The OpenAltaRica Platform

GETTING STARTED

Sumary

This document quickly presents the OpenAltaRica platform dedicated to MBSA (Model-Based Safety Assessment) and based on the AltaRica 3.0 modeling language.

TABLE OF CONTENTS

1. Preliminaries

AltaRica 3.0 is the new version of the modeling language AltaRica dedicated to risk analysis (safety, reliability, performance) of complex systems.

The OpenAltaRica platform provides several tools to perform such analyses. This platform is protected by French laws and provided under a specific license. For any information or problem, see the OpenAltaRica forum or contact us. Finally if ever, report a bug (if not available, see the OpenAltaRica forum or contact us).

This document presents the overall process to install and configure the OpenAltaRica platform. According to the evolution of the platform, this document will change (a lot). Please check you are up to date for both this document and the platform.

2. Installation

The OpenAltaRica platform is (will be ☺) available for Windows, Linux and macOS operating systems. The following describes the entirely process to install the OpenAltaRica Platform on your operating system. For any information or problem, see the OpenAltaRica forum or contact us.

To download and install the OpenAltaRica platform, go to the OpenAltaRica downloads page and download the suited package according to your operating system.

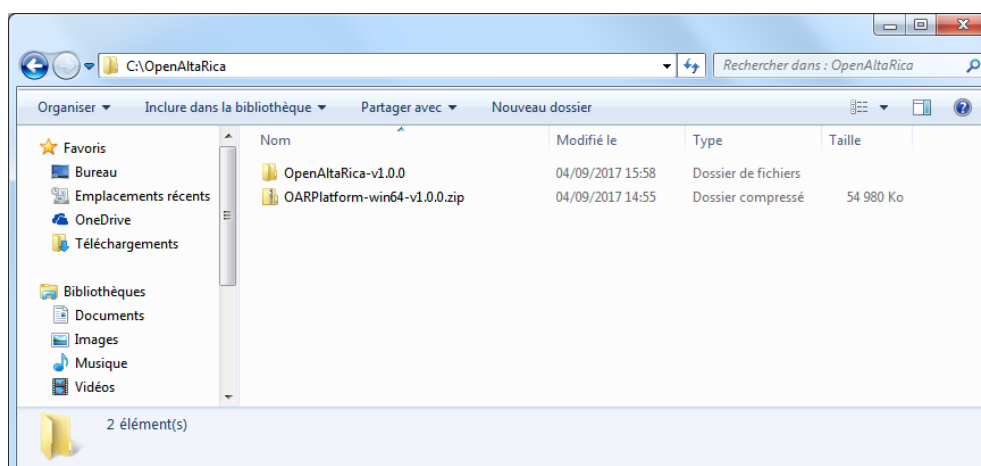## 1. Windows operating system

Extract the downloaded compressed file into the folder of your choice:

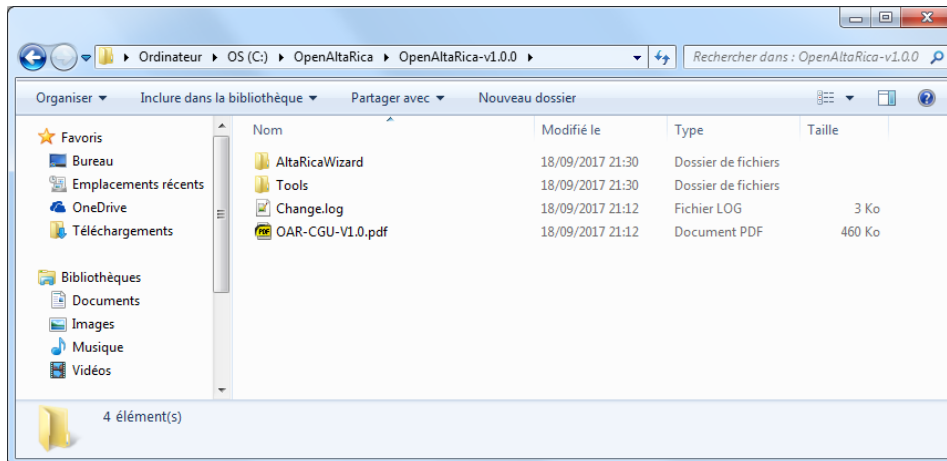<div align="center">e.g. 'C:\OpenAltaRica'</div>

You can rename this extracted folder, e.g. 'OpenAltaRica-v1.0.0';

WARNING:

- ensure you have full "Read" and "Execute" permissions on this folder;
- ensure the absolute file system path to this folder (also including this folder) does not include spaces and parenthesis;


The OpenAltaRica Platform directory contains several elements:

- The folder 'AltaRicaWizard', including the AltaRica 3.0 editor to design AltaRica 3.0 models;
- The folder 'Tools' including tools to perform analyses on AltaRica 3.0 models or other models (e.g. fault trees);
- The file 'Change.log' providing information about modifications between the previous version and the actual;
- The file 'OAR-CGU-v1.0.pdf' providing the terms of use (in French) for default Users.

Go to the directory 'AltaRicaWizard'. The binary file 'AltaRicaWizard.exe' (or 'AltaRicaWizard' if files' extension are not shown) is the dedicated editor to design AltaRica 3.0 models.
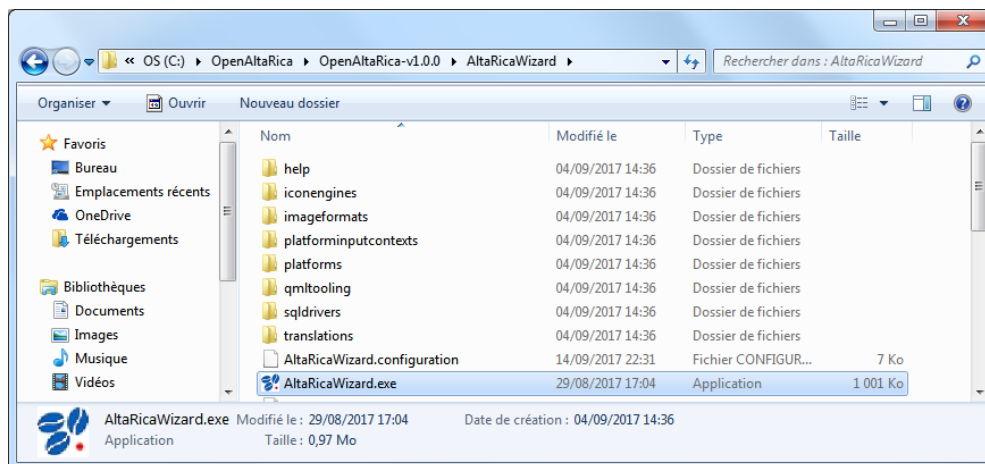
2. Linux operating system
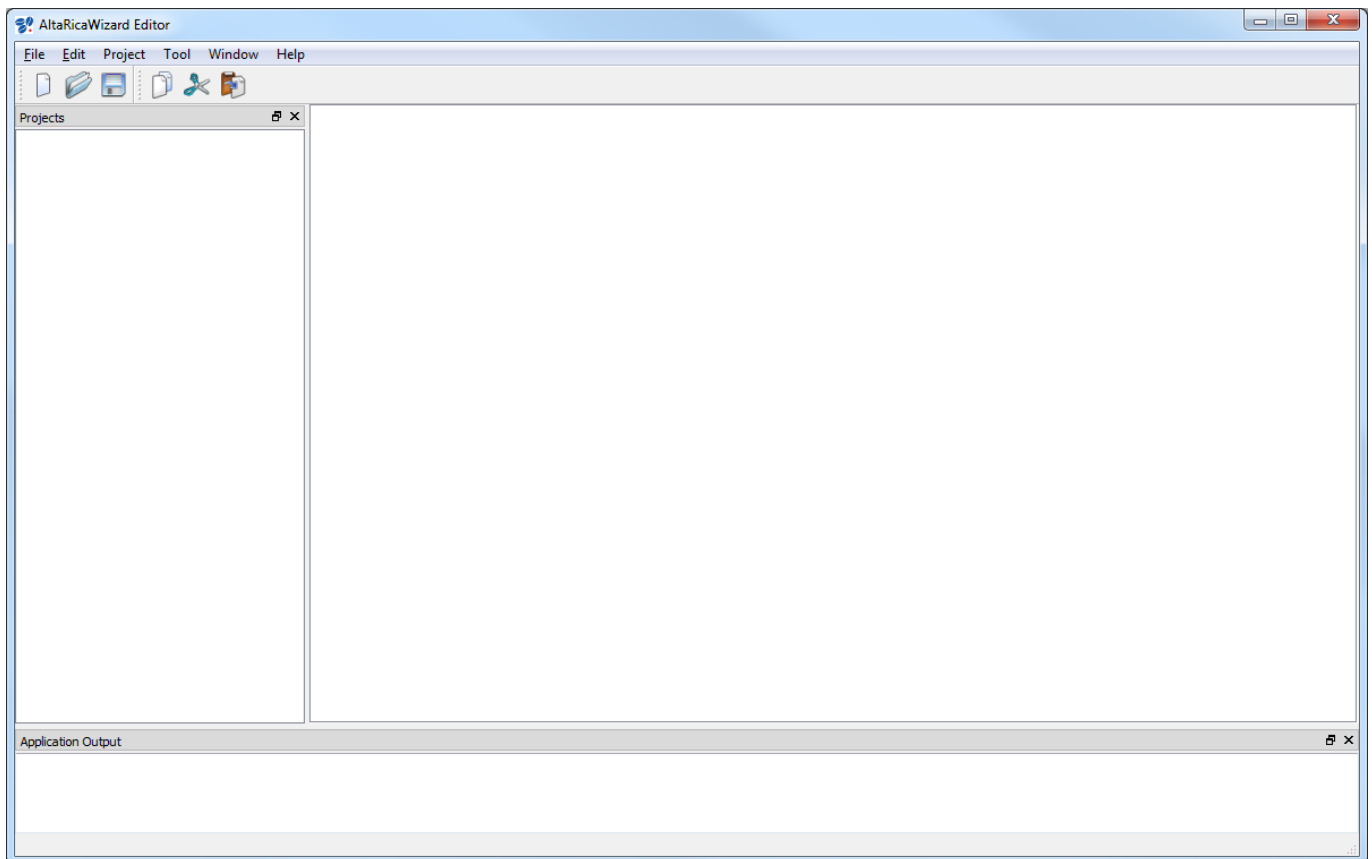
Coming soon!

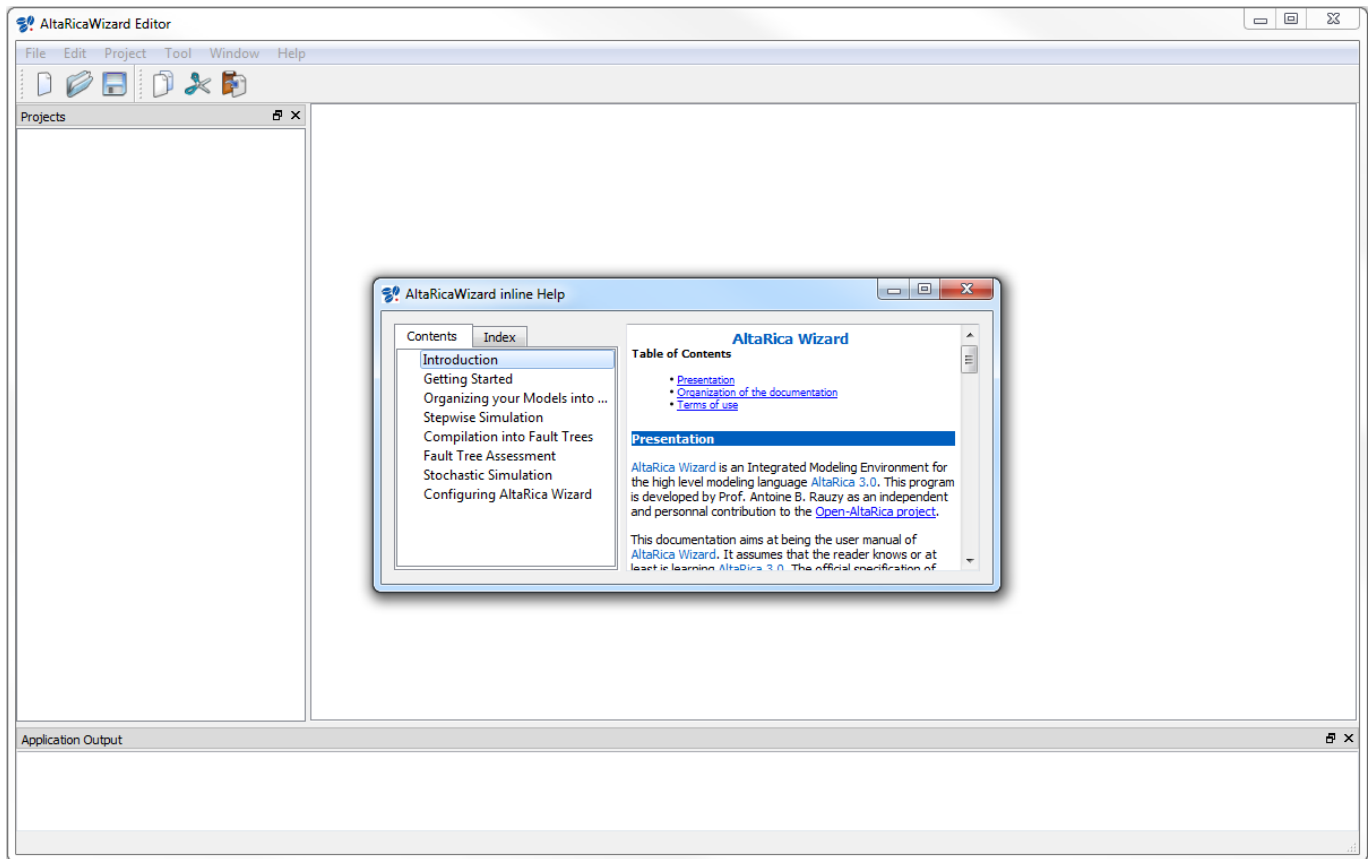## 3. macOS operating system

Coming soon!

3. Uses

Go to the directory 'AltaRicaWizard' and start the binary 'AltaRicaWizard'
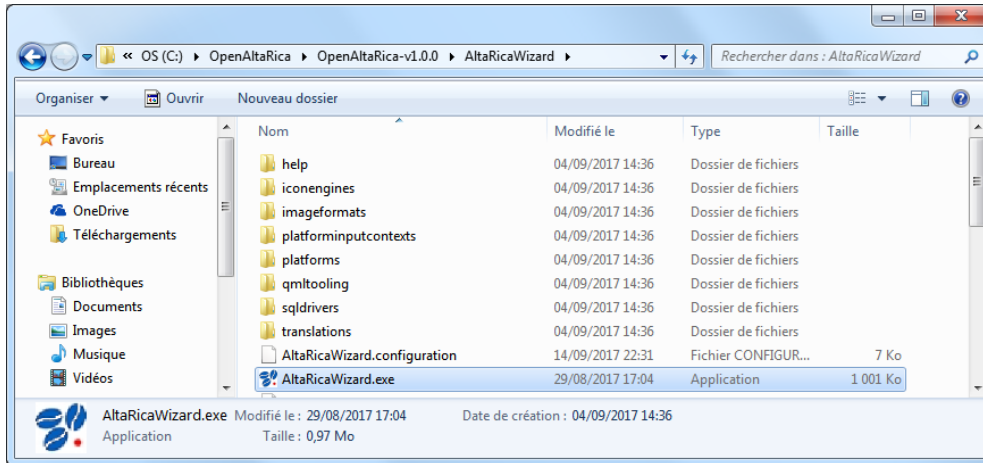
A new window opens, it is the dedicated editor to design AltaRica 3.0 models:

The first and important thing to remember is the help: Click on 'Help' and select 'Help'.
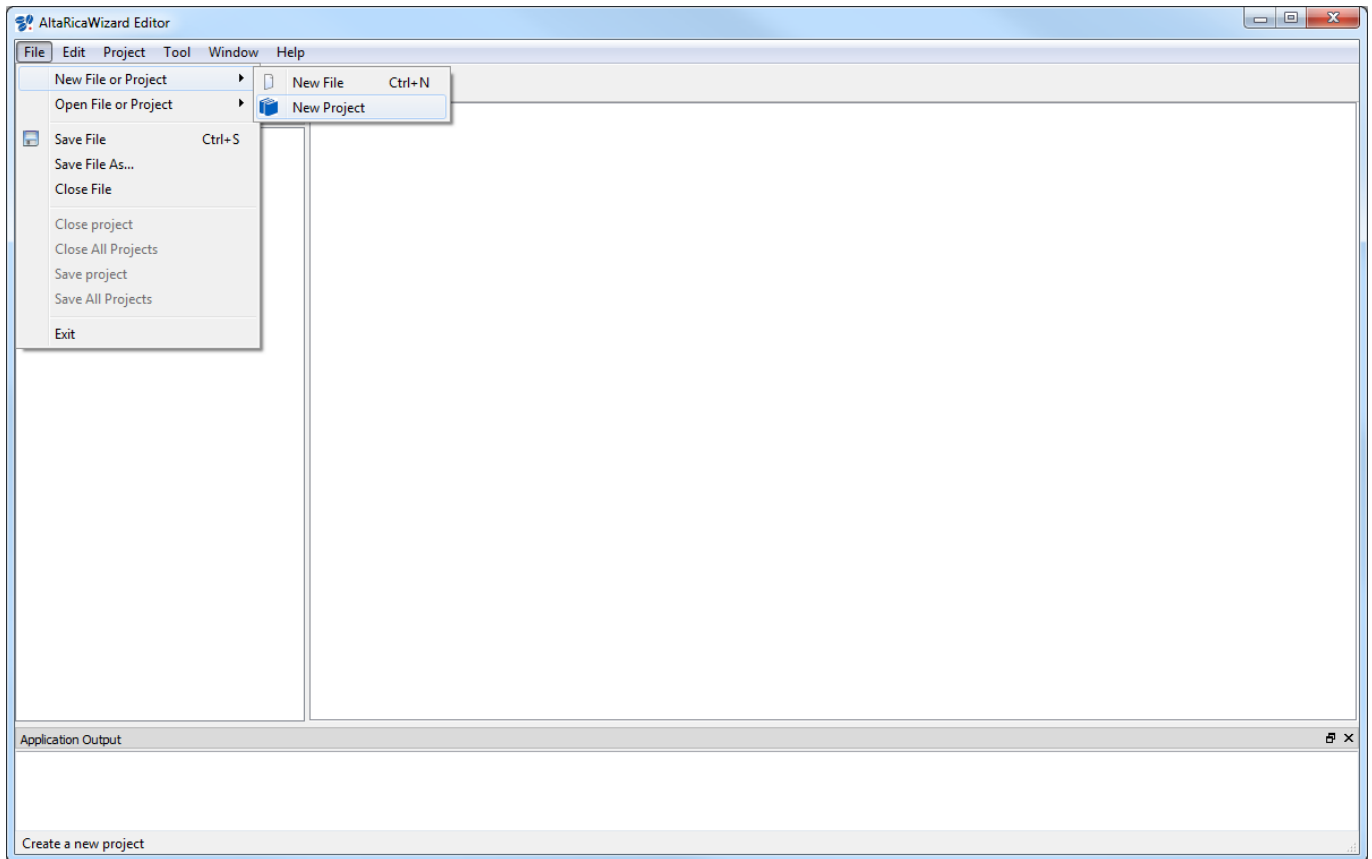
After launching this AltaRicaWizard, you can now use it.

4. Create a new project
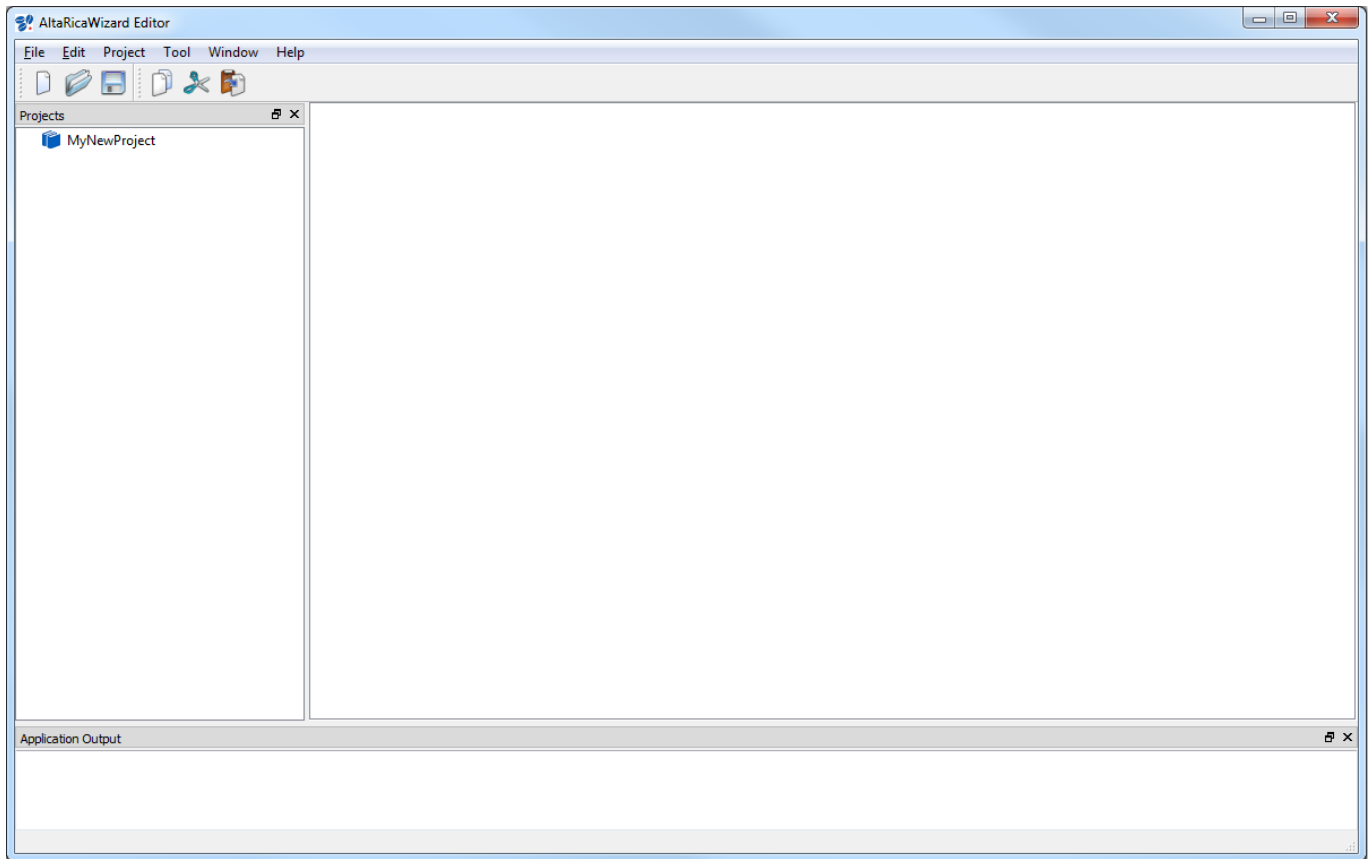
To create a new project, follow these steps:

1. Click on 'File', select 'New File or Project' and 'New Project';

2. A new window opens. Enter the name of your new project (e.g. 'MyNewProject'), click on 'Browse…' to define its location. The button 'Create new directory' must be clicked if you want to create the directory according to the name of your project. Finally click on 'OK';
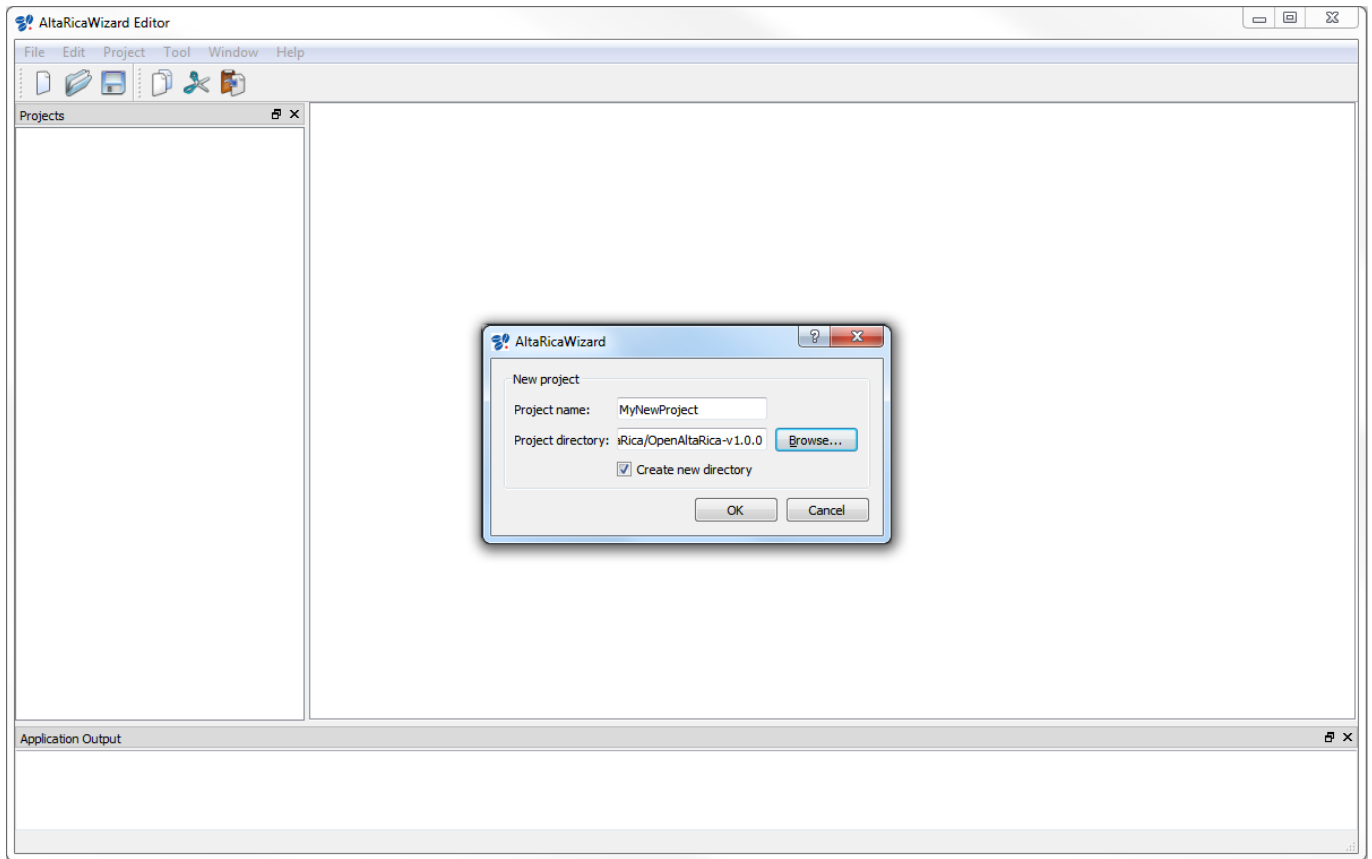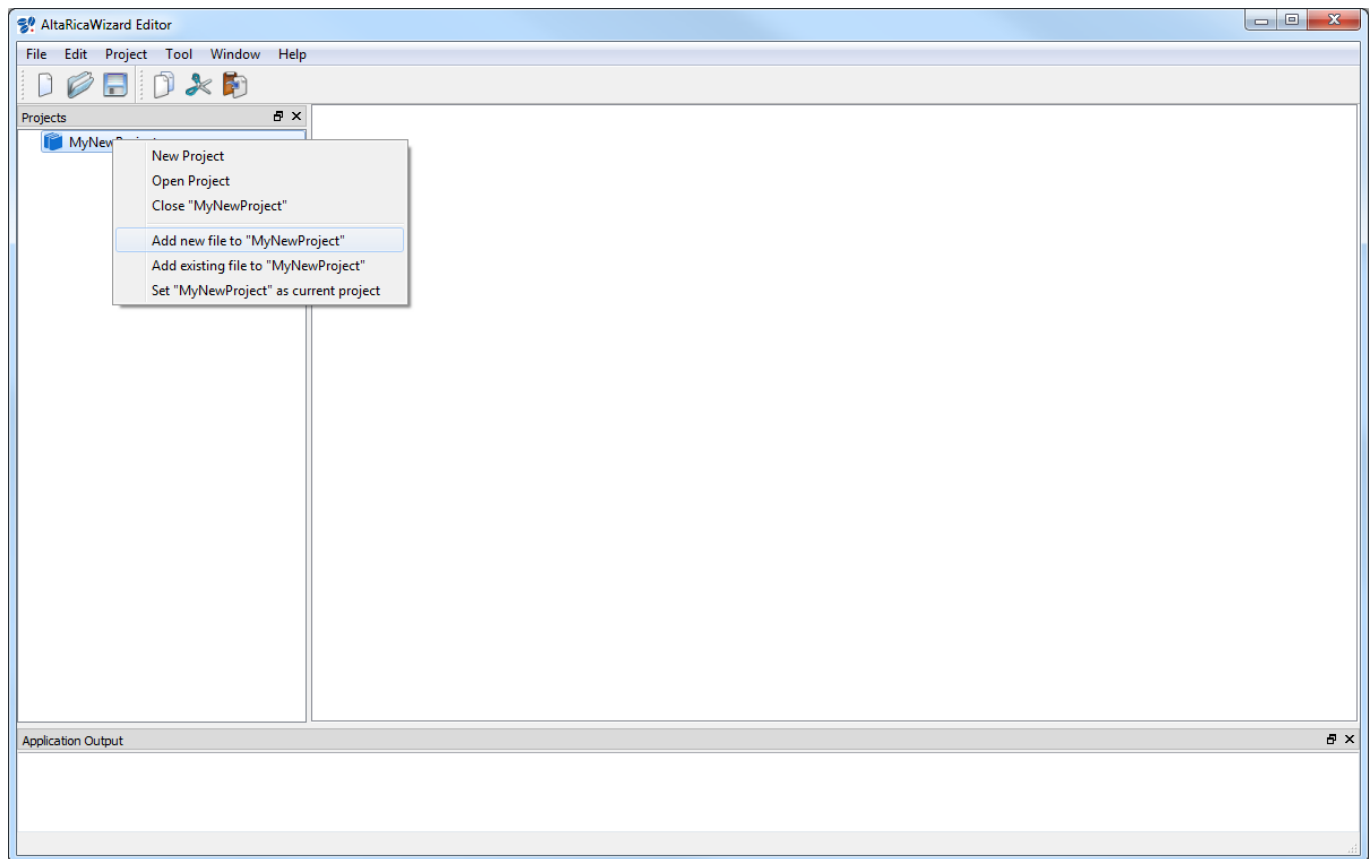
3. The 'Projects' explorer part of the AltaRicaWizard contains the new project (e.g. 'MyNewProject'). By default, it contains nothing.
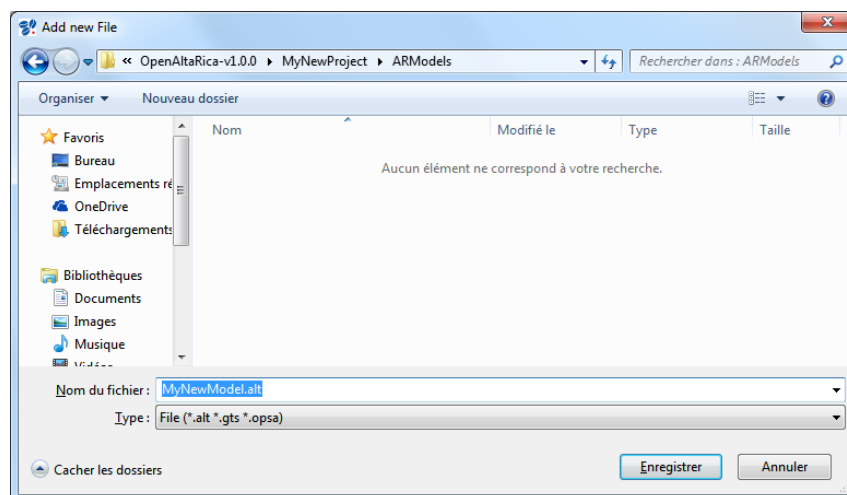
5. Create a new AltaRica 3.0 model

To create a new AltaRica 3.0 model into an existing project, follow these steps:
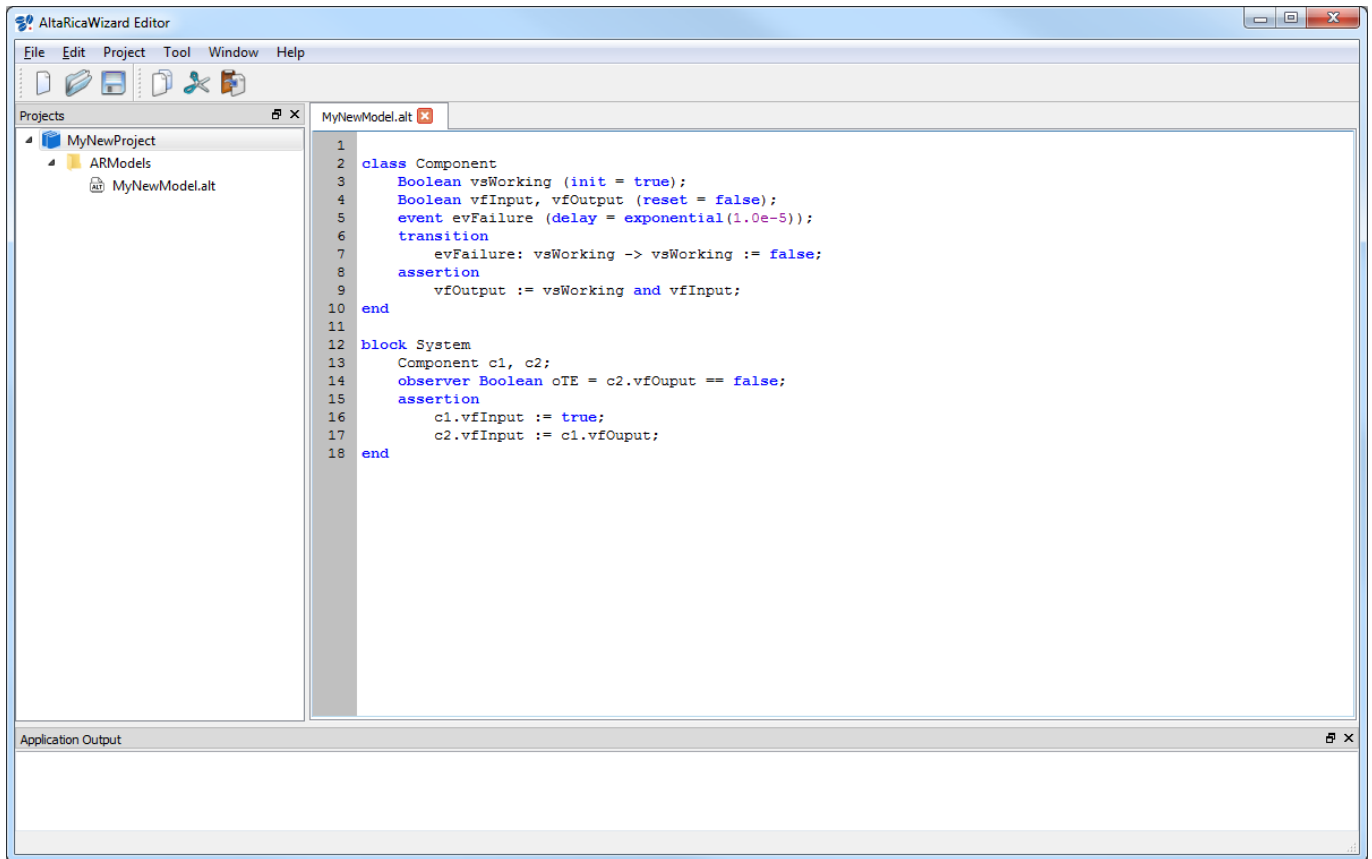
1. Into the 'Projects' explorer part of the AltaRicaWizard, right-click on your project (e.g. 'MyNewProject') and select 'Add new file to';

2. A new window opens. Edit the name of your model (e.g. '`MyNewModel.alt`'); you can create it into a sub-directory (e.g. '`ARModels`') and click on '`Save`' (or '`Enregistrer`');
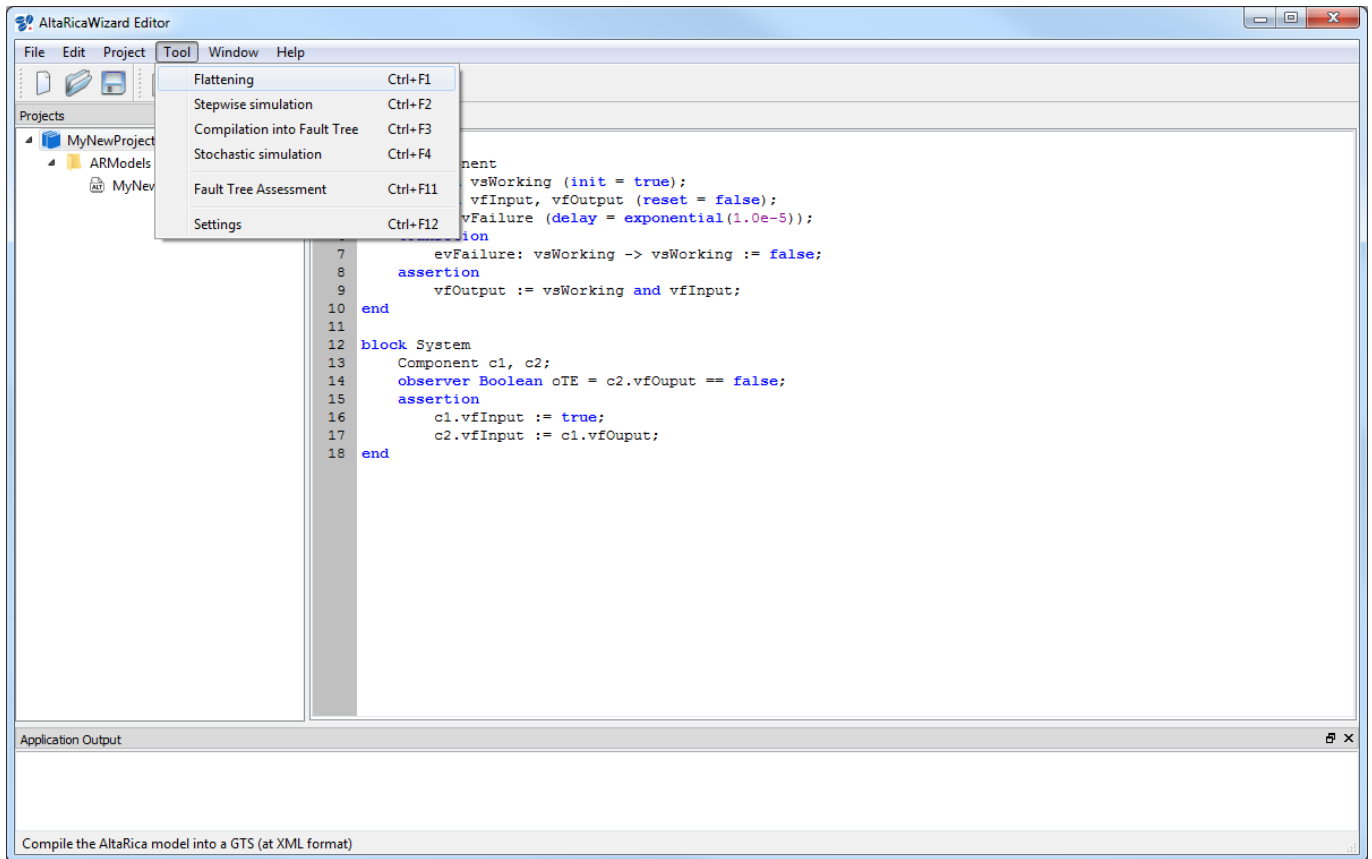

3. Your new AltaRica 3.0 model is now created into your project and you can now edit it.
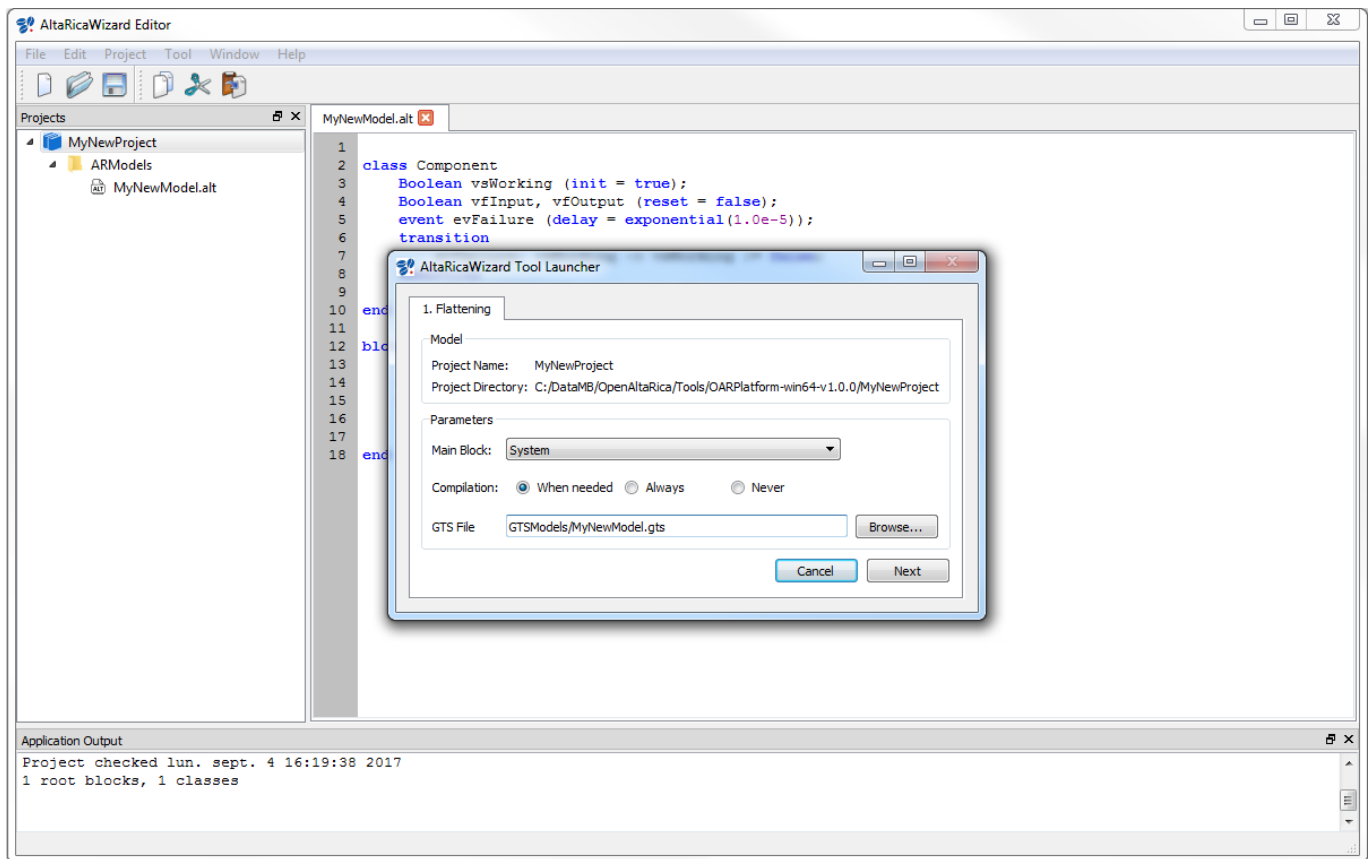
6. Compile an AltaRica 3.0 model

In order to be assessed, an AltaRica 3.0 model must be compiled (more precisely flattened). Click on 'Tool' and select 'Flattening'.

A first static analysis is realized on your model and if no error, a new window opens. You can choose where the flattened model is created (e.g. 'GTSModels/MyNewModel.gts'). Then click on 'Next'

The AltaRica 3.0 compiler is launched and if no error occurs, you can see 'Compile AltaRica model: OK.' into the Console part of the AltaRicaWizard.

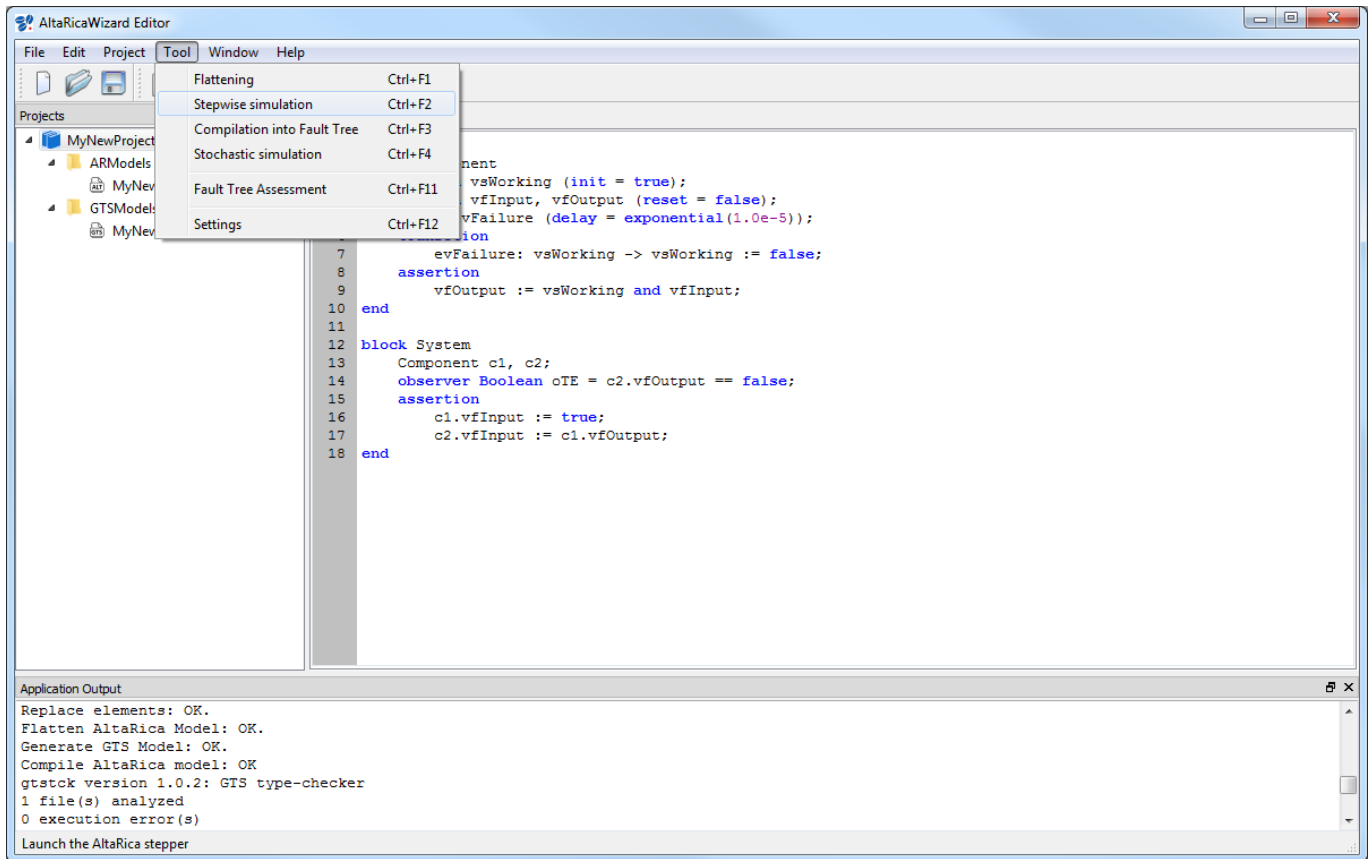The flattened model is created and inserted within your project (e.g. 'GTSModels/MyNewModel.gts'); it is a XML file.
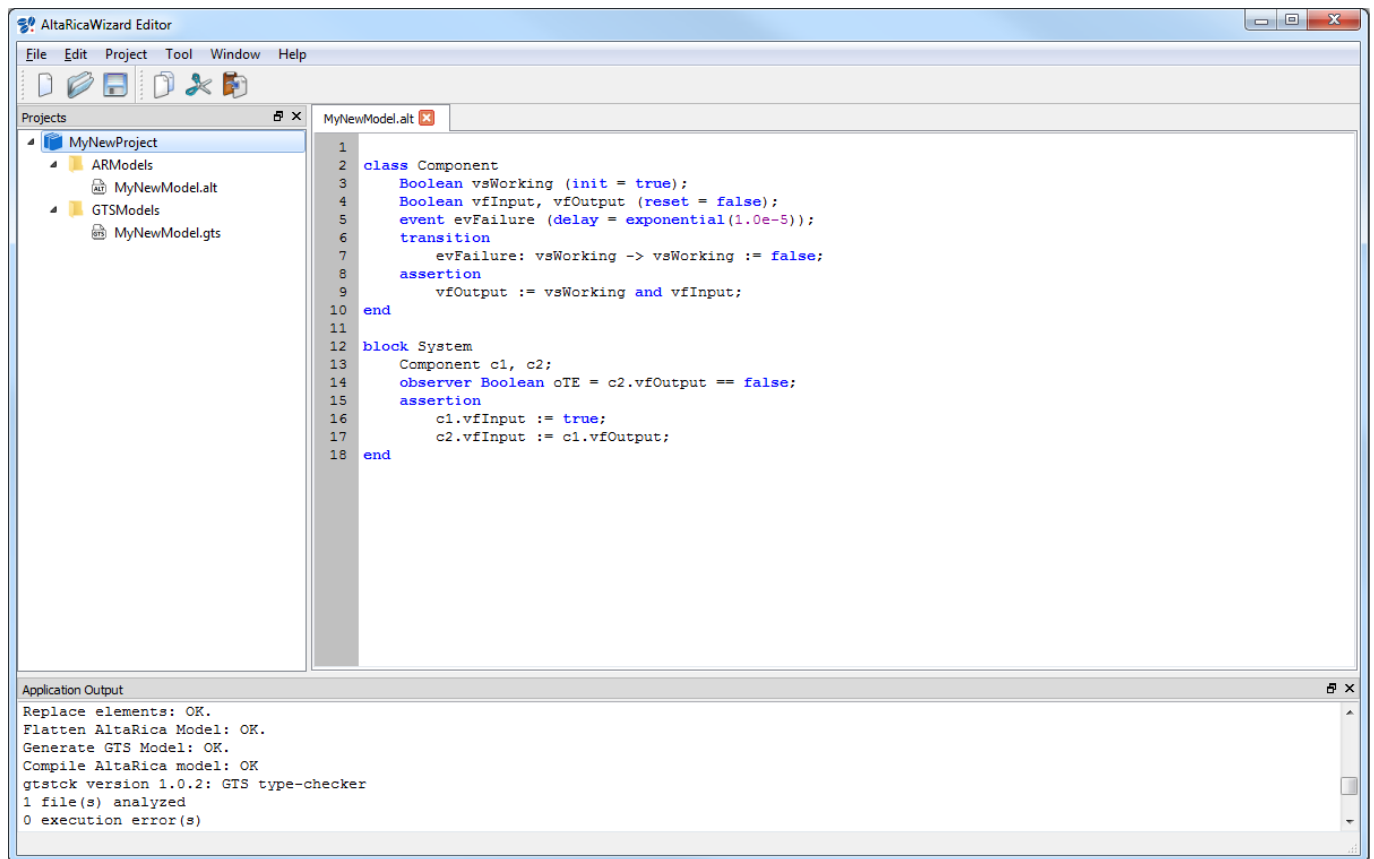
In case of errors into your model, please refer to the AltaRica 3.0 language specification.

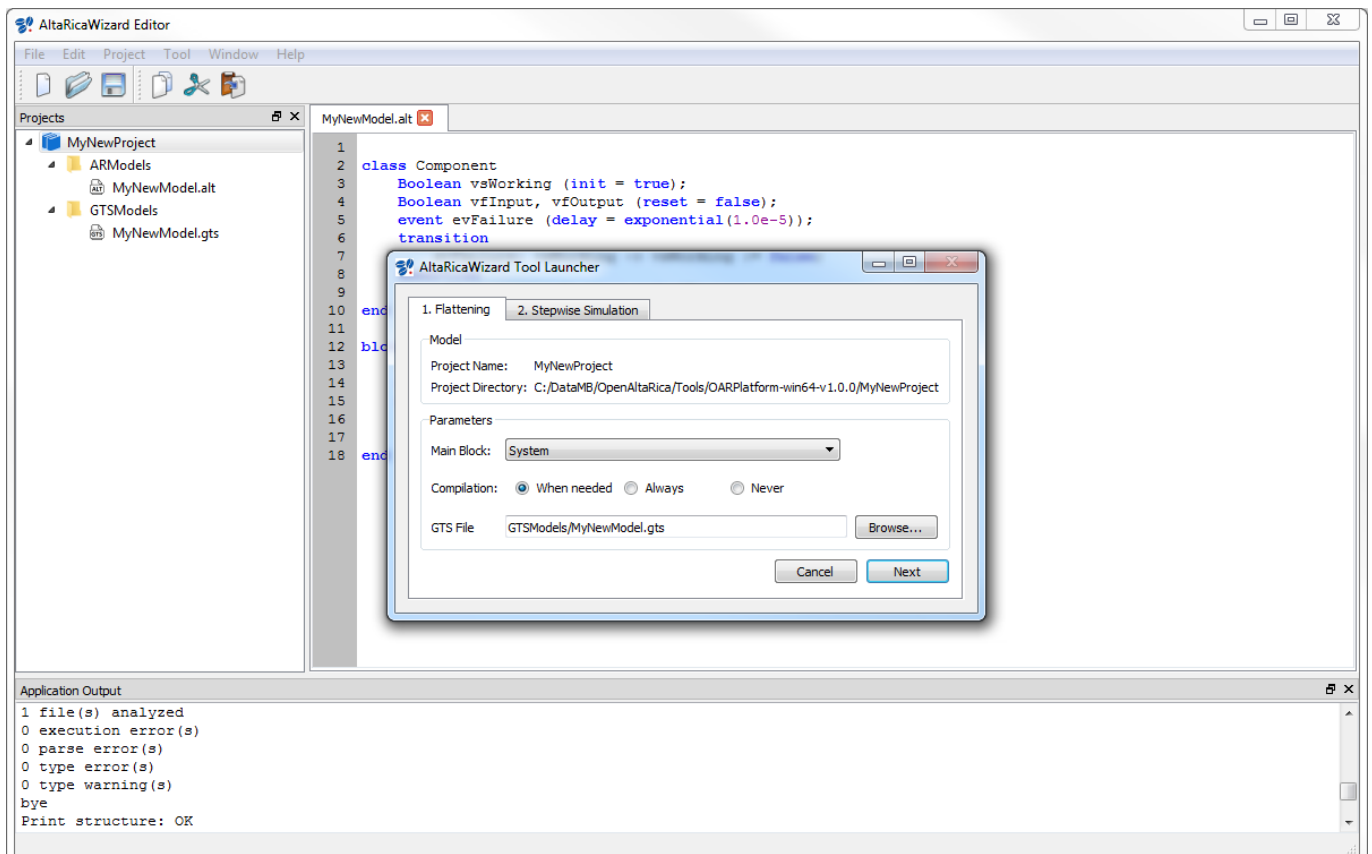This compilation step can be realized when launching a specific assessment tool.
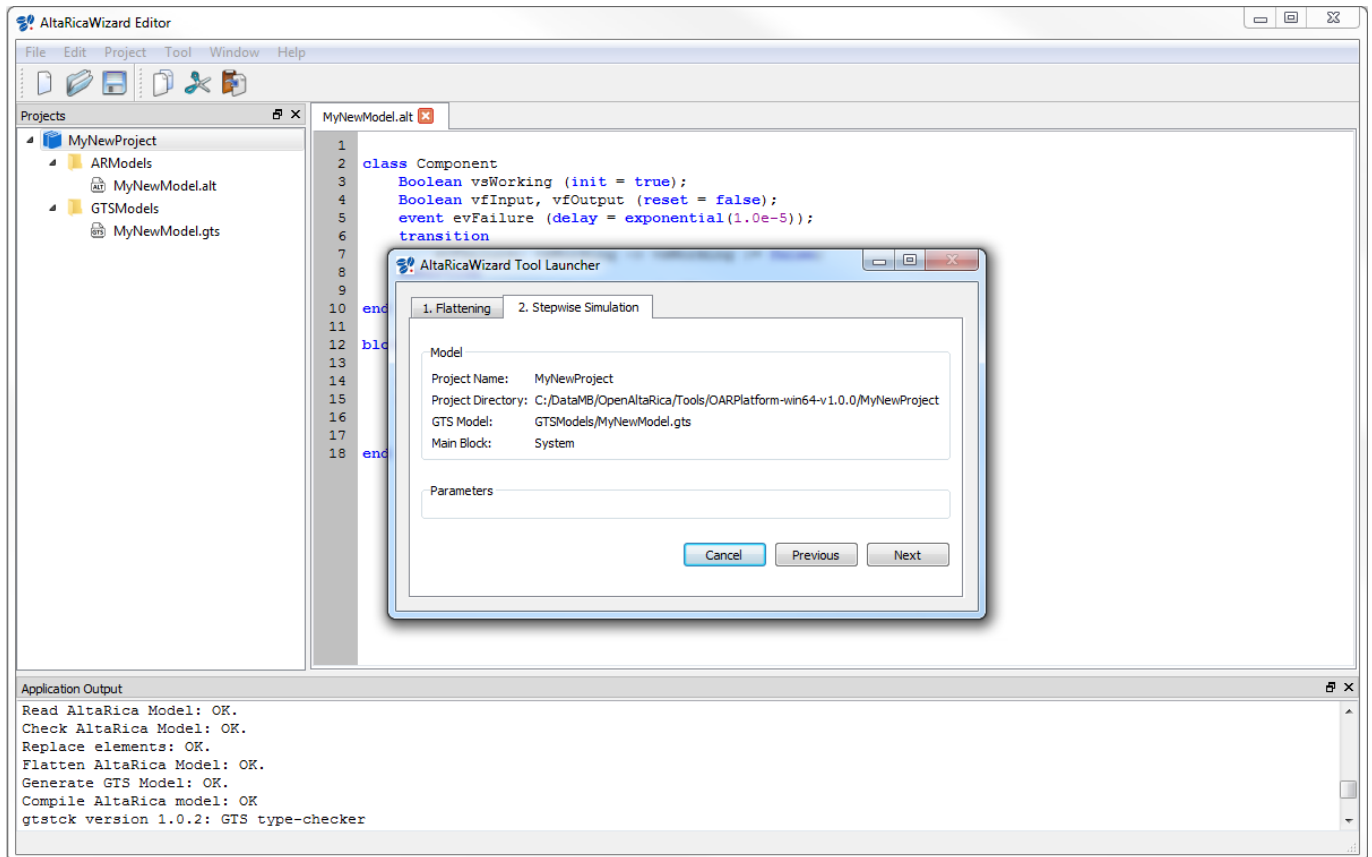
7. Perform a 'stepwise' simulation

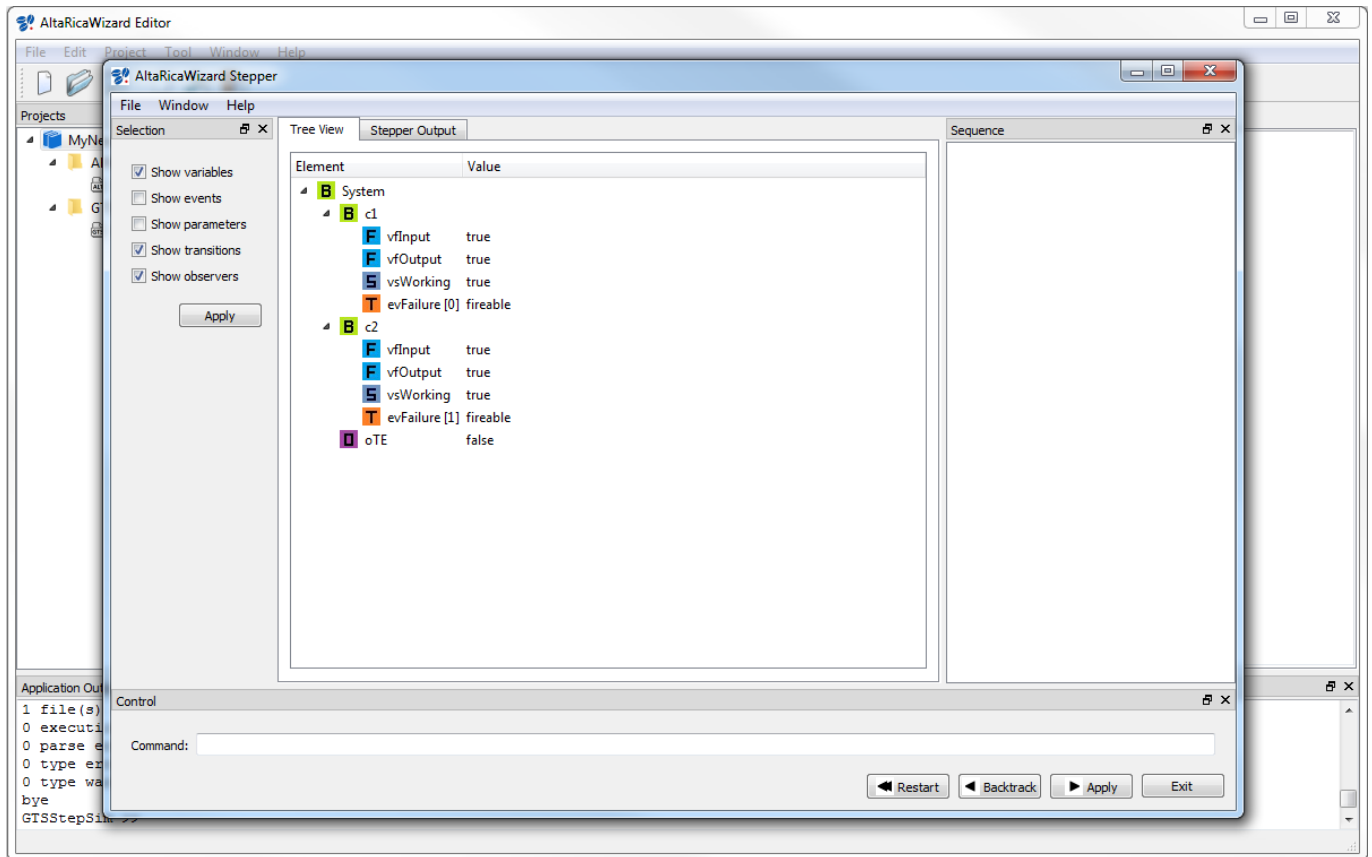Click on 'Tool' and select 'Stepwise simulation'

A new window opens. The first part performs a compilation of your model (see part. 6); click on 'Next'.

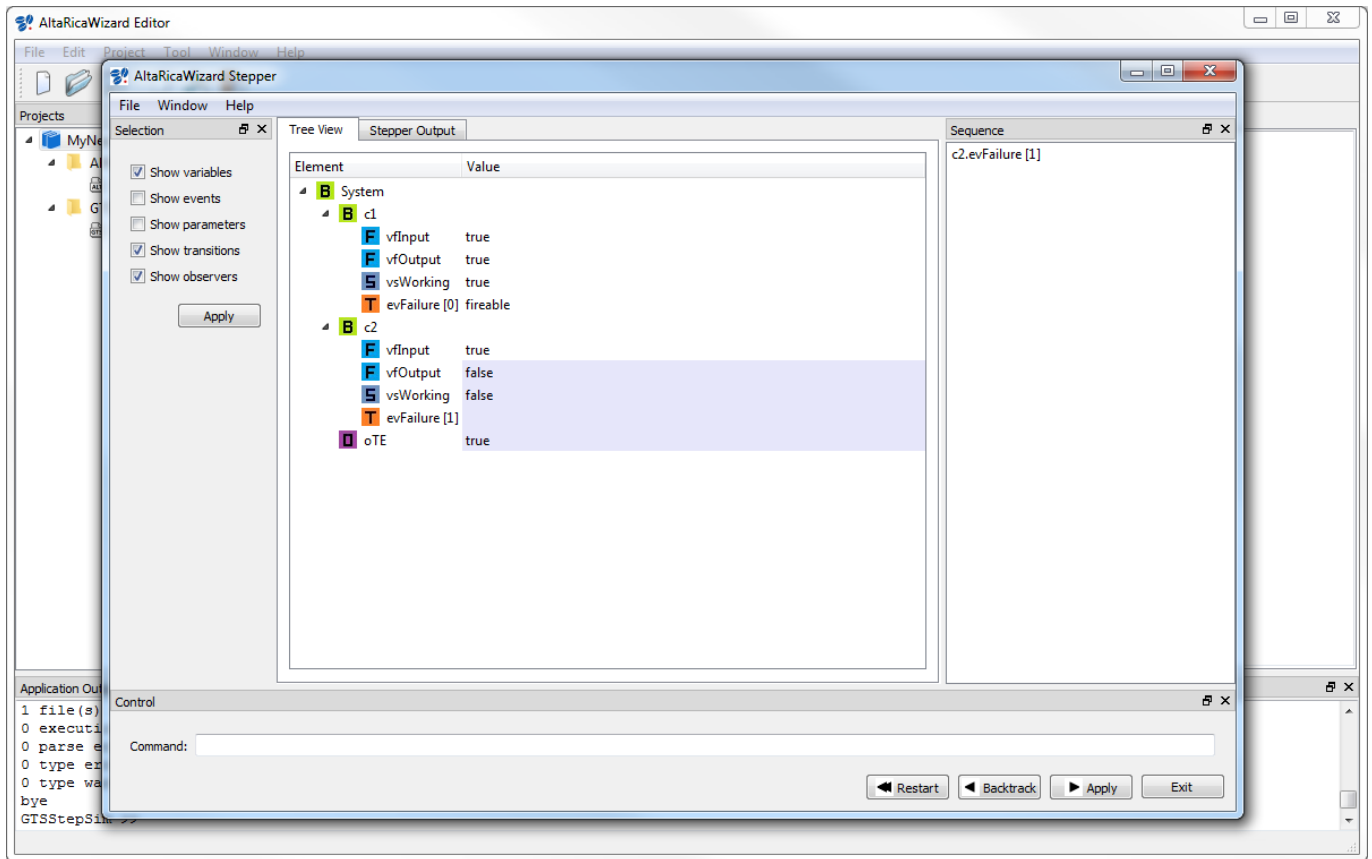The second part just shows information. Click on 'Next'

A new window opens: it is a graphical interface for the stepwise simulator.

You can select which information you want to be shown: variables, events, etc.

To fire a transition, just click on it (the orange square with the letter 'T'). If a transition can be fired, it's value is 'fireable'.
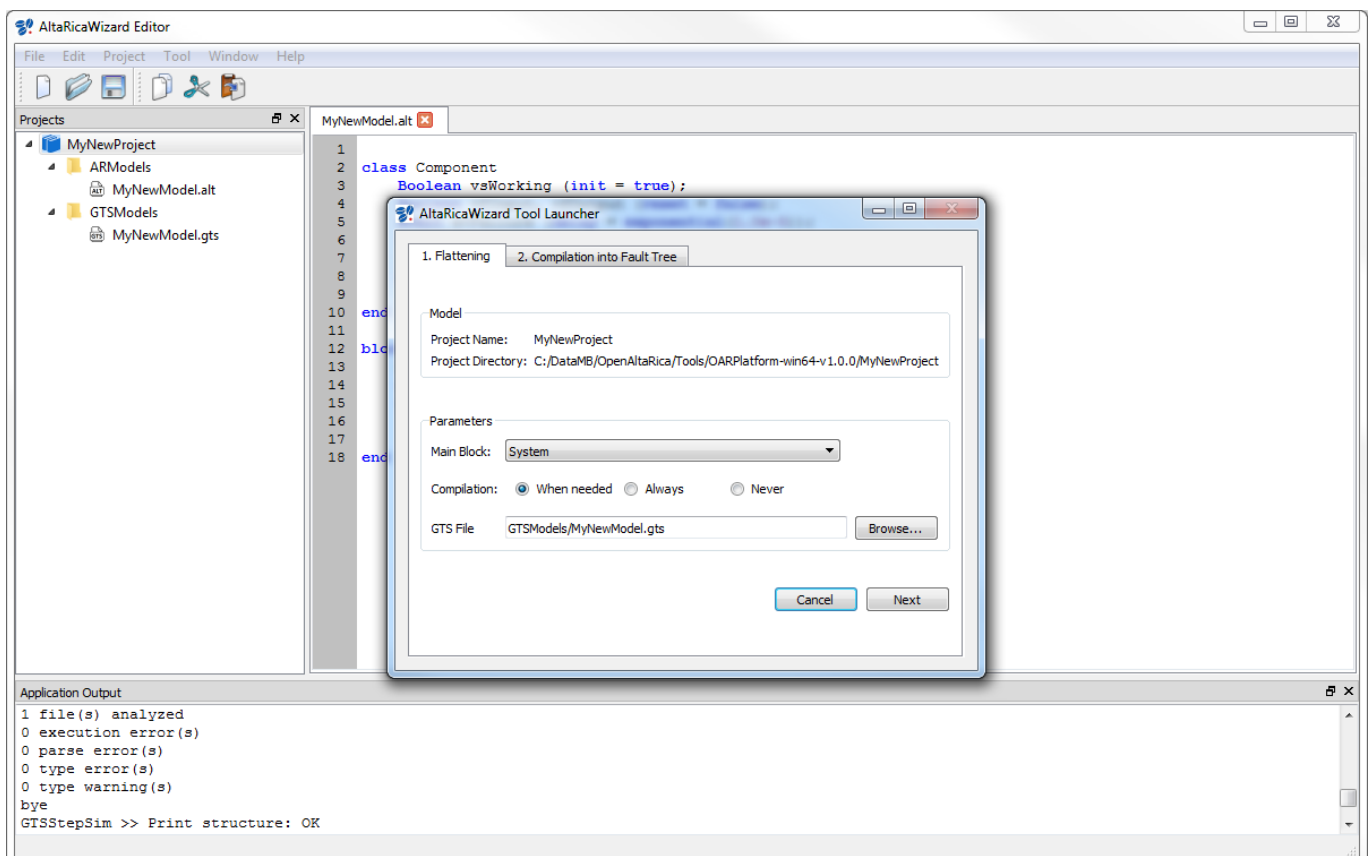
8. Perform a fault tree assessment from an AltaRica 3.0 model

Click on 'Tool' and select 'Compilation into Fault Tree'
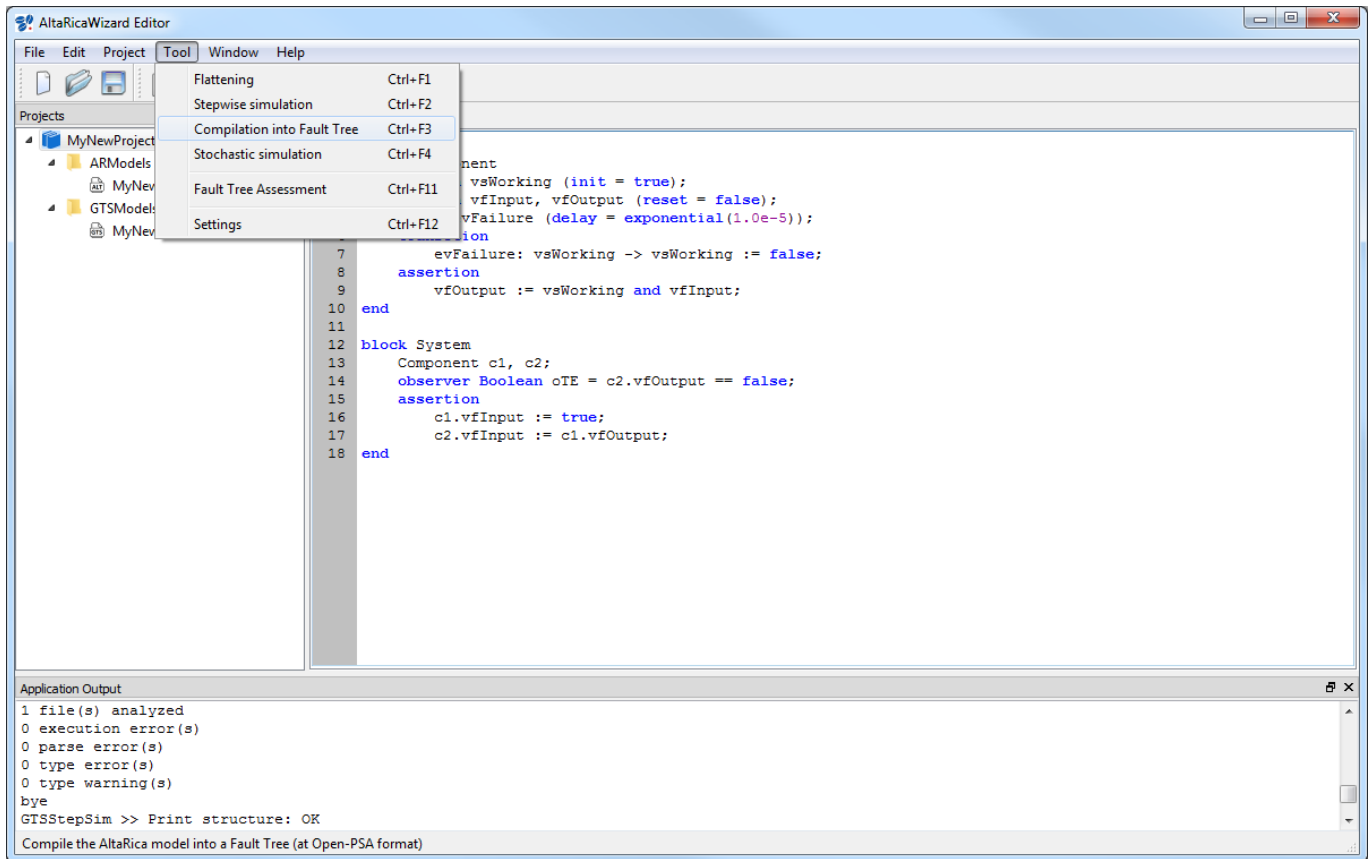
A new window opens. The first part performs a compilation of your model (see part. 6); click on 'Next'.

Into the second part, you can define where the file containing the fault trees is created (e.g. 'FaultTrees/MyNewModel.opsa'). These fault trees are generated according Boolean observers defined into your AltaRica 3.0 model (by default, one Boolean observer corresponds to one top event of a fault tree).

You can also directly launch the fault tree assessment tool by clicking the dedicated button. You have to choose the top event you want to study into the observer part (and also the value).
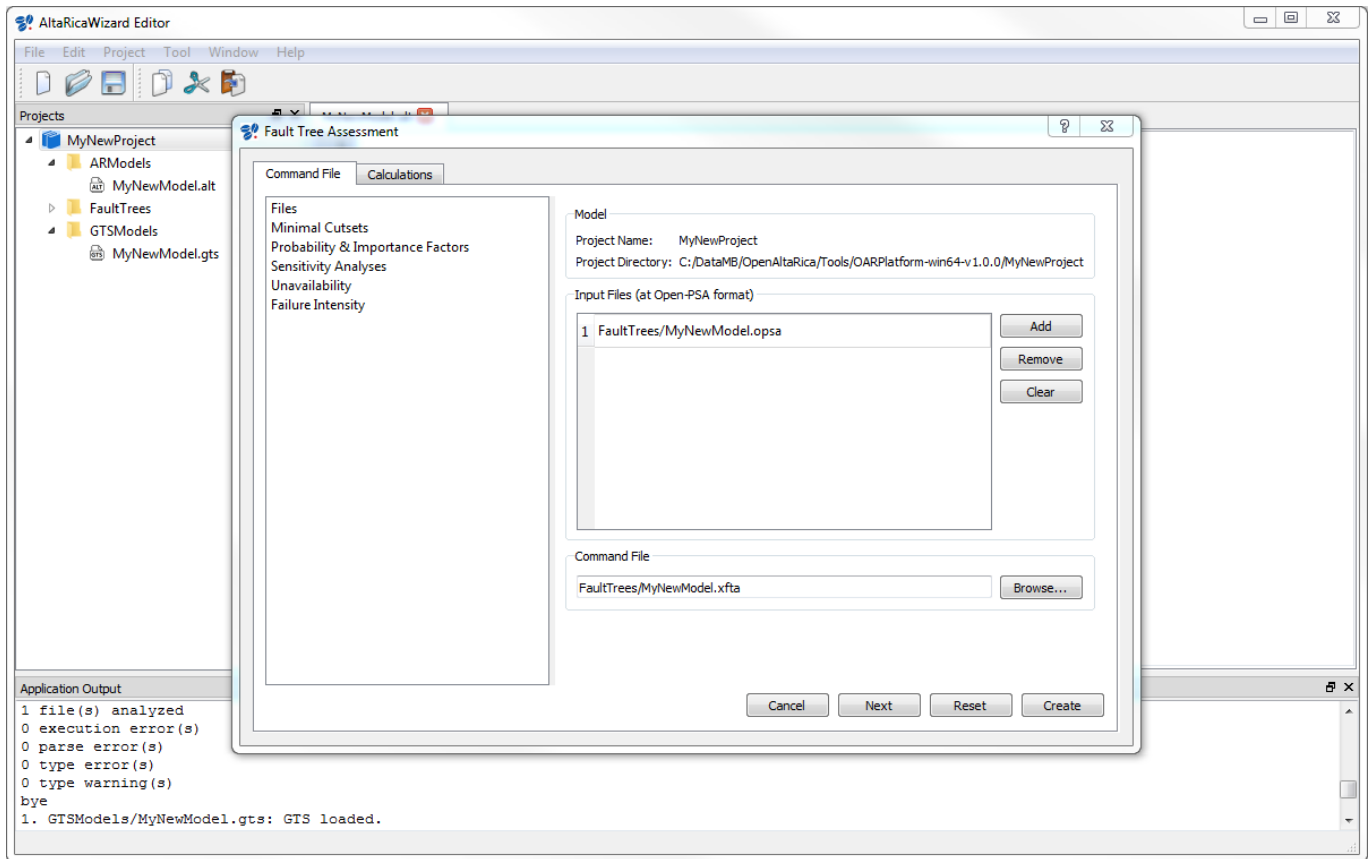
Then click on 'Next'.
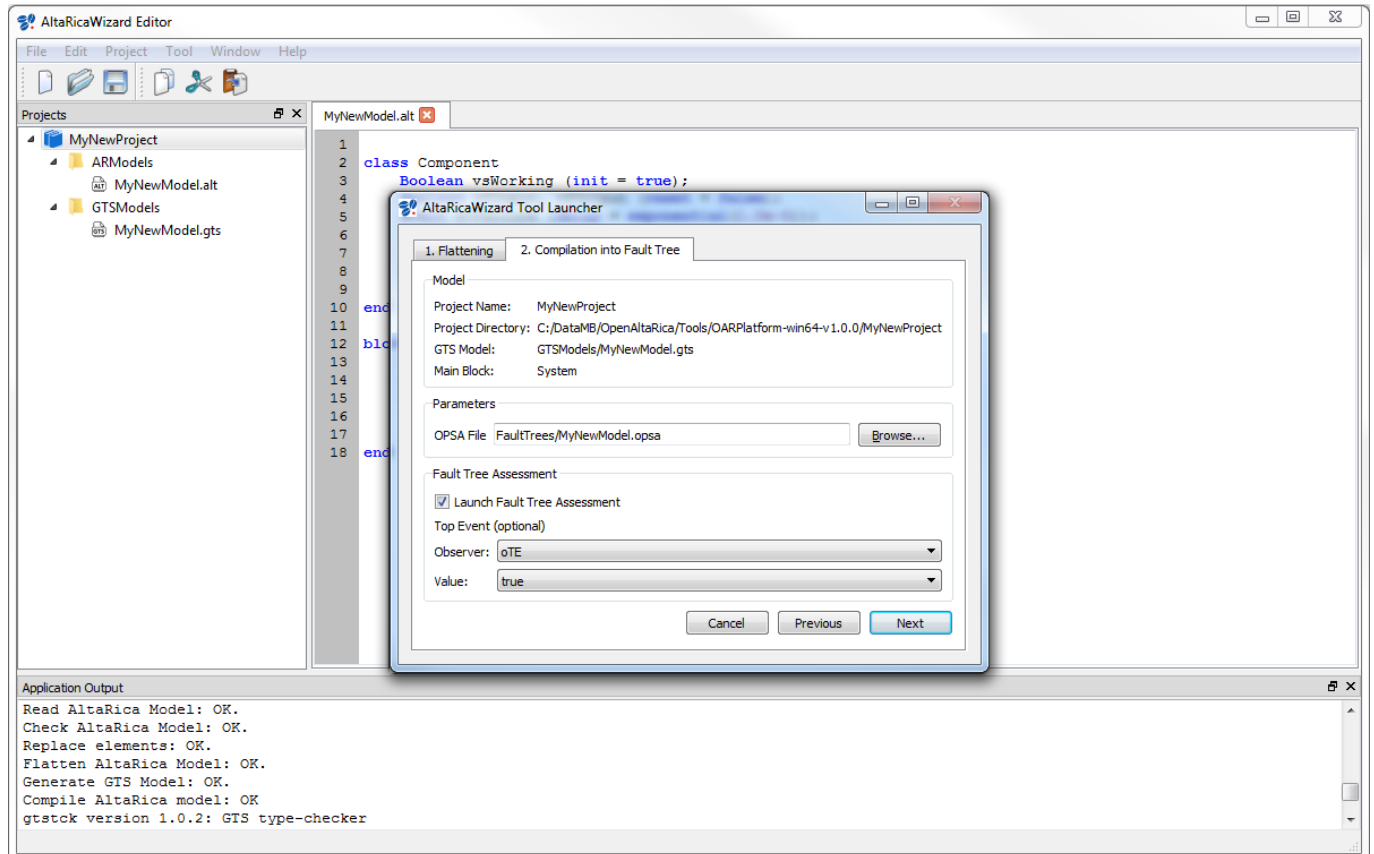
A new window opens. It is the fault tree assessment user interface.

Into the 'Files' menu, you have to choose:

- The input file containing the fault trees from your ArtaRica 3.0 model (e.g. 'FaultTrees/MyNewModel.opsa').
- The command file for the fault tree assessment tool (e.g. 'FaultTrees/MyNewModel.xfta').
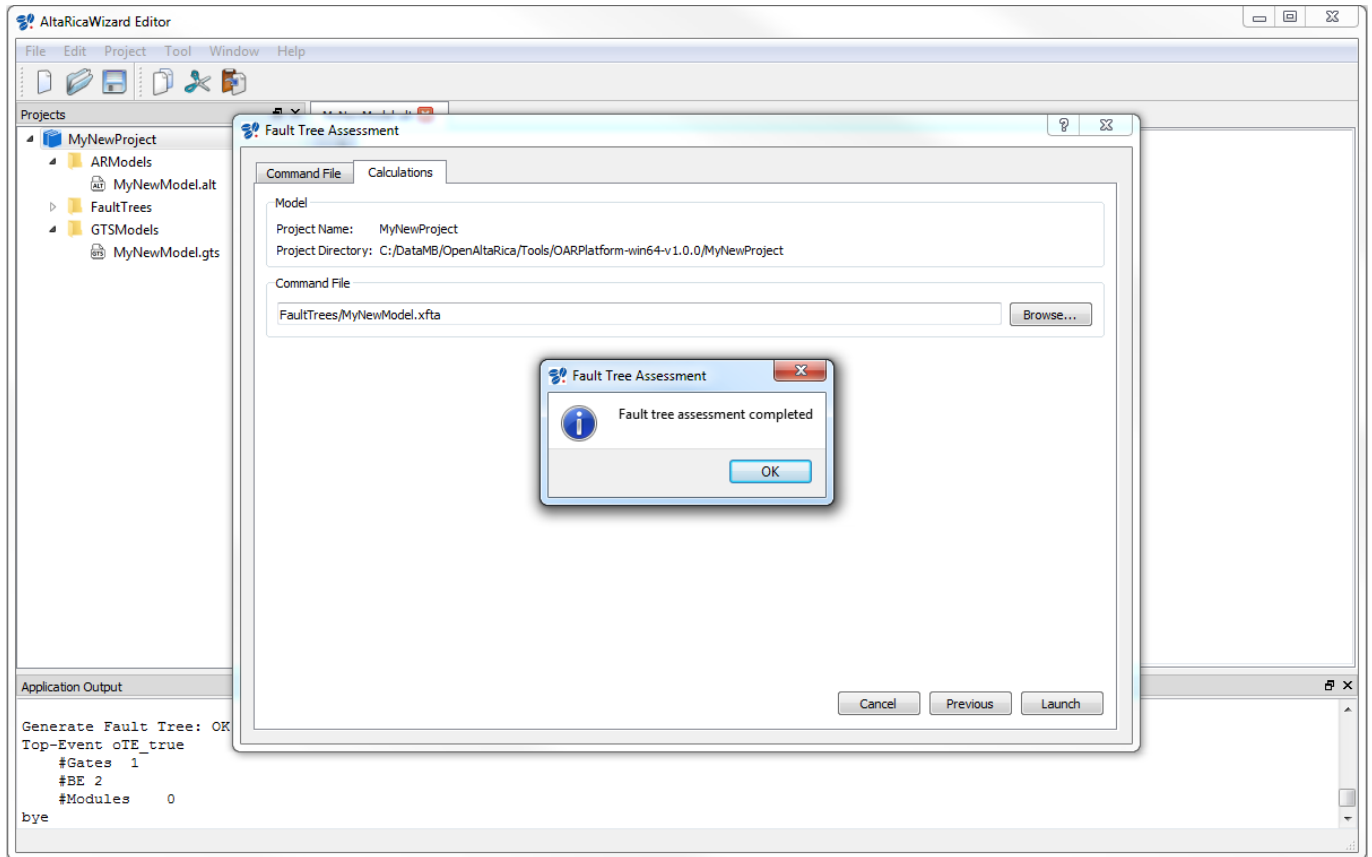
Into other menues (Minimal Cutsets, Probability & Importance Factors, etc.) you can choose indicators to compute and which files to save results (e.g. 'FaultTrees/Results.txt' for minimal cutsets with Orders).
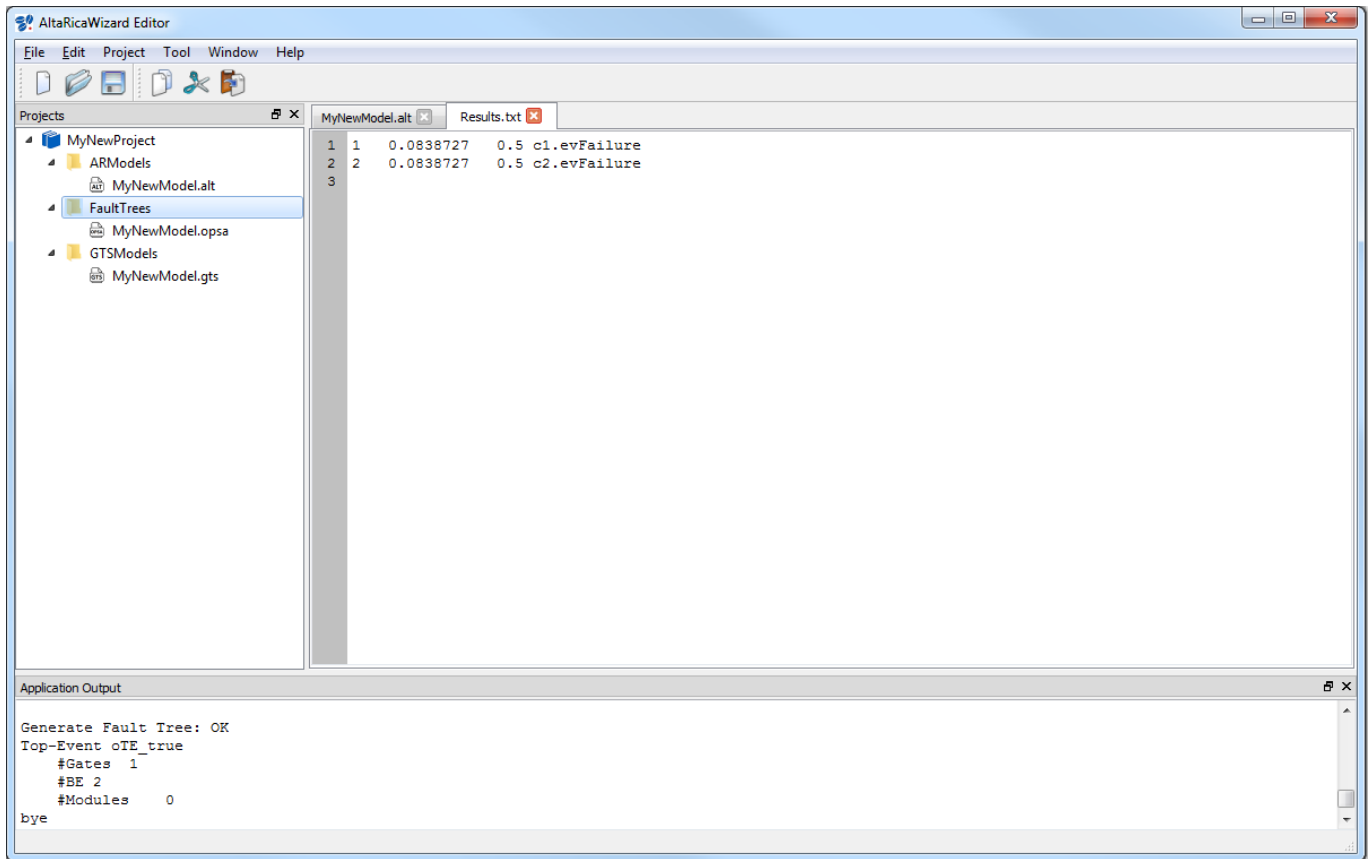
Then click on 'Create'
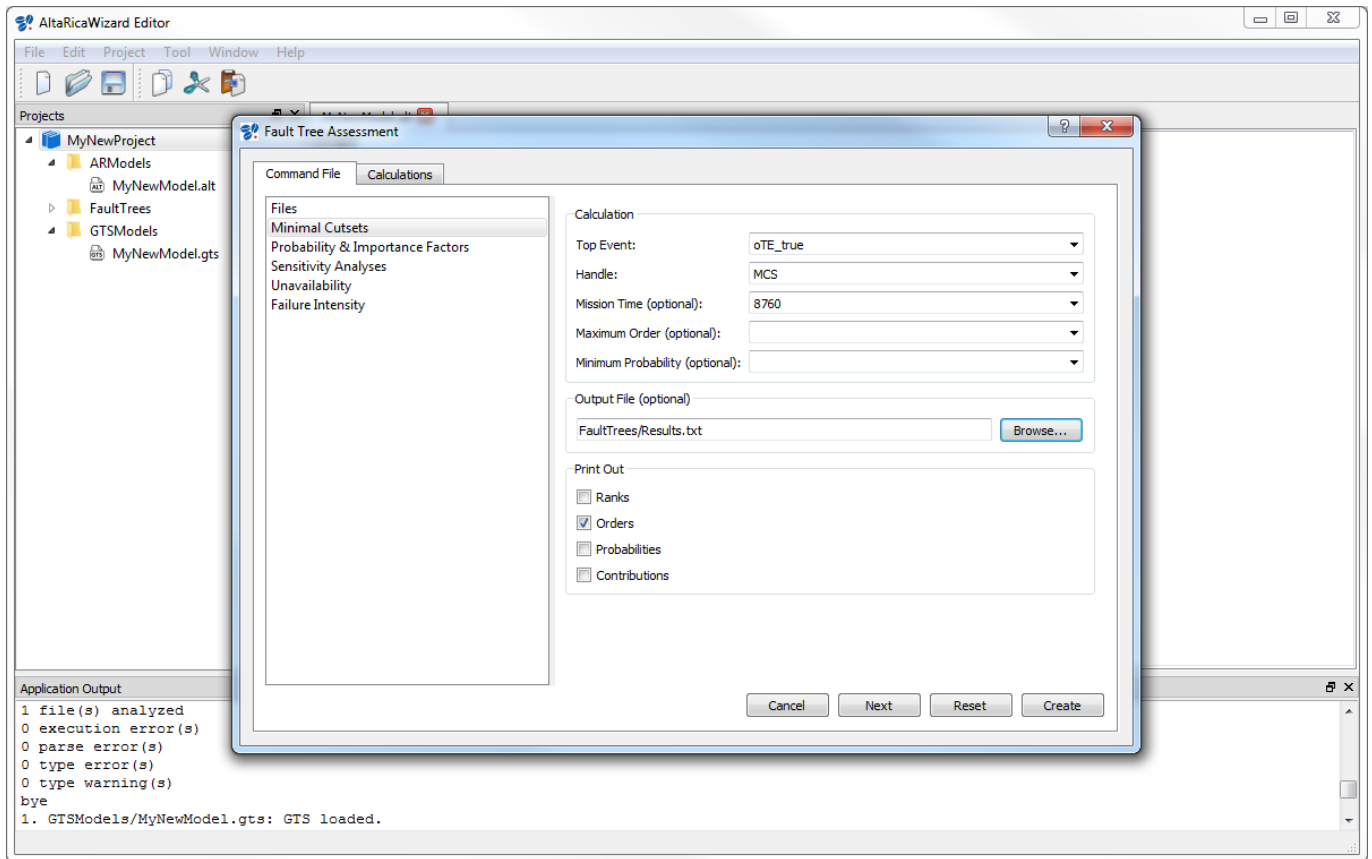
Then into the 'Calculations' part, click on 'Launch'.
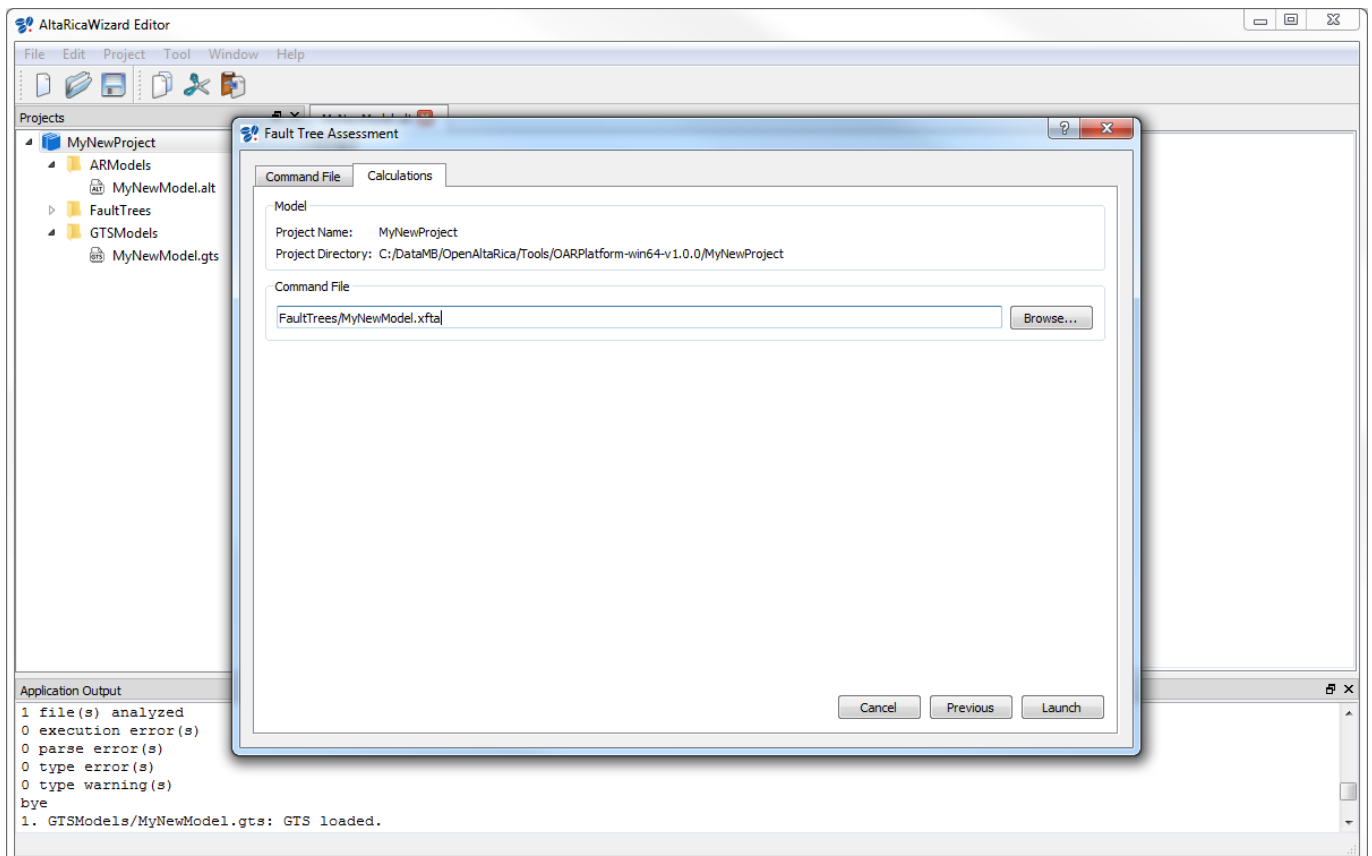
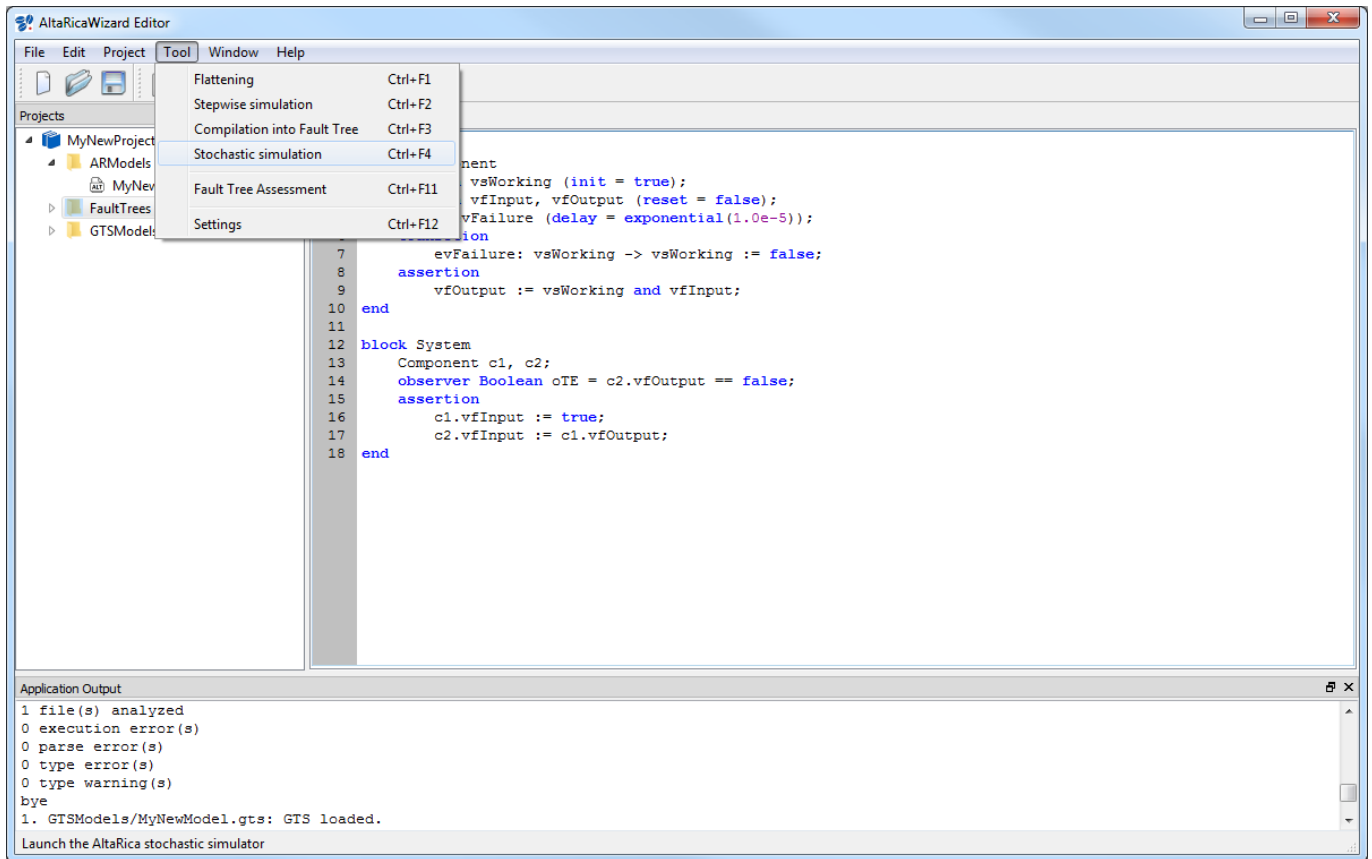If everything is OK, you have the following window:

Finally you can show the file(s) of results by openning it (them) with the menu: click on 'File', select 'Open File or Project', 'Open File' and choose your file (e.g. 'FaultTrees/Results.txt').
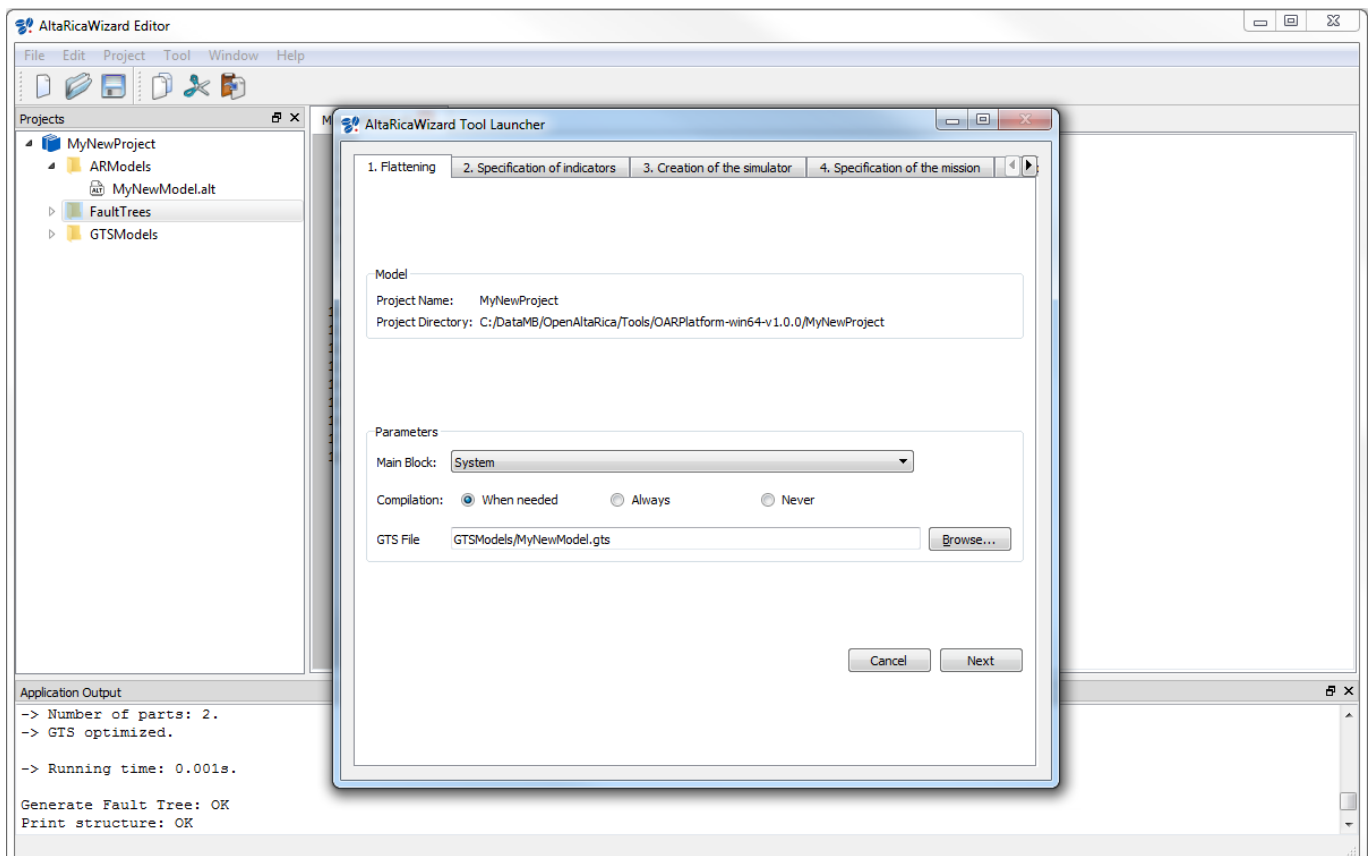
9. Perform a stochastic simulation

Click on 'Tool' and select 'Stochastic simulation'

A new window opens. The first part performs a compilation of your model (see part. 6); click on 'Next'.

The second part allows you to define indicators (e.g. when the observer 'oTE' of the AltaRica 3.0 has the value 'true'): click add and fill up into the new opened window.
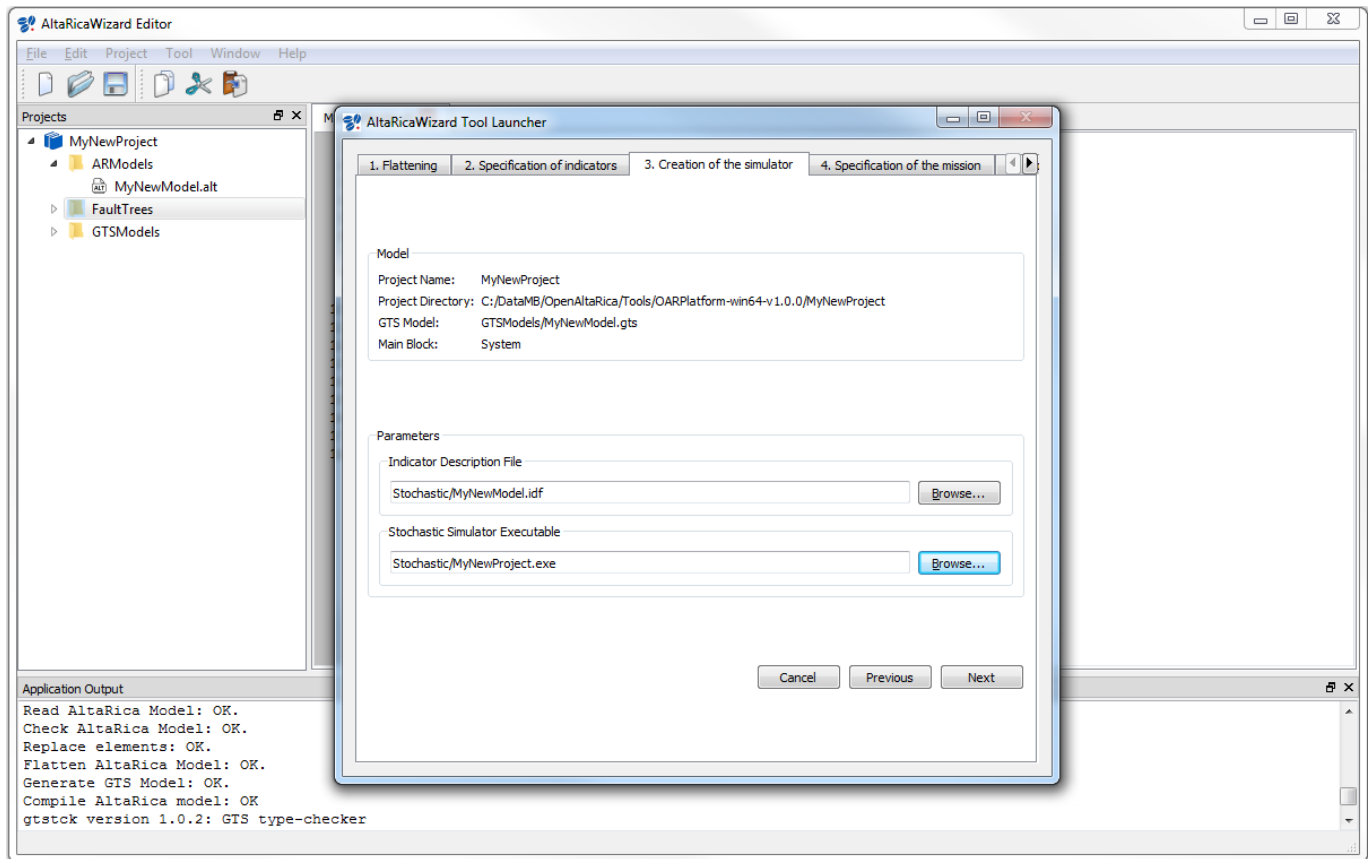
You can choose where to save the file defining indicators for the stochastic simulation (e.g. 'Stochastic/MyNewModel.idf').
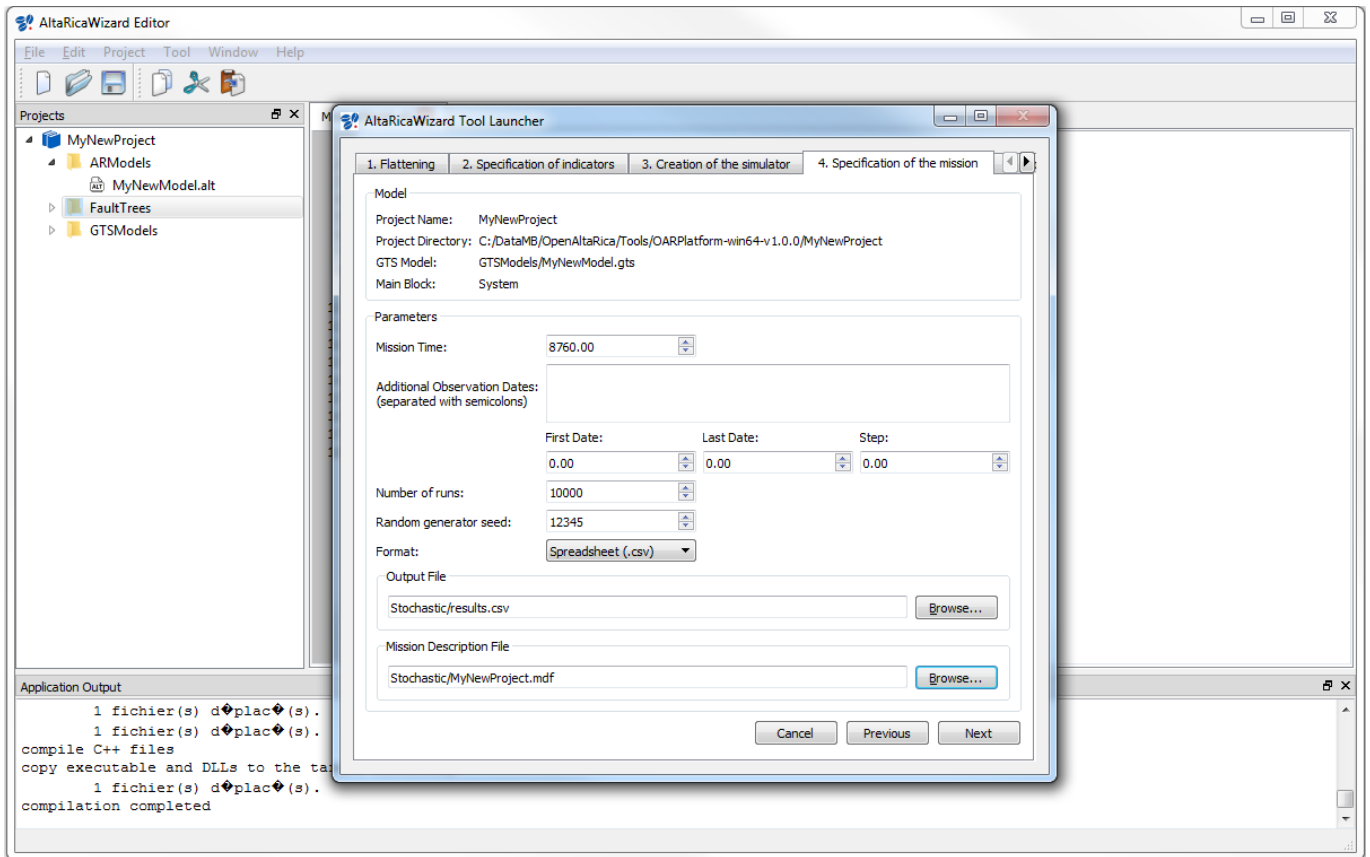
Then click on 'Next'
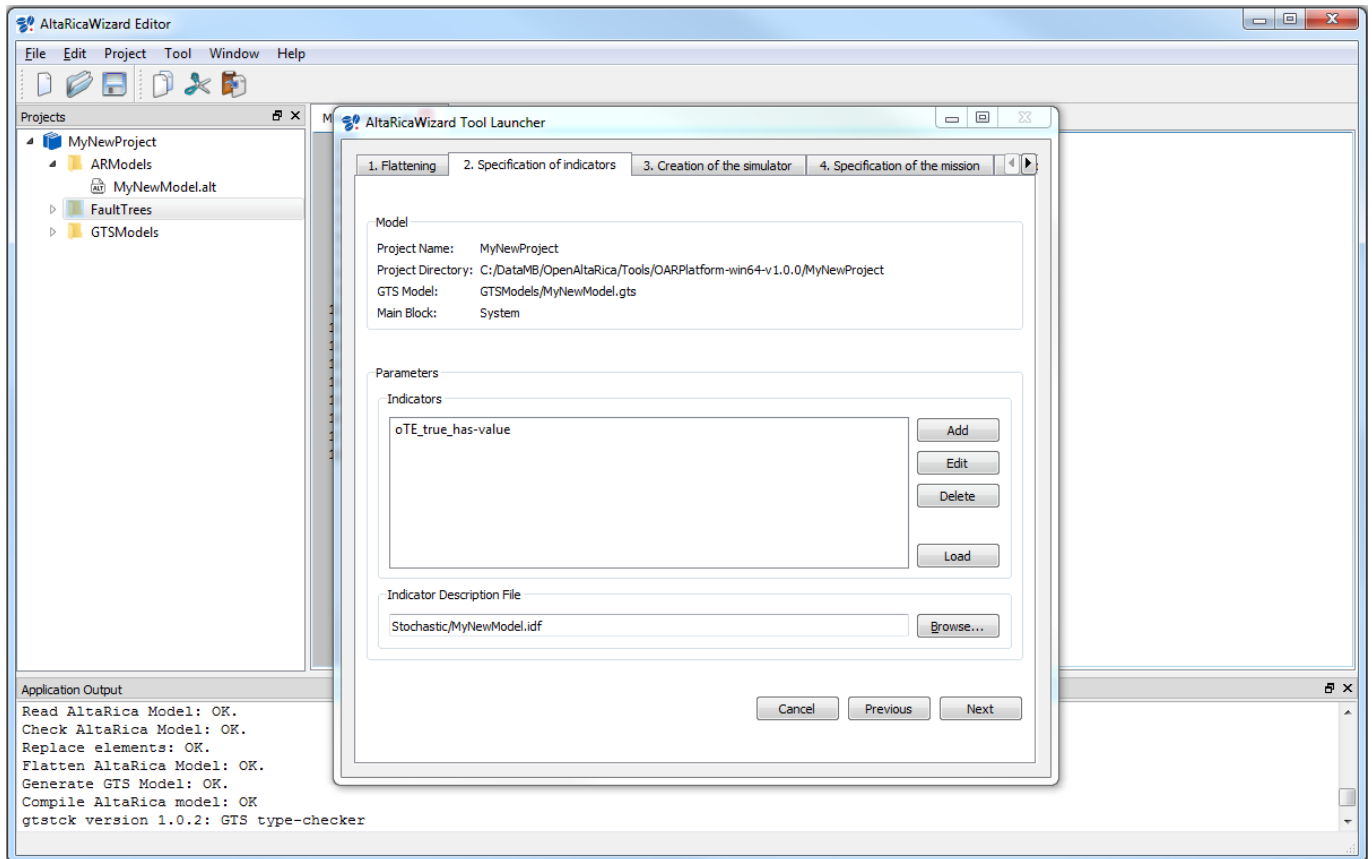
The third part allows you to define where to create the stochastic simulator (e.g. 'Stochastic/MyNewProject.exe').
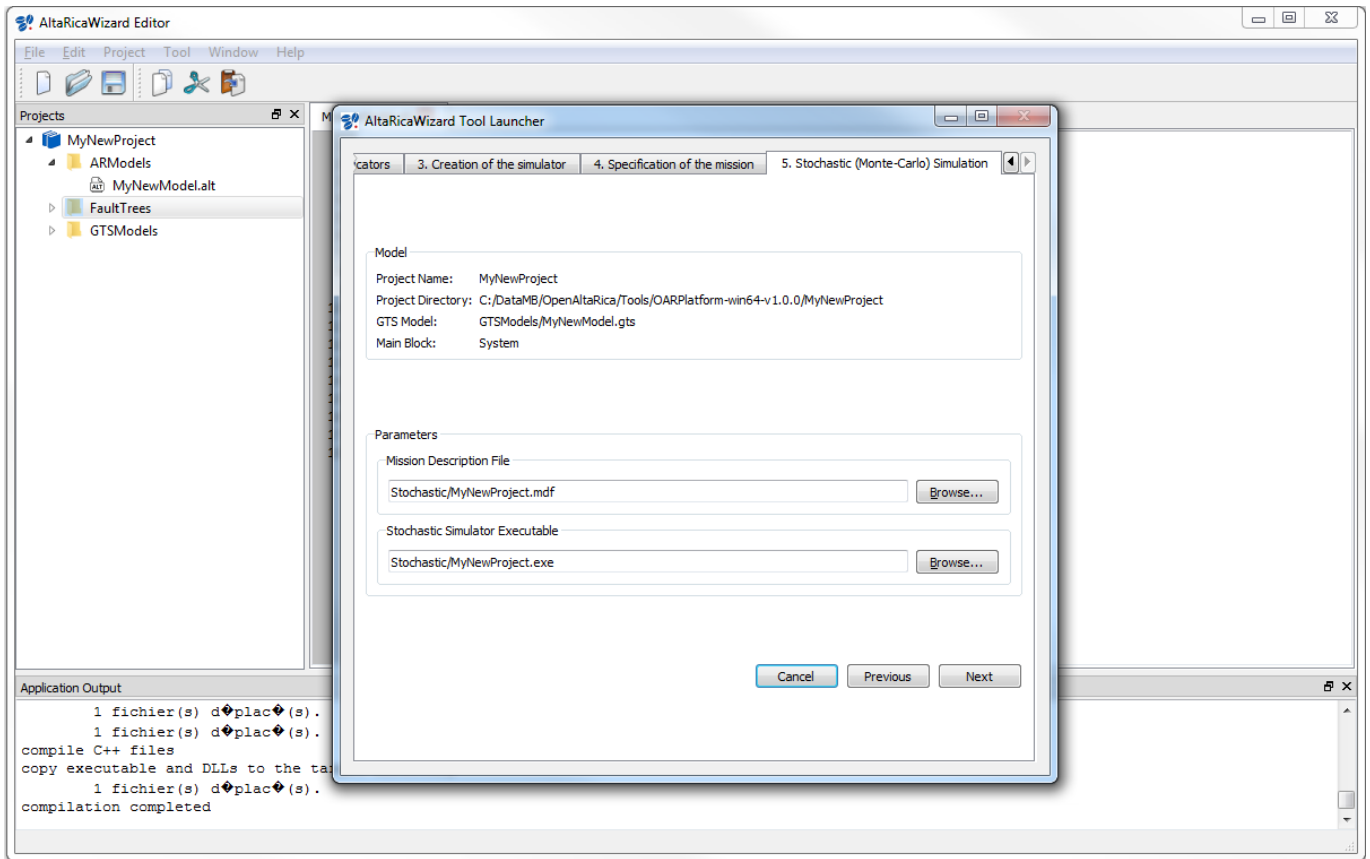
Then click on 'Next'.

The fourth part allows you to specify the number of simulations, the mission time and other information.

You can also specify the output file for the results (e.g. 'Stochastic/results.csv').

Then, click on 'Next'.

The fifth and last part just summarizes the differents previous elements. Click on 'Next' and the stochastic simulation is launched.

If everything is Ok, a specific new window opens. Click on '`OK`'.
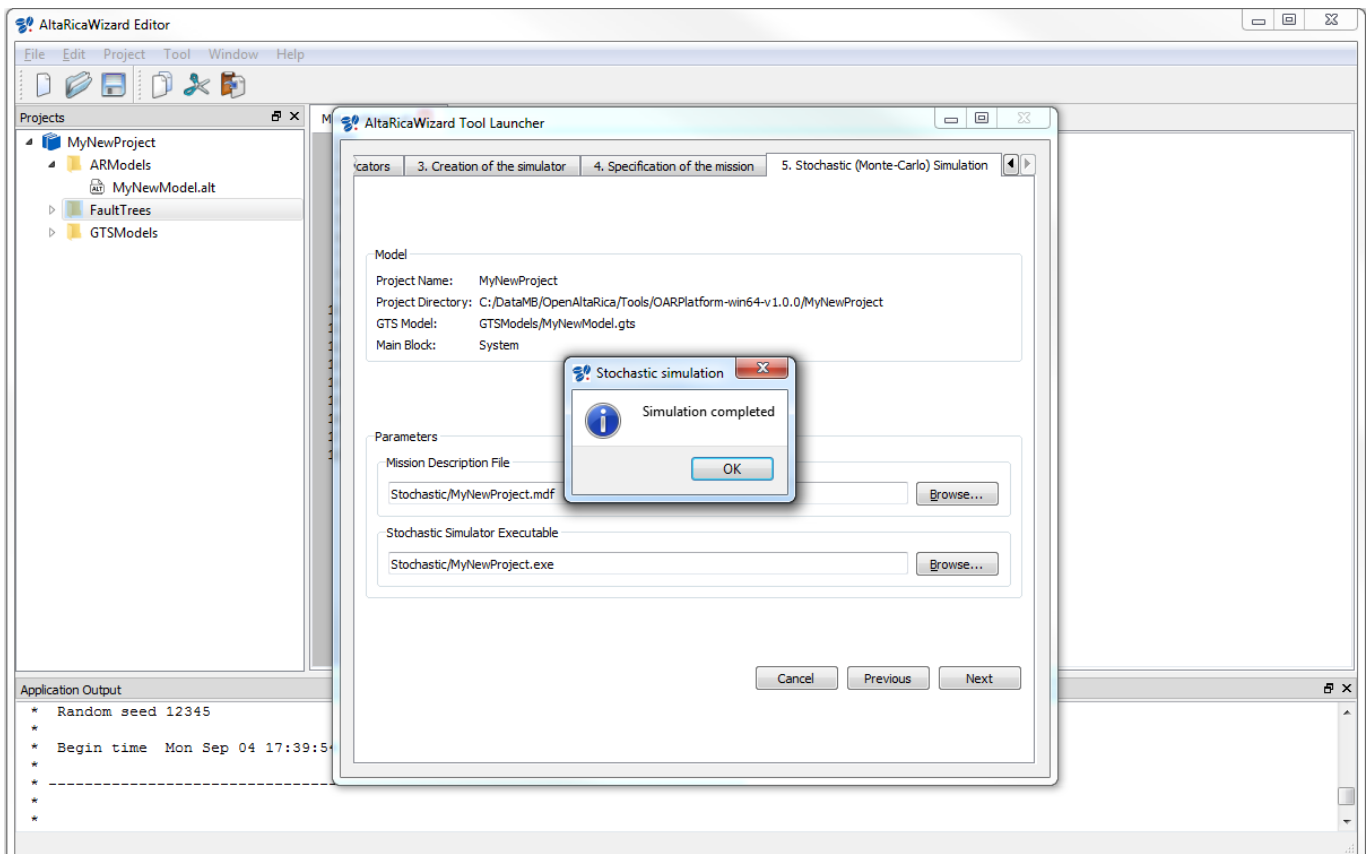
The result of the simulation is printed into the defined file (e.g. 'Stochastic/results.csv'). You can open it with a specific spreadsheet tool (e.g. LibreOffice Calc, Microsoft Excel, etc.).
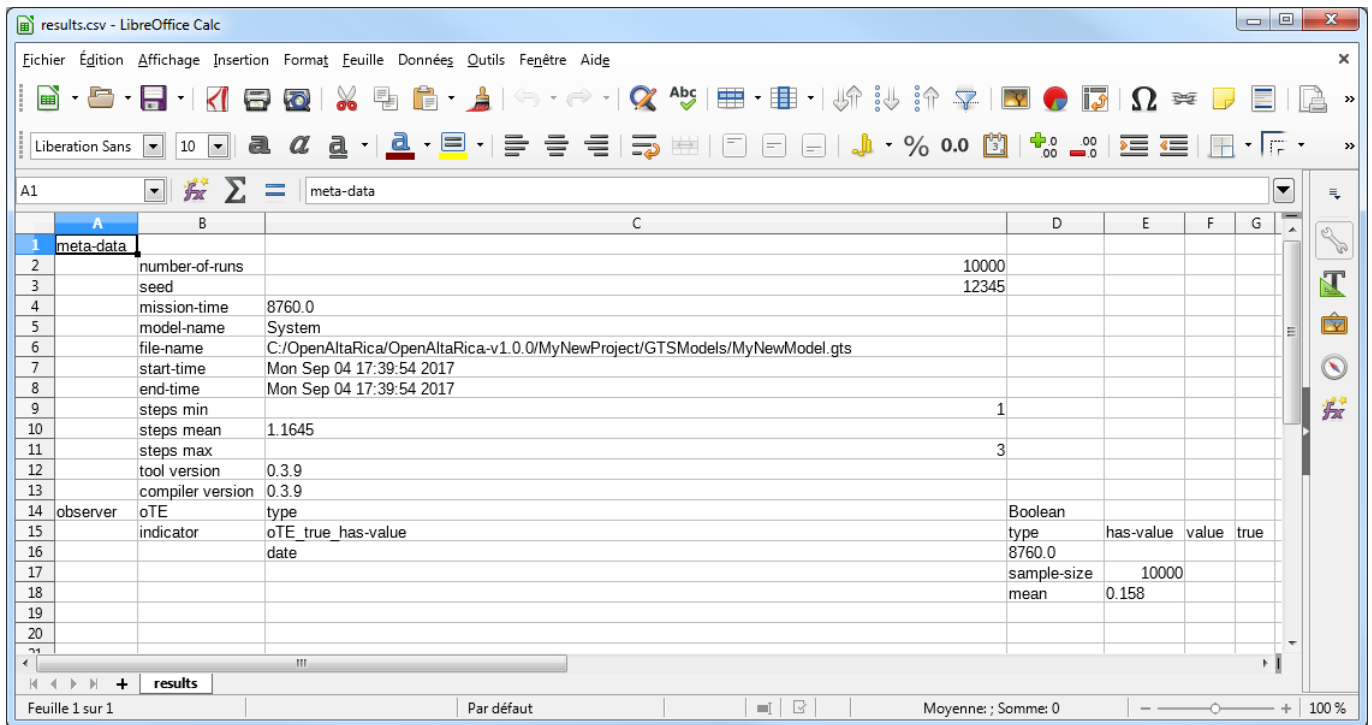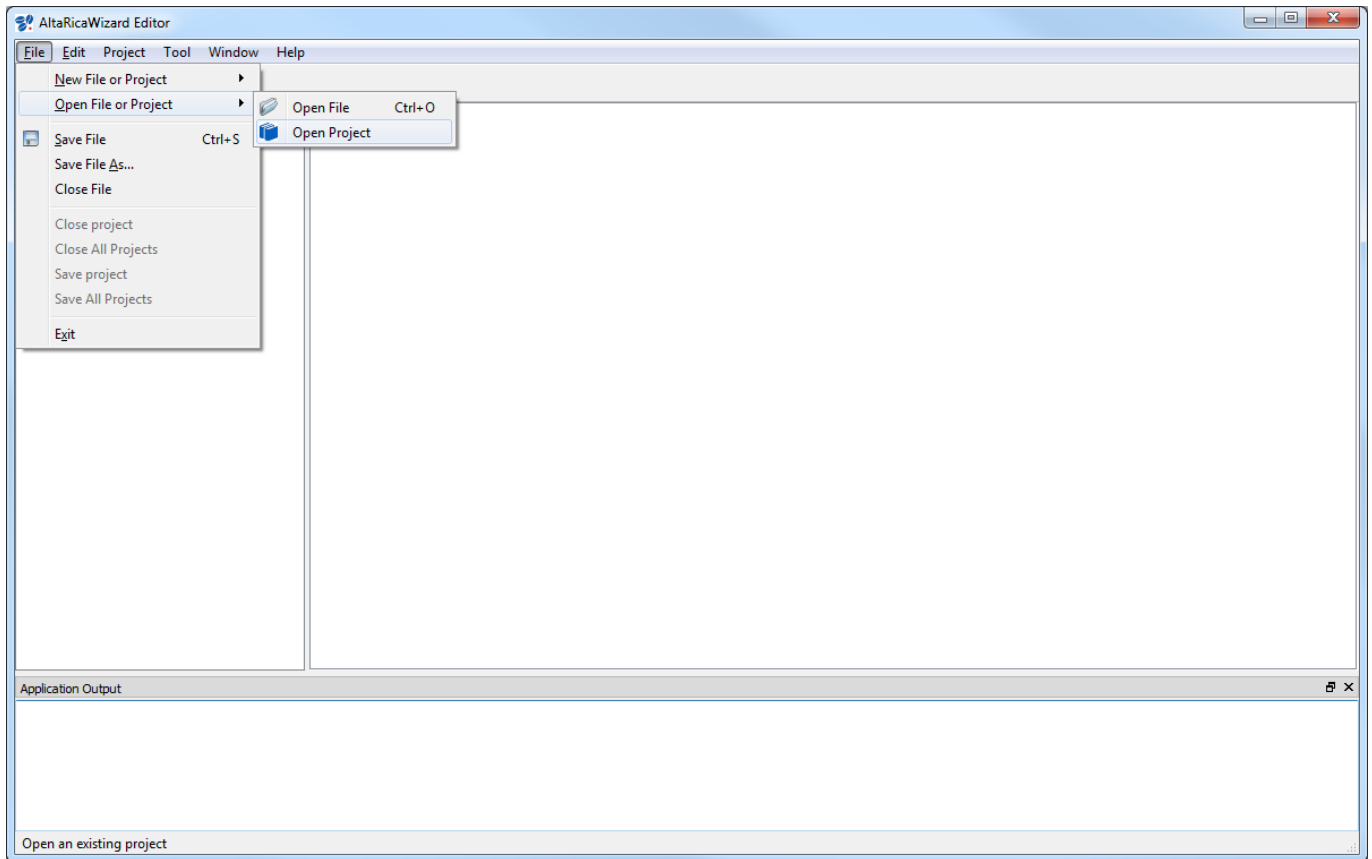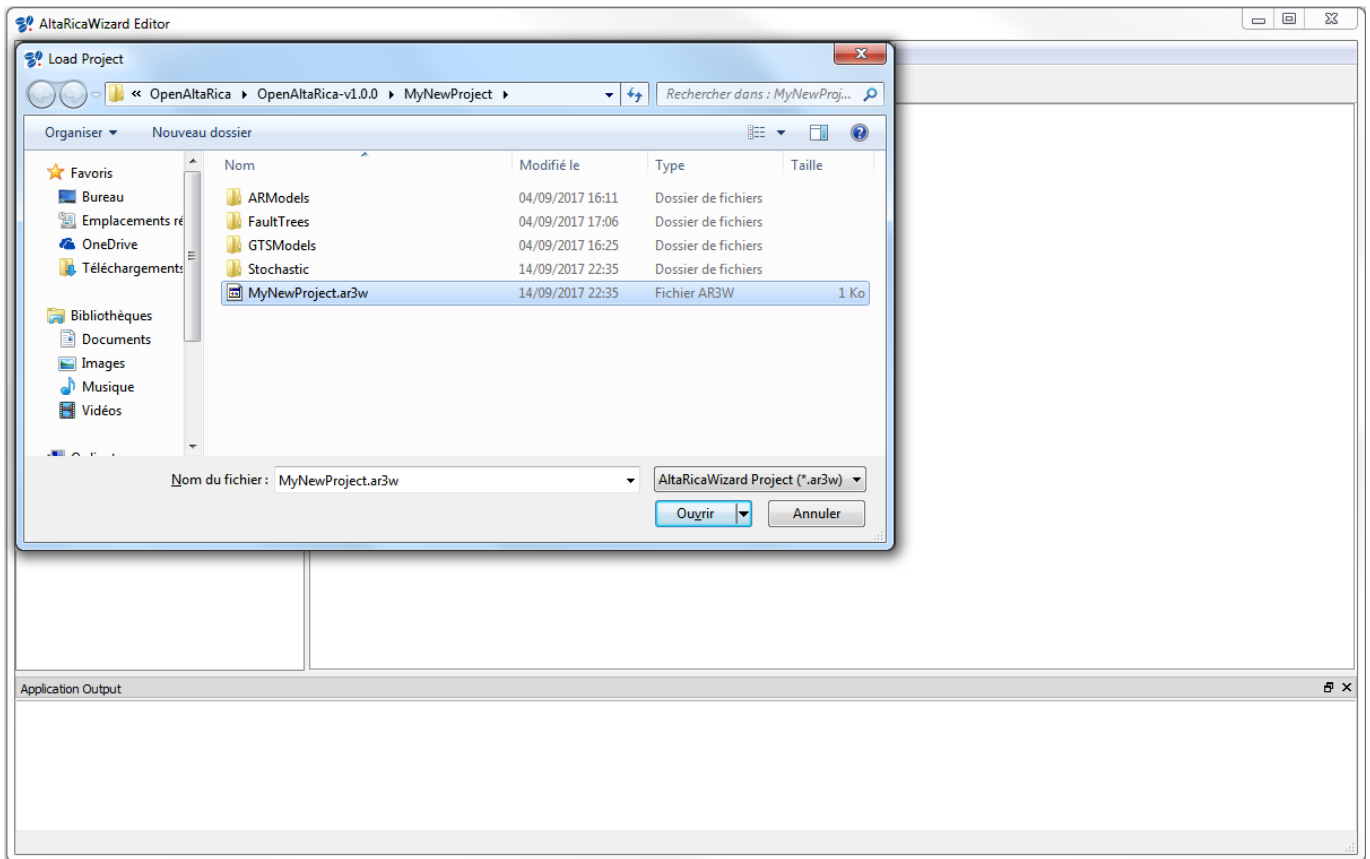

10.   Open an existing project

Assuming you have an existing project (e.g. 'MyNewProject').

Follow these steps:

1. Click on 'File', select 'Open File or Project' and 'Open Project';

2. A new window opens. Select the project you want to open (e.g. 'MyNewProject') and then click on 'Open' (or 'Ouvrir').

3. The Projects explorer part of the AltaRicaWizard contains the opened project (e.g. 'MyNewProject').