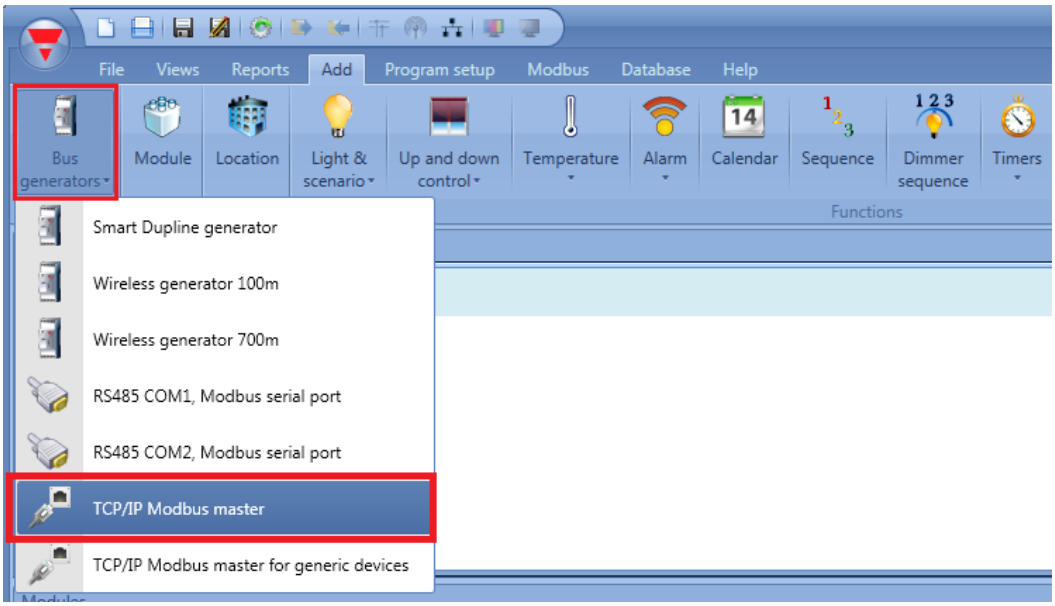
The *Modbus TCP output bridge* function is used to link a Modbus TCP command with a Modbus slave device and with an input signal to trigger the delivery of the Modbus command to the specified slave device.

To set up a Modbus TCP output bridge function, the user has to select *Modbus output bridge* function from the *Basic* menu, as shown below:

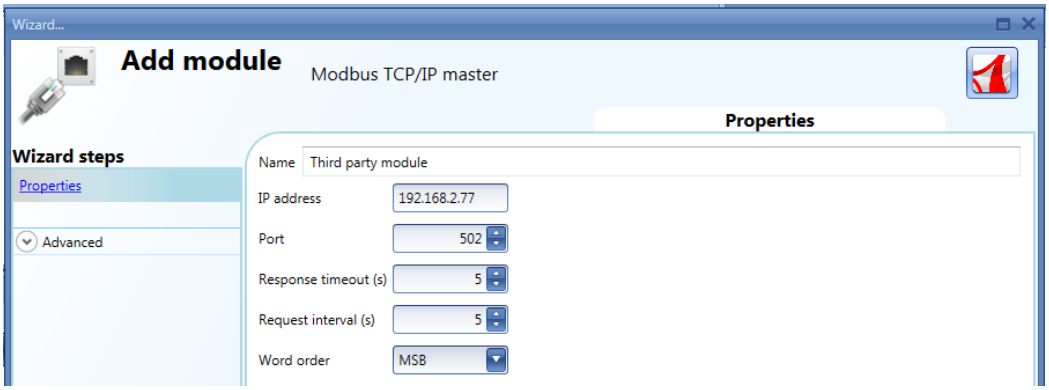


17.6.8 How to send the Modbus TCP command manually

The user has to add a TCP/IP Modbus master interface from the *Bus generators* menu

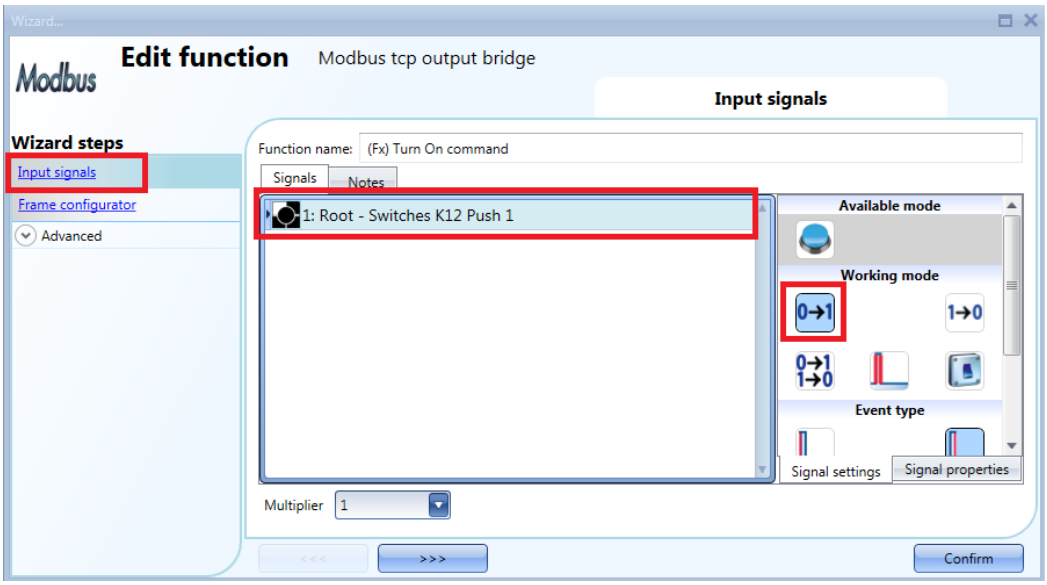


In the TCP/IP Modbus master interface the user has to define the IP address of the Modbus device which is to receive the commands sent by the UWP 3.0 controller.



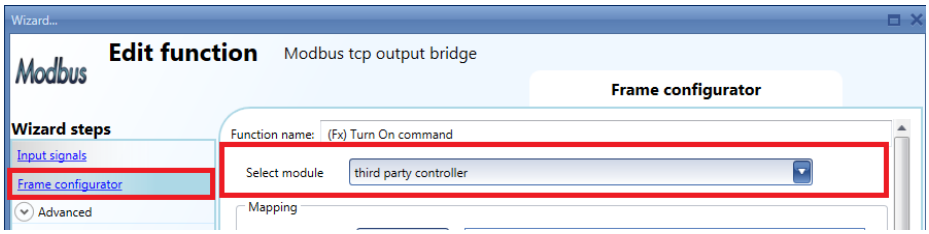
IP address: enter the address of the slave device
Port: enter the port used for the Modbus communication
Response timeout (s): timeout settings
Request interval (s): time between execution of each scheduled frame
Word order: is possible to define the byte order when writing multiple registers MSB means (MSB-LSB), LSB means (LSB-MSB)

To add the input signal, the user has to select the *Input signal* field, then double click on the *Signals* window and select the input signal from the list of those available.

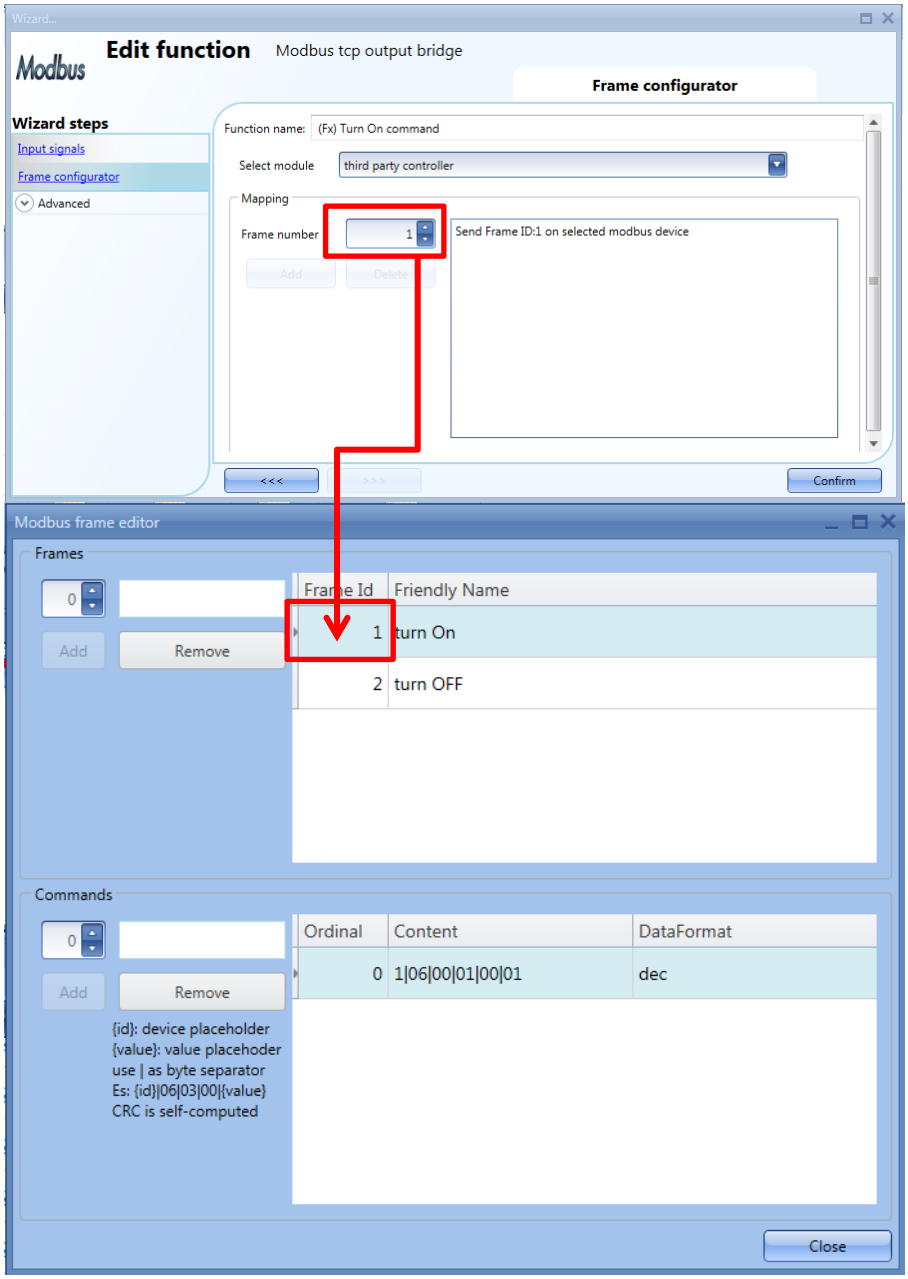


The Modbus TCP command will be sent when the push button is pressed, as shown in the red rectangle above.

In the *Select module* field of the *Frame configurator* window, the user has to select the Modbus TCP master device, as shown below:



In the *Frame number* field, the user has to select the Modbus command according to the *Frame Id* field, and click on the *Confirm* button to save the configuration, (see below).

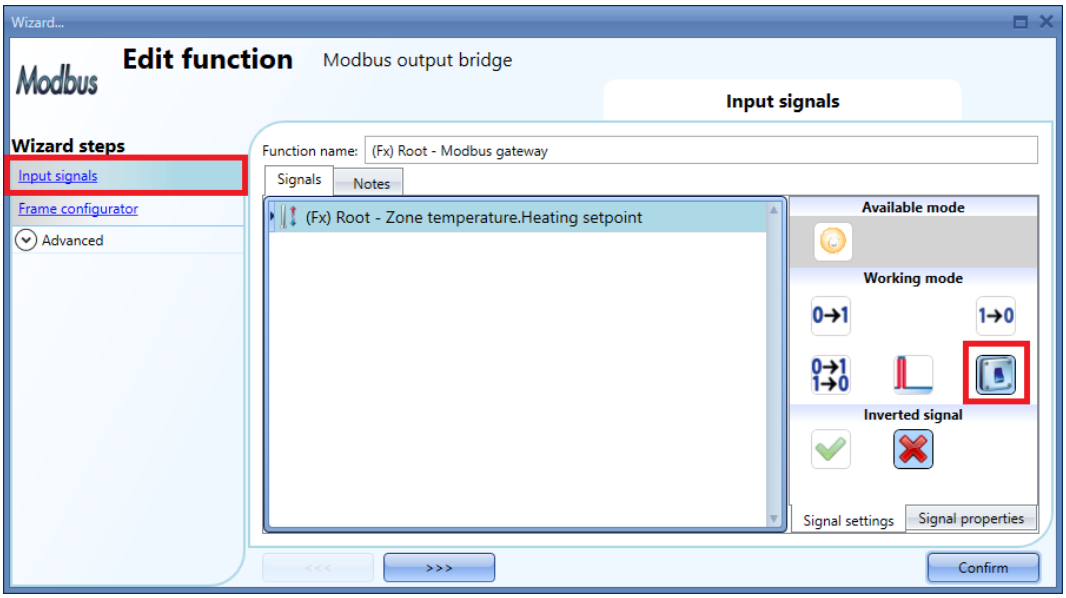


As a result of this configuration, each time the pushbutton is pressed, the Modbus TCP command *Turn ON device* will be sent to the Modbus device selected in the *Modbus TCP output bridge* function.

17.6.9 How to send the Modbus TCP command automatically

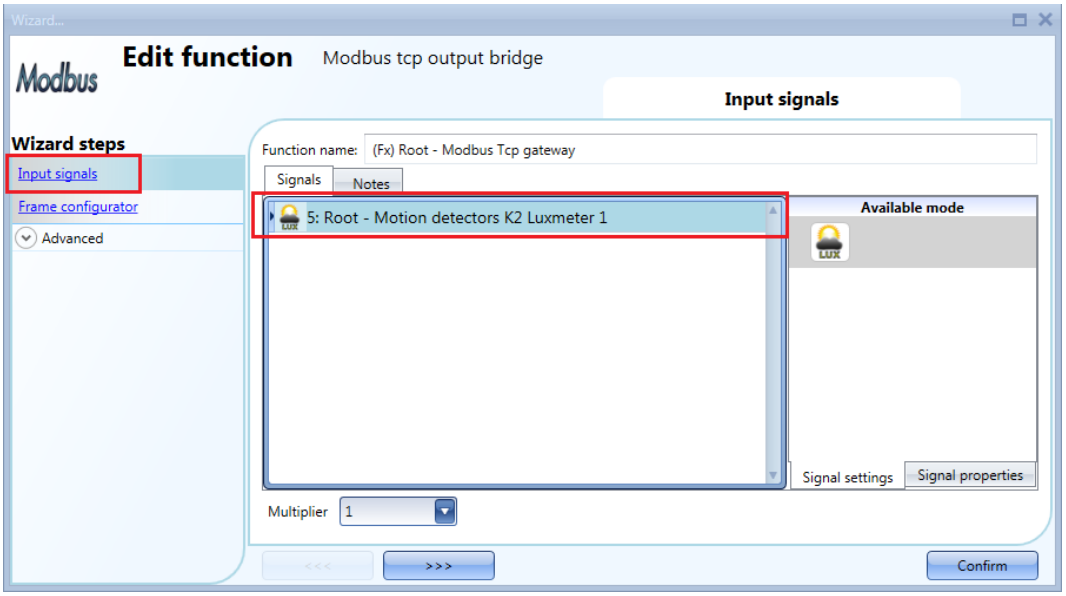
A Modbus TCP command can be sent automatically to a selected Modbus device, whenever the Modbus TCP command is edited with the placeholder *{value}* in the Data field of the frame editor window.

Example
In the example below, the *Zone temperature.Heating setpoint* signal is added to the *Modbus output bridge* function in the Input signals window, as shown below:

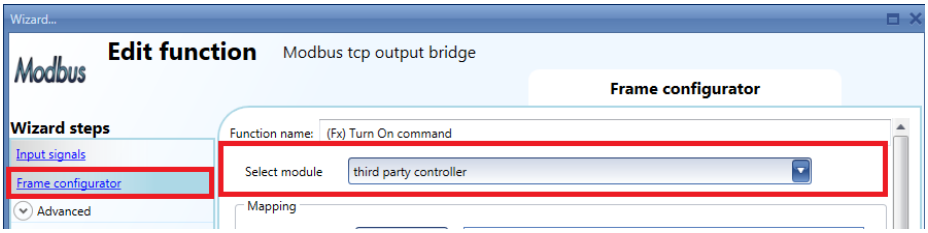


In the *Working mode* field of the Signals setting window, the *Every change* mode is selected (see the red rectangle above): whenever the *Zone temperature.Heating setpoint* signal changes, the Modbus TCP command will be sent to the Modbus device.

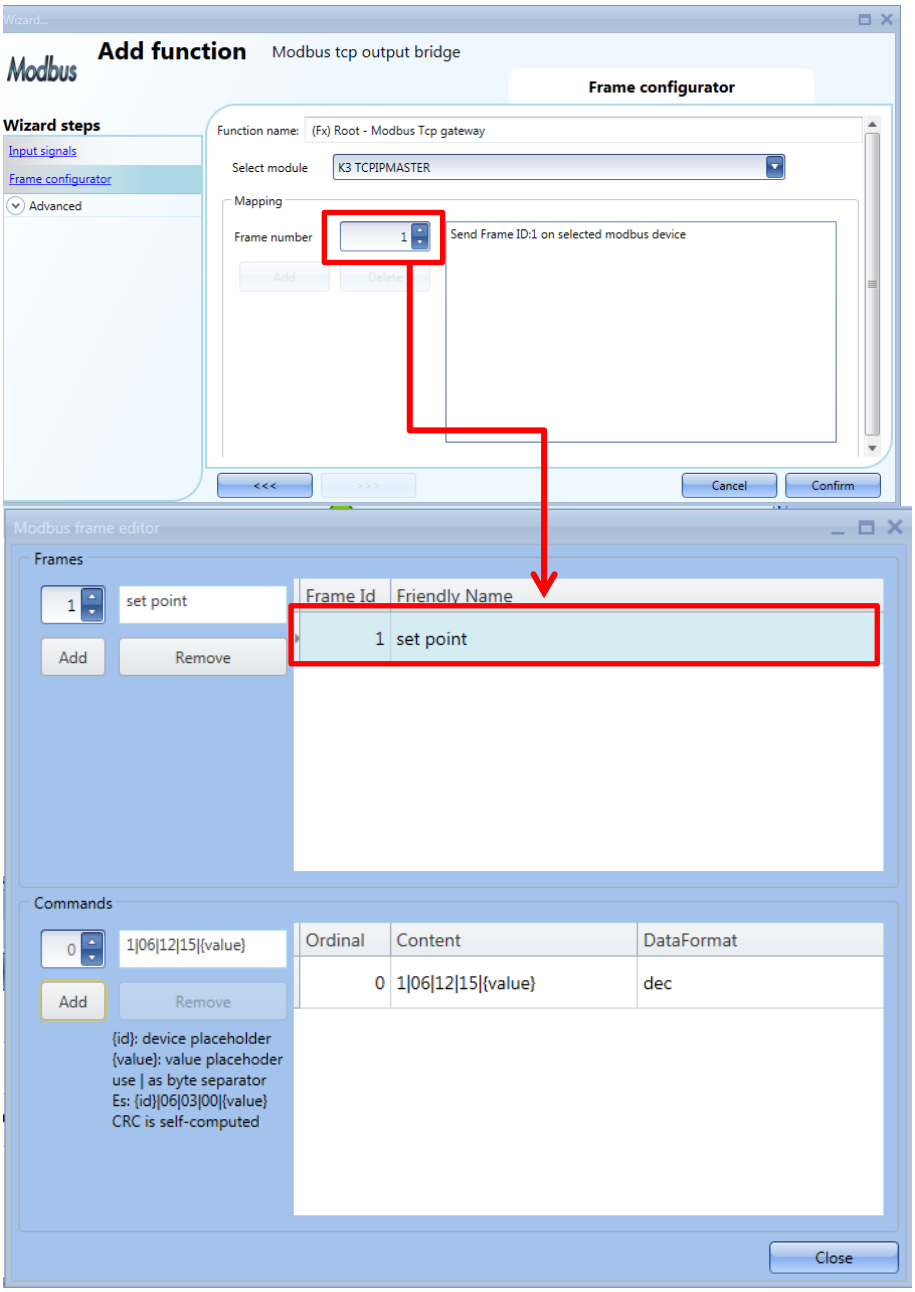
It is possible to add an analogue value as an input signal. In this case, every time the analogue signal changes its value, a new command will be sent to the device.



In the *Select module* field of the *Frame configurator* window, the user has to select the Modbus serial device, as shown below:



In the *Frame number* field, the user has to select the Modbus TCP command according to the *Frame Id* field, as shown in the *Modbus frame editor* function.



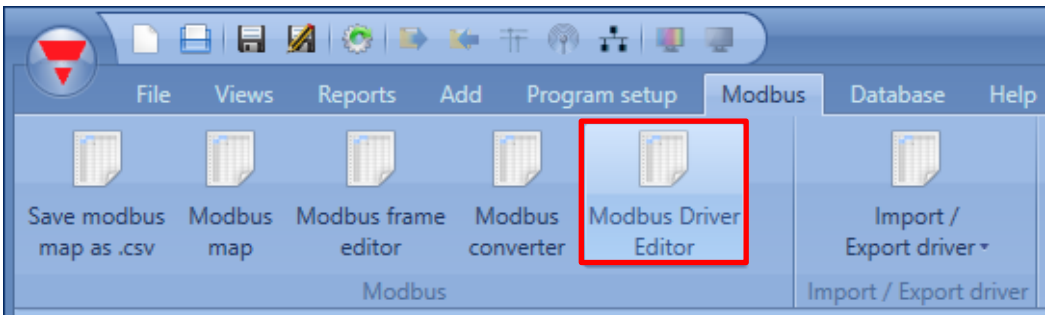
As shown above, the *Heating active setpoint command* ***1|06|12|15|{value}*** will be sent to the Modbus device with MODBUS ID 1 every time its value changes: as a result of this automation, the holding register indicated in the Modbus TCP command will be written with the new value.

18 Modbus Driver Editor tool

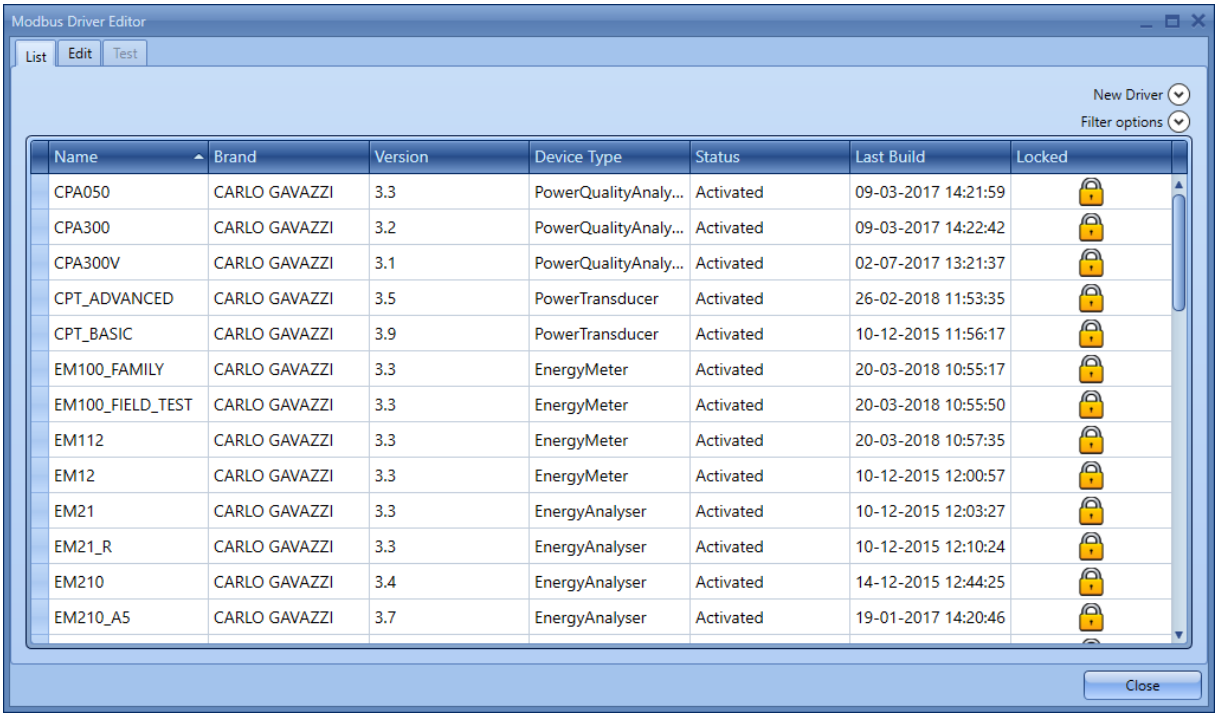
The *Modbus Driver Editor* tool is a flexible platform aimed at:

- developing **Modbus drivers** for UWP 3.0,
- connecting Modbus meters either via RS485 or Ethernet and
- gathering data from them according to their Modbus protocol.

To open the *Modbus Driver Editor* tool, the user has to select the **Modbus Driver Editor** icon from the **Modbus** menu (see the picture):



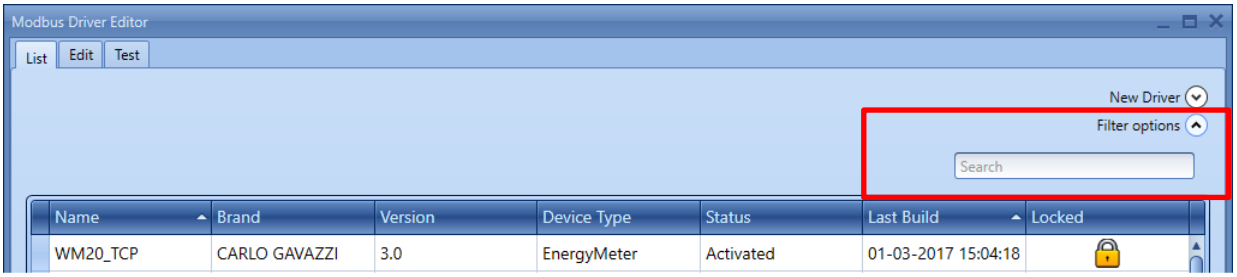
The *Modbus Driver Editor (MDE)* window will appear as follows:



Name	Brand	Version	Device Type	Status	Last Build	Locked
CPA050	CARLO GAVAZZI	3.3	PowerQualityAnaly...	Activated	09-03-2017 14:21:59	
CPA300	CARLO GAVAZZI	3.2	PowerQualityAnaly...	Activated	09-03-2017 14:22:42	
CPA300V	CARLO GAVAZZI	3.1	PowerQualityAnaly...	Activated	02-07-2017 13:21:37	
CPT_ADVANCED	CARLO GAVAZZI	3.5	PowerTransducer	Activated	26-02-2018 11:53:35	
CPT_BASIC	CARLO GAVAZZI	3.9	PowerTransducer	Activated	10-12-2015 11:56:17	
EM100_FAMILY	CARLO GAVAZZI	3.3	EnergyMeter	Activated	20-03-2018 10:55:17	
EM100_FIELD_TEST	CARLO GAVAZZI	3.3	EnergyMeter	Activated	20-03-2018 10:55:50	
EM112	CARLO GAVAZZI	3.3	EnergyMeter	Activated	20-03-2018 10:57:35	
EM12	CARLO GAVAZZI	3.3	EnergyMeter	Activated	10-12-2015 12:00:57	
EM21	CARLO GAVAZZI	3.3	EnergyAnalyser	Activated	10-12-2015 12:03:27	
EM21_R	CARLO GAVAZZI	3.3	EnergyAnalyser	Activated	10-12-2015 12:10:24	
EM210	CARLO GAVAZZI	3.4	EnergyAnalyser	Activated	14-12-2015 12:44:25	
EM210_A5	CARLO GAVAZZI	3.7	EnergyAnalyser	Activated	19-01-2017 14:20:46	

18.1 How to manage the filters in the Modbus Driver Editor window

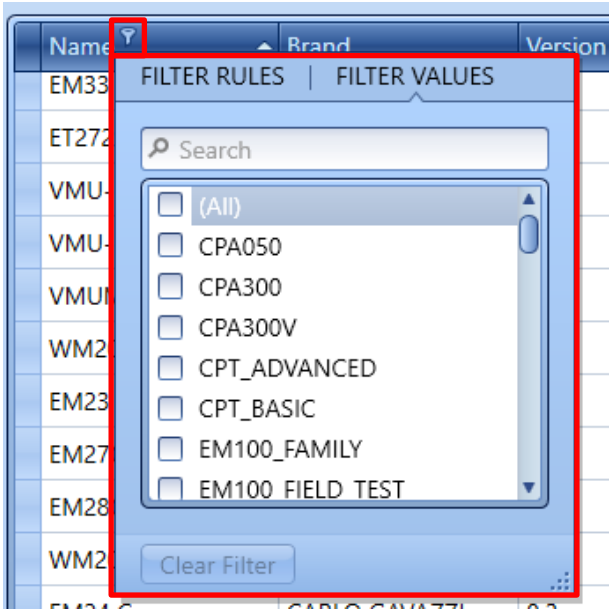
The user can sort or filter the available *Modbus drivers* using the *Filter options*: clicking on , a *Search box* will appear under the *Filter options* field:



The user can enter the search criteria and the system will provide all the searched *Modbus drivers*: these can be sorted in ascending or descending order by clicking on the header fields.

Note: A small ▾ / ▴ icon will indicate which order has been applied.

The user can also access the advanced filter options, clicking on (available in each field of the header bar):



In order to remove the applied filters, the user has to click on *Clear filter*.

18.2 Modbus Driver Editor overview

The *Modbus Driver Editor* window has three tabs: The ***List*** tab is the default one.

The following table describes each tab:

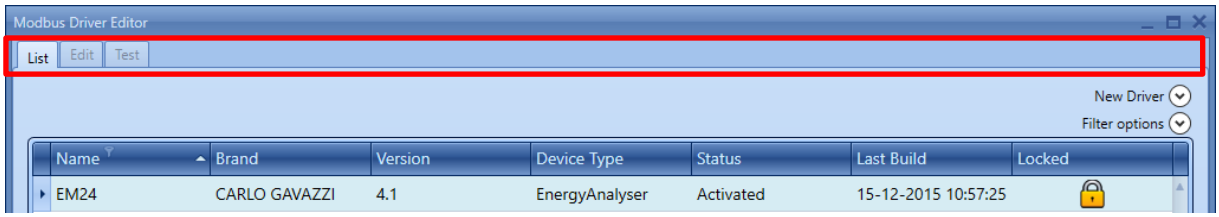
Tab name	Description
List	It shows the list of the available *Modbus drivers*, displaying their status.
Edit	It shows the inner hierarchy of a Modbus driver for a specific device and the editing parameters **(Tree view** and **Editing menu).**
Test	It shows the options to perform the *Driver test* (see ***How to test a driver***).

The access to the different tabs depends on to the *Modbus driver* status:
e.g.

- if a driver is being edited, it is possible to access the *Edit* tab;
- if a driver has been compiled (**BuildOK** status), it is possible to access the *Test* tab.

18.2.1 List Tab overview

In the *List* tab, all the available *Modbus drivers* are displayed. The list may vary according to the firmware release of the UWP 3.0 Tool software.



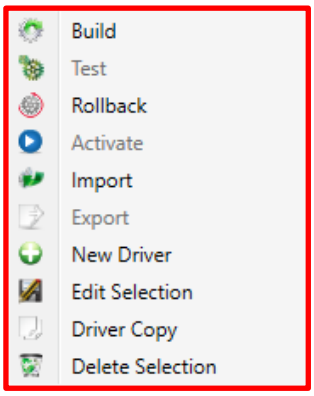
Each *Modbus driver* is represented by the following fields:

Field name	Description
Name	It shows the device name (e.g. the Part Number) for which the *Driver* is made
Brand	It shows the manufacturer's name
Version	It shows the Driver *version* of the related part number
Device Type	It shows the *Device type*
Status	It shows the Driver *status*: **Status** — **Description** Editing — The driver is being edited, but it has not been successfully compiled. BuildOk — The driver has been successfully compiled and it is available to be tested. It is not available to be used to connect Modbus devices to UWP 3.0. Activated — The driver has been successfully compiled and activated. It is available to be used to connect Modbus devices to UWP 3.0. Build Error — The driver has not been compiled correctly. The user has to repeat the Build procedure.
Last Build	It shows the date of the last build.

Field name	Description
Locked	The drivers that have the yellow padlock are locked: it is not possible to edit Carlo Gavazzi's locked drivers. Nonetheless, it is possible to copy a Carlo Gavazzi's driver and edit it. **N.B: The drivers that have a yellow padlock are delivered with the software installer.**

18.2.2 Contextual menu Driver options

Right-clicking on a *Modbus driver* present in the list, the following menu will appear:



The available options are clickable and they depend on the status of the selected driver.
The following table helps the user to know when they are available:

Option (Action)	Description	Editing	BuildOK	Activated
Build	To compile a driver, highlighting any error or warning. At each build process, this field will be increased by 0.1.	✔	✔	✔
Test	To start the testing process: this process is aimed at checking that the compiled driver communicates with the connected Modbus devices, according to the expectations. *Note: This option automatically opens the Test tab.*	✖	✔	✔
Rollback	To bring the driver back to the last valid compiled version. All changes that have been made after the build process will be lost, even if the user has saved them.	✔	✖	✖
Activate	To activate a driver (that must be compiled). The activation permits to have the Modbus driver available: it can be used to add the related device in the configuration.	✖	✔	–
Import	To import a driver (XML format), the UCS software or the VMU-C.	*Note: The **Import** is always available, independently from the point where the user right-clicks*		

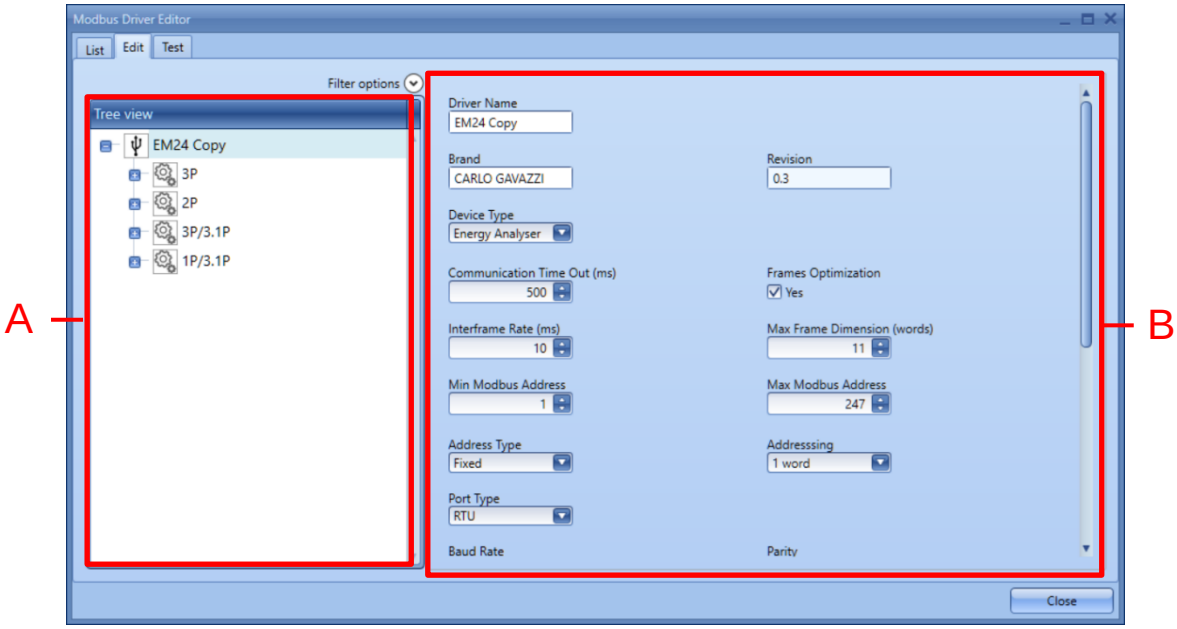
Option (Action)	Description	Editing	BuildOK	Activated
Export	To export a driver (XML format). Once exported, the file is available to be imported in another UWP 3.0 Tool or VMU-C.	✖	✔	✔
New Driver	To create a new driver: this action opens another page.	*The **New driver** action is always available, independently from the point where the user right-clicks*		
Edit Selection	To edit a driver. It is possible to copy a Carlo Gavazzi's driver and edit it.	✔	✔	✔
		***N.B.:** It is not possible to edit the Carlo Gavazzi's locked drivers*		
Driver copy	To create a new item as a copy of the existing selected driver. It is possible to copy a Carlo Gavazzi's driver and edit it.	✔	✔	✔
Delete Selection	To delete a driver from the *List* tab.	✔	✔	✔
		***N.B.:** It is not possible to delete the Carlo Gavazzi's locked drivers*		

18.2.3 Edit tab overview

The *Edit* tab allows the user to create and modify a *Modbus driver*. It can be accessed by:

- Double clicking on a *Modbus driver*;
- Right clicking on the *Modbus driver* and clicking on the *Edit Selection* option from the *contextual menu*;
- Selecting a *Modbus driver* from the *List* and clicking on the *Edit* tab;

N.B: Those actions can be performed only for unlocked (with green padlock) *Modbus drivers* present in the *List* tab.

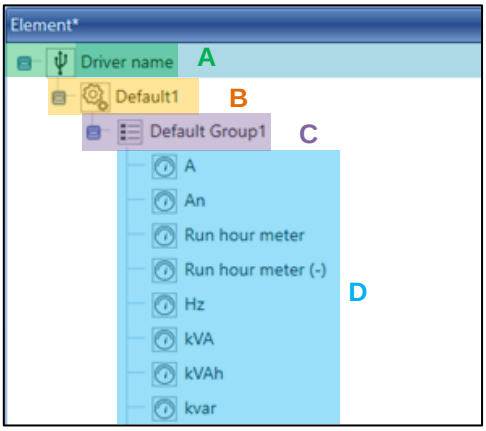


When the user accesses the *Edit* tab, the following areas will be available:

Area	Description
A	The left panel contains the hierarchical **tree view** of the entire Driver structure. The available levels are the following ones: **Level** / **Description** DRIVER: Root level. Here, the main parameters of the field bus connection are defined. CONFIGURATION: Any pattern meant as a group of variables that identifies a specific operation mode of the target Modbus device. GROUP: Any group of variables with a logical link among them. *Note: in the case of multi-meters, any single meter is considered as a group.* VARIABLE: Any single variable in a group. At this level, the *Modbus register* settings, scales, ranges and validation rules are defined. CUSTOM CHART: Any *Custom chart* based on the available set of variables to embed in the driver. The *Custom chart* will be available for whatever UWP 3.0 that uses the driver.
B	The main area shows the parameters, according to the selection.

Example of a Modbus driver hierarchy

The example below shows the hierarchy of a *Modbus driver*; the following picture describes the parent-child relationship:



[A] Driver
It defines the communication parameters of the *Modbus device*.

[B] Configuration
It defines the working mode of the *Modbus device*. There can be several configurations for each device.

[C] Group
It represents a *Load* item. The number of *Groups* depends on the configuration *(e.g. 3-Phase or 1-Phase systems).*

(D) Variables set
It represents the list of variables: this list is automatically defined by the *Group type*.

Each level has a specific set of parameters that the user has to provide according to the *Modbus register map*.
According to the selected level, the available options change, as described in the following table:

Option	Description	Options available for level
Build	See ***Contextual menu Driver options***	A B C D
Test		
Rollback		
Add configuration	To add a new configuration for the Modbus device	A B C D
Add group	To add a new group for the selected device, that represents a *Load* of the Modbus device *(e.g. Energy Meter)*	B C D
Add Custom chart	To add a new *Custom chart* for the selected *Group*	Only B
Add Variable	To add a new *variable* for the selected *Group*	Only C
Delete Selection	To delete the selected level. *Note: This operation affects also the sub-levels that belong the main level.*	B C D
Copy Configuration	To delete the selected level: the entire hierarchy will be copied	Only B
Copy Group	To delete the selected level: the entire hierarchy will be copied	Only C
Copy Custom chart	To delete the selected level: the entire hierarchy will be copied	Only B
Save	To save all the changes.	A B C D

18.2.4 [A] DRIVER level overview

1.2.4.1 Configuring the communication parameters (DRIVER level)

To edit a *Modbus driver*, the first step is to configure the relevant communication parameters:

N.B.: Not all the parameters are needed. The user has to refer to the *Modbus register map* of the device.

The following table describes the communication parameters:

Parameter	Description	TCP /RTU
Driver Name	Name that identifies the device in the *List* tab.	Both
Brand	Name that identifies the device manufacture in the *List* tab.	Both
Device Type	A sort of label that permits to identify at a glance the type of driver. By clicking on *Device Type*, the *drivers* are grouped by type.	Both
Revision	The driver revision: the number is automatically updated whenever a change is applied.	Both
Communication Time Out (ms)	The max. time interval during which a response to a command (set in the target Modbus slave) is expected. *Note: 500 ms is the default value.*	Both
Frames Optimization	If flagged, the frames will be grouped together.	Both
Interframe Rate (ms)	Time interval between 2 Modbus frame readings. *Note: 10ms is the default value that corresponds to standard 3,5 chars.*	Both
Max Frame Size (Words)	The size of the frame in words. *Note: according to this parameter, the tool splits the map in frames.*	Both
Min/Max Address	Range of available Modbus address. *Note: Some Modbus devices accept a restricted set of addresses.*	Both

<table>
<tr><th>Parameter</th><th>Description</th><th>TCP /RTU</th></tr>
<tr><td>Address Type
<u>(See details)</u></td><td>The modality under which the driver will be presented to the user when it is added to the UWP 3.0 configuration.

The values are as follows:
<table>
<tr><th>Address Type</th><th>Description</th></tr>
<tr><td>Fixed</td><td>Valid in most cases</td></tr>
<tr><td>Consecutive</td><td>Valid for devices that embed multiple sub-devices with consecutive Modbus addresses</td></tr>
<tr><td>Free</td><td>Valid for devices that embed multiple sub-devices with free Modbus addresses</td></tr>
</table>
Note: The default choice (Fixed) in most cases is the suggested one.</td><td>Both</td></tr>
<tr><td>Register address size</td><td>▪ 1 Byte
▪ 1 Word (default value)
▪ 2 Words</td><td>Both</td></tr>
<tr><td>Port Type</td><td>The communication ports that have to be used for the devices:
▪ RTU for Modbus RTU communication via RS485 port
▪ TCP for Modbus TCP/IP communication via Ethernet port</td><td>Both</td></tr>
<tr><td>Baud rate</td><td>Range of allowed baud-rate values
<table>
<tr><th>Type</th><th>Default</th><th>Values</th></tr>
<tr><td>int16</td><td>7 [9600bps]</td><td>0=110, 1=150, 2=300, 3=600, 4=1200, 5=2400, 6=4800, 7=9600, 8=19200, 9=38400, 10=57600, 11=115200, 12=256000</td></tr>
</table></td><td>RTU</td></tr>
<tr><td>Parity</td><td>Range of allowed parity</td><td>RTU</td></tr>
<tr><td>Data bits</td><td>Range of allowed data bits</td><td>RTU</td></tr>
<tr><td>Stop bits</td><td>Range of allowed stop bits</td><td>RTU</td></tr>
<tr><td>Default Port Configuration</td><td>For each of them, the user can select the default communication value that will be shown during the device configuration.</td><td>RTU</td></tr>
</table>

18.2.5 Things to know

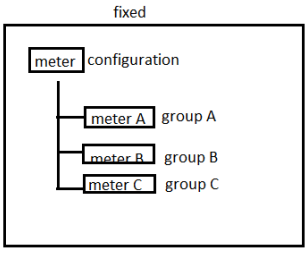
1.2.5.1 Address Type

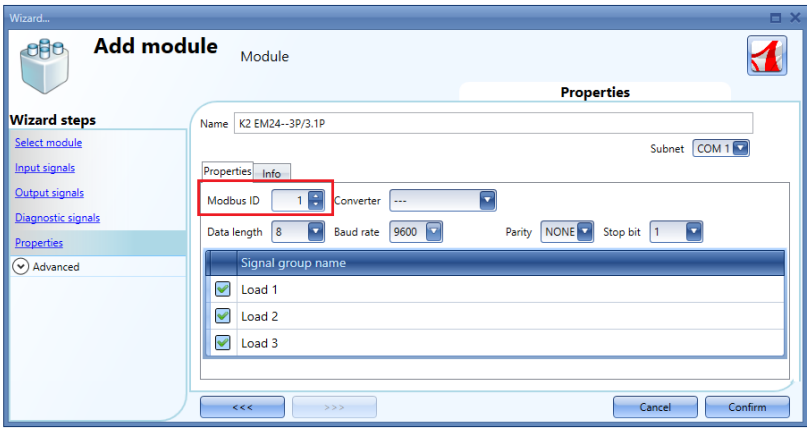
The *Address Type* defines the layout that the driver will assume when the relevant device is imported in the UWP 3.0: the device will be configured according to the driver. In order to define which is the right value to select during the driver editing, the user can consider the following examples.

