Software and Manuals © 1992, 2013 Adroit Technologies

Information in this document is subject to change without notice and does not in any way commit Adroit Technologies. The software described in this document is supplied under a License Agreement. The software may only be copied in accordance with the agreement. No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, recording, or information storage and retrieval systems for any purpose other than the licensee’s own use, without the express permission of Adroit Technologies.

Adroit is a trademark of Adroit Technologies, RSA
Microsoft is a registered trademark of Microsoft Corporation, USA
Windows is a trademark of Microsoft Corporation, USA
Table of Content

1. Purpose of this document ................................................................. 2

2. Prerequisites needed to work through this Quick Start Guide .......... 2

3. Theory ............................................................................................ 2
   3.1. General Overview ................................................................. 2
   3.2. Front End or Plant Floor Devices (FED's/PLC's/RTU's etc) .... 3
   3.3. Device Driver (PLC Communications) .................................. 3
   3.4. Agent Server (I/O Server) .................................................... 4

4. Needed configuration steps ............................................................. 5
   4.1. Configure the Agent Server .................................................. 5
   4.2. Configure your PLC ............................................................ 7

5. Engineering .................................................................................... 9
   5.1. Open your MAPS designer ................................................... 9
   5.2. Discover the MAPS Designer and Create a new project ....... 11
   5.3. Create a display .................................................................... 12
   5.4. Create a tag .......................................................................... 12
   5.5. Create a graphic form and display the value of a tag .......... 16
   5.6. Write a value to a tag ........................................................... 22
   5.7. Log and trend a tag ............................................................... 23
   5.8. Alarm a tag ......................................................................... 27
   5.9. Working with vector graphics and behaviours (animations) .. 29
   5.10. Create a navigation structure ............................................. 33
   5.11. Working with Security and profiles .................................... 36

6. Running your project ..................................................................... 40
1. Purpose of this document

The purpose of this document is to give you a quick start into the MAPS SCADA software. After you’ve worked through this document you should understand the basics of MAPS SCADA. The reader of this document requires basic knowledge around:

- Mitsubishi Electric’s Programmable Logic Controllers (PLC)
- Basic notions in Supervisory Control and Data Acquisition Systems (SCADA)
- Basic notions around Ethernet networks (configuring the IP address and the subnet of his PC network adapter, and understand what it implies)

The purpose of this document is NOT to give the reader an introduction to PLC and SCADA systems. It does NOT explain how to program the Mitsubishi Electric PLC platforms.

2. Prerequisites needed to work through this Quick Start Guide.

To work through this Quick Start Guide you’ll need:

- 2 to 3 hours time.
- To install MAPS. Run the Express Installation from the DVD and follow the instruction.
- An Ethernet cable to connect your PLC to your computer
- If you’re an experienced Mitsubishi Electric PLC user (you know how to configure the MC protocol) you’ll need:
  - the Mitsubishi GX IEC Developer or GX Works 2 to configure your PLC.
  - A Mitsubishi Q Series PLC with an Ethernet port or an external Ethernet card.
- You’re new to Mitsubishi:
  - You’ll need GX IEC Developer
  - A Q Series CPU with Ethernet on board. (Ex. Q03UDE)
  - A USB cable to program the CPU

3. Theory

3.1. General Overview

In order to understand MAPS SCADA, it is important to understand the basic architecture of the product. The most important aspect is to understand that the SCADA consists of two main applications:

- The Agent Server (I/O Server) – Agent (Tag) configuration, data acquisition, logging, alarming, data manipulation; and

- The User Interface (UI) application – data acquisition and serving of data to graphic forms from Agent Server (I/O Server) and other sources of data (ex: databases), graphics design and graphic form storage
  - Graphics Server – data acquisition and Graphic Forms server
  - Designer – application to build Graphic Forms.
  - Operator – application to view Graphic Forms.
The function of a SCADA/HMI product such as MAPS SCADA is to communicate to a Front End Device/Field or Plant Floor Devices using a protocol driver. Drivers then access real-time data and feed it into an Agent (Tag) of a certain type in the MAPS SCADA Agent Server.

The Agent Server, which is the real-time database, handles the real-time values and if configured, will log and/or alarm the data. This data can also be displayed within the User Interface(s).

We will deal with each of these aspects separately here:

3.2. Front End or Plant Floor Devices (FED’s/PLC’s/RTU’s etc)

Front End Device hardware is designed to measure real-time variables and put these values into its own real-time database. Most modern devices support communications and can be interfaced to using a known or defined protocol. The MAPS SCADA supports over 100 different protocol drivers. A complete list of available protocols is available on the MAPS CD or can be downloaded from the Adroit Technologies website (www.adroit.co.za).

Examples of Front End Devices include:

- Programmable Logic Controllers (Q-Series and L-Series PLC)
- Remote Telemetry Units (RTUs)
- Energy Meters
- Dataloggers

3.3. Device Driver (PLC Communications)

The protocol driver is a piece of software (DLL) that essentially plugs into the MAPS SCADA Agent Server. The driver knows how to communicate to the FED that you wish to get data from. Hence there are many different protocol drivers available. There’re well over 100 different drivers developed for MAPS SCADA. We also support OPC, the industry standard
for Process Data adopted by all popular device vendors providing effortless connecting. A list of the available drivers is available on our website.

**IMPORTANT:** All drivers are not automatically installed when installing MAPS SCADA for the first time. You need to know which device you wish to communicate to, its required protocol driver and then go through a separate installation process after completely installing the MAPS SCADA product onto your PC. This process will be covered later.

All drivers are available on the installation CD or can be downloaded from our website.

### 3.4. Agent Server (I/O Server)

The Agent Server (AS) is the real-time database or engine of the MAPS SCADA. This is where all the Agents (Tags) are configured, where all scanning using the protocol drivers happens along with all the logging and alarming. When you run the Agent Server application, it loads the following:

- All the product base files (DLLs)
- The database as selected in Setup (.WGP)
- Drivers loaded and configured in the configuration

![Figure 2: The Agent Server window on Start-up](image)

You can see that it lists all the DLLs as it loads them, and at the bottom it indicates which .WGP file it loads. It also shows you what licensing has been configured.

**NOTE:** MAPS SCADA is licensed using a security dongle known as a HASP. This is a USB device that plugs into the Server machine and is configured according to the order received. MAPS SCADA is licensed in two dimensions: the first is the number of Scan Points to be
communicated through the drivers, and the second is the number of Remote Connections that can concurrently connect to the Agent Server.

The options available on scan points are 30, 75, 150, 300, 750, 1500, 2500, 5000, 25,000 and system limited. There is no limitation on the number of Remote Connections.

A Scan Point is essentially one data point (value) in the front end device that you are communicating to. There are some exceptions namely when you use special internal agents such as OLEDB, Scripting and SNMP which each consume a scan point per Agent that is used. Internal tags such as log tags and alarm tags do not consume scan points.

**TIP**: We recommend extensive project planning and consultation with your MAPS SCADA sales representative to ensure accurate licensing. A comprehensive licensing document is available to you on request.

### 4. Needed configuration steps

#### 4.1. Configure the Agent Server

You need to configure your Adroit Agent Server in order to be able to communicate to a Mitsubishi PLC.

We’ll now install the Mitsubishi Electric Q Series Ethernet driver.

Goto “Start” or “All Programs” -> “Adroit 7” -> “Adroit Setup”

The following window should be displayed.

![Figure 3: Adroit Setup window](image)

Click the tab “Drivers”. You should know see the following view.
Right click the “Mitsubishi Q/QnA Series Ethernet driver” and select “Add Device”.

You’ll be prompted for a name. We’ll choose “PLC”.

The next window will prompt you to configure the device called PLC. You should see the following window.

In this window we’ll set up the IP address and the port number of the PLC we want to connect to.

I’ve set the IP address to 192.168.3.39 and left the port number to HEX 500 (DEC 1280), which is the standard port number for the MC protocol that this driver is using.

**IMPORTANT:**
The IP address of your PLC needs to be in the same subnet as your network adapter’s IP address.
Example: 192.168.3.10 for your PC and 192.168.3.39 for your PLC.

**NOTE:**
The Mitsubishi onboard Ethernet port comes with a standard IP address set to IP 192.168.3.39 (which can be changed any time through the PLC engineering tool)
You should now see the following window. The symbol is yellow because the device is not scanning the PLC yet. But we’ll change that later on in this Guide.

![Figure 6: The PLC has been set up.]

### 4.2. Configure your PLC

If you know how to setup the MC Protocol you can skip the sub chapter 4.2. However don’t forget to reset your CPU after you’ve downloaded the parameters to you CPU.

Now you’ll need to configure your PLC. We’ll demonstrate it using GX IEC Developer and a Q-CPU with onboard Ethernet port. However:

- the procedure using GXWorks 2 is very similar.
- The configuration of the Ethernet module is described in the drivers help file that can be find by clicking “Help” in the window showed in “Figure 5: Configuration window for the “PLC” device”

Start GX IEC Developer and create a new project. Select the CPU you’ll be working with and create the project. From the project tree left double click the PLC item (also called PLC parameters) as shown below.

![Figure 7: Select PLC parameters]

In the window to will open go to the very right (last) tab called “Built-in Ethernet port” You should see the following window.
Enter the IP address that you’ve entered in the window showed under Figure 5: Configuration window for the "PLC" device then select the “Binary code” radio button, check “Enable online Change (FTP,MC protocol)” and click the button “Open settings”.

The following window will open.

Set the first line as shown in Figure 9: Built-in Ethernet port open settings.

Click “End” and then “End” again.

Now compile your project (Menu “Project” -> “Build All”). Now we need to establish a communication to the PLC through GX IEC Developer. Therefore go to the menu “Online” -> “Transfer Setup” -> “Ports”. You should see the following window.
Click the button “PLC direct coupled setting”. A window will open. Make sure USB is selected and click OK.

Hit the button “Connection test”. The following message should be displayed to inform you that the PLC connection was successful.

Click “OK” and then “OK”.

We’ll now download the PLC project.

Click the download button from the menu bar as shown in the following figure.

Confirm that you want to stop the plc and download the project and after a certain time you should get the feedback that the project has been downloaded successfully.

Now reset your PLC by opening the cover from the left side. Turn the dip switch to the left for 5 seconds and release it and switch it back to the RUN position.

Your PLC is now ready to communicate with your SCADA project.

5. Engineering

5.1. Open your MAPS designer

First start your “Agent Server” and your “MAPS Server”. The sequence in which you do this doesn’t matter.

If you have a MAPS license installed on your computer start the “Adroit Agent Server”

To Start the “Adroit Agent Server” goto “Start” or \rightarrow “All Programs” \rightarrow “Adroit 7” \rightarrow “Adroit Agent Server”
If you don’t have a license start the “Adroit Agent Server (time licensed)”
To Start the “Adroit Agent Server (time licensed)” goto “Start” or “�数” -> “All Programs” -> “Adroit 7” -> “Adroit Agent Server (time licensed)”

You can start the MAPS Server as an application or as a service. The difference between an application and a service is that the application needs a user to be logged on to run and the service can be started by the system user and therefore don’t need a user to be logged in.

When you start the MAPS Server as an application you’ll see the following window:

![Figure 13: MAPS started as an application](image1)

When you start MAPS as a service you’ll see the MAPS Server icon in the system tray

![Figure 14: MAPS Server service in the system tray](image2)

The MAPS Server icon has the following status:
- for MAPS Server service stopped and NOT executed.
- for MAPS Server service starting
- for MAPS Server service started and executed

**IMPORTANT:**
- The default install of MAPS will register the MAPS Server Service and start the service on windows startup.
- MAPS Server service and the MAPS Server application can’t be executed at the same time.
- We’ll use and start the MAPS Server service in this quick start guide. To start the MAPS Server check if the MAPS Server icon in the system tray is green (�� ). If yes your MAPS Server is running. If it’s red ( 红 ) right click the icon (🪐 ) and choose “start service”.

**HINT:**
- Should the MAPS Server service be disabled/not starting then start the MAPS Server as an application (goto “Start” or “�数” -> “All Programs” -> “MAPS” -> “MAPS Server”).
Now that all server programs have been started we can start the MAPS Designer

Goto “Start” or “All Programs” -> “MAPS” -> “MAPS Designer”

You’ll be prompted to login to the MAPS designing environment. Use therefore your windows credentials that you used to install MAPS. Then click “Finish”

**NOTE:**
The user account needs local administrator privileges.

You’re now logged into the MAPS Designer and ready to work with MAPS.

### 5.2. Discover the MAPS Designer and Create a new project

Once in the Designer you should see the following window.

![Figure 15: The MAPS Designer](image)

The MAPS Designer view can be customized. You can dock windows to the different panes of the master window. All you need to do is to click the title of the window and while you leave the mouse button down you can move the window around and dock it to the top, the bottom, the right or the centre. You can also drop it into an existing window and create a tabbed view where every tab represents a window. Feel free to play around with it.

Once you finished playing reset the layout. To reset the layout go to the ribbon “view” and select “Reset to Default Layout”.

On the left side of your environment you’ll see a docked window called MAPS. It has 3 tabs at the button. Click the one called “Enterprise Manager”.

You should see the following.
Right Click “Projects” (blue underlined in the Figure 16: Enterprise Manager) and select “Add a Project”. Call your project “myPlant”.

In the tree view of the Enterprise Manager you’ll see under Projects a project called “myPlant”.

Click the “+” next to “myPlant”. You'll see that your Project doesn't contain anything but the “Translation” item. We’re about to change that now.

5.3. Create a display

A graphic form is a display that will contain the visual representation. You can create several graphic forms and call them from an existing graphic form. But we'll deal with that later on.

From the Enterprise Manager right click your recently created project “myPlant” and select create Graphic Form. Call the graphic form “StartupView”.

5.4. Create a tag

We’ll now create a tag. In MAPS SCADA however we don't directly create tags. We use something called an Agent. The agent is managed by the Adroit Agent Server (the IO Server) that we started at the beginning. The Agent provides an extra layer of functionality that can and should be configured. It can be compared to a function block:

- You link a value from the PLC to the Agent
- The Agent will process this value to its outputs with respect to the parameters you’ve entered.
- The outputs and inputs of the agent are available in the SCADA application.
- The inputs and outputs of the agent are called slots or tags. Whenever we mention tags or slots we refer to these inputs. An Agent doesn’t have a value. It provides value through the slots/tags

Let me give you an example based on the analog agent:

The analog agent has a slot/tag called “rawValue”. We’ll link the address “D0” of the Q-PLC to that slot/tag (we’ll show how to do that later). If D0 has not been scaled in the PLC you could tell the agent to do that for you. The Analog Agent would then output the scaled value to the slot/tag “value” of the Agent.

You could also define HighHigh, High, Low, LowLow and rate of change limits. The Agent would then trigger the slots/tags of type bit called HighHigh, High, Low, LowLow and rate of change if
the respective limit has been reached. These bits can easily be linked into the Alarm Agent and trigger an alarm in the integrated Alarm viewer object (We’ll show that later on as well).

We’ll now create an Analog Agent called MOTORPUMPFLOW and scan it against the address “D0” in the PLC.

Go to your MAPS Designer. Select the “Enterprise Manager” in the left window if not already selected. Under “Datasources” (almost at the top of the tree) right click “Adroit” and select “Open Configurator”. You should see the following window.

![Adroit Agent Configurator](image)

In that window select the “Type” “Analog” from the pull down menu. Then under “Name” at the top enter “MOTORPUMPFLOW” you can also enter a description if you want. Then click “Add” second button from the top on the right.

You should now see the following.

![MOTORPUMPFLOW Analog Agent](image)

While MOTORPUMPFLOW is highlighted in blue click “Edit”. You’ll get to the configuration window of the Agent where we can for example set the HighHigh, High, Low, LowLow and rate of change limits. Set the parameters as shown in the window below.
Figure 19: Configure the Analog Agent

We'll now explain the meaning of the configuration (not all just the part we're using. For more information please refer to the integrated help file and type in the search dialog "Analog Agent"):

- **Device Span** is the possible value range of the PLC value available under the address D0 (or the possible range of the slot/tag "rawValue" of the Analog Agent to be more precise)
- **Engineering Span** is the possible range of the scaled value that the Analog Agent will provide (that will be the output to the slot/tag "value" of the Analog Agent).
- In our case putting the device span to 0 – 1.000 and the engineering span to 0 – 10.000 means that if D0 is 10 then the scaled value will be 100. It also means that D0 can't be less than 0 or greater than 1.000 AND that the scaled value can be less than 0 or greater than 10.000.
- **Under Alarms** we've set the High-High, High, Low and Low-Low. Once the scaled value (engineering span) has reached the respective limit the respective bit in the agent will be set. As soon as the scaled value is within the respective limit the respective bit will be reset.
- In our case once the scaled value reaches 8.000 the high bit of the agent will be set. Once the scaled value is less than 8.000 the high bit will be reset.
- **Under units** we've entered L/h.

Now click "OK".

We'll now scan the address D0 of the Mitsubishi Q-PLC and write this value periodically to the "rawValue" slot/tag of our Analog Agent. This is what we call "scan" a PLC tag to an agent.

Back in the main configurator window (as shown in Figure 18: MOTORPUMPFLOW Analog Agent) click the button "Scan" (5th from the bottom on the right side)

You should see the following window.
In the Tag scanning configuration window you’ll see that the Analog Agent created has been pre-selected. The slot to which the PLC tag should be scanned to, has been pre-selected as well (“rawValue”).

We’ll first of all select the Device which is our PLC. This could also be an OPC Server or a third party PLC.

As we set up the Adroit Agent Server (see figure Figure 6: The PLC has been set up.) we called the PLC device “PLC”. We’ll now choose that device from the pull down menu in the Tag Scanning Configurator (Figure 20: Scan a PLC tag to an agent). If you’re prompted to create it automatically just click “Yes”.

Configure the window as shown in the window below.

Then click the button “Scan”. Now you see the following window
The Agent MOTORPUMPFLOW is now on scan.

We’ll now explain the meaning of the parameters we just set:

- **Under Address** we entered the address of the tag we wanted to read/write from/to the PLC. (Note in Figure 22 under address you’ll see D1 and not D0. That’s because once you’ve clicked scan the address gets incremented by 1. If you look in the field below you’ll see a tag that is scan to D0. That’s our tag)
- **Under Scan rate (ms)** we defined the cycle the Agent Server will actually scan the PLC for that address. In our case the value from the PLC will be read and written every 500ms.
- **Output enabled** is an option that explicitly allows the Analog Agent to write values back to the PLC. If you want a value to be read only leave that box unchecked.

We’ve now created an Analog Agent that reads the address D0 in the Q-PLC. The value read will be scaled with a factor 10. We’ve also set up some limits and a unit.

Click OK and close the configurator (cross in the upper right corner). You’ll be prompted to save the changes. Click “Yes”.

Finally refresh your Adroit Datasource in Enterprise Manager tree. You should do that every time you create a new tag to make it visible in the enterprise manager.

To refresh the Adroit datasource right click “Adroit” under “Datasources” of your Enterprise Manager and then select “refresh”. That’s it.

Now we’ll play with the Agent and its tags/slots.

### 5.5. Create a graphic form and display the value of a tag

While in the MAPS Designer go to the “Enterprise Manager”. In the Enterprise Manager tree go to the project “myPlant” (see figure below).
Right click the Project “myPlant”. Select “Create Graphic Form”. You'll see under “myPlant” a new item called “Graphic Form”. Change its name to “Main”. To do so select the Graphic Form in the tree and press F2. Then enter the name.

You now have a graphic form called “Main” in your project called “myPlant”. However your Enterprise Manager view has disappeared. You now see the “Toolbox” view instead as shown in the figure below.

This view has seven groups. We'll not explain the content of every single one. The important ones are “Vectors” and “Windows Forms”. We'll use the Vectors in this tutorial.

The advantage of Vectors is that they can be grouped easily and they support layers. That means you can decide at any time which vector or vector group should be on top or in the background. This concept of layers is not supported by the Windows Forms. So if you’re not going to make use of layers then we’d rather recommend you to use the Windows Forms as they provide more functionality.

Now let’s create a display based on a vector. Press the “+” next to the group called “Vectors” in you toolbox and select the “Text” vector object. (See figure below)
Now draw it on your display. You should now have something like seen in the figure below.

If you look closer to that object you’ll see that it has a text in it called “COText” and it has a black border. If we take a closer look at the border we’ll see 8 white squares outside the borders and 4 smaller squares on the border corners.

If you click the big squares and hold your mouse down you’ll be able while dragging your mouse around to resize the Text vector object. But it will also resize the text inside and change its ratio.

If you want to keep the text size and its ratio you should click and hold your mouse down on the smaller squares. That will resize the text box without resizing the text or its ratio.

If you want to undo any changes made to the object just click undo in the upper left corner of the MAPS Designer.

Now let’s go to the “Design” Menu in our ribbon at the top of the MAPS Designer. See figure below.

In that menu you can format the text in the text vector object. Let’s increase the text size to 12 instead of 9.

Now let’s go to the “Behaviors” menu. See figure below.
We'll not explain all the behaviors. If you want a detailed explanation on the behaviors please go to the help file (Question mark in the upper right corner of the MAPS Designer) and enter “Behaviors” in the search field of the help dialog or the name of the behavior you want to know more about.

Behaviors can be assigned to objects. You can associate several behaviors to one object.

If you want to assign a behavior to an object select the object in the MAPS Designer and then click the behavior you want to assign. A configuration dialog will open and enable you to configure the behavior.

Let's assign a “Display Value” behavior to our text vector object.

Therefore select the text vector object and then click the behavior “Display Value”. The following configuration dialog should display.

We created the agent MOTORPUMPFLOW. And now we’ll display the scaled value of this agent in our text vector object.

There’re 2 fields:

- Input Data Element: This is the address of the tag/slot (MOTORPUMPFLOW). We'll browse it in a minute.
- String Formatting: Here we’ll configure the format of the value. Since we’re working with a D0 register, we’ll be working with non-floating point values. So we’ll adjust the formatting to not display anything after the decimal point. We’ll do that in a minute as well.

Let’s assign the tag/slot to the behavior. Click the “…” next to the Input Data Element field. The “Data Element Selection” window should open.

Click the “+” next to “Adroit” under “Datasources” (we usually call this the Adroit Datasource). Now click the “+” next to the “Agentgroup”. In the list find the item Analog and click “+”. You should see the Agent “MOTORPUMPFLOW”. Select this item.

**NOTE on Agents and tags/slots:**
At the bottom of the window you’ll see a string (see figure below).
This string is “Adroit.Analog.MOTORPUMPFLOW.value”. This is the address of the scaled value provided by the Analog Agent called “MOTORFLOWPUMP” hosted on the Adroit Agent Server.

Remember the Agent provides values over tags/slots? The Agent itself is not a tag! If you click on the “+” next to the “MOTORPUMPFLOW” you’ll see a whole list of tags/slots. Somewhere in that list you’ll find “rawValue” and “value”. If you clicked “rawValue” you would be able to show the value in the PLC and not the scaled value. You could also show the High-High limit (and even change it) by select that tag/slot.

When you click an agent in the Data Element Selection window it will automatically select the tag/slot called “value” (scaled value) for you.

Click the Agent “MOTORPUMPFLOW” to make sure you’ve selected the slot/tag called “value” and then click finish.

Now back in the main configuration window of the Display Value Behavior window we’ll format the value to be displayed. Click on the “…” next to the field called String Formatting. You should see the following “Format String Editor” dialog.
Put the “decimal places” to 0 instead of 2 and click “Finish”.

Your window should look like this now.

Then click “Finish” again.

How to get an overview of the assigned “Behaviour” and edit an already assigned one is part of chapter 5.9

You’ve now configured the text vector object to display the scaled value of the MOTORPUMPFLOW agent.

Let’s preview that and see if it works. In your Graphic Form Window you’ll find a menu with a feature called “Preview” (Preview) click and see what happens. You should see a “0” in your text box (text vector object). See figure below
To switch back to Design mode and leave the Preview mode click “Design” ( ▶️ Design ).

**NOTE:**
In Preview mode you can’t edit graphic forms. Always make sure you left the Preview mode if you want to make changes to your Graphic Forms.

Now we’re going to change the value of MOTORPUMPFLOW in the PLC.

### 5.6. Write a value to a tag

To write a value to a tag we’ll assign a behaviour called “Operator Action” to our text vector object.

Select the text vector object and click in the “Behaviours” Menu “Operator Action”.

You should see a similar window as below.

Select under Data Element the “value” slot/tag provided by the “MOTORPUMPFLOW” agent. (Click the “…” next to the field Data Element and just select the MOTORPUMPFLOW agent from the list as this will automatically select the “value” slot).

Leave the minimum value to 0.
Change the maximum value to 10000 (remember we set the scaled value to 10000?)

Under “Action Type” choose the radio button “Data Entry”.
Your Window should look like this now

![Figure 35: Configured Operator Action Behavior](image)

Click "Finish". We’ve now configured our text vector object to write a value to the register D0 of the Q-Series PLC. To enter a value all you need to do is to click the text vector object at runtime. You’ll then be prompted to enter a value. This value has to be within the entered min/max values (0 to 10,000).

Hit “Preview” and try it out. Don’t forget to hit “Design” before you go on with this guide.

Now that we display and write our tag lets log and trend it.

5.7. Log and trend a tag

Logging a tag is very simple. You can choose to log to a database or a proprietary file. We’ll log to the proprietary file as this doesn’t require any database.

Go to the Enterprise Manager and right click the “Adroit” Datasource. Select “Open Configurator”. The Agent Configurator should open.

Under “Type” select “Analog” from the drop down box. Now the MOTORPUMPFLOW Agent should appear. Select so it’s highlighted in blue (see figure below)
Then click “Log” (third button from the bottom on the right side)

The following window called “Edit Datalog Agent” should display

All you need to do to activate data logging is to switch the radio button at the top from “stop” to “start”. That will start the data logging.

So just hit the “start” radio button now to launch the data logging.

**NOTE:**
If you want to log to a SQL database, just click the butt “SQL…” (middle right of the window) and follow the instructions. Once finished you’ll have a SQL connection string in the datasource field instead of “C:\AdroitProject\Data\wg”.

If you want more information around the data logging please refer to the help file.

We’re now logging the scaled value of the PLC register D0.
Let’s set up a trend to display the historical data
Leave the Agent Configuration Window opened and get back to the designer.
Change the “Enterprise Manager” view to the “Toolbox” view (click the “Toolbox” tab).
You should now see the “Toolbox” view like on the figure on page 17.
In the “Toolbox” select the Group called “Data”. Select the object called “LineChart” and draw it on your display.
A configuration Wizard should open. See figure below.

We’ll not explain the different parameters in the wizard. If you wish to learn more about it please refer to the help files (click the question mark “?” at the right top of the desiger)
Let’s go through the wizard now:
Give the Chart a title (Example MotorPumpFlow Trend) and click “Next”
In the following window click the “Values” tab (See figure below)
Next to the “Y Values” field click the “…” and select the Analog Agent called… right: MOTORPUMPFLOW

Just select the Agent as the slot/tag “value” will automatically get selected.

Back in the wizard you should see something similar to the following figure.

Now click “Finish”. Hit the “Preview” button. If you start entering values through the text vector object you should see how your trend curve changes. You should even see historical data.
NOTE:
Once you’ve set up a tag to be logged the trend will automatically display the logged data. There’s no need to establish a relation between the logged data and the trend. It’s done seamlessly by MAPS.

We’ll now start working with the alarms.

5.8. Alarm a tag

You might remember that when we configured the scaling in our MOTORPUMPFLOW agent we also set up the High-High, High, Low and Low-Low limits. (See page 14). We also explained that once one of these limits would be exceeded the respective bit/tag/slot with the same name would be set. The bit will be reset as soon as the value of the agent is within the limits again.

We’ll now trigger an alarm on these bits.

If your Agent Configurator is still opened just go to it or open it while right clicking your “Adroit” Datasource and selecting “Open Configurator”

Select “Analog” from the pull down menu called Type and you should see the same as on page 24, Figure 36.

Select the Agent “MOTORPUMPFLOW” so it gets highlighted in blue (like on page 24 Figure 36).

Now click the button “Alarm” (fourth from the bottom on the right side)

You should the window as shown below.

![Figure 41: Alarm configuration](image)

In the field called “Available” you’ll see all the bits that can be triggered from the Analog Agent called MOTORPUMPFLOW.
Remember we set up the High-High, High, Low and Low-Low limits up earlier (page 14)
The bits in the “Available” list are exactly the bits that will be set and reset once the “value” slot/tag of the MOTORPUMPFLOW agent is out or within these limits.

We’ll now select the “High” bit and move it over to the “Current” list with the “<<” button. Then we uncheck the checkbox called “Acknowledgment required”. That means as soon as the alarm is gone it also gets removed from the alarm list. (You don’t need to acknowledge it)
Your window should now look like the following.

![Configure High Alarm](image1)

**Figure 42: Configure the “High” Alarm**

Now we’ll do the same for the High-High bit but this time we tick the checkbox “Acknowledgment required”. That means that the alarm will change color once it’s gone but will remain in the list till the alarm gets acknowledged.

You should see something similar to the window below.

![Configure High-High Alarm](image2)

Click “OK” and then close the Agent Configurator. Answer “Yes” when prompted to save the changes.
If you wish to learn more on alarms please refer to the help files.

Now that we’ve set up the alarm conditions, we’ll now display the alarm. We’ll therefore create a new graphic form. Go to the Enterprise Manager view and under projects right click “myPlant” and select “Create Graphic Form”. Rename the Graphic Form to Alarms.

Now go to the Toolbox view and select the group called “Adroit” (You might need to scroll down a bit or minimize some other groups). From that group select the “AlarmViewer” Object and draw it on the graphic from “Alarms” that we’ve just created.

You should see something like the following.

![Figure 43: AlarmView Object on the graphic form “Alarms”](image)

Save the form while clicking the “!” icon.

Go to the graphic form called “Main” (select the tab between “Start Page” and “Alarms” in the Figure 43: AlarmView Object on the graphic form “Alarms”)

Save that form as well.

Let’s preview what we’ve done so far.

Go to the graphic form called “Main” and preview it. Click the text vector object and enter the value 9500.

Select the other graphic form called “Alarms” and preview it you should see both alarms in it High and High-High. You can now put the value back to 5000 and go back to the “Alarms” graphic form and preview it. You should now see that the High alarm has disappeared but the High-High is still in the list with a green background.

Remember we set the High alarm up and unchecked the “Acknowledgment required”. That’s why the alarm is not in the alarmviewer anymore.

We’ll now introduce you to working with vector graphics so you can use the nice vector graphics from our library.

**5.9. Working with vector graphics and behaviours (animations)**

Vector graphics provides a way to draw graphics. It enables resizing without any quality loss. Typical vector graphics can be lines, rectangles and circles and vector text. These can be grouped to a group of vectors. Even groups of vectors can be grouped to a new group. A vector group can therefore become very complex and constituted of many vectors. However if you want to animate a vector group you need to know and select which part of the vector group
you want to animate. Do you want to animate a sub group or the whole vector group? The last thing you want to do is ungroup all the vectors to be able to select the group or vector you want to animate.

In MAPS we have a simple solution for that. The “Content” view.

The “content” view shows you all vector sub groups and vectors within a vector group. Let’s illustrate this with an example.

Create a new graphic form in your project called “myPlant”. Call the graphic form “pump”.

Now go to the Enterprise Manager view. Scroll to the bottom of the tree. You should see “Shapes Wizards and Examples”. Double click it. Then double click “Static Graphics”. Scroll down to the section called “Pumps” and double click it. Drag and drop the “Standard Pump” on the screen. Select “Paste Template” (the option in the middle)

We’ve now dropped a nice looking pump on our graphic form. Let’s play with it a bit. Hold “Shift” on your keyboard and drag over the lower left corner of the Pump till you see a black double arrow. Click and resize the Pump. You’ll see that the Pump will be resized with respect to its initial ratio.

Now you should have a bigger pump.

Let’s now display the Content view.

Go to the ribbon and select the tab called View. (like in the figure below)

Click the lower left icon in the content section of the “View” menu. This will display the “Content” View on the right side of your MAPS Designer. (See figure below)
If you now click the Pump you’ll see the vector structure of the pump in the Content view. This should look like this in the content view.

We can now select from the content view the vector or vector group. The selection will get highlighted on the pump in the graphic form. Let’s try it. Select “PE_231_Copy1”. This will highlight the right side of the pump.

We’ll now assign a “Color Behavior” to this part of the Pump. It should get green if the high bit is not set and red if the high bit is set.

Right click in the content view the “PE_231_Copy1” and select “Add New Color Behavior”. Browse the Input Data Element. This time we’ll have to choose the slot as we don’t want the “value” slot. Expand the MOTORPUMPFLOW Agent and search for the slot/tag called “High” (see figure below).
Figure 47: Select the "High" slot/tag from the MOTORPUMPFLOW Agent

Select the slot/tag called “High” and click “Finish”. Back in the Configuration window of the Color Behavior go to the ranges and set them up like in the figure below.

Figure 48: Configure Color Behavior

Click “Finish”.

Select the slot/tag called “High” and click “Finish”. Back in the Configuration window of the Color Behavior go to the ranges and set them up like in the figure below.
Let’s have a look at the content window again. When you expand the “PE_231_Copy1” (if it’s not already the case) you’ll see at its bottom that there’s an item called “color”. This is the behavior we’ve just created. If you double click it you can actually change the parameters of the behavior and save them.

Let’s preview the Pump.

![Figure 49: Pump Color Behavior Preview](image)

Let’s now finalize this little application by creating a navigation structure.

5.10. Create a navigation structure

There’re several ways to implement navigation in MAPS. You can assign an “Execute Command” behaviour to a button that opens a graphic form for example. We’ll not work with this here but if you’re interested to learn more about that behaviour please use the help files and search for the behaviour “execute command”.

