

MANUAL

INL/EXT-15-34123

Draft Release

Printed January 2015

RAVEN User Manual

Prepared by
Idaho National Laboratory
Idaho Falls, Idaho 83415

The Idaho National Laboratory is a multiprogram laboratory operated by Battelle Energy Alliance for the United States Department of Energy under DOE Idaho Operations Office. Contract DE-AC07-05ID14517.

Approved for unlimited release.



Issued by the Idaho National Laboratory, operated for the United States Department of Energy by Battelle Energy Alliance.

NOTICE: This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government, nor any agency thereof, nor any of their employees, nor any of their contractors, subcontractors, or their employees, make any warranty, express or implied, or assume any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represent that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government, any agency thereof, or any of their contractors or subcontractors. The views and opinions expressed herein do not necessarily state or reflect those of the United States Government, any agency thereof, or any of their contractors.

Printed in the United States of America. This report has been reproduced directly from the best available copy.



INL/EXT-15-34123
Draft Release
Printed January 2015

RAVEN User Manual

**Cristian Rabiti Andrea Alfonsi
Joshua Cogliati Diego Mandelli
Robert Kinoshita Sonat Sen**

Acknowledgment

Thanks to the following people for their contributions in the drawing up of this Manual:

- Ivan Rinaldi;
- Alessandro Bandini.

1 Introduction

RAVEN is a generic software framework to perform parametric and probabilistic analysis based on the response of complex system codes. The initial development was aimed at providing dynamic risk analysis capabilities to the thermohydraulic code RELAP-7, currently under development at Idaho National Laboratory (INL). Although the initial goal has been fully accomplished, RAVEN is now a multi-purpose probabilistic and uncertainty quantification platform, capable of agnostically communicating with any system code.

The agnosticism of RAVEN is inherent in its provision of Application Programming Interfaces (APIs). These APIs allow RAVEN to interact with any code as long as all the parameters that need to be perturbed are accessible by input files or via python interfaces. RAVEN is capable of investigating system response and input space using various sampling schemes such as Monte Carlo, grid, or Latin hypercube. However, its strength lies in system feature discovery such as: constructing limit surfaces, separating regions of the input space leading to system failure, and using dynamic supervised learning techniques.

The development of RAVEN started in 2012, when, within the Nuclear Energy Advanced Modeling and Simulation (NEAMS) program, the need to provide a modern risk evaluation framework arose. RAVEN's principal assignment is to provide the necessary software and algorithms in order to employ the concepts developed by the Risk Informed Safety Margin Characterization (RISMC) program. RISMC is one of the pathways defined within the Light Water Reactor Sustainability (LWRS) program.

In the RISMC approach, the goal is not just to identify the frequency of an event potentially leading to a system failure, but the proximity (or lack thereof) to key safety-related events. Hence, the approach is interested in identifying and increasing the safety margins related to those events. A safety margin is a numerical value quantifying the probability that a safety metric (e.g. peak pressure in a pipe) is exceeded under certain conditions.

The initial development of RAVEN has been focused on providing dynamic risk assessment capability to RELAP-7, currently under development at INL and, likely, future replacement of the RELAP5-3D code. Most of the capabilities implemented having RELAP-7 as a principal focus are easily deployable to other system codes. For this reason, several side activities are currently ongoing for coupling RAVEN with software such as RELAP5-3D, etc. The aim of this document is to detail the input requirements for RAVEN focusing on the input structure.

2 Manual Formats.

- *Python Coding:*

```
class AClass():  
    def aMethodImplementation(self):  
        pass
```

- *XML input example:*

```
Example:  
-----  
<MainXMLBlock>  
    ...  
    <aXMLnode name='' anObjectName'' anAttribute='' aValue''>  
        <aSubNode>body</aSubNode>  
    </aXMLnode>  
    ...  
</MainXMLBlock>
```

- *Bash Commands:*

```
cd trunk/raven/  
./raven_libs_script.sh  
cd ../../
```

3 Installation

3.1 Framework Source Install

3.1.1 Unpacking The Source

Untar the source install (if there is more than one version of the source tarball, the full filename will need to be used instead of *):

```
tar -xvzf raven_framework_*_source.tar.gz
```

3.1.2 Dependencies

RAVEN has a number of dependencies that must be installed before it can be used, the recommended minimum versions are:

1. numpy-1.7.0
2. hdf5-1.8.12
3. Cython-0.18
4. h5py-2.2.1
5. scipy-0.12.0
6. scikit-learn-0.14.1
7. matplotlib-1.4.0

Older versions may work, but it is highly recommended to use these minimum versions. You may install these dependencies yourself, or by running the `raven_libs_script.sh` script provided within the RAVEN distribution:

```
cd full_path_to_raven_distribution/raven/  
./raven_libs_script.sh  
cd ../../
```

3.1.3 Compilation

First, CROW must be compiled. This can be done either with the python setup or with a makefile.

Using the setup.py file:

```
cd trunk/crow/  
python setup.py build_ext build install --user
```

Using the makefile:

```
cd trunk/crow/  
make -f Makefile.linux
```

3.1.4 Running the Test Suite

Next, the tests should be run (you may need to change the `cd` command if you are not in the `crow` directory):

```
cd ../raven/  
./run_tests --re=framework --skip-config-check
```

The output should describe why any tests failed. At the end, there should be a line that looks similar to the output below:

```
8 passed, 19 skipped, 0 pending, 0 failed
```

3.2 Ubuntu Framework Install

3.2.1 Dependencies

Install the dependencies using the command:

```
sudo apt-get install libtool python-dev swig g++ python3-dev \  
python-numpy python-sklearn python-h5py
```

3.2.2 Installation

Untar the binary (if there is more than one version of the binary, the full filename will need to be used instead of *):


```
tar -xvzf raven_framework*_ubuntu.tar.gz
```

3.2.3 Running the Test Suite

Run the tests:

```
cd trunk/raven/  
./run_tests --re=framework --skip-config-check
```

The output should describe why any tests failed. At the end, there should be a line that looks similar to the output below:

```
8 passed, 19 skipped, 0 pending, 0 failed
```

3.3 Fedora Framework Install

3.3.1 Dependencies

Install the dependencies:

```
yum install swig libtool gcc-c++ python-devel python3-devel \  
numpy h5py scipy python-scikit-learn python-matplotlib-qt4
```

Dan: [\[\[Do we want the equivalent command for Ubuntu?\]\]](#) Andrea: [\[\[I think so. Please ask Josh.\]\]](#)

Optionally, if you want to be able to edit and rebuild the manual, you can install T_EX Live and its related packages:

```
yum install texlive texlive-subfigure texlive-stmaryrd
```

3.3.2 Installation

Untar the binary (if there is more than one version of the binary, the full filename will need to be used instead of *):

```
tar -xvzf raven_framework*_fedora.tar.gz
```

3.3.3 Running The Test Suite

Run the tests:

```
cd trunk/raven/  
./run_tests --re=framework --skip-config-check
```

The output should describe why any tests failed. At the end, there should be a line that looks similar to the output below:

```
8 passed, 19 skipped, 0 pending, 0 failed
```

3.4 OSX Framework Install

Open up the file `raven_libs_framework_and_crow.dmg`. Next, Open up the `raven_libs.pkg` inside, and install it. The files will be installed into `/opt/raven_libs`. Your `.bash_profile` will be modified to source the `/opt/raven_libs/environments/raven_libs_profile` file. This file sets up the environment variables `PYTHONPATH` and `PATH` so that the `raven_framework` command can be used.

3.5 MOOSE and RAVEN Source Install

3.5.1 MOOSE, RELAP-7, and Other Dependencies

First, MOOSE should be installed. Follow the instructions for MOOSE listed here: <http://mooseframework.org/getting-started/>.

Next, if the portion of RAVEN in C++ is desired, RELAP-7 needs to be installed. Follow the RELAP-7 instructions available at <https://hpcgitlab.inl.gov/idaholab/relap-7>

Ensure that MOOSE and RELAP-7 are installed in the same directory level.

3.5.2 Obtaining CROW and RAVEN

Then clone CROW and RAVEN:

```
git clone git@hpcgitlab.inl.gov:idaholab/crow.git  
git clone git@hpcgitlab.inl.gov:idaholab/raven.git
```

Install the RAVEN dependencies via one of the methods mentioned for the RAVEN framework (see Section 3.1.2).

3.5.3 Compilation

Then compile RAVEN:

```
cd raven
make
```

3.5.4 Running The Test Suite

Then run the tests:

```
./run_tests
```

The output should describe why any tests failed. At the end, there should be a line that looks similar to the output below:

```
8 passed, 19 skipped, 0 pending, 0 failed
```

4 Running RAVEN

The RAVEN code is a mixture of C++, C, and Python software. The entry point resides on the Python side and is accessible via a command line interface. After following the instructions in the previous Section, RAVEN is ready to be used. The RAVEN driver is contained in the folder “raven/framework.” To run RAVEN, open a terminal and use the following command (replace `<inputFileName.xml>` with your RAVEN input file):

```
python raven/framework/Driver.py <inputFileName.xml>
```

5 Raven Input Structure

The RAVEN code does not have a fixed calculation flow, since all of its basic objects can be combined in order to create a user-defined calculation flow. Thus, its input (XML) is organized in different XML blocks, each with a different functionality. The main input blocks are as follows:

- **<Simulation>**: The root node containing the entire input, all of the following blocks fit inside the *Simulation* block.
- **<RunInfo>**: Specifies the calculation settings (number of parallel simulations, etc.).
- **<Distributions>**: Defines distributions needed for describing parameters, etc.
- **<Samplers>**: Sets up the strategies used for exploring an uncertain domain.
- **<Functions>**: Details interfaces to external user-defined functions and modules.
- **<Models>**: Specifies codes, ROMs, post-processing analysis, etc. the user will be building and/or running.
- **<Steps>**: Combines other blocks to detail a step in the RAVEN workflow including I/O and computations to be performed.
- **<Data>**: Specifies internal data objects used by RAVEN.
- **<Databases>**: Lists the HDF5 databases used as input/output to a RAVEN run.
- **<OutStreamManager>**: Visualization and Printing system block.

Each of these blocks are explained in dedicated sections in the following chapters.

6 RunInfo

In the **RunInfo** block, the user specifies how the overall computation should be run. This block accepts several input settings that define how to drive the calculation and set up, when needed, particular settings for the machine the code needs to run on (e.g. queueing system, if not PBS, etc.). In the following subsections, we explain all the keywords and how to use them in detail.

6.1 RunInfo: Input of Calculation Flow

This sub-section contains the information regarding the XML nodes used to define the settings of the calculation flow that is being performed through RAVEN:

- **<WorkingDir>**, *string, required field*, specifies the absolute or relative (with respect to the location where RAVEN is run from) path to a directory that will store all the results of the calculations and where RAVEN looks for the files specified in the block **<Files>**. *Default = None*.
- **<Files>**, *comma separated string, required field*, lists the paths to any files required by the code.
- **<batchSize>**, *integer, required field*, specifies the number of parallel runs executed simultaneously (e.g., the number of driven code instances, e.g. RELAP5-3D, that RAVEN will spawn at the same time). *Default = 1*.
- **<Sequence>**, *comma separated string, required field*, is an ordered list of the step names that RAVEN will run (see Section 7).
- **<NumThreads>**, *integer, optional field*, can be used to specify the number of threads RAVEN should associate when running the driven software. For example, if RAVEN is driving a code named “FOO,” and this code has multi-threading support, this block is used to specify how many threads each instance of FOO should use (e.g. “FOO --n-threads=N” where N is the number of threads). *Default = 1 (or None when the driven code does not have multi-threading support)*.
- **<NumMPI>**, *integer, optional field*, can be used to specify the number of MPI CPUs RAVEN should associate when running the driven software. For example, if RAVEN is driving a code named “FOO,” and this code has MPI support, this block specifies how many MPI CPUs each instance of FOO should use (e.g. “mpiexec FOO -np N” where N is the number of CPUs). *Default = 1 (or None when the driven code does not have MPI support)*.
- **<totalNumCoresUsed>**, *integer, optional field*, is the global number of CPUs RAVEN is going to use for performing the calculation. When the driven code has MPI and/or

multi-threading support and the user specifies `NumThreads > 1` and `NumMPI > 1`, then `totalNumCoresUsed` is set according to the following formula:

`totalNumCoresUsed = NumThreads * NumMPI * batchSize`. *Default = 1.*

- **<precommand>**, *string, optional field*, specifies a command that needs to be inserted before the actual command that is used to run the external model (e.g., `mpiexec -n 8 precommand ./externalModel.exe (...)`). *Default = None.*
- **<postcommand>**, *string, optional field*, specifies a command that needs to be appended after the actual command that is used to run the external model (e.g., `mpiexec -n 8 ./externalModel.exe (...) postcommand`). *Default = None.*
- **<MaxLogFileSize>**, *integer, optional field*, specifies the maximum size of the log file in bytes. Every time RAVEN drives a code/software, it creates a logfile of the code's screen output. *Default = Inf. (N.B. This flag is not implemented yet.)*
- **<deleteOutExtension>**, *comma separated string, optional field*, specifies, if a run of an external model has not failed, which output files should be deleted by their extension (e.g., **<deleteOutExtension>**`txt, pdf`**</deleteOutExtension>** will delete all generated txt and pdf files). *Default = None.*
- **<delSucLogFiles>**, *boolean, optional field*, when True and the run of an external model has not failed (return code = 0), deletes the associated log files. *Default = False.*

6.2 RunInfo: Input of Queue Modes

In this sub-section, all of the keywords (XML nodes) for setting the queue system are reported.

- **<mode>**, *string, optional field*, can specify which kind of protocol the parallel environment should use. RAVEN currently supports two pre-defined “modes”:
 - **pbsdsh**: this “mode” uses the pbsdsh protocol to distribute the program running; more information regarding this protocol can be found in [1]. Mode “pbsdsh” automatically knows when it needs to generate a `qsub` command, by inquiring the machine environment:
 - * If RAVEN is executed in the HEAD node of an HPC system, RAVEN generates a `qsub` command, and instantiates and submits itself to the queue system.
 - * If the user decides to execute RAVEN from an “interactive node” (a certain number of nodes that have been reserved in interactive PBS mode), RAVEN, using the “pbsdsh” system, is going to utilize the reserved resources (CPUs and nodes) to distribute the jobs, but will not generate a `qsub` command.

- **mpi**: this “mode” uses `mpirexec` to distribute the running program; more information regarding this protocol can be found in [2]. Mode “MPI” can either generate a `qsub` command or can execute on selected nodes. In order to make the “mpi” mode generate a `qsub` command, an additional keyword (xml sub-node) needs to be specified:

- * If RAVEN is executed in the HEAD node of an HPC system, the user needs to input a sub-node, `<runQSUB>`, right after the specification of the mpi mode (i.e. `<mode>mpi<runQSUB/></mode>`). If the keyword is provided, RAVEN generates a `qsub` command, instantiates itself, and submits itself to the queue system.
- * If the user decides to execute RAVEN from an “interactive node” (a certain number of nodes that have been reserved in interactive PBS mode), RAVEN, using the “mpi” system, is going to utilize the reserved resources (CPUs and nodes) to distribute the jobs, but, will not generate a `qsub` command.

When the user decides to run in “mpi” mode without making RAVEN generate a `qsub` command, different options are available:

- * If the user decides to run on the local machine (either in local desktop/workstation or a remote machine), no additional keywords are needed (i.e. `<mode>mpi</mode>`).
- * If the user is running on multiple nodes, the node ids have to be specified:
 - the node ids can be specified in an external text file (node ids separated by blank space). This file needs to be provided in the XML node `<mode>`, introducing a sub-node named `<nodefile>` (e.g. `<mode>mpi<nodefile>/tmp/nodes</nodefile></mode>`).
 - the node ids can be contained in an environmental variable (node ids separated by blank space). This variable needs to be provided in the `<mode>` XML node, introducing a sub-node named `<nodefileenv>` (e.g. `<mode>mpi<nodefileenv>NODEFILE</nodefileenv></mode>>`).
 - If none of the above options are used, RAVEN will attempt to find the nodes’ information in the environment variable `PBS_NODEFILE`.
- **<NumNode>**, *integer, optional field*, specifies the number of nodes RAVEN should request when running on High Performance Computing (HPC) systems. *Default = None*.
- **<CustomMode>**, *xml node, optional field*, is an xml node where “advanced” users can implement newer “modes.” Please refer to sub-section 6.3 for advanced users.
- **<queueingSoftware>**, *string, optional field*. RAVEN has support for the PBS queueing system. If the platform provides a different queueing system, the user can specify its name here (e.g., PBS PROFESSIONAL, etc.). *Default = PBS PROFESSIONAL*.
- **<expectedTime>**, *colum separated string, requested field (pbsdsh mode)*, specifies the time the whole calculation is expected to last. The syntax of this node is *hours:minutes:seconds* (e.g. 40:10:30 equals 40 hours, 10 minutes, 30 seconds). After this period of time, the

HPC system will automatically stop the simulation (even if the simulation is not completed). It is preferable to rationally overestimate the needed time. *Default = None*.

6.3 RunInfo: Advanced Users

This sub-section addresses some customizations of the running environment that are possible in RAVEN. Firstly, all the keywords reported in the previous sections can be pre-defined by the user in an auxiliary XML input file. Every time RAVEN gets instantiated (i.e. the code is run), it looks for an optional file, named “default_runinfo.XML” contained in the “\home\username\.raven\” directory (i.e. “\home\username\.raven\default_runinfo.XML”). This file (same syntax as the RunInfo block defined in the general input file) will be used for defining default values for the data in the RunInfo block. In addition to the keywords defined in the previous sections, in the **<RunInfo>** node, an additional keyword can be defined:

- **<DefaultInputFile>**, *string, optional field*. In this block, the user can change the default xml input file RAVEN is going to look for if none have been provided as a command-line argument. *Default = “test.xml”*.

As already mentioned, this file is read to define default data for the RunInfo block. This means that all the keywords defined here will be overridden by any values specified in the actual RAVEN input file.

In section refsubsec:runinfoModes, it has been explained how RAVEN can handle the queue and parallel systems. If the currently available “modes” are not suitable for the user’s system (workstation, HPC system, etc.), it is possible to define a custom “mode” modifying the **<RunInfo>** block as follows:

```
<RunInfo>
...
<CustomMode file="newMode.py" class="NewMode">
    aNewMode
</CustomMode>
<mode>aNewMode</mode>
...
</RunInfo>
```

The python file should define a class that inherits from `Simulation.SimulationMode` of the RAVEN framework and overrides the necessary functions. Generally, `modifySimulation` will be overridden to change the precommand or postcommand parts which will be added before and after the executable command. An example Python class is given below with the functions that can and should be overridden:

```

import Simulation
class NewMode(Simulation.SimulationMode):
    def doOverrideRun(self):
        # If doOverrideRun is true, then use runOverride instead of
        # running the simulation normally.
        # This method should call simulation.run somehow
        return True

    def runOverride(self):
        # this can completely override the Simulation's run method
        pass

    def modifySimulation(self):
        # modifySimulation is called after the runInfoDict has been
        # setup and allows the mode to change any parameters that
        # need changing. This typically modifies the precommand and
        # the postcommand that are put before/after the command.
        pass

    def XMLread(self, XMLNode):
        # XMLread is called with the mode node, and can be used to
        # get extra parameters needed for the simulation mode.
        pass

```

6.4 RunInfo: Examples

Here we present a few examples using different components of the RunInfo node:

```

<RunInfo>
  <WorkingDir>externalModel</WorkingDir>
  <Files>lorentzAttractor.py</Files>
  <Sequence>MonteCarlo</Sequence>
  <batchSize>100</batchSize>
  <NumThreads>4</NumThreads>
  <mode>mpi</mode>
  <NumMPI>2</NumMPI>
</RunInfo>

```

This examples specifies the working directory (WorkingDir) where the necessary file (Files) is located and to run a series of 100 (batchSize) Monte-Carlo calculations (Sequence).

MPI mode (mode) is used along with 4 threads (NumThreads) and 2 MPI processes per run (NumMPI).

7 Steps

The core of the RAVEN calculation flow is employed in the **Step** system. The **Step** is in charge of assembling different entities in RAVEN (e.g. Samplers, Models, DataBases, etc.) in order to perform a task defined by the kind of step being used. A sequence of different **Steps** represents the calculation flow and core of the analysis the user wants to perform.

Before analyzing each **Step** type, it is worth some time to briefly explain how a general **Step** entity is organized and the concept of a “role” in its definition. In the following example, a general example of a **Step** is reported:

```
<Simulation>
...
<Steps>
...
  <WhateverStepType name=' aName' >
    <Role1 class=' aMainClassType'
      type=' aSubType' >userDefinedName1</Role1>
    <Role2 class=' aMainClassType'
      type=' aSubType' >userDefinedName2</Role2>
    <Role3 class=' aMainClassType'
      type=' aSubType' >userDefinedName3</Role3>
    <Role4 class=' aMainClassType'
      type=' aSubType' >userDefinedName4</Role4>
  </WhateverStepType>
...
</Steps>
...
</Simulation>
```

As made evident above, independently of the type, each **Step** consists of a list of entities organized into “Roles.” Each role represents a behavior the entity (object) will assume during the evaluation of the **Step**. In RAVEN, several different roles are available:

- **Input** represents the input of the **Step**. The allowable input objects depend on the type of **Model** in the **Step**.
- **Output** defines how to represent the results of an action performed by the **Model**. It is generally one of the following types: **Datas**, **DataBases**, or **OutputStreamManager**.

- **Model** represents a physical or mathematical system or behavior. The object used in this role defines the compatibility of the different **Inputs** and **Outputs** listed in this step.
- **Sampler** defines a sampling strategy to be performed.
- **Function** is an extremely important role. For example, when performing adaptive sampling it can be used to represent the metric of the transition regions. Also, it can be used to collapse information coming from a **Model**.
- **ROM** defines an acceleration Reduced Order Model to use for a **Step**.
- **SolutionExport** represents the container of the eventual output of a **Sampler**.

Depending on the **Step** type, different combinations of these roles can be used. For this reason, it is important to analyze each **Step** type in details.

7.1 SingleRun

The **SingleRun** is the simplest step the user can use to assemble a calculation flow. It is aimed to perform a single action, employed by a **Model**. For example, it can be used to run a single job (Code Model) and collect the outcome(s) in a “**Datas**” object of type **TimePoint** or **History** (see Section 10 for more details on available data representations).

The specifications of this Step must be defined within a **<SingleRun>** XML block. This XML node has the following definable attributes:

- **name**, *required string attribute*, user-defined name of this **Step**. **N.B.** This name can be used to reference this specific entity in the **<RunInfo>** block, under the **<Sequence>** node.
- **pauseAtEnd**, *optional boolean/string attribute*, if True (True values = True, yes, y, t, si, dajie), the code will pause at the end of the step, waiting for a user signal to continue. This is used in case one or more of the **Outputs** are of type **OutputStreamManager**. For example, it can be used when an **OutputStreamManager** of type **Plot** is output to the screen. Thus, allowing the user to interact with the **Plot** (e.g. rotate the figure, change the scale, etc.). *Default = False*.

In the **<SingleRun>** input block, the user needs to specify the objects needed for the different allowable roles. This step accepts the following roles:

- **<Input>**, *string, required parameter*, names an entity (defined elsewhere in the input) that will be used as input for the model specified in this step. This XML node accepts the following attributes:

- **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. For example, 'Files', 'Datas', 'DataBases', etc.
- **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the main object class. For example, if the **class** attribute is 'Datas', the **type** attribute might be 'TimePointSet'. N.B. The class 'Files' has no type (i.e. **type**='').

N.B. The **class** and, consequently, the **type** usable for this role depends on the particular **<Model>** being used. In addition, the user can specify as many **<Input>** nodes as needed.

- **<Model>**, *string, required parameter*, names an entity defined elsewhere in the input file to be used as a model for this step. This XML node accepts the following attributes:
 - **class**, *required string attribute*, main object class type. For this role, only 'Models' can be used.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the Models object class. For example, the **type** attribute might be 'Code', 'ROM', etc.
- **<Output>**, *string, required parameter* names an entity defined elsewhere in the input to use as the output for the **Model**. This XML node recognizes the following attributes:
 - **class**, *required string attribute*, main object class type. For this role, only 'Datas', 'DataBases', and 'OutputStreamManager' can be used.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the main object class. For example, if the **class** attribute is 'Datas', the **type** attribute might be 'TimePointSet'.

N.B. The number of **<Output>** nodes is unlimited.

Example:

```
<Steps>
...
<SingleRun name='StepName' pauseAtEnd='false'>
  <Input    class='Files'
            type=''>anInputFile.i</Input>
  <Input    class='Files'                type=''>aFile</Input>
  <Model    class='Models'               type='Code'>aCode</Model>
  <Output   class='DataBases' type='HDF5'>aDataBase</Output>
  <Output   class='Datas'
            type='History'>aData</Output>
</SingleRun>
```

...
</Steps>

7.2 MultiRun

The **MultiRun** step allows the user to assemble the calculation flow of an analysis that requires multiple “runs” of the same model. This step is generally used when the input (space) of the model needs to be perturbed by a particular sampling strategy.

The specifications of this type of step must be defined within a **<MultiRun>** XML block. This XML node recognizes the following list of attributes:

- **name**, *required string attribute*, user-defined name of this Step. **N.B.** As with other objects, this name can be used to reference this specific entity in the **<RunInfo>** block, under the **<Sequence>** node.
- **pauseAtEnd**, *optional boolean/string attribute*, if True (True values = True, yes, y, t, si, dajie), the code will pause at the end of the step, waiting for a user signal to continue. This is used in case one or more of the **Outputs** are of type **OutputStreamManager**. For example, it can be used when an **OutputStreamManager** of type **Plot** is output to the screen. Thus, allowing the user to interact with the **Plot** (e.g. rotate the figure, change the scale, etc.).
- **sleepTime**, *optional float attribute*, in this attribute the user can specify the waiting time (seconds) between two subsequent inquiries of the status of the submitted job (i.e. check if a run has finished). *Default = 0.05*.

In the **<MultiRun>** input block, the user needs to specify the objects that need to be used for the different allowable roles. This step accepts the following roles:

- **<Input>**, *string, required parameter*, names an entity to be used as input for the model specified in this step. This XML node accepts the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object’s type used in the input. For example, ‘**Files**’, ‘**Datas**’, ‘**DataBases**’, etc.
 - **type**, *required string attribute*, the actual entity type. This attribute specifies the object type within the main object class. For example, if the **class** attribute is ‘**Datas**’, the **type** attribute might be ‘**TimePointSet**’. **N.B.** The class ‘**Files**’ has no type (i.e. **type**=‘ ’).

N.B. The **class** and, consequently, the **type** usable for this role depend on the particular **<Model>** being used. In addition, the user can specify as many **<Input>** nodes as needed.

- **<Model>**, *string, required parameter* names an entity defined elsewhere in the input that will be used as the model for this step. This XML node recognizes the following attributes:

- **class**, *required string attribute*, main object class type. For this role, only '**Models**' can be used.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the **Models** object class. For example, the **type** attribute might be '**Code**', '**ROM**', etc.
- **<Sampler>**, *string, required parameter* names an entity defined elsewhere in the input file to be used as a sampler. As mentioned in Section 9, the **Sampler** is in charge of defining the strategy to characterize the input space. This XML node recognizes the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used. Only '**Samplers**' can be used for this role.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the **Samplers** object class. For example, the **type** attribute might be '**MonteCarlo**', '**Adaptive**', '**AdaptiveDET**', etc. See Section 9 for all the different types currently supported.
 - **<SolutionExport>**, *string, optional parameter* identifies an entity to be used for exporting key information coming from the **Sampler** object during the simulation. This XML node accepts the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. For this role, only '**Datas**' can be used.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the **Datas** object class. For example, the **type** attribute might be '**Code**', '**ROM**', etc.

N.B. Whether or not it is possible to export the **Sampler** solution depends on the **type**. Currently, only the **Samplers** in the '**Adaptive**' category can export their solution into a **<SolutionExport>** entity. For example, if **<Sampler>** is of type '**Adaptive**', the **<SolutionExport>** needs to be of type '**TimePointSet**' and it will contain the coordinates, in the input space, that belong to the "Limit Surface."
 - **<Output>**, *string, required parameter* identifies an entity to be used as output for this step. This XML node recognizes the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. For this role, only '**Datas**', '**DataBases**', and '**OutputStreamManager**' may be used.
 - **type**, *required string attribute*, the actual entity type. This attribute specifies the object type within the main object class. For example, if the **class** attribute is '**Datas**', the **type** attribute might be '**TimePointSet**'.

N.B. The number of **<Output>** nodes is unlimited.

Example:

```
<Steps>
...
<MultiRun name='StepName1' pauseAtEnd='False' sleepTime='0.01'>
  <Input class='Files' type=''>anInputFile.i</Input>
  <Input class='Files' type=''>aFile</Input>
  <Sampler class='Samplers' type='Grid'>aGridName</Sampler>
  <Model class='Models' type='Code'>aCode</Model>
  <Output class='DataBases' type='HDF5'>aDataBase</Output>
  <Output class='Datas' type='History'>aData</Output>
</MultiRun >
<MultiRun name='StepName2' pauseAtEnd='True' sleepTime='0.02'>
  <Input class='Files' type=''>anInputFile.i</Input>
  <Input class='Files' type=''>aFile</Input>
  <Sampler class='Samplers' type='Adaptive'>anAS</Sampler>
  <Model class='Models' type='Code'>aCode</Model>
  <Output class='DataBases' type='HDF5'>aDataBase</Output>
  <Output class='Datas' type='History'>aData</Output>
  <SolutionExport class='Datas' type='TimePointSet'>
    aTPS
  </SolutionExport>
</MultiRun>
...
</Steps>
```

7.3 IOStep

As the name suggests, the **IOStep** is the step where the user can perform input/output operations among the different I/O entities available in RAVEN. This step type is used to:

- construct/update a *DataBase* from a *Datas* object, and vice versa;
- construct/update a *DataBase* or a *Datas* object from CSV files contained in a directory;
- stream the content of a *DataBase* or a *Datas* out through an **OutStream** object (see section 14).

The specifications of this type of step must be defined within an **<IOStep>** XML block. This XML node can accept the following attributes:

- **name**, *required string attribute*, user-defined name of this Step. **N.B.** As for the other objects, this is the name that can be used to refer to this specific entity in the `<RunInfo>` block, under the `<Sequence>` node.
- **pauseAtEnd**, *optional boolean/string attribute*, if True (True values = True, yes, y, t, si, dajie), the code will pause at the end of the step, waiting for a user signal to continue. This is used in case one or more of the **Outputs** are of type **OutputStreamManager**. For example, it can be used when an **OutputStreamManager** of type **Plot** is output to the screen. Thus, allowing the user to interact with the **Plot** (e.g. rotate the figure, change the scale, etc.). *Default = False*.

In the `<IOStep>` input block, the user specifies the objects that need to be used for the different allowable roles. This step accepts the following roles:

- **<Input>**, *string, required parameter*, names an entity that is going to be used as a source (input) from which the information needs to be extracted. This XML node recognizes the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. As already mentioned, the allowable main classes are '**Datas**' and '**DataBases**'.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the main object class. For example, if the **class** attribute is '**Datas**', the **type** attribute might be '**TimePointSet**'.
- **<Output>**, *string, required parameter* names an entity to be used as the target (output) where the information extracted in the input will be stored. This XML node needs to contain the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. The allowable main classes are '**Datas**', '**DataBases**', and '**OutputStreamManager**'.
 - **type**, *required string attribute*, the actual entity type. This attribute specifies the object type within the main object class. For example, if the **class** attribute is '**OutputStreamManager**', the **type** attribute might be '**Plot**'.

This step acts as a “transfer network” among the different RAVEN storing (or streaming) objects. The number of `<Input>` and `<Output>` nodes is unlimited, but should match. This step assumes a 1-to-1 mapping (e.g. first `<Input>` is going to be used for the first `<Output>`, etc.).

N.B. This 1-to-1 mapping is not present when `<Output>` nodes are of **class** '**OutputStreamManager**'.

In this case, the user needs to provide in the `<Input>` nodes, all of the “Datas” objects linked to the **OutputStreamManager** objects (see the example below): **Dan: [[I don't understand?]]**

```

<Steps>
...
<IOStep name='OutStreamStep'>
  <Input class='Datas' type='Histories'>aHistories</Input>
  <Input class='Datas' type='TimePointSet'>aTPS</Input>
  <Output class='OutStreamManager' type='Plot'>plot_hist
</Output>
  <Output class='OutStreamManager' type='Print'>print_hist
</Output>
  <Output class='OutStreamManager' type='Print'>print_tps
</Output>
  <Output class='OutStreamManager' type='Print'>print_tp
</Output>
</IOStep>
<IOStep name='PushDatasIntoDataBase'>
  <Input class='Datas' type='Histories'>aHistories</Input>
  <Input class='Datas' type='TimePointSet'>aTPS</Input>
  <Output class='DataBases' type='HDF5'>aDataBase</Output>
  <Output class='DataBases' type='HDF5'>aDataBase</Output>
</IOStep>
<IOStep name='ConstructDatasFromDataBase'>
  <Input class='DataBases' type='HDF5'>aDataBase</Input>
  <Input class='DataBases' type='HDF5'>aDataBase</Input>
  <Output class='Datas' type='Histories'>aHistories</Output>
  <Output class='Datas' type='TimePointSet'>aTPS</Output>
</IOStep>
...
</Steps>

```

7.4 RomTrainer

The **RomTrainer** step type performs the training of a Reduced Order Model. The specifications of this step must be defined within a **<RomTrainer>** block. This XML node accepts the attributes:

- **name**, *required string attribute*, user-defined name of this step. **N.B.** As for the other objects, this is the name that can be used to refer to this specific entity in the **<RunInfo>** block under **<Sequence>**.

In the **<RomTrainer>** input block, the user will specify the objects needed for the different allowable roles. This step accepts the following roles:

- **<Input>**, *string, required parameter* names an entity to be used as a source (input) from which the information needs to be extracted. This XML node accepts the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. The only allowable main class is 'Datas'.
 - **type**, *required string attribute*, the actual entity type. This attribute specifies the object type within the main object class. For example, the **type** attribute might be 'TimePointSet'. **N.B.** Since the ROMs currently present in RAVEN are not time-dependent, the only allowable types are 'TimePoint' and 'TimePointSet'.
- **<Output>**, *string, required parameter*, names a ROM entity that is going to be trained. This XML node recognizes the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main objects type used in the input. The only allowable main class is 'Models'.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the main object class. The only type accepted here is, currently, 'ROM'.

Example:

```
<Steps>
...
<RomTrainer name='aStepName' >
  <Input class='Datas' type='TimePointSet' >aTPS</Input>
  <Output class='Models' type='ROM' >aROM</Output>
</RomTrainer>
...
</Steps>
```

7.5 PostProcess

The **PostProcess** step is used to post-process data or manipulate RAVEN entities. It is aimed at performing a single action that is actually employed by a **Model** of type **PostProcessor**.

The specifications of this type of step is defined within a **<PostProcess>** XML block. This XML node specifies the following attributes:

- **name**, *required string attribute*, user-defined name of this Step. **N.B.** As for the other objects, this is the name that can be used to refer to this specific entity in the `<RunInfo>` block under the `<Sequence>` node.
- **pauseAtEnd**, *optional boolean/string attribute*, if True (True values = True, yes, y, t, si, dajie), the code will pause at the end of the step, waiting for a user signal to continue. This is used in case one or more of the **Outputs** are of type **OutputStreamManager**. For example, it can be used when an **OutputStreamManager** of type **Plot** is output to the screen. Thus, allowing the user to interact with the **Plot** (e.g. rotate the figure, change the scale, etc.). *Default = False.*

In the `<PostProcess>` input block, the user needs to specify the objects needed for the different allowable roles. This step accepts the following roles:

- **<Input>**, *string, required parameter*, names an entity to be used as input for the model specified in this step. This XML node accepts the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. For example, '**Files**', '**Datas**', '**DataBases**', etc.
 - **type**, *required string attribute*, the actual entity type. This attribute specifies the object type within the main object class. For example, if the **class** attribute is '**Datas**', the **type** attribute might be '**TimePointSet**'. **N.B.** The class '**Files**' has no type (i.e. **type**='').

N.B. The **class** and, consequently, the **type** usable for this role depends on the particular type of **PostProcessor** being used. In addition, the user can specify as many `<Input>` nodes as needed by the model.

- **<Model>**, *string, required parameter*, names an entity to be used as a model for this step. This XML node recognizes the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input. For this role, only '**Models**' can be used.
 - **type**, *required string attribute*, the actual entity type. This attribute needs to specify the object type within the '**Models**' object class. The only type accepted here is '**PostProcessor**'.
- **<Output>**, *string, required/optional parameter*, names an entity to be used as output for the PostProcessor. The necessity of this XML block and the types of entities that can be used as output depend on the type of **PostProcessor** that has been used as a **Model** (see section 13.5). This XML node specifies the following attributes:
 - **class**, *required string attribute*, main object class type. This string corresponds to the tag of the main object's type used in the input.

- **type**, *required string attribute*, the actual entity type. This attribute specifies the object type within the main object class. For example, if the **class** attribute is 'Datas', the **type** attribute might be 'TimePointSet'.

N.B. The number of **<Output>** nodes is unlimited.

Example:

```
<Steps>
...
<PostProcess name='PP1'>
  <Input class='Ddatas' type='TimePointSet' >aData</Input>
  <Model class='Models' type='PostProcessor'>aPP</Model>
  <Output class='Files' type=''>anOutputFile.csv</Output>
</PostProcess>
...
</Steps>
```

8 Distributions

Dan: [[Do we want to provide the equations of each distribution?]]

RAVEN provides support for several probability distributions. Currently, the user can choose among all of the most important 1-dimensional distributions and N -dimensional ones, either custom or multivariate.

The user will specify the probability distributions, that need to be used during the simulation, within the **<Distributions>** XML block:

```
<Simulation>
  ...
  <Distributions>
    <!-- All the necessary distributions will be listed here -->
  </Distributions>
  ...
</Simulation>
```

In the next two sub-sections, the input requirements for all of the distributions are reported.

8.1 1-Dimensional Probability Distributions

This sub-section is organized in two different parts: 1) continuous 1-D distributions and 2) discrete 1-D distributions. These two chapters cover all the requirements for using the different distribution entities.

8.1.1 1-Dimensional Continuous Distributions

In this paragraph all the 1-D distributions', currently available in RAVEN, are reported.

Firstly, all the probability distributions functions in the code can be truncated by using the following keywords:

```
<lowerBound>aFloatValue</lowerBound>
<upperBound>aFloatValue</upperBound>
```

Obviously, each distribution already defines its valid domain (e.g. Normal distribution, $[-\infty, +\infty]$).

RAVEN currently provides support for 12 1-Dimensional distributions. In the following paragraphs, all the input requirements are reported and commented.

8.1.1.1 Beta Distribution

The **Beta** distribution is defined on the interval $[0, 1]$ parameterized by two positive shape parameters, denoted by α and β , that appear as exponents of the random variable. The distribution domain can be changed, specifying new boundaries, to fit the user's needs. Its support is $x \in (0, 1)$.

The specifications of this distribution must be defined within a **<Beta>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<alpha>**, *float, required parameter*, first shape parameter.
- **<beta>**, *float, required parameter*, second shape parameter.
- **<low>**, *float, optional parameter*, lower domain boundary.
- **<high>**, *float, required parameter*, upper domain boundary.

Example:

```
<Distributions>
...
<Beta name='aUserDefinedName' >
  <low>aFloatValue</low>
  <high>aFloatValue</high>
  <alpha>aFloatValue</alpha>
  <beta>aFloatValue</beta>
</Beta>
...
</Distributions>
```

8.1.1.2 Exponential Distribution

The **Exponential** distribution can be used to model the time between independent events happening at a constant average time. Its support is $x \in [0, +\infty)$.

The specifications of this distribution must be defined within an **<Exponential>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following child:

- **<lambda>**, *float, required parameter*, rate parameter.

Example:

```
<Distributions>
...
<Exponential name='aUserDefinedName' >
  <lambda>aFloatValue</lambda>
</Exponential>
...
</Distributions>
```

8.1.1.3 Gamma Distribution

The **Gamma** distribution is a two-parameter family of continuous probability distributions. The common exponential distribution and χ -squared distribution are special cases of the gamma distribution. Its support is $x \in (0, +\infty)$.

The specifications of this distribution must be defined within a **<Gamma>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<alpha>**, *float, required parameter*, shape parameter.
- **<beta>**, *float, required parameter*, 1/scale or the inverse scale parameter;
- **<low>**, *float, optional parameter*, lower domain boundary.

Example:

```

<Distributions>
...
<Gamma name='aUserDefinedName' >
  <alpha>aFloatValue</alpha>
  <beta>aFloatValue</beta>
  <low>aFloatValue</low>
</Gamma>
...
</Distributions>

```

8.1.1.4 Logistic Distribution

The **Logistic** distribution is similar to the normal distribution with a CDF that is an instance of a logistic function. It resembles the normal distribution in shape but has heavier tails (higher kurtosis). Its support is $x \in (-\infty, +\infty)$.

The specifications of this distribution must be defined within a **<Logistic>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<location>**, *float, required parameter*, the distribution mean.
- **<scale>**, *float, required parameter*, scale parameter that is proportional to the standard deviation.

Example:

```

<Distributions>
...
<Logistic name='aUserDefinedName' >
  <location>aFloatValue</location>
  <scale>aFloatValue</scale>
</Logistic>
...
</Distributions>

```

8.1.1.5 LogNormal Distribution

The **LogNormal** distribution is a distribution with the logarithm of the random variable being normally distributed. Its support is $x \in (0, +\infty)$.

The specifications of this distribution must be defined within a **<LogNormal>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<mean>**, *float, required parameter*, the distribution mean or expected value (in log-scale).
- **<sigma>**, *float, required parameter*, standard deviation.

Example:

```
<Distributions>
...
<LogNormal name='aUserDefinedName' >
  <mean>aFloatValue</mean>
  <sigma>aFloatValue</sigma>
</LogNormal>
...
</Distributions>
```

8.1.1.6 Normal Distribution

The **Normal** distribution (or Gaussian) is an extremely useful continuous distribution. Its utility is due to the central limit theorem, which states that, under mild conditions, the mean of many random variables independently drawn from the same distribution is distributed approximately normally, irrespective of the form of the original distribution. Its support is $x \in (-\infty, +\infty)$.

The specifications of this distribution must be defined within a **<Normal>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<mean>**, *float, required parameter*, the distribution mean or expected value.
- **<sigma>**, *float, required parameter*, the standard deviation.

Example:

```
<Distributions>
...
<Normal name='aUserDefinedName' >
  <mean>aFloatValue</mean>
  <sigma>aFloatValue</sigma>
</Normal>
...
</Distributions>
```

8.1.1.7 Triangular Distribution

The **Triangular** distribution is a continuous distribution that has a triangular shape for its PDF. It is often used where the distribution is only vaguely known. Like the uniform distribution, upper and lower limits are “known,” but a “best guess,” of the mode or center point is also added. It has been recommended as a “proxy” for the beta distribution. Its support is $lower \leq x \leq upper$.

The specifications of this distribution must be defined within a **<Triangular>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<apex>**, *float, required parameter*, “best guess”, also called, peak factor.
- **<min>**, *float, required parameter*, domain lower boundary.
- **<max>**, *float, required parameter*, domain upper boundary.

Example:

```
<Distributions>
...
<Triangular name='aUserDefinedName' >
  <apex>aFloatValue</apex>
```

```

    <min>aFloatValue</min>
    <max>aFloatValue</max>
  </Triangular>
  ...
</Distributions>

```

8.1.1.8 Uniform Distribution

The **Uniform** distribution is a continuous distribution with a rectangular-shaped PDF. It is often used where the distribution is only vaguely known, but upper and lower limits are known. Its support is $lower \leq x \leq upper$.

The specifications of this distribution must be defined within a **<Uniform>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<low>**, *float, required parameter*, domain lower boundary.
- **<high>**, *float, required parameter*, domain upper boundary.

Example:

```

<Distributions>
  ...
  <Uniform name='aUserDefinedName' >
    <low>aFloatValue</low>
    <high>aFloatValue</high>
  </Uniform>
  ...
</Distributions>

```

8.1.1.9 Weibull Distribution

The **Weibull** distribution is a continuous distribution that is often used in the field of failure analysis; in particular, it can mimic distributions where the failure rate varies over time. If the failure rate is:

- constant over time, then $k = 1$, suggests that items are failing from random events;
- decreases over time, then $k < 1$, suggesting “infant mortality”;
- increases over time, then $k > 1$, suggesting “wear out” - more likely to fail as time goes by.

Its support is $x \in [0, +\infty)$.

The specifications of this distribution must be defined within a **<Weibull>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<k>**, *float, required parameter*, shape parameter.
- **<lambda>**, *float, required parameter*, scale parameter.

Example:

```
<Distributions>
...
<Weibull name='aUserDefinedName' >
  <lambda>aFloatValue</lambda>
  <k>aFloatValue</k>
</Weibull>
...
</Distributions>
```

8.1.2 1-Dimensional Discrete Distributions.

RAVEN currently supports 3 discrete distributions. In the following paragraphs, the input requirements are reported.

8.1.2.1 Bernoulli Distribution

The **Bernoulli** distribution is a discrete distribution of the outcome of a single trial with only two results, 0 (failure) or 1 (success), with a probability of success p . It is the simplest building block on which other discrete distributions of sequences of independent Bernoulli trials can be based. Basically, it is the binomial distribution ($k = 1, p$) with only one trial. Its support is $k \in 0, 1$.

The specifications of this distribution must be defined within a **<Bernoulli>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following child:

- **<p>**, *float, required parameter*, probability of success.

Example:

```
<Distributions>
...
<Bernoulli name='aUserDefinedName'>
  <p>aFloatValue</p>
</Bernoulli>
...
</Distributions>
```

8.1.2.2 Binomial Distribution

The **Binomial** distribution is the discrete probability distribution of the number of successes in a sequence of n independent yes/no experiments, each of which yields success with probability p . Its support is $k \in 0, 1, 2, \dots, n$.

The specifications of this distribution must be defined within a **<Binomial>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following children:

- **<p>**, *float, required parameter*, probability of success.
- **<n>**, *integer, required parameter*, number of experiments.

Example:

```
<Distributions>
...
<Binomial name='aUserDefinedName'>
  <n>aFloatValue</n>
```

```

    <p>aFloatValue</p>
  </Binomial>
  ...
</Distributions>

```

8.1.2.3 Poisson Distribution

The **Poisson** distribution is a discrete probability distribution that expresses the probability of a given number of events occurring in a fixed interval of time and/or space if these events occur with a known average rate and independently of the time since the last event. Its support is $k \in 1, 2, 3, 4, \dots$

The specifications of this distribution must be defined within a **<Poisson>** XML block. This XML node accepts one attribute:

- **name**, *required string attribute*, user-defined name of this distribution. **N.B.** As with other objects, this identifier can be used to reference this specific entity from other input blocks in the XML.

This distribution can be initialized with the following child:

- **<mu>**, *float, required parameter*, mean rate of events/time.

Example:

```

<Distributions>
  ...
  <Poisson name='aUserDefinedName' >
    <mu>aFloatValue</mu>
  </Poisson>
  ...
</Distributions>

```

8.2 N-Dimensional Probability Distributions

We have the multivariate normal distributions and 3 different types of user-input *ND* distributions. These types depend on the type of interpolation scheme requested by the user. The input requirements are explained in the following:

Andrea:[[Diego I choose you!]]

9 Samplers

The sampler is probably the most important entity in the RAVEN framework. Indeed, it performs the driving of the specific sampling strategy and, hence, determines the effectiveness of the analysis, from both an accuracy and computational point of view. The samplers, that are available in RAVEN, can be categorized in three main classes:

- **Once-through**
- **Dynamic Event Tree (DET)**
- **Adaptive**

Before analyzing each sampler in details, it is important to mention that each type has a similar syntax to input the variables that need to be “sampled”. In the example below the variable “variableName” is going to be sampled by the Sampler “WhateverSampler” using the distribution “aDistribution”.

```
-----  
<Simulation>  
...  
  <Samplers>  
    ...  
    <WhateverSampler name='whatever'>  
      ...  
      <variable name='variableName'>  
        ...  
        <distribution>aDistribution</distribution>  
        ...  
      </variable>  
      ...  
    </WhateverSampler>  
    ...  
  </Samplers>  
  ...  
</Simulation>  
-----
```

As reported in section 15, the variable naming syntax, for external driven codes, depends on the way the “code interface” has been implemented. For example, if the code has an input structure

like the one reported below, the variable name would be “ $I - Level|II - Level|variable$ ”; in this way, the relative code interface (and input parser) will know which variable needs to be perturbed and the “recipe” to access to it. As reported in 15, its syntax is chosen by the developer of the “code interface” and is implemented in the interface only (no modifications are needed in the RAVEN code).

```
-----
Example Input:
[I-Level]
  [./II-Level]
    variable = xxx
  [../]
[]
XML relative
-----
Example XML block:
<variable name=' 'I-Level|II-Level|variable' '>
  <distribution>exampleDistribution</distribution>
</variable>
-----
```

9.1 Once-through Samplers.

The once-through sampler category collects all the strategies that perform the sampling of the input space without exploiting, through dynamic learning approaches, the information made available from the outcomes of calculation previously performed (adaptive sampling) and the common system evolution (patterns) that different sampled calculations can generate in the phase space (dynamic event tree). In the RAVEN framework, five different and well-known “once-through” samplers are available:

- **Monte Carlo (MC)**
- **Stratified**
- **Grid Based**
- **Response Surface Design of Experiment**
- **Factorial Design of Experiment**

From a practical point of view, these sampling strategies represent different ways to perturb the input space. In the following paragraphs, the input requirements and a small explanation of the different sampling methodologies are reported.

9.1.1 Monte Carlo.

Monte-Carlo sampling approach is one of the most well-known and used approaches to perform exploration of the input space. The main idea, behind it, is the random perturbation of the input space accordingly with uniform or parameter-based probability density functions.

The specifications of this Sampler must be defined within the xml block `< MonteCarlo >`. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **limit**, *required integer attribute*, number of MonteCarlo samples needs to be generated;
- **initial seed**, *optional integer attribute*, initial seeding of random number generator. *Default = random seed*;
- **reseedAtEachIteration**, *optional boolean/string attribute*, perform a re-seeding for each sample generated (True values = True, yes, y, t, si, dajie). *Default = False*;

In the **MonteCarlo** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called `< variable >`:

- `< variable >`, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- `< distribution >`, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8.

Example:

```
-----
Example:
-----
<Samplers>
...
  <MonteCarlo name='MCname' limit='10' initial_seed='200286'
    reseedAtEachIteration='false'>
    <variable name='var1'>
      <distribution>***</distribution>
    </variable>
  </MonteCarlo>
...
</Samplers>
-----
```

9.1.2 Grid.

Grid sampling approach is probably the simplest exploration approach that can be employed to explore the uncertain domain. The main idea, behind it, is the construction of a N-Dimensional grid, where each dimension is represented by the uncertain variables. This approach performs the sampling of each node of the grid. The sampling of the grid consists in evaluating the answer of the system under all the possible combinations among the different variables' values, with respect a predefined discretization metric. In RAVEN two discretization metrics are available: 1) Cumulative Distribution Function, and 2) Value. Thus, the grid meshing can be inputted either in probability or in absolute values.

The specifications of this Sampler must be defined within the xml block `< Grid >`. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);

In the **Grid** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called `< variable >`:

- `< variable >`, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- `< distribution >`, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;
- `< grid >`, **float or space separated floats, required field.** The content of this xml node depends on the definition of the associated attributes:
 - * **type**, *required string attribute*, user-defined discretization metric type: 1) *CDF*, the grid is going to be specified based on Cumulative Distribution Function probability thresholds, and 2) *value*, the grid is going to be provided inputting absolute variable values;
 - * **construction**, *required string attribute*, how the grid needs to be constructed, independently by the its type (i.e. *CDF* or *value*).

Based on the *construction* type, the content of the `< grid >` xml node and the requirements for other attributes change:

- * *construction* = “equal”. The grid is going to be constructed equally-spaced (type = “value”) or equally-probable (type == “CDF”). This construction type requires the definition of additional attributes:

- **steps**, *required integer attribute*, number of equally-spaced/probable discretization steps.
- **upperBound**, *required float attribute*, the upper limit of the grid. NB. This attribute must be specified if the **lowerBound** has not been defined;
- **lowerBound**, *required float attribute*, the lower limit of the grid. NB. This attribute must be specified if the **upperBound** has not been defined;

This construction type requires that the content of the xml node `< grid >` represents the step size (either in probability or value). The attributes **lowerBound** and **upperBound** are mutually exclusive (only one of them can be specified):

If the *upperBound* is present, the grid lower bound is going to be at the *upperBound* – *steps * stepSize*

If the *lowerBound* is present, the grid upper bound is going to be at the *lowerBound* + *steps * stepSize*. The lower and upper bounds are checked against the associated `< distribution >` bounds. If one or both of them fell/s outside the distribution's bounds, the code is going to raise an error.

- * *construction* = "custom". The grid is going to directly be specified by the user. No additional attributes are needed.

This construction type requires that the xml node `< grid >` contains the actual mesh bins. For example, if the grid "type" is "CDF", in the body of `< grid >`, the user is going to specify CDF probability thresholds (nodalization in probability).

Example:

```

<Samplers>
...
<Grid name='Gridname'>
  <variable name='var1'>
    <distribution>***</distribution>
    <grid      type='value' construction='equal' steps='100'
      lowerBound='1.0'>0.2</grid>
  </variable>
  <variable name='var2'>
    <distribution>***</distribution>
    <grid      type='CDF' construction='equal' steps='5'
      lowerBound='0.0'>0.2</grid>
  </variable>
  <variable name='var3'>
    <distribution>***</distribution>
    <grid      type='value' construction='equal' steps='100'
      upperBound='21.0'>0.2</grid>
  </variable>

```

```

<variable name='var4'>
  <distribution>***</distribution>
  <grid type='CDF' construction='equal' steps='5'
    upperBound='1.0'>0.2</grid>
</variable>
<variable name='var5'>
  <distribution>***</distribution>
  <grid type='value' construction='custom'>0.2 0.5
    10.0</grid>
</variable>
<variable name='var6'>
  <distribution>***</distribution>
  <grid type='CDF' construction='custom'>0.2 0.5
    1.0</grid>
</variable>
</Grid>
...
</Samplers>
-----

```

9.1.3 Stratified.

Stratified sampling approach is a method for the exploration of the input space that basically consists in dividing the uncertain domain into subgroups before sampling. In the “stratified” sampling, these subgroups must be:

- mutually exclusive: every element in the population must be assigned to only one stratum (subgroup);
- collectively exhaustive: no population element can be excluded.

Then simple random sampling or systematic sampling is applied within each stratum. It is worth to notice that the well-known Latin Hyper Cube sampling represents a specialization of the Stratified approach, when the domain strata are constructed in equally-probable CDF bins.

The specifications of this Sampler must be defined within the xml block `< Stratified >`. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);

In the **Stratified** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called *< variable >*:

- *< variable >*, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- *< distribution >*, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;
- *< grid >*, **float or space separated floats, required field.** The content of this xml node depends on the definition of the associated attributes:
 - * **type**, *required string attribute*, user-defined discretization metric type: 1) *CDF*, the grid is going to be specified based on Cumulative Distribution Function probability thresholds, and 2) *value*, the grid is going to be provided inputting absolute variable values;
 - * **construction**, *required string attribute*, how the grid needs to be constructed, independently by the its type (i.e. *CDF* or *value*).

Based on the *construction* type, the content of the *< grid >* xml node and the requirements for other attributes change:

- * *construction* = “equal”. The grid is going to be constructed equally-spaced (type = “value”) or equally-probable (type == “CDF”). This construction type requires the definition of additional attributes:
 - **steps**, *required integer attribute*, number of equally-spaced/probable discretization steps.
 - **upperBound**, *required float attribute*, the upper limit of the grid. NB. This attribute must be specified if the **lowerBound** has not been defined;
 - **lowerBound**, *required float attribute*, the lower limit of the grid. NB. This attribute must be specified if the **upperBound** has not been defined;

This construction type requires that the content of the xml node *< grid >* represents the step size (either in probability or value). The attributes **lowerBound** and **upperBound** are mutually exclusive (only one of them can be specified):

If the *upperBound* is present, the grid lower bound is going to be at the *upperBound – steps * stepSize*

If the *lowerBound* is present, the grid upper bound is going to be at the *lowerBound + steps * stepSize* The lower and upper bounds are checked against the associated *< distribution >* bounds. If one or both of them fell/s outside the distribution’s bounds, the code is going to raise an error.

- * *construction* = “*custom*”. The grid is going to directly be specified by the user. No additional attributes are needed.

This construction type requires that the xml node `< grid >` contains the actual mesh bins. For example, if the grid “type” is “CDF”, in the body of `< grid >`, the user is going to specify CDF probability thresholds (nodalization in probability).

As it is inferable from above, the input specifications for the **Stratified** sampler are similar the Grid sampler ones. It is important to mention again that for each zone (grid mesh) only a point, randomly selected, is picked and not all the nodal combinations (like in the Grid sampling).

Example:

```

-----
Example:
-----
<Samplers>
...
  <Stratified name='StratifiedName'>
    <variable name='var1'>
      <distribution>***</distribution>
      <grid type='CDF' construction='equal' steps='5'
        lowerBound='0.0'>0.2</grid>
    </variable>
    <variable name='var2'>
      <distribution>***</distribution>
      <grid type='value' construction='equal' steps='100'
        upperBound='21.0'>0.2</grid>
    </variable>
    <variable name='var3'>
      <distribution>***</distribution>
      <grid type='CDF' construction='custom'>0.2 0.5
        1.0</grid>
    </variable>
  </Stratified>
...
</Samplers>
-----

```

9.1.4 Response Surface Design.

Response Surface Design approach is one of the most common Design of Experiment (DOE) methodology that are currently in use. It explores the relationships between several explanatory variables and one or more response variables. The main idea of RSM is to use a sequence of

designed experiments to obtain an optimal response. RAVEN Currently employs two different algorithms that can be classified within this methodology family:

- **Box-Behnken** design: this methodology has the aim to achieve the following goals:
 - Each factor, or independent variable, is placed at one of three equally spaced values, usually coded as -1, 0, +1. (At least three levels are needed for the following goal);
 - The design should be sufficient to fit a quadratic model, that is, one containing squared terms and products of two factors;
 - The ratio of the number of experimental points to the number of coefficients in the quadratic model should be reasonable (in fact, their designs kept it in the range of 1.5 to 2.6);
 - The estimation variance should more or less depend only on the distance from the center (this is achieved exactly for the designs with 4 and 7 factors), and should not vary too much inside the smallest (hyper)cube containing the experimental points.

Each design can be thought of as a combination of a two-level (full or fractional) factorial design with an incomplete block design. In each block, a certain number of factors are put through all combinations for the factorial design, while the other factors are kept at the central values.

- **Central Composite** design: the design consists of three distinct sets of experimental runs:
 - A factorial (perhaps fractional) design in the factors studied, each having two levels;
 - A set of center points, experimental runs whose values of each factor are the medians of the values used in the factorial portion. This point is often replicated in order to improve the precision of the experiment;
 - A set of axial points, experimental runs identical to the centre points except for one factor, which will take on values both below and above the median of the two factorial levels, and typically both outside their range. All factors are varied in this way.

This methodology is useful for building a second order (quadratic) model for the response variable without needing to use a complete three-level factorial experiment.

All the parameters, needed for setting up the algorithms reported above, must be defined within the xml block `< ResponseSurfaceDesign >`. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);

In the **ResponseSurfaceDesign** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called *< variable >*:

- *< variable >*, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- *< distribution >*, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;
- *< boundaries >*, **xml node, required field.** Within this block the boundaries for this variable are defined. This xml-node needs to contain the attribute:
 - * **type**, *required string attribute*, how the boundaries are defined. This attribute can be:
 - either *CDF*: the boundaries are going to be provided as probability CDF thresholds
 - or *value*: the boundaries are going to be provided as variable values.

Within the *< boundaries >* xml block, the following sub-nodes need to be provided:

- * *< lower >*, **float, required field.** The lower limit of this variable;
- * *< upper >*, **float, required field.** The upper limit of this variable.

The main xml block *< ResponseSurfaceDesign >* needs to contain an additional sub-node called *< ResponseSurfaceDesignSettings >*. In this sub-node, the user needs to specify different settings, depending on the algorithm needs to be used:

- *< type >*, **string, required field.** This xml node needs to contain the name of the algorithm needs to be used. Based on the chosen algorithm, other nodes need to be defined:
 - *< type > BoxBehnken < type/ >*. If Box-Behnken is specified, the following additional node might be inputted:
 - * *< ncenters >*, **integer, optional field.** The number of center points to include in the box. If this parameter is not specified, then a pre-determined number of points are automatically included. *Default = None*.

NB. In order to employ the “Box-Behnken” design, at least 3 variables must be inputted.

- *< type > CentralComposite < type/ >*. If Central Composite is specified, the following additional nodes might be inputted:

- * *< centers >, comma separated integers, optional field..* The number of center points to be included. This block needs to contain 2 integers values separated by a comma. The first entry represents the number of centers to be added for the factorial block; the second one the one for the star block. *Default = 4,4.*
- * *< alpha >, string, optional field..* In this node, the user might decides how α factor needs to be determined. Two options are available: 1) *alpha = orthogonal*, for orthogonal design, or 2) *alpha = rotatable*, for rotatable design. *Default = orthogonal.*
- * *< face >, string, optional field..* In this node, the user defines how faces should be constructed. Three options are available: 1) *face = circumscribed*, for circumscribed facing; 2) *face = inscribed*, for inscribed facing; 3) *face = faced*, for faced facing. *Default = circumscribed.*

NB. In order to employ the “Central Composite” design, at least 2 variables must be inputted.

Example:

<Samplers>

...

```

<ResponseSurfaceDesign name='BoxBehnkenRespDesign' >
  <ResponseSurfaceDesignSettings>
    <type>BoxBehnken</type>
    <ncenters>***</ncenters>
  </ResponseSurfaceDesignSettings>
  <variable name='var1' >
    <distribution >Gauss1</distribution>
    <boundaries type="CDF">
      <lower>0.0</lower>
      <upper>1.0</upper>
    </boundaries>
  </variable>
  <!-- N.B. at least 3 variables need to inputted
        in order to employ this algorithm
  -->
</ResponseSurfaceDesign>
<ResponseSurfaceDesign name='CentralCompositeRespDesign' >
  <ResponseSurfaceDesignSettings>
    <type>CentralComposite</type>
    <centers>***, ***</centers>
    <alpha>orthogonal</alpha>
    <face>circumscribed</face>
  </ResponseSurfaceDesignSettings>

```

```

<variable name='var4' >
  <distribution >Gauss1</distribution>
  <boundaries type="CDF">
    <lower>***</lower>
    <upper>***</upper>
  </boundaries>
</variable>
<!-- N.B. at least 2 variables need to inputted
      in order to employ this algorithm
-->
</ResponseSurfaceDesign>
...
</Samplers>
-----

```

9.1.5 Factorial Design.

Factorial Design method is an important method to determine the effects of multiple variables on a response. Factorial design can reduce the number of experiments one has to perform by studying multiple factors simultaneously. Additionally, it can be used to find both main effects (from each independent factor) and interaction effects (when both factors must be used to explain the outcome). Factorial design tests all possible conditions. Because factorial design can lead to a large number of trials, which can become expensive and time-consuming, factorial design is best used for a small number of variables with few states (1 to 3). Factorial design works well when interactions between variables are strong and important and where every variable contributes significantly. RAVEN currently employs three different algorithms that can be classified within this methodology family:

- **General Full Factorial** design: this methodology explore the input space investigating all possible combinations of a set of factors (variables).
- **2-Level Fractional-Factorial** design: this methodology consists of a carefully chosen subset (fraction) of the experimental runs of a full factorial design. The subset is chosen so as to exploit the sparsity-of-effects principle to expose information about the most important features of the problem studied, while using a fraction of the effort of a full factorial design in terms of experimental runs and resources.
- **Plackett-Burman** design: this method is used to identify the most important factors early in the experimentation phase when complete knowledge about the system is usually unavailable. It is an efficient screening method to identify the active factors (variables) using as

few samples as possible. In Plackett-Burman designs, main effects have a complicated confounding relationship with two-factor interactions. Therefore, these designs should be used to study main effects when it can be assumed that two-way interactions are negligible.

All the parameters, needed for setting up the algorithms reported above, must be defined within the xml block `< FactorialDesign >`. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);

In the **FactorialDesign** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called `< variable >`:

- `< variable >`, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- `< distribution >`, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;
- `< grid >`, **float or space separated floats, required field.** The content of this xml node depends on the definition of the associated attributes:
 - * **type**, *required string attribute*, user-defined discretization metric type: 1) *CDF*, the grid is going to be specified based on Cumulative Distribution Function probability thresholds, and 2) *value*, the grid is going to be provided inputting absolute variable values;
 - * **construction**, *required string attribute*, how the grid needs to be constructed, independently by the its type (i.e. *CDF* or *value*).

Based on the *construction* type, the content of the `< grid >` xml node and the requirements for other attributes change:

- * *construction* = “equal”. The grid is going to be constructed equally-spaced (type = “value”) or equally-probable (type == “CDF”). This construction type requires the definition of additional attributes:
 - **steps**, *required integer attribute*, number of equally-spaced/probable discretization steps.
 - **upperBound**, *required float attribute*, the upper limit of the grid. NB. This attribute must be specified if the **lowerBound** has not been defined;

- **lowerBound**, *required float attribute*, the lower limit of the grid. NB. This attribute must be specified if the **upperBound** has not been defined;

This construction type requires that the content of the xml node `< grid >` represents the step size (either in probability or value). The attributes **lowerBound** and **upperBound** are mutually exclusive (only one of them can be specified):

If the *upperBound* is present, the grid lower bound is going to be at the $upperBound - steps * stepSize$

If the *lowerBound* is present, the grid upper bound is going to be at the $lowerBound + steps * stepSize$. The lower and upper bounds are checked against the associated `< distribution >` bounds. If one or both of them fell/s outside the distribution's bounds, the code is going to raise an error.

- * *construction = "custom"*. The grid is going to directly be specified by the user. No additional attributes are needed.

This construction type requires that the xml node `< grid >` contains the actual mesh bins. For example, if the grid "type" is "CDF", in the body of `< grid >`, the user is going to specify CDF probability thresholds (nodalization in probability).

The main xml block `< FactorialDesign >` needs to contain an additional sub-node called `< FactorialSettings >`. In this sub-node, the user needs to specify different settings, depending on the algorithm needs to be used:

- `< type >`, *string, required field..* This xml node needs to contain the name of the algorithm needs to be used. Based on the chosen algorithm, other nodes need to be defined:
 - `< type > full < type/ >`. Full factorial design. If "full" is specified, no additional nodes need to be inputted. NB. The Full factorial design does not have any limitations in the number of discretization bins can be inputted in the `< grid >` xml node for each `< variable >`.
 - `< type > 2levelFract < type/ >`. Two-levels Fractional-Factorial design. If "level-Fract" is specified, the following additional nodes must be inputted:
 - * `< gen >`, *space separated strings, required field..* In this block the confounding mapping needs to inputted. By instance, in this block the user defines the decisions on a fraction of the full-factorial by allowing some of the factor main effects to be confounded with other factor interaction effects. This is done by defining an alias structure that defines, symbolically, these interactions. These alias structures are written like "C = AB" or "I = ABC", or "AB = CD", etc. These define how one column is related to the others.
 - * `< genMap >`, *space separated strings, required field..* In this block the user defines the mapping between the "gen" symbolic aliases and the variables that have been inputted in the `< FactorialDesign >` main block.

NB. The Two-levels Fractional-Factorial design is limited to 2 discretization bins that can be inputted in the `< grid >` xml node for each `< variable >`.

- `< type > pb < type/ >`. Plackett-Burman design. If “pb” is specified, no additional nodes need to be inputted.
NB. The Full factorial design does not have any limitations in the number of discretization bins can be inputted in the `< grid >` xml node for each `< variable >`.

Example:

<Samplers>

...

```

<FactorialDesign name='fullFactorial'>
  <FactorialSettings>
    <type>full</type>
  </FactorialSettings>
  <variable name='var1' >
    <distribution >***</distribution>
    <grid          type='value' construction='custom'
      >0.02 0.03 0.5</grid>
  </variable>
  <variable name='var2' >
    <distribution >***</distribution>
    <grid          type='CDF'
      construction='custom'>0.5 0.7 1.0</grid>
  </variable>
</FactorialDesign>
<FactorialDesign name='2levelFractFactorial'>
  <FactorialSettings>
    <type>2levelFract</type>
    <gen>a,b,ab</gen>
    <genMap>var1,var2,var3</genMap>
  </FactorialSettings>
  <variable name='var1' >
    <distribution >***</distribution>
    <grid          type='value' construction='custom'
      >0.02 0.5</grid>
  </variable>
  <variable name='var2' >
    <distribution >***</distribution>
    <grid          type='CDF'
      construction='custom'>0.5 1.0</grid>
  </variable>
  <variable name='var3'>
    <distribution >***</distribution>

```

```

        <grid type='value' upperBound='4'
            construction='equal' steps='1'>0.5</grid>
    </variable>
</FactorialDesign>
<FactorialDesign name='pbFactorial'>
    <FactorialSettings>
        <type>pb</type>
    </FactorialSettings>
    <variable name='var1' >
        <distribution >***</distribution>
        <grid type='value' construction='custom'
            >0.02 0.5</grid>
    </variable>
    <variable name='VarGauss2' >
        <distribution >***</distribution>
        <grid type='CDF'
            construction='custom'>0.5 1.0</grid>
    </variable>
</FactorialDesign>
...
</Samplers>

```

9.2 Dynamic Event Tree (DET) Samplers.

The Dynamic Event Tree methodologies are designed to take the timing of events explicitly into account, which can become very important especially when uncertainties in complex phenomena are considered. Hence, the main idea of this methodology is to let a system code determine the pathway of an accident scenario within a probabilistic “environment”. In this methodologies’ family a continuous monitoring of the system evolution in the phase space is needed. In order to use the DET-based methods, the generic driven code needs to have, at least, an internal trigger system and, consequentially, a “restart” capability. In the RAVEN framework, three different DET samplers are available:

- **Dynamic Event Tree (DET)**
- **Hybrid Dynamic Event Tree (HDET)**
- **Adaptive Dynamic Event Tree (ADET)**

The ADET methodology represents an hybrid between the DET and Adaptive sampling approaches. For this reason, its input requirements are reported in the Adaptive Samplers' section (9.3).

9.2.1 Dynamic Event Tree.

Dynamic Event Tree sampling approach is a sampling strategy that is designed to take the timing of events, in transient/accident scenarios, explicitly into a account.

From an application point of view, a N-Dimensional grid is built on the CDF space. A single simulation is spooned and a set of triggers is added to the system code control logic. Every time a trigger gets activated (one of the CDF thresholds in the grid is overpassed), a new set of simulations (branches) is spooned. Each branch carries its conditional probability. In RAVEN code, the triggers are defined specifying a grid, using a predefined discretization metric. In RAVEN two discretization metrics are available: 1) Cumulative Distribution Function, and 2) Value. Thus, the trigger thresholds can be inputted either in probability or in absolute values.

The specifications of this Sampler must be defined within the xml block `< DynamicEventTree >`. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **print_end_xml**, *optional string/boolean attribute*, this attribute controls the dumping of a “summary” of the DET performed in an xml external output. *Default = False*.
- **maxSimulationTime**, *optional float attribute*, this attribute controls the maximum “mission” time of the simulation underneath. *Default = None*.

In the **DynamicEventTree** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called `< variable >`:

- `< variable >`, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- `< distribution >`, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;
- `< grid >`, **float or space separated floats, required field.** The content of this xml node depends on the definition of the associated attributes:

- * **type**, *required string attribute*, user-defined discretization metric type: 1) *CDF*, the grid is going to be specified based on Cumulative Distribution Function probability thresholds, and 2) *value*, the grid is going to be provided inputting absolute variable values;
- * **construction**, *required string attribute*, how the grid needs to be constructed, independently by the its type (i.e. *CDF* or *value*).

Based on the *construction* type, the content of the `< grid >` xml node and the requirements for other attributes change:

- * *construction* = “*equal*”. The grid is going to be constructed equally-spaced (*type* = “*value*”) or equally-probable (*type* == “*CDF*”). This construction type requires the definition of additional attributes:
 - **steps**, *required integer attribute*, number of equally-spaced/probable discretization steps.
 - **upperBound**, *required float attribute*, the upper limit of the grid. NB. This attribute must be specified if the **lowerBound** has not been defined;
 - **lowerBound**, *required float attribute*, the lower limit of the grid. NB. This attribute must be specified if the **upperBound** has not been defined;

This construction type requires that the content of the xml node `< grid >` represents the step size (either in probability or value). The attributes **lowerBound** and **upperBound** are mutually exclusive (only one of them can be specified):

If the *upperBound* is present, the grid lower bound is going to be at the *upperBound* – *steps * stepSize*

If the *lowerBound* is present, the grid upper bound is going to be at the *lowerBound* + *steps * stepSize*. The lower and upper bounds are checked against the associated `< distribution >` bounds. If one or both of them fell/s outside the distribution’s bounds, the code is going to raise an error.

- * *construction* = “*custom*”. The grid is going to directly be specified by the user. No additional attributes are needed.

This construction type requires that the xml node `< grid >` contains the actual mesh bins. For example, if the grid “type” is “*CDF*”, in the body of `< grid >`, the user is going to specify CDF probability thresholds (nodalization in probability).

Example:

```

<Samplers>
...
<DynamicEventTree name='DETname' >
  <variable name='var1' >
    <distribution>***</distribution>
    <grid      type='value' construction='equal' steps='100'
      lowerBound='1.0'>0.2</grid>
  
```

```

</variable>
<variable name='var2'>
  <distribution>***</distribution>
  <grid type='CDF' construction='equal' steps='5'
    lowerBound='0.0'>0.2</grid>
</variable>
<variable name='var3'>
  <distribution>***</distribution>
  <grid type='value' construction='equal' steps='100'
    upperBound='21.0'>0.2</grid>
</variable>
<variable name='var4'>
  <distribution>***</distribution>
  <grid type='CDF' construction='equal' steps='5'
    upperBound='1.0'>0.2</grid>
</variable>
<variable name='var5'>
  <distribution>***</distribution>
  <grid type='value' construction='custom'>0.2 0.5
    10.0</grid>
</variable>
<variable name='var6'>
  <distribution>***</distribution>
  <grid type='CDF' construction='custom'>0.2 0.5
    1.0</grid>
</variable>
</DynamicEventTree>
...
</Samplers>
-----

```

9.2.2 Hybrid Dynamic Event Tree.

Hybrid Dynamic Event Tree sampling approach is a sampling strategy that represents an evolution of the Dynamic Event Tree method for the simultaneous exploration of the epistemic and aleatory uncertain space. In similar approaches, the uncertainties are generally treated employing a Monte-Carlo sampling approach (epistemic) and DET methodology (aleatory). The HDET methodology, developed within the RAVEN code, can reproduce the capabilities employed by this approach, but provides additional sampling strategies to the user. The epistemic or epistemic-like uncertainties can be sampled through the following strategies:

- Monte-Carlo;
- Grid sampling;
- Stratified (e.g., Latin Hyper Cube).

From a practical point of view, the user defines the parameters that need to be sampled by one or more different approaches. The HDET module samples those parameters creating a N-Dimensional Grid characterized by all the possible combinations of the input space coordinates coming from the different sampling strategies. Each coordinate in the input space represents a separated and parallel standard DET exploration of the uncertain domain. The HDET methodology allows the user to completely explore the uncertain domain employing one methodology. The addition of Grid sampling strategy among the approaches usable, allow the user to perform a discrete parametric study, under aleatory and epistemic uncertainties.

Regarding the input requirements, the HDET sampler is a “sub-type” of the *< DynamicEventTree >* sampler. For this reason, its specifications must be defined within the xml block *< DynamicEventTree >*. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **print_end_xml**, *optional string/boolean attribute*, this attribute controls the dumping of a “summary” of the DET performed in an xml external output. *Default = False*;
- **maxSimulationTime**, *optional float attribute*, this attribute controls the maximum “mission” time of the simulation underneath. *Default = None*.

In the **DynamicEventTree** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called *< variable >*:

- *< variable >*, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- *< distribution >*, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;
- *< grid >*, **float or space separated floats, required field.** The content of this xml node depends on the definition of the associated attributes:

- * **type**, *required string attribute*, user-defined discretization metric type: 1) *CDF*, the grid is going to be specified based on Cumulative Distribution Function probability thresholds, and 2) *value*, the grid is going to be provided inputting absolute variable values;
- * **construction**, *required string attribute*, how the grid needs to be constructed, independently by the its type (i.e. *CDF* or *value*).

Based on the *construction* type, the content of the `< grid >` xml node and the requirements for other attributes change:

- * *construction* = “equal”. The grid is going to be constructed equally-spaced (*type* = “value”) or equally-probable (*type* == “CDF”). This construction type requires the definition of additional attributes:
 - **steps**, *required integer attribute*, number of equally-spaced/probable discretization steps.
 - **upperBound**, *required float attribute*, the upper limit of the grid. NB. This attribute must be specified if the **lowerBound** has not been defined;
 - **lowerBound**, *required float attribute*, the lower limit of the grid. NB. This attribute must be specified if the **upperBound** has not been defined;

This construction type requires that the content of the xml node `< grid >` represents the step size (either in probability or value). The attributes **lowerBound** and **upperBound** are mutually exclusive (only one of them can be specified):

If the *upperBound* is present, the grid lower bound is going to be at the *upperBound* – *steps * stepSize*

If the *lowerBound* is present, the grid upper bound is going to be at the *lowerBound* + *steps * stepSize*. The lower and upper bounds are checked against the associated `< distribution >` bounds. If one or both of them fell/s outside the distribution’s bounds, the code is going to raise an error.

- * *construction* = “custom”. The grid is going to directly be specified by the user. No additional attributes are needed.
This construction type requires that the xml node `< grid >` contains the actual mesh bins. For example, if the grid “type” is “CDF”, in the body of `< grid >`, the user is going to specify CDF probability thresholds (nodalization in probability).

In order to activate the **Hybrid Dynamic Event Tree** sampler, the main xml block `< DynamicEventTree >` needs to contain, at least, an additional sub-node called `< HybridSamplerSettings >`. As already mentioned, the user can combine the Monte-Carlo, Stratified and Grid approaches in order to create a “pre-sampling” N-Dimensional grid, from whose nodes a standard DET method is employed. For this reason, the user can specify at maximum three `< HybridSamplerSettings >` sub-nodes (i.e. one for each of the available once-through samplers). This sub-node needs to contain the attribute:

- **type**, *required string attribute*, type of pre-sampling strategy needs to be used. Available are: 1) MonteCarlo, 2) Grid, and 3) Stratified.

Independently on the type of “pre-sampler” that has been specified, the *< HybridSamplerSettings >* must contain the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called *< variable >*:

- *< variable >*, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- *< distribution >*, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;

If the pre-sampling strategy is either **Grid** or **Stratified**, within the *< variable >* blocks, the user needs to specify the sub-node *< grid >*. As for the standard DET, the content of this xml node depends on the definition of the associated attributes:

- **type**, *required string attribute*, user-defined discretization metric type: 1) *CDF*, the grid is going to be specified based on Cumulative Distribution Function probability thresholds, and 2) *value*, the grid is going to be provided inputting absolute variable values;
- **construction**, *required string attribute*, how the grid needs to be constructed, independently by the its type (i.e. *CDF* or *value*).

Based on the *construction* type, the content of the *< grid >* xml node and the requirements for other attributes change:

- *construction* = “*equal*”. The grid is going to be constructed equally-spaced (type = “value”) or equally-probable (type == “CDF”). This construction type requires the definition of additional attributes:
 - **steps**, *required integer attribute*, number of equally-spaced/probable discretization steps.
 - **upperBound**, *required float attribute*, the upper limit of the grid. NB. This attribute must be specified if the **lowerBound** has not been defined;
 - **lowerBound**, *required float attribute*, the lower limit of the grid. NB. This attribute must be specified if the **upperBound** has not been defined;

This construction type requires that the content of the xml node *< grid >* represents the step size (either in probability or value). The attributes **lowerBound** and **upperBound** are mutually exclusive (only one of them can be specified):

If the *upperBound* is present, the grid lower bound is going to be at the *upperBound* –

$steps * stepSize$

If the *lowerBound* is present, the grid upper bound is going to be at the $lowerBound + steps * stepSize$. The lower and upper bounds are checked against the associated *< distribution >* bounds. If one or both of them fell/s outside the distribution's bounds, the code is going to raise an error.

- *construction = "custom"*. The grid is going to directly be specified by the user. No additional attributes are needed.

This construction type requires that the xml node *< grid >* contains the actual mesh bins. For example, if the grid "type" is "CDF", in the body of *< grid >*, the user is going to specify CDF probability thresholds (nodalization in probability).

Example:

```
<Samplers>
...
<DynamicEventTree name='HybridDETname' print_end_xml="True">
  <HybridSamplerSettings type='MonteCarlo' limit='2'>
    <variable name='***' >
      <distribution>***</distribution>
    </variable>
    <variable name='***' >
      <distribution>***</distribution>
      <grid type='CDF' construction='equal' steps='1'
        lowerBound='0.1'>0.1</grid>
    </variable>
  </HybridSamplerSettings>
  <HybridSamplerSettings type='Grid'>
    <!-- Point sampler way (directly sampling the
      variable) -->
    <variable name='***' >
      <distribution>***</distribution>
      <grid type='CDF' construction='equal' steps='1'
        lowerBound='0.1'>0.1</grid>
    </variable>
    <variable name='***' >
      <distribution>***</distribution>
      <grid type='CDF' construction='equal' steps='1'
        lowerBound='0.1'>0.1</grid>
    </variable>
  </HybridSamplerSettings>
  <HybridSamplerSettings type='Stratified'>
```

```

<!-- Point sampler way (directly sampling the variable
) -->
<variable name='***' >
  <distribution>***</distribution>
  <grid type='CDF' construction='equal' steps='1'
    lowerBound='0.1'>0.1</grid>
</variable>
<variable name='***' >
  <distribution>***</distribution>
  <grid type='CDF' construction='equal' steps='1'
    lowerBound='0.1'>0.1</grid>
</variable>
</HybridSamplerSettings>
<!-- DYNAMIC EVENT TREE INPUT (it goes outside an inner
block like HybridSamplerSettings) -->
<Distribution name='***' >
  <distribution >***</distribution>
  <grid type='CDF' construction='custom'>0.1 0.8</grid>
</Distribution>
</DynamicEventTree>
...
</Samplers>
-----

```

9.3 Adaptive Samplers.

The Adaptive Samplers' family provides the possibility to perform smart sampling (also known as adaptive sampling) as an alternative to classical "once-through" techniques. The motivation is that system simulations are often computationally expensive, time-consuming, and high dimensional with respect to the number of input parameters. Thus, exploring the space of all possible simulation outcomes is infeasible using finite computing resources. During simulation-based probabilistic risk analysis, it is important to discover the relationship between a potentially large number of input parameters and the output of a simulation using as few simulation trials as possible. This is a typical context for performing adaptive sampling where a few observations are obtained from the simulation, a reduced order model (ROM) is built to represent the simulation space, and new samples are selected based on the model constructed. The reduced order model (see section 13.3) is then updated based on the simulation results of the sampled points. In this way, it is attempted to gain the most information possible with a small number of carefully selected sampled points, limiting the number of expensive trials needed to understand features of the system space. In the following, the specific use case of identifying the limit surface, i.e. the boundaries in the simulation space between system failure and system success (see Figure 4) is analyzed. For this scope

two classes of algorithms are considered:

- Model-based algorithms;
- Data-based algorithms.

In the first class, the built reduced order model aims to approximate the real response function of the system as function of the input parameters. Once it is built, it is used to search for the points that are in the proximity of the limit surface using contour reconstruction based algorithms. Response function can be built using Support Vector Machines or Kriging based interpolators. On the other side, data-based algorithms do not build a response function based reduced order model but determine the location of the limit surface directly from the neighborhood graph constructed from the training data, without any dependencies on a particular prediction model. These algorithms begin the search of the limit surface by directly building a neighborhood structure as the surrogate model on the initial training data. It then creates a candidate set by first obtaining linearly interpolated points along spanning edges of the graph, and introducing a random perturbation along all dimensions to these points.

Currently, RAVEN provides support for two adaptive algorithm:

- Adaptive Sampler for Limit Surface Search;
- Adaptive Dynamic Event Tree.

In the following paragraphs, the input requirements and a small explanation of the different sampling methodologies are reported.

9.3.1 Adaptive Sampler.

Adaptive Sampler approach is an advanced methodology that employs a smart sampling around transition zones that determine a change in the status of the system (Limit Surface). To perform such sampling, RAVEN uses Reduced Order Models for predicting, in the input space, the location(s) of these transitions, in order to accelerate the exploration of the input space in proximity of the Limit Surface.

The specifications of this Sampler must be defined within the xml block `< Adaptive >`. This xml-node needs to contain the attribute:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);

In the **Adaptive** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called *< variable >*:

- *< variable >*, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-node needs to be specified:

- *< distribution >*, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8.

In addition to the *< variable >* nodes, the main xml node *< Adaptive >* needs to contain two supplementary sub-nodes:

- *< Convergence >*, **float, required field.** Convergence tolerance. The meaning of this tolerance depends on the definition of other attributes that might be defined in this xml node:
 - **limit**, *optional integer attribute*, the maximum number of adaptive samples. *Default = infinitive*;
 - **forceIteration**, *optional boolean attribute*, this attribute controls if at least a number of iterations equal to **limit** must be performed. *Default = False* ;
 - **weight**, *optional string attribute*, this attribute defines on what the convergence check need to be performed.
 - * *weight = probability*, the convergence is checked in terms of probability (Cumulative Distribution Function);
 - * *weight = none*, the convergence is checked on the hyper-volume in terms of absolute values.

Default = probability;

- **persistence**, *optional integer attribute*, this option is an additional convergence check. It represents the number of times the computed error needs to be below the inputted tolerance, in order to consider the algorithm in converged condition. *Default = 0*;
- **subGridTol**, *optional string attribute*, in this attribute the user can define a tolerance for constructing a sub-grid on which the acceleration ROM is going to be tested. *Default = None*;

Summarizing, this xml node contains the information that are needed in order to control this sampler convergence criterion.

- **Assembler Objects** These objects are either required or optional depending on the functionality of the Adaptive Sampler. The objects must be listed with a rigorous syntax that, except for the xml node tag, is common among all the objects. Each of these nodes must contain 2 attributes that are used to map those within the simulation framework:

- **class**, *required string attribute*, it is the main “class” the listed object is from. For example, it can be “Models”, “Functions”, etc;
- **type**, *required string attribute*, it is the object identifier or sub-type. For example, it can be “ROM”, “External”, etc.

The **Adaptive Sampler** approach requires or optionally accepts the following objects’ types:

- `< ROM >`, **string, optional field.** If inputted, the body of this xml node must contain the name of a ROM defined in the `< Models >` block (see section 13.3);
- `< Function >`, **string, required field.** The body of this xml block needs to contain the name of an External Function defined within the `< Functions >` main block (see section 12). This object represents the boolean function that defines the transition boundaries. This function must implement a method called `_residuumSign(self)`, that returns either -1 or 1, depending on the system conditions (see section 12);
- `< TargetEvaluation >`, **string, required field.** The target evaluation object represents the container where the system evaluations are stored. From a practical point of view, this xml node must contain the name of a Data defined in the `< Datas >` block (see section ??). The adaptive sampling accepts “Datas” of type “TimePoint” and “TimePointSet” only.

Example:

XML INPUT:

```

<Samplers>
...
<Adaptive name = 'AdaptiveName'>
  <ROM                      class = 'Models'      type = 'ROM'
                        >ROMname</ROM>
  <Function                class = 'Functions' type = 'External'
                        >FunctionName</Function>
  <TargetEvaluation class = 'Datas'      type =
    'TimePointSet'>DataName</TargetEvaluation>
<Convergence limit = '3000' subGridTol= '0.001'
  forceIteration = 'False' weight = 'none' persistence = '5'>
  1e-2
</Convergence>

```

```

    <variable name = 'var1'>
        <distribution>***</distribution>
    </variable>
    <variable name = 'var2'>
        <distribution>***</distribution>
    </variable>
    <variable name = 'var3'>
        <distribution>***</distribution>
    </variable>
</Adaptive>
...
</Samplers>

```

Example:

EXTERNAL FUNCTION:

```

def __residuumSign(self):
    if self.whatEverValue < self.OtherValue :
        return 1
    else:
        return -1
-----

```

9.3.2 Adaptive Dynamic Event Tree.

Adaptive Dynamic Event Tree approach is an advanced methodology that employs a smart sampling around transition zones that determine a change in the status of the system (Limit Surface), using the support of a Dynamic Event Tree methodology. The main idea of the application of the previously explained adaptive sampling approach to the DET comes from the observation that the DET, when evaluated from a Limit Surface perspective, is intrinsically adaptive. For this reason, it appears natural to use the DET approach to perform a goal-function oriented pre-sampling of the input space.

To perform such sampling, RAVEN uses Reduced Order Models for predicting, in the input space, the location(s) of these transitions, in order to accelerate the exploration of the input space in proximity of the Limit Surface.

The specifications of this Sampler must be defined within the xml block `< AdaptiveDynamicEventTree >`. This xml-node needs to contain the attributes:

- **name**, *required string attribute*, user-defined name of this Sampler. N.B. As for the other

objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);

- **print_end_xml**, *optional string/boolean attribute*, this attribute controls the dumping of a “summary” of the DET performed in an xml external output. *Default = False*;
- **maxSimulationTime**, *optional float attribute*, this attribute controls the maximum “mission” time of the simulation underneath. *Default = None*.
- **mode**, *optional string attribute*, adaptive DET mode. This attribute controls when the adaptive search needs to begin. Two options are available:
 - *mode = post*, if this option is activated, the sampler firstly performs a standard Dynamic Event Tree. At end of it, it uses the outcomes to start the adaptive search in conjunction with the DET support;
 - *mode = online*, if this option is activated, the adaptive search starts at begin, during the initial standard Dynamic Event Tree. Whenever a transition is detected, the **Adaptive Dynamic Event Tree** starts its goal-oriented search using the DET as support;

Default = post.

- **updateGrid**, *optional boolean attribute*, update DET grid flag. If it is true, each Adaptive Request is going to update the meshing of the initial DET grid. *Default = True*.

In the **AdaptiveDynamicEventTree** input block, the user needs to specify the variables need to be sampled. As already mentioned, these variables are inputted within consecutive xml blocks called *< variable >*:

- *< variable >*, xml node, required parameter. This xml-node needs to contain the attribute:
 - **name**, *required string attribute*, user-defined name of this variable.

In the variable node, the following xml-nodes need to be specified:

- *< distribution >*, **string, required field.** Name of the distribution that is associated to this variable. Its name needs to be contained in the **Distributions** block explained in sections 8;
- *< grid >*, **float or space separated floats, required field.** The content of this xml node depends on the definition of the associated attributes:
 - * **type**, *required string attribute*, user-defined discretization metric type: 1) *CDF*, the grid is going to be specified based on Cumulative Distribution Function probability thresholds, and 2) *value*, the grid is going to be provided inputting absolute variable values;
 - * **construction**, *required string attribute*, how the grid needs to be constructed, independently by its type (i.e. *CDF* or *value*).

Based on the *construction* type, the content of the `< grid >` xml node and the requirements for other attributes change:

- * *construction* = “equal”. The grid is going to be constructed equally-spaced (*type* = “value”) or equally-probable (*type* == “CDF”). This construction type requires the definition of additional attributes:
 - **steps**, *required integer attribute*, number of equally-spaced/probable discretization steps.
 - **upperBound**, *required float attribute*, the upper limit of the grid. NB. This attribute must be specified if the **lowerBound** has not been defined;
 - **lowerBound**, *required float attribute*, the lower limit of the grid. NB. This attribute must be specified if the **upperBound** has not been defined;

This construction type requires that the content of the xml node `< grid >` represents the step size (either in probability or value). The attributes **lowerBound** and **upperBound** are mutually exclusive (only one of them can be specified):

If the *upperBound* is present, the grid lower bound is going to be at the *upperBound* – *steps * stepSize*

If the *lowerBound* is present, the grid upper bound is going to be at the *lowerBound* + *steps * stepSize*. The lower and upper bounds are checked against the associated `< distribution >` bounds. If one or both of them fell/s outside the distribution’s bounds, the code is going to raise an error.

- * *construction* = “custom”. The grid is going to directly be specified by the user. No additional attributes are needed.

This construction type requires that the xml node `< grid >` contains the actual mesh bins. For example, if the grid “type” is “CDF”, in the body of `< grid >`, the user is going to specify CDF probability thresholds (nodalization in probability).

In addition to the `< variable >` nodes, the main xml node `< AdaptiveDynamicEventTree >` needs to contain two supplementary sub-nodes:

- `< Convergence >`, *float, required field*. Convergence tolerance. The meaning of this tolerance depends on the definition of other attributes that might be defined in this xml node:
 - **limit**, *optional integer attribute*, the maximum number of adaptive samples. *Default = infinitive*;
 - **forceIteration**, *optional boolean attribute*, this attribute controls if at least a number of iterations equal to **limit** must be performed. *Default = False* ;
 - **weight**, *optional string attribute*, this attribute defines on what the convergence check need to be performed.
 - * *weight* = *probability*, the convergence is checked in terms of probability (Cumulative Distribution Function);

- * *weight = none*, the convergence is checked on the hyper-volume in terms of absolute values.

Default = probability;

- **persistence**, *optional integer attribute*, this option is an additional convergence check. It represents the number of times the computed error needs to be below the inputted tolerance, in order to consider the algorithm in converged condition. *Default = 0*;
- **subGridTol**, *optional string attribute*, in this attribute the user can define a tolerance for constructing a sub-grid on which the acceleration ROM is going to be tested. *Default = None*;

Summarizing, this xml node contains the information that are needed in order to control this sampler convergence criterion.

- **Assembler Objects** These objects are either required or optional depending on the functionality of the Adaptive Sampler. The objects must be listed with a rigorous syntax that, except for the xml node tag, is common among all the objects. Each of these nodes must contain 2 attributes that are used to map those within the simulation framework:

- **class**, *required string attribute*, it is the main “class” the listed object is from. For example, it can be “Models”, “Functions”, etc;
- **type**, *required string attribute*, it is the object identifier or sub-type. For example, it can be “ROM”, “External”, etc.

The **Adaptive Sampler** approach requires or optionally accepts the following objects’ types:

- *< ROM >*, **string, optional field.** If inputted, the body of this xml node must contain the name of a ROM defined in the *< Models >* block (see section 13.3);
- *< Function >*, **string, required field.** The body of this xml block needs to contain the name of an External Function defined within the *< Functions >* main block (see section 12). This object represents the boolean function that defines the transition boundaries. This function must implement a method called *_residuumSign(self)*, that returns either -1 or 1, depending on the system conditions (see section 12);
- *< TargetEvaluation >*, **string, required field.** The target evaluation object represents the container where the system evaluations are stored. From a practical point of view, this xml node must contain the name of a Data defined in the *< Datas >* block (see section ??). The adaptive sampling accepts “Datas” of type “TimePoint” and “TimePointSet” only.

Example:

XML INPUT:

```

<Samplers>
...
<AdaptiveDynamicEventTree name = 'AdaptiveName'>
  <ROM                                class = 'Models'      type = 'ROM'
                                >ROMname</ROM>
  <Function                        class = 'Functions' type = 'External'
                                >FunctionName</Function>
  <TargetEvaluation class = 'Datas'      type =
    'TimePointSet'>DataName</TargetEvaluation>
  <Convergence limit = '3000' subGridTol= '0.001'
    forceIteration = 'False' weight = 'none' persistence =
    '5'>
    1e-2
  </Convergence>
  <variable name = 'var1'>
    <distribution>***</distribution>
    <grid type='CDF' construction='custom'>0.1 0.8</grid>
  </variable>
  <variable name = 'var2'>
    <distribution>***</distribution>
    <grid type='CDF' construction='custom'>0.1 0.8</grid>
  </variable>
  <variable name = 'var3'>
    <distribution>***</distribution>
    <grid type='CDF' construction='custom'>0.1 0.8</grid>
  </variable>
</AdaptiveDynamicEventTree>
...
</Samplers>

```

```

EXTERNAL FUNCTION:
def __residuumSign(self):
    if self.whatEverValue < self.OtherValue :
        return 1
    else:
        return -1
-----

```


10 Datas

As one could infer from the previous chapters, different entities in the RAVEN code interact with each other in order to create, ideally, an infinite number of different calculation flows. These interactions are made possible through a data handling system that each entity understands. This system, neglecting the grammar imprecision, is called the “Datas” system. The `Datas` tag is a container of data objects of different type that can be constructed during the execution of a particular calculation flow. These data objects can be used as input or output for a particular Model (see Roles’ meaning in section 13), etc. Currently, RAVEN supports 4 different data types, each with a particular conceptual meaning:

- **TimePoint:** The *TimePoint* entity, as the name suggests, describes the state of the system in a certain point in time. In other words, it can be considered a mapping between a set of parameters in the input space and the resulting outcomes in the output space at a particular time.
- **TimePointSet:** The *TimePointSet* object is a collection of individual *TimePoint(s)*. It can be considered a mapping between multiple sets of parameters in the input space and the resulting sets of outcomes in the output space at a particular point in time.
- **History:** The *History* entity describes the temporal evolution of the state of the system within a certain input domain.
- **Histories:** The *Histories* object is a collection of individual *History* objects. It can be considered a mapping between multiple sets of parameters in the input space and the resulting sets of temporal evolutions in the output space.

As noted above, each data object represents a mapping between a set of parameters and the resulting outcomes. The data objects are defined within the main XML block called `<Datas>`:

```
<Simulation>
...
<Datas>
  <TimePoint name='***'>...</TimePoint>
  <TimePointSet name='***'>...</TimePointSet>
  <History name='***'>...</History>
  <Histories name='***'>...</Histories>
</Datas>
...
</Simulation>
```

This example also demonstrates the specification of each data type must be defined within one of the following xml nodes (each corresponding to the data types previously listed):

- <TimePoint>
- <TimePointSet>
- <History>
- <Histories>

Independent of the type of data, the respective XML node has the following attributes:

- **name**, *required string attribute*, user-defined name of this data object. **N.B.** As with other objects, this name can be used to refer to this specific entity from other input blocks in the XML.
- **inputTs**, *optional integer attribute*, This attribute can be used to specify at which “time step” the input space needs to be retrieved. **N.B.** If the user wants to take those conditions from the end of the simulation, she/he can directly input “-1”. *Default = 0*.
- **operator**, *optional string attribute*, The operator attribute is aimed to perform simple operations on the data to be stored. The 3 options currently available are *operator = max*, *operator = min*, *operator = average*, with obvious meaning. *Default = None*;
- **hierarchical**, *optional boolean attribute*, If True this data is going to be constructed, if possible, in an hierarchical fashion. *Default = False*;

In each XML node (e.g. < TimePoint > or < Histories >), the user needs to specify the following sub nodes:

- < Inputs >, *comma separated string, required field..* List of input parameters this data is connected to. NB. In case the “Data” type is either *TimePoint* or *History*, this XML node must contain the attribute **history**, where the name of the associated history needs to be placed;
- < Outputs >, *comma separated string, required field..* List of output parameters this data is connected to;

```
-----  
<Datas>  
  <TimePoint name='***' inputTs='-1' operator='max'  
    hierarchical='False'>  
    <Input history='a_history_name'>***,***,***</Input>  
    <Output>***,***</Output>  
  </TimePoint>
```

```
<TimePointSet name='***'>
  <Input>***,***,***</Input>
  <Output>***,***</Output>
</TimePointSet>
<History name='***'>
  <Input history='a_history_name'>***,***,***</Input>
  <Output>***,***</Output>
</History>
<Histories name='***'>
  <Input>***,***,***</Input>
  <Output>***,***</Output>
</Histories>
</Datas>
```

11 Databases

The RAVEN framework provides the capability to store and retrieve data to/from an external database. Currently RAVEN has support for only a database type called **HDF5**. This database, depending on the data format is receiving, will organize itself in a “parallel” or “hierarchical” fashion. The user can create as many database objects as needed. The DataBase objects are defined within the main XML block called `< DataBases >`:

```
-----  
<Simulation>  
  ...  
  <DataBases>  
    ...  
    <HDF5 name="***" />  
    <HDF5 name="***" />  
    ...  
  </DataBases>  
  ...  
</Simulation>  
-----
```

The specifications of each DataBase of type HDF5 needs to be defined within the XML block `< HDF5 >`, that needs (or not) to contain the attributes:

- **name**, *required string attribute*, user-defined name of this Data. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **directory**, *optional string attribute*, this attribute can be used to specify a particular directory path where to create the database, if no *filename* is specified, or from where open an already existing one, if *filename* is provided. *Default = raven/framework/DataBaseStorage*;
- **filename**, *optional string attribute*, this attribute can be used to specified the filename of an HDF5 that already exists in the *directory*. This is the only way to let RAVEN know that an HDF5 should be opened and not overwritten. NB. When this attribute is not specified, the newer database filename is going to be named *name.h5*. *Default = None*;
- **compression**, *optional string attribute*, compression algorithm to be used. Available are:
 - *compression = gzip*, best where portability is required. Good compression, moderate speed;
 - *compression = lzf*, Low to moderate compression, very fast.

Default = no compression;

```
-----  
<DataBases>  
  <HDF5 name="***" directory=' 'path_to_a_dir' '  
    compression=' 'lzf' ' />  
  <HDF5 name="***" filename=' 'existing_hdf5.h5' ' />  
</DataBases>  
-----
```

12 Functions **Dan: [[Can we rename this?]]**

The RAVEN code provides support for the usage of user-defined external functions. These functions are python modules, with a format that is automatically interpretable by the RAVEN framework. For example, users can define their own method to perform a particular post-processing activity and the code will be embedded and use the function as though it were an active part of the code itself. In this section, the XML input syntax and the format of the accepted functions are fully specified.

The specifications of an external function must be defined within the XML block **<External>**. This XML node requires the following attributes:

- **name**, *required string attribute*, user-defined name of this function. **N.B.** As with other objects, this name can be used to refer to this specific entity from other input blocks in the XML.
- **file**, *required string attribute*, absolute or relative path specifying the code associated to this function. **N.B.** If a relative path is specified, it must be relative with respect to where the user is running the instance of RAVEN.

In order to make the RAVEN code aware of the variables the user is going to manipulate/use in her/his own python function, the variables need to be specified in the **<External>** input block. The user needs to input, within this block, only the variables directly used by the external code and not the local variables that the user does not want to, for example, those stored in a RAVEN internal object. These variables are named within consecutive **<variable>** XML nodes:

- **<variable>**, *string, required parameter*. In the body of this XML node, the user needs to specify the name of the variable. This variable needs to match a variable used/defined in the external python function.

When the external function variables are defined, at runtime, RAVEN initializes them and keeps track of their values during the simulation. Each variable defined in the **<External>** block is available in the function as a python “self.” In the following, an example of a user-defined external function is reported (python module and its relative XML input specifications).

Example Python Function:

```
import numpy as np

def residuumSign(self):
    if self.var1 < self.var2 :
        return 1
    else:
        return -1
```

Example XML Input:

```
...
<Functions>
  ...
  <External name='whatever' file='path_to_python_file'>
    ...
    <variable>var1</variable>
    <variable>var2</variable>
    ...
  </External>
  ...
</Functions>
...
</Simulation>
```

13 Models

In the RAVEN code a crucial entity is represented by a Model. A model is an object that employs a mathematical representation of a phenomenology, either physical or of other nature (e.g. statistical operators, etc.). From a practical point of view, it can be seen, as a “black box” that, given an input, returns an output.

In the RAVEN code, a strict classification of the different models is present. As obviously, each “class” of models is represented by the definition reported above, but it can be further classified based on the peculiar functionalities:

- **Code.** This “class” is the representation of an external system code that employs an high fidelity physical model;
- **Dummy.** The “Dummy” object is a model that acts as “transfer” tool. The only action it performs is transferring the the information in the input space (inputs) into the output space (outputs). For example, it can be used to check the effect of a Sampling strategy, since its outputs are the sampled parameters’ values (input space) and a counter that keeps track of the number of times an evaluation has been requested;
- **ROM.** A ROM is a mathematical model of fast solution trained to predict a response of interest of a physical system. The “training” process is performed by “sampling” the response of a physical model with respect variation of its parameters subject to probabilistic behavior. The results (outcomes of the physical model) of those sampling are fed into the algorithm representing the ROM that tunes itself to replicate those results;
- **ExternalModel.** As the name suggests, an external model is an entity that is embedded in the RAVEN code at run time. This object allows the user to create a python module that is going to be treated as a predefined internal model object;
- **PostProcessor.** The post-processor “class” of objects is the container of all the actions that can be performed to manipulate and process the data in order to extract key information, such as statistical quantities, etc.

Before analyzing each model in details, it is important to mention that each type needs to be contained in the main XML node `< Models >`, as reported below:

Example:

```
-----  
<Simulation>  
...
```



```

<Models>
  ...
  <WhateverModel name='whatever' >
    ...
  </WhateverModel>
  ...
</Models>
...
</Simulation>

```

In the following sub-sections each **Model** type is fully analyzed and described.

13.1 Code

As already mentioned, the model **Code** is the representation of an external system software that employs an high fidelity physical model. The link between RAVEN and the driven code is performed at run time, through coded interfaces that are the responsible of transferring the information from the code to RAVEN and vice versa. In section 15 all the available interfaces are reported and, for advanced users, section ?? explains how to couple a newer code.

The specifications of this Model must be defined within the xml block `< Code >`. This XML node needs to contain the attributes:

- **name**, *required string attribute*, user-defined name of this Model. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **subType**, *required string attribute*, in this attribute the user selects the code that needs to be associated to this Model. NB. See section 15 to check which codes are currently supported.

In the **Code** input block, the following XML sub-nodes are available:

- `< executable >`, *string, required field..* In this node, the user needs to specify the path of the executable to be used. NB. In this node, either the absolute or relative path can be inputted;
- `< alias >`, *string, optional field..* In the `< alias >` block the user can specify aliases for some variables of interest coming from the code this model refers to. These aliases can be used in the whole input to refer to the code variables. In the body of this node the user specifies the name of the variable that RAVEN will look for in the output files of the code. The actual alias, usable throughout the input, are instead defined in the attribute **variable**. NB. The user can specify as many aliases as needed. *Default = None*.

Example:

```
-----  
<Simulation>  
...  
<Models>  
...  
  <Code name='***' subType='RAVEN_Driven_code'>  
    <executable>path_to_executable</executable>  
    <alias variable='internal_variable_name1'>  
      External_Code_Variable_Name_1  
    </alias>  
    <alias variable='internal_variable_name2'>  
      External_Code_Variable_Name_2  
    </alias>  
  </Code>  
...  
</Models>  
...  
</Simulation>  
-----
```

13.2 Dummy.

The model **Dummy** is an object that acts as “transfer” tool. The only action it performs is transferring the the information in the input space (inputs) into the output space (outputs). For example, it can be used to check the effect of a Sampling strategy, since its outputs are the sampled parameters’ values (input space) and a counter that keeps track of the number of times an evaluation has been requested.

The specifications of this Model must be defined within the xml block `< Dummy >`. This XML node needs to contain the attributes:

- **name**, *required string attribute*, user-defined name of this Model. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **subType**, *required string attribute*, this attribute must be kept empty.

If this model, in a *Step*, is associated to a *Data* with the role of **Output**, it expects that one of the output parameters of such *Data* is identified by the keyword “OutputPlaceholder” (see section 7).

Example:

```
-----
```

```

<Simulation>
  ...
  <Models>
    ...
    <Dummy name='***' subType='' />
    ...
  </Models>
  ...
</Simulation>

```

13.3 ROM

A Reduced Order Model (ROM) is a mathematical model of fast solution trained to predict a response of interest of a physical system. The “training” process is performed by “sampling” the response of a physical model with respect variation of its parameters subject, for example, to probabilistic behavior. The results (outcomes of the physical model) of those sampling are fed into the algorithm representing the ROM that tunes itself to replicate those results. RAVEN supports several different types of ROMs, both internally developed and imported through an external library called “SciKitLearn” [3]. Currently in RAVEN the Reduced Order Models are classified in 4 main “classes” that, once chosen, provide access to several different algorithms:

- **NDspline;**
- **NDinvDistWeigth;**
- **microSphere;**
- **SciKitLearn.**

The specifications of this Model must be defined within the XML block $\langle ROM \rangle$. This XML node needs to contain the attributes:

- **name**, *required string attribute*, user-defined name of this Model. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **subType**, *required string attribute*, in this attribute the user defines which of the main “classes” needs to be used, choosing among the previously reported types. Obviously, this choice conditions the subsequent the required and/or optional $\langle ROM \rangle$ sub nodes.

In the **ROM** input block, the following XML sub-nodes are required, independently on the “main” class inputted in the attribute *subType*:

- *< Features > , comma separated string, required field..* In this node, the user needs to specify the names of the features of this ROM. NB. These parameters are going to be requested for the training of this object (see section ??);
- *< Target > , comma separated string, required field..* This XML node contains a comma separated list of the targets of this ROM. By Instance, these parameters are the Figure of Merits this ROM is supposed to predict. NB. These parameters are going to be requested for the training of this object (see section ??).

As already mentioned, all the types and meaning of the remaining sub-nodes depend on the main “class” type specified in the attribute *subType*. In the following sections the specifications of each type are reported.

13.3.1 NDspline.

The main “class” NDspline contains a single ROM type, based on a N-Dimensional spline interpolation/extrapolation. The spline interpolation is a form of interpolation where the interpolant is a special type of piecewise polynomial called a spline. The interpolation error can be made small even when using low degree polynomials for the spline. Spline interpolation avoids the problem of Runge’s phenomenon, in which oscillation can occur between points when interpolating using high degree polynomials.

In order to use this Reduced Order Model, the *< ROM >* attribute *subType* needs to be “ND-spline” (i.e. *subType* = “NDspline”). No further XML sub-nodes are required.

NB. This ROM type must be trained from a Regular Cartesian Grid. By instance, it can only be trained from the outcomes of a Grid Sampling strategy.

Example:

```

-----
<Simulation>
...
  <Models>
    ...
    <ROM name='***' subType='NDspline'>
      <Features>***, ***, ***</Features>
      <Target>***, ***</Target>
    </ROM>
    ...
  </Models>
  ...
</Simulation>
-----

```

13.3.2 NDinvDistWeigth.

The main “class” NDinvDistWeigth contains a single ROM type, based on a N-Dimensional Inverse Distance Weighting formulation. Inverse Distance Weighting (IDW) is a type of deterministic method for multivariate interpolation with a known scattered set of points. The assigned values to unknown points are calculated with a weighted average of the values available at the known points. In order to use this Reduced Order Model, the $\langle ROM \rangle$ attribute *subType* needs to be “NDinvDistWeigth” (i.e. *subType* = “NDinvDistWeigth”). The specification of the ROM “NDinvDistWeigth” needs to be completed inputting, within the main XML node $\langle ROM \rangle$, of the following sub-node:

- $\langle p \rangle$, *integer, required field.* This node contains an *integer* > 0 that represents the “power parameter”. For the choice of value for $\langle p \rangle$, it is necessary to consider the degree of smoothing desired in the interpolation/extrapolation, the density and distribution of samples being interpolated, and the maximum distance over which an individual sample is allowed to influence the surrounding ones (lower p means greater importance for points faraway).

Example:

```
-----  
<Simulation>  
...  
<Models>  
...  
  <ROM name='***' subType='NDinvDistWeigth'>  
    <Features>***, ***, ***</Features>  
    <Target>***</Target>  
    <p>3</p>  
  </ROM>  
...  
</Models>  
...  
</Simulation>  
-----
```

13.3.3 MicroSphere.

Not yet functional. Its validity for prediction purposes needs to be still assessed.

13.3.4 SciKitLearn.

The main “class” SciKitLearn represents the container of several Reduced Order Models that are available in RAVEN through the external library SciKitLearn [3].

In order to use this Reduced Order Model, the $\langle ROM \rangle$ attribute *subType* needs to be “SciKitLearn” (i.e. *subType* = “SciKitLearn”). The specifications of the ROM “SciKitLearn” depends on value assumed by the following sub-node within the main XML node $\langle ROM \rangle$:

- $\langle SKLtype \rangle$, **vertical bar (|) separated string , required field.**.. This nodes contains a string that represents the ROM type that needs to be used. As mentioned, its format is, for example, $\langle SKLtype \rangle mainSKLclass | algorithm \langle /SKLtype \rangle$: the first word (before symbol |) represents the main class of algorithms; the second word (after symbol |) represents the specific algorithm.

Based on the $\langle SKLtype \rangle$ several different algorithms are available. In the following paragraphs a brief explanation and the input requirements are reported for each of them.

13.3.4.1 Linear Models.

The LinearModels’ type of algorithms implement generalized linear models. It includes Ridge regression, Bayesian Regression, Lasso and Elastic Net estimators computed with Least Angle Regression and coordinate descent. It also implements Stochastic Gradient Descent related algorithms. In the following, all the linear models available in RAVEN are reported.

13.3.4.1.1 Linear Model: Automatic Relevance Determination regression

The *Automatic Relevance Determination* regressor is a hierarchical Bayesian approach where there are hyperparameters which explicitly represent the relevance of different input features. These relevance hyperparameters determine the range of variation for the parameters relating to a particular input, usually by modelling the width of a zero-mean Gaussian prior on those parameters. If the width of that Gaussian is zero, then those parameters are constrained to be zero, and the corresponding input cannot have any effect on the predictions, therefore making it irrelevant. ARD optimizes these hyperparameters to discover which inputs are relevant. In order to use the *Automatic Relevance Determination* regressor, the user needs to set the sub-node $\langle SKLtype \rangle ARDRegression \langle /SKLtype \rangle$. In addition to this XML node, several other need (or not) be inputted:

- $\langle n_{iter} \rangle$, **integer, optional field.**.. Maximum number of iterations. *Default* = 300;
- $\langle tol \rangle$, **float, optional field.**.. Stop the algorithm if w has converged. *Default* = 1.e-3;

- $\langle \alpha_1 \rangle$, **float, optional field.** Hyper-parameter : shape parameter for the Gamma distribution prior over the alpha parameter. *Default = 1.e-6*;
- $\langle \alpha_2 \rangle$, **float, optional field.** Hyper-parameter : inverse scale parameter (rate parameter) for the Gamma distribution prior over the alpha parameter. *Default = 1.e-6*;
- $\langle \lambda_1 \rangle$, **float, optional field.** Hyper-parameter : shape parameter for the Gamma distribution prior over the lambda parameter. *Default = 1.e-6*;
- $\langle \lambda_2 \rangle$, **float, optional field.** Hyper-parameter : inverse scale parameter (rate parameter) for the Gamma distribution prior over the lambda parameter. *Default = 1.e-6*;
- $\langle compute_score \rangle$, **boolean, optional field.** If True, compute the objective function at each step of the model. *Default = False*;
- $\langle threshold_lambda \rangle$, **float, optional field.** Threshold for removing (pruning) weights with high precision from the computation. *Default = 1.e+4*;
- $\langle fit_intercept \rangle$, **float, optional field.** whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered). *Default = True*;
- $\langle normalize \rangle$, **boolean, optional field.** If True, the regressors X will be normalized before regression. *Default = False*;
- $\langle verbose \rangle$, **boolean, optional field.** Verbose mode when fitting the model. *Default = False*;

13.3.4.1.2 Linear Model: Bayesian ridge regression

The *Bayesian ridge regression* estimates a probabilistic model of the regression problem as described above. The prior for the parameter w is given by a spherical Gaussian:

$$p(w|\lambda) = \mathcal{N}(w|0, \lambda^{-1}\mathbf{I}_p) \quad (1)$$

The priors over α and λ are chosen to be gamma distributions, the conjugate prior for the precision of the Gaussian. The resulting model is called Bayesian Ridge Regression, and is similar to the classical Ridge. The parameters w , α and λ are estimated jointly during the fit of the model. The remaining hyperparameters are the parameters of the gamma priors over α and λ . These are usually chosen to be non-informative. The parameters are estimated by maximizing the marginal log likelihood. In order to use the *Bayesian ridge regression* regressor, the user needs to set the subnode $\langle SKLtype \rangle$ *BayesianRidge* $\langle /SKLtype \rangle$. In addition to this XML node, several others need (or not) to be inputted:

- $\langle n_iter \rangle$, **integer, optional field.** Maximum number of iterations. *Default = 300*;

- $\langle tol \rangle$, **float, optional field**.. Stop the algorithm if w has converged. *Default* = $1.e-3$;
- $\langle alpha_1 \rangle$, **float, optional field**.. Hyper-parameter : shape parameter for the Gamma distribution prior over the alpha parameter. *Default* = $1.e-6$;
- $\langle alpha_2 \rangle$, **float, optional field**.. Hyper-parameter : inverse scale parameter (rate parameter) for the Gamma distribution prior over the alpha parameter. *Default* = $1.e-6$;
- $\langle lambda_1 \rangle$, **float, optional field**.. Hyper-parameter : shape parameter for the Gamma distribution prior over the lambda parameter. *Default* = $1.e-6$;
- $\langle lambda_2 \rangle$, **float, optional field**.. Hyper-parameter : inverse scale parameter (rate parameter) for the Gamma distribution prior over the lambda parameter. *Default* = $1.e-6$;
- $\langle compute_score \rangle$, **boolean, optional field**.. If True, compute the objective function at each step of the model. *Default* = *False*;
- $\langle threshold_lambda \rangle$, **float, optional field**.. Threshold for removing (pruning) weights with high precision from the computation. *Default* = $1.e+4$;
- $\langle fit_intercept \rangle$, **float, optional field**.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered). *Default* = *True*;
- $\langle normalize \rangle$, **boolean, optional field**.. If True, the regressors X will be normalized before regression. *Default* = *False*;
- $\langle verbose \rangle$, **boolean, optional field**.. Verbose mode when fitting the model. *Default* = *False*;

13.3.4.1.3 Linear Model: Elastic Net

The *Elastic Net* is a linear regression with combined L1 and L2 priors as regularizer. It minimizes the objective function:

$$1/(2 * n_{samples}) * ||y - Xw||_2^2 + alpha * l1_ratio * ||w||_1 + 0.5 * alpha * (1 - l1_ratio) * ||w||_2^2 \quad (2)$$

In order to use the *Elastic Net* regressor, the user needs to set the sub-node $\langle SKLtype \rangle$ *ElasticNet* $\langle /SKLtype \rangle$. In addition to this XML node, several others need (or not) to be inputted:

- $\langle alpha \rangle$, **float, optional field**.. Constant that multiplies the penalty terms. $alpha = 0$ is equivalent to an ordinary least square, solved by the LinearRegression object. *Default* = 1.0 ;
- $\langle l1_ratio \rangle$, **float, optional field**.. The ElasticNet mixing parameter, with $0 \leq l1_ratio \leq 1$. For $l1_ratio = 0$ the penalty is an L2 penalty. For $l1_ratio = 1$ it is an L1 penalty. For $0 < l1_ratio < 1$, the penalty is a combination of L1 and L2. *Default* = 0.5 ;

- $\langle fit_intercept \rangle$, **boolean, optional field.** Whether the intercept should be estimated or not. If False, the data is assumed to be already centered. *Default = True*;
- $\langle normalize \rangle$, **boolean, optional field.** If True, the regressors X will be normalized before regression. *Default = False*;
- $\langle max_iter \rangle$, **integer, optional field.** The maximum number of iterations. *Default = 300*;
- $\langle tol \rangle$, **float, optional field.** The tolerance for the optimization: if the updates are smaller than tol, the optimization code checks the dual gap for optimality and continues until it is smaller than tol.. *Default = 1.0e-4*;
- $\langle warm_start \rangle$, **boolean, optional field.** When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.. *Default = False*;
- $\langle positive \rangle$, **float, optional field.** When set to True, forces the coefficients to be positive. *Default = False*.

13.3.4.1.4 Linear Model: Elastic Net CV

The *Elastic Net CV* is a linear regression similar to Elastic Net model but with an iterative fitting along a regularization path. The best model is selected by cross-validation.

In order to use the *Elastic Net CV* regressor, the user needs to set the sub-node $\langle SKLtype \rangle$ *ElasticNetCV* $\langle /SKLtype \rangle$. In addition to this XML node, several others need (or not) to be inputted:

- $\langle l1_ratio \rangle$, **float, optional field.** Float flag between 0 and 1 passed to ElasticNet (scaling between l1 and l2 penalties). For $l1_ratio = 0$ the penalty is an L2 penalty. For $l1_ratio = 1$ it is an L1 penalty. For $0 < l1_ratio < 1$, the penalty is a combination of L1 and L2 This parameter can be a list, in which case the different values are tested by cross-validation and the one giving the best prediction score is used. Note that a good choice of list of values for $l1_ratio$ is often to put more values close to 1 (i.e. Lasso) and less close to 0 (i.e. Ridge), as in [.1, .5, .7, .9, .95, .99, 1]. *Default = 0.5*;
- $\langle eps \rangle$, **float, optional field.** Length of the path. $eps=1e-3$ means that $alpha_{min}/alpha_{max} = 1e - 3$. *Default = 0.001*;
- $\langle n_alphas \rangle$, **integer, optional field.** Number of alphas along the regularization path, used for each $l1_ratio$. *Default = 100*;
- $\langle precompute \rangle$, **boolean or string, optional field.** Whether to use a precomputed Gram matrix to speed up calculations. If set to 'auto' let us decide. The Gram matrix can also be passed as argument. Available are [True — False — 'auto' — array-like]. *Default = 1.0*;

- *< max_iter > , integer, optional field..* The maximum number of iterations. *Default = 300;*
- *< tol > , float, optional field..* The tolerance for the optimization: if the updates are smaller than tol, the optimization code checks the dual gap for optimality and continues until it is smaller than tol.. *Default = 1.0e-4;*
- *< positive > , float, optional field..* When set to True, forces the coefficients to be positive. *Default = False.*

13.3.4.1.5 Linear Model: Least Angle Regression model

The *Least Angle Regression model* (LARS) is a regression algorithm for high-dimensional data. LARS algorithm provides a means of producing an estimate of which variables to include, as well as their coefficients, when a response variable is determined by a linear combination of a subset of potential covariate.

In order to use the *Least Angle Regression model* regressor, the user needs to set the sub-node *< SKLtype > Lars < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< n_nonzero_coef > , integer, optional field..* Target number of non-zero coefficients. *Default = 500;*
- *< fit_intercept > , boolean, optional field..* Whether the intercept should be estimated or not. If False, the data is assumed to be already centered. *Default = True;*
- *< verbose > , boolean, optional field..* Verbose mode when fitting the model. *Default = False;*
- *< precompute > , boolean or string, optional field..* Whether to use a precomputed Gram matrix to speed up calculations. If set to 'auto' let us decide. The Gram matrix can also be passed as argument. Available are [True — False — 'auto' — array-like]. *Default = 1.0;*
- *< normalize > , boolean, optional field..* If True, the regressors X will be normalized before regression. *Default = False;*
- *< eps > , float, optional field..* The machine-precision regularization in the computation of the Cholesky diagonal factors. Increase this for very ill-conditioned systems. Unlike the tol parameter in some iterative optimization-based algorithms, this parameter does not control the tolerance of the optimization. *Default = 2.22e-16;*
- *< fit_path > , boolean, optional field..* If True the full path is stored in the coef_path_ attribute. If you compute the solution for a large problem or many targets, setting fit_path to False will lead to a speedup, especially with a small alpha. *Default = True.*

13.3.4.1.6 Linear Model: Cross-validated Least Angle Regression model

The *Cross-validated Least Angle Regression model* is a regression algorithm for high-dimensional data. It is similar to LARS method, but the best model is selected by cross-validation.

In order to use the *Cross-validated Least Angle Regression model* regressor, the user needs to set the sub-node `< SKLtype > LarsCV < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< fit_intercept >`, **boolean, optional field.** Whether the intercept should be estimated or not. If False, the data is assumed to be already centered. *Default = True*;
- `< verbose >`, **boolean, optional field.** Verbose mode when fitting the model. *Default = False*;
- `< normalize >`, **boolean, optional field.** If True, the regressors X will be normalized before regression. *Default = False*;
- `< precompute >`, **boolean or string, optional field.** Whether to use a precomputed Gram matrix to speed up calculations. If set to 'auto' let us decide. The Gram matrix can also be passed as argument. Available are [True — False — 'auto' — array-like]. *Default = 1.0*;
- `< max_iter >`, **integer, optional field.** The maximum number of iterations. *Default = 300*;
- `< max_n_alphas >`, **integer, optional field.** The maximum number of points on the path used to compute the residuals in the cross-validation. *Default = 1000*;
- `< eps >`, **float, optional field.** The machine-precision regularization in the computation of the Cholesky diagonal factors. Increase this for very ill-conditioned systems. Unlike the tol parameter in some iterative optimization-based algorithms, this parameter does not control the tolerance of the optimization. *Default = 2.22e-16*;

13.3.4.1.7 Linear Model trained with L1 prior as regularizer (aka the Lasso)

The *Linear Model trained with L1 prior as regularizer (Lasso)* is a shrinkage and selection method for linear regression. It minimizes the usual sum of squared errors, with a bound on the sum of the absolute values of the coefficients.

In order to use the *Linear Model trained with L1 prior as regularizer (Lasso)* regressor, the user needs to set the sub-node `< SKLtype > Lasso < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alpha >`, **float, optional field.** Constant that multiplies the L1 term. Defaults to 1.0. $\alpha = 0$ is equivalent to an ordinary least square, solved by the LinearRegression object. For numerical reasons, using $\alpha = 0$ with the Lasso object is not advised and you should prefer the LinearRegression object.

- *< fit_intercept >* , **boolean**. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- *< normalize >* , **boolean, optional, default False**.. If True, the regressors X will be normalized before regression.
- *< precompute >* , **True — False — ‘auto’ — array-like**.. Whether to use a precomputed Gram matrix to speed up calculations. If set to 'auto' let us decide. The Gram matrix can also be passed as argument. For sparse input this option is always True to preserve sparsity.
- *< max_iter >* , **int, optional field**.. The maximum number of iterations
- *< tol >* , **float, optional field**.. The tolerance for the optimization: if the updates are smaller than tol, the optimization code checks the dual gap for optimality and continues until it is smaller than tol.
- *< warm_start >* , **boolean, optional field**.. When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.
- *< positive >* , **boolean, optional field**.. When set to True, forces the coefficients to be positive.

13.3.4.1.8 Lasso linear model with iterative fitting along a regularization path

The *Lasso linear model with iterative fitting along a regularization path* is Lasso linear model with iterative fitting along a regularization path

In order to use the *Lasso linear model with iterative fitting along a regularization path* regressor, the user needs to set the sub-node *< SKLtype > LassoCV < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< eps >* , **float, optional field**.. Length of the path. eps=1e-3 means that $\alpha_{\min} / \alpha_{\max} = 1e-3$.
- *< n_alphas >* , **int, optional field**.. Number of alphas along the regularization path
- *< alphas >* , **numpy array, optional field**.. List of alphas where to compute the models. If None alphas are set automatically
- *< precompute >* , **true — False — ‘auto’ — array-like**.. Whether to use a precomputed Gram matrix to speed up calculations. If set to 'auto' let us decide. The Gram matrix can also be passed as argument.
- *< max_iter >* , **int, optional field**.. The maximum number of iterations
- *< tol >* , **float, optional field**.. The tolerance for the optimization: if the updates are smaller than tol, the optimization code checks the dual gap for optimality and continues until it is smaller than tol.

- `< verbose >` , **boolean or integer**.. Amount of verbosity.
- `< positive >` , **boolean, optional field**.. If positive, restrict regression coefficients to be positive

13.3.4.1.9 Lasso model fit with Least Angle Regression

The *Lasso model fit with Least Angle Regression* is a Cross-validated Least Angle Regression model

In order to use the *Cross-validated Least Angle Regression model* regressor, the user needs to set the sub-node `< SKLtype > LarsCV < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< fit_intercept >` , **boolean**.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< verbose >` , **boolean or integer, optional field**.. Sets the verbosity amount
- `< normalize >` , **boolean, optional, default False**.. If True, the regressors X will be normalized before regression.
- `< precompute >` , **True — False — 'auto' — array-like**.. Whether to use a precomputed Gram matrix to speed up calculations. If set to 'auto' let us decide. The Gram matrix can also be passed as argument.
- `< max_iter >` , **integer, optional field**.. Maximum number of iterations to perform.
- `< max_n_alphas >` , **integer, optional field**.. The maximum number of points on the path used to compute the residuals in the cross-validation
- `< eps >` , **float, optional field**.. The machine-precision regularization in the computation of the Cholesky diagonal factors. Increase this for very ill-conditioned systems.

13.3.4.1.10 Cross-validated Lasso, using the LARS algorithm p

The *Cross-validated Lasso, using the LARS algorithm* is a Cross-validated Lasso, using the LARS algorithm

In order to use the *Cross-validated Lasso, using the LARS algorithm* regressor, the user needs to set the sub-node `< SKLtype > LassoLarsCV < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< fit_intercept >` , **boolean**.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< verbose >` , **boolean or integer, optional field**.. Sets the verbosity amount

- *< normalize >* , **boolean, optional, default False**.. If True, the regressors X will be normalized before regression.
- *< precompute >* , **True — False — ‘auto’ — array-like**.. Whether to use a precomputed Gram matrix to speed up calculations. If set to ‘auto’ let us decide. The Gram matrix can also be passed as argument.
- *< max_iter >* , **integer, optional field**.. Maximum number of iterations to perform.
- *< max_n_alphas >* , **integer, optional field**.. The maximum number of points on the path used to compute the residuals in the cross-validation
- *< eps >* , **float, optional field**.. The machine-precision regularization in the computation of the Cholesky diagonal factors. Increase this for very ill-conditioned systems.

13.3.4.1.11 Lasso model fit with Lars using BIC or AIC for model selection

The *Lasso model fit with Lars using BIC or AIC for model selection* is Lasso model fit with Lars using BIC or AIC for model selection. The optimization objective for Lasso is: $(1/(2 * n_samples)) * ||y - Xw||_2^2 + alpha * ||w||_1$. AIC is the Akaike information criterion and BIC is the Bayes Information criterion. Such criteria are useful to select the value of the regularization parameter by making a trade-off between the goodness of fit and the complexity of the model. A good model should explain well the data while being simple.

In order to use the *Lasso model fit with Lars using BIC or AIC for model selection* regressor, the user needs to set the sub-node *< SKLtype > LassoLarsIC < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< criterion >* , **‘bic’ — ‘aic’** .. The type of criterion to use.
- *< fit_intercept >* , **boolean**.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- *< verbose >* , **boolean or integer, optional field**.. Sets the verbosity amount
- *< normalize >* , **boolean, optional, default False**.. If True, the regressors X will be normalized before regression.
- *< precompute >* , **True — False — ‘auto’ — array-like**.. Whether to use a precomputed Gram matrix to speed up calculations. If set to ‘auto’ let us decide. The Gram matrix can also be passed as argument.
- *< max_iter >* , **integer, optional field**.. Maximum number of iterations to perform. Can be used for early stopping.

- `< eps >`, **float, optional field**.. The machine-precision regularization in the computation of the Cholesky diagonal factors. Increase this for very ill-conditioned systems. Unlike the `tol` parameter in some iterative optimization-based algorithms, this parameter does not control the tolerance of the optimization.

13.3.4.1.12 Ordinary least squares Linear Regression

The *Ordinary least squares Linear Regression* is a method for estimating the unknown parameters in a linear regression model, with the goal of minimizing the differences between the observed responses in some arbitrary dataset and the responses predicted by the linear approximation of the data

In order to use the *Ordinary least squares Linear Regression* regressor, the user needs to set the sub-node `< SKLtype > LinearRegression < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< fit_intercept >`, **boolean, optional field**.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< normalize >`, **boolean, optional field, default False**.. If True, the regressors X will be normalized before regression.

13.3.4.1.13 Logistic Regression

The *Logistic Regression* implements L1 and L2 regularized logistic regression using the liblinear library. It can handle both dense and sparse input. Use C-ordered arrays or CSR matrices containing 64-bit floats for optimal performance; any other input format will be converted (and copied).

In order to use the *Logistic Regression* regressor, the user needs to set the sub-node `< SKLtype > LogisticRegression < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< penalty >`, **string, 'l1' or 'l2'**.. Used to specify the norm used in the penalization.
- `< dual >`, **boolean**.. Dual or primal formulation. Dual formulation is only implemented for l2 penalty. Prefer `dual=False` when `n_samples < n_features`.
- `< C >`, **float, optional field (default: 1.0)**.. Inverse of regularization strength; must be a positive float. Like in support vector machines, smaller values specify stronger regularization.
- `< fit_intercept >`, **boolean (default: True)**.. Specifies if a constant (a.k.a. bias or intercept) should be added the decision function.

- *< intercept_scaling > , float, optional field (default: 1.0)*.. when self.fit _intercept is True, instance vector x becomes [x, self.intercept _scaling], i.e. a “synthetic” feature with constant value equals to intercept _scaling is appended to the instance vector. The intercept becomes intercept _scaling * synthetic feature weight Note! the synthetic feature weight is subject to l1/l2 regularization as all other features. To lessen the effect of regularization on synthetic feature weight (and therefore on the intercept) intercept _scaling has to be increased
- *< class_weight > , dict, ‘auto’, optional*.. Over-/undersamples the samples of each class according to the given weights. If not given, all classes are supposed to have weight one. The ‘auto’ mode selects weights inversely proportional to class frequencies in the training set.
- *< random_state > , int seed, RandomState instance, or None (default)*.. The seed of the pseudo random number generator to use when shuffling the data.
- *< tol > , float, optional field*.. Tolerance for stopping criteria.

13.3.4.1.14 Multi-task Lasso model trained with L1/L2 mixed-norm as regularizer

The *Multi-task Lasso model trained with L1/L2 mixed-norm as regularizer* is a regressor where the optimization objective for Lasso is: $(1/(2 * n_samples)) * ||Y - XW||_F^2 + alpha * ||W||_21$

Where: $||W||_21 = \sum_i \sqrt{\sum_j w_{ij}^2}$ i.e. the sum of norm of each row.

In order to use the *Multi-task Lasso model trained with L1/L2 mixed-norm as regularizer* regressor, the user needs to set the sub-node *< SKLtype > MultiTaskLasso < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< alpha > , float, optional field*.. Constant that multiplies the L1/L2 term. Defaults to 1.0
- *< fit_intercept > , boolean*.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- *< normalize > , boolean, optional field (default: False)*.. If True, the regressors X will be normalized before regression.
- *< max_iter > , int, optional field*.. The maximum number of iterations
- *< tol > , float, optional field*.. The tolerance for the optimization: if the updates are smaller than tol, the optimization code checks the dual gap for optimality and continues until it is smaller than tol.
- *< warm_start > , boolean, optional field*.. When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.

13.3.4.1.15 Multi-task ElasticNet model trained with L1/L2 mixed-norm as regularizer

The *Multi-task ElasticNet model trained with L1/L2 mixed-norm as regularizer* is a regressor where the optimization objective for MultiTaskElasticNet is: $(1/(2 * n_samples)) * ||Y - XW||_{Fro}^2 + alpha * l1_ratio * ||W||_1 + 0.5 * alpha * (1 - l1_ratio) * ||W||_{Fro}^2$ Where: $||W||_1 = \sum_i \sqrt{\sum_j w_{ij}^2}$ i.e. the sum of norm of each row.

In order to use the *Multi-task ElasticNet model trained with L1/L2 mixed-norm as regularizer* regressor, the user needs to set the sub-node `< SKLtype > MultiTaskElasticNet < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alpha >`, **float, optional field.** Constant that multiplies the L1/L2 term. Defaults to 1.0
- `< l1_ratio >`, **float.** The ElasticNet mixing parameter, with $0 < l1_ratio \leq 1$. For $l1_ratio = 0$ the penalty is an L1/L2 penalty. For $l1_ratio = 1$ it is an L1 penalty. For $0 < l1_ratio < 1$, the penalty is a combination of L1/L2 and L2.
- `< fit_intercept >`, **boolean.** whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< normalize >`, **boolean, optional field (default False).** If True, the regressors X will be normalized before regression.
- `< max_iter >`, **int, optional field.** The maximum number of iterations
- `< tol >`, **float, optional field.** The tolerance for the optimization: if the updates are smaller than tol, the optimization code checks the dual gap for optimality and continues until it is smaller than tol.
- `< warm_start >`, **boolean, optional field.** When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.

13.3.4.1.16 Orthogonal Mathching Pursuit model (OMP)

The *Orthogonal Mathching Pursuit model (OMP)* is a type of sparse approximation which involves finding the "best matching" projections of multidimensional data onto an over-complete dictionary D.

In order to use the *Orthogonal Mathching Pursuit model (OMP)* regressor, the user needs to set the sub-node `< SKLtype > OrthogonalMatchingPursuit < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< n_nonzero_coefs >`, **int, optional field.** Desired number of non-zero entries in the solution. If None (by default) this value is set to 10% of n_features.

- `< tol >` , **float, optional field.**.. Maximum norm of the residual. If not None, overrides `n_nonzero_coefs`.
- `< fit_intercept >` , **boolean, optional field.**.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< normalize >` , **boolean, optional field.**.. If False, the regressors X are assumed to be already normalized.
- `< precompute >` , **True, False, 'auto', default 'auto'.**.. Whether to use a precomputed Gram and Xy matrix to speed up calculations. Improves performance when `n_targets` or `n_samples` is very large. Note that if you already have such matrices, you can pass them directly to the fit method.

13.3.4.1.17 Cross-validated Orthogonal Mathching Pursuit model (OMP)

The *Cross-validated Orthogonal Mathching Pursuit model (OMP)* is a regressor similar to OMP which has good performance in sparse recovery.

In order to use the *Cross-validated Orthogonal Mathching Pursuit model (OMP)* regressor, the user needs to set the sub-node `< SKLtype > OrthogonalMatchingPursuitCV < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< fit_intercept >` , **boolean, optional field.**.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< normalize >` , **boolean, optional field.**.. If False, the regressors X are assumed to be already normalized.
- `< max_iter >` , **integer, optional field.**.. Maximum numbers of iterations to perform, therefore maximum features to include 10% of `n_features` but at least 5 if available.
- `< cv >` , **cross-validation generator, optional.**.. see `sklearn.cross_validation`. If None is passed, default to a 5-fold strategy
- `< verbose >` , **boolean or integer, optional.**.. Sets the verbosity amount

13.3.4.1.18 Passive Aggressive Classifier

The *Passive Aggressive Classifier* is a principled approach to linear classification that advocates minimal weight updates i.e., the least required so that the current training instance is correctly classified.

In order to use the *Passive Aggressive Classifier* classifier, the user needs to set the sub-node `< SKLtype > PassiveAggressiveClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< C >, float..` Maximum step size (regularization). Defaults to 1.0.
- `< fit_intercept >, boolean..` Whether the intercept should be estimated or not. If False, the data is assumed to be already centered. Defaults to True.
- `< n_iter >, int, optional field..` The number of passes over the training data (aka epochs). Defaults to 5.
- `< shuffle >, boolean, optional field..` Whether or not the training data should be shuffled after each epoch. Defaults to False.
- `< random_state >, int seed, RandomState instance, or None (default)..` The seed of the pseudo random number generator to use when shuffling the data.
- `< verbose >, integer or boolean, optional field..` The verbosity level
- `< loss >, string, optional field..` The loss function to be used: hinge: equivalent to PA-I in the reference paper. squared_hinge: equivalent to PA-II in the reference paper.
- `< warm_start >, boolean, optional field..` When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.

13.3.4.1.19 Passive Aggressive Regressor

The *Passive Aggressive Regressor* is similar to the Perceptron in that they do not require a learning rate. However, contrary to the Perceptron, they include a regularization parameter C

In order to use the *Passive Aggressive Regressor* regressor, the user needs to set the sub-node `< SKLtype > PassiveAggressiveRegressor < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< C >, float..` Maximum step size (regularization). Defaults to 1.0.
- `< epsilon >, float..` If the difference between the current prediction and the correct label is below this threshold, the model is not updated.
- `< fit_intercept >, boolean..` Whether the intercept should be estimated or not. If False, the data is assumed to be already centered. Defaults to True.
- `< n_iter >, int, optional field..` The number of passes over the training data (aka epochs). Defaults to 5.
- `< shuffle >, boolean, optional field..` Whether or not the training data should be shuffled after each epoch. Defaults to False.
- `< random_state >, int seed, RandomState instance, or None (default)..` The seed of the pseudo random number generator to use when shuffling the data.

- `< verbose > , int, optional field.` The verbosity level
- `< loss > , string, optional field.` The loss function to be used: `epsilon_insensitive`: equivalent to PA-I in the reference paper. `squared_epsilon_insensitive`: equivalent to PA-II in the reference paper.
- `< warm_start > , boolean, optional field.` When set to `True`, reuse the solution of the previous call to `fit` as initialization, otherwise, just erase the previous solution.

13.3.4.1.20 Perceptron

The *Perceptron* is an algorithm for supervised classification of an input into one of several possible non-binary outputs. It is a type of linear classifier, i.e. a classification algorithm that makes its predictions based on a linear predictor function combining a set of weights with the feature vector. The algorithm allows for online learning, in that it processes elements in the training set one at a time.

In order to use the *Perceptron* classifier, the user needs to set the sub-node `< SKLtype > Perceptron < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< penalty > , None, 'l2' or 'l1' or 'elasticnet'.` The penalty (aka regularization term) to be used. Defaults to `None`.
- `< alpha > , float.` Constant that multiplies the regularization term if regularization is used. Defaults to `0.0001`
- `< fit_intercept > , boolean.` Whether the intercept should be estimated or not. If `False`, the data is assumed to be already centered. Defaults to `True`.
- `< n_iter > , int, optional field.` The number of passes over the training data (aka epochs). Defaults to `5`.
- `< shuffle > , boolean, optional field.` Whether or not the training data should be shuffled after each epoch. Defaults to `False`.
- `< random_state > , int seed, RandomState instance, or None (default).` The seed of the pseudo random number generator to use when shuffling the data.
- `< verbose > , int, optional field.` The verbosity level
- `< eta0 > , double, optional field.` Constant by which the updates are multiplied. Defaults to `1`.
- `< class_weight > , dict, class label.` Preset for the `class_weight` fit parameter. Weights associated with classes. If not given, all classes are supposed to have weight one. The “auto” mode uses the values of `y` to automatically adjust weights inversely proportional to class frequencies.

- *< warm_start > , **boolean, optional field.*** When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.

13.3.4.1.21 Randomized Lasso

The *Randomized Lasso* works by resampling the train data and computing a Lasso on each resampling. In short, the features selected more often are good features. It is also known as stability selection.

In order to use the *Randomized Lasso* regressor, the user needs to set the sub-node *< SKLtype > RandomizedLasso < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< alpha > , **float, ‘aic’, or ‘bic’, optional.*** The regularization parameter alpha parameter in the Lasso. Warning: this is not the alpha parameter in the stability selection article which is scaling.
- *< scaling > , **float, optional field.*** The alpha parameter in the stability selection article used to randomly scale the features. Should be between 0 and 1.
- *< sample_fraction > , **float, optional field.*** The fraction of samples to be used in each randomized design. Should be between 0 and 1. If 1, all samples are used.
- *< n_resampling > , **int, optional field.*** Number of randomized models.
- *< selection_threshold > , **float, optional field.*** The score above which features should be selected.
- *< fit_intercept > , **boolean, optional field.*** whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- *< verbose > , **boolean or integer, optional field.*** Sets the verbosity amount
- *< normalize > , **boolean, optional field (default True).*** If True, the regressors X will be normalized before regression.
- *< precompute > , **True — False — ‘auto’.*** Whether to use a precomputed Gram matrix to speed up calculations. If set to ‘auto’ let us decide. The Gram matrix can also be passed as argument.
- *< max_iter > , **int, optional field.*** Maximum number of iterations to perform in the Lars algorithm.
- *< eps > , **float, optional field.*** The machine-precision regularization in the computation of the Cholesky diagonal factors. Increase this for very ill-conditioned systems. Unlike the ‘tol’ parameter in some iterative optimization-based algorithms, this parameter does not control the tolerance of the optimization.

- `< random_state > , int, RandomState instance or None, optional (default=None)`.. If int, random_state is the seed used by the random number generator; If RandomState instance, random_state is the random number generator; If None, the random number generator is the RandomState instance used by np.random.

13.3.4.1.22 Randomized Logistic Regression

The *Randomized Logistic Regression* works by resampling the train data and computing a LogisticRegression on each resampling. In short, the features selected more often are good features. It is also known as stability selection.

In order to use the *Randomized Logistic Regression* regressor, the user needs to set the sub-node `< SKLtype > RandomizedLogisticRegression < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< C > , float, optional field (default=1)`.. The regularization parameter C in the LogisticRegression.
- `< scaling > , float, optional field (default=0.5)`.. The alpha parameter in the stability selection article used to randomly scale the features. Should be between 0 and 1.
- `< sample_fraction > , float, optional field (default=0.75)`.. The fraction of samples to be used in each randomized design. Should be between 0 and 1. If 1, all samples are used.
- `< n_resampling > , int, optional field (default=200)`.. Number of randomized models.
- `< selection_threshold > , float, optional field (default=0.25)`.. The score above which features should be selected.
- `< fit_intercept > , boolean, optional field`.. whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< verbose > , boolean or integer, optional field (default=True)`. Sets the verbosity amount
- `< normalize > , boolean, optional field (default=True)`.. If True, the regressors X will be normalized before regression.
- `< tol > , float, optional field (default=1e-3)`.. tolerance for stopping criteria of LogisticRegression
- `< random_state > , int, RandomState instance or None, optional (default=None)`.. If int, random_state is the seed used by the random number generator; If RandomState instance, random_state is the random number generator; If None, the random number generator is the RandomState instance used by np.random.

13.3.4.1.23 Linear least squares with l2 regularization

The *Linear least squares with l2 regularization* solves a regression model where the loss function is the linear least squares function and regularization is given by the l2-norm. Also known as Ridge Regression or Tikhonov regularization. This estimator has built-in support for multi-variate regression (i.e., when y is a 2d-array of shape $[n_samples, n_targets]$).

In order to use the *Linear least squares with l2 regularization* regressor, the user needs to set the sub-node `< SKLtype > Ridge < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alpha >`, **float, array-like..** shape = $[n_targets]$ Small positive values of alpha improve the conditioning of the problem and reduce the variance of the estimates. Alpha corresponds to $(2 * C)^{-1}$ in other linear models such as LogisticRegression or LinearSVC. If an array is passed, penalties are assumed to be specific to the targets. Hence they must correspond in number.
- `< fit_intercept >`, **boolean..** Whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< max_iter >`, **int, optional field..** Maximum number of iterations for conjugate gradient solver. The default value is determined by `scipy.sparse.linalg`.
- `< normalize >`, **boolean, optional field (default False)..** If True, the regressors X will be normalized before regression.
- `< solver >`, **'auto', 'svd', 'cholesky', 'lsqr', 'sparse_cg'..** Solver to use in the computational routines:
 - 'auto' chooses the solver automatically based on the type of data.
 - 'svd' uses a Singular Value Decomposition of X to compute the Ridge coefficients. More stable for singular matrices than 'cholesky'.
 - 'cholesky' uses the standard `scipy.linalg.solve` function to obtain a closed-form solution.
 - 'sparse_cg' uses the conjugate gradient solver as found in `scipy.sparse.linalg.cg`. As an iterative algorithm, this solver is more appropriate than 'cholesky' for large-scale data (possibility to set `tol` and `max_iter`).
 - 'lsqr' uses the dedicated regularized least-squares routine `scipy.sparse.linalg.lsqr`. It is the fastest but may not be available in old scipy versions. It also uses an iterative procedure.

All three solvers support both dense and sparse data.

13.3.4.1.24 Classifier using Ridge regression

The *Classifier using Ridge regression* is a classifier based on Linear least squares with l2 regularization

In order to use the *Classifier using Ridge regression* classifier, the user needs to set the sub-node `< SKLtype > RidgeClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alpha > , float..` Small positive values of alpha improve the conditioning of the problem and reduce the variance of the estimates. Alpha corresponds to $(2 * C)^{-1}$ in other linear models such as LogisticRegression or LinearSVC.
- `< class_weight > , dic, optional field..` Weights associated with classes in the form `class.label : weight`. If not given, all classes are supposed to have weight one.
- `< fit_intercept > , boolean..` Whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< max_iter > , int, optional field..` Maximum number of iterations for conjugate gradient solver. The default value is determined by `scipy.sparse.linalg`.
- `< normalize > , boolean, optional field (default False)..` If True, the regressors X will be normalized before regression.
- `< solver > , 'auto', 'svd', 'cholesky', 'lsqr', 'sparse_cg'..` Solver to use in the computational routines. 'svd' will use a Singular value decomposition to obtain the solution, 'cholesky' will use the standard `scipy.linalg.solve` function, 'sparse_cg' will use the conjugate gradient solver as found in `scipy.sparse.linalg.cg` while 'auto' will chose the most appropriate depending on the matrix X. 'lsqr' uses a direct regularized least-squares routine provided by `scipy`.
- `< tol > , float..` Precision of the solution.

13.3.4.1.25 Ridge classifier with built-in cross-validation

The *Ridge classifier with built-in cross-validation* performs Generalized Cross-Validation, which is a form of efficient Leave-One-Out cross-validation. Currently, only the `n_features < n_samples` case is handled efficiently.

In order to use the *Ridge classifier with built-in cross-validation* classifier, the user needs to set the sub-node `< SKLtype > RidgeClassifierCV < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alphas > , numpy array of shape [n_alphas]..` Array of alpha values to try. Small positive values of alpha improve the conditioning of the problem and reduce the variance of the estimates. Alpha corresponds to $(2 * C)^{-1}$ in other linear models such as LogisticRegression or LinearSVC.

- `< fit_intercept >` , **boolean**.. Whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< normalize >` , **boolean, optional field (default False)**.. If True, the regressors X will be normalized before regression.
- `< scoring >` , **string, callable or None, optional, default: None**.. A string (see model evaluation documentation) or a scorer callable object / function with signature `scorer(estimator, X, y)`.
- `< cv >` , **cross-validation generator, optional**.. If None, Generalized Cross-Validation (efficient Leave-One-Out) will be used.
- `< class_weight >` , **dic, optional field**.. Weights associated with classes in the form `class.label : weight`. If not given, all classes are supposed to have weight one.

13.3.4.1.26 Ridge regression with built-in cross-validation

The *Ridge regression with built-in cross-validation* performs Generalized Cross-Validation, which is a form of efficient Leave-One-Out cross-validation.

In order to use the *Ridge regression with built-in cross-validation* regressor, the user needs to set the sub-node `< SKLtype > RidgeCV < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alphas >` , **numpy array of shape [n_alphas]** .. Array of alpha values to try. Small positive values of alpha improve the conditioning of the problem and reduce the variance of the estimates. Alpha corresponds to $(2 * C)^{-1}$ in other linear models such as LogisticRegression or LinearSVC.
- `< fit_intercept >` , **boolean**.. Whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).
- `< normalize >` , **boolean, optional field (default False)**.. If True, the regressors X will be normalized before regression.
- `< scoring >` , **string, callable or None, optional, default: None**.. A string (see model evaluation documentation) or a scorer callable object / function with signature `scorer(estimator, X, y)`.
- `< cv >` , **cross-validation generator, optional field**.. If None, Generalized Cross-Validation (efficient Leave-One-Out) will be used.
- `< gcv_mode >` , **None, 'auto', 'svd', 'eigen', optional field**.. Flag indicating which strategy to use when performing Generalized Cross-Validation. Options are:

- ‘auto’ : use svd if $n_samples \leq n_features$ or when X is a sparse matrix, otherwise use eigen
- ‘svd’ : force computation via singular value decomposition of X (does not work for sparse matrices)
- ‘eigen’ : force computation via eigendecomposition of $X^T X$

The ‘auto’ mode is the default and is intended to pick the cheaper option of the two depending upon the shape and format of the training data.

- `< store_cv_values >` , **boolean (default=False)**.. Flag indicating if the cross-validation values corresponding to each alpha should be stored in the `cv_values` attribute (see below). This flag is only compatible with `cv=None` (i.e. using Generalized Cross-Validation).

13.3.4.1.27 Linear classifiers (SVM, logistic regression, a.o.) with SGD training

The *Linear classifiers (SVM, logistic regression, a.o.) with SGD training* implements regularized linear models with stochastic gradient descent (SGD) learning: the gradient of the loss is estimated each sample at a time and the model is updated along the way with a decreasing strength schedule (aka learning rate). SGD allows minibatch (online/out-of-core) learning, see the `partial_fit` method. This implementation works with data represented as dense or sparse arrays of floating point values for the features. The model it fits can be controlled with the `loss` parameter; by default, it fits a linear support vector machine (SVM). The regularizer is a penalty added to the loss function that shrinks model parameters towards the zero vector using either the squared euclidean norm L2 or the absolute norm L1 or a combination of both (Elastic Net). If the parameter update crosses the 0.0 value because of the regularizer, the update is truncated to 0.0 to allow for learning sparse models and achieve online feature selection.

In order to use the *Linear classifiers (SVM, logistic regression, a.o.) with SGD training* WHAT, the user needs to set the sub-node `< SKLtype > SGDClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< loss >` , **str, ‘hinge’, ‘log’, ‘modified_huber’, ‘squared_hinge’, ‘perceptron’, or a regression loss: ‘squared_loss’, ‘huber’, ‘epsilon_insensitive’, or ‘squared_epsilon_insensitive’**.. The loss function to be used. Defaults to ‘hinge’, which gives a linear SVM. The ‘log’ loss gives logistic regression, a probabilistic classifier. ‘modified_huber’ is another smooth loss that brings tolerance to outliers as well as probability estimates. ‘squared_hinge’ is like hinge but is quadratically penalized. ‘perceptron’ is the linear loss used by the perceptron algorithm. The other losses are designed for regression but can be useful in classification as well; see `SGDRegressor` for a description.
- `< penalty >` , **str, ‘l2’ or ‘l1’ or ‘elasticnet’**.. The penalty (aka regularization term) to be used. Defaults to ‘l2’ which is the standard regularizer for linear SVM models. ‘l1’ and ‘elasticnet’ might bring sparsity to the model (feature selection) not achievable with ‘l2’.

- `< alpha >, float..` Constant that multiplies the regularization term. Defaults to 0.0001
- `< l1_ratio >, float..` The Elastic Net mixing parameter, with $0 \leq l1_ratio \leq 1$. $l1_ratio=0$ corresponds to L2 penalty, $l1_ratio=1$ to L1. Defaults to 0.15.
- `< fit_intercept >, boolean..` Whether the intercept should be estimated or not. If False, the data is assumed to be already centered. Defaults to True.
- `< n_iter >, int, optional field..` The number of passes over the training data (aka epochs). Defaults to 5.
- `< shuffle >, boolean, optional field..` Whether or not the training data should be shuffled after each epoch. Defaults to False.
- `< random_state >, int seed, RandomState instance, or None (default)..` The seed of the pseudo random number generator to use when shuffling the data.
- `< verbose >, int, optional field..` The verbosity level
- `< epsilon >, int, optional field..` The number of CPUs to use to do the OVA (One Versus All, for multi-class problems) computation. -1 means ‘all CPUs’. Defaults to 1.
- `< learning_rate >, string, optional field..` The learning rate: constant: $\eta = \eta_0$ optimal: $\eta = 1.0 / (t + t_0)$ [default] invscaling: $\eta = \eta_0 / \text{pow}(t, \text{power}_t)$
- `< eta0 >, double..` The initial learning rate for the ‘constant’ or ‘invscaling’ schedules. The default value is 0.0 as η_0 is not used by the default schedule ‘optimal’.
- `< power_t >, double..` The exponent for inverse scaling learning rate [default 0.5].
- `< class_weight >, dict, class label..` Preset for the class_weight fit parameter. Weights associated with classes. If not given, all classes are supposed to have weight one. The “auto” mode uses the values of y to automatically adjust weights inversely proportional to class frequencies.
- `< warm_start >, boolean, optional field..` When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.

13.3.4.1.28 Linear model fitted by minimizing a regularized empirical loss with SGD

The *Linear model fitted by minimizing a regularized empirical loss with SGD* is a model where SGD stands for Stochastic Gradient Descent: the gradient of the loss is estimated each sample at a time and the model is updated along the way with a decreasing strength schedule (aka learning rate). The regularizer is a penalty added to the loss function that shrinks model parameters towards the zero vector using either the squared euclidean norm L2 or the absolute norm L1 or a combination of both (Elastic Net). If the parameter update crosses the 0.0 value because of the regularizer,

the update is truncated to 0.0 to allow for learning sparse models and achieve online feature selection. This implementation works with data represented as dense numpy arrays of floating point values for the features.

In order to use the *Linear model fitted by minimizing a regularized empirical loss with SGD* model, the user needs to set the sub-node `< SKLtype > FW_NAME < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< loss >`, **str**, *'squared_loss', 'huber', 'epsilon_insensitive', or 'squared_epsilon_insensitive'*.. The loss function to be used. Defaults to 'squared_loss' which refers to the ordinary least squares fit. 'huber' modifies 'squared_loss' to focus less on getting outliers correct by switching from squared to linear loss past a distance of epsilon. 'epsilon_insensitive' ignores errors less than epsilon and is linear past that; this is the loss function used in SVR. 'squared_epsilon_insensitive' is the same but becomes squared loss past a tolerance of epsilon.
- `< penalty >`, **str**, *'l2' or 'l1' or 'elasticnet'*.. The penalty (aka regularization term) to be used. Defaults to 'l2' which is the standard regularizer for linear SVM models. 'l1' and 'elasticnet' might bring sparsity to the model (feature selection) not achievable with 'l2'.
- `< alpha >`, **float**.. Constant that multiplies the regularization term. Defaults to 0.0001
- `< l1_ratio >`, **float**.. The Elastic Net mixing parameter, with $0 \leq \text{l1_ratio} \leq 1$. $\text{l1_ratio}=0$ corresponds to L2 penalty, $\text{l1_ratio}=1$ to L1. Defaults to 0.15.
- `< fit_intercept >`, **boolean**.. Whether the intercept should be estimated or not. If False, the data is assumed to be already centered. Defaults to True.
- `< n_iter >`, **int, optional field**.. The number of passes over the training data (aka epochs). Defaults to 5.
- `< shuffle >`, **boolean, optional field**.. Whether or not the training data should be shuffled after each epoch. Defaults to False.
- `< random_state >`, **int seed, RandomState instance, or None (default)**.. The seed of the pseudo random number generator to use when shuffling the data.
- `< verbose >`, **int, optional field**.. The verbosity level.
- `< epsilon >`, **float**.. Epsilon in the epsilon-insensitive loss functions; only if loss is 'huber', 'epsilon_insensitive', or 'squared_epsilon_insensitive'. For 'huber', determines the threshold at which it becomes less important to get the prediction exactly right. For epsilon-insensitive, any differences between the current prediction and the correct label are ignored if they are less than this threshold.
- `< learning_rate >`, **string, optional field**.. Learning rate: constant: $\eta = \eta_0$ optimal: $\eta = 1.0/(t+t_0)$ invscaling: $\eta = \eta_0 / \text{pow}(t, \text{power_t})$ [default]

- `< eta0 >` , **double**.. The initial learning rate [default 0.01].
- `< power_t >` , **double, optional field**.. The exponent for inverse scaling learning rate [default 0.25].
- `< warm_start >` , **boolean, optional field**.. When set to True, reuse the solution of the previous call to fit as initialization, otherwise, just erase the previous solution.

13.3.4.1.29 Compute Least Angle Regression or Lasso path using LARS algorithm

The *Compute Least Angle Regression or Lasso path using LARS algorithm* is a regressor where the optimization objective for the case `method='lasso'` is: $(1/(2 * n_samples)) * ||y - Xw||_2^2 + \alpha * ||w||_1$ in the case of `method='lars'`, the objective function is only known in the form of an implicit equation

In order to use the *Compute Least Angle Regression or Lasso path using LARS algorithm* regressor, the user needs to set the sub-node `< SKLtype > lars_path < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< X >` , **array, shape: (n_samples, n_features)**.. Input data.
- `< y >` , **array, shape: (n_samples)**.. Input targets.
- `< max_iter >` , **integer, optional (default=500)**.. Maximum number of iterations to perform, set to infinity for no limit.
- `< gram >` , **None, 'auto', array, shape: (n_features, n_features), optional**.. Precomputed Gram matrix ($X' * X$), if 'auto', the Gram matrix is precomputed from the given X, if there are more samples than features.
- `< alpha_min >` , **float, optional field (default=0)**.. Minimum correlation along the path. It corresponds to the regularization parameter alpha parameter in the Lasso.
- `< method >` , **'lar', 'lasso', optional (default='lar')**.. Specifies the returned model. Select 'lar' for Least Angle Regression, 'lasso' for the Lasso.
- `< eps >` , **float, optional (default="np.finfo(np.float).eps")**.. The machine-precision regularization in the computation of the Cholesky diagonal factors. Increase this for very ill-conditioned systems.
- `< verbose >` , **int (default=0)**.. Controls output verbosity.

13.3.4.1.30 Compute Lasso path with coordinate descent

The *Compute Lasso path with coordinate descent* is a regressor where the Lasso optimization function varies for mono and multi-outputs. For mono-output tasks it is: $(1/(2 * n_samples)) * ||y - Xw||_2^2 + alpha * ||w||_1$ For multi-output tasks it is: $(1/(2 * n_samples)) * ||Y - XW||_{Fro}^2 + alpha * ||W||_21$ Where: $||W||_21 = \sum_i \sqrt{\sum_j w_{ij}^2}$ i.e. the sum of norm of each row.

In order to use the *Compute Lasso path with coordinate descent* regressor, the user needs to set the sub-node `< SKLtype > lasso_path < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< X >` , **array-like, sparse matrix, shape (n_samples, n_features)**.. Training data. Pass directly as Fortran-contiguous data to avoid unnecessary memory duplication. If y is mono-output then X can be sparse.
- `< y >` , **ndarray, shape = (n_samples,)**, or **(n_samples, n_outputs)**.. Target values
- `< eps >` , **float, optional field**.. Length of the path. eps=1e-3 means that alpha_min / alpha_max = 1e-3
- `< n_alphas >` , **int, optional field**.. Number of alphas along the regularization path
- `< alphas >` , **ndarray, optional field**.. List of alphas where to compute the models. If None alphas are set automatically
- `< precompute >` , **True — False — ‘auto’ — array-like**.. Whether to use a precomputed Gram matrix to speed up calculations. If set to 'auto' let us decide. The Gram matrix can also be passed as argument.
- `< Xy >` , **array-like, optional field**.. $Xy = np.dot(X.T, y)$ that can be precomputed. It is useful only when the Gram matrix is precomputed.
- `< copy_X >` , **boolean, optional field (default True)**.. If True, X will be copied; else, it may be overwritten.
- `< coef_init >` , **array, shape (n_features,) — None**.. The initial values of the coefficients.
- `< verbose >` , **bool or integer**.. Amount of verbosity.

13.3.4.1.31 Stability path based on randomized Lasso estimates

In order to use the *Stability path based on randomized Lasso estimates* regressor, the user needs to set the sub-node `< SKLtype > lasso_stability_path < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< X >` , **array-like, shape = [n_samples, n_features]**.. training data.

- *< y > , array-like, shape = [n_samples]*.. target values.
- *< scaling > , float, optional, default=0.5*.. The alpha parameter in the stability selection article used to randomly scale the features. Should be between 0 and 1.
- *< random_state > , integer or numpy.random.RandomState, optional*.. The generator used to randomize the design.
- *< n_resampling > , int, optional, default=200*.. Number of randomized models.
- *< n_grid > , int, optional, default=100*.. Number of grid points. The path is linearly reinterpolated on a grid between 0 and 1 before computing the scores.
- *< sample_fraction > , float, optional, default=0.75*.. The fraction of samples to be used in each randomized design. Should be between 0 and 1. If 1, all samples are used.
- *< eps > , float, optional field*.. Smallest value of alpha / alpha_max considered
- *< verbose > , boolean or integer, optional field*.. Sets the verbosity amount

13.3.4.1.32 Gram Orthogonal Matching Pursuit (OMP)

The *NAME* Solves *n_targets* Orthogonal Matching Pursuit problems using only the Gram matrix $X.T * X$ and the product $X.T * y$.

In order to use the *NAME* regressor, the user needs to set the sub-node *< SKLtype > orthogonal_mp_gram < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< Gram > , array, shape (n_features, n_features)*.. Gram matrix of the input data: $X.T * X$
- *< Xy > , array, shape (n_features,) or (n_features, n_targets)* .. Input targets multiplied by $X: X.T * y$
- *< n_nonzero_coefs > , int*.. Desired number of non-zero entries in the solution. If None (by default) this value is set to 10% of *n_features*.
- *< tol > , float*.. Maximum norm of the residual. If not None, overrides *n_nonzero_coefs*.
- *< norms_squared > , array-like, shape (n_targets)*.. DESCRIPTION

13.3.4.2 Support Vector Machines.

In machine learning, **Support Vector Machines** (SVMs, also support vector networks) are supervised learning models with associated learning algorithms that analyze data and recognize patterns, used for classification and regression analysis. Given a set of training examples, each marked as

belonging to one of two categories, an SVM training algorithm builds a model that assigns new examples into one category or the other, making it a non-probabilistic binary linear classifier. An SVM model is a representation of the examples as points in space, mapped so that the examples of the separate categories are divided by a clear gap that is as wide as possible. New examples are then mapped into that same space and predicted to belong to a category based on which side of the gap they fall on. In addition to performing linear classification, SVMs can efficiently perform a non-linear classification using what is called the kernel trick, implicitly mapping their inputs into high-dimensional feature spaces. In the following, all the SVM models available in RAVEN are reported.

13.3.4.2.1 Linear Support Vector Classifier

The *Linear Support Vector Classifier* is similar to SVC with parameter `kernel='linear'`, but implemented in terms of `liblinear` rather than `libsvm`, so it has more flexibility in the choice of penalties and loss functions and should scale better (to large numbers of samples). This class supports both dense and sparse input and the multiclass support is handled according to a one-vs-the-rest scheme. In order to use the *Linear Support Vector Classifier* classifier, the user needs to set the sub-node `< SKLtype > LinearSVC < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< C >`, **boolean, optional field (default=1.0)**.. Penalty parameter C of the error term.
- `< loss >`, **string, 'l1' or 'l2' (default='l2')**.. Specifies the loss function. 'l1' is the hinge loss (standard SVM) while 'l2' is the squared hinge loss.
- `< penalty >`, **string, 'l1' or 'l2' (default='l2')**.. Specifies the norm used in the penalization. The 'l2' penalty is the standard used in SVC. The 'l1' leads to `coef_vectors` that are sparse.
- `< dual >`, **boolean (default=True)**.. Select the algorithm to either solve the dual or primal optimization problem. Prefer `dual=False` when `n_samples` \gg `n_features`.
- `< tol >`, **float, optional field**.. Tolerance for stopping criteria
- `< multi_class >`, **string, 'ovr' or 'crammer_singer' (default='ovr')**.. Determines the multi-class strategy if `y` contains more than two classes. `ovr` trains `n_classes` one-vs-rest classifiers, while `crammer_singer` optimizes a joint objective over all classes. While `crammer_singer` is interesting from an theoretical perspective as it is consistent it is seldom used in practice and rarely leads to better accuracy and is more expensive to compute. If `crammer_singer` is chosen, the options `loss`, `penalty` and `dual` will be ignored.
- `< fit_intercept >`, **boolean, optional field (default=True)**.. Whether to calculate the intercept for this model. If set to false, no intercept will be used in calculations (e.g. data is expected to be already centered).

- *< intercept_scaling > , float, optional field (default=1)..* when `self.fit.intercept` is `True`, instance vector `x` becomes `[x, self.intercept_scaling]`, i.e. a “synthetic” feature with constant value equals to `intercept_scaling` is appended to the instance vector. The intercept becomes `intercept_scaling * synthetic feature weight` Note! the synthetic feature weight is subject to l1/l2 regularization as all other features. To lessen the effect of regularization on synthetic feature weight (and therefore on the intercept) `intercept_scaling` has to be increased
- *< class_weight > , dict, ‘auto’, optional field..* Set the parameter `C` of class `i` to `class_weight[i]*C` for SVC. If not given, all classes are supposed to have weight one. The ‘auto’ mode uses the values of `y` to automatically adjust weights inversely proportional to class frequencies.
- *< verbose > , int, optional field (default=0)..* Enable verbose output. Note that this setting takes advantage of a per-process runtime setting in `liblinear` that, if enabled, may not work properly in a multithreaded context.
- *< random_state > , int seed, RandomState instance, or None (default)..* The seed of the pseudo random number generator to use when shuffling the data.

13.3.4.2.2 C-Support Vector Classification

The *C-Support Vector Classification* is based on `libsvm`. The fit time complexity is more than quadratic with the number of samples which makes it hard to scale to dataset with more than a couple of 10000 samples. The multiclass support is handled according to a one-vs-one scheme. In order to use the *C-Support Vector Classification* classifier, the user needs to set the sub-node *< SKLtype > SVC < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< C > , float, optional field (default=1.0)..* Penalty parameter `C` of the error term.
- *< kernel > , string, optional (default='rbf')..* Specifies the kernel type to be used in the algorithm. It must be one of ‘linear’, ‘poly’, ‘rbf’, ‘sigmoid’, ‘precomputed’ or a callable. If none is given, ‘rbf’ will be used. If a callable is given it is used to precompute the kernel matrix.
- *< degree > , int, optional field (default=3)..* Degree of the polynomial kernel function (‘poly’). Ignored by all other kernels.
- *< gamma > , float, optional field (default=0.0)..* Kernel coefficient for ‘rbf’, ‘poly’ and ‘sigmoid’. If `gamma` is 0.0 then `1/n_features` will be used instead.
- *< coef0 > , float, optional field (default=0.0)..* Independent term in kernel function. It is only significant in ‘poly’ and ‘sigmoid’.
- *< probability > , boolean, optional field (default=False)..* Whether to enable probability estimates. This must be enabled prior to calling `fit`, and will slow down that method.

- *< shrinking >* , **boolean, optional field (default=True)**.. Whether to use the shrinking heuristic.
- *< tol >* , **float, optional field (default=1e-3)**.. Tolerance for stopping criterion.
- *< cache_size >* , **float, optional field**.. Specify the size of the kernel cache (in MB)
- *< class_weight >* , **dict, 'auto', optional**.. Set the parameter C of class i to class_weight[i]*C for SVC. If not given, all classes are supposed to have weight one. The 'auto' mode uses the values of y to automatically adjust weights inversely proportional to class frequencies.
- *< verbose >* , **boolean (default: False)**.. Enable verbose output. Note that this setting takes advantage of a per-process runtime setting in libsvm that, if enabled, may not work properly in a multithreaded context.
- *< max_iter >* , **int, optional field (default=-1)**.. Hard limit on iterations within solver, or -1 for no limit.
- *< random_state >* , **int seed, RandomState instance, or None (default)**.. The seed of the pseudo random number generator to use when shuffling the data for probability estimation.

13.3.4.2.3 Nu-Support Vector Classification

The *Nu-Support Vector Classification* is similar to SVC but uses a parameter to control the number of support vectors. The implementation is based on libsvm.

In order to use the *Nu-Support Vector Classification* classifier, the user needs to set the sub-node *< SKLtype > NuSVC < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< nu >* , **float, optional field (default=0.5)**.. An upper bound on the fraction of training errors and a lower bound of the fraction of support vectors. Should be in the interval (0, 1].
- *< kernel >* , **string, optional field (default='rbf')**.. Specifies the kernel type to be used in the algorithm. It must be one of 'linear', 'poly', 'rbf', 'sigmoid', 'precomputed' or a callable. If none is given, 'rbf' will be used. If a callable is given it is used to precompute the kernel matrix.
- *< degree >* , **int, optional field (default=3)**.. degree of kernel function is significant only in poly, rbf, sigmoid
- *< gamma >* , **float, optional field (default=0.0)**.. kernel coefficient for rbf and poly, if gamma is 0.0 then 1/n_features will be taken.
- *< coef0 >* , **float, optional field (default=0.0)**.. independent term in kernel function. It is only significant in poly/sigmoid.

- *< probability >* , **boolean, optional field (default=False)**.. Whether to enable probability estimates. This must be enabled prior to calling fit, and will slow down that method.
- *< shrinking >* , **boolean, optional field (default=True)**.. Whether to use the shrinking heuristic.
- *< tol >* , **float, optional field (default=1e-3)**.. Tolerance for stopping criterion.
- *< cache_size >* , **float, optional field**.. Specify the size of the kernel cache (in MB)
- *< verbose >* , **boolean, optional field (default=False)**.. Enable verbose output. Note that this setting takes advantage of a per-process runtime setting in libsvm that, if enabled, may not work properly in a multithreaded context.
- *< max_iter >* , **int, optional field (default=-1)**.. Hard limit on iterations within solver, or -1 for no limit.
- *< random_state >* , **int seed, RandomState instance, or None (default)**.. The seed of the pseudo random number generator to use when shuffling the data for probability estimation.

13.3.4.2.4 Support Vector Regression

The *Support Vector Regression* is an epsilon-Support Vector Regression. The free parameters in the model are C and epsilon. The implementations is a based on libsvm.

In order to use the *Support Vector Regression* regressor, the user needs to set the sub-node *< SKLtype > SVR < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< C >* , **float, optional field (default=1.0)**.. penalty parameter C of the error term.
- *< epsilon >* , **float, optional field (default=0.1)**.. epsilon in the epsilon-SVR model. It specifies the epsilon-tube within which no penalty is associated in the training loss function with points predicted within a distance epsilon from the actual value.
- *< kernel >* , **string, optional field (default='rbf')**.. Specifies the kernel type to be used in the algorithm. It must be one of 'linear', 'poly', 'rbf', 'sigmoid', 'precomputed' or a callable. If none is given, 'rbf' will be used. If a callable is given it is used to precompute the kernel matrix.
- *< degree >* , **float, optional field (default=3.0)**.. degree of kernel function is significant only in poly, rbf, sigmoid
- *< gamma >* , **float, optional field (default=0.0)**.. kernel coefficient for rbf and poly, if gamma is 0.0 then 1/n.features will be taken.

- `< coef0 >` , **float, optional field (default=0.0)**.. independent term in kernel function. It is only significant in poly/sigmoid.
- `< probability >` , **boolean, optional field (default=False)**.. Whether to enable probability estimates. This must be enabled prior to calling fit, and will slow down that method.
- `< shrinking >` , **boolean, optional field (default=True)**.. Whether to use the shrinking heuristic.
- `< tol >` , **float, optional field (default=1e-3)**.. Tolerance for stopping criterion.
- `< cache_size >` , **float, optional field**.. Specify the size of the kernel cache (in MB)
- `< verbose >` , **boolean (default: False)**.. Enable verbose output. Note that this setting takes advantage of a per-process runtime setting in libsvm that, if enabled, may not work properly in a multithreaded context.
- `< max_iter >` , **int, optional field (default=-1)**.. Hard limit on iterations within solver, or -1 for no limit.
- `< random_state >` , **int seed, RandomState instance, or None (default)**.. The seed of the pseudo random number generator to use when shuffling the data for probability estimation.

13.3.4.3 Multi Class.

Multiclass classification means a classification task with more than two classes; e.g., classify a set of images of fruits which may be oranges, apples, or pears. Multiclass classification makes the assumption that each sample is assigned to one and only one label: a fruit can be either an apple or a pear but not both at the same time. In the following, all the multi-class models available in RAVEN are reported.

13.3.4.3.1 One-vs-the-rest (OvR) multiclass/multilabel strategy

The *One-vs-the-rest (OvR) multiclass/multilabel strategy*, also known as one-vs-all, consists in fitting one classifier per class. For each classifier, the class is fitted against all the other classes. In addition to its computational efficiency (only `n_classes` classifiers are needed), one advantage of this approach is its interpretability. Since each class is represented by one and one classifier only, it is possible to gain knowledge about the class by inspecting its corresponding classifier. This is the most commonly used strategy and is a fair default choice.

In order to use the *One-vs-the-rest (OvR) multiclass/multilabel* classifier, the user needs to set the sub-node `< SKLtype > multiClass | OneVsRestClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< estimator >` , **boolean, required field**.. An estimator object implementing fit and one of decision_function or predict_proba. This XML node needs to contain the attributes:

- **estimatorType**, *required string attribute*, this attribute an other reduced order mode type that needs to be used for the construction of the multi-class algorithms; each sub-sequential node depends on the chosen ROM;

13.3.4.3.2 One-vs-one multiclass strategy

The *One-vs-one multiclass strategy* consists in fitting one classifier per class pair. At prediction time, the class which received the most votes is selected. Since it requires to fit $n_classes * (n_classes - 1) / 2$ classifiers, this method is usually slower than one-vs-the-rest, due to its $O(n_classes^2)$ complexity. However, this method may be advantageous for algorithms such as kernel algorithms which do not scale well with $n_samples$. This is because each individual learning problem only involves a small subset of the data whereas, with one-vs-the-rest, the complete dataset is used $n_classes$ times.

In order to use the *One-vs-one multiclass* classifier, the user needs to set the sub-node `< SKLtype > multiClass | OneVsOneClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< estimator >`, **boolean, required field..** An estimator object implementing fit and one of decision_function or predict_proba. This XML node needs to contain the attributes:
 - **estimatorType**, *required string attribute*, this attribute an other reduced order mode type that needs to be used for the construction of the multi-class algorithms; each sub-sequential node depends on the chosen ROM;

13.3.4.3.3 Error-Correcting Output-Code multiclass strategy

The *Error-Correcting Output-Code multiclass strategy* consists in representing each class with a binary code (an array of 0s and 1s). At fitting time, one binary classifier per bit in the code book is fitted. At prediction time, the classifiers are used to project new points in the class space and the class closest to the points is chosen. The main advantage of these strategies is that the number of classifiers used can be controlled by the user, either for compressing the model ($0 < code_size < 1$) or for making the model more robust to errors ($code_size > 1$).

In order to use the *Error-Correcting Output-Code multiclass* classifier, the user needs to set the sub-node `< SKLtype > multiClass | OutputCodeClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< estimator >`, **xml node, required field..** An estimator object implementing fit and one of decision_function or predict_proba. This XML node needs to contain the attributes:
 - **estimatorType**, *required string attribute*, this attribute an other reduced order mode type that needs to be used for the construction of the multi-class algorithms; each sub-sequential node depends on the chosen ROM;
- `< code_size >`, **float, required field..** ercentage of the number of classes to be used to create the code book. A number between 0 and 1 will require fewer classifiers than one-vs-the-rest. A number greater than 1 will require more classifiers than one-vs-the-rest.

13.3.4.4 Naive Bayes.

Naive Bayes methods are a set of supervised learning algorithms based on applying Bayes' theorem with the “naive” assumption of independence between every pair of features. Given a class variable y and a dependent feature vector x_1 through x_n , Bayes' theorem states the following relationship:

$$P(y | x_1, \dots, x_n) = \frac{P(y)P(x_1, \dots, x_n | y)}{P(x_1, \dots, x_n)} \quad (3)$$

Using the naive independence assumption that

$$P(x_i | y, x_1, \dots, x_{i-1}, x_{i+1}, \dots, x_n) = P(x_i | y), \quad (4)$$

for all i , this relationship is simplified to

$$P(y | x_1, \dots, x_n) = \frac{P(y) \prod_{i=1}^n P(x_i | y)}{P(x_1, \dots, x_n)} \quad (5)$$

Since $P(x_1, \dots, x_n)$ is constant given the input, we can use the following classification rule:

$$P(y | x_1, \dots, x_n) \propto P(y) \prod_{i=1}^n P(x_i | y) \Downarrow \quad (6)$$

$$\hat{y} = \arg \max_y P(y) \prod_{i=1}^n P(x_i | y), \quad (7)$$

and we can use Maximum A Posteriori (MAP) estimation to estimate $P(y)$ and $P(x_i | y)$; the former is then the relative frequency of class y in the training set. The different naive Bayes classifiers differ mainly by the assumptions they make regarding the distribution of $P(x_i | y)$. In spite of their apparently over-simplified assumptions, naive Bayes classifiers have worked quite well in many real-world situations, famously document classification and spam filtering. They require a small amount of training data to estimate the necessary parameters. (For theoretical reasons why naive Bayes works well, and on which types of data it does, see the references below.) Naive Bayes learners and classifiers can be extremely fast compared to more sophisticated methods. The decoupling of the class conditional feature distributions means that each distribution can be independently estimated as a one dimensional distribution. This in turn helps to alleviate problems stemming from the curse of dimensionality. On the flip side, although naive Bayes is known as a decent classifier, it is known to be a bad estimator, so the probability outputs from `predict_proba` are not to be taken too seriously. In the following, all the Naive Bayes available in RAVEN are reported.

13.3.4.4.1 Gaussian Naive Bayes

The *Gaussian Naive Bayes strategy* implements the Gaussian Naive Bayes algorithm for classification. The likelihood of the features is assumed to be Gaussian:

$$P(x_i | y) = \frac{1}{\sqrt{2\pi\sigma_y^2}} \exp\left(-\frac{(x_i - \mu_y)^2}{2\sigma_y^2}\right) \quad (8)$$

The parameters σ_y and μ_y are estimated using maximum likelihood.

In order to use the *Gaussian Naive Bayes strategy*, the user needs to set the sub-node `< SKLtype > naiveBayes | GaussianNB < /SKLtype >`. No additional XML nodes are needed to be inputted.

13.3.4.4.2 Multinomial Naive Bayes

The *Multinomial Naive Bayes* implements the naive Bayes algorithm for multinomially distributed data, and is one of the two classic naive Bayes variants used in text classification (where the data are typically represented as word vector counts, although tf-idf vectors are also known to work well in practice). The distribution is parametrized by vectors $\theta_y = (\theta_{y1}, \dots, \theta_{yn})$ for each class y , where n is the number of features (in text classification, the size of the vocabulary) and θ_{yi} is the probability $P(x_i | y)$ of feature i appearing in a sample belonging to class y . The parameters θ_y is estimated by a smoothed version of maximum likelihood, i.e. relative frequency counting:

$$\hat{\theta}_{yi} = \frac{N_{yi} + \alpha}{N_y + \alpha n} \quad (9)$$

where $N_{yi} = \sum_{x \in T} x_i$ is the number of times feature i appears in a sample of class y in the training set T , and $N_y = \sum_{i=1}^{|T|} N_{yi}$ is the total count of all features for class y . The smoothing priors $\alpha \geq 0$ accounts for features not present in the learning samples and prevents zero probabilities in further computations. Setting $\alpha = 1$ is called Laplace smoothing, while $\alpha < 1$ is called Lidstone smoothing.

In order to use the *Multinomial Naive Bayes strategy*, the user needs to set the sub-node `< SKLtype > naiveBayes | MultinomialNB < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alpha >, float, optional field..` Additive (Laplace/Lidstone) smoothing parameter (0 for no smoothing). *Default = 1.0*;
- `< fit_prior >, boolean, required field..` Whether to learn class prior probabilities or not. If false, a uniform prior will be used. *Default = False*;
- `< class_prior >, array-like float (n_classes), optional field..` Prior probabilities of the classes. If specified the priors are not adjusted according to the data. *Default = None*.

13.3.4.4.3 Bernoulli Naive Bayes

The *Bernoulli Naive Bayes* implements the naive Bayes training and classification algorithms for data that is distributed according to multivariate Bernoulli distributions; i.e., there may be multiple features but each one is assumed to be a binary-valued (Bernoulli, boolean) variable. Therefore, this class requires samples to be represented as binary-valued feature vectors; if handed any other

kind of data, a *Bernoulli Naive Bayes* instance may binarize its input (depending on the binarize parameter). The decision rule for Bernoulli naive Bayes is based on

$$P(x_i | y) = P(i | y)x_i + (1 - P(i | y))(1 - x_i) \quad (10)$$

which differs from multinomial NB's rule in that it explicitly penalizes the non-occurrence of a feature i that is an indicator for class y , where the multinomial variant would simply ignore a non-occurring feature. In the case of text classification, word occurrence vectors (rather than word count vectors) may be used to train and use this classifier. *Bernoulli Naive Bayes* might perform better on some datasets, especially those with shorter documents. It is advisable to evaluate both models, if time permits.

In order to use the *Bernoulli Naive Bayes* strategy, the user needs to set the sub-node `< SKLtype > naiveBayes | BernoulliNB < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< alpha >`, **float, optional field..** Additive (Laplace/Lidstone) smoothing parameter (0 for no smoothing). *Default = 1.0*;
- `< binarize >`, **float, optional field..** Threshold for binarizing (mapping to booleans) of sample features. If None, input is presumed to already consist of binary vectors.. *Default = None*;
- `< fit_prior >`, **boolean, optional field..** Whether to learn class prior probabilities or not. If false, a uniform prior will be used. *Default = False*;
- `< class_prior >`, **array-like float (n.classes), optional field..** Prior probabilities of the classes. If specified the priors are not adjusted according to the data. *Default = None*.

13.3.4.5 Neighbors.

The *Neighbors* class provides functionality for unsupervised and supervised neighbors-based learning methods. Unsupervised nearest neighbors is the foundation of many other learning methods, notably manifold learning and spectral clustering. Supervised neighbors-based learning comes in two flavors: classification for data with discrete labels, and regression for data with continuous labels. The principle behind nearest neighbor methods is to find a predefined number of training samples closest in distance to the new point, and predict the label from these. The number of samples can be a user-defined constant (k-nearest neighbor learning), or vary based on the local density of points (radius-based neighbor learning). The distance can, in general, be any metric measure: standard Euclidean distance is the most common choice. Neighbors-based methods are known as non-generalizing machine learning methods, since they simply “remember” all of its training data (possibly transformed into a fast indexing structure such as a Ball Tree or KD Tree.).

In the following, all the Neighbors' models available in RAVEN are reported.

13.3.4.5.1 Nearest Neighbors

The *Nearest Neighbors* implements unsupervised nearest neighbors learning. It acts as a uniform interface to three different nearest neighbors algorithms: BallTree, KDTree, and a brute-force algorithm.

In order to use the *Nearest Neighbors* strategy, the user needs to set the sub-node `< SKLtype > neighbors | NearestNeighbors < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< n_neighbors >`, **integer, optional field.**.. Number of neighbors to use by default for k_neighbors queries.. *Default = 5*;
- `< radius >`, **float, optional field.**.. Range of parameter space to use by default for :meth`radius_neighbors` queries. *Default = 1.0*;
- `< algorithm >`, **string, optional field.**.. Algorithm used to compute the nearest neighbors:
 - *ball_tree* will use BallTree;
 - *kd_tree* will use KDtree;
 - *brute* will use a brute-force search;
 - *auto* will attempt to decide the most appropriate algorithm based on the values passed to fit method.

NB. fitting on sparse input will override the setting of this parameter, using brute force. *Default = auto*;

- `< leaf_size >`, **integer, optional field.**.. Leaf size passed to BallTree or KDTree. This can affect the speed of the construction and query, as well as the memory required to store the tree. The optimal value depends on the nature of the problem. *Default = 30*;
- `< p >`, **integer, optional field.**.. Parameter for the Minkowski metric. When $p = 1$, this is equivalent to using manhattan distance (L1), and euclidean distance (L2) for $p = 2$. For arbitrary p, minkowski distance (L_p) is used. *Default = 2*.

13.3.4.5.2 K Neighbors Classifier

The *K Neighbors Classifier* is a type of instance-based learning or non-generalizing learning: it does not attempt to construct a general internal model, but simply stores instances of the training data. Classification is computed from a simple majority vote of the nearest neighbors of each point: a query point is assigned the data class which has the most representatives within the nearest neighbors of the point. It implements learning based on the k nearest neighbors of each query point, where k is an integer value specified by the user.

In order to use the *K Neighbors Classifier*, the user needs to set the sub-node `< SKLtype > neighbors | KNeighborsClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- *< n_neighbors > , integer, optional field..* Number of neighbors to use by default for k_neighbors queries.. *Default = 5;*
- *< weights > , string, optional field..* Weight function used in prediction. Possible values:
 - *uniform* : uniform weights. All points in each neighborhood are weighted equally;
 - *distance* : weight points by the inverse of their distance. in this case, closer neighbors of a query point will have a greater influence than neighbors which are further away.

Default = uniform;

- *< radius > , float, optional field..* Range of parameter space to use by default for :meth‘radius_neighbors‘ queries. *Default = 1.0;*
- *< algorithm > , string, optional field..* Algorithm used to compute the nearest neighbors:
 - *ball_tree* will use BallTree;
 - *kd_tree* will use KDtree;
 - *brute* will use a brute-force search;
 - *auto* will attempt to decide the most appropriate algorithm based on the values passed to fit method.

NB. fitting on sparse input will override the setting of this parameter, using brute force.
Default = auto;

- *< metric > , string, optional field..* the distance metric to use for the tree. The default metric is minkowski, and with $p=2$ is equivalent to the standard Euclidean metric. *Default = minkowski;*
- *< leaf_size > , integer, optional field..* Leaf size passed to BallTree or KDTree. This can affect the speed of the construction and query, as well as the memory required to store the tree. The optimal value depends on the nature of the problem. *Default = 30;*
- *< p > , integer, optional field..* Parameter for the Minkowski metric. When $p = 1$, this is equivalent to using manhattan distance (L1), and euclidean distance (L2) for $p = 2$. For arbitrary p , minkowski distance (L_p) is used. *Default = 2.*

13.3.4.5.3 Radius Neighbors Classifier

The *Radius Neighbors Classifier* is a type of instance-based learning or non-generalizing learning: it does not attempt to construct a general internal model, but simply stores instances of the training data. Classification is computed from a simple majority vote of the nearest neighbors of each point: a query point is assigned the data class which has the most representatives within the nearest neighbors of the point. It implements learning based on the number of neighbors within a fixed radius r of each training point, where r is a floating-point value specified by the user.

In order to use the *Radius Neighbors Classifier*, the user needs to set the sub-node `< SKLtype > neighbors | RadiusNeighbors < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< n_neighbors >` , **integer, optional field.**.. Number of neighbors to use by default for `k_neighbors` queries.. *Default = 5*;
- `< weights >` , **string, optional field.**.. Weight function used in prediction. Possible values:
 - *uniform* : uniform weights. All points in each neighborhood are weighted equally;
 - *distance* : weight points by the inverse of their distance. in this case, closer neighbors of a query point will have a greater influence than neighbors which are further away.

Default = uniform;

- `< radius >` , **float, optional field.**.. Range of parameter space to use by default for `:meth'radius_neighbors'` queries. *Default = 1.0*;
- `< algorithm >` , **string, optional field.**.. Algorithm used to compute the nearest neighbors:
 - *ball_tree* will use BallTree;
 - *kd_tree* will use KDtree;
 - *brute* will use a brute-force search;
 - *auto* will attempt to decide the most appropriate algorithm based on the values passed to fit method.

NB. fitting on sparse input will override the setting of this parameter, using brute force.
Default = auto;

- `< metric >` , **string, optional field.**.. the distance metric to use for the tree. The default metric is minkowski, and with $p=2$ is equivalent to the standard Euclidean metric. *Default = minkowski*;
- `< leaf_size >` , **integer, optional field.**.. Leaf size passed to BallTree or KDTree. This can affect the speed of the construction and query, as well as the memory required to store the tree. The optimal value depends on the nature of the problem. *Default = 30*;
- `< p >` , **integer, optional field.**.. Parameter for the Minkowski metric. When $p = 1$, this is equivalent to using manhattan distance (L1), and euclidean distance (L2) for $p = 2$. For arbitrary p , minkowski distance (L_p) is used. *Default = 2*;
- `< outlier_label >` , **integer, optional field.**.. Label, which is given for outlier samples (samples with no neighbors on given radius). If set to None, ValueError is raised, when outlier is detected. *Default = None*.

13.3.4.5.4 K Neighbors Regressor

The *K Neighbors Regressor* can be used in cases where the data labels are continuous rather than discrete variables. The label assigned to a query point is computed based the mean of the labels of its nearest neighbors. It implements learning based on the k nearest neighbors of each query point, where k is an integer value specified by the user.

In order to use the *K Neighbors Regressor*, the user needs to set the sub-node `< SKLtype > neighbors | KNeighborsRegressor < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< n_neighbors >` , **integer, optional field.**.. Number of neighbors to use by default for k_neighbors queries.. *Default = 5*;
- `< weights >` , **string, optional field.**.. Weight function used in prediction. Possible values:
 - *uniform* : uniform weights. All points in each neighborhood are weighted equally;
 - *distance* : weight points by the inverse of their distance. in this case, closer neighbors of a query point will have a greater influence than neighbors which are further away.

Default = uniform;

- `< radius >` , **float, optional field.**.. Range of parameter space to use by default for :meth‘radius_neighbors’ queries. *Default = 1.0*;
- `< algorithm >` , **string, optional field.**.. Algorithm used to compute the nearest neighbors:
 - *ball_tree* will use BallTree;
 - *kd_tree* will use KDtree;
 - *brute* will use a brute-force search;
 - *auto* will attempt to decide the most appropriate algorithm based on the values passed to fit method.

NB. fitting on sparse input will override the setting of this parameter, using brute force.
Default = auto;

- `< metric >` , **string, optional field.**.. the distance metric to use for the tree. The default metric is minkowski, and with p=2 is equivalent to the standard Euclidean metric. *Default = minkowski*;
- `< leaf_size >` , **integer, optional field.**.. Leaf size passed to BallTree or KDTree. This can affect the speed of the construction and query, as well as the memory required to store the tree. The optimal value depends on the nature of the problem. *Default = 30*;
- `< p >` , **integer, optional field.**.. Parameter for the Minkowski metric. When $p = 1$, this is equivalent to using manhattan distance (L1), and euclidean distance (L2) for $p = 2$. For arbitrary p, minkowski distance (L_p) is used. *Default = 2*.

13.3.4.5.5 Radius Neighbors Regressor

The *Radius Neighbors Regressor* can be used in cases where the data labels are continuous rather than discrete variables. The label assigned to a query point is computed based the mean of the labels of its nearest neighbors. It implements learning based on the neighbors within a fixed radius r of the query point, where r is a floating-point value specified by the user.

In order to use the *Radius Neighbors Regressor*, the user needs to set the sub-node `< SKLtype > neighbors | RadiusNeighborsRegressor < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< n_neighbors >` , **integer, optional field.**.. Number of neighbors to use by default for `k_neighbors` queries.. *Default = 5*;
- `< weights >` , **string, optional field.**.. Weight function used in prediction. Possible values:
 - *uniform* : uniform weights. All points in each neighborhood are weighted equally;
 - *distance* : weight points by the inverse of their distance. in this case, closer neighbors of a query point will have a greater influence than neighbors which are further away.

Default = uniform;

- `< radius >` , **float, optional field.**.. Range of parameter space to use by default for `:meth'radius_neighbors'` queries. *Default = 1.0*;
- `< algorithm >` , **string, optional field.**.. Algorithm used to compute the nearest neighbors:
 - *ball_tree* will use `BallTree`;
 - *kd_tree* will use `KDtree`;
 - *brute* will use a brute-force search;
 - *auto* will attempt to decide the most appropriate algorithm based on the values passed to fit method.

NB. fitting on sparse input will override the setting of this parameter, using brute force.
Default = auto;

- `< metric >` , **string, optional field.**.. the distance metric to use for the tree. The default metric is *minkowski*, and with $p=2$ is equivalent to the standard Euclidean metric. *Default = minkowski*;
- `< leaf_size >` , **integer, optional field.**.. Leaf size passed to `BallTree` or `KDTree`. This can affect the speed of the construction and query, as well as the memory required to store the tree. The optimal value depends on the nature of the problem. *Default = 30*;
- `< p >` , **integer, optional field.**.. Parameter for the Minkowski metric. When $p = 1$, this is equivalent to using manhattan distance (L1), and euclidean distance (L2) for $p = 2$. For arbitrary p , minkowski distance (L_p) is used. *Default = 2*;

- `< outlier_label >` , **integer, optional field.**.. Label, which is given for outlier samples (samples with no neighbors on given radius). If set to None, ValueError is raised, when outlier is detected. *Default = None.*

13.3.4.5.6 Nearest Centroid Classifier

The *Nearest Centroid classifier* is a simple algorithm that represents each class by the centroid of its members. It also has no parameters to choose, making it a good baseline classifier. It does, however, suffer on non-convex classes, as well as when classes have drastically different variances, as equal variance in all dimensions is assumed.

In order to use the *Nearest Centroid Classifier*, the user needs to set the sub-node `< SKLtype > neighbors | NearestCentroid < /SKLtype >`. In addition to this XML node, another might be inputted:

- `< n_neighbors >` , **float, optional field.**.. Threshold for shrinking centroids to remove features. *Default = None.*

13.3.4.6 Quadratic Discriminant Analysis.

The *Quadratic Discriminant Analysis* is a classifier with a quadratic decision boundary, generated by fitting class conditional densities to the data and using Bayes' rule. The model fits a Gaussian density to each class.

In order to use the *Quadratic Discriminant Analysis Classifier*, the user needs to set the sub-node `< SKLtype > qda | QDA < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< priors >` , **array-like (n_classes), optional field.**.. Priors on classes. *Default = None;*
- `< reg_param >` , **float, optional field.**.. Regularizes the covariance estimate as $(1 - \text{reg_param}) * \text{Sigma} + \text{reg_param} * \text{Identity}(n_features)$. *Default = 0.0.*

13.3.4.7 Tree.

Decision Trees (DTs) are a non-parametric supervised learning method used for classification and regression. The goal is to create a model that predicts the value of a target variable by learning simple decision rules inferred from the data features.

- Some advantages of decision trees are:
- Simple to understand and to interpret. Trees can be visualized.

- Requires little data preparation. Other techniques often require data normalization, dummy variables need to be created and blank values to be removed. Note however that this module does not support missing values.
- The cost of using the tree (i.e., predicting data) is logarithmic in the number of data points used to train the tree.
- Able to handle both numerical and categorical data. Other techniques are usually specialized in analyzing datasets that have only one type of variable.
- Able to handle multi-output problems.
- Uses a white box model. If a given situation is observable in a model, the explanation for the condition is easily explained by boolean logic. By contrast, in a black box model (e.g., in an artificial neural network), results may be more difficult to interpret.
- Possible to validate a model using statistical tests. That makes it possible to account for the reliability of the model.
- Performs well even if its assumptions are somewhat violated by the true model from which the data were generated.

The disadvantages of decision trees include:

- Decision-tree learners can create over-complex trees that do not generalise the data well. This is called overfitting. Mechanisms such as pruning (not currently supported), setting the minimum number of samples required at a leaf node or setting the maximum depth of the tree are necessary to avoid this problem.
- Decision trees can be unstable because small variations in the data might result in a completely different tree being generated. This problem is mitigated by using decision trees within an ensemble.
- The problem of learning an optimal decision tree is known to be NP-complete under several aspects of optimality and even for simple concepts. Consequently, practical decision-tree learning algorithms are based on heuristic algorithms such as the greedy algorithm where locally optimal decisions are made at each node. Such algorithms cannot guarantee to return the globally optimal decision tree. This can be mitigated by training multiple trees in an ensemble learner, where the features and samples are randomly sampled with replacement.
- There are concepts that are hard to learn because decision trees do not express them easily, such as XOR, parity or multiplexer problems.
- Decision tree learners create biased trees if some classes dominate. It is therefore recommended to balance the dataset prior to fitting with the decision tree.

In the following, all the linear models available in RAVEN are reported.

13.3.4.7.1 Decision Tree Classifier

The *Decision Tree Classifier* is a classifier that is based on the decision tree logic.

In order to use the *Decision Tree Classifier*, the user needs to set the sub-node `< SKLtype > tree | DecisionTreeClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< criterion >` , **string, optional field.** The function to measure the quality of a split. Supported criteria are “gini” for the Gini impurity and “entropy” for the information gain. *Default = gini*;
- `< splitter >` , **string, optional field.** The strategy used to choose the split at each node. Supported strategies are “best” to choose the best split and “random” to choose the best random split. *Default = best*;
- `< max_features >` , **int, float or string, optional field.** The number of features to consider when looking for the best split:
 - If int, then consider max_features features at each split.
 - If float, then max_features is a percentage and $\text{int}(\text{max_features} * \text{n_features})$ features are considered at each split.
 - If “auto”, then $\text{max_features} = \text{sqrt}(\text{n_features})$;
 - If “sqrt”, then $\text{max_features} = \text{sqrt}(\text{n_features})$;
 - If “log2”, then $\text{max_features} = \log_2(\text{n_features})$;
 - If None, then $\text{max_features} = \text{n_features}$.

NB. The search for a split does not stop until at least one valid partition of the node samples is found, even if it requires to effectively inspect more than max_features features.

Default = None;

- `< max_depth >` , **integer, optional field.** The maximum depth of the tree. If None, then nodes are expanded until all leaves are pure or until all leaves contain less than min_samples_split samples. Ignored if max_samples_leaf is not None. *Default = None*;
- `< min_samples_split >` , **integer, optional field.** The minimum number of samples required to split an internal node. *Default = 2*;
- `< min_samples_leaf >` , **integer, optional field.** The minimum number of samples required to be at a leaf node. *Default = 1*;
- `< max_leaf_nodes >` , **integer, optional field.** Grow a tree with max_leaf_nodes in best-first fashion. Best nodes are defined as relative reduction in impurity. If None then unlimited number of leaf nodes. If not None then max_depth will be ignored. *Default = None*.

13.3.4.7.2 Decision Tree Regressor

The *Decision Tree Regressor* is a Regressor that is based on the decision tree logic.

In order to use the *Decision Tree Regressor*, the user needs to set the sub-node `< SKLtype > tree | DecisionTreeRegressor < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< criterion >` , **string, optional field.** The function to measure the quality of a split. Supported criteria are “gini” for the Gini impurity and “entropy” for the information gain. *Default = gini*;
- `< splitter >` , **string, optional field.** The strategy used to choose the split at each node. Supported strategies are “best” to choose the best split and “random” to choose the best random split. *Default = best*;
- `< max_features >` , **int, float or string, optional field.** The number of features to consider when looking for the best split:
 - If int, then consider max_features features at each split.
 - If float, then max_features is a percentage and $\text{int}(\text{max_features} * \text{n_features})$ features are considered at each split.
 - If “auto”, then $\text{max_features} = \text{sqrt}(\text{n_features})$;
 - If “sqrt”, then $\text{max_features} = \text{sqrt}(\text{n_features})$;
 - If “log2”, then $\text{max_features} = \log_2(\text{n_features})$;
 - If None, then $\text{max_features} = \text{n_features}$.

NB. The search for a split does not stop until at least one valid partition of the node samples is found, even if it requires to effectively inspect more than max_features features.

Default = None;

- `< max_depth >` , **integer, optional field.** The maximum depth of the tree. If None, then nodes are expanded until all leaves are pure or until all leaves contain less than min_samples_split samples. Ignored if max_samples_leaf is not None. *Default = None*;
- `< min_samples_split >` , **integer, optional field.** The minimum number of samples required to split an internal node. *Default = 2*;
- `< min_samples_leaf >` , **integer, optional field.** The minimum number of samples required to be at a leaf node. *Default = 1*;
- `< max_leaf_nodes >` , **integer, optional field.** Grow a tree with max_leaf_nodes in best-first fashion. Best nodes are defined as relative reduction in impurity. If None then unlimited number of leaf nodes. If not None then max_depth will be ignored. *Default = None*.

13.3.4.7.3 Extra Tree Classifier

The *Extra Tree Classifier* is an extremely randomized tree classifier. Extra-trees differ from classic decision trees in the way they are built. When looking for the best split to separate the samples of a node into two groups, random splits are drawn for each of the `max_features` randomly selected features and the best split among those is chosen. When `max_features` is set 1, this amounts to building a totally random decision tree.

In order to use the *Extra Tree Classifier*, the user needs to set the sub-node `< SKLtype > tree | ExtraTreeClassifier < /SKLtype >`. In addition to this XML node, several others need (or not) to be inputted:

- `< criterion >` , **string, optional field.** The function to measure the quality of a split. Supported criteria are “gini” for the Gini impurity and “entropy” for the information gain. *Default = gini*;
- `< splitter >` , **string, optional field.** The strategy used to choose the split at each node. Supported strategies are “best” to choose the best split and “random” to choose the best random split. *Default = best*;
- `< max_features >` , **int, float or string, optional field.** The number of features to consider when looking for the best split:
 - If int, then consider `max_features` features at each split.
 - If float, then `max_features` is a percentage and `int(max_features * n_features)` features are considered at each split.
 - If “auto”, then `max_features=sqrt(n_features)`;
 - If “sqrt”, then `max_features=sqrt(n_features)`;
 - If “log2”, then `max_features=log2(n_features)`;
 - If None, then `max_features=n_features`.

NB. The search for a split does not stop until at least one valid partition of the node samples is found, even if it requires to effectively inspect more than `max_features` features.

Default = None;

- `< max_depth >` , **integer, optional field.** The maximum depth of the tree. If None, then nodes are expanded until all leaves are pure or until all leaves contain less than `min_samples_split` samples. Ignored if `max_samples_leaf` is not None. *Default = None*;
- `< min_samples_split >` , **integer, optional field.** The minimum number of samples required to split an internal node. *Default = 2*;
- `< min_samples_leaf >` , **integer, optional field.** The minimum number of samples required to be at a leaf node. *Default = 1*;

- *< max_leaf_nodes > , integer, optional field..* Grow a tree with max_leaf_nodes in best-first fashion. Best nodes are defined as relative reduction in impurity. If None then unlimited number of leaf nodes. If not None then max_depth will be ignored. *Default = None.*

13.3.4.7.4 Extra Tree Regressor

The *Extra Tree Regressor* is an extremely randomized tree regressor. Extra-trees differ from classic decision trees in the way they are built. When looking for the best split to separate the samples of a node into two groups, random splits are drawn for each of the max_features randomly selected features and the best split among those is chosen. When max_features is set 1, this amounts to building a totally random decision tree.

In order to use the *Extra Tree Regressor*, the user needs to set the sub-node *< SKLtype > tree | ExtraTreeRegressor < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- *< criterion > , string, optional field..* The function to measure the quality of a split. Supported criteria are “gini” for the Gini impurity and “entropy” for the information gain. *Default = gini;*
- *< splitter > , string, optional field..* The strategy used to choose the split at each node. Supported strategies are “best” to choose the best split and “random” to choose the best random split. *Default = best;*
- *< max_features > , int, float or string, optional field..* The number of features to consider when looking for the best split:
 - If int, then consider max_features features at each split.
 - If float, then max_features is a percentage and int(max_features * n_features) features are considered at each split.
 - If “auto”, then max_features=sqrt(n_features);
 - If “sqrt”, then max_features=sqrt(n_features);
 - If “log2”, then max_features=log2(n_features);
 - If None, then max_features=n_features.

NB. The search for a split does not stop until at least one valid partition of the node samples is found, even if it requires to effectively inspect more than max_features features.

Default = None;

- *< max_depth > , integer, optional field..* The maximum depth of the tree. If None, then nodes are expanded until all leaves are pure or until all leaves contain less than min_samples_split samples. Ignored if max_samples_leaf is not None. *Default = None;*

- *< min_samples_split > , integer, optional field..* The minimum number of samples required to split an internal node. *Default = 2;*
- *< min_samples_leaf > , integer, optional field..* The minimum number of samples required to be at a leaf node. *Default = 1;*
- *< max_leaf_nodes > , integer, optional field..* Grow a tree with max_leaf_nodes in best-first fashion. Best nodes are defined as relative reduction in impurity. If None then unlimited number of leaf nodes. If not None then max_depth will be ignored. *Default = None.*

13.3.4.8 Gaussian Process.

Gaussian Processes for Machine Learning (GPML) is a generic supervised learning method primarily designed to solve regression problems. The advantages of Gaussian Processes for Machine Learning are:

- The prediction interpolates the observations (at least for regular correlation models);
- The prediction is probabilistic (Gaussian) so that one can compute empirical confidence intervals and exceedance probabilities that might be used to refit (online fitting, adaptive fitting) the prediction in some region of interest;
- Versatile: different linear regression models and correlation models can be specified. Common models are provided, but it is also possible to specify custom models provided they are stationary.

The disadvantages of Gaussian Processes for Machine Learning include:

- It is not sparse. It uses the whole samples/features information to perform the prediction;
- It loses efficiency in high dimensional spaces – namely when the number of features exceeds a few dozens. It might indeed give poor performance and it loses computational efficiency;
- Classification is only a post-processing, meaning that one first need to solve a regression problem by providing the complete scalar float precision output y of the experiment one attempt to model.

In order to use the *Gaussian Process Regressor*, the user needs to set the sub-node *< SKLtype > GaussianProcess | GaussianProcess < /SKLtype >*. In addition to this XML node, several others need (or not) to be inputted:

- **< regr > , *string, optional field.*** A regression function returning an array of outputs of the linear regression functional basis. The number of observations `n_samples` should be greater than the size `p` of this basis. Available built-in regression models are: 'constant', 'linear', 'quadratic'. *Default = constant*;
- **< corr > , *string, optional field.*** A stationary autocorrelation function returning the autocorrelation between two points `x` and `x'`. Default assumes a squared-exponential autocorrelation model. Built-in correlation models are: 'absolute_exponential', 'squared_exponential', 'generalized_exp', 'cubic', 'linear'. *Default = squared_exponential*;
- **< beta0 > , *float, array-like, optional field.*** The regression weight vector to perform Ordinary Kriging (OK). *Default = Universal Kriging*;
- **< storage_mode > , *string, optional field.*** A string specifying whether the Cholesky decomposition of the correlation matrix should be stored in the class (`storage_mode = 'full'`) or not (`storage_mode = 'light'`). *Default = full*;
- **< verbose > , *boolean, optional field.*** boolean specifying the verbose level. *Default = False*;
- **< theta0 > , *float, array-like, optional field.*** An array with shape `(n_features,)` or `(1,)`. The parameters in the autocorrelation model. If `thetaL` and `thetaU` are also specified, `theta0` is considered as the starting point for the maximum likelihood estimation of the best set of parameters. *Default = [1e-1]*;
- **< thetaL > , *float, array-like, optional field.*** An array with shape matching `theta0`'s. Lower bound on the autocorrelation parameters for maximum likelihood estimation. *Default = None*;
- **< thetaU > , *float, array-like, optional field.*** An array with shape matching `theta0`'s. Upper bound on the autocorrelation parameters for maximum likelihood estimation. *Default = None*;
- **< normalize > , *boolean, optional field.*** Input `X` and observations `y` are centered and reduced wrt means and standard deviations estimated from the `n_samples` observations provided. *Default = True*;
- **< nugget > , *float, optional field.*** Introduce a nugget effect to allow smooth predictions from noisy data. The nugget is added to the diagonal of the assumed training covariance; in this way it acts as a Tikhonov regularization in the problem. In the special case of the squared exponential correlation function, the nugget mathematically represents the variance of the input values. *Default = 10. * MACHINE_EPSILON*;
- **< optimizer > , *string, optional field.*** A string specifying the optimization algorithm to be used. Available optimizers are: 'fmin_cobyla', 'Welch'. *Default = fmin_cobyla*;

- `< random_start >`, *integer, optional field.*. The number of times the Maximum Likelihood Estimation should be performed from a random starting point. The first MLE always uses the specified starting point (theta0), the next starting points are picked at random according to an exponential distribution (log-uniform on [thetaL, thetaU]). *Default = 1*;
- `< random_state >`, *integer, optional field.*. Seed of the internal random number generator. *Default = random.*

Example:

```
-----
<Simulation>
...
<Models>
...
  <ROM name='***' subType='SciKitLearn'>
    <Features>***,***</Features>
    <SKLtype>linear_model|LinearRegression</SKLtype>
    <Target>***</Target>
    <fit_intercept>***</fit_intercept>
    <normalize>***</normalize>
  </ROM>
...
</Models>
...
</Simulation>
-----
```

13.4 External Model

As the name suggests, an external model is an entity that is embedded in the RAVEN code at run time. This object allows the user to create a python module that is going to be treated as a predefined internal model object. In other words, the **External Model** is going to be treated by RAVEN as a normal external Code (e.g. it is going to be called in order to compute a whatever quantity, based on a whatever input).

The specifications of an External Model must be defined within the XML block `< ExternalModel >`. This XML node needs to contain the attributes:

- **name**, *required string attribute*, user-defined name of this External Model. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (XML);
- **subType**, *required string attribute*, This string must be kept empty;

- **ModuleToLoad**, *required string attribute*, file name with its absolute or relative path. NB. If a relative path is specified, it must be noticed it is relative with respect to where the user runs the code.

In order to make the RAVEN code aware of the variables the user is going to manipulate/use in its own python Module, the variables need to be specified in the `< ExternalModel >` input block. The user needs to input, within this block, only the variables that RAVEN needs to be aware of (i.e. the variables are going to directly be used by the code) and not the local variables that the user does not want to, for example, store in a RAVEN internal object. These variables are inputted within consecutive XML blocks called `< variable >`:

- `< variable >`, string, required parameter. In the body of this XML node, the user needs to specify the name of the variable. This variable needs to match a variable used/defined in the external python model.

When the external function variables are defined, at run time, RAVEN initialize those and take track of their values during the simulation. Each variable defined in the `< ExternalModel >` block is available in the module (each method implemented) as a python “self”.

In the External Python module, the user can implement all the methods that are needed for the functionality of the model, but only the following methods are going, if present, called by the framework:

- ***def _readMoreXML***, *OPTIONAL METHOD*, This method can be implemented by the user if the XML input that belongs to this External Model needs to be extended to contain other information. The information read needs to be stored in “self” in order to be available to all the other methods (e.g. if the user needs to add a couple of newer XML nodes with information needed by the algorithm implemented in the “run” method);
- ***def initialize***, *OPTIONAL METHOD*, Initialization method. In this function the user can implement all the actions need to be performed at the initialization stage;
- ***def createNewInput***, *OPTIONAL METHOD*, Method to create a new input with the information coming from the RAVEN framework. In this function the user can retrieve the information coming from the RAVEN framework, during the employment of a calculation flow, and use them to construct a new input that is going to be transferred to the “run” method;
- ***def run***, *REQUIRED METHOD*, This is the actual location where the user needs to implement the model action (e.g. resolution of a set of equations, etc.). This function is going to receive the Input(or Inputs) generated either by the External Model “createNewInput” method or the internal RAVEN one;

In the following sub-sections, all the methods are going to be analyzed in detail.

13.4.1 Method: def _readMoreXML.

As already mentioned, the **readMoreXML** method can be implemented by the user if the XML input that belongs to this External Model needs to be extended to contain other information. The information read needs to be stored in “self” in order to be available to all the other methods (e.g. if the user needs to add a couple of newer XML nodes with information needed by the algorithm implemented in the “run” method). If this method is implemented in the **External Model**, RAVEN is going to call it when the node `< ExternalModel >` is found parsing XML input file. The method receives from RAVEN an attribute of type “xml.etree.ElementTree”, containing all the sub-nodes and attribute of the XML block `< ExternalModel >`.

In the following an example is reported:

```
-----
<Simulation>
...
  <Models>
    ...
    <ExternalModel name='AnExtModule'
                        subType=' '
                        ModuleToLoad='path_to_external_module'>
      <variable>sigma</variable>
      <variable>rho</variable>
      <variable>outcome</variable>
      <!--
        here we define other nodes RAVEN does not read
        automatically
        => We need to implement, in the external module
            ``AnExtModule``
            the readMoreXML method
      -->
      <newNodeWeNeedToRead>
        whatNeedsToBeRead
      </newNodeWeNeedToRead>
    </ExternalModel>
    ...
  </Models>
  ...
</Simulation>
```

```
-----
def _readMoreXML(self, xmlNode) :
  # the xmlNode is passed in by RAVEN framework
  # <newNodeWeNeedToRead> is unknown (in the RAVEN framework)
```



```

# we have to read it on our own
# get the node
ourNode = xmlNode.find('`newNodeWeNeedToRead`')
# get the information in the node
self.ourNewVariable = ourNode.text
# end function
-----

```

13.4.2 Method: `def initialize`.

The **`initialize`** method can be implemented in the **External Model** in order to initialize some variables needed by it. For example, it can be used to compute a quantity needed by the “run” method before performing the actual calculation). If this method is implemented in the **External Model**, RAVEN is going to call it at the initialization stage of each “Step” (see section 7. RAVEN will communicate, thorough a set of method attributes, all the information that are generally needed to perform a initialization:

- `runInfo`, a dictionary containing information regarding how the calculation is set up (e.g. number of processors, etc.). It contains the following attributes:
 - *DefaultInputFile*, Default input file to use
 - *SimulationFiles*, the xml input file
 - *ScriptDir*, the location of the pbs script interfaces
 - *FrameworkDir*, the directory where the framework is located
 - *WorkingDir*, the directory where the framework should be running
 - *TempWorkingDir*, the temporary directory where a simulation step is run
 - *NumMPI*, the number of mpi process by run
 - *NumThreads*, Number of Threads by run
 - *numProcByRun*, Total number of core used by one run (number of threads by number of mpi)
 - *batchSize*, number of contemporaneous runs
 - *ParallelCommand*, the command that should be used to submit jobs in parallel (mpi)
 - *numNode*, number of nodes
 - *procByNode*, number of processors by node
 - *totalNumCoresUsed*, total number of cores used by driver
 - *quequingSoftware*, quequing software name

- *stepName*, the name of the step currently running
 - *precommand*, Add to the front of the command that is run
 - *postcommand*, Added after the command that is run
 - *delSucLogFiles*, If a simulation (code run) has not failed, delete the relative log file (if True)
 - *deleteOutExtension*, If a simulation (code run) has not failed, delete the relative output files with the listed extension (comma separated list, for example: 'e,r,txt')
 - *mode*, Running mode. Curently the only modes supported are pbsdsh and mpi
 - *expectedTime*, How long the complete input is expected to run
 - *logfileBuffer*, logfile buffer size in bytes
- *inputs*, a list of all the inputs that have been specified in the “Step” is using this model.

In the following an example is reported:

```
-----
def initialize(self,runInfo,inputs):
    # Let's suppose we just need to initialize some variables
    self.sigma = 10.0
    self.rho    = 28.0
    # end function
-----
```

13.4.3 Method: **def createNewInput.**

The **createNewInput** method can be implemented by the user to create a new input with the information coming from the RAVEN framework. In this function the user can retrieve the information coming from the RAVEN framework, during the employment of a calculation flow, and use them to construct a new input that is going to be transferred to the “run” method. The new input created needs to be returned to RAVEN (i.e. “return NewInput”). RAVEN communicates, thorough a set of method attributes, all the information that are generally needed to create a new input: *myInput,samplerType,**Kwargs*

- *inputs*, python list, a list of all the inputs that have been defined in the “Step” is using this model;
- *samplerType*, string, the type of Sampler, if a Sampling strategy is employed; None otherwise;

- Kwargs, dictionary, a dictionary containing several informations (that can change based on the “Step” type). If a Sampling strategy is employed, this dictionary contains another one identified by the keyword “SampledVars”, in which the variables perturbed by the sampler are reported;

NB. If the “Step” that is using this Model has as input(s) an object of main class type “Datas” (see section ??), the internal “createNewInput” method is going to convert it in a dictionary of values. In the following an example is reported:

```
-----
def createNewInput(self,inputs,samplerType,**Kwargs):
    # in here the actual createNewInput of the
    # model is implemented
    if samplerType == ``MonteCarlo``:
        variable = inputs[``something``]*inputs[``something2``]
    else:
        variable = inputs[``something``]/inputs[``something2``]
    return variable*Kwargs[``SampledVars``][``aSampledVar``]
-----
```

13.4.4 Method: def run.

As stated previously, the only method the MUST be present in an External Module is the **run** function. In this function , the user needs to implement the algorithm that RAVEN has to execute. The **run** method is generally called after having inquired the “createNewInput” method (either the internal or the user-implemented one). The only attribute this method is going to receive by is a Python list of inputs (the inputs coming from the “createNewInput” method). If the user wants RAVEN to collect the results of this method, the outcomes of interest need to be stored in “self”. NB. RAVEN is trying to collect the values of the variables listed in the XML block *< ExternalModel >* only.

In the following an example is reported:

```
-----
def run(self,Input):
    # in here the actual run of the
    # model is implemented
    input = Input[0]
    self.outcome = self.sigma*self.rho*input[``whatEver``]
-----
```

13.5 PostProcessor

A Post-Processor (PP) can be considered as an Action performed on a set of data or other type of objects. Most of the post-processors contained in RAVEN, employ a mathematical operation on the data given as “input”. RAVEN supports several different types of PPs. Currently in RAVEN the following Post-Processors are available:

- **BasicStatistics;**
- **ComparisonStatistics;**
- **SafestPoint;**
- **LimitSurface;**
- **PrintCSV;**
- **LoadCsvIntoInternalObject.**

The specifications of this Model must be defined within the XML block `< PostProcessor >`. This XML node needs to contain the attributes:

- **name**, *required string attribute*, user-defined name of this Model. N.B. As for the other objects, this is the name that can be used to refer to this specific entity from other input blocks (xml);
- **subType**, *required string attribute*, in this attribute the user defines which of the post-processors needs to be used, choosing among the previously reported types. Obviously, this choice conditions the subsequent the required and/or optional `< PostProcessor >` sub nodes.

As already mentioned, all the types and meaning of the remaining sub-nodes depend on the post-processor type specified in the attribute *subType*. In the following sections the specifications of each type are reported.

13.5.0.1 BasicStatistics.

The **BasicStatistics** post-processor is the container of the algorithms to compute all the most important statistical quantities.

In order to use the *BasicStatistics* post-processor, the user needs to set the sub-node `< subType > BasicStatistics < /subType >`. In addition to this XML node, several others need (or not) to be inputted:

- `< what >`, *comma separated string, required field.*. List of quantities need to be computed. Currently the quantities available are:

- **covariance matrix**, Covariance matrix;
- **NormalizedSensitivity**, Matrix of normalized sensitivity coefficients;
- **sensitivity**, Matrix of sensitivity coefficients;
- **pearson**, Matrix of sensitivity coefficients;
- **expectedValue**, Expected value;
- **sigma**, Standard deviation;
- **variationCoefficient**, Coefficient of variation (sigma/expected value);
- **variance**, Variance;
- **skewness**, Skewness;
- **kurtois**, Kurtois;
- **median**, Data median;
- **percentile**, 95 percentile.

If all the quantities need to be computed, the user can input in the body of `< what >` the string “all”.

- `< parameters >`, **comma separated string, required field..** List of the parameters on which the previous operations need to be applied on (e.g., massFlow, Temperature);
- `< methodsToRun >`, **comma separated string, optional field..** The method names of an external Function that need to be run before computing any of the predefined quantities. If this XML node is inputted, the `< Function >` node must be present. *Default = None*;
- **Assembler Objects** These objects are either required or optional depending on the functionality of the Adaptive Sampler. The objects must be listed with a rigorous syntax that, except for the xml node tag, is common among all the objects. Each of these nodes must contain 2 attributes that are used to map those within the simulation framework:
 - **class**, *required string attribute*, it is the main “class” the listed object is from;
 - **type**, *required string attribute*, it is the object identifier or sub-type.

The **BasicStatistics** post-processor approach optionally accepts the following object type:

- `< Function >`, **string, required field..** The body of this xml block needs to contain the name of an External Function defined within the `< Functions >` main block (see section 12). This object needs to contain the methods listed in the node `< methodsToRun >`;

Example: _____

```

<Simulation>
...
<Models>
...
  <PostProcessor name="***" subType='BasicStatistics'
    debug='true' >
    <!-- ==> you can here specify what type of figure of
      merit
      you need to compute
      <what>expectedValue,sigma,variance,kurtois,pearson,covariance<
    -->
    <what>expectedValue</what>
    <parameters>x01,x02</parameters>
    <methodsToRun>failureProbability</methodsToRun>
  </PostProcessor>
...
<Models>
...
<Simulation>
-----

```

13.5.0.2 ComparisonStatistics.

To be finalized.

13.5.0.3 SafestPoint.

The **SafestPoint** post-processor provides the coordinates of the farthest point from the limit surface that is given as an input. The safest point coordinates are expected values of the coordinates of the farthest points from the limit surface in the space of the "controllable" variables based on the probability distributions of the "non-controllable" variables.

The term "controllable" identifies those variables that are under control during the system operation, while the "non-controllable" variables are stochastic parameters affecting the system behaviour randomly.

The "SafestPoint" post-processor requires the set of points belonging to the limit surface, which must be given as an input. The probability distributions as "Assembler Objects" are required in the "Distribution" section for both "controllable" and "non-controllable" variables.

The sampling method used by the "SafestPoint" is a "value" or "CDF" grid. At present only the "equal" grid type is available.

- *< Distribution >*: the probability distributions of the "controllable" and "non-controllable" variables are required. These are **Assembler Objects**, each of these nodes must contain 2 attributes that are used to map those within the simulation framework:
 - **class**, *required string attribute*, it is the main "class" the listed object is from;
 - **type**, *required string attribute*, it is the object identifier or sub-type.
- *< controllable >*: the list of the controllable variables is given here. Each variable is associated with its name and the two items below:
 - *< distribution >*: the name of the probability distribution associated with the controllable variable is specified here.
 - *< grid >*: type, number of steps and tolerance of the sampling grid are defined here.
- *< non – controllable >*: the list of the non-controllable variables is given here. Each variable is associated with its name and the two items below:
 - *< distribution >*: the name of the probability distribution associated with the non-controllable variable is specified here.
 - *< grid >*: type, number of steps and tolerance of the sampling grid are defined here.

Example:

```

-----
<Simulation>
...
  <Models>
    ...
    <PostProcessor name='SP' subType='SafestPoint'>
      <Distribution class = 'Distributions' type =
        'Normal'>x1_dst</Distribution>
      <Distribution class = 'Distributions' type =
        'Normal'>x2_dst</Distribution>
      <Distribution class = 'Distributions' type =
        'Normal'>gammay_dst</Distribution>
      <controllable>
        <variable name = 'x1'>
          <distribution>x1_dst</distribution>
          <grid type = 'value' steps = '20'>1</grid>
        </variable>
        <variable name = 'x2'>
          <distribution>x2_dst</distribution>
          <grid type = 'value' steps = '20'>1</grid>
        </variable>
      </controllable>
    </PostProcessor>
  </Models>
</Simulation>

```

```

    </controllable>
    <non-controllable>
      <variable name = 'gammay'>
        <distribution>gammay_dst</distribution>
        <grid type = 'value' steps = '20'>2</grid>
      </variable>
    </non-controllable>
  </PostProcessor>
  ...
</Models>
...
</Simulation>
-----

```

13.5.0.4 LimitSurface.

The **LimitSurface** post-processor is aimed to identify the transition zones that determine a change in the status of the system (Limit Surface).

In order to use the *LimitSurface*, the user needs to set the sub-node `< subType > LimitSurface < /subType >`. In addition to this XML node, several others need (or not) to be inputted:

- `< parameters >` , **comma separated string, required field..** List of the parameters that define the uncertain domain and from which the LS needs to be computed;
- `< tollerance >` , **float, optional field..** Absolute value converge tollerance. This value defines the coarsens of the evaluation grid. *Default= 1.e-4*;
- **Assembler Objects** These objects are either required or optional depending on the functionality of the Adaptive Sampler. The objects must be listed with a rigorous syntax that, except for the xml node tag, is common among all the objects. Each of these nodes must contain 2 attributes that are used to map those within the simulation framework:
 - **class**, *required string attribute*, it is the main “class” the listed object is from. For example, it can be “Models”, “Functions”, etc;
 - **type**, *required string attribute*, it is the object identifier or sub-type. For example, it can be “ROM”, “External”, etc.

The **LimitSurface** post-processor requires or optionally accepts the following objects’ types:

- `< ROM >` , **string, optional field..** If inputted, the body of this xml node must contain the name of a ROM defined in the `< Models >` block (see section 13.3);

- `< Function >`, **string, required field**..The body of this xml block needs to contain the name of an External Function defined within the `< Functions >` main block (see section 12). This object represents the boolean function that defines the transition boundaries. This function must implement a method called `_residuumSign(self)`, that returns either -1 or 1, depending on the system conditions (see section 12).

Example:

```
-----
<Simulation>
...
<Models>
...
<PostProcessor name="computeLimitSurface"
  subType='LimitSurface' debug='True'>
  <what>all</what>
  <parameters>x0,y0</parameters>
  <parameters>x0,y0</parameters>
  <ROM      class='Models'                type='ROM'
    >Acc</ROM>
  <!--You can add here a ROM defined in Models block.If
    not Present,
    a nearest algorithm is going to be used
  -->
  <Function class='Functions'              type='External'
    >goalFunctionForLimitSurface</Function>
  </PostProcessor>
  ...
</Models>
...
</Simulation>
-----
```

13.5.0.5 PrintCSV.

TO BE MOVED TO STEP "IOSTEP"

13.5.0.6 LoadCsvIntoInternalObject.

TO BE MOVED TO STEP "IOSTEP"

14 OutStream system

The PRA and UQ framework provides the capabilities to visualize and dump out the data that are generated, imported (from a system code) and post-processed during the analysis. These capabilities are contained in the "OutStream" system. Actually, two different OutStream types are available:

- **Print**, module that lets the user dump the data contained in the internal objects;
- **Plot**, module, based on Matplotlib [4], aimed to provide advanced plotting capabilities.

Both the types listed above only accept as "input" a *Data* object type. This choice has been taken since the "*Datas*" system (see section 10) has the main advantages, among the others, of ensuring a standardized approach for exchanging the data/meta-data among the different framework entities. Every module can project its outcomes into a *Data* object. This provides, to the user, the capability to visualize/dump all the modules' results. As already mentioned [put reference to the xml input section], the RAVEN framework input is based on the **Extensible Markup Language (XML)** format. Thus, in order to activate the "*OutStream*" system, the input needs to contain a block identified by the "< *OutStreamManager* >" tag (as shown below).

```
-----  
<OutStreamManager>  
  <!-- "OutStream" objects that need to be created-->  
</OutStreamManager>  
-----
```

In the "OutStreamManager" block an unlimited number of "Plot" and "Print" sub-blocks can be inputted. The input specifications and the main capabilities for both types are reported in the following sections.

14.1 Printing system

The Printing system has been created in order to let the user dump the data, contained in the internal data objects (see [reference to Data(s) section]), out at anytime during the calculation. Currently, the only available output is a **Comma Separated Value (CSV)** file. In the near future, an XML formatted file option will be available. This will facilitate the exchanging of results and provide the possibility to dump the solution of an analysis and "restart" another one constructing a *Data* from scratch. The XML code, that is reported below, shows different ways to request a *Print* OutStream. The user needs to provide a name for each sub-block (XML attribute). These names are then used in the *Steps*' blocks in order to activate the Printing options at anytime. As shown in the examples below, every *Print* block must contain, at least, the two required tags:

- `< type >`, the output file type (csv or xml). *Note, only csv is currently available*
- `< source >`, the *Data* name (one of the *Data* defined in the “*Datas*” block)

If only these two tags are provided (as in the “first-example” below), the output file will be filled with the whole content of the “d-name” *Data*.

```
-----
<OutputStreamManager>
  <Print name='first_example'>
    <type>csv</type>
    <source>d-name</source>
  </Print>
  <Print name='second-example'>
    <type>csv</type>
    <source>d-name</source>
    <variables>Output</variables>
  </Print>
  <Print name='third-example'>
    <type>csv</type>
    <source>d-name</source>
    <variables>Input</variables>
  </Print>
  <Print name='forth-example'>
    <type>csv</type>
    <source>d-name</source>
    <variables>Input | var-name-in, Output | var-name-out</variables>
  </Print>
</OutputStreamManager>
-----
```

If just few parts of the `< source >` are important for a particular analysis, the additional XML tag `< variables >` can be provided. In this block, the variables that need to be dumped must be inputted, in a comma separated format. The available options, for the `< variables >` sub-block, are listed below:

- **Output**, the output space will be dumped out (see “second-example”)
- **Input**, the input space will be dumped out (see “third-example”)
- **Input—var-name-in/Output—var-name-out**, only the particular variables “var-name-in” and “var-name-out” will be reported in the output file (see “forth-example”)

Note that all the XML tags are case-sensitive but not their content.

14.2 Plotting system

The Plotting system provides all the capabilities to visualize the analysis outcomes, in real-time or at the post-processing stage. The system is based on the Python library Matplotlib [4]. Matplotlib is a 2D/3D plotting library which produces publication quality figures in a variety of hardcopy formats and interactive environments across platforms. This external tool has been wrapped in the RAVEN framework, and is usable by the user. Since it was unfeasible to support, in the source code, all the interfaces for all the available plot types, the RAVEN Plotting system directly provide a formatted input structure for 11 different plot types (2D/3D). The user may request a plot not present among the supported ones, since the RAVEN Plotting system has the capability to construct on the fly the interface for a Plot, based on XML instructions. This capability will be discussed in the sub-section ??.

14.2.1 Plot input structure

In order to create a plot, the user needs to add, within the `< OutStreamManager >` block, a `< Plot >` sub-block. As for the *Print* OutStream, the user needs to specify a name as attribute of the plot. This name will then be used to request the plot in the *Steps*' block. In addition, the Plot block may need the following attributes:

- **dim**, *required integer attribute*, define the dimensionality of the plot: 2 (2D) or 3 (3D)
- **interactive**, *optional bool attribute (default=False)*, specify if the Plot needs to be interactively created (real-time screen visualization)
- **overwrite**, *optional bool attribute (default=False)*, if the plot needs to be dumped into picture file/s, does the code need to overwrite them every time a new plot (with the same name) is requested?

As shown, in the XML input example below, the body of the Plot XML input contains two main sub-nodes:

- `< actions >`, where general control options for the figure layout are defined (see [])
- `< plot_settings >`, where the actual plot options are provided

These two main sub-block are discussed in the following paragraphs.

14.2.1.1 “Actions” input block

The input in the `< actions >` sub-node is common to all the Plot types, since, in it, the user specifies all the controls that need to be applied to the figure style. This block must be unique in the definition of the `< Plot >` main block. In the following list, all the predefined “actions” are reported:

- `< how >`, comma separated list of output types:

- *screen*, show the figure on the screen in interactive mode
 - *pdf*, save the figure as a Portable Document Format file (PDF)
 - *png*, save the figure as a Portable Network Graphics file (PNG)
 - *eps*, save the figure as a Encapsulated Postscript file (EPS)
 - *pgf*, save the figure as a LaTeX PGF Figure file (PGF)
 - *ps*, save the figure as a Postscript file (PS)
 - *gif*, save the figure as a Graphics Interchange Format (GIF)
 - *svg*, save the figure as a Scalable Vector Graphics file (SVG)
 - *jpeg*, save the figure as a jpeg file (JPEG)
 - *raw*, save the figure as a Raw RGBA bitmap file (RAW)
 - *bmp*, save the figure as a Windows bitmap file (BMP)
 - *tiff*, save the figure as a Tagged Image Format file (TIFF)
 - *svgz*, save the figure as a Scalable Vector Graphics file (SVGZ)
- *< title >*, as the name suggests , within this block the user can specify the title of the figure. In the body, few other keywords (required and not) are present:
 - *< text >*, string type, title of the figure
 - *< kwargs >*, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example *< param1 > '1stKeyword' : 45 < /param1 >* will be converted into a dictionary, *< param2 > [56,67] < /param2 >* into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.title” method in [4].

- *< label_format >*, within this block the default scale formatting can be modified. In the body, few keywords can be specified (all optional):
 - *< style >*, string, the style of the number notation, 'sci' or 'scientific' for scientific, 'plain' for plain notation. Default = scientific

- *< scilimits >*, tuple, (m, n), pair of integers; if style is 'sci', scientific notation will be used for numbers outside the range 10^m to 10^n . Use (0,0) to include all numbers. NB. The value for this keyword, needs to be inputted between brackets [for example, (5,6)]. Default = (0,0)
- *< useOffset >*, bool or double, if True, the offset will be calculated as needed; if False, no offset will be used; if a numeric offset is specified, it will be used. Default = False
- *< axis >*, string, the axis where to apply the defined format, 'x', 'y' or 'both'. Default = 'both'. NB. If this action will be used in a 3-D plot, the user can input 'z' as well and 'both' will apply this format to all three axis.
- *< figure_properties >*, within this block the user specifies how to customize the figure style/quality. Thus, through this “action” the user has got full control on the quality of the figure, its dimensions, etc. This control is performed by the following keywords:
 - *< figsize >*, tuple (optional), (width, height), in inches
 - *< dpi >*, integer, dots per inch
 - *< facecolor >*, string, set the figure background color (please refer to “matplotlib.figure.Figure” in [4] for a list of all the colors available)
 - *< edgecolor >*, string, the figure edge background color (please refer to “matplotlib.figure.Figure” in [4] for a list of all the colors available)
 - *< linewidth >*, self explainable keyword
 - *< frameon >*, bool, if False, suppress drawing the figure frame
- *< range >*, the range “action” allows to specify the ranges of all the axis. All the keywords in the body of this block are optional:
 - *< ymin >*, double (optional), lower boundary for y axis
 - *< ymax >*, double (optional), upper boundary for y axis
 - *< xmin >*, double (optional), lower boundary for x axis
 - *< xmax >*, double (optional), upper boundary for x axis
 - *< zmin >*, double (optional), lower boundary for z axis. NB. Obviously, this keyword is effective in 3-D plots only
 - *< zmax >*, double (optional), upper boundary for z axis. NB. Obviously, this keyword is effective in 3-D plots only
- *< camera >*, the camera item is available in 3-D plots only. Through this “action”, it is possible to orientate the plot as wished. The controls are:
 - *< elevation >*, double (optional), stores the elevation angle in the z plane

- *< azimuth >*, double (optional), stores the azimuth angle in the x,y plane
- *< scale >*, the scale block allows the specification of the axis scales:
 - *< xscale >*, string (optional), scale of the x axis. Three options are available: “linear”, “log”, “symlog”. Default = linear
 - *< yscale >*, string (optional), scale of the y axis. Three options are available: “linear”, “log”, “symlog”. Default = linear
 - *< zscale >*, string (optional), scale of the z axis. Three options are available: “linear”, “log”, “symlog”. Default = linear. NB. Obviously, this keyword is effective in 3-D plots only
- *< add_text >*, same as title
- *< autoscale >*, the autoscale block is a convenience method for simple axis view autoscaling. It turns autoscaling on or off, and then, if autoscaling for either axis is on, it performs the autoscaling on the specified axis or axes. The following keywords are available:
 - *< enable >*, bool (optional), True turns autoscaling on, False turns it off. None leaves the autoscaling state unchanged. Default = True
 - *< axis >*, string (optional), string, the axis where to apply the defined format, 'x', 'y' or 'both'. Default = 'both'. NB. If this action will be used in a 3-D plot, the user can input 'z' as well and 'both' will apply this format to all three axis.
 - *< tight >*, bool (optional), if True, set view limits to data limits; if False, let the locator and margins expand the view limits; if None, use tight scaling if the only artist is an image, otherwise treat tight as False.
- *< horizontal_line >*, this “action” provides the ability to draw a horizontal line in the current figure. This capability might be useful, for example, if the user wants to highlight a trigger, function of a variable. The following keywords are settable:
 - *< y >*, double (optional), y coordinate. Default = 0
 - *< xmin >*, double (optional), starting coordinate on the x axis. Default = 0
 - *< xmax >*, double (optional), ending coordinate on the x axis. Default = 1
 - *< kwargs >*, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The `kwargs` block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56,67] < /param2 >` into a list, etc.). For reference regarding the available `kwargs`, see “`matplotlib.pyplot.axhline`” method in [4].

NB. This capability is not available for 3-D plots.

- `< vertical_line >`, similarly to the “`horizontal_line`” action, this block provides the ability to draw a vertical line in the current figure. This capability might be useful, for example, if the user wants to highlight a trigger, function of a variable. The following keywords are settable:

- `< x >`, double (optional), x coordinate. Default = 0
- `< ymin >`, double (optional), starting coordinate on the y axis. Default = 0
- `< ymax >`, double (optional), ending coordinate on the y axis. Default = 1
- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The `kwargs` block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56,67] < /param2 >` into a list, etc.). For reference regarding the available `kwargs`, see “`matplotlib.pyplot.axvline`” method in [4].

NB. This capability is not available for 3-D plots.

- `< horizontal_rectangle >`, this “action” provides the possibility to draw, in the current figure, a horizontally orientated rectangle . This capability might be useful, for example, if the user wants to highlight a zone in the plot. The following keywords are settable:
- `< ymin >`, double (required), starting coordinate on the y axis
 - `< ymax >`, double (required), ending coordinate on the y axis
 - `< xmin >`, double (optional), starting coordinate on the x axis. Default = 0
 - `< xmax >`, double (optional), ending coordinate on the x axis. Default = 1
 - `< kwargs >`, within this block the user can specify optional parameters with the following format:


```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56,67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.axhspan” method in [4].

NB. This capability is not available for 3-D plots.

- `< vertical_rectangle >`, this “action” provides the possibility to draw, in the current figure, a vertically orientated rectangle . This capability might be useful, for example, if the user wants to highlight a zone in the plot. The following keywords are settable:
 - `< xmin >`, double (required), starting coordinate on the x axis
 - `< xmax >`, double (required), ending coordinate on the x axis
 - `< ymin >`, double (optional), starting coordinate on the y axis. Default = 0
 - `< ymax >`, double (optional), ending coordinate on the y axis. Default = 1
 - `< kwargs >`, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56,67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.axvspan” method in [4].

NB. This capability is not available for 3-D plots.

- `< axes_box >`, this keyword controls the axes’ box. No body and its value can be ‘on’ or ‘off’.
- `< axis_properties >`, this block is used to set axis properties. There are not fixed keywords. If only a single property needs to be set, it can be specified as body of this block, otherwise a

dictionary-like string needs to be provided. For reference regarding the available keys, refer to “matplotlib.pyplot.axis” method in [4].

- `< grid >`, this block is used to define a grid that needs to be added in the plot. The following keywords can be inputted:
 - `< b >`, double (required), starting coordinate on the x axis
 - `< which >`, double (required), ending coordinate on the x axis
 - `< axis >`, double (optional), starting coordinate on the y axis. Default = 0
 - `< kwargs >`, within this block the user can specify optional parameters with the following format:

```
-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----
```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56,67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.grid” method in [4].

14.2.1.2 “plot_settings” input block

The sub-block identified by the keyword `< plot_settings >` is used to define the plot characteristics. Within this sub-section at least a `< plot >` block must be present. the `< plot >` sub-section may not be unique within the `< plot_settings >` definition; the number of `< plot >` sub-blocks is equal to the number of plots that need to be placed in the same figure. For example, in the following XML cut, a “line” and a “scatter” type are combined in the same figure.

```
-----
<OutputStreamManager>
  <Plot name='example2PlotsCombined' dim='2'>
    <actions>
      <!-- Actions -->
    </actions>
    <plot\_settings>
      <plot>
        <type>line</type>
        <x>d-type|Output|x1</x>
        <y>d-type|Output|y1</y>
      </plot>
```

```

    <plot>
      <type>scatter</type>
      <x>d-type|Output|x2</x>
      <y>d-type|Output|y2</y>
    </plot>
    <xlabel>label X</xlabel>
    <ylabel>label Y</ylabel>
  </plot\_settings>
</Plot>
</OutputStreamManager>

```

As already mentioned, within the `< plot_settings >` block, at least a `< plot >` sub-block needs to be inputted. Independently from the plot type, some keywords are mandatory:

- `< type >`, string, required parameter, the plot type (for example, line, scatter, wireframe, etc.);
- `< x >`, string, required parameter, the parameter needs to be considered as x coordinate;
- `< y >`, string, required parameter, the parameter needs to be considered as y coordinate;
- `< z >`, string required parameter (3D plots only), the parameter needs to be considered as z coordinate.

In addition other plot-dependent keywords, reported in section 14.2.1.3, can be provided.

Under the `< plot_settings >` block other keywords, common to all the plots the user decided to combine in the figure, can be inputted, such as:

- `< xlabel >`, string, optional parameter, x axis label;
- `< ylabel >`, string, optional parameter, y axis label;
- `< zlabel >`, string, optional parameter (3D plots only), z axis label;
- `< colorMap >`, string, the parameter needs to be used to define a color map.

As already mentioned, the Plot system accepts as parameter (i.e., x, y, z, colorMap) the **Datas** object only. Considering the structure of “Datas”, the parameters are inputted as follow:

DataObjectName|Input(or)Output|variableName.

If the “variableName” contains the symbol |, it must be surrounded by brackets:

DataObjectName|Input(or)Output|(whatever|variableName).

14.2.1.3 Predefined Plotting System: 2D/3D

As already mentioned above, the Plotting system provides specialized input structure for several different kind of plots:

- *2 Dimensional plots:*

- *scatter*. 2 dimensional scatter plot. It used to create a scatter plot of x vs y, where x and y are sequences of numbers of the same length;
- *line*. 2 dimensional line plot. It used to create a line plot of x vs y, where x and y are sequences of numbers of the same length;
- *histogram*. 2 dimensional histogram plot. Compute and draw the histogram of x. It must be noticed that this plot accepts only the XML node $\langle x \rangle$ even if it is considered as 2D plot type;
- *stem*. 2 dimensional stem plot. A stem plot plots vertical lines at each x location from the baseline to y, and places a marker there;
- *step*. 2 dimensional step plot;
- *pseudocolor*. 2 dimensional scatter plot. It creates a pseudocolor plot of a two dimensional array. The two dimensional array is built creating a mesh from $\langle x \rangle$ and $\langle y \rangle$ data, in conjunction with the data inputted in $\langle colorMap \rangle$;
- *contour*. 2 dimensional contour plot. It plots the contour lines. Contour plot is built creating a plot from $\langle x \rangle$ and $\langle y \rangle$ data, in conjunction with the data inputted in $\langle colorMap \rangle$;
- *filledContour*. 2 dimensional contour plot. It plots the filled contour leveles. Filled contour plot is built creating a plot from $\langle x \rangle$ and $\langle y \rangle$ data, in conjunction with the data inputted in $\langle colorMap \rangle$;

- *3 Dimensional plots:*

- *scatter*. 3 dimensional scatter plot. It is used to create a scatter plot of (x,y) vs z, where x, y, z are sequences of numbers of the same length;
- *line*. 3 dimensional line plot. It used to create a line plot of (x,y) vs z, where x, y, z are sequences of numbers of the same length;
- *stem*. 3 dimensional stem plot. It creates a 3 Dimensional stem plot of (x,y) vs z;
- *surface*. 3 dimensional surface plot. Create a surface plot of (x,y) vs z. By default it will be colored in shades of a solid color, but it also supports color mapping;
- *wireframe*. 3 dimensional wire-frame plot. No color mapping is supported;
- *tri-surface*. 3 dimensional tri-surface plot. It is a surface plot with automatic triangulation;

- *contour3D*. 3 dimensional contour plot. It plots the contour lines. Contour plot is built creating a plot from $\langle x \rangle$, $\langle y \rangle$ and $\langle z \rangle$ data, in conjunction with the data inputted in $\langle colorMap \rangle$
- *filledContour3D*. 3 dimensional filled contour plot. It plots the filled contour leveles. Filled contour plot is built creating a plot from $\langle x \rangle$, $\langle y \rangle$ and $\langle z \rangle$ data, in conjunction with the data inputted in $\langle colorMap \rangle$;
- *histogram*. 3 dimensional histogram plot. Compute and draw the histogram of x and y. It must be noticed that this plot accepts only the XML nodes $\langle x \rangle$ and $\langle y \rangle$ even if it is considered as 3D plot type

As already mentioned, the settings for each plot type are inputted within the XML block called $\langle plot \rangle$. The sub-nodes that can be inputted depends on the plot type: each plot type has its own set of parameters that can be specified.

In the following sub-sections all the options for the plot types listed above are reported.

14.2.2 2D & 3D Scatter plot.

In order to create a “scatter” plot, either 2D or 3D, the user needs to write in the $\langle type \rangle$ body the keyword “scatter”. In order to customize the plot, the user can define the following XML sub-nodes:

- $\langle s \rangle$, **integer, optional field..** Size in points². *Default = 20*;
- $\langle c \rangle$, **string, optional field..** color or sequence of color. $\langle c \rangle$ can be a single color format string, or a sequence of color specifications of length N, or a sequence of N numbers to be mapped to colors using the cmap and norm specified via kwargs. Note that $\langle c \rangle$ should not be a single numeric RGB or RGBA sequence because that is indistinguishable from an array of values to be colormapped. $\langle c \rangle$ can be a 2-D array in which the rows are RGB or RGBA;
- $\langle marker \rangle$, **string, optional field..** Marker type. *Default = o*;
- $\langle alpha \rangle$, **string, optional field..** The alpha blending value, between 0 (transparent) and 1 (opaque). *Default = None* ;
- $\langle linewidths \rangle$, **string, optional field..** Widths of lines. Note that this is a tuple, and if you set the linewidths argument you must set it as a sequence of floats. *Default = None*;
- $\langle kwargs \rangle$, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56, 67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.scatter” method in [4].

14.2.3 2D & 3D Line plot.

In order to create a “line” plot, either 2D or 3D, the user needs to write in the `< type >` body the keyword “line”. In order to customize the plot, the user can define the following XML sub-nodes:

- `< interpolationType >`, **string, optional field..** Type of interpolation algorithm to use for the data. Available are “nearest”, “linear”, “cubic”, “multiquadric”, “inverse”, “gaussian”, “Rbflinear”, “Rbfcubic”, “quintic”, “thin_plate”. *Default = linear*;
- `< interpPointsX >`, **integer, optional field..** Number of points need to be used for interpolation of x axis;
- `< interpPointsY >`, **integer, optional field..** Number of points need to be used for interpolation of y axis (only 3D line plot);

14.2.4 2D & 3D Histogram plot.

In order to create a “histogram” plot, either 2D or 3D, the user needs to write in the `< type >` body the keyword “histogram”. In order to customize the plot, the user can define the following XML sub-nodes:

- `< bins >`, **integer or array like, optional field..** Number of bins if an integer is inputted or a sequence of edges if a python list is defined. *Default = 10*;
- `< normed >`, **boolean, optional field..** if True normalize the histogram to 1. *Default = False*;

- *< weights >, **sequence, optional field.***.. An array of weights, of the same shape as x. Each value in x only contributes its associated weight towards the bin count (instead of 1). If *normed* is True, the weights are normalized, so that the integral of the density over the range remains 1. *Default = None*;
- *< cumulative >, **boolean, optional field.***.. If True, then a histogram is computed where each bin gives the counts in that bin plus all bins for smaller values. The last bin gives the total number of datapoints. If *normed* is also True then the histogram is normalized such that the last bin equals 1. If *cumulative* evaluates to less than 0 (e.g., -1), the direction of accumulation is reversed. In this case, if *normed* is also True, then the histogram is normalized such that the first bin equals 1. *Default = False* ;
- *< histtype >, **string, optional field.***.. The type of histogram to draw:
 - **bar** is a traditional bar-type histogram. If multiple data are given the bars are arranged side by side;
 - **barstacked** is a bar-type histogram where multiple data are stacked on top of each other;
 - **step** generates a lineplot that is by default unfilled;
 - **stepfilled** generates a lineplot that is by default filled;

Default = bar;
- *< align >, **string, optional field.***.. Controls how the histogram is plotted.
 - **left** bars are centered on the left bin edge;
 - **mid** bars are centered between the bin edges;
 - **right** bars are centered on the right bin edges;

Default = mid;
- *< orientation >, **string, optional field.***.. Orientation of the histogram:
 - **horizontal**;
 - **vertical**;

Default = vertical;
- *< rwidth >, **float, optional field.***.. The relative width of the bars as a fraction of the bin width. *Default = None*;
- *< log >, **boolean, optional field.***.. Set a log scale. *Default = False*;
- *< color >, **string, optional field.***.. Color of the histogram. *Default = blue*;

- *< stacked >, boolean, optional field..* If True, multiple data are stacked on top of each other If False multiple data are arranged side by side if histtype is 'bar' or on top of each other if histtype is 'step'. *Default = False*;
- *< kwargs >*, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example *< param1 > '1stKeyword' : 45 < /param1 >* will be converted into a dictionary, *< param2 > [56,67] < /param2 >* into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.hist” method in [4].

14.2.5 2D & 3D Stem plot.

In order to create a “stem” plot, either 2D or 3D, the user needs to write in the *< type >* body the keyword “stem”. In order to customize the plot, the user can define the following XML sub-nodes:

- *< linefmt >, string, optional field..* Line style. *Default = b-;*
- *< markerfmt >, string, optional field..* Marker format. *Default = bo;*
- *< basefmt >, string, optional field..* Base format. *Default = r-;*
- *< kwargs >*, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example *< param1 > '1stKeyword' : 45 < /param1 >* will be converted into a dictionary, *< param2 > [56,67] < /param2 >* into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.stem” method in [4].

14.2.6 2D Step plot

In order to create a 2D “step” plot, the user needs to write in the `< type >` body the keyword “step”. In order to customize the plot, the user can define the following XML sub-nodes:

- `< where >`, **string, optional field..** Positioning:
 - **pre**, the interval from $x[i]$ to $x[i+1]$ has level $y[i+1]$;
 - **post**, that interval has level $y[i]$;
 - **mid**, the jumps in y occur half-way between the x -values;

Default = mid;

- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```
-----  
<kwargs>  
  <param1>value1</param1>  
  <param2>value2</param2>  
</kwargs>  
-----
```

The `kwargs` block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56, 67] < /param2 >` into a list, etc.). For reference regarding the available `kwargs`, see “`matplotlib.pyplot.step`” method in [4].

14.2.7 2D Pseudocolor plot

In order to create a 2D “pseudocolor” plot, the user needs to write in the `< type >` body the keyword “pseudocolor”. In order to customize the plot, the user can define the following XML sub-nodes:

- `< interpolationType >`, **string, optional field..** Type of interpolation algorithm to use for the data. Available are “nearest”, “linear”, “cubic”, “multiquadric”, “inverse”, “gaussian”, “Rbflinear”, “Rbfcubic”, “quintic”, “thin_plate”. *Default = linear;*
- `< interpPointsX >`, **integer, optional field..** Number of points need to be used for interpolation of x axis;
- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56, 67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.pcolor” method in [4].

14.2.8 2D Contour or filledContour plots.

In order to create a 2D “Contour” or “filledContour” plot, the user needs to write in the `< type >` body the keyword “contour” or “filledContour”, respectively. In order to customize the plot, the user can define the following XML sub-nodes:

- `< number_bins >`, **integer, optional field.**.. Number of bins. *Default = 5*;
- `< interpolationType >`, **string, optional field.**.. Type of interpolation algorithm to use for the data. Available are “nearest”, “linear”, “cubic”, “multiquadric”, “inverse”, “gaussian”, “Rbflinear”, “Rbfcubic”, “quintic”, “thin_plate”. *Default = linear*;
- `< interpPointsX >`, **integer, optional field.**.. Number of points need to be used for interpolation of x axis;
- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56, 67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.contour” method in [4].

14.2.9 3D Surface Plot.

In order to create a 3D “Surface” plot, the user needs to write in the `< type >` body the keyword “surface”. In order to customize the plot, the user can define the following XML sub-nodes:

- `< rstride >`, **integer, optional field.**. Array row stride (step size). *Default = 1*;
- `< cstride >`, **integer, optional field.**. Array column stride (step size). *Default = 1*;
- `< cmap >`, **string, optional field.**. Color map. *Default = jet*;
- `< antialiased >`, **boolean, optional field.**. Antialiased rendering. *Default = False*;
- `< linewidth >`, **integer, optional field.**. Widths of lines. Note that this is a tuple, and if you set the linewidths argument you must set it as a sequence of floats. *Default = 0*;
- `< interpolationType >`, **string, optional field.**. Type of interpolation algorithm to use for the data. Available are “nearest”, “linear”, “cubic”, “multiquadric”, “inverse”, “gaussian”, “Rbflinear”, “Rbfcubic”, “quintic”, “thin_plate”. *Default = linear*;
- `< interpPointsX >`, **integer, optional field.**. Number of points need to be used for interpolation of x axis;
- `< interpPointsY >`, **integer, optional field.**. Number of points need to be used for interpolation of y axis;
- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```
-----  
<kwargs>  
  <param1>value1</param1>  
  <param2>value2</param2>  
</kwargs>  
-----
```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56, 67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.contour” method in [4].

14.2.10 3D Wireframe Plot.

In order to create a 3D “Wireframe” plot, the user needs to write in the `< type >` body the keyword “wireframe”. In order to customize the plot, the user can define the following XML sub-nodes:

- `< rstride >`, **integer, optional field.**.. Array row stride (step size). *Default = 1*;
- `< cstride >`, **integer, optional field.**.. Array column stride (step size). *Default = 1*;
- `< cmap >`, **string, optional field.**.. Color map. *Default = jet*;
- `< interpolationType >`, **string, optional field.**.. Type of interpolation algorithm to use for the data. Available are “nearest”, “linear”, “cubic”, “multiquadric”, “inverse”, “gaussian”, “Rbflinear”, “Rbfcubic”, “quintic”, “thin_plate”. *Default = linear*;
- `< interpPointsX >`, **integer, optional field.**.. Number of points need to be used for interpolation of x axis;
- `< interpPointsY >`, **integer, optional field.**.. Number of points need to be used for interpolation of y axis;
- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56, 67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.contour” method in [4].

14.2.11 3D Tri-surface Plot.

In order to create a 3D “Tri-surface” plot, the user needs to write in the `< type >` body the keyword “tri-surface”. In order to customize the plot, the user can define the following XML sub-nodes:

- `< color >`, **string, optional field.**.. Color of the surface patches. *Default = b*;
- `< shade >`, **boolean, optional field.**.. Apply shading. *Default = False*;
- `< cmap >`, **string, optional field.**.. Color map. *Default = jet*;
- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `< param2 > [56, 67] < /param2 >` into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.contour” method in [4].

14.2.12 3D Contour or filledContour plots.

In order to create a 3D “Contour” or “filledContour” plot, the user needs to write in the `< type >` body the keyword “contour3D” or “filledContour3D”, respectively. In order to customize the plot, the user can define the following XML sub-nodes:

- `< number_bins >`, **integer, optional field.**.. Number of bins. *Default = 5*;
- `< interpolationType >`, **string, optional field.**.. Type of interpolation algorithm to use for the data. Available are “nearest”, “linear”, “cubic”, “multiquadric”, “inverse”, “gaussian”, “Rbflinear”, “Rbfcubic”, “quintic”, “thin_plate”. *Default = linear*;
- `< interpPointsX >`, **integer, optional field.**.. Number of points need to be used for interpolation of x axis;
- `< interpPointsY >`, **integer, optional field.**.. Number of points need to be used for interpolation of y axis;
- `< kwargs >`, within this block the user can specify optional parameters with the following format:

```

-----
<kwargs>
  <param1>value1</param1>
  <param2>value2</param2>
</kwargs>
-----

```

The kwargs block is able to convert whatever string into a python type (for example `< param1 > '1stKeyword' : 45 < /param1 >` will be converted into a dictionary, `<`

param2 > [56, 67] < /*param2* > into a list, etc.). For reference regarding the available kwargs, see “matplotlib.pyplot.contour” method in [4].

14.2.13 Example XML input.

```
<OutputStreamManager>
  <Plot name='2DHistoryPlot' dim='2' interactive='False'
    overwrite='False'>
    <actions>
      <how>pdf,png,eps</how>
      <title>
        <text>***</text>
      </title>
    </actions>
    <plot_settings>
      <plot>
        <type>line</type>
        <x>stories|Output|time</x>
        <y>stories|Output|pipe1_Hw</y>
        <kwargs>
          <color>green</color>
          <label>pipe1-Hw</label>
        </kwargs>
      </plot>
      <plot>
        <type>line</type>
        <x>stories|Output|time</x>
        <y>stories|Output|pipe1_aw</y>
        <kwargs>
          <color>blue</color>
          <label>pipe1-aw</label>
        </kwargs>
      </plot>
      <xlabel>time [s]</xlabel>
      <ylabel>evolution</ylabel>
    </plot_settings>
  </Plot>
</OutputStreamManager>
```

15 Existing Interfaces

15.1 RELAP5 Interface

15.1.1 Files

In the `< Files >` section, as specified before, there should be specified all the files needed for the code to run. In the case of RELAP5 most of the times the files needed are the following:

- RELAP5 Input file
- Table file or files that RELAP needs to run
- The Relap5 executable

Example:

```
<Files>X10.i,tpfh2o,r5executable.x</Files>
```

It is a good practice to put inside the working directory all of these files and also:

- the RAVEN input file
- the license for the executable of RELAP5

15.1.2 Sequence

In the `< Sequence >` section there should be specified the names of the steps declared in the `< Steps >` block. So if we called the first multirun: "Grid_Sampler" and the second multirun: "MC_Sampler" in the sequence section we should see this:

```
<Sequence>Grid_Sampler,MC_Sampler</Sequence>
```

15.1.3 batchSize and mode

For the `< batchSize >` and `< mode >` sections please take a look at the RunInfo block in the previous chapters.

15.1.4 RunInfo

After all of these blocks are filled out here is a standard example of what a RunInfo block can look like:

```
<RunInfo>
  <WorkingDir>~/working_dir</WorkingDir>
  <Files>inputfilerelap.i,tpfh2o,executablerelap5.x</Files>
  <Sequence>Grid_Sampler,MC_Sampler</Sequence>
  <batchSize>1</batchSize>
    <mode>pbsdsh</mode>
  <expectedTime>1:00:00</expectedTime>
  <ParallelProcNumb>1</ParallelProcNumb>
</RunInfo>
```

15.1.5 Models

For the `< Models >` block here is a standard example of what can be used to use RELAP5 as the external model:

```
<Models>
  <Code name='MyRELAP'
    subType='Relap5'><executable>~/path_to_the_executable</executable></Code>
</Models>
```

15.1.6 Distributions

In the `< Distribution >` block are defined the distributions that are going to be used for the sampling of the variables defined in the `< Samplers >` block. For all the possible distributions and all their possible inputs please see the chapter about Distributions. Here we give a general example of three different distributions, with the names:

```
<Distributions debug='True'>
  <Triangular name='BPfailtime'>
    <apex>5.0</apex>
    <min>4.0</min>
    <max>6.0</max>
  </Triangular>
  <LogNormal name='BPrepairtime'>
    <mean>0.75</mean>
```



```

        <sigma>0.25</sigma>
    </LogNormal>
    <Uniform name='ScalFactPower'>
        <low>1.0</low>
        <hi>1.2</hi>
    </Uniform>
</Distributions>

```

It is good practice to name the distribution something similar to what kind of variable is going to be sampled, since there might be many variables with the same kind of distributions but different input parameters.

15.1.7 Samplers

In the `< Samplers >` block we want to define the variables that are going to be sampled. **Example:** We want to do the sampling of 3 variables:

- Battery Fail Time
- Battery Repair Time
- Scaling Factor Power Rate

We are going to sample these 3 variables using two different sampling methods: grid and Monte-Carlo.

In RELAP5 the sampler reads the variable in this way: Given the name, the first number is the card number, the second number is the word number. In this example we are sampling:

- For card 0000588 (trip) the word 6 (battery failure time)
- For card 0000575 (trip) the word 6 (battery repair time)
- For card 20210000 (reactor power) the word 4 (reactor scaling factor)

We proceed to do so for both the Grid sampling and the MonteCarlo sampling.

```

<Samplers debug='True'>
    <Grid name='Grid_Sampler' initial_seed='210491'>
        <variable name='0000588:6'>
            <distribution>BPfailtime</distribution>

```

```

        <grid type='value' construction='equal'
            lowerBound='0.0' steps='10'>2880</grid>
    </variable>
    <variable name='0000575:6'>
        <distribution>BPrepairtime</distribution>
        <grid type='value' construction='equal'
            lowerBound='0.0' steps='10'>2880</grid>
    </variable>
        <variable name='20210000:4'>
            <distribution>ScalFactPower</distribution>
            <grid type='value' construction='equal'
                lowerBound='1.0' steps='10'>0.02</grid>
    </Grid>
    <MonteCarlo name='MC_Sampler' limit='1000'>
        <variable name='0000588:6'>
            <distribution>BPfailtime</distribution>
        </variable>
        <variable name='0000575:6'>
            <distribution>BPrepairtime</distribution>
        </variable>
        <variable name='20210000:4'>
            <distribution>ScalFactPower</distribution>
        </variable>
    </MonteCarlo>
</Samplers>

```

It can be seen that each variable is connected with a proper distribution defined in the *< Distributions >* block. Here is how the Input would be read for the first variable:

We are sampling a variable situated in word 6 of the card 0000588 using a Grid sampling method. The distribution that this variable is following is a Triangular distribution (see section above). We are sampling this variable beginning from 0.0 in 10 *equal* steps of 2880.

15.1.8 Steps

For a RELAP interface the steps that are probably going to be used are the *< MultiRun >*. First we need to name the step: this name is one the ones used in the *< Sequence >* block. In this case: Grid_Sampler and MC_Sampler.

```

<MultiRun name='Grid_Sampler' debug='True'
    re-seeding='210491'>

```

With this step with need to import all the files needed for the simulation:

- RELAP input file
- Relap 5 Executable
- elements tables – tpfh2o

```
<Input class='Files' type=''>inputrelap.i</Input>
<Input class='Files' type=''>r5executable.x</Input>
<Input class='Files' type=''>tpfh2o</Input>
```

We then need to define which model it is used:

```
<Model class='Models' type='Code'>MyRELAP</Model>
```

We then need to specify which Sampler is used, and this can be done as follows:

```
<Sampler class='Samplers' type='Grid'>Grid_Sampler</Sampler>
```

And lastly we need to specify what kind of Output the used wants. For example the user might want to make a DataBase (in RAVeN the DataBase crated is a HDF5 file. Here is a classical example:

```
<Output class='DataBases' type='HDF5'>MC_out</Output>
```

Following is the example of two MultiRun steps which use different sampling methods (grid and montecarlo), and creating two different DataBases for each one:

```
<Steps debug='True'>
<MultiRun name='Grid_Sampler' debug='True'
  re-seeding='210491'>
  <Input class='Files' type=''>X10.i</Input>
  <Input class='Files' type=''>r5executable.x</Input>
  <Input class='Files' type=''>tpfh2o</Input>
  <Model class='Models' type='Code'>MyRELAP</Model>
  <Sampler class='Samplers' type='Grid'>Grid_Sampler</Sampler>
  <Output class='DataBases' type='HDF5'>Grid_out</Output>
</MultiRun>
```

```

<MultiRun name='MC_Sampler' debug='True'
  re-seeding='210491'>
  <Input class='Files' type=''
    >X10.i</Input>
  <Input class='Files' type=''
    >r5executable.x</Input>
  <Input class='Files' type=''
    >tpfh2o</Input>
  <Model class='Models' type='Code'
    >MyRELAP</Model>
  <Sampler class='Samplers' type='MonteCarlo'
    >MC_Sampler</Sampler>
  <Output class='DataBases' type='HDF5'
    >MC_out</Output>
</MultiRun>
</Steps>

```

15.1.9 DataBases

As shown in the `< Steps >` block, the code is creating two DataBases called Grid_out and MC_out. So the user needs to input the following

```

<DataBases>
  <HDF5 name="Grid_out"/>
  <HDF5 name="MC_out"/>
</DataBases>

```

As listed before, this will create two DataBases. The files are .h5 extension files.

15.2 RELAP7 Interface

15.2.1 Files

In the `< Files >` section, as specified before, there should be specified all the files needed for the code to run. In the case of RELAP7 most of the times the files needed are the following:

- RELAP7 Input file
- Control Logic file

Example:

```
<Files>nat_circ,control_logic.py</Files>
```

15.2.2 Models

For the `< Models >` block RELAP7 uses the RAVEN executable. Here is a standard example of what can be used to use RELAP7 as the model:

```
<Models>
  <Code name=' MyRAVEN'
    subType=' RAVEN' ><executable>~path/to/RAVEN-opt</executable></Code>
</Models>
```

15.2.3 Distributions

The `< Distributions >` block, when using RELAP7 has to be specified also through the control logic. Given the names of the distributions, and their parameters, a python file should be used for the control logic. For example, it is required the sampling of a normal distribution for the primary pressure in RELAP7.

```
<Distributions>
  <Normal name="Prim_Pres">
    <mean>1000000</mean>
    <sigma>100<sigma/>
  </Normal>
</Distributions>
```

The python file associated to it should look like this:

```
def initial_function(monitored, controlled, auxiliary)
    print("monitored",monitored,"controlled",
        controlled,"auxiliary",auxiliary)

    controlled.pressure_in_pressurizer =
        distributions.Prim_Pres.getDistributionRandom()
    return
```

15.2.4 Samplers

In the `< Samplers >` block there are going to be defined both which kind of sampling method used, the variables not sampled inside the control logic and of course which distributions will follow the chosen sampling method. For the example it was chosen a MonteCarlo sampling with a 500 runs. The global initial pressure wasn't specified in the control logic so it is sampled in this block. It is also specified that such initial pressure has to follow the same distribution as the primary pressure.

```
<Samplers>
  <MonteCarlo name="MC_samp" limit="500">
    <variable name="GlobalParams|global_init_P">
      <distribution>Prim_Pres</distribution>
    </variable>
  </MonteCarlo>
</Samplers>
```

References

[1]

[2] M. P. Forum, “Mpi: A message-passing interface standard,” tech. rep., Knoxville, TN, USA, 1994.

[3]

[4] J. D. Hunter, “Matplotlib: A 2d graphics environment,” *Computing In Science & Engineering*, vol. 9, no. 3, pp. 90–95, 2007.