1) Address Type - Fixed

In the *Properties* tab of the Tool, the user has to set the Modbus ID: this value will be assigned to all the *Groups* and the relevant variables.

E.g. **Load1** is set with Modbus ID 1, **Load2** is set with Modbus ID 1, **Load3** is set with Modbus ID 1.

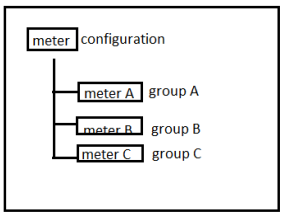


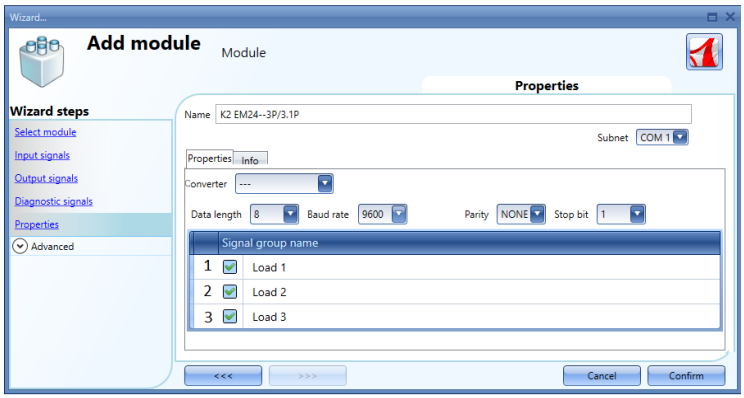


2) Address type - Consecutive

In the *Properties* tab of the Tool, the user has to set the Modbus ID for the first *Group*: incremental ID values will be assigned to the *Groups* and the relevant variables.

E.g. **Load1** is set with Modbus ID 2, **Load2** is automatically set with Modbus ID 1, **Load3** is automatically set with Modbus ID 1.

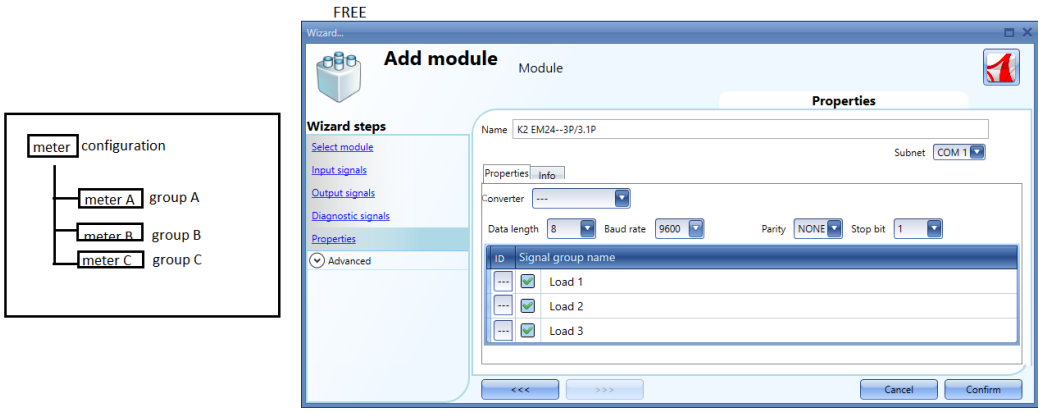




3) Address type - Free

In the *Properties* tab of the Tool, the user can set whatever Modbus ID value for each *Group*.

E.g. **Load1** is manually set with Modbus ID 2, **Load2** is manually set with Modbus ID 4, **Load3** is manually set with Modbus ID 5.

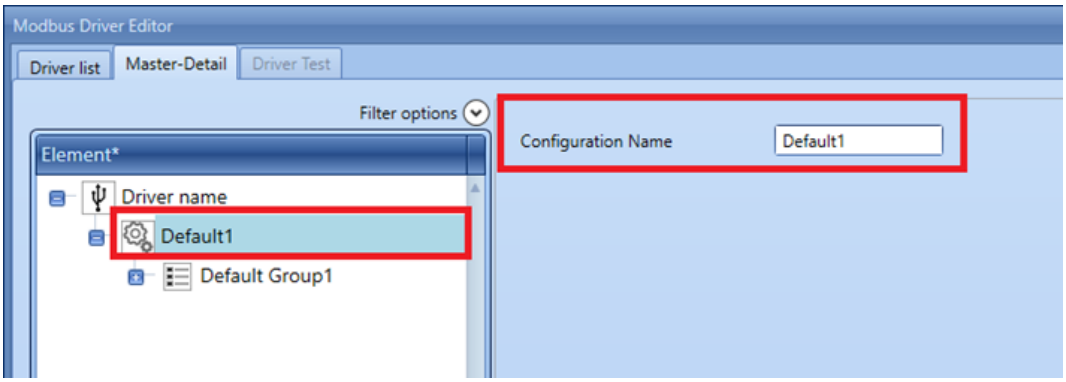


18.2.6 [B] CONFIGURATION level overview

Configurations are different working modes in the same *Modbus device*. Some meters may have different working modes, which allow them to work in 3-phase or 1-phase systems. For this reason, a *Modbus driver* of a *Modbus device* may contain different configurations.

Note: For further information, see ***How to Add a Modbus device in the configuration****.*

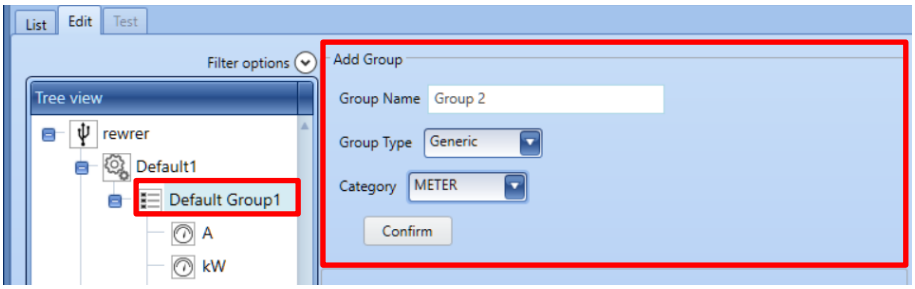
A configuration can be added, edited and deleted from the *Edit* tab of the *Modbus Driver Editor* tool: the procedure *(Part3 – Adding/editing* a Configuration (B)*)* may vary according to the type of the Modbus device whose driver has to be developed.



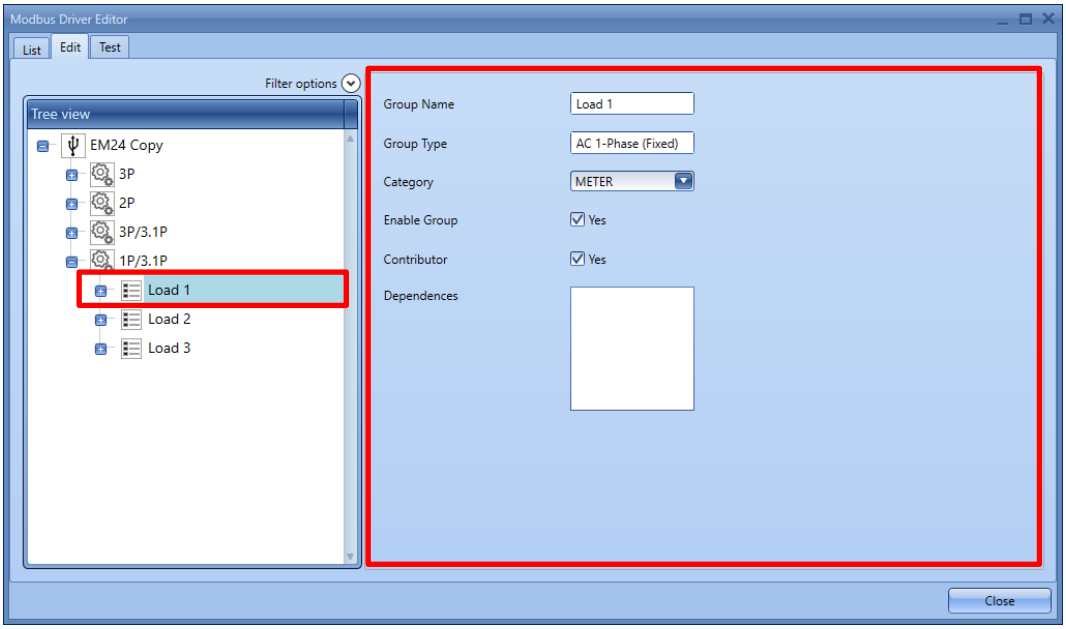
18.2.7 [C] Group level overview

Groups are set of variables in the same configuration. Some meters may have different groups in the same configuration, usually corresponding to a sub-meter. In this section of the interface, it is possible to add, edit, save and delete a *Group* (*Part4 – Adding/editing/coping* a Group (C)).

Add a new Group



Edit a Group

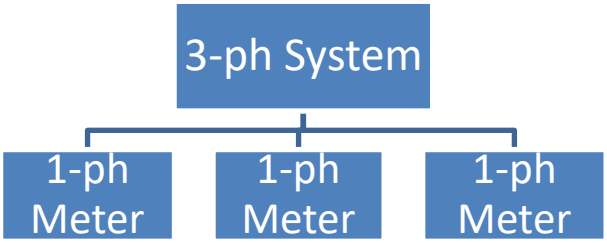
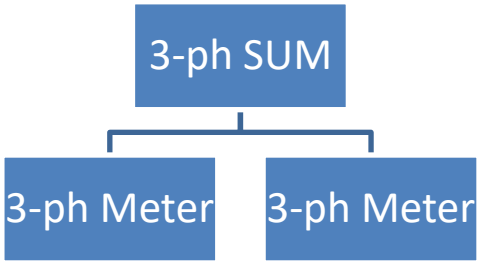


The *Group* is a container of variables and its available parameters are as follows:

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td>Group name</td><td>It identifies the Group in the relevant Configuration.</td></tr>
<tr><td>Group type</td><td>It can be selected among these options:
<table>
<tr><th>Group Type</th><th>Description</th></tr>
<tr><td>Generic</td><td>For variables that derive neither from AC nor DC meters.</td></tr>
<tr><td>AC 3-phase</td><td>For standard 3-phase energy meters or power analysers</td></tr>
<tr><td>AC 1-phase (free)</td><td>For standard 1-phase energy meters or power analysers that have to be linked to a specific phase during the set-up process; the phase is defined at the group level.</td></tr>
<tr><td>AC 1-phase (fixed)</td><td>For standard 1-phase energy meters or power analysers that have to be linked to a specific phase during the set-up process; the phase defined at the configuration level.</td></tr>
<tr><td>AC 1-phase(L1)
AC 1-phase(L2)
AC 1-phase (L3)</td><td>For standard 1-phase energy meters or power analysers linked to a specific phase (L1, L2 or L3).</td></tr>
<tr><td>DC</td><td>For direct current energy meters or power analysers</td></tr>
<tr><td>Digital</td><td>For group of variables aimed at managing digital inputs or variables status.</td></tr>
</table>
Note: Once added, the Group type cannot be changed but the user can customize the variables list.</td></tr>
<tr><td>Category</td><td>It defines the structure of the device.
The user has to select one of the following categories:
<table>
<tr><th>Category</th><th>Description</th></tr>
<tr><td>METER</td><td>To add a group of energy variables (default value).
A predefined set of variables will be added according to the Group type.</td></tr>
<tr><td>SYSTEM</td><td>To add a SYSTEM group of variables (in the case of Modbus devices: multiple groups are dedicated to single phases and one group contains SYSTEM variables).</td></tr>
<tr><td>SUM</td><td>To add a SUM group of variables, meant as a sum of the other sub-meters (groups) that have to be selected in the Dependency field.

Note: Only for AC or DC meters.</td></tr>
</table></td></tr>
<tr><td>Enable</td><td>If the Enable check-box field is flagged, it will appear during the configuration of the Modbus device in the UWP 3.0 Tool.</td></tr>
<tr><td>Contributor (Deprecated)</td><td>It indicates if the Group contributes or not to the totalizing functions of VMU-C. This permits to avoid adding the same contribution twice in the case of SUM groups.

Note: It is pointless in the UWP 3.0 management.</td></tr>
</table>

Parameter	Description
Dependency	It links individual groups by defining a “parent” group such as SYSTEM. The user has to select the METER groups (e.g. an Energy Meter that is able to manage 2x 3-Phase SYSTEMS) that have to be represented as SUM.   The first example is a structure based on METER and SYSTEM items. The METERs are 1-phase, while SYSTEM includes 3-phase System variables. SYSTEM is used in those meters where it is possible to split 1-phase system variables into 3-phase system variables and treat a 3-phase meter as the union of three 1-phase groups and 1 system group. The last example is typically used in those 3-phase multi-meters where multiple 3-phase channels are available in the same physical device (e.g. Carlo Gavazzi’s EM27x family). In this case, in the Modbus map it is often available a virtual meter that is the virtual SUM of the single 3-phase channels: to treat it correctly in the Modbus driver, it is necessary to mark it as SUM.

1.2.7.1 Things to know - Group Type and set of Variables

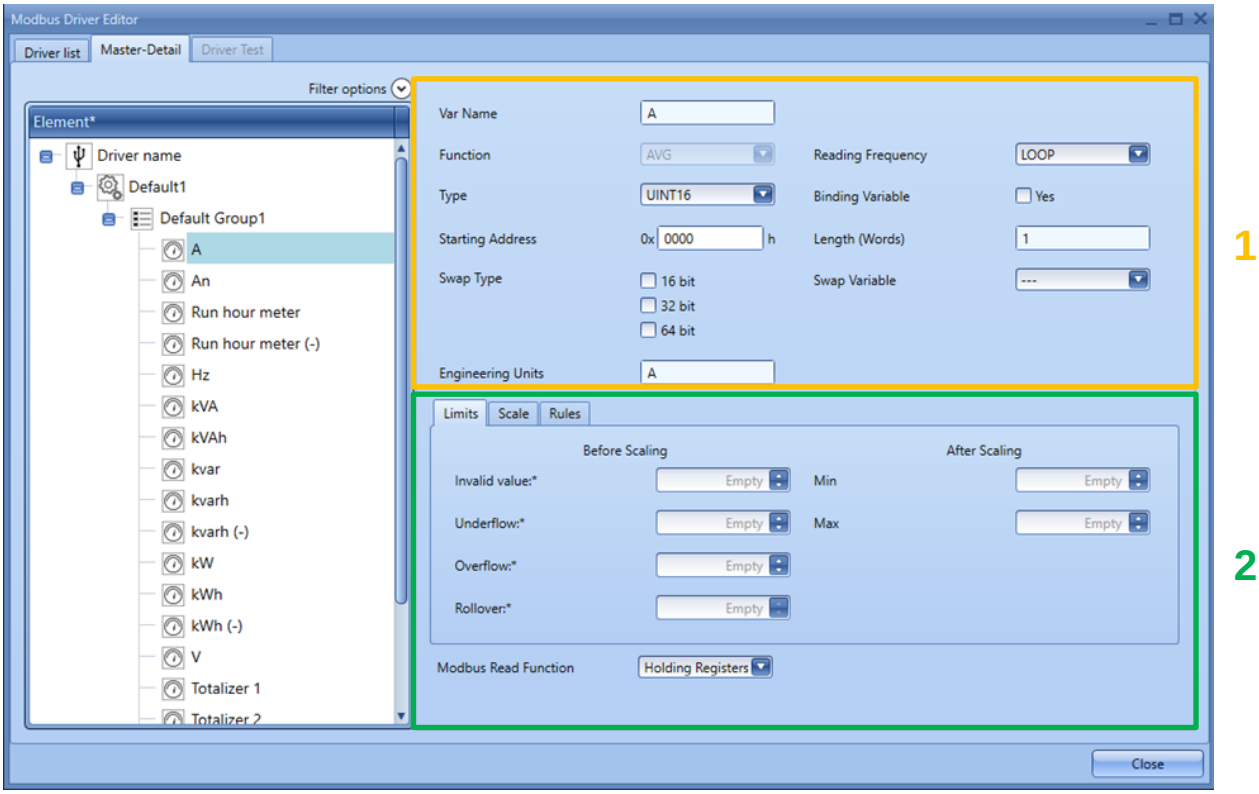
A selection of templates is available for the further development of the driver. The available templates include a wide range of variables for the specific target device (e.g. AC 3-phase includes a typical range of variables for an advanced 3-Phase analyser).

The variables list will appear according to the *Group*.

Group Type	Set of Variables
AC 3-Phase	A L1, A L2, A L3, Run hour meter, Run hour meter (-), Hz, kVA L1, kVA L2, kVA L3, kW L1, kW L2, kW L3, kWh L1, kWh L2, kWh L3, V L-N sys, V L1-N, V L2-N, VL3-N, V L-L sys, kW sys, kvar sys, kvar L2, kvar L3, kVA sys, kWh ac, V L1-L2, V L2-L3, V L3-L1, PF sys, PF L1, PF L2, PF L3, Phase Sequence, THD A L1, THD A L2, THD A L3, THD V L1-N, kvar L1, kvarh L1, kvarh L2, kvarh L3, THD V L2-N, THD V L3-N, kvarh (-) L1, kvarh (-) L2, kvarh (-) L3, kWh (-) L1, kWh (-) L2, kWh (-) L3, kVAh L1, kVAh L2, kVAh L3, kWh (-), kvarh, kvarh (-), kvarh (C), kvarh (L), A sys, kvar dmd
AC 1-Phase	A, An, Run hour meter, Run hour meter (-), Hz, kVA, kVAh, kvar, kvar (-), kW, kWh, kWh (-), V, Totalizer 1, Totalizer 2, Totalizer 3, PF, THDA, THDV
DC	A, V, kW, kWh
Totalizer	Totalizer 1, Totalizer 2, Totalizer 3
Generic	Generic Var 1, Generic Var 2, Generic Var 3, Generic Var 4, Generic Var 5, Generic Var 6, Generic Var 7, Generic Var 8, Generic Var 9, Generic Var 10, Generic Var 11, Generic Var 12, Generic Var 13, Generic Var 14, Generic Var 15, Generic Var 16, Generic Var 17, Generic Var 18, Generic Var 19, Generic Var 20, Generic Var 21, Generic Var 22, Generic Var 23, Generic Var 24, Generic Var 25
Digital	Digital, Multi-state

18.2.8 [D] Variable level overview

In this section, it is possible to add/edit/save/delete a *variable*, which is always contained in a *Group*.
A predefined set of variables is automatically created according to the *Group Type*.
There are two groups of settings: the parameters in the **part [1]** define how to read the variable.



The following table shows the **part [1]** parameters

Parameter	Description
Var Name	It shows the variable name. It can be filled in manually only for a *Generic* variable. Otherwise, the value is automatically set by the software according to the *Group* (*Load*).
Function	It is a function to apply to the samples in the UWP 3.0 (during the sampling time interval). **Function Type** / **Description** / **Electric** / **Generic** / **Control** / **Digital** AVG / Values that have to be averaged / ☑ / / / / COUNTER / Values that have to be averaged / ☑ / / / / DMD / Values that have to be averaged / ☑ / / / / INFO / Values that have to be averaged / / ☑ / / / SWAP / Values that have to be averaged / / / ☑ / SCALE / Values that have to be averaged / / / ☑ / ALARM / Values that have to be averaged / / / / ☑ *Note: For all the Electric variables that are automatically added to the Group, the Function value is set by default and it cannot be changed.*
Type	It defines the *Data Type* of the target variable, as shown in the Modbus map (see *Modbus Data Types* details).
Starting Address	It defines the *Initial hex address* of the target variable, as shown in the Modbus map.
Swap Type	Any combination of swaps at 16/32/64 bits level (see *SWAP conversion*).

Parameter	Description
Engineering Units	Every Modbus variable must be assigned a unit as the data are transmitted without dimensions. The Engineering Unit is automatically set for the *Electric* variables. *Note: A wide range of units is available.*
Reading frequency	It defines the frequency used by the UWP 3.0 for the register reading. The available options are: • **LOOP***:* for the variables, the register is continuously read. This should be selected for instant values and energy variables; • **ONCE**: for register reading only during the Modbus device / UWP 3.0 start-up phase (e.*g. Read Serial number once during the start-up phase).*
Binding variable	It can be flagged only for *Control* variable and it can be used to link together the *Control* variable and the *Electrical* variables to Swap/scale, according to the type of those variables. All of them will be read in the same Modbus frame. This kind of variable is useful in case the UWP3.0 has to compute a new value starting from other registers. The variables will be linked to the target ones: this operation is helpful when some variables are used together for calculations and when the values have to be read in the same frame.
Length (Words)	It defines how many registers have to be read. It appears in the Modbus map of the *Modbus device.*
Swap Variable	The user can select a *Control* variable in order to swap the registers of the variable, according to its value. **If the Control variable value is… / Then the swap…** 0 – HAS TO BE MANAGED Other than 0 – HAS NOT TO BE MANAGED
Modbus read function	The data types for Modbus are divided into variables that monitor, control, and carry out other functions. See the Modbus protocol documentation of the Modbus device that matches each logic code. **Function code / Description** Holding register (0x03) – The UWP 3.0 reads the Register using Modbus function 03 Input register (0x04) – The UWP 3.0 reads the Register using Modbus function 04 Read coil (0x01) – The UWP 3.0 reads the Register using Modbus function 01

18.2.9 Things to know

1.2.9.1 Modbus Data Types

The identification of the data type is necessary to develop the *Modbus driver*. Depending on the device documentation, the data type may be determined according to anyone of the above criteria.

The variables are represented by integers or floating numbers, with 2 complement notations in case of "signed" format. See the following table:

Type Format	Description	Bits	Range
INT16	Integer	16	-32768 .. 32767
UINT16	Unsigned integer	16	0 .. 65535
INT32	Double integer	32	-2^{31} .. 2^{31}
UINT32	Unsigned double integer	32	0 .. $2^{32}-1$
UINT64	Unsigned long integer	64	0 .. $2^{64}-1$
INT64	Long Integer	64	
FLOAT	IEEE754 SP Single-precision floating-point	32	$-(1+[1-2^{-23}])x2^{127}$ .. 2^{128}
Hour		32	

Type Format	Description	Bits	Range
UByteLow	Unsigned Low Byte *Note: It will be used the low Byte of the Word*	8	
UByteHigh	Unsigned High Byte *Note: It will be used the high Byte of the Word*	8	
ByteLow	Unsigned Low Byte *Note: It will be used the low Byte of the Word*	8	
ByteHigh	Unsigned High Byte *Note: It will be used the high Byte of the Word*	8	
DOUBLE	IEEE754 DP Double-precision floating-point	64	2.2e-308 ... 1.79e308

1.2.9.2 **SWAP conversion**

The swap is an operation that converts the Modbus address. Depending on the selected data type, it allows to properly order all the data bytes of the frame to read correctly the data.

Request frame: [01h] [03h] [00h] [00h] [00h] [02h] [C4h] [0Bh]

Reply frame: [01h] [03h] [04h] [08h] [FCh] [12h] [00h] [38h] [63h]

Data bytes: [08h] [FCh] [12h] [00h]

[A] [B] [C] [D]

The following table provides different examples of conversion:

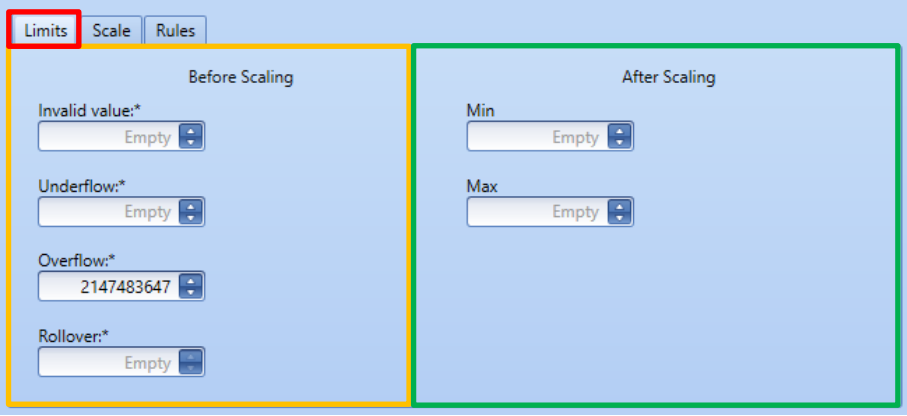
Conversion	Result
SWAP 16	SWAP16 can be used to swap the nibbles of a byte: [B][A] [D][C] → [FCh] [08h] [00h] [12h]
SWAP 32	SWAP32 can be used to swap bytes of a Word: [C][D] [A][B] → [12h] [00h] [08h] [FCh]
SWAP 16 + 32	SWAP16+32 can be combined together: [D][C] [B][A] → [00h] [12h] [FCh] [08h]
SWAP 64	In the case of registers that are 4 words long, also the SWAP 64 can be used: [A][B] [C][D] [E][F] [G][H] [G][H] [E][F] [C][D] [A][B]

2 [2D] The lower part of the screen - Parameters

The lower part of the screen contains the parameters that can be used to apply mathematical operations to the target variable. There are 3 tabs: *Limits*, *Scale* and *Rules.*

Limits tab

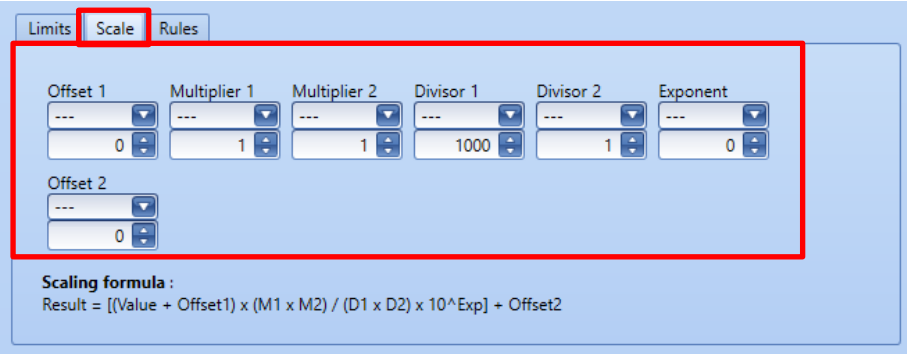
In the *Limits* tab, the user can enter the value aimed at rejecting invalid read values.
This operation can be done before and/or after having read the registers that contain the raw values.



The table below describes the meaning of each field:

	Parameter	Description
Before Scaling	Invalid value	Special decimal value indicating invalid reading
	Underflow	Special decimal value indicating Underflow
	Overflow	Special decimal value indicating Overflow
	Rollover	Decimal value indicating Rollover value
After Scaling	Min	If the read value exceeds these limits, that value will be not considered.
	Max	

Scale tab



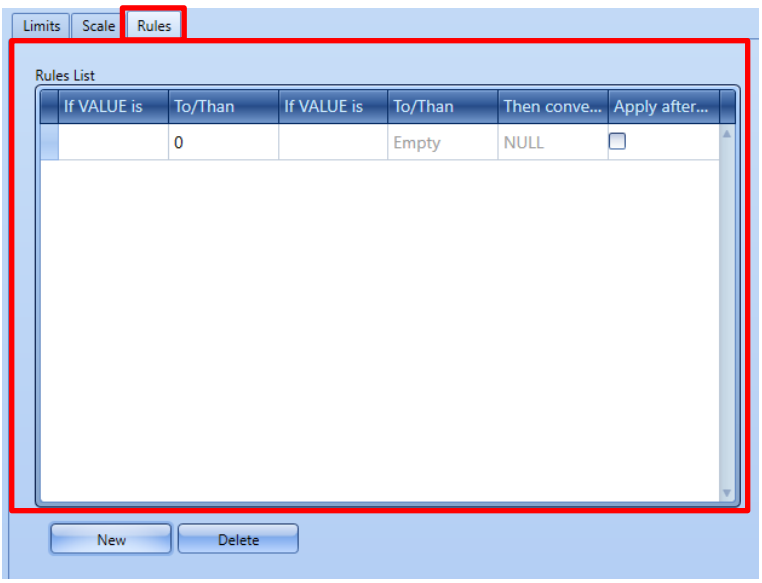
In the *Scale* tab, the user can enter the value aimed at rejecting invalid read values. This operation can be done before and/or after reading the registers that contain the raw values.

Enter a divisor or factor to adjust the applied value to the current value (e.g. correct positioning of the decimal point).

The following table describes the *Scale* tab parameters and the *Scaling formula:*

Parameter	Description
Offset1 Multiplier1 Multiplier2 Divisor1 Divisor2 Exponent Offset2	Scaling formula: it allows to define whatever combination of registers to calculate a variable on the basis of different registers *(e.g. reading + scaling).* *[(Value + Offset1) x (M1 x M2) / D1 x D2) x 10^Exp] + Offset2*

Rules tab



In the *Rules* tab, the user can enter the value aimed at rejecting invalid read values. This operation can be done before and/or after reading the registers that contains the raw values.

The following table describes the *Rules* tab parameters:

Rules list	List of configured rules
If VALUE is	Operator used to evaluate the condition
To/Than	Value to be checked
If VALUE is	Operator to evaluate the condition
To/Than	Value to be checked
Then convert to	Result of the evaluation
Apply rule after scaling	The rule is applied after the scaling (it is applied before by default)

3 Bitfield variable

In the Bitfield variable, the values of single bits are the important elements because each of them represents a specific status (e.g. Bit 0 = Power ON/OFF, Bit 1 = Communication ON/OFF etc).

The purpose of the user is to assign the correct meaning to the target bit in the selected Bitfield.

See the following table:

BIT POSITION	DESCRIPTION
0	Power ON/OFF
1	Communication ON/OFF
2	Communication Error
3	Faulty system
4	Over current alarm
5	Not assigned
6	Not assigned
7	Not assigned

Examples:
10000000 = Power ON
10100000 = Communication Error (with power ON)

Note: With these types of variables, the Modbus Read Function to use is the ***Read coil (0x01)****.*

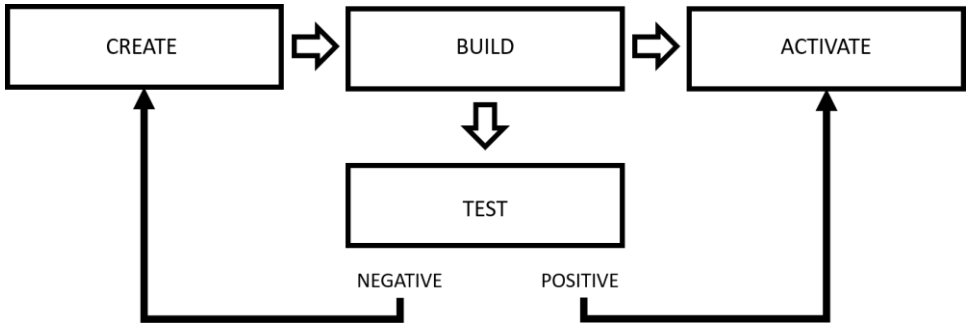
4 Multi-state variable

The multi-state variable is used to represent finite set of values (e.g. the range of alert status of an inverter).
Each value has a specific meaning that depends on the target device. The user has to specify each single value to completely define the variable.

MULTI-STATE VALUE	DESCRIPTION
10	Over current alarm
15	Grid connection alarm
20	Communication Error
23	Overheating alarm
28	General failure

18.3 How to create a Modbus driver – Real Use case (EM24-DIN)

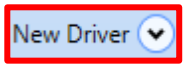
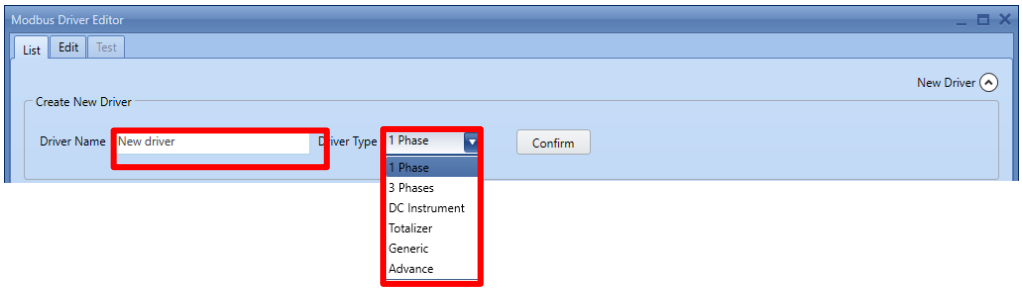
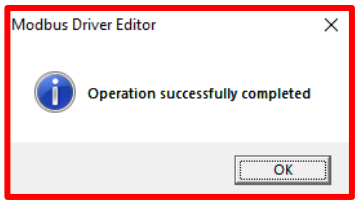
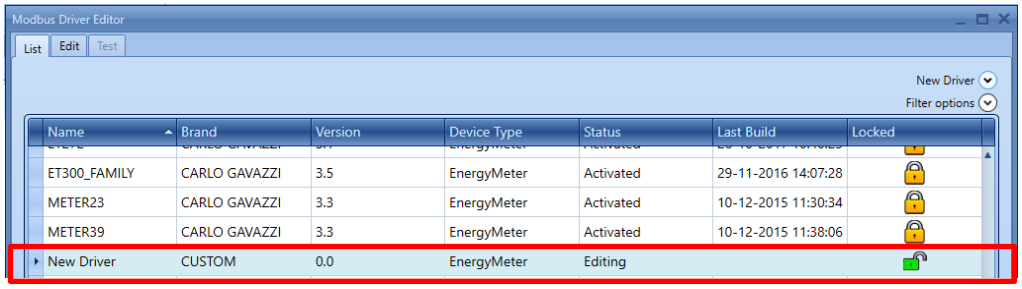
The diagram below represents the process the user has to follow to develop a driver for a Modbus device. The procedure is quite the same for the creation/editing, except for a few steps.



The following procedure is related to the creation of a commonly used **Energy meter 3-Phase**, within a couple of widely used configurations.

18.3.1 Part1 - Creating/editing a Modbus driver

Once the new *Driver* has been created, the *Modbus driver editor* (MDE) switches to the **tree view** to allow the configuration of the relevant parameters.
A list of driver templates is available during the creation of a new driver: the template is compiled by VMU-C according to the target driver in order to define the correct set of variables.

Step	Action
1	In the *List* tab, click on  to expand the *Create New Driver* menu.
2	In the *Create New Driver* area: • Enter the name of the Driver in the *Driver Name* field; • Select the type from the *Driver Type* combo-box. 
3	Click on *Confirm* to save the changes.  A pop up will appear to notify the successful operation.
4	A *New Driver* will be added to the *List* tab; it should have a green padlock. 
5	Edit the *Driver* following one of these actions: • Double click on an Item from the list; • Right-click on one of the items from the list and click on the *Edit Selection* option from the *contextual menu*; • Select an item from the list and access the *Edit* tab. **N.B: The following actions can be performed only for unlocked (with the green padlock) Modbus drivers in the List tab.**

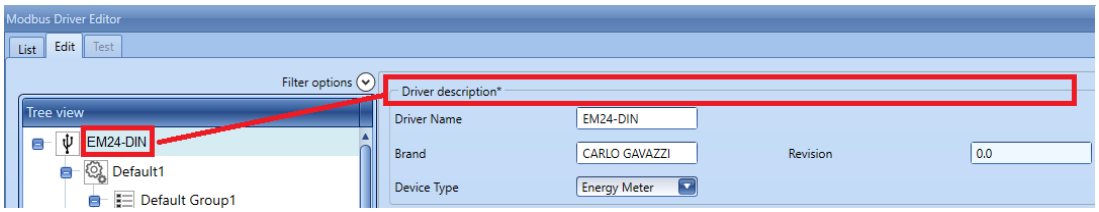
For further information, see the *Part3 – Adding/editing* a Configuration (B) procedures.

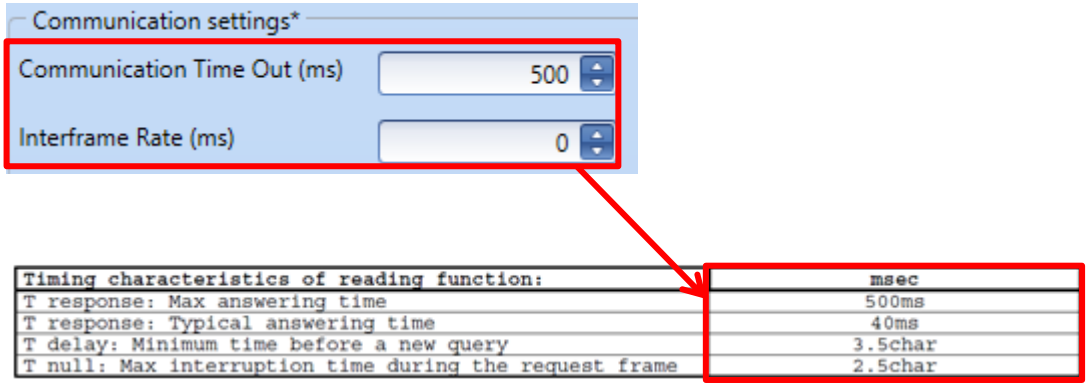
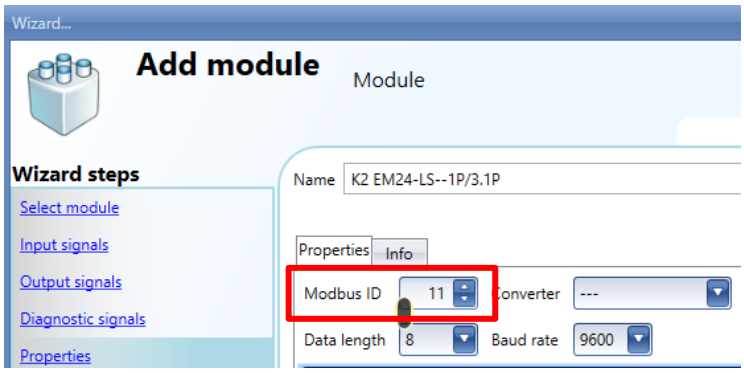
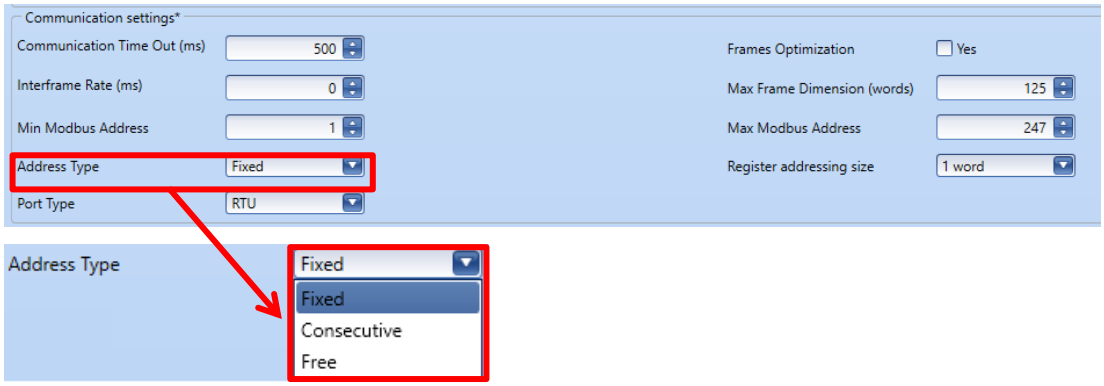
18.3.2 Part2 – Setting the device parameters (A)

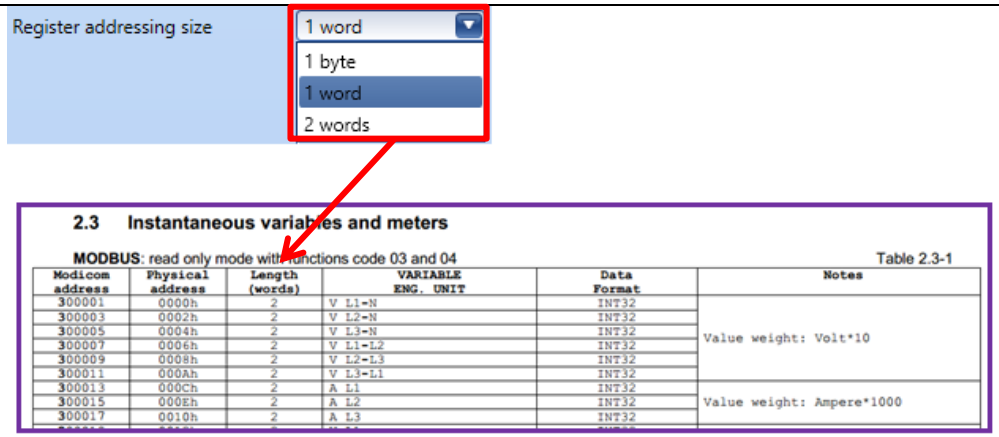
The communication parameters must be set according to the Modbus device specifications, such as the media connection or the register type: to get this information, the user has to refer to the *Modbus map* of the device.

Follow the steps described below to configure the parameters (for more details, see *Things to know*).

Note: In the following example, an EM24-DIN device has been considered.

Step	Action
1	In the *Driver description* area: • Edit the *Driver Name* of the device. • Enter the name of the device manufacturer (*Brand*). • Select the *Device Type* from the combo-box.  *NB: The Revision field cannot be edited. Its value will increase each time a driver is compiled.*
2	In the *Communication settings,* the *Frames Optimization* and the *Max Frame Dimension (words)* have to be set according to the limits set in the target device communication protocol or, possibly, in the communication network (e.g. Some protocol converters or gateways have some restrictions in Modbus frames sizes).

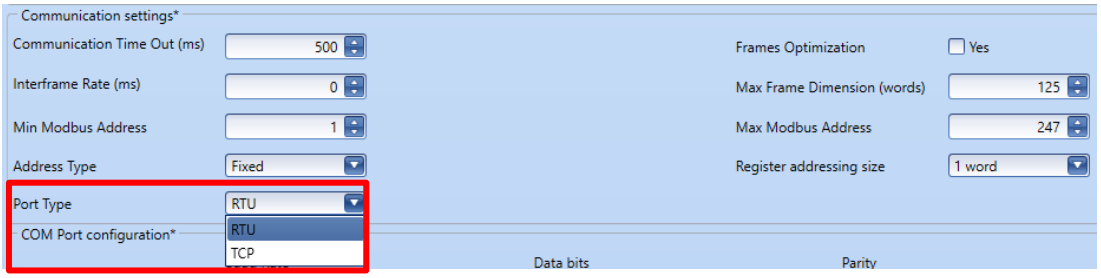
3	In the *Communication settings* area, enter the: • *Communication Time Out (ms);* • *Interframe Rate (ms).* 
4	In the *Min Modbus Address* and *Max Modbus Address* (*Communication settings*), set the valid Modbus ID range. This will restrict the available range during the device setting up, after having imported the driver (see the picture below). 
5	Select an *Address Type*. 
6	In the *Register address size* combo-box, set the size of the register.



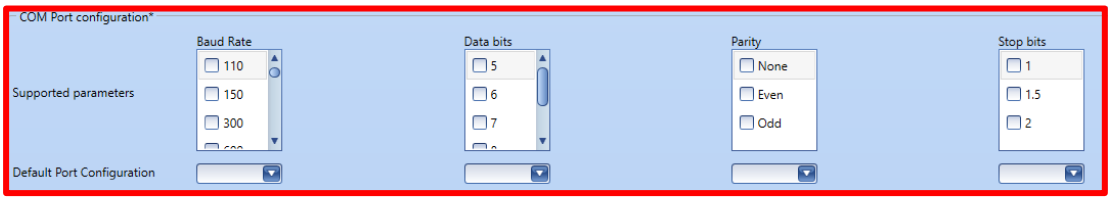
7

In the *Port Type*, set the option used to connect the Modbus device to the UWP 3.0:

- RTU for serial communication via RS-485 port
- TCP for communication via Ethernet port



ONLY for RTU connection, set the parameters in the *COM Port Configuration* section:

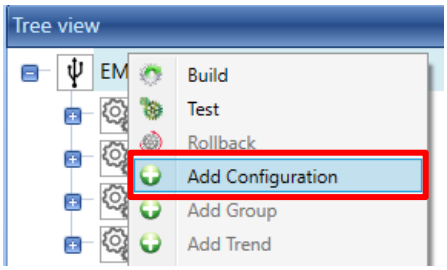
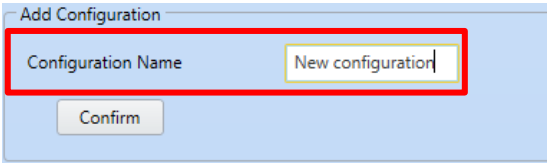
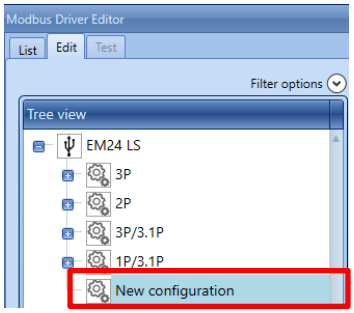


18.3.3 Part3 – Adding/editing a Configuration (B)

In this section of the interface, it is possible to add, edit, save and delete a configuration. The configuration is a container and it has only a parameter: its name.
For complex instruments, it is possible to have multiple configurations, meant as different *Modbus maps* to access the same device.

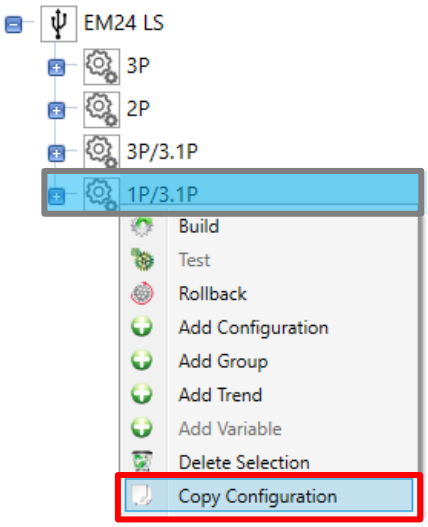
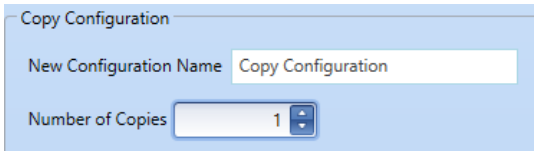
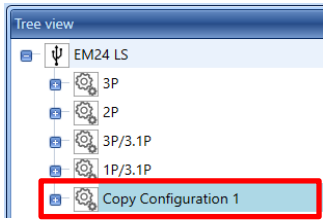
5 Adding a new Configuration

To add a new configuration, follow this procedure:

Step	Action
1	In the **tree view** in the *Edit* tab, select the device level.
2	Right-click and select *Add Configuration* option from the contextual menu. 
3	In the right part of the *Edit* tab, enter the name of the *Configuration.* 
3	Click on *Confirm* to add the new configuration: it will appear in the **tree view** area. 

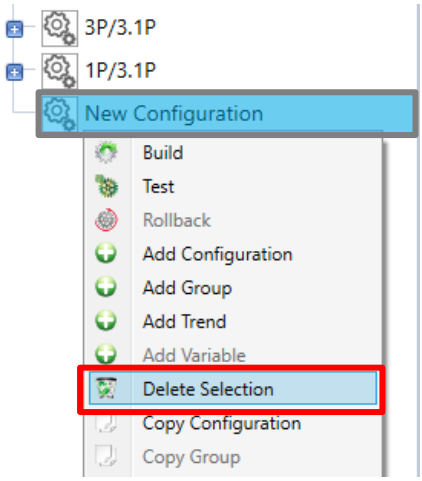
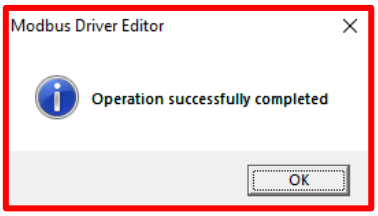
6 Copying an existing configuration

To copy a configuration, follow this procedure:

Step	Action
1	In the **tree view** in the *Edit* tab, select the configuration to copy.
2	Right-click and select *Copy Configuration* option from the contextual menu. 
3	In the right part of the *Edit* tab, enter the name of the configuration and the number of copies to create. 
4	Click on *Confirm* to copy the element in the driver. 

7 Deleting a configuration

To delete a configuration, follow this procedure:

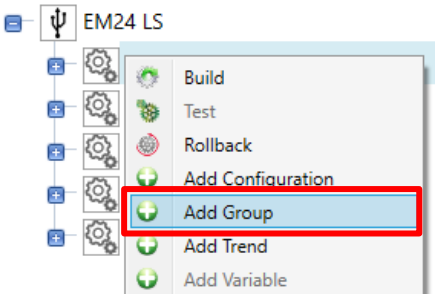
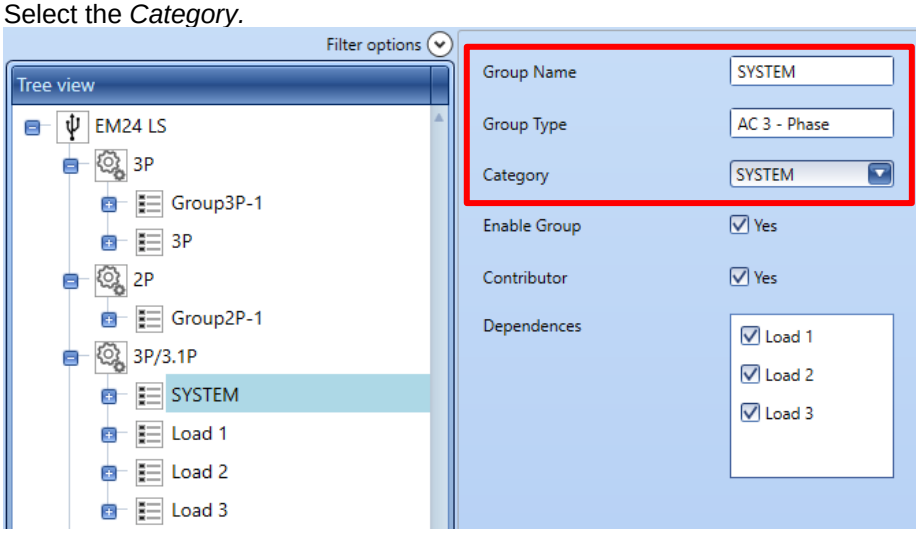
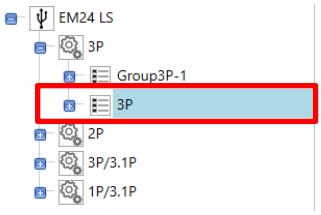
Step	Action
1	In the **tree view** in the *Edit* tab, select the configuration level to remove.
2	Right-click and select *Delete Selection* from the contextual menu.  A pop up will notify that the operation is complete. 

18.3.4 Part4 – Adding/editing/coping a Group (C)

To add/edit a group, at least a configuration must exist. A configuration can have several groups and usually a group is a *Load* element.

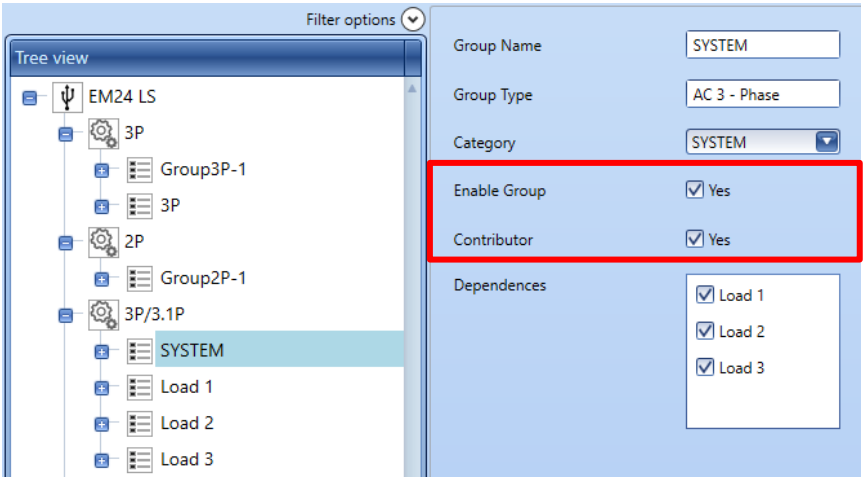
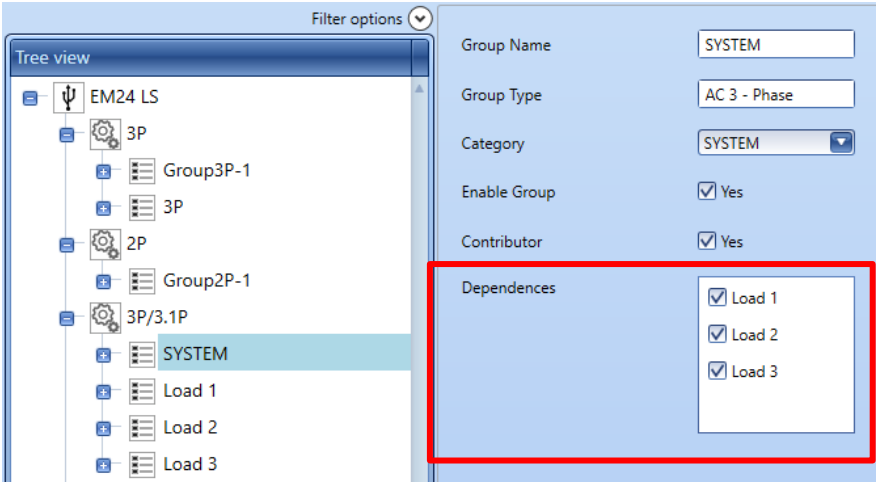
8 **Adding a new Group**

Follow the procedure described below to add a new group to a configuration:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Configuration* element to edit.
2	Right-click and select *Add Group* from the contextual menu. 
3	In the right part of the window: • Enter the name of the *Group;* • Select the *Group Type;* • Select the *Category.* 
4	Click on *Confirm* to add the new *Group* to the driver: it will appear in the **tree view** area. 

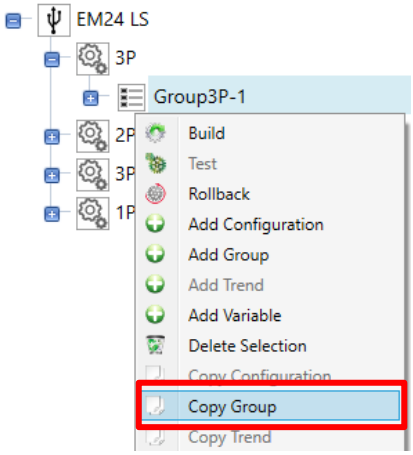
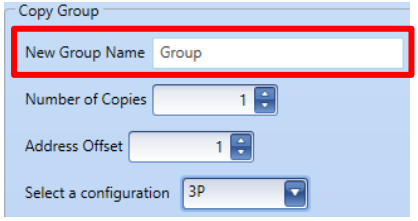
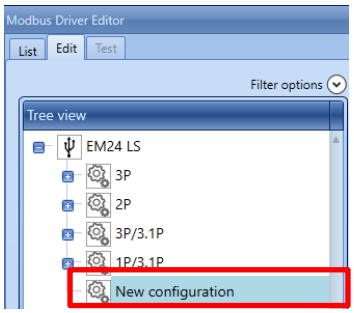
9 Editing an existing Group

Follow the procedure described below to edit an existing group:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* element to change.
2	In the right part of the *Edit* tab, change the: • *Group Name;* • *Group Type;* • *Category.* *Note: see the previous procedure.*
3	Flag the: • *Enable Group* to read all the *Electric Variables* that belong to the selected *Group;* • *Contributor(Deprecated)* to keep the compatibility with VMU-C. 
4	From the *Dependency* field, select the Sub-system (the Groups) to represent as SUM.  *Note: Only for SUM and SYSTEM.*

10 Coping an existing Group

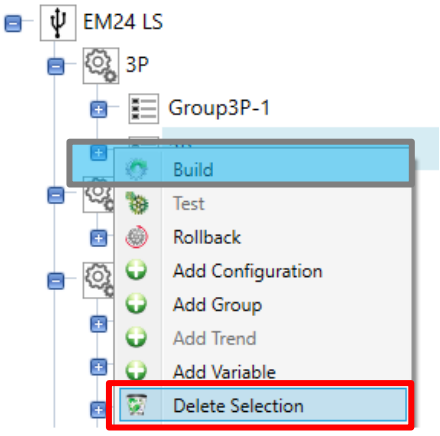
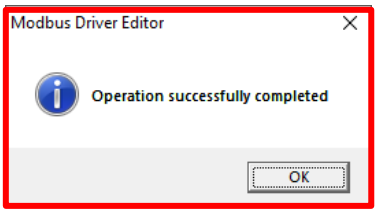
Follow the procedure below to copy an existing *Group*:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Configuration* element to be copied.
2	Right-click and select *Copy Group* from the contextual menu. 
3	In the right part of the *Edit* tab, enter the name of the *Group.* 
4	In the *Number of Copies* field, set the number of copies to create.
5	In the *Address Offset* field, enter the OFFSET increment for each group to be copied
6	In the *Select a configuration* combo-box, select the configuration model that must be applied to the group copy.
7	Click on *Confirm* to add the *new Group* to the driver: it will appear in the tree-view area. 

11 Deleting a Group

Warning. Whenever a group is deleted, all the relevant variables and *Custom Charts* will be deleted.

Follow this procedure to delete a *Group*:

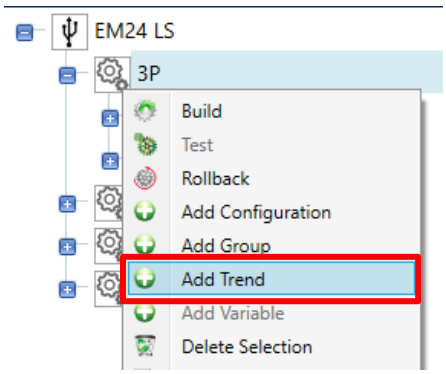
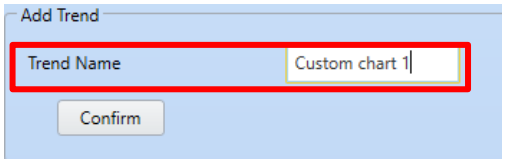
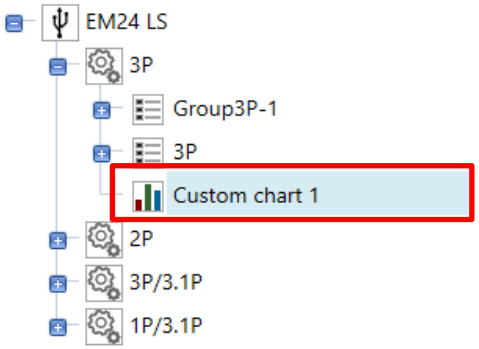
Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* to delete.
2	Right-click and select *Delete Selection* from the contextual menu.
3	 A pop up will notify that the operation is complete. 

18.3.5 Part4a - Adding/editing a Trend (or Custom Chart)

A trend (or *Custom chart)* is a set of variables from one or more groups that are displayed in the Web App at the same time, in the same chart. In this document, Trend and Custom Chart have the same meaning.
If a *Custom chart* is linked to a Modbus driver, the former will be immediately available to be used without needing to be configured afterwards.

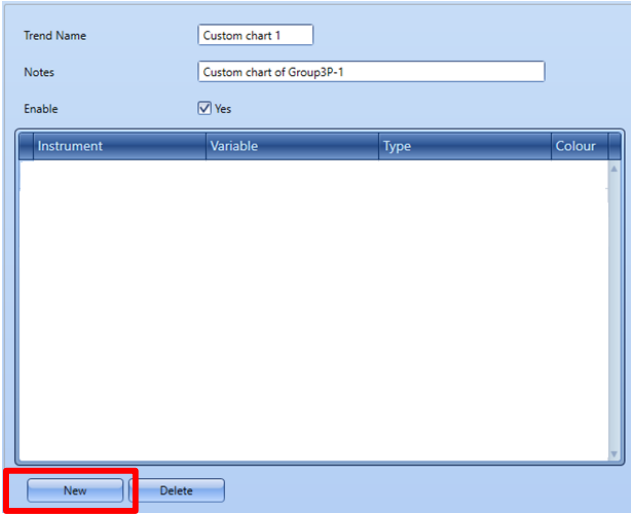
Note: It is possible to link one or more Custom Charts to a driver.

Follow the procedure below to add a *Custom chart*:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Configuration* to be add.
2	Right-click and select *Add Group* option from the contextual menu. 
3	In the right part of the *Edit* tab, enter the name of the *Configuration.* 
4	Click on *Confirm* to add the *new Group* to the driver: it will appear in the tree-view area. 

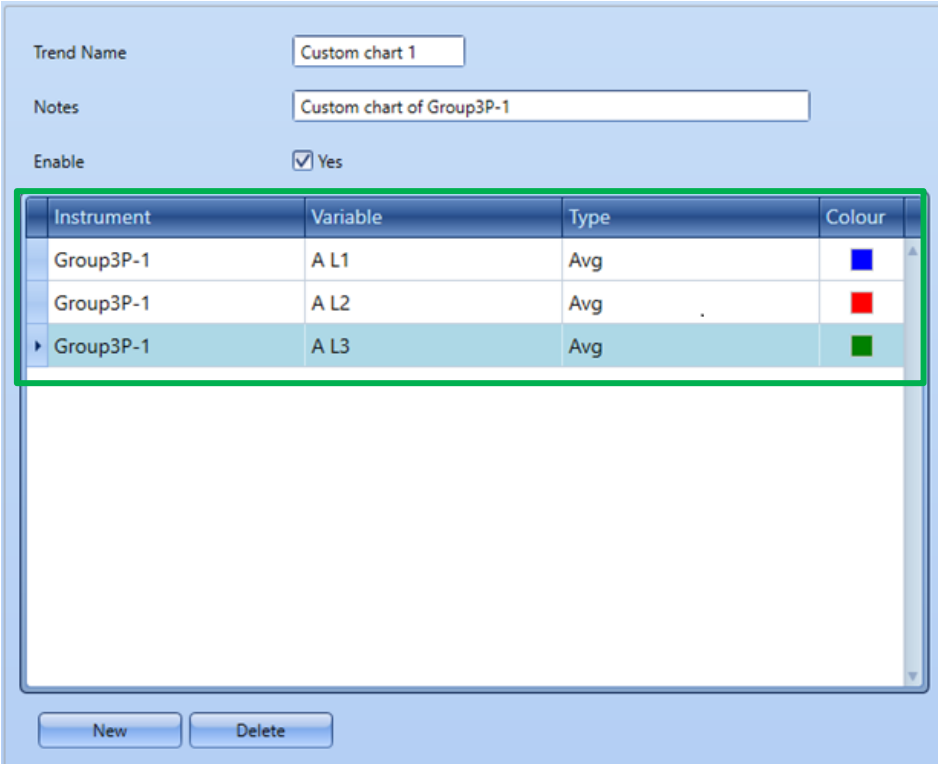
12 Editing a Custom chart

Follow the procedure below:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Custom chart* to edit.
2	Click on *New* to add a new item. 

3

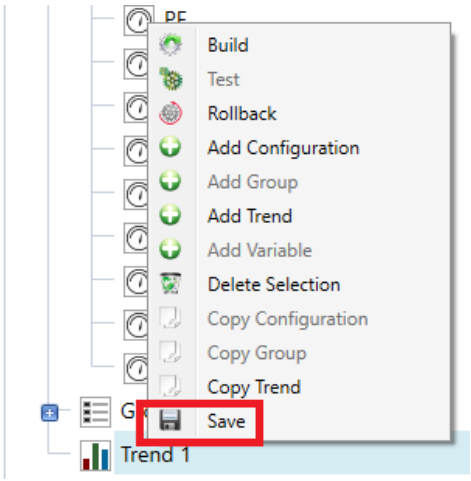
In the right side of the window, enter the chart elements.



Field	Description	Action
Instrument	Container group for the target variable	
Variable		
Type	Average Max Min	VMU-C stores 3 series for each variable, maximum, minimum and average
Colour		
Variable	Target variable	
Type		

4

Right-click on the modified chart to save it.



18.3.6 Part5 – Adding/editing a variable (D)

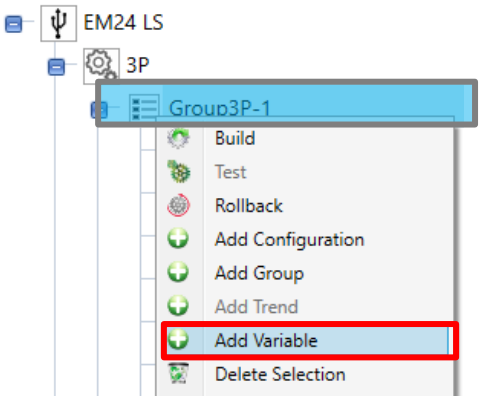
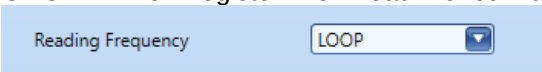
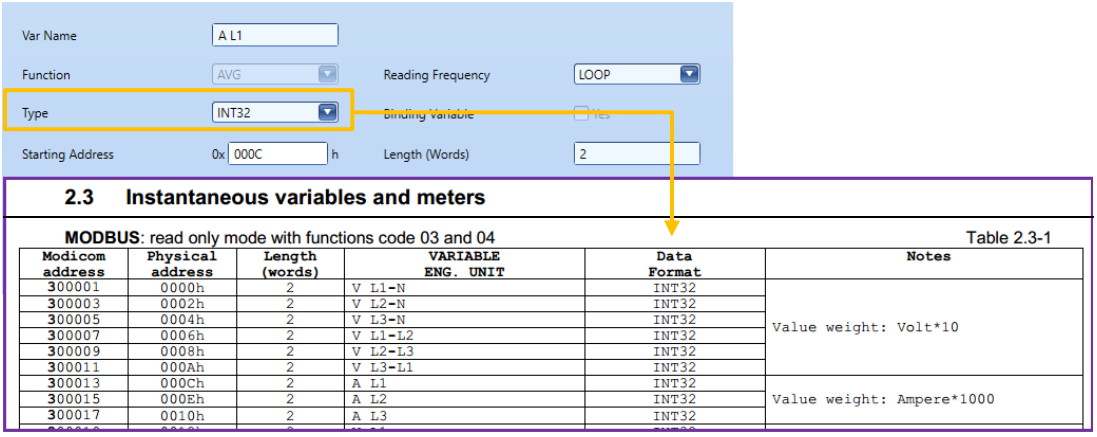
There are different types of variables:

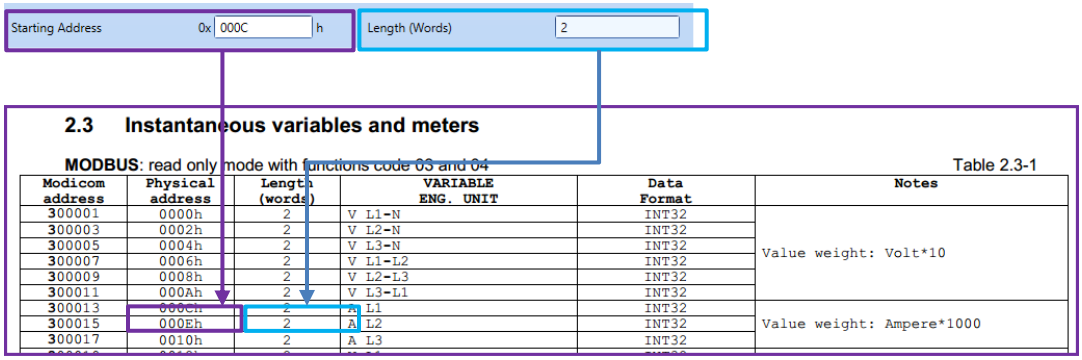
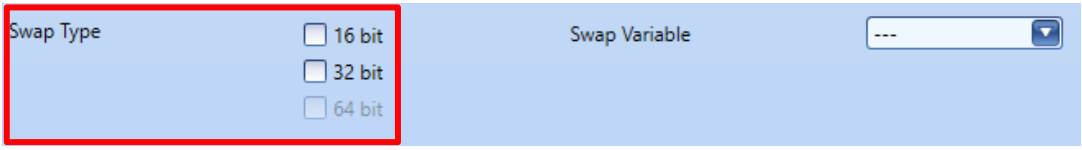
1. Electric variable
2. Control variable
3. Generic variable.

The following chapters will describe how to add each of them.

13 Adding/Editing an Electric variable

Follow the procedure below to add a new variable to the selected *Group*:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* where the *Variable* has to be added. 
2	From the *Reading frequency* combo-box, select: **LOOP**: The register is read every cycle **ONCE**: The register is read once at the start-up of the Modbus device 
3	Select the *Type* according to the available protocol documentation for the target device. 

4	Set the *Starting address* and the *Length (words)*: 
6	The *Swap type* can be chosen among the available choices and a *Swap variable* can be indicated to trigger the swap. 

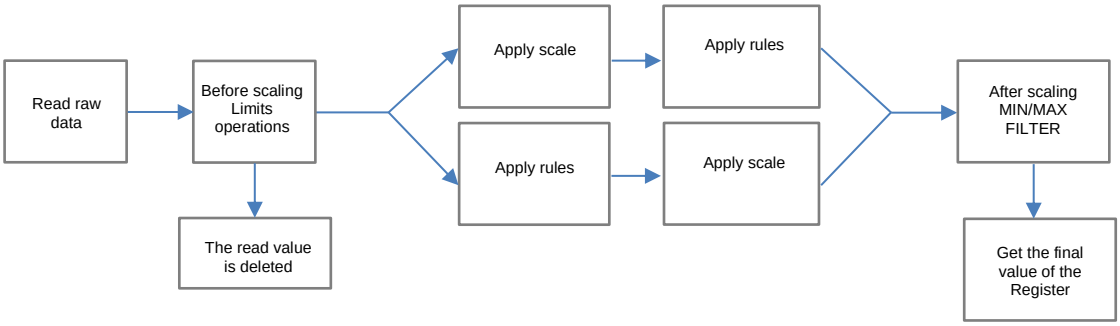
14 Editing an Electric variable

Follow the procedure below to add a new variable to the selected *Group*:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Variable* element.
2	Follow the previously described steps (2-6).
3	**If the variable has to be…** — **Then go to…** Scaled — ***Scaling a variable*** Filtered — ***Filtering undesired values*** Transformed according to rules — ***Adding/edit rules to a variable***

18.3.7 How to apply the calculation

The diagram below shows how the user can apply calculation on a read raw data in order to get the final wanted value.

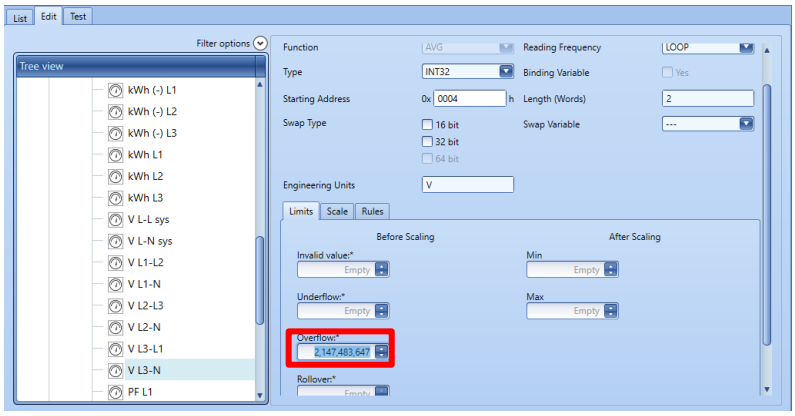


These logical and mathematical operations can be executed for each variable, as indicated in the above diagram.

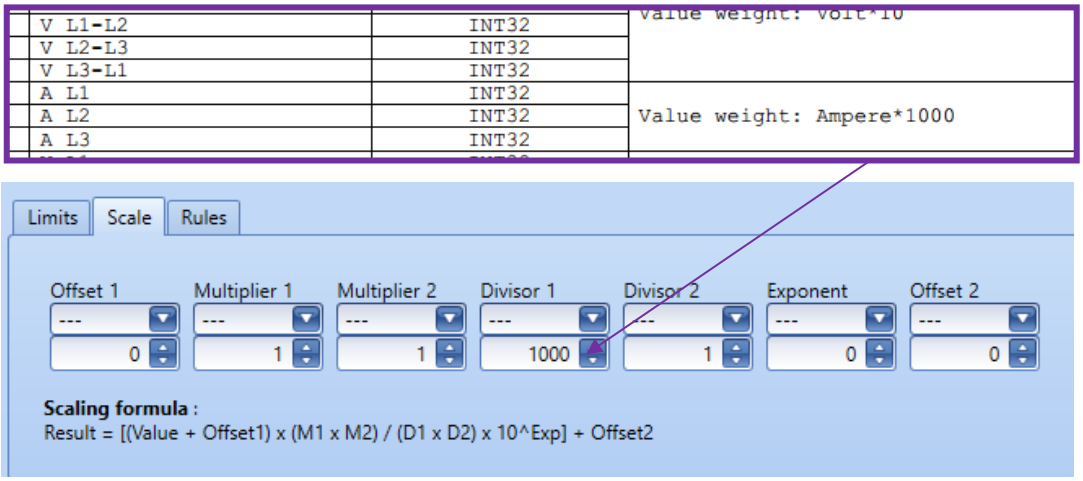
According to the needs, follow one or more step (scale, filter or apply rules).

15 Filtering undesired values

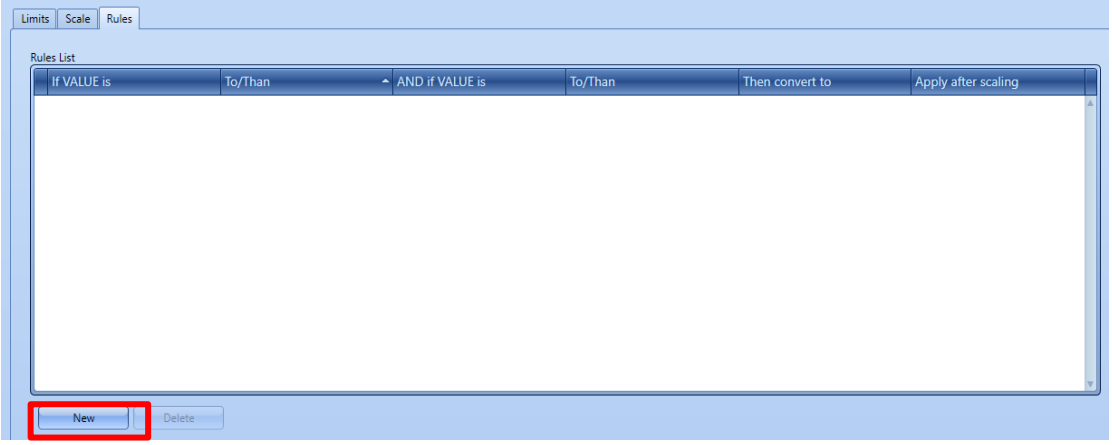
It is necessary when one or more specific values are used in specific conditions. These fields appear in the *Limits* tab.
If the input is above the maximum value, the display shows “EEEE”.
The overflow indication relevant variable is 0x7FFFh (= 2147483647 decimal).

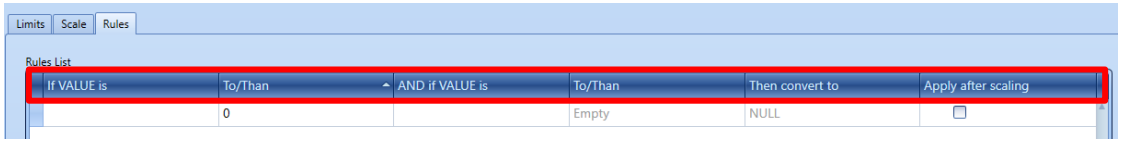
Step	Action
1	Start editing the target variable.
2	Select the *Limits* tab.
3	Complete the relevant OVERFLOW field with the aforementioned value: 

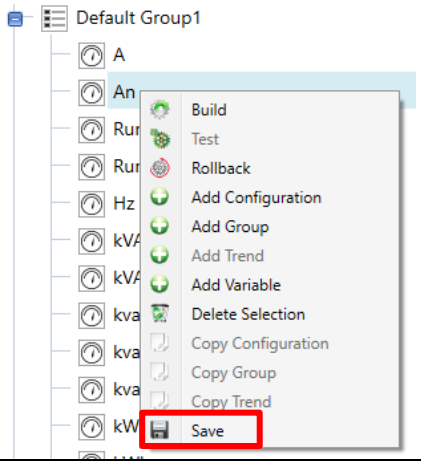
16 Scaling a variable

Step	Action
1	Start Editing the target variable
2	Select the *Scale* tab.
3	 Complete the Scaling formula fields with any necessary value, either fixed or register based; in the latter case, define a CONTROL VARIABLE, following the relevant instructions (see ***Other Variable***).

17 Adding/edit rules to a variable

Step	Action
1	From the *Rules* tab, click on *New* to add a new item. 

<table>
<tr><td>2</td><td>Set the rules.


<table>
<tr><th>Rules list</th><th>List of configured rules</th></tr>
<tr><td>If VALUE is</td><td>Operator to be used to evaluate the condition</td></tr>
<tr><td>To/Than</td><td>Value to be checked</td></tr>
<tr><td>If VALUE is</td><td>Operator to be used to evaluate the condition</td></tr>
<tr><td>To/Than</td><td>Value to be checked</td></tr>
<tr><td>Then convert to</td><td>Result of the evaluation</td></tr>
<tr><td>Apply rule after scaling</td><td>The rule is applied after the scaling (by default it is applied before)</td></tr>
</table></td></tr>
<tr><td>3</td><td>Right-click on the modified variable to save it.

</td></tr>
</table>

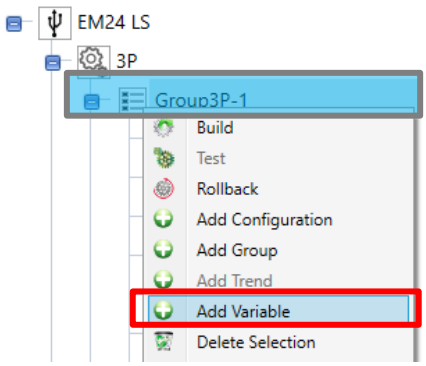
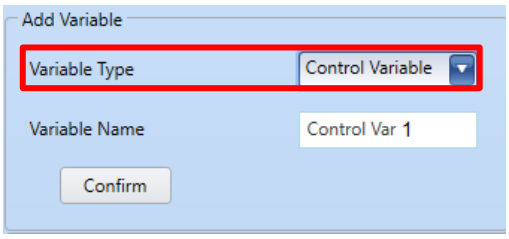
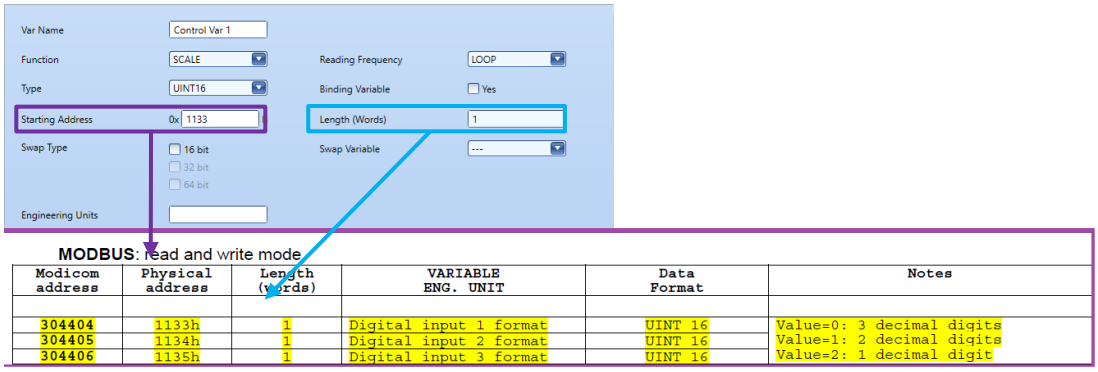
18.3.8 Other Variable

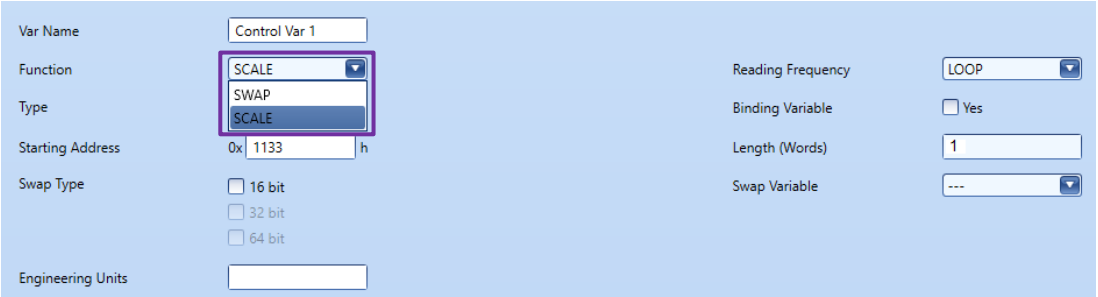
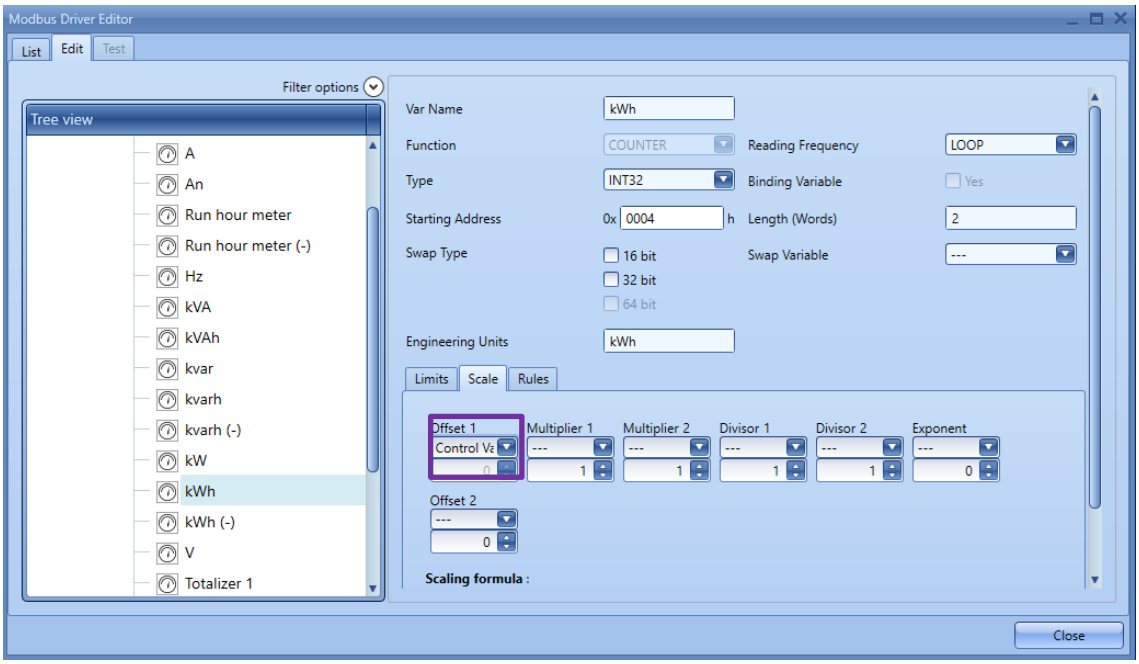
Control variables are available in some Modbus devices that, according to their value, can be used to:

- Scale an electric variable – procedure
- Swap an electric variable – procedure

18 Adding a Control Variable to scale an Electric variable

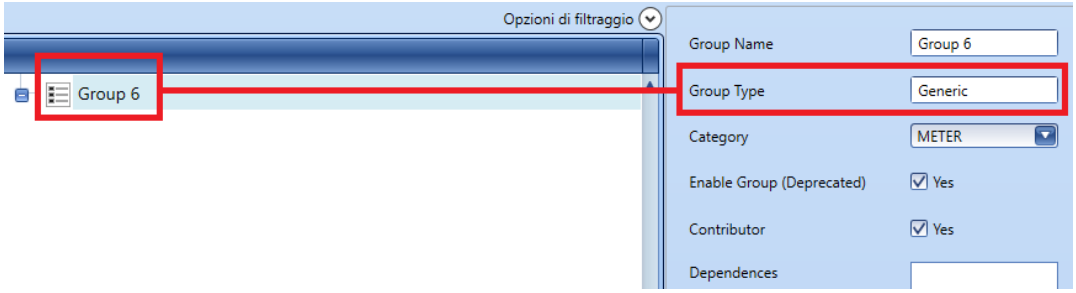
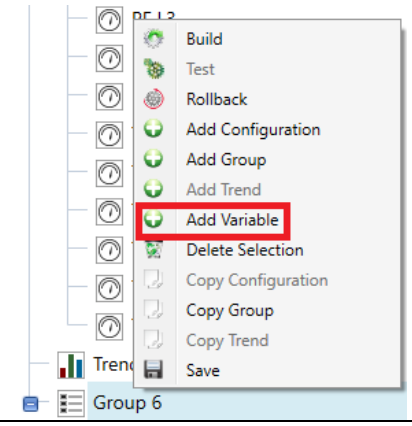
Follow the procedure below to add a new *Control variable* to the selected *Group*:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* where the *Variable* has to be added. 
2	In the right side of the **window**, select *Control Variable* from the *Variable Type* combo-box. 
3	Click on *Confirm* to save the variable.
4	Set the *Starting address* and the *Length (words)*:  **Serial protocols**

5	From the *Function* combo box, select *Scale.* 
6	Add or edit the variable to be scaled and compile the *Scale* tab, including the control variable and any additional item in the scaling formula reported in the tab. 

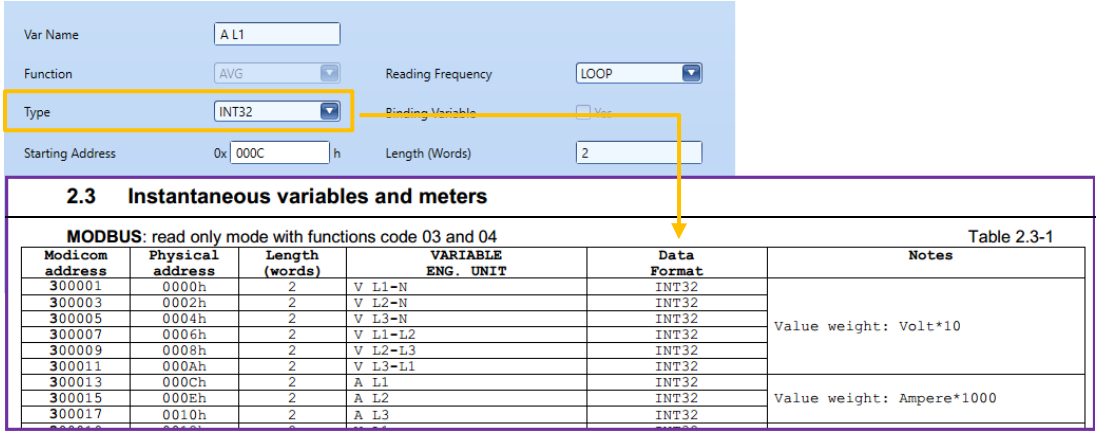
19 Adding a Generic variable

Follow the procedure below to add a *new Generic variable* to the selected *Group*:

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* where the *Variable* has to be added. 
2	Right click to select *Add Variable.* 
3	From the *Variable Type* combo box on the right part of the window, select the *Generic Variable.*
4	Click on *Confirm* to add the variable.
5	From the *Reading frequency* combo-box, select: **LOOP**: The register is read every cycle **ONCE**: The register is read once at the start-up of the Modbus device

6

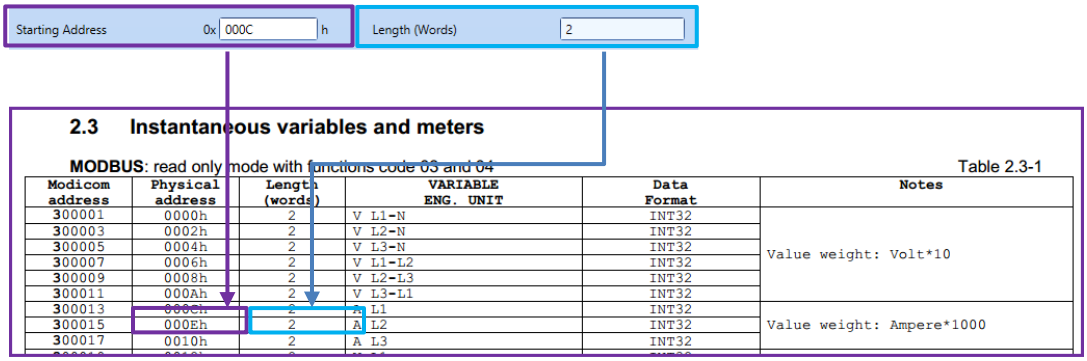
Select the *Type* according to the protocol documentation of the target device.



Serial protocols

7

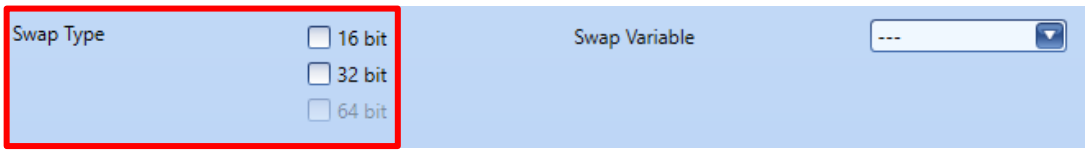
Set the *Starting address* and the *Length (words):*



Serial protocols

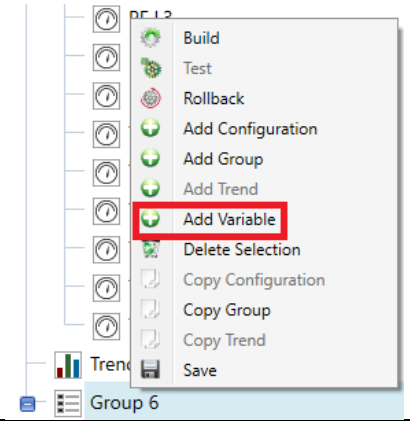
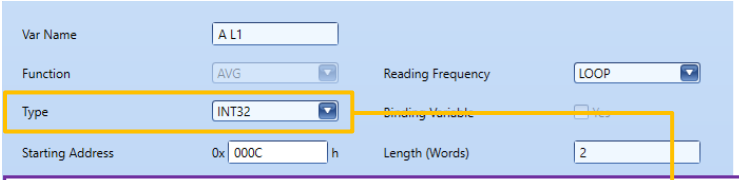
8

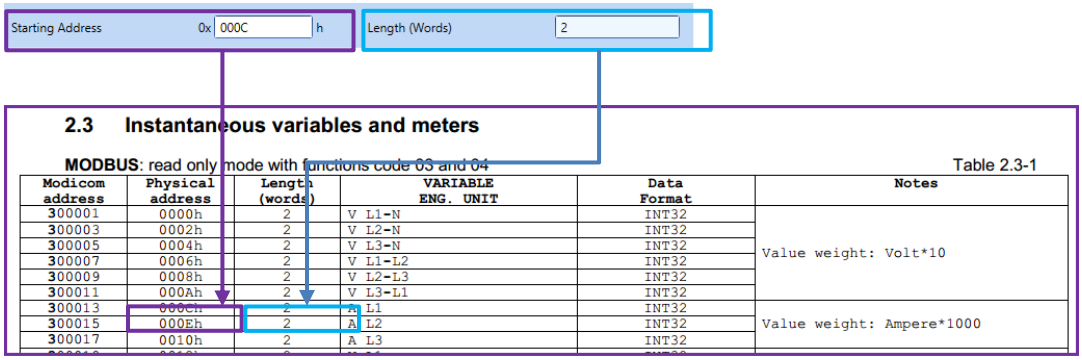
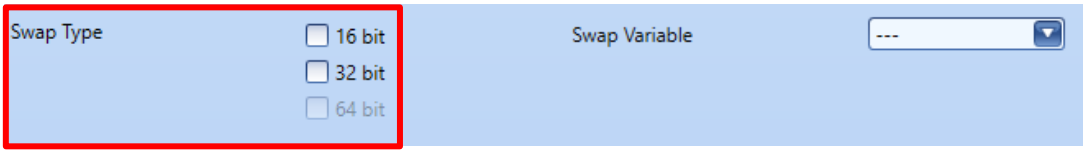
The *Swap type* can be chosen among the available choices and a *Swap variable* can be indicated to trigger the swap.



20 Adding an Info Variable

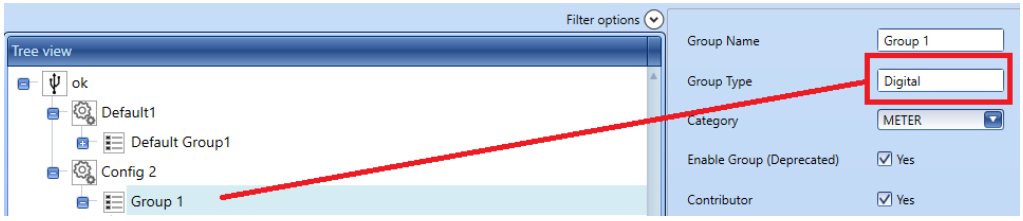
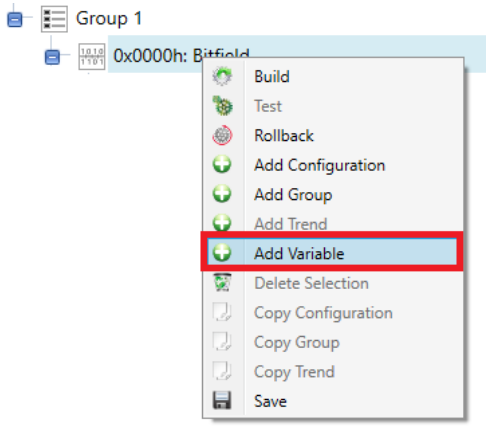
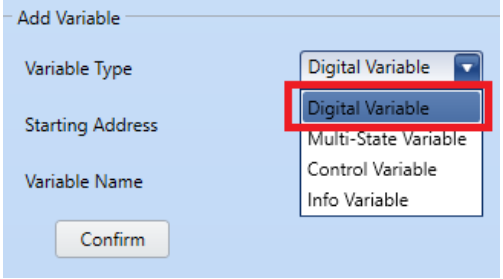
Follow this procedure to add an *Info variable*:

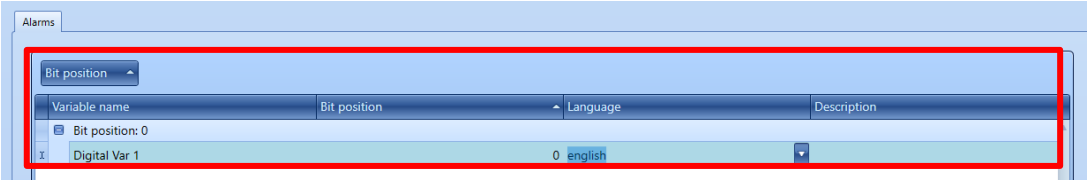
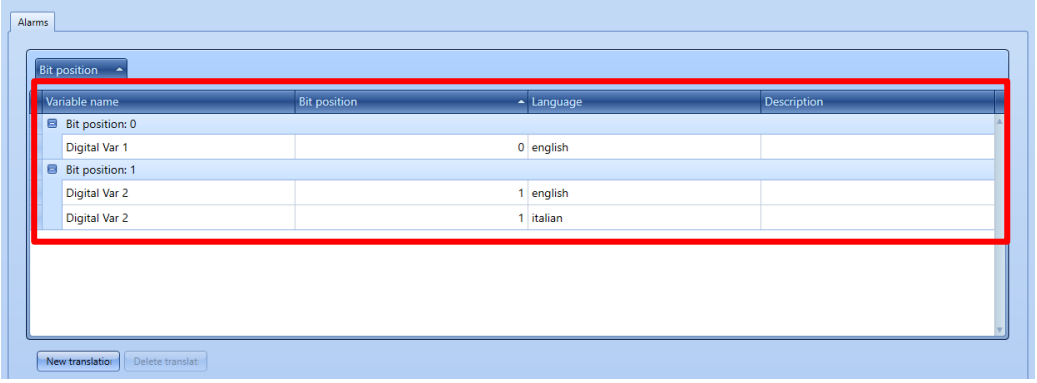
Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* where the *Variable* has to be added.
2	Right click to select *Add Variable.* 
3	From the *Variable Type* combo box on the right part of the window, select the *Info Variable.*
4	Click on *Confirm* to add the variable.
5	From the *Reading frequency* combo-box, select: **LOOP**: The register is read every cycle **ONCE**: The register is read once at the start-up of the Modbus device
~~6~~	Select the *Type* according to the protocol documentation of the target device  **2.3 Instantaneous variables and meters** **MODBUS**: read only mode with functions code 03 and 04 — Table 2.3-1 Modicom address / Physical address / Length (words) / VARIABLE ENG. UNIT / Data Format / Notes 300001 0000h 2 V L1-N INT32 300003 0002h 2 V L2-N INT32 300005 0004h 2 V L3-N INT32 300007 0006h 2 V L1-L2 INT32 300009 0008h 2 V L2-L3 INT32 300011 000Ah 2 V L3-L1 INT32 Notes: Value weight: Volt*10 300013 000Ch 2 A L1 INT32 300015 000Eh 2 A L2 INT32 300017 0010h 2 A L3 INT32 Notes: Value weight: Ampere*1000 **Serial protocols**

7	Set the *Starting address* and the *Length (words):*  **Serial protocols**
8	The *Swap type* can be chosen among the available choices and a *Swap variable* can be indicated to trigger the swap. 

21 Adding a Digital Variable

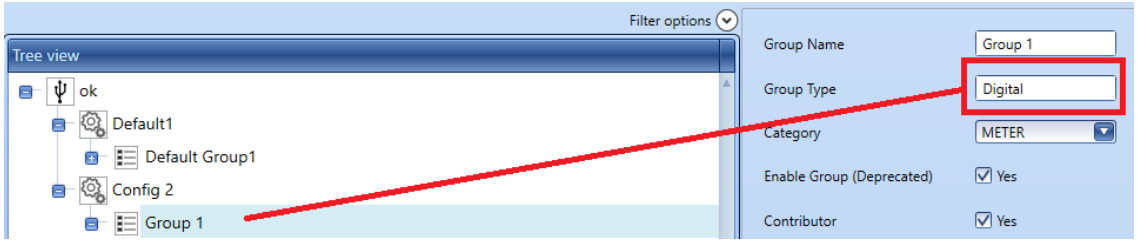
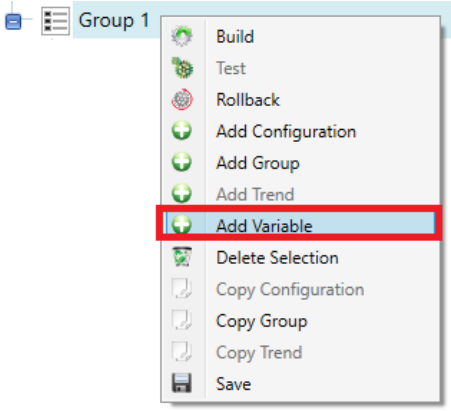
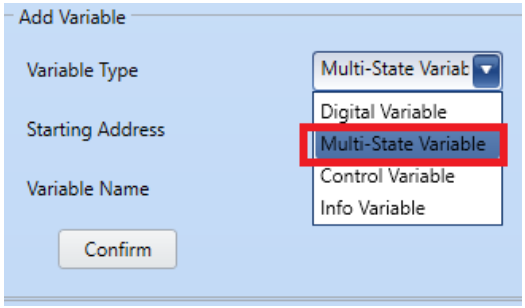
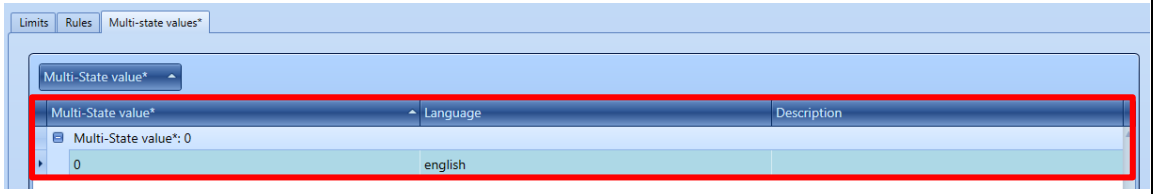
Follow this procedure to add a Digital variable

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* where the *Variable* has to be added 
2	Right click on the *Digital group* to select *Add Variable.* 
3	From the *Variable Type* combo-box on the right side, select the *Digital Variable.* 
4	Select the *Bit position* to describe (see the next step)

5	Describe the signal - associated to the specific Bit - that has to be represented.  *Note: The Modbus Read Function for this variable is the Read Coil Status (0x01).*
6	Clicking on the *Bitfield* field on the left side, it is possible to see a summary of every *bit position* description. 

22 Adding a Multi-State variable

Follow this procedure to add a Multi-state variable

Step	Action
1	In the **tree view** in the *Edit* tab, select the *Group* where the *Variable* has to be added 
2	Right click on the *Digital group* to select *Add Variable* 
3	From the *Variable Type* combo-box on the right side, select the *Multi-State Variable.* 
4	Describe the signal associated to the specific value – acquired by the variable.  *Notes:* • *The Modbus Read Function for this variable is the Holding Registers (0x03)* • *When a Multi-State value is changed, a new group of translations is created (starting by the default language).*

These operations can be done for Modbus driver.

18.3.9 Building a Modbus driver

An existing driver can be built by means of the BUILD DRIVER action available in the context menu. The BUILD action checks the driver mistakes and compiles the driver structure by taking into account the correct Modbus frame configuration.
If any error is found, the process exits without building a driver, and errors / warnings are reported to the user.

18.3.10 Copying a Modbus driver

For further information, go to ***Copying an existing configuration***.

18.3.11 Activating a Modbus driver

An existing driver can be built by means of the BUILD DRIVER action available in the context menu. The BUILD action checks the driver mistakes and compiles the driver structure by taking into account the correct Modbus frame configuration.
If any error is found, the process exits without building a driver, and errors / warnings are reported to the user.

18.3.12 Testing a Modbus driver

An existing driver can be built by means of the BUILD DRIVER action available in the context menu. The BUILD action checks the driver mistakes and compiles the driver structure by taking into account the correct Modbus frame configuration.
If any error is found, the process exits without building a driver, and errors / warnings are reported to the user.

18.3.13 Deleting a driver

An existing driver may be deleted by the relevant DELETE action in the context menu. Only those that have been created by the user can be deleted. The drivers created by Carlo Gavazzi as part of the firmware distribution cannot be deleted.

18.3.14 Importing a driver

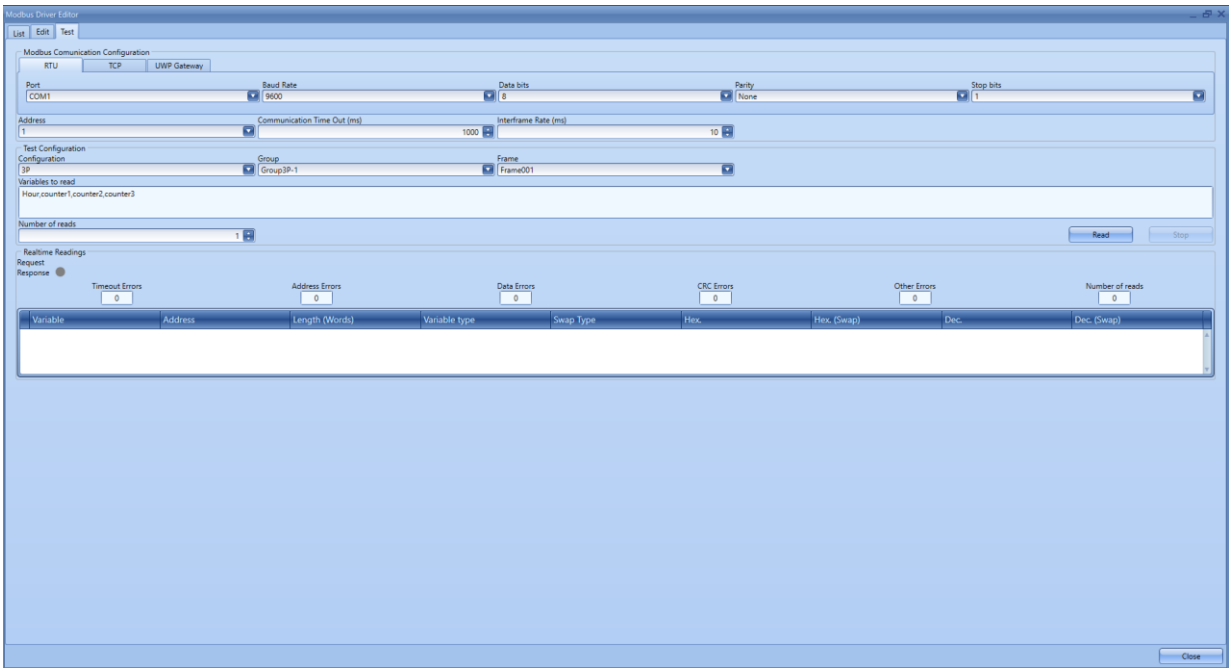
An existing driver may be imported clicking on IMPORT in the context menu.

18.3.15 Exporting a driver

Thanks to this function, it is possible to export the driver as an XML file. Once the file has been exported, it will be available for being imported in another UWP 3.0 system.


```xml
<?xml version="1.0" encoding="UTF-8"?>
<EOS_driver xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance" xsi:noNamespaceSchemaLocation="driver_schema.xsd">
  <general>
    <driver_id>729862703746</driver_id>
    <name>EM210</name>
    <brand>CUSTOM</brand>
    <driver_version>1.7</driver_version>
    <protocol>0</protocol>
    <instrum>EM210</instrum>
    <instrum_code>0</instrum_code>
    <data_max_size>10</data_max_size>
    <extraoption1></extraoption1>
    <extraoption2></extraoption2>
    <notes></notes>
    <imgsmall></imgsmall>
    <imgbig></imgbig>
    <autoscan>0</autoscan>
    <code_address>0</code_address>
    <errskipthres>3</errskipthres>
    <errskipcount>5</errskipcount>
    <driver_date>1435334809</driver_date>
    <idtypelabel>0</idtypelabel>
    <addresstype>2</addresstype>
    <slvaddrtype>0</slvaddrtype>
  </general>
  <configurations>
    <configsnum>1</configsnum>
    <configuration>
      <idDrvConfig>1</idDrvConfig>
      <name>3P</name>
      <groupsnum>1</groupsnum>
      <group>
        <idDrvGroup>1</idDrvGroup>
        <category>2</category>
```

18.3.1 How to test a driver

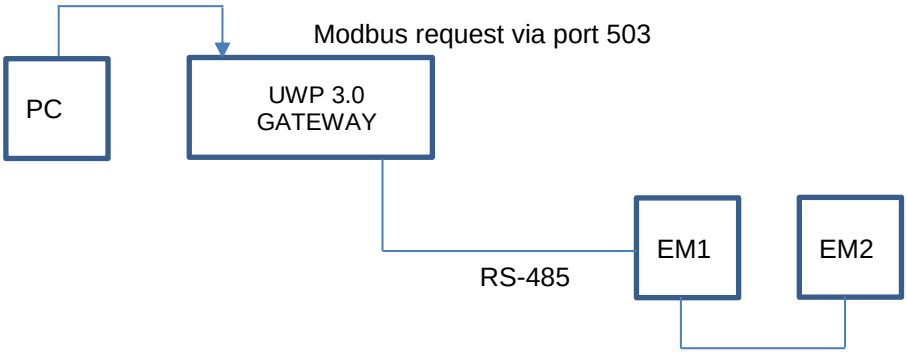


An existing driver can be tested without activating it, using the relevant testing function. The testing function allows to test interactively a Modbus Driver by reading on demand the desired Modbus frames with the selected variables. The result of single or multiple read actions is available as summary statistics (highlighting communication errors) and contents of the last read frame.
The variable values are available both in decimal and hexadecimal format, in standard and swapped format.
Please note that also the bitfield variables are displayed as decimal/hexadecimal and the relevant values must be translated into binary bitfields by the user.

The driver testing tool allows to execute tests in 3 use cases:

- RTU= in the case of a Modbus device connected to pc via Modbus converter to the PC where this software is installed
- TCP/IP= in the case of a Modbus device connected to the same Ethernet network of the PC where this software is installed

- UWP GATEWAY= (bridge) in the case of a Modbus device connected to one of the RS485 ports of the UWP 3.0 unit (see the diagram below)



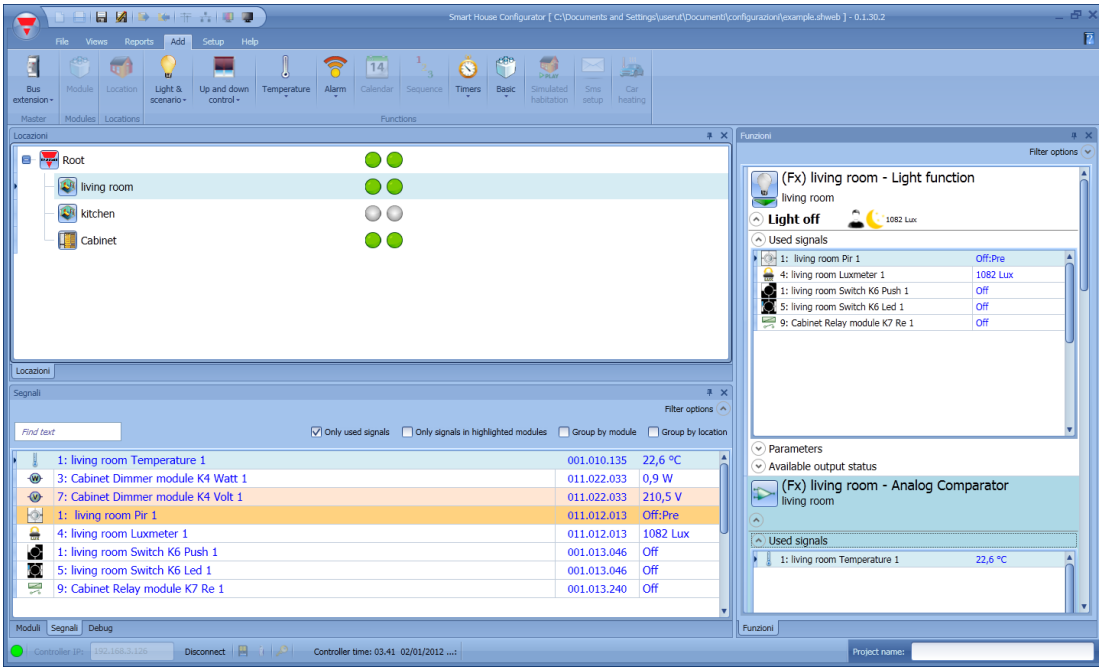
19 Live signals

19.1 How live signals work

The smart house system provides information about its working status and makes it available by using a feature called *Live signals* in the UWP 3.0 Tool.
The main functionalities of the live signals are described below.

19.1.1 Checking the status of all the analogue and digital values

The user can check at any time the status of all analogue values coming from all the sensors in the installation, such as temperature, light level, wind speed, voltage, current and so on.
The status of all the digital signals is presented to the user, so he can check if a PIR sensor is detecting movement or presence, if the output relay of the heating system is working properly or if a generic input contact is on or off.

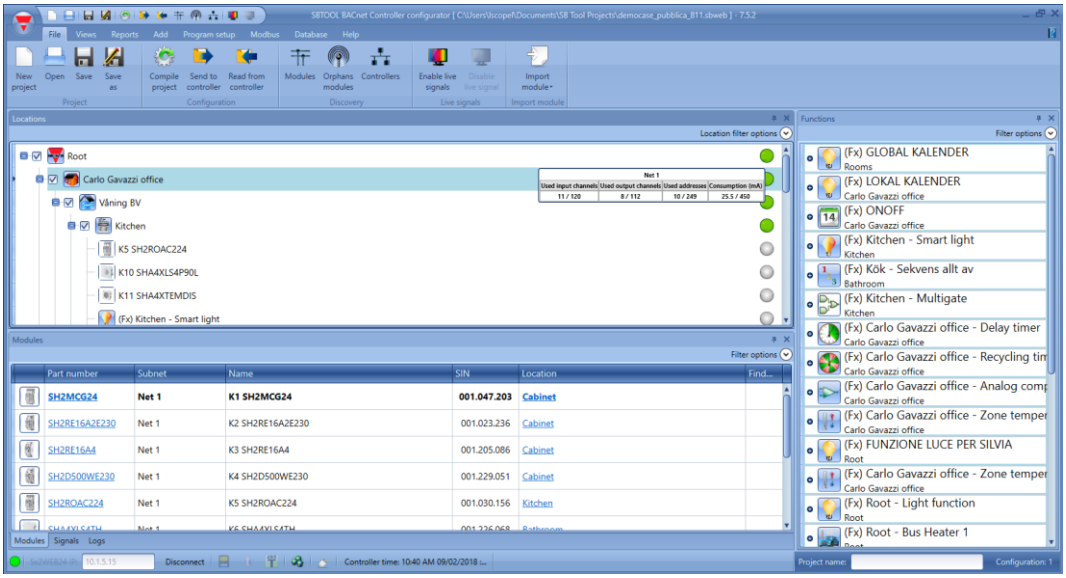


19.1.2 Tuning the parameters of the functions

During commissioning the installer can change the parameters of the functions, for example the lux level value for the activation/deactivation of a light function, or the settings for the PIR actions in a dimmable light when movement is detected, also many other configuration parameters.

19.1.3 Checking the Bus diagnostic

During commissioning, if any trouble should occur, the system provides a powerful diagnostic function in order to make the fault-finding much easier: the bus is always monitored, giving information about short circuits, bus voltage and bus noise level, as shown in the picture below.



19.1.4 How the quality index works

The quality index of the bus signals is shown in the UWP 3.0 tool for each Smart Dupline and WiDup modules, when the live signals are enabled. The quality index is represented by a percentage number between 0 (Low quality) and 100 (High quality) and it shows the communication quality between a module and the relevant master generator, such as SH2MCG24 or SH2WBU230x.

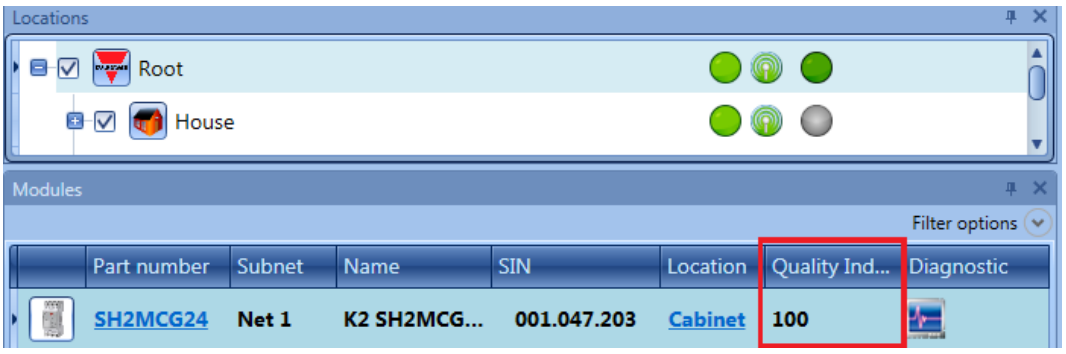
19.1.4.1 Checking the quality index for a Smart Dupline wired network

The quality index for a Smart Dupline network is represented by the ratio between signals without noise and signals with noise. Usually the noise is generated by the activation or deactivation of inductive loads and contactors. In such situations we recommend using shielded cables and keep the wire away from contactors, relays, motors and other inductive noise sources. (Please refer to the Hardware manual for instructions).

As concerning the quality index shown in the *Live signals* field for each module, the higher the number the better the communication quality to the master. When one or more Smart Dupline modules are affected by noise, the quality index value is immediately updated by the tool.
The value slowly goes back to 100 only when no channels are noisy. A quality index below 100 occurs when a Smart Dupline network is affected by critical conditions and it anticipates the evidence of network issues. Thanks to this, the installer will be informed in order to make the fault-finding much easier.

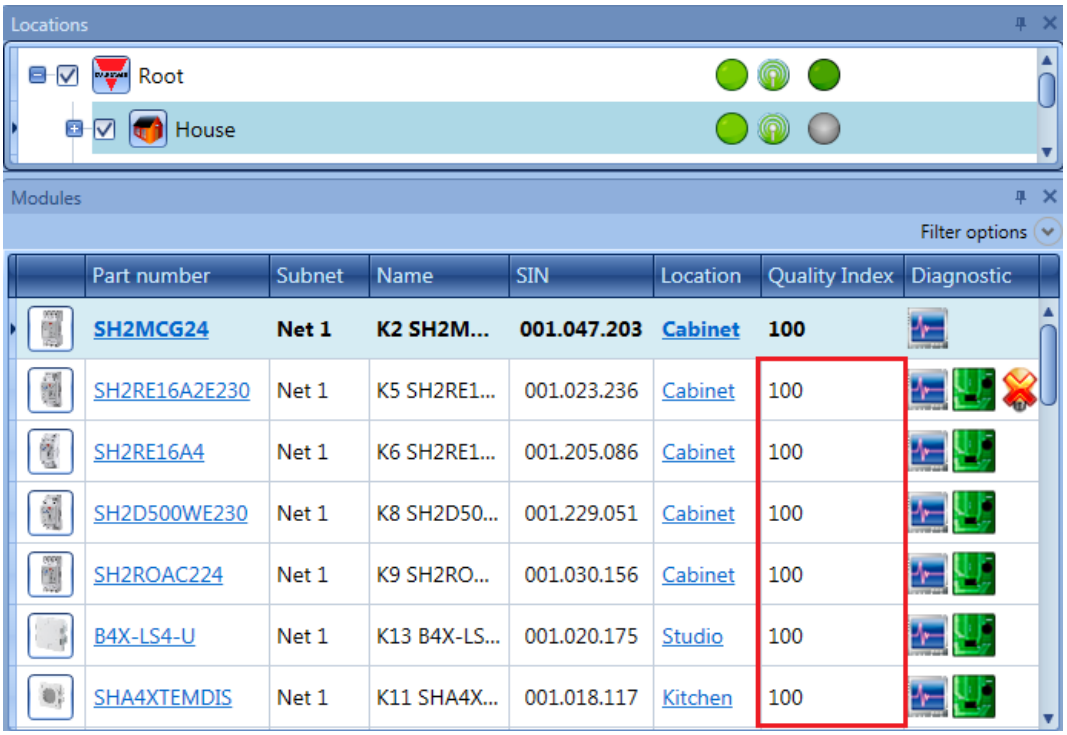
19.1.4.2 How to check the quality index of a Smart Dupline master generator

The quality index level shown for the SH2MCG24 master generator represents the “healthy” status of all the input channels managed. This parameter is calculated by the UWP 3.0 controller and it is shown in the UWP 3.0 tool with a percentage number between 0 and 100. The picture below shows the quality index for a SH2MCG24 module.



19.1.4.3 How to check the quality index of a Smart Dupline module

The quality index level shown for a Smart Dupline module represents the “healthy” status of its Smart Dupline output channels. This parameter is calculated by the UWP 3.0 controller and it is shown in the UWP 3.0 tool with a percentage number between 0 and 100. The picture below shows the quality index of the Smart Dupline modules that are part of the *Net1* network.



19.1.4.4 How to check the quality index of a wireless module

The quality index for a wireless module is calculated considering the distance between the SH2WBU230x and a wireless module and the power ratio for the radio communication from the receiver to the transmitter.

For the wireless modules the quality index is represented by two different values: the transmission quality index, as shown in the picture below, in the TX field and the receiver quality index, as shown in the picture below, in the RX field.

	Onderdeel n...	Subnet	Naam	SIN	Plaats	Diagnostic	Quality Index
	SHJWEM16A23	draadloos 2	Kwh WCD	020.132.173	App. 206		TX:77 RX:100
	SHJWEM16A23	draadloos 2	KWh badkamer	020.132.178	App. 206		TX:29 RX:74
	SHJWEM16A23	draadloos 2	KWh keuken	020.133.014	App. 206		TX:61 RX:77
	SHJWEM16A23	draadloos 2	KWh koof	020.130.174	App. 206		TX:96 RX:77
	SHJWEM16A23	draadloos 2	KWh woonkamer	020.132.185	App. 207		TX:100 RX:100
	SHJWEM16A23	draadloos 2	KWh WCD	020.133.012	App. 207		TX:100 RX:100
	SHJWEM16A23	draadloos 2	KWh badkamer	020.132.253	App. 207		TX:90 RX:100

19.1.4.4.1 Rx value for a wireless module

The RX quality index is calculated directly by each wireless module, analysing the received radio frames sent from the SH2WBU230x: if all the frames sent are received correctly, the RX value is 100 (high quality). A value below 100, indicates that some frames are not received by the wireless module. Anyway the protocol is designed to repeat not received commands.
A SHJWD200WExxxx, a SHJWEM16Axxx or a SHJWRE10AExxx should be used as a router/repeater when the RX value is below 50.

19.1.4.4.2 TX value for a wireless module

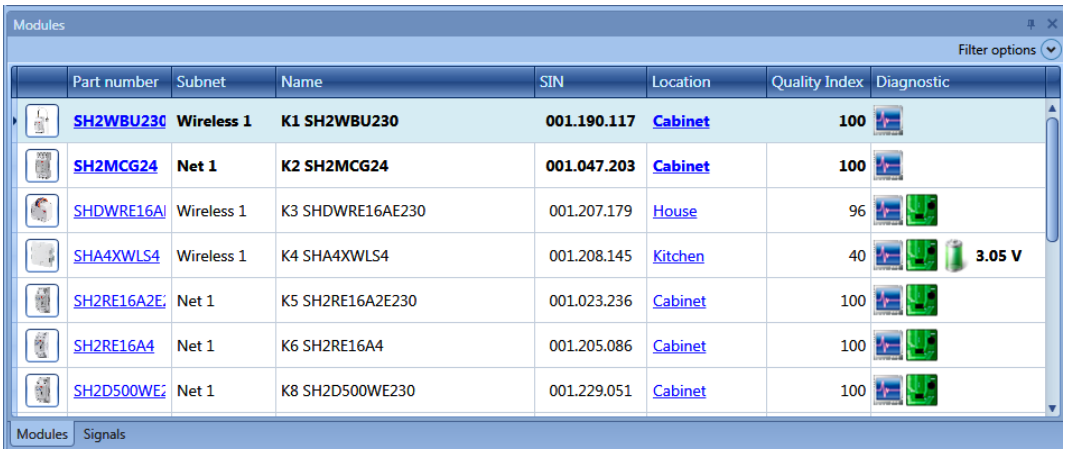
The TX quality index of a wireless module is calculated by the SH2WBU230x, analysing the radio frames received by the wireless module. If all the frames sent are received correctly, the TX value is 100 (high quality). A value below 100, indicates that some frames are not received by the wireless base unit. If a wireless module is connected to the SH2WBU230x via a router, the TX value is referred to the quality index of the wireless module used as a router in the configuration.

When the *RX/TX* index is below 50, we suggest extending the operating distance using a SHJWD200WExxxx, a SHJWEM16Axxx or a SHJWRE10AExxx as router/repeater.

In the example shown below, a router/repeater should be used for the wireless module marked in red to extend the operating distance.

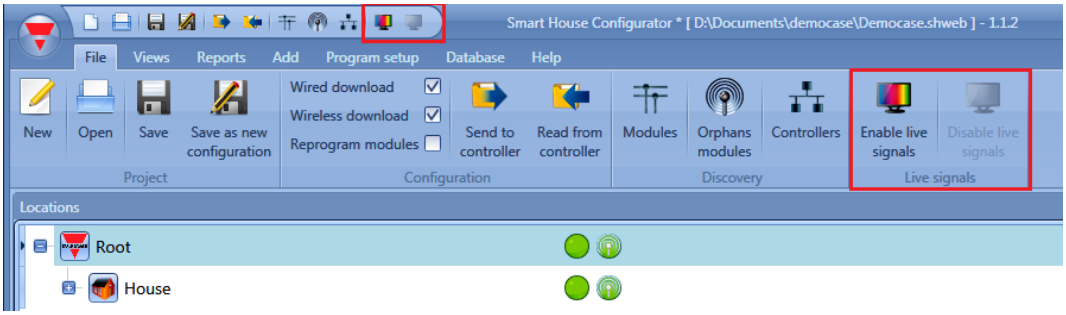
19.1.5 Checking the status of each module:

The presence of the programmed devices is always monitored in order to give an immediate message if any fault should occur. At all times each module is monitored to check if it is alive, working well and without any alarm/warning message such as overcurrent or over-temperature. The system can also advise the user when a lamp, or in general a load, is broken or not working anymore, reminding the user to change or repair it.

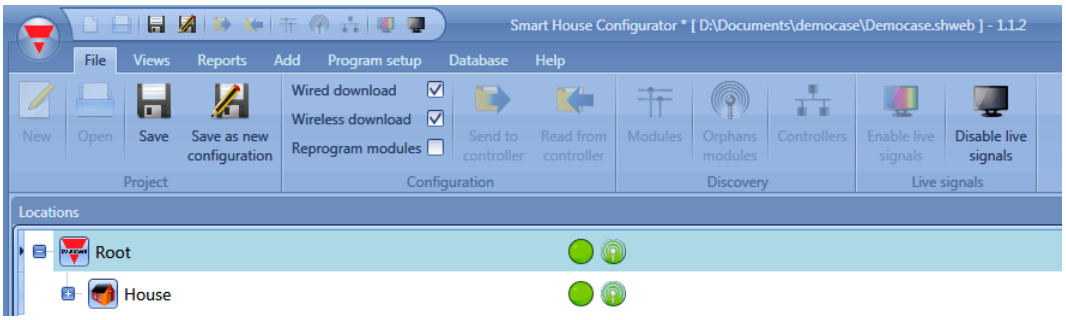


19.2 How to enable/disable *live signals*

Live signals can be enabled/disabled at any time by selecting the relevant icon filed in the main menu or by using the icon at the top of the tool. Live signals is available only when the tool is connected to the UWP 3.0.



When *Live signals* is activated, all the other actions are disabled (only saving the project is allowed). The user must disable *Live signals* to start again, using all the functions.

