

---

# **HABBY Documentation**

***Release 1***

**Diane von Gunten, Yann Le Coarer and Fabrice Zaoui**

**May 11, 2017**



# CONTENTS

<b>1</b>	<b>How to execute HABBY</b>	<b>3</b>
<b>2</b>	<b>Main() and source code</b>	<b>5</b>
<b>3</b>	<b>Graphical interface</b>	<b>7</b>
3.1	Main_windows of HABBY . . . . .	7
3.2	Hydrological information - GUI . . . . .	13
3.3	Figure Option - GUI . . . . .	21
3.4	The Stathab model - GUI . . . . .	22
3.5	FStress model - GUI . . . . .	24
3.6	Estimhab - GUI . . . . .	26
3.7	Information Biological and Run habitat . . . . .	27
3.7.1	Biological data - Estimhab . . . . .	28
<b>4</b>	<b>Calculation of fish's habitat</b>	<b>31</b>
4.1	Hec-ras model 1D . . . . .	31
4.1.1	Notes on hec-ras outputs . . . . .	38
4.2	Hec-ras model 2D . . . . .	39
4.3	Mascaret . . . . .	42
4.4	River 2D . . . . .	47
4.5	Rubar . . . . .	48
4.6	Telemac . . . . .	53
4.7	LAMMI . . . . .	54
4.8	Load HABBY hdf5 file . . . . .	57
4.8.1	Form of the hdf5 files . . . . .	60
4.9	Velocity distribution . . . . .	60
4.10	Create a grid . . . . .	63
4.11	Estimhab -source . . . . .	71
4.12	Stathab - source . . . . .	72
4.13	FStress - source . . . . .	76
4.14	Substrate . . . . .	79
4.15	Merge the grid . . . . .	81
4.16	Biological Info . . . . .	84
4.17	Calculation habitat . . . . .	86
4.18	Create Paraview Files . . . . .	89
4.19	Function for the command lines . . . . .	90
<b>5</b>	<b>Various notes</b>	<b>91</b>
5.1	The legend of the plots are not shown. . . . .	91
5.2	Translation of HABBY . . . . .	91

5.3	Create a .exe . . . . .	92
5.4	Logging . . . . .	93
5.5	Command line . . . . .	94
5.6	Git - code management . . . . .	95
5.7	Write the documentation . . . . .	96
5.8	License of used python modules . . . . .	96
<b>6</b>	<b>Indices and tables</b>	<b>99</b>
	<b>Python Module Index</b>	<b>101</b>

HABBY is a program to estimate the habitat of fish using various hydrological models and preference curve as input.



## HOW TO EXECUTE HABBY

### To execute HABBY:

- Go to folder which contains habby.py using the command line.
- Open the command line and type python habby.py.

The python version should be 3.4. HABBY should also function with most of the python 3 distributions.

If a module is missing, it is possible to install it using pip (“pip install -m *module\_name*”). Obviously, pip needs to be installed, which should be done by default in python 3.4. If you want to be sure to have the same version of the module than originally, go to the folder zen\_file/wheel with the command line and install the missing module from there (something similar to “pip install -m *.whl*”). Not all modules are in this folder, only the ones which were difficult to install.





## **MAIN( ) AND SOURCE CODE**

The source code is separated in two folders: one folder which contain the code source for the graphical user interface (GUI) and one folder for the rest of the code source.

The dependency between the different part of the source code can be visualized in the mindmap `real_GUI.xmind` (xmind should be installed).

The main of HABBY is `habby.py`. It has the usual form for an application using PyQt5. The `main()` creates an application of `QWidget` and call the `Main_Windows` class, which we will discuss shortly. The last line closes the application.

It is also possible to call `habby` from the command line without the GUI. For this, the script called `habby_cmd.py` is used.



## GRAPHICAL INTERFACE

Here is the list of all modules contains in the src\_GUI folder.

### 3.1 Main\_windows of HABBY

in src\_GUI/Main\_Windows\_1.py

**class** src\_GUI.Main\_windows\_1.**CentralW**(*rech, path\_prj, name\_prj, lang\_bio*)

This class create the different tabs of the programm, which are then used as the central widget by the class MainWindows.

#### Parameters

- **rech** – A bollean which is True if the tabs for the “research option” are shown. False otherwise.
- **path\_prj** – A string with the path to the project xml file
- **name\_prj** – A string with the name of the project
- **lang\_bio** – A string with the word ‘English’, ‘French’ (or an other language). It is used to find the langage in which the biological info should be shown. So lang\_bio should have the same form than the attribute “langage” in xml preference file.

#### Technical comments

In the attribute list, there are a series of name which finish by “tab” such as stathab\_tab or output\_tab. Each of these names corresponds to one tab and a new name should be added to the attributes to add a new tab.

During the creation of the class, each tab is created. Then, the signals to show the figures are connected between this class and all the children classes which need it (often this are the classes used to load the hydrological data). When a class emits the signal “show\_fig”, CentralW collect this signal and show the figure, using the showfig function.

Show\_fig is mostly a “plt.show()”. To avoid problem between matplotlib and PyQt, it is however important that matplotlib use the backend “Qt5Agg” in the .py where the “plt.plot” is called. Practically, this means modifying the matplotlib import.

Showfig shows only one figure. To show all existing figures, one can call the function show\_fig2 from the menu. Show\_fig2 call the instance child\_win of the class ShowImageW to open a new Windows with all figure. However, this would only show the figure without any option for the zoom.

Then we call a function which connects all the signals from each class which need to write into the log. It is a good policy to create a “send\_log” signal for each new important class. As there are a lot of signal to connect, these connections are written in the function “connect\_signal\_log”, where the signal for a new class can be added.

When this is done, the info for the general tab (created before) is filled. If the user has opened a project in HABBY before, the name of the project and the other info related to it will be shown on the general tab. If the general tab is modified in the class `WelcomeW()`, this part of the code which fill the general tab will probably needs to be modified.

Finally, each tab is filled. The tabs have been created before, but there were empty. Now we fill each one with the adequate widget. This is the link with many of the other classes that we describe below. Indeed, many of the widget are based on more complicated classes created for example in `hydro_GUI_2.py`.

Then, we create an area under it for the log. Here HABBY will write various infos for the user. Two things to note here: a) we should show the end of the scroll area. b) The size of the area should be controlled and not be changing even if a lot of text appears. Hence, the `setSizePolicy` should be fixed.

The `write_log()` and `write_log_file()` method are explained in the section about the log.

**`add_all_tab()`**

This function add the different tab to habby (used by `init` and by `save_project`)

**`connect_signal_log()`**

connect all the signal linked to the log. This is in a function only to improve lisibility.

**`init_iu()`**

A function to initilize an instance of `CentralW`. Called by `__init__()`.

**`optfig()`**

A small function which open the output tab. It contains the different options for the figures. Output should be the 5th tab, otherwise it will not work.

**`scrollldown()`**

Move the scroll bar to the bottow if the `ScrollArea` is getting bigger

**`showfig()`**

A small function to show the last figure

**`showfig2()`**

A function to see all saved figures without possibility to zoom

**`update_hydro_hdf5_name()`**

This is a short function used to read all the hydrological data (contained in an hdf5 files) available in one project.

When these files are read, they are added to the drop-down menu on the hydrological tab an on the substrate tab. On the substrate Tab, if we have more than one hdf5 file, the first item is blank to insure that the user actively choose the hdf5 to reduce the risk of error (Otherwise the user might merge the substrate and an hydrological hdf5 without seeing that he needs to select the right hydrological hdf5).

This tasks should be in a function because an update to this list can be triggered by the loading of a new hydrological data. The class `Hydro2W()` and `substrateW()` noticed this through the signal `drop_hydro` send by the hydrological class. The signal `drop_hydro` is connected to this function is in the class `CentralW` in `MainWindows.py`.

**`write_log(text_log)`**

A function to write the different log. Please read the section of the doc on the log.

**Parameters `text_log`** – the text which should be added to the log (a string)

- if `text_log` start with # -> added it to `self.l2 (QLabel)` and the `.log` file (comments)
- if `text_log` start with restart -> added it `restart_nameproject.txt`
- if `text_log` start with WARNING -> added it to `self.l2 (QLabel)` and the `.log` file
- if `text_log` start with ERROR -> added it to `self.l2 (QLabel)` and the `.log` file

- if text\_log start with py -> added to the .log file (python command)
- if text\_log start with nothing -> just print to the QLabel
- if text\_log out from stdout -> added it to self.l2 (QLabel) and the .log file (comments)

if logon = false, do not write in log.txt

**write\_log\_file** (*text\_log*, *pathname\_logfile*)

A function to write to the .log text. Called by write\_log.

#### Parameters

- **text\_log** – the text to be written (string)
- **pathname\_logfile** – the path+name where the log is

**class** src\_GUI.Main\_windows\_1.**CreateNewProject** (*lang*, *path\_trans*, *file\_langue*)

A class which is used to help the user to create a new project

**init\_iu** ()

**save\_project**

a signal to save the project

**send\_log**

A PyQt signal used to write the log

**setfolder** ()

This function is used by the user to select the folder where the xml project file will be located.

**class** src\_GUI.Main\_windows\_1.**EmptyTab**

This class is used to fill empty tabs with something during the developement. It will not be use in the final version.

**addtext** ()

This function print a string on the command line. This is useful if you need to check if a button (or similar) is connected.

**init\_iu** ()

Used in the initialization.

**class** src\_GUI.Main\_windows\_1.**MainWindows**

The class MainWindows contains the menu and the title of all the HABBY windows. It also create all the widgets which can be called during execution

#### Technical comments and walk-through

First, we load the user setting using Qsettings: The settings by default of Qsettings are the name of the program (HABBY) and the name of the organization which develops the program (irstea). I have added three user settings (the name of the last project loaded into HABBY, the path to this project and the language used). The Qsetting are stored in the registry in Windows. Qsettings also function with Apple and Linux even if the information is stored differently

We set up the translation next. The translation of HABBY in different language is explained in more detail in the section “Translation of HABBY”. We give here the path to the data related to the translation. More precisely, we indicate here the path to the translation data and the name of the qm file containing the data related to the translation in each language. If a new qm is added for a new language, it should be added here to the list.

Now, two important attributes are defined: self.name\_prj and self.path\_prj. These attribute will be communicated to children classes. For each project, an xml file is created. This “project” file should be called name\_prj.xml and should be situated in the path indicated by self.path\_prj.

We call the central\_widget which contains the different tabs.

We create the menu of HABBY calling the function `my_menu_bar()`.

Two signal are connected, one to save the project (i.e to update the xml project file) and another to save an ESTIMHAB calculation.

We show the created widget.

**`change_name_project ()`**

This function is used to change the name of the current project. To do this, it copies the xml project with a new name and copy the file for the log with a new name

**`clear_log ()`**

Clear the log in the GUI.

**`closeEvent (event)`**

Close the program better than before (where it used to crash about 1 times in ten). It is not really clear why.

**Parameters** `event` – managed by the operating system.

**`close_project ()`**

This function close the current project without opening a new project

**`close_rech ()`**

Close the additional research menu (see `open_rech` for more information). For the moment, **ONLY** works with two research tabs. Modify this function if a different number of tab is needed.

**`do_log (save_log)`**

Save or not save the log

**Parameters** `save_log` – an int which indicates if the log should be saved or not

- 0: do not save log

- 1: save the log in the .log file and restart file

**`empty_project ()`**

This function open a new empty project

**`erase_pict ()`**

All files contained in the folder indicated by `path_im` will be deleted.

From the menu of HABBY, it is possible to ask to erase all files in the folder indicated by `path_im` (usually `figure_HABBY`). Of course, this is a bit dangerous. So the function asks the user for confirmation. However, it is practical because you do not have to go to the folder to erase all the images when there are too many of them.

**`init_ui ()`**

Used by `__init__()` to create an instance of the class `MainWindows`

**`my_menu_bar ()`**

This function creates the menu bar of HABBY.

**`new_project ()`**

This function open an empty project and guide the user to create a new project, using a new Windows of the class `CreateNewProject`

**`open_project ()`**

This function is used to open an existing habby project by selecting an xml project file. Called by `my_menu_bar()`

**open\_recent\_project (j)**

This function open a recent project of the user. The recent project are listed in the menu and can be selected by the user. When the user select a recent project to open, this function is called. Then, the name of the recent project is selected and the method save\_project() is called.

**Parameters** j – This indicates which project should be open, based on the order given in the menu

**open\_rech ()**

Open the additional research tab, which can be used to create Tab with more experimental contents.

Indeed, it is possible to show extra tab in HABBY. These supplementary tab correspond to open for researcher. The plan is that these options are less tested than other mainstream options. It is not clear yet what will be added to these options, but the tabs are already there when it will be needed.

**save\_project ()**

A function to save the xml file with the information on the project

**Technical comments**

This function saves or creates the xml file related to the projet. In this xml file, there are the path and the name to all files related to the project, notably the hdf5 files containing the hydrological data.

To find or create the xml file, we use the attribute self.path\_prj and self.name\_prj. If the path to the project directory is not found an error appears. The error is here sent though additional windows (to be sure that the user notice this problem), using the Qmessage module. The user should give the general info about the project in the general tab of HABBY and they are collected here. User option (using Qsetting) is next updated so that the user will find his project open the next time it opens HABBY.

When HABBY open, there are therefore two choice: a) This is a new project b) the project exists already. If the project is new, the xml file is created and general information is written in this file. In addition, the text file which are necessary to log the action of HABBY are created now. This part of the reason why it is not possible to run other part of HABBY (such as loading hydrological data) before a project is saved. In addition, it would create a lot of problems on where to store the data created. Hence, a project is needed before using HABBY. If the project exists already (i.e. the name and the path of the project have not been modified), the xml file is just updated to change its attributes as needed.

Interesting path are a) the biology path (named “biology” by default) which contains the biological information such as the preference curve and b) the path\_im which is the path where all figures and most outputs of HABBY is saved. If path\_im is not given, HABBY automatically create a folder called figures when the user creates a new project. The user can however change this path if he wants. It also create other similar folders to store different type of outputs. The next step is to communicate to all the children widget than the name and path of the project have changed.

This function also changes the title of the Windows to reflect the project name and it adds the saved project to the list of recent project if it is not part of the list already. Because of this the menu must updated.

Finally the log is written (see “log and HABBY in the command line).

**save\_project\_estimhab ()**

A function to save the information linked with Estimhab in an hdf5 file.

**Technical comments**

This function save the data and result from the estimhab calculation. It would look more logic if it was in the esimhab.py script, but it was easier to call it from here instead of in the child class.

This function get all estimhab input, create an hdf5 file using h5py and save the data in the hdf5. One specialty of hdf5 is that is cannot use Unicode. Hence all string have to be passed to ascii using the encode function. The size of each data should also be known.

Finally, we save the name and path of the estimhab file in the xml project file.

**save\_project\_if\_new\_project ()**

This function is used to save a project when the project is created from the other Windows CreateNewProject. It can not be in the new\_project function as the new\_project function call CreateNewProject().

**setlangue (nb\_lang)**

A function which change the language of the programme. It change the menu and the central widget. It uses the self.lang attribute which should be set to the new language before calling this function.

**Parameters** **nb\_lang** – the number representing the language (int)

- 0 is for English
- 1 for French
- n for any additionnal language

**test\_entry\_float (var\_in)**

An utility function to test if var\_in are float or not the boolean self.does\_it\_work is used to know if the functions run until the end.

**Parameters** **var\_in** – the QLineEdit which contains the data (so var\_in.text is a string)

**Returns** the tested variable var\_in

**class** src\_GUI.Main\_windows\_1.**ShowImageW (path\_prj, name\_prj)**

The widget which shows the saved images. Used only to show all the saved figure together iwhtout zoom or other options.

**Technical comments**

The ShowImageW() class is used to show all the figures created by HABBY. It is a class which can only be called from the menu (In Option/Option Image). This is not the usual way of opening a figure which is usually done by plt.show from matplotlib. This is the way to look at all figures together, which can be useful, even if zooming is not possible anymore.

To show all image, HABBY open a separate window and show the saved image in .png format. Currently, the figures shown are in .png, but other formats could be used. For this, one can change the variable self.imtype.

An important point for the ShowImageW class is where the images were saved by the functions which created them. In HABBY, all figures are saved in the same folder called “path\_im”. One “path\_im” is chosen at the start of each project. By default, it is the folder “Figure\_Habby”, but the user can modify this folder in the window created by ShowImageW(). The function for this is called “change\_folder”, also in ShowImageW(). The path\_im is written in the xml project file. The different functions which create image read this path and send the figure created to this folder. ShowImageW() reads all figure of “.png” type in the “path\_im” folder and show the most recent figure. The user can use the drop-down menu to choose to see another figure. The names of the figure are added to the drop-down menu in the function update\_namefig. The function “selectionchange” changes the figure shown based on the user action.

**change\_folder ()**

A function to change the folder where are stored the image (i.e., the path\_im)

**init\_iu ()**

Used in the initialization.

**selectionchange (i)**

A function to change the figure shown by ShowImageW() :return:

**send\_log**

A PyQt signal used to write the log

**update\_namefig ()**

This function add the different figure name to the drop-down list.



**class** `src_GUI.Main_windows_1.WelcomeW(path_prj, name_prj)`  
 The class `WelcomeW()` creates the first tab of HABBY (the tab called “General”). This tab is there to create a new project or to change the name, path, etc. of a project.

**change\_name**  
 A signal to change the name of the project for MainWindows

**init\_iu()**  
 Used in the initialization of a new instance of the class `WelcomeW()`

**new\_proj\_signal**  
 A PyQt signal used to open a new project

**open\_example()**  
 This function will be used to open a project example for HABBY, but the example is not prepared yet

**open\_proj**  
 A signal for MainWindows to open an existing project

**save\_signal**  
 A PyQt signal used to save the project

**send\_log**  
 A PyQt signal used to write the log

**setfolder()**  
 This function is used by the user to select the folder where the xml project file will be located. This is used in the case where the project exist already. A similar function is in the class `CreateNewProject()` for the case where the project is new.

## 3.2 Hydrological information - GUI

in `src_GUI/hydro_GUI_2.py`

This python module contains the class which forms the hydrological tab in HABBY. It contains the information for the graphical interface and make the link with the scripts used for the hydrological calculations.

**class** `src_GUI.hydro_GUI_2.FreeSpace`  
 Bases: `PyQt5.QtWidgets.QWidget`

Simple class with empty space, just to have only `QWidget` in the stack.

### Technical comment

The idea of this class is that the user see a free space when it opens the “Hydro” Tab instead of directly seeing one of the hydraulic model. The goal is to avoid the case where a user tries to load data before selecting the real model. For example, if a user wants to load mascaret data and that an item is selected by default in the stack of classes related to hydrology (such as `HEC-RAS1D`), it might be logical for the user to try to load masacret data using the `HEC-RAS` class. Because of the `FreeSpace` class, he actually has to select the model he wants to load.

**class** `src_GUI.hydro_GUI_2.HEC_RAS1D(path_prj, name_prj)`  
 Bases: `src_GUI.hydro_GUI_2.SubHydroW`

The class `Hec_ras 1D` is there to manage the link between the graphical interface and the functions in `src/hec_ras06.py` which loads the `hec-ras` data in 1D. The class `HEC_RAS1D` inherits from `SubHydroW()` so it have all the methods and the variables from the class `ubHydroW()`. The class `hec-ras 1D` is added to the `self.stack` of `Hydro2W()`. So the class `Hec-Ras 1D` is called when the user is on the hydrological tab and click on `hec-ras1D` as hydrological model.

**init\_iu()**

This function is called by `__init__()` during the initialization.

**Technical comment**

The `self.attributexml` variable is the name of the attribute in the xml file. To load a hec-ras file, one needs to give to HABBY one file containing the geometry data and one file containing the simulation result. The name and path to these two file are saved in the xml project file under the attribute given in the `self.attributexml` variable.

The variable `self.extension` is a list of list of the accepted file type. The first list is for the file with geometry data. The second list is the extension of the files containing the simulation results.

Using the function `self.was_model_loaded_before`, HABBY write the name of the hec-ras files which were loaded in HABBY in the same project before.

Hec-Ras is a 1.5D model and so HABBY create a 2D grid based on the 1.5D input. The user can choose the interpolation type and the number of extra profile. If the interpolation type is “interpolation by block”, the number of extra profile will always be one. See `manage_grid.py` for more information on how to create a grid.

We add a `QLineEdit` with the proposed name for the created hdf5 file. The user can modified this name if wished so.

**load\_hec\_ras\_gui()**

A function to execute the loading and saving of the HEC-ras file using `Hec_ras.py`

**Technical comments**

This function is called when the user press on the button `self.load_b`. It is the function which really calls the load function for `hec_ras`. First, it updates the xml project file. It adds the name of the new file to xml project file under the attribute indicated by `self.attributexml`. It also gets the `path_im` by reading the `path_im` in the xml project file. Then it check if the user want to create the figure or not. If `self.cb.isChecked()`, 2D figures should be created. If we want to create the 1D figure, the option `show_all_fig` should be selected in the figure option. It also manages the log as explained in the section about the log. It loads the hec-ras data as explained in the section on `hec_ras06.py` and creates the grid as explained in the `manage_grid.py` based on the interpolation type wished by the user (linear, nearest neighbor or by block). The variable `self.name_hdf5()` is taken from the GUI.

**class** `src_GUI.hydro_GUI_2.HEC_RAS2D` (*path\_prj, name\_prj*)

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

The class `hec_RAS2D` is there to manage the link between the graphical interface and the functions in `src/hec_ras2D.py` which loads the `hec_ras2D` data in 2D. It inherits from `SubHydroW()` so it have all the methods and the variables from the class `SubHydroW()`. It is very similar to `RUBAR2D` class and it has the same problem about node/cell which will need to be corrected.

**init\_iu()**

This method is used to by `__init__()` during the initialization.

**load\_hec\_2d\_gui()**

This function calls the function which load hecras 2d and save the names of file in the project file. It is similar to the function to load `rubar2D`. It open a second thread to avoid freezing the GUI.

When this function starts, it also starts a timer. Every three seconds, the timer run the function `send_data()` which is the class `SubHydroW()`. This function checks if the thread is finished and, it is finished, manage figure and errors.

**class** `src_GUI.hydro_GUI_2.HabbyHdf5` (*path\_prj, name\_prj*)

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

This class is used to load hdf5 hydrological file done by HABBY on another project. If the project was lost, it is there possible to just add a along hdf5 file to the current project without having to pass to the original hydrological files.

#### **get\_new\_hydro\_hdf5 ()**

This is a function which allows the user to select an hdf5 file containing the hydrological data from a previous project and add it to the current project. It modifies the xml project file and test that the data is in correct form by loading it. The hdf5 should have the same form than the hydrological data created by HABBY in the method save\_hdf5 of the class SubHydroW.

#### **init\_iu ()**

Used in the initialization by \_\_init\_\_()

**class** src\_GUI.hydro\_GUI\_2.**Hydro2W**(path\_prj, name\_prj)

Bases: PyQt5.QtWidgets.QWidget

The class Hydro2W is the second tab of HABBY. It is the class containing all the classes/Widgets which are used to load the hydrological data.

List of model supported by Hydro2W: files separately. However, sometime the file was not found \* Telemac (2D) \* Hec-Ras (1.5D et 2D) \* Rubar BE et 2(1D et 2D) \* Mascaret (1D) \* River2D (2D)

#### **Technical comments**

To call the different classes used to load the hydrological data, the user selects the name of the hydrological model from a QComboBox call self.mod. The method 'selection\_change' calls the class that the user chooses in self.mod. All the classes used to load the hydrological data are created when HABBY starts and are kept in a stack called self.stack. The function selection\_change() just changes the selected item of the stack based on the user choice on self.mod.

Any new hydrological model should also be added to the stack and to the list of models contained in self.mod (name of the list: self.name\_model).

In addition to the stack containing the hydrological information, hydro2W has two buttons. One button open a QMessageBox() which give information about the models, using the method "give\_info\_model". It is useful if a special type of file is needed to load the data from a model or to give extra information about one hydrological model. The text which is shown on the QMessageBox is given in one text file for each model. These text file are contained in the folder 'model\_hydro' which is in the HABBY folder. For the moment, there are models for which no text files have been prepared. The text file should have the following format:

- A short sentence with general info
- The keyword: MORE INFO
- All other infomation which are needed.

The second button allows the user to load an hdf5 file containing hydrological data from another project. As long as the hdf5 is in the right format, it does not matter from which hydrological model it was loaded from or even if this hydrological model is supported by HABBY.

#### **give\_info\_model ()**

A function to show extra information about each hydrological model. The information should be in a text file with the same name as the model in the model\_hydo folder. General info goes as the start of the text file. If the text is too long, add the keyword "MORE INFO" and add the longer text afterwards. The message box will show the supplementary information only if the user asks for detailed information.

#### **init\_iu ()**

Used in the initialization by \_\_init\_\_()

#### **selectionchange (i)**

Change the shown widget which represents each hydrological model (all widget are in a stack)

**Parameters** **i** – the number of the model (0=no model, 1=hecras1d, 2= hecras2D,...)

**send\_log**

A PyQt signal to send the log.

**class** `src_GUI.hydro_GUI_2.Mascaret (path_prj, name_prj)`

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

The class Mascaret is there to manage the link between the graphical interface and the functions in `src/mascaret.py` which loads the Mascaret data in 1D. It inherits from `SubHydroW()` so it have all the methods and the variables from the class `SubHydroW()`. It is similar to the HEC-Ras1D class (see this class for more information). However, mascaret is 1D model, so the loading of mascaret has one additional step compared to the hec-ras load: The velocity must be distributed along the profile. For this, the `load_masaret_gui` call the `self.distribute_velocity` function. In addition, it prepares the manning value which is necessary to distribute the velocity. The user has two choices to input the manning value. The easiest one is just to give a value constant for the whole river. In the second choice, the user loads a text file with a serie of lines with the following info: p, dist, n where p is the profile number (starting at zero), dist is the distance in meter along the profile and n in the manning value (see the method `load_manning_text` of the class `SubHydroW` for more information)

**init\_iu()**

Used in the initialization by `__init__()`

**load\_mascaret\_gui()**

The function is used to load the mascaret data, calling the function contained in the script `mascaret.py`. It also create a 2D grid from the 1D data and distribute the velocity. All of theses tasks are done on a second thread to avoid freezing the GUI.

**class** `src_GUI.hydro_GUI_2.River2D (path_prj, name_prj)`

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

The class River2D is there to manage the link between the graphical interface and the functions in `src/river2D.py` which loads the River2D data in 2D.

**Technical comments**

The class River2D inherits from `SubHydroW()` so it have all the methods and the variables from the class `SubHydroW()`. It is similar generally to the hec-ras2D class. However, the hydrological model River2D create one file per time step. Hence, it is necessary to have a way to load all the files automatically. Loading one file after one file would be annoying. There are four functions to manage the large number of file:

- `add_all_file`: find all files in a folder selected by the user.
- `add_file_river2D`: add just one selected file
- `Remove_all_file`: remove all selected files
- `Remove_file`: remove one selected file

None of this four functions load the data, it just add the name and path of the files to be loaded to `self.namefile` and `self.pathfile`. Generally, in HABBY, we load hydrological data in two steps: a) select the files, b) load the data. For river2D, the step b) is done by the function `load_river2d_gui()`. This function is similar to the one used by `Rubar2D`. It has the same problem about the grid which is identical for all time steps and which contains all reaches together. So a temporary correction was applied. Data in River2D is given on the nodes as in HABBY.

**add\_all\_file()**

The function finds all `.cdg` file in one directory to add there names to the list of files to be loaded

**add\_file\_river2d()**

This function is used to add one file to the list of file to be loaded. It let the user select one or more than

one file, prepare some data for it and update the QWidgetList with the name of the file contained in the variable `self.namefile`.

We can not call `show_dialog()` directly here as the user can select more than one file

#### **`add_file_to_list()`**

This function to add all file contained in `self.namefile` to the QWidgetlist. Called by `add_file_river2D` and `add_all_file`.

#### **`init_iu()`**

used by `__init__` in the initialization

#### **`load_river2d_gui()`**

This function is used to load the river 2d data. It use a second thread to avoid freezing the GUI

#### **`remove_all_file()`**

This function removes all files from the list of files to be loaded and from the QlistWidget.

#### **`remove_file()`**

This is small function to remove one or more `.cdg` file from the list of files to be loaded and from the QlistWidget.

### **Technical Comments**

The function `selectedIndexes` does not return an int but an object called `QModelIndex`. We should start removing object from the end of the list to avoid problem. However, it is not possible to sort `QModelIndex`. Hence, It is necessary to use the `row()` function to get the index as int.

```
class src_GUI.hydro_GUI_2.Rubar1D(path_prj, name_prj)
```

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

The class `Rubar1D` is there to manage the link between the graphical interface and the functions in `src/rubar.py` which loads the `Rubar1D` data in 1D. It inherits from `SubHydroW()` so it have all the methods and the variables from the class `SubHydroW()`. It is very similar to `Mascaret` class.

#### **`init_iu()`**

Used in the initialization by `__init__()`

#### **`load_rubar1d()`**

A function to execute the loading and saving the the rubar file using `rubar.py`. After loading the data, it distribute the velocity along the profiles by calling `self.distribute_velocity()` and it created the 2D grid by calling the method `self.grid_and_interpo`.

```
class src_GUI.hydro_GUI_2.Rubar2D(path_prj, name_prj)
```

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

The class `Rubar2D` is there to manage the link between the graphical interface and the functions in `src/rubar.py` which loads the `RUBAR` data in 2D. It inherits from `SubHydroW()` so it have all the methods and the variables from the class `SubHydroW()`. The form of the function is similar to `hec-ras`, but it does not have the part about the grid creation as we look here as the data created in 2D by `RUBAR`.

#### **`init_iu()`**

used by `__init__()` in the initialization.

#### **`load_rubar()`**

A function to execture the loading and saving the the rubar file using `rubar.py`. It is similar to the `load_hec_ras_gui()` function. Obviously, it calls `rubar` and not `hec_ras` this time. A small difference is that the `rubar2D` outputs are only given in one grid for all time steps and all reaches. Moreover, it is necessary to cut the grid for each time step as a function of the wetted area and maybe to separate the grid by reaches. Another problem is that the data of `Rubar2D` is given on the cells of the grid and not the nodes. So we use linear interpolation to correct for this.

A second thread is used to avoid “freezing” the GUI.

**propose\_next\_file()**

This function proposes the second RUBAR file when the first is selected. Indeed, to load rubar, we need one file with the geometry data and one file with the simulation results. If the user selects a file, this function looks if a file with the same name but with the extension of the other file type exists in the selected folder. This could be done for all hydrological models, but the function is harder to write when more than one extension is possible, so it has not been done yet.

**class** `src_GUI.hydro_GUI_2.SubHydroW(path_prj, name_prj)`  
Bases: `PyQt5.QtWidgets.QWidget`

SubHydroW is class which is the parent of the classes which can be used to open the hydrological models. This class is a bit special. It is not called directly by HABBY but by the classes which load the hydrological data and which inherits from this class. The advantage of this architecture is that all the children classes can use the methods written in SubHydroW(). Indeed, all the children classes load hydrological data and therefore they are similar and can use similar functions.

In other word, there are MainWindows() which provides the windows around the widget and Hydro2W which provide the widget for the hydrological Tab and one class by hydrological model to really load the model. The latter classes have various methods in common, so they inherit from SubHydroW, this class.

**dis\_enable\_nb\_profile()**

This function enable and disable the QLineEdit where the user gives the number of additional profile needed to create the grid and the related QLabel. If the user choose the interpolation by bloc, the QLineEdit will be disabled. If it chooses linear or nearest neighbour interpolation, it will be enabled. Careful, this function only works with 1D and 1.5D model.

**drop\_hydro**

A PyQtsignal signal for the substrate tab so it can account for the new hydrological info.

**find\_path\_hdf5()**

A function to find the path where to save the hdf5 file. Careful a simialar one is in estimhab\_GUI.py. By default, path\_hdf5 is in the project folder in the folder ‘fichier\_hdf5’.

**find\_path\_im()**

A function to find the path where to save the figures. Careful a simialar one is in estimhab\_GUI.py. By default, path\_im is in the project folder in the folder ‘figure’.

This is practical to have in a function form as it should be called repeatably (in case the project have been changed since the last start of HABBY).

**find\_path\_input()**

A function to find the path where to save the input file. Careful a simialar one is in estimhab\_GUI.py. By default, path\_input indicates the folder ‘input’ in the project folder.

**gethdf5\_name\_gui()**

This function get the name of the hdf5 file for the hydrological and write down in the QLineEdit on the GUI. It is possible to have more than one hdf5 file for a model type. For example, we could have created two hdf5 based on hec-ras output. The default here is to write the last model loaded. It is the same default behaviour than for the function was\_model\_loaded\_before(). To keep the coherence between the filename and hdf5 name, a change in this behaviour should be reflected in both function.

This function calls the function get\_hdf5\_name in the load\_hdf5.py file

**load\_manning\_text()**

This function loads the manning data in case where manning number is not simply a constant. In this case, the manning parameter is given in a .txt file. The manning parameter used by 1D model such as mascaret or Rubar BE to distribute velocity along the profiles. The format of the txt file is “p, dist, n” where p is the

profile number (start at zero), dist is the distance along the profile in meter and n is the manning value (in SI unit). One point per line so something like:

0, 150, 0.035

0, 200, 0.025

1, 120, 0.035, etc.

White space is neglected and a line starting with the character # is also neglected.

A very similar function to this ones exists in `func_for_cmd`. It is used to so the same thing but called from the `cmd`. Changes should be copied in both functions if necessary.

**read\_attribute\_xml** (*att\_here*)

A function to read the text of an attribute in the xml project file.

**Parameters** *att\_here* – the attribute name (string).

**save\_xml** (*i=0, append\_name=False*)

A function to save the loaded data in the xml file.

This function adds the name and the path of the newly chosen hydrological data to the xml project file. First, it open the xml project file (and send an error if the project is not saved, or if it cannot find the project file). Then, it opens the xml file and add the path and name of the file to this xml file. If the model data was already loaded, it adds the new name without erasing the old name IF the switch `append_name` is True. Otherwise, it erase the old name and replace it by a new name. The variable “i” has the same role than in `show_dialog`.

**Parameters**

- *i* – a int for the case where there is more than one file to load
- **append\_name** – A boolean. If True, the name found will be append to the existing name in the xml file, instead of replacing the old name by the new name.

**send\_data** ()

This function is call regularly by the methods which have a second thread (so moslty the function to load the hydrological data). To call this functin regularly, the variable `self.timer` of `QTimer` type is used. The variable `self.timer` is connected to this function in the initiation of `SubHydroW()` and so in the initiation of all class which inherits from `SubHydroW()`.

This function just wait while the thread is alive. When it has terminated, it creates the figure and the error messages.

**send\_err\_log** ()

This function sends the errors and the warnings to the logs. The stdout was redirected to `self.mystdout` before calling this function. It only sends the hundred first errors to avoid freezing the GUI. A similar function exists in `estimhab_GUI.py`. Correct both if necessary.

**send\_log**

A `PyqtSignal` to write the log.

**show\_dialog** (*i=0*)

A function to obtain the name of the file chosen by the user. This method open a dialog so that the user select a file. This file is NOT loaded here. The name and path to this file is saved in an attribute. This attribute is then used to loaded the file in other function, which are different for each children class. Based on the name of the chosen file, a name is proposed for the hdf5 file.

**Parameters** *i* – a int for the case where there is more than one file to load

**show\_fig**

A `PyQtSignal` to show the figure.

**was\_model\_loaded\_before** (*i=0, many\_file=False*)

A function to test if the model loaded before. If yes, it updates the attributes and the widgets of the hydrological model on consideration.

#### Parameters

- **i** – an int used in cases where there is more than one file to load (geometry and output for example)
- **many\_file** – A boolean. If true this function will load more than one file, separated by ‘,’. If False, it will only load the file of one model (see the comment below).

#### Technical comment

This method opens the xml project file and look in the attribute of the xml file to see if data from the hydrological model have been loaded before. If yes, the name of the data is written on the GUI of HABBY in the Widget related to the hydrological model. Now, there are often more than one data loaded. This method allows choosing what should be written. There are two different cases to be separated: a) We have loaded two different models (like two rivers modeled by HEC-RAS) b) One model type needs two data files (like HEC-RAS would need a geometry and output data). For the case a), the default is to write only the last model loaded. If this default behaviour is changed, the behaviour of `gethdf5_name_GUI` should also be changed. If we wish to write all data, the switch “many\_file” should be True. This switch is also useful for the river2D model, because this model creates one output file per time step. For the case b), the argument “i” (which is an int) allows us to choose which data type should be shown. “i” is in the order of the `self.attributexml` variable. The definition of this order is given in the definition of the class of each hydrological model.

**class** `src_GUI.hydro_GUI_2.SubstrateW` (*path\_prj, name\_prj*)

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

This is the widget used to load the substrate. It is practical to re-use some of the methods from `SubHydroW`. So this class inherits from `SubHydroW`.

#### **drop\_merge**

A pyqtSignal which signals that merged hydro data is ready. The signal is for the `bioinfo_tab` and is collected by `MainWindows1.py`.

#### **get\_att\_name** ()

A function to get the attribute name of the shapefile which contains the substrate data. It is given by the user in the GUI.

#### **get\_attribute\_from\_shp** ()

This function opens a shapefile and obtains the attribute. It then updates the GUI to reflect this and also updates the label as needed.

#### **init\_iu** ()

Used in the initialization by `__init__()`.

#### **load\_sub\_gui** (*const\_sub=False*)

This function is used to load the substrate data. The substrate data can be in two forms: a) in the form of a shp file from ArcGIS (or another GIS-program). b) in the form of a text file (x,y, substrate data line by line). Generally this function has some similarities to the functions used to load the hydrological data and it re-uses some of the methods developed for them.

It is possible to have a constant substrate if `const_sub= True`. In this case, an hdf5 is created with only the default value marked. This form of hdf5 file is then managed by the merge function.

**Parameters** **const\_sub** – If True, a constant substrate is being loaded. Usually it is set to False.



**log\_txt** (*code\_type*)

This function gives the log for the substrate in text form. this is in a function because it is used twice in the function `load_sub_gui()`

**send\_merge\_grid** ()

This function calls the function `merge_grid` in `substrate.py`. The goal is to have the substrate and hydrological data on the same grid. Hence, the hydrological grid will need to be cut to the form of the substrate grid.

This function can be slow so it call on a second thread.

**update\_sub\_hdf5\_name** ()

This function update the QComBox on substrate data which is on the substrate tab. The similiar function for hydrology is in `Main_Windows_1.py` as it is more practical to have it there to collect all the signals.

**class** `src_GUI.hydro_GUI_2.TELEMATC` (*path\_prj*, *name\_prj*)

Bases: `src_GUI.hydro_GUI_2.SubHydroW`

The class `Telemac` is there to manage the link between the graphical interface and the functions in `src/selafin_habby1.py` which loads the `Telemac` data in 2D. It inherits from `SubHydroW()` so it have all the methods and the variables from the class `SubHydroW()`. It is very similar to `RUBAR2D` class, but data from `Telemac` is on the node as in `HABBY`.

**init\_iu** ()

Used by `__init__()` during the initialization.

**load\_telemac\_gui** ()

The function which call the function which load `telemac` and save the name of files in the project file

### 3.3 Figure Option - GUI

in `src_GUI/output_fig_GUI.py`

This python module lets the user select various options to create the figures, notably the colormap or the size of the text. It is also where the user select the needed outputs.

`src_GUI.output_fig_GUI.create_default_figoption` ()

This function creates the default dictionary of option for the figure.

`src_GUI.output_fig_GUI.load_fig_option` (*path\_prj*, *name\_prj*)

This function loads the figure option saved in the xml file and create a dictionary will be given to the functions which create the figures to know the different options chosen by the user. If the options are not written, this function uses data by default which are in the fonction `create_default_fig_options()`.

#### Parameters

- **path\_prj** – the path to the xml project file
- **name\_prj** – the name to this file

**Returns** the dictionary containing the figure options

**class** `src_GUI.output_fig_GUI.outputW` (*path\_prj*, *name\_prj*)

The class which support the creation and management of the output. It is notably used to select the options to create the figures.

**init\_iu** ()

**save\_option\_fig** ()

A function which save the options for the figures in the xml project file. The options for the figures are

contained in a dictionary. The idea is to give this dictionary in argument to all the function which create figures. In the xml project file, the options for the figures are saved under the attribute "Figure\_Option".

**send\_log**

A PyQtsignal used to write the log.

## 3.4 The Stathab model - GUI

in `src_GUI/stathab_GUI.py`

**class** `src_GUI.stathab_GUI.StathabW`(*path\_prj, name\_prj*)

The class to load and manage the widget controlling the Stathab model.

**Technical comments**

The class `StathabW` makes the link between the data prepared by the user for Stathab and the Stathab model which is in the `src` folder (`stathab_c.py`) using the graphical interface. Most of the Stathab input are given in form of text file. For more info on the preparation of text files for stathab, read the document called 'stathabinfo.pdf'. To use Stathab in HABBY, all Stathab input should be in the same directory. The user select this directory (using the button "loadb") and HABBY tries to find the file it needs. All found files are added to the list called "file found". If file are missing, they are added to the "file still needed" list. The user can then select the fishes on which it wants to run stathab, then it run it by pressing on the "runb" button.

If file were loaded before by the user in the same project, `StathabW` looks for them and load them again. Here we can have two cases: a) the data was saved in hdf5 format (as it is done when a stathab run was done) and the path to this file noted in the xml project file. b) Only the name of the directory was written in the xml project file, indicated that data was loaded but not saved in hdf5 yet. HABBY manages both cases.

Next, we check in the xml project file where the folder to save the figure (`path_im`) is. In case, there are no `path_im` saved, Stathab create one folder to save the figure outputs. This should not be the usual case. Generally, `path_im` is created with the xml project file, but you cannot be sure.

There is a list of error message which are there for the case where the data which was loaded before do not exist anymore. For example, somebody erased the directory with the Stathab data in the meantime. In this case, a pop-up message open and warn the user.

An important attribute of `StathabW()` is `self.mystathab`. This is an object for the stathab class. The stathab model, which is in the form of a class and not a function, will be run on this object.

**add\_all\_fish()**

This function add the name of all known fish (the ones in `Pref.txt`) to the `QListWidget`. Careful, a similar function exists in `Fstress_GUI`. Modify both if needed.

**add\_fish()**

This function add the name of one fish species to the selected list of fish species.

**change\_riv\_type()**

This function manage the changes which needs to happens to the GUI when the user want to pass from tropical river to temperate river and vice-versa. Indeed the fish species and the input files are not the same for tropical and temperate river.

**init\_iu()****load\_from\_hdf5\_gui()**

This function calls from the GUI the `load_stathab_from_hdf5` function. In addition to call the function to load the hdf5, it also updates the GUI according to the info contained in the hdf5.

**Technical comments**

This function updates the Qlabel similarly to the function “load\_from\_txt\_gui()”. It also loads the data calling the load\_stathab\_from\_hdf5 function from the Stathab class in src. The info contains in the hdf5 file are now in the memory in various variables called self.mystathab.”something”. HABBY used them to update the GUI. First, it updates the list which contains the name of the reaches (self.list\_re.). Next, it checks that each of the variable needed exists and that they contain some data. Afterwards, HABBY looks which preference file to use. Either, it will use the default preference file (contained in HABBY/biology) or a custom preference prepared by the user. This custom preference file should be in the same folder than the hdf5 file. When the preference file was found, HABBY reads all the fish type which are described and add their name to the self.list\_f list which show the available fish to the user in the GUI. Finally it checks if all the variables were found or if some were missing

#### **load\_from\_txt\_gui ()**

The main roles of load\_from\_text\_gui() are to call the load\_function of the stathab class (which is in stathab\_c.py in the folder src) and to call the function which create an hdf5 file. However, it does some modifications to the GUI before.

#### **Technical comments**

Here is the list of the modifications done to the graphical user interface before calling the load\_function of Stathab.

First, it updates the label. Because a new directory was selected, we need to update the label containing the directory’s name. We only show the 30 last character of the directory name. In addition, we also need to update the other label. Indeed, it is possible that the data used by Stathab would be loaded from an hdf5 file. In this case, the labels on the top of the list of file are slightly modified. Here, we insure that we are in the “text” version since we will load the data from text file.

Next, it gets the name of all the reach and adds them to the list of reach name. For this, it calls a function from the stathab class (in src). Then, it looks which files are present and add them to the list which contains the reach name called self.list\_re.

Afterwards, it checks if the files needed by Stathab are here. The list of file is given in the self.end\_file\_reach list. The form of the file is always the name of the reach + one item of self.end\_file\_reach. If it does not find all files, it add the name of the files not found to self.list\_needed, so that the user can be aware of which file he needs. The exception is Pref.txt. If HABBY do not find it in the directory, it uses the default “Pref.txt”. All files (apart from Pref.txt) should be in the same directory. If the river is temperate, the files needed are not the same than if the river is in the tropic. This is accounted using the variable rivint. If rivint is zero, the river is temperate and this function looks for the file needed for temperate type (list of file contained in self.end\_file\_reach and self.name\_file\_allreach). If riverin is equal to 1 or 2, the river is tropical (list of file contained in self.end\_file\_reach\_top and biological data in the stathab folder in the biology folder (many files). If the river is temperate, all preference coeff are in one file called Pref.txt (also in the stathab folder of the biology folder).

Then, it calls a method of the Stathab class (in src) which reads the “pref.txt” file and adds the name of the fish to the GUI. If the “tropical river” option is selected, it looks which preference file are present in self.path\_bio\_stathab. The name of the tropical preference file needs to be in this form: YuniYh\_XXX.csv and YuniYh\_XX.csv for the univariate case and YbivYXXX.csv for the bivariate where XX is the three letter fish code from ONEMA and Y is whatever string. The form of the preference file is the form from the R version of stathab 2.

Next, if all files are present, it loads the data using the method written in Stathab (in the src folder). When the data is loaded, it creates an hdf5 file from this data and save the name of this new hdf5 file in the xml project file (also using a method in the stathab class).

Finally, it sends the log info as explained in the log section of the documentation

#### **reach\_selected ()**

A function which indicates which files are linked with which reach.

**Technical comment**

This is a small function which only impacts the GUI. When a Stathab model has more than one reach, the user can click on the name of the reach. When he does this, HABBY selects the first file linked with this reach and shows it in `self.list_f`. This first file is highlighted and the list is scrolled down so that the files linked with the selected reach are shown. This function manages this. It is connected with the list `self.list_re`, which is the list with the name of the reaches.

**`remove_fish()`**

This function remove the name of one fish species to the selected list of fish species.

**`run_stathab_gui()`**

This is the function which calls the function to run the Stathab model. First it read the list called `self.list_s`. This is the list with the fishes selected by the user. Then, it calls the function to run stathab and the one to create the figure if the figures were asked by the user. Finally, it writes the log.

**`select_dir()`**

This function is used to select the directory and find the files to load stathab from txt files. It calls `load_from_txt_gui()` when done.

**`select_hdf5()`**

This function allows the user to choose an hdf5 file as input from Stathab.

**Technical comment**

This function is for example useful if the user would have created an hdf5 file for a Stathab model in another project and he would like to send the same model on other fish species.

This function writes the name of the new hdf5 file in the xml project file. It also notes that the last data loaded was of hdf5 type. This is useful when HABBY is restarting because it is possible to have a directory name and the address of an hdf5 file in the part of the xml project file concerning Stathab. HABBY should know if the last file loaded was this hdf5 or the files in the directory. Finally, it calls the function to load the hdf5 called `load_from_hdf5_gui`.

**`send_err_log()`**

Send the errors and warnings to the logs. It is useful to note that the stdout was redirected to `self.mystdout`.

**`send_log`**

A PyQtsignal used to write the log.

**`show_fig`**

A PyQtsignal used to show the figures.

## 3.5 FStress model - GUI

in `src_GUI/fstress_GUI.py`

**class** `src_GUI.fstress_GUI.FstressW`(*path\_prj*, *name\_prj*)

This class provides the graphical user interface for the habby version of Fstress. The Fstress model is described in `fstress.py`. FstressW just loads the data given by the user. He/She can write this data in the GUI or loads it from files. The following files are needed:

- `listriv.txt` the list of the river (if the file does not exist, the river is called `river1`).
- `rivqwh.txt` discharge, width, height for two discharge (measured) for each rivers.
- `rivdeb.txt` max and min discharge.

FStress inherits from the class StatModUseful. This class is a “parent” class with some functions which are the same for estimhab and fstress. Hence, we can re-use these function without re.writing them (a bit like SubHydroW)

#### **add\_all\_fish()**

This function add the name of all known fish (the ones in Pref.txt) to the QListWidget. This function was copied from the one in SStathab\_GUI.py

#### **erase\_name()**

This function erases the data from the river selected by the user.

#### **init\_iu()**

This function is used to initialized an instance of the FstressW() class. It is called by \_\_init\_\_(). It is very similar to EstihabW but it is possible to get more than one river and it can load the data from folder.

#### **load\_all\_fish()**

This function find the preference file, load the preference coefficient for each invertebrate and show their name on QListWidget. it is run at the start of the program. FStress cannot be run as long as a preference file is not found.

#### **load\_data\_fstress(rind)**

This function loads the data for fstress and add it to the variable self.qrange and self.qhw for the river rind.  
:param rind: The indices of the river is self.river\_name. So it is which river should be loaded

#### **load\_hdf5\_fstress()**

This function loads an hdf5 file in the fstress format and add it to the project. This hdf5 file was not part of the project previously.

#### **load\_txt()**

In this function, the user select a qhw.txt or a listriv.txt file. This files are loaded and written on the GUI. If a listriv.txt is selected, the river in listriv.txt are loaded. If a qhw.txt is loaded and if there are more than one qhw.txt in the folder, we ask the user if all files must be loaded. The needed text files are the following:

- listriv.txt is a text file with one river name by line (not necessary)
- rivnameqhw.txt is a text file with minimum two lines which the measured discharge, height and width for 2 measurement (necessary)
- rivnamedeb.txt is a text file which has two lines. The first line is the minimum discharge and the second is the maximum discharge to be modelled. It is chosen by the user (necessary to run, but can be given by the user on the GUI)

All data should be in SI unit.

#### **modify\_name()**

This function is used to modify the name of a river. It will only be saved if FStress is run. Otherwise it is not kept in the data.

#### **runsave\_fstress()**

This function save the data related to FStress and call the model Fstress. It is the method which makes the link between the GUI and fstress.py.

#### **save\_river\_data()**

This function save the data for one river based on the data from the GUI (i.e., after modification by the user). It can be used to save the data given directly by the user or modified by him.

#### **show\_data\_one\_river()**

This function shows the qhw and the [qmin, qmax] data on the GUI for the river selected by the user. The river must have been loaded before.

#### **update\_list\_riv()**

This function is a small function to update the QCombobox which contains the river name

**was\_loaded\_before()**

This function looks in the xml project file if an hdf5 exists already. If yes, it loads this data and shows it on the GUI.

## 3.6 Estimhab - GUI

in `src_GUI/estimhab_GUI.py`

**class** `src_GUI.estimhab_GUI.EstimhabW`(*path\_prj, name\_prj*)

The Estimhab class provides the graphical interface for the version of the Estimhab model written in HABBY. The Estimhab model is described elsewhere. EstimhabW() just loads the data for Estimhab given by the user.

**change\_folder()**

A small method to change the folder which indicates where is the biological data

**init\_iu()**

This function is used to initialize an instance of the EstimhabW() class. It is called by `__init__()`.

### Technical comments and walk-through

First we looked if some data for Estimhab was saved before by a user. If yes, we will fill the GUI with the information saved before. Estimhab information is saved in hdf5 file format and the path/name of the hdf5 file is saved in the xml project file. So we open the xml project file and look if the name of an hdf5 file was saved for Estimhab. If yes, the hdf5 file is read.

The format of the hdf5 file is relatively simple. Each input data for Estimhab has its own dataset (qmes, hmes, wmes, q50, qrange, and substrate). Then, we have a list of strings which are a code for the fish species which were analyzed. All the data contained in the hdf5 file is loaded into a variable.

The different labels are written on the graphical interface. Then, two QListWidget are modified. The first list contains all the fish species on which HABBY has info (see XML Estimhab format for more info). The second list is the fish selected by the user on which Estimhab will be run. Here, we link these lists with two functions so that the user can select/deselect fish using the mouse. The function names are `add_fish()` and `remove_fish()`.

Then, we fill the first list. HABBY looks up all files of xml type in the "Path\_bio" folder (the one indicated in the xml project file under the attribute "Path\_bio"). The names are then modified so that only the name of the species appears (and not the full path). We set the layout with all the different QLineEdit where the user can write the needed data.

Estimhab model is saved using a function situated in `MainWindows_1.py` (frankly, I am not so sure why I put the save function there, but anyway). So the save button just sends a signal to MainWindows here, which saves the data.

**open\_estimhab\_hdf5()**

This function opens the hdf5 data created by estimhab

**run\_estimhab()**

A function to execute Estimhab by calling the estimhab function.

### Technical comment

This is the function making the link between the GUI and the source code proper. The source code for Estimhab is in `src/Estimhab.py`.

This function loads in memory the data given in the graphical interface and calls the Estimhab model. The data could be written by the user now or it could be data which was saved in the hdf5 file before and loaded when HABBY was open (and the init function called). We check that all necessary data is present and that

the data given makes sense (e.g., the minimum discharge should not be bigger than the maximal discharge, the data should be a float, etc.). We then remove the duplicate fish species (in case the user select one specie twice) and the Estimhab model is called. The log is then written (see the paragraph on the log for more information). Next, the figures created by Estimhab are shown. As there is only a short number of outputs for Estimhab, we create a figure in all cases (it could be changed by adding a checkbox on the GUI like in the Telemac or other hydrological class).

#### **save\_signal\_estimhab**

PyQtSignal to save the Estimhab data.

#### **class src\_GUI.estimhab\_GUI.StatModUseful**

This class is not called directly by HABBY, but it is the parent class of EstihabW and FstressW. As fstress and estimhab have a similar graphical user interface, this architecture allows to re-use some functions between the two classes, which saves a bit of coding.

#### **add\_fish()**

The function is used to select a new fish species (or invertebrate)

#### **find\_path\_hdf5\_est()**

A function to find the path where to save the hdf5 file. Careful a similar one is in hydro\_GUI\_2.py and in stathab\_c. By default, path\_hdf5 is in the project folder in the folder 'fichier\_hdf5'.

#### **find\_path\_im\_est()**

A function to find the path where to save the figures. Careful there is similar function in hydro\_GUI\_2.py. Do not mix it up

**Returns** path\_im a string which indicates the path to the folder where are save the images.

#### **find\_path\_input\_est()**

A function to find the path where to save the input file. Careful a similar one is in estimhab\_GUI.py. By default, path\_input indicates the folder 'input' in the project folder.

#### **find\_path\_output\_est()**

A function to find the path where to save the shapefile, paraview files and other future format. By default, path\_out is in the project folder in the folder 'other\_output'.

#### **find\_path\_text\_est()**

A function to find the path where to save the hdf5 file. Careful a similar one is in estimhab\_GUI.py. By default, path\_hdf5 is in the project folder in the folder 'fichier\_hdf5'.

#### **remove\_fish()**

The function is used to remove fish species (or invertebrates species)

#### **send\_err\_log()**

This function sends the errors and the warnings to the logs. The stdout was redirected to self.mystdout before calling this function. It only sends the hundred first errors to avoid freezing the GUI. A similar function exists in hydro\_GUI\_2.py. Correct both if necessary.

#### **send\_log**

PyQtSignal to write the log.

#### **show\_fig**

PyQtSignal to show the figures.

## 3.7 Information Biological and Run habitat

in src\_GUI/bio\_info\_GUI.py

This python module is where the biological info is managed and shown to the user. It is also where the user can run the habitat calculation.

**class** `src_GUI.bio_info_GUI.BioInfo` (*path\_prj, name\_prj, lang='French'*)

This class contains the tab with the biological information (the curves of preference). It inherits from `StatModUseful`. `StatModuseful` is a `QWidget`, with some practical signal (`send_log` and `show_fig`) and some functions to find `path_im` and `path_bio` (the path where to save image) and to manage lists.

**init\_iu** ()

Used in the initialization by `__init__()`

**run\_habitat\_value** ()

This function runs HABBY to get the habitat value based on the data in a “merged” hdf5 file and the chosen preference files.

**select\_fish** ()

This function select the fish which corresponds at the chosen criteria by the user. The type of criteria is given in the list `self.keys` and the criteria is given in `self.cond1`. The condition should exactly match the criteria. Sign such as `*` does not work.

**show\_hydrosignature** ()

This function make the link with function in `bio_info.py` which allows to load and plot the data related to the hydrosignature.

**show\_info\_fish** ()

This function shows the useful information concerning the selected fish on the GUI

**show\_pref** ()

This function shows the image of the preference curve of the selected xml file. For this it calls, the functions `read_pref` and `figure_pref` of `bio_info.py`. Hence, this function just makes the link between the GUI and the functions effectively doing the image.

**update\_merge\_list** ()

This function goes in the projet xml file and gets all available merged data. Usually, it is called by `Substrate()` (when finished to merge some data) or at the start of HABBY.

### 3.7.1 Biological data - Estimhab

The biological data, i.e., the preference curves of Estimhab, are saved in xml files situated in the folder given by the path written in the xml project file under the attribute `Path_bio`. By default, it is HABBY/biology. It is possible to change this folder using the GUI.

Estimhab is a statistical model, which functions using mathematical regressions. The different regressions (or preference curve) of each fish are described in an xml file whose format is given here.

Conceptually, the regressions  $R$  are of two types:

- Type 0  $R = C * Q^{m1} * \exp(m2*Q)$
- Type 1  $R = C * (1+m1*\exp(m2*Q))$

Where  $Q$  is the discharge,  $m1$  and  $m2$  are coefficients which depend on the fish type, and  $C$  is a constant which depends on the stream characteristic and the fish type.

The constant  $C$  is of the form  $C = a + \sum a_i * \ln(S_i)$  where  $a$  and  $a_i$  are coefficients which depend on the fish type.  $S_i$  are particular stream characteristics. Which characteristics should be used is a function of the fish type and is so given in the xml file. The value of  $S_i$  is a function of the stream and is calculated by the program.

In the xml file,

- Attribute `coeff_q`: Give the main coefficients of the regression ( $m1$  and  $m2$ )



- Attribute `func_q` : Give the type of regression R used. Type 0 and type 1, as described above, are known by HABBY.
- Attribute `coeff_const`: Give the coefficient used to construct the constant C (a, a1, a2, a3,...). The number of coefficient differs for each fish, but should be at least one.
- Attribute `var_const`: Give which type of stream characteristics is used. This is not the value of the particular characteristic, but only which type is used. The following list of type is accepted:
  - 0 for Q50, natural median discharge
  - 1 for H50, the height of the stream at q50
  - 2 for L50, the width of the stream at q50
  - 3 for V50, the velocity of the stream at q50
  - 4 for Re50, the discharge divided by 10 times the width at Q50
  - 5 for Fr50, the Froude number at Q50
  - 6 for Dh50, the mean substrate height divided by h50
  - 7 for Exp(Dh50). Erase the log() of this particular term of the constant



## CALCULATION OF FISH'S HABITAT

The src folder contains the python module which are not linked with the graphical user interface.

### 4.1 Hec-ras model 1D

in src/Hec\_ras06.py

This module contains the functions used to load the outputs from the hec-ras model in 1.5D.

```
src.Hec_ras06.coord_profile_non_georeferenced(data_bank_all, data_dist_all,  
                                              data_river_all, data_profile_all,  
                                              nb_pro_reach)
```

This is a function to create the coordinates of the profile in the non-georeferenced case. This function is called by open geo\_file(). Hypothesis: The profile are straight and perpendicular to the river. The last profile is at the end of the river.

#### Parameters

- **data\_bank\_all** – distance along the profile of bank station
- **data\_dist\_all** – the distance between the profile (left, center channel, right)
- **data\_river\_all** – the coordinate of the river
- **data\_profile\_all** – the (d,z) data of the profile
- **nb\_pro\_reach** – the number of profile by reach

**Returns** the coordinates of the profile

#### Technical comments

For each profile, we create an array composed of five points: Start of profile, left bank, intersection between river and profile, right bank and end of profile. The intersection with the river is directly given as input to the function. Then we find the vector perpendicular to this river and we get the four other points on the same line.

To get the distance for these four other point, we must be careful to pass from the distance given in meter and the distance in the model coordinates (scaled between [0, 1] usually). The way to go from one coordinate system to another is to use the “alpha” variable. We only need to correct distance, no problem with a system of coordinate which would not be in the same direction (as the data is given along a profile). The river passes in the middle of the right and left bank, so we can find where is left and right bank is. Because we know the total length of the profile, we can also find the beginning and end of the profile.

```
src.Hec_ras06.figure_xml(data_profile, coord_pro_old, coord_r, xy_h_all, zone_v_all, pro, path_im,  
                        fig_opt, nb_sim=0, name_profile='no_name', coord_p2=-99)
```

A function to plot the results of the loading of hec-ras data.

#### Parameters

- **data\_profile** – (list with np.array)
- **coord\_pro\_old** – (x,y) data of the profile (list with np.array)
- **coord\_r** – (x,y) data of the river (list with np.array)
- **xy\_h\_all** – (x,y, h) for the height data for each simulation (list with np.array)
- **zone\_v\_all** – (x,y, v) for the velocity data. velocity is by zone of profile. for each simulation. the (x,y) indicates the start of the zone which end with the next velocity
- **pro** – a list of int with the profile whcih should be plotted [2,3,4]
- **nb\_sim** – which simulation should be plotted. In fact, it often relates to the time step.
- **name\_profile** – a list of string with the name of the profiles
- **coord\_p2** – the data of the profile when non geo-referenced, optional
- **path\_im** – the path where the figure should be saved (string)
- **fig\_opt** – the figure options

### Technical comments

We first choose the size of the font to be written. At term, it should be given by the options.

Two main groups of figure will be done: One list of figure with the form of the profil, the water height, and the velocity for the chosen profiles and one (x,y) view of the position of each profile.

We chose the time step to be written (the variable nb\_sim here). The variable pro is a list which says which profiles are to be plotted. Hence, we get the velocity and water height for the time step and profile of interest.

To plot the velocity, we first get the distance along the profile where the water level cut the profile elevation. This is the variable xint1 and xint2. We then get the velocity data for the region under the water. We add three points for velocity at 0, xint1 and xint2. We then used the step function to plot the vceloity. Because of the added point, we will have a zero velocity from 0 to xint1, then the velocity data, then again zeros from xint2 to the end.

To plot the elevation of the profile, we plot the variable xz and we use the function fill\_between to fill in blue the region under water. This function creates a line at the water elevation and fills in blue between this line and the profile elevation. We add some titles and save the figures.

For the second type of figure (view in x,y coordinates), We first plot the river position which is saved in the coord\_r variable. Then we plot the coordinate of each profile and their names. If the name of the profile is not known, we plot the profile number. We also plot the position of each velocity data and height data (as it could be useful). If the figure gets too complicated, this can be taken away by changing the two lines which finish with height or velocity as comment. We add some titles and save the figures.

```
src.Hec_ras06.find_coord_height_velocity(coord_pro, data_profile, vel, wse, nb_sim,
                                         max_vel_dist=0)
```

This function finds the coordinates of the height/velocity. In hec-ras outputs the data are often written in the form (profile, distance along the profile, data). This function passes this type of information in the usual coordinate form.

### Parameters

- **coord\_pro** – the coordinate (x,y) of the profile. List of np.array.
- **data\_profile** – data concening the geometry of the profile, notably its elevation (x,z). List of np.array.
- **vel** – the velocity data. List of np.array.
- **wse** – the water surface elevation. List of np.array.

- **nb\_sim** – the number of simulation in case there is more than one
- **max\_vel\_dist** – the minimum number of velocity point by ten meter before a warnings appears

**Returns** for each simulation, a list of np.array representing (x,y,v) and (x,y,h)

#### Technical comments

This is a function called after having loaded the data. Hec-Ras present the data in (profil, distance along profile, data) form for the height. For the velocity, it is similar but the distance is given by a number between 0 and 1 (0 is the start of the profile, 1 is the end of the profile). This function transforms this data in the form (x,y, dist, data) using the (x,y) coordinates given in the coord\_pro variable. In other word, we have the coordinate of the profile, not of the coordinates of the height and velocity data.

First, we get the distance between all points in (x,y) system. Then, we get the length of the profile in meter or feet. It is possible to have a (x,y) coordinate system in a different unit. Hence, the length of the profil is valid for the (profile, distance along profile, data) view. We multiply the velocity distance data by this length. Hence, the distance information is now in meter or feet along the profile for water height and velocity.

There are some lines added to account for the last and first points of the profile (annoying in hec-ras). We then calculate the new coordinates. For each velocity and water height point, we find the last known point in the (x,y) coordinates. We do a vectorial addition from this point plus the vector between this point and the next multiplied by the distance from this point to the point that we tried to calculate. The variable alpha is used to pass from one coordinate system to the next one.

Careful the height is on the node and the velocity is by zone.

`src.Hec_ras06.get_rid_of_white_space(stream_str)`

This is a small function to get rid of white space at the end of name which could contain white space. Not used anymore as str.strip() functions well. But, as it was done already, we let it here.

**Parameters** **stream\_str** – the name of the string

**Returns** the same name without white space.

`src.Hec_ras06.load_xml(xml_file, path)`

This is a function used by openxml\_file and.opengml\_file to load an xml file.

#### Parameters

- **xml\_file** – the name of an xml file (string)
- **path** – the path where the xml file is (string)

**Returns** the loaded data from the XML file in the form of the root of the xml file.

`src.Hec_ras06.main()`

This is not the main() of HABBY. This function is used to test this module independently of the rest of HABBY.

`src.Hec_ras06.open_geofile(geo_file, path)`

This function opens the geometry file (.g0X) from Hecr-rad. It extracts the (x,z) from each profile and the (x,y) if georeferenced,

#### Parameters

- **geo\_file** – the name of the Hec-Ras geometry file (string)
- **path** – the path to the geo file (string)

**Returns** A list with each river profile (each profile is represented by a numpy array with the x and the altitude of each point in the profile), the coordinate of the profile (list of np.array), the coordinate of the river and the name of the reaches/ river in the file order (list of string)

### Technical comments

The geofile is a text file with contains the geographical information. Because it is written to be read by human, it is complicated to load and regular expression are needed. It is written profile by profile.

Generally, to give a position, hec-ras indicates the profile number and the distance along this profile. In addition, data can be georeferenced or not. If it is geo-referenced we have some data in an (x,y) form. Otherwise, we only have geometrical data in the form (profile, dist).

First, for each profile, we get the elevation of the points forming each profile in the form (dist, elevation). The list of elevation for each profile starts with the keyword “Sta/Elev”. The data found in the text file is in a string format. We use the function `pass_in_float_from_geo` to pass it in float. It is usually done using the function `float`. However, there are cases where there are no space between two number. However, in this case, the number of character per number is constant. In this case, we separate the number first.

Then, we get the coordinate of the river. If no coordinate are available the river is assumed to be straight. Next, we get the bank limit (even if we do not really used afterwards), and the name of the reach. It is also important to save the order in which the names of the reach are given. Indeed, we want this order to be the same in all functions, but they can be different between the geo file and the data output.

Next, we want to get the position (x,y) of each profile. If it is georeferenced, we will be able to get this position directly from the file and put it in the `data_dist_str` variable. We will then pass it to `float`. If not, we will use the function `coord_profil_non_georeferenced` to estimate the position of the profile (see below).

If the profile is not georeferenced, it is important to have the distance between two profile, so we extract the information from the geo file in all cases (georeferenced or not). The last profile of a reach does not have a distance to the next (not existing) profile. However, if a profile does not have a distance to the next profile and is not the last profile, we ignore this profile. It is usually not a problem because this profile is usually not a “real” profile, but the representation of a bridge or a culvert.

```
src.Hec_ras06.open_hec_hec_ras_and_create_grid(name_hdf5, path_hdf5, name_prj,
                                                path_prj, model_type, namefile, pathfile,
                                                interpo_choice, path_im, save_fig1d,
                                                pro_add=1, q=[], print_cmd=False,
                                                fig_opt=[])
```

This function open the hec\_ras data and creates the 2D grid from the 1.5 data. It is called by the class `HEC_RAS1D` in a second thread to not freeze the GUI.

### Parameters

- **name\_hdf5** – the name of the hdf5 to be created (string)
- **path\_hdf5** – the path to the hdf5 to be created (string)
- **model\_type** – the name of the model (hec\_ras in most case, but given as argument in case we change the form of the name)
- **name\_prj** – the name of the project (string)
- **path\_prj** – the path of the project
- **namefile** – the name of the geo file and the data file, which contains respectively geographical data and the output data (see `open_hec_ras()` for more precision) -> list of string
- **pathfile** – the absolute path to the file chosen into namefile
- **interpo\_choice** – the interpolation type (int: 0,1,2 or 3). See `grid_and_interpo()` for more details.
- **path\_im** – the path to where to save the image
- **save\_fig1d** – create and save the figure related to the loading of the data (profile and so on)

- **pro\_add** – the number of additional profile (one used for interpolation\_choice 1 and 2)
- **q** – used in the second thread
- **print\_cmd** – if True the print command is directed in the cmd, False if directed to the GUI
- **fig\_opt** – the options to crete the figure if save\_fig1d is True

**\*\* Technical comments\*\***

This function redirect the sys.stdout. The point of doing this is because this function will be call by the GUI or by the cmd. If it is called by the GUI, we want the output to be redirected to the windows for the log under HABBY. If it is called by the cmd, we want the print function to be sent to the command line. We make the switch here.

```
src.Hec_ras06.open_hecras(geo_file, res_file, path_geo, path_res, path_im, save_fig=False,
                           fig_opt=[])
```

This function will open HEC-RAS outputs, i.e. the .geo file and the outputs (either .XML, .sdf or .rep) from HEC-RAS. All arguments from this function are string.

#### Parameters

- **geo\_file** – the name of .goX (example .go3) file which is an output from hec-ras containing the profile data
- **res\_file** – the name of O0X.xml file for the name of the .sdf file or the name of the .rep file (output data)
- **path\_res** – path to the result file
- **path\_geo** – path to the geo file
- **path\_im** – the path to the folder where the images should be saved
- **save\_fig** – if True image is saved
- **fig\_opt** – the figure option is save\_fig is True

**Returns** coord\_pro (for each profile, x,y,elev, dist along the profile), vh\_pro (for each profile, dist along the profile, water height, velocity). Both variable are a list of numpy array.

#### How to obtain the input files

To obtain the xml file in HEC-RAS version 4:

- open the project in HEC-RAS.
- click on File , then export geometry and result (RAS Mapper), then OK

To obtain the sdf file in HEC-RAS version 5 which should be used if the model is georeferenced:

- click on File, then Export GIS data
- Export all reaches (select Reaches to export -. Full List -> Ok)
- Export all needed profile (select Profile to export -> Select all -> ok)

To obtain the report file .rep in HEC-RAS version which should be used if the model is NOT geo-referenced

- click on File, generate report
- Select Flow data and Geometry data in input data and, in Specific Table, select Flow distribution and Cross section Table

### Technical comments

This is function which loads the `hec_ras` inputs in 1D for the version 4 and 5 of HEC-RAS. It accepts different type of `hec-ras` output as input and calls the appropriate sub-function for each input file. The geometrical data is always given in the `geo` file (with the extension `g01`, `G01`, `g02`, `G02`, `g03`, etc.). The output data can be in an `xml` file for the `hec-ras` in the version 4, an `sdf` file for `hec-ras` in version 5 or a `.rep` file in the version 5 if the model is not georeferenced. The `xml` file is the format which has been tested the most.

First, it loads the geometrical data. Then it select the function to load the output data and loads it. Then, it transforms the loaded data in a (x,y) coordinates system. Indeed, most of the data in `hec-ras` is given by indicating a profile (which crossed the modelled river) and the distance along this profile. For HABBY, it is better to get (x,y) coordinates. Then it create figure if asked by the switch “`save_fig`”. Finally, it updates the forms of the output to be coherent with the `dist_velocity_hecras` function. This way, in HABBY, the output from `mascaret` and `rubar` after the velocity distribution have the same form than the output from `hec-ras`, which is useful afterwards to save all these data in the `hdf5` file.

`src.Hec_ras06.open_repfile` (*report\_file*, *reach\_name*, *path*, *data\_profile*, *data\_bank*)

A function to open the report file (`.rep`) from HEC-RAS. To obtain the report file, see the doc of the function `open_hecras`.

#### Parameters

- **report\_file** – a string with the name of the report file (`.rep`)
- **reach\_name** – a list of string containing the name of the reaches/ivers in the order of the `geo` file, which might not be the order of the `sdf` file.
- **path** – the path where the report file is stored (string)
- **data\_profile** – the data from each profile from the `geofile` (output from the `open_geofile` function)
- **data\_bank** – the position of the bank limit (output from the `open_geofile` function)

**Returns** velocity and the water surface elevation for each river profiles in a list of `np.array`, the number of simulation (int) and the name of the river profile (list of string)

### Technical comments and walk-through

This function is used to open output from models which were not geo-referenced in `hec-ras v5`. It cannot be used if the model was georeferenced (or at least one should make some tests before).

First, we obtain the water height. Then, we obtain the number of time step (which is called the number of simulation by `hec-ras`). To get the number of time step, we count each outputs given (one by profiles) and we divided it by the number of profile in the river. It is a bit indirect, but I did not find a simpler solution.

We get the name of each profile and reach. Then, we get the velocity data. We have in a case which is not geo-referenced. By consequence, there are only three velocities: one the left bank, one in the main river channel and one the right bank. Next we get the distance along the profile for these three velocities. Finally, we use the function `reoder_reach` for the same reason than in `open_sdf` and `open_xml`.

`src.Hec_ras06.open_sdf` (*sdf\_file*, *reach\_name*, *path*)

This is a function to load `.sdf` file from HEC-RAS v5 used if the model is georeferenced. To find how to obtain the `sdf` file, read the doc of `open_hecras`.

#### Parameters

- **sdf\_file** – the name of the `sdf` file (string)
- **reach\_name** – a list of string containing the name of the reaches/ivers in the order of the `geo` file which might not be the one of the `sdf` file. Output from `open_geofile`.
- **path** – the path where the file is stored (string)



**Returns** velocity, water height, river\_name, number of time step (nb\_sim)

#### Technical comments

To strat loading the sdf file, we open the sdf file. It is mostly a text file. Then we find velocity data and we pass this velocity data from string to float. The process is a bit similar to the one used in the function open\_geofile with a healthy dose of regular expressions. We do this again for height data.

We also extract the name of the river, reaches and profile. The number of simulation (nb\_sim) is a bit confusing for a variable name. In fact, it is the number of time step. Hec-Ras considers that one simulation is the simulation for one time step. Hence, nb\_sim is more or less nb\_timestep.

As in the xml file, we finally re-order the data as in the geo file. Indeed, it is possible to have different order between the reaches in the geo file and in the sdf file. Here, we use the function reorder\_reach for this.

`src.Hec_ras06.open_xmlfile(xml_file, reach_name, path)`

This function open the xml file from HEC-RAS v4 to get the velocity and water surface elevation. To know how to obtain this xml file, read the doc of open\_hecras.

#### Parameters

- **xml\_file** – the name of O0X.xml file from HEC-RAS. (string)
- **reach\_name** – a list of string containing the name of the reaches/rivers in the order of the geo file which might not be the one of the xml file.
- **path** – path to the xml file (string)

**Returns** velocity and the water surface elevation for each river profiles (list of np.array), the number of simulation(int) and the name of the river profile (list of string)

#### Technical comments

To load the xml file, we first call the load\_xml function. It is a function which check that the xml file is well formed and which return the “root” part for the xml. With this “root”, it is possible to load other part of the xml file using the Etree module.

Then, we load the velocity and water height data from the xml file. We also load the name of the profiles and of the reach names. Next, we pass the data into float. For each velocity of height point, we get its position along the profile (see below for format) and the value at this point.

Finally, we re-order the data as in the geo file. Indeed, it is possible to have different order between the reaches in the geo file and in the xml file. The last part of this function is there to order all the data as in the geo file. There is a function reorder\_reach which does something similar, but could not be used by the output from the xml file (it is slightly different). However the reorder\_reach function and this part of the open\_xml function is very similar.

`src.Hec_ras06.pass_in_float_from_geo(data_str, len_number)`

This is a function to pass the string data into float for open\_geofile() and open\_sdf(). It is in a function because it is possible that two number are not separated by a space in the input data.

#### Parameters

- **data\_str** – the data in a string form
- **len\_number** – the number of digit for one number (int)

**Returns** a np.array of float with 2 columns (x,y) or (x,z)

`src.Hec_ras06.reorder_reach(wse, vel, riv_name, reach_name, reach_str, stream_str, nb_sim)`

The order of the reach in HABBY is in the order given in the geo file. However, it can be given in any order in the other file. (xml, sdf, rep,...). This function re-order the reaches based on their name.

#### Parameters

- **wse** – water height data (list of np.array for each profile)
- **vel** – velocity data (list of np.array for each profile)
- **riv\_name** – the name of the profile (yeah I know it is not really logical as a name)
- **reach\_name** – the name of the reach and stream (stream,reach) in the geo file order
- **reach\_str** – the name of the reach in the analysed file order
- **stream\_str** – the name of the stream in the analysed file order
- **nb\_sim** – the number of simulation

**Returns** wse, vel, riv\_name all re-ordered

#### Technical comments

The reach name should not have white space at the end/start but can have white space into them.

`src.Hec_ras06.update_output (zone_v, coord_pro_old, data_profile, xy_h, nb_pro_reach_old)`

This function updates the form of the output so it is coherent with mascaret and rubar after the lateral distribution of velocity for these two models. There are three important changes. First, coord\_pro contains dist along the profile (x) and height in addition to the coordinates. Secondly, vh\_pro contains only height if height is above or equal to zero. Thirdly, a point is created at the water limits and v and height are given at the same points. nb\_pro\_reach is also modified as in mascaret. We want to modify it so it start by zero and is additive, i.e., that it gives total number of profile before, not the number of profile by reach.

#### Parameters

- **zone\_v** – (x,y, dist along profile, v) for each time step. However, the zone are the one from the models. They are different than the one from xy\_h, which is unpractical for the rest of HABBY.
- **coord\_pro\_old** – the (x,y) coordinate for the profile
- **data\_profile** – the distance along the profile and height of each profile
- **xy\_h** – the water height
- **nb\_pro\_reach\_old** – the number of the profile by reach in the old form.

**Returns** coord\_pro, vh\_pro, nb\_pro\_reach

[doc to be finished]

### 4.1.1 Notes on hec-ras outputs

- Data in HEC-RAS can be geo-referenced or not georeferenced. It is advised to geo-reference all model in HEC-RAS. If the model is not geo-referenced, the function makes some assumptions to load the data: 1) the river profile are straight and perpendicular to the river. 2) the last profile is at the end of the river.
- To geo-reference a model in hec\_ras: In the “geometric data” window, GIS tool, GIS Cut Line, Accept Display location, choose all profile
- Numerical data are sometime not separated (0.4556 0.3453233.454 05.343). In this case, the number of digit is assumed to be 8 for the profile and 16 for the river coordinates.
- Part of the profile can be vertical: The function also functions in this case.
- There is sometimes more than one reach in the modelled river and these reaches sometimes form loops: The function load each reach one after the other.
- The river reaches are sometimes not in the same order in the xml file and in the .goX file. The order of the .goX is used by the function. Reach are automatically re-ordered.

- If the river is straight, the coordinates of the river are given differently. The function try to load the river in the “straight” style if the usual style fail.
- The .goX file includes data on bridges and culvert. Currently, the function neglects this information.
- Sometimes distances between profiles are not given in the .goX file. The function neglects the distance data of this profile as long as it is not the last profile.
- The velocity data for the end and the beginning of the river profile is indicated by a large number (example 1.23e35 or -1.234e36). The function considers that velocity info is situated at the start of the profile if  $x > 1e30$  and at the end of the profile if  $x > 1e30$ .
- There are two concepts called “profile” in HEC-RAS: The river profiles and the simulation profiles. The river profiles are the geometry perpendicular to the river and the simulation profile are the different simulations.
- Data in many of the example cases of HEC-RAS are in foot and miles. 1 miles = 5280 foot, and not 1000 foot.

## 4.2 Hec-ras model 2D

in src/Hec\_ras2D.py

This module contains the functions used to load the outputs from the hec-ras model in 2D.

```
src.hec_ras2D.figure_hec_ras2d(v_all, h_all, elev_all, coord_p_all, coord_c_all, ikle_all,
                                path_im, time_step=[0], flow_area=[0], max_point=-99)
```

This is a function to plot figure of the output from hec-ras 2D. This function is only used to debug, not directly by HABBY.

### Parameters

- **v\_all** – a list of np array representing the velocity at the center of the cells
- **h\_all** – a list of np array representing the water depth at the center of the cells
- **elev\_all** – a list of np array representing the minimum elevation of each cells
- **coord\_p\_all** – a list of np array representing the coordinates of the points of the grid
- **coord\_c\_all** – a list of np array representing the coordinates of the centers of the grid
- **ikle\_all** – a list of np array representing the connectivity table one array by flow area
- **time\_step** – which time\_step should be plotted (default, the first one)
- **flow\_area** – which flow\_area should be plotted (default, the first one)
- **max\_point** – the number of cell to be drawn when reconstructing the grid (it might long)
- **path\_im** – the path where the figure should be saved

### Technical comment

This function creates three figures which represent: a) the grid of the loaded models b) the water height and c) the velocity.

The two last figures will be modified when the data will be loaded by node and not by cells. So we will not explain them here as they should be re-written.

The first figure is used to plot the grid. If we would plot the grid by drawing one side of each triangle separately, it would be very long to draw. To optimize the process, we use the prepare\_grid function.

```
src.hec_ras2D.get_triangular_grid_hecras(ikle_all, coord_c_all, point_all, h, v)
```

In Hec-ras, it is possible to have non-triangular cells, often rectangular cells. This function transform the “mixed” grid to a triangular grid. For this, it uses the centroid of each cell with more than three side and it create a triangle

by side (linked with the center of the cell). A similar function exists in `rubar.py`, but, as there are only one reach in `rubar` and because `ikle` is different in `hec-ras`, it was hard to marge both functions together.

#### Parameters

- **ikle\_all** – the connectivity table by reach (list of `np.array`)
- **coord\_c\_all** – the coordinate of the centroid of the cell by reach
- **point\_all** – the points of the grid
- **h** – data on water height by reach by time step
- **v** – data on velocity by reach by time step

**Returns** the updated `ikle`, `coord_c` (the center of the cell , must be updated ) and `xy` (the grid coordinate)

```
src.hec_ras2D.load_hec_ras2d(filename, path)
```

The goal of this function is to load 2D data from Hec-RAS in the version 5.

#### Parameters

- **filename** – the name of the file containg the results of HEC-RAS in 2D. (string)
- **path** – the path where the file is (string)

**Returns** velocity and height at the center of the cells, the coordinate of the point of the cells, the coordinates of the center of the cells and the connectivity table. Each output is a list of numpy array (one array by 2D flow area)

#### How to obtain the input file

The file neede as input is an hdf5 file (.hdf) created automatically by Hec-Ras. There are many .hdf created by Hec-Ras. The one to choose is the one with the extension `p0X.hdf` (not `g0x.hdf`). It is usually the largest file in the results folder.

#### Technical comments

Outputs from HEC-RAS in 2D are in the hdf5 format. However, it is not possible to directly use the output of HEC-RAS as an hdf5 input for HABBY. Indeed, even if they are both in hdf5, the formats of the hdf5 files are different (and would miss some important info for HABBY). So we still need to load the HEC-RAS data in HABBY even if in 2D.

This function call the function `get_triangular grid` which is in `rubar.py`.

#### Walk-through

The name and path of the file is given as input to the `load_hec_ras_2D` function. Usually this is done by the class `HEC_RAS()` in the GUI. We load the file using the `h5py` module. This module opens and reads hdf5 file.

Then we can read different part of the hdf5 file when we know the address of it (this is a bit like a file system). In hdf5 file of Hec-RAS, this first thing is to get the names of the flow area in “Geometry/2D Flow Area”. In general, this is the name of each reach, but it could be lake or pond also. In an hdf5 file, to see the name of the member in a group, use: `list(“group”.keys())`

Then, we go to “Geometry/2D Flow Area/<name>/FacePoint Coordinates” to get the points forming the grid. We can also get the connectivity table (or `ikle`) to the path “Geometry/2D Flow Area/<name>/Cells Face Point Indexes” We also get the elevations of the cells. However, this is just the minimum elevation of the cells, so it is to be used only for a quick estimation. We then get the water depth by cell. The velocity is given by face of the cells and is averaged to get it on the middle of the cells.

To get Hec-Ras data by nodes, it is necessary to intepolate the data. There is a function to do this in `manage_grid_8`.

```
src.hec_ras2D.load_hec_ras_2d_and_cut_grid(name_hdf5, filename, path, name_prj,
                                           path_prj, model_type, nb_dim, path_hdf5,
                                           q=[], print_cmd=False)
```

This function calls `load_hec_ras_2d` and the `cut_2d_grid` function. Hence, it loads the data, pass it from cell to node (as data output in hec-ras is by cells) and it cut the grid to get only the wetted area. This was done before in the `HEC_RAS2D` Class in `hydro_gui_2.py`, but it was necessary to create a separate function to called this task in a second thread to avoid freezing the GUI.

#### Parameters

- **name\_hdf5** – the base name of the created hdf5 (string)
- **filename** – the name of the file containing the results of HEC-RAS in 2D. (string)
- **path** – the path where the file is (string)
- **name\_prj** – the name of the project (string)
- **path\_prj** – the path of the project
- **model\_type** – the name of the model such as Rubar, hec-ras, etc. (string)
- **nb\_dim** – the number of dimension (model, 1D, 1,5D, 2D) in a float
- **path\_hdf5** – A string which gives the adress to the folder in which to save the hdf5
- **q** – used by the second thread to get the error back to the GUI at the end of the thread
- **print\_cmd** – If True will print the error and warning to the cmd. If False, send it to the GUI.

**\*\* Technical comments\*\***

This function redirect the `sys.stdout`. The point of doing this is because this function will be call by the GUI or by the cmd. If it is called by the GUI, we want the output to be redirected to the windows for the log under HABBY. If it is called by the cmd, we want the print function to be sent to the command line.

```
src.hec_ras2D.main()
```

Used to test this module independantly of HABBY.

```
src.hec_ras2D.prepare_grid(ikle, coord_p, max_point=-99)
```

This is a function to put in the new form the data forming the grid to accelerate the plotting of the grid. This function creates a list of points of the grid which are re-ordered compared to the usual list of grid point (the variable `coord_p` here). These points are reordered so that it is possible to draw only one line to form the grid (one point can appears more than once). The grid is drawn as one long line and not as a succession of small lines, which is quicker. When this new list is created by `prepare_function()`, it is send back to `figure-hec_ras_2D` and plotted.

#### Parameters

- **ikle** – the connectivity table
- **coord\_p** – the coordinates of the point
- **max\_point** – if the grid is very big, it is possible to only plot the first points, up to `max_points` (int)

**Returns** a list of x and y coordinates ordered.

```
src.hec_ras2D.scatter_plot(coord, data, data_name, my_cmap, sl, t)
```

The function to plot the scatter of the data. Will not be used in the final version, but can be useful to plot data by cells.

#### Parameters

- **coord** – the coordinates of the point

- **data** – the data to be plotted (np.array)
- **data\_name** – the name of the data (string)
- **my\_cmap** – the color map (string with matplotlib colormap name)
- **s1** – the size of the dot for the scatter
- **t** – the time step being plotted

## 4.3 Mascaret

in `src/mascaret.py`

This module contains the functions used to load the outputs from the mascaret model.

`src.mascaret.correct_duplicate(seq, send_warn, idfun=None)`

It is possible to have a vertical line on a profile (different h, identical x). This is not possible for HABBY and the 2D grid. So this function correct duplicates along the profile.

A similar function exists in `rubar`, for the case where input is (x,y) coordinates and not distance along the profile. This function is inspired by <https://www.peterbe.com/plog/uniqifiers-benchmark>

It should be tested more as `manage_grid` sometime still send warning about duplicate data in profile.

### Parameters

- **seq** – the list to be corrected (list)
- **send\_warn** – a bool to avoid printing certain warning too many time
- **idfun** – support an optional transform function (not used)

**Returns** the profile data without duplicate and the boolean which manages the warning.

`src.mascaret.define_stream_network(node_number, start_node, end_node, angles, nb_pro_reach, nb_reach, abscisse)`

This function extracts the stream network from the node and angle data. This is used if we have more than one reach to define the geometry of the junction.

### Parameters

- **node\_number** – the start/end number of the reaches for each nodes (list of list)
- **start\_node** – the numbers indicating the start of each reach (list)
- **end\_node** – the numbers indicating the end of each reach
- **angles** – for each node the angle between the reach
- **nb\_pro\_reach** – the number of profile by reach
- **nb\_reach** – the number of reach
- **abscisse** – the distance along the river of each reach

**Returns** the river coordinates and the unit vector indicating the river direction

`src.mascaret.figure_mascaret(coord_pro, coord_r, xhzv_data, on_profile, nb_pro_reach, fig_opt, name_pro, name_reach, path_im, pro, plot_timestep=[-1], reach_plot=[0])`

The function to plot the figures related to mascaret.

### Parameters

- **coord\_pro** – the coordinates (x,y,h, dist along the river) of the profiles

- **coord\_r** – the coordinate (x,y) of the river
- **name\_pro** – the name of the profile
- **name\_reach** – the name of the reach
- **on\_profile** – which result are on the profile. Some output are not the profiles.
- **nb\_pro\_reach** – the number of profile by reach (careful this is the number of profile, not the number of output)
- **fig\_opt** – the figure option
- **xhzhv\_data** – the height and velocity (x,h,v) list by time step
- **profile** (*pro*) – which profile to be plotted (list of int)
- **plot\_timestep** – which timestep to be plotted
- **reach\_plot** – the reach to be plotted for the river view
- **path\_im** – the path where to save the figure

`src.mascaret.find_node (node_number, reach_to_find)`

This function finds which node is a stream end or a stream start. It is associated by the function `define_stream_network()`

#### Parameters

- **node\_number** – the list of list of the reaches linked with one node
- **reach\_to\_find** – the number indicating the start or end of the reach

**Returns** the node number, ordered as in the xcas file

`src.mascaret.flat_coord_pro (coord_pro)`

This function is not used anymore.

The variable `coord_pro` was a list of profile by reach. Finally, it was useful to have each profile one after the other with accounting for the reach. So we stop to use this function whose goal was to pass from one form of `coord_pro` to the other form (with or without reach information).

**Parameters** **coord\_pro** – the list of profile (x,y,h, dist along the river) by reach

**Returns** `coord_pro_f`: a list of profile without the reach information. The list is flatten

`src.mascaret.get_geo_name_from_xcas (file_gen, path_gen)`

This function gets the name of the .geo file from the .xcas xml file. It is not used yet, but it could be useful in the GUI to simplify the loading of mascaret. The user would not need to give the name of the geo and the xcas files separately. However, it is not written in yet.

#### Parameters

- **file\_gen** – the xcas file
- **path\_gen** – the path to the xcas file

**Returns** the name of the .geo file (no path indicated)

`src.mascaret.get_name_from_cas (file_gen, path_gen)`

This function gets the name of the .geo file from the .cas text file. It is not used yet, but it could be useful in the GUI to simplify the loading of mascaret. The user would not need to give the name of the geo and the cas files separately. However, it is not written in yet.

#### Parameters

- **file\_gen** – the name of .cas file (string)

- **path\_gen** – the path to the cas file (string)

**Returns** the name of the .geo file (no path indicated)

`src.mascaret.is_this_res_on_the_profile(abscisse, xhzv_data_all)`

The output of mascaret can be given at points of the river where there is no profile. The function here says which results are on the profiles. All profiles are linked with an output, but some output are not linked with a profile.

#### Parameters

- **abscisse** – the distance between each profile (list of float)
- **xhzv\_data\_all** – the outputs from mascaret by time step

**Returns** a list of bool of the length of xhzv\_data, True on profile, False not on profile

#### Technical comment

In the mascaret outputs, some rounding are suprising. For example, 0.49 can be transformed to 0.50 in an otehr file (not 0.5). To avoid this type of problem, we says that outputs with a distance smaller than 3cm of the profile are on the profile. If there are more than one output by profile, we takes the output which is the closest to the profile.

`src.mascaret.load_mascaret(file_gen, file_geo, file_res, path_gen, path_geo, path_res)`

The function is used to load the mascaret data. It load the geofile and the general file. Then, it re-forms the geometrical data. Next, it loads the output data from mascaret. Fianally, it looks which outputs is close to a profile and which outputs is not linked with a profile as there are some outputs given between profiles.

#### Parameters

- **file\_gen** – the xcas .xml file giving general info about the model (string)
- **file\_geo** – the file containing the profile data (.geo) (string)
- **file\_res** – the files containing the mascaret output in the Optyca format (.opt) (string)
- **path\_gen** – the path to the xcas file or .cas file (string). By default, choose the xcas file.
- **path\_geo** – the path to the geo file (string)
- **path\_res** – the path to the res file (string)

**Returns** the coordinates of the profile (x,y,z, dist along the profile), the coordinate of the river (x,y), name of reach and profile, data height and velocity (list by time step), list of bollean indicating which data is on the profile and the number of profile by reach.

`src.mascaret.load_mascaret_and_create_grid(name_hdf5, path_hdf5, name_prj, path_prj,  
model_type, namefile, pathfile, in-  
terpo_choice, manning_data, nb_point_vel,  
show_fig_1D, pro_add, q=[], path_im='.',  
print_cmd=False)`

This function is used to load the mascaret data by calling the load\_mascaret() function and to create the grid by calling the grid\_and\_interpo function in manage\_grid\_8. This function is called in a second thread by the class Mascaret() in Hydro\_grid\_2(). It also distribute the velocity by calling dist\_vitess2.

#### Parameters

- **name\_hdf5** – the name of the hdf5 to be created (string)
- **path\_hdf5** – the path to the hdf5 to be created (string)
- **model\_type** – the name of the model (mascaret in most case, but given as argument in case we change the form of the name)
- **name\_prj** – the name of the project (string)



- **path\_prj** – the path of the project
- **namefile** – the name of the geo file and the data file, which contains respectively geographical data and the output data (see `open_hec_ras()` for more precision) -> list of string
- **pathfile** – the absolute path to the file chosen into namefile -> list of string
- **interpo\_choice** – the interpolation type (int: 0,1,2 or 3). See `grid_and_interpo()` for more details.
- **manning\_data** – Contains the manning data. It can be in an array form (variable) or as a float (constant)
- **nb\_point\_vel** – the number of velocity point by whole profile
- **show\_fig\_1D** – A boolean. If True, image from the 1D data are created and saved
- **q** – used by the second thread.
- **pro\_add** – the number of additional profile (one used for interpolation\_choice 1 and 2)
- **path\_im** – the path where to save the figure
- **print\_cmd** – if True the print command is directed in the cmd, False if directed to the GUI

**\*\* Technical comments\*\***

This function redirect the `sys.stdout`. The point of doing this is because this function will be called by the GUI or by the cmd. If it is called by the GUI, we want the output to be redirected to the windows for the log under HABBY. If it is called by the cmd, we want the print function to be sent to the command line. We make the switch here.

`src.mascaret.main()`

Used to test this module separately.

`src.mascaret.open_geo_mascaret(file_geo, path_geo)`

This function load the mascaret geo file. Generally, the profile are not geo-referenced when using this function.

#### Parameters

- **file\_geo** – the name of the geo file (string)
- **path\_geo** – the path to the geo file (string)

**Returns** the profile data (x,y), profile name (list of string), brief name (list of string), the number of profile in each reach and distance along the river/abscisse (list)

`src.mascaret.open_res_file(file_res, path_res)`

The function to load the output from mascaret (.opt file). The format is Optyca.

#### Parameters

- **file\_res** – the name of the .opt file (string)
- **path\_res** – the path to this file (string)

#### Returns

`src.mascaret.open_rub_file(file_res, path_res)`

The function to open the binary output file from mascaret (.rub format).

#### Parameters

- **file\_res** – the name of the rub binary file (string)
- **path\_res** – the path to this file (string)

**Returns** xhzy\_data, timestep

#### Technical comment

The binary output file was done using a program written in FORTRAN. So there are often suprising octet which are added to the binary file. Be careful before changing anything.

`src.mascret.profil_coord_non_georef` (*coord\_pro, coord\_r, nr, nb\_pro\_reach, bt=None*)

This function gets the coordinates (x,y) of the profile as mascret outputs are not georeferenced.

Hypothesis: The river and the profile are straight. The profile is perpendicular to the river. The river pass at the minimum elevation of the river bed. If there is a distinction between the main bed the secondary bed is given, we take the minimum elevation of the main bed

The origin of the coordinate system is the start of the river.

#### Parameters

- **coord\_pro** – the coordinate of the profile. This variable is not in the general coordinate system, just distance along the profile and bed elevation (p, dist, h)
- **coord\_r** – the river coordinates
- **n** – the vector indicating the river direction
- **nb\_pro\_reach** – the number of profile by reach (additive)
- **bt** – optional, it indicates which points in the profiles are in the minor/major bed

**Returns** the velocity and height data, the timestep

`src.mascret.river_coord_non_georef_from_cas` (*file\_gen, path\_gen, abcisse, nb\_pro\_reach*)

Get the coordinates of the river based on the cas text file. If there are only one river, this is an easy task as the river is straight. If there are more than one reach, the junctions and the angles between the reach could be managed using the `define_stream_network` function and the information in the .cas file.

#### Parameters

- **file\_gen** – the .cas file whcih contains general info (string)
- **path\_gen** – the path to this file (string)
- **abcisse** – ditance along the profiles
- **nb\_pro\_reach** – the number of reach by profile

**Returns** the river coordinate and the unit vector indicating the river direction

`src.mascret.river_coord_non_georef_from_xcas` (*file\_gen, path\_gen, abcisse, nb\_pro\_reach*)

Get the coordinates of the river based on the xcas xml file. If there are only one river, this is an easy task as the river is straight. If there are more than one reach, the junctions and the angles between the reach could be managed using the `define_stream_network` function and the information in the .xcas file.

#### Parameters

- **file\_gen** – the .xcas file with the information concerning the reach (string)
- **path\_gen** – the path to the xcas file (string)
- **abcisse** – the distance along the river
- **nb\_pro\_reach** – the number of profile by reach

**Returns** coord\_r the coordinate of the river

## 4.4 River 2D

in `src/river2D.py`

This module contains the functions used to load the outputs from the River2D model.

`src.river2d.figure_river2d(xyzhv, ikle, path_im, t=0)`

A function to plot the output from river 2d. Need `hec-ras2d` as import because it re-used most of the plot from this script. It is only used to debug. It is not used directly by HABBY.

Plot only one time step because river 2d output have one file by time step.

### Parameters

- **xyzhv** – the x,y, coordinates of the node (h,v are nodal output in river 2d), the river bed, the water height and the velocity (one data by column, row are node)
- **ikle** – connectivity table
- **path\_im** – the path where to save the figure
- **t** – the time step which is being plotted

### Returns

`src.river2d.get rid_of_lines(datahere, nb_data)`

There are lines which are useless in the `cdg` file. This function is used to correct `ikle` and `data_node`

### Parameters

- **datahere** – the data with the empty lines
- **nb\_data** – `nb_node` or `nb_el`

**Returns** `datahere` without the useless lines

`src.river2d.load_river2d_and_cut_grid(name_hdf5, namefiles, paths, name_prj, path_prj, model_type, nb_dim, path_hdf5, q=[], print_cmd=False)`

This function loads the river2d data and cut the grid to the wet area. Originally, this function was in the class `River2D()` in `hydro_GUI_2`. This function was added as it was practical to have a second thread to avoid freezing the GUI.

### Parameters

- **name\_hdf5** – the base name of the created `hdf5` (string)
- **namefiles** – the names of all the `cdg` file (list of string)
- **paths** – the path to the files (list of string).
- **name\_prj** – the name of the project (string)
- **path\_prj** – the path of the project
- **model\_type** – the name of the model such as `Rubar`, `hec-ras`, etc. (string)
- **nb\_dim** – the number of dimension (model, 1D, 1,5D, 2D) in a float
- **path\_hdf5** – A string which gives the adress to the folder in which to save the `hdf5`
- **q** – used to send the error back from the second thread (can be used to send other variable too)
- **print\_cmd** – if `True` the print command is directed in the `cmd`, `False` if directed to the GUI

```
src.river2d.load_river2d_cdg(file_cdg, path)
```

The file to load the output data from River2D. Careful the input data of River2D has the same ending and nearly the same format as the output. However, it is necessary to have the output here. River2D gives one cdg. file by timestep. Hence, this function read only one timestep. HABBY read all time step by calling this function once for each time step.

#### Parameters

- **file\_cdg** – the name of the cdg file (string)
- **path** – the path to this file (string).

**Returns** the velocity and height data, the coordinate and the connectivity table.

```
src.river2d.main()
```

Used to test this module.

## 4.5 Rubar

in src/rubar.py

This module contains the functions used to load the Rubar data in 2D and 1D.

```
src.rubar.correct_duplicate_xy(seq3D, send_warn, idfun=None)
```

It is possible to have a vertical line on a profile (different h, identical x). This is not possible for HABBY and the 2D grid. So this function correct duplicates along the profile.

A similar function exists in mascaret, for the case where the input is the distance along the profile and not (x,y) coordinates. This function is inspired by <https://www.peterbe.com/plog/uniqifiers-benchmark>.

It should be tested more as manage\_grid sometime still send warning about duplicate data in profile.

#### Parameters

- **seq3D** – the list to be corrected in this case (x,y,z,dist along the profile)
- **send\_warn** – a bool to avoid printing the warning too many time
- **idfun** – support an optional transform function (not tested)

**Returns** the list without duplicate and the boolean which helps manage the warnings

```
src.rubar.figure_rubar1d(coord_pro, lim_riv, data_xhzv, name_profile, path_im, pro, plot_timestep,
                        nb_pro_reach=[0, 10000000000], fig_opt={})
```

The function to plot the loaded RUBAR 1D data (Rubar BE).

#### Parameters

- **coord\_pro** – the coordinate of the profile (x, y, z, dist along the river)
- **lim\_riv** – the right bank, river center, left bank
- **data\_xhzv** – the data by time step with x the distance along the river, h the water height and v the velocity
- **cote** – the altitude of the river center
- **name\_profile** – the name of the profile
- **path\_im** – the path where to save the image
- **pro** – the profile number which should be plotted
- **plot\_timestep** – which timestep should be plotted

- **nb\_pro\_reach** – the number of profile by reach
- **fig\_opt** – the dictionnary with the figure option

**Returns** none

`src.rubar.figure_rubar2d(xy, coord_c, ikle, v, h, path_im, time_step=[-1])`

This functions plots the rubar 2d data. This function is only used to debug. It is not used directly by Habby.

**Parameters**

- **xy** – coordinates of the points
- **coord\_c** – the center of the point
- **ikle** – connectivity table
- **v** – speed
- **h** – height
- **path\_im** – the path where to save the figure
- **time\_step** – The time step which will be plotted

`src.rubar.get_triangular_grid(ikle, coord_c, xy, h, v)`

In Rubar, it is possible to have non-triangular cells. It is possible to have a grid composed of a mix of pentagonal, 4-sided and triangular cells. This function transform the “mixed” grid to a triangular grid. For this, it uses the centroid of each cell with more than three side and it create a triangle by side (linked with the center of the cell). A similar function exists in hec-ras2D.py, but, as there is only one reach in rubar and because ikle is different in hec-ras, it was hard to marge both functions together.

**Parameters**

- **ikle** – the connectivity table (list)
- **coord\_c** – the coordinate of the centroid of the cell (list)
- **xy** – the points of the grid (np.array)
- **h** – data on water height
- **v** – data on velocity

**Returns** the updated ikle, coord\_c (the center of the cell , must be updated ) and xy (the grid coordinate)

`src.rubar.load_coord_1d(name_rbe, path)`

the function to load the rbe file, which is an xml file. The gives the geometry of the river system.

**Parameters**

- **name\_rbe** – The name fo the rbe file (string)
- **path** – the path to this file (string)

**Returns** the coordinates of the profiles and the coordinates of the right bank, center of the river, left bank (list of np.array with x,y,z coordinate), name of the profile (list of string), dist along the river (list of float) number of cells (int)

`src.rubar.load_dat_2d(geofile, path)`

This function is used to load the geomtery info for the 2D case, using the .dat file The .dat file has the same role than the .mai file but with more information (number of side and more complicated connectivity table).

**Parameters**

- **geofile** – the .dat file which contain the connectivity table and the (x,y)

- **path** – the path to this file

**Returns** connectivity table, point coordinates, coordinates of the cell centers

`src.rubar.load_data_1d(name_data_vh, path, x)`

This function loads the output data for Rubar BE (in 1D). The geometry data should be loaded before using this function.

**Parameters**

- **name\_data\_vh** – the name of the profile.ETUDE file (string)
- **path** – the path to this file
- **x** – the distance along the river (from the .geo file)

**Returns** data x, velocity height, cote for each time step (list of np.array), time step

`src.rubar.load_mai_1d(mailfile, path)`

This function is not used anymore. It was used to load the coordinate of the 1D data. It might become useful again in the case where we found a Rubar model with more than one reach (which we do not have yet).

**Parameters**

- **mailfile** – the name of the file which contain the (x,z) data
- **path** – the path to this file

**Returns** x of the river, np.array and the number of mail

`src.rubar.load_mai_2d(geofile, path)`

The function to load the geomtery info for the 2D case when we use the .mai file. It would also be possible to use the .dat file. In fact, it is advised to use the dat file when possible as there are more info in the .dat file.

**Parameters**

- **geofile** – the .mai file which contain the connectivity table and the (x,y)
- **path** – the path to this file

**Returns** connectivity table, point coordinates, coordinates of the cell centers

`src.rubar.load_rubar1d(geofile, data_vh, pathgeo, pathdata, path_im, savefig, fig_opt=[])`

the function to load the RUBAR BE data (in 1D).

**Parameters**

- **geofile** – the name of .rbe file which gives the coordinates of each profile (string)
- **data\_vh** – the name of the profile.ETUDE file which contains the height and velocity data (string)
- **pathgeo** – the path to the geofile - string
- **pathdata** – the path to the data\_vh file
- **path\_im** – the file where to save the image
- **savefig** – a boolean. If True create and save the figure.
- **fig\_opt** – A dictionary with the figure option

**Returns** coordinates of the profile (x,y,z dist along the profile) coordinates (x,y) of the river and the bed, data xhzv by time step where x is the distance along the river, h the water height, z the elevation of the bed and v the velocity

```
src.rubar.load_rubar1d_and_create_grid(name_hdf5, path_hdf5, name_prj, path_prj,
                                       model_type, namefile, pathfile, interpo_choice, man-
                                       ning_data, nb_point_vel, show_fig_1D, pro_add,
                                       q=[], path_im='.', print_cmd=False)
```

This function is used to load rubar 1d data by calling the load\_rubar1d() function and to create the grid by calling the grid\_and\_interpo function in manage\_grid\_8. This function is called in a second thread by the class Mascaret() in Hydro\_grid\_2(). It also distribute the velocity by calling dist\_vitesse2.

#### Parameters

- **name\_hdf5** – the name of the hdf5 to be created (string)
- **path\_hdf5** – the path to the hdf5 to be created (string)
- **name\_prj** – the name of the project (string)
- **path\_prj** – the path of the project
- **model\_type** – the name of the model (rubar in most case, but given as argument in case we change the form of the name)
- **namefile** – the name of the geo file and the data file, which contains respectively geographical data and the output data (see open\_hec\_ras() for more precision) -> list of string
- **pathfile** – the absolute path to the file chosen into namefile -> list of string
- **interpo\_choice** – the interpolation type (int: 0,1,2 or 3). See grid\_and\_interpo() for more details.
- **manning\_data** – Contains the manning data. It can be in an array form (variable) or as a float (constant)
- **nb\_point\_vel** – the number of velocity point by whole profile
- **show\_fig\_1D** – A boolean. If True, image from the 1D data are created and saved
- **q** – used by the second thread.
- **path\_im** – the path where to save the figure
- **print\_cmd** – if True the print command is directed in the cmd, False if directed to the GUI

**\*\* Technical comments\*\***

This function redirect the sys.stdout. The point of doing this is because this function will be call by the GUI or by the cmd. If it is called by the GUI, we want the output to be redirected to the windows for the log under HABBY. If it is called by the cmd, we want the print function to be sent to the command line. We make the switch here.

```
src.rubar.load_rubar2d(geofile, tpsfile, pathgeo, pathtps, path_im, save_fig)
```

This is the function used to load the RUBAR data in 2D.

#### Parameters

- **geofile** – the name of the .mai or .dat file which contains the connectivity table and the coordinates (string)
- **tpsfile** – the name of the .tps file (string)
- **pathgeo** – path to the geo file (string)
- **pathtps** – path to the tps file which contains the outputs (string)
- **path\_im** – the path where to save the figure (string)
- **save\_fig** – a boolean indicating if the figures should be created or not

**Returns** velocity and height at the center of the cells, the coordinate of the point of the cells, the coordinates of the center of the cells and the connectivity table.

```
src.rubar.load_rubar2d_and_create_grid(name_hdf5, geofile, tpsfile, pathgeo, pathtps,  
                                       path_im, name_prj, path_prj, model_type, nb_dim,  
                                       path_hdf5, q=[], print_cmd=False)
```

This is the function used to load the RUBAR data in 2D, to pass the data from the cell to the node using interpolation and to save the whole in an hdf5 format

#### Parameters

- **name\_hdf5** – the base name of the created hdf5 (string)
- **geofile** – the name of the .mai or .dat file which contains the connectivity table and the coordinates (string)
- **tpsfile** – the name of the .tps file (string)
- **pathgeo** – path to the geo file (string)
- **pathtps** – path to the tps file which contains the outputs (string)
- **path\_im** – the path where to save the figure (string)
- **name\_prj** – the name of the project (string)
- **path\_prj** – the path of the project
- **model\_type** – the name of the model such as Rubar, hec-ras, etc. (string)
- **nb\_dim** – the number of dimension (model, 1D, 1,5D, 2D) in a float
- **path\_hdf5** – A string which gives the adress to the folder in which to save the hdf5
- **q** – used by the second thread to get the error back to the GUI at the end of the thread
- **print\_cmd** – if True the print command is directed in the cmd, False if directed to the GUI

```
src.rubar.load_tps_2d(tpsfile, path, nb_cell)
```

The function to load the output data in the 2D rubar case. The geometry file (.mai or .dat) should be loaded before.

#### Parameters

- **tpsfile** – the name of the file with the data for the 2d case
- **path** – the path to the tps file.
- **nb\_cell** – the number of cell extracted from the .mai file

**Returns** v, h, timestep (all in list of np.array)

```
src.rubar.m_file_load_coord_1d(geofile_name, pathgeo)
```

This function loads the m.ETUDE file which is based on .st format from cemagref. When we use the M.ETUDE file instead of the rbe file, more than one reach can be studied but the center and side of the river is not indicated anymore.

#### Parameters

- **geofile\_name** – The name to the m.ETUDE file (string)
- **pathgeo** – the path to this file (string)

**Returns** the coordinates of the profiles (list of np.array with x,y,z coordinate), name of the profile (list of string), dist along the river (list of float), number of profile by reach



```
src.rubar.main()
    Used to test this module
```

## 4.6 Telemac

in src/selafin\_habby1.py

This module contains the functions used to load the Telemac data.

**class** src.selafin\_habby1.**Selafin** (*filename*)  
 Selafin file format reader for Telemac 2D. Create an object for reading data from a slf file. Adapted from the original script 'parserSELAFIN.py' from the open Telemac distribution.

**Parameters** **filename** – the name of the binary Selafin file

**addcontent** (*fileName, times, values*)

**appendcoretimeslf** (*t*)

**appendcorevarsslf** (*varsor*)

**appendheaderslf** ()

Write the header file

**getheaderfloatsslf** ()

Get the mesh coordinates

**getheaderintegersslf** ()

Get dimensions and descriptions (mesh)

**getheadermetadataslf** ()

Get header information

**gettimehistoryslf** ()

Get the timesteps

**getvalues** (*t*)

Get the values for the variables at time t

**getvariablesat** (*frame, varindexes*)

Get the values for the variables at a particular time step

**putcontent** (*fileName, times, values*)

src.selafin\_habby1.**getendianfromchar** (*fileslf, nchar*)

**Get the endian encoding** "<" means little-endian ">" means big-endian

src.selafin\_habby1.**getfloattypefromfloat** (*fileslf, endian, nfloat*)

Get float precision

src.selafin\_habby1.**load telemac** (*namefilel, pathfilel*)

A function which load the telemac data using the Selafin class.

**Parameters**

- **namefilel** – the name of the selafin file (string)
- **pathfilel** – the path to this file (string)

**Returns** the velocity, the height, the coordinate of the points of the grid, the connectivity table.

```
src.selafile_habby1.load_telemac_and_cut_grid(name_hdf5, namefile, pathfile, name_prj,  
                                              path_prj, model_type, nb_dim, path_hdf5,  
                                              q=[], print_cmd=False)
```

This function calls the function `load_telemac` and call the function `cut_2d_grid()`. Originally, this function was part of the `TELEMAC` class in `Hydro_GUI_2.py` but it was separated to be able to have a second thread, which is useful to avoid freezing the GUI.

#### Parameters

- **name\_hdf5** – the base name of the created hdf5 (string)
- **namefile** – the name of the selafile file (string)
- **pathfile** – the path to this file (string)
- **name\_prj** – the name of the project (string)
- **path\_prj** – the path of the project
- **model\_type** – the name of the model such as Rubar, hec-ras, etc. (string)
- **nb\_dim** – the number of dimension (model, 1D, 1,5D, 2D) in a float
- **path\_hdf5** – A string which gives the adress to the folder in which to save the hdf5
- **q** – used by the second thread to get the error back to the GUI at the end of the thread
- **print\_cmd** – if True the print command is directed in the cmd, False if directed to the GUI

```
src.selafile_habby1.plot_vel_h(coord_p2, h, v, path_im, timestep=[-1])
```

a function to plot the velocity and height which are the output from TELEMAC. It is used to debug. It is not used directly by HABBY.

#### Parameters

- **coord\_p2** – the coordinates of the point forming the grid
- **h** – the water height
- **v** – the velocity
- **path\_im** – the path where the image should be saved (string)
- **timestep** – which time step should be plotted

## 4.7 LAMMI

in `src/lammi.py`

This module contains functions used to load data from LAMMI. For more information on LAMMI, please see the pdf document `LAMMI.pdf`

```
src.lammi.coord_lammi(dist_all, vel_all, height_all, sub_all, length_all)
```

This function takes the data from the lammi outputs and get the coordinate for the river. It also reform the data to put it in the needed for HABBY (as the other 1.5D hydraulic model as hec\_ras).

To get the coordinates, we assume that the river is straight, that each facies is one after the other and that the river passes by the deepest point of the profile. In addition we assume that the profile are straight and perpendicular to the river.

We loop through all the profiles for all reach all time steps. For each profile, the x coordinate is identical for all point of the profile and is calculated using `length_all`. When a new reach starts, a meter is added to the x

coordinate. To find the y coordainte, we first pass from cell data (in lammi) to point data. The poitn are in the center of the cell. In addition, one point is added at the start an end of the profile with a water heighth of zero. Then, we find the height water height and we assume that the passes there. There is the zeros y-ccordinates.

#### Parameters

- **dist\_all** – the distance along profile by reach (or facies) and by time step
- **vel\_all** – the velocity along profile by reach (or facies) and by time step
- **height\_all** – the height along profile by reach (or facies) and by time step
- **sub\_all** – the substrate data along profile by reach (or facies) and by time step. Eacu substrate data is a list of eight number representing the percentage of each of the eight subtrate class.
- **length\_all** – the distance between profile

**Returns** coord\_pro, nb\_pro\_reach and vh\_pro in the same form as in hec-ras (final form)

`src.lammi.fig_lammi(vh_pro, coord_pro, nb_pro_reach, pro_num, sim_num, fig_opt, path_im)`

This function create a figure with the loaded lammi data. It work only for one time steps given by the number sim\_num.

#### Parameters

- **vh\_pro** – dist along the profile, height, vel
- **coord\_pro** – x,y, dist along profile, height
- **nb\_pro\_reach** – the number of profile by reach
- **pro\_num** – the profile to plot
- **sim\_num** – the time step (or simulation) to plot
- **fig\_opt** – the option for the figure
- **path\_im** – path path where to save the figure

`src.lammi.get_transect_filename(facies_path, facies_name, transect_path, transect_name, new_dir)`

For each facies, we obtain the name of the transect file and the length of this reach

#### Parameters

- **facies\_path** – the path the facies.txt file
- **facies\_name** – the name of the facies file, usually 'Facies.txt'
- **transect\_path** – the path to the transect.txt path
- **transect\_name** – the name of the transect file, usually 'Transect.txt'
- **new\_dir** – If the folder with the transect have been moved, this argument allos it to be corrected without modification to transect.txt

**Returns** the length of each transect (arranged by facies and station) and the filename with the transect info

`src.lammi.load_lammi(facies_path, transect_path, path_im, new_dir='', fig_opt=[], transect_name='Transect.txt', facies_name='Facies.txt')`

This function loads the data from the LAMMI model. A description of the LAMMI model is available in the documentation folder (LAMMIGuideMetho.pdf).

#### Parameters

- **transect\_path** – the path to the transect.txt path

- **facies\_path** – the path the facies.txt file
- **path\_im** – the path where to save the image
- **fig\_opt** – the figure option
- **new\_dir** – if necessary, the path to the resultat file (.prn file). Be default, use the one in transect.txt
- **transect\_name** – the name of the transect file, usually ‘Transect.txt’
- **facies\_name** – the name of the facies file, usually ‘Facies.txt’

### Returns

### Technical Comments

LAMMI is organised around group of transects. Transect are river profile which describe the river geometry. In LAMMI, there are four way of grouping transect. The facies is the a group a transect which is considered by HABBY to form a reach. The facies can then begroup in station. HABBY do not considered station directly, but it is possible to use the function “load\_station” to get the station info if needed. The group Secteur are used in case where water is brought to the river.

To load LAMMI data, we first load the facies file, which gives which transect are in which facies. Then, we use the transect file to know the length of each transect (length between transects along the river) and the name of the file containing the transect precise data. The name of the file is an absolute path to the file. This can be annoying if one want to move the files. Hence, we add the variable new\_dir which correct the transect file in case the files containing the transect data have been moved (they should however all be in the same directory). This is done by the function get\_transect\_name().

Then it uses the function load\_transect\_data to read all this data , file by file. Consequently, we have the data in memory but no(x,y) coordinate. In addition, this data is is in the different form than in the other hydraulic model.

To obtain the coordainte of the river and to put the data is the form usually needed by HABBY for 1.5D model (coord\_pro, vh\_pro, nb\_pro\_reach), we use the coord\_lammi() function.

`src.lammi.load_station(station_path, station_name)`

This function loads the station data from the LAMMI model. This is the data contains in Station.txt. It is not used by HABBY but it could be useful.

### Parameters

- **station\_path** – the path to the station.txt file
- **transect\_path** – the path to the transect.txt path
- **facies\_path** – the path the facies.txt file
- **new\_dir** – if necessary, the path to the resultat file (.prn file). Be default, use the one in transect.txt
- **station\_name** – the name of the station file, usually ‘Station.txt’
- **transect\_name** – the name of the transect file, usually ‘Transect.txt’
- **facies\_name** – the name of the facies file, usually ‘Facies.txt’

**Returns** the length of the station (list of float) and the id of the facies for each station (list of list)

`src.lammi.load_transect_data(fac_filename_all)`

This function loads the transect data. In this data, there are the substrate, the height and the velocity data.

**Parameters** **fac\_filename\_all** – the list of transect name organized by facies

```
src.lammi.main()
```

Used to test this module

## 4.8 Load HABBY hdf5 file

in `src/load_hdf5.py`

This module contains some functions to load and manage hdf5 input/outputs.

```
src.load_hdf5.copy_files(names, paths, path_input)
```

This function copied the input files to the project file. The input files are usually contains in the input project file. It is ususally done on a second thread as it might be long.

For the moment this function cannot send warning and error to the GUI. As input shoudl have been cheked before by HABBY, this should not be a problem.

### Parameters

- **names** – the name of the files to be copied (list of string)
- **paths** – the path to these files (list of string)
- **path\_input** – the path where to send the input (string)

```
src.load_hdf5.get_all_filename(dirname, ext)
```

This function gets the name of all file with a particular extension in a folder. Useful to get all the output from one hydraulic model.

### Parameters

- **dirname** – the path to the directory (string)
- **ext** – the extension (.txt for example). It is a string, the point needs to be the first character.

**Returns** a list with the filename (filename+dir) for each extension

```
src.load_hdf5.get_hdf5_name(model_name, name_prj, path_prj)
```

This function get the name of the hdf5 file containg the hydrological data for an hydrological model of type `model_name`. If there is more than one hdf5 file, it choose the last one. Tha path is the path from the project folder. Hencem, it is not the absolute path.

### Parameters

- **model\_name** – the name of the hydrological model as written in the attribute of the xml project file
- **name\_prj** – the name of the project
- **path\_prj** – the path to the project

**Returns** the name of the hdf5 file

```
src.load_hdf5.load_hdf5_hyd(hdf5_name_hyd, path_hdf5='', merge=False)
```

A function to load the 2D hydrological data contains in the hdf5 file in the form required by HABBY. f `hdf5_name_sub` is an absolute path, the `path_prj` is not used. If it is a relative path, the path is composed of the path to the project (`path_prj`) composed with `hdf5_name_sub`.

### Parameters

- **hdf5\_name\_hyd** – filename of the hdf5 file (string)
- **path\_hdf5** – the path to the hdf5 file
- **merge** – If merge is True. this is a merged file with substrate data added

**Returns** the connectivity table, the coordinates of the point, the height data, the velocity data on the coordinates.

`src.load_hdf5.load_hdf5_sub(hdf5_name_sub, path_hdf5)`

A function to load the substrate data contained in the hdf5 file. It also manage the constant cases. If `hdf5_name_sub` is an absolute path, the `path_prj` is not used. If it is a relative path, the path is composed of the path to the 'hdf5' folder (`path_prj/fichier_hdf5`) composed with `hdf5_name_sub`. it manages constant and variable (based on a grid) cases. The code should be of `cemagref` type and the data is given as coarser and dominant.

#### Parameters

- **hdf5\_name\_sub** – path and file name to the hdf5 file (string)
- **path\_prj** – the path to the hdf5 file

`src.load_hdf5.open_hdf5(hdf5_name)`

This is a function which open an hdf5 file and check that it exists. it does not load the data. It only opens the files. :param `hdf5_name`: the path and name of the hdf5 file (string)

`src.load_hdf5.save_hdf5(name_hdf5, name_prj, path_prj, model_type, nb_dim, path_hdf5, ikle_all_t, point_all_t, point_c_all_t, inter_vel_all_t, inter_h_all_t, xhzh_data=[], coord_pro=[], vh_pro=[], nb_pro_reach=[], merge=False, sub_pg_all_t=[], sub_dom_all_t=[])`

This function save the hydrological data in the hdf5 format.

#### Parameters

- **name\_hdf5** – the base name for the hdf5 file to be created (string)
- **name\_prj** – the name of the project (string)
- **path\_prj** – the path of the project
- **model\_type** – the name of the model such as `Rubar`, `hec-ras`, etc. (string)
- **nb\_dim** – the number of dimension (model, 1D, 1.5D, 2D) in a float
- **path\_hdf5** – A string which gives the adress to the folder in which to save the hdf5
- **ikle\_all\_t** – the connectivity table for all discharge, for all reaches and all time steps
- **point\_all\_t** – the point forming the grid, for all reaches and all time steps
- **point\_c\_all\_t** – the point at the center of the cells, for all reaches and all time steps
- **inter\_vel\_all\_t** – the velocity for all grid point, for all reaches and all time steps (by node)
- **inter\_h\_all\_t** – the height for all grid point, for all reaches and all time steps (by node)
- **xhzh\_data** – data linked with 1D model (only used when a 1D model was transformed to a 2D)
- **coord\_pro** – data linked with 1.5D model or data created by `dist_vist` from a 1D model (profile data)
- **vh\_pro** – data linked with 1.5D model or data created by `dist_vist` from a 1D model (velocity and height data)
- **nb\_pro\_reach** – data linked with 1.5D model or data created by `dist_vist` from a 1D model (nb profile)
- **merge** – If True, the data is coming from the merging of substrate and hydrological data.

- **sub\_pg\_all\_t** – the data of the coarser substrate given on the merged grid by cell. Only used if merge is True.
- **sub\_dom\_all\_t** – the data of the dominant substrate given on the merged grid by cells. Only used if merge is True.

### Technical comments

This function could look better inside the class SubHydroW where it was before. However, it was not possible to use it on the command line and it was not practical for having two thread (it is impossible to have a method as a second thread)

This function creates an hdf5 file which contains the hydrological data. First it creates an empty hdf5. Then it fill the hdf5 with data. For 1D model, it fill the data in 1D (the original data), then the 1.5D data created by `dist_vitess2.py` and finally the 2D data. For model in 2D it only saved 2D data. Hence, the 2D data is the data which is common to all model and which can always be loaded from a hydrological hdf5 created by HABBY. The 1D and 1.5D data is only present if the model is 1D or 1.5D. Here is some general info about the created hdf5:

- Name of the file: `name_hdf5 + date/time.h5`. For example, `test4_HEC-RAS_25_10_2016_12_23_23.h5`.
- Position of the file: in the folder `figure_habby` currently (probably in a project folder in the final software)
- Format of the hdf5 file:
  - `Dats_gen`: number of time step and number of reach
  - `Data_1D`: `xhzv_data_all` (given profile by profile)
  - `Data_15D`: `vh_pro`, `coord_pro` (given profile by profile in a dict) and `nb_pro_reach`.
  - `Data_2D`: For each time step, for each reach: `ikle`, `point`, `point_c`, `inter_h`, `inter_vel`

If a list has elements with a changing number of variables, it is necessary to create a dictionary to save this list in hdf5. For example, a dictionary will be needed to save the following list: `[[1,2,3,4], [1,2,3]]`. This is used for example, to save data by profile as we can have profile with more or less points. We also note in the hdf5 attribute some important info such as the project name, path to the project, hdf5 version. This can be useful if an hdf5 is lost and is not linked with any project. We also add the name of the created hdf5 to the xml project file. Now we can load the hydrological data using this hdf5 file and the xml project file.

Hdf5 file do not support unicode. It is necessary to encode string to write them.

```
src.load_hdf5.save_hdf5_sub(path_hdf5, path_prj, name_prj, sub_pg, sub_dom,
                           ikle_sub=[], coord_p=[], name_hdf5='', constsub=False,
                           model_type='SUBSTRATE', return_name=False)
```

This function creates an hdf5 with the substrate data. This hdf5 does not have the same form than the hdf5 file used to store hydrological or merge data. This hdf5 store the substrate data alone before it is merged with the hydrological info. The substrate info should be given in the `cemagref` code.

### Parameters

- **path\_hdf5** – the path where the hdf5 file should be saved
- **path\_prj** – the project path
- **name\_prj** – the name of the project
- **sub\_pg** – the coarser part of the substrate (array with length of `ikle` if `const_sub` is False, a float otherwise)
- **sub\_dom** – the dominant part of the substrate (array with length of `ikle` if `const_sub` is False, a float otherwise)
- **ikle\_sub** – the connectivity table for the substrate (only if `constsub = False`)

- **coord\_p** – the point of the grid of the substrate (only if `constsub = False`)
- **name\_hdf5** – the name of the substrate hdf5 file (without the timestamp). If not given, a default name is used.
- **constsub** – If True the substrate is a constant value
- **model\_type** – the attribute for the xml file (usually SUBSTRATE)
- **return\_name** – If True this function return the name of the substrate hdf5 name

## 4.8.1 Form of the hdf5 files

Here is the actual form of the hdf5 containing the 2D hydrological data.

- Number of timestep: `Data_gen/Nb_timestep`
- Number of reach: `Data_gen/Nb_reach`
- Connectivity table for the whole profile: `Data_2D/Whole_Profile/Reach_<r>/ikle`
- Connectivity table for the wetted area (by time step): `Data_2D/Timestep<t>/Reach_<r>/ikle`
- Coordinates for the whole profile: `Data_2D/Whole_Profile/Reach_<r>/point_all`
- Coordinates for the wetted area (by time steps): `Data_2D/Timestep<t>/Reach_<r>/point_all`
- Data for the velocity: `Data_2D/Timestep<t>/Reach_<r>/inter_vel_all`
- Data for the height: `Data_2D/Timestep<t>/Reach_<r>/inter_h_all`

Here is the actual form of the hdf5 containing the substrate data.

- the coordinate of the point forming the substrate “grid”: `coord_p_sub/`
- the connectivity table of the substrate “grid”: `ikle_sub/`
- Substrate data; not done yet

## 4.9 Velocity distribution

in `src/dist_vitesse2.py`

The goal of this list of function is to distribute the velocity along the cross-section for 1D model such as mascaret or Rubar BE. Hec-Ras outputs do not need to uses this type of function as they are already distributed along the profiles.

The method of velocity distribution in HABBY is similar to the one used by Hec-Ras to distribute velocity.

```
src.dist_vistess2.dist_velocity_hecras (coord_pro, xhzv_data_all, manning_pro, nb_point=-99, eng=1.0, on_profile=[])
```

This function distribute the velocity along the profile using the method from hec-ras which is described in the hydraulic reference manual p 4-20 (Flow distribution calculation)

### Parameters

- **coord\_pro** – the coordinates and elevation of the river bed for each profile (x,y, h, dist along the profile) this list is flatten No reach info.
- **xhzv\_data\_all** – water height and velocity at each profile, 1D
- **manning\_pro** – the manning coefficient for zone between point of each profile. For a particular profile, the length of `manning_pro` is the length of `coord_pro[0]`



- **nb\_point** – number of velocity points (-99 takes the number of measured elevation as the number of velocity points).
- **eng** – in case the output from hec-ras are in US unit (eng=1 for SI unit and 1.486 for US unit)
- **on\_profile** – Mascaret also gives outputs in points between profile. on\_profile is true if the results are close or on the profile (less than 3cm of difference). This is not important for rubar or other models

**Returns** the velocity for each profile by time step (x,v)

#### Technical comment and walk-through

First, we decide on which point along the profile we will calculate the velocity. This is controlled by the variable nb\_point. If nb\_point=-99, we will calculate the velocity at the same point than the profile (i.e., the velocity will be calculated at each point on which the elevation of the profile was measured). There are cases where this is not adequate. Let's imagine for example a rectangular canal. The calculation would only give two velocity points, which is not enough. So, it is possible to give the number of velocity point on which the calculation must be made, using the variable nb\_point.

Currently, the velocity points are determined by dividing the whole profile in nb\_point segments. This means that some velocity point are not used afterwards because they are in the dry part of the profile and that it is not possible to select for a part of the profile where more velocity points would be calculated. This could be modified in the future if it is judged necessary.

To determine the point where velocity should be calculated we need to get two array: one "x" array, the distance along the profile and one "h" array, the elevation of the profile at this point. As we choose the position of the velocity point as regularly placed along the profile, the "x" array is easy to determine using linspace. For the "h" array, we use the hypothesis that the elevation of the profile changes linearly between the measured elevation points. We find between which elevation point are the new point and we use a linear interpolation to find the new "h". To find between which points we are, we use the bisect.bisect function. It is a bit like the np.where function, but it is quicker when the array is ordered (as it is the case here).

Then, we get the manning array as created by the get\_manning\_arr and the get\_manning function. It should be a float.

Next, we cut the profile to keep only the part under water. For this, we do two things: First we had a point on the profile where h==0. We should account for the fact that we might have "islands" (part of the profile which are dry, but surrounded by water on both side.). So we cannot only look which part are dry, we need to look for each point where we pass from "wet to dry" or from "dry to wet". At this place, we add one point where h=0. For these new points water height is obviously known, but x (the distance along profile) should be determined. It is determined assuming a linear change between the measured points of the profile.

If the profile is not entirely dry, we will now distribute the velocity along the profile. First, for each part of the profile where velocity will be calculated, it looks where is the higher height (like if this part of the profile is going up or down). Next, we calculate the area, the wetted perimeter and the hydraulic radius of each part of the profile. By combining this geometrical information with the manning parameter, we can calculate the conveyance of each part of the profile. See manual of hec-ras p.4-20 for the conveyance definition.

We now calculate the conveyance of the whole profile. Normally, the sum of the conveyance of the part is higher than the total conveyance. The next part of the script corrects for this, using the ratio of the total conveyance and the sum of the parts of the conveyance. Next, we calculate the velocity using the modelled energy slope (Sf) and the manning equation. We then then add a velocity of zero where there are no water (velocity is not defined at his point).

```
src.dist_vitesse2.distribute_velocity(manning_data, nb_point_vel, coord_pro, xhsv_data,
                                     on_profile=[])
```

This function make the link between the GUI and the functions of dist\_vitesse2. It is used by 1D model, notably rubar and masacret.

Dist vites needs a manning parameters. It can be given by the user in two forms: a constant (float) or an array created by the function `load_manning_text`.

#### Parameters

- **manning\_data** – the manning data as a float (constant) or as an array (variable)
- **nb\_point\_vel** – the number of velocity point asked (if -99, the same number of point than the profile will be used)
- **coord\_pro** – the form of the profil (x,y, h, dist along the profile)
- **xhzhv\_data** – the velocity and height data
- **on\_profile** – Mascaret also gives outputs in points between profile. `on_profile` is true if the results are close or on the profile (less than 3cm of difference). This is not important for rubar or other models

```
src.dist_vites2.get_manning(manning1, nb_point, nb_profil, coord_pro)
```

This function creates an array with the manning value when a single float is given, so when the manning value is a constant for the whole river.

#### Parameters

- **manning1** – the manning value (can be a value or an array)
- **nb\_point** – the number of velocity point by profile
- **nb\_profil** – the number of profile
- **coord\_pro** – necessary if the number is -99 as we need to know the length of each profile

#### Technical comments

The function `dist_velocity_hec_ras` needs a manning array with a length equal to the number of profile, where each row (representing a profile) have one value by velocity point which will be calculated. This function creates an array of this form based on a float. It creates a list of manning value which is identical for each point of the river. It can be used for the cases where the same number of point is asked for each profile or for the case where the number of point is defined by the form of the profile (`nb_point = -99`).

```
src.dist_vites2.get_manning_arr(manning_arr, nb_point, coord_pro)
```

This function create the manning array when manning data is loaded using a text file. In this case, the manning value do not needs to be a constant.

#### Parameters

- **manning\_arr** – the data for manning
- **nb\_point** – the number of velocity point by profile
- **coord\_pro** – x,y,dist

#### Technical comment

The user creates a txt file with a list of manning info. Each manning value is given the following way: the profile, the distance along the profile and the manning value. One value by line in SI unit.

This function automatically fills the missing value, so that the user do not needs to give each manning value. He can describe one profile and this profile will be replicated until the next profile written in the text file.

```
src.dist_vites2.main()
```

Used to test this module.

```
src.dist_vistess2.plot_dist_vit (v_pro, coord_pro, xhzv_data, plot_timestep, pro, name_pro=[],
                                on_profile=[], zone_v_all=[], data_profile=[], xy_h_all=[])
```

This is a function to plot the distribution of velocity and the elevation of the profile. It is quite close to the similar function which is in hec-ras (see this function for a more detailed explanation)

It can be used to test the program if we provide the variable `zone_v_all` where `zone_v_all` is an hec-ras output with a velocity distribution. In this case, it would plot the comparison between the output from this script and the output from hec-ras. Of course, for this, it is necessary to have prepared the 1D output from hec-ras (using the function `preparetest_velocity`) and to have the same points on which to calculate the velocity.

#### Parameters

- **v\_pro** – the calculated velocity distribution by time step
- **coord\_pro** – the coordinate of the profiles
- **xhzv\_data** – the output data from the model, before the velocity distribution
- **plot\_timestep** – which time step to be plotted
- **name\_pro** – the name of the profile (optional just for the title)
- **pro** – which profile to be plotted
- **on\_profile** – select the data which is on the profile
- **zone\_v** – output from hec-ras used to test `dist_vitesse`
- **data\_profile** – output from hec-ras used to test `dist_vitesse`
- **xy\_h** – output from hec-ras used to test `dist_vitesse`

```
src.dist_vistess2.preparetest_velocity (coord_pro, vh_pro_orr, v_in)
```

This is a debugging function. It takes as input the output from the hec-ras model and gives a 1D velocity as output. This is only to test this program. It will not be used by HABBY directly. To use this function, it is necessary to use the function to load hec-ras data from HABBY, so that the hec-ras data is in the right form. The 1D-velocity is assumed to be the velocity at the lowest part of the profile. This is where a 1D-model would estimate the position of the river (the lowest part of the river bed).

A complicated point to test the program is to put the velocity point at the same point than hec-ras. As hec-ras calculate velocity between zones and not on one point, this is more or less impossible to do with precision. However, one can count the number of velocity zone and give this as an input to `dist_velocity_hecras()` for the variable `nb_point`. However, both line will not be exactly at the same place. The results should however be close enough.

#### Parameters

- **coord\_pro** – the coordinate of the profile (x,y,h,dist along profile)
- **vh\_pro\_orr** – the velocity distribution which is the output from hec ras (produced by `hec-ras06.py`)
- **v\_in** – the uni-dimensional velocity

## 4.10 Create a grid

in `src/manage_grid_8`

This module is composed of the functions used to manage the grid, notably to create 2D grid from the output from 1D model.

There are two main way to go from data in 1.5D in a profile form to a 2D grid:

- through the usage of the triangle module in `create_grid()`.
- through the definition of a middle profile used as a guide to create the grid in `create_grid_only_one_profile()`.

For an in-depth explanation on how to create the grids, please see the pdf document `More info on the grid`

`src.manage_grid_8.add_point(point_all, point)`

To manage the substrate data, we modify the hydrological grid to avoid to have cells with two substrate type. This function add one coordinate point to the list of coordinates which compose the hydrological grid. This point is the intersection between one side of one triangluar cell of the hydrological grid and one side of the sibstrate layer (which is a shp). It only adds this intersection point if it is not already in `point_all`.

#### Parameters

- **point\_all** – the coordinates of the hydrological grid
- **point** – one intersection point between substrat and hydrological grids

**Returns** the updated `point_all` (the coordinates of the hydrological grid)

`src.manage_grid_8.create_dummy_substrate(coord_pro, sqrtnp)`

For testing purposes, it can be useful to create a substrate input even if one does not exist. This substrate is compose of n triangle situated on the rivers in the same coodinates system.

#### Parameters

- **coord\_pro** – the coordinate of each profile
- **sqrtnp** – the number of point which will compose one side of the new substrate grid (so the total number of point is `sqrtnb squared`).

**Returns** dummy `coord_sub`, `ikle_sub`

`src.manage_grid_8.create_grid(coord_pro, extra_pro, coord_sub, ikle_sub, nb_pro_reach=[0, 10000000000.0], vh_pro_t=[], q=[], pnnew_add=1)`

It creates a grid from the `coord_pro` data using the triangle module. It creates the grid up to the end of the profile if `vh_pro_t` is not present or up to the water limit if `vh_pro_t` is present

#### Parameters

- **q** – used in the secondary process when we do not call this function directly, but we call it in a second process so that the GUI do not crash if something go wrong (not used anymore in this form)
- **coord\_pro** – the profile coordinates (x,y, h, dist along) the profile
- **extra\_pro** – the number of “extra” profiles to be added between profile to simplify the grid
- **coord\_sub** – (not used anymore) the coordinate of the point forming the substrate layer (often created with `substrate.load_sub`)
- **ikle\_sub** – (not used anymore) the connectivity table of the substrate grid (often created with `substrate.load_sub`)
- **nb\_pro\_reach** – the number of reach by profile starting with 0
- **vh\_pro\_t** – the velocity and height of the water (used to cut the limit of the river).
- **pnnew\_add** – (not used anymore) a parameter to cut the substrate side in smaller part (improve grid quality) in the form dist along profile, h , v for the analyzed time step. f not given, gird is contructed on the whole profile.

**Returns** connectivity table and grid point

#### Form of the function in summary

- if `vh_pro_t`:
  - find coordinate under water and used this to update `coord_pro`
  - see if there is islands, find the island limits and the holes indicating the inside/outside of the islands
- find the point which give the end/start of the segment defining the grid limit
- find all point which need to be added to the grid and add extra profile if needed
- based on the start/end points and the island limits, create the segments which gives the grid limit
- triangulate and so create the grid
- flag point which are overlapping in two grids

For more info, see the document “More info on the grid”.

```
src.manage_grid_8.create_grid_only_1_profile(coord_pro, nb_pro_reach=[0,
10000000000.0], vh_pro_t=[])
```

This function creates the grid from the `coord_pro` data using one additional profil in the middle. No triangulation. The interpolation of the data is done in this function also, contrarily to `create_grid()`.

#### Parameters

- **coord\_pro** – the profile coordinates (x,y, h, dist along) the profile
- **nb\_pro\_reach** – the number of profile by reach
- **vh\_pro\_t** – the data with heigh and velocity, giving the river limits

**Returns** the connexivity table, the coordinate of the grid, the centroid of the grid, the velocity data on this grid, the height data on this grid.

For more info on this function, see the document “More info on the grid”.

```
src.manage_grid_8.cut_2d_grid(ikle, point_all, water_height, velocity)
```

This function cut the grid of the 2D model to have correct wet surface. If we have a node with  $h < 0$  and other node(s) with  $h > 0$ , this function cut the cells to find the wetted perimeter, assuming a linear decrease in the water elevation. This function works for one time steps and for one reach

#### Parameters

- **ikle** – the connectivity table of the 2D grid
- **point\_all** – the coordinate of the points
- **water\_height** – the water height data given on the nodes
- **velocity** – the velocity given on the nodes

**Returns** the update connectivity table, the coordinate of the point, the height of the water and the velocity on the updated grid

```
src.manage_grid_8.cut_2d_grid_all_reach(ikle_all, point_all, inter_height_all, inter_vel_all)
```

This function is just use to call `cut_2d-grid` for all reach. So that if we have a river with more than reach, we do not need to add a for loops to call for all reach. Sometime it can save place. This can be only use for one time step.

#### Parameters

- **ikle\_all** – the connectivity table of the 2D grid for all reach
- **point\_all** – the coordinate of the points for all reach
- **inter\_height\_all** – the water height data given on the nodes for all reach
- **inter\_vel\_all** – the velocity given on the nodes for all reach

**Returns** the update connectivity table, the coordinate of the point, the height of the water and the velocity on the updated grid for all reaches

`src.manage_grid_8.find_profile_between(coord_pro_p0, coord_pro_p1, nb_pro, trim=True)`

Find n profile between two profiles which are not straight. This function is useful to create the grid from 1D model as profile in 1D model are often far away from another.

**Parameters**

- **coord\_pro\_p0** – the coord\_pro (x,y,h, z) of the first profile
- **coord\_pro\_p1** – the coord\_pro (x,y,h, z) of the second profile
- **nb\_pro** – the number of profile to add
- **trim** – If True cut the end and start of profile to avoid to have part of the grid outside of the water limit

**Returns** a list with the updated profiles

`src.manage_grid_8.get_crossing_segment_sub(p1sub, p2sub, lim_here, lim_by_reachr, point_all, island, ind_seg_sub_ini=[0])`

This function looks at one substrate segment and find the crossing points of this segment with the different segment which composed the hydrological grid. This function is useful to cut the grid as a function of the form of the substrate layer (to avoid having cells in the hydrological grid which have two substrate value).

If island switch is True, lim\_here is the limit of the island, so inside the polygon is outside the river. If island is false, lim\_here is the limit of the reach under investigation

**Parameters**

- **p1sub** – the start point of the substrate segment
- **p2sub** – the end point of the substrate segment
- **lim\_here** – the reach?island limit given in the coordinate system
- **lim\_by\_reachr** – the limits for reach r which will be given to triangle given by point\_all indices.
- **point\_all** – all the point (coordinates) which will be given to triangle
- **island** – a boolean indicating if we are on an island or not
- **ind\_seg\_sub\_ini** – the indices of the first segment add by p1sub et p2sub by the reach. Only used island = true

**Returns** the updated point\_all and lim\_by\_reach

`src.manage_grid_8.get_new_point_and_cell_1_profile(coord_pro_p, vh_pro_t_p, point_mid_x, point_mid_y, point_all, ikle, point_c, dir)`

This function is use by create\_grid\_one\_profile. It creates the grid for one profile (one “line” of triangle). To create the whole grid this function is called for each profile.

**Parameters**

- **coord\_pro\_p** – the coordinates of the profile
- **vh\_pro\_t\_p** – the height and velocity data of the profile analysed
- **point\_mid\_x** – the x coordinate of the points forming the middle profile
- **point\_mid\_y** – the y coordinate of the points forming the middle profile
- **point\_all** – the point of the grid

- **ikle** – the connectivity table of the grid
- **point\_c** – the central point of each cell
- **dir** – in which direction are we going around the profile (upstream/downstream)

**Returns** point\_all, ikle, point\_c (the centroid of the cell)

For more info, see the document “More info on the grid”.

```
src.manage_grid_8.grid_and_interpo(vh_pro, coord_pro, nb_pro_reach, interpo_choice,
                                   pro_add=1)
```

This function forms the link between GUI and the various grid and interpolation functions. Is called by the “loading” function of hec-ras 1D, Mascaret and Rubar BE. It used to be a method in hydro\_GUI2, but we have to move it as a function to create a second thread. Hence, the high amount of parameter.

#### Parameters

- **vh\_pro** – Velcoity and height data
- **coord\_pro** – the position of the profile
- **nb\_pro\_reach** – the number of profile by reach
- **interpo\_choice** – an int which gives the choice of interpolatin (see below)
- **pro\_add** – the number of profile for be added (for interpoation method 1 and 2)

#### Technical comments

Here are the list of the interpolation choice:

- 0 Use the function create\_grid\_only\_1\_profile() from manage\_grid\_8.py for all time steps.
- 1 Use the function create\_grid() from manage\_grid\_8.py for all time steps followed by a linear interpolation
- 2 Use the function create\_grid() from manage\_grid\_8.py for all time steps followed by a nearest neighbour interpolation
- 3 Use create\_grid() for the whole profile, make a linear interporlation on this grid for all time step and use the cut\_2d\_grid to get a grid with only the wet profile for all time step (This part was only started. It was not finished.)

For the interpolation case 1 and 2, it is possible that the triangle module crashes if the geometry of the river is too complicated. Generally, the interpolation method 1 and 2 gives smoother results with more control over the interpolation option and the size of the cells. However, these two interpolation methods are more sensitive to the inputs, especially if the river has a lot of “island” (strongly anastomotic). So the interpolation method 0 is more adequate in this case.

```
src.manage_grid_8.inside_polygon(seg_poly, point)
```

This function find if a point is inside a polygon, using a ray casting algorythm.

#### Parameters

- **seg\_poly** – the segmentS forming the polygon
- **point** – the point which is indide or outside the polygon

**Returns** True is the point is inside the polygon, false otherwise

```
src.manage_grid_8.interp_weights(xyz, uvw)
```

This fuction is used by the function pass\_grid\_cell\_to\_node\_lin(). To optimize the interpolation when more than one time step is done on the same grid, the first step of scipy.griddata.interolate are done here and are called only once for all time step. Hence, this function is the first part of a quicker “scipy.interpolate.griddata”

#### Parameters

- **xyz** –
- **uvw** –

`src.manage_grid_8.interpo_linear` (*point\_all, coord\_pro, vh\_pro\_t*)

Using `scipy.griddata`, this function interpolates the 1.5 D velocity and height to the new grid. It can be used for only one time step. The interpolation is linear. It is usually called after `create_grid` have been called.

#### Parameters

- **point\_all** – the coordinate of the grid point
- **coord\_pro** – the coordinate of the profile. It should be coherent with the coordinate from `vh_pro`. To insure this, pass `coord_pro` through the function “`create_grid`” with the same `vh_pro` as input
- **vh\_pro\_t** – for each profile, dist along the profile, water height and velocity at a particular time step

**Returns** the new interpolated data for velocity and water height

`src.manage_grid_8.interpo_nearest` (*point\_all, coord\_pro, vh\_pro\_t*)

Using `scipy.griddata`, this function interpolates the 1.5 D velocity and height to the new grid. It can be used for only one time step. The interpolation is nearest neighbours. It is usually called after `create_grid` have been called.

#### Parameters

- **point\_all** – the coordinate of the grid point
- **coord\_pro** – the coordinate of the profile. It should be coherent with the coordinate from `vh_pro`. To insure this, pass `coord_pro` through the function “`create_grid`” with the same `vh_pro` as input
- **vh\_pro\_t** – for each profile, dist along the profile, water height and velocity at a particular time step

**Returns** the new interpolated data for velocity and water height

`src.manage_grid_8.interpolate_opti` (*values, vtx, wts*)

This function is called by `interp_weights()`. It is used in the optimization of the function `pass_grid_cell_to_node_lin()`. This idea of this optimization is to not re-do some calculation when many interpolation are done on the same grid.

#### Parameters

- **values** –
- **vtx** –
- **wts** –

`src.manage_grid_8.intersection_seg` (*p1hyd, p2hyd, p1sub, p2sub, col=True, wig=1e-07*)

This function finds if there is an intersection between two segment (AB and CD). Idea from : <http://stackoverflow.com/questions/563198/how-do-you-detect-where-two-line-segments-intersect> It is based on the calculation of the cross-product  $z = 0$  for 2D

Careful there is many function using this function, so change here should be thought about. There is a precision management, so everything smaller than  $10^{-8}$  is ok

#### Parameters

- **p1hyd** – point A
- **p2hyd** – point B



- **p1sub** – point C
- **p2sub** – point D
- **col** – if True, colinear segment crossed. If false, they do not cross
- **wig** – “wiggle room”, how precise should the calculation be (careful, complicated!)

**Returns** intersect (True or False) and the crossing point (if True, empty is False)

`src.manage_grid_8.linear_h_cross(p1, p2, h1, h2)`

This function is called by `cut_2D_grid`. It find the intersection point along a side of the triangle if part of a cells is dry.

#### Parameters

- **p1** – the coordinate (x,y) of the first point
- **p2** – the coordinate (x,y) of the first point
- **h1** – the water height at p1 (might be negative or positive)
- **h2** – the water height at p2 (might be negative or positive)

**Returns** the intersection point

`src.manage_grid_8.main()`

Used to test this module

`src.manage_grid_8.newp(p0, p1, extra_pro)`

This function find the start/end of the added profile. If only one profile is needed, it is just the point in the middle of the start/end of the profile. If more than one profile is needed, there are linearly distributed. This function only give the start and the end of the profile, the profile in full are constructed using `find_profile_between()`

#### Parameters

- **p0** – the point at the profile p
- **p1** – the point at the profile p-1
- **extra\_pro** – the number of extra profile needed

**Returns** the start/end of the new profile

`src.manage_grid_8.pass_grid_cell_to_node_lin(point_all, coord_c, vel_in, height_in, warn1=True, vtx_all=[], wts_all=[])`

HABBY uses nodal information. Some hydraulic models have only output on the cells. This function pass from cells information to nodal information. The interpolation is linear and the cell centroid is used as the point where the cell information is carried. It can be used for one time step only.

#### Parameters

- **point\_all** – the coordinates of grid points (new grid here)
- **coord\_c** – the coordinates of the centroid of the cells (old grid here)
- **vel\_in** – the velocity data by cell
- **height\_in** – the height data by cell
- **warn1** – if True, show the warning (usually warn1 is True for t=0, False afterwards)
- **vtx\_all** – if it exists it means that the same grid was interpolated before. This info can be reused to speed up the interpolation of multiple time step. (optional, need wts)
- **wts\_all** – if it exists it means that the same grid was interpolated before. This info can be reused to speed up the interpolation of multiple time step. (optional, need vtx)

**Returns** velocity and height data by node

#### Technical Comment

This function can be very slow when a lot of time step needs to be interpolated if done directly with `scipy.interpolate`. It was optimized for this case: <http://stackoverflow.com/questions/20915502/speedup-scipy-griddata-for-multiple-interpolations-between-two-irregular-grids>

```
src.manage_grid_8.plot_grid(point_all_reach, ikle_all, lim_by_reach, hole_all, overlap,
                             point_c_all=[], inter_vel_all=[], inter_h_all=[], path_im=[],
                             coord_pro2=[])
```

This is a function to plot a grid and the output. It is mostly used to debug the grid creation. Contrarily to the more simple function `plot_grid_simple`, it is possible to plot the position of the holes (which indicates the dry area), the limits of the reaches used by triangle, the overlap between two reaches, and so on.

#### Parameters

- **point\_all\_reach** – the grid point by reach
- **ikle\_all** – the connectivity table by reach
- **lim\_by\_reach** – the segment giving the limits of the grid
- **hole\_all** – the coordinates of the holes
- **overlap** – the point of each reach which are also on an other reach
- **point\_c\_all** – the centroid of each element
- **inter\_vel\_all** – the interpolated velocity for each reach
- **inter\_h\_all** – the interpolated height
- **path\_im** – the path where to save the image

```
src.manage_grid_8.plot_grid_simple(point_all_reach, ikle_all, fig_opt, inter_vel_all=[],
                                   inter_h_all=[], path_im=[], merge_case=False,
                                   time_step=0)
```

This is the function to plot grid output for one time step. The data is one the node. A more complicated function exists to plot the grid and additional information (`manage_grid_8.plot_grid()`) in case there are needed to debug. The present function only plot the grid and output without more information.

#### Parameters

- **point\_all\_reach** – the coordinate of the point. This is given by reaches.
- **ikle\_all** – the connectivity table. This is given by reaches.
- **inter\_vel\_all** – the velocity data. This is given by reaches.
- **inter\_h\_all** – the height data. This is given by reaches.
- **path\_im** – the path where the figure should be saved
- **merge\_case** – If True, we plot data from grid with merged substrate and hydrological data
- **time\_step** – time step to be added to the title

**Param** `fig_opt`: the dictionary with the different options to create the figures

```
src.manage_grid_8.update_coord_pro_with_vh_pro(coord_pro, vh_pro_t)
```

The points describing the profile elevation and the points where velocity is measured might not be the same. Additionally, part of the profile might be dry and we have added points giving the wetted limit in `vh_pro_t`. They were are not in the original profil (`coord_pro`). In this function, `coord_pro` is recalculated to account for these modifications. It is used by `create_grid()` and `create_grid_one_profile`, but only if `vh_pro_t` exists.

**Parameters**

- **coord\_pro** – the original coord\_pro
- **vh\_pro\_t** – the value and position of h and velocity measurement with the river limits

**Returns** updated coord\_pro

More information in the document “More info on the grid” (linked above)

## 4.11 Estimhab -source

in src/estimhab.py

The module contains the Estimhab model. For an explanation on the estimhab model, please see the pdf document `estimhab2008`

```
src.estimhab.estimhab(qmes, width, height, q50, qrange, substrat, path_bio, fish_name, path_im,
                        pict=False, fig_opt={})
```

This the function which forms the Estimhab model in HABBY. It is a reproduction in python of the excel file which forms the original Estimhab model.. Unit in meter and m<sup>3</sup>/sec

**Parameters**

- **qmes** – the two measured discharge
- **width** – the two measured width
- **height** – the two measured height
- **q50** – the natural median discharge
- **qrange** – the range of discharge
- **substrat** – mean height of substrat
- **path\_im** – the path where the image should be saved
- **path\_bio** – the path to the xml file with the information on the fishes
- **fish\_name** – the name of the fish which have to be analyzed
- **pict** – if true the figure is shown. If false, the figure is not shown
- **fig\_opt** – a dictionary with the figure option

**Returns** habitat value and useful surface (VH and SPU) as a function of discharge

**Technical comments and walk-through**

First, we get all the discharges on which we want to calculate the SPU (surface ponderée utile), using the inputs from the user.

Next we use hydrological rating curves (info on google if needed) to get the height and the width of the river for all discharge. The calculation is based on the width and height of the river measured at two discharges (given by the user).

Next, we get other parameters which are used in the preference curves such as the Froude number of the mean discharge or the Reynolds number.

Next, we load the fish data contains in the xml files in the biology folder. Careful, this is not the xml project file. This are the xml files described above in the “Class EstimhabW” section. There are one xml file per fish and they described the preference curves. For the argumentation on the form of the relationship, report yourself to the documentation of Estimhab (one pdf file should in the folder “doc “ in HABBY).

Then, we calculate the habitat values (VH and SPU). Finally, we plot the results in a figure and we save it as a text file.

```
src.estimhab.main()
```

Used to test this module.

```
src.estimhab.pass_to_float_estimhab(var_name, root)
```

This is a function to pass from an xml element to a float

#### Parameters

- **root** – the root of the open xml file
- **var\_name** – the name of the attribute in the xml file

**Returns** the float data

## 4.12 Stathab - source

in src/stathab\_c

This module contains the function used to run the model stathab. For an explanation on the form of the stathab input, please see the pdf document `stathabinfo`

```
class src.stathab_c.Stathab(name_prj, path_prj)
```

The class for the Stathab model

```
create_hdf5()
```

A function to create an hdf5 file from the loaded txt. It creates “name\_prj”\_STATHAB.h5, an hdf5 file with the info from stathab

```
dengauss(x)
```

gaussian density, used only for debugging purposes. This is not used in Habby, but can be useful if scipy is not available (replace all `stat.norm.cdf` with `dengauss`)

**Parameters** **x** – the parameter of the gaussian

**Returns** the gaussian density

```
dist_h(sh0, h0, bornh, h)
```

The calculation of height distribution across the river. The distribution is a mix of an exponential and gaussian.

#### Parameters

- **sh0** – the sh of the original data sh is the parameter of the distribution, gives the relative importance of gaussian and exp distribution
- **h** – the mean height data
- **h0** – the mean height
- **bornh** – the limits of each class of height

**Returns** disth the distribution of heights across the river for the mean height h.

```
dist_h_trop(v, h, mean_slope)
```

This function calculates the height distribution for steep tropical stream based on the R code from Girard et al. (`stathab_hyd_steep`). The frequency distribution is based on empirical data which is given in the list of numbers in the codes below. The final frequency distribution is in the form:  $t \cdot x f_1 + (1-t) \cdot x f$  where  $t$  is a function of the froude number and the mean slope of the station.

The height limits are considered constant here (contrarily to `dist_h` where they are given in the parameter `bornh`).

#### Parameters

- **v** – the velocity for this discharge
- **h** – the height for this discharge
- **mean\_slope** – the mean slope of the station (usually in the `data_ii[0]` variable)

**Returns** the distribution of height

**dist\_v** (*h, d, bornv, v*)

The calculation of velocity distribution across the river. The distribution is a mix of an exponential and gaussian.

#### Parameters

- **h** – the height which is related to the mean velocity *v*
- **d** – granulo moyenne
- **bornv** – the born of the velocity
- **v** – the mean velocity

**Returns** the distribution of velocity across the river

**dist\_v\_trop** (*v, h, h\_waterfall, length\_stat*)

This function calculate the velocity distribution for steep tropical stream based on the R code from Girard et al. (`stathab_hyd_steep`). The frequency distribution is based on empirical data which is given in the list of numbers in the codes below. The final frequency distribution is in the form:  $t \times f1 + (1-t) \times f$  where *t* depends on the ratio of the length of station and the height of the waterfall.

#### Parameters

- **v** – the velocity for this discharge
- **h** – the height for this discharge
- **h\_waterfall** – the height of the waterfall
- **length\_stat** – the length of the station

**Returns** the distribution of velocity

**find\_path\_hdf5\_stat** ()

A function to find the path where to save the hdf5 file. Careful a similar one is in `hydro_GUI_2.py` and in `estimhab_GUI`. By default, `path_hdf5` is in the project folder in the folder 'fichier\_hdf5'.

**find\_sh0** (*disthmesr, h0*)

the function to find `sh0`, using a minimization technique. Not used because the output was string. Possibly an error on the bornes? We replaced this function by the function `find_sh0_maxvrais()`.

#### Parameters

- **disthmesr** – the measured distribution of height
- **h0** – the measured mean height

**Returns** the optimized `sh0`

**find\_sh0\_maxvrais** (*disthmesr, h0*)

the function to find `sh0`, using the maximum of vraisemblance. This function aims at reproducing the results from the c++ code. Hence, no use of `scipy`

**Parameters**

- **disthmesr** – the measured distribution of height
- **h0** – the measured mean height

**Returns** the optimized sh0

**load\_stathab\_from\_hdf5()**

A function to load the file from an hdf5 whose name is given in the xml project file. If the name of the file is a relative path, use the path\_prj to create an absolute path.

It works for tropical and temperate rivers. It checks the river type in the hdf5 files and used this river type regardless of the one currently used by the GUI. The method load\_hdf5 in stathab\_GUI get the value of self.riverint from the object mystathab to check the coherence between the GUI and the loaded hdf5.

**load\_stathab\_from\_txt(reachname\_file, end\_file\_reach, name\_file\_allreach, path)**

A function to read and check the input from stathab based on the text files. All files should be in the same folder. The file Pref.txt is read in run\_stathab. If self.fish\_chosen is not present, all fish in the preference file are read.

**Parameters**

- **reachname\_file** – the file with the name of the reaches to study (usually listirv)
- **end\_file\_reach** – the ending of the files whose names depends on the reach (with .txt or .csv)
- **name\_file\_allreach** – the name of the file common to all reaches
- **path** – the path to the file

**Returns** the inputs needed for run\_stathab

**power\_law(qwh\_r)**

The function to calculate power law for discharge and width  $\ln(h_0 = a_1 + a_2 \ln(Q))$

**Parameters** **qwh\_r** – an array where each line in one observatino of Q, width and height

**Returns** the coeff of the regression

**save\_xml\_stathab(no\_hdf5=False)**

The function which saves the function related to stathab in the xml projext files

**Parameters** **no\_hdf5** – If True, no hdf5 file was created (usually because Stathab crashed at some points)

**savefig\_stahab(show\_class=True)**

A function to save the results in text and the figure. If the argument show\_class is True, it shows an extra figure with the size of the different height, granulo, and velocity classes. The optional figure only works when stathab1 for temperate river is used.

**savetxt\_stathab()**

A function to save the stathab result in .txt form

**stathab\_calc(path\_pref='.', name\_pref='Pref.txt')**

The function to calculate stathab output.

**Parameters**

- **path\_pref** – the path to the preference file
- **name\_pref** – the name of the preference file

**Returns** the biological preference index (np.array of [reach, specices, nbclaq] size), surface or volume by class, etc.

**stathab\_trop\_biv** (*path\_bio*)

This function calculate the stathab outputs for the bivariate preference file in the case where the river is steep and in the tropical regions (usually the islands of Reunion and Guadeloupe).

**Parameters** *path\_bio* –

**Returns**

**stathab\_trop\_univ** (*path\_bio*, *by\_vol*)

This function calculate the stathab outputs for the univariate preference file in the case where the river is steep and in the tropical regions (usually the islands of Reunion and Guadeloupe).

**Parameters**

- **path\_bio** – the path to the preference file usually biology/stathab
- **by\_vol** – If True the output is by volum (VPU instead of SPU) from the velcoity pref file

**Returns** the SPU or VPU

**test\_stathab** (*path\_ori*)

A short function to test part of the outputs of stathab in temperate rivers against the C++ code, It is not used in Habby but it is practical to debug.

**Parameters** *path\_ori* – the path to the files from stathab based on the c++ code

**test\_stathab\_trop\_biv** (*path\_ori*)

A short function to test part of the outputs of the stathab tropical rivers against the R code in the bivariate mode. It is not used in Habby but it is practical to debug. Stathab\_trop+biv should be executed before. For the moment only the fish SIC is tested.

**Parameters** *path\_ori* – the path to the output files from stathab based on the R code

**test\_stathab\_trop\_uni** (*path\_ori*, *by\_vel=True*)

A short function to test part of the outputs of the stathab tropical rivers against the R code in the univariate mode. It is not used in Habby but it is practical to debug. Stathab\_trop\_uni should be executed before. For the moment only the fish SIC is tested.

**Parameters**

- **path\_ori** – the path to the output files from stathab based on the R code
- **by\_vel** – If True, the velcoity-based vpu is used. Otherise, it is height-based spu

**src.stathab\_c.load\_float\_stathab** (*filename*, *check\_neg*)

A function to load float with extra checks

**Parameters**

- **filename** – the file to load with the path
- **check\_neg** – if true negative value are not allowed in the data

**Returns** data if ok, -99 if failed

**src.stathab\_c.load\_namereach** (*path*, *name\_file\_reach='listriv'*)

A function to only load the reach names (useful for the GUI). The extension must be .txt or .csv

:param *path* : the path to the file listriv.txt or listriv.csv :param *name\_file\_reach*: In case the file name is not listriv :return: the list of reach name

**src.stathab\_c.load\_pref** (*filepref*, *path*)

The function loads the different pref coefficient contained in filepref, for the temperate river from Stathab

**Parameters**

- **filepref** – the name of the file (usually Pref.txt)
- **path** – the path to this file

**Returns** the name of the fish, a np.array with the differen coeff

`src.stathab_c.load_pref_trop_biv (code_fish, path)`

This function loads the bivariate preference files for tropical rivers. The name of the file must be in the form of `xbiv_XXX.csv` where XXX is the three-letters fish code and x is whatever string.

**Parameters**

- **code\_fish** – the code for the fish name in three letters (such as ASC)
- **path** – the path to files

**Returns** the bivariate preferences

`src.stathab_c.load_pref_trop_uni (code_fish, path)`

This function loads the preference files for the univariate data. The file with the univariate data should be in the form of `xuni-h_XXX` where XX is the fish code and x is whatever string. The assumption is that the filename for velocity is very similar to the filename for height. In more detail that the string uni-h is changed to uni-v in the filename. Otherwise, the file are csv file with two columns: First is velocity or height, the second is the preference data.

**Parameters**

- **code\_fish** – the code for the fish name in three letters (such as ASC)
- **path** – the path to files

**Returns** the height data and velcoity data (h, pref) and (v,pref)

`src.stathab_c.main ()`

used to test this module.

## 4.13 FStress - source

in `src/fstress.py`

This module contains the function used to run the model FStress. For an explanation on the form of the FStress text input, please see the last page of the pdf document `stathabinfo`

`src.fstress.denstress (k, m, nbst)`

This function calculates the stress distribution function for FStress. This distribution has generally the form of  $k \cdot \exp() + (1-k) \cdot \text{Sigma}(x-m)$

**Parameters**

- **k** – the first parameter of the distribution
- **m** – the second parameter of the disitribution
- **nbst** – the number of stress class in the distribution

**Returns** the stress disitrbution for the (m,k) parameters

`src.fstress.figure_fstress (qmod_all, vh_all, name_inv, path_im, name_river, fig_opt={})`

This function creates the figures for Fstress, notably the suitability index as a function of discharge for all rivers

**Parameters**

- **qmod\_all** – the modelled discharge for each river



- **vh\_all** – the suitability index for each invertebrate species for each river
- **name\_inv** – The four letter code of each selected invertebrate
- **path\_im** – the path where to save the figure
- **name\_river** – the name of the river
- **fig\_opt** – the figure option in a dictionary

`src.fstress.fstress_test (qmod_all, vh_all, name_inv, name_river, path_rre)`

This function compares the output of the C program of FStress and the output of this script. It is not used by HABBY, but it is practical to debug.

#### Parameters

- **qmod\_all** – the modelled discharge for each river
- **vh\_all** – the suitability index for each invertebrate species for each river
- **name\_inv** – The four letter code of each selected invertebrate
- **name\_river** – the name of the river
- **path\_rre** – the path to the C output

`src.fstress.func_stress (vm, h, tau)`

This function calculates the distribution of stress on the bottom of the river based on height and velocity at one discharge. In other words, it calculates the distribution of the “hemispheres”. This function is mainly a copy of the stress function contained in the `vites2.c` of the C source of FStress.

#### Parameters

- **vm** – the velocity for this discharge value
- **h** – the height for this discharge value
- **tau** – the constraint values

**Returns** the stress distribution for this discharge

`src.fstress.main ()`

This is not the `main()` of HABBY. This local function is used to test the FStress model.

`src.fstress.read_fstress_hdf5 (hdf5_name, hdf5_path)`

This function reads an hdf5 file related to FStress and extracts the relevant information.

#### Parameters

- **hdf5\_name** – the name of the hdf5 file with the information related to FStress
- **hdf5\_path** – the path to this file

**return:** `[[q,w,h], [q,w,h]]` for each river, `[qmin,qmax]` for each river, the river names, and the selected fish

`src.fstress.read_pref (path_bio, name_bio)`

This function loads and reads the preference file for FStress.

#### Parameters

- **path\_bio** – the path to the preference file
- **name\_bio** – the name of the preference file

**Returns** the name invertebrate and their preference coefficient

```
src.fstress.run_fstress(data_hydro, qrange, riv_name, inv_select, pref_all, name_all, name_prj,  
                        path_prj)
```

This function run the model FStress for HABBY. FStress is based on the model of Nicolas Lamouroux. This model estimates suitability indices for invertebrate in relation with shear stress distributions. However, shear stress do not needs to be measured. It is statistically estimated based on velocity and height measurement.

#### Parameters

- **riv\_name** – the name of the river-> string
- **data\_hydro** – the hydrological data (q,w,h for each river in riv name) -> list of list
- **qrange** – the qmin and qmax for each river [qmin,qmax] -> list of list
- **inv\_select** – the name of the selected invetebate
- **pref\_all** – the preference data for all invertebrate
- **name\_all** – the four letter code of all possible invertebrate
- **path\_prj** – the path to the project-> string
- **name\_prj** – the name of the project-> string

```
src.fstress.save_fstress(path_hdf5, path_prj, name_prj, name_bio, path_bio, riv_name,  
                        data_hydro, qrange, fish_list)
```

This function saves the data related to the fstress model in an hdf5 file and write the name of this hdf5 file in the xml project file.

#### Parameters

- **path\_hdf5** – the path where to sdave the hdf5-> string
- **path\_prj** – the path to the project-> string
- **name\_prj** – the name of the project-> string
- **name\_bio** – the name of the preference file-> string
- **path\_bio** – the path to the preference file-> string
- **riv\_name** – the name of the river-> string
- **data\_hydro** – the hydrological data (q,w,h for each river in riv name) -> list of list
- **qrange** – the qmin and qmax for each river [qmin,qmax] -> list of list
- **fish\_list** – the name of the selected invertebrate (! no fish) -> list of string

```
src.fstress.write_txt(qmod_all, vh_all, name_inv, path_txt, name_river)
```

This function writes the txt outputs for FStress

#### Parameters

- **qmod\_all** – the modelled discharge for each river
- **vh\_all** – the suitability indoex for each invertebrate species for each river
- **name\_inv** – The four letter code of each selected invetebate
- **path\_txt** – the path where to save the text file
- **name\_river** – the name of the river

## 4.14 Substrate

in `src/substrate.py`

This module contains the function to load and manage the substrate data.

```
src.substrate.create_dummy_substrate_from_hydro(h5name, path, new_name, code_type,
                                                attribute_type, nb_point=200,
                                                path_out='.')
```

This function takes an hydrological hdf5 as inputs and create a shapefile of substrate and a text file of substrate which can be used as input for habby. The substrate data is random. So it is mainly useful to test an hydrological input.

The created shape file is rectangular with a size based on min/max of the hydrological coordinates. The substrate grid is not the same as the hydrological grids (which is good to test the programm). In addition, one side of the shp is smaller than the hydrological grid to test the ‘default substrate’ option (which is used if the substrate shapefile is not big enough).

### Parameters

- **h5name** – the name of the hydrological hdf5 file
- **path** – the path to this file
- **new\_name** – the name of the create shape file without the shp (string)
- **code\_type** – the code type for the substrate (Sandre, Cemagref or EDF)
- **attribute\_type** – if the substrate is given in the type coarser/dominant/(0) or in percentage (1)
- **nb\_point** – the number of point needed (more points results in smaller substrate form)
- **path\_out** – the path where to save the new substrate shape

```
src.substrate.edf_to_cemagref(records)
```

This function passes the substrate data from the code type ‘EDF’ to the code type ‘Cemagref’. As the code 1 from EDF can be the code 2 or code 1 in Cemagref, a code 1 in EDF is code 1 in Cemagref half of the time and code 2 the other half of the time. This function is not optimized yet. For the definition of the code, see the tabular at the 14 of the LAMMI manual. THIS IS NOT RIGHT AS CLASS ARE NOT SPEAREATED IDENTICALLY, TO BE CORRECTED !!!

**Parameters** **records** – the substrate data in code edf

**Returns** the substrate data in code cemagref

```
src.substrate.fig_substrate(coord_p, ikle, sub_pg, sub_dom, path_im, fig_opt={}, xtxt=[-99],
                           ytxt=[-99], subtxt=[-99])
```

The function to plot the substrate data, which was loaded before. This function will only work if the substrate data is given using the cemagref code.

### Parameters

- **coord\_p** – the coordinate of the point
- **ikle** – the connectivity table
- **sub\_pg** – the information on substrate by element for the ‘coarser part’
- **sub\_dom** – the information on substrate by element for the ‘dominant part’
- **fig\_opt** – the figure option as a doctionnary
- **xtxt** – if the data was given in txt form, the original x data

- **ytxt** – if the data was given in txt form, the original y data
- **subtxt** – if the data was given in txt form, the original sub data
- **path\_im** – the path where to save the figure

`src.substrate.get_all_attribute(filename, path)`

This function open a shapefile and get the list of all the attribute which is contains in it.

#### Parameters

- **filename** – the name of the shpae file
- **path** – the path to this shapefile

**Returns** a list with the name and information of all attribute. The form of each element of the list is [name(str), type (F,N), int, int]

`src.substrate.get_useful_attribute(attributes)`

This function find if the substrate was given in percentage or coarser/dominant/accesory and get the attribute or header name useful to load the substrate. If the subtrate is given by percentage, it is important to get attribute in order (S1, S2, S3) and not (S3, S1,S2) for this function and the next functions.

**Parameters** **attributes** – The list of attribute from the shp file or the header list from the text file

**Returns** The attribute type as int (percentage=1 and coarser/... = 0 and failed = -99) and the attribute names If we are in the attribute type 0, attribute list is in the order of: coarser, dominant, and accessory

`src.substrate.load_sub_shp(filename, path, code_type, dominant_case=0)`

A function to load the substrate in form of shapefile.

#### Parameters

- **filename** – the name of the shapefile
- **path** – the path where the shapefile is
- **code\_type** – the type of code used to define the substrate (string)
- **dominant\_case** – an int to manage the case where the transformation form percentage to dominnat is unclear (two maximum percentage are equal from one element). if -1 take the smallest, if 1 take the biggest, if 0, we do not know.

**Returns** grid in form of list of coordinate and connectivity table (two list) and an array with substrate type and a boolean which allows to get the case where we have tow dominant case

`src.substrate.load_sub_txt(filename, path, code_type)`

A function to load the substrate in form of a text file. The text file must have 4 columns x,y coordinate and xoarser substrate type, dominant substrate type, no header or title. It is transform to a grid using a voronoi transformation.

#### Parameters

- **filename** – the name of the shapefile
- **path** – the path where the shapefile is
- **code\_type** – the type of code used to define the substrate (string)

**Returns** grid in form of list of coordinate and connectivity table (two list) and an array with substrate type and (x,y,sub) of the original data

`src.substrate.main()`

Used to test this module.

`src.substrate.open_shp(filename, path)`

This function open a ArcGIS shpaefile.

#### Parameters

- **filename** – the name of the shapefile
- **path** – the path to this shapefile

**Returns** a shapefile object as define in the model

`src.substrate.sandre_to_cemagref(records)`

This function passes the substrate data from the code type “Sandre” to the code type “Cemagref”. This function is not optimized yet. For the definition of the code, see the tabular at the 14 of the LAMMI manual.

**Parameters** **records** – the substrate data in code samdre

**Returns** the substrate data in code cemagref

## 4.15 Merge the grid

in `src/mesh_grid2.py`

This module contains the function to merge the substrate grid and the hydrological grid. It cut the hydrological grid to the form of each of the substrate. An important hypothesis is that each polygon forming the substrate should convex.

`src.mesh_grid2.create_merge_grid(ikle, coord_p, data_sub_pg, data_sub_dom, vel, height, ikle_sub, default_data, data_crossing, sub_cell)`

A function to update the grid after finding the crossing points. It also get the `substrate_data` for each cell of the new grid.

#### Parameters

- **ikle** – the hydrological grid to be merge with the substrate grid
- **coord\_p** – the coordinate of the point of the hydrological grid
- **data\_sub\_pg** – the coarser substrate data by hydrological cell (3 information by cells realted to the three points)
- **data\_sub\_dom** – the dominant substrate data by hydrological cell (3 information by cells realted to the three points)
- **vel** – the velocity (one time step, one reach) for each point in `coord_p`
- **height** – the water height (one time step, one reach) for each point in `coord_p`
- **ikle\_sub** – the connectivity table for the substrate
- **default\_data** – the default substrate data
- **data\_crossing** – the hydrological element with a crossing and the info for this crossing (a list of list)

**Returns** the new grid

#### Technical comments

This function corrects all element of the grids where a crossing point have been found by the function `find_sub_and_cross()`

There are three cases:

- 1.one crossing point -> no change

2.two crossing points and substrate point inside -> done manually. We take the two crossing point and the side on which the crossing is done. Based on this, we correct the grid.

3.more than two crossing point on the elements -> We call the external module triangle to re-do some triangulations into the element. This last case covers many possible cases, but it is slow. To optimize, we can think about writing more individual cases. To follow the border of each substrate cell in the “special cell”, we do one triangulation by substrate element, so we can have two or three triangulation. It is also important there that the substrate is convex as the triangulation is not constrained.

```
src.mesh_grid2.fig_merge_grid(point_all_both_t,    ikle_both_t,    path_im,    ikle_orr=[],  
                               point_all_orr=[])
```

A function to plot the grid after it was merged with the substrate data. It plots one time step at the time.

#### Parameters

- **point\_all\_both\_t** – the coordinate of the points of the updated grid
- **ikle\_both\_t** – the connectivity table
- **path\_im** – the path where the image should be saved
- **ikle\_orr** – the original ikle
- **point\_all\_orr** – the original point\_all

```
src.mesh_grid2.find_sub_and_cross(ikle_sub, coord_p_sub, ikle, coord_p, data_sub_pg,  
                                   data_sub_dom)
```

A function which finds where the crossing points are. Crossing points are the points on the triangular side of the hydrological grid which cross with a side of the substrate grid. The algorithm is based on finding if points of one element are in the same polygon using a ray casting method. We assume that the polygon forming the substrate grid are convex. Otherwise it would not work in all cases. We could do a small function to test or correct this. We also neglect the case where a substrate cell at the border of the substrate grid is fully in a hydrological cell.

IMPORTANT: polygon should be convex.

#### Parameters

- **ikle\_sub** – the connectivity table for the substrate
- **coord\_p\_sub** – the coordinates of the points forming the substrate
- **ikle** – the connectivity table for the hydrology
- **coord\_p** – the coordinate of the hydrology
- **data\_sub\_dom** – the substrate data by substrate cell (dominant)
- **data\_sub\_pg** – the substrate data by substrate cell (coarser)

**Returns** the new substrate grid (ikle\_sub, coord\_p\_sub, data\_sub\_pg, data\_sub\_dom, sub\_cell), the data for the crossing point (hydrological element with a crossing, crossing point, substrate element linked with the crossing point, point of substrate inside, substrate element linked with the substrate point, side of the crossing points, substrate element link with hydro\_point).

```
src.mesh_grid2.get_new_vel_height_data(newp, point_old, data_old)
```

This function gets the height and velocity data for a new point in or on the side of an element. It does an average of the data (velocity or height) given at the node of the original (old) elements. This average is weighted as a function of the distance of the point.

#### Parameters

- **newp** – the coordinates of the new points
- **point\_old** – the coordinates of the three old points (would work with more than three)
- **data\_old** – the data for the point in point\_old

**Returns** the new data

`src.mesh_grid2.inside_trigon(pt, p0, p1, p2)`

This function check if a point is in a triangle using the barycentric coordinates.

**Parameters**

- **pt** – the point to determine if it is in or not
- **p0** – the first point of triangle
- **p1** – the second point of triangle
- **p2** – the third point of triangle

**Returns** A boolean (Ture if pt inside of triangle)

`src.mesh_grid2.intersec_cross(hyd1, hyd2, sub1, sub2)`

A function function to calculate the intersection, segment are not parrallel, used in case where we know that the intersection exists

**Parameters**

- **hyd1** – the first hydrological point
- **hyd2** – the second
- **sub1** – the first substrate point
- **sub2** – the second

**Returns** intersection

`src.mesh_grid2.main()`

Used to test this module.

`src.mesh_grid2.merge_grid_and_save(hdf5_name_hyd, hdf5_name_sub, path_hdf5, default_data, name_prj, path_prj, model_type, q=[], print_cmd=False)`

This function call the merging of the grid between the grid from the hydrological data and the substrate data. It then save the merged data and the substrate data in a common hdf5 file. This function is called in a second thread to avoid freezin gthe GUI. This is why we have this extra-function just to call `save_hdf5()` and `merge_grid_hydro_sub()`.

**Parameters**

- **hdf5\_name\_hyd** – the name of the hdf5 file with the hydrological data
- **hdf5\_name\_sub** – the name of the hdf5 with the substrate data
- **path\_hdf5** – the path to the hdf5 data
- **default\_data** – The substrate data given in the region of the hydrological grid where no substrate is given
- **name\_prj** – the name of the project
- **path\_prj** – the path to the project
- **model\_type** – the type of the “model”. In this case, it is just ‘SUBSTRATE’
- **q** – used to share info with the GUI when this thread have finsihed (`print_cmd = False`)
- **print\_cmd** – If False, print to the GUI (usually False)

`src.mesh_grid2.merge_grid_hydro_sub(hdf5_name_hyd, hdf5_name_sub, path_hdf5, default_data=1, path_prj='')`

After the data for the substrate and the hydrological data are loaded, they are still in different grids. This

functions will merge both grid together. This is done for all time step and all reaches. If a constant substrate is there, the hydrological hdf5 is just copied.

#### Parameters

- **hdf5\_name\_hyd** – the name of the hdf5 file with the hydrological data
- **hdf5\_name\_sub** – the name of the hdf5 with the substrate data
- **path\_hdf5** – the path to the hdf5 data
- **default\_data** – The substrate data given in the region of the hydrological grid where no substrate is given
- **path\_prj** – the path to the project

**Returns** the connectivity table, the coordinates, the substrated data, the velocity and height data all in a merge form.

## 4.16 Biological Info

in `src/bio_info.py`

This module contains the script to show and manage the biological models (preference curves) which are in the biology folder.

`src.bio_info.change_unit (data, unit)`

This function modify the unit of the data to SI unit. Currently it accept the following unit : Centimeter, Meter, CentimeterPerSecond, MeterPerSecond, Millimeter, “Code EVHA 2.0 (GINOT 1998)”

#### Parameters

- **data** – the data which has to be change to SI unit
- **unit** – the unit code

`src.bio_info.create_and_fill_database (path_bio, name_database, attribute)`

This function create a new database when Habby starts. The goal of creating a database is to avoid freezing the GUI when info on the preference curve are asked. So it is possible to select one curve and have information without seeing too much of a delay.

This is not used anymore by HABBY as the xml file is really small. It could however be useful if the xml file becomes too big. In this case, this function could be called if modification are found in the `pref_file` folder and would create a database.

The attribute can be modified, but they should all be of text type. It is also important to keep stage at the first attribute. The modified attribute should reflect the attribute of the xml file. If it not possible, lines should be added in the “special case” attributes”. The main table with the data is called `pref_bio`.

#### Parameters

- **path\_bio** – the path to the biological information (usually `./biology`)
- **name\_database** – the name of the database (string) without the path
- **attribute** – the attribute in the database (only text type)

**Returns** a boolean (True if everthing ok, False otherwise)

`src.bio_info.execute_request (path_bio, name_database, request)`

This function execute the SQL request given in the string called request. it saves the found data in a variable. The idea is to use this function for `SELELCT X FROM X WHERE ...` , not really to handle all possible request.



It also opens and close the database `name_database` to do this. This is not used anymore by HABBY as we do not use a database. It could however be useful if the xml file becomes too big.

#### Parameters

- **path\_bio** – the path to the biological information (usually `./biology`)
- **name\_database** – the name of the database (string) without the path
- **request** – the SQL request in a string form

**Returns** the result

`src.bio_info.figure_pref (height, vel, sub, code_fish, name_fish, stade, get_fig=False)`

This function is used to plot the preference curves.

#### Parameters

- **height** – the height preference data (list of list)
- **vel** – the height preference data (list of list)
- **sub** – the height preference data (list of list)
- **code\_fish** – the three letter code which indiate which fish species is analyzed
- **name\_fish** – the name of the fish analyzed
- **stade** – the name of the stade analyzed (ADU, JUV, ...)
- **get\_fig** – usually False, If True return the figure (to modified it more)

`src.bio_info.get_stage (names_bio, path_bio)`

This function loads all the stages present in a list of xml preference files (JUV, ADU, etc) and the latin name of the fish species. All the files should be in the same folder indicated by `path_bio`. It is mainly used by `habby_cmd` but it can be useful in other cases also.

#### Parameters

- **names\_bio** – A list of xml biological preference file
- **path\_bio** – the path to the xml preference files (usually `./biology`)

**Returns** the stages in a list of string

`src.bio_info.load_evha_curve (filename, path)`

This function is used to load the preference curve in the EVHA form . It will be useful to create xml preference file, but it is not used directly by HABBY. This function does not have much control on user input as it is planned to be used only by people working on HABBY. The order of the data in the file must be height, velocity, substrate

#### Parameters

- **filename** – the name of file containing the preference curve for EVHA
- **path** – the path to this file

**Returns** preference for height, vel, sub in a list of list form, name of the fish, code, stade and description

`src.bio_info.load_xml_name (path_bio, attributes)`

This function looks for all preference curves found in the `path_bio` folder. It extract the fish name and the stage. to be corrected if more than one language. The name of attribute\_acc should be coherent with the one from the xml file apart from the common name which should be in the form `language_common_name` (so we can write something in the GUI to get all language if we get something else than English or French)

#### Parameters

- **path\_bio** – the path to the biological function
- **attributes** – the list of attribute which should be possible to search from the GUI or, more generally which should be in data-fish which is returned.

**Returns** the list of stage/fish species with the info from [name for GUI, s, xmlfilename, attribute\_acc without s]

`src.bio_info.main()`

Used to test the module on the biological preference

`src.bio_info.plot_hydrosignature(xmlfile)`

This function plots the hydrosignature in the vclass and hclass given in the xml file. It does only work if: units are SI (meter and m/s) and if the order of data is ‘velocity increasing and then height of water increasing’.

**Parameters** **xmlfile** – the path and name of the xmlfile

`src.bio_info.read_pref(xmlfile)`

This function reads the preference curve from the xml file and get the substrate, height and velocity data. It return the data in meter. Unit for space can be in centimeter, milimeter or meter. Unit for time should be in second. The unit attribute of the xml files should be coherent with the data.

**Parameters** **xmlfile** – the path and name to the xml file (string)

**Returns** height, vel, sub, code\_fish, name\_fish, stade

`src.bio_info.test_evah_xml_pref(path_xml, path_evha)`

This function is used to visually compared the evha (.PRF) curve and the xml curve based on the evah curve. Obviously the xml file should contain the same data than the evha curve (when the xml file are based on the evah curve). An important assumption of this function is that the data in the xml file is in the order: fry, juvenile, adult.

**Parameters**

- **path\_xml** – the path to the folder which contains the xml files
- **path\_evha** – the path to the evha folder which contains the PRF files

## 4.17 Calculation habitat

in `src/calcul_hab.py`

This module calculates the habitat value for the fish and creates the outputs (text files, shapefile, and figures).

`src.calcul_hab.calc_hab(merge_name, path_merge, bio_names, stages, path_bio, opt)`

This function calculates the habitat value. It loads substrate and hydrology data from an hdf5 files and it loads the biology data from the xml files. It is possible to have more than one stage by xml file (usually the three stages are in the xml files). There are more than one method to calculate the habitat so the parameter `opt` indicate which metho to use. 0-> usde coarser substrate, 1 -> use dominant substrate

**Parameters**

- **merge\_name** – the name of the hdf5 with the results
- **path\_merge** – the path to the merged file
- **bio\_names** – the name of the xml biological data
- **stages** – the stage chosen (youngs, adults, etc.). List with the same length as `bio_names`.
- **path\_bio** – The path to the biological folder (with all files given in `bio_names`)
- **opt** – an int from 0 to n. Gives which calculation method should be used

**Returns** the habiatat value for all species, all time, all reach, all cells.

```
src.calcul_hab.calc_hab_norm(ikle_all_t, point_all_t, vel, height, sub, pref_vel, pref_height,
                             pref_sub)
```

This function calculates the habitat suitability index ( $f(H)xf(v)xf(sub)$ ) for each and the SPU which is the sum of all habitat suitability index weighted by the cell area for each reach. It is called by `clac_hab_norm`.

#### Parameters

- **ikle\_all\_t** – the connectivity table for all time step, all reach
- **point\_all\_t** – the point of the grid
- **vel** – the velocity data for all time step, all reach
- **height** – the water height data for all time step, all reach
- **sub** – the substrate data (can be coarser or dominant substrate based on function's call)
- **pref\_vel** – the preference index for the velocity (for one life stage)
- **pref\_sub** – the preference index for the substrate (for one life stage)
- **pref\_height** – the preference index for the height (for one life stage)

**Returns** vh of one life stage, area, habitat value

```
src.calcul_hab.find_pref_value(data, pref)
```

This function finds the preference value associated with the data for each cell. For this, it finds the last point of the preference curve under the data and it makes a linear interpolation with the next data to find the preference value. As preference value is sorted, it uses the module `bisect` to accelerate the process.

#### Parameters

- **data** – the data on the cell
- **pref** – the pref data [pref, class data]

```
src.calcul_hab.save_hab_fig_spu(area_all, spu_all, name_fish, path_im, name_base)
```

This function creates the figure of the spu as a function of time for each reach. if there is only one time step, it reverse to a bar plot. Otherwise it is a line plot.

#### Parameters

- **area\_all** – the area for all reach
- **spu\_all** – the “surface pondere utile” (SPU) for each reach
- **name\_fish** – the list of fish latin name + stage
- **path\_im** – the path where to save the image
- **name\_base** – a string on which to base the name of the files

```
src.calcul_hab.save_hab_shape(name_merge_hdf5, path_hdf5, vh_data, vel_data, height_data,
                             name_fish_sh, path_shp, name_base)
```

This function create the output in the form of a shapefile. It creates one shapefile by time step. It put all the reaches together. If there is overlap between reaches, it does not care. It create an attribute table with the habitat value, velocity, height, substrate coarser, substrate dominant. It also create a shapefile 0 with the whole profile without data.

The name of the column of the attribute table should be less than 10 character. Hence, the variable `name_fish` has been adapted to be shorter. The shorter `name_fish` is called `name_fish_sh`.

#### Parameters

- **name\_merge\_hdf5** – the name of the hdf5 merged file

- **path\_hdf5** – the path to the hydrological hdf5 data
- **vel\_data** – the velocity by reach by time step on the cell (not node!)
- **height\_data** – the height by reach by time step on the cell (not node!)
- **vh\_data** – the habitat value data by species by reach by time step
- **name\_fish\_sh** – the list of fish latin name + stage
- **path\_shp** – the path where to save the shp file
- **name\_base** – a string on which to base the name of the files

```
src.calcul_hab.save_hab_txt(name_merge_hdf5, path_hdf5, vh_data, vel_data, height_data,  
                           name_fish, path_txt, name_base)
```

This function prints the text output. We create one set of text files by time step. Each Reach is separated by the key word REACH followed by the reach number (starting from 0). There are three files by time steps: one file which gives the connectivity table (starting at 0), one file with the point coordinates in the coordinate systems of the hydraulic models (x,y), one file which gives the results. In all three files, the first column is the reach number. In the results files, the next columns are velocity, height, substrate, habitat value for each species.

#### Parameters

- **name\_merge\_hdf5** – the name of the hdf5 merged file
- **path\_hdf5** – the path to the hydrological hdf5 data
- **vel\_data** – the velocity by reach by time step on the cell (not node!)
- **height\_data** – the height by reach by time step on the cell (not node!)
- **vh\_data** – the habitat value data by species by reach by time step
- **name\_fish** – the list of fish latin name + stage
- **path\_txt** – the path where to save the text file
- **name\_base** – a string on which to base the name of the files

```
src.calcul_hab.save_spu_txt(area_all, spu_all, name_fish, path_txt, name_base)
```

This function creates a text file with the following columns: the time step, the reach number, the area of the reach and the spu for each fish species.

#### Parameters

- **area\_all** – the area for all reach
- **spu\_all** – the “surface pondere utile” (SPU) for each reach
- **name\_fish** – the list of fish latin name + stage
- **path\_txt** – the path where to save the text file
- **name\_base** – a string on which to base the name of the files

```
src.calcul_hab.save_vh_fig_2d(name_merge_hdf5, path_hdf5, vh_all_t_sp, path_im, name_fish,  
                             name_base, time_step=-1)
```

This function creates a 2D map of the habitat value for each species at the time step asked. All reaches are plotted on the same figure.

#### Parameters

- **name\_merge\_hdf5** – the name of the hdf5 merged file
- **path\_hdf5** – the path to the hydrological hdf5 data
- **vh\_all\_t\_sp** – the habitat value for all reach all time step all species

- **path\_im** – the path where to save the figure
- **name\_fish** – the name and stage of the studied species
- **name\_base** – the string on which to base the figure name
- **time\_step** – which time step should be plotted

## 4.18 Create Paraview Files

in `src/new_create_vtk.py` in `src/evtk.py` in `src/hl.py` in `src/vtk.py` in `src/xml_vk.py`

These modules contain the scripts to create Paraview input which are binary xml-based vtu files. This part is heavily based on the module `Pyevtk` created by Paulo Herrera (<https://bitbucket.org/pauloh/pyevtk>). Hence, the only script written for HABBY is in `new_create_vtk.py`. This script then called one function in `hl.py` which then called the three other scripts as in `Pyevtk` (not documented here).

```
src.new_create_vtk.habitat_to_vtu(file_name_base, path_out, path_hdf5, name_hdf5,
                                vh_all_t_sp, height_c_data, vel_c_data, name_fish, bi-
                                nary_data=False)
```

This function creates paraview input in the new, non-legacy xml format. This function called the `evtk` class written by Paulo Herrera, which is available at <https://bitbucket.org/pauloh/pyevtk/downloads/>

The format for paraview input file is described in `paraview_file_format.pdf` in the `doc` folder of HABBY.

The data of the paraview file created here is usually not in a binary form, but it could be put in binary if the switch `binary_data` is `True`. The idea of paraview for a binary file is to keep the same usual xml file. The data is encoded to base64. Before the binary array, there is a 32-bits integer which contains the data length in bytes. More info: <http://www.earthmodels.org/software/vtk-and-paraview/vtk-file-formats>.

Paraview can handle a group of file which compose an output with than one time step. This is the reason to create a `pvd` files which list the files composing all the time steps (one file by time step)

### Parameters

- **file\_name\_base** – the base to create the name of the vtu file
- **path\_hdf5** – the path to the hdf5 hydro file (to load the grid)
- **name\_hdf5** – the name of the hdf5 containing the grid
- **vh\_all\_t\_sp** – the habitat data by reach, time step, species
- **height\_c\_data** – the height by cell by reach by time step
- **vel\_c\_data** – the velocity by cell by reach by time step
- **name\_fish** – the name of fish and stage
- **path\_out** – the path where to save the data
- **binary\_data** – If `True` will create binary data. DO NOT WORK YET, let it to `FALSE`

```
src.new_create_vtk.main()
```

Used to test this module

```
src.new_create_vtk.writePVD(fileName, fileNames)
```

This function write the file which indicates to paraview how to group the file. This “grouping” file is `pvd` file. With this file, paraview can open all the time steps together with one clic. This function is heavily inspired by the class given at the adress: [https://github.com/cfinch/Shocksolution\\_Examples/blob/master/Visualization/vtktools.py](https://github.com/cfinch/Shocksolution_Examples/blob/master/Visualization/vtktools.py). Because `Element Tree` was not used in the class from `Shocksolution`, we use `minidom` here instead of `Element Tree`.

**Parameters**

- **fileName** – the name of the pvd file
- **fileNames** – the names of all the files for this time step (usually only for one reach but could be changed).

## 4.19 Function for the command lines

in `src/func_for_cmd.py`

This module contains functions which are only used by `habby_cmd.py`. Theoretically, it is not needed but it is sometimes practical. It is not great idea to add to many functions here however because it is more complicated to maintain.

`src.func_for_cmd.load_fstress_text(path_fstress)`

This function loads the data for `fstress` from text files. The data is composed of the name of the river, the discharge range, and the [discharge, height, width]. To read the files, the file `listriv.txt` is given. From then, the function looks for the other files in the same folder. The other files are `rivdeb.txt` and `rivqwh.txt`. If more than one river is given in `listriv.txt`, it loads the data for all rivers.

There is a very similar function as a method in the class `FStressW()` in `fstress_GUI.py` but it is used by the GUI and it includes a way to select the file using the GUI. Changes should be copied in both functions if necessary.

**Parameters** `path_fstress` – the path to the `listriv.txt` function (the other file should be in the same folder)

`src.func_for_cmd.load_manning_txt(filename_path)`

This function loads the manning data in case where manning number is not simply a constant. In this case, the manning parameter is given in a .txt file. The manning parameter used by 1D model such as mascaret or Rubar BE to distribute velocity along the profiles. The format of the txt file is “p, dist, n” where p is the profile number (start at zero), dist is the distance along the profile in meter and n is the manning value (in SI unit). White space is neglected and a line starting with the character # is also neglected.

There is a very similar function as a method in the class `Sub_HydroW()` in `hydro_GUI.py` but it is used by the GUI and it includes a way to select the file using the GUI and it used a lot of class attribute. So it cannot be used by the command line. Changes should be copied in both functions if necessary.

**Parameters** `filename_path` – the path and the name of the file containing the manning data

**Returns** the manning as an array form

## VARIOUS NOTES

### 5.1 The legend of the plots are not shown.

Generally HABBY is able to save the figure while showing the legend (which is often outside of the figure) appropriately. However, the figure shown to the user by matplotlib often have the legend outside of the visible area. To see the figure fully, one can modify the axes in the option of the figure (the menu in the axes on the top of the figure).

### 5.2 Translation of HABBY

In HABBY, it is possible to translate all strings which are in a python file (.py) which is in the src\_GUI folder. It should be possible to translate also strings which are in a .py file which is in the .src folder if one modifies the .pro file, but this is not done yet. Also, it might not be necessary because ./src contains code which is not linked with the graphical interface. Hence, English might be sufficient here.

To add a new string to translate:

- Code as usual and write the string in English.
- Add `self.tr()` around the string `a = QLabel(self.tr("My message"))`
- If the code is in a new python file (like the .py was just created), open the `habby_trans.pro` file which is `src_GUI`. Then add the line `SOURCES+= new_file.py` where `new_file.py` is the new python file.
- If you want to add a new language, add the line `TRANSLATIONS += Zen_ES.ts` in the case you want to add Spanish or any other language.
- Copy the files `ZEN_EN.ts` and `ZEN_FR.ts` from `HABBY/translation` to `/src_GUI`
- In the `src_GUI` folder, run the following command on the cmd: `pylupdate5 habby_trans.pro`. it will work if `pylupdate` is installed.
- It should update the .ts file (which is an xml file)
- Copy both .ts file back to `HABBY/translation`
- Open Qt linguist. This is a program that you need to install before. Open the French .ts file. The English should not need translation.
- Translate as needed and save in Qt Linguist.
- A .qm file is the binary representing the .ts file with all the translation. To create .qm file, type (in the cmd) `lrelease file.ts`. It will create a file.qm file
- Run HABBY. The string should be updated.

### In the code

If the user asked for a new language, we need to reload the translator with the following lines:

```
app = QApplication.instance()
self.languageTranslator.load(file.qm, self.path_trans)
app.installTranslator(self.languageTranslator)
```

with the appropriate name for “file.qm”.

In HABBY, the list of the name of all qm file are in the variable `self.file_langue` in class `MainWindows`. Hence, we can follow the selected language using an integer `self.lang` (0 for English and 1 for French). We can now call `self.file_langue[self.lang]` to get the qm file in the right language. If a new language is added, it is necessary to add one string to this list and to modify the menu. If the new language is also present in the xml preference file (which contains the biological info), it is also necessary to update the variable “`bioinfo_tab.lang`” from `central_widget` in the function `setlangue` from `Main_Windows()`. If this is done, the description of the xml preference file in the “Biology” tab will be shown in the selected language. Otherwise, it will be the first language found.

When the translator has been created, it is necessary to re-do all Widgets and Windows. This is not a problem when we open HABBY, but it can be a bit of work if the user asks for a change in language when HABBY is running. This is the function of the `setlangue` function. This function would work for all language (it takes an integer as input to know which language to use), but it needs to be modified if one modifies the `Main_Windows` Class strongly (notably if one add signals). The language should be saved in the user setting using `Qsettings` as it is done at the end of the `setlangue` function.

## 5.3 Create a .exe

Here are step to create a .exe using PyInstaller:

- install Pyinstaller (pip install pyinstaller)
- cd “folder with source code”
- pyinstaller.exe [option] habby.py, with the option `–onefile` to get only one .exe and `–windowed` to not have the cmd which opens with the application.

Here are some common problems:

- `ImportError: (No module named ‘PyQt5.QtGui’)`: Copy the folder platform with `qwindows.dll` and add to the `set_up.py` “includes”: [“PyQt5.QtCore”, “PyQt5.QtGui”]
- This application fails to start because ... the Qt platform pugin windows: Copy the folder platform with `qwindows.dll` in it
- `ImportError: h5Py` “includes”: [“h5py”, “h5py.defs”, “h5py.utils”, “h5py.h5ac”, “h5py.\_proxy” ] etc if necessary
- Intel MKL fatal error copy the .dll missing (or just find an old dist and copy all mkl stuff) AND the `libomp5md.dll`
- The translation does not work: Add the translation folder into the dist folder
- Do not find `log0.txt` (or crash when saving project): create a folder called `src_GUI`, copy the files `log0.txt` and `restart_log0.txt` from the `src_GUI` folder in the python module

Practically:

- go to the folder called executable
- run `createexe.bat` (only on Windows)
- copy all files in the `mklall` folder to the folder `dist/habby`



- run habby by writing habby on the cmd
- test and correct problems. It can be long!

## 5.4 Logging

### General information

There are two different logs for HABBY. By default, the first one is called “name\_project”.log and the second is called restart/\_’name\_project’.log. Their name and path can be changed in the xml project file. Both file are text file.

The first log is in the form of a python file with comments. If python and the necessary modules are installed on the machine, this log can be renamed “name.py” and started as a python file. In the command line, the following command should be used: python name.py. This file can be modified to create a new script to use HABBY in a different ways. For this, python syntax should be used.

The second log, called restart/\_’name\_project’.log, has limited functionalities but allows to re-start the HABBY simulation from the command line, without the need for python. Format of this file is described below. It is aimed to be readable and easily modifiable. To use the restart file, type in the command line: python habby\_cmd.py restart/\_’name\_project’.log.

This part genreally needs more revisions and tests.

### Type of log and format

Currently, there are five types of outputs, which can be sent to the log:

- Comment, which should start with #. They will be sent to the python-type log file and to the GUI of Habby.
- Errors, which should start with word “Error”. They will be sent to the python-type log file and to the GUI of Habby. In the GUI, they will appear in red.
- Warnings, which should start with the word “Warning”. They will be sent to the python-type log file and to the GUI of Habby. In the GUI, they will appear in orange.
- Restart info, which should start with the word “restart”. They will be sent to the restart/\_’name\_project’.log. The format will be developed afterwards.
- All types of text which do not start with these code words are only shown to the GUI of Habby.
- Python code, which should start with the line py followed by four spaces. It will be sent to the python-type log file. It is usually a function which is part of Habby code. The different arguments of the function should be given in the preceding lines.

### Example

Let’s write to the log a function which takes an integer and a string as input. The function is in the module called habby1, which is imported by default in the .log file. The strings to send as log would be:

- “# this my fancy function”
- “py my\_int = ” + str(my\_int\_in\_code)
- “py my\_string = ” + my\_string\_in\_the \_code+ “”
- “py habby1.myfunc(my\_int, my\_string)”

A comment should be added before each chunk of python code to improve the readability.

### Update the log

Let’s consider a scenario where a new function has been written in a non-GUI module (class or function) and has to be called in the GUI in a method of a class. Let’s call the new function new\_func and the class in the GUI my\_class.

To create a new line of log for `new_func`, one should follow these steps:

- A `PyQtSignal` with a string as argument should be added to `my_class`: `send_log = pyqtSignal(str, name='send_log')`
- If a log should be sent directly from `my_class` (for example, to say that `new_func` has been called), the signal should be emitted: `self.send_log.emit('# new_func has been called')`
- In the new function, error and warning are written as follows: `print("Error: here is an error.n")` or `print("Warning: This is just a warning.n")`
- In `my_class`, error and warning are collected by redirecting `stdout` to a string. The following lines of code should be added around the calling of `my_func()`:
  - `sys.stdout = mystdout = StringIO() # redirect stdout`
  - `my_func(my_int,my_string)`
  - `sys.stdout = sys.__stdout__ # re-sent stdout to the cmd`
  - `str_found = mystdout.getvalue() # get all warning, error, text,...`
  - `str_found = str_found.split('n') # separate each message`
  - `for i in range(0, len(str_found)):`
    - \* `if len(str_found[i]) > 1:`
      - `self.send_log.emit(str_found[i]) #send the text`
- To import `StringIO`, the following statement is needed at the start of the code: `from io import StringIO`
- If `new_func` is called from the command line, `stdout` will not be redirected and the errors or warnings will be printed on the cmd as usual. `Stderr` should be re-directed in a similar manner if needed.
- The signal should be collected in the function `connect_signal_log` in the `Main_Windows_1.py`. For this, a line should be added in the function: `* self.my_class.send_log.connect(self.write_log)`

#### restart file

The format of the restart file is based on the format asked by `habby_cmd.py`. More information on this format in `habby_cmd.py` (notably in the part `list_command`).

## 5.5 Command line

The module `habby_cmd` is used to call `habby` from the command line or to use the restart file to run `habby`. This module can also be used to run `HABBY` on a bunch of files contained in a folder.

`habby_cmd.all_command(all_arg, name_prj, path_prj, path_bio)`

This function is used to call `HABBY` from the command line. The general form is to call: `habby_cmd command_name input1 input2 .. input n`. The list of the `command_name` is given in the documentation and by calling the command `"python habby_cmd.py LIST_COMMAND"`. This function is usually called directly by the `main()` or it is called by the function `restart` which read a list of function line by line. Careful, new command cannot contain the symbol `":"` as it is used by `restart`.

#### Parameters

- **all\_arg** – the list of argument (`sys.argv` more or less)
- **name\_prj** – the name of the project, created by default by the `main()`
- **path\_prj** – the path to the project created by default by the `main()`
- **path\_bio** – the path to the project

`habby_cmd.habby_on_all` (*all\_arg, name\_prj, path\_prj, path\_bio*)

This function is used to execute a command from `habby_cmd` on all files in a folder. The form of the command should be something like “`habby_cmd ALL COMMAND path_to_file/*.ext arg2 ag3`” with the arguments adated to the specific command.

In other words, the command should be the usual command with the keyword `ALL` before and with the name of the input files remplace by `*.ext` . where `ext` is the extension of the files. It is better to not add an output name. Indeed default name for output includes the input file name, which is practical if different files are given as input. If the default is overided, the same name will be applied, only the time stamps will be different. To be sure to not overwrite a file, this function waits one second between each command. Only the input argument should contains the string `*`. Otherwise, other commands would be treated as input files.

If there is more than one type of input, it is important that the name of the file are the name (or at least that there are in the same alphabetical order).

If more than one extension is possible (example `g01, g02, g03, etc.` in `hec-ras`), remplace the changing part of the extension with the symbol `*` (so `path_to_folder/*.g0*` `arg1 argn`). If the name of the file changed in the extension as in `RUBAR` (where the file have the name `PROFIL.file`), just change for `PROFIL.*` or something similar. Generally the matching is done using the function `glob`, so the shell-type wildcard can be used.

#### Parameters

- **all\_arg** – the list of argument (`sys.argv` without the argument `ALL` so [`sys.argv[0]`, `sys.argv[2]`, `sys.argv[n]`])
- **name\_prj** – the name of the project, created by default by the `main()`
- **path\_prj** – the path to the project created by default bu the `main()`
- **path\_bio** – the path to the project

`habby_cmd.habby_restart` (*file\_comm, name\_prj, path\_prj, path\_bio*)

This function reads a list of command from a text file called `file_comm`. It then calls `all_command` one each line which does contain the symbol `”:`” . If the lines contains the symbol `”:`”, it considered as an input. Careful, the input should be in order!!!! The info on the left and sight of the symbol `”:`” are just there so an human can read them more easily. Space does not matters here. We try to write the restart file created automatically by HABBY in a “nice” layout, but it just to read it more easily.

#### Parameters

- **file\_comm** – a string wehich gives the name of the restart file (with the path)
- **name\_prj** – the name of the project, created by default by the `main()`
- **path\_prj** – the path to the project created by default bu the `main()`
- **path\_bio** – the path to the project

`habby_cmd.main` ()

This is the main for HABBY when called from the command line. It can call `restart` (read a list of command from a file) or read a command written on the `cmd` or apply a command to a type of file (key word `ALL` before the command and name of the file with asterisk). For more complicated case, one can directly do a python script using the function from HABBY.

## 5.6 Git - code management

### Pour commencer:

- Choisir un dossier sur l’ordinateur local ou va se trouver les fichiers sources.
- `cd` « dossier avec les codes source»

- `git config --global user.name « username »`
- `git config --global user.email « mail »`
- `git init`
- lier le repertoire local avec le repertoire distant sur `forge.irstea.fr`

### Pour mettre une nouvelle version sur le site web

- `cd « dossier avec les codes source »`
- `git pull` (prend la dernière version à jour sur le site et mets tous les fichiers ensemble) ou `git fetch` (prend juste les derniers fichiers sans mettre tous les fichiers ensemble).
- `git add 'my_file.py ou .pyc'` (choisit les fichiers qui doivent être envoyé), le signe `*` fonctionne.
- `git log` (donne l'historique)
- `git status` (donne les nouveaux fichiers locaux)
- `git commit -m « description »` (commit localement)
- `git push`

### Pour ajouter une nouvelle branche

- Donc pour avoir une partie du travail sépare du reste
- `git checkout -b [branchname]` pour créer la branche et y travailler
- `git checkout [branchname]` pour y travailler

## 5.7 Write the documentation

Habby uses Sphinx to document the code. Sphinx uses the docstring given in each function. Hence, it is necessary to write a docstring for each function which has to be documented.

To update the html documentation, go to the doc folder and execute the command: “make html”.

To update the Latex documentation, use the commande “make latex” and use Miketex to create the pdf. `rts2pdf` does not work with Python 3.

To add text in the documentation, modify the `index.rst` file in the doc folder. To add a new module to the documentation, add the module as written in the `index.rts` file in the doc folder. To add text comment, the `index.rts` file can also be directly modified.

It is important to keep the formatting and the alignment.

If the module is in a new folder, the address of the folder must be added to the `config.py` file. It is better to not use absolute path for this, so it is possible to move the documentation on another computer. If the documentation does not run on a new computer, check the path given in the `config.py` file.

In the docstring, add as many blank lines as possible (in reasonable limit). This is easier for the formatting. To make a bullet list, one should use a tab and the symbol “\*”. Using only the symbol “\*” will fail.

To add a new title, do not start the title or the line of symbol under the title with a blank space.

## 5.8 License of used python modules

- h5py: BSD License

- Element tree (XML): MIT License
- numpy: BSD License
- matplotlib: BSD License
- PyQt5: GNU License
- Scipy: BSD license
- shutil:



## INDICES AND TABLES

- `genindex`
- `modindex`
- `search`





## PYTHON MODULE INDEX

### h

habby\_cmd, 94

### s

src, 31

src.bio\_info, 84

src.calcul\_hab, 86

src.dist\_vistess2, 60

src.estimhab, 71

src.fstress, 76

src.func\_for\_cmd, 90

src.Hec\_ras06, 31

src.hec\_ras2D, 39

src.lammi, 54

src.load\_hdf5, 57

src.manage\_grid\_8, 64

src.mascaret, 42

src.mesh\_grid2, 81

src.new\_create\_vtk, 89

src.river2d, 47

src.rubar, 48

src.selafin\_habby1, 53

src.stathab\_c, 72

src.substrate, 79

src\_GUI, 7

src\_GUI.bio\_info\_GUI, 28

src\_GUI.estimhab\_GUI, 26

src\_GUI.fstress\_GUI, 24

src\_GUI.hydro\_GUI\_2, 13

src\_GUI.Main\_windows\_1, 7

src\_GUI.output\_fig\_GUI, 21

src\_GUI.stathab\_GUI, 22

## INDEX

- add\_all\_file() (src\_GUI.hydro\_GUI\_2.River2D method), 16
- add\_all\_fish() (src\_GUI.fstress\_GUI.FstressW method), 25
- add\_all\_fish() (src\_GUI.stathab\_GUI.StathabW method), 22
- add\_all\_tab() (src\_GUI.Main\_windows\_1.CentralW method), 8
- add\_file\_river2d() (src\_GUI.hydro\_GUI\_2.River2D method), 16
- add\_file\_to\_list() (src\_GUI.hydro\_GUI\_2.River2D method), 17
- add\_fish() (src\_GUI.estimhab\_GUI.StatModUseful method), 27
- add\_fish() (src\_GUI.stathab\_GUI.StathabW method), 22
- add\_point() (in module src.manage\_grid\_8), 64
- addcontent() (src.selafin\_habby1.Selafin method), 53
- addtext() (src\_GUI.Main\_windows\_1.EmptyTab method), 9
- all\_command() (in module habby\_cmd), 94
- appendcoretimeslf() (src.selafin\_habby1.Selafin method), 53
- appendcorevarsslf() (src.selafin\_habby1.Selafin method), 53
- appendheaderslf() (src.selafin\_habby1.Selafin method), 53
- BioInfo (class in src\_GUI.bio\_info\_GUI), 28
- calc\_hab() (in module src.calcul\_hab), 86
- calc\_hab\_norm() (in module src.calcul\_hab), 87
- CentralW (class in src\_GUI.Main\_windows\_1), 7
- change\_folder() (src\_GUI.estimhab\_GUI.EstimhabW method), 26
- change\_folder() (src\_GUI.Main\_windows\_1.ShowImageW method), 12
- change\_name (src\_GUI.Main\_windows\_1.WelcomeW attribute), 13
- change\_name\_project() (src\_GUI.Main\_windows\_1.MainWindows method), 10
- change\_riv\_type() (src\_GUI.stathab\_GUI.StathabW method), 22
- change\_unit() (in module src.bio\_info), 84
- clear\_log() (src\_GUI.Main\_windows\_1.MainWindows method), 10
- close\_project() (src\_GUI.Main\_windows\_1.MainWindows method), 10
- close\_rech() (src\_GUI.Main\_windows\_1.MainWindows method), 10
- closeEvent() (src\_GUI.Main\_windows\_1.MainWindows method), 10
- connect\_signal\_log() (src\_GUI.Main\_windows\_1.CentralW method), 8
- coord\_lammi() (in module src.lammi), 54
- coord\_profile\_non\_georeferenced() (in module src.Hec\_ras06), 31
- copy\_files() (in module src.load\_hdf5), 57
- correct\_duplicate() (in module src.mascaret), 42
- correct\_duplicate\_xy() (in module src.rubar), 48
- create\_and\_fill\_database() (in module src.bio\_info), 84
- create\_default\_figoption() (in module src\_GUI.output\_fig\_GUI), 21
- create\_dummy\_substrate() (in module src.manage\_grid\_8), 64
- create\_dummy\_substrate\_from\_hydro() (in module src.substrate), 79
- create\_grid() (in module src.manage\_grid\_8), 64
- create\_grid\_only\_1\_profile() (in module src.manage\_grid\_8), 65
- create\_hdf5() (src.stathab\_c.Stathab method), 72
- create\_merge\_grid() (in module src.mesh\_grid2), 81
- CreateNewProject (class in src\_GUI.Main\_windows\_1), 9
- cut\_2d\_grid() (in module src.manage\_grid\_8), 65
- cut\_2d\_grid\_all\_reach() (in module src.manage\_grid\_8), 65
- define\_stream\_network() (in module src.mascaret), 42
- dengauss() (src.stathab\_c.Stathab method), 72
- denstress() (in module src.fstress), 76
- dis\_enable\_nb\_profile() (src\_GUI.hydro\_GUI\_2.SubHydroW method), 18
- dist\_h() (src.stathab\_c.Stathab method), 72
- dist\_h\_trop() (src.stathab\_c.Stathab method), 72
- dist\_v() (src.stathab\_c.Stathab method), 73
- dist\_v\_trop() (src.stathab\_c.Stathab method), 73

dist\_velocity\_hecras() (in module src.dist\_vistess2), 60  
 distribute\_velocity() (in module src.dist\_vistess2), 61  
 do\_log() (src\_GUI.Main\_windows\_1.MainWindows method), 10  
 drop\_hydro (src\_GUI.hydro\_GUI\_2.SubHydroW attribute), 18  
 drop\_merge (src\_GUI.hydro\_GUI\_2.SubstrateW attribute), 20  
 edf\_to\_cemagref() (in module src.substrate), 79  
 empty\_project() (src\_GUI.Main\_windows\_1.MainWindows method), 10  
 EmptyTab (class in src\_GUI.Main\_windows\_1), 9  
 erase\_name() (src\_GUI.fstress\_GUI.FstressW method), 25  
 erase\_pict() (src\_GUI.Main\_windows\_1.MainWindows method), 10  
 estimhab() (in module src.estimhab), 71  
 EstimhabW (class in src\_GUI.estimhab\_GUI), 26  
 execute\_request() (in module src.bio\_info), 84  
 fig\_lammi() (in module src.lammi), 55  
 fig\_merge\_grid() (in module src.mesh\_grid2), 82  
 fig\_substrate() (in module src.substrate), 79  
 figure\_fstress() (in module src.fstress), 76  
 figure\_hecras2d() (in module src.hecras2D), 39  
 figure\_mascaret() (in module src.mascaret), 42  
 figure\_pref() (in module src.bio\_info), 85  
 figure\_river2d() (in module src.river2d), 47  
 figure\_rubar1d() (in module src.rubar), 48  
 figure\_rubar2d() (in module src.rubar), 49  
 figure\_xml() (in module src.Hec\_ras06), 31  
 find\_coord\_height\_velocity() (in module src.Hec\_ras06), 32  
 find\_node() (in module src.mascaret), 43  
 find\_path\_hdf5() (src\_GUI.hydro\_GUI\_2.SubHydroW method), 18  
 find\_path\_hdf5\_est() (src\_GUI.estimhab\_GUI.StatModUseful method), 27  
 find\_path\_hdf5\_stat() (src.stathab\_c.Stathab method), 73  
 find\_path\_im() (src\_GUI.hydro\_GUI\_2.SubHydroW method), 18  
 find\_path\_im\_est() (src\_GUI.estimhab\_GUI.StatModUseful method), 27  
 find\_path\_input() (src\_GUI.hydro\_GUI\_2.SubHydroW method), 18  
 find\_path\_input\_est() (src\_GUI.estimhab\_GUI.StatModUseful method), 27  
 find\_path\_output\_est() (src\_GUI.estimhab\_GUI.StatModUseful method), 27  
 find\_path\_text\_est() (src\_GUI.estimhab\_GUI.StatModUseful method), 27  
 find\_pref\_value() (in module src.calcul\_hab), 87  
 find\_profile\_between() (in module src.manage\_grid\_8), 66  
 find\_sh0() (src.stathab\_c.Stathab method), 73  
 find\_sh0\_maxvrais() (src.stathab\_c.Stathab method), 73  
 find\_sub\_and\_cross() (in module src.mesh\_grid2), 82  
 flat\_coord\_pro() (in module src.mascaret), 43  
 FreeSpace (class in src\_GUI.hydro\_GUI\_2), 13  
 fstress\_test() (in module src.fstress), 77  
 FstressW (class in src\_GUI.fstress\_GUI), 24  
 func\_stress() (in module src.fstress), 77  
 get\_all\_attribute() (in module src.substrate), 80  
 get\_all\_filename() (in module src.load\_hdf5), 57  
 get\_att\_name() (src\_GUI.hydro\_GUI\_2.SubstrateW method), 20  
 get\_attribute\_from\_shp() (src\_GUI.hydro\_GUI\_2.SubstrateW method), 20  
 get\_crossing\_segment\_sub() (in module src.manage\_grid\_8), 66  
 get\_geo\_name\_from\_xcas() (in module src.mascaret), 43  
 get\_hdf5\_name() (in module src.load\_hdf5), 57  
 get\_manning() (in module src.dist\_vistess2), 62  
 get\_manning\_arr() (in module src.dist\_vistess2), 62  
 get\_name\_from\_cas() (in module src.mascaret), 43  
 get\_new\_hydro\_hdf5() (src\_GUI.hydro\_GUI\_2.HabbyHdf5 method), 15  
 get\_new\_point\_and\_cell\_1\_profil() (in module src.manage\_grid\_8), 66  
 get\_new\_vel\_height\_data() (in module src.mesh\_grid2), 82  
 get\_rid\_of\_lines() (in module src.river2d), 47  
 get\_rid\_of\_white\_space() (in module src.Hec\_ras06), 33  
 get\_stage() (in module src.bio\_info), 85  
 get\_transect\_filename() (in module src.lammi), 55  
 get\_triangular\_grid() (in module src.rubar), 49  
 get\_triangular\_grid\_hecras() (in module src.hecras2D), 39  
 get\_useful\_attribute() (in module src.substrate), 80  
 get\_endianfromchar() (in module src.selafin\_habby1), 53  
 get\_floattypefromfloat() (in module src.selafin\_habby1), 53  
 gethdf5\_name\_gui() (src\_GUI.hydro\_GUI\_2.SubHydroW method), 18  
 getheaderfloatsself() (src.selafin\_habby1.Selafin method), 53  
 getheaderintegersself() (src.selafin\_habby1.Selafin method), 53  
 getheadermetadataself() (src.selafin\_habby1.Selafin method), 53  
 gettimehistoryself() (src.selafin\_habby1.Selafin method), 53  
 getvalues() (src.selafin\_habby1.Selafin method), 53  
 getvariablesat() (src.selafin\_habby1.Selafin method), 53  
 give\_info\_model() (src\_GUI.hydro\_GUI\_2.Hydro2W method), 15

grid\_and\_interpo() (in module src.manage\_grid\_8), 67  
 habby\_cmd (module), 94  
 habby\_on\_all() (in module habby\_cmd), 94  
 habby\_restart() (in module habby\_cmd), 95  
 HabbyHdf5 (class in src\_GUI.hydro\_GUI\_2), 14  
 habitat\_to\_vtu() (in module src.new\_create\_vtk), 89  
 HEC\_RAS1D (class in src\_GUI.hydro\_GUI\_2), 13  
 HEC\_RAS2D (class in src\_GUI.hydro\_GUI\_2), 14  
 Hydro2W (class in src\_GUI.hydro\_GUI\_2), 15  
  
 init\_iu() (src\_GUI.bio\_info\_GUI.BioInfo method), 28  
 init\_iu() (src\_GUI.estimhab\_GUI.EstimhabW method), 26  
 init\_iu() (src\_GUI.fstress\_GUI.FstressW method), 25  
 init\_iu() (src\_GUI.hydro\_GUI\_2.HabbyHdf5 method), 15  
 init\_iu() (src\_GUI.hydro\_GUI\_2.HEC\_RAS1D method), 13  
 init\_iu() (src\_GUI.hydro\_GUI\_2.HEC\_RAS2D method), 14  
 init\_iu() (src\_GUI.hydro\_GUI\_2.Hydro2W method), 15  
 init\_iu() (src\_GUI.hydro\_GUI\_2.Mascaret method), 16  
 init\_iu() (src\_GUI.hydro\_GUI\_2.River2D method), 17  
 init\_iu() (src\_GUI.hydro\_GUI\_2.Rubar1D method), 17  
 init\_iu() (src\_GUI.hydro\_GUI\_2.Rubar2D method), 17  
 init\_iu() (src\_GUI.hydro\_GUI\_2.SubstrateW method), 20  
 init\_iu() (src\_GUI.hydro\_GUI\_2.TELEMAT method), 21  
 init\_iu() (src\_GUI.Main\_windows\_1.CentralW method), 8  
 init\_iu() (src\_GUI.Main\_windows\_1.CreateNewProject method), 9  
 init\_iu() (src\_GUI.Main\_windows\_1.EmptyTab method), 9  
 init\_iu() (src\_GUI.Main\_windows\_1.ShowImageW method), 12  
 init\_iu() (src\_GUI.Main\_windows\_1.WelcomeW method), 13  
 init\_iu() (src\_GUI.output\_fig\_GUI.outputW method), 21  
 init\_iu() (src\_GUI.stathab\_GUI.StathabW method), 22  
 init\_ui() (src\_GUI.Main\_windows\_1.MainWindows method), 10  
 inside\_polygon() (in module src.manage\_grid\_8), 67  
 inside\_trigon() (in module src.mesh\_grid2), 83  
 interp\_weights() (in module src.manage\_grid\_8), 67  
 interpo\_linear() (in module src.manage\_grid\_8), 68  
 interpo\_nearest() (in module src.manage\_grid\_8), 68  
 interpolate\_opti() (in module src.manage\_grid\_8), 68  
 intersec\_cross() (in module src.mesh\_grid2), 83  
 intersection\_seg() (in module src.manage\_grid\_8), 68  
 is\_this\_res\_on\_the\_profile() (in module src.mascaret), 44  
  
 linear\_h\_cross() (in module src.manage\_grid\_8), 69  
 load\_all\_fish() (src\_GUI.fstress\_GUI.FstressW method), 25  
 load\_coord\_1d() (in module src.rubar), 49  
 load\_dat\_2d() (in module src.rubar), 49  
 load\_data\_1d() (in module src.rubar), 50  
 load\_data\_fstress() (src\_GUI.fstress\_GUI.FstressW method), 25  
 load\_evha\_curve() (in module src.bio\_info), 85  
 load\_fig\_option() (in module src\_GUI.output\_fig\_GUI), 21  
 load\_float\_stathab() (in module src.stathab\_c), 75  
 load\_from\_hdf5\_gui() (src\_GUI.stathab\_GUI.StathabW method), 22  
 load\_from\_txt\_gui() (src\_GUI.stathab\_GUI.StathabW method), 23  
 load\_fstress\_text() (in module src.func\_for\_cmd), 90  
 load\_hdf5\_fstress() (src\_GUI.fstress\_GUI.FstressW method), 25  
 load\_hdf5\_hyd() (in module src.load\_hdf5), 57  
 load\_hdf5\_sub() (in module src.load\_hdf5), 58  
 load\_hec\_2d\_gui() (src\_GUI.hydro\_GUI\_2.HEC\_RAS2D method), 14  
 load\_hec\_ras2d() (in module src.hec\_ras2D), 40  
 load\_hec\_ras\_2d\_and\_cut\_grid() (in module src.hec\_ras2D), 40  
 load\_hec\_ras\_gui() (src\_GUI.hydro\_GUI\_2.HEC\_RAS1D method), 14  
 load\_lammi() (in module src.lammi), 55  
 load\_mai\_1d() (in module src.rubar), 50  
 load\_mai\_2d() (in module src.rubar), 50  
 load\_manning\_text() (src\_GUI.hydro\_GUI\_2.SubHydroW method), 18  
 load\_manning\_txt() (in module src.func\_for\_cmd), 90  
 load\_mascaret() (in module src.mascaret), 44  
 load\_mascaret\_and\_create\_grid() (in module src.mascaret), 44  
 load\_mascaret\_gui() (src\_GUI.hydro\_GUI\_2.Mascaret method), 16  
 load\_namereach() (in module src.stathab\_c), 75  
 load\_pref() (in module src.stathab\_c), 75  
 load\_pref\_trop\_biv() (in module src.stathab\_c), 76  
 load\_pref\_trop\_uni() (in module src.stathab\_c), 76  
 load\_river2d\_and\_cut\_grid() (in module src.river2d), 47  
 load\_river2d\_cdg() (in module src.river2d), 47  
 load\_river2d\_gui() (src\_GUI.hydro\_GUI\_2.River2D method), 17  
 load\_rubar() (src\_GUI.hydro\_GUI\_2.Rubar2D method), 17  
 load\_rubar1d() (in module src.rubar), 50  
 load\_rubar1d() (src\_GUI.hydro\_GUI\_2.Rubar1D method), 17  
 load\_rubar1d\_and\_create\_grid() (in module src.rubar), 50  
 load\_rubar2d() (in module src.rubar), 51  
 load\_rubar2d\_and\_create\_grid() (in module src.rubar), 52

[load\\_stathab\\_from\\_hdf5\(\)](#) ([src.stathab\\_c.Stathab](#) method), 74  
[load\\_stathab\\_from\\_txt\(\)](#) ([src.stathab\\_c.Stathab](#) method), 74  
[load\\_station\(\)](#) (in module [src.lammi](#)), 56  
[load\\_sub\\_gui\(\)](#) ([src\\_GUI.hydro\\_GUI\\_2.SubstrateW](#) method), 20  
[load\\_sub\\_shp\(\)](#) (in module [src.substrate](#)), 80  
[load\\_sub\\_txt\(\)](#) (in module [src.substrate](#)), 80  
[load\\_telemac\(\)](#) (in module [src.selafln\\_habby1](#)), 53  
[load\\_telemac\\_and\\_cut\\_grid\(\)](#) (in module [src.selafln\\_habby1](#)), 53  
[load\\_telemac\\_gui\(\)](#) ([src\\_GUI.hydro\\_GUI\\_2.TELEMAC](#) method), 21  
[load\\_tps\\_2d\(\)](#) (in module [src.rubar](#)), 52  
[load\\_transect\\_data\(\)](#) (in module [src.lammi](#)), 56  
[load\\_txt\(\)](#) ([src\\_GUI.fstress\\_GUI.FstressW](#) method), 25  
[load\\_xml\(\)](#) (in module [src.Hec\\_ras06](#)), 33  
[load\\_xml\\_name\(\)](#) (in module [src.bio\\_info](#)), 85  
[log\\_txt\(\)](#) ([src\\_GUI.hydro\\_GUI\\_2.SubstrateW](#) method), 20  
  
[m\\_file\\_load\\_coord\\_1d\(\)](#) (in module [src.rubar](#)), 52  
[main\(\)](#) (in module [habby\\_cmd](#)), 95  
[main\(\)](#) (in module [src.bio\\_info](#)), 86  
[main\(\)](#) (in module [src.dist\\_vistess2](#)), 62  
[main\(\)](#) (in module [src.estimhab](#)), 72  
[main\(\)](#) (in module [src.fstress](#)), 77  
[main\(\)](#) (in module [src.Hec\\_ras06](#)), 33  
[main\(\)](#) (in module [src.hec\\_ras2D](#)), 41  
[main\(\)](#) (in module [src.lammi](#)), 56  
[main\(\)](#) (in module [src.manage\\_grid\\_8](#)), 69  
[main\(\)](#) (in module [src.mascaret](#)), 45  
[main\(\)](#) (in module [src.mesh\\_grid2](#)), 83  
[main\(\)](#) (in module [src.new\\_create\\_vtk](#)), 89  
[main\(\)](#) (in module [src.river2d](#)), 48  
[main\(\)](#) (in module [src.rubar](#)), 52  
[main\(\)](#) (in module [src.stathab\\_c](#)), 76  
[main\(\)](#) (in module [src.substrate](#)), 80  
[MainWindows](#) (class in [src\\_GUI.Main\\_windows\\_1](#)), 9  
[Mascaret](#) (class in [src\\_GUI.hydro\\_GUI\\_2](#)), 16  
[merge\\_grid\\_and\\_save\(\)](#) (in module [src.mesh\\_grid2](#)), 83  
[merge\\_grid\\_hydro\\_sub\(\)](#) (in module [src.mesh\\_grid2](#)), 83  
[modify\\_name\(\)](#) ([src\\_GUI.fstress\\_GUI.FstressW](#) method), 25  
[my\\_menu\\_bar\(\)](#) ([src\\_GUI.Main\\_windows\\_1.MainWindows](#) method), 10  
  
[new\\_proj\\_signal](#) ([src\\_GUI.Main\\_windows\\_1.WelcomeW](#) attribute), 13  
[new\\_project\(\)](#) ([src\\_GUI.Main\\_windows\\_1.MainWindows](#) method), 10  
[newp\(\)](#) (in module [src.manage\\_grid\\_8](#)), 69  
  
[open\\_estimhab\\_hdf5\(\)](#) ([src\\_GUI.estimhab\\_GUI.EstimhabW](#) method), 26  
[open\\_example\(\)](#) ([src\\_GUI.Main\\_windows\\_1.WelcomeW](#) method), 13  
[open\\_geo\\_mascaret\(\)](#) (in module [src.mascaret](#)), 45  
[open\\_geofile\(\)](#) (in module [src.Hec\\_ras06](#)), 33  
[open\\_hdf5\(\)](#) (in module [src.load\\_hdf5](#)), 58  
[open\\_hec\\_hec\\_ras\\_and\\_create\\_grid\(\)](#) (in module [src.Hec\\_ras06](#)), 34  
[open\\_hecras\(\)](#) (in module [src.Hec\\_ras06](#)), 35  
[open\\_proj](#) ([src\\_GUI.Main\\_windows\\_1.WelcomeW](#) attribute), 13  
[open\\_project\(\)](#) ([src\\_GUI.Main\\_windows\\_1.MainWindows](#) method), 10  
[open\\_recent\\_project\(\)](#) ([src\\_GUI.Main\\_windows\\_1.MainWindows](#) method), 10  
[open\\_rech\(\)](#) ([src\\_GUI.Main\\_windows\\_1.MainWindows](#) method), 11  
[open\\_repfile\(\)](#) (in module [src.Hec\\_ras06](#)), 36  
[open\\_res\\_file\(\)](#) (in module [src.mascaret](#)), 45  
[open\\_rub\\_file\(\)](#) (in module [src.mascaret](#)), 45  
[open\\_sdffile\(\)](#) (in module [src.Hec\\_ras06](#)), 36  
[open\\_shp\(\)](#) (in module [src.substrate](#)), 80  
[open\\_xmlfile\(\)](#) (in module [src.Hec\\_ras06](#)), 37  
[optfig\(\)](#) ([src\\_GUI.Main\\_windows\\_1.CentralW](#) method), 8  
[outputW](#) (class in [src\\_GUI.output\\_fig\\_GUI](#)), 21  
  
[pass\\_grid\\_cell\\_to\\_node\\_lin\(\)](#) (in module [src.manage\\_grid\\_8](#)), 69  
[pass\\_in\\_float\\_from\\_geo\(\)](#) (in module [src.Hec\\_ras06](#)), 37  
[pass\\_to\\_float\\_estimhab\(\)](#) (in module [src.estimhab](#)), 72  
[plot\\_dist\\_vit\(\)](#) (in module [src.dist\\_vistess2](#)), 62  
[plot\\_grid\(\)](#) (in module [src.manage\\_grid\\_8](#)), 70  
[plot\\_grid\\_simple\(\)](#) (in module [src.manage\\_grid\\_8](#)), 70  
[plot\\_hydrosignature\(\)](#) (in module [src.bio\\_info](#)), 86  
[plot\\_vel\\_h\(\)](#) (in module [src.selafln\\_habby1](#)), 54  
[power\\_law\(\)](#) ([src.stathab\\_c.Stathab](#) method), 74  
[prepare\\_grid\(\)](#) (in module [src.hec\\_ras2D](#)), 41  
[preparetest\\_velocity\(\)](#) (in module [src.dist\\_vistess2](#)), 63  
[profil\\_coord\\_non\\_georef\(\)](#) (in module [src.mascaret](#)), 46  
[propose\\_next\\_file\(\)](#) ([src\\_GUI.hydro\\_GUI\\_2.Rubar2D](#) method), 18  
[putcontent\(\)](#) ([src.selafln\\_habby1.Selafln](#) method), 53  
  
[reach\\_selected\(\)](#) ([src\\_GUI.stathab\\_GUI.StathabW](#) method), 23  
[read\\_attribute\\_xml\(\)](#) ([src\\_GUI.hydro\\_GUI\\_2.SubHydroW](#) method), 19  
[read\\_fstress\\_hdf5\(\)](#) (in module [src.fstress](#)), 77  
[read\\_pref\(\)](#) (in module [src.bio\\_info](#)), 86  
[read\\_pref\(\)](#) (in module [src.fstress](#)), 77  
[remove\\_all\\_file\(\)](#) ([src\\_GUI.hydro\\_GUI\\_2.River2D](#) method), 17

[remove\\_file\(\)](#) (src\_GUI.hydro\_GUI\_2.River2D method), [17](#)  
[remove\\_fish\(\)](#) (src\_GUI.estimhab\_GUI.StatModUseful method), [27](#)  
[remove\\_fish\(\)](#) (src\_GUI.stathab\_GUI.StathabW method), [24](#)  
[reorder\\_reach\(\)](#) (in module src.Hec\_ras06), [37](#)  
[River2D](#) (class in src\_GUI.hydro\_GUI\_2), [16](#)  
[river\\_coord\\_non\\_georef\\_from\\_cas\(\)](#) (in module src.mascaret), [46](#)  
[river\\_coord\\_non\\_georef\\_from\\_xcas\(\)](#) (in module src.mascaret), [46](#)  
[Rubar1D](#) (class in src\_GUI.hydro\_GUI\_2), [17](#)  
[Rubar2D](#) (class in src\_GUI.hydro\_GUI\_2), [17](#)  
[run\\_estmihab\(\)](#) (src\_GUI.estimhab\_GUI.EstimhabW method), [26](#)  
[run\\_fstress\(\)](#) (in module src.fstress), [77](#)  
[run\\_habitat\\_value\(\)](#) (src\_GUI.bio\_info\_GUI.BioInfo method), [28](#)  
[run\\_stathab\\_gui\(\)](#) (src\_GUI.stathab\_GUI.StathabW method), [24](#)  
[runsav\\_fstress\(\)](#) (src\_GUI.fstress\_GUI.FstressW method), [25](#)  
  
[sandre\\_to\\_cemagref\(\)](#) (in module src.substrate), [81](#)  
[save\\_fstress\(\)](#) (in module src.fstress), [78](#)  
[save\\_hab\\_fig\\_spu\(\)](#) (in module src.calcul\_hab), [87](#)  
[save\\_hab\\_shape\(\)](#) (in module src.calcul\_hab), [87](#)  
[save\\_hab\\_txt\(\)](#) (in module src.calcul\_hab), [88](#)  
[save\\_hdf5\(\)](#) (in module src.load\_hdf5), [58](#)  
[save\\_hdf5\\_sub\(\)](#) (in module src.load\_hdf5), [59](#)  
[save\\_option\\_fig\(\)](#) (src\_GUI.output\_fig\_GUI.outputW method), [21](#)  
[save\\_project](#) (src\_GUI.Main\_windows\_1.CreateNewProject attribute), [9](#)  
[save\\_project\(\)](#) (src\_GUI.Main\_windows\_1.MainWindows method), [11](#)  
[save\\_project\\_estimhab\(\)](#) (src\_GUI.Main\_windows\_1.MainWindows method), [11](#)  
[save\\_project\\_if\\_new\\_project\(\)](#) (src\_GUI.Main\_windows\_1.MainWindows method), [11](#)  
[save\\_river\\_data\(\)](#) (src\_GUI.fstress\_GUI.FstressW method), [25](#)  
[save\\_signal](#) (src\_GUI.Main\_windows\_1.WelcomeW attribute), [13](#)  
[save\\_signal\\_estimhab](#) (src\_GUI.estimhab\_GUI.EstimhabW attribute), [27](#)  
[save\\_spu\\_txt\(\)](#) (in module src.calcul\_hab), [88](#)  
[save\\_vh\\_fig\\_2d\(\)](#) (in module src.calcul\_hab), [88](#)  
[save\\_xml\(\)](#) (src\_GUI.hydro\_GUI\_2.SubHydroW method), [19](#)  
[save\\_xml\\_stathab\(\)](#) (src.stathab\_c.Stathab method), [74](#)  
[savefig\\_stahab\(\)](#) (src.stathab\_c.Stathab method), [74](#)  
  
[savetxt\\_stathab\(\)](#) (src.stathab\_c.Stathab method), [74](#)  
[scatter\\_plot\(\)](#) (in module src.hec\_ras2D), [41](#)  
[scrollsdn\(\)](#) (src\_GUI.Main\_windows\_1.CentralW method), [8](#)  
[Selafin](#) (class in src.selafin\_habby1), [53](#)  
[select\\_dir\(\)](#) (src\_GUI.stathab\_GUI.StathabW method), [24](#)  
[select\\_fish\(\)](#) (src\_GUI.bio\_info\_GUI.BioInfo method), [28](#)  
[select\\_hdf5\(\)](#) (src\_GUI.stathab\_GUI.StathabW method), [24](#)  
[selectionchange\(\)](#) (src\_GUI.hydro\_GUI\_2.Hydro2W method), [15](#)  
[selectionchange\(\)](#) (src\_GUI.Main\_windows\_1.ShowImageW method), [12](#)  
[send\\_data\(\)](#) (src\_GUI.hydro\_GUI\_2.SubHydroW method), [19](#)  
[send\\_err\\_log\(\)](#) (src\_GUI.estimhab\_GUI.StatModUseful method), [27](#)  
[send\\_err\\_log\(\)](#) (src\_GUI.hydro\_GUI\_2.SubHydroW method), [19](#)  
[send\\_err\\_log\(\)](#) (src\_GUI.stathab\_GUI.StathabW method), [24](#)  
[send\\_log](#) (src\_GUI.estimhab\_GUI.StatModUseful attribute), [27](#)  
[send\\_log](#) (src\_GUI.hydro\_GUI\_2.Hydro2W attribute), [16](#)  
[send\\_log](#) (src\_GUI.hydro\_GUI\_2.SubHydroW attribute), [19](#)  
[send\\_log](#) (src\_GUI.Main\_windows\_1.CreateNewProject attribute), [9](#)  
[send\\_log](#) (src\_GUI.Main\_windows\_1.ShowImageW attribute), [12](#)  
[send\\_log](#) (src\_GUI.Main\_windows\_1.WelcomeW attribute), [13](#)  
[send\\_log](#) (src\_GUI.output\_fig\_GUI.outputW attribute), [22](#)  
[send\\_log](#) (src\_GUI.stathab\_GUI.StathabW attribute), [24](#)  
[send\\_merge\\_grid\(\)](#) (src\_GUI.hydro\_GUI\_2.SubstrateW method), [21](#)  
[setfolder\(\)](#) (src\_GUI.Main\_windows\_1.CreateNewProject method), [9](#)  
[setfolder\(\)](#) (src\_GUI.Main\_windows\_1.WelcomeW method), [13](#)  
[setlangue\(\)](#) (src\_GUI.Main\_windows\_1.MainWindows method), [12](#)  
[show\\_data\\_one\\_river\(\)](#) (src\_GUI.fstress\_GUI.FstressW method), [25](#)  
[show\\_dialog\(\)](#) (src\_GUI.hydro\_GUI\_2.SubHydroW method), [19](#)  
[show\\_fig](#) (src\_GUI.estimhab\_GUI.StatModUseful attribute), [27](#)  
[show\\_fig](#) (src\_GUI.hydro\_GUI\_2.SubHydroW attribute), [19](#)  
[show\\_fig](#) (src\_GUI.stathab\_GUI.StathabW attribute), [24](#)



show\_hydrosignature() (src\_GUI.bio\_info\_GUI.BioInfo method), 28  
 show\_info\_fish() (src\_GUI.bio\_info\_GUI.BioInfo method), 28  
 show\_pref() (src\_GUI.bio\_info\_GUI.BioInfo method), 28  
 showfig() (src\_GUI.Main\_windows\_1.CentralW method), 8  
 showfig2() (src\_GUI.Main\_windows\_1.CentralW method), 8  
 ShowImageW (class in src\_GUI.Main\_windows\_1), 12  
 src (module), 31  
 src.bio\_info (module), 84  
 src.calcul\_hab (module), 86  
 src.dist\_vistess2 (module), 60  
 src.estimhab (module), 71  
 src.fstress (module), 76  
 src.func\_for\_cmd (module), 90  
 src.Hec\_ras06 (module), 31  
 src.hec\_ras2D (module), 39  
 src.lammi (module), 54  
 src.load\_hdf5 (module), 57  
 src.manage\_grid\_8 (module), 64  
 src.mascaret (module), 42  
 src.mesh\_grid2 (module), 81  
 src.new\_create\_vtk (module), 89  
 src.river2d (module), 47  
 src.rubar (module), 48  
 src.selafln\_habby1 (module), 53  
 src.stathab\_c (module), 72  
 src.substrate (module), 79  
 src\_GUI (module), 7  
 src\_GUI.bio\_info\_GUI (module), 28  
 src\_GUI.estimhab\_GUI (module), 26  
 src\_GUI.fstress\_GUI (module), 24  
 src\_GUI.hydro\_GUI\_2 (module), 13  
 src\_GUI.Main\_windows\_1 (module), 7  
 src\_GUI.output\_fig\_GUI (module), 21  
 src\_GUI.stathab\_GUI (module), 22  
 Stathab (class in src.stathab\_c), 72  
 stathab\_calc() (src.stathab\_c.Stathab method), 74  
 stathab\_trop\_biv() (src.stathab\_c.Stathab method), 74  
 stathab\_trop\_univ() (src.stathab\_c.Stathab method), 75  
 StathabW (class in src\_GUI.stathab\_GUI), 22  
 StatModUseful (class in src\_GUI.estimhab\_GUI), 27  
 SubHydroW (class in src\_GUI.hydro\_GUI\_2), 18  
 SubstrateW (class in src\_GUI.hydro\_GUI\_2), 20  
 TELEMAT (class in src\_GUI.hydro\_GUI\_2), 21  
 test\_entry\_float() (src\_GUI.Main\_windows\_1.MainWindows method), 12  
 test\_evah\_xml\_pref() (in module src.bio\_info), 86  
 test\_stathab() (src.stathab\_c.Stathab method), 75  
 test\_stathab\_trop\_biv() (src.stathab\_c.Stathab method), 75  
 test\_stathab\_trop\_uni() (src.stathab\_c.Stathab method), 75  
 update\_coord\_pro\_with\_vh\_pro() (in module src.manage\_grid\_8), 70  
 update\_hydro\_hdf5\_name() (src\_GUI.Main\_windows\_1.CentralW method), 8  
 update\_list\_riv() (src\_GUI.fstress\_GUI.FstressW method), 25  
 update\_merge\_list() (src\_GUI.bio\_info\_GUI.BioInfo method), 28  
 update\_namefig() (src\_GUI.Main\_windows\_1.ShowImageW method), 12  
 update\_output() (in module src.Hec\_ras06), 38  
 update\_sub\_hdf5\_name() (src\_GUI.hydro\_GUI\_2.SubstrateW method), 21  
 was\_loaded\_before() (src\_GUI.fstress\_GUI.FstressW method), 25  
 was\_model\_loaded\_before() (src\_GUI.hydro\_GUI\_2.SubHydroW method), 19  
 WelcomeW (class in src\_GUI.Main\_windows\_1), 12  
 write\_log() (src\_GUI.Main\_windows\_1.CentralW method), 8  
 write\_log\_file() (src\_GUI.Main\_windows\_1.CentralW method), 9  
 write\_txt() (in module src.fstress), 78  
 writePVD() (in module src.new\_create\_vtk), 89