EDF R&D

Fluid Dynamics, Power Generation and Environment Department
Single Phase Thermal-Hydraulics Group

6, quai Watier
F-78401 Chatou Cedex

Tel: 33 1 30 87 75 40
Fax: 33 1 30 87 79 16

MARCH 2024

code_saturne documentation

code_saturne version 8.0 tutorial: simple junction

contact: saturne-support@edf.fr

TABLE OF CONTENTS

Part I

Introduction

1 Introduction

1.1 code_saturne short presentation

code_saturne is a system designed to solve the Navier-Stokes equations in the cases of 2D, 2D ax-isymmetric or 3D flows. Its main module is designed for the simulation of flows which may be steady or unsteady, laminar or turbulent, incompressible or potentially dilatable, isothermal or not. Scalars and turbulent fluctuations of scalars can be taken into account. The code includes specific modules, referred to as "specific physics", for the treatment of lagrangian particle tracking, semi-transparent radiative transfer, gas, pulverized coal and heavy fuel oil combustion, electricity effects (Joule effect and electric arcs) and compressible flows. code_saturne relies on a finite volume discretization and allows the use of various mesh types which may be hybrid (containing several kinds of elements) and may have structural non-conformities (hanging nodes). This code_saturne GUI version is architectured to provide users a logical approach to process CFD simulation. The following figure I.1 code_saturne GUI. You will find 3 main zones :

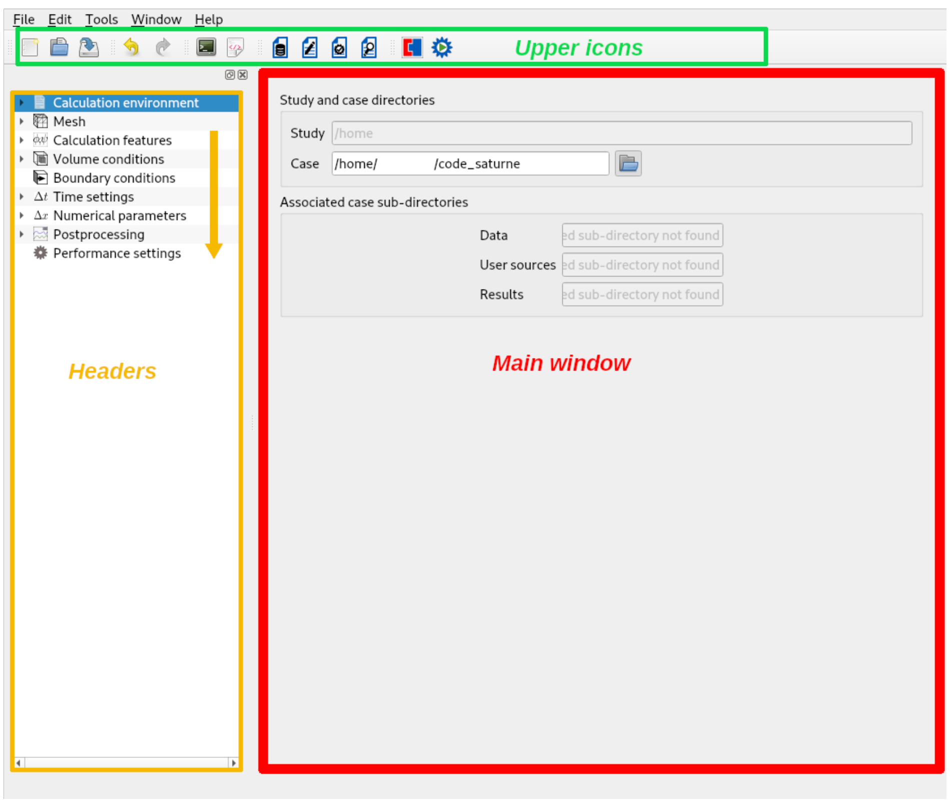


Figure I.1: code_saturne GUI

1. **Upper icons - green zone** : User can manage case file (from the creation to the computation)
2. **Header tabs window - orange zone** : User can access and define all mandatory settings to perform CFD analyzis
3. **Main window - red zone** : User can set parameters for every selected tab

code_saturne tutorials follow a logical process for every analyzis. User should begin by **Calculation envrionement** tab and finish by **Performance settings** tab before running computation.

1.2 About this document

The present document is a tutorial for code_saturne version 8.0. It presents a simple test case and guides the future code_saturne user step by step into the preparation and the computation of that case.

The test case directory, containing the necessary meshes and data is available in the `examples` directory.

This tutorial focuses on the procedure and the preparation of the code_saturne computations. For more elements on the structure of the code and the definition of the different variables, it is higly recommended to refer to the user manual.

1.3 code_saturne copyright informations

Part II

Simple junction testcase

1 Study description

1.1 Study creation and preparation

The first thing to do is to prepare the computation directories. You will find all tutorial folders in the examples directory 🗀 `examples`. Here, the study directory 🗀 `simple_junction` will contain a single calculation directory 🗀 `case1`.

Create the study 🗀 `simple_junction` and the 🗀 `case1`. There are three ways to create the study :

1. Within SALOME module CFDStudy -as explained in the Shear driven cavity tutorial-
2. With code_saturne-via the terminal-
3. With code_saturne-via the GUI (Graphic User Interface)-

The second option can be done by typing the following commands in your terminal:

```
$ code_saturne create -s simple_junction -c case1
```

Then code_saturne Graphical User Interface (GUI) can be launched by typing the command lines as below:

```
$ cd simple_junction/case1/DATA
$ ./code_saturne gui &
```

And the following window opens (fig II.1).

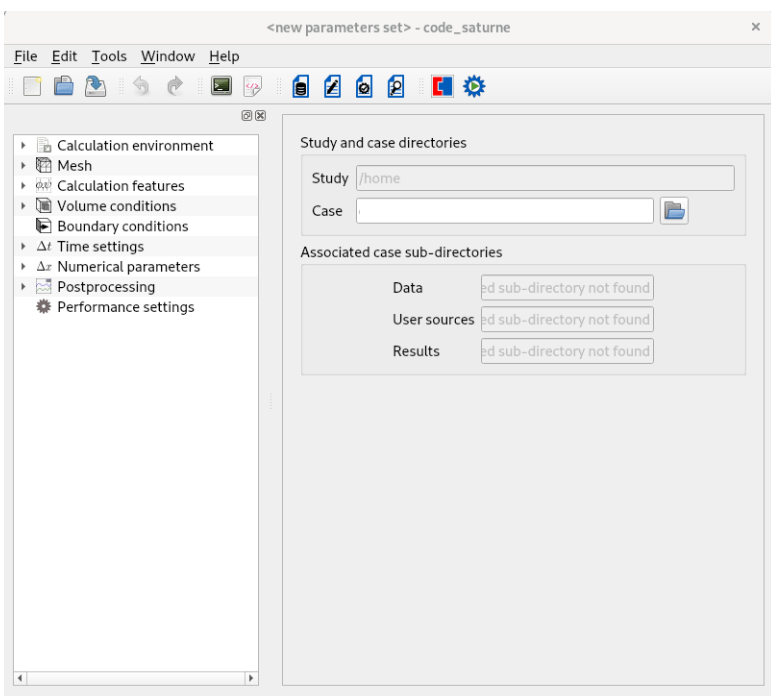


Figure II.1: code_saturne (GUI) graphic window

The third option can be done by first launching code_saturne GUI then creating and opening new cases and or meshes by clicking on File ⟩ New Case as follow (fig II.2):

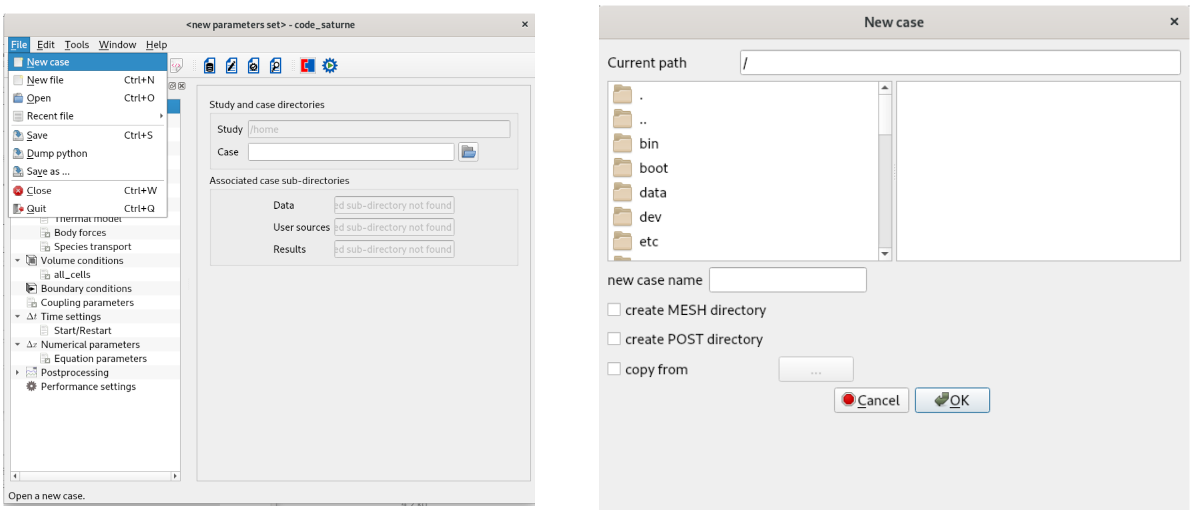


Figure II.2: code_saturne (GUI) graphic window - case creation

The mesh files, available in the examples directory, should be copied in the directory 🗀`MESH/`, by the command line as follows or by your favorite explorer:

```
$ cd simple_junction/MESH/
$ cp .../examples/1-simple_junction/mesh/downcomer.med .
```

If you do use SALOME here is a helping note for you :

SALOME interface helping note: Once the mesh is copied in the directory 🗀`MESH`, you can update the object browser (open a contextual menu by a right-click on the study name or the case name in the object browser, and left-clik on the entry Update Object Browser).

The mesh can then be directly displayed in the VTK viewer (the open viewer when module CFDStudy is active). To do so, follow these steps:

- In the object browser of *SALOME*, right-click on the mesh of the study (in the directory 🗀 `MESH` of the study), then select *'Convert to MED'*. A med file should be generated in the same directory;
- Right-click on this med file, then select *'Export in SMESH'*. A heading **Mesh** should appear in the object browser;
- Under this heading, right-click on the mesh name and then *'Display mesh'*;

1.2 Objective

The aim of this case is to train the user of code_saturne on an oversimplified 2D junction including an inlet, an outlet, walls and symmetries.

1.3 Description of the configuration

The configuration is two-dimensional.

It consists of a simple junction as shown on figure II.3. The flow enters through a hot inlet into a cold environment and exits as indicated on the same figure. This geometry can be considered as a very rough approximation of the cold branch and the downcomer of the vessel in a nuclear pressurized water reactor. The effect of temperature on the fluid density is not taken into account in this first example.

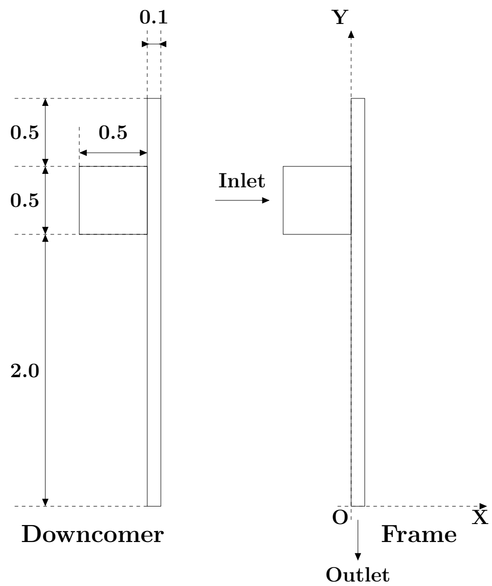


Figure II.3: Geometry of the downcomer

1.4 Characteristics

Characteristics of the geometry and the flow:

Height of downcomer	$H = 3.00\ m$
Thickness of downcomer	$E_d = 0.10\ m$
Diameter of the cold branch	$D_b = 0.50\ m$
Inlet velocity of fluid	$V = 1\ m.s^{-1}$

Table II.1: Characteristics of the geometry

Physical characteristics of fluid:

The initial water temperature in the domain is equal to 20°C. The inlet temperature of water in the cold branch is 300°C. Water characteristics are considered constant and their values taken at 300°C and $150 \times 10^5\ Pa$:

- Density: $\rho = 725.735\ kg.m^{-3}$
- Dynamic viscosity: $\mu = 0.895 \times 10^{-4}\ kg.m^{-1}.s^{-1} = 8.951 \times 10^{-5}\ Pa.s$
- Specific heat: $C_p = 5\,483\ J.kg^{-1}.K^{-1}$
- Thermal conductivity $= 0.02495\ W.m^{-1}.K^{-1}$

1.5 Mesh characteristics

Figure II.4 shows a global view of the downcomer mesh. This two-dimensional mesh is composed of 700 cells, which is very small compared to those used in real studies. This is a deliberate choice so that tutorial calculations run fast.

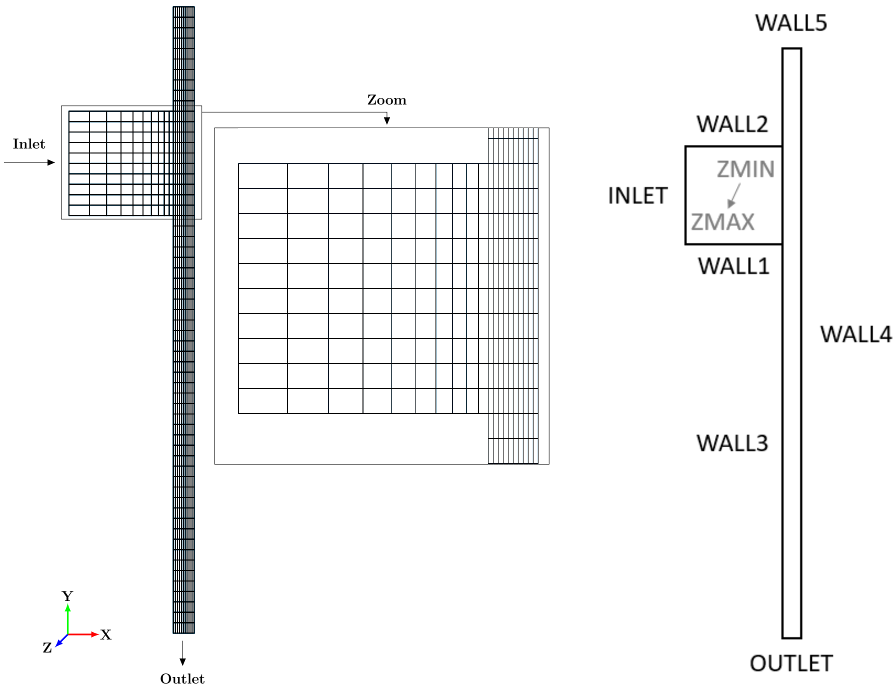


Figure II.4: Mesh and groups of boundary faces

Note that here the case is two-dimensional but code_saturne always operates on three-dimensional mesh elements (cells). The present mesh is composed of a layer of hexahedrons created from the 2D mesh shown on figure II.4 by extrusion (elevation) in the z direction. The virtual planes parallel to Oxy will have **slipping** (**symmetry**) conditions to account for the two-dimensional character of the configuration.

Type: structured mesh

Coordinates system: cartesian, origin on the edge of the main pipe at the outlet level, on the nozzle side (figure II.3)

Group definition: see figure II.4. To specify boundary conditions on the boundary faces of the mesh, the **groups** have to be identified. To do so, a name is first assign to a group of boundary faces during the definition of the geometry. This group is reused during the generation of the mesh to associate the list of boundary elements to the given name.

2 CASE 1: Basic calculation

2.1 Calculation options

Most of the options used in this calculation are default options of code_saturne. Some none default options are listed below:

→ Time settings: steady algorithm (local time step) (Velocity-Pressure algorithm is the SIMPLEC one)

→ Turbulence model: $k-\varepsilon$ LP (Linear Production)

→ Temperature activated with no gravity (acts like a passive scalar)

→ Physical properties: uniform and constant

2.2 Initial and boundary conditions

→ Initialization: none (default values)

The boundary conditions are defined as follows:

- **Flow inlet**: Dirichlet condition, an inlet velocity of 1 $m.s^{-1}$ and an inlet temperature of 300°C are imposed
- **Outlet**: default values
- **Walls**: default values

Figure II.4 shows the groups used for boundary conditions and table II.2 defines the correspondance between the group names and the type of boundary condition to use.

Do not forget to enter the value of the hydