BBIPED Platform

Author : BBIPED Support (bbiped@bcamath.org)
Date : June/2014
Version of Document : V1.5.0
Version of GUI : V1.5.0

Collaborator

The BBIPED Platform was developed within the project " Development of an efficient, flexible and innovative CFD Computational Platform to optimally simulate and design industrial products and processes", co-funded by:

Table of Contents

Graphical Interface User Guide

Motivation:

✔ The main motivation is to provide an standardized **interface** for the whole process, integrating the main tools used by the engineers.

✔ A set of tools will be integrated under the final version, and one of them is the **SU2 tool**

✔ For the **1st GUI version**, the **interface** will address the provision of a set of **basic functionality** to graphically **connect with SU2 tool and its resources**

✔ The **BBiped platform** will integrate several tools that belongs to the whole process, as it is shown in the graphic

✔ The **1st version** will be focused on the interface definition to **integrate the SU2 tool**

This document is intended to explain the basic functionality of the BBIPED platform.

In this case, the graphical interface (GUI) is related to the 1st version of the platform. This is, only the integration of SU2 has been done. The main idea behind this GUI is shown in the picture. The SU2 has been selected since it currently has no interface at all. The researches shall go through configuration files, with no automatic selection of properties and variables, loosing control in the changes.

Features (Version 1.5.0)

- ✔ **Common platform** that homogenizes the usage of SU2 tool within the CFD process
- ✔ Easing **project version** & associated documentation storage and track within a common folder structure
- ✔ Possibility to run the configuration using different Solver Engines
- ✔ Possibility to define projects for Multizone Execution
- ✔ Possibility to compare the differences among the configuration files in Multizone Projects
- ✔ The **Solver** configuration file features:
  - ✔ Easing the configuration of the simulation with different **views**
    - ✔ **Basic configuration view**: Only for those engineers that need to change some values, but they have no depth knowledge about Solver
    - ✔ **Advanced configuration view**: For expert users for a more customized configuration
    - ✔ **Personalized configuration view** based on a custom template (defined by the user)
  - ✔ **Automatic generation of configuration files** for simulation according the user input
  - ✔ **Automatic detection of the boundary names**, loaded from your mesh file. This prevents naming errors and keep the boundary configuration error-safe
  - ✔ **Automatic run your Solver** with your own configuration and mesh files from the platform
  - ✔ Currently **support for SU2**, and customized solver engines supporting Virtual and Multizone execution
  - ✔ Get the **help** associated to each variable
- ✔ The **SU2 simulation features** :
  - ✔ **Graphical evolution** of the simulation
  - ✔ **Save** your simulation evolution image
  - ✔ **Customized graphical views** based on the different user parameters
  - ✔ **Integration with Paraview** or any similar technology

Technical Requirements

This GUI is only tested to run under the following technical requirements :

✔ OS 64 bit

✔ Linux Ubuntu 12.04

Note : This platform cannot be run under OS 32 bit

✔ Qt libraries (v4.8)

✔ LibQwt v 6.0 installed.

Note : If you get the error that this library is missing, you need to go to the Ubuntu SW center to download the corresponding version. Please, be aware of the version of this library, it is the stable version v6.0.0

Other Tools

✔ The GUI can automatically connect to different tools. Please, go to the official site to download the stable version

✔ **Meshing**. The default viewer is Salome. This version of the GUI has been tested to open the **Salome V7.2.0**.

✔ Salome v7.2.0. For this tool in Ubuntu, it is needed to add it to the current path. To keep the configuration, the best way is to add this lines to your .bashrc file:

✔ export SALOME_HOME="/your path/appli_V7_2_0"

✔ *export PATH=$PATH:$ SALOME_HOME*

✔ *The executable shall be called runAppli (the same name as the installation provides).*

✔ You can select a different viewer to be launched from the BBIPED platform

✔ **The Solver Engine.** By default, this GUI is intended to provide support for the **SU2 tool** (tested till version 2.04)

***Note :** Please, follow the instruction details of SU2. At the end of the installation process, please, configure the following environment variables into your bashrc file (see an example):*

✔ *export SU2_RUN="/usr/local/bin" (only if you have your binaries here)*

✔ *export SU2_HOME="/your path/SU2v2.0/trunk"*

✔ *export PATH=$PATH:$SU2_RUN*

Note :These variables can be copied and pasted from the output result of the SU2 installation. Notices that this avoids the need to copy and paste or to provide the full path to the executable of SU2_CFD.

✔ **The Viewer tool. Paraview** has been considered for viewing the final results. If you install it through the Ubuntu Software Center, make sure that you can call it

from console, if not, then you will need to manually select in the configuration section of BBIPED. This must be done in those cases where you manually download Paraview and compile it for your machine

Installing the GUI

A customized installer for the BBIPED platform is provided. This installer has been made using InstallJammer facility.

Installing the GUI : Using installer

The BBIPED GUI can be installed by means of the installer (see picture).

1. Unzip the file
2. Double click on the executable Installer
3. Follow the instructions of the graphical installer

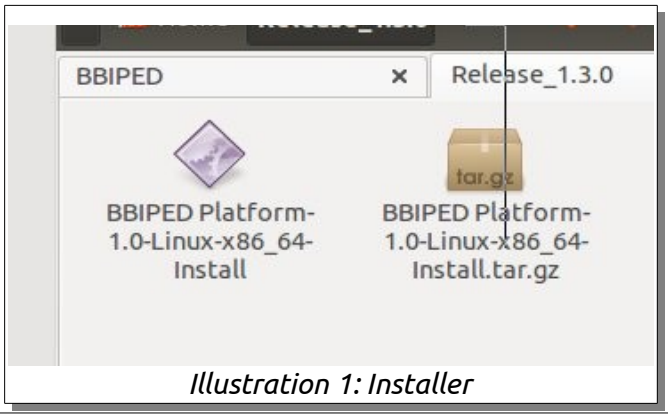

Illustration 1: Installer

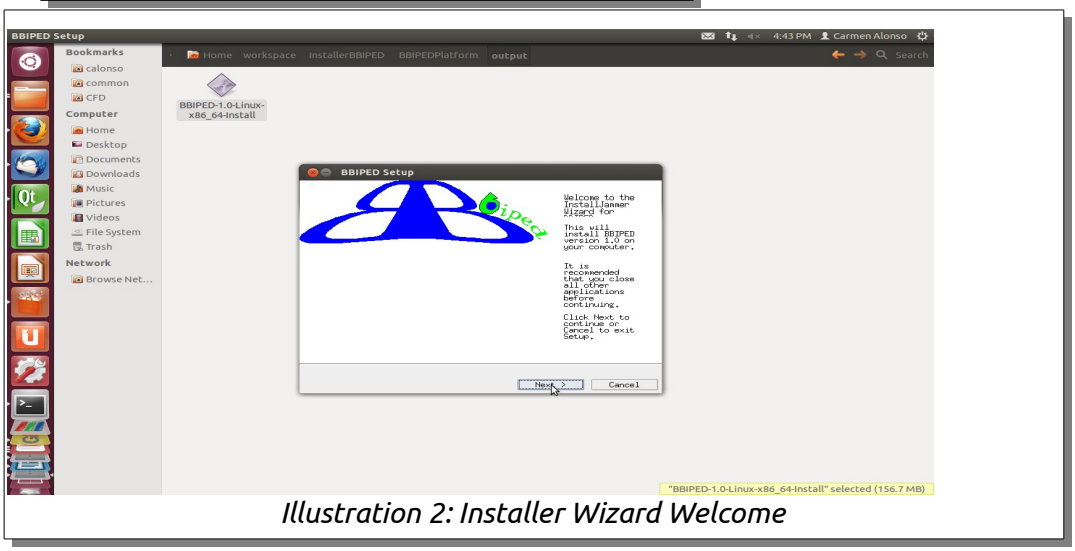

Illustration 2: Installer Wizard Welcome

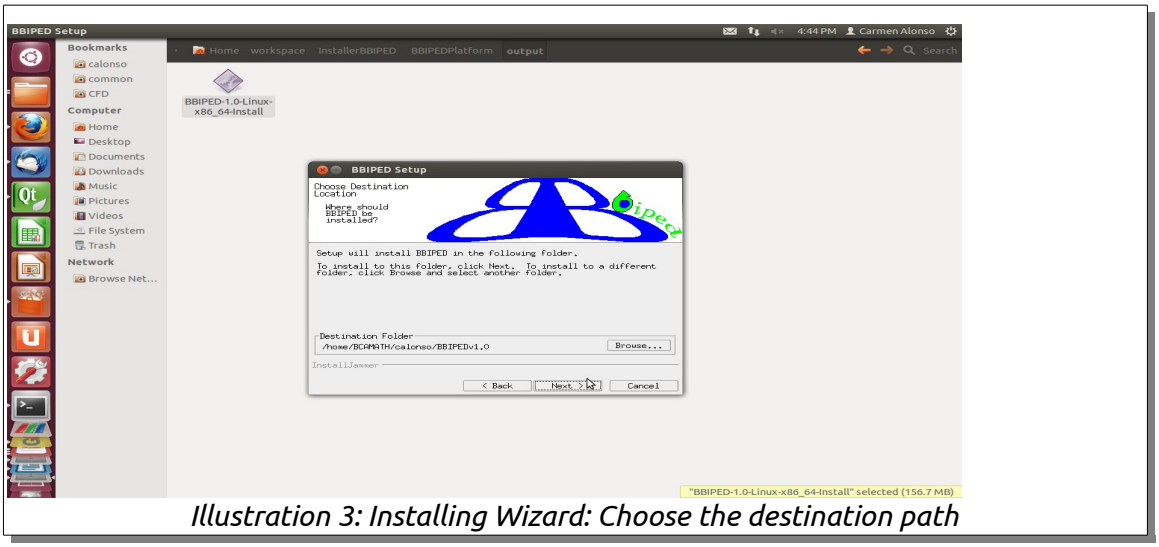

Illustration 3: Installing Wizard: Choose the destination path

Once the tool is installed, you will see the following files in the path where you installed it:

