



# ETACE Virtual Appliance User Manual

*Gregor Böhl, Philipp Harting, Sander van der Hoog, Bielefeld University  
Chair for Economic Theory and Computational Economics (ETACE)  
February 21, 2017*

## Contents

1	Purpose and Overview . . . . .	2
2	Programmes, Documentation and the Eurace@Unibi Model . . . . .	2
3	Performance and the Linux system . . . . .	2
4	Quick Starter Guide . . . . .	3
5	How to use the Simulation GUI for customized Simulation Experiments . . . . .	5
6	Flame Modeling Environment . . . . .	23
7	Licensing . . . . .	25

## 1 Purpose and Overview

This User Manual for the ETACE Virtual Appliance describes how to conduct economic analyses with the Eurace@Unibi model using the programmes available on the VA and its respective tools, with regard to the general workflow of the Flexible Large-scale Agent-based Modelling Environment (FLAME<sup>1</sup>).

The virtual appliance has been created at ETACE, the Chair for Economic Theory and Computational Economics, at Bielefeld University. The intention behind this software collection is to make every step related to the initialization, execution, modification and analysis of the Eurace@Unibi agent-based simulation model as easy as possible. We hence address here the issue of reproducibility of simulation-based research (Stodden, 2010).

## 2 Programmes, Documentation and the Eurace@Unibi Model

This software package is based on the SliTaz distribution of free software, that includes the Linux kernel. The corresponding documentation is included in the software.

Programs provided in the Virtual Appliance (including their dependencies):

Xparser GUI	: Parser for FLAME models.
GNU GCC compiler	: C compiler for model + framework code.
Flame Editor	: Generate model.xml file, XML description of model.
Population GUI	: Generates initialization files, population description.
Simulation GUI	: Settings for simulation experiments and data analysis.

Apart from standard dependencies such as GNU GCC, the relevant documentation to these programmes can be found in the **Documentation** folder on the Desktop.

The following versions of the Eurace@Unibi model are included and can be found in the **Models** folder on the Desktop:

Dawid et al., 2011	: full source code of Eurace@Unibi 1.0
Dawid and Gemkow, 2013	: main, model.xml, 0.pop & 0.xml for reproduction
Dawid et al., 2014b	: full source code of the model used for the paper
Dawid et al., 2014a	: full source code of the model used for the paper

For the following papers, a pre-configured ready-to-run experiment can be found in the **./Preconfigured Experiments** folder:

Dawid et al., 2014a	: ./Preconfigured Experiments/WP_Dawid_Harting_Neugart_2014/
---------------------	--

## 3 Performance and the Linux system

The virtual appliance is set up with a minimum performance configuration. You can increase the performance of the VA especially by increasing the number of processors and the size of the allocated memory. However, the performance is limited by the configuration of the host system. To change these:

---

<sup>1</sup> <http://www.flame.ac.uk/>

- Enable **hardware virtualization support** in your own system's BIOS to run in multi-core mode (normally under menu "CPU" in BIOS).
- In the settings of the virtual machine itself (i.e. in the client of Oracle's VirtualBox but **not inside** this virtual appliance), you can change the number of assigned CPU cores and RAM memory.



The default memory size allocated to the VA might not be sufficient for the post-processing of large simulations with big data bases. In this case, in order to successfully finish the R processes that read the data and generate the plots of the simulations, it can be necessary to increase the size of the allocated memory.

To get root access in a terminal, type `su` with password `root`. The Super User is likewise `root` with password `root`. All the relevant files have been placed on the Desktop (`/home/eurace/Desktop`), whereas additional libraries (Libmboard, R, Python, GSL) have been installed directly into the system.

There is also the possibility to mount a shared folder in order to facilitate the exchange of files between the VA and the host system. A separate instruction how to mount a shared folder can be found on the dedicated webpage of the VA.

## 4 Quick Starter Guide

### The Desktop

After launching, you should see the desktop of the VA. The desktop contains folders and program launchers. The folders are

- **Documentation**; this folder contains the licenses of the ETACE Virtual Appliance, as well as manuals for GUIs and applications used in the VA. Furthermore, it contains a description of the Eurace@Unibi model together with several research papers based on the model.
- **src**; this folder contains source files of applications.
- **Models**; this folder hosts source code and executables of different versions of the Eurace@Unibi model.
- **exper**; this is an empty folder for storing new simulation experiments of the user.
- **Preconfigured Experiments**; this folder contains ready-to-run simulation setups for exact paper reproduction.



The launchers can be used to launch different GUIs:

- **PopulationGUI**; to create an initial state as input of a simulation.



- AgentGUI; to change the model structure.
- xparserGUI; to parse and compile the model.
- SimulationGUI; for setting up and starting simulations as well as post-processing simulation data.



Only the SimulationGUI is required for reproducing the experiments of the paper.

### Step-by-step Instruction: How to run a first simulation and to plot simple time series with the standard version of the model

The following guide provides a short and simple instruction to get users quickly accustomed to the basic operations of the VA. The guide describes step-by-step how to use the Simulation GUI to run a simulation with the standard model implementation of the Eurace@Uni model and, as an illustration, how to plot two time series, aggregate output and the unemployment rate. For more advanced applications, especially how to set up simulation experiments with a systematic variation of model parameters, the reader is referred to Section 5 of this Manual.

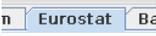
1. Double click the launcher of the Simulation GUI.
2. Choose/ or create a workspace, i.e. specify a folder in which the simulation data and figures are stored (default: /home/eurace/Desktop/exper).
3. If the workspace folder has not been used before and therefore does not contain saved settings, you are asked to enter the following:
  - Path to the model.xml file: Use the file browser to enter the file with the following path: /home/eurace/Desktop/Models/EuraceUnibi\_Model\_1.0/eurace\_model.xml.
  - Path to the main executable: Use the file browser to enter the file with the following path: /home/eurace/Desktop/Models/EuraceUnibi\_Model\_1.0/main.
  - Path to the start state of the simulation: Use the file browser to enter the file with the following path: /home/eurace/Desktop/Models/EuraceUnibi\_Model\_1.0/its/0\_after\_2k\_transient.xml.
4. Now you should see the main dialog window of the Simulation GUI. Modify the **number of iterations** and the **number of batch runs** to your preferred settings.

 When editing text fields, new values have to be confirmed by hitting enter.

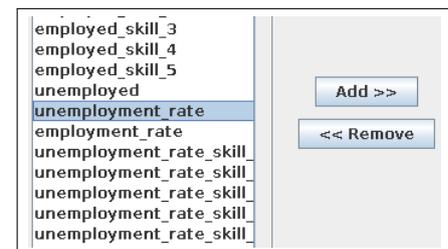
Number of batch Runs	<input type="text" value="10"/>
Number of Iterations	<input type="text" value="5000"/>
Number of Processes	<input type="text" value="1"/>

5. Select the agent *Eurostat* in the table. This agent contains the two time series unemployment rate and monthly output that we want to plot. It should be noted that the Simulation GUI saves only the data from the selected agents.

Agent	Record	Period	Phase
Firm	<input type="checkbox"/>		
House...	<input type="checkbox"/>		
Mall	<input type="checkbox"/>		
IGFirm	<input type="checkbox"/>		
Eurostat	<input checked="" type="checkbox"/>	20	0
Bank	<input type="checkbox"/>		
Govern...	<input type="checkbox"/>		
Centra...	<input type="checkbox"/>		
Clearin...	<input type="checkbox"/>		
snaps...	<input type="checkbox"/>		

6. Switch to the tab **Plotting Settings** 
7. Within the Plotting tab, choose the Eurostat tab: 
8. Select the menu **Time Series** → **Add/Remove Variable** of the Plotting tab. You should see a dialog window for choosing the variables to be plotted.

9. Select the variables `monthly_output` and `unemployment_rate` from the list of variables and add them to the empty list on the right-hand side. By clicking the right mouse you can order the list of variables alphabetically. To confirm your choice click **Apply Changes**.



10. Click first **Build Experiment** and then **Run Experiment** on the bottom of the main dialog window. Now the simulations start and you see a progress bar. If the simulation is finished, you can find the figures in the folder `Plots/Time_Series/Single` and `Plots/Time_Series/Batch`.

## 5 How to use the Simulation GUI for customized Simulation Experiments

The SimulationGUI has been developed to have an all-in-one user interface for setting up, running, and analyzing simulations of models implemented in FLAME.

The general design of the GUI acknowledges that setting up and analyzing simulation experiments are two separated tasks, the first done before the simulation and the second after the simulations have finished. These tasks are also separated in the GUI, which is achieved by providing two separated tabs, one for the simulation settings and another for the post-processing of the data. The Simulation GUI helps you to perform the following steps:

### Design of the Computational Experiment

1. Setting the environment constants (model parameters)
2. Selection of data storage options: agents to store, at which frequency (every iteration, every month,...)
3. Running a set of batch runs

4. Launching the simulation
5. After the simulation has finished and all data has been produced, the final stage is the data analysis stage. For this, we again make use of the Simulation & Analysis GUI.

## 5.1 Getting Started: Define the Work Space and Enter basic Settings

The GUI works with a workspace environment. This means there is a folder in the file system of the VA that is defined by the user as the location where all files related to a simulation experiment are located.

**How to set the Workspace** There are two possibilities to set the current workspace:

1. Set the workspace immediately after launching the GUI: a file browser opens after launching the GUI which can be used to load an existing workspace or to create a new workspace.
  - Load an existing workspace: use the file browser to browse to the top level folder of the workspace. Click **Open** to load the settings. The GUI reads the saved settings of the workspace and opens the main dialog window.
  - Create a new workspace: use the file browser to browse to a location in the file system where the new workspace should be created. Use the new folder button to create and name the workspace. In the following, the user has to set some basic settings manually. Use the file browsers to set the path to
    - the model xml file `eurace_model.xml`
    - the main executable (`main`)
    - the initial start state (`0.xml`)
2. Use the **Experiment** menu of the main menu bar and click **Load** to open an existing workspace. Use the file browser to browse to the top level folder of the workspace and load the workspace by clicking **Open**. To create a new workspace, click **New**. Use the file browser to browse to the location where the new workspace should be created. The new workspace is initialized with the current settings.

**Save the current state of the workspace** The current state of the workspace can be saved by using the **Save** item of the **Experiment** menu. The previously saved settings are overwritten. Alternatively, one can create a new workspace by using the **Save as** item of the **Experiment** menu. In this case, one gets a new branch of the workspace without overwriting the old settings.

**Importing and Exporting special configurations** There can be the situation that the user wants to flexibly use different settings as e.g. plotting settings without switching between workspaces. For this case, the user can use the import and export feature of the GUI. Use the **Import/-Export** item of the **Experiment** menu to export and import settings without branching the workspaces. The settings which can be imported/exported are

- **Plotting Settings:** Exporting and importing the settings related to the data analysis. This is to use different plotting profiles.

- Exporting: Use the file browser to define a location to which the plotting settings are exported. The GUI writes a xml file with the currently used plotting settings. The name of the file has to be specified by the user.
- Importing: Use the file browser to browse to the export location. Select and import the xml file in which the plotting settings have been exported.
- **Parameter Settings:** Export and import parameter settings in order to switch between different parametrizations of the model.
  - Exporting: Use the file browser to define a location to which the parameter settings are exported. The GUI writes a xml file with currently used parametrization of the model. The name of the file has to be specified by the user.
  - Importing: Use the file browser to browse to the export location. Select and import the xml file in which the parameter settings have been exported.

**Setting the required file paths manually** When creating a new workspace from scratch, the user is automatically asked to set the correct path to several file resources required by the GUI. However, these paths can also be set manually. It should be noted that these path are mandatory and have to be set correctly before setting up an experiment.

Use the **Set Paths** item of the **Settings** menu to check or change paths used for the simulation. The paths to be set are

Set Path to Model.xml file
Set Model Executable
Set initial Data File {0.xml}
Set path to R Scripts
Set path to xparser

- path to the eurace\_model.xml file.
- path to the main executable.
- path to the 0.xml file.
- path to the top level folder of the R scripts. In the VA the full path of this folder is `/home/eurace/Desktop/src/JavaGUI/src/Data_Analysis_GUI_Serial/`.
- path to the executable of the xparser. In the VA the full path is `/home/eurace/Desktop/src/xparser-0.17.1/xparser`.



For the pre-configured experiments in `./Preconfigured Experiments/`, all paths have already been set correctly.

## 5.2 How to setup an Experiment

**Simulation Settings** **Plotting Settings**

The **Simulation Settings** tab, which is the first tab on the main dialog window, can be used to specify the settings that are related to the simulations. There are basically three different kinds of settings to be can specified:

Simulation Settings | Plotting Settings

Run Simulations  Do not run Simulations

Number of batch Runs   
 Number of Iterations   
 Number of Processes

Run only one Batch  
 Parameter Variation with one Parameter

probability tec...
0.0
0.1
0.2
0.3
1.1

Do not compress  
 Compress Databases and remove Original  
 Compress Databases and keep Original  
 Remove decompressed Database  
 Decompress Database

Agent	Record	Period	Phase
Firm	<input checked="" type="checkbox"/>	20	0
House...	<input type="checkbox"/>		
Mail	<input type="checkbox"/>		
IGFirm	<input type="checkbox"/>		
Eurostat	<input checked="" type="checkbox"/>	20	0
Bank	<input type="checkbox"/>		
Govern...	<input type="checkbox"/>		
Centra...	<input type="checkbox"/>		
Clearin...	<input type="checkbox"/>		
snaps...	<input type="checkbox"/>		

Write all agent variables to Database  Yes  No

1. General setup of the simulations, i.e. the length of the simulation (number of iterations), the number of batch runs and the parametrization of the simulated model.
2. Experiment mode, i.e. whether or not the model is run multiple times with a specified parameter being varied.
3. Data management; which simulation output will be stored for the post-simulation analysis.