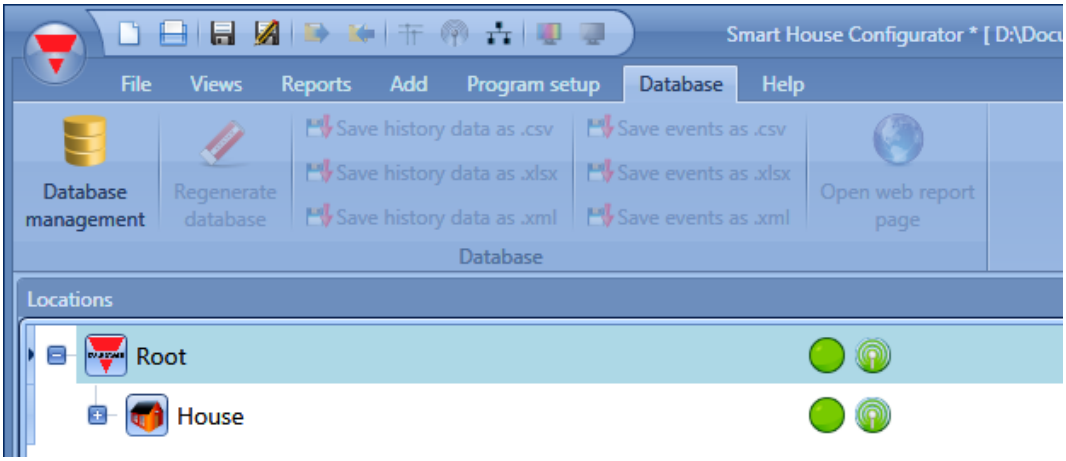


19.2.1 How to allow *Live signal*s to show an analogue value

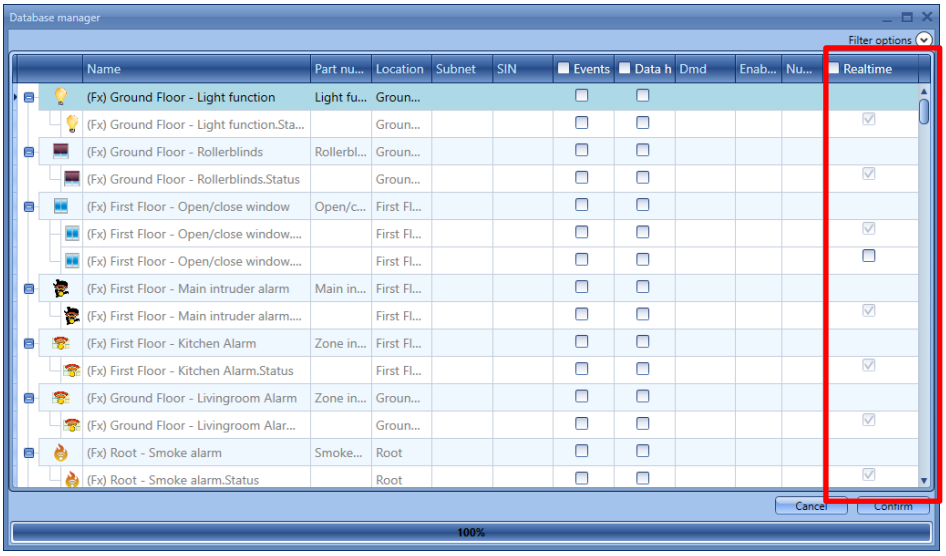
The UWP 3.0 Tool automatically shows in *Live signals* all the values that are used in the functions of the project. If a signal is not used in a function, it must be added in *Database management* to see it in *Live signals.*

The steps to add a variable to the database are:

1) Click on *Database* and select *Database management*


Smart House Configurator * [D:\Docu
File
Views
Reports
Add
Program setup
Database
Help
Database management
Regenerate database
Save history data as .csv
Save history data as .xlsx
Save history data as .xml
Save events as .csv
Save events as .xlsx
Save events as .xml
Open web report page
Database
Locations
Root
House

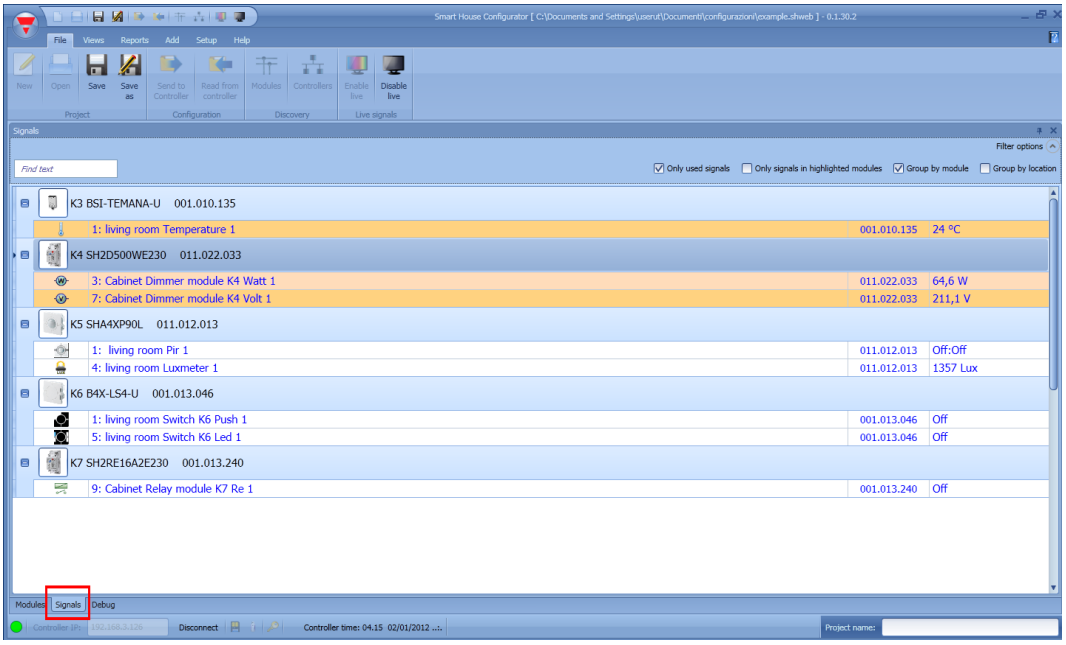
2) In the *Realtime* column, the user has to enable the checkbox next to each signal, such as analogue and digital signals and function statues that have to be added., as shown in the red rectangle in the picture below:



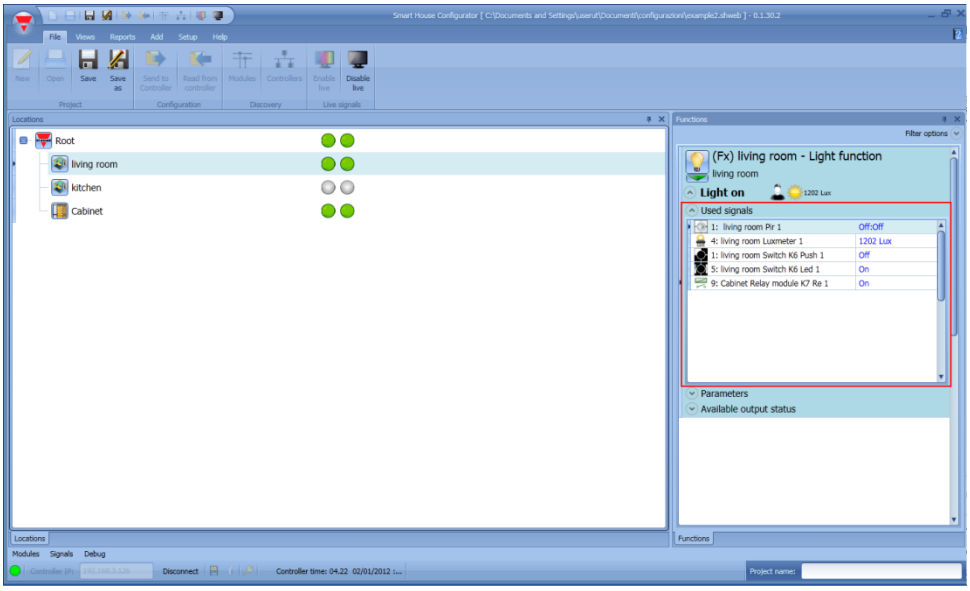
3) *Confirm* and write the configuration to the UWP 3.0

19.3 How to see values and status of the functions

There are two ways to check the status of a signal: the first is to select the *Signals* window at the bottom of the UWP 3.0 tool and find the required signal in the list. The user can apply the filter to search for the signal by name, by module type or according to the location where it is placed.
Each time a value changes, the UWP 3.0 tool shows the change, marking the relevant row in light orange in order to easily recognize the update.

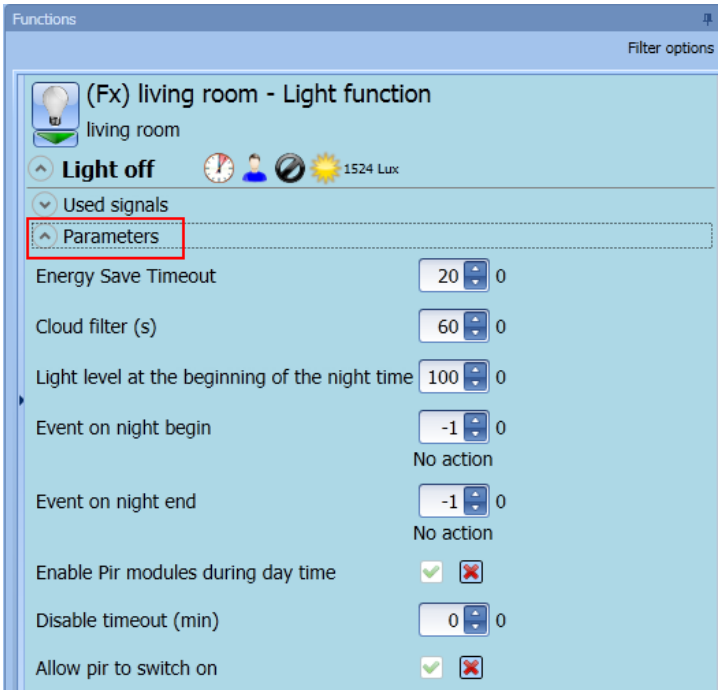


The second way is to click on the function where the signal is used and select *Used signals* in order to have a global view of all the signals used in the function.



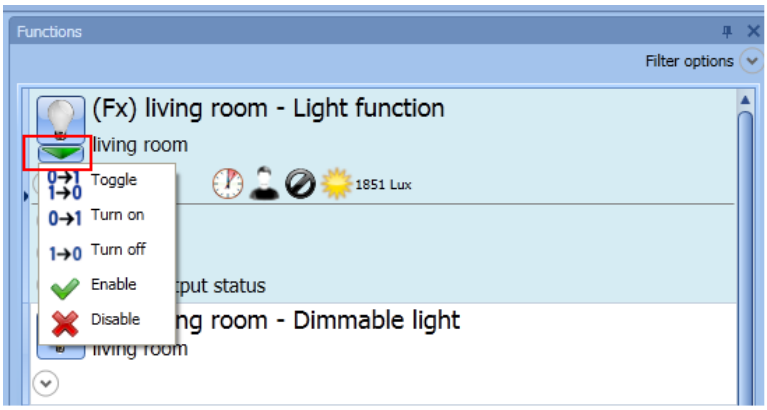
19.4 How to tune the parameters of a function

The user can change the parameters of each function by using the relevant file *Parameters* below the function icon: all the modified values are automatically changed, also in the configuration file.



19.5 How to change the status of a function

The user can access a list of commands in order to perform different actions when *Live signals* is active. The menu of available actions can be shown by clicking on the green arrow under each function icon.



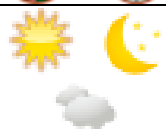
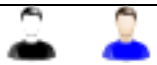
When the menu is open, the user can select the action to be performed.

19.5.1 On/off light

The available actions with *Live signals* are listed in the table below:

Icons	Action
	By clicking on the icon next to the name of the function, the light is turned on/off in toggle mode.
0→1 1→0	Turn the light on/off in toggle mode
0→1	Turn the light on
1→0	Turn the light off
	Enable automations
	Disable automations

In the table below, the list of available icons for the light function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the light is off, the yellow icon indicates that the light is on.
Energy save timer		The icons change according to the value of the energy-save timer.
Luxmeter		The sun icon indicates that the daytime condition is active, the moon icon indicates that the night-time condition is active, the cloud indicates that the cloud filter is in progress. Next to the icon the value of the measured light level is also shown.
Pir		The blue icon is active when the PIR detects presence, otherwise the black icon is shown.
Disable automation		The black icon is shown when the automation is enabled, the red icon is shown when the automation is disabled.
Force signal on		The padlock is closed when the light is forced on, otherwise it is open.
Force signal off		The padlock is closed when the light is forced off, otherwise it is open.

19.5.2 Dimmable light

The actions that the user can carry out using the live signal are explained in the table below:

Icons	Action
	By clicking on the icon next to the name of the function, the light is turned on/off in toggle mode.
S1	Switch the light on at scenario 1 (S1)
1→0	Turn the light off
S2	Switch the light on at scenario 2 (S2)
S3	Switch the light on at scenario 3 (S3)
S4	Switch the light on at scenario 4 (S4)
S5	Switch the light on at scenario 5 (S5)
%	Set the light percentage using the slider
A	Start sequence A
B	Start sequence B
	Enable automations
	Disable automations

In the table below the list of available icons for the dimmable light function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the light is off, the yellow icon indicates that the light is on.
Energy save timer		The icons change according to the value of the energy-save timer.
Luxmeter		The sun icon indicates that the daytime condition is active, the moon icon indicates that the night-time condition is active, the cloud indicates that the cloud filter is in progress. Next to the icon the value of the measured light level is also shown.
Pir		The blue icon is active when the PIR detects presence, otherwise the black icon is shown.
Disable automation		The black icon is shown when the automation is enabled, the red icon is shown when the automation is disabled.
Force signal on	ON ON	The padlock is closed when the light is forced on, otherwise it is open.
Force signal off	OFF OFF	The padlock is closed when the light is forced off, otherwise it is open.

19.5.3 Zone intruder alarm

The actions that the user can carry out using *Live signals* are shown in the table below:

Icons	Action
0→1	Remove override reset
1→0	override reset

In the table below, the list of available icons for the zone alarm function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates when the zone alarm is disarmed, the coloured icon indicates that the zone is armed.

19.5.4 Main intruder alarm

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	Arm the intruder alarm
1→0	Disarm the intruder alarm
✔	Enable reset
✘	Disable reset

In the table below, the list of available icons for the main alarm function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates when the alarm is disarmed, the coloured icon indicates when the alarm is armed.

19.5.5 Smoke alarm

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1 1→0	Activate/deactivate the reset condition
	Enable reset
	Disable reset

In the table below, the list of available icons for the smoke alarm function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that no alarm is detected, the coloured icon indicates that an alarm is detected.
Disable automation		The black icon is shown when the reset condition is not activated, the red icon is shown when the reset condition is activated.
Force signal on	ON ON	The padlock is closed when the smoke alarm is forced on, otherwise it is open.
Force signal off	OFF OFF	The padlock is closed when the smoke alarm is forced off, otherwise it is open.

19.5.6 Water alarm

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1 1→0	Activate/deactivate the reset condition
	Enable reset
	Disable reset

In the table below, the list of available icons for the water alarm function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that no alarm is detected, the coloured icon indicates that an alarm is detected.
Disable automation		The black icon is shown when the reset condition is deactivated, the red icon is shown when the reset condition is activated.
Force signal on	ON ON	The padlock is closed when the water alarm is forced on, otherwise it is open.
Force signal off	OFF OFF	The padlock is closed when the water alarm is forced off (the function stays off even if water is detected), otherwise it is open.

19.5.7 Siren function

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	Reset the siren

In the table below, the list of available icons for the siren function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the siren is off, the blue icon indicates that the siren is on.

19.5.8 Sequence function

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	Start sequence
1→0	Stop sequence
✔	Enable automations
✖	Disable automations

In the table below, the list of available icons for the sequence function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the sequence is not running, the coloured icon indicates that the sequence is running.

19.5.9 Rollerblind function

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	Fully up
1→0	Fully down
	Stop movement
%	Set the movement/tilting position
	Enable automations
	Disable automations

In the table below, the list of available icons for the roller blind function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the blind is halted, the coloured icon indicates that the blind is moving.
Luxmeter		The sun icon indicates that the blinds are managed according to the light level, the cloud indicates that the cloud filter is in operation. Next to the icon the value of the measured light level is also shown.
Wind sensor		According to the wind speed the icon changes, indicating weak, average and strong wind.
Disable automation		The black icon is shown when the automation is enabled, the red icon is shown when the automation is disabled
Force signal on	ON ON	The padlock is closed when the blind is forced to the safe position, otherwise it is opened.
Force signal off	OFF OFF	The padlock is closed when the blind is forced to the non-safe position, otherwise it is opened.

19.5.10 Window function

The actions that the user can carry out using the live signals are explained in the table below:

Icons	Action
0→1	Fully open
1→0	Fully closed
	Stop movement
%	Set the movement/tilting position value
	Enable automations
	Disable automations

In the table below, the list of available icons for the window function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the window is halted, the coloured icon indicates that the window is moving.
Wind sensor		The sun icon indicates that the window is managed according to the light level, the cloud indicates that the cloud filter is in operation. Next to the icon the value of the measured light level is also shown.
Disable automation		According to the wind speed the icon changes, indicating weak, average and strong wind.
Force signal on	ON ON	The black icon is shown when the automation is enabled, the red icon is shown when the automation is disabled
Force signal off	OFF OFF	The padlock is closed when the blind is forced to the safe position, otherwise it is opened.

19.5.11 Zone Temperature

The actions that the user can carry out using the live signal are explained in the table below:

Icons	Action
	Switch heating off Switch cooling off
	Select heating set point1 Select cooling set point1
	Select heating set point2 Select cooling set point2
	Select heating set point3 Select cooling set point3
	Enable automations
	Disable automations

In the table below, the list of available icons for the zone temperature function is shown:

Object	Icons	Icon description
Fx status		The grey icon is shown when the timer is off, the coloured icon is shown when the timer is on.

19.5.12 Heating system temperature

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1 1→0	Turn the heating system on/off in toggle mode
0→1	Turn the heating system on
1→0	Turn the heating system off
	Enable automations
	Disable automations

19.5.13 Cooling system temperature

The actions that the user can carry out using *Live signals* are explained in the table below:

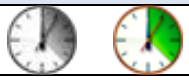
Icons	Action
0→1 1→0	Turn the cooling system on/off in toggle mode
0→1	Turn the cooling system on
1→0	Turn the cooling system off
✔	Enable automations
✖	Disable automations

19.5.14 Delay timer

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	The trigger signal is simulated as active (pushbutton kept pressed)
1→0	The trigger signal is simulated as non-active (pushbutton released)

In the table below, the list of available icons for the delay timer function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the timer is not running, the coloured icon indicates that the timer is on.

19.5.15 Recycling timer

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	Start the timer
1→0	Stop the timer

In the table below, the list of available icons for the recycling timer function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the timer is not running, the coloured icon indicates that the timer is on.

19.5.16 Interval timer

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	Start the interval timer
1→0	Stop the interval timer

In the table below, the list of available icons for the interval timer function is shown:

Object	Icons	Icon description
Fx status		The grey icon indicates that the timer is not running, the coloured icon indicates that the timer is on.

19.5.17 Simulated habitation

The actions that the user can carry out using *Live signals* are explained in the table below:

Icons	Action
0→1	Start the simulated habitation
1→0	Stop the simulated habitation

In the table below, the list of available icons for the simulated habitation function is shown:

Object	Icons	Icon description
Fx status		The grey icon is shown when the simulated habitation is not active, the coloured icon is shown when the simulated habitation is active.

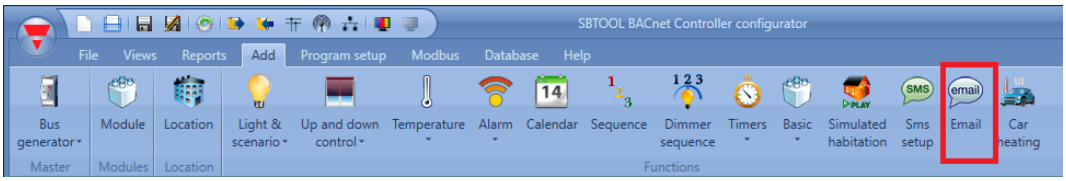
19.5.18 Analogue comparator

In the table below, the list of available icons for the analogue comparator function is shown:

Object	Icons	Icon description
Fx status		The grey icon is shown when the comparator is not active, the coloured icon is shown when the comparator is active.

20 How to set the Email function

The *Mail* function can be used to advise that a function has changed its status. To add the *Email* function the user should select *Email* from the *Add* menu (see picture below). The new function will be added into the selected location.

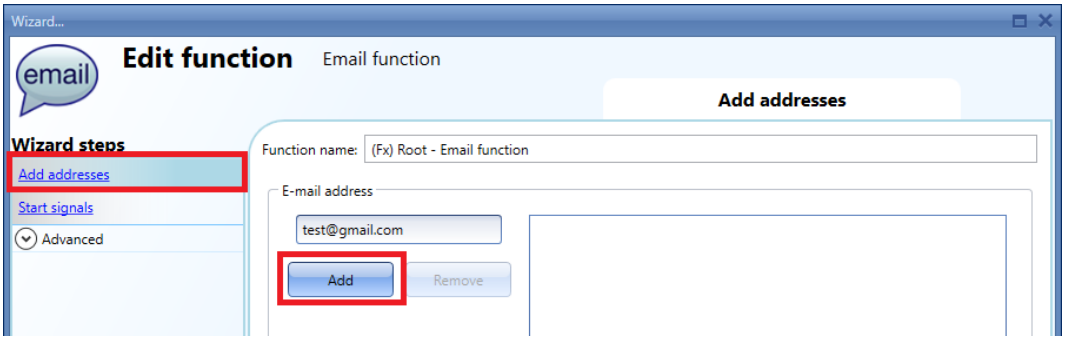


20.1 How to configure the email account

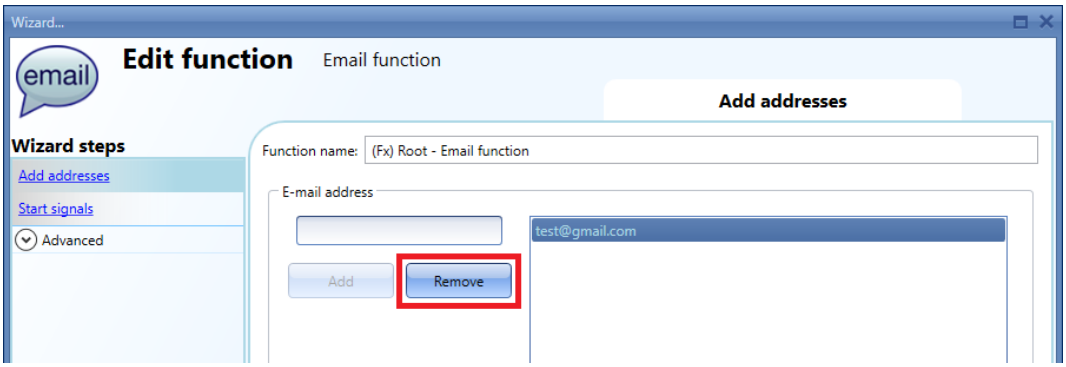
The user has to configure the email settings in the *Default system settings* menu, (see also the paragraph *General settings* for more info).

20.2 How to manage the accounts in the contacts list

The *Email* function can send emails to different accounts according to different purposes: for example, one address could be configured to receive the statuses of some functions. To add a new receiver address, the user has to click on *Add addresses*, type the address and then click on *Add*: all the added addresses will be displayed in the mail configuration list (see picture below).

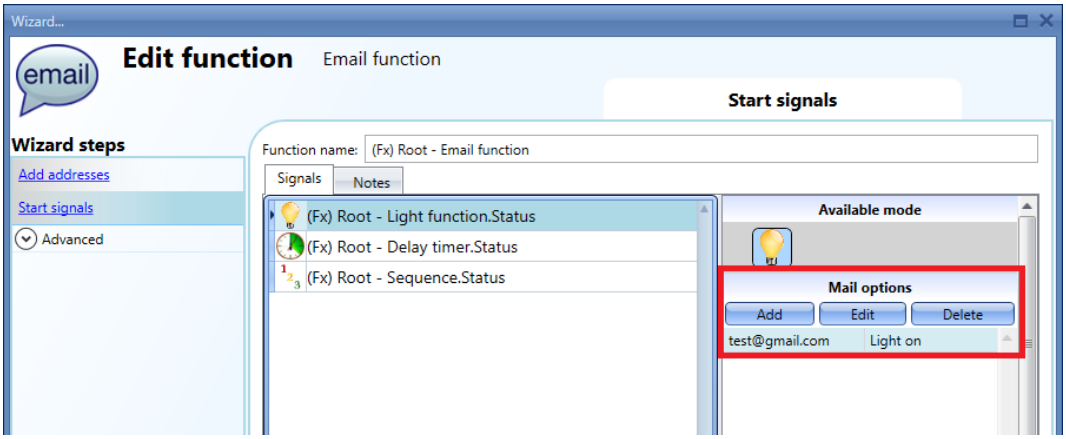


To delete an address, select it in the addresses list and then click on the *Remove* button as shown below.



20.3 How to configure an outgoing mail

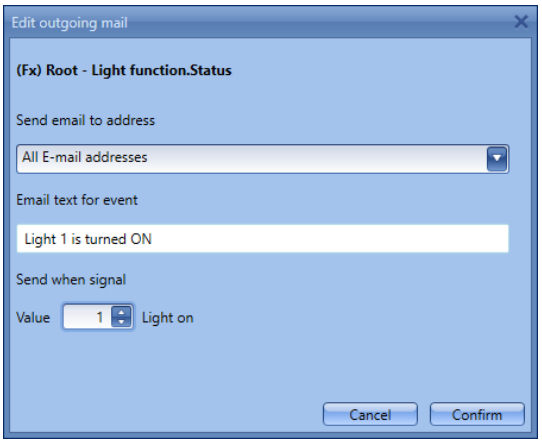
In the *Start signals* field, the user has to add all the functions that result in the emails being sent. Once a function is added, the user can access *Mail options* by clicking on it.



The available options are:

- Add a new action according to the status of the function
- Edit an already added action
- Delete an action

When the *Add* icon is selected, the following setup-window is shown:



Fill in the boxes with the necessary information:

1: Send email to address: This contains the address the email is sent to. The list of addresses added in the Email configuration will be prompted. If all of them are selected, the email will be sent to all.

2: Email text for event: in this field the user can type a text which informs the receiver about what is happening.

3: Send when signal: in this field the user selects the event that will cause the email to be sent. In the example on the left, an email with the text "Light 1 is turned ON" will be sent when the status of that light changes from off to on.

Note: In this field the user selects the event that will cause the email to be sent. In the example on the left, an email with the text "Light 1 is turned ON" will be sent when the status of that light changes from off to on. Please consider the following exceptions:

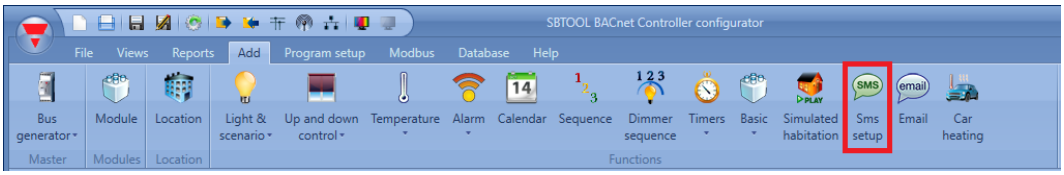
1- *Sequence* functions: a delay higher than 1 seconds must be set between one step and the next one in order the email will be sent;

2- *Dimmer* functions: the email will not be sent when the *Dimmer* function status changes from OFF to S1 scenario;

21 How to set the GSM function

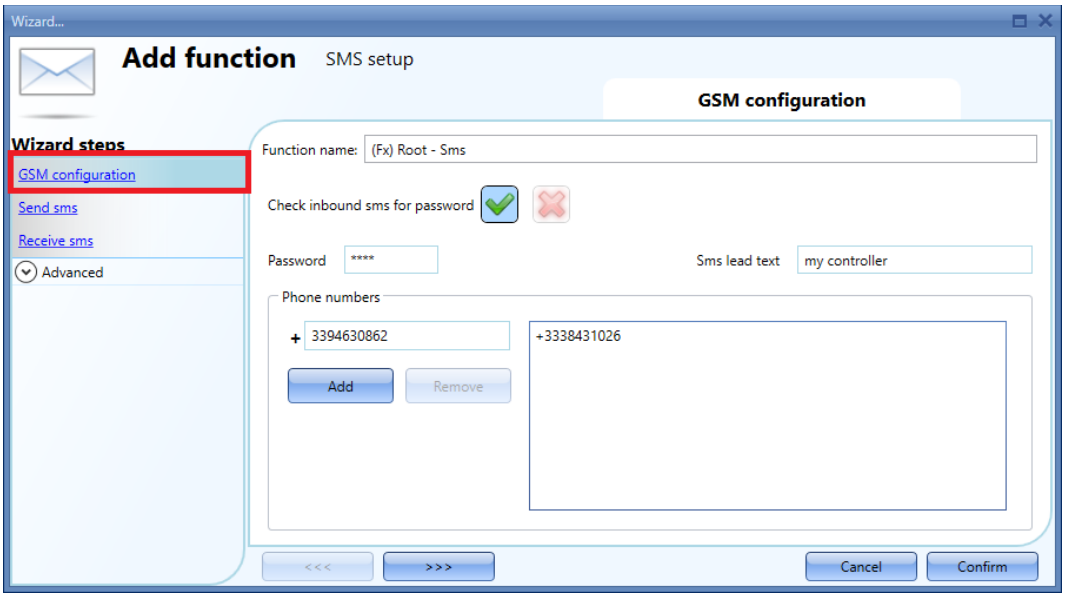
To use the GSM function, the SH2UMMF124 or the SH2DSP24 module has to be connected to the UWP 3.0.
This has to be configured as described in *General settings*.
The SMS function can be used to check the status of a function or an analogue variable, to activate a function or to be advised about the change of the status of a function.

To add the *SMS function,* the user has to select *SMS setup* from the *Add* menu (see picture below). The new function will be added into the selected location.



21.1 GSM configuration

In the first step of the function, the user has to set up the GSM configuration.



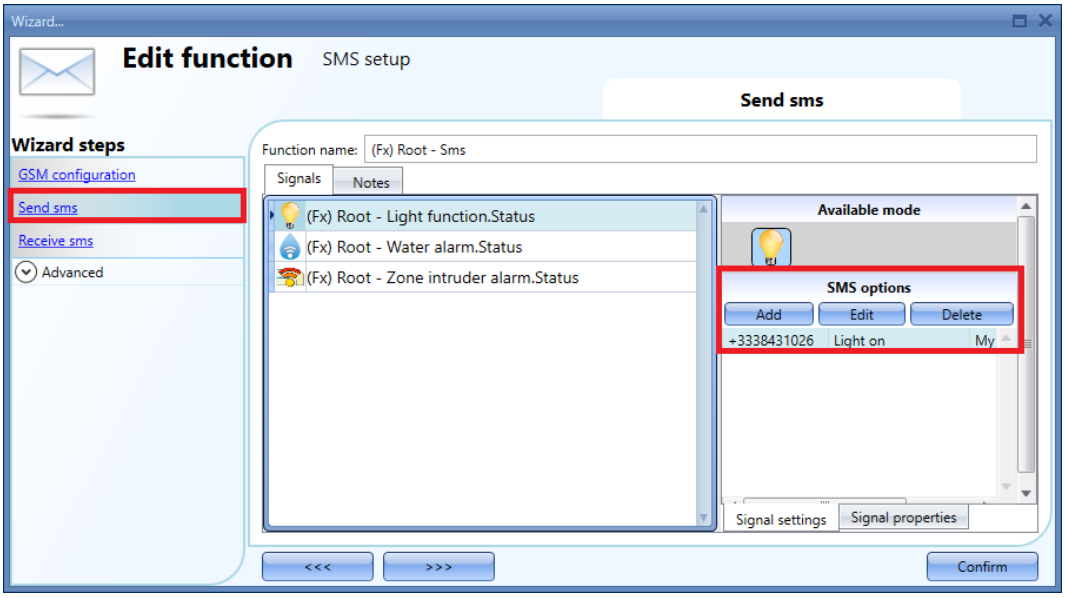
Check inbound sms for password
When enabled (green V selected), the sms that is received by the system must contain the password written in the *Password* box. The password can contain numbers and letters, which are not case sensitive.

SMS lead text
Here the user can type the header of the message sent by the smart-house system and displayed on the phone.
Example: according to the picture above, each SMS sent by the UWP 3.0 will start with “my controller”.

Phone numbers
All the telephone numbers which the SMS have to be sent to and received by, must be added in this box. Each added number must start with the country code of the receiver (e.g. +39 for Italy).

21.2 How to set up the smart-house system to send SMS

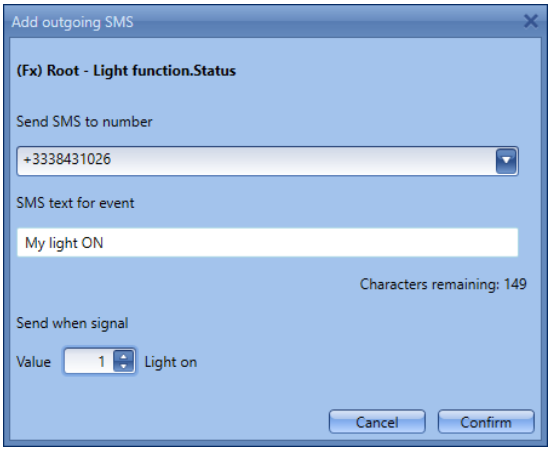
In the *Send SMS* field, the user has to add all the functions that result in the SMS being sent. Once a function is added, the user can access the *SMS options* by clicking on it (see below).



When the function is added, the possible options are:

- Add a new action according to the status of the function
- Edit an already added action
- Delete an action

When the Add icon is selected, the following setup-window is shown:



Fill in the boxes with the necessary information:

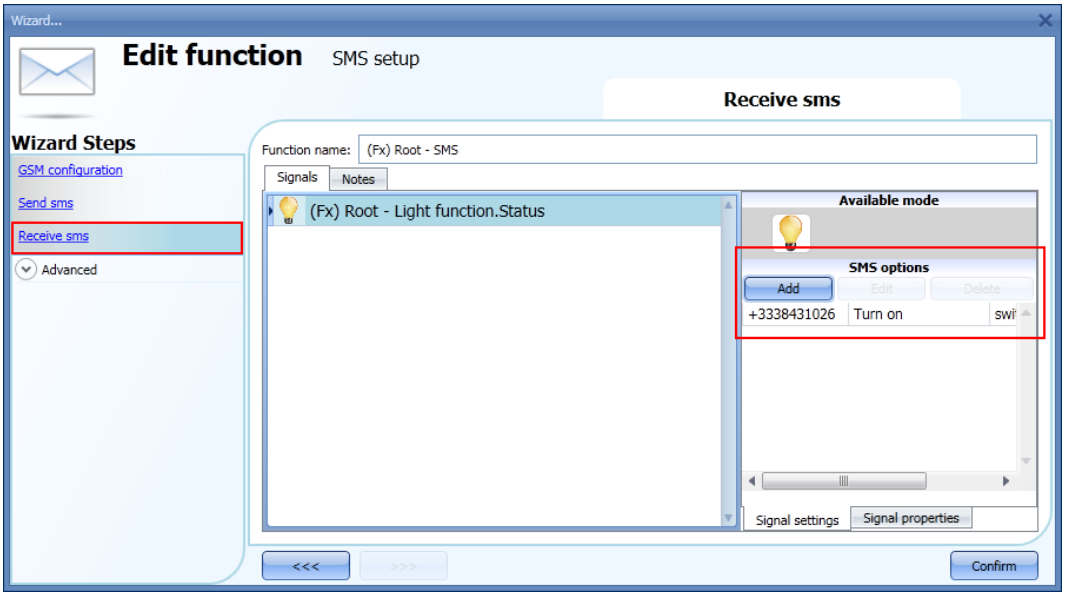
1: Send SMS to number: This contains the number the SMS is sent to. The list of the numbers added in the SMS configuration will be prompted. If all of them are selected, the SMS will be sent to all.

2: SMS text for event: in this field the user can type a text that informs the receiver about what is happening.

3: Send when signal: in this field the user selects the event that will cause the SMS to be sent. In the example on the left, an SMS with the text “My light on” will be sent when the status of that light changes from off to on.

21.3 How to set up the UWP 3.0 to receive an SMS

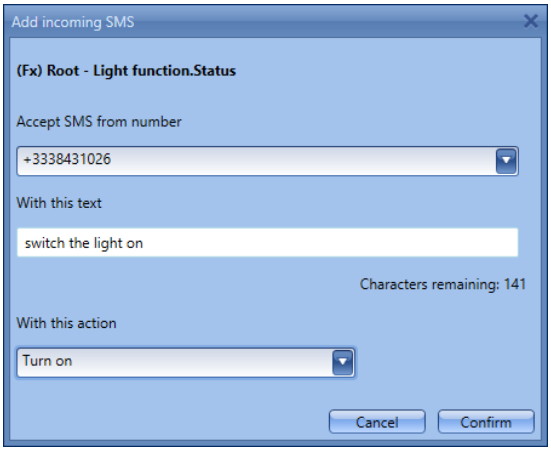
In the *Receive SMS* field, the user has to add all the functions that can be controlled by an SMS. Once a function is added, by a single click on the function the user can access the SMS options (see picture below).



When the function is added, the possible options are:

- Add a new action according to the status of the function
- Edit an already added action
- Delete an action

When the *Add* icon is selected, the following setup-window is shown:



Fill in the boxes with the necessary information:

1: Accept SMS from number: This contains the number the SMS comes from. The list of the numbers added in the SMS configuration will be prompted. If all of them are selected, the SMS will be received by all.

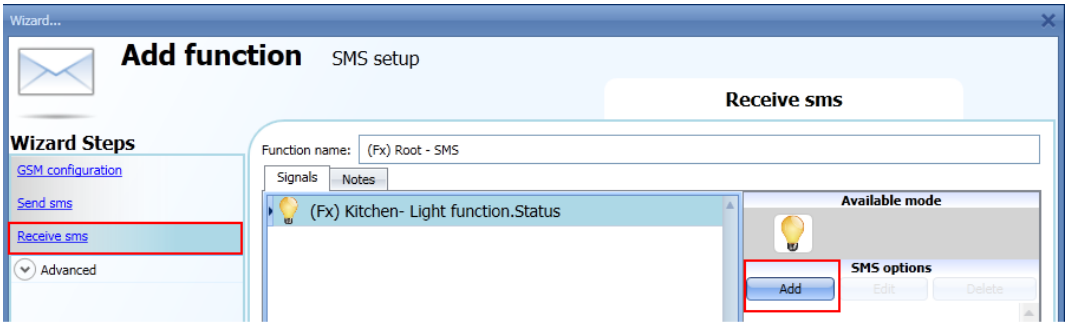
2: With this text: in this field the user types the text that is to be received: if the received text does not match the one written here, the SMS will not be processed.

3: With this action in this field the user selects the event the system will carry out after receiving this SMS. In the example on the left, an SMS received with the text “switch the light on”, will turn the light on.

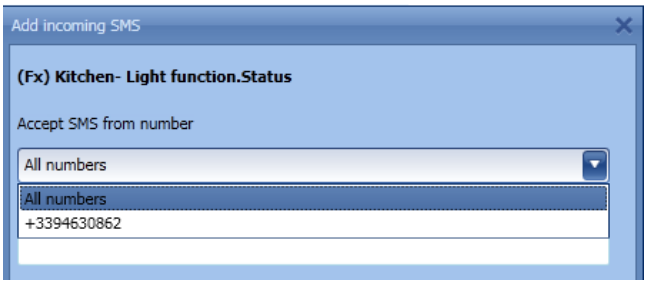
Example 1

In the kitchen a function called *Light-kitchen* is to be controlled by using the SMS.

This function has to be added in Receive sms (see below).

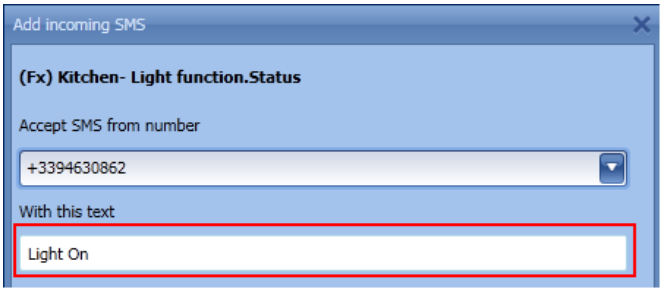


By selecting the function and by clicking on the *Add* pushbutton, the following window will appear.

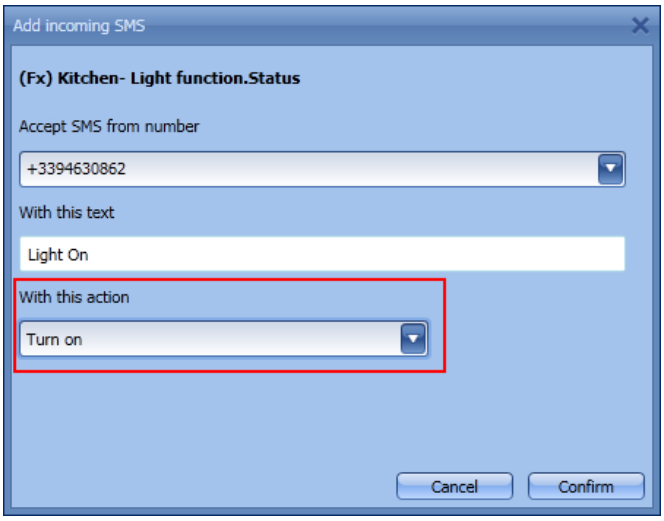


Select the phone number from which you want to control the function.

Then, fill in the text to be used to activate the function.

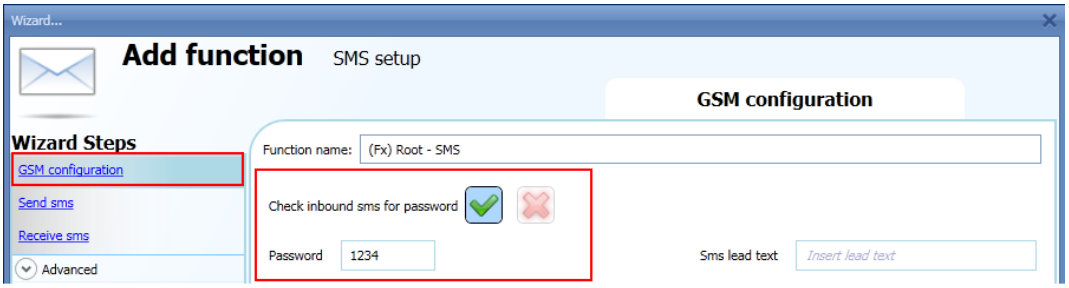


Select the required action of the SMS, e.g. “Turn on”.



The SMS to be sent to the UWP 3.0 controller is: (assuming that the phone number of the SIM card in the GSM module SH2UMMF124 or Dongle modem together with the SH2DSP24 module is +393338431022, country code +39).

Prepare the SMS with the text “Light on”
Send this SMS to the phone number “+393338431022”.
When the GSM unit receives this message, the function Kitchen-light will be switched on.



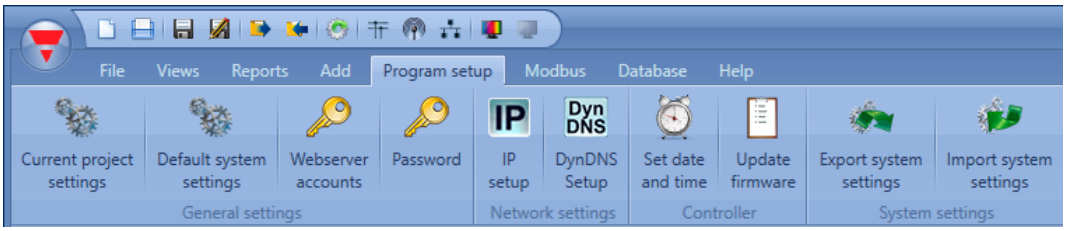
If a password is selected in the GSM configuration (e.g. PSW 1234), the text to be sent should look like this: “1234,Light on”
Please note that a comma is used to separate the password and the command text, and no space is allowed before or after the comma.

NB. All received texts are automatically converted into lowercase letters

To complete the settings of the modem, see also paragraph General settings.

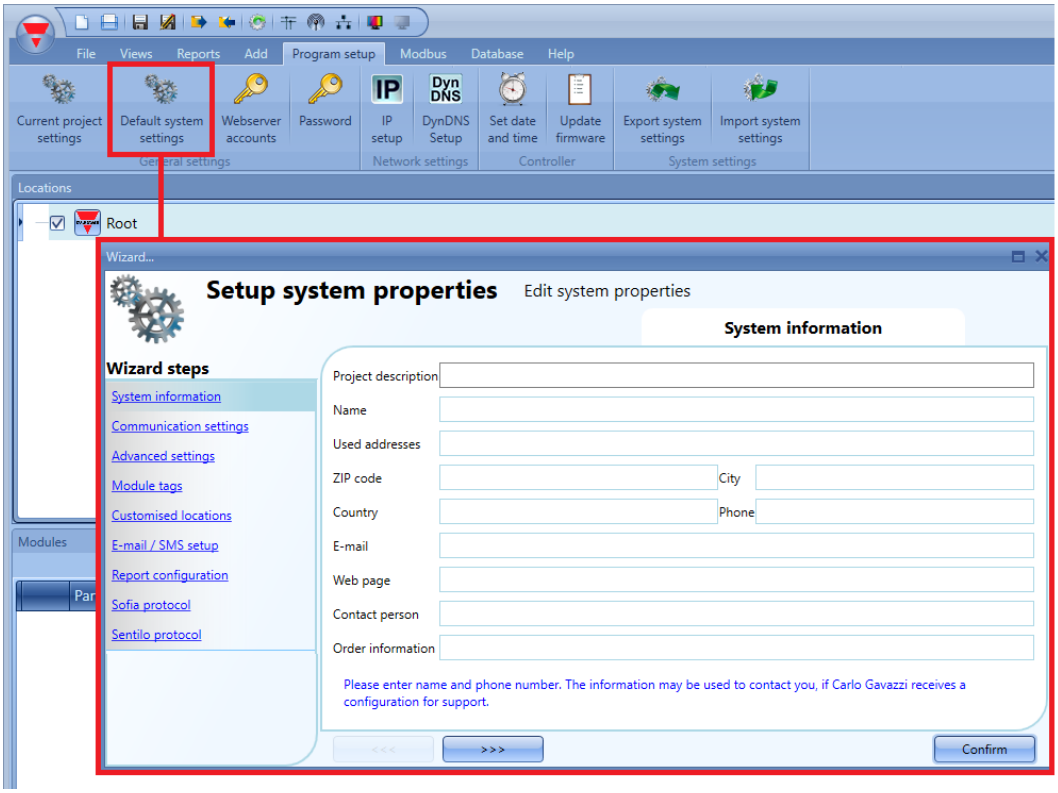
22 General settings

In the *Program setup* menu, the user can configure the settings relevant to a specific project, the general settings of the installer, the webserver accounts, the password settings, the IP address of the UWP 3.0, date and time of the UWP 3.0 and the DynDNS parameters when the modem is used. They can also update the firmware and *Import* and/or *Export* the *system* settings.



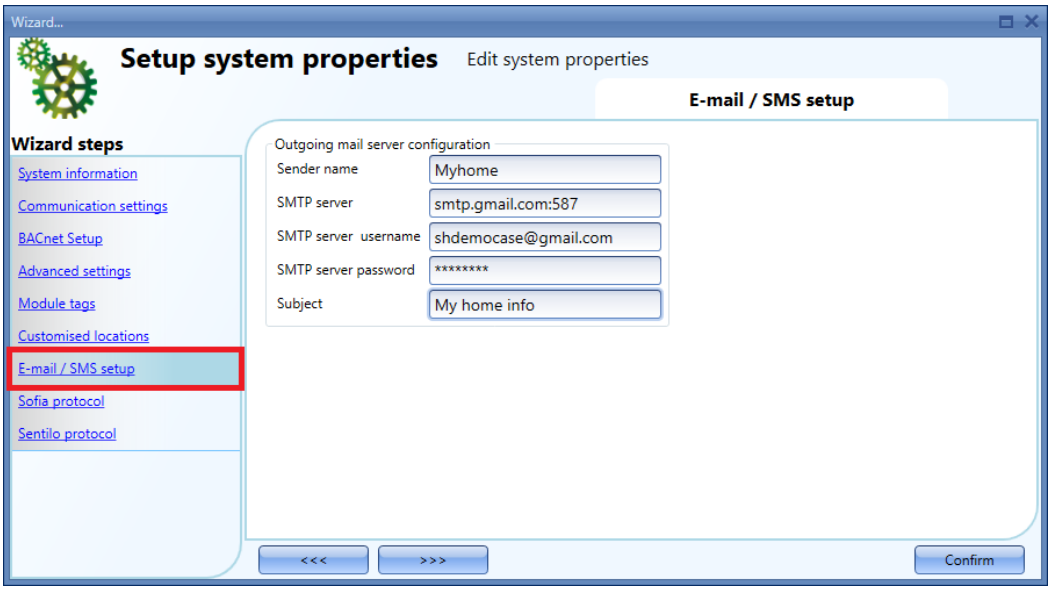
22.1 How to configure the general settings

By clicking on the *Default system settings*, the following window will appear: all the settings configured here will be available in all the new projects created, without needing to repeat the operations every time.



22.1.1 E-mail /SMS setup

In *Properties,* the user should configure the parameters of the email account so that the system can send emails.



The user must enter the *Sender name* by typing the name used for the sender (e.g. My home).
The user must complete *SMTP server* with the address of the server used for sending the email.
The user must complete *SMTP server username* with the email address used for sending the email.
The user must complete *SMTP server password* with the password for the email account.
The user must complete *Subject* with the name used as the subject for outgoing emails.

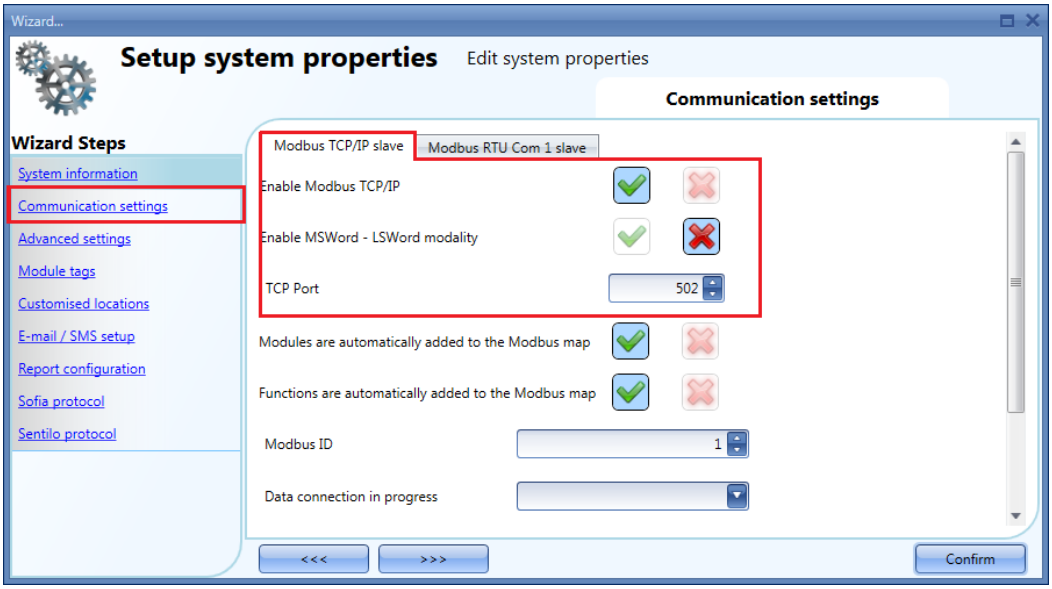
Note: usually the mail service uses port 25, although some providers have changed this port to another one in order to limit SPAM: for instance, the GMAIL account uses port 587.
To set a different port, the SMTP server must be followed by "**:587**": for instance, smtp.gmail.com:587.
Please check the provider requirements when configuring your mail settings.

22.1.2 System information

In *System information* the installer has to fill in their personal data.

22.1.3 How to set Modbus TCP/IP settings

In the field *Communication settings* the user can define the communication parameters relevant to the Modbus TCP/IP and the modem.



Enable Modbus TCP/IP
When the green V is selected, the function is enabled and the system will answer the Modbus TCP/IP requests received on the selected TCP port.

Enable MSWord-LSWord modality
If this option is enabled, in the double word values, the most and least significant words are swapped. By default, the least significant word is sent first.

TCP Port
The user can select the communication port for the Ethernet connection (by default it is 502).

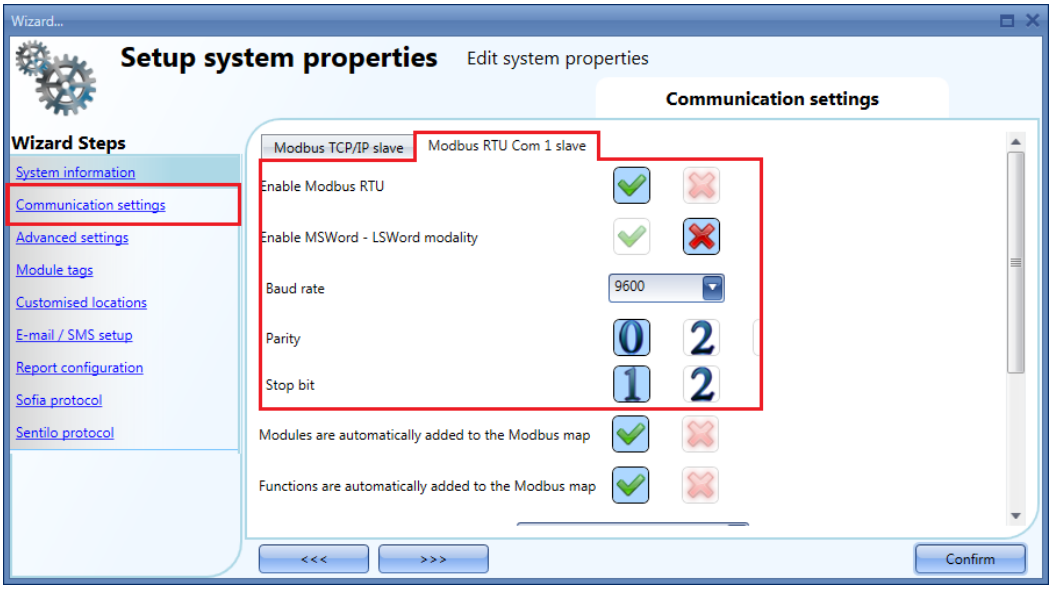
Modules are automatically added to the Modbus map
If this option is enabled, each module added to the configuration will be automatically added to the Modbus map. By default this option is disabled and the user has to add each module manually.

Functions are automatically added to the Modbus map
If this option is enabled, each function in the configuration will be automatically added to the Modbus map. By default this option is disabled and the user has to add each function manually.

Modbus ID
The user can select the Modbus ID of the controller. The ID can be any number from 0 to 250.

22.1.4 How to set Modbus RTU properties

In *Communication settings* the user can define the communication parameters relevant to the Modbus RTU Com 1 port - slave.



Enable Modbus RTU
When the green V is selected, the function is enabled and the system will answer the Modbus RTU requests received on the RS485 port.

Enable MSWord-LSWord modality
If this option is enabled, in the double word values, the most and least significant words are swapped. By default the least significant word is sent first.

Baud rate
The user can select the baud rate for Com1 port (default is 9600).

Parity
The user can select the parity bit for Com1 port (default is 0).

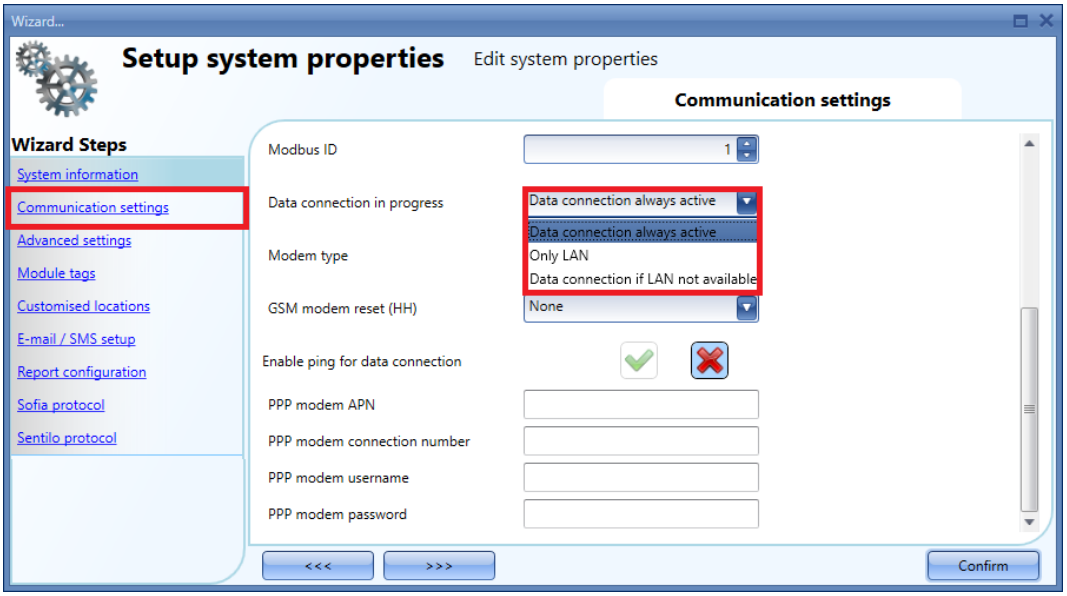
Stop bit
The user can select the stop bit for Com1 port (default is 1).

22.1.5 How to configure the modem

The User can connect to the UWP 3.0 Tool via Internet by using a Dongle USB modem connected to the UWP 3.0, using the accessory SH2DSP24. In the *Communication settings* window, the user can configure the parameters.

Data connection in progress
The user can set three different actions when the modem(*) is connected to the UWP 3.0.

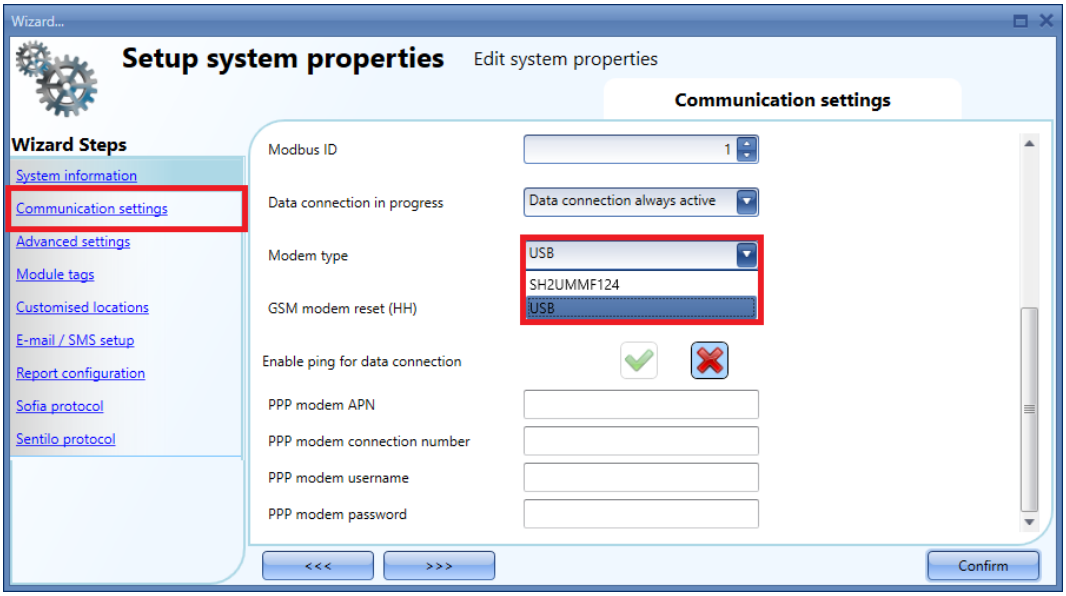
- *Data connection always active*: the system always uses the data connection to access the Internet.
- *Only LAN*: the internet connection is not managed by means of the modem; the modem is only used to send and receive SMS.
- *Data connection if LAN not available*: usually the system works using the LAN connection. If the LAN connection is not available, the system uses the modem data connection and it will continue using it until a new configuration is written or the system is reset.



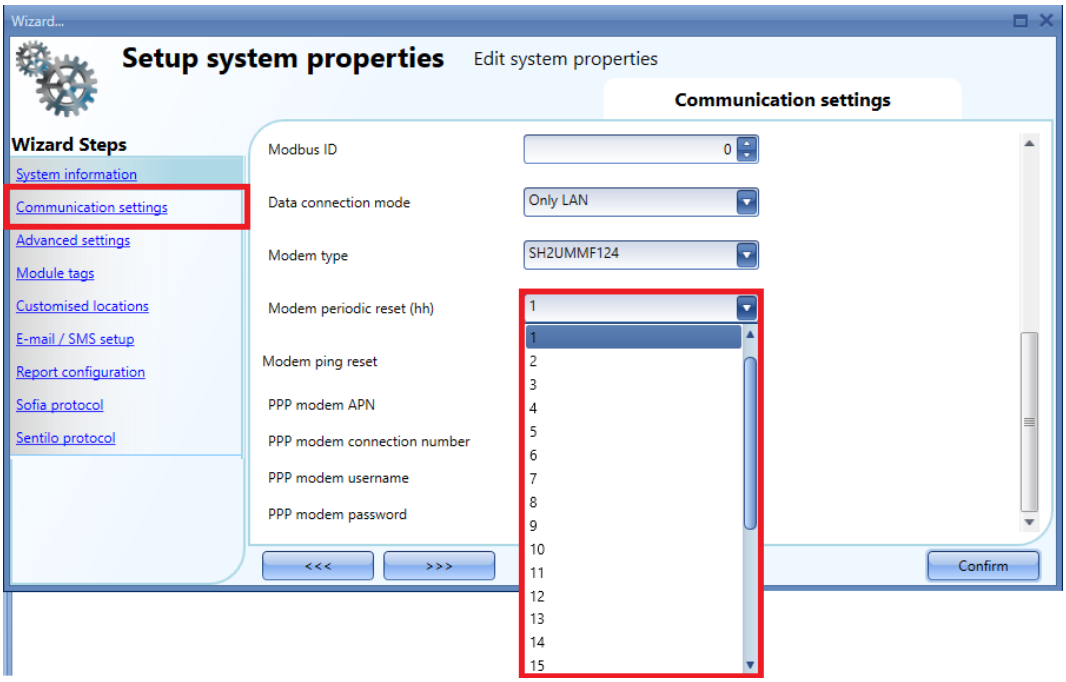
Modem type
In the *Modem type* field of the window, as shown in the red rectangle in the picture below, the user has to select the modem used in the project:

- when *SH2UMMF124*(*) is selected, the universal mobile modem is configured;
- when *USB* is selected, the USB Dongle modem connected to the SH2DSP24 module is configured;

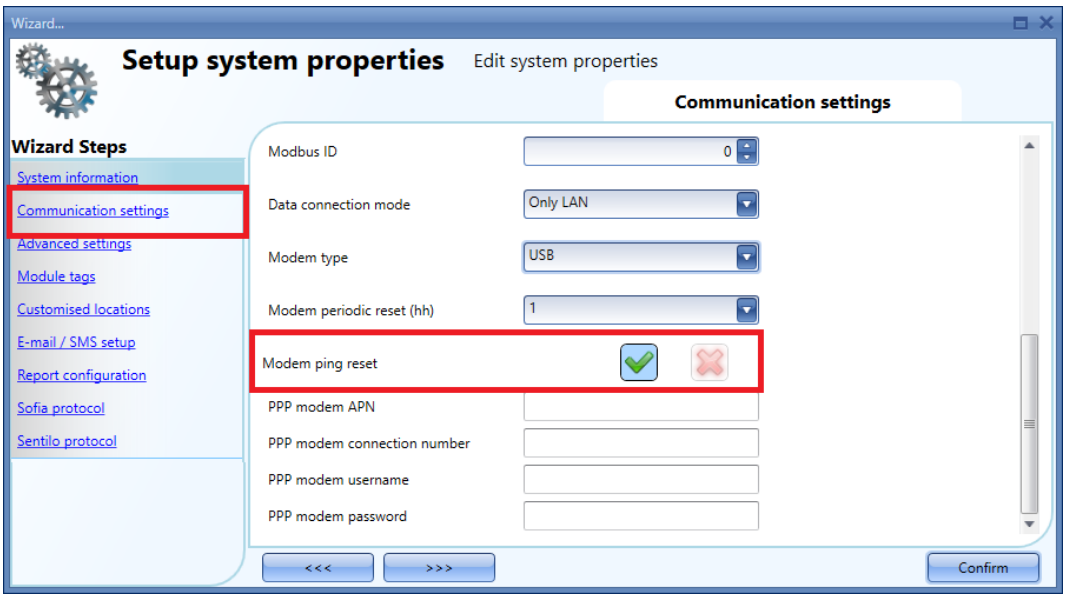
(*) The SH2UMMF124 universal mobile modem is no longer available: it is mentioned here only for compatibility with the past.



Modem periodic reset (hh)
The modem is reset every specified number of hours; the user can select a value from 1 to 24 hours. When *None* value is selected, the option is disabled. This option is useful when the modem is used for the SMS function and it means that the modem is always available for sending and receiving messages.



Modem ping reset
When this option is enabled (the green *V* is selected), the system constantly checks that remote servers are reachable: should they not be, the modem is automatically reset and the Internet connection is re-established. This is useful when the data connection of the modem is used to access the Internet and helps to maintain for as long as possible the dynamic IP address delivered by the Internet service provider (ISP).



The use of the *Modem periodic reset (hh)* and/or the *Modem ping reset* options depends on the functions the user has configured in the project. There are different possible scenarios, so please consider the following recommendations:

- When the modem is used to access the Internet, the *Modem periodic reset (hh)* option should not be activated (the *None* value should be selected). Frequent resets of the modem might create inefficiency to the network.

- If *Data connection if LAN not available* is enabled, we suggest also enabling the *Modem ping reset* option, because every time the system switches from the LAN to the modem, it allows a dynamic IP to be obtained.

PPP modem APN (***mandatory***)
The user must add the NTP server of the Mobile Service Provider used by the modem.
Note: some providers do not need other info to be added, such as modem connection number, username and password, so please check the provider requirements when you configure the modem settings.

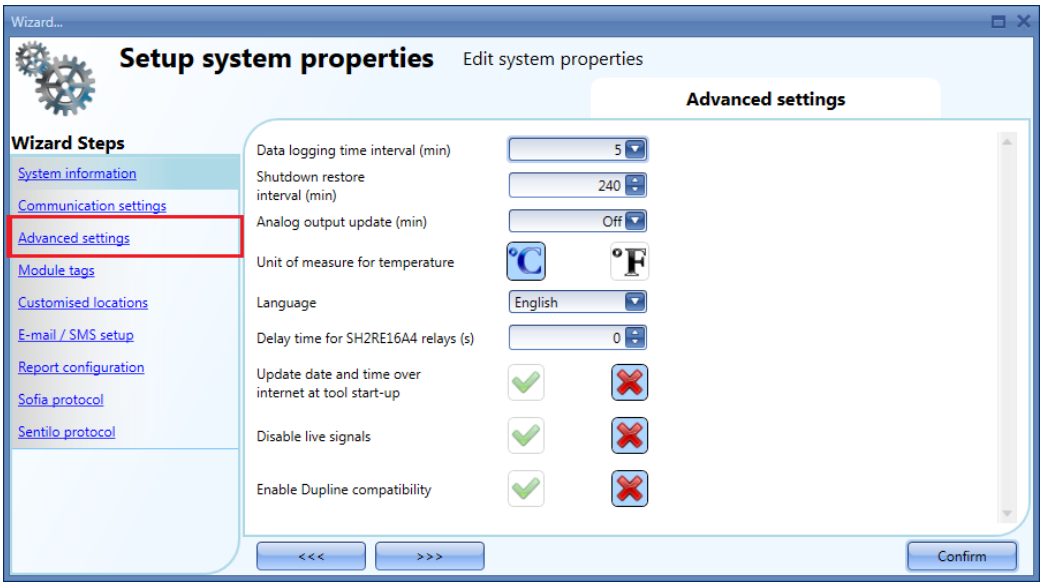
PPP modem connection number
(*) fill in only if specified by the Service Provider

PPP modem username
(*) fill in only if specified by the Service Provider

PPP modem password
(*) fill in only if specified by the Service Provider
(*) = Not mandatory: it depends on the Mobile Service Provider.

22.1.6 Advanced settings

In *Advanced settings* the user can set:



Datalogging time interval (min):
All logged variables are sampled and stored in the Database according to the time set.

Shutdown restore interval (min):
It is possible to set a timer for the auto-restart of some functions after a power-down of the master unit UWP 3.0. The timer starts counting when the supply goes off. If the supply comes back on within this time, the system will go back to the on status it had before the power-off. Otherwise, it will start in an off status. The status of each function is stored in the memory every minute.

The functions with “back-up” capability are the following:

- On/off light : at power on the function is set at the status it had before the power off.
- Dimmable light: if the function is configured with an energy-save timer, at power on the function is always switched at the last saved scenario and the energy-save timer is retriggered.
- Temperature system: at power on the function is set at the status it had before the power off.
- Zone temperature: at power on the function is switched on with the previous heating/cooling setpoint
- Main alarm: at the power on the function is set at the status it had before the power off.
- Playback: at power on the function is set at the status it had before the power off.
- Sequence: at power on the function is set at the status it had before the power off.
- Car heating: at power on the function is set at the status it had before the power off.

The functions that do not have back-up capability are:

- Rollerblinds
- Window control
- Timer
- Smoke alarm
- Water alarm
- Siren
- All basic functions

Analogue output update (min):
All the Analogue outputs are updated according to the time set.

Unit of measurement for temperature:
The user can select Celsius or Fahrenheit.

Language:
Here the user can select the language of the UWP 3.0 Tool. Once selected, the software has to be closed and then opened again to have the new language.

Update date and time over internet at tool start up:
When this option is enabled (green V selected), the tool synchronizes the date and time via internet.

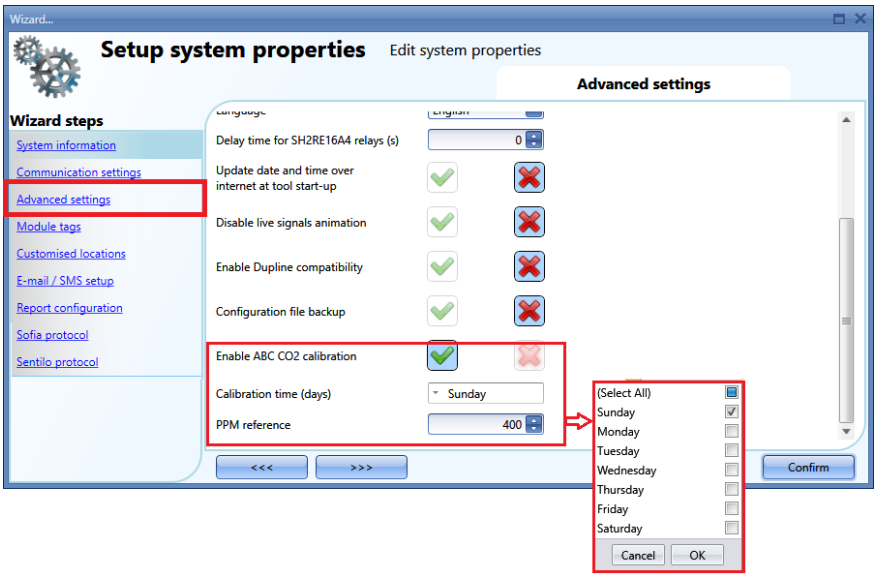
Disable live signals:
When this option is enabled (green V selected), the tool does not show graphical animation when the Live signal is active.

Enable Dupline compatibility:
When the green V is selected, the feature is enabled. The UWP 3.0 tool starts running with the standard Dupline support: new functions will be available that can manage those modules not powered by smart-Dupline.

Configuration file backup:
When the green *V* is selected, the feature is enabled. A progressive configuration file is saved in the folder *…/Document/UWP 3.0 tool projects* with the date and time information each time the project is compiled.

Enable ABC CO2 calibration:
When at least one SHSUCOxxx module is present in the configuration, the user can enable the ABC Calibration for the CO2 sensor.

In order to activate the ABC Calibration, the green *V* has to be selected in the *Enable ABC CO2 calibration* field then, the *Number of days* and *Reference ppm* fields have to be set (see the red rectangle in the picture below).



Calibration time (days): select here the week days during which the system will read the minimum value to associate to the reference edited in *PPM reference*

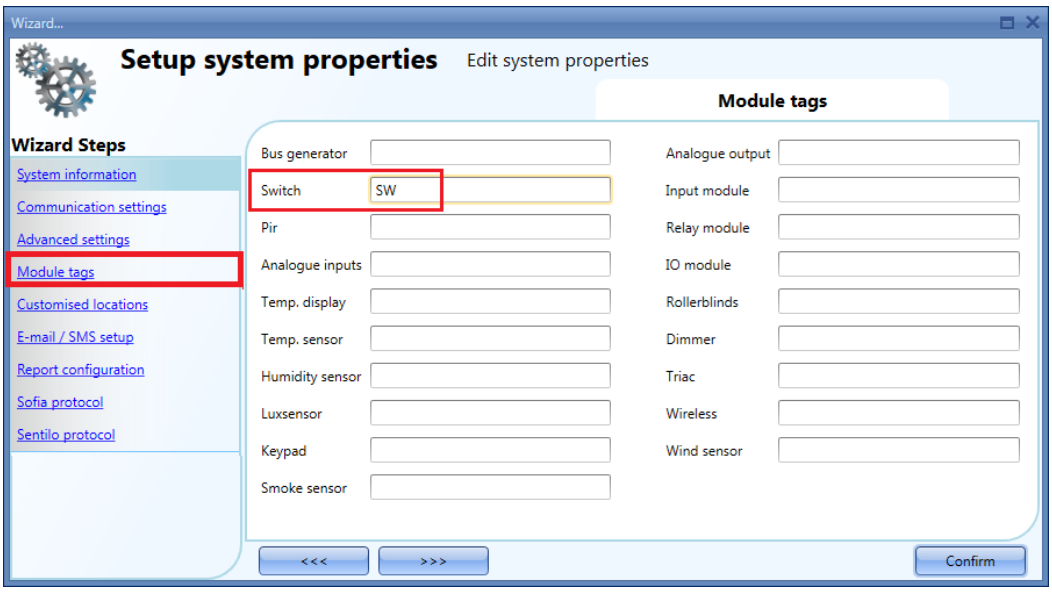
PPM reference: this is the value that SHSUCOxxx associates to the lowest CO2 level read during the night of the days selected in *Calibration time (days)*

If *Enable ABC CO2 calibration* is enabled, the SHSUCOxxx will adjust the read value according to the minimum value set in *Reference ppm.*
If *Enable ABC CO2 calibration* is not enabled, the SHSUCOxxx will adjust the read value according to the factory calibration.

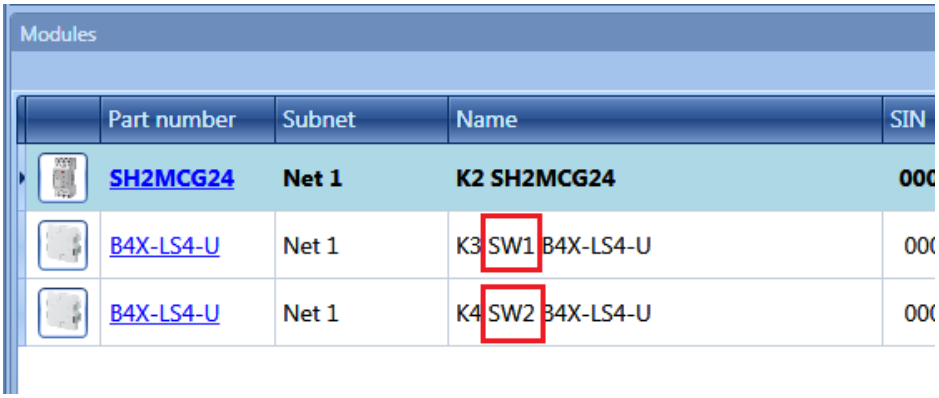
22.1.7 How to add module tags

In this window the user can define the name for each group of modules. This name will be used when a new module is added to the project and then it will also be reported in the module list report file.

Example: in the figure below, the module tag SW is added to the light switch family.



When a new light switch module is added, the name tag will be linked to a number that is increased each time a module of that group is added: for example, the first light switch module will be preceded by *SW1*, the second one by *SW2* and so on so forth (see picture below).

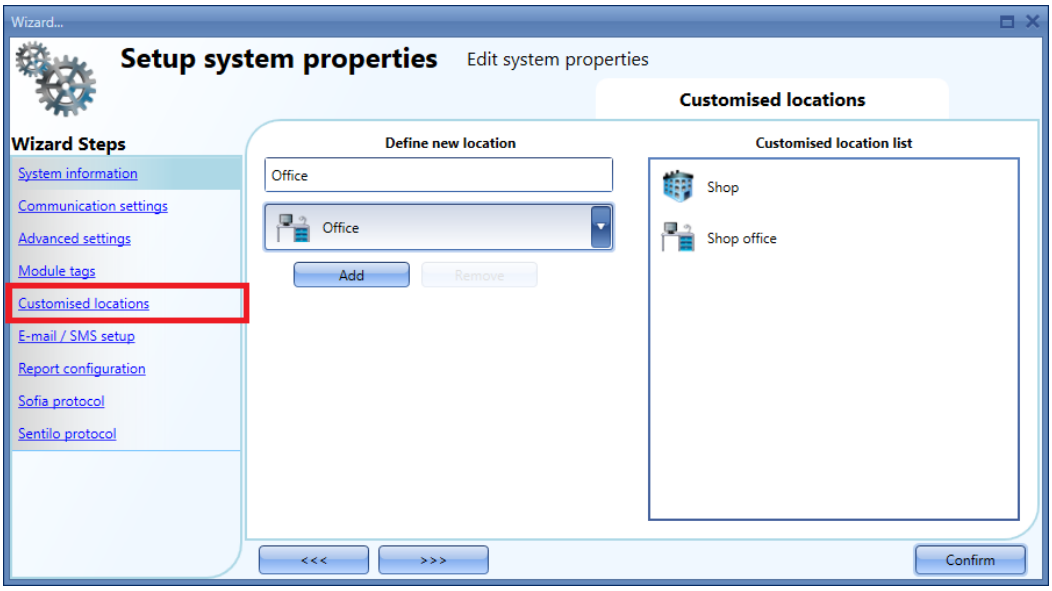


The same name tag will also be used in the module li*st* report file, as shown in the example below.

N.	Family	Part number	Name	S
1	Master Generator	SH2MCG24	K2 SH2MCG24	0
2	Switches	B4X-LS4-U	K3 SW1 B4X-LS4-U	0
3	Switches	B4X-LS4-U	K4 SW2 B4X-LS4-U	0

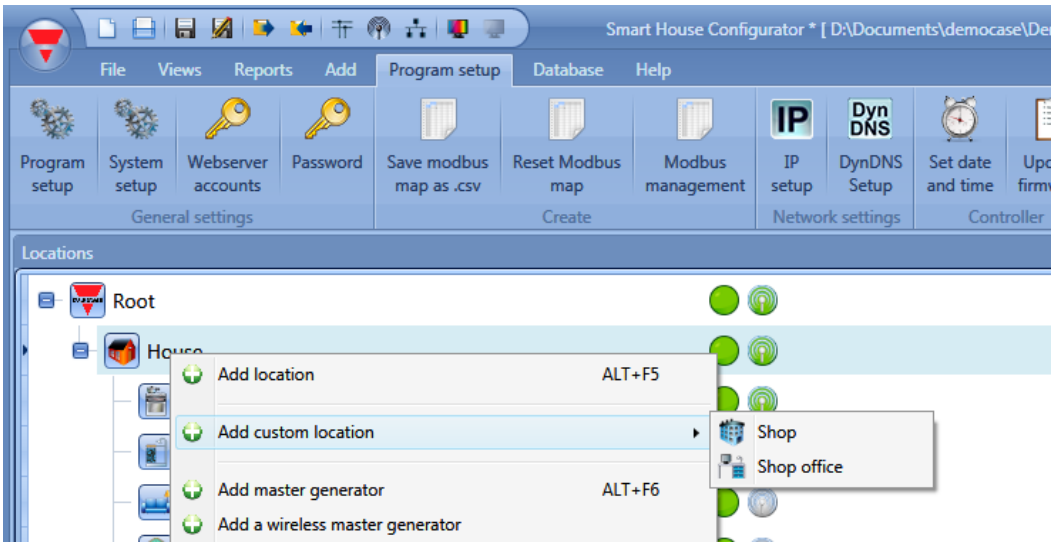
22.1.8 How to create customized locations

In the menu *Customized locations,* the user can define his customised locations.



The name and the icon of the location have to be selected, then by clicking on *Add* the new location is added to the list on the right.

To use these, right click on the Location area and select *Add custom location*: the new ones will be available.

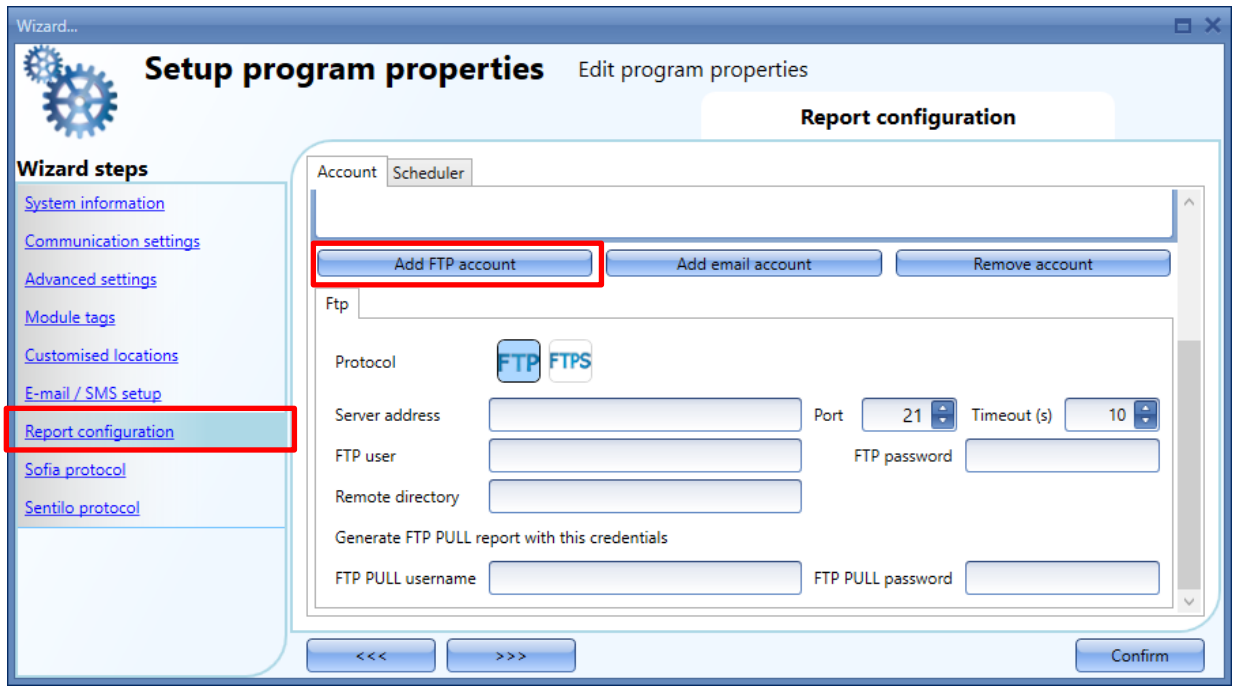


22.2 How to configure the Report

In the *Account* tab of the *Report configuration* window, the user can manage the FTP/SMTP accounts that can be used to send the reports according to schedulers.

22.2.1 How to add an FTP account

The user can add an FTP account by clicking the *Add FTP account* button, as shown below:



The following settings have to be defined:

Protocol: the type of protocol of the FTP account: FTP or FTPS

Server address: this field has to be filled-in with the address of the FTP server to which the report files will be sent

Port: usually the FTP service uses port **21** while the FTPS uses port **22**: however, the port that the server listens to for FTP connections can be any port not already reserved for another service, and this is also configured by the server administrator.

Timeout(s): specifies the timeout, in seconds, in which the FTP account will try to connect to the FTP server before timing out

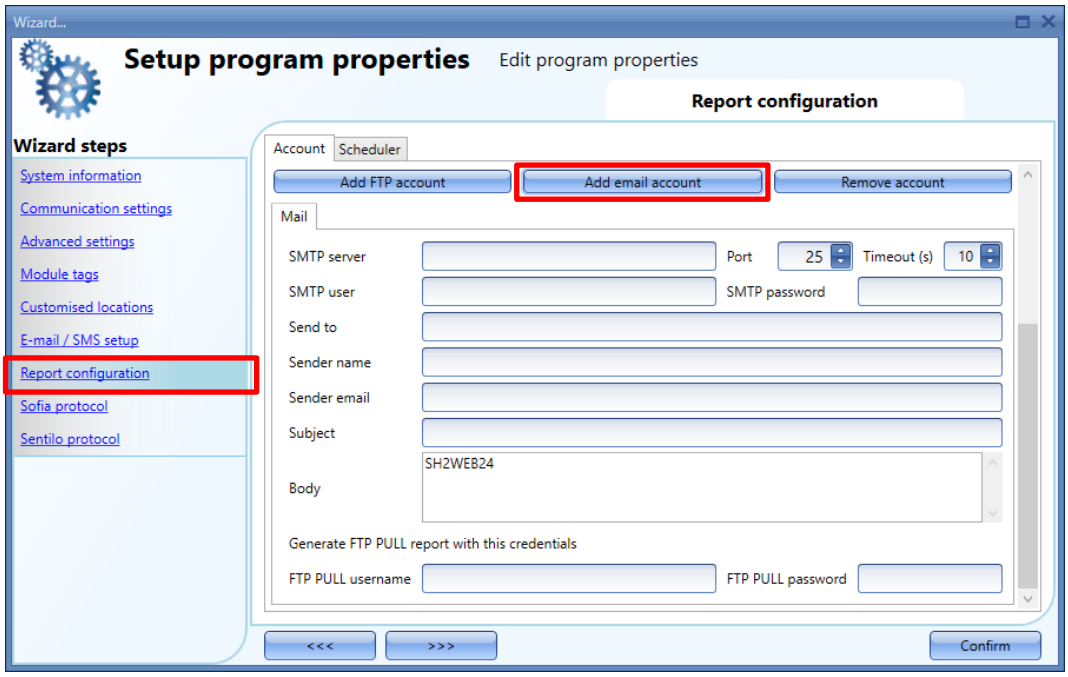
FTP user and FTP password: the user must enter complete *FTP user* and *FTP password* fields with valid credentials to access the remote FTP directory

Remote directory: this field contains the directory of the FTP server where the report files have to be stored

FTP PULL username and password: these settings have to be defined with valid credentials for FTP PULL services

22.2.2 How to add an email account

The user can add an email account by clicking the *Add email account* button, as shown below:



The following settings have to be defined:

SMTP server: the user must complete the SMTP server field with the address of the server used for sending the email

Port: usually the mail service uses port 25, although some providers have changed this port to another one in order to limit SPAM: for instance, the GMAIL account uses port 587.
Please check the provider requirements when configuring an SMTP account

Timeout(s): this field specifies the timeout, in seconds, in which the SMTP account will try to connect to the SMTP server before timing out

SMTP username and SMTP password: the user must complete the *SMTP server username* and *password* fields with valid credentials for the email account

Send to: the user has to complete the *Send to* field with a valid email address to which the report files are to be sent

Sender name: the user can enter the Sender name by typing the name used for the sender *(e.g. Web-app)*

Sender email: this field contains the address the email is sent from

Subject: the user can specify the subject to be indicated in the report

Body: in this box the user can type a text which informs the receiver about the content of the report files

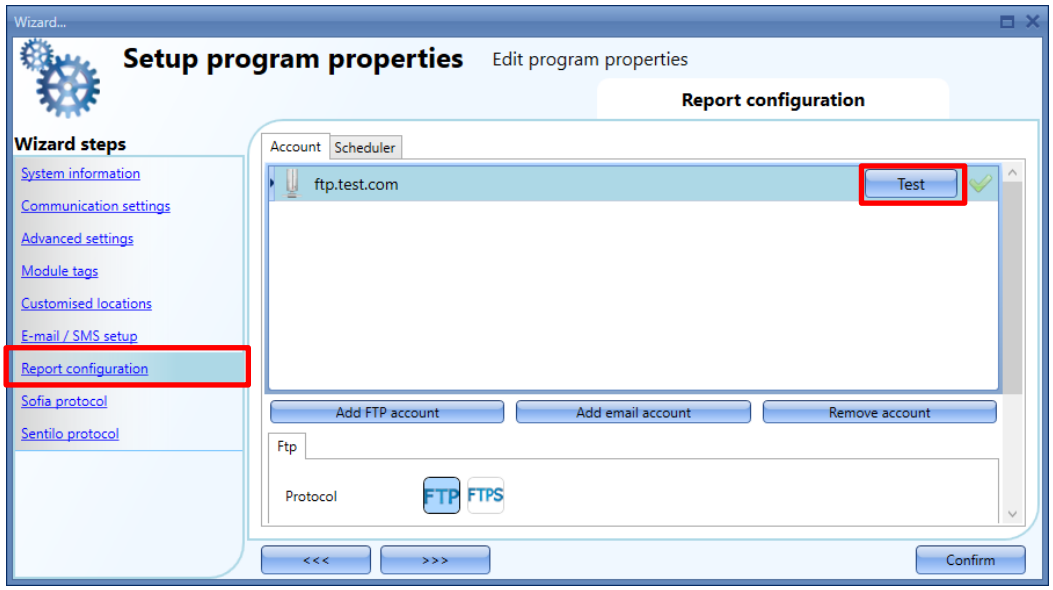
FTP PULL username and password: these settings have to be defined with valid credentials for FTP PULL services

22.2.3 How to remove an existing account

In the *Account* tab the user can select the account which has to be removed: by clicking on the *Remove account* button the account will be deleted.

22.2.4 How to test the FTP / email connection

In the *Account* tab, once the fields have been compiled for the FTP accounts and/or the email accounts, as described above, to test the communication click on the *Test* button, as shown in the red box below:

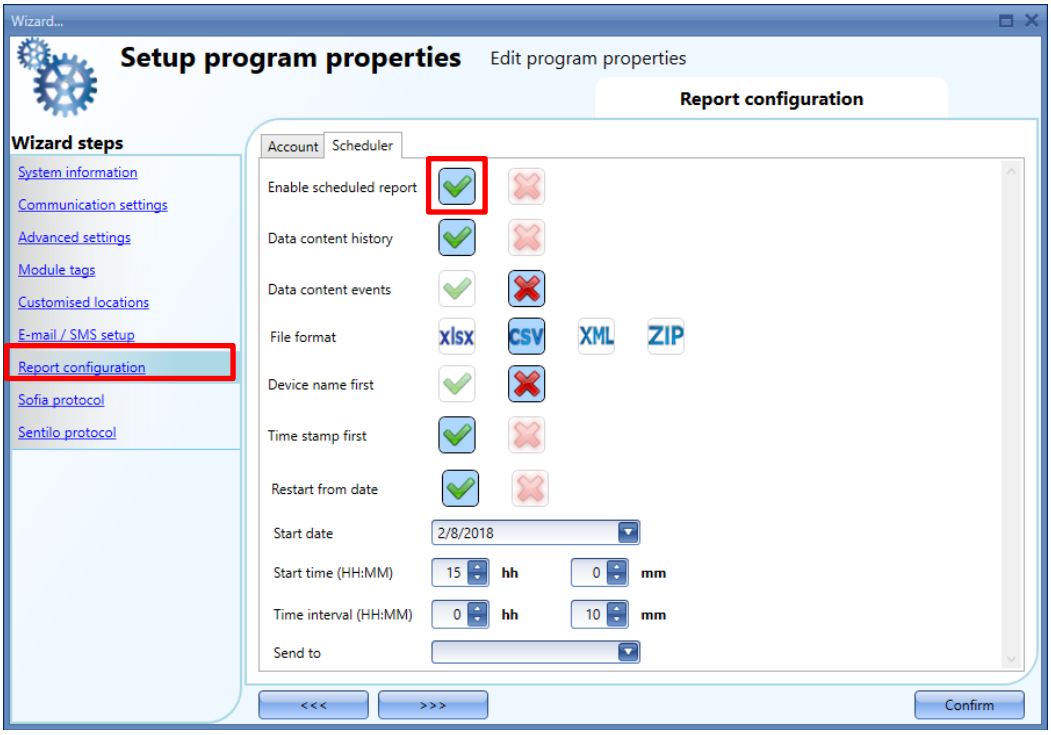


The communication test will be immediately started: the response from the test will be displayed by the icon on the right:

- a *green V* icon indicates the communication test is successful
- a *red X* icon indicates the communication test is not successful: by hovering with the mouse over this, a tooltip will show the error details

22.2.5 How to enable the Report scheduler

In the *Scheduler* tab of the *Report configuration* window, select the *green V* tick to enable the scheduler, as shown below:



The following settings have to be defined:

Data content history: the user has to select what kind of logged file has to be sent

***Data content events*:** the user has to select what kind of logged file has to be sent

***File format*:** the type of file the user wants to receive when the report is generated: XLSX, CSV or XML. Select ZIP to save the report in a ZIP archive

***Device name first*:** the name of the report file will start with the device name information,
e.g. UWP 3.0 Serialnumber_2017-10-08_15.28.01_S_data.zip

Time stamp first: the name of the report file will start with the time stamp information,
e.g. 2017-10-08_23.59.59_UWP 3.0 Serialnumber_T_all.zip

Restart date: if this option is enabled, the user can specify in the ***Start date*** and ***Start time (HH:MM)*** fields the date and time the report service will be activated

Time interval (HH:MM): the user has to define how often the file has to be sent.

Send to: the user can select the account where the report files have to be sent, selecting from the FTP accounts or email accounts configured.

22.3 How to configure the Project settings

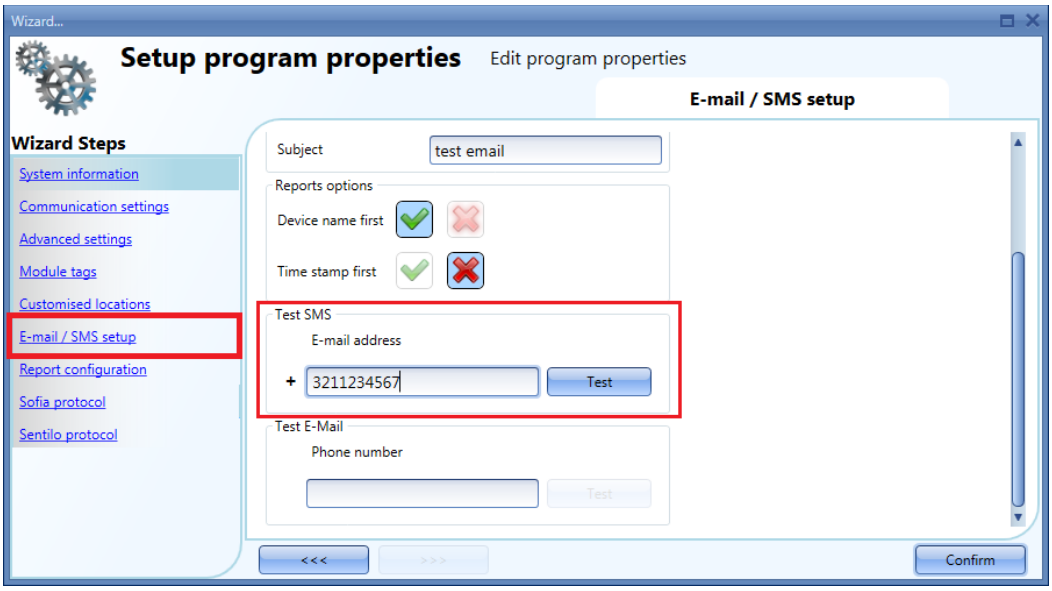
By clicking on *Project settings,* a window identical to the *System settings* will appear: each menu is the same as the one seen above, the only difference being that all the configurations and settings are relevant to the specific project that is under development at this moment, and will not be copied into new projects.

22.3.1 How to test Email and SMS

If the user wants to test the EMail and SMS functionality regardless the presence of the *Email* function and the *SMS* function in the configuration, he can do the test directly from the *Project settings* window. Moreover, any Email address and any mobile-phone number can be used to receive a test SMS.

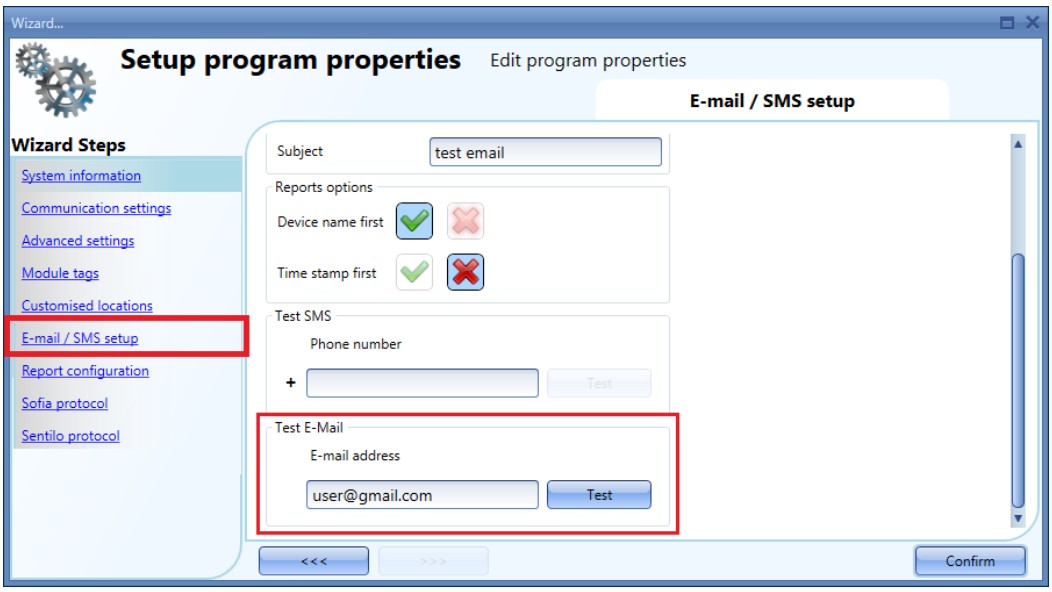
22.3.1.1 Test SMS

1) A modem has to be connected to the UWP 3.0. This does not have to be configured for the SMS functionality.
2) In the *Test SMS* field of the *Email / SMS setup* menu, add the mobile-phone number that has to start with the Country code (e.g. +39 for Italy). ***The user must be careful to review the mobile-phone number correctly, no error message will appear if the number is not correct.***
3) Press the *Test* button: an SMS will be sent by the system to the mobile-phone. An example is shown in the picture below.



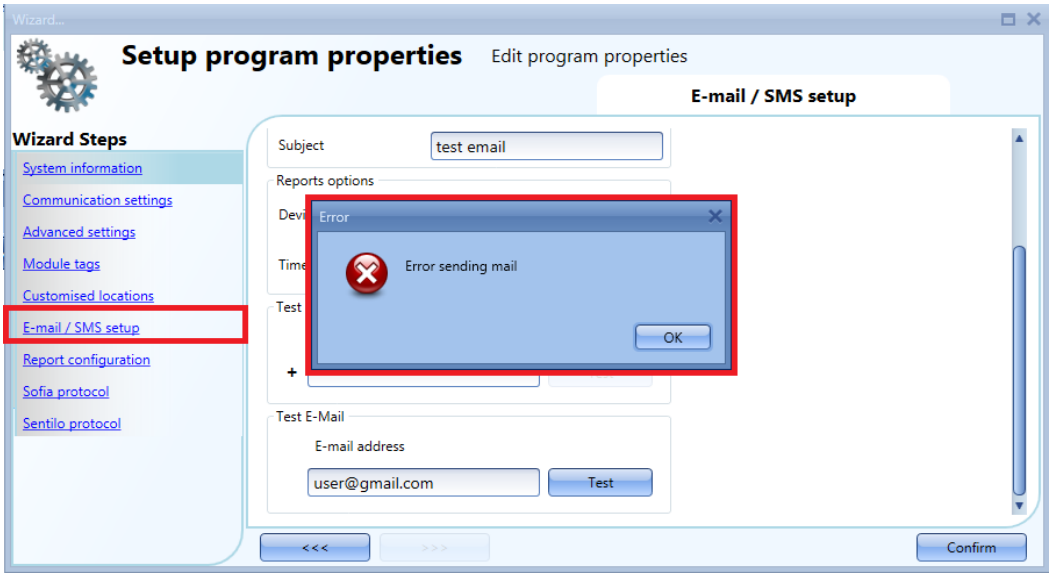
22.3.1.2 Test E-mail

1) In the *Project settings* menu, configure the settings of the e-mail account so that the system can send e-mails (Fill in the fields *Outgoing mail server configuration*)
2) Write the configuration into the UWP 3.0
3) In the *Test E-Mail* field of the *Email / SMS setup* menu, add the e-mail address to which the email has to be sent and press the *Test* button: a verification e-mail will be sent to this address by the system. An example is shown in the picture below.
4) Check if you have received this email



This is a fake e-mail used to test the e-mail sending from:
UWP 3.0 name: test
UWP 3.0 IP Address: 192.168.2.76
UWP 3.0 MAC Address: 00:19:EE:10:12:40 at Tue Sep 8 10:27:44 2015

If the settings parameters of the E-mail account are not correct, an error message will appear, as shown in the red rectangle in the picture below.

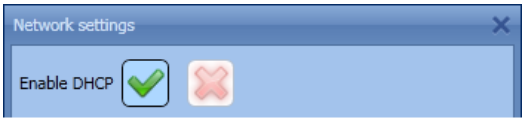


Should this happen, the user has to check all the parameters and repeat the procedure described above.

22.4 How to configure the IP address of the master unit UWP 3.0

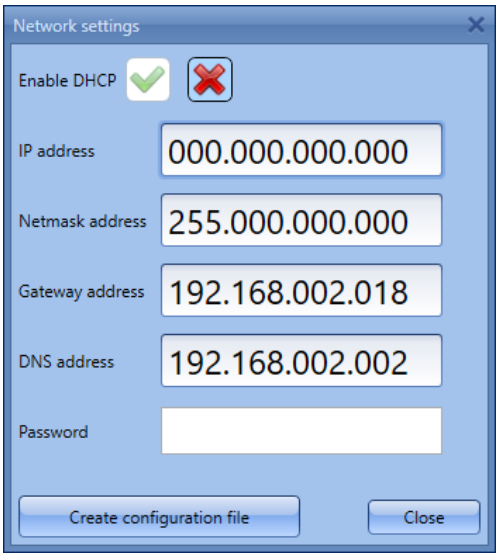
In the program setup, by clicking on the IP setup icon the user can enter the controller IP settings.

The user can enable the controller DHCP client by selecting the green V icon *Enable DHCP*:



Or set a static IP address, Netmask, Gateway and DNS address by clicking on the red cross icon.

NB: When you set up a fixed IP, the Gateway address and DNS address are mandatory, and the UWP 3.0 Tool automatically adds the Gateway and DNS address by obtaining the missing information from the PC settings.



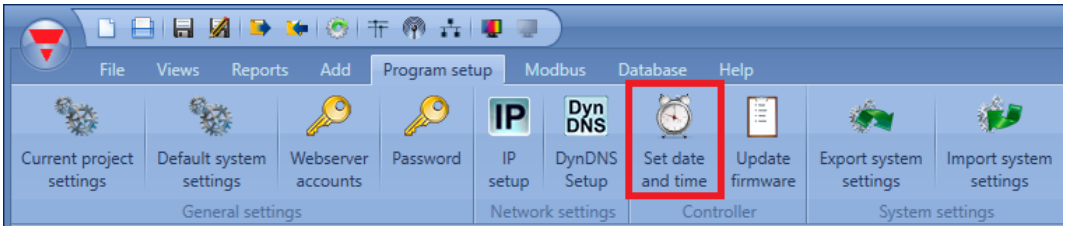
Once you have entered the required field, click on *Create Configuration File:* the UWP 3.0 Tool will create the file *controller_settings.shcfg for UWP 3.0 Tool and controller_settings.sbcfg* for UWP 3.0 Tool that you must copy into a USB stick or a microSD.

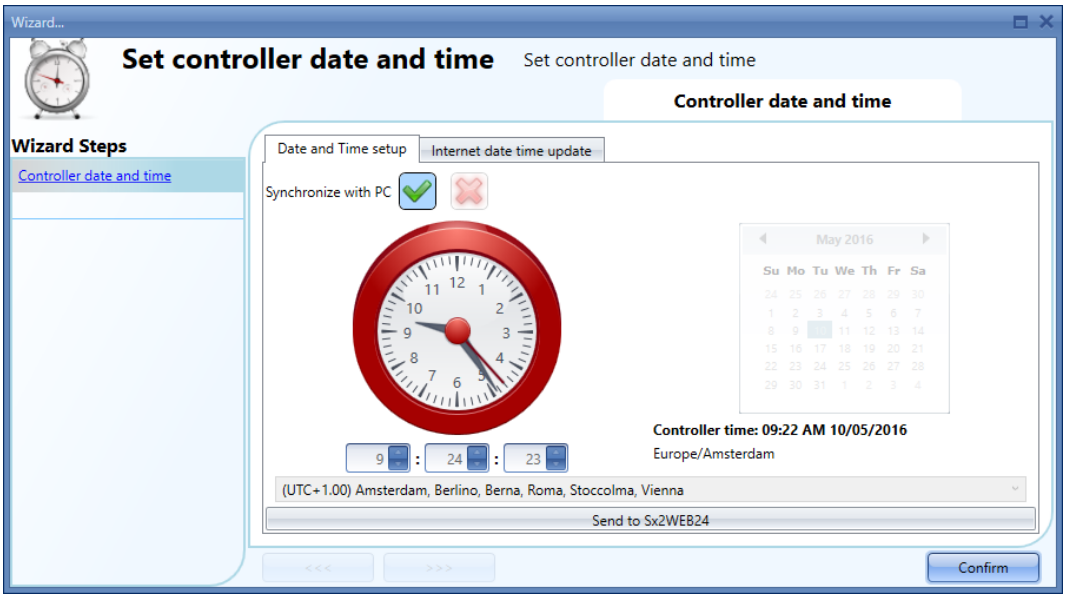
When you insert the USB stick into the UWP 3.0, it will automatically read the file. The time necessary for the operation is about 5 seconds: when the blue USB LED is lit, remove the USB card. Now the new IP settings are stored into the master unit.

N.B. This operation cannot be carried out if a modem is connected and configured in the project. If a USB modem or the SH2UMMF124 is connected, the USB ports (the one on the UWP 3.0 and the one on the SH2DSP24) cannot be used to change the IP address.
The USB port in the SH2DSP24 is available only for the USB Dongle modem.

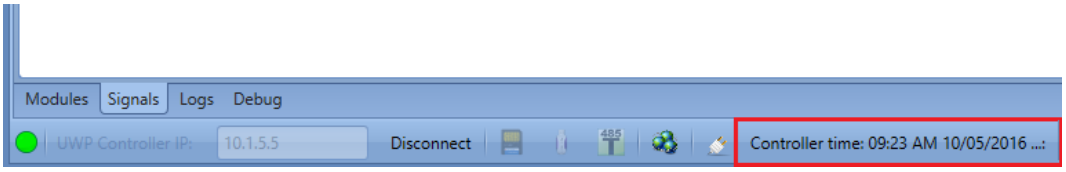
22.5 How to configure the date and time of the master unit UWP 3.0

In the program setup, by clicking on the *Set date and time* icon the user can display the controller date and time settings window.

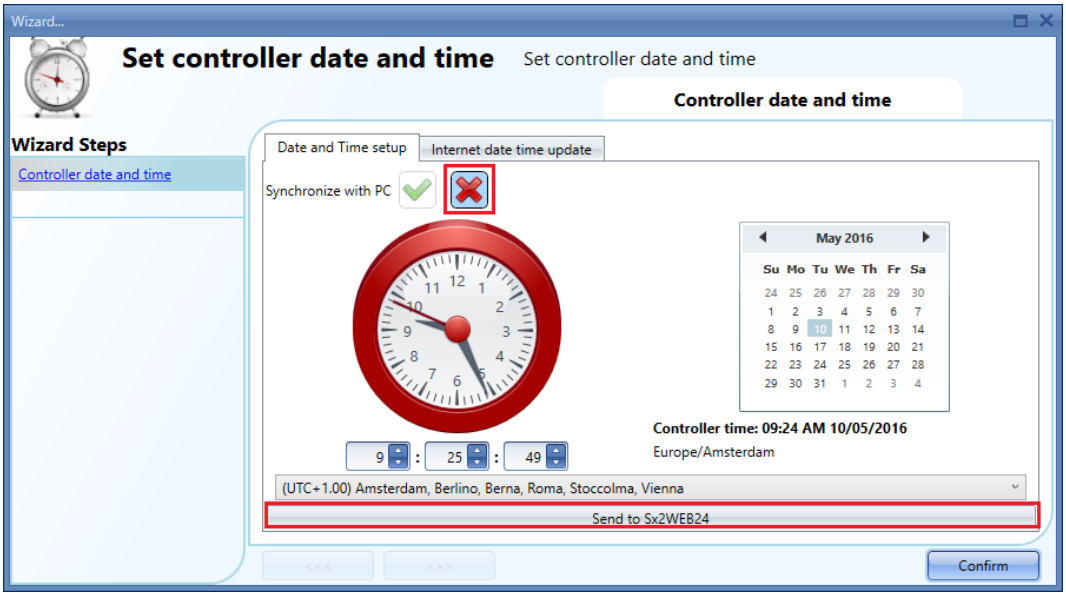




By clicking on the green V, *Synchronise with PC* and by clicking on *Send to UWP 3.0*, the Tool synchronizes the UWP 3.0's clock with your PC clock. If your PC clock is not correct, this time error will be transferred to the controller.
The current time and date of the UWP 3.0 can be seen at the bottom of the UWP 3.0 tool (see picture below).



If the user wants to manually set the clock, the red cross icon *Synchronise with PC* must be selected, then it will be possible to manually change the date and time and send the new data to the UWP 3.0 by clicking on *Send to UWP 3.0* (see picture below).

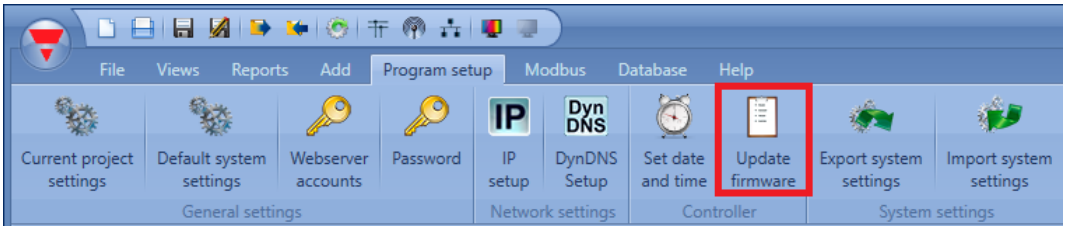


22.5.1 Internet date and time update

The user can also select to constantly synchronize the DATE and TIME data to an NTP server; this service also allows the UWP 3.0 to know in which geographic area it has been installed.
To allow the UWP 3.0 to connect to the selected servers, port 22 has to be NAT-forwarded in the router.

22.6 How to update the firmware in the master unit UWP 3.0

In the program setup, by clicking on the *Update firmware* button, the user opens a 'find' dialogue window that allows him to choose the required firmware file.



The ports 80, 10000, 10001 and 52325 are used for firmware upgrade procedure. Ensure that ports **are open** in the network devices (e.g. Firewall). Please contact your system administrator for further information.

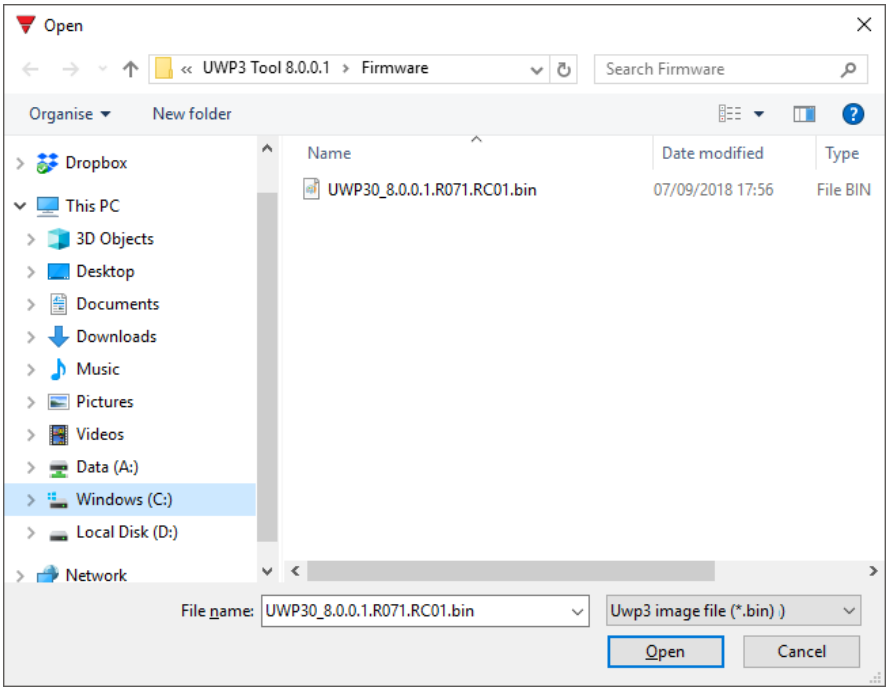
N.B: Before updating the firmware, this procedure must be followed in order to update old configurations:

1. Read the actual configuration stored in the UWP 3.0 and save it with the new UWP 3.0 Tool into your PC

2. Update the firmware following the procedure described below

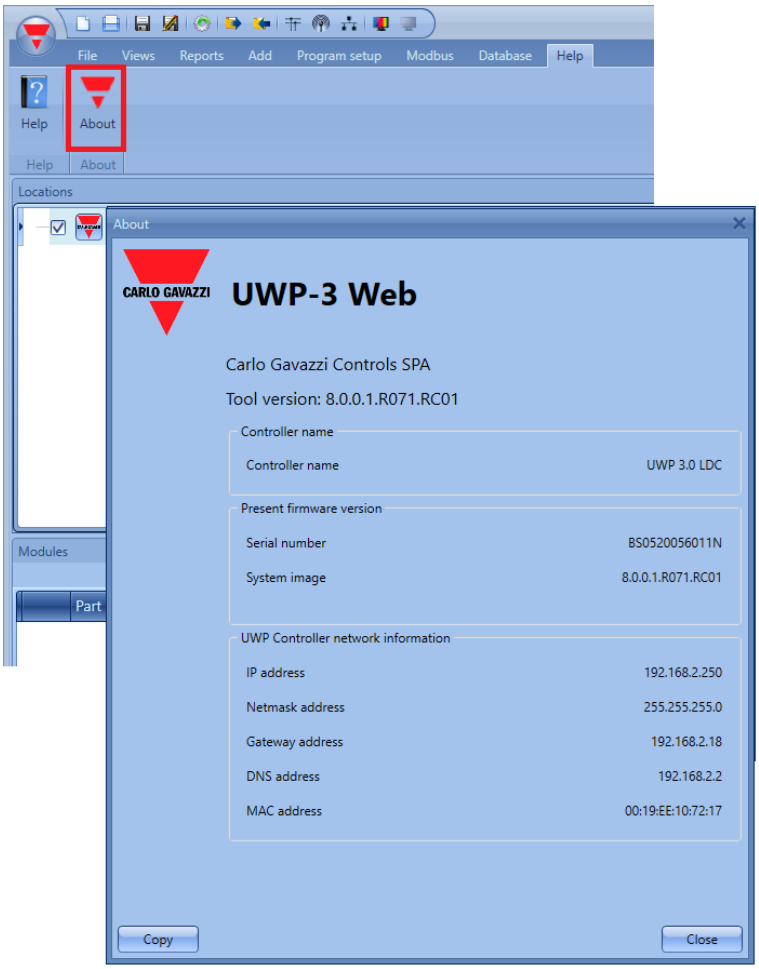
3. Write again the configuration into the UWP 3.0

The user can find the firmware in the relevant folder in the installation directory of the UWP 3.0 Tool.

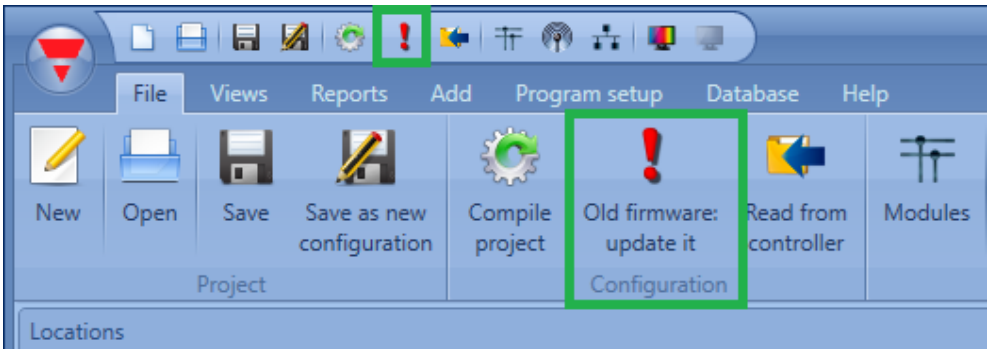


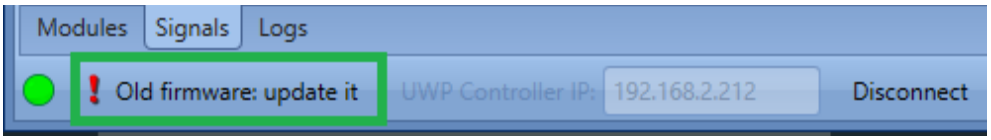
After having selected the correct file, the UWP 3.0 Tool automatically starts updating the UWP 3.0: once this operation is finished, the UWP 3.0 is automatically rebooted.

The current firmware version in the controller can be seen in the *Help* menu (see picture below).



If a new firmware release is available for the UWP 3.0, an icon showing a red exclamation mark is shown. By clicking on this icon, a window for the update immediately opens.



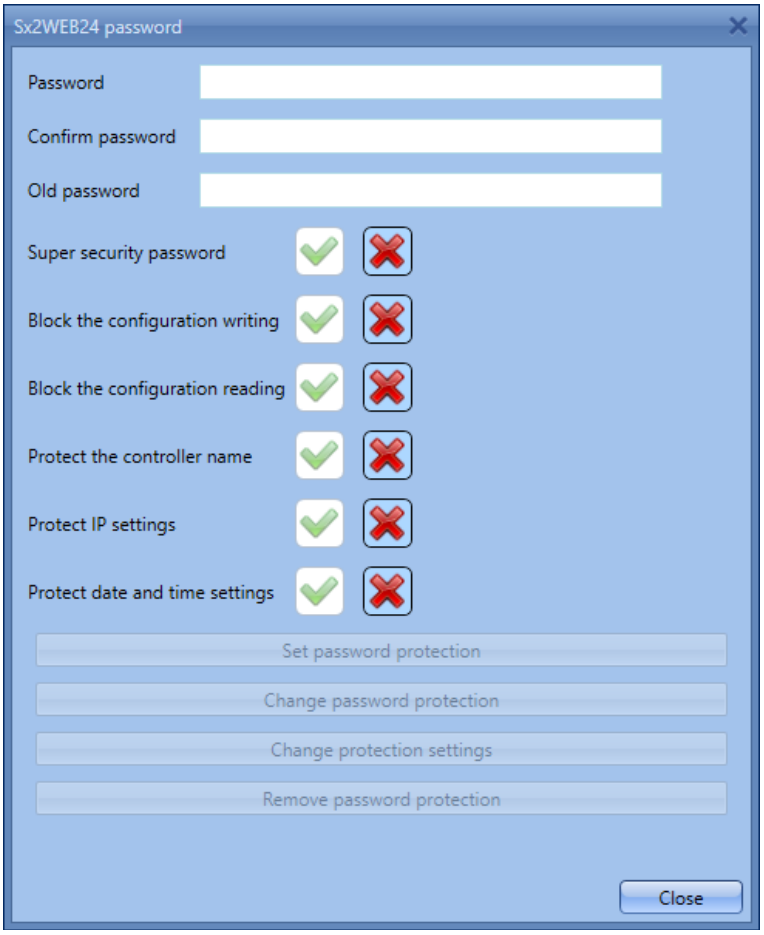


N.B: We strongly recommend updating the UWP 3.0 when new firmware is available.

22.7 How to set the password protection in the Controller

In the program setup, by clicking on the *Password* button, the user opens a window that allows him to manage the protection of the UWP 3.0.

By clicking on the *Password* icon, the UWP 3.0 Tool opens the settings window (see picture below):



Password: in this field the user fills in the new password for the UWP 3.0 protection.

Password confirm: the new password must be repeated.

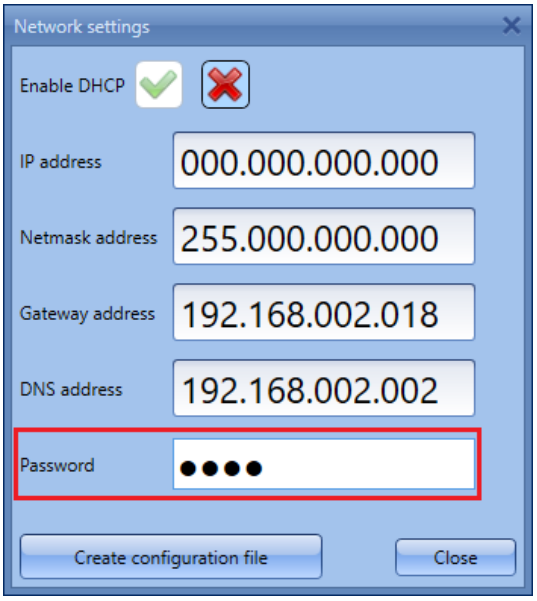
Old password: if the UWP 3.0 has a password and the user needs to change it for a new one, the old password must be entered in this field.

Super security password: when the green V is selected, if the user forgets the password, the only way to unlock the UWP 3.0 is to send it back to the factory.

Block the configuration writing: when the green V is selected, nobody connected to the UWP 3.0 can write the configuration until the password is entered and the UWP 3.0 is unlocked.
Block the configuration reading: when the green V is selected, nobody connected to the UWP 3.0 can read the configuration until the password is entered and the UWP 3.0 is unlocked.

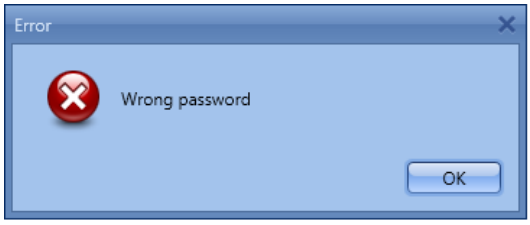
Protect the controller name: when the green V is selected, it is not possible to change the controller name.

Protect IP settings: when the green V is selected, the user has to type the password in the IP settings window. If the password is wrong the IP settings file is created, but when the USB is connected to the UWP 3.0, the IP settings are not changed (see picture below).



Protect date and time settings: when the green V is selected, it is not possible to change the date and time of the controller until the password is entered and the controller is unlocked.

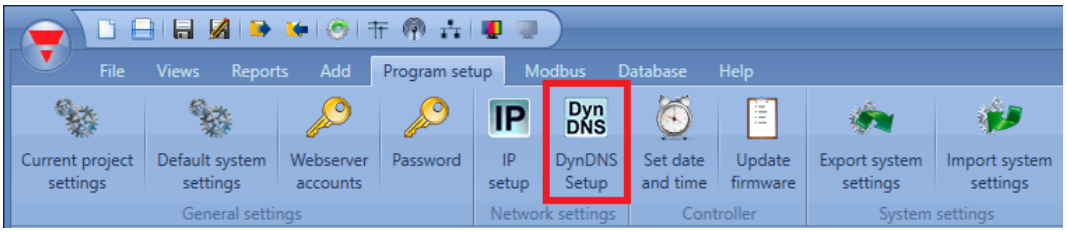
When the controller is protected by a password, the user can connect to the UWP 3.0 and then type the password in the bottom of the UWP 3.0 Tool, then press on the lock icon (shown in red in the picture below).



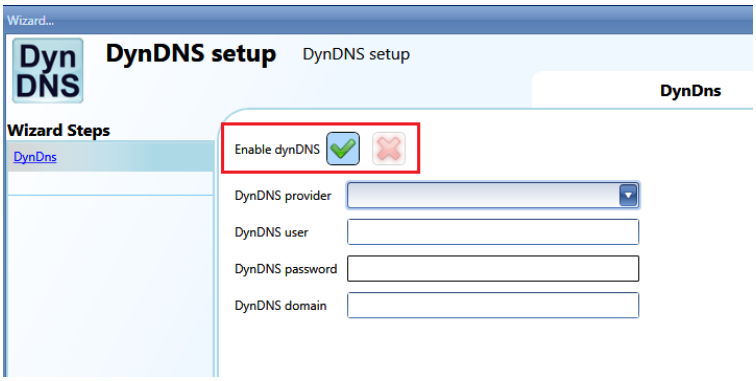
If the typed password is wrong, the tool will show an error message, while if it is correct ,the icon status will be changed to unlocked.

22.8 How to set the DynDNS parameters when using the modem

To enable the management of the dynamic public IP address, the user needs to click on the icon *DynDNS* in the *Program setup* menu: the icon is active only when the UWP 3.0 Tool is connected to a UWP 3.0. These settings will allow access to the UWP 3.0 by using the dynamic IP delivered by the Internet Service Provider from which the SIM card has been bought.

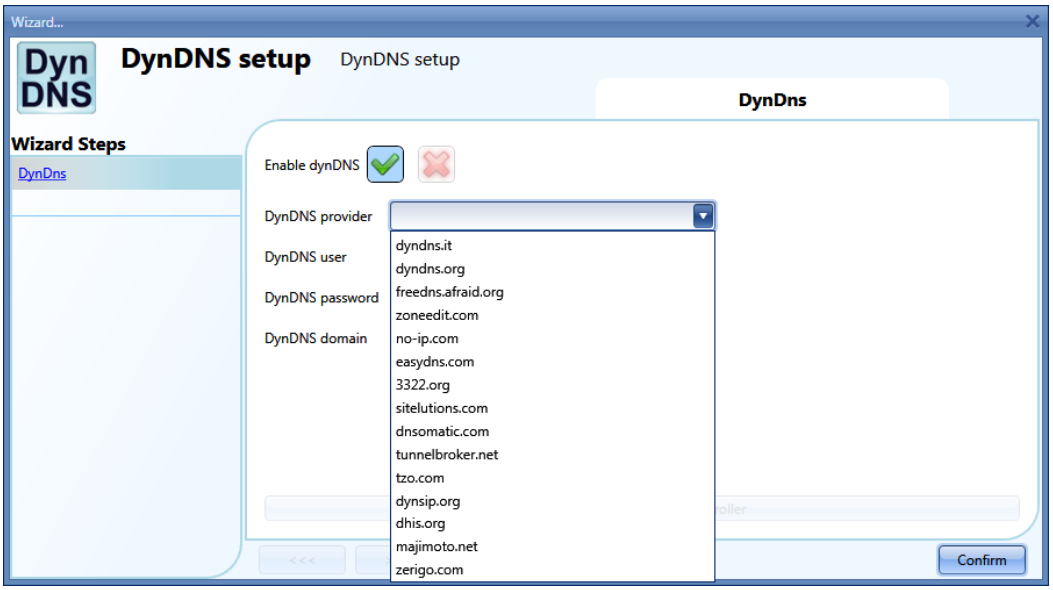


The following window has to be compiled to enable the service in the UWP 3.0:



First the DynDNS service has to be enabled by selecting the green V.

DynDNS provider: in this field the user has to select the host used. Many are available, as shown in the picture below.

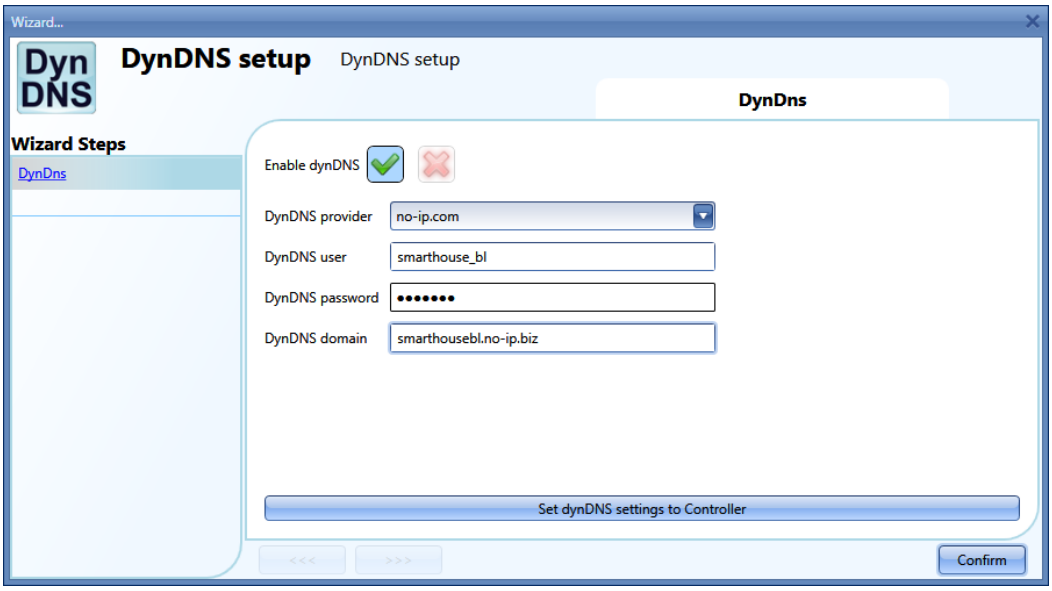


DynDNS user
Type in the user name provided during account registration.

DynDNS password
Type in the password provided during account registration.

DynDNS domain
Type in the domain defined during account registration.

In the picture below, an example is shown where the account has been created with the free host no-ip.com.



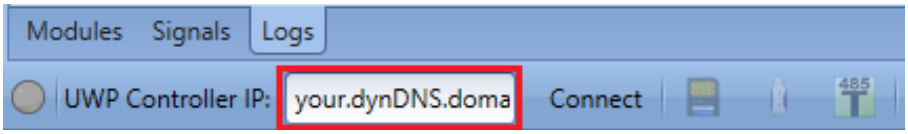
Once the setup is completed, to make it active the user has to send it to the connected UWP 3.0 by clicking on *Set DynDNS settings to Controller*.

N.B.: the DynDNS service works when a modem is connected, so carefully follow the modem setup in the *Communication settings* menu.

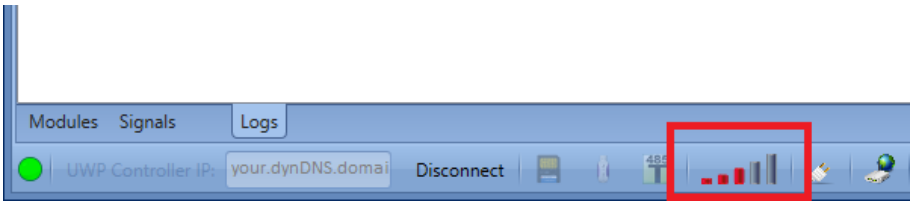
22.8.1 How to connect to the controller by means of a modem

When the user has to connect to the controller for the first time, the modem connection is not available, **so first it has to be configured.** Please refer to the section *How to configure the modem* in this manual for more information.

In the IP address bar, the user can type in the dynamic IP address delivered by the ISP, or they can directly fill in the *DynDNS domain* registered (see *How to set the DynDNS parameters when using the modem*): the second option is preferable since the UWP 3.0 Tool is able to auto-update the dynamic IP address delivered for the DynDNS account each time the modem is reset.

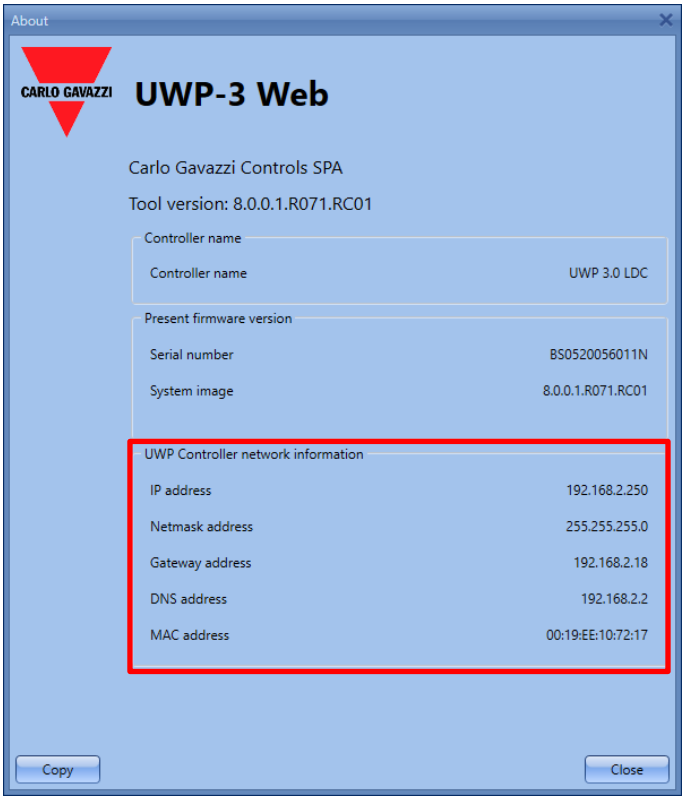


When a connection is established with the modem, the network signal strength will be shown by a set of bars, as shown in the picture below. The higher the number of red bars, the stronger the network signal.



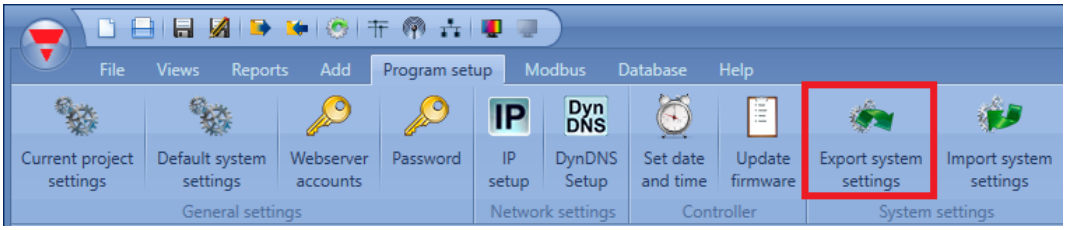
22.8.2 How to see the dynamic IP address

The present dynamic IP address, assigned by the Internet Service Provider, can be seen in the *Help* menu in the *UWP 3.0 network information* fields, as shown in the red rectangle in the picture below.

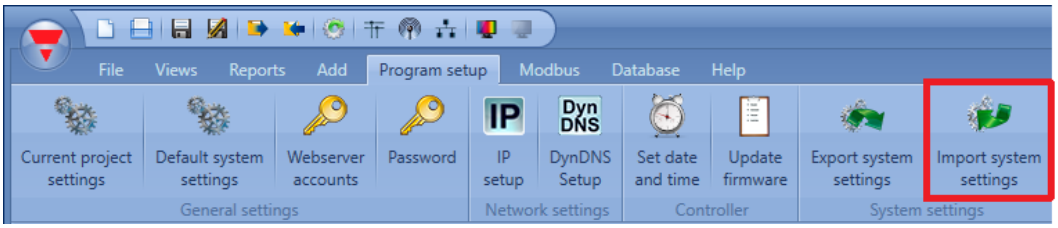


22.9 How to import/export the UWP 3.0 Tool settings

To save the settings defined in the menu *System settings,* a file can be created by clicking on the icon *Export system settings* as shown in the picture below.

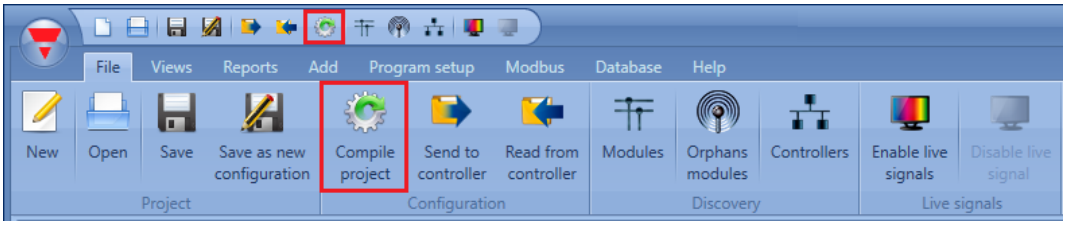


To import the settings, click on the icon *Import system settings* and select the file saved during the “export” operation.



22.10 The Compile project function

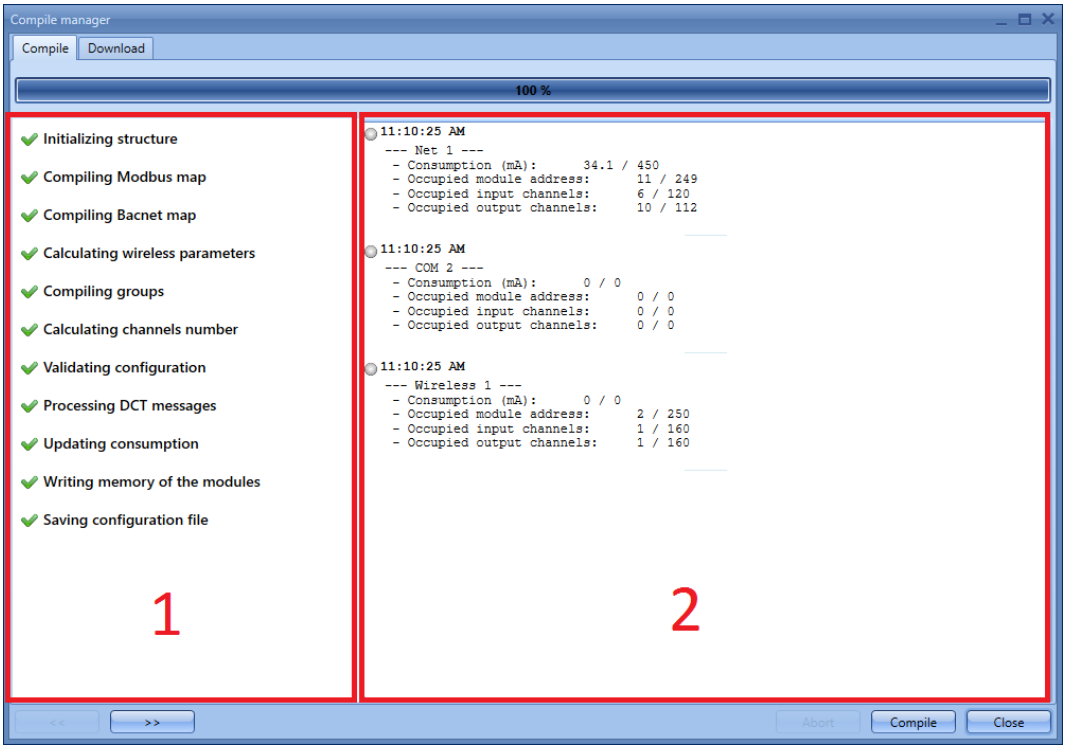
To compile a project, the user has to click on the *Compile project* button in the *File* menu of the UWP 3.0 Tool
(see picture below).



The *Compile project* function is the first step of the *download configuration* process that helps the user to check if a project has been done properly: only if a configuration is compiled successfully, it is eligible to be sent to the UWP 3.0 controller, otherwise by means of the *Compile manager* window the user will be informed concerning the warning and /or error occurred and the actions that must be taken to fix the configuration.

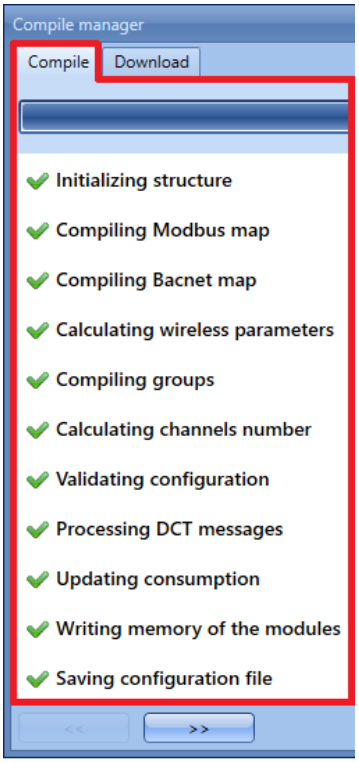
22.10.1 How to compile a project

By clicking on the icon *Compile project* the *Compile manager* window appears, as displayed in the picture below and the system starts to compile the project automatically.

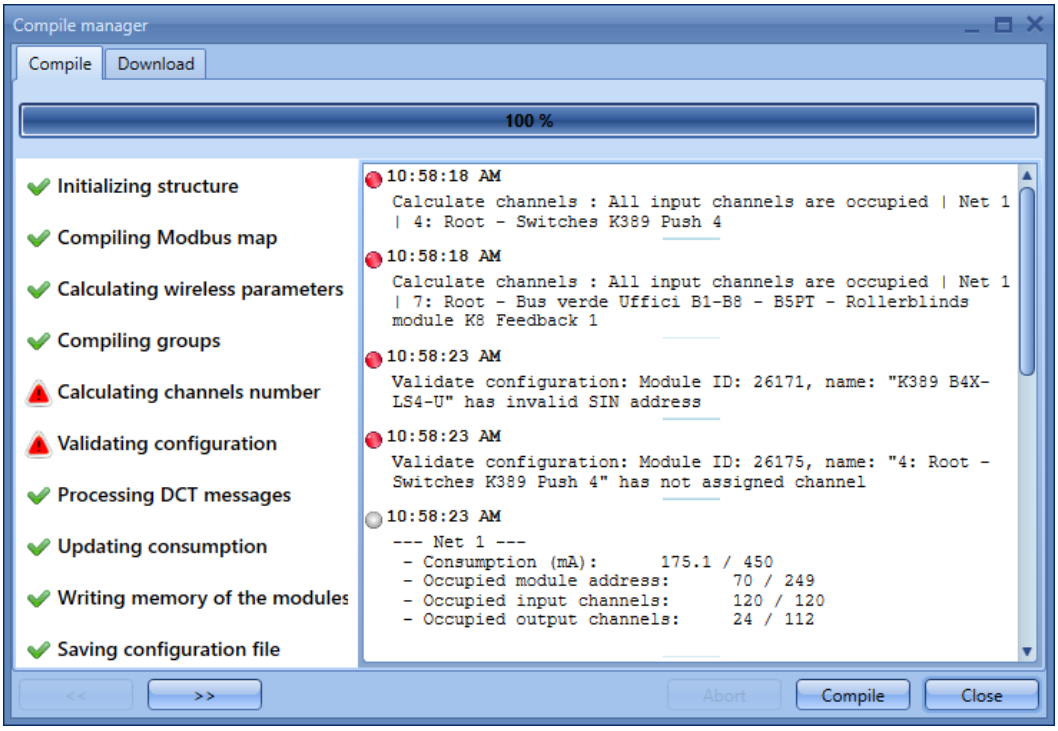


While compiling, the user can check the status of all the tasks on the left side of the *Compile manager* window, as presented in the picture above in the **Area** number **1**.

A configuration that is compiled successfully has all the steps with the green *V* icons, as shown in the example below. The project is eligible to be sent to the UWP 3.0 controller.



On the other way around, if any issue occurs during the *Compile* process, the icon will be shown for any of the tasks that fails. Should this occur, on the right side of the *Compile manager* window a detailed message will guide the user to fix the error.



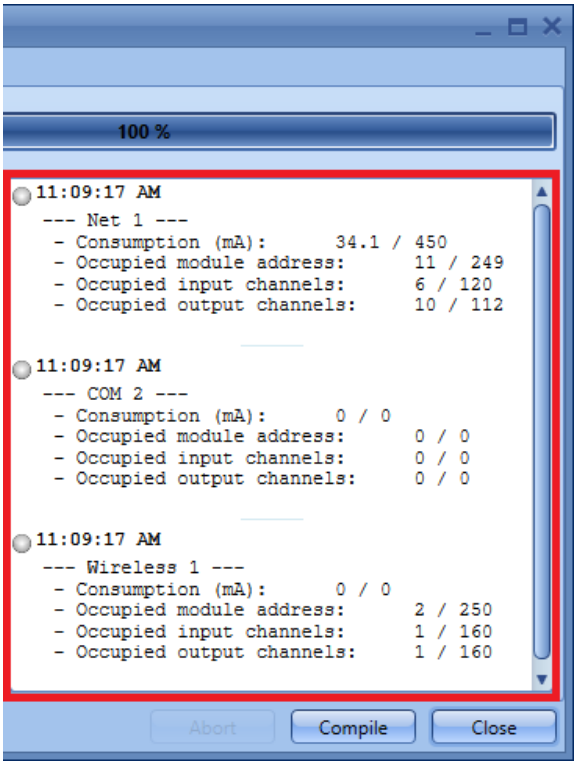
22.10.2 How to fix an error in the configuration

When one or more tasks are not executed successfully in the *Compile manager* window, the following table can be used for trouble shooting.

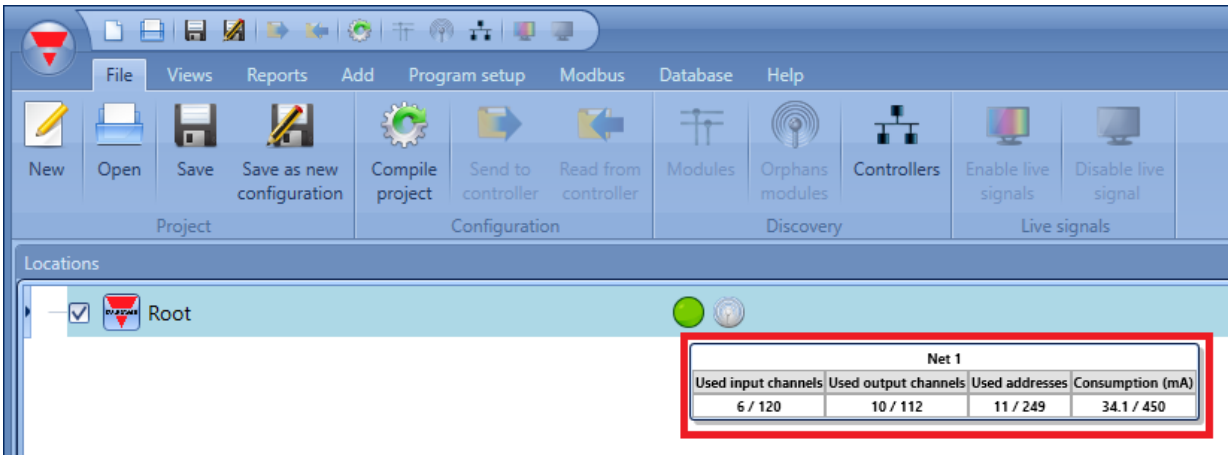
Task Name	Description	Actions
Initializing structure	This is the preliminary step to collect the data	No action is available
Compiling Modbus map	This step checks that the Modbus map is generated according to the current project specifications. The user will be notified about the Modbus addresses that are in conflicts	The user has to fix the conflicts in the Modbus map
Compiling Bacnet map (only for UWP 3.0 Tool)	This step checks that the BACNet map is generated according to the current project specifications. The user will be notified about the BACnet addresses that are in conflicts	The user has to fix the conflicts in the BACnet map
Calculating wireless parameters	This step checks that every wireless module in the project is associated to the proper SH2WBU230N. It will be executed only when at least one SH2WBU230N is present in the project	The user has to fix the routing configuration on the properties of the wireless base unit (SH2WBU230x)
Compiling groups	This step calculates the consumption of the dimmer groups in the project	The user has to contact Carlo Gavazzi for technical support
Calculating channels number	This step calculates the Dupline channels consumption. The user will be notified when the maximum limit of the input/output channels of a bus generator is reached	The module has to be moved to the first free subnet or a new bus generator has to be added
Validating configuration	This step checks if the configuration is eligible to be sent to the controller. The user will be notified about the functions that have been not configured properly, for example if a Dimmable light function is generated without any output dimmer signals	The user has to complete the functions that have not been configured properly
Updating consumption	This step calculates the consumption for all the Smart Dupline networks. The user will be notified when one of the subnet is overloaded	A new Smart Dupline generator (i.e. a SH2MCG24) must be added

22.10.3 How to check the network consumption

On the right side of the *Compile manager* window, it is presented a detailed summary of the resources used by each subnet, to give a clear indication of channels occupation and current consumption (see the example below).



The same summary is also presented by mouse hovering the small round icon representing each network in the *Location* window: a tool tip appears with the updated information.



To update the consumption values shown on the summary, the project must be compiled before.

22.10.4 How to understand when a new compilation is needed

The project must be compiled when anything is changed: a module is moved from one function to another, a module/function/location is added, etc..

The user will be notified when this is required with a small *Compile project* icon appearing in the bottom right side of the UWP 3.0 Tool (close to the *Project name* field). See the picture below.

Once it is completed successfully, by clicking on the *Download* button in the *Compile manager* window, the project is sent to the UWP 3.0 controller.

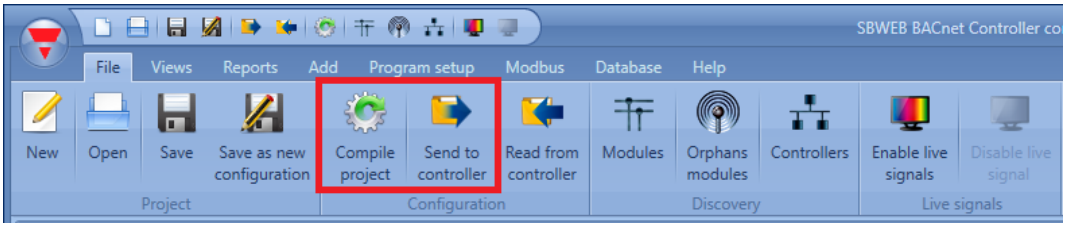
22.11 How to read/write the project in the UWP 3.0

When the UWP 3.0 Tool is connected to the UWP 3.0, the user can write and read projects.
N.B. The *Compile project* function must be executed at least one time before sending a configuration to the UWP 3.0 controller. Please refer to the previous chapter.

There are two different options to send a project to the UWP 3.0:

1) By clicking on *Send to controller* button from *File* menu;

2) By clicking on *Compile project* button from *File* menu;



The differences between *Compile project* and *Send to controller* are*:*

Compile project
- The user can select the modules and the whole network has to be selected manually
- Download process has to be done manually

Send to controller
- The compile and download processes are executed automatically
- It is not possible to select what to program, since the system does it and selects only the differences against the previous sent configuration
- It programs the memory only of new added modules

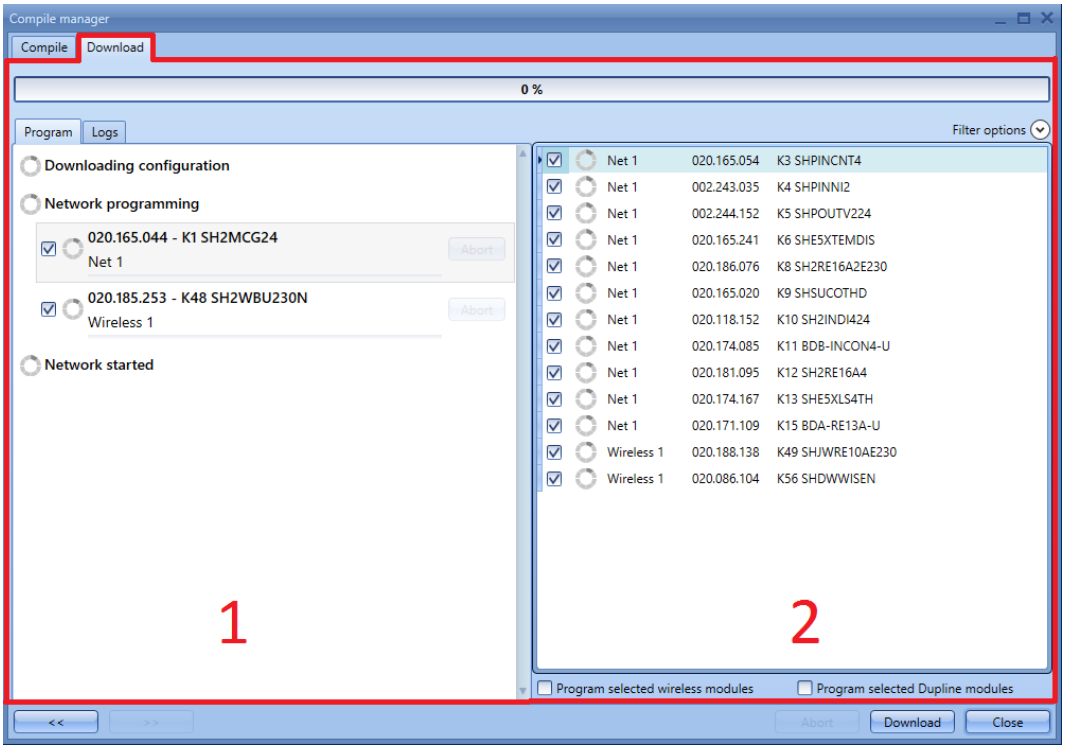
The system checks automatically what are the modules, functions and locations that have to be programmed.

When a new project is sent to the UWP 3.0 for the first time, every object (modules, location, function, Modbus/BACnet maps, parameters, etc…) the system will perform a total network programming: all the subnets and the related modules present in the project will be programmed.

Next time the configuration is modified, for example a new module is added and it is used in a new function, the UWP 3.0 tool will program only this new module. Thanks to the selective download functionality the download operation is faster and furthermore, during the writing operation, the user can continue with the project development.

22.11.1 How to send a configuration from the *Compile manager* window

To send a configuration to the controller, the user can click on the *Compile project* button in the *File* menu: only when the compile process is completed successfully, it is possible to send the project into the UWP 3.0. Clicking on the *Download* tab from the *Compile manager* window, the *Download configuration* window appears, as displayed in the picture below.



On the left side of the *Download configuration* window (**Area1**) all the networks are listed, while on the right side (**Area2**), all the modules are shown. Each module is presented with the following information: network it is connected to, SIN number and the name.

There are two different writing options that can be selected:

Program selected Dupline modules: when this is checked, the software will completely program the memory of the selected smart Dupline modules, otherwise only the addresses of the selected modules will be programmed

Program selected wireless modules: when this is checked, the software will completely program the memory of the selected wireless modules, otherwise only the addresses of the selected modules will be programmed

22.11.2 How to manually select the modules/network to program

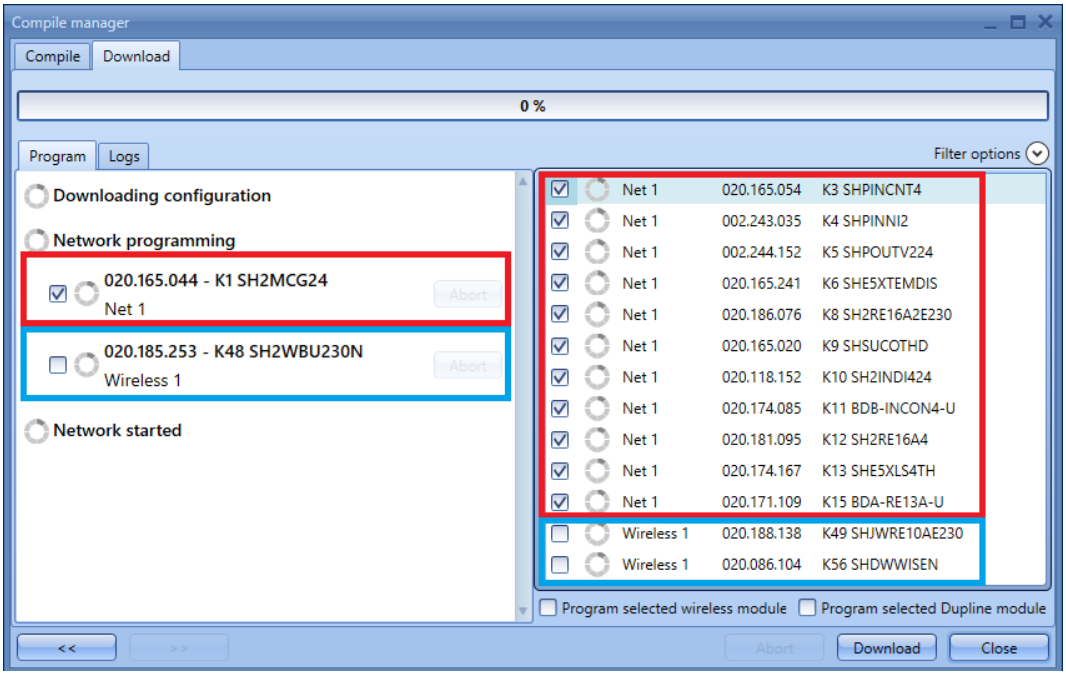
The system, as described in the previous page of this manual, checks automatically what are the modules/networks that have to be programmed.

However, in the *Download* section on the left side of the window, the user can select or change **manually** what modules / networks have to be programmed.

The checked box next to each network indicates the ones that have been automatically selected by the system. The user can select which to program by clicking on the relevant symbol.

- When the check box is checked, at least one module of the network will be programmed;
- When the check box is unchecked, none of the modules of this network will be programmed;

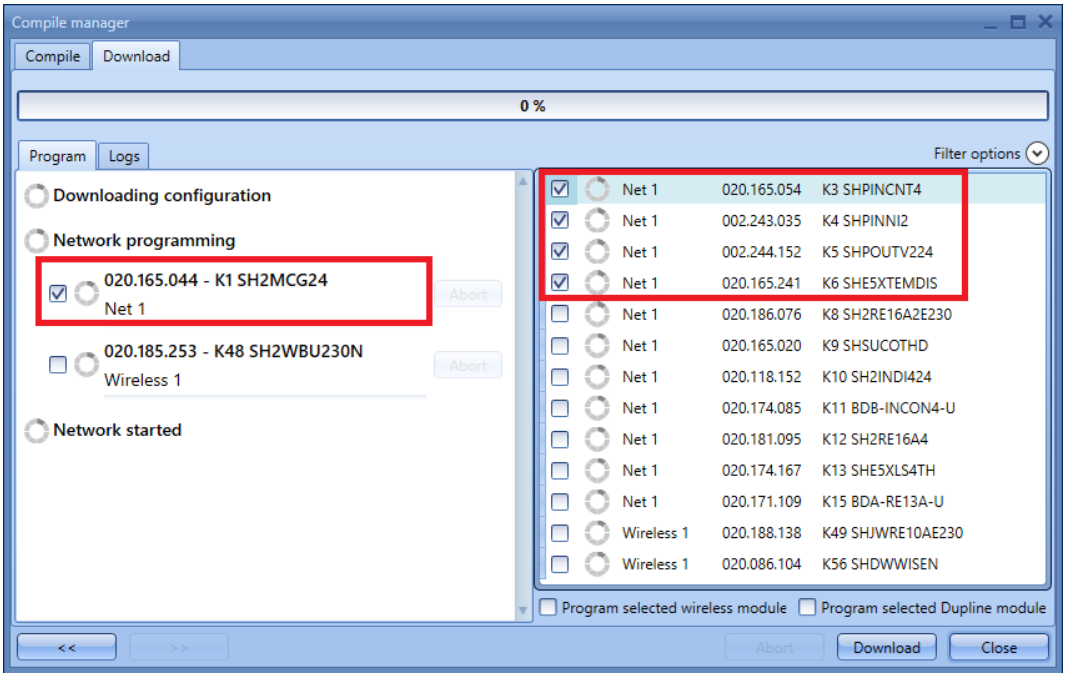
Example
In the example below, the network *Net1* is selected: all the relevant modules are selected as well and they will be programmed (red rectangles); the modules part of the subnet *Wireless1* will not be programed since the network *Wireless1* is unchecked (blue boxes).



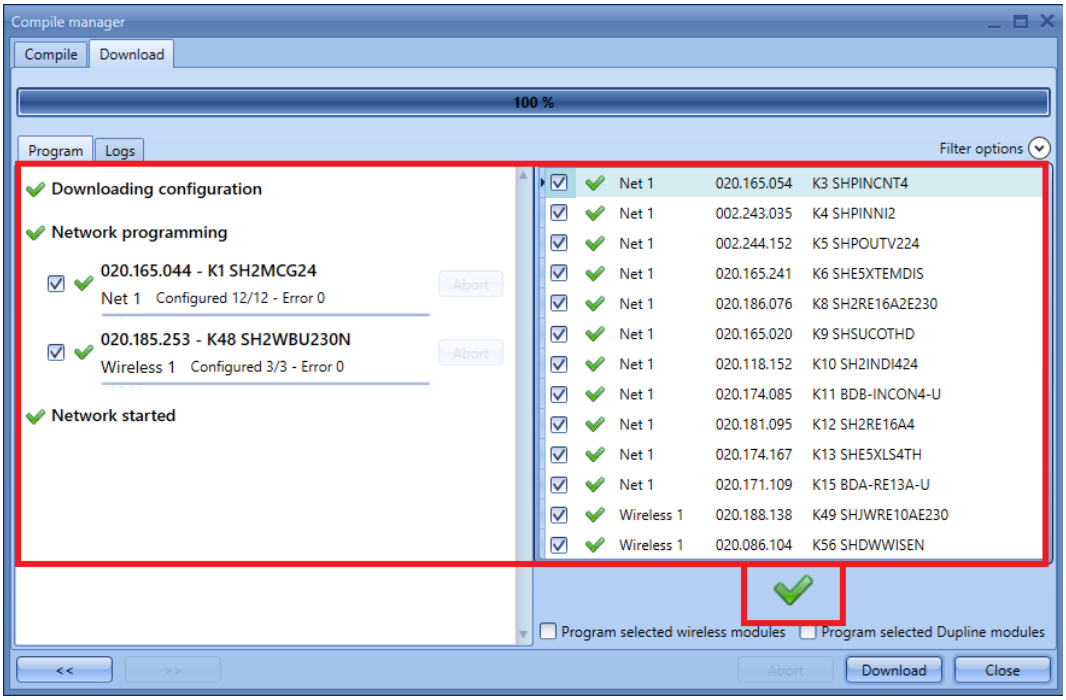
On the right part of the *Download* window, the user can select the modules that have to be programmed:

- When the check box is checked the module will be programmed;
- When the check box is unchecked, the module will not be programmed;

As displayed in the example below, only the first four modules of the newtwork *Net1* have been selected, next time the download operations will be executed, only these modules will be programmed.



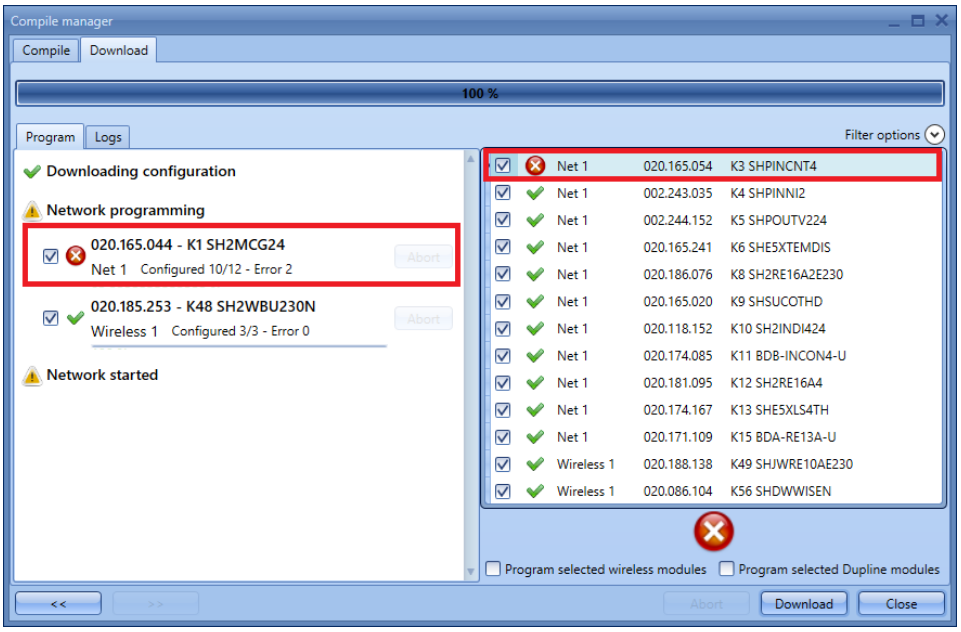
By clicking on the *Download* button, the system starts sending the project into the UWP 3.0.
During this operation, moving or minimizing the *Download* window, it is possible to go on making changes to the configuration.



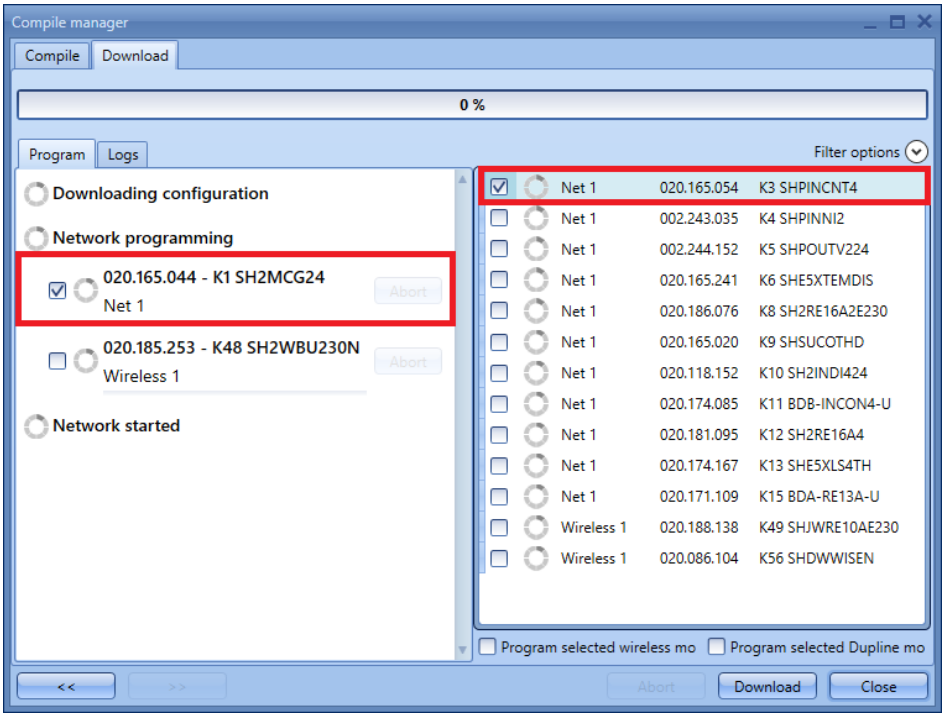
When the upload is finished and all the modules have been programmed correctly (a green V icon is present for each module and network), a green *V* icon is shown in the center of the right side of the window. The user can close the *Download* window by clicking the on the *Close* button.

22.11.3 How to program again the modules in error

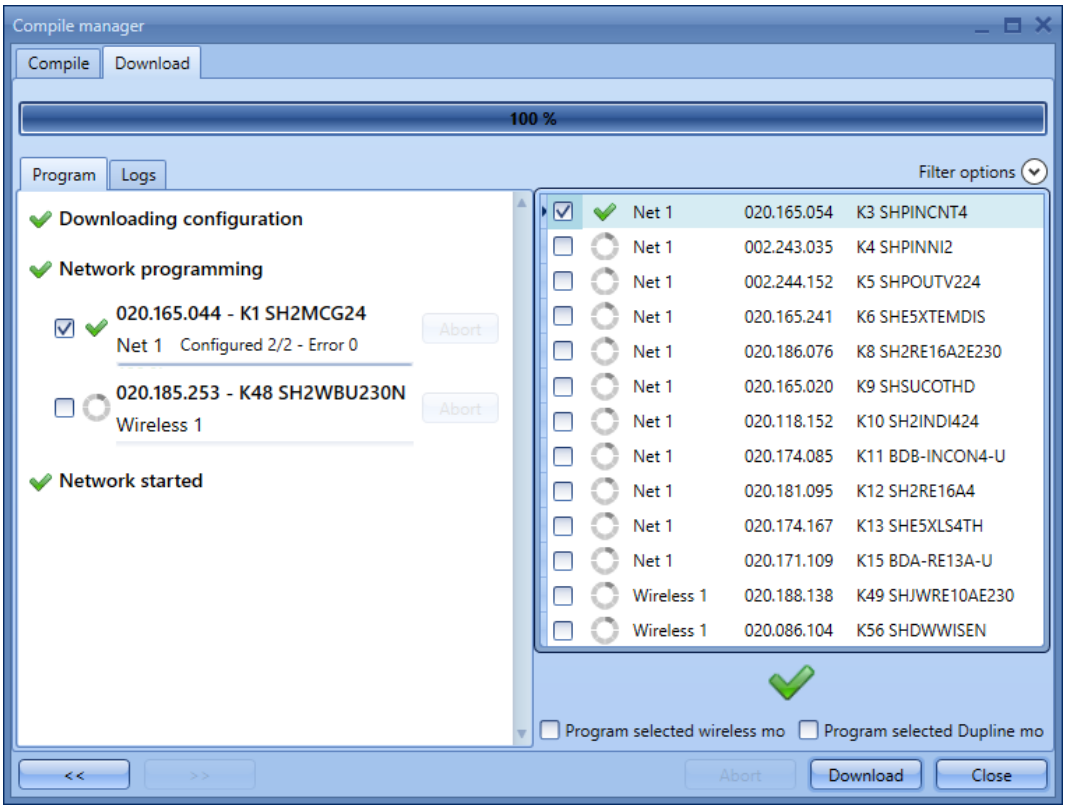
Any module that has not been correctly updated is presented with a small red *X* icon close to the module and to the relevant subnet. In the example below, the module *K3* of the network *Net1* has not been programmed.



The selective download functionality detects automatically what are the modules that have not been programmed, so they will be selected automatically next time the *Compile project* function is executed. As displayed in the picture below, the faulty module *K3* is ready to be re-programmed.

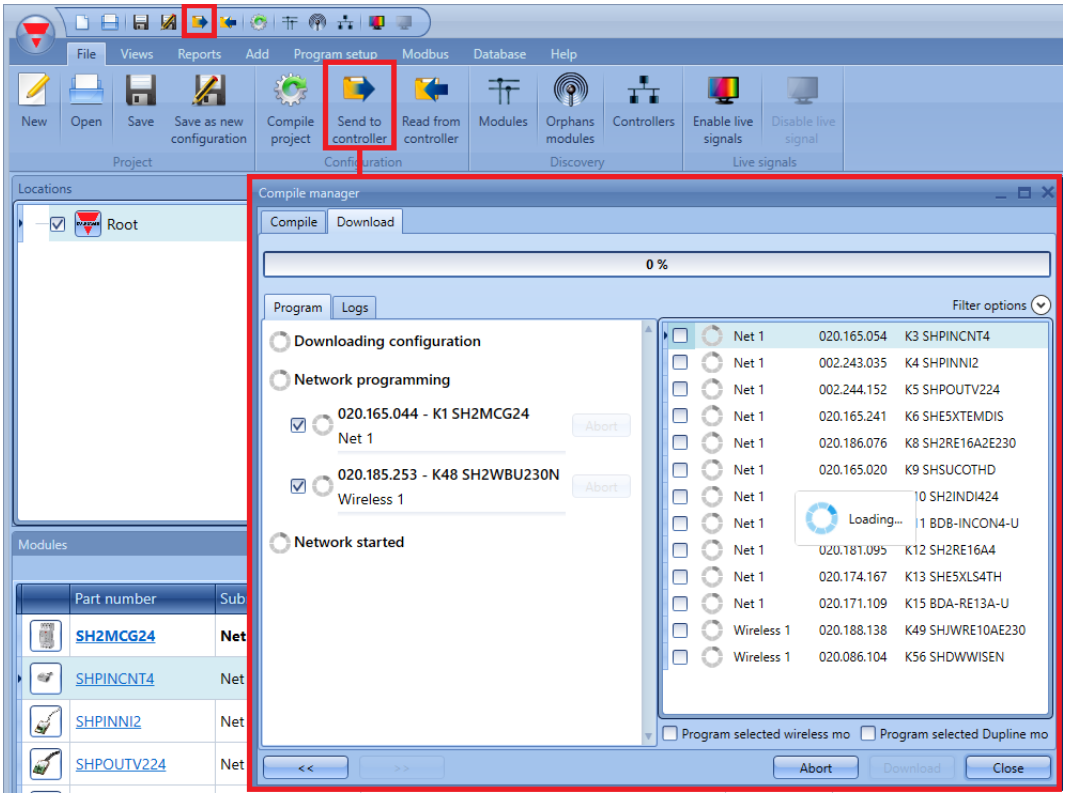


Clicking on the *Download* button, only this module will be programmed (see the picture below).



22.11.4 How to write a project with *Send to controller* function

Unlike the writing mode of the *Compile project* button, by clicking on the icon *Send to controller* the download operation starts immediately after compiling and saving it.



In this case, the system selects the objects to be programmed and the user cannot change anything. Only the memory of new modules will be programmed, while as for the other already existing modules, only the address will be programmed.
So if a setting of the module is changed, for example the colour of the feedback LED in the light switches, it is better to follow these steps:

1) Click on *Compile project*
2) Select the modules that have to be programmed
3) Check the field *Program selected Dupline modules or Program selected wireless modules*
4) Execute the download.

To summarise, the automatic download is fast since it programs only new functions/objects, but regarding the modules that are not new, no changes are done.

22.11.5 How to manage the writing operations with a modem connection

N.B: We strongly recommend carrying out any writing operations only when the UWP 3.0 Tool is connected to the UWP 3.0 via Ethernet or mini-USB cable.

Despite this, the user may write a configuration via modem, after waiting up to 5 minutes before reconnecting to the controller after any writing operations.

22.12 Programming the wireless modules

When the user starts writing a project with wireless modules into a UWP 3.0, the programming may take some minutes: this time is due to background actions of the wireless modules.
The modules stay in a sleeping mode and they wake up every few minutes to communicate their status to the wireless base unit: only when they wake up, the software can program them. The sleeping time of the relay module is one minute, the time for the pushbutton is three minutes.
It is possible to wake up a pushbutton without waiting by pressing it.