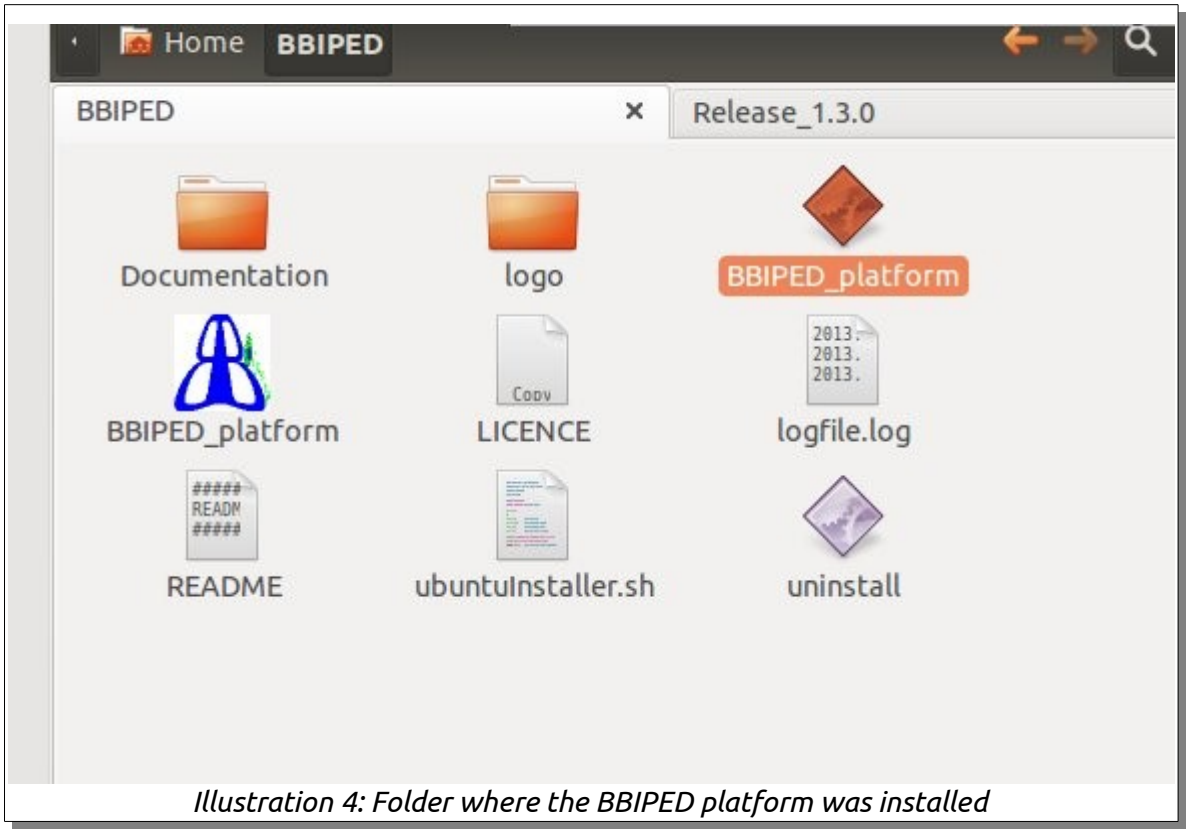

Illustration 4: Folder where the BBIPED platform was installed

To execute the tool you can click any of the BBIPED_platform elements. Uninstalling the tool is easily managed by the uninstall executable

The GUI in a nutshell

When you launch the GUI, the first thing you notice is that everything is disabled. This is because you need to create or open an project to start working.

The layout of the GUI is shown in the following image :

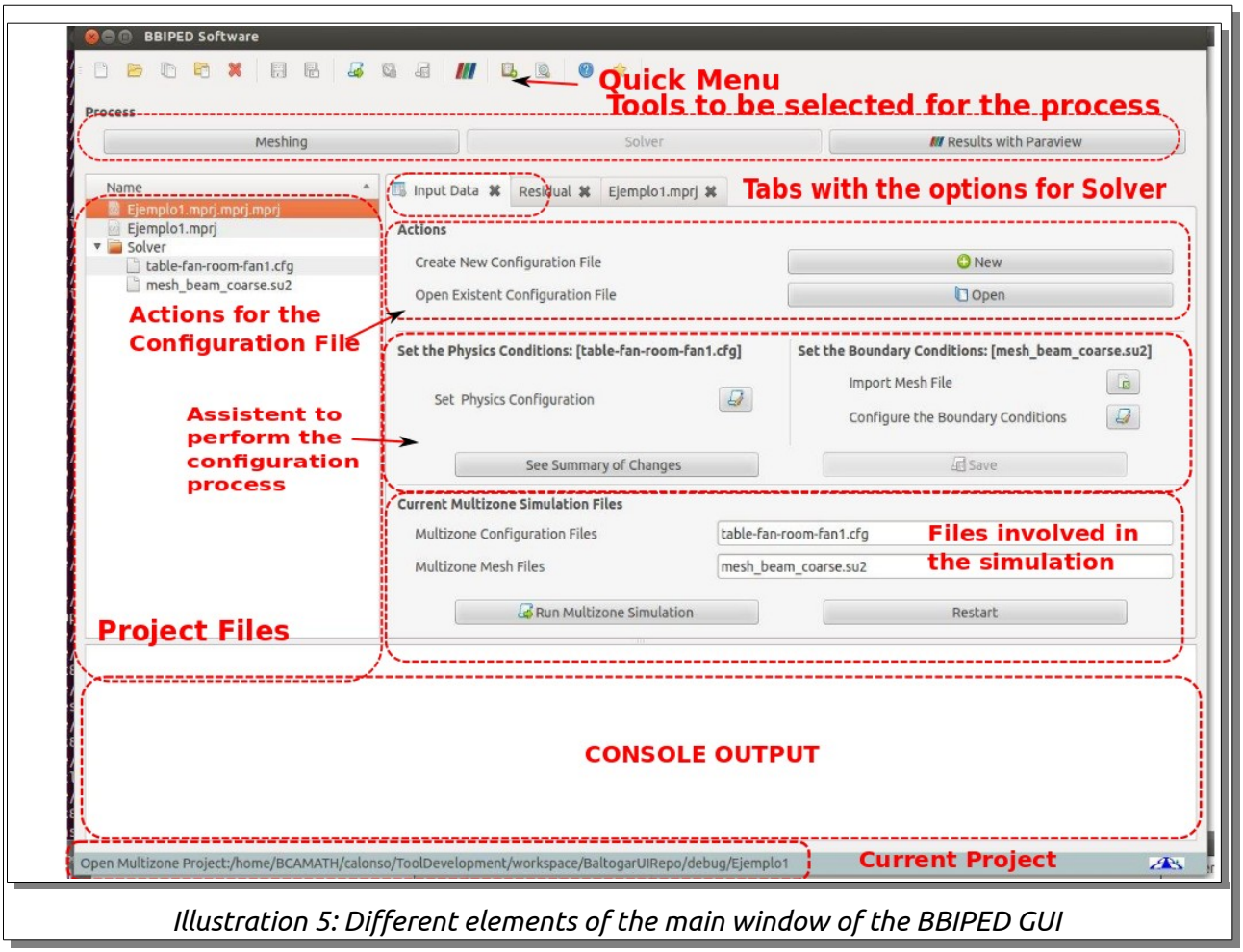

Illustration 5: Different elements of the main window of the BBIPED GUI

There is a full menu, that appears only when the whole tool is maximized. But, there is also a quick menu to access the main functionality. There is an area where the project tree is shown and where all the project files are shown. A Console area shows the output when the running process is made. Moreover, The tabs for the Solver functionality are enabled with different elements to help the user configuration. Please, see the following sections for a detailed explanation.

Setting up the Solver Engine

This version allows to use different user predefined Tools. You can go through the menu Edit → Preferences

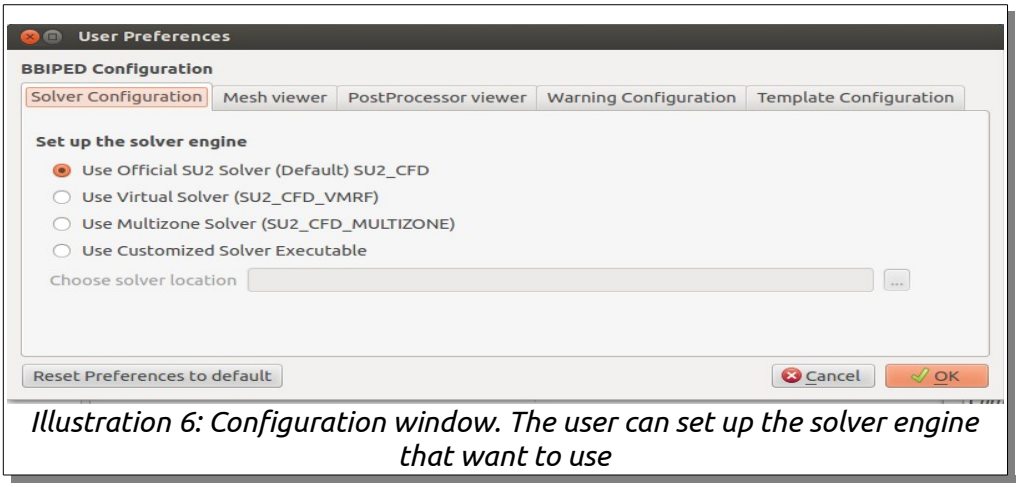

Illustration 6: Configuration window. The user can set up the solver engine that want to use

- ✔ **Solver Configuration:** The user can select their own Solver Engines. This solve engines must be compatible or based on the SU2 structure for variable definition.
- ✔ **Mesh Viewer:** By default, BBIPED calls Salome Tool as a mesh viewer. The user can select other mesh viewer tools
- ✔ **PostProcessor viewer:** By default, BBIPED calls Paraview tool. The user can select other postprocessor viewer tools
- ✔ **Warning Configuration:** The different warning windows that appear in the BBIPED platform can be enabled or disabled in this tab
- ✔ **Template Configuration:** A user can define its own template for the configuration file, in order to show only those variables that really needs. To use the template, you must select it in this tab.

Project Details

The project details, this is which files are belonging to the current project can be also checked in a dedicated window, go to the menu

Edit → Project Settings

or through the quick icon. In this menu, you can delete the pair of configuration file + mesh file you want to remove from the current project configuration. Moreover, the user can customize the project details to automatically appear in the generated files.

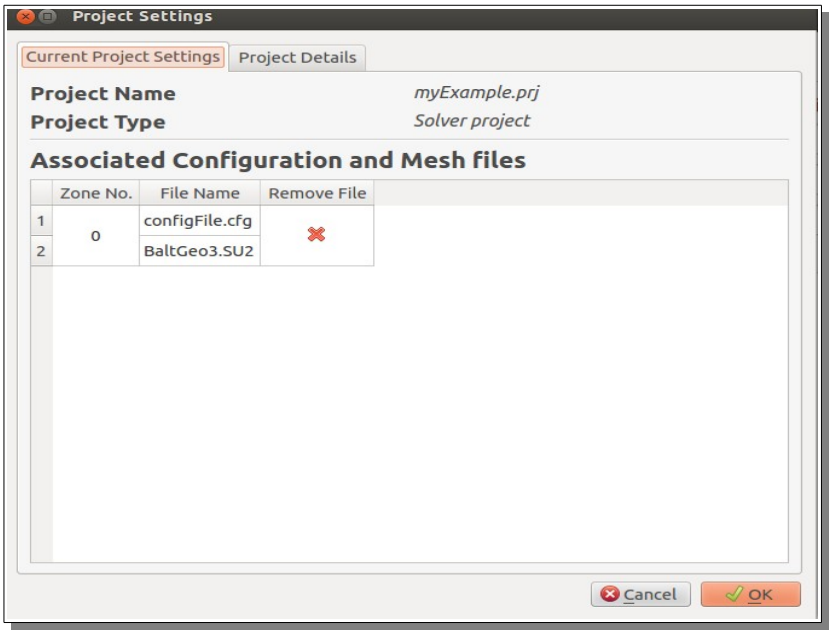

Illustration 7: Project Settings Dialog. Here, the user can see which files are currently in the project

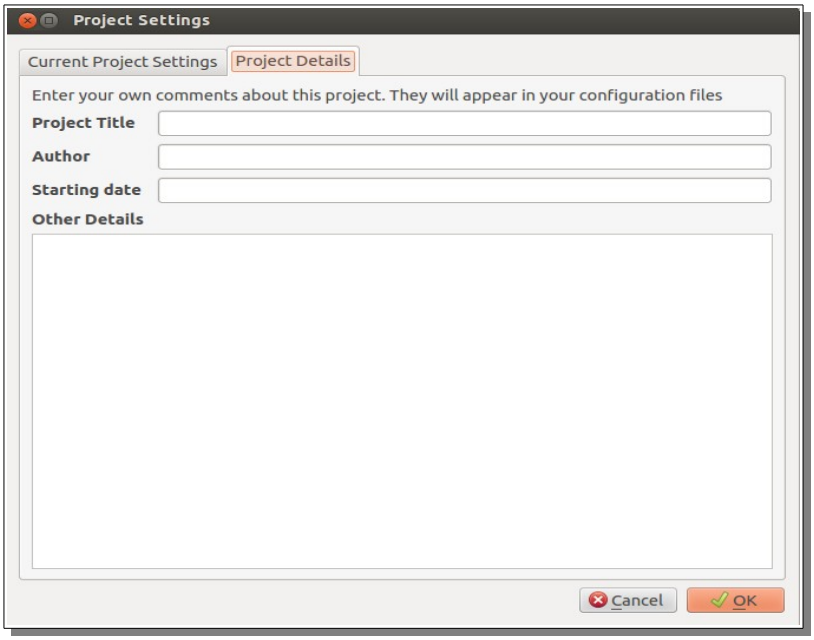

Illustration 8: Project Settings Dialog. Here, the user can customize the information of the associated project, like author, date, or project title

The Project

As a general basis, any project will have these elements :

- ✔ A project file
  - ➔ **Single Project:** one configuration file + 1 mesh file → Project with extension of .prj
  - ➔ **Multizone Project:** Several configuration files and mesh files → Project with .mprj extension
- ✔ A Solver folder that contains:
  - ➔ The configuration and mesh file(s). Once a running is made, then the correspondent files generated by the Solver will be stored in that folder.

If double click in the project file(* .prj or *.mprj) , or a configuration file (*.cfg), a new tab will be open the text version of those files.

HOWTO: Create/Open a project

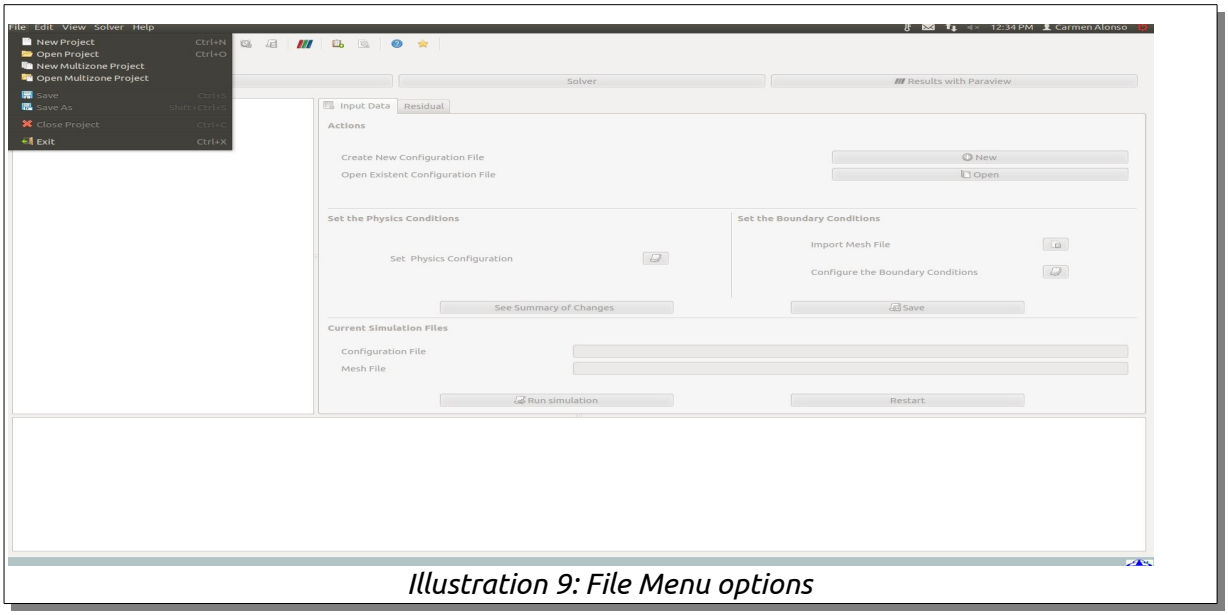

Illustration 9: File Menu options

The creation and opening of projects can be done from the main menu or the quick menu. Two types of projects are allowed: the traditional simple project or specific multizone projects. The simple project implies that only a single configuration file could be run at the same time within a project. However, **multizone projects** are designed to run several configuration files at the same time using the specific **Multizone Solver Engine.** Once you select your option, some dialogs will ask you for providing some specific information.

HOWTO: Open a simple Project

For a single project, a dialog will appear asking for the name of the project and the location, as follows :

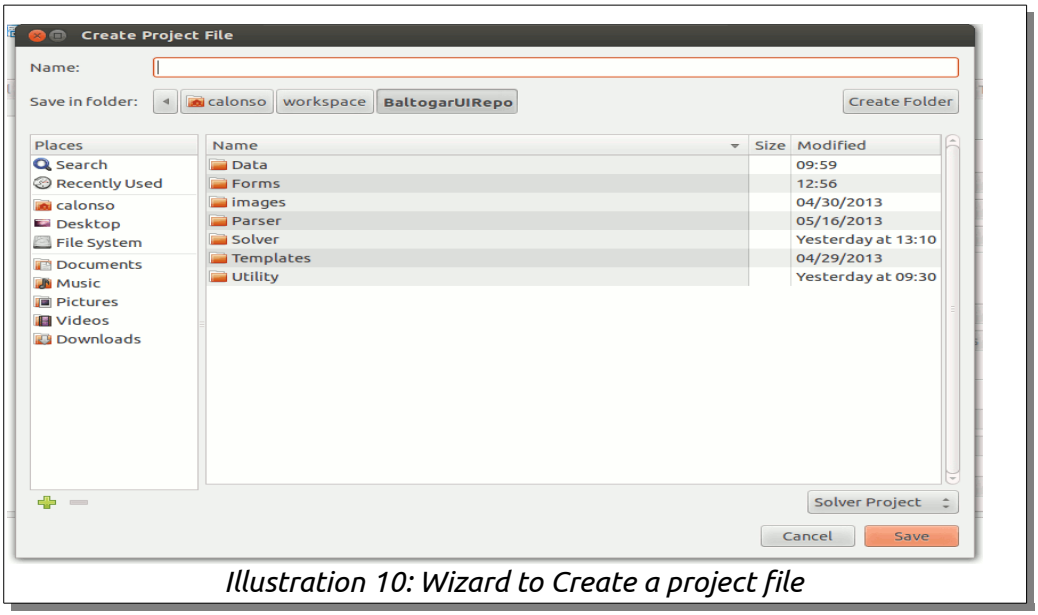

Illustration 10: Wizard to Create a project file

The creation or opening of a project, will be shown in the Project tree Area of your window, where you can see the folders of the different tools (and their contents) as well as the project file. As a convention, the name of this file is the name of the project with a .prj extension. If you open an existent project, all the Solver variables and files will be automatically loaded.

HOWTO: Create/Open a multizone Project

A multizone project is a project where several configuration files will be run using the specific Multizone Solver Engine (by default SU2_CFD_MULTIZONE). So, the creation of this type of multizone project is slightly different from the simple one. The Creation of a New Multizone Project dialog looks like:

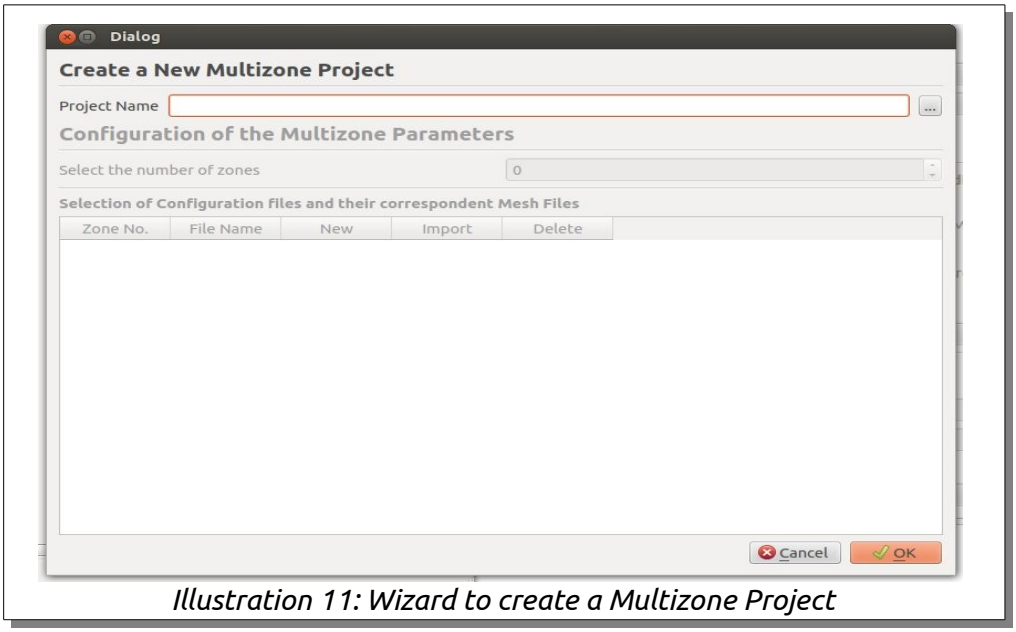

Illustration 11: Wizard to create a Multizone Project

The user needs to provide a project name and location trough the tool button. Then, a number of zones must be selected. Per each zone a configuration file + a mesh file must be selected, under the table files that are created in the table below the selection number field. See an example below:

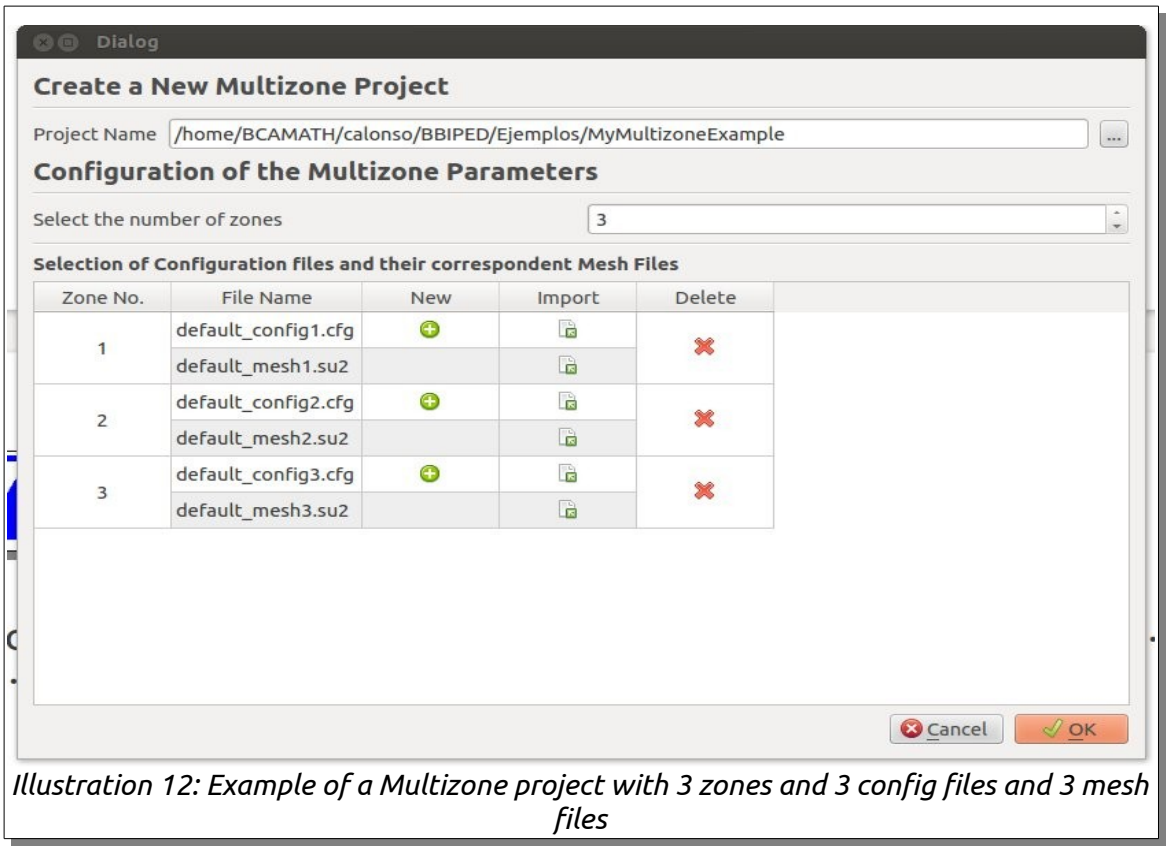

Illustration 12: Example of a Multizone project with 3 zones and 3 config files and 3 mesh files

In this example, for each configuration zone a new row with default values has been created. Each zone must need to define a configuration file and its correspondent mesh file. Notice that this configuration could be changed later, but at least one configuration file and mesh file must be provided.

There are 3 possibilities for each configuration file:

- to create a new one with default values,
- to import an existing one
- to delete it[1]

Existing mesh files can be only imported to the projects.

Once all the parameters are defined, then, when the user press ok button, the configuration file and the mesh files are analysed in order to look for inconsistencies among the boundary configuration from the configuration file and the mesh files. This issue is important specially when the configuration files are imported. If any inconsistency is found a new dialog will appear for each file . The user will have the opportunity of:

1 Please, notice that deleting a configuration file implies to delete the correspondent mesh file connection. It won't be deleted in their physical location.

✔ **Update (Repair the inconsistency)**, by manually fixing the values of the configuration file with the actual names of the mesh variables found in the mesh file

- You will need to select the values from the combos in the table (see figure below).