### 5.2.1 General Setup

#### Launch simulations and plotting or plotting only

Run Simulations  Do not run Simulations

These radio buttons define whether the GUI launches both, the simulations and the plotting process or the plotting process only. This feature can be helpful if the simulation data has already been generated and the user wants to change only plotting settings.

#### Set number of batch runs

Number of batch Runs

Enter the number of batch runs. For each parameter value of the experiment, the GUI launches the corresponding number of simulations. The runs only differ by the random seed used.

 Entering new values in any text field **must always** be confirmed by hitting enter.

**Note on random seeds:** The random seed is a parameter (model constant) in the model.xml file called RND\_SEED that needs to be initialized (or else defaults to zero). This allows us to include the value of the random seed in the 0.xml file, and then use it as an input into the simulation. This way, for all runs we use a different random seed, and still have full reproducibility of the simulation results, because the value of the seed is stored in the initialization files.

#### Set number of iterations

Enter the number of iterations. Each simulation run is executed for the indicated number of iterations. In the Eurace@Unibi model, one iteration corresponds to one day. Considering only working days, 5 iterations constitute a week, 20 iterations a month and 240 iterations a year. A good starting point for a standard experiment is 5000 iterations, which corresponds to 20 years.

Number of Iterations

### Set number of process threads

Number of Processes

This pull-down menu can be used to set the number of simulation processes. This is helpful to reduce the simulation wall-time of an experiment.

 It is recommended to execute simulations in multiple process threads but to limit the number of threads to the number of cores with which the VA is configured.

### Change the parametrization of the model

This button opens a new dialog window which can be used to change the value of any model parameter.

Parameter	Value
total regions	2
id_debug_probe	50
print_log	0
print_debug_bankruptcy	0
print_debug_eurostat	0
print_debug_ch	0
print_debug_afm	0
print_debug_afm_ch	0
print_debug_exp1	0
print_debug_file_exp1	0
print_debug_file_exp2	0
policy_exp1	0
print_debug_gov	0
print_debug_household	0
print_debug_credit	0
print_debug_consumption	0
print_debug_finance	0
print_debug_production	0
print_debug	0
print_debug_market_research	0
policy_exp_energy_shock	0
policy_exp_stabilization_subsidy	0
policy_exp_stabilization_tax	0
const_bankruptcy_idle_period	240
days_per_month	20
xml_cloned	0
individual_learning	0
const_wage_wealth_ratio	0.200000
const_income_tax_rate	0.050000
gamma_const	9.000000
gamma_quality	1.000000
depreciation_rate	0.010000
discount_rate	0.020000
mark_up	0.200000
lambda	1.000000
target_savings_rate	0.100000
carrot_consumption_parameter	0.010000
quantil_normal_distribution	0.524400
firm_planning_horizon	10
adaption_delivery_volume	0.100000
inv_inertia	10.000000
gamma_logit_vintage_choice	30.000000
delta_init_vintage_choice	3.000000

- New parameter values can be entered in the table.

 Hit return after entering a new value.

- Confirm the changes and close the dialog window.
- Discard the changes and close the dialog window.
- Reset **all** changes to the default values of the experiment saved in the 0.xml file.

## 5.2.2 Experiment Mode

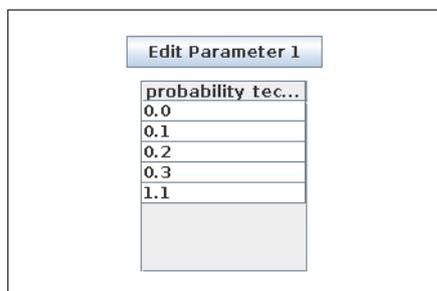
### Select whether or not a model parameter is varied

These radio buttons can be used to switch between two experiment modes:



1. *Run only one Batch*: run a batch only with the current parameter setup.
2. *Parameter Variation with one Parameter*: a selected model parameter is varied among specified values

### Select the model parameter for the experiment

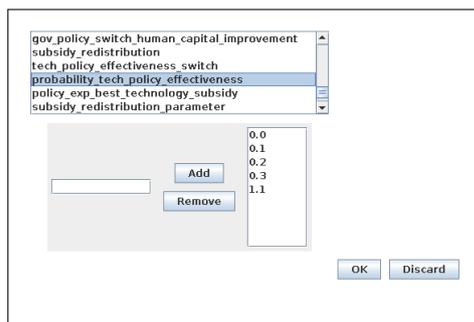


Clicking Edit Parameter 1 opens a dialog window in which the user can select the parameter to be varied in the experiment as well as can define the parameter values used in the experiment. This button is only active if the radio button *Parameter Variation with one Parameter* is selected.

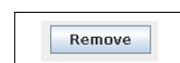
- Scroll through the list of parameters and choose the parameter for the experiment by clicking on that parameter.



Clicking the right mouse button orders the list alphabetically.



- Enter one value or multiple, by comma separated values in the text field.
- Select one or more values and click remove to remove these values from the list.
- Click OK to confirm all changes and close the dialog window.
- Discard the changes and close the dialog window.



### 5.2.3 Data management

#### Compress or decompress the databases

Do not compress  
 Compress Databases and remove Original  
 Compress Databases and keep Original  
 Remove decompressed Database  
 Decompress Database

These radio buttons can be used to define whether or not the databases should be compressed after finishing all jobs. Depending on the storage settings, compressing databases can save hard disk space. These radio buttons can be used in combination with the **Do not run** radio button to compress and decompress data bases without running the simulations again.

### Data recording of agents

This table specifies which agent type should be recorded and at which frequency.

Agent	Record	Period	Phase
Firm	<input checked="" type="checkbox"/>	20	0
House...	<input type="checkbox"/>		
Mall	<input type="checkbox"/>		
IGFirm	<input type="checkbox"/>		
Eurostat	<input checked="" type="checkbox"/>	20	0
Bank	<input type="checkbox"/>		
Govern...	<input type="checkbox"/>		
Centra...	<input type="checkbox"/>		
Clearin...	<input type="checkbox"/>		
snaps...	<input type="checkbox"/>		

- By checking the check boxes in the Record column, the user selects the agent type to be recorded.
- The user can determine the periodicity of data recording (20 by default, which means the data is recorded on a monthly base) and the phase shift (0 by default, which means in combination with period 20 that data is recorded at iteration 20, 40, 60 etc.).
- The snapshot option allows to record a full snapshot of the agent population with the specified phase and periodicity.



Hit return after entering a new value.

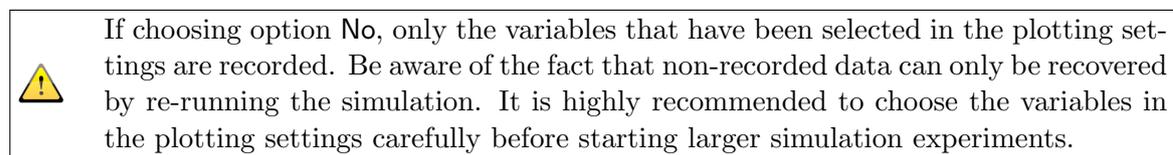
### Record all agent variables

These radio buttons are important for the space requirement of the VA.

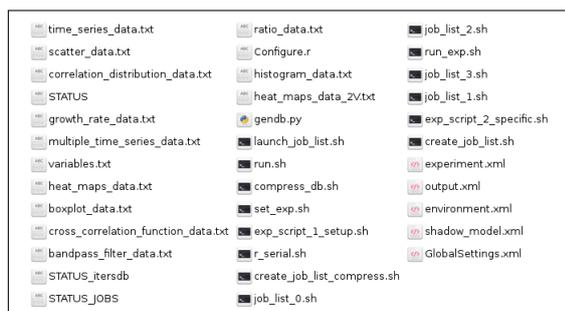


- If **Yes** is selected, then the full memory of each selected agent is written to disk. As one typically needs only a small subset of agent's memory variables for the post-simulation analysis, this setting can imply a waste of hard disk space especially if running large simulations.

- If **No** is selected, then only a small subset of agents' memory variables are recorded. The selection of memory variables to be recorded is based on the selection for the plotting settings (see Section 5.4).

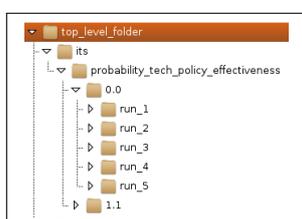


### 5.3 How to build and Run experiments



- Before starting a simulation, the simulation has to be built by clicking **Build Experiment**. This means the configuration of the experiment, which has been entered through the GUI, is translated into files (\*.sh, \*.xml and \*.txt files) which can be read by the simulator to run the simulations, the R software to analyze the data and by the operating system for file operations and to control the sequence of activities.
- The GUI writes these scripts in the top level folder of the work space.

- Clicking **Run Experiment** launches the actual simulation experiment. A simulation experiment is a two step process:



1. The simulation of the model. The simulator (`main` executable) is executed and writes the simulation data as output to file. The progress of the simulation is indicated by a progress bar which automatically pops up when the simulation is being started. For writing the simulation data, the GUI creates a folder with a specific hierarchy in the workspace with `its` as top level folder. Each simulation run is written in a specific sub folder of the hierarchy whose path relative to `its` can be used to identify the particular run. The simulation data is first written into iteration specific xml files (e.g. `20.xml`, `40.xml`, etc.). In a second step, these data files are translated into SQL databases and finally deleted.



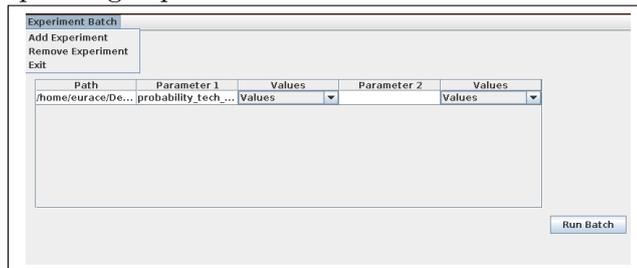
Closing terminals or the GUI ends the simulation. Terminals or the GUI should only be closed if the user wants to terminate the simulation.

2. The data processing. The SQL databases are read by R in order to carry out a user defined data analysis. The results of the data analysis are written to files that are stored in a folder system below the `Plots` folder, which is created after the simulations have finished. The `log` folder, which is also created during the data analysis, contains the console output of the R scripts and can be used to read error messages in case the figures have not been created correctly.

It should be noted that without specifying additional settings for the post-simulation analysis, the Simulation GUI runs the simulator to generate simulation data but it does not generate any plot that graphically describes the simulation output. How to generate plots is described in the next Subsection 5.4.

**How to run multiple experiments in a row** There is also a possibility to run more than one ready-to-run experiment automatically without launching each of them separately. Use the **Run Batch** item of the **Experiment** menu. This opens a dialog window for selecting ready-to-run experiments that can be executed without separate launching.

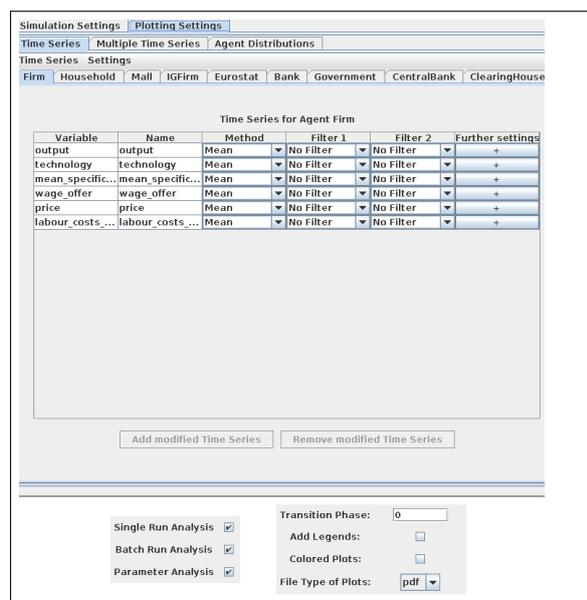
- Click **Add Experiment** in the menu of the dialog window to add an experiment to the list. Use the file browser to load the experiment.xml file of the corresponding experiment which is located in the workspace folder of the experiment.



- If the list of experiments is complete, click **Run Batch**. The GUI starts to run the first experiment on the list and continues with the next after finishing the previous experiment.

## 5.4 How to set up the Post-Simulation Analysis

The GUI provides basically two ways of analyzing the simulation data of an experiment. The one is to consider aggregated time series and the other distributions of agents' memory variables. In order to configure the post-simulation analysis, the user selects the Plotting Settings tab of the GUI. The user can switch between three tabs:



1. **Time Series** to plot time series of single variables
2. **Multiple Time Series** to plot multiple time series in a common plot using the same scale.
3. **Agent distributions** to create distribution plots such as box plots or histograms for specified iterations.

The plotting process is automatically started after the simulations have finished. However, if the simulation data already exists, the plotting can also be started without running simulations. Therefore, one has to select the **Do not run Simulation** radio button on the **Simulation Settings** tab. In this case, the simulation step is skipped and the plotting starts immediately.



### 5.4.1 Basic plotting settings

At the bottom of the plotting tab there are elements that can be used to set some basic plotting settings.



- The check boxes **Single Run Analysis**, **Batch Run Analysis** and **Parameter Analysis** can be used to opt whether the post-simulation analysis should include an analysis of single runs, of batch runs and a detailed analysis based on the varied parameters.

- **Transition Phase:** This allows to cut off an initial transient of the time series.



This is not the same as the transient phase used to create the initial state file (0.xml). This transient phase is not stored.

- **Add legends:** Specifies whether legends are added to plots.
- **Colored Plots:** Specifies whether the lines of the plots are colored or black.
- **File Type of Plots:** Select the file type (at the moment only the pdf format).

### 5.4.2 How to plot Single Time Series



In order to plot single time series, one has to switch to the **Time Series** tab of the *Plotting Settings*. The tab **Time Series** itself contains a set of tabs, each tab for an agent type of the model. To generate time series for different agent types, one can simply switch between the agent tabs.



In order to generate time series for an agent type, one has to make sure that this particular agent type has been selected in the data recording table on the **Simulation Setting** tab.

**Defining Time Series** To generate a time series of a memory variable of an agent, the user has to specify settings. Besides the agent type and the name of the variable, those settings include a method which is applied to aggregate the data and filters used to draw a sub sample of the population featuring certain characteristics. The agent tab contains a table listing the currently selected time series. The table also provides an interface to edit the time series settings.

Time Series for Agent Firm						
Variable	Name	Method	Filter 1	Filter 2	Further setti...	
output	output	Mean	No Filter	No Filter	+	
technology	technology	Mean	No Filter	No Filter	+	
mean_specific...	mean_specific...	Mean	No Filter	No Filter	+	
wage_offer	wage_offer	Mean	No Filter	No Filter	+	
price	price	Mean	No Filter	No Filter	+	
labour_costs_...	labour_costs_...	Mean	No Filter	No Filter	+	

- **Variable:** The name of the variable used in the model.
- **Name:** A customized name of the time series. Change the name by editing the corresponding field of the table. This name is used as the file name when generating the plots and will appear on the y-axis of the plot.



Hit return after editing a time series name.

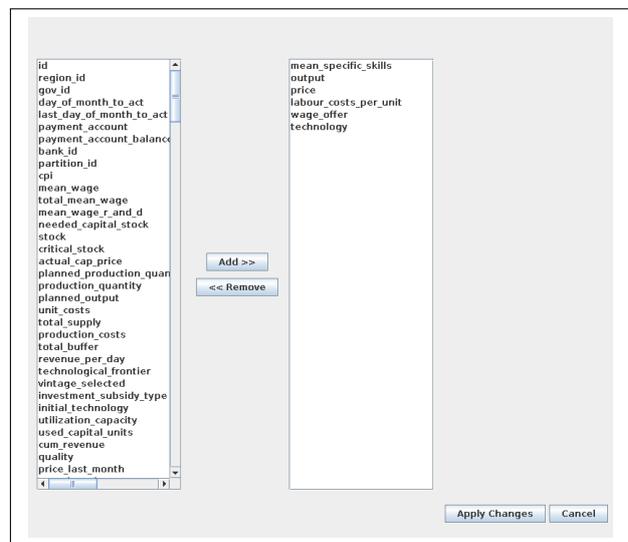
- **Method** Pull-down menu to define the function applied to aggregate the time series. One can use
  - Mean
  - Median
  - Standard deviation
  - Percentage standard deviation
  - Sum
  - Minimum
  - Maximum
  - Weighted averages; this method can only be selected after specifying a weighting factor in the **Settings** menu.
- **Filter 1** and **Filter 2** The user can apply two filters on the time series. The filters can be defined in the **Settings** menu.
- **Further Settings:** Clicking the button opens a dialog window to define:
  - **tmin** and **tmax:** Defines the limits to the x-axis, i.e. the range in terms of iterations within which the time series is plotted.
  - **Lower Bound** and **Upper Bound:** Customized limits of the y-axis.



The same memory variable can be used for different time series.

**Adding new Time Series** There are two possibilities to add new time series. Which one to use depends on whether or not the memory variable from which a time series should be plotted has already been selected for plotting time series.

- If the variable **has not yet been used** for plotting time series: Click the **Add/Remove Time Series** item from the **Time Series** menu bar. This opens a dialog window, which can be used to select variables from the variable list of the considered agent. Mark one or more variables and click **Add** to select these variables as time series. One can also remove time series which should not be plotted any more by marking them and clicking **Remove**. Clicking **Apply Changes** confirms the changes and closes the dialog window.

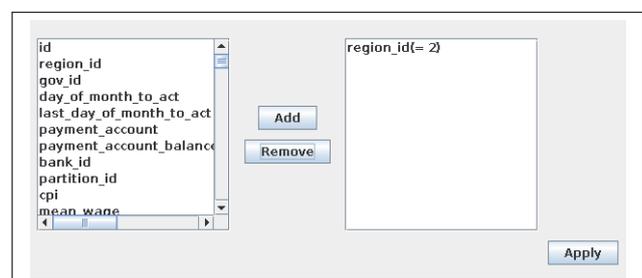


The list of variables can be ordered alphabetically by clicking the right mouse button.

- If the variable **has already been used** for plotting time series: Mark the time series in the table and click the **Add Modified Time Series** button. The GUI adds a new line to the table containing a new time series of this variable with the default settings. The name of the new time series is the name of the variable plus a number as each time series need a unique name.



**Defining filters:** Filters can be defined by clicking the **Filter** item in the **Settings** menu, which opens a dialog window for editing filters. Filters can be used to define sub populations of an agent type by **filtering out** the agents that do **not** have the requirement defined by the filter. By clicking on one variable from the list, one selects the filter *variable*. Another window pops up to enter the filter options (filter *value* and filter *method*). The filter *methods* are:



- Ignore agents with  $variable = filter\ value$
- Ignore agents with  $variable > filter\ value$
- Ignore agents with  $variable < filter\ value$
- Ignore agents with  $variable \neq filter\ value$

By clicking **Apply** the settings are confirmed and the new filter can be selected from the pull-down menu **Filter 1** and **Filter 2** in the time series table.

#### Define a default settings profile:

Click **Default Settings** in the **Settings** menu which opens a dialog window in which the default settings can be defined. The default settings are those settings that are automatically chosen when defining new time series.

#### Define weighting factor for weighted averages:

Click **Select Weighting Factors for Means** in the **Settings** menu, which opens a dialog window in which weighting factors can be defined to compute weighted averages. Select variables from the list to have the possibility to compute averages over the population weighted by this variables. The additional aggregation methods are can then be selected as additional options in the pull-down menu of the time series table. A possible application is a price index for which an average price weighted by firms' sales is computed.

### 5.4.3 How to plot Multiple Time Series



All time series defined on the **Time Series** tab are plotted as a single time series graph in a single plot. If you want to plot more than one of these time series together in one plot, one you use the **Multiple Time Series** tab to define those multiple time series.



Only time series that have been selected for plotting as single time series can be plotted in a multi plot.

The **Multiple Time Series** tab contains a table listing the currently used multiple time series and the corresponding settings.

Multiple ...	Tmin	Tmax	Compone...	Lower Limit	Upper Limit
mts_tec...	1	28000	List o... ▼	<input type="checkbox"/>	<input type="checkbox"/>

- **Multiple Time Series Name:** The name of the time series; the name is a combination of the prefix **mts** and the names of the included time series. This name is used as file name when generating the plots.
- **Tmin** and **Tmax** as minimum and maximum iteration numbers to appear in the plot.
- **Components:** This shows a list of the time series.
- **Lower Limit** and **Upper Limit:** Customized limits for the y-axis.

**Add Multi Time Series** In order to create a new multi time series or to edit an existing one, you can select the **Multiple Time Series** menu and click **Add Time Series** (or **Edit Time Series**). This opens a dialog window in which a multiple time series can be specified.

- Double click on the agent type from which the time series should be added.
- After choosing the agent, you can now select the time series from the list of available variables.
- Confirm the settings by **OK**.



The time series are plotted by using the same scale of the y axis.

**Defining default settings** To define default settings, use the **Default Settings** item of the menu **Settings**. One can set default values for the minimum iteration and the maximum iteration.

### 5.4.4 How to plot Agent Distributions

Time Series Multiple Time Series **Agent Distributions**

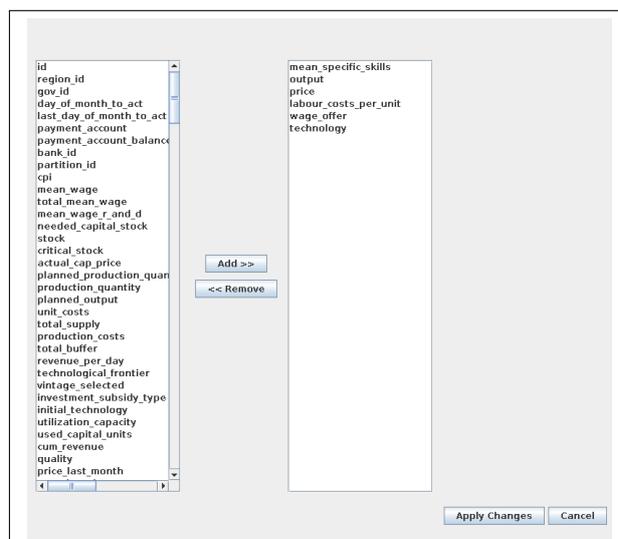
The **Agent Distribution** tab can be used to analyze the distribution of agent variables in the population. The distribution of a variable is plotted as a box plot and as a histogram for a point in time or for a discrete sequence that has to be specified by the user. By default, the distribution is plotted for the last iteration of the simulation.

To analyze the distribution of an agent variable, switch to the corresponding agent. There is a table listing the variables currently used for plotting and which filters are applied, i.e. the properties of the considered sample of the population.

Variable	Name	Filter 1	Filter 2
unit_costs	unit_costs	No Filter	No Filter
output	output	No Filter	No Filter

**Adding new Distributions** There are two possibilities to add a new distribution. Which one to use depends on whether or not the memory variable for which a distribution should be plotted has already been selected for plotting distributions.

- If the variable **has not yet been used**: Click the **Add/Remove variables** item from the **Agent Distribution** menu bar. This opens a dialog window, which can be used to select variables from the variable list of the considered agent. Mark one or more variables and click **Add**. One can also remove distributions from the list by marking them and clicking **Remove**. Clicking **Apply Changes** confirms the changes and closes the dialog window.



The list of variables can be ordered alphabetically by clicking the right mouse button.

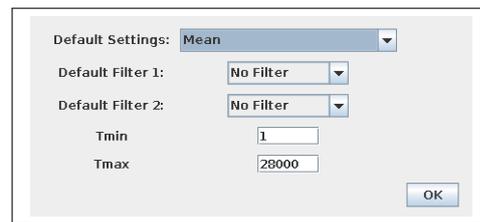
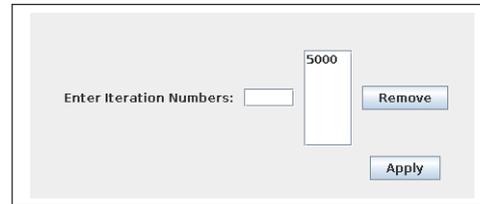
- If the variable **has already been used**: Mark the distribution in the table and click the **Modify variable** button. The GUI adds a new line to the table featuring the default settings. The name of the new distribution is the name of the variable plus a number as each distribution needs a unique identifier.



**Defining filters:** same as for adding new variables for time series plotting (see Page 17)

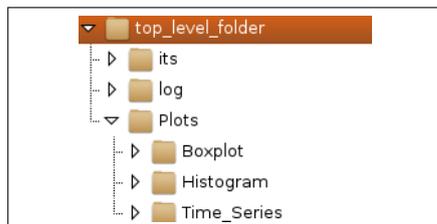
**Specifying time snapshots for distributions:** Click the **Iterations** item in the **Settings** menu, which opens a dialog window in which the iterations can be selected for which distributions are plotted. One can enter a sequence of iterations by entering comma separated values.

**Define default setting profile:** Click the **Default Setting** item in the **Settings** menu, which opens a dialog window in which the default settings can be defined. The default settings are those settings that are automatically chosen when defining new time series.



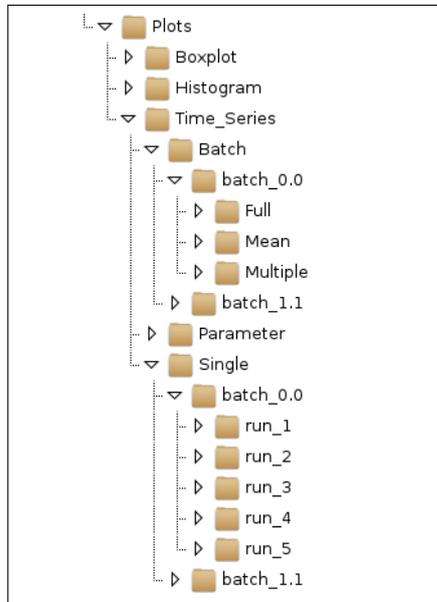
### 5.4.5 Plotting Output

The GUI launches the plotting process automatically after the simulations have finished. During the plotting process, the R jobs generate plots and write these plots to files in a specific sub folder structure. These sub folders can be found in the **Plots** folder located in the workspace. The **Plots** folder contains three sub folders:



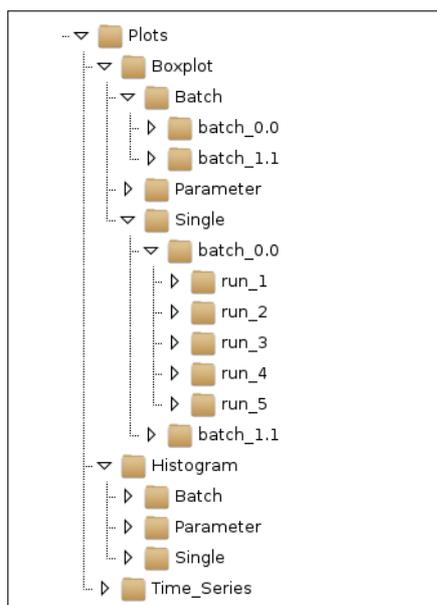
- **Boxplot**, which contains the plots showing the agent distribution as box plots.
- **Histogram**, which contains the plots showing the agent distribution as histograms.
- **Time\_Series**, which contains the single and multiple time series.

The **Time\_Series** folder has three sub folders:



- **Single:** This folder contains for each single run of each batch all single time series plots and multiple time series plots.
- **Batch:** Summarizes the time series plots of the batch analysis. The directory name of the batch sub folders indicate for which parameter value the batch plots have been generated. There are three sub folders
  - **Full:** Contains plots that show all time series of the batch runs.
  - **Mean:** Contains plots which depict the *average time series* of a batch in a single plot. The time series are averaged across the batch runs.
  - **Multiple:** Contains the multiple time series; the time series included in the plots are averaged across the batch runs.
- **Parameter:** Contains time series plots that shows for each of the considered parameter values the average time series of the batch in a common graph. This can be used to compare the effects of a variation of the parameter of the experiment.

The folder structure for distributions is the same for boxplots and histograms and is almost the same as used for time series.



- **Batch:** Summarizes the distribution plots of the batch analysis. The directory name of the batch sub folders indicate for which parameter value the batch plots have been generated.
- **Parameter:** This folder contains distribution plots showing in one plot the distributions for each batch of the considered range of parameter values.
- **Single:** This folder contains distributions for each single run of the batches.

## 6 Flame Modeling Environment

The Eurace@Unibi model is implemented in the Flame development framework. We provide the full source code for the Eurace@Unibi 1.0 model. If you want to get a deeper understanding of this code, make changes (i.e. implement a new policy rule) or experiment with different initial populations, continue reading.

A Flame model development cycle goes through several stages, as illustrated in Fig.1.

### 6.1 What is FLAME?

In a nutshell:

- FLAME is a program generator. It generates a simulation executable from C and XML files.
- FLAME is a domain-specific language programmed in C, that provides facilities for higher-level programming of economic and biological models.
- It uses the markup language XMML (X-Machine Modeling Language) for the declaration of function and variables, and template files in C to generate the final C code of the model.
- All model functions are written in plain C code (a small subset of C), while the scheduling of agents, functions and messages is coded in XML, with the help of an easy-to-use FLAME Editor.
- The XML model file is then parsed by the Xparser, producing C code for the simulator. This C code is then compiled together with the user-provided C code for the agent functions, which generates the simulation executable.
- Libmboard (Message Board Library) provides facilities for message passing.

### 6.2 Flame Model Workflow

A detailed workflow can be found below. For producing new models quickly, we note that:

- To view a model and its agents, functions, messages, etc., use the FLAME Editor and open the `model.xml` file.
- To generate initial data in an `0.xml` file, use the Population GUI. Also use this to view or edit pre-existing `.pop` files (population description files). These files can be found in the same `/its` folder as the default `0.xml` file.
- In order to keep this image lightweight we do not provide a front-end for C programming. To view and edit the code, you can use `vi`, `nano` or `leafpad`, all available from the terminal. If required, additional programmes can be installed using the SliTaz package manager `tazpkg`. For further detail on that, we refer here to the SliTaz documentation.
- Once you have created a new `model.xml` or `0.xml` file, use the `xparser` to parse and compile the model. After double-clicking the desktop shortcut, choose the `model.xml` file (usually called `eurace_model.xml`). After x-parsing the model you will be prompted to press ENTER to compile the model<sup>2</sup>.

---

<sup>2</sup> For advanced users: the GUI version of the xparser does not provide any additional options. If you require these, you will have to run the xparser from the command line: `/home/eurace/Desktop/Models/xparser-gsl/xparser`. Options include: `-p`: parallel code, `-f`: final production mode.

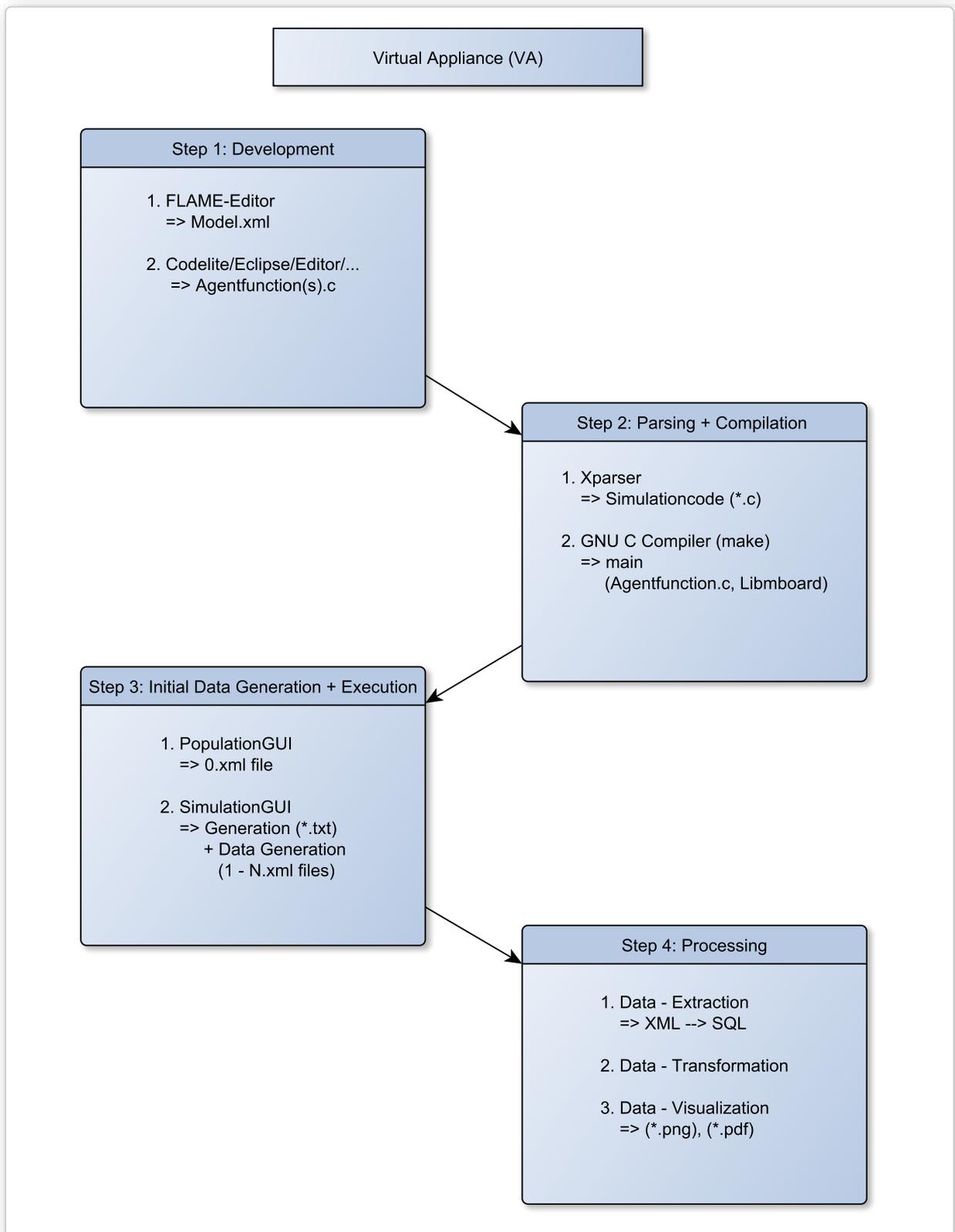


Fig. 1: Overview of the Flame Workflow and Programmes in the ETACE Virtual Appliance.

### 6.3 Flame Editor: Model Design Stage

The model design stage starts with setting up the general model hierarchy. The entire model can be subdivided into several modules, each having an internal structure that adheres to the XML definition. Each module will contain a well-defined description of the agents as X-Machines. In fact, each module itself is also an X-Machine, and the model as a whole can also again be construed as X-Machine (*c.f.* Holcombe, 1988; Balanescu et al., 1999).<sup>3</sup>

In the next stage we define the agent types, their memory variables, functions, and the activation structure of the functions (scheduling of functions can be either time- or event-based, or both). At this stage we also define message definitions, and set the input and output messages for the functions. In addition, the environment constants (fixed model parameters) can be defined. The complete XML structure can be tested for consistency before writing any actual code by using the Flame Editor GUI developed by Simon Coakley from Sheffield University.

After Xparsing the full model, the model design stage ends and the hierarchical structure can be inspected through a birds-eye view of the model in a stategraph, an example of which is given in Figure 2. Such a stategraph shows for every agent its states and the transition functions between states. It also shows the branching of agent activities, depending on time conditions (monthly or yearly activation) or on event-based conditions (a memory variable of an agent). In addition, the stategraph also shows the flow of information between agents. In principle, FLAME also allows agents of a specific type to send messages to agents of the same type.

### 6.4 Population GUI: Agent Population Instantiation

After the entire model design stage is done, the next stage is to initialize our agent population. This initialization is done using the PopGUI. This stage consists of setting the size of the agent population, the number of regions, and the subpopulation of agents in each region. We can initialize all model constants and agent memory variables. For each agent memory variable we can define relationships on the initialization values of other memory variables of the same agent or of other agents, or on the values of model constants. It is possible to validate the complete set of relationships before instantiating the population. At the end of this stage, a complete population description file (called `0.pop` file) has been generated, with all the interdependencies between the agents' initial values resolved. From this, the input file, called `0.xml` file, is generated.

In the case you want to use FLAME to develop a new model, a tutorial and further explanatory materials can be found in the `Documentation` folder. To explain this in detail would go beyond the scope of this user manual.

## 7 Licensing

The "ETACE Virtual Appliance" is made available under the GNU General Public License. This does not affect any contributions to the Appliance by others, which are included with permission of the authors and whose rights remain non-infringed upon.

Use of the source code of the Eurace@Unibi model is guided by the End-User License Agreement (EULA), that can be found in the "Documentation" folder of the Software Package. If you cannot find this file in your copy of the Software Package, or if you claim any further rights on some parts of this work, please contact [ETACE@wiwi.uni-bielefeld.de](mailto:ETACE@wiwi.uni-bielefeld.de).

<sup>3</sup> The XMML of FLAME has been defined as a hierarchical DTD (Document Type Definition). This means that FLAME accepts a nested model definition consisting of sub-models, which again may consist of sub-sub-models, and so on and so forth. This is possible due to the fact that a collection of Communicating Stream X-Machines can again be considered an CSX-Machine.

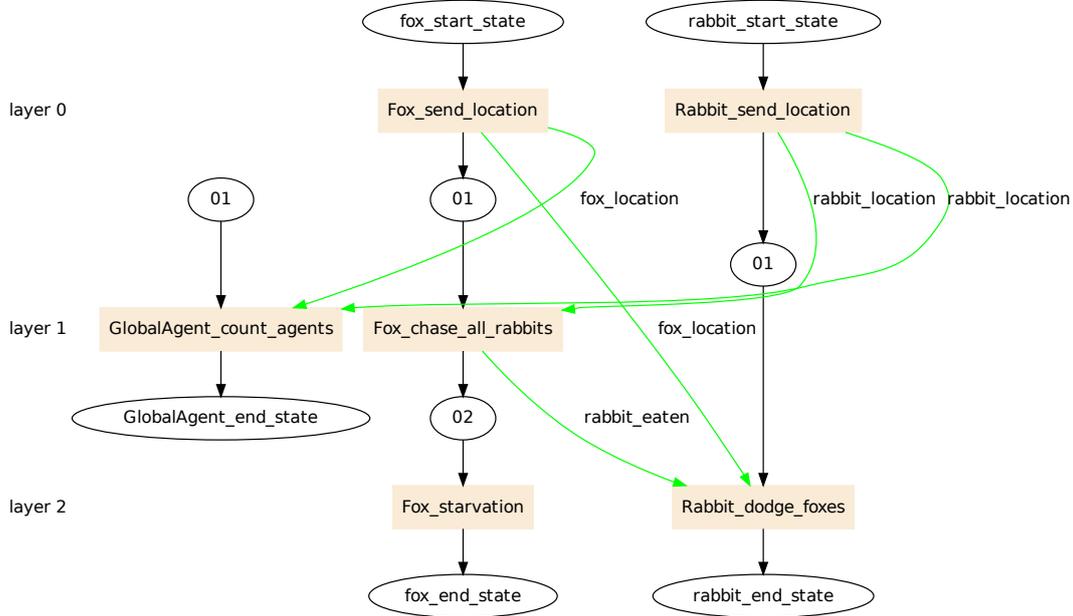


Fig. 2: State graph of a simple predator-prey model with foxes and rabbits.

A full list of all the contributors to this Appliance or the included software, as well as their respective licenses can likewise be found in the "Documentation" folder.

## References

- Balanescu, T., Cowling, A. J., Georgescu, H., Gheorghe, M., Holcombe, M., and Vertan, C. (1999). Communicating Stream X-machines systems are no more than X-machines. *In Twelfth International Symposium on Fundamentals of Computation Theory (FCT'99), Iasi*, 5:494–507.
- Dawid, H. and Gemkow, S. (2013). How do social networks contribute to wage inequality? insights from an agent-based analysis. *Industrial and Corporate Change*.
- Dawid, H., Gemkow, S., Harting, P., van der Hoog, S., and Neugart, M. (2011). The eurace@unibi model: An agent-based macroeconomic model for economic policy analysis. Available from [www.wiwi.uni-bielefeld.de/vpl1/projects/eurace/eurace-unibi.html](http://www.wiwi.uni-bielefeld.de/vpl1/projects/eurace/eurace-unibi.html).
- Dawid, H., Harting, P., and Neugart, M. (2014a). Cohesion policy and inequality dynamics: Insights from a heterogeneous agents macroeconomic model. *DFG Research Center (SFB) 882 From Heterogeneities to Inequalities*, 34.
- Dawid, H., Harting, P., and Neugart, M. (2014b). Economic convergence: policy implications from a heterogeneous agent model. *Journal of Economic Dynamics and Control*, 44:54–80.
- Holcombe, M. (1988). X-machines as a basis for dynamic system specification. *Software Engineering Journal*, 3(2):69–76.

---

Stodden, V. C. (2010). Reproducible research: Addressing the need for data and code sharing in computational science. *Computing in Science & Engineering*, 12(5):8–12.