We’ll show you how easily you can generate a whole navigation structure with the “TemplateGONavToolbar” Object.

Let’s start.

First create a new graphic form in the “myPlant” Project section and call it “Startup”. Go to the Toolbox view and select the group called “Templates”. Select the object called “TemplateGONavToolbar”. Draw it on the graphic form called “Startup” using as much space as you can. A context menu will appear. Select “Browse for Toolbar Graphic Forms”.

NOTE:
This context menu can be accesses anytime by clicking the smart tag in the upper right corner of each object. See figure below

![Figure 50: Access the context menu of an object through the Smart Tag](image)

Once you’ve selected “Browse for Toolbar Graphic Forms” you see the following window.
In that window select “Add Project” and select the “myPlant” Project.

You should now see something like this.

Now select the “Startup” (highlighted in blue in Figure 52) and click the remove button (third from the left).

Now we need to append the graphic form addresses in the column called Graphic Form Name (the first column)
Double click the white field left of the field called “Alarms” and select the “Alarms” graphic form. See figure below.

![Configuration: Graphic Form Selection](image)

**Figure 53: Select a graphic form**

Then click “Finish” and do the same for the “Main” and “Pump” graphic form.

You should now have the same view as in the figure below.
You have now setup a menu bar with the menu items “Alarms”, “Main” and “Pump” and defined that when they get clicked the respective graphic form will get loaded.

Finally click “Finish” and save the display.

You have created your navigation structure.

We’ll preview this in the MAPS Operator after we’ve set up the profile for the application.

5.11. Working with Security and profiles

MAPS uses the Windows security. That means that all users and groups are provided from the Windows operating system. The privileges assigned to a user or a group will be set up in MAPS. When you installed MAPS the user that was logged in to install MAPS will automatically be added to the so called “list of allowed users”. We’ll now have a look at that list.

Go to your Enterprise Manager. Expand the Management section. Then expand the “Profile” and the “Security” section. Then in the “Security” section expand the “Users and Groups” section and then the “Users” section. You should see the same as in the figure below.
Right click the item called “Allowed Users” and select “Set Allowed Users”. You should see the following window.

In the right list you have all available users on your windows machine. In my window I have a “mapsoper” and “mapsuser” for example. I could now with the “<<” button shift these users to the “selected users” which means they would be available to MAPS. I could then assign privileges to these users and make sure that when the user “mapsoper” logs in he wouldn’t see certain sections of my application or the runtime environment (MAPS Operator) would start in full screen mode and can’t be closed.

In the left side you should the user that you used to install MAPS with (your own user).
NOTE:
In bigger applications we would recommend you to use groups and assign privileges to groups and not directly to a user. That way it’s easier to handle your security concept. What we have just described for the users applies in the same manner when you use groups.

Let’s close the window “Set Allowed Users” and open (double click) the item called “Default” from the section “Management” -> “Profile”.

You should see a window called Profile Configuration. See figure below.

![Profile Configuration](image)

Figure 57: Profile Configuration

You’ll see at the bottom that your user account belongs to the default profile. That means that all settings made to the default profile will apply to your user account.

Let’s change the default profile’s settings.

Let’s select the “Operator Settings” section on the upper left side.

You should see the following window.
In this section we can setup the way the MAPS runtime environment (MAPS Operator) will be started for the default profile. You’ll find a detailed explanation in the help file of every property in this list. We’ll only change the following:

- DefaultGraphicFormPath (the default graphic form that we want to be loaded)
- Full Screen (when set to True will start the MAPS Operator in full screen mode)
- StartupWindowState (maximized, minimized, normal)
- InitialOperatorMode (disables the Enterprise Manager view at runtime when set to false)

Set your window to look like the following.
Click “Finish”.

We have now setup the MAPS Operator to open the “Startup” graphic form when a user or a group is linked to the “Default Profile”. We have further allowed to close the MAPS Operator and not to “always have it on top” (which means other windows can be put in front of the MAPS Operator). And finally we’ve disabled the Enterprise Manager View at runtime.

Let’s now start the MAPS Operator.

**6. Running your project**

We’ll now run the MAPS Operator which is the runtime environment of MAPS.

Therefore goto “Start” or \(\text{Ctrl} \) - “All Programs” -> “MAPS” -> “MAPS Operator”

You’ll be prompted to login. Use the account you used to login to the MAPS Designer.

You should now see the following window.
You can now click the different displays “Alarms”, “Main” and “Pump” to navigate between each of them and interact with your application.

Once you’ve finished right click the MAPS Operator and select “Exit” to close the “MAPS Operator”.

You are now ready to continue your own journey into this great product.

Other resources that will help you along the way are:

- The help files in MAPS
- The Training Manual
- Online videos and tutorials on our website
- Product Technical Description document
- Forum on www.adroit.co.za. We strongly recommend you to register (It’s free)
- Help desk on support@adroit.co.za or your local distributor