✔ To **keep** all the old value names as their are, for a manual fixing later

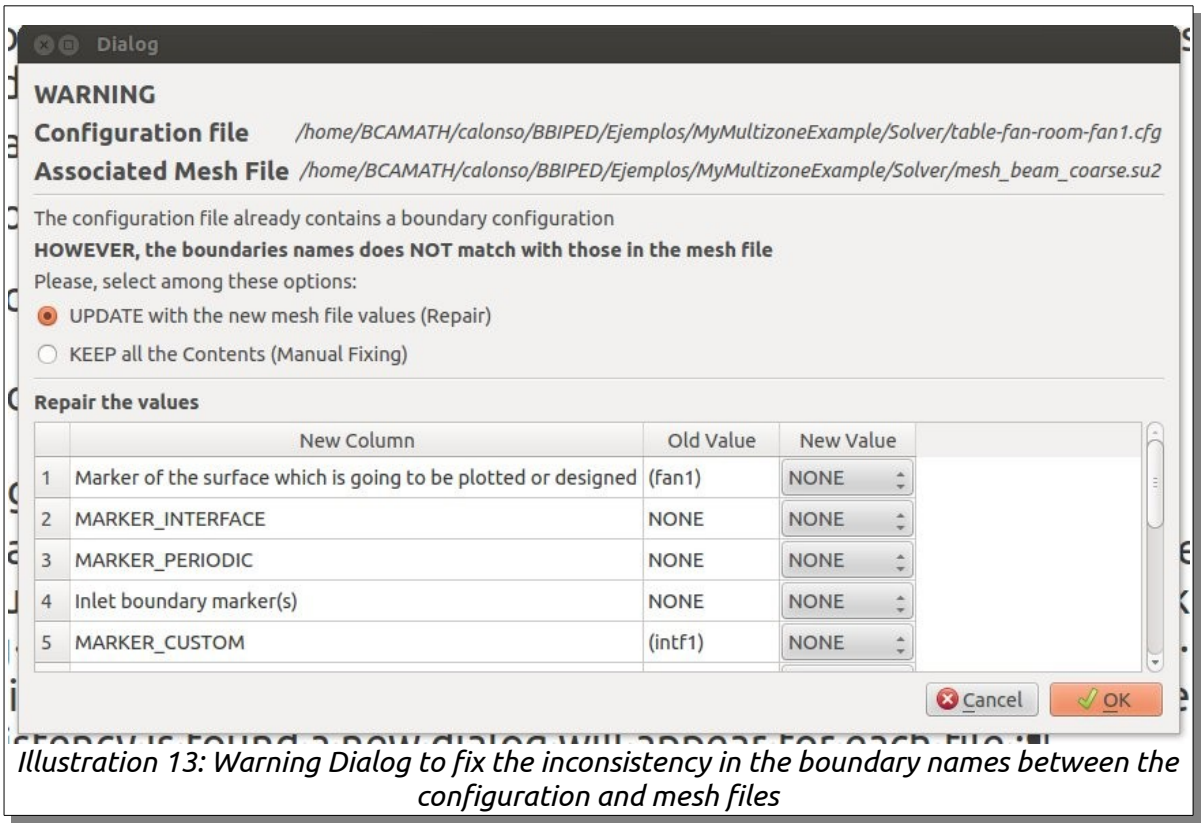

Illustration 13: Warning Dialog to fix the inconsistency in the boundary names between the configuration and mesh files

The SOLVER

The open or creation of a new project is compulsory to access to the Solver facilities. Once you have your project opened or created, the tool buttons are enabled. In the following sections, the main functionality shown in the tabs will be explained. The Solver tool button, once is clicked, enables the tabs for input data and the residual tab where you can see the graphical evolution of any of the run simulations.

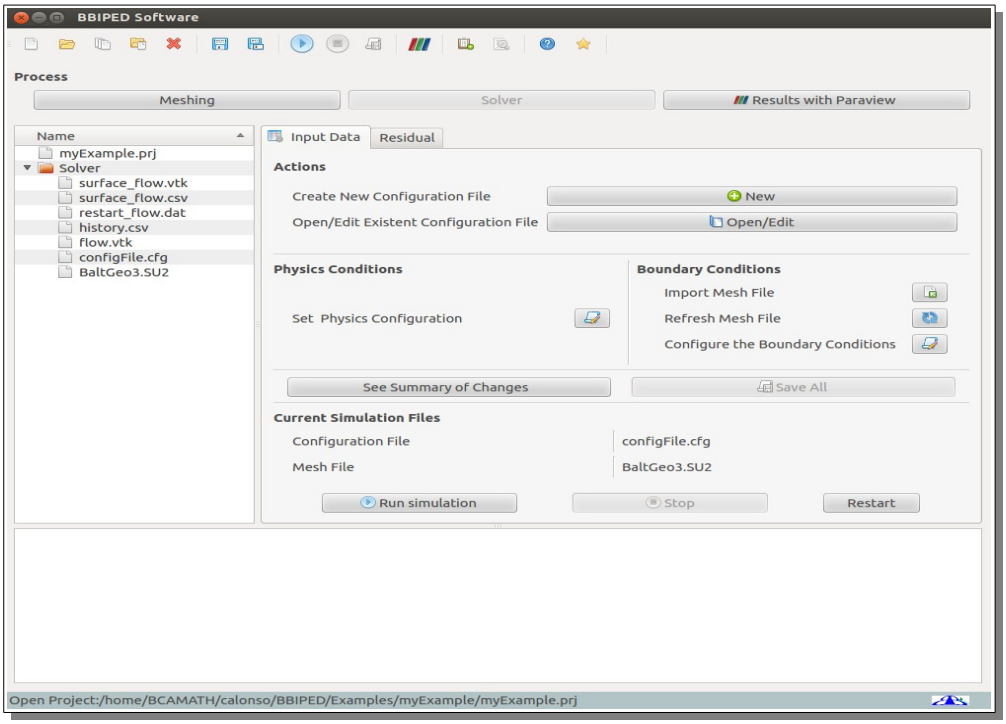

Illustration 14: Simple Solver Example (No Multizone support)

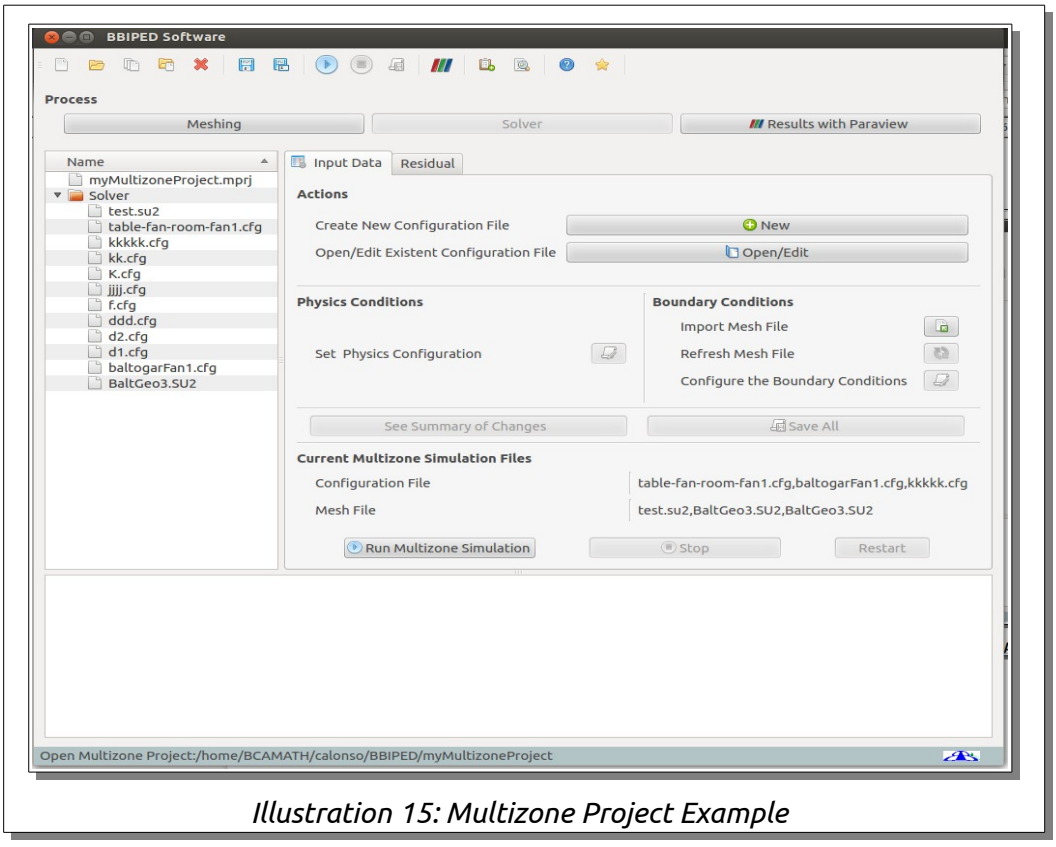

Illustration 15: Multizone Project Example

Solver Tabs

For the Solver, two main tabs have been implemented, one for the configuration and running of the simulation, and another one to provide some graphical evolution of the residual values during the simulation.

The **Input Data Tab (IDT)** is splitted up into several areas. :

1. Actions
2. Change Variables in Configuration File
3. Save Current Changes
4. Current Simulation Files

Both the simple and multizone projects have exactly the same possibilities for the IDT. Notice that for simple Solver projects, if there is a configuration file and a mesh file, they are both automatically opened. In the case of multizone projects, the user must open the correspondent configuration file project, and if there is an associated Mesh file it will be automatically loaded.

HOWTO : Set up a configuration File (Actions)

The « **Actions »** area allows the user to :

- Create a **new** configuration file
  - A mesh file will be asked to be provided !
- **Open or Edit** an existent configuration file . This file **MUST** be in the same folder as the project, otherwise, it will be copied onto it

  For **Multizone projects**, the open of an existent configuration file it doesn't mean copying again into the folder.

HOWTO : Set up the variables, the Physics and the Boundaries

The Physics and the Boundary conditions can be set up with different assistants. In the case of the Physics, the user can choose among a basic and an advanced view.

- The **basic** Configuration view, shows the basic elements for **NON-EXPERT** users
- The **Advanced** Configuration View shows all the variables that can be currently configured. You **MUST** be an expert
- A **personalised** configuration View will show the variables selected by the user in a template. A Template file must be defined first, and loaded, otherwise this option won't be available.

There are the possibility to search for specific variables. The search function looks within the variable names.

On the left side, there are the main groups. For each group, a set of configurable variables appears on the right side with the allowed options for each one. Customized help for each variable can be obtained clicking on the help icon in each row.

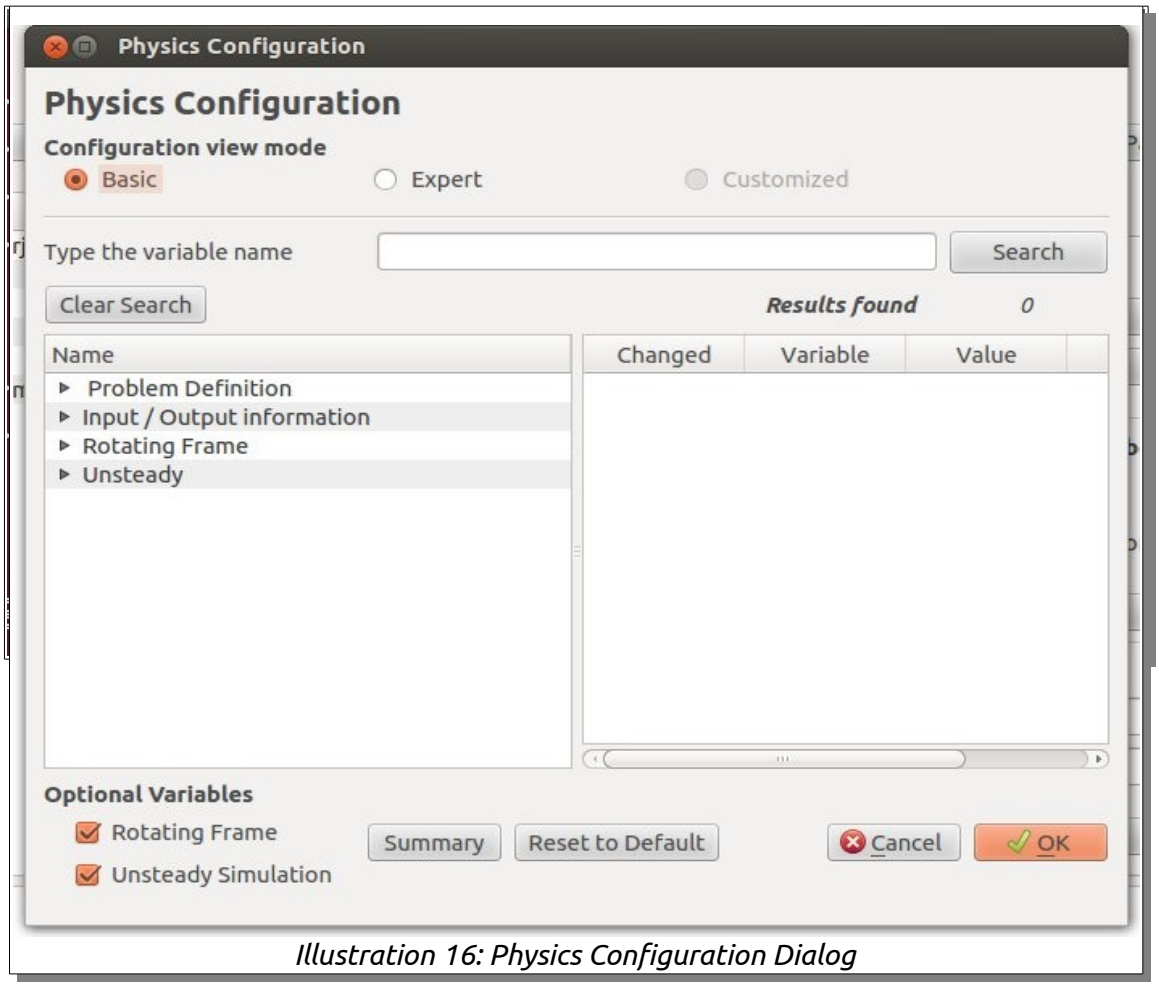

Illustration 16: Physics Configuration Dialog

The **Rotating** and **Unsteady** variables can be selected on the bottom left side, and their correspondent menus will appear

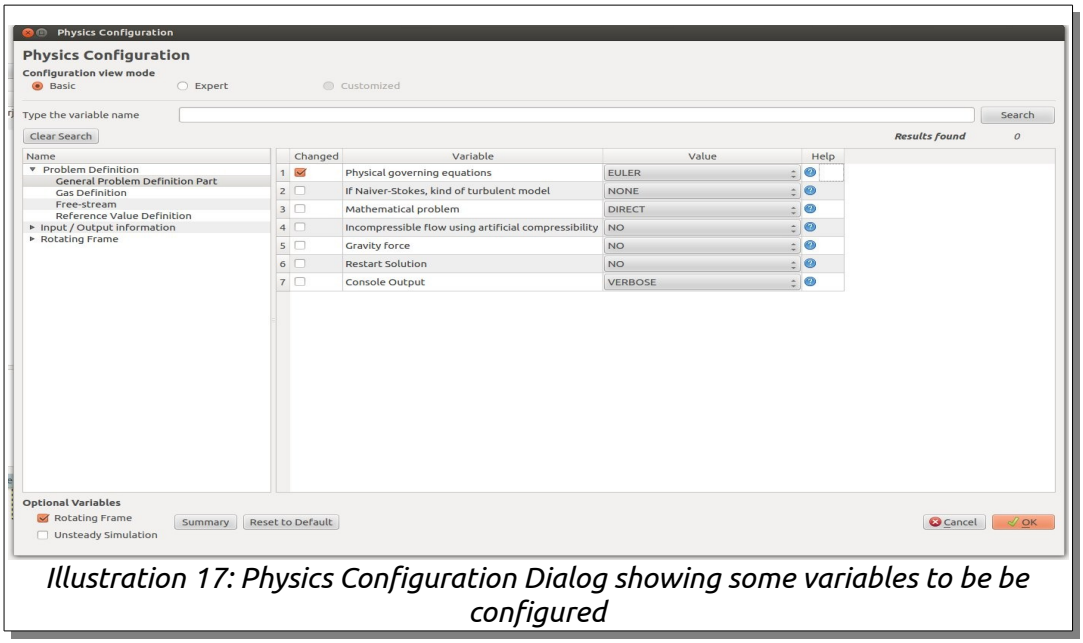

Illustration 17: Physics Configuration Dialog showing some variables to be be configured

Once you select a new value or modify any value, a check will appear in the table. In any

case, the **Summary** button will show you the changes in a quick tabular format. Quick changes can be made and saved prior closing the **Summary button**.

In case you need to reset all the values to default, the button « RESET TO DEFAULT » will reset all the variables of the configuration file to default. Please, notice that this button resets **ALL** the variables, not only those appearing at that precise moment.

The **Advanced View** provides more options, regarding the Numerics behind the Solver configuration.

In any case, you need to press **Save** or **Save As** buttons to store your changes into the correspondent file, before running the simulation

HOWTO: The Boundary Configuration Menu

The **boundary conditions** are set up according with a **specific mesh file.** So, for the configuration of the boundary conditions, the user must import the correspondent mesh file (in this case under the SU2 format). Then, the program will detect automatically the boundary names and the user will be able to set up the correspondent values in the correspondent window. See an example below :

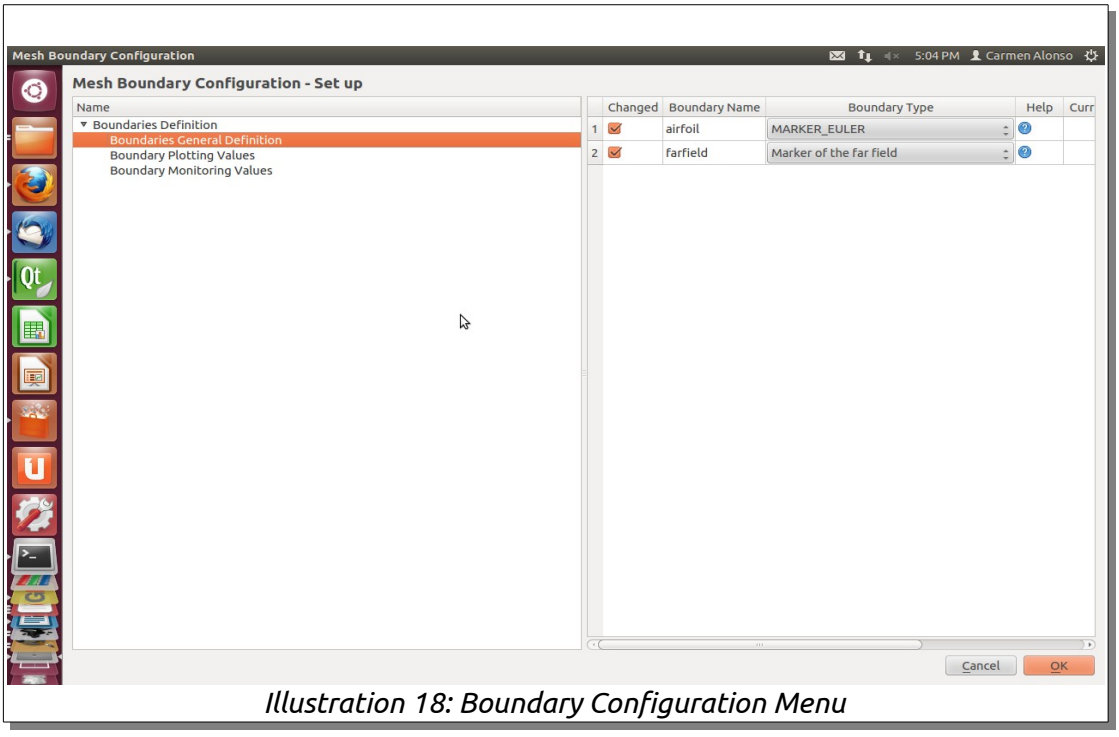

Illustration 18: Boundary Configuration Menu

HOWTO:Comparing different configuration files in multizone projects

If you want to see the different values of your variables among the configuration files in a multizone project you can use the following tool from the menu (Solver → Comparison tool). Any modification in the values you made will be added to the actual files.

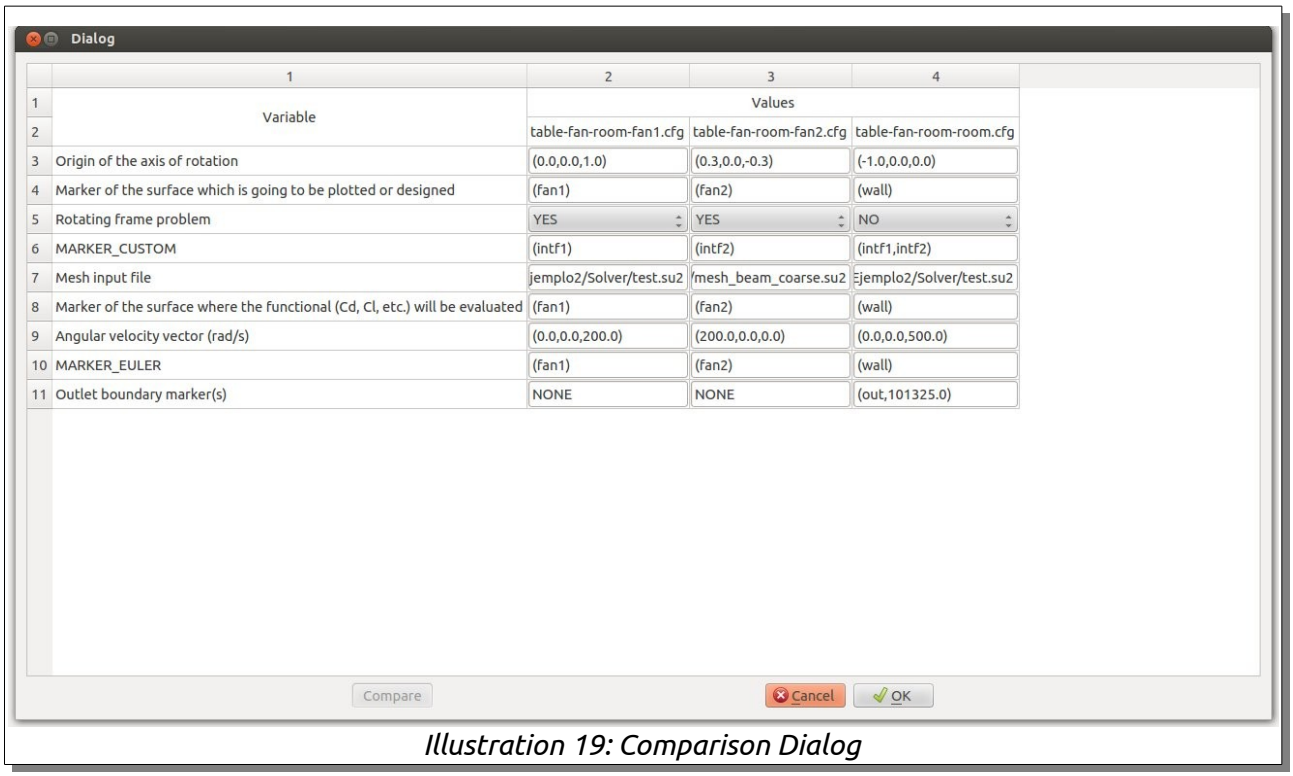

Illustration 19: Comparison Dialog

HOWTO: Running a Simulation

Once the configuration file has been saved, the simulation can starts. The user must click on the run button. If everything is ok, the Solver output can be seen into the console. The current simulation can be stop at any moment (in the main menu, or the quick menu element). Once it has been stopped, this simulation could be initialized from scratch (**Run** button) or restart from a previous simulation (**RESTART**) but only if this option has been enabled in the configuration file. Notice that all the generated files of the simulation are stored in the project folder, under the Solver folder.

The multizone running will run all the configuration files that appear in the IDT under the current simulation files.

The Residual Tab

The residual tab shows the graphical evolution of the simulation

HOWTO: The Residual Tab plot

Int he residual tab, is it possible to see the evolution of the simulation. The legend items are checkable, allowing the user to see all, some or none of the variables. The graphical evolution image can be saved to a pdf file from the Menu:

Solver-> Save Residual Image

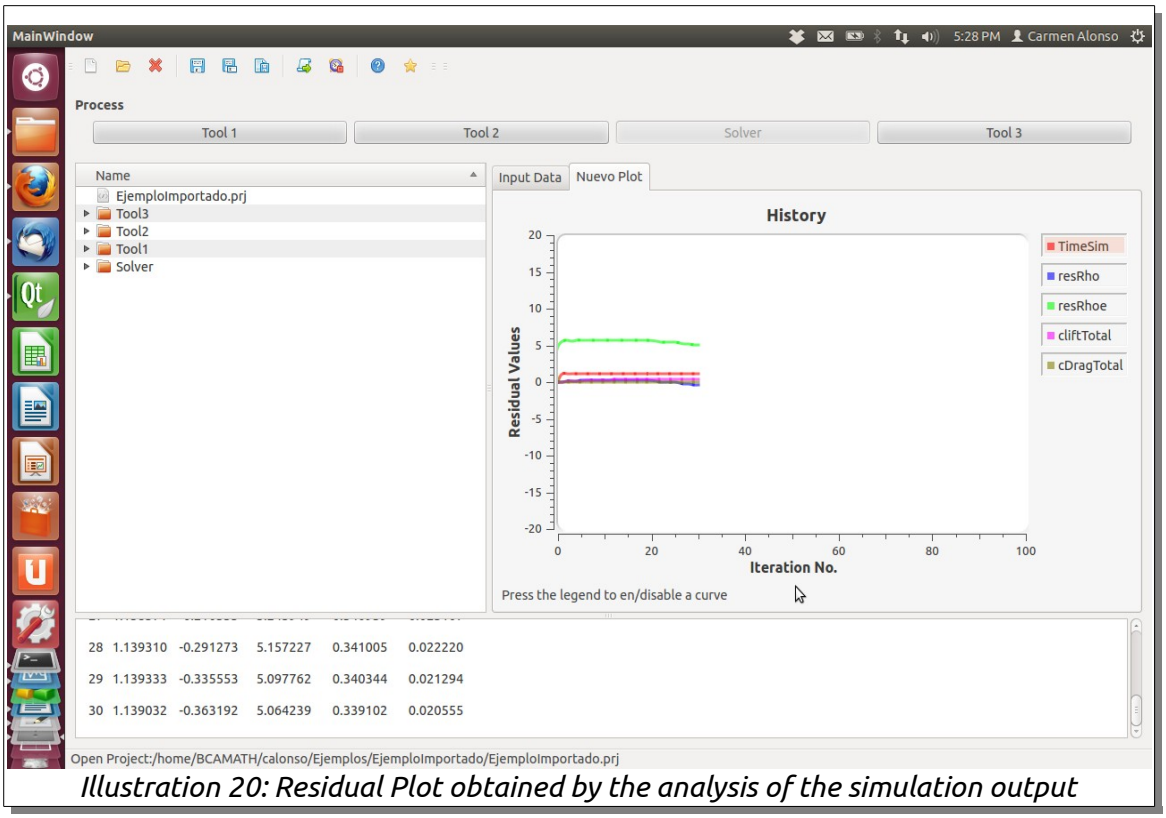

Illustration 20: Residual Plot obtained by the analysis of the simulation output

Creating our own Template

BBIPED allows the user to create a customized template, selecting only those variables to be shown in the **Physics configuration Menu.** Notice that the value associated to those variables that are not shown will be set to their default values.

The template can be created from scratch through the menu:

Solver → Create Configuration Template

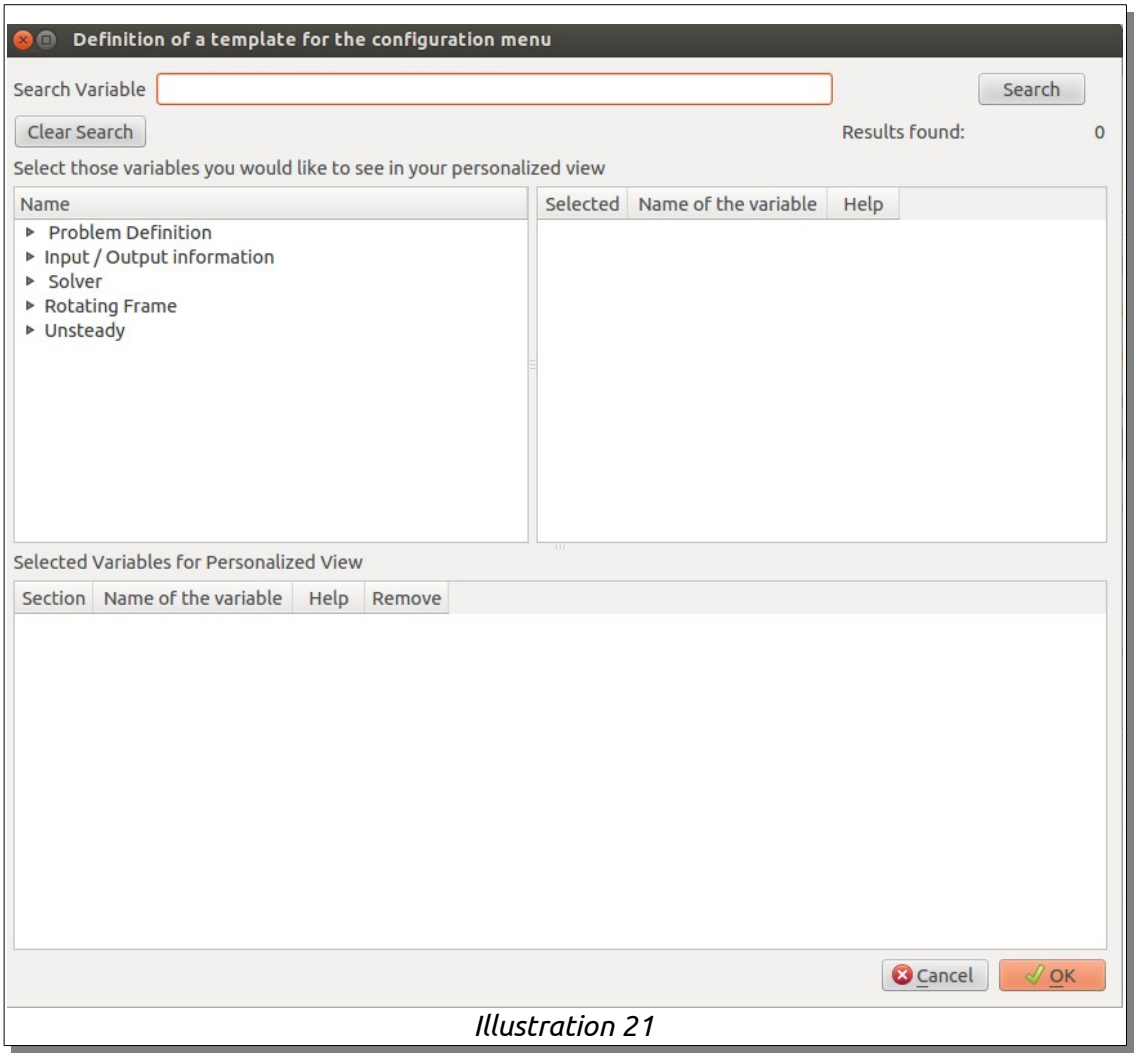

Illustration 21

Once the template has been saved, it is needed to be loaded in order to be used. It can be loaded through the Menu:

Solver → Load Configuration Template

or through the Preferences Menu

Edit → Preferences

where you can set the Template in the Template Tab

Any Template file can be edited at any time and updated according user needs, use the menu

Solver → Edit Configuration Template

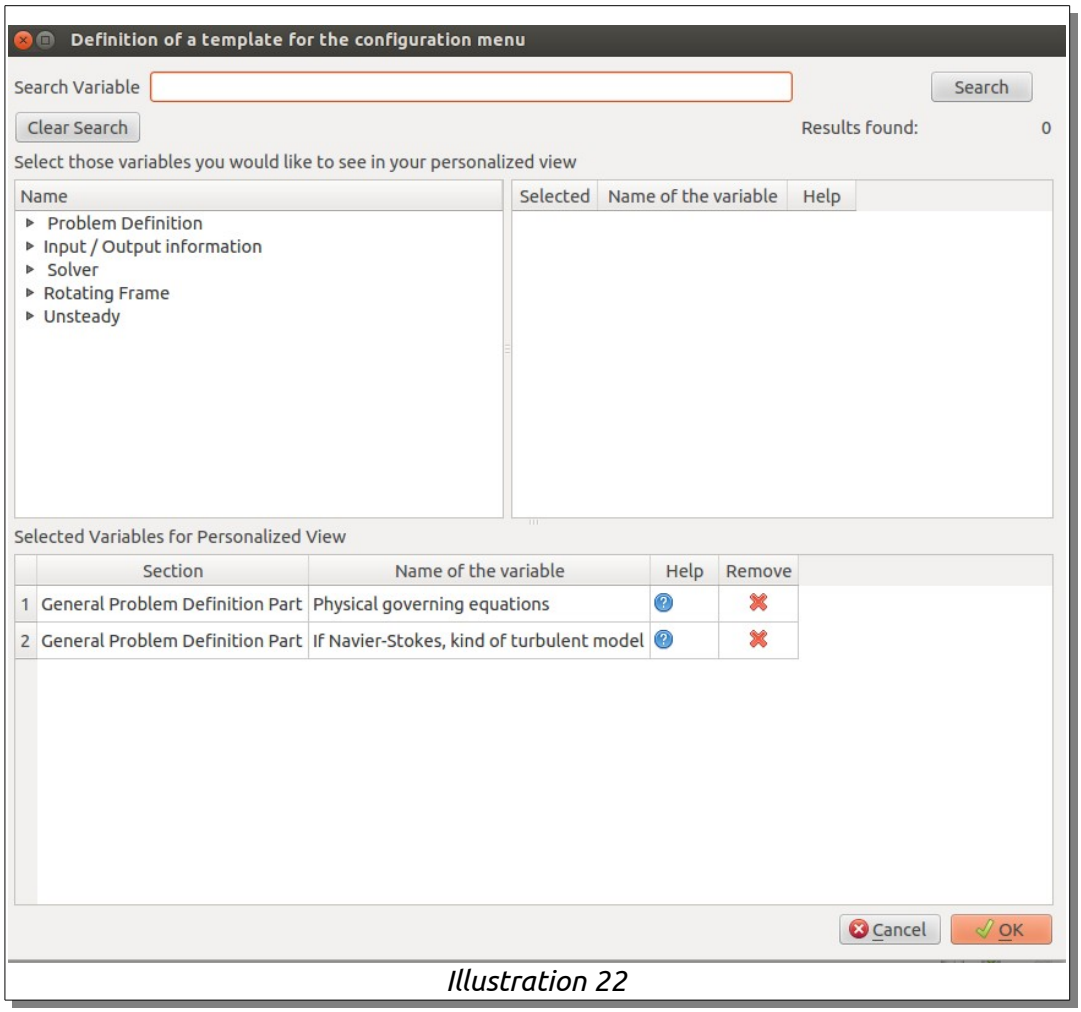

Illustration 22

BBIPED Platform Documentation

The BBIPED platform is an open source platform. Some documentation for users and developers are available.

The developers guide

A developers guide can be directly accessed from the GUI Help menu. The documentation has been generated by means of Doxygen and it is available in web and pdf formats.

Acknowledgements

We would like to thanks for their contribution and support to **BCAM** and **Baltogar** :

This project was co-funded by **Diputación Foral de Bizkaia**

This job has partly funded under the research project "***Development of an efficient, flexible and innovative CFD Computational Platform to optimally simulate and design industrial products and processes***" (**BFA/DFB - 6/12/TK/2012/00020**), funded by the **Department of Economic Promotion of the Biscay Foral Council**, through the funding plan for **Research Projects in Excellence Centres 2012**

This work has been developed using the following technologies:

The installer of this GUI has been made using **InstallJammer** (http://www.installjammer.com/)

QT and Qwt

For the Documentation, we used the **Doxygen** tool

The images of the menus belongs to **Fatcow Co**. (http://www.fatcow.com/free-icons)

This GUI serves as common framework to launch the following tools: