EcosimPro 5.2.0
Quick Start Guide

School of Aeronautical and Space Engineering

Technical University of Madrid

In cooperation with Empresarios Agrupados

Pablo Sierra Heras

Supervisors:

Juan Manuel Tizón Pulido (Technical University of Madrid),
Javier Vilá Vacas (Empresarios Agrupados),
José Francisco Moral Moral (Empresarios Agrupados).

Contents

Getting Started

1 EcosimPro

EcosimPro and PROOSIS are powerful mathematical tools capable of modelling any kind of dynamic system represented by differential-algebraic equations (DAE) or ordinary-differential equations (ODE) and discrete events. They are integrated visual environments that provide intuitive tools to make simulation easy.

The tools provide an object-oriented acausal approach to create reusable component libraries. They are based on symbolic and numerical methods capable of processing complex systems of differential-algebraic equations. Their smart wizards provide modellers with an easy way to build consistent mathematical models.

EcosimPro and PROOSIS are used by leading companies in the Aerospace and Energy sectors. The European Space Agency (ESA) has chosen EcosimPro as its recommended tool for simulation in several fields, including propulsion, environmental control systems and life support (ECLSS) and power systems. Over the last 15 years, ESA has turned to EcosimPro to model, among others, complex systems of the International Space Station (ISS), rocket propulsion systems and biological systems.

The European Space Propulsion System Simulation (ESPSS) is an ESA toolkit to model rocket and satellite propulsion systems. It consists of a set of libraries based on the EcosimPro simulation environment.

The ESPSS toolkit provides components and functions for the simulation of launch vehicle and spacecraft propulsion systems able to work under transient and steady conditions. ESPSS also includes a complete database of fluids to be used as propellants, pressurizing fluids or other applications.

NOTE: PROOSIS is a Propulsion Object-Oriented Simulation Software tool based on EcosimPro. It provides additional features for modelling gas turbines.

2 Working in EcosimPro

This chapter will show you how to create models based on existing libraries so you will be able to generate and simulate the schematics that represent the different physical systems.

You simulate a physical system in EcosimPro from an existing library following these 5 steps:

Figure 1 - Steps to simulate a physical system

Steps 1 and 2 are explained in section *6 Graphical Modelling*, step 3, in section *7 Partition Management*, step 4, in section *8 Experiment Management*, and, finally, step 5, in section *9 Simulation*.

3 Key Concepts

EcosimPro has five fundamental concepts:

- **Components:** These represent models of the systems simulated by means of variables, differential-algebraic equations, topology and event-based behaviour. A component is the equivalent of the "class" concept in object-oriented programming.

- **Ports:** These define a set of variables to be interchanged in connections and the behaviour and restrictions when there are connections between more than two ports. For instance, an electrical connection type uses voltage and current as variables to be used in connections. The connection port avoids having to connect individual variables; instead, sets of variables are managed together.

- **Partitions:** To simulate a component, you first have to define its associated mathematical model; this is called a partition. A component may have more than one partition. For example, if a component has several different boundary conditions, depending on the set of variables selected, each set of variables produces a different mathematical model, or partition. The next step is to generate experiments for each partition. The partition defines the causality of the final model.

- **Experiments:** The experiments performed for each partition of the component are the different simulation cases. They may be trivial to calculate a steady state experiment or they may be very complex to calculate an experiment with many steady and transient states changing multiple variables in the model.

- **Libraries:** All components are classified by disciplines into libraries.

4 Workspaces

The workspace contains a set of libraries related to the specific simulation environment that you are working with. Each user can create personal workspaces to work in his or her particular simulation environment.

When a workspace is active, its libraries and their components will be displayed in the workspace area.

4.1 Creating a new Workspace

This is what the EcosimPro program looks like when it is first opened:

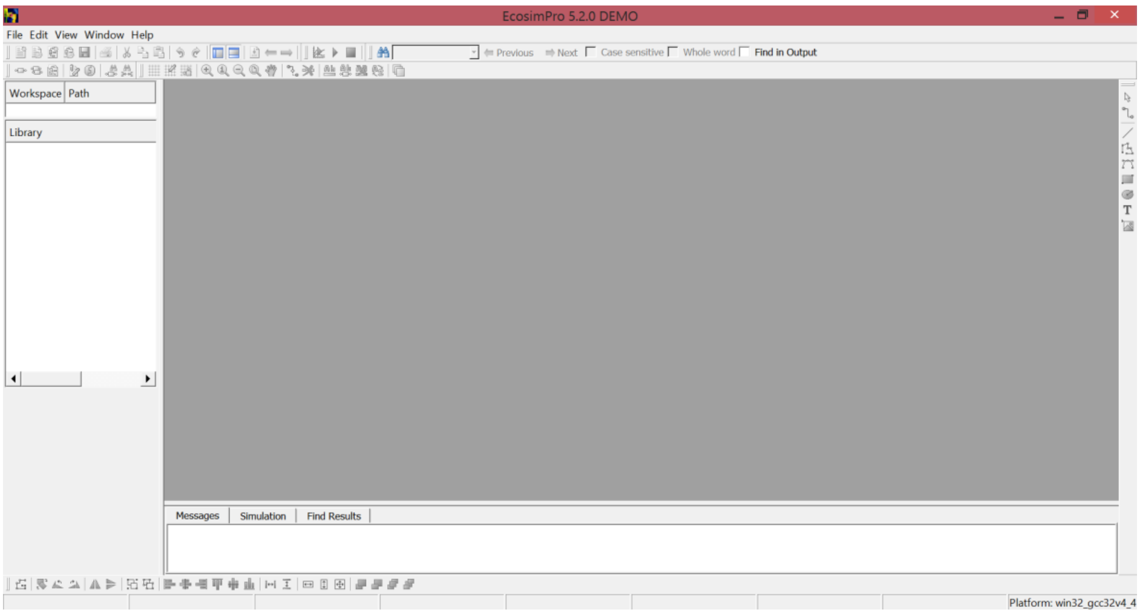


Figure 2 - Initial aspect of the EcosimPro program

To use the program, the fist step is to create a new workspace. To do this, click on **File** > **New** > **Workspace.**

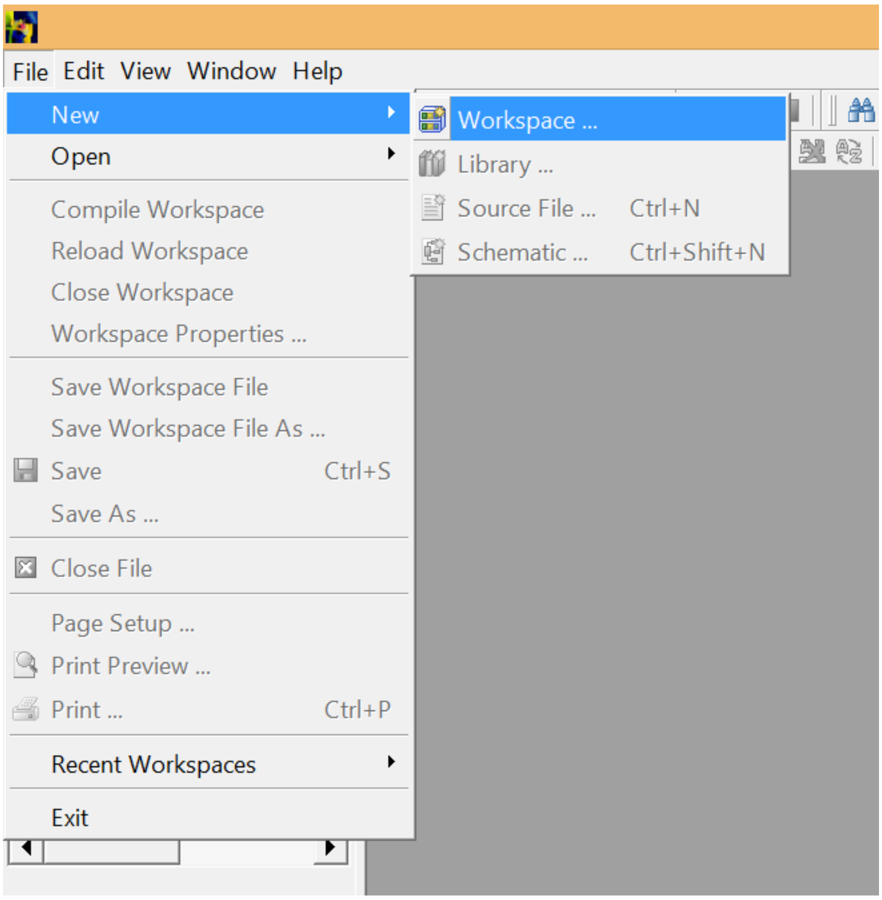


Figure 3 - Open a new Workspace

In the **New workspace** dialog box, type in the name of the new workspace and the path where the associated file is to be saved.

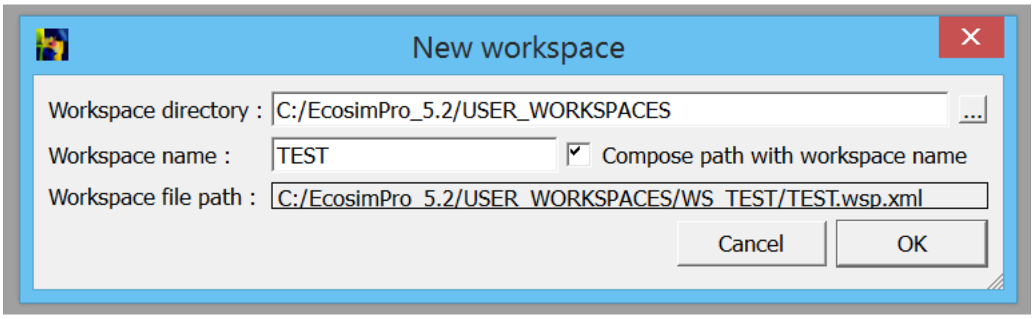


Figure 4 - Give a name to the workspace

4.2 Opening a Workspace

It is also possible to work with an existing workspace. To open an existing workspace, click on **File** > **Open** > **Workspace.**

This opens the corresponding dialog box. To specify the workspace to be used, first locate it (**.wsp.xml** file) and then open it. Once the workspace has been selected, the libraries and their components will be displayed in the workspace area.

4.3 Closing a Workspace

This option allows you to close the current workspace, and is available on the **File** > **Close Workspace** menu or on the pop-up menu of the workspace area.

5 Libraries

A library is a collection of components related to a specific simulation environment (Electrical, Mechanical, Thermal, etc).

The **Libraries area** consists of 5 tabs that group the files and text and graphical modelling functionalities, as well as the different types of experiments that can be designed based on the modelling.

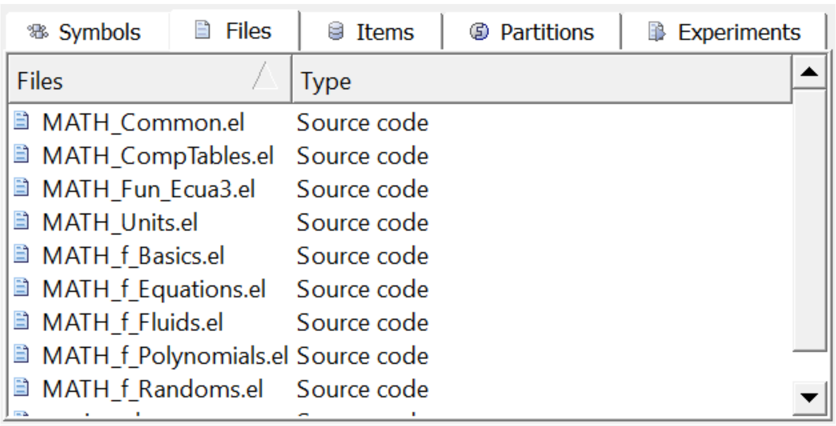


Figure 5 - Libraries area

The various components in a library are modelled using EL Language. EL provides everything you need to model a component or set of components in a user-friendly manner (for further information, see the EL Modelling Language manual). EL Language is not explained in this manual.

5.1 Creating a new Library

To create a new library, click **File** > **New** > **Library.**

In the **New library** dialog box, give the new library an identifier and type the path to the directory where the library components will be saved.

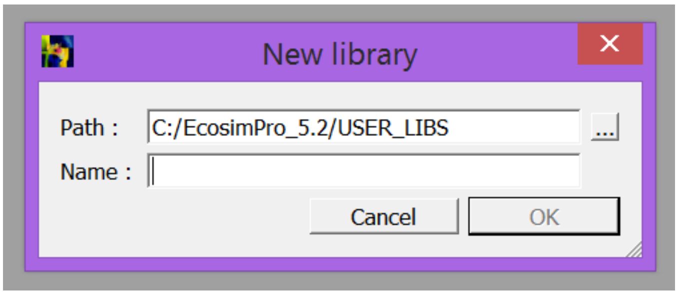


Figure 6 - Giving a name to the library

Once you have given the library an identifier, the **Area** is automatically updated and the library you have just created appears.

Use the **Editing Area** to begin modelling the components that will belong to the library you have just created.

5.2 Opening a Library

To open an existing library, click **File** > **Open** > **Library.**

The open dialog box appears for you to specify the library (**.lsp.xml** file) to be used.

Once the library is open, it will be displayed at the top of the workspace area and the files, items and other components belonging to it will be displayed at the bottom.

5.3 Closing a Library

To close a library, go into the library you wish to close and right-click with the mouse (a context menu will appear), click the **Close** command.

Selecting this option deletes the library from the workspace but not from the system. To access the library again click **File** > **Open** > **Library.**

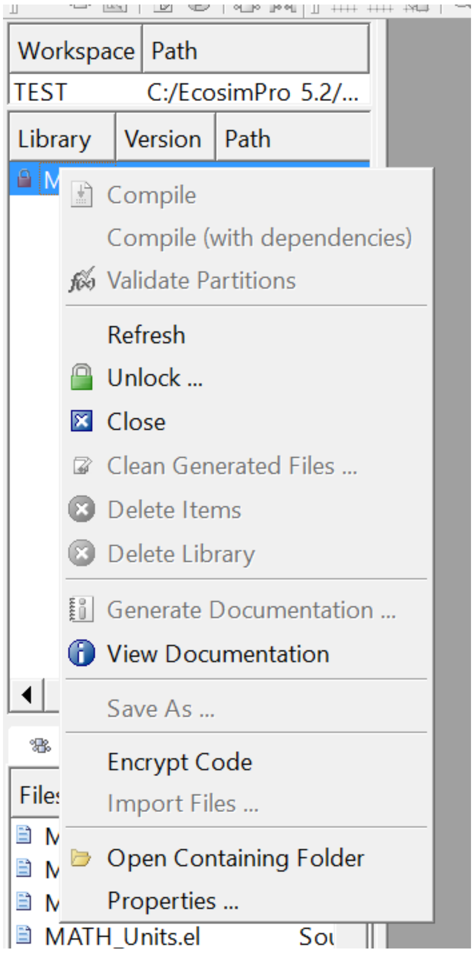


Figure 7 - Closing a library

6 Graphical Modelling

Topological components can be created in two ways: by writing EL code (by means of a **Source File**, and which is not explained in this manual) or by building graphical models called schematics.

The methods are equivalent during the modelling phase, as they produce the same results; however, the graphical method is much simpler, faster and more intuitive.

There are two basic concepts in graphical modelling:

- **Symbol:** a graphical representation of a component or port that has been previously compiled.

- **Schematic:** a graph that uses symbols and relations between them to build a new topographical component.

6.1 Schematics

In EcosimPro all components can be created using EL even those with very complex topologies. But you don't have to learn the details of EL because EcosimPro makes it easy to create complex (and simple) components graphically.

Schematics are graphical models that join components in topological formations to create other higher-level components that can be compiled.

Schematics are composed, therefore, of interrelated subcomponent instances. These instances are represented by symbols. The interrelations of these instances are created by way of connectors that join the ports of one instance to those of the others.

The following figure shows a component that has been created graphically by means of a schematic.

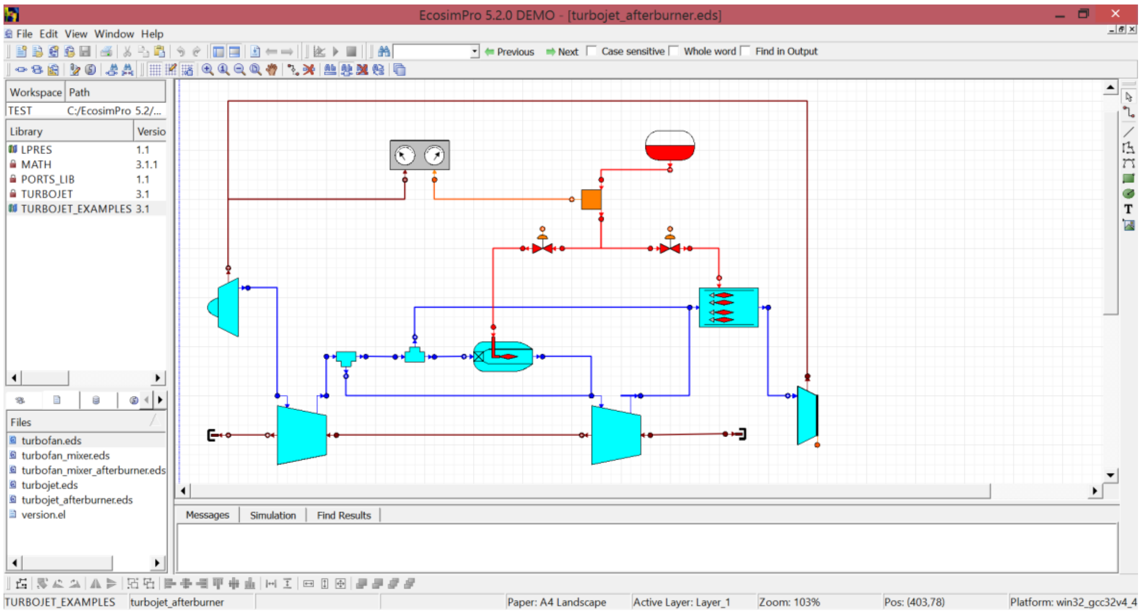


Figure 8 - Example of a schematic

The properties of schematics and possible operations on them are described in detail below.

6.2 Creating and opening a Schematic

To create a new schematic, click on  **New Schematic** on the general toolbar of the application.

In the **New Schematic** dialog box, give the new schematic an identifier and choose the library where you want to include the created component.

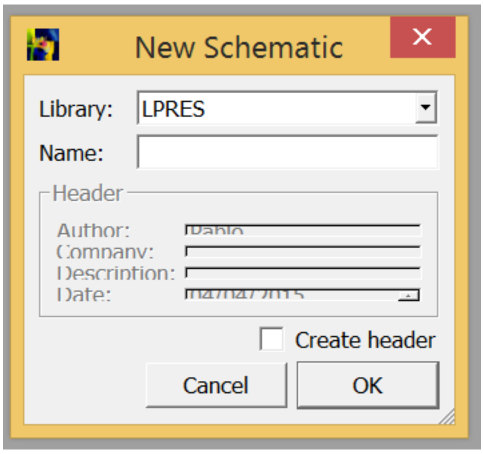


Figure 9 - Giving a name to the schematic

All the schematics of a library are accessible directly from the library file list which can be found in the **Files tab** of the library. The list can comprise source codes files (EL) and schematics.

To open a schematic, from the **Files tab** of the selected library, double click on the schematic.

6.3 Editing Schematics

The edition of a schematic allows you to create graphical displays formed by:

- Instances of symbols of other components and ports.
- Connectors that link components and ports.
- Basic graphical figures (Not explained in this manual).

The final result of the edition is a file that can be saved and compiled, in such a way that a new component with a topology formed by the other components is generated with it.

Adding Symbols

On the left-hand side of the window is the palette of available components (one icon for each component). The schematic must be created inside a library, but it can use components from other libraries. To do this, select the library with the components you want, and the palette containing them automatically appears in the **Symbols tab**.

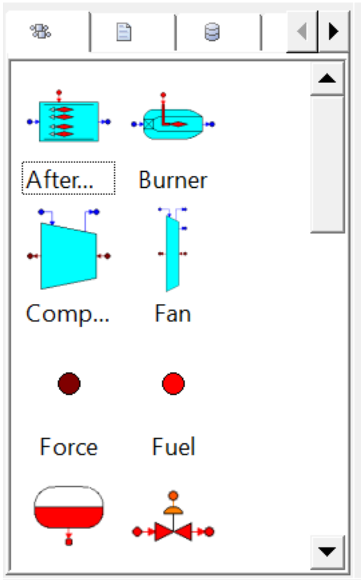


Figure 10 - Palette of available components

You can then drag and drop components from the palette onto the canvas.

Connecting Symbols

A connection between instances is a union of ports for transferring input and output data between them. To make a connection:

- Select the **Draw Connector** option on the Drawing toolbar.

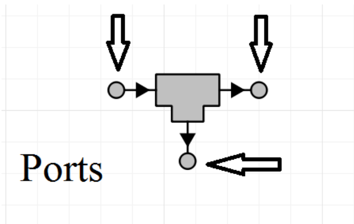


Figure 11 - Ports

- Situate the cursor over the origin port of the connector.
    - An anchor cursor appears; this means that you are squarely over the port.
    - If you hold the cursor still, information associated with the port will be shown in a tooltip.

- Left click on the port of origin to define the first point of the connection and continue with as many left clicks as intermediate points you think you need.

- Situate your cursor over the end port of the connector

    - An anchor cursor appears; this means that you are squarely over the port.
    - If you hold the cursor still, information associated with the port will be shown in a tooltip.

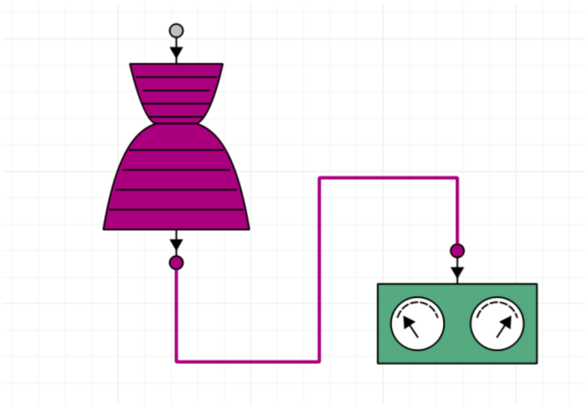


Figure 12 - Connection between two components

6.4 Attribute Editor

EcosimPro provides visual editors to enter the object properties. To edit an object, double-click on it. The figure below shows an example:

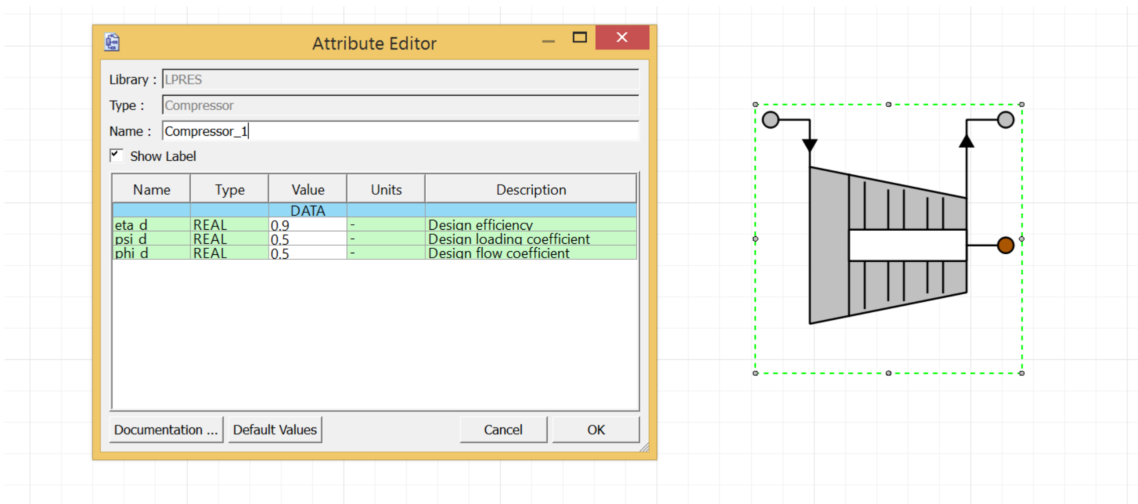


Figure 13 - Attribute Editor

The object editor accepts different data types: REAL, INTEGER, ENUMERATION, BOOLEAN and tables (1D, 2D and 3D). These editors allow you to modify the attributes (name, parameters, data…) of an object instance intuitively.

When a parameterized port or component is instantiated, the value of the construction parameters must be specified and they cannot be changed during the simulation. One of the ways in which they differ from data is that the values of data can change during an experiment, but construction parameters never do. Construction parameters behave as initialization constants of the component (without their values, it cannot be generated).

Variables are calculated in the simulation, they are not attributes of the component. Hence, they cannot be fixed here.

6.5 Schematic Compilation

Finally, selecting the **Compile** option creates a new component in EcosimPro, if there are no mistakes. The new component appears in the **Items tab** of the library. This component is then simulated normally, as if it had been created using the EcosimPro language (EL).

If the compilation was correct, the **Message window** should read like this:

Figure 14 - Compilation OK

The schematic has been compiled once it has been edited, so that the model topology and characteristics it represents become an item that can then be used in other components.

7 Partition Management

Partitions are mathematical representations of a physical model under a given set of conditions. Each component can have as many associated partitions as combinations of conditions can be defined for that component.

A component represents a specific physical system defined by its equation system which, in general, is not mathematically closed and needs an additional set of conditions to be able to calculate all the variables (e.g., a set of boundary variables when there are more variables than equations). Moreover, it could contain sets of implicit equations, non-linear equation systems for which it is not possible to establish an explicit solution sequence or high index problems that require the establishment of specific solution mechanisms. Additionally, it is possible to calculate the values of certain data (values that are known hypotheses of a model) so other sets of conditions are fulfilled. These are usually called design models.

7.1 Creating different types of Partitions

Default Partitions

Default partitions are those generated automatically by the program, in this sense, the better the modelling libraries, the better the default partitions. In this type of partition the program selects, if necessary, which derivative values should be excluded, which boundary conditions are the most suitable and/or which variables should be used as algebraic in the non-linear equations systems.

To create a default partition, right-click on the component the partition is being created for. A contextual menu will appear, like the one below, where you can select the **New Default Partition** option.

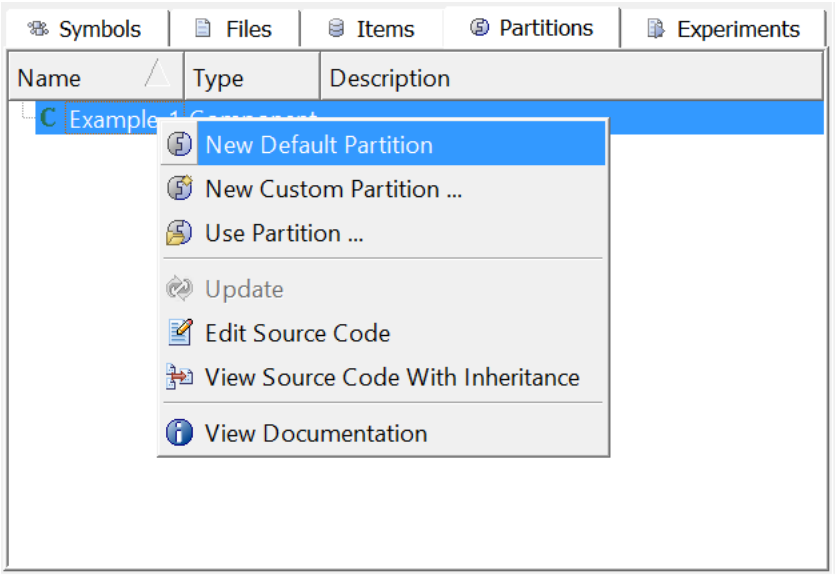


Figure 15 - Create a new default partition

The **Partition Name** dialog will appear, like the one below, and will ask for a name to identify the partition. Click **OK** to save the partition or **Cancel** if you don't want to create it.

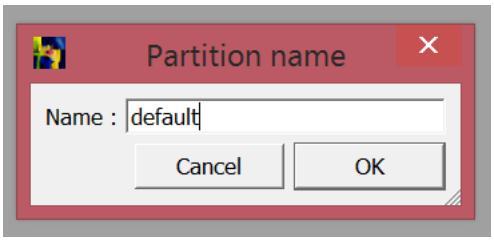


Figure 16 - Give a name to the partition

The workspace area partition tree will be updated automatically to show the newly created partition.

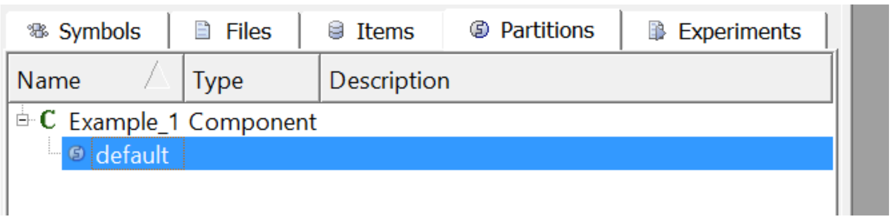


Figure 17 - Created partition

The set of excluded variables in the process of creating a default partition for a given component are shown as suggested variables if the partition is edited or if the partition is created using the **New Custom Partition** option.

Custom Partitions

Custom partitions are created by means of a wizard that guides the user through the process of generating a partition. This type of partitions can be created using the **New Custom Partition** option. See section *7.3 Steps in the Partition wizard*, which explains the different steps of the partition wizard.

To create a custom partition, right-click on the component the partition is being created for. Select the **New Custom Partition** option on the contextual menu.

7.2 Editing, validating and deleting Partitions

Once a partition has been created, it can be modified, validated and deleted by simply right-clicking on it. A contextual menu will be displayed where you can select **Edit** to modify any option, **Validate**, to check that the selected options are still valid after any changes are made to the modelling of the component or **Delete**, to delete it.

7.3 Steps in the Partition wizard

The partition wizard is used to create a new partition and to edit an existing one. It offers a dialog that allows dealing with the different problems that can appear when creating a valid mathematical model.

The complete set of wizard panels is as follows:

- Partition settings panel
- Panel to manage the properties of variables, for instance, the alias (optional)
- Panel to transform data into unknown variables (optional when creating the partition; it will appear only if necessary while editing the partition)
- High index problems panel (will appear only if necessary)
- Boundary conditions selection panel (will appear only if necessary)
- Algebraic variables selection panel (will appear only if necessary)
- Final panel, used to display a summary of the user selection and to give a name to the partition

During the partition creation process, the wizard intelligently and automatically inform the user in each case which variables are the most mathematically suitable for solving each problem. However, this is only an aid to the user who, at any time, can select those that best suit his needs, delegating to the wizard the sole task of checking that the selection is acceptable. The following sections review the different steps of the partition wizard.

Partition settings panel

The purpose of this panel (**PARTITION SETTINGS**) is to change basic and advanced settings related to partitions. The panel of the wizard corresponding to the basic settings is shown below.

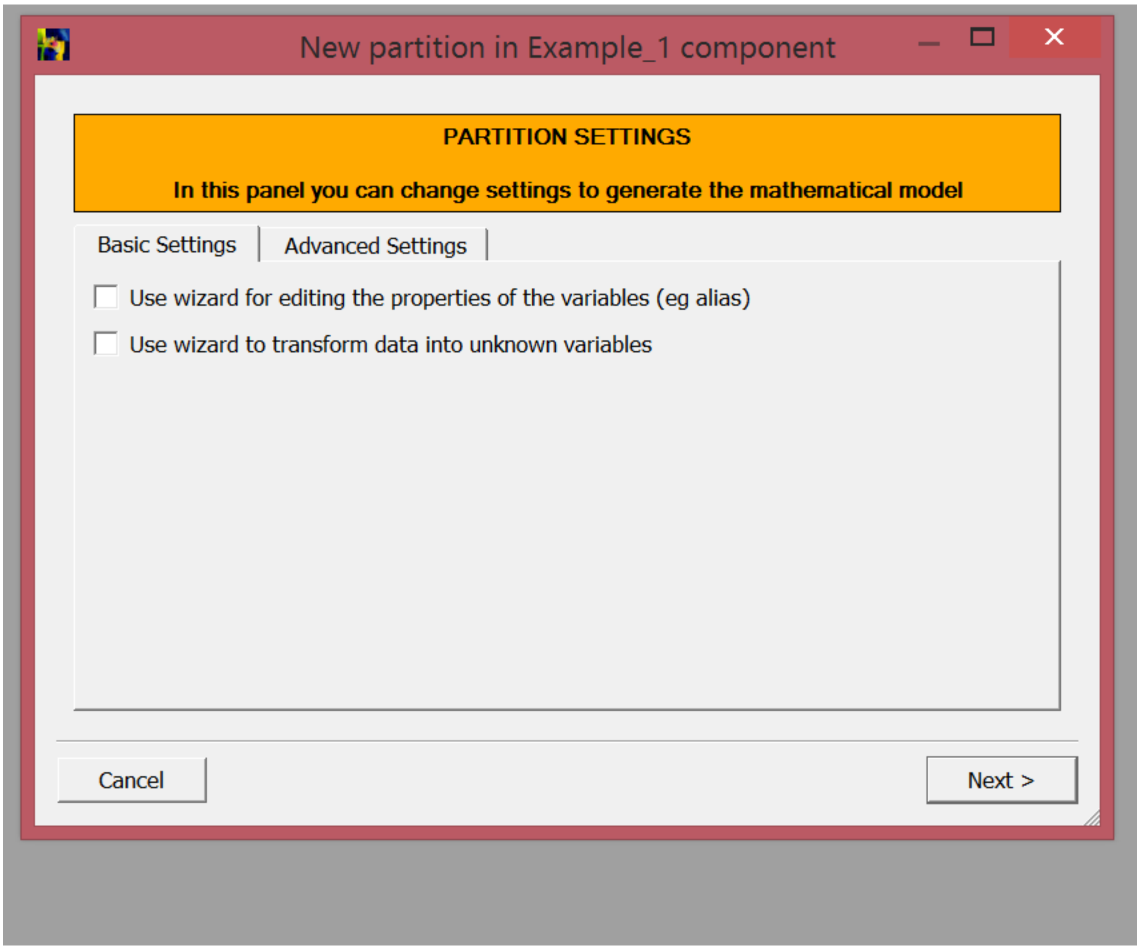


Figure 18 - Partition settings panel

As this is a basic manual, just click **Next >**.

If the first of the basic settings (**Use wizard for editing the properties of the variables**) is selected, the panel for managing the properties of variables (**PROPERTY WIZARD**) will appear.

If the second of the basic settings (**Use wizard to transform data into unknown variables**) is selected, the panel for transforming data into unknown variables (**DATA TRANSFORMATION WIZARD**) will appear. This panel is not explained in this manual.

The high index problems panel (**HIGH-INDEX WIZARD**) is not explained in this manual either.

Panel for transforming data into unknown variables

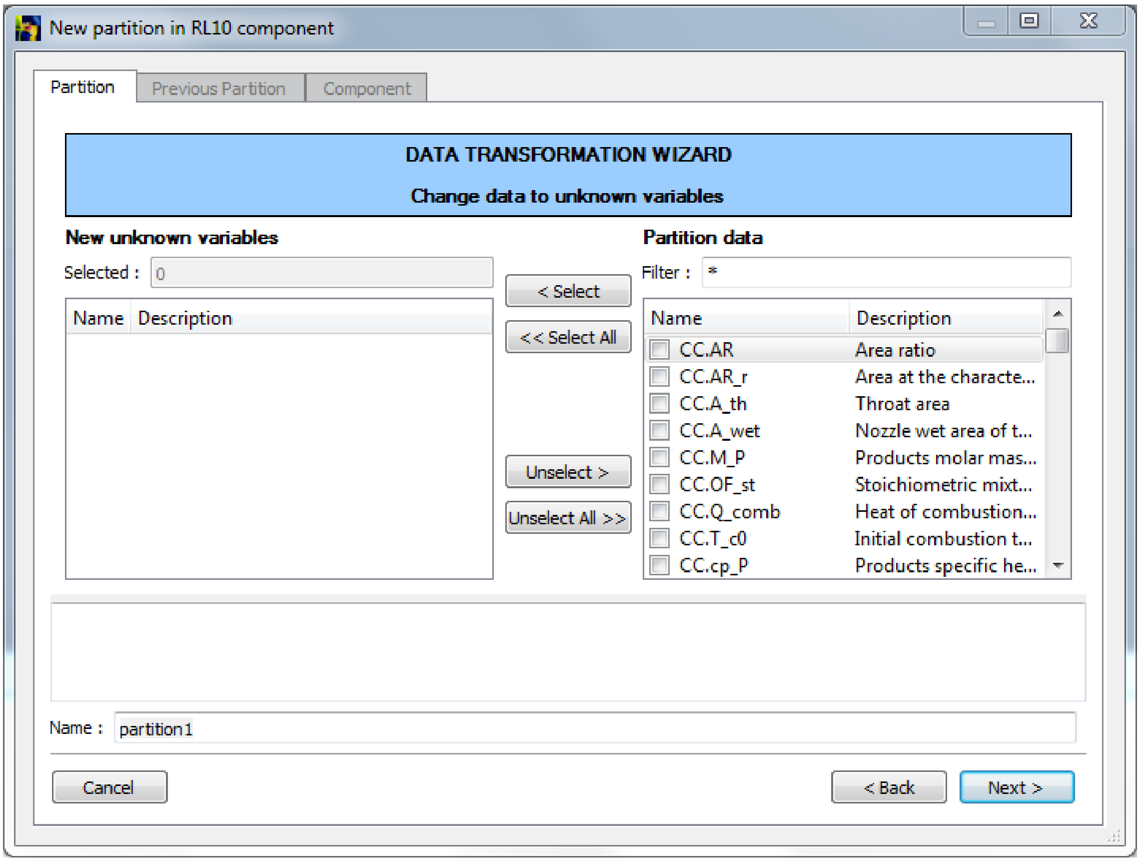


Figure 19 - Panel for transforming data into unknown variables

In this step, a specific amount of data can be released so that it can be calculated or designed on the basis of a series of extra conditions; i.e. variables which are considered to be known in a specific component become unknown and consequently the number of boundary conditions needed increases. These additional boundary conditions are called design conditions.

The **Design variables area** displays the set of data that has been released, now considered as unknown variables, while the **Component data area** displays the complete list of component data.

To convert component data to variables select them by checking the box on the left of the name and clicking the **< Select** button, or double-click on any of them.

All the variables displayed in the **Component data area** can be converted at one time by clicking the **<< Select All** button. To unselect data, check the box to the left of the name and then click the **Unselect >** button, or double-click on each one.

Filters can be established for the variables that are displayed in the **Component data area** to facilitate searches. To do this, write a sequence of characters representative of the name you are looking for. This can include the "?" and "*" wildcards to replace one or several characters respectively.

Information, incident and error messages that may appear during the selection process are displayed in the output area of the lower part of the wizard.

Boundary conditions selection panel

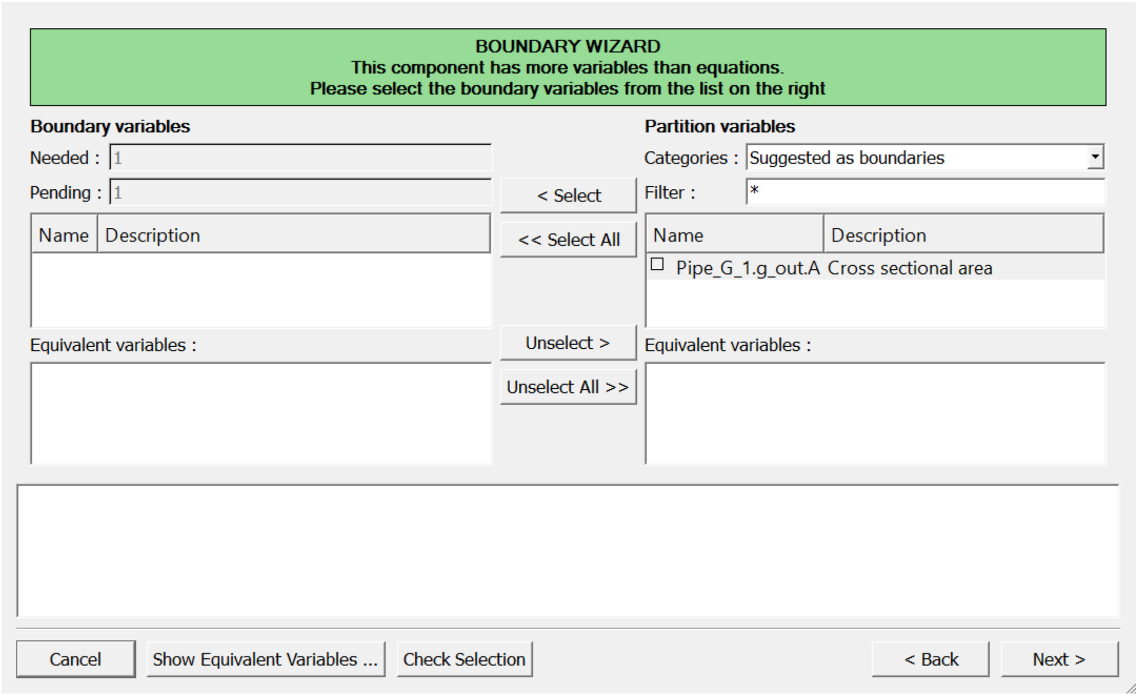


Figure 20 - Boundary conditions section panel

This wizard (**BOUNDARY WIZARD**) appears when there are more variables than equations in a mathematical model. If this happens, it is necessary to select a set of variables equal to the difference between the number of variables and equations, which will be used as model boundary conditions.

The total number of variables that should be selected as boundary conditions and those that are still missing are shown on the left and a set of options that helps with selecting the boundary conditions is displayed on the right.

The **Boundary variables area** displays the set of variables that have been selected to be used as boundary conditions, while the **Partition variables area** displays the set of variables established by the **Categories** criterion.

These options include **All**, which displays all the component variables, **Able to be selected**, which displays only those that comply with the minimum selection requirements. **External**, for those port variables that can be selected as boundary conditions and **Suggested as boundaries**, for the set of variables proposed by the tool.

In the event that there is any equivalence between variables, the set of variables that is considered to be the same as the one selected is displayed below each of these areas.

To convert candidate variables to boundary conditions, select them by checking the box to the left of the name and clicking the **< Select** button, or by double-clicking on each one.

All the variables displayed in the **Partition variables area** can be converted at one time by clicking on the **<< Select All** button.

To unselect a variable as a boundary condition, check the box on the left of the name and then click the **Unselect >** button, or double-click on each one.

Filters can be established for the variables that are displayed in the **Component variables area** to facilitate searches. To do this, write a sequence of characters representative of the name you are looking for. This can include the "?" and "*" wildcards to replace one or several characters respectively.

Information, incident and error messages that may appear during the selection process are shown in the output area in the lower part of the wizard.

Algebraic variables selection panel

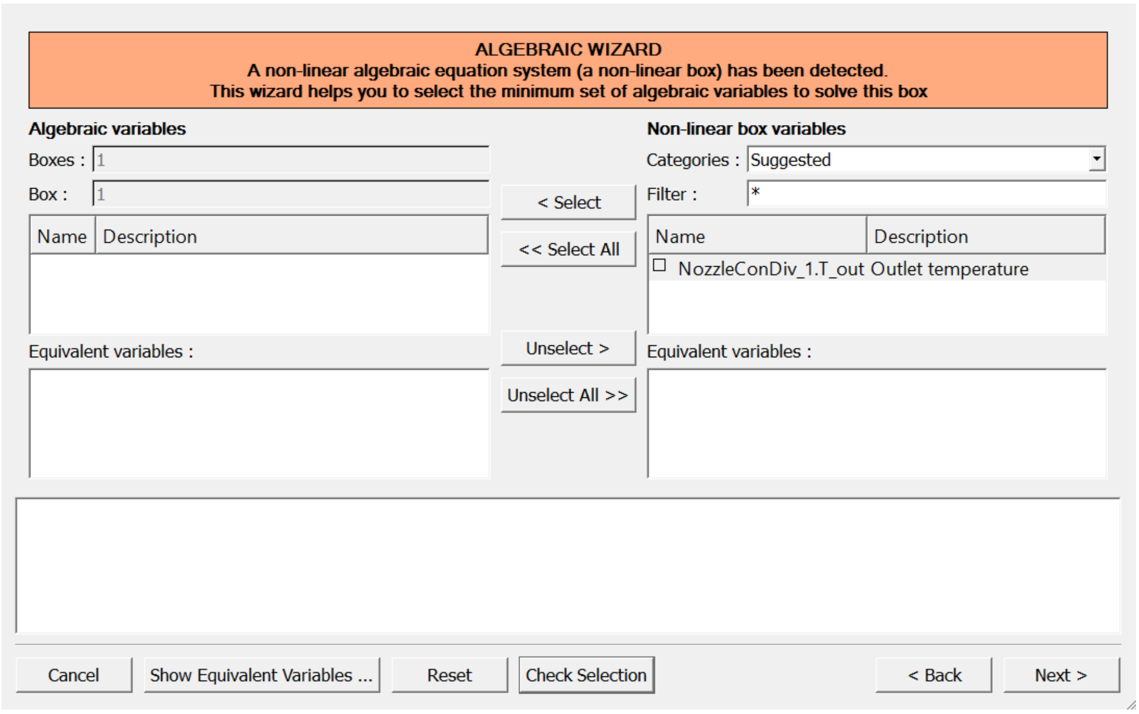


Figure 21 - Algebraic variables selection panel

This wizard (**ALGEBRAIC WIZARD**) appears when there is a set of implicit equations in a mathematical model that have to be solved by iteration. In this case, it will be necessary to select the variable or the set of variables that will be used for the iterative process as algebraic variables. For this, an initial value will need to be taken.

The total number of implicit or non-linear equation systems (called non-linear boxes) and the number of the one being solved at the moment are displayed on the left, while there is a combination with a series of options that helps with the selection of algebraic variables on the right.

The **Algebraic variables area** displays the set of variables that have been selected as iteration variables to solve the system. The **Non-linear box variables area** displays the set of variables established by the **Categories** criterion.

These options include **Box vars**, which displays all the equation system variables, and **Suggested**, which displays the set of variables proposed by the tool.

In the event that there is any equivalence between variables, the set of variables that is considered to be the same as the one selected is displayed below each of these areas.

To convert box variables into iteration variables, select them by checking the box to the left of the name and click on the **< Select** button, or double-click on each one.

All the variables displayed in the **Algebraic variables area** can be converted at one time by clicking on the **<< Select All** button.

To unselect an iteration variable, check the box to the left of the name and then click on the **Unselect >** button, or double-click on each one.

Filters can be established for the variables that are displayed in the **Non-linear box variables area** to facilitate searches. To do this, write a sequence of characters representative of the name you are looking for. This can include the "?" and "*" wildcards to replace one or several characters respectively.

Information, incident and error messages that may appear during the selection process are displayed in the output area of the lower part of the wizard.

Summary panel

This is the final panel of the wizard. It provides a summary including both automatic and user decisions taken as the wizard progressed. You can also give a name to the partition. Press the **Finish** button to generate the partition.

8 Experiment Management

Once the system to be simulated has been modelled and a partition created (a mathematical model), the next step is to configure the simulation. In EcosimPro this is done through EL language in so-called experiments, where we can perform steady state, parametric and transient calculations, impose initial conditions, etc.

Only steady state calculations are explained in this manual.

The normal operations done on experiments are their creation, the editing of the EL code and, finally, their execution or running, either in the graphic model using the graphics assistant (**Monitor**) or in batch simulation.

All of these operations are done from the **Experiments tab**.

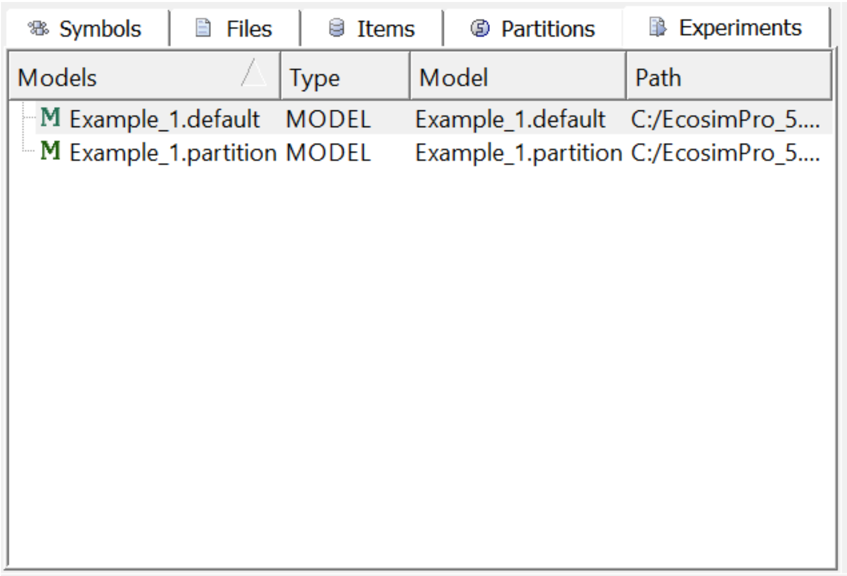


Figure 22 - Experiments tab

This is the only section of the manual in which EL Language is used.

8.1 Creating an Experiment

To create an experiment we have to go to the partition tree and then right-click on the partition where we want to create it. From the menu which appears, we select the **New Experiment** option.

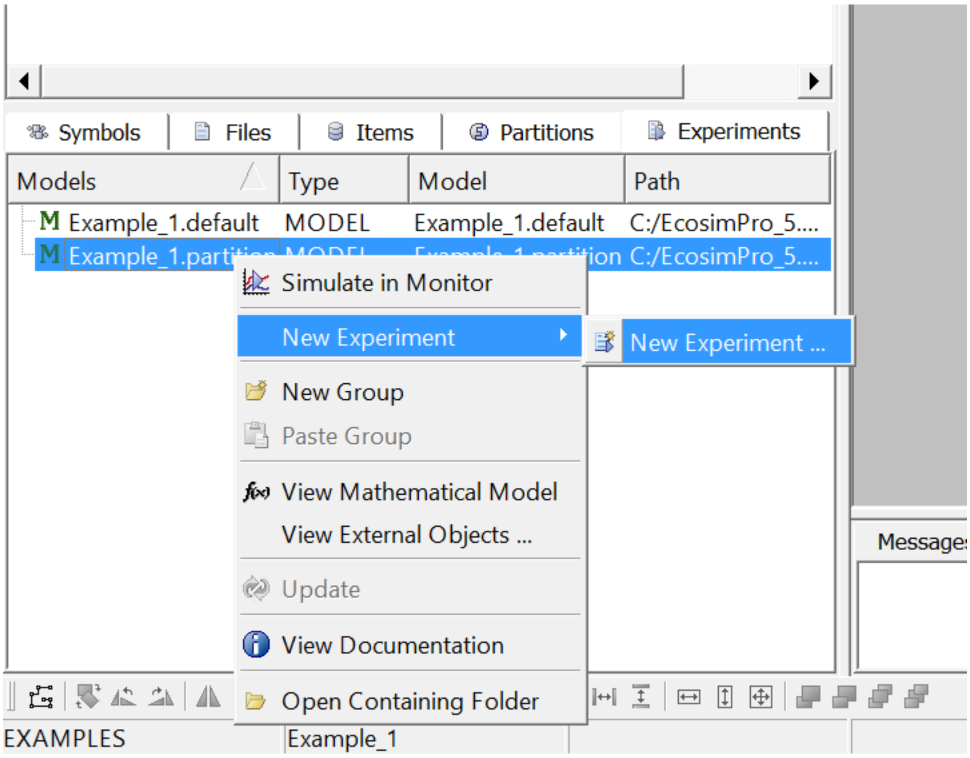


Figure 23 - Create a new experiment

A dialog box will then appear in which we type in the name of the new experiment.

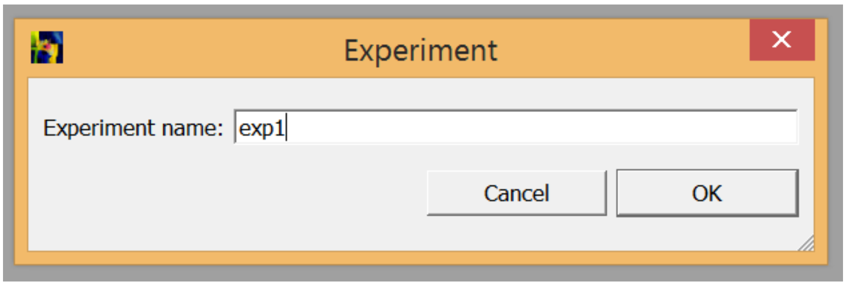


Figure 24 - Give a name to the experiment

Click OK to launch a new edit window with the default code for the experiment.

```
EXPERIMENT exp1 ON Example_1.partition
   DECLS
   OBJECTS
   INIT
      -- initial values for algebraics
      NozzleConDiv_1.T_out = 0

   BOUNDS
      -- Set equations for boundaries: boundVar = f(TIME;...)
      NozzleConDiv_1.gn_out.A_th = 1

   BODY
      -- report results in file reportAll.rpt
      REPORT_TABLE("reportAll.rpt", "*")
      -- Set the tolerances (relative and absolute);
      REL_ERROR = 1e-006
      ABS_ERROR = 1e-006
      -- Integrate the model
      TIME = 0
      TSTOP = 15
      CINT = 0.1
      INTEG()
END EXPERIMENT
```

Figure 25 - Generated code

At the same time, the experiment we have just created will appear as a new item in the partition tree:

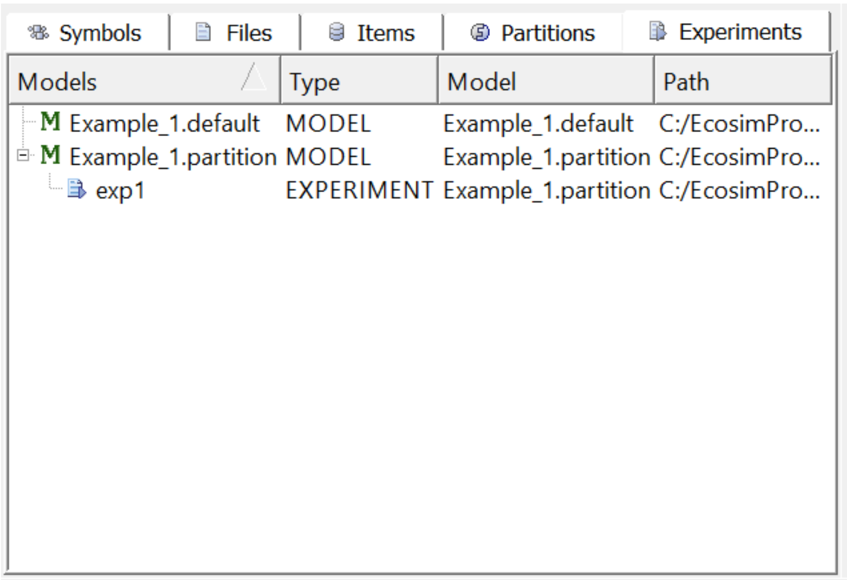


Figure 26 - Created experiment

8.2 Editing and Compilation of Experiments

Once the experiment is created, it is time to modify the generated code to impose the desired boundary conditions...

1. The piece of code headed by the keyword `BODY` must be deleted and replaced by `STEADY()`. This is how we tell the program that a steady state calculation is required in the experiment.

2. The initial values for algebraics are included in the piece of code headed by the keyword `INIT`. These values have to be replaced by others which we believe will be close to what they should be after calculating. These values are not the final values of the variable, but just an initial starting point of the iterations.

3. The piece of code headed by the keyword `BOUNDS` includes the values of variables considered as boundary variables in the partition. Replace these values with the ones you want them to be. These values are the final values of the variables.

The values given to the boundary variables don't have to be constant, they can be variable relationships over time. To refer to time in EL Language, write `TIME`.

```
BOUNDS
   -- Set equations for boundaries: boundVar = f(TIME;...)
   F = 500*step(TIME,5)

```

Figure 27 - Example of time-dependent condition

NOTE: `step(TIME,5)` is an internal function of the EcosimPro program which means an unitary step in the variable `TIME` when the value of the variable is 5.

Finally, to indicate the results that you want to obtain from the experiment, write below the word `STEADY()`:

```
PRINT(" $variable ")
```

The program will write, once the experiment is run, the value of the variable as a message.

Among the "" can write more things in order, for example, to identify the variable:

```
PRINT(" variable_1 = $variable ")
```

In this case, the message will be: `variable_1 = variable_value`

This is an example:

```
EXPERIMENT exp1 ON example_1.partition
   DECLS
   OBJECTS
   INIT
      -- initial values for algebraics
      CombCha_OG_FG_1.p_out = 100
      CombCha_OG_FG_1.T_out = 300

   BOUNDS
      -- Set equations for boundaries: boundVar = f(TIME;...)
      CombCha_OG_FG_1.AR = 2
      CombCha_OG_FG_1.gi_F.W = 0.1
      CombCha_OG_FG_1.gi_O.W = 0.5
      LPRES.Altitude = 0

   BODY
      STEADY()

      PRINT(" ")
      PRINT("-----------------------------------------------------------------")
      PRINT(" ")
      PRINT("  RESULTADOS:")
      PRINT(" ")
      PRINT("  A_out = $CombCha_OG_FG_1.A_out")
      PRINT("  A_th = $CombCha_OG_FG_1.A_th")
      PRINT("  T_c = $CombCha_OG_FG_1.T_c")


END EXPERIMENT
```

Figure 28 - Example of an experiment

To know how to write a variable in EL Language:

- If the variable is associated with a port: `component_name.port_name.variable_name`

- If the variable is not associated with a port, but is as variable within the component itself: `component_name.variable_name`

Once the code has been modified, it can be saved or directly compiled (which also involves an associated save operation).

The new code can be compiled by clicking  on the standard toolbar, or by highlighting the experiment in the tree and selecting the **Compile** command.

Errors may occur when the experiment code is being compiled. All compilation messages will appear in the general **Output** of the tool.

9 Simulation

As soon as an experiment has been compiled correctly it can be simulated in two different ways: simulation in monitor and batch simulation from the interface (without using **Monitor**). These will be explained below in more detail.

9.1 Batch Simulation

For batch simulation right-click on the experiment to be simulated. A menu will appear; select the **Simulate** option (don't confuse it with **Simulate in Monitor**).

The experiment will start and the associated messages will appear in the general **Output** under the **Experiments tab**.

The experiment open in the active window can also be simulated through batch simulation by clicking on the button on the tool bar or by pressing F5.

9.2 Simulation in Monitor

To simulate an experiment using Monitor, we must first open the associated window by selecting the **Simulate in Monitor** option from the menu.

Monitor will open in another window. If this is the first time the experiment is being simulated it will be blank and there will be no associated graphical configuration.

The experiment open in the active window can also be simulated in the monitor by clicking on the button on the tool bar or by pressing F6.

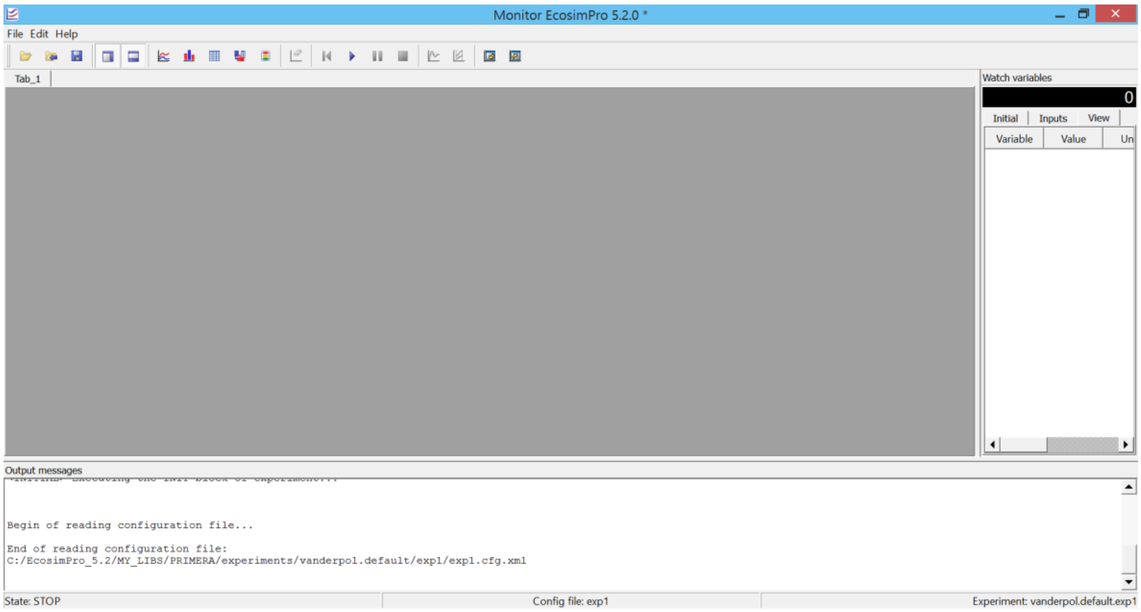
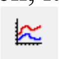


Figure 29 - Monitor

Once the monitor is open, it is possible to create plots in addition to other options. To create a plot, press the  button. The **New plot** dialog box appears. Select the variables that are shown on the "Y" axis. If nothing else is changed, the "X" axis is the TIME.

To change the variable of the "X" axis, select the Variable option, next to the TIME option at the end of the dialog box, click on the variable that you want to be in the "X" axis and then press the **<< Select** button, but this must be done **before selecting the variables in the "Y" axis**.

Then, press **OK** and a blank plot will appear.

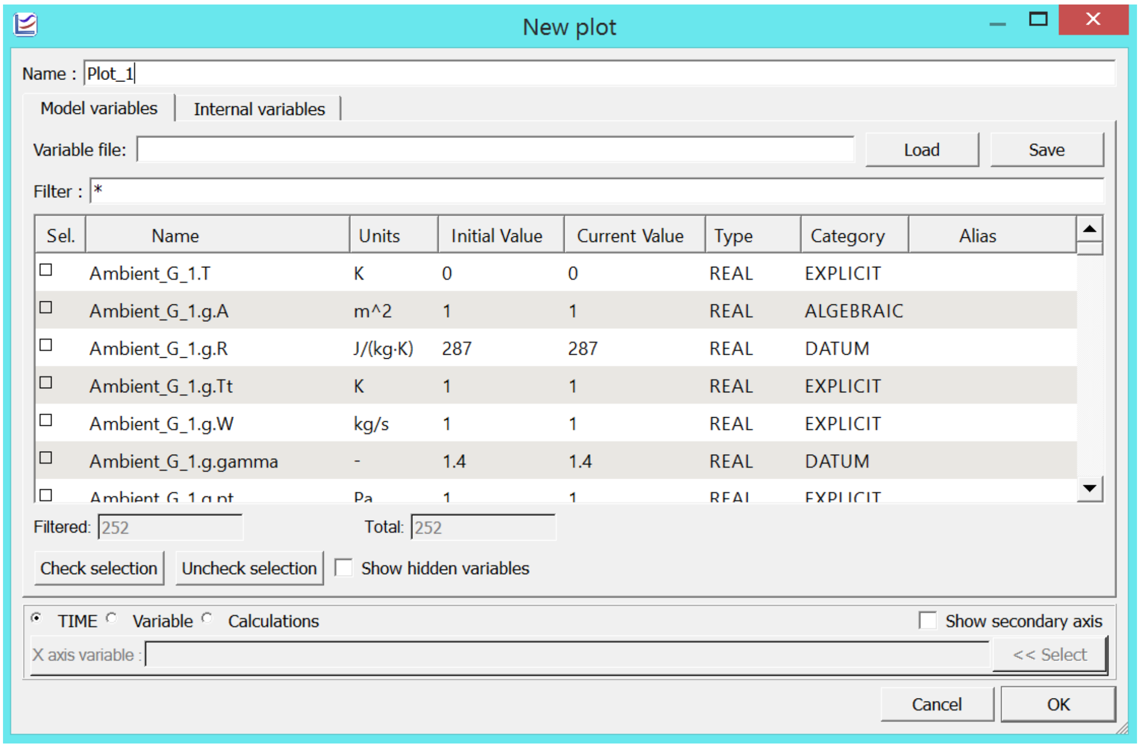


Figure 30 - New plot

Finally, press  to run the simulation.

The experiment will start and the associated messages will appear in the general **Output messages area** (at the end of the **Monitor**).

The **Monitor** has other possible uses that are not explained in this manual. Still, it is a very intuitive environment so you should be able to handle it easily.

For example, the **Watch variables area**, situated on the right-hand side of the **Monitor**, is one environment in which it is advisable to test. The main task of the **Watch** is to display the variable, data and initial values of the experiment in an organized manner, enabling them to be modified wherever possible. Right clicking on **Watch variables area** will open a menu. Select **Edit watch** to choose the variables that you want in it. In addition to selecting the variables, all the results of the experiment can be seen in the dialog box that appears once the simulation has been run.

References

ECOSIMPRO 5.2 USER MANUAL. Empresarios Agrupados. Edition 2013

ECOSIMPRO 5.2 INSTALLATION AND GETTING STARTED. Empresarios Agrupados. Edition 2013

INTRODUCCIÓN AL MODELADO Y SIMULACIÓN CON ECOSIMPRO. Francisco Vázquez, Jorge Jiménez, Juan Garrido and Antonio Belmonte. 2010, Pearson

www.ecosimpro.com

Table of figures