autoSIM-200 – FAQs

Contents

Error codes during the license registration process

During the license registration process, one of the following error messages may appear:

Error code	Description for the error	Solution
-1	Invalid code (not a license code).	Find the correct license code and re-register the license.
-9	The license registration server is not responding.	Run autoSIM with administrator privileges to register the license.
-10	License registration error. License already registered and in use.	Register a different license code on that PC.

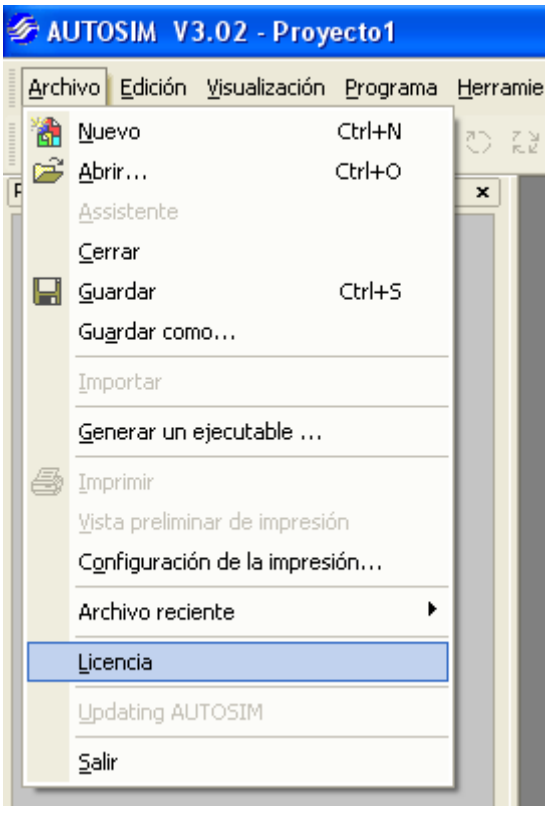
Retrieving the autoSIM licence

There are two possible scenarios:

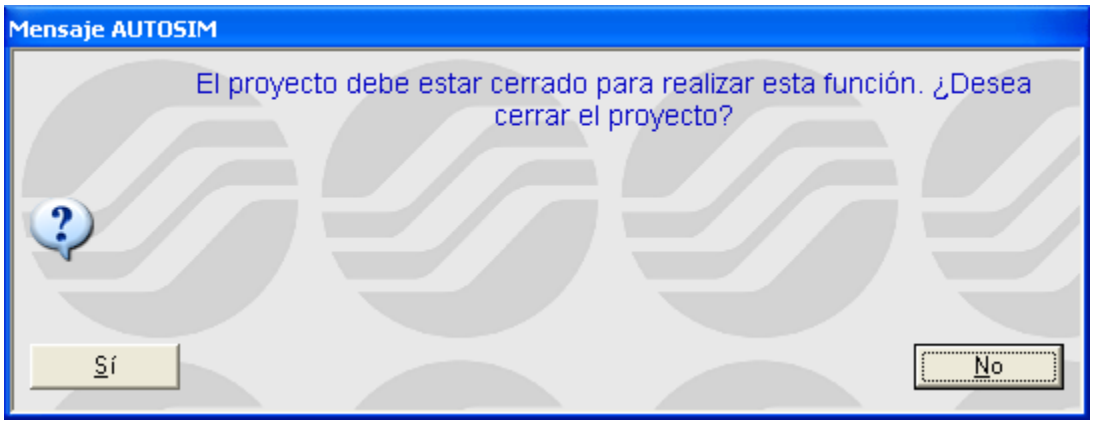
1. autoSIM is installed on the computer and is working.
2. autoSIM is installed on the computer but is not working (possibly because it was uninstalled incorrectly).

CASE 1: autoSIM is installed on the computer and is working.

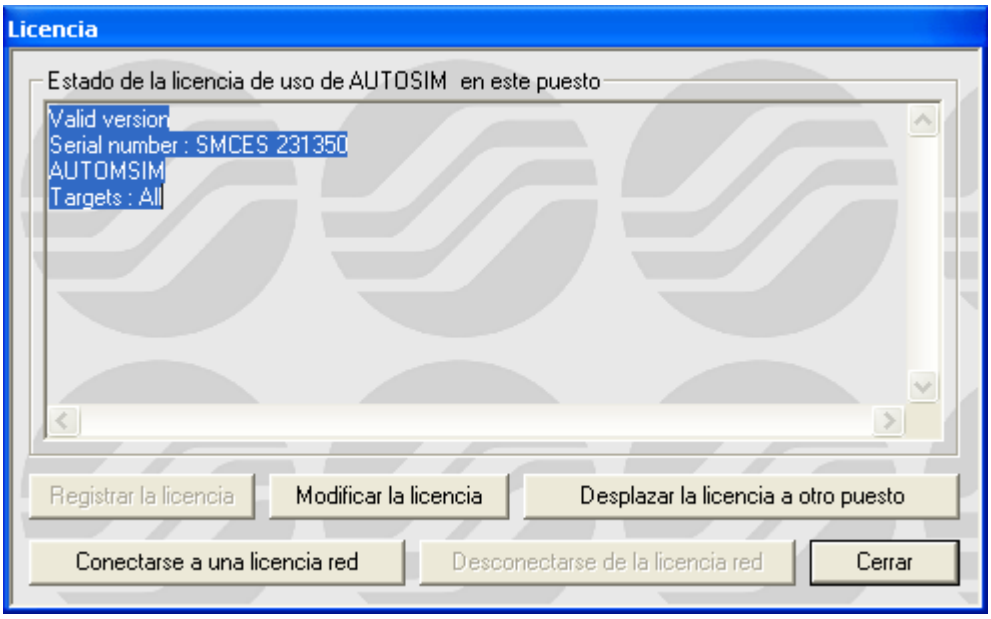
1. Open autoSIM.
2. Click on **'Licence**' in the **'File'** menu.



3. Close the project by clicking the **'Yes'** button.

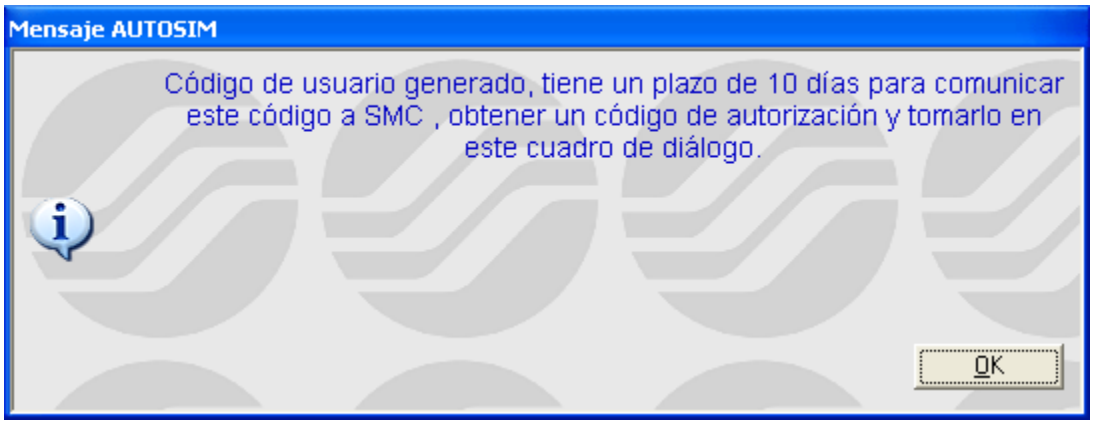


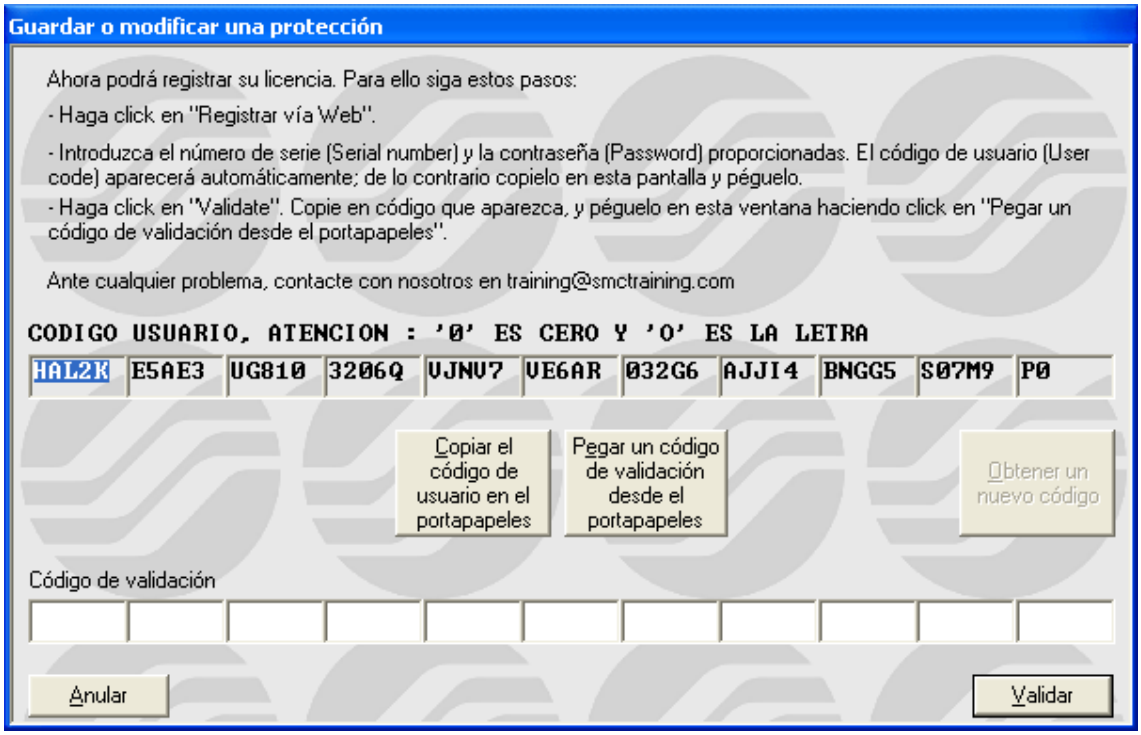
4. The following window appears:



As shown in this window, you can see the licence serial number that has been registered on this PC.

5. Now, click on **'Modify licence'** and then click **'OK'** (a new User Code will be generated).





6. Save this code and send it to the email address support.training@smc.com along with the serial number and password.

Using that User Code, *SMC International Training* generates a new Validation Code, which must be entered in the relevant field in the previous window.

CASE 2: autoSIM is installed on the computer but is not working (incorrect uninstallation).

1. Send the serial number and password to the email address support.training@smc.com so that a new licence can be generated.

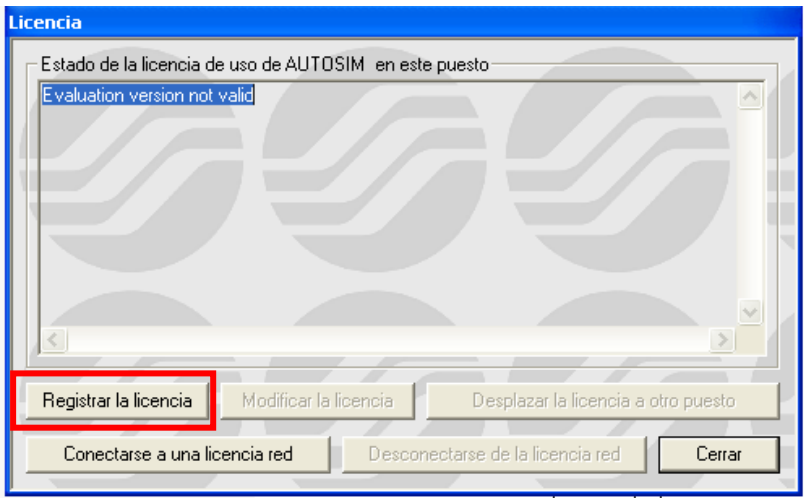
SMC International Training will generate a new serial number and a new password (i.e. a new licence), which must be entered after installing autoSIM.

Transferring the autoSIM licence from one computer to another

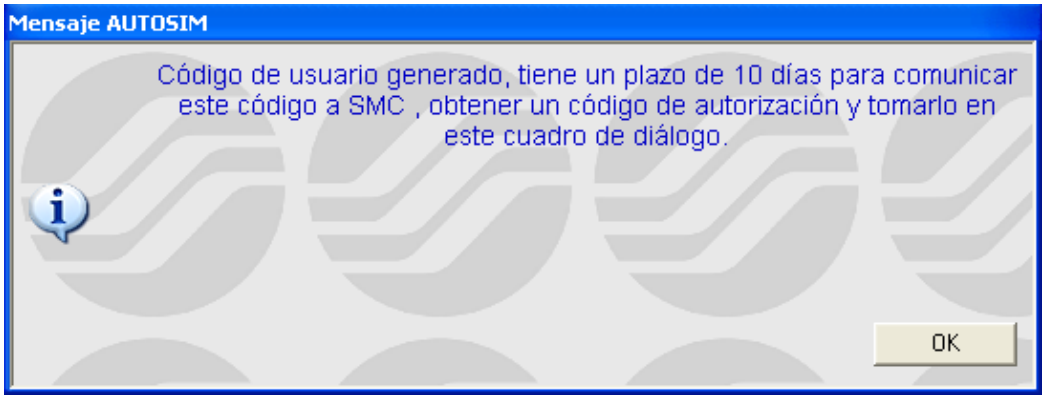
The **source PC** is the one on which the licence is already installed. Similarly, **the destination PC** is the one to which the licence is being transferred.

1. Install autoSIM on the **destination PC**.
2. Generate a User Code on the **destination PC** and copy this code. To do this, click on the *File/Licence* menu.

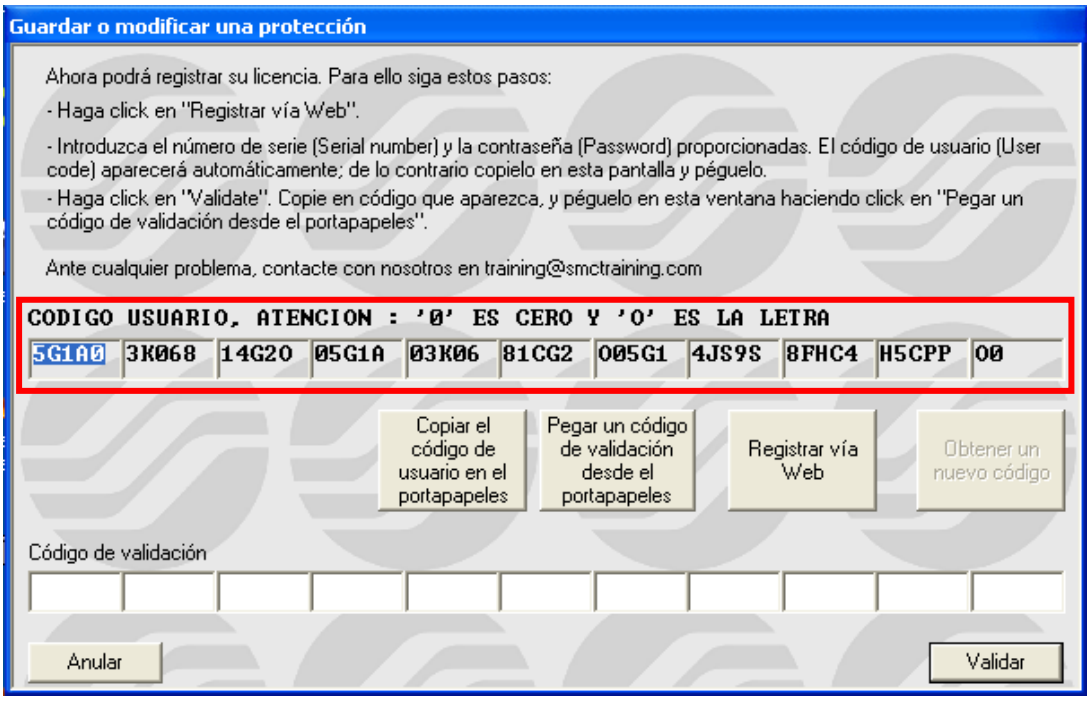
A new window will appear where you must select the **'Register licence'** option.



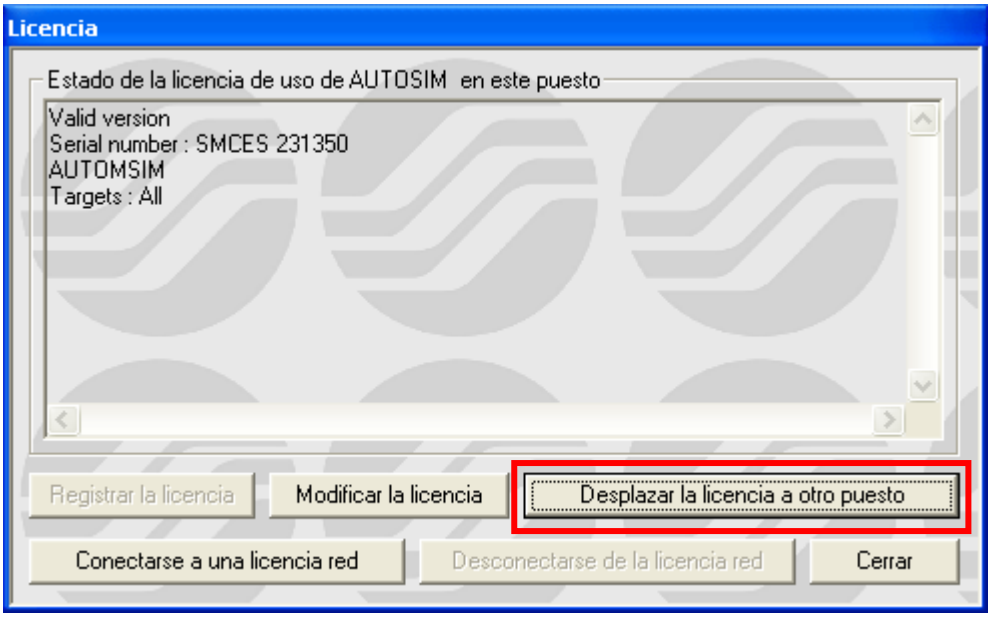
Once this option has been selected, autoSIM displays a message stating that a User Code valid for 10 days has been generated.



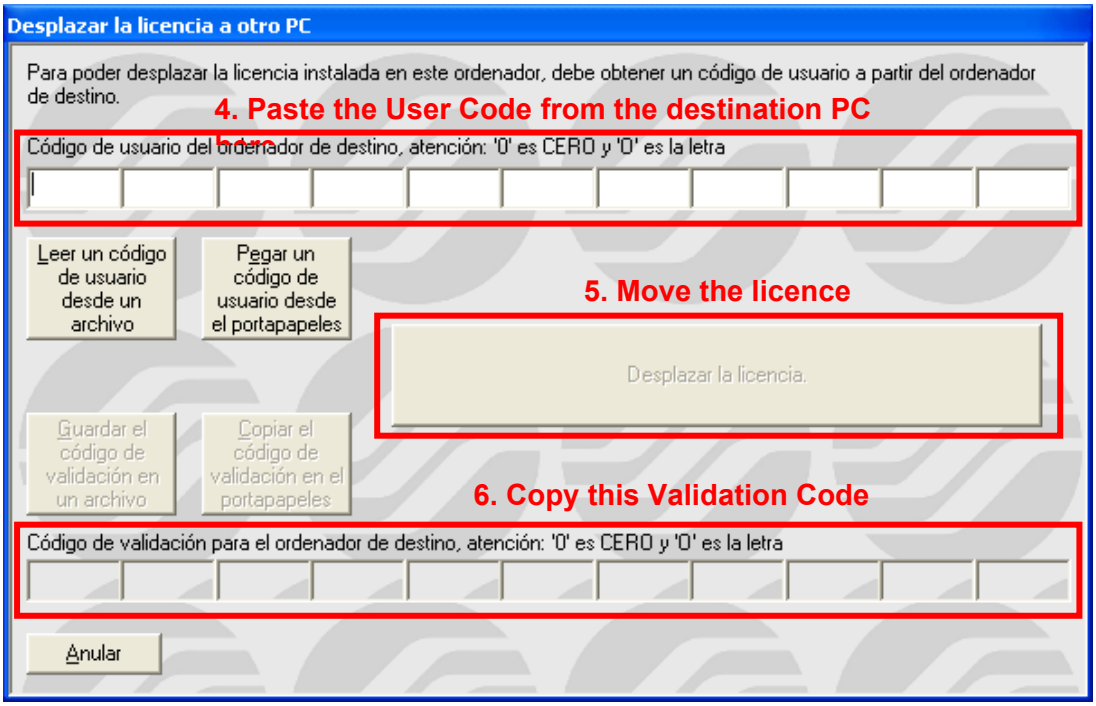
Click the **OK** button to generate the user code. Copy this code to the clipboard.



3. On the **source PC**, go to the *File/Licence* menu and select the **'Move licence to another workstation**' option.



4. Paste the User Code from the **destination PC** into the **source PC**.

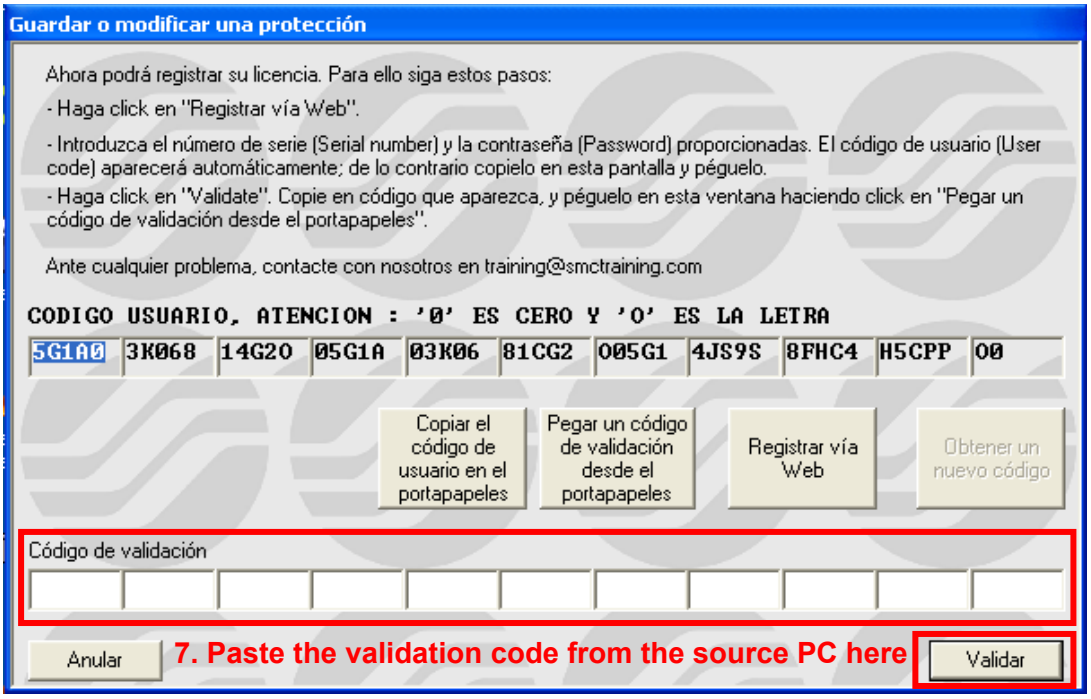


5. On the **source PC**, click on **'Transfer the licence**'.

autoSIM generates a validation code.

6. Copy this validation code from the **source PC**.

7. Paste the validation code from **the source PC** into the **destination PC**.
8. Click on the **'Validate'** button to complete the transfer process.

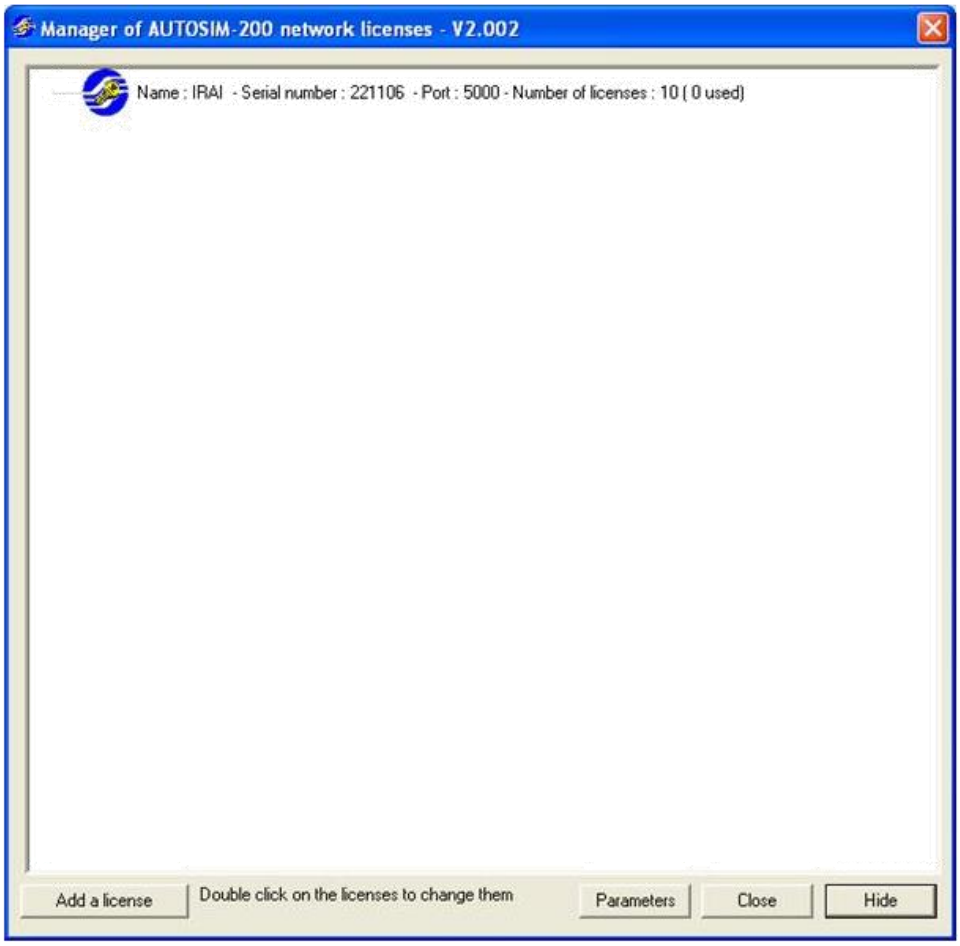


8. Validate licence

Register a network licence on a local server

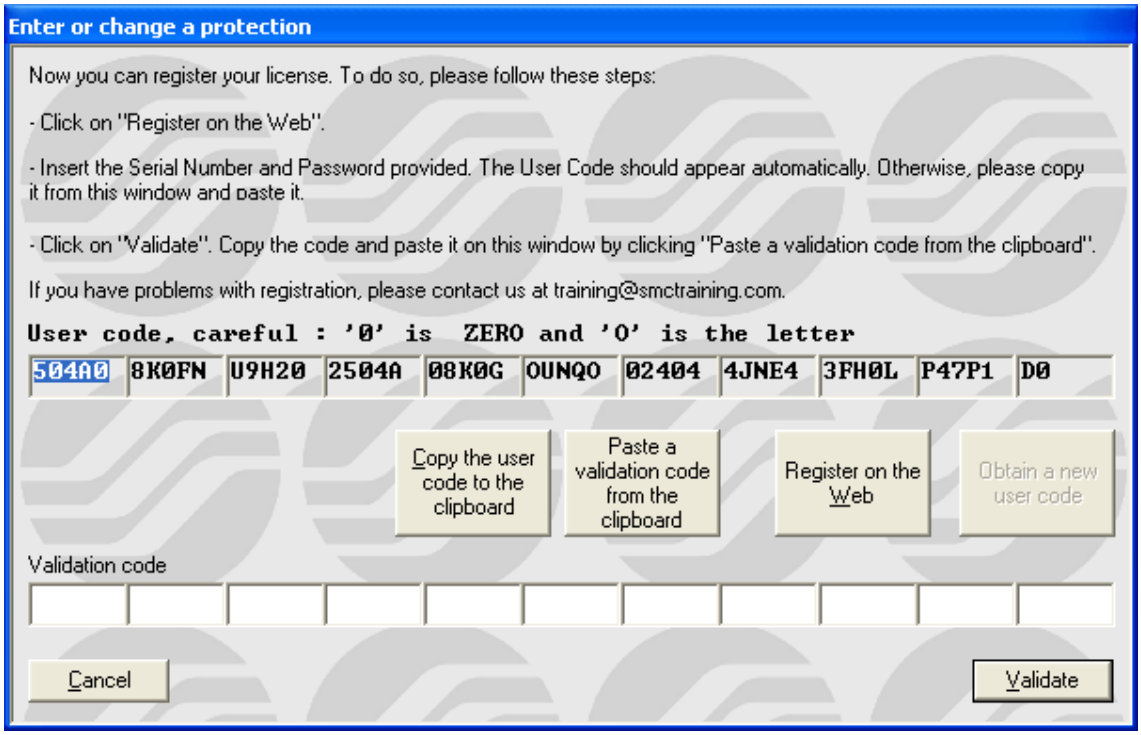
The ***Network Licence Manager*** application manages network licences when they are installed on a local server.

This application must be run from a PC acting as a server (which must support the TCP/IP protocol). Once the Network Licence Manager has been launched, it minimises and is identified only by the key-shaped icon in the Windows taskbar (bottom right, next to the date). To open the Network Licence Manager, double-click on that icon (see image).

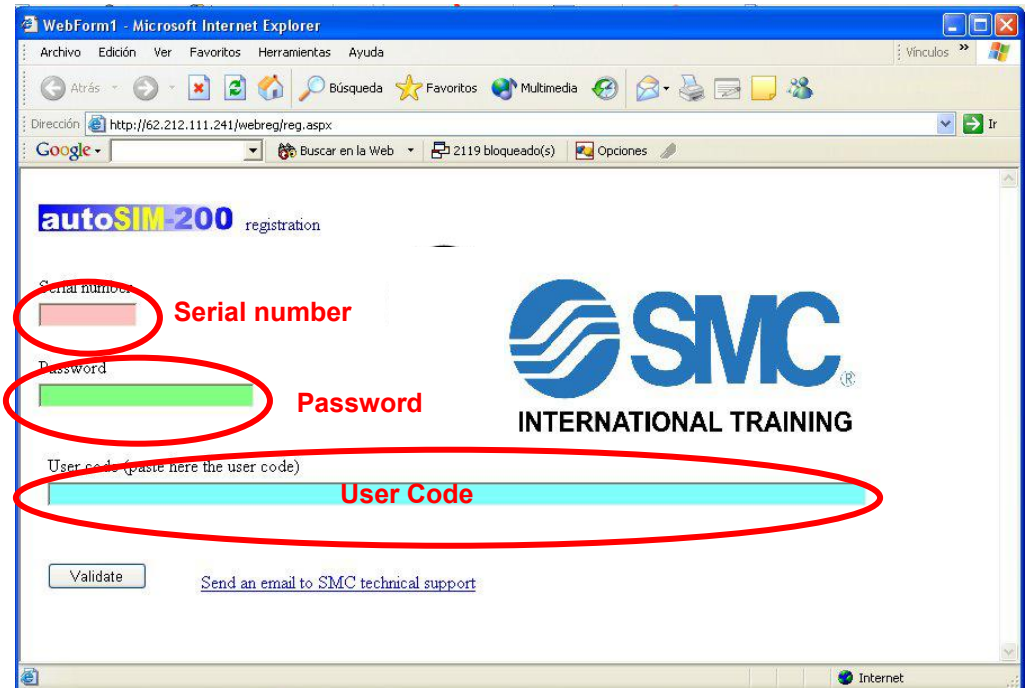


The network licence manager **can manage up to 16** different **licences**. A network licence contains a predefined number of users and is characterised by a set of rights (relating to the number of logins and logouts). For each licence, the maximum number of possible users and the number of connected users (using autoSIM-200 at the same time) are displayed in a tree structure. Each licence is associated with a port number (by default, a numerical value starting with 5000). The first port number used can be configured by clicking the **'Parameters'** button.

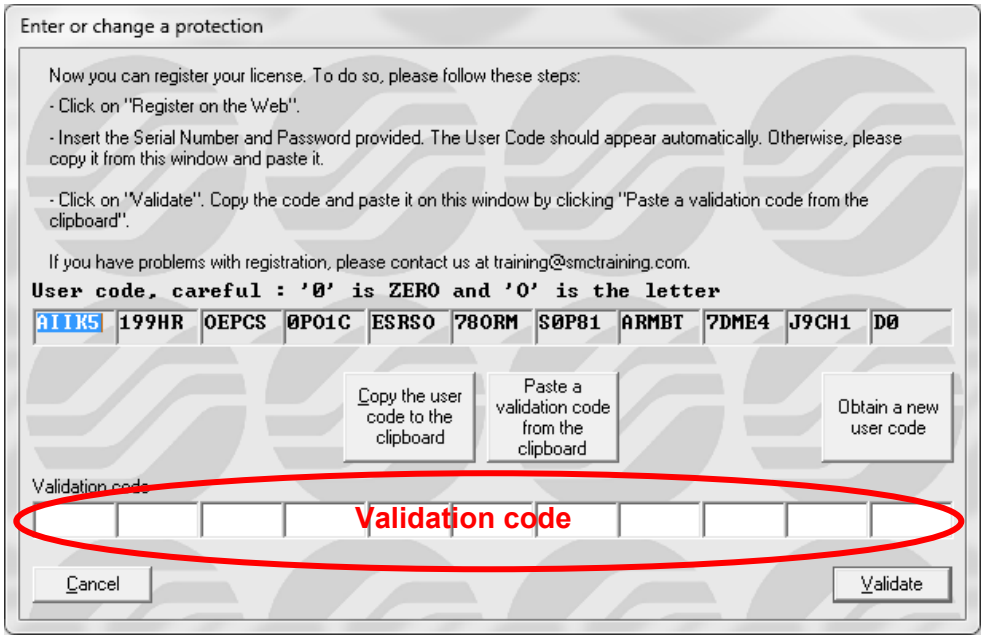
To add a licence, click the **'Add a licence**' button. autoSIM-200 generates a User Code for that PC and that licence.



To register this licence, click on the **'Register on the Web**' button (an internet connection is required). A window like the one shown below will appear, where you must enter the serial number and password provided by SMC International Training, and the User Code generated by autoSIM-200.



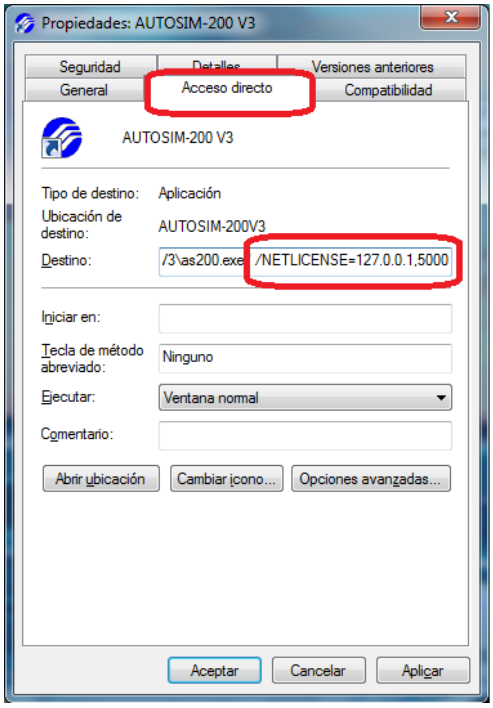
Once all fields have been completed, click on the **'Validate'** button. A validation code will be generated, which must be entered into autoSIM-200 (see figure).



Click on the **'Validate'** button to register the licence.

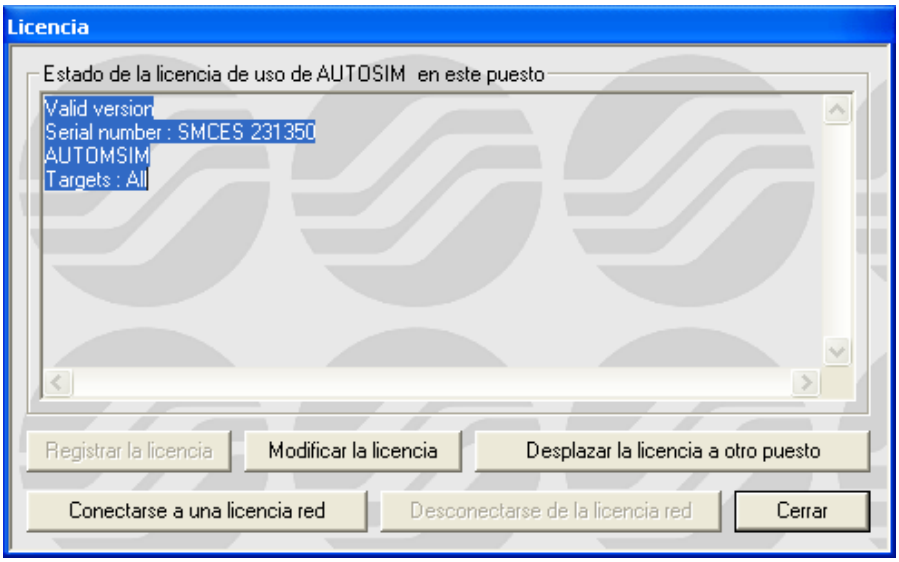
To open the autoSIM software from the server PC, you must enter the text shown below in the **Properties** of the shortcut on the desktop:

/NETLICENSE=127.0.0.1,5000 (where 5000 is the port number selected for the network licence).

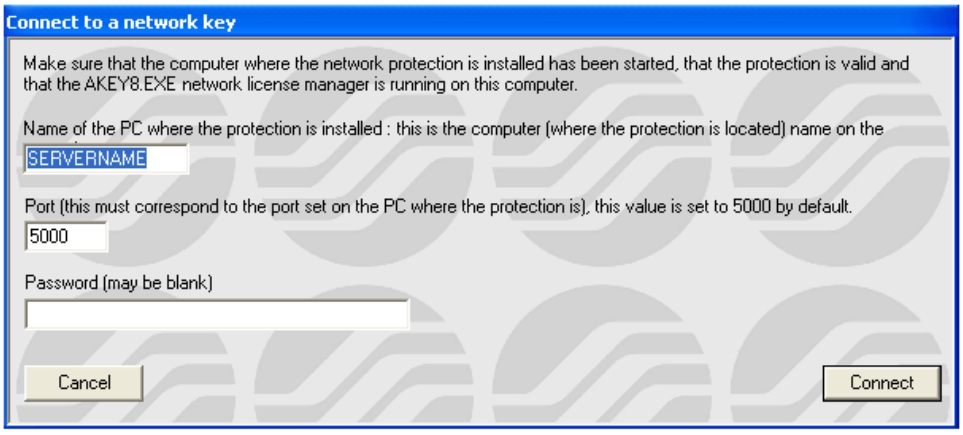

Propiedades: AUTOSIM-200 V3
Seguridad
Detalles
Versiones anteriores
General
Acceso directo
Compatibilidad
AUTOSIM-200 V3
Tipo de destino:
Aplicación
Ubicación de destino:
AUTOSIM-200V3
Destino:
/3\as200.exe
/NETLICENSE=127.0.0.1,5000
Iniciar en:
Tecla de método abreviado:
Ninguno
Ejecutar:
Ventana normal
Comentario:
Abrir ubicación
Cambiar icono...
Opciones avanzadas...
Aceptar
Cancelar
Aplicar

Connecting to a network licence

To connect to a network licence, go to **the File/Licences** menu (this must be done on the client PC, which is the PC you wish to connect to the network licence). Close the current project and click on **'Connect to a network licence'** from the client PC.



You must provide the name of the PC on which the network licence has been registered, as well as the port number corresponding to the required licence. The ***Network Licence Manager*** is used to manage multiple licences on PCs within a TCP/IP network.

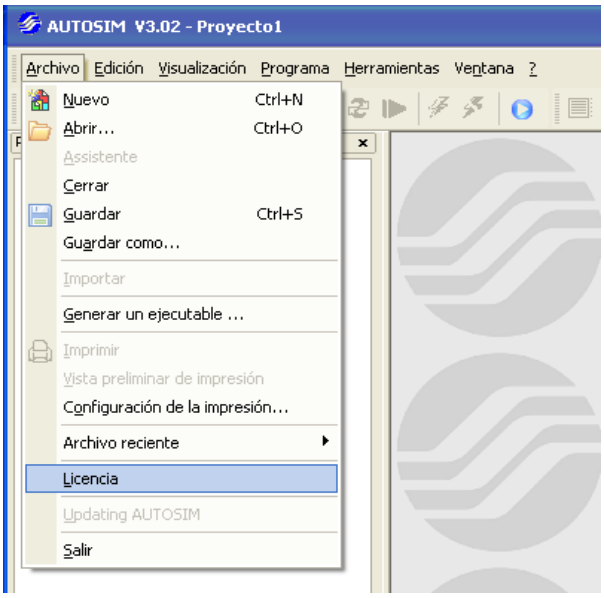


Click the **Connect** button to connect the PC to the network licence.

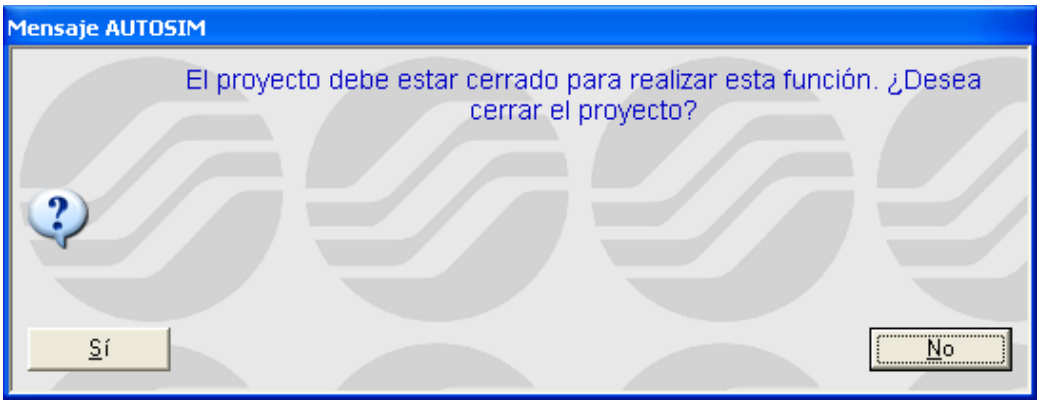
To view the number of licences in use, go to the ***Network Licence Manager*** on the server PC, where all connected licences and available licences are displayed.

Corporate licence installation

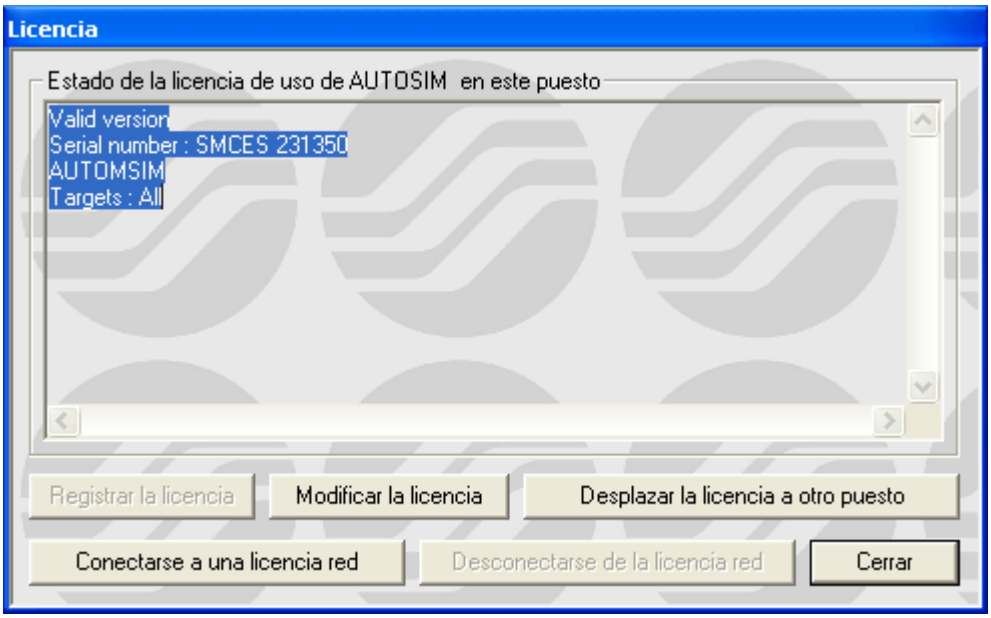
1. Open the **File/Licence** menu.



2. Close the project by clicking **Yes**.

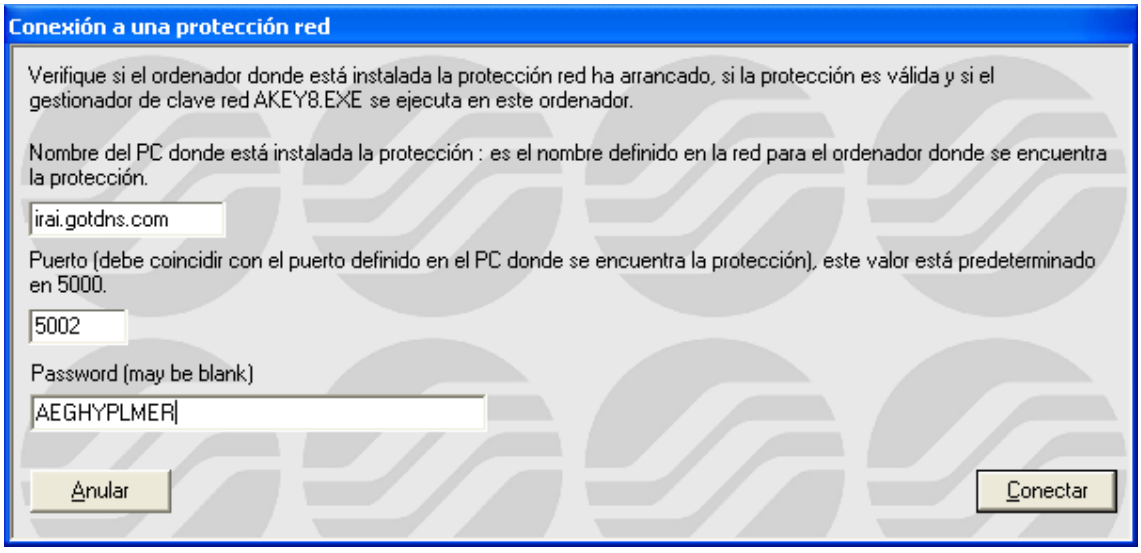


3. Click on '**Connect to a network licence**'.

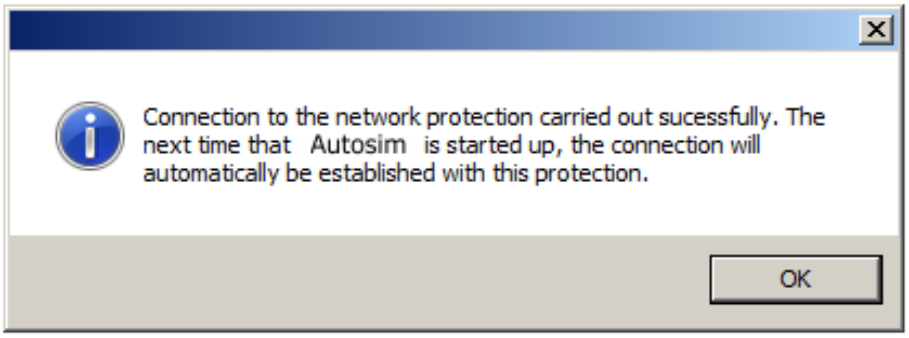


4. Enter the following information:
   - **PC name:** irai.gotdns.com
   - **Port:** 5002
   - **Password:** AEGHYPLMER

(These details may vary depending on the licence)



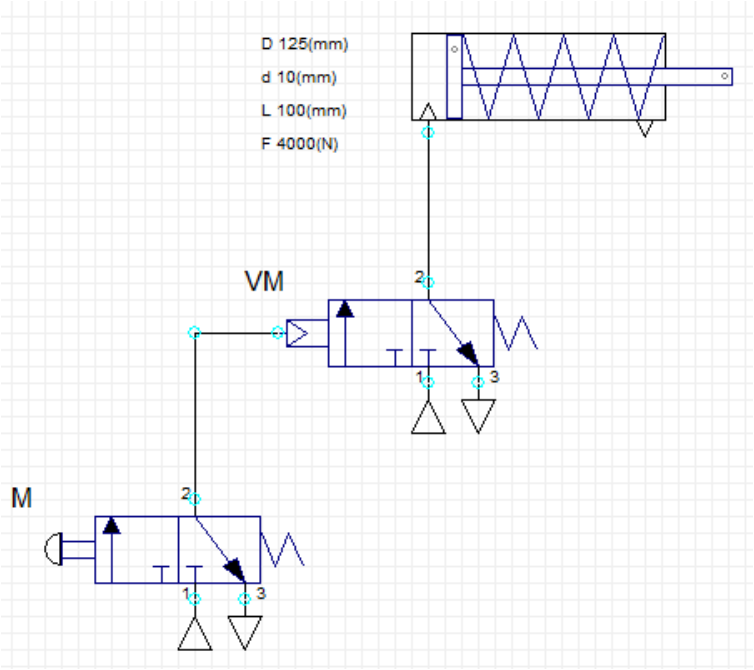
5. The following window will appear to confirm that the connection to the licence has been established successfully:



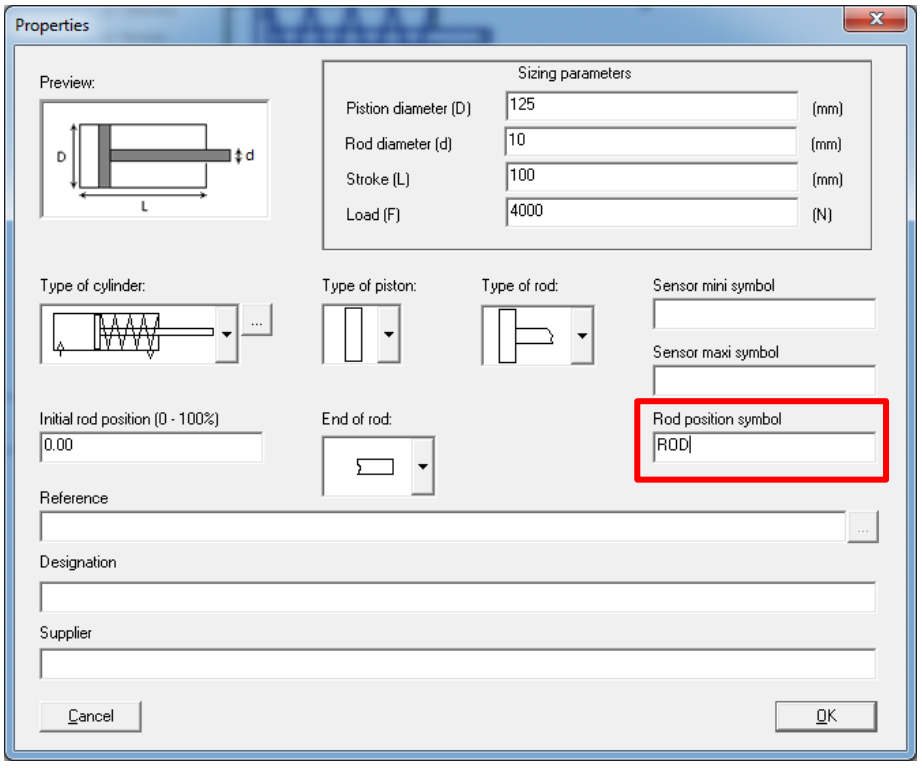
- If this window does not appear, check your internet connection.
- The computer must be connected to the Internet whilst autoSIM is in use.

How to create a phase-space diagram

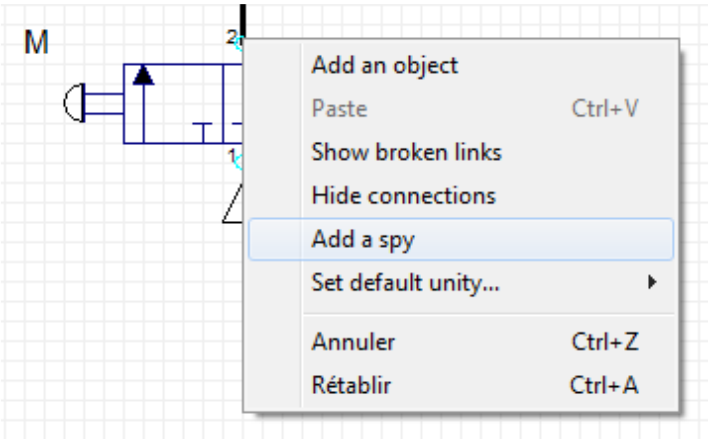
Create the phase-space diagram for the following circuit:



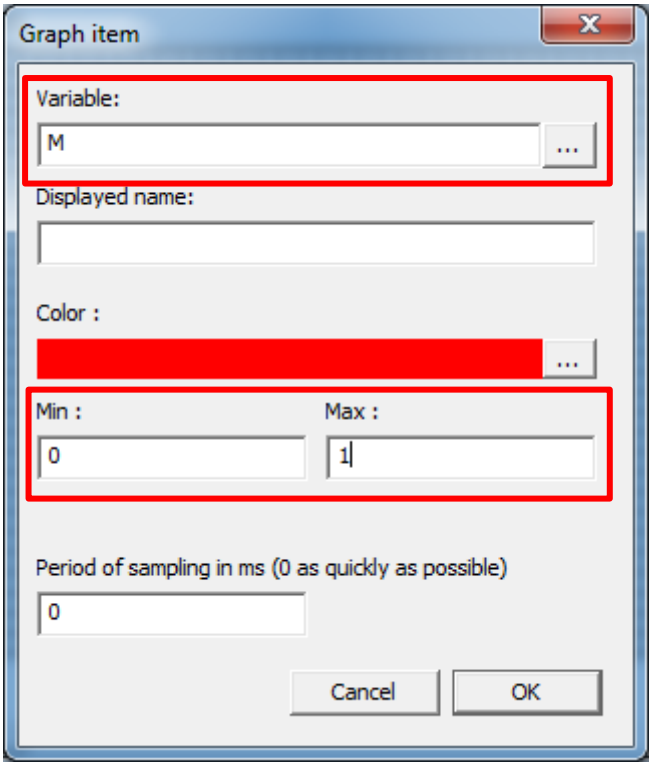
1. Open the cylinder's Properties and enter a name in the **'Rod position symbol'** field.



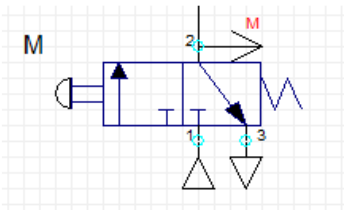
2. Place the mouse cursor over the output (2) of valve **M** and right-click. Select the **'Add a** spy' option.



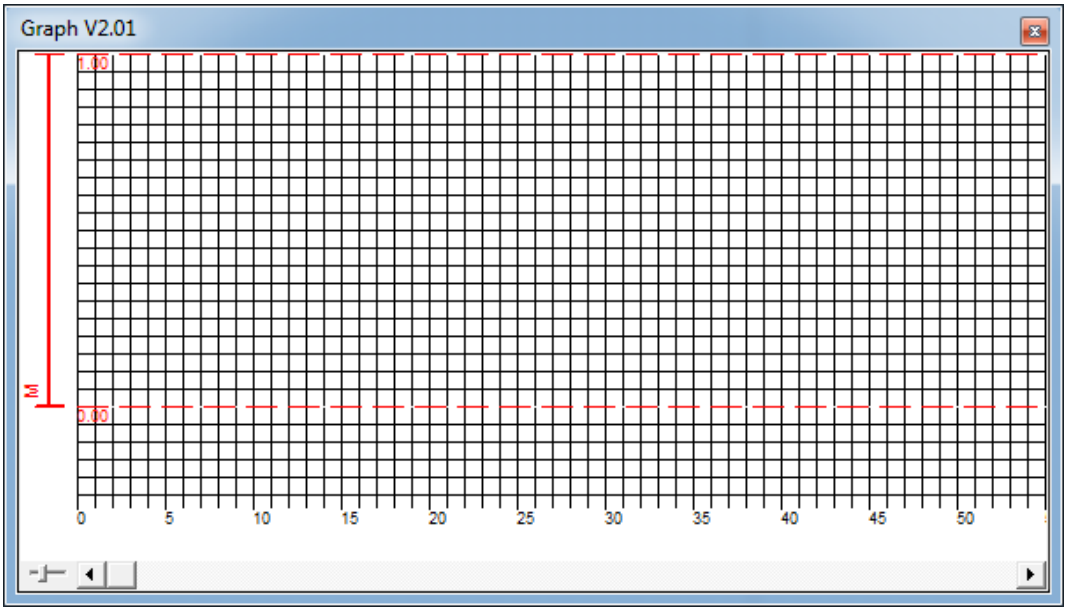
3. Select a name for this spy, in this case **M**. Set a maximum and minimum value for this new variable (for a valve, the values range from 0 to 1).



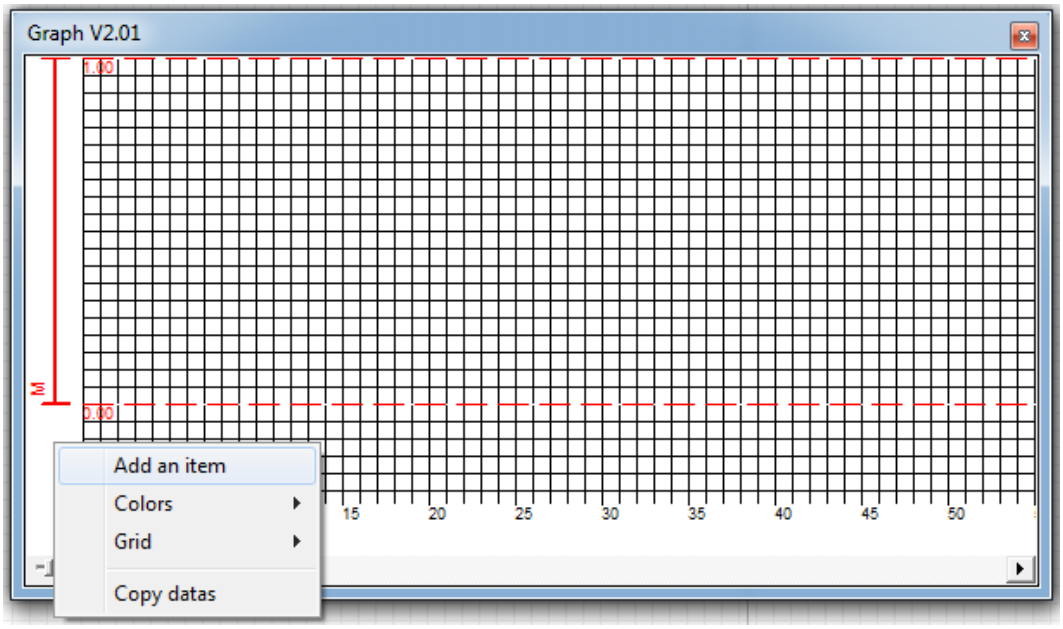
The result looks like this:



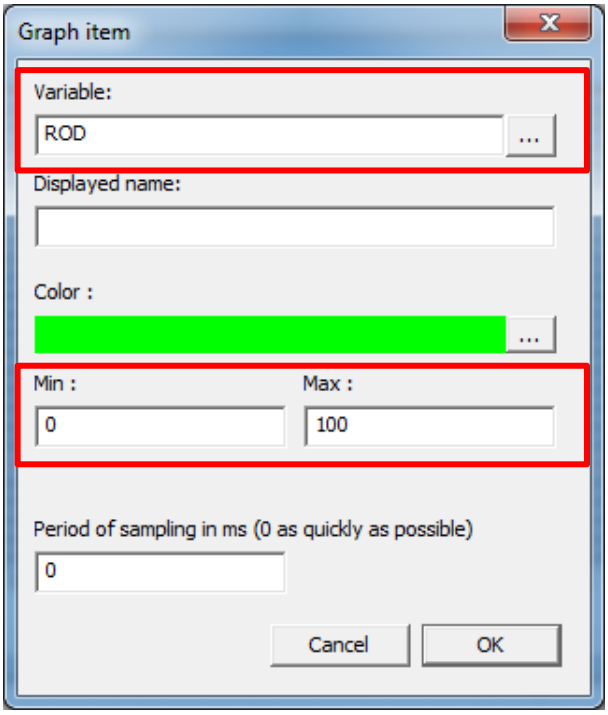
Where autoSIM creates a graph showing the phase-space diagram.



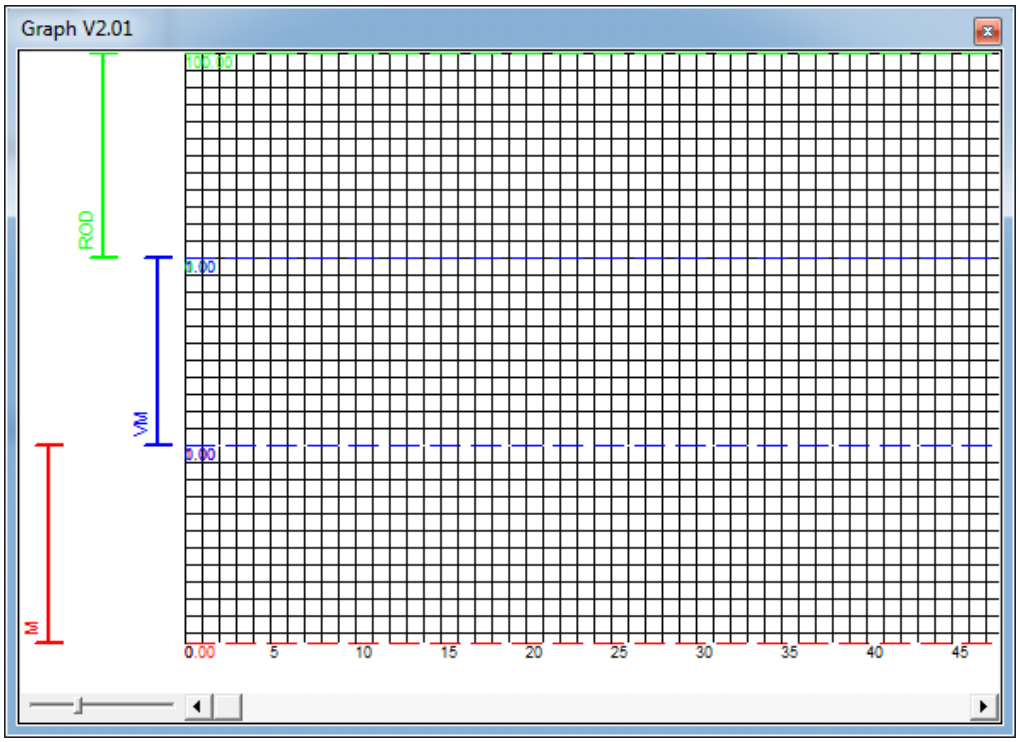
4. To add the cylinder variable created earlier, right-click on the graph and select the **'Add an item'** option.



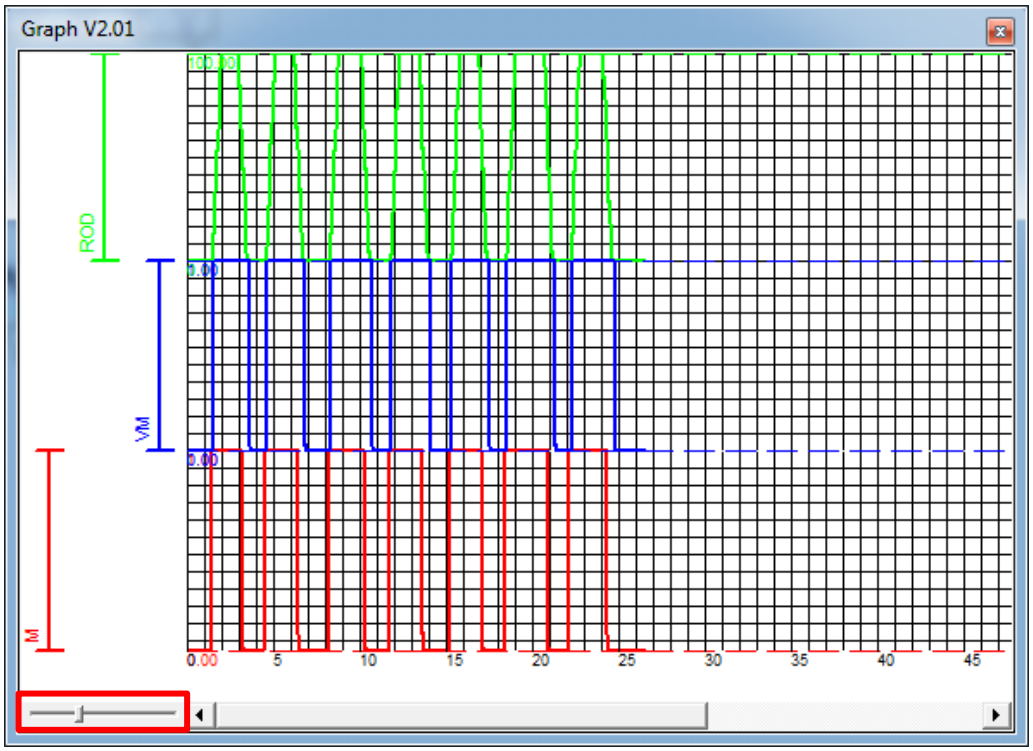
5. Enter a name for this variable (in this case, use the same name as before, **ROD**). For a cylinder, the maximum value is 100 and the minimum is 0 (as the graph displays this value as a percentage).



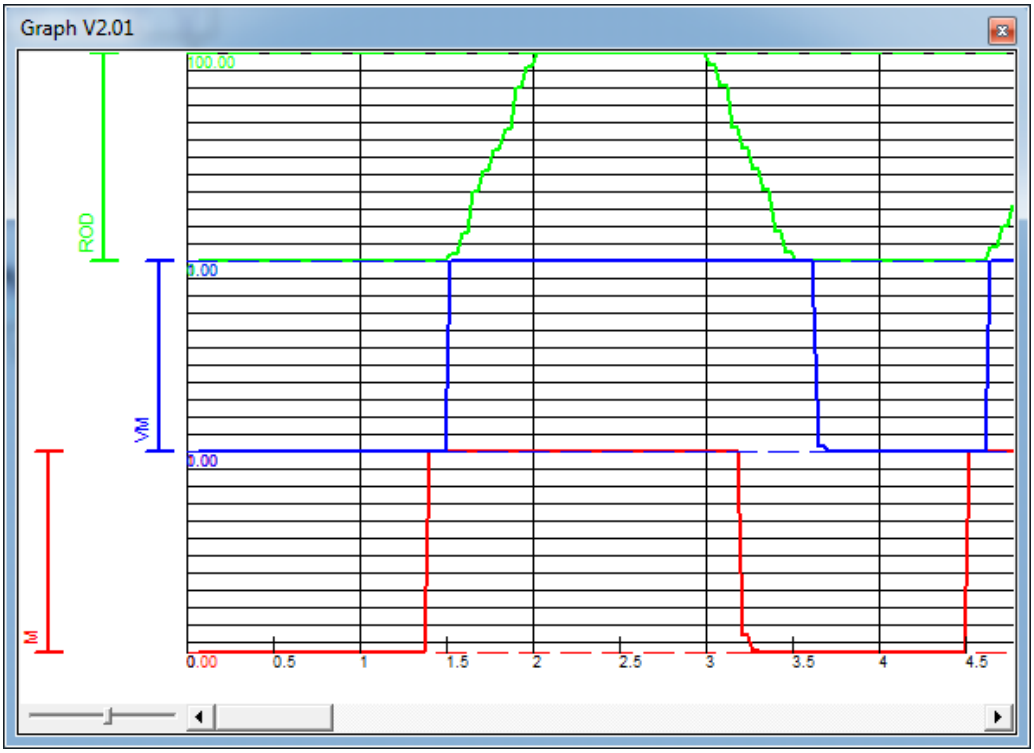
The final result of the graph is shown below, displaying the variables **M**, **VM** and **ROD**.



6. Click on the **'Go!'** button to view the phase-space diagram.

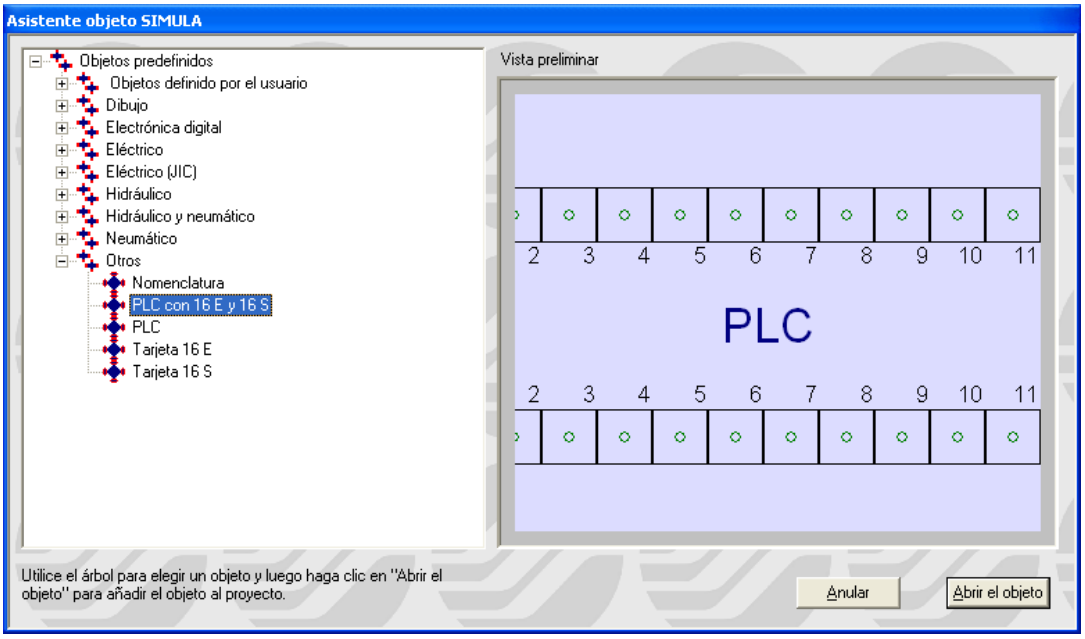


You can view the phase-space diagram in detail by moving the slider located at the bottom left of the graph.

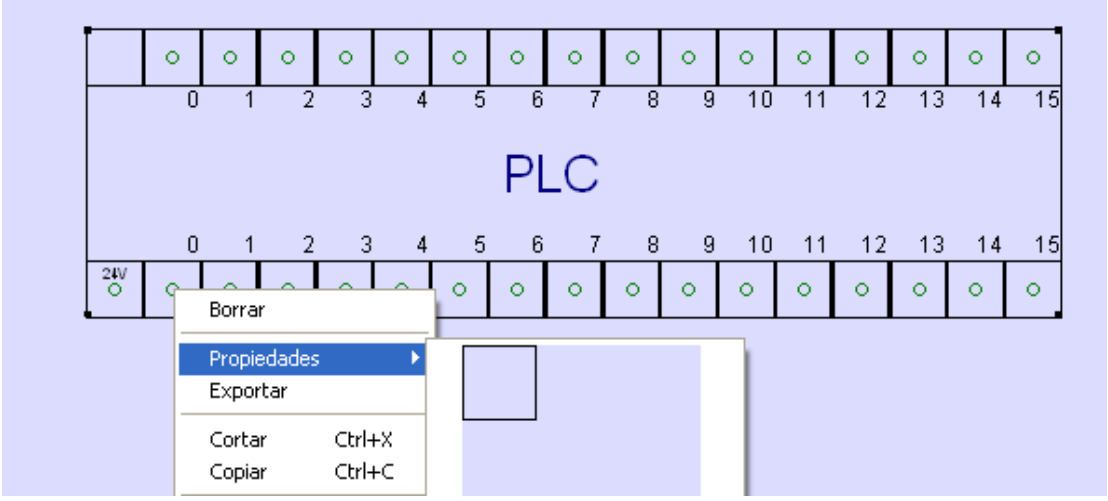


Inserting the Virtual PLC object into SIMULA

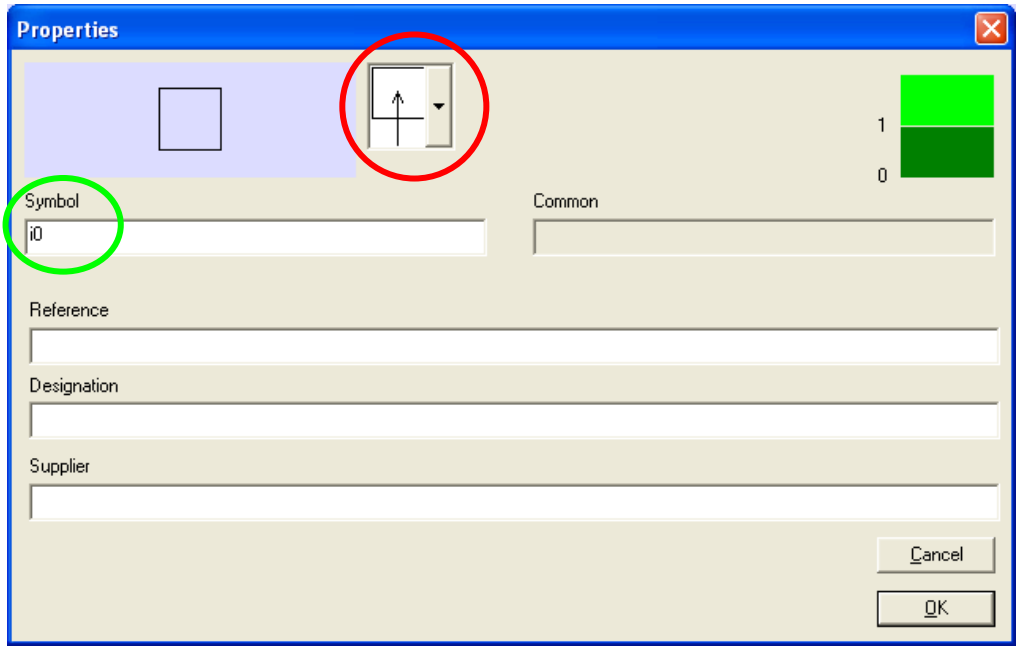
1. The first step is to insert a PLC object into *SIMULA*. In this example, the **PLC** object **with 16E and 16S** has been inserted from the **'Others'** menu.



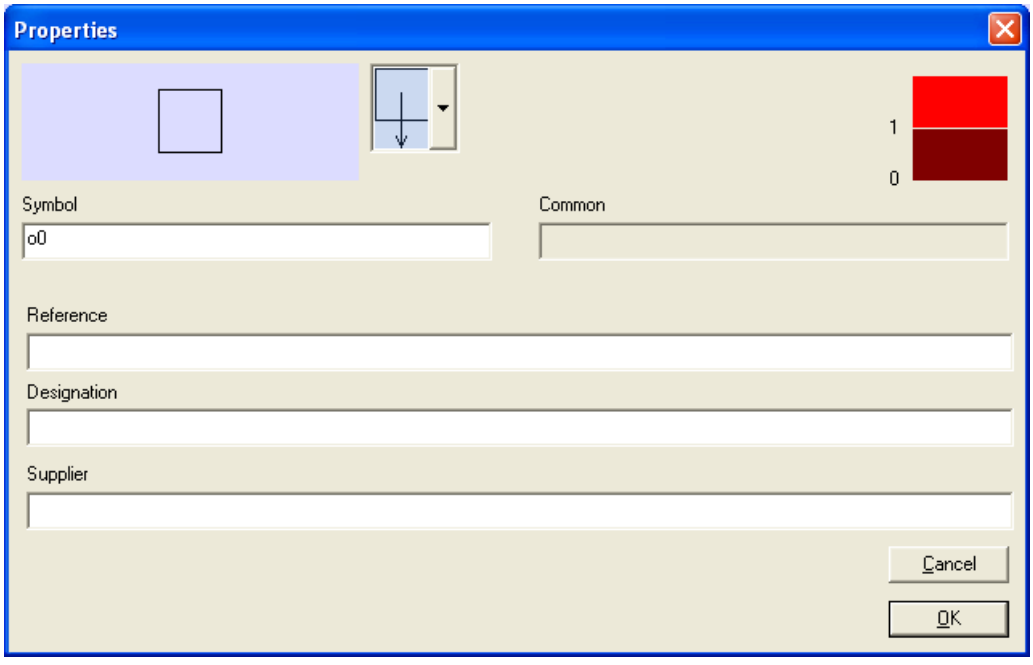
2. Click once with the left mouse button on the PLC object. Once the PLC is selected, hover the mouse over the input or output you wish to define (see figure) and right-click:



3. Select the first of the elements that appear in the **Properties** section. This opens the **Block** element's properties window, which in this case is defined as an input (see red circle in the figure).

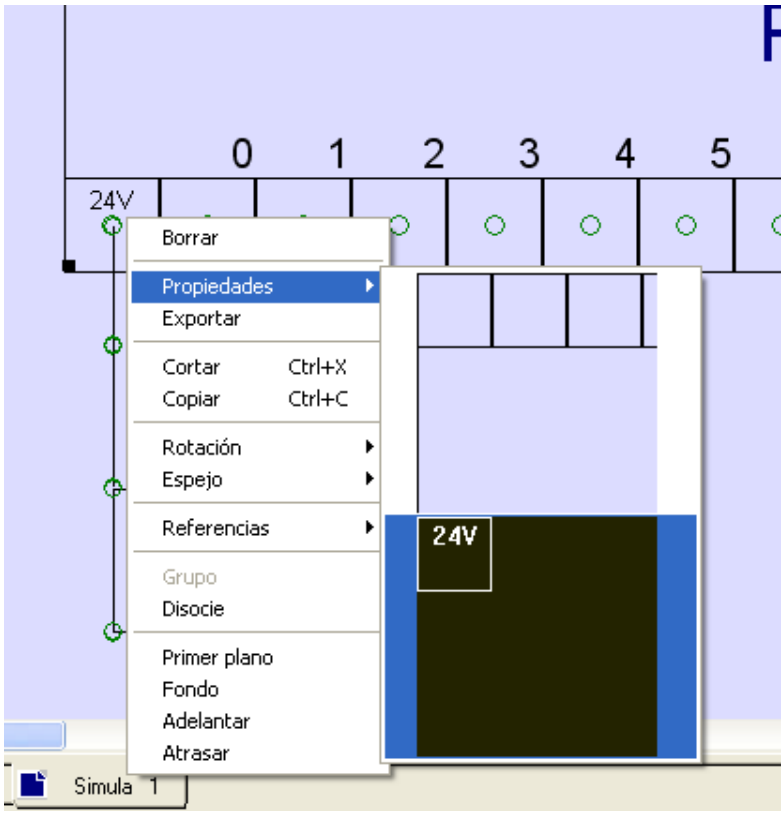


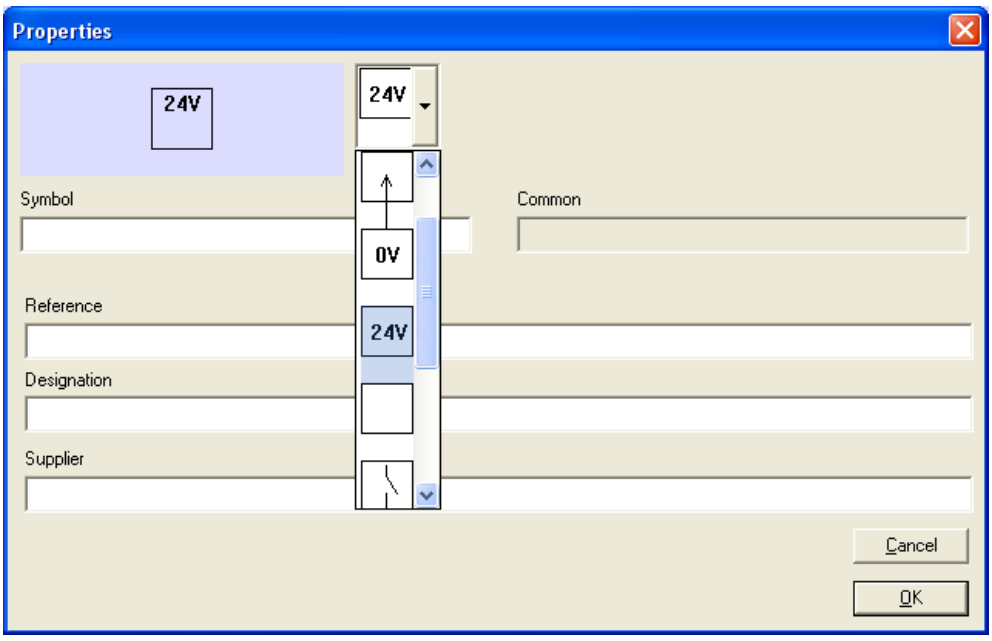
4. Enter a name that will subsequently refer to the PLC input (see the green circle in the figure above). For outputs, the procedure is the same (see the figure below).



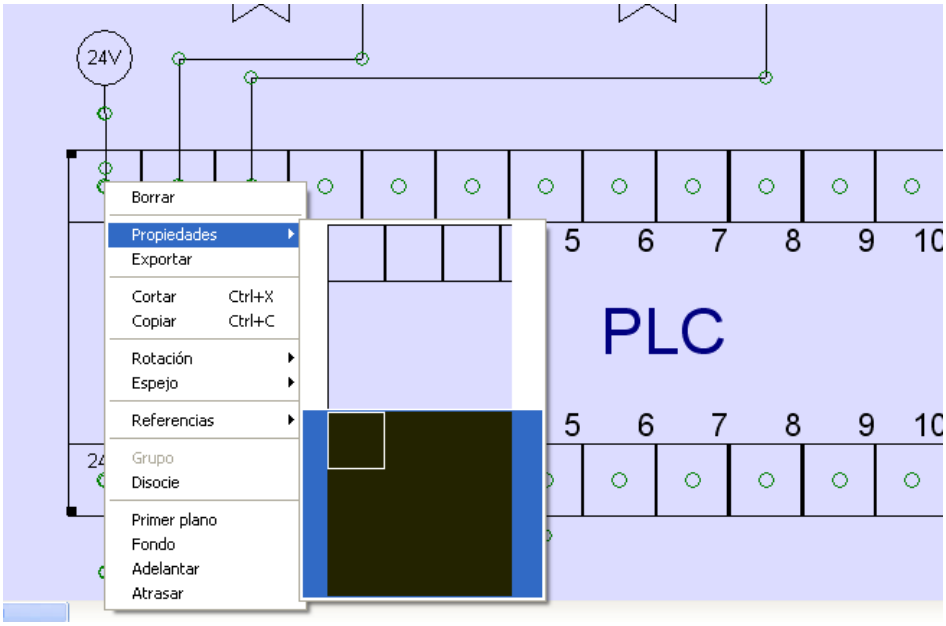
5. Finally, you must enter the common connections for the PLC. The configuration depends on whether they are inputs or outputs.

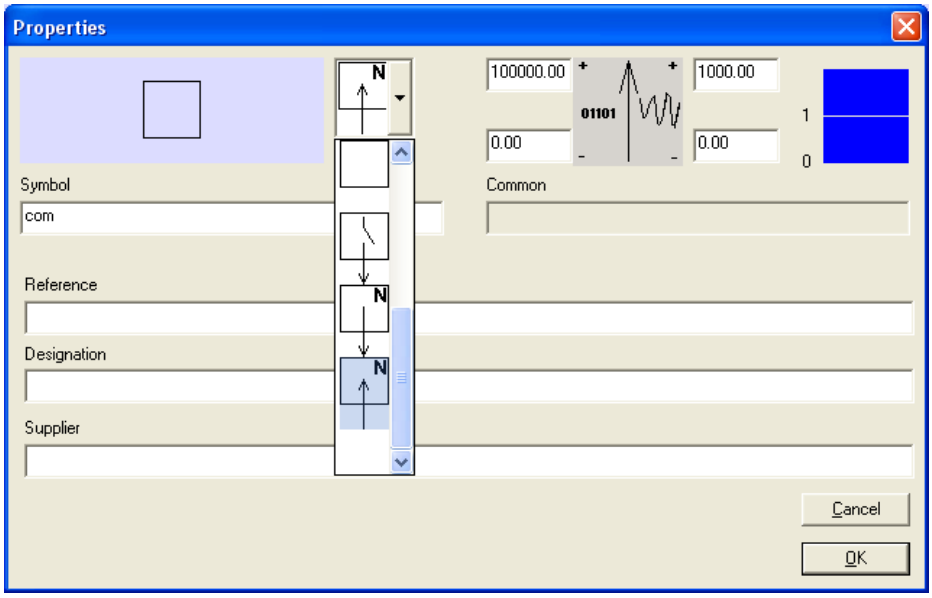
For **inputs,** you need to supply 24 V DC to the PLC so that it can power the contacts. To do this, double-click the block with the right mouse button and select the 24 V element (see figures below):





For the **outputs**, it depends on whether the common terminal is positive (NPN) or negative (PNP). In other words, you must insert a block with the PLC's **common** terminal **property** (leave the default settings in autoSim as they are). If **a positive common (NPN)** is set, the common is supplied with 24 V DC and the PLC outputs go to the solenoid valves, which are connected to 0 V DC. If **a negative common (PNP)** is used, the common is supplied with 0 VDC and the PLC outputs go to the solenoid valves, which are connected to 24 VDC (see figures). In this example, a positive common (NPN) connection has been used:

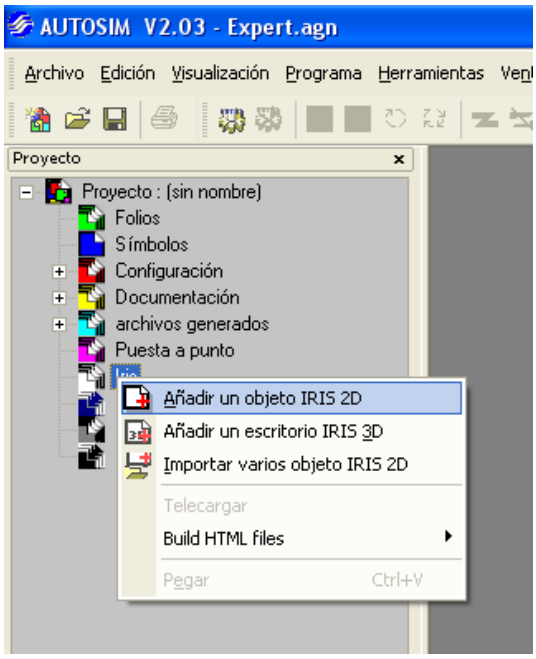




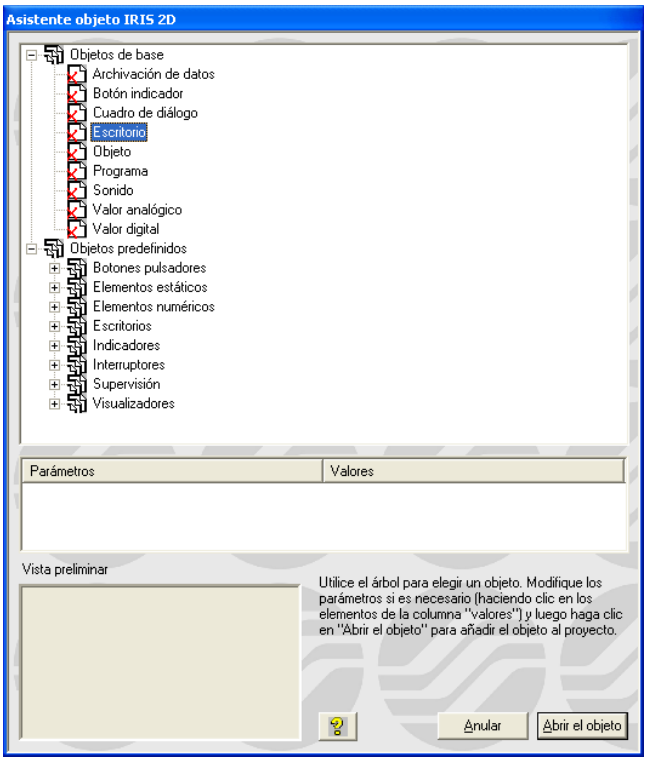
Once the PLC has been defined in the SIMULA section, it can be programmed using *Ladder*, *Grafcet* or *Logigramme* in the PROGRAMAS programming section.

IRIS 2D project example

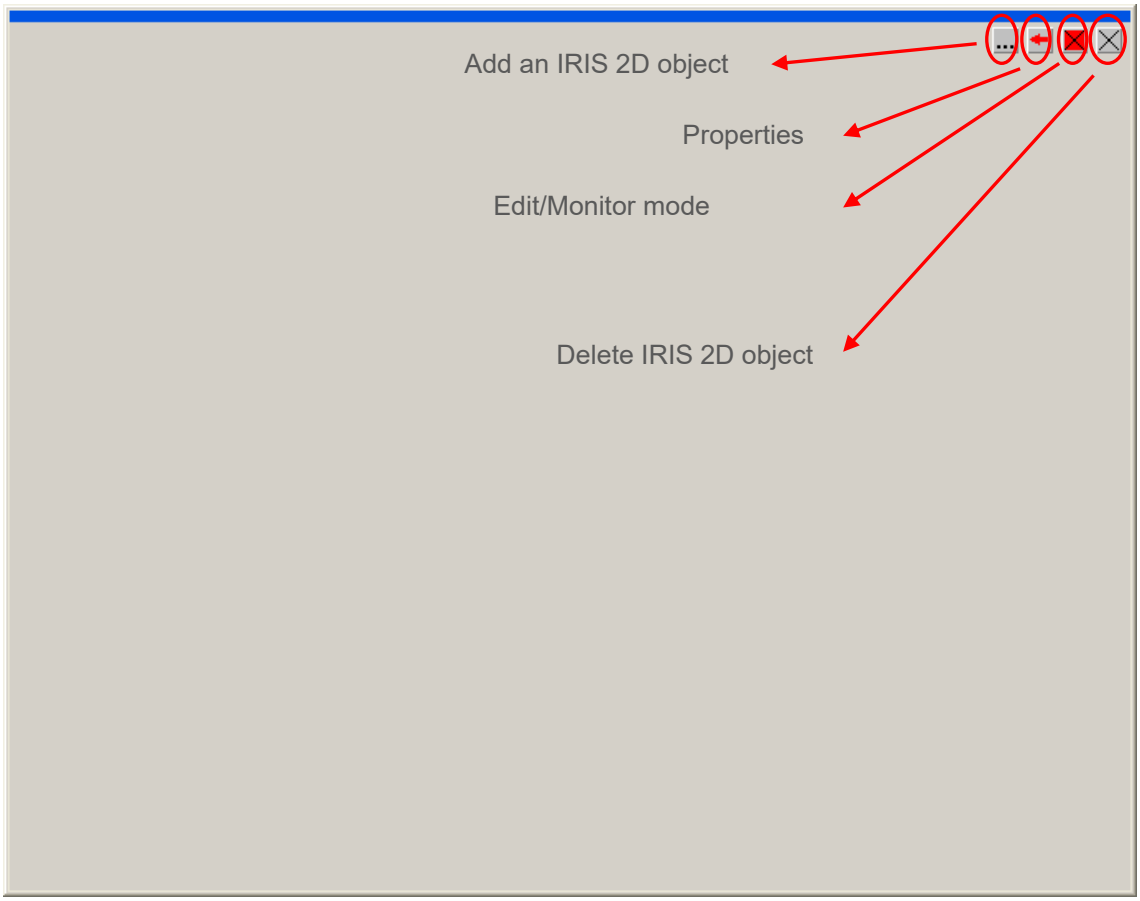
1. Create the IRIS 2D console. To do this, right-click on ***'Iris'*** and select the '**Add an IRIS 2D object**' option.



2. Add the **Desktop** object from the **Basic Objects** menu.



The Desktop object comprises the following options:

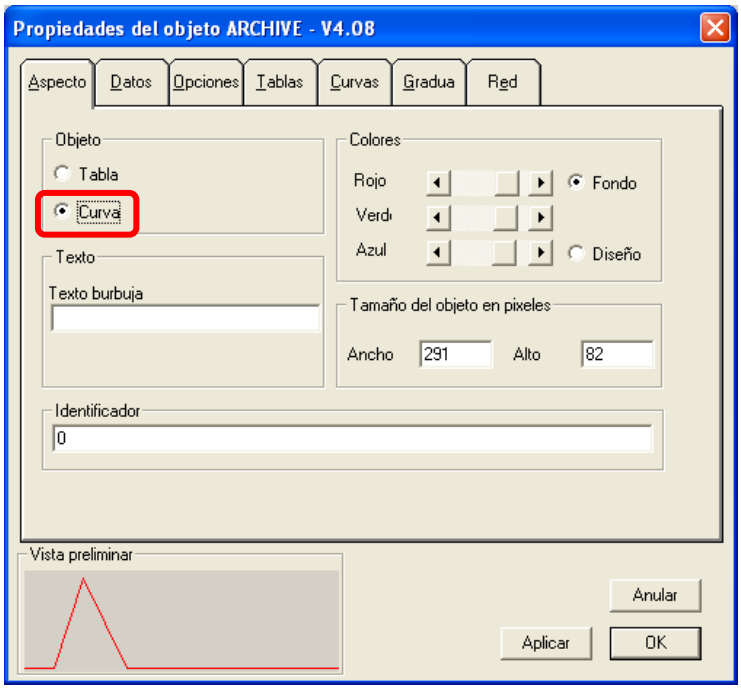


- **Add a 2D IRIS object.** The 2D IRIS library opens, allowing you to select any 2D IRIS object you wish to add to this workspace.
- **Properties.** The object's properties window opens, allowing the user to modify the IRIS 2D object's settings: appearance, size, perform actions, etc.
- **Edit/Monitoring mode.** Toggles the display of the IRIS 2D object between edit mode and monitoring mode (you can also right-click on the object to switch between these modes).
- **Delete IRIS 2D object.** Deletes the IRIS 2D object from the project.

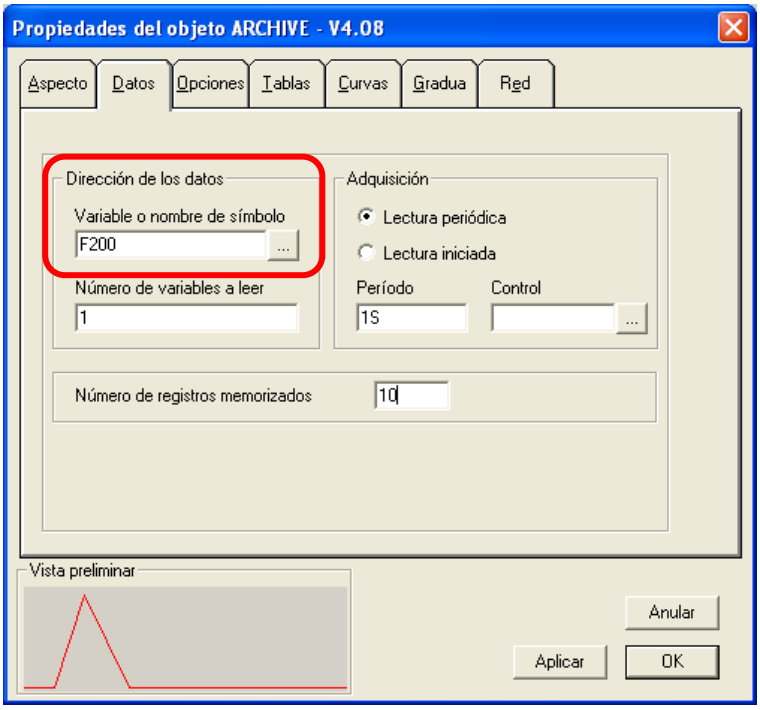
In this example, a graph is created to show the values of a variable, which increases when one button is pressed and decreases when another button is pressed. Furthermore, this value is shown on a *display*.

3. Add the graph to the IRIS 2D desktop. This object is defined under the name **'Data** Archiving' in the **'Basic Objects'** menu.
4. Open the object's **Properties** to modify its appearance and characteristics (you must select the **'Curve'** option).

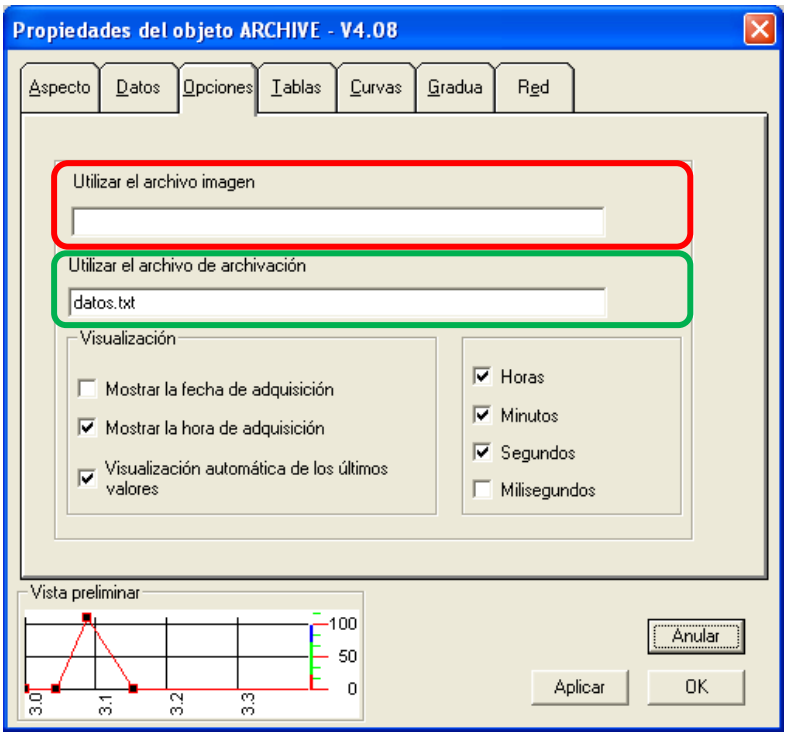
On the **Appearance** tab, you can change the appearance of this object: Table or Curve. You can also change the colour and size of the object.



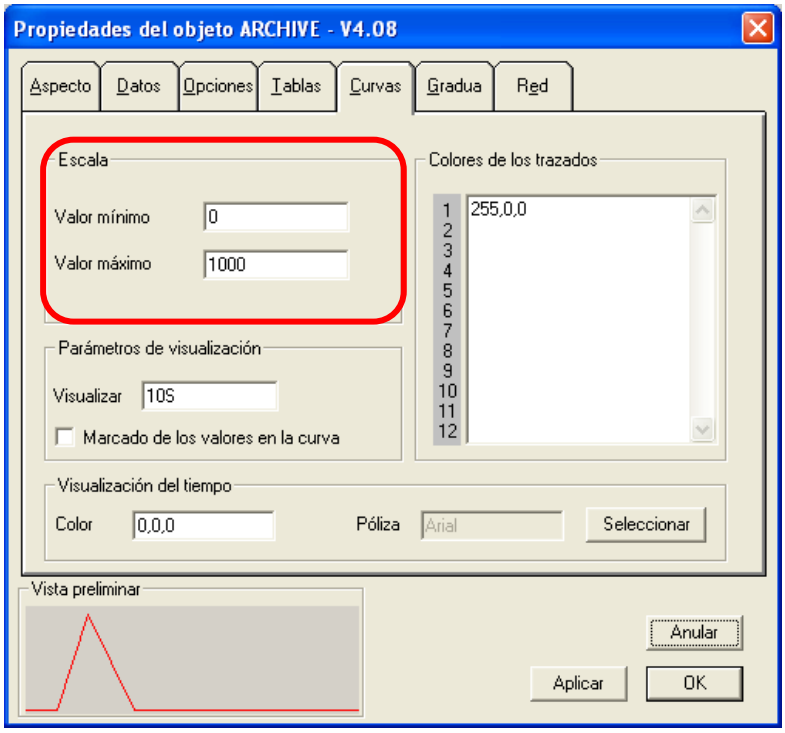
On the **Data** tab, you can select the memory address to be plotted on the graph (in this case, select the *float* variable F200).



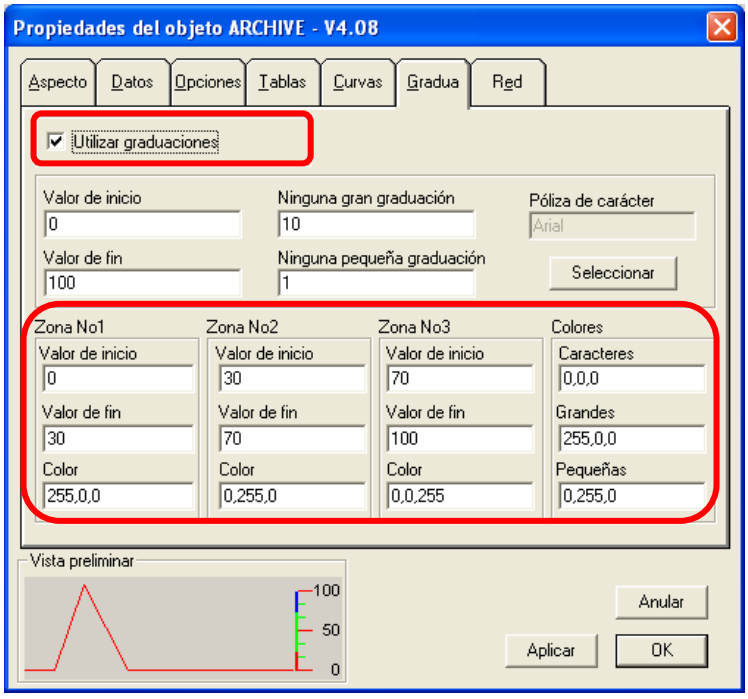
On the **Options** tab, you can select the file from which the data is read (highlighted in red), the file in which the data is saved (highlighted in green) and various options, such as displaying the date and time:



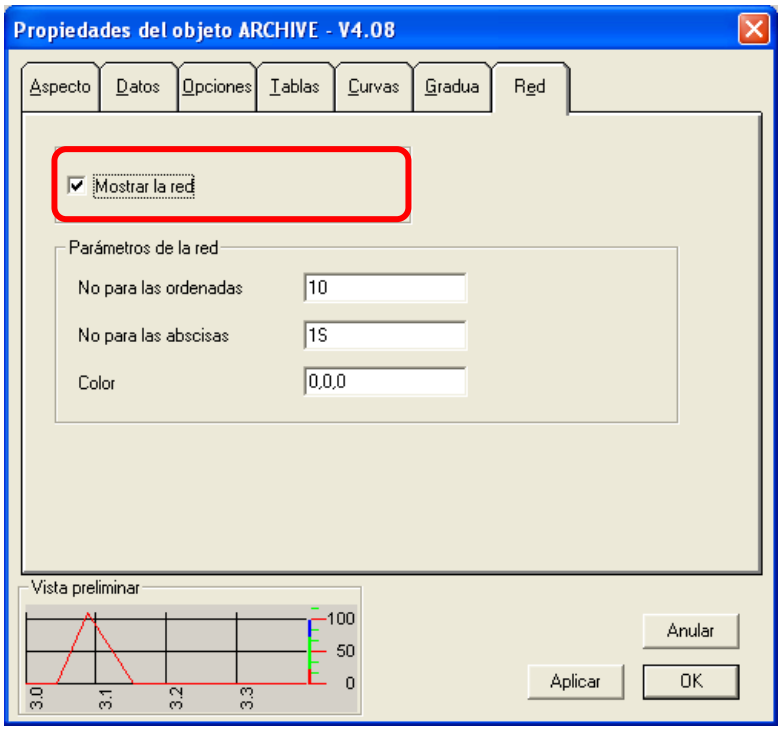
On the **Curves** tab, you can define the scale of the values to be displayed (the **Tables** tab is used when the **Table** option is selected for data collection):



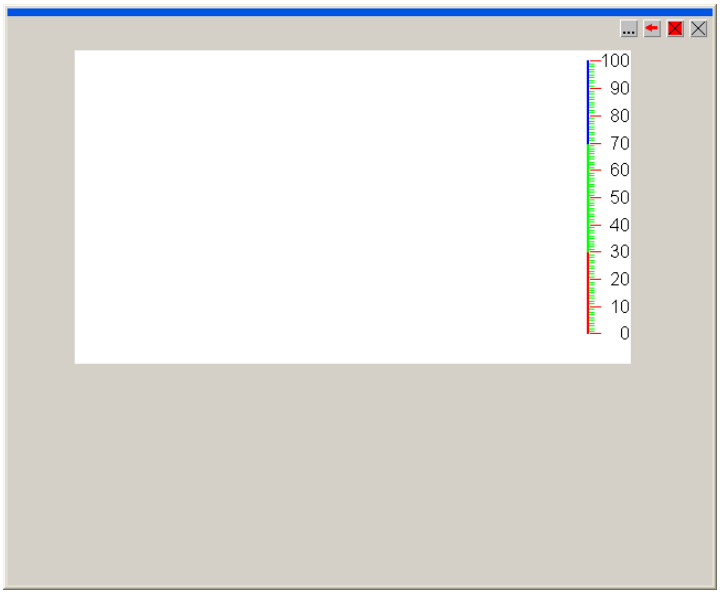
On the **Graduation** tab, you can define the colour of the graduations; in other words, you can specify different colours to be displayed on the scale graduations.



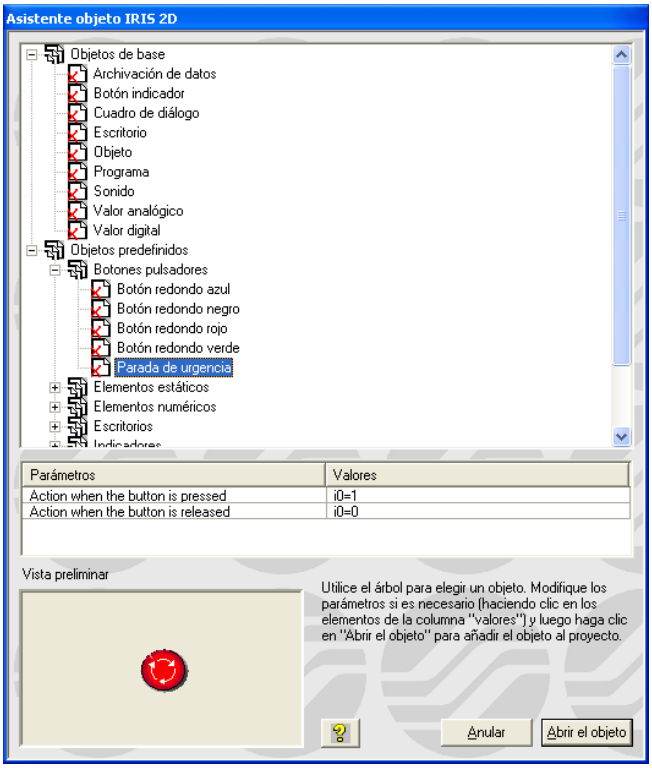
The **Grid** tab allows you to enable or disable a grid to be displayed on the curve.



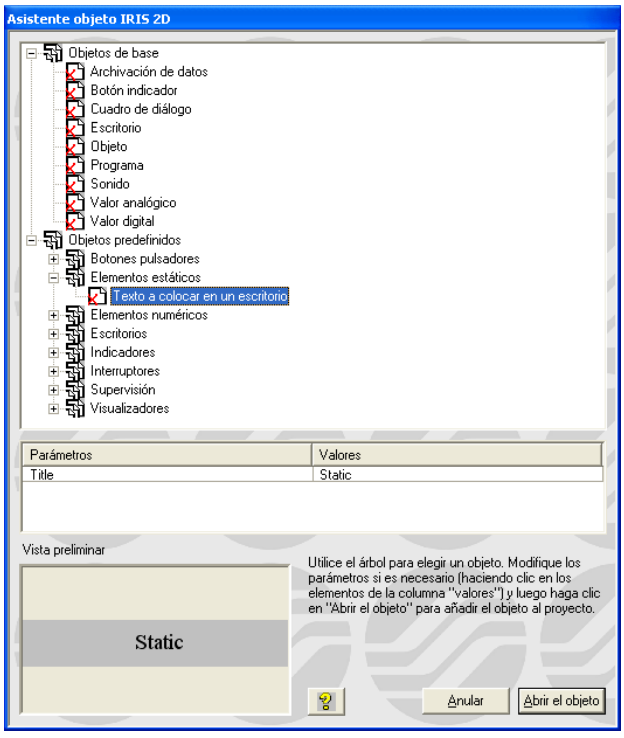
The image below shows what the resulting graph looks like.



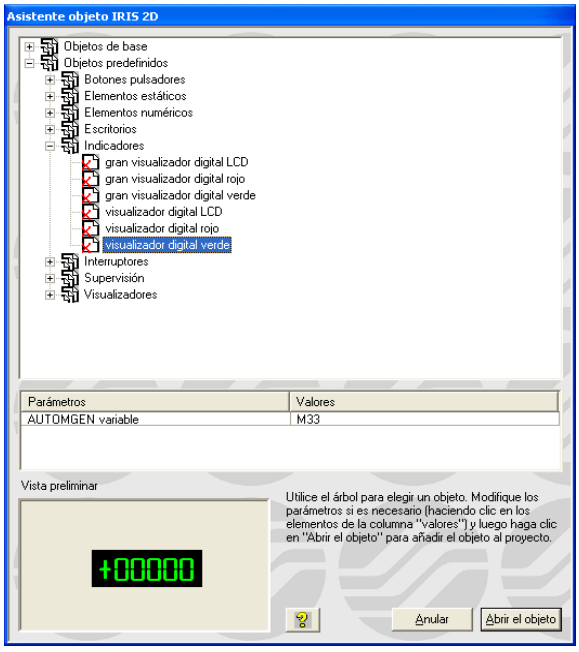
5. Add the control buttons to the IRIS 2D desktop. Select the **Emergency Stop** object from the **Predefined Objects/Push Buttons** menu.



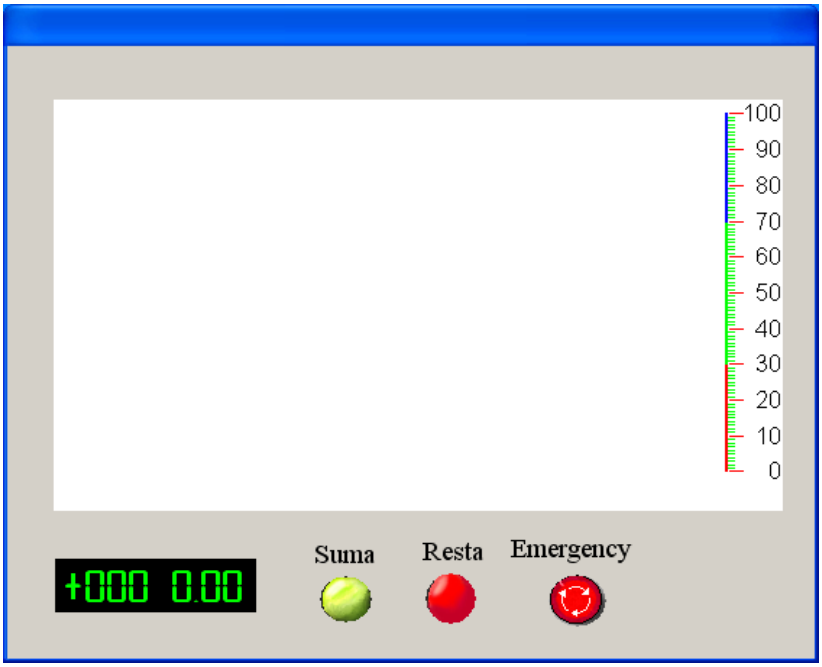
6. Add labels, in the form of text, to identify each of the buttons. Select the **'Text'** object **to place on a desktop** from the **'Predefined Objects/Static Elements'** menu.



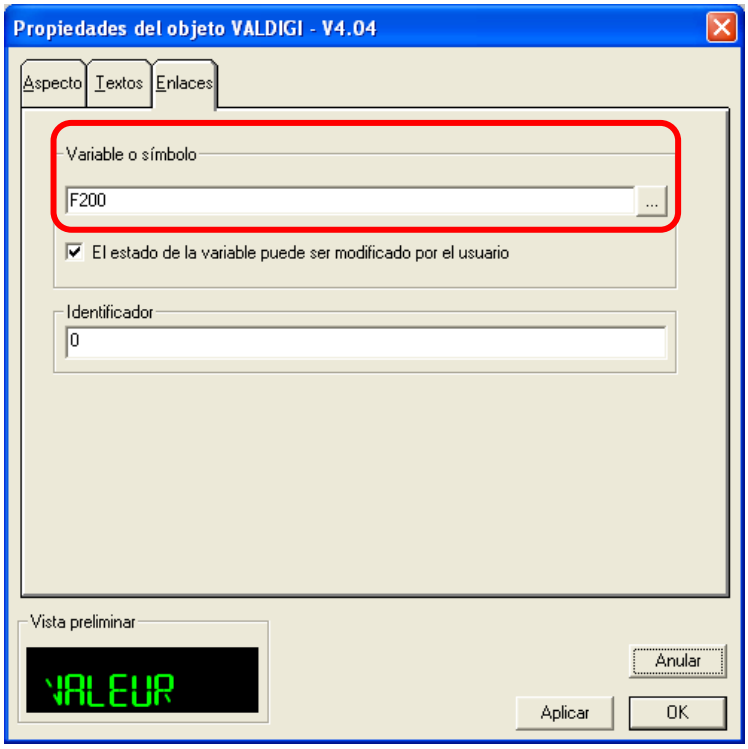
7. Insert the *display* object where the value of the curve variable (F200) is shown. Select the **'Green Digital Display'** object from the **'Predefined Objects/Indicators'** menu.



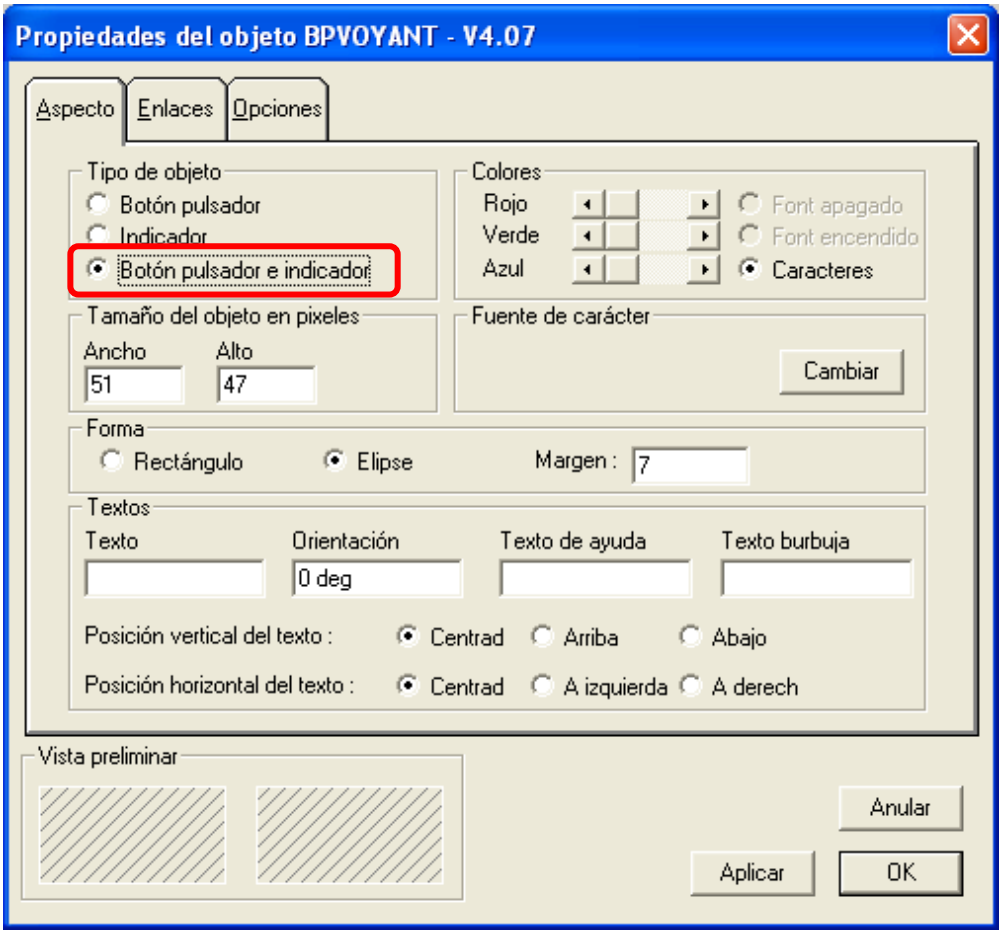
The final appearance of the IRIS 2D desktop is as follows:



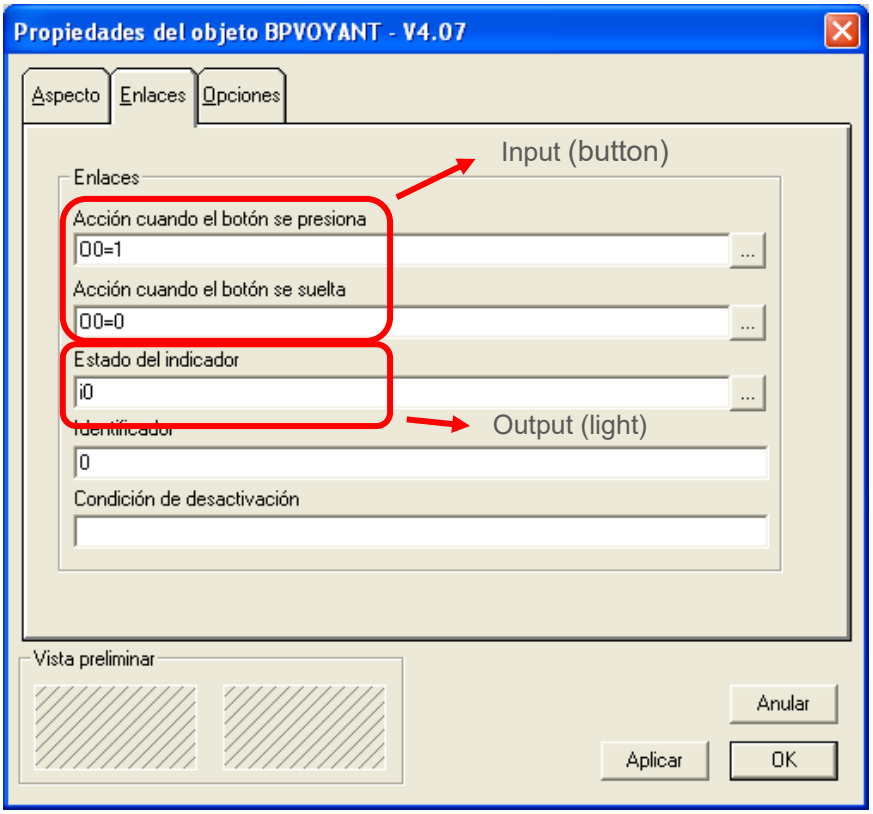
8. Synchronise the curve variable with the *display* variable. To do this, enter the memory address F200 in the **Links** tab of the *display* object's properties (boxed in red):



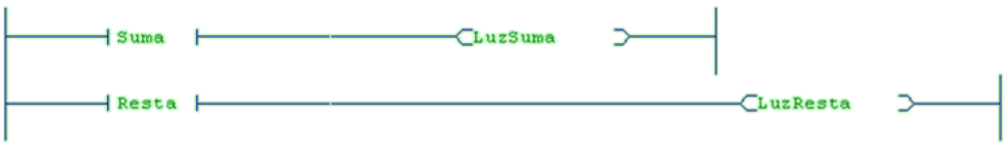
9. Define the properties of each of the buttons. Select the **'Pushbutton and indicator'** option under **'Object type'** in the **'Appearance'** menu.

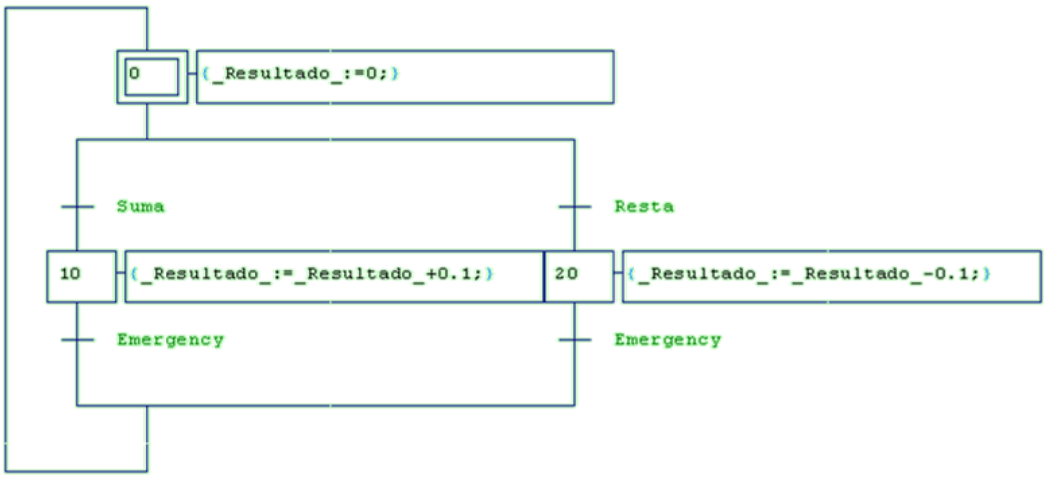


10. Synchronise the IRIS 2D buttons with the memory addresses of the programme that controls the sequence of the current project. Go to the button **Properties** and enter the memory addresses in the fields **'Action when button is pressed**', **'Action when button is released'** and **'Indicator** status' within the **Links** menu (see image).



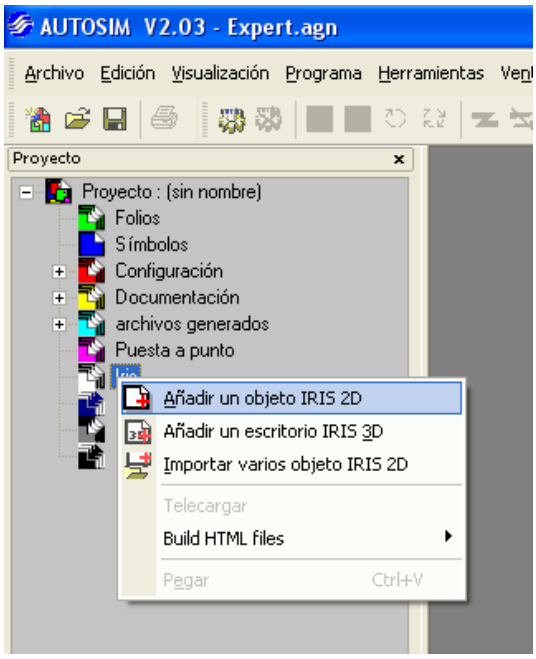
11. Write the control programme. You can choose between different programming languages: **Grafcet**, **Ladder**, **Logigramme** or **Structured Text**. In this example, the programme has been written in Grafcet and Ladder.



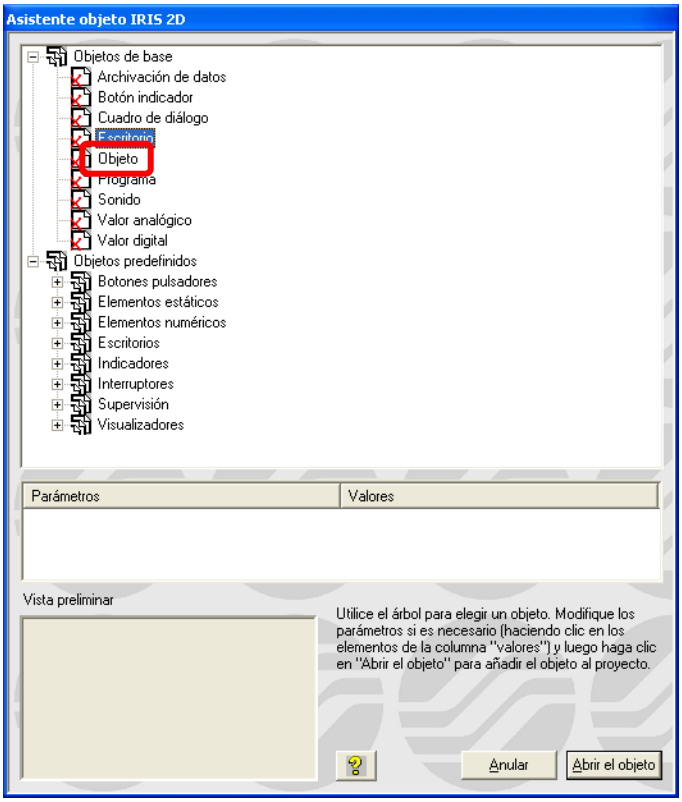


Apply a linear movement to an IRIS 2D object

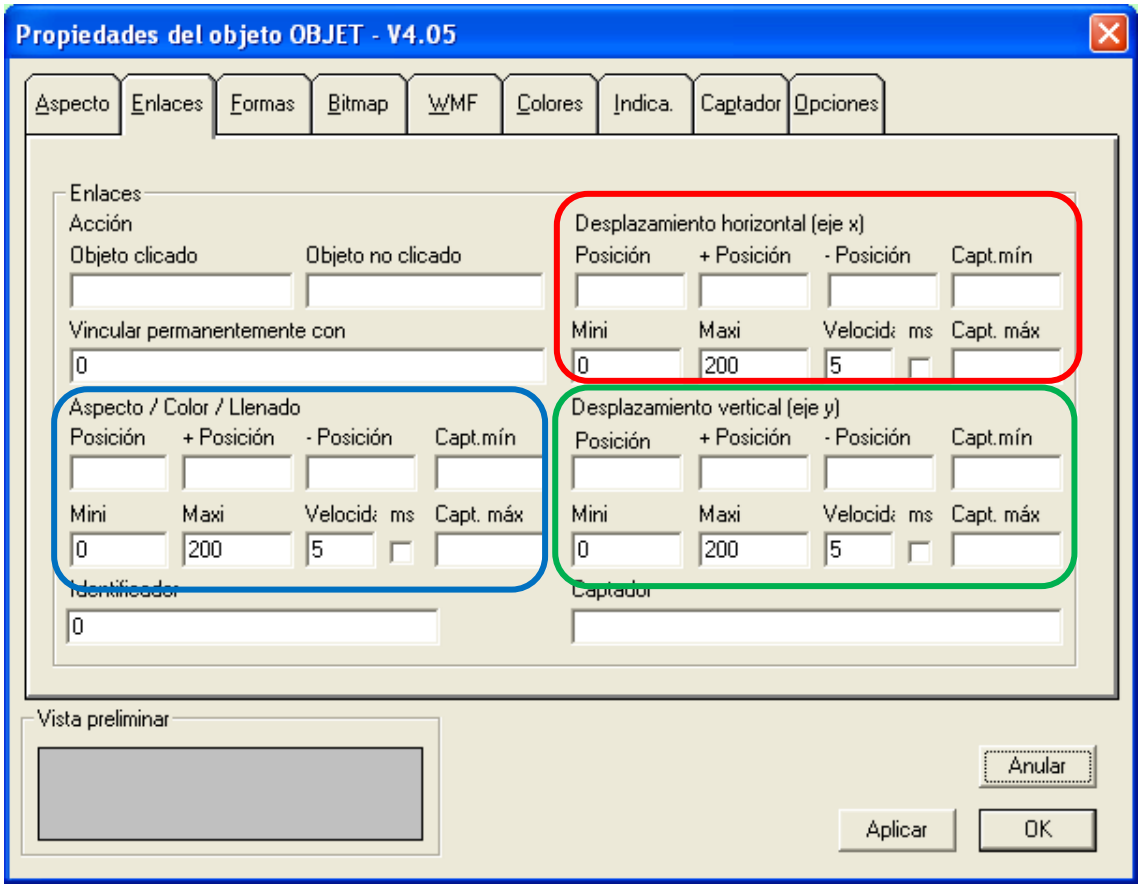
1. Create the IRIS 2D console. To do this, right-click on ***Iris*** and select the option **'Add an IRIS 2D object**'.



2. Add the **'Object'** from the **'Basic Objects**' menu.



3. Go to the object's properties and open the **Links** tab.



- **Appearance / Colour / Fill (boxed in blue).** This is used to change the colour of the object, for example, a line extending like a cylinder or a fluid reservoir being filled.
- **Horizontal displacement (X-axis) (highlighted in red).** This is used to move the object horizontally along the X-axis.
- **Vertical displacement (Y-axis) (highlighted in green).** This is used to move the object vertically along the Y-axis.

The meaning of each field is detailed below:

- **Position.** This is the level or position of the object (a numerical memory address must be entered, as this field represents an analogue input).
- **+Position.** Activation of filling or movement in the positive direction (+). A Boolean memory address must be entered. When this variable is enabled (TRUE), the object begins to fill or move from the initial position (**Mini)** to the final position (**Maxi**).
- **-Position.** Activation of filling or movement in the positive direction (-). A Boolean memory address must be entered. When this variable is set to TRUE, the object begins to empty or move from the final position (**Maxi)** to the initial position (**Mini**).
- **Mini.** Initial position of the object (a constant must be entered).
- **Maxi.** Final position of the object (a constant must be entered).

- **Capt min.** Start position detector. A Boolean memory address must be entered; this is set to TRUE when the object is at this start position.
- **Capt max.** End position detector. A Boolean memory address must be entered, which is set to TRUE when the object is at that end position.
- **Speed.** This is the speed at which the object moves (a constant must be entered).

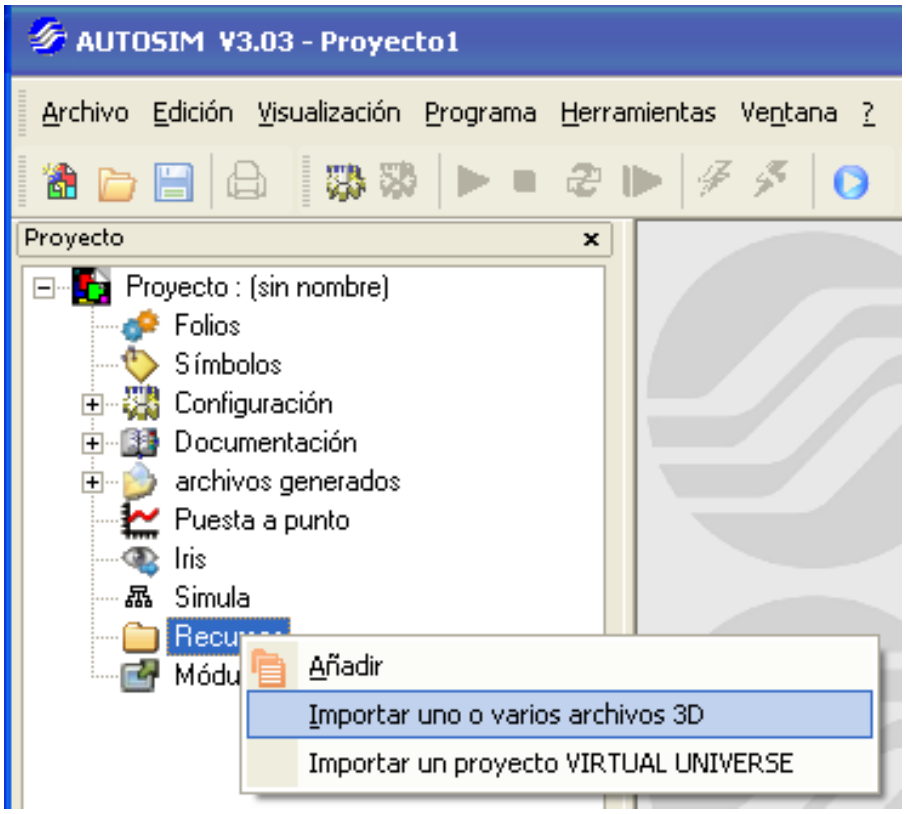
Importing 3D drawings into autoSIM

autoSIM is capable of importing 3D CAD drawings created using two of the most common software programmes: 3DStudio and SolidWorks.

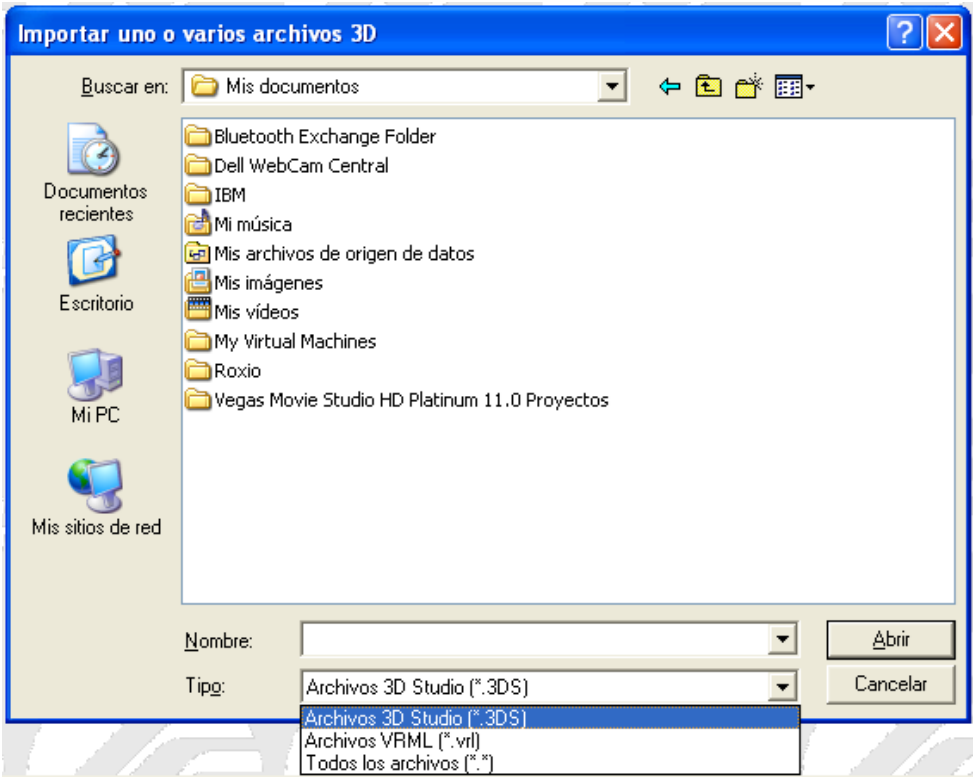
The format used by autoSIM by default for handling this type of file is '.X', defined by Microsoft DIRECTX 8.

However, as the format used by autoSIM is not the most common, autoSIM includes a format converter capable of converting '.3DS' (3DStudio) and '.wrl' (SolidWorks, save the file in VRML format) files to '.X' (DIRECTX 8).

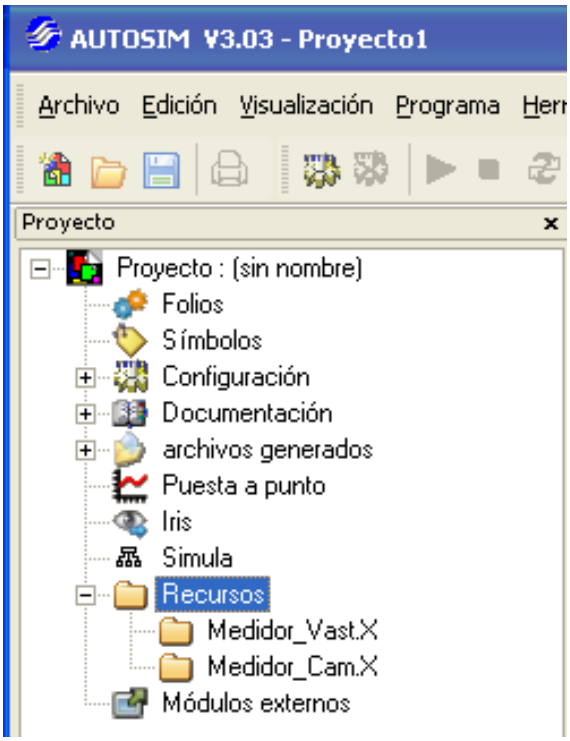
1. To import the drawing into autoSIM, right-click on **'Resources'** and select the option **'Import one or more 3D files**'.



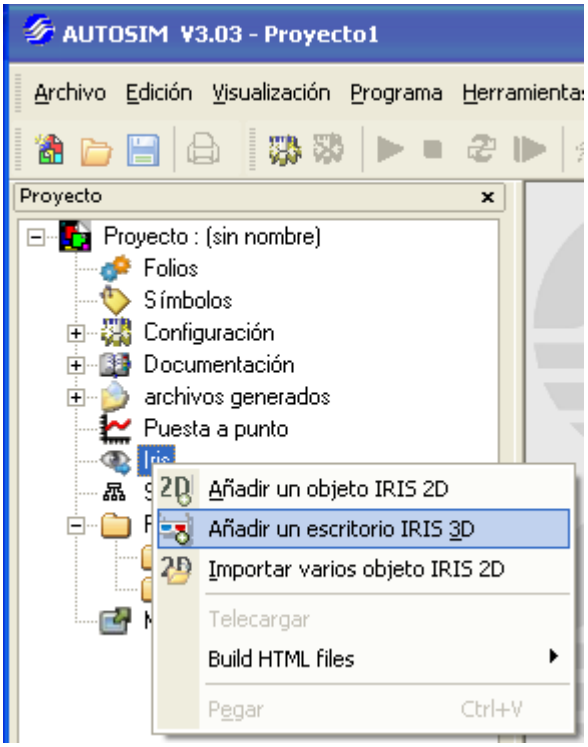
2. Select the type of drawing file you wish to import (**SolidWorks** – *file*.wrl – or **3DStudio** – *file*.3ds –).



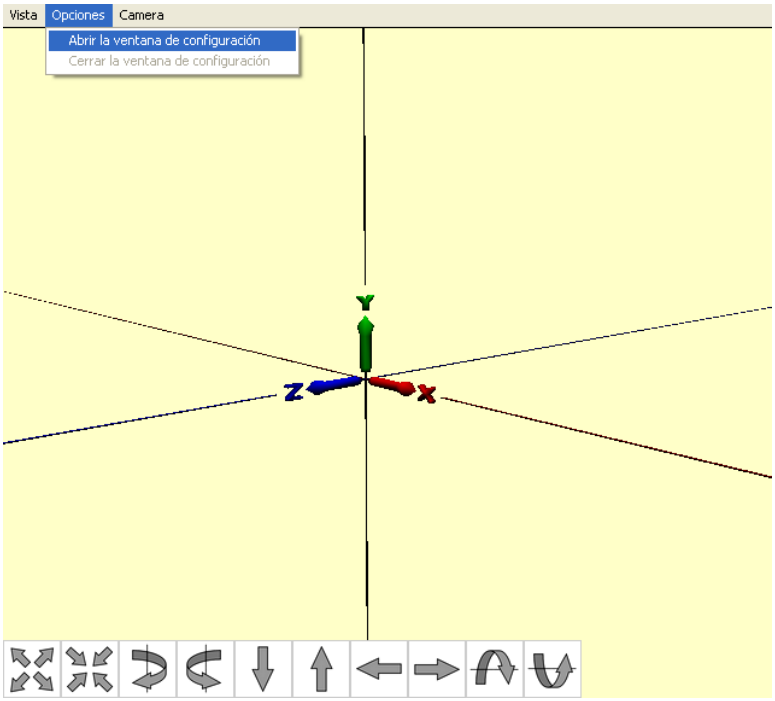
3. Locate the 3D file in the folder where it was saved and click the **Open** button. The 3D drawings have been added to the project.



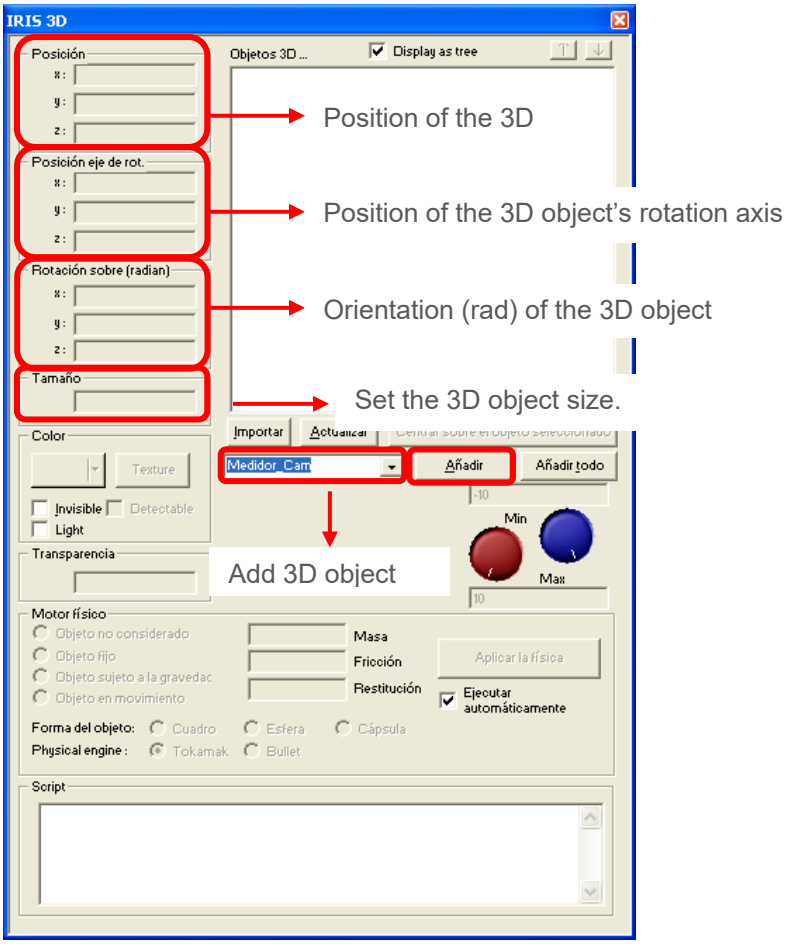
4. **Add an IRIS 3D desktop** to the project.



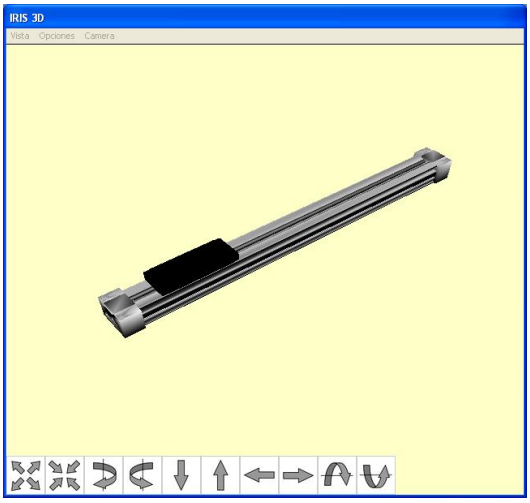
5. Open the IRIS 3D desktop settings window. To do this, go to **Options/Open Settings Window**.



6. From the configuration window, you can insert the models into the desktop and modify their properties. Select the model and click **Add** to import it into the project:



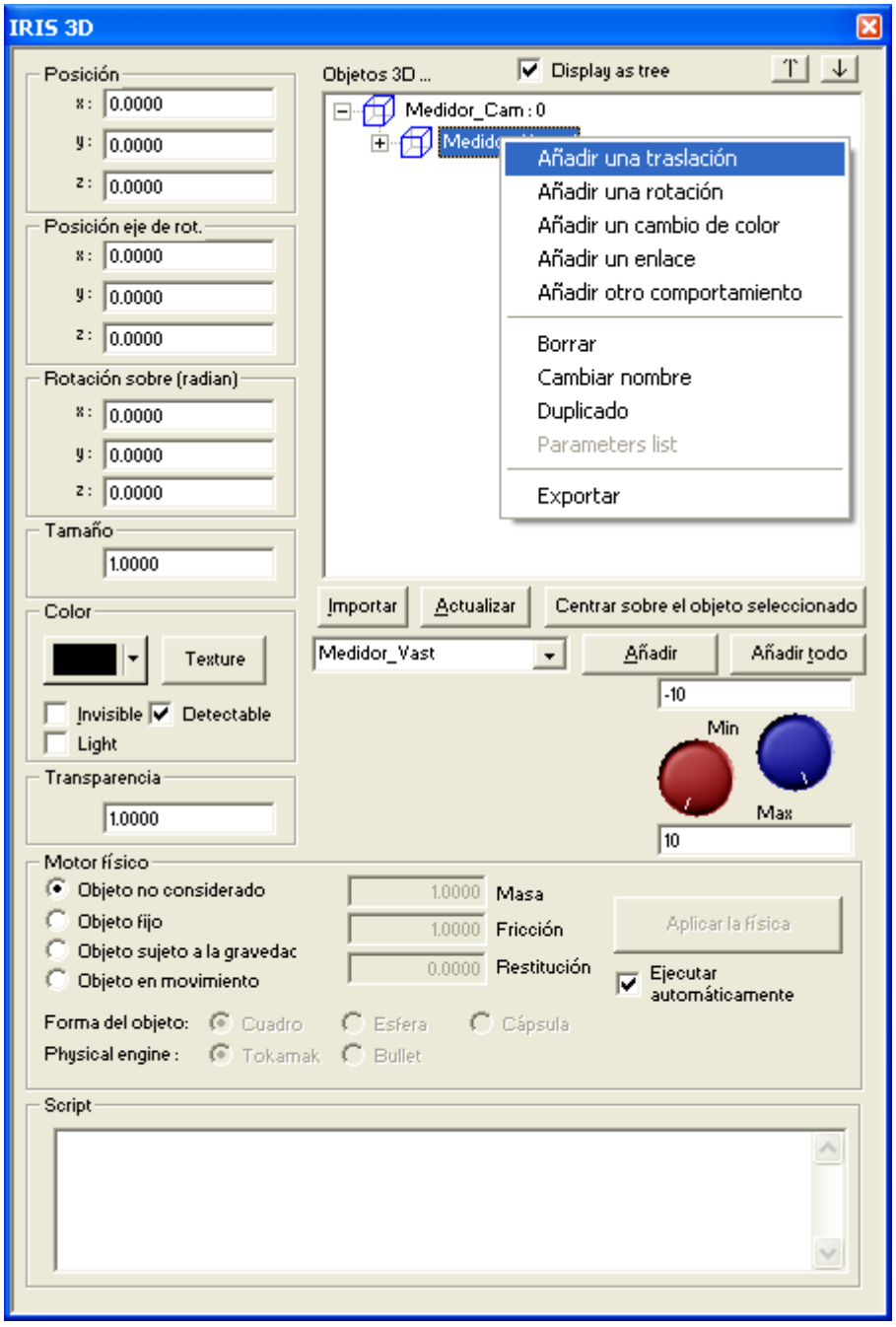
7. Once added, the 3D model is displayed on the IRIS 3D desktop.



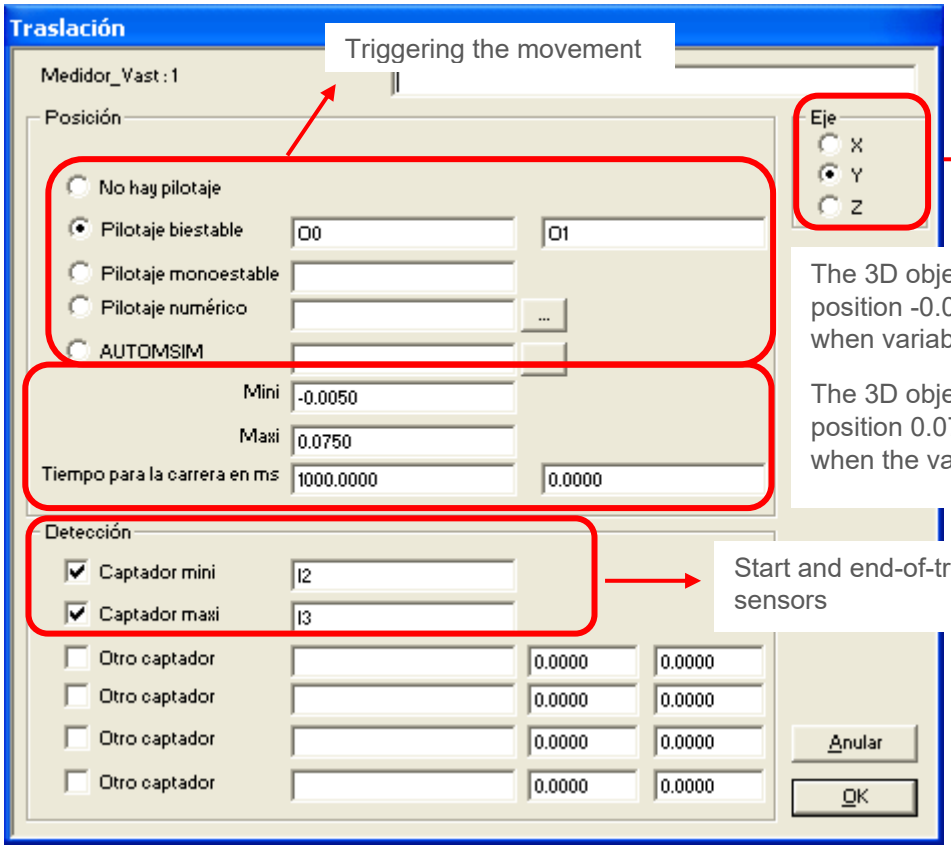
Once all the 3D models have been added to the project and positioned as required, you can add actions such as translation, rotation and colour changes, etc.

Adding an action to a 3D object:

1. Select the 3D object and right-click on it (see figure). Select **'Add a translation'** to perform a linear movement along a desired axis.



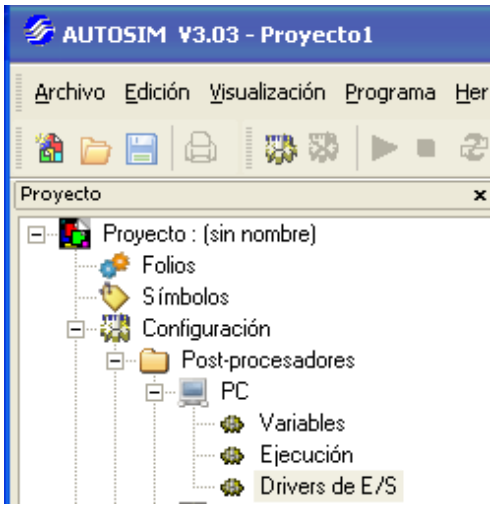
In this example, a rodless cylinder has been added using two models, *Medidor_Cam.3ds* and *Medidor_Vast.3ds*, where one acts as the sleeve (fixed part) and the other as the table (moving part). Therefore, a translation movement is applied **only** to the moving part. This movement can be actuated via a monostable or bistable valve. Furthermore, limit switches can be added to detect when the part reaches the start or end position.


Traslación
Triggering the movement
Medidor_Vast : 1
Posición
No hay pilotaje
Pilotaje biestable
O0
O1
Pilotaje monoestable
Pilotaje numérico
AUTOMSIM
Mini
-0.0050
Maxi
0.0750
Tiempo para la carrera en ms
1000.0000
0.0000
Eje
X
Y
Z
Direction of movement
The 3D object will move from position -0.005 to 0.075 in 1000 ms when variable O0 is set to 1.
The 3D object will move from position 0.075 to -0.005 in 1000 ms when the variable O1 is set to 1.
Detección
Captador mini
I2
Captador maxi
I3
Start and end-of-travel sensors
Otro captador
Otro captador
Otro captador
Otro captador
0.0000
Anular
OK

Communication with the PN4443 data acquisition card

autoSIM allows a physical connection to a data acquisition card. The model number of this card is **PN4443**. The configuration process is as follows:

1. Double-click on **Configuration/Post-Processors/PC/I/O Drivers**.



2. Select the **PN4443** card driver.

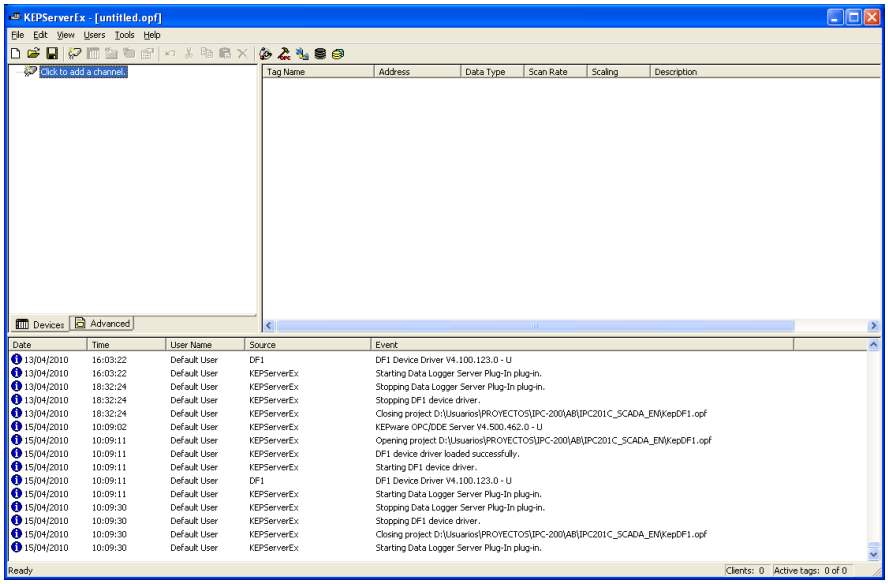
3. It is not necessary to configure the driver as this is done automatically via autoSIM. The mapping of autoSIM variables to the card's inputs and outputs is as follows:

autoSIM-200	SAI2443
I0-%I0	I0
I1-%I1	I1
I2-%I2	I2
I3-%I3	I3
I4-%I4	I4
I5-%I5	I5
I6-%I6	I6
I7-%I7	I7
autoSIM-200	SAI2443
O0-%Q0	O0
O1-%Q1	O1
O2-%Q2	O2
O3-%Q3	O3
O4-%Q4	O4
O5-%Q5	O5
O6-%Q6	O6
O7-%Q7	O7

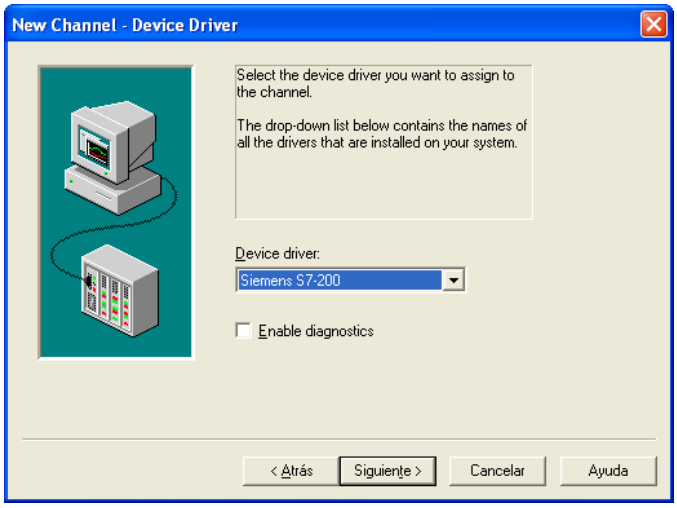
Communication with the KEPware KEPServer OPC server

autoSIM enables the exchange of signals with an external OPC server. The OPC server used in this example is ***KEPware KEPServer EX***. To establish this communication, follow the procedure below:

1. Open the **KEPware** OPC server and create a new project with the following characteristics.

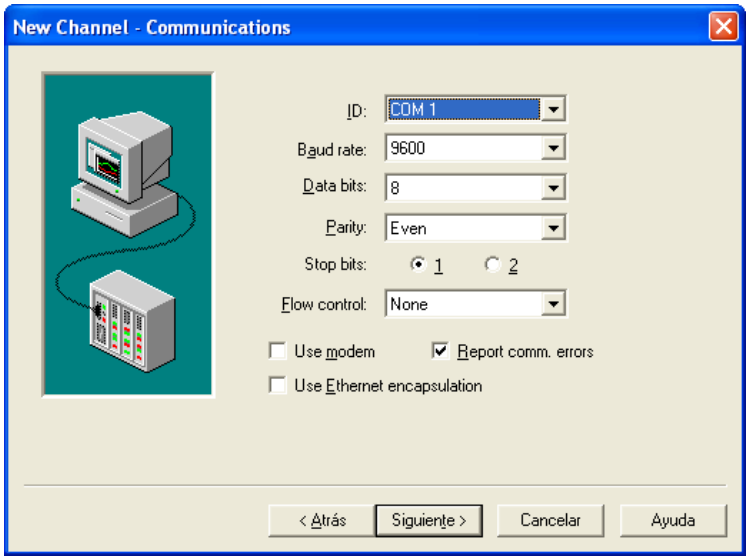


2. Click on '**Click to add a channel**'. The following window appears.

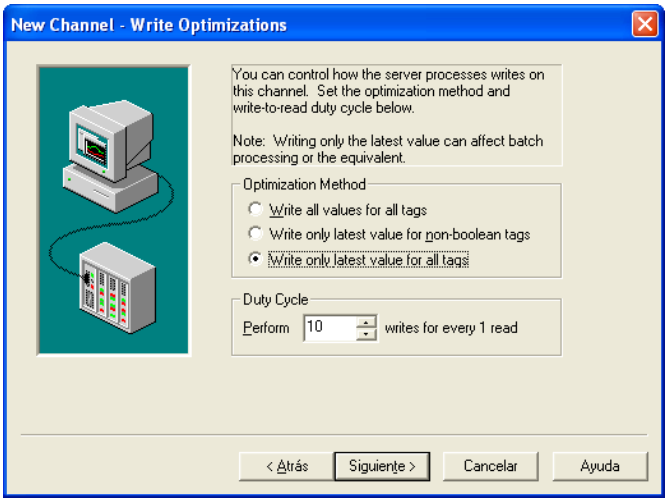


In this document, we will be establishing a connection between autoSIM and a Siemens PLC from the S7-300 family, so select the **Siemens S7-300** driver. If you wish to connect to another PLC, you will need to select a different driver (for a full list of options, see the link: http://www.kepware.com/Products/products_OPCServers.asp).

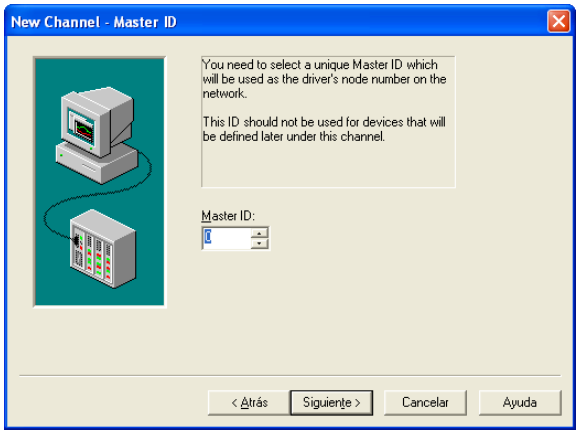
3. Click **Next** to continue. Another window appears.



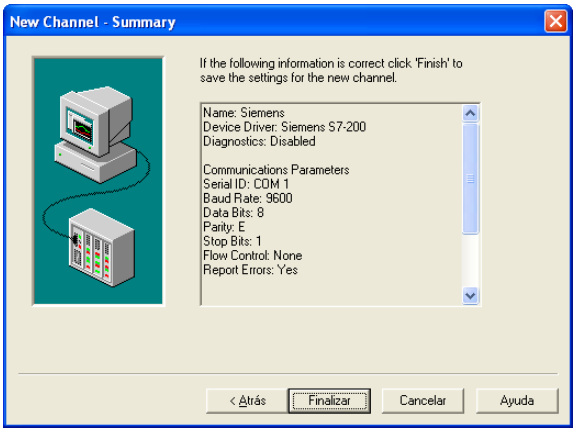
4. Here, you must select the serial port to be used for communication. In this case, the default settings will be used. Click **Next** to continue.



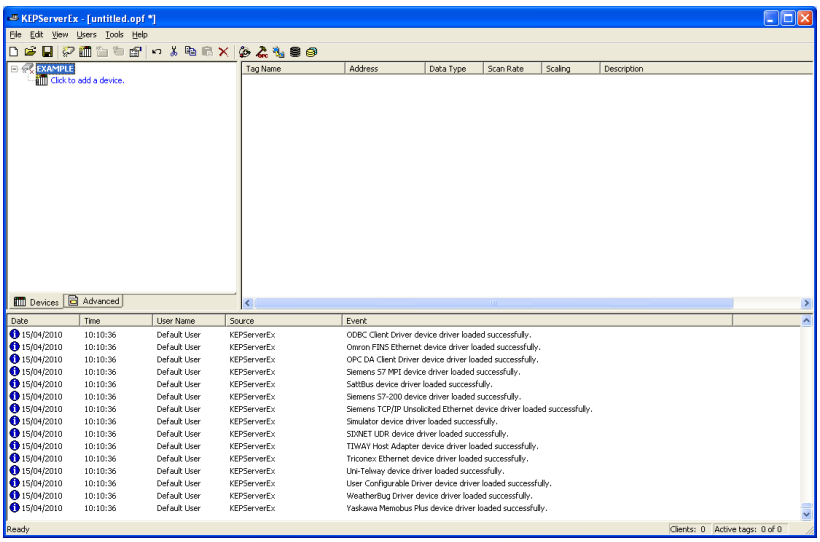
5. Leave the default settings and click **Next** to continue.



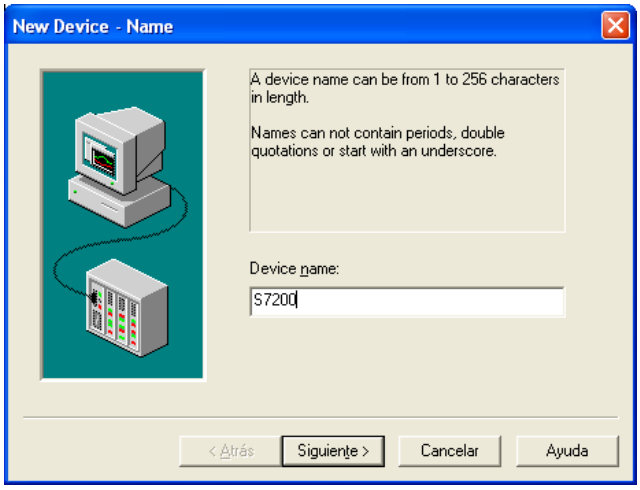
6. Leave the default settings and click **Next** to continue.



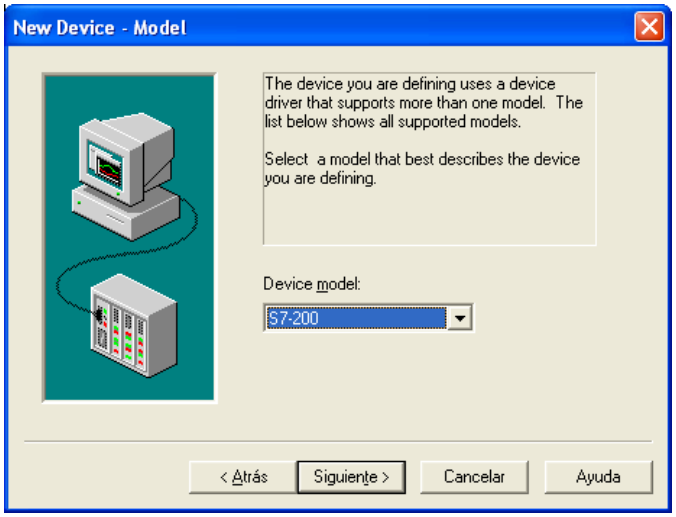
7. Click **Finish** to complete the channel configuration process.



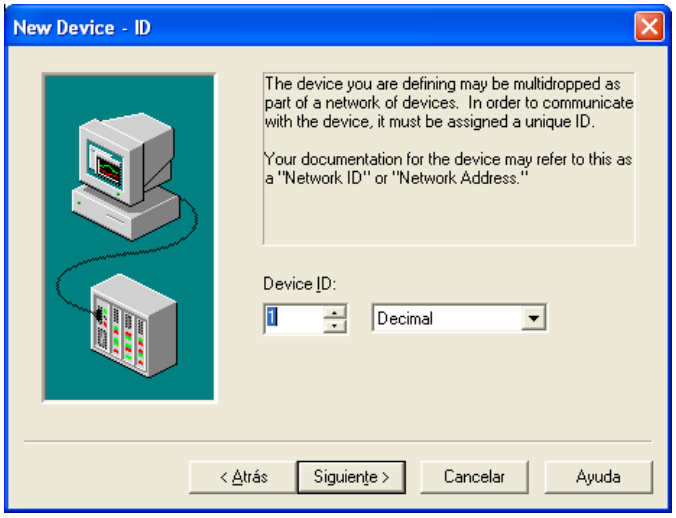
8. Now, you must add the device by clicking on '**Click to add a device**'.



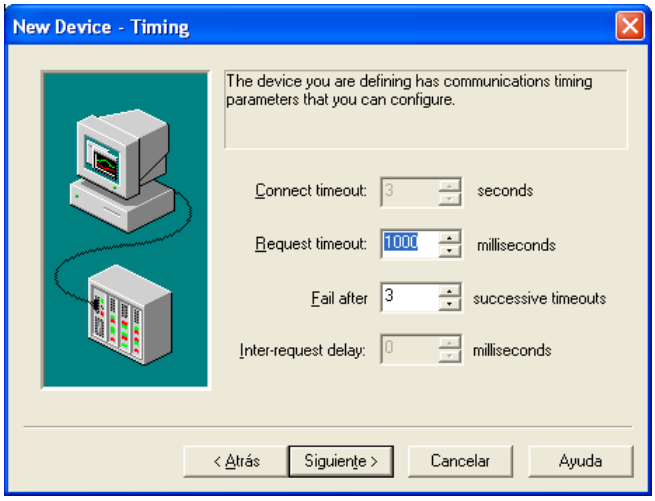
9. In this case, a Siemens S7-300 PLC is being used. Click **Next** to continue.



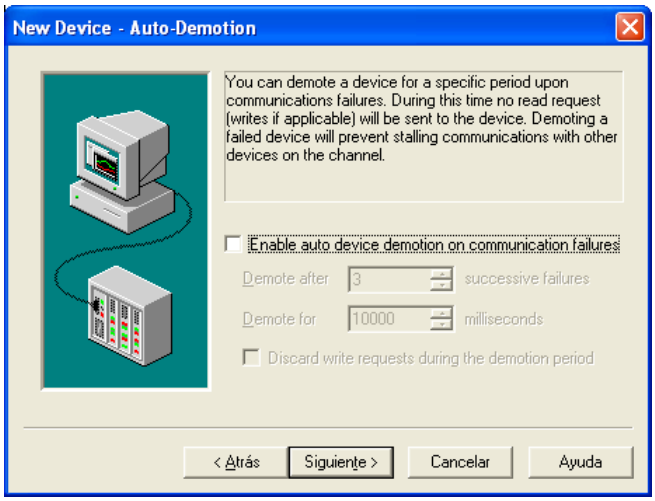
10. Select the **S7-300** option and click **Next** to continue.



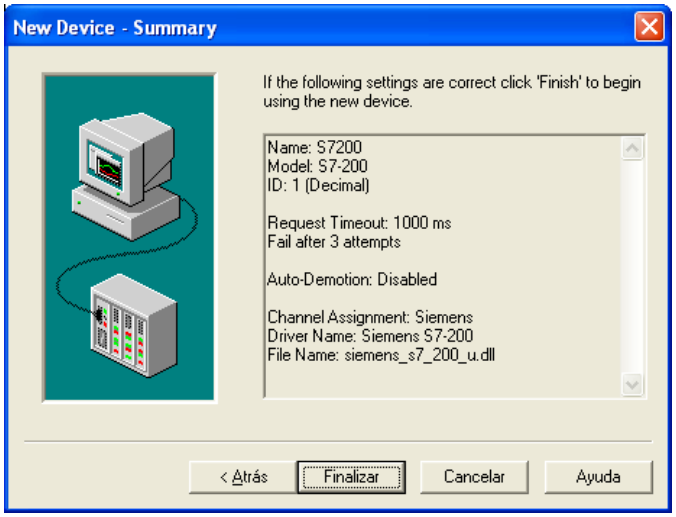
11. Enter a value for the *'Device ID'* parameter. In this case, the default values are retained. Click **Next** to continue.



12. Leave the default values and click **Next** to continue.

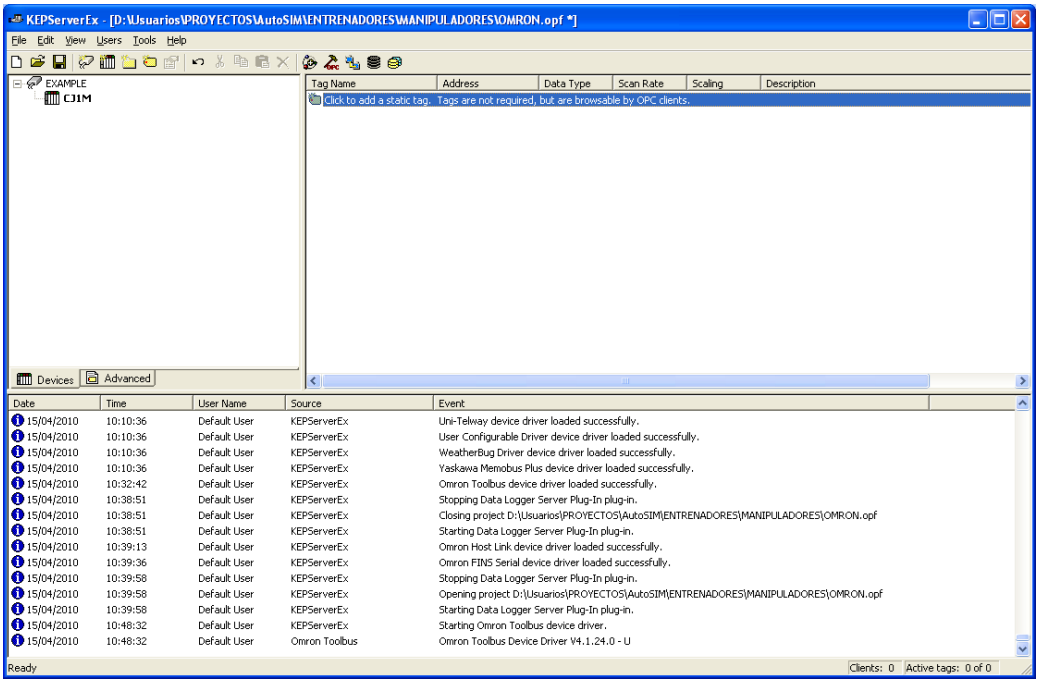


13. Leave the default values and click **Next** to continue.

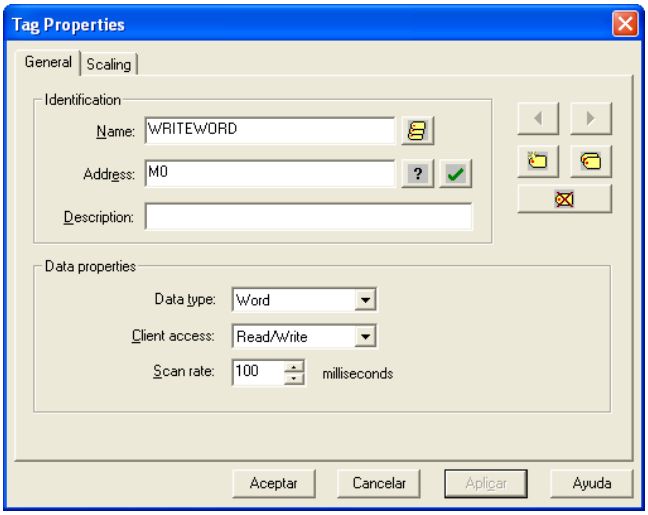


14. Click **Finish** to complete the device configuration.

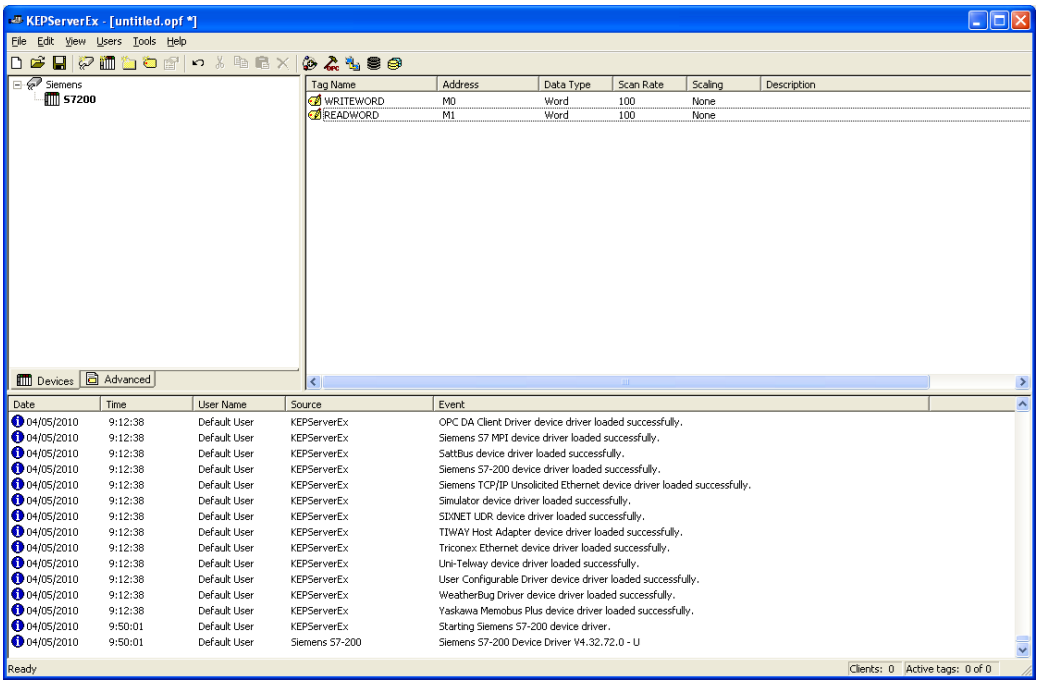
15. Configuring the communication variables between autoSIM and a Siemens PLC. Click **'Click to add a static** tag' to add a new communication variable.



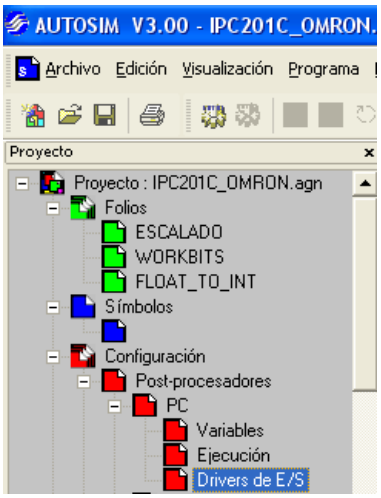
When you do this, the following window appears (in this example, data is read from one word and written to another).



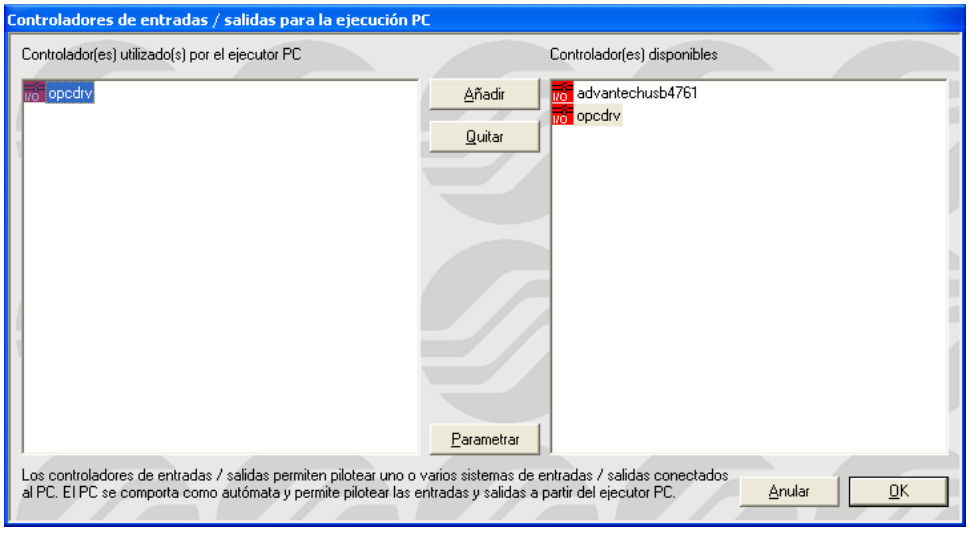
Address **MW0** is used for read operations and address **MW1** for write operations. With this, the OPC server is ready for communication.



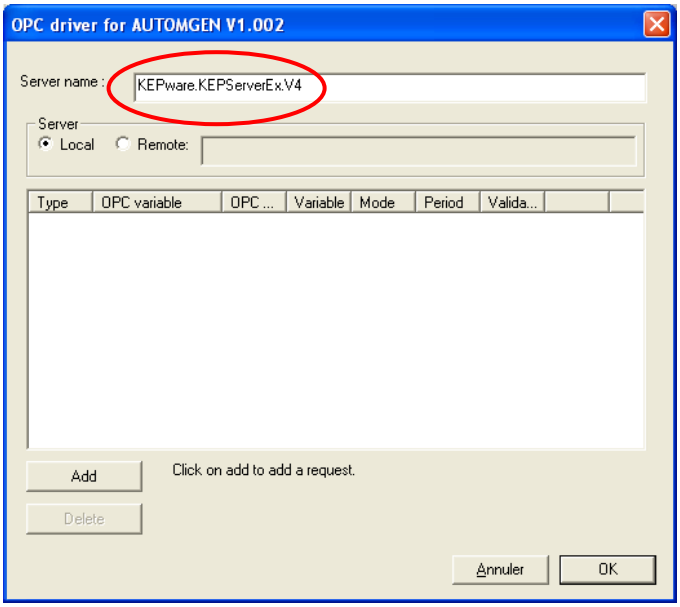
16. Once this is done, you must configure autoSIM as an OPC client. To do this, click on **Configuration/Post-Processors/PC/I/O Drivers**.



The following window appears.

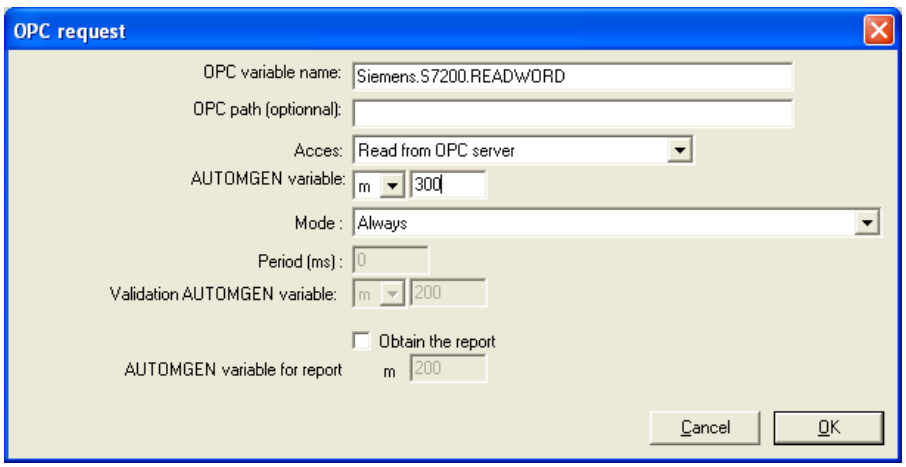


17. Select the **'opcdvr'** option and click on **'Configure'** to set up the driver and define the communication variables. When you do this, the following window appears.



The server name is **KEPware.KEPServerEx.VX** (this name cannot be changed; X is the OPC server version number). Click **OK** to accept the changes. When you run the simulation, the KEPware OPC server opens automatically.

18. All that remains is to tell autoSIM what the communication variables are. To do this, click on the **'Add'** button, whereupon autoSIM displays the following window.



You must enter the full name of the OPC server variable exactly as it appears. In the case of a read operation, the name of the variable created is **Siemens.S7300.READWORD**. The name must contain the full path to the variable. As you wish to read from the OPC server, you must select the **'Read from OPC server'** option in the *'Access'* field. Furthermore, this variable is stored

in the autoSIM variable **m300**, so select this variable in the corresponding field, *'AUTOMGEN variable*'. Select the **'Always'** option under *'Mode*'. Click **'OK'** to accept the settings.

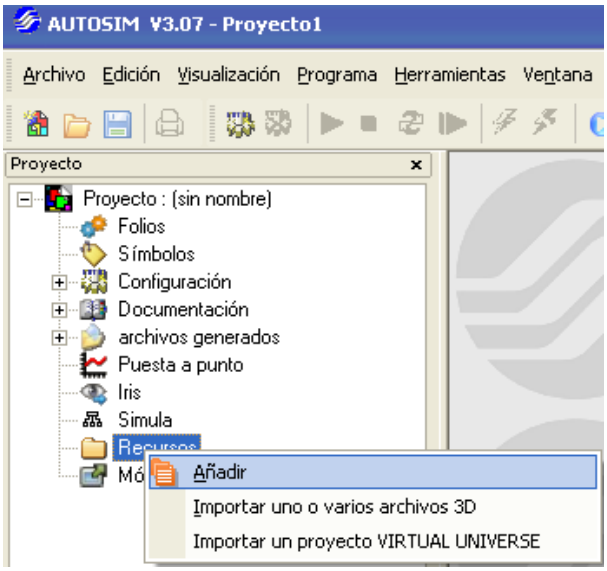
For writing, the full name of the variable is **Siemens.S7300.WRITEWORD**. As you wish to write to the OPC server, select the **'Write to OPC server'** option in the *'Access'* field. Furthermore, this variable is stored in the autoSIM variable **m301**, so select this variable in the corresponding field, *'AUTOMGEN variable*'. Select the **'Always'** option under *'Mode*'. Click **'OK'** to accept this configuration.

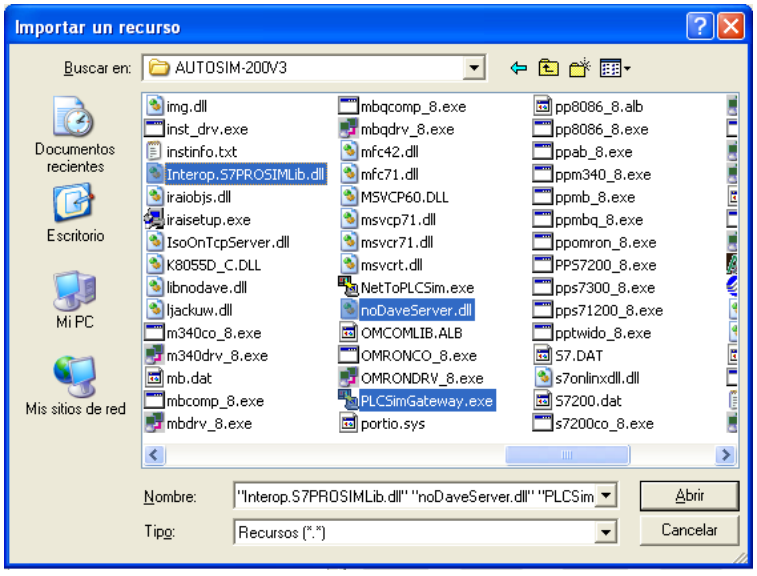
Once this is done, the system is ready to communicate with a Siemens S7-300 PLC.

Communication with the PLCSim V4.5 simulator

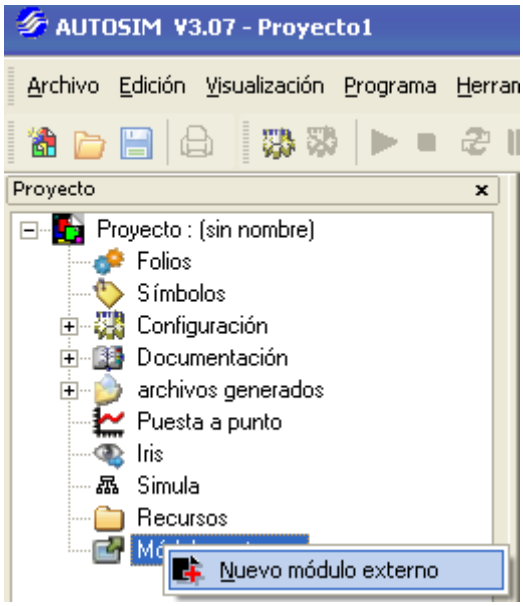
All the **.dll* libraries and the *PLCSimGateway.exe* executable required for this communication are located in the autoSIM directory: *C:\Program Files\SMC\AUTOSIM-200V4*.

1. Add the libraries *noDaveServer.dll* and *Interop.S7PROSIMLib.dll*, and the executable *PLCSimGateway.exe*, to the autoSIM project. To do this, navigate to the '**Resources**' section in the project tree. Right-click and select '**Add**'. Select the files mentioned above and click '**Open**'.

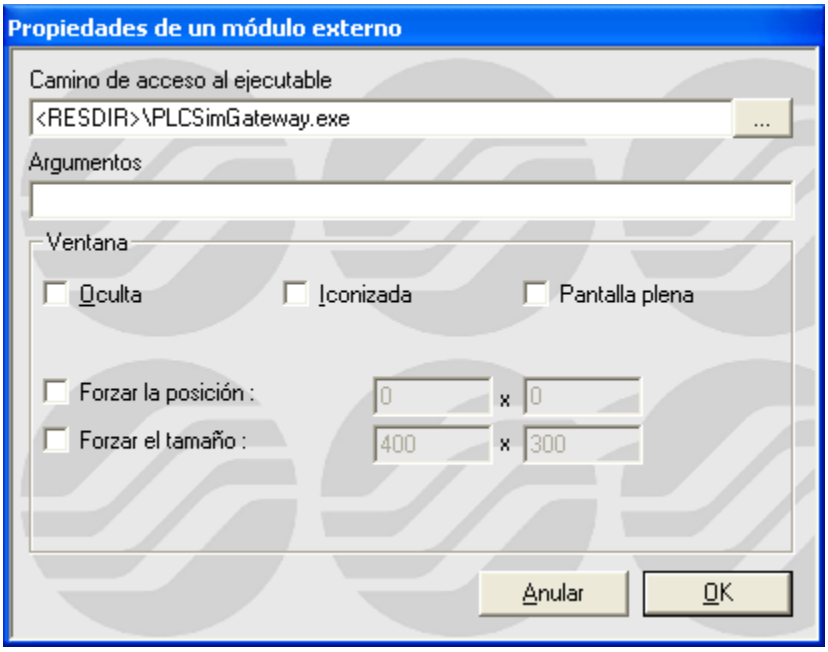




2. Right-click on **'External** Modules' in the project tree and select the **'New External Module'** option.



3. A new window will appear where you must enter the following command: **<RESDIR>\PLCSimGateway.exe**.



4. Configure the communication driver for PLCSim from autoSIM. To do this, go to **Settings/Post-Processors/PC/I/O Drivers**.

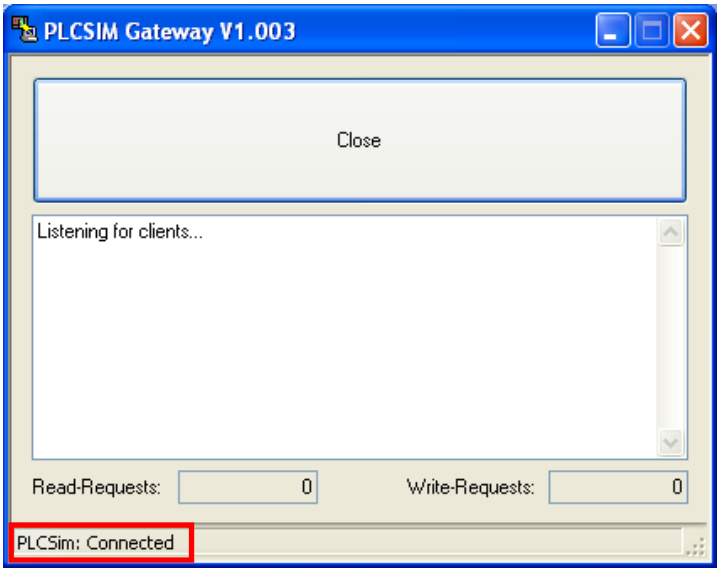
a. Open the **plcsimdrv** driver.
b. Enter the IP address where PLCSim is located (enter **'localhost'** if it is on the same PC).
c. Enter the port number (the default is 5500).
d. Add the variables that will be used in the communication. For each one, you must define:

- **PLCSim variable name:** The PLCSim address of the variable to be controlled (this may be: E127.0, M10.2, MW5, MD100, AD4, etc.).
- **Access:** Access type; choose between read or write mode.
- **Variable:** This is the autoSIM address to which this variable is to be linked.
- **Mode:** You can specify whether the exchange should take place continuously, periodically, or when an event occurs.
- **Obtain a report:** If this option is enabled, you can check the communication status (this is a word that is set to 0 if communication is OK and to a value other than 0 if an error has occurred).

5. Open the Siemens STEP7 V5.5 software or later and load the programme created in the PLCSim simulator.

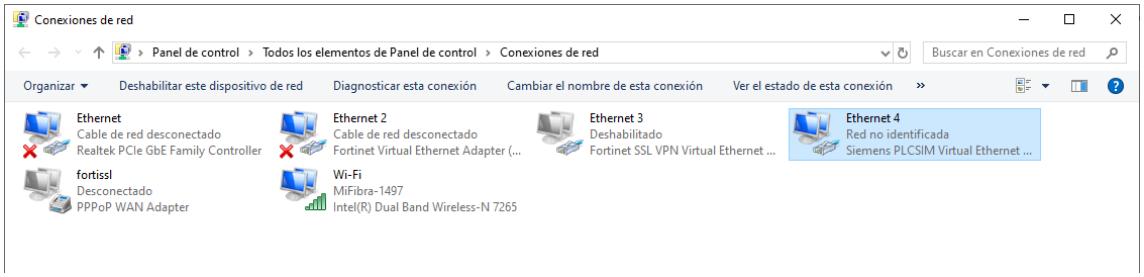
6. Set the PLCSim simulator to RUN mode.

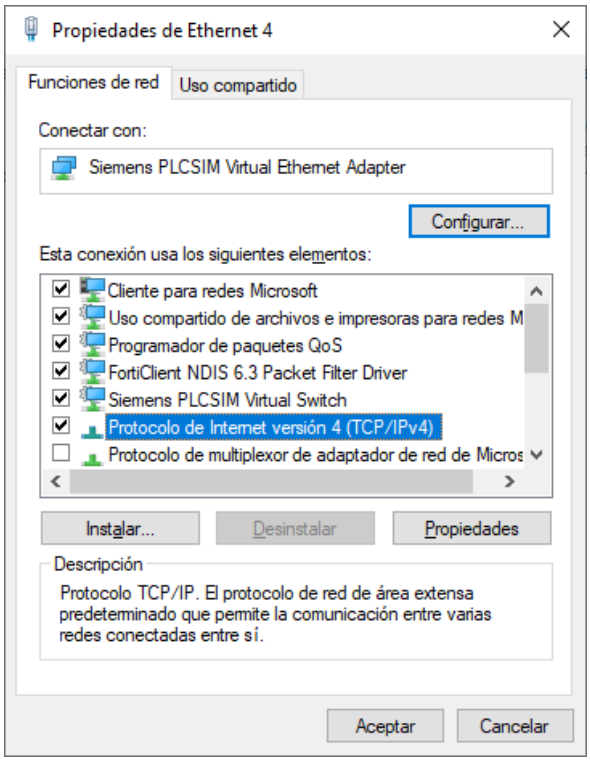
7. Start the autoSIM simulation using the **'Go!'** button. The following window appears, showing the communication status:

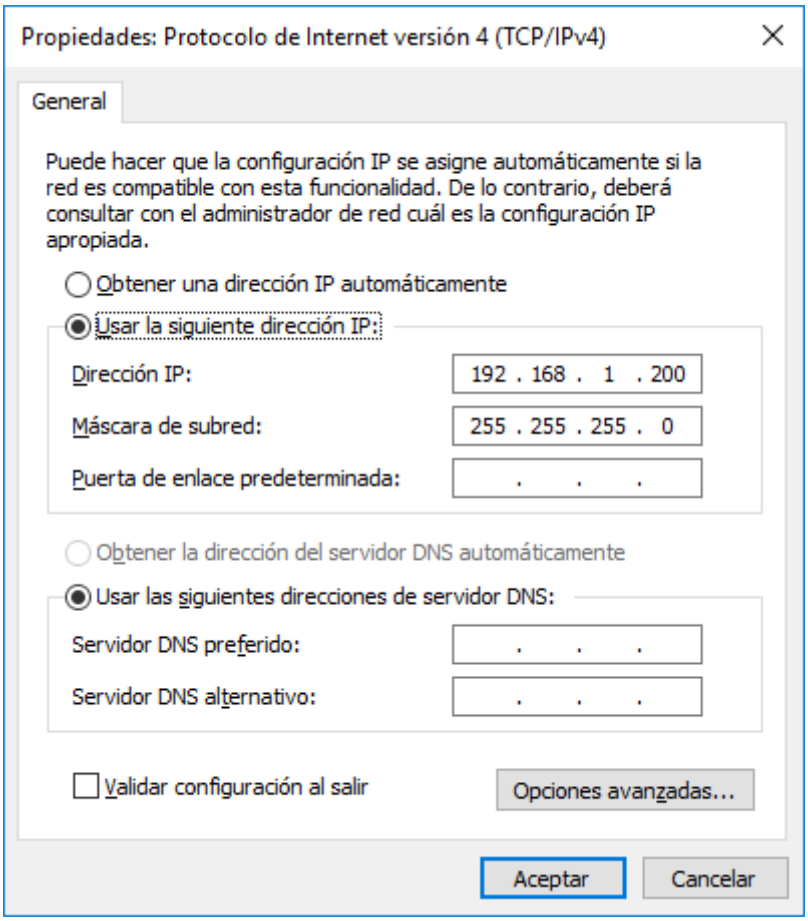


Connection between *PLCSim Advanced* and TIA PORTAL

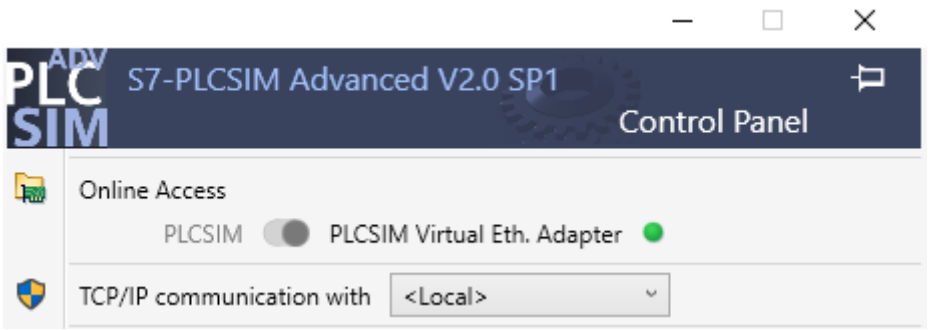
1. Install *PLCSim Advanced V1* or later on the computer.
2. Open the Internet connection called *'Siemens PLCSIM Virtual Ethernet Adapter'* (this connection is created when you install *PLCSim Advanced*) and select a static IP address (this will be the computer's IP address).



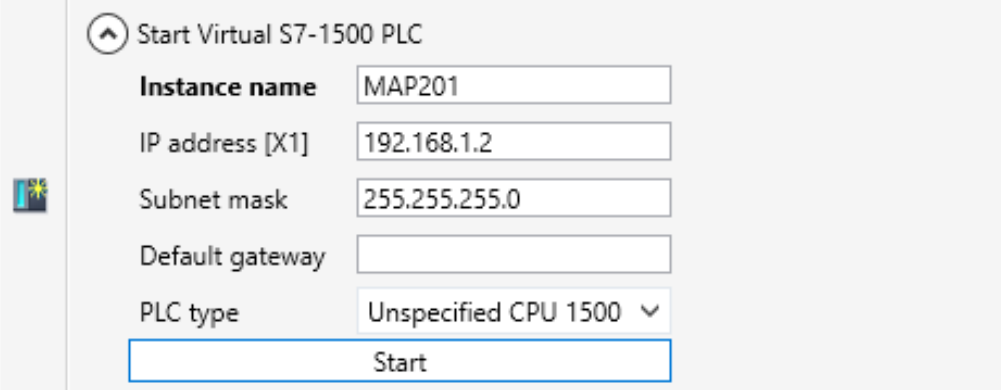




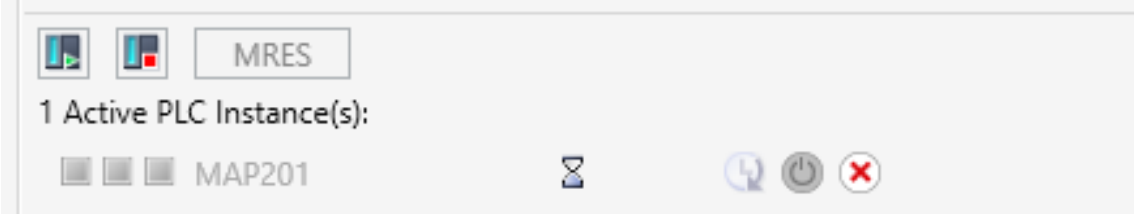
3. Open *PLCSim Advanced* (in this manual, the version of *PLCSim Advanced* used is V2 SP1).
4. Select *'Online access* (PLCSIM Virtual Eth. Adapter)' and *'TCP/IP communication with* (<Local>)'.



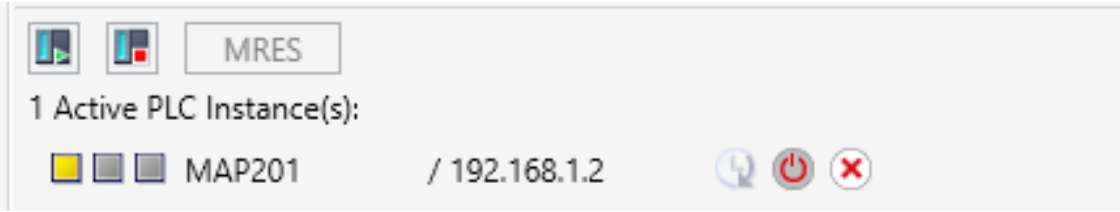
5. Create a new instance in *PLCSim Advanced*. Enter a name, an IP address (this must be the PLC's IP address), a subnet mask and select the PLC type (in this case, select *'Unspecified CPU 1500'*).



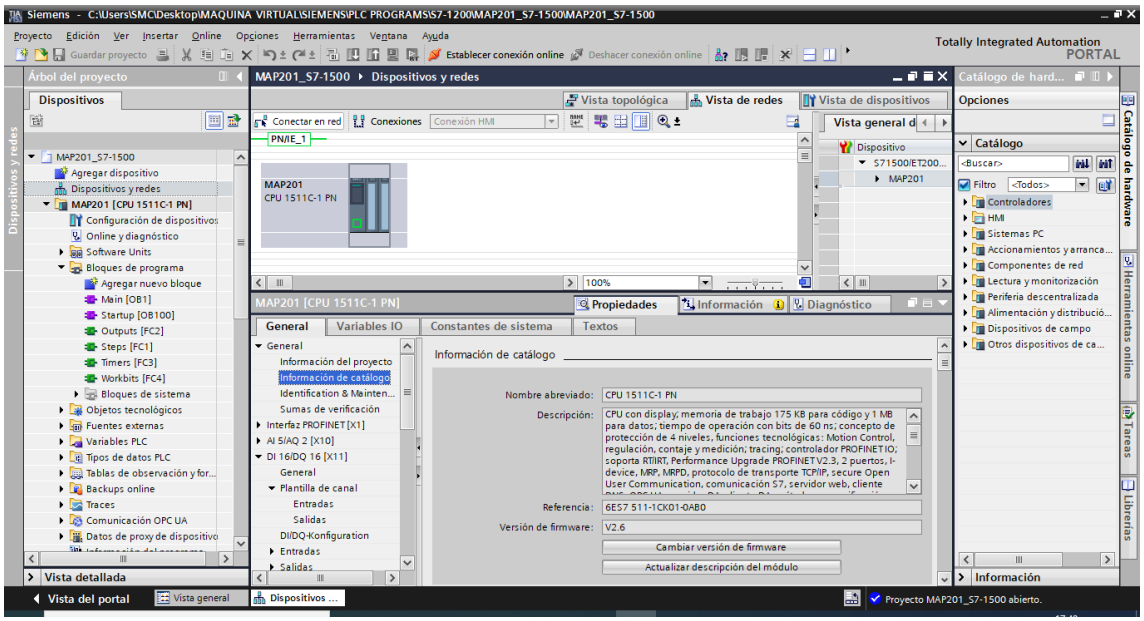
6. Click the *Start* button and wait until the instance has opened.



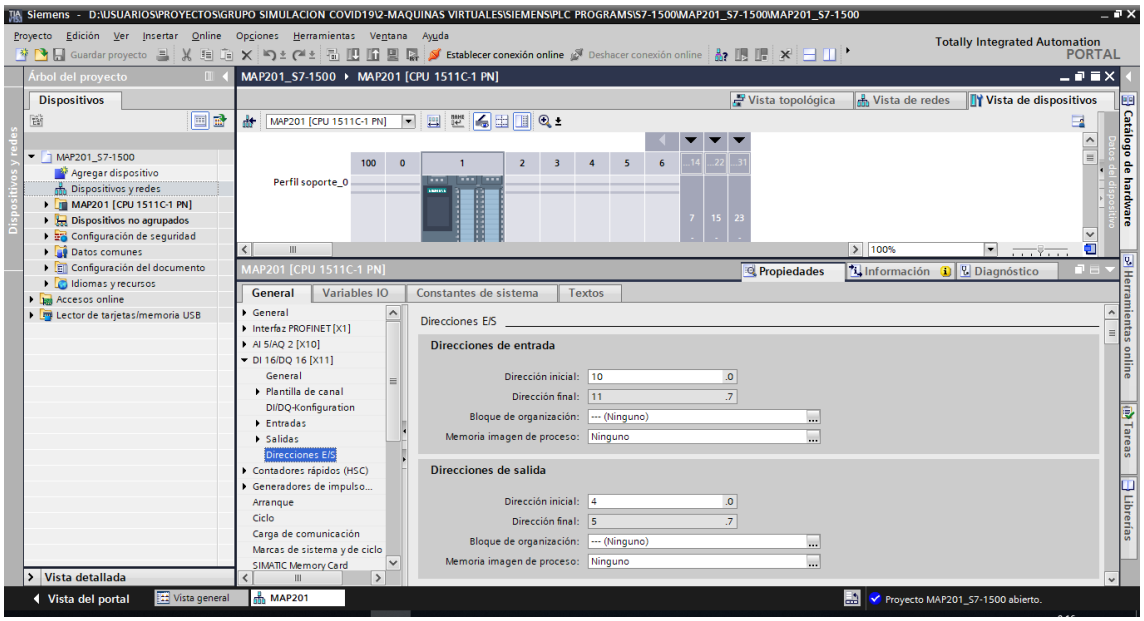
At this point, the instance has been opened and the PLC is in STOP mode.



7. Open *TIA PORTAL* (in this manual, the version of *TIA PORTAL* used is V15.1).

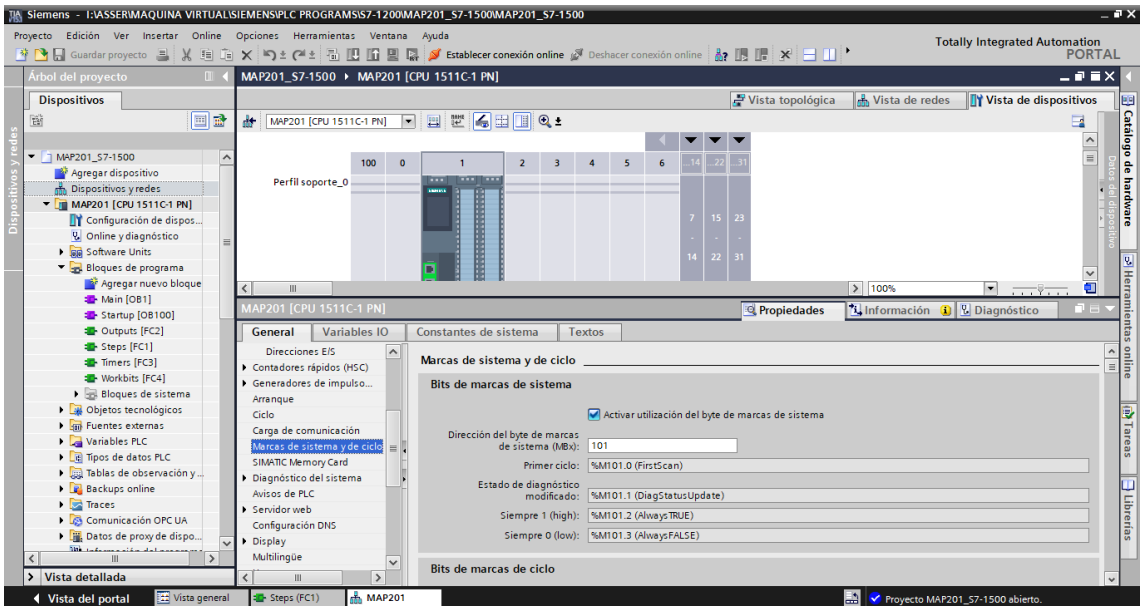


8. Create a new project. Due to the limitation of *PLCSim Advanced* mentioned earlier, you must choose a PLC from the S7-1500 family.
9. Go to the PLC properties and assign a value to the physical addresses for both digital inputs and outputs.

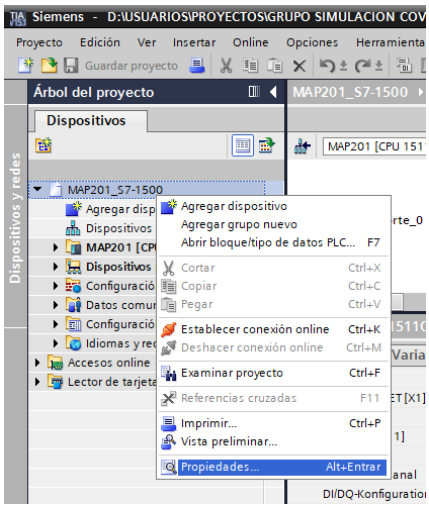


To specify the physical input/output addresses, you must modify the *'Process Image* Memory' field. You must select the 'None' option to access the user-defined physical addresses (this option is set to *'Automatic* Update' by default).

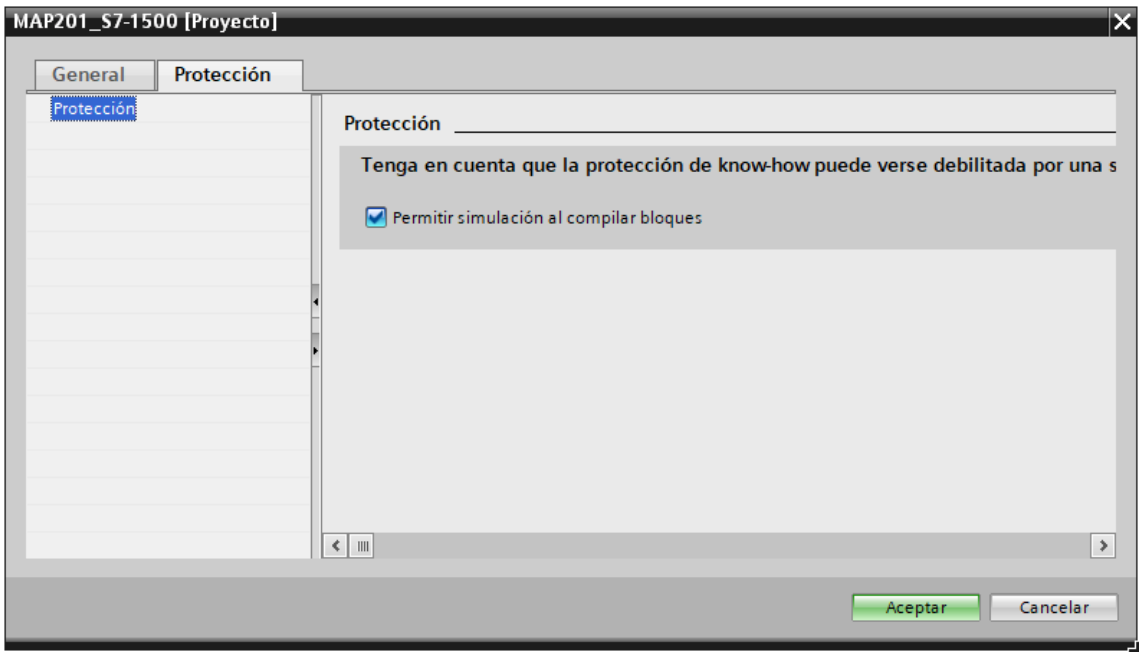
10. As the PLC has been modified, the cycle marks must be reactivated. To do this, go to the PLC properties and, in the *"System and cycle* marks" section, tick the box *"Enable use of the system mark* byte" and enter the value *101* (for example) in the *"System mark byte address (MBx)"* field:



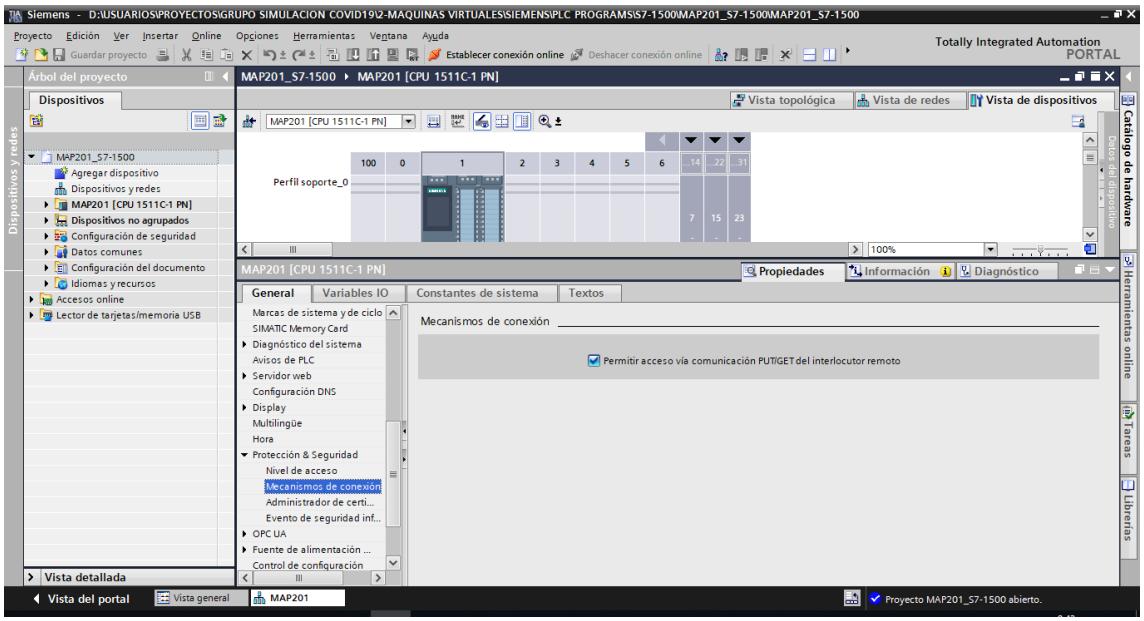
11. Check that the project allows programme blocks to be simulated. To do this, go to the project's "Properties":



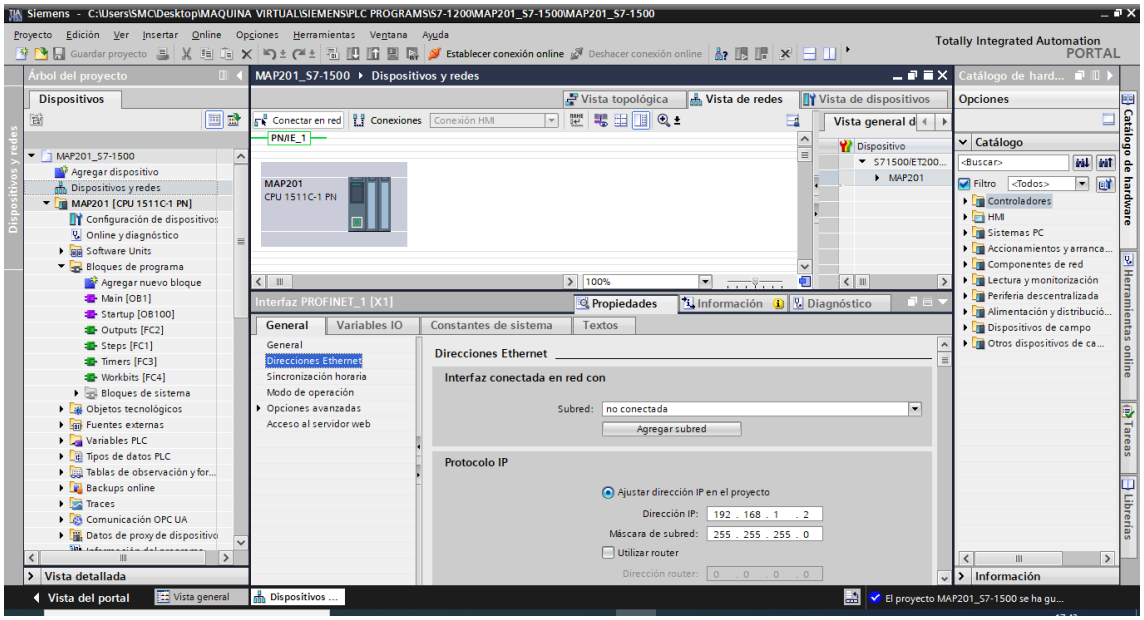
Select the “Protection” tab and tick the box *“Allow simulation when compiling blocks*”.



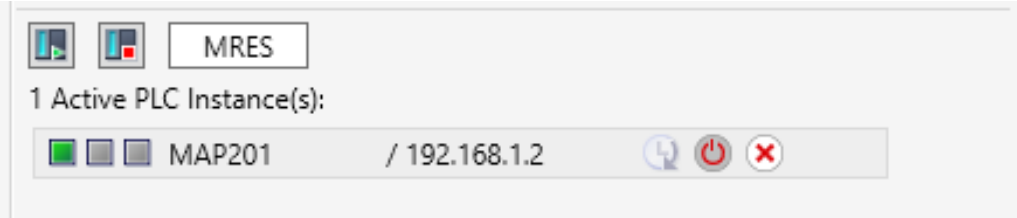
12. Ensure that the option to communicate the project with an external device has been enabled. To do this, go to the PLC's properties and, in the *"Protection &* Security" section, tick the box *"Allow access via PUT/GET communication with external device*":



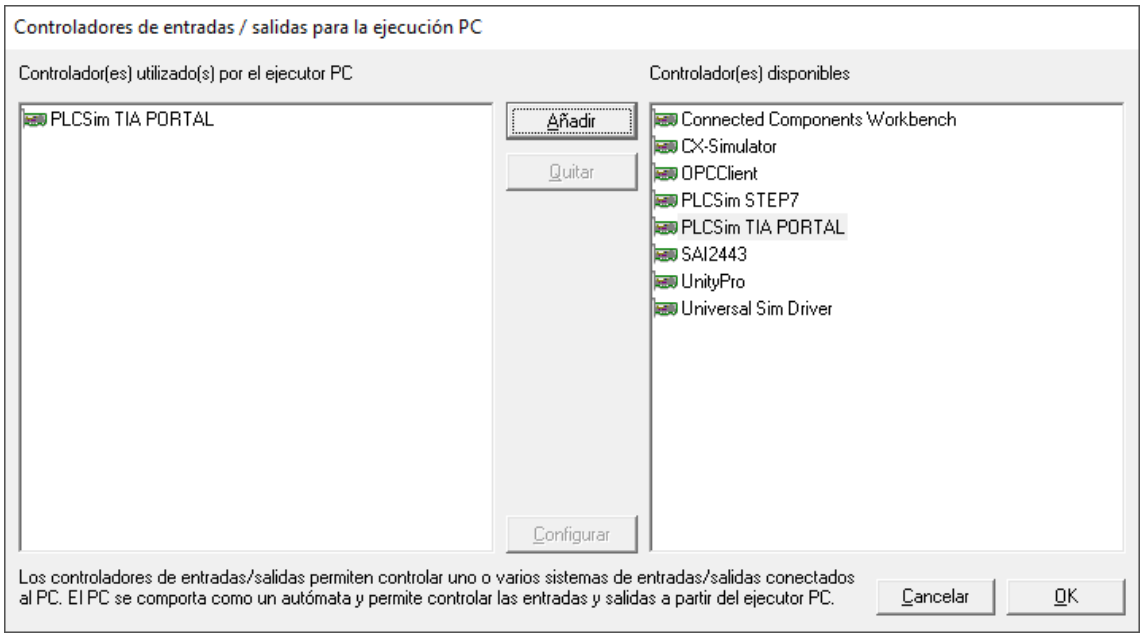
13. Go to the PLC's Ethernet port properties and assign an IP address to the PLC; this must match the one assigned to the instance created in *PLCSim Advanced*.



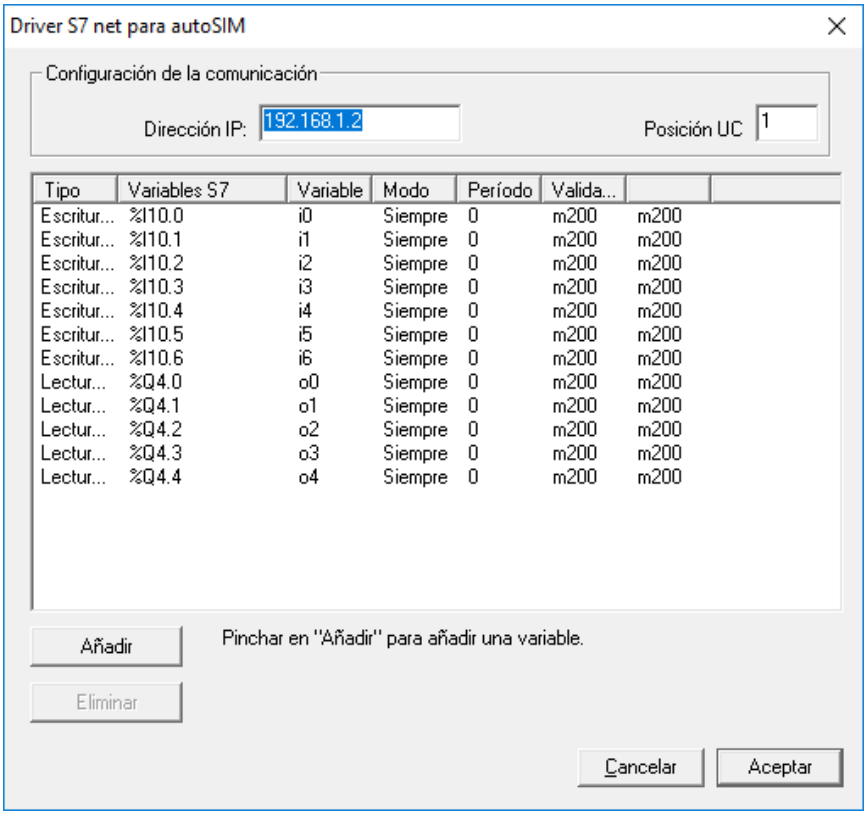
14. Load the project created in PLCSim Advanced and ensure that the PLC instance created in *PLCSim Advanced* is in RUN mode.



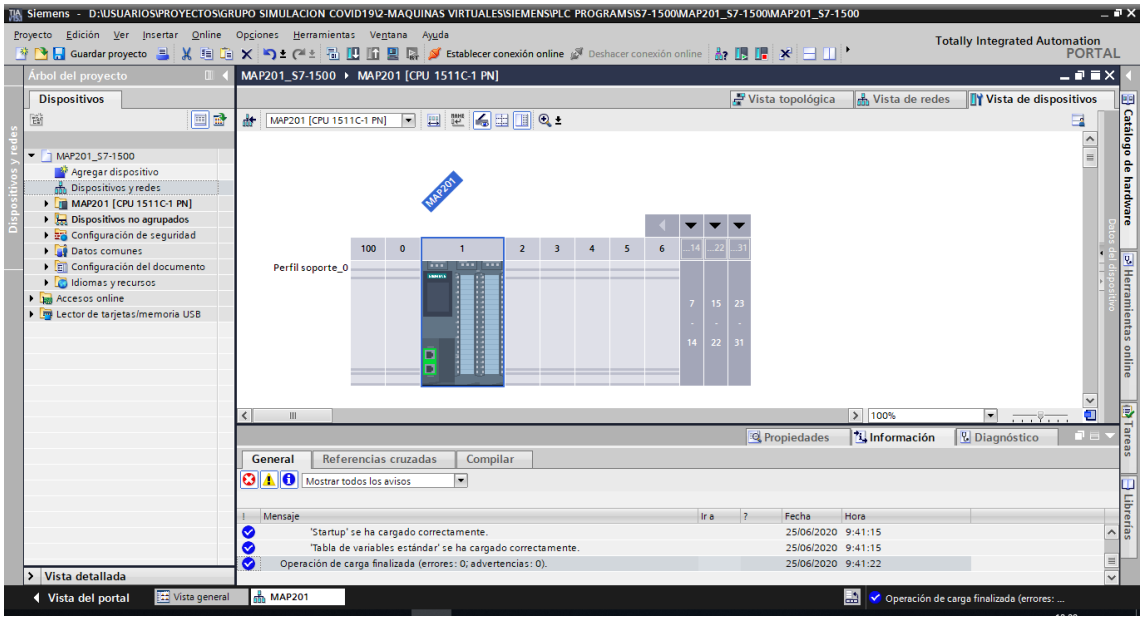
15. Open autoSIM-200.
16. Select the appropriate driver to establish the connection between *autoSIM-200* and *PLCSim Advanced*. To do this, go to *Configuration/Post-processors/PC/I/O Drivers*, select the driver named **PLCSim TIA PORTAL** and click the *"Add"* button.



17. Double-click on the *PLCSim TIA PORTAL* driver or select the driver and click the *"Configure"* button.

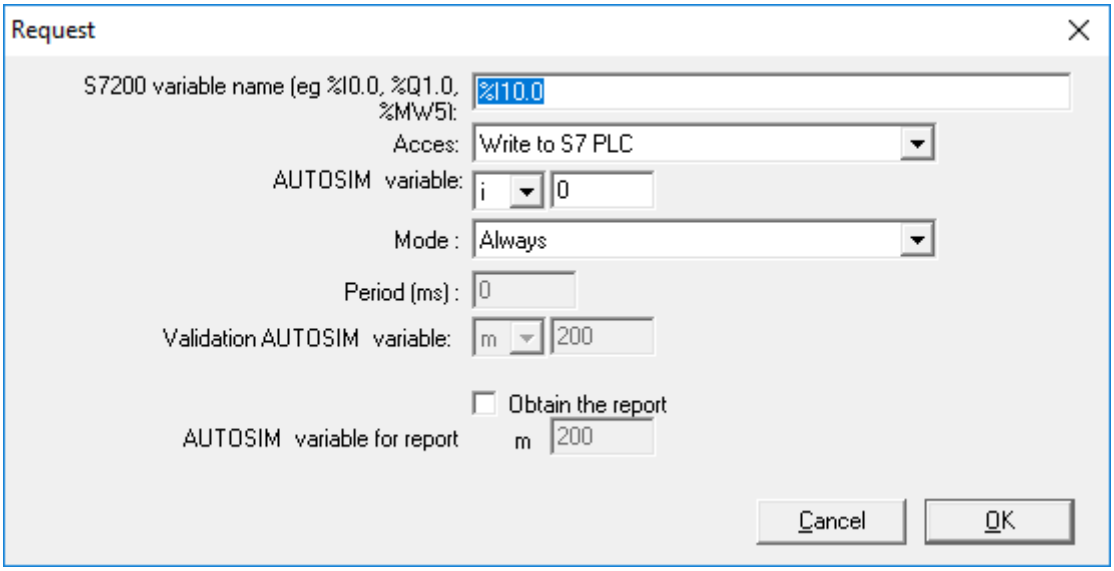


18. Enter the IP address of the PLC to which you wish to connect the Digital Model and the PLC's CPU slot (this is the slot occupied by the PLC's CPU in the rack; you can check this slot in TIA PORTAL).

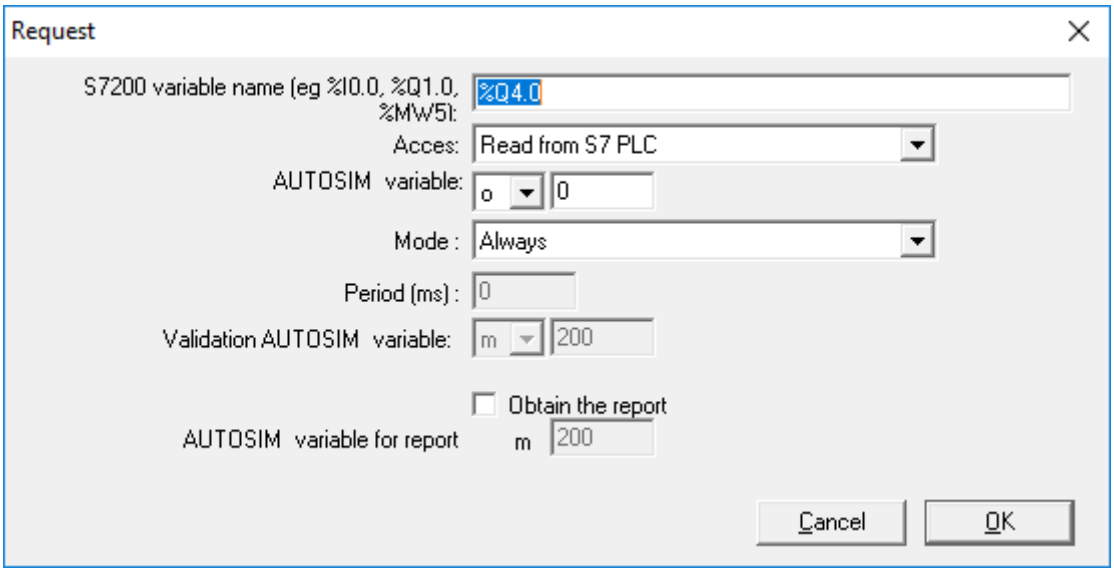


19. Click the "Add" button to insert the variables you wish to exchange between *autoSIM-200* and *PLCSim Advanced*.

Physical input:



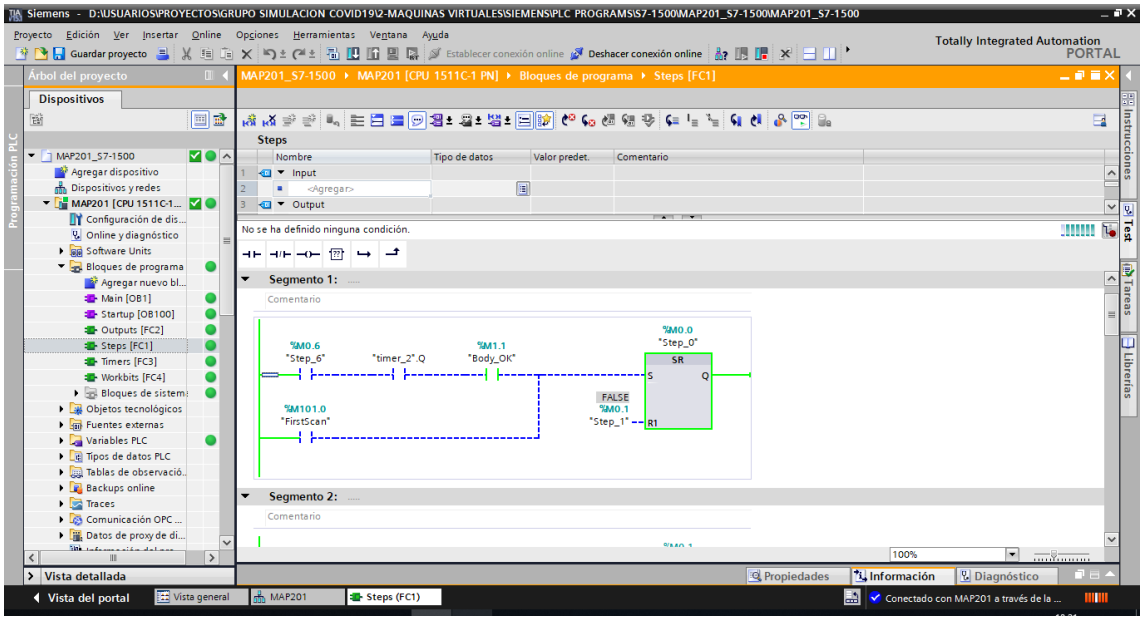
Physical output:



Click on the **'Go!'** button to start communication.

When the *"Dynamic* Display" icon starts spinning and no compilation errors have appeared, this means that the connection between *autoSIM-200* and *PLCSim Advanced* has been established correctly.

The user can now go online with the PLC and check whether the programme they have developed is correct using the simulations previously generated in the autoSIM software.

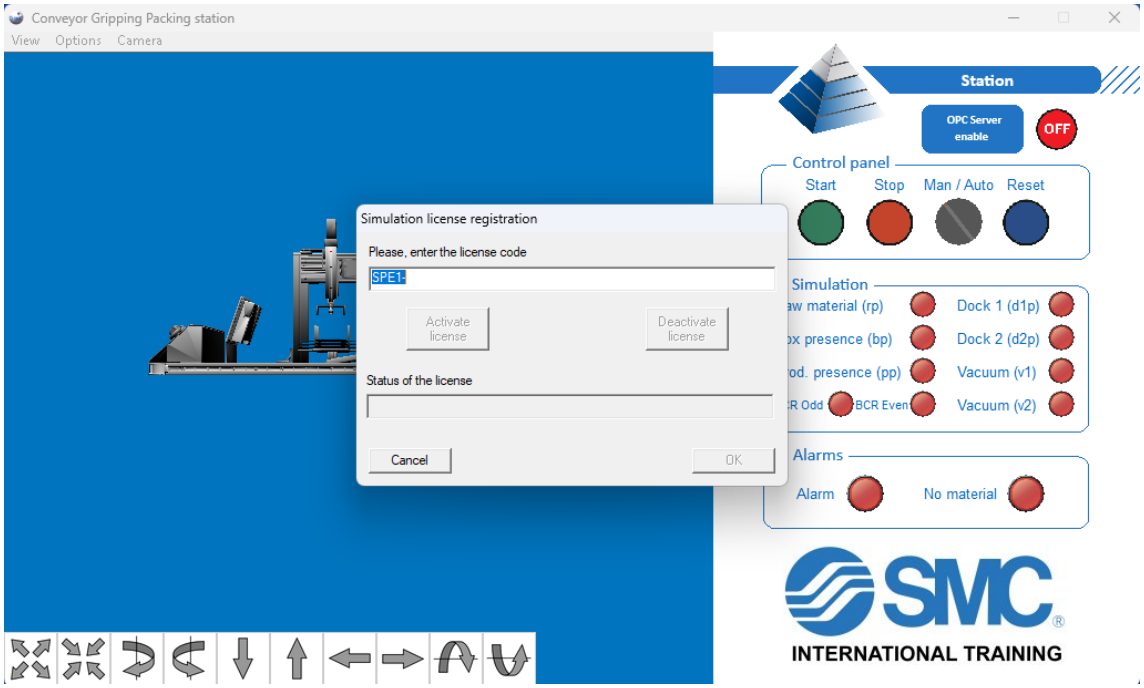


Activating licences for 3D simulations

To activate a simulation licence, follow the procedure described below:

1. Double-click on the autoSIM-200 simulation file *"simulation_name.agn"*.
2. Enter the password "smcit" to open the simulation.

3. The following window will appear:



4. Enter the licence number in the field provided. Click the "Activate licence" button, then click "OK".

The user can check the licence status at any time in the “Licence status” field.

5. autoSIM-200 opens the simulation.

When you are no longer going to use the simulation, it is advisable to deactivate it so that the licence can be used on another PC. To do this, open the ‘.agn’ file and click the ‘Deactivate licence’ button. Once this is done, the licence will be available for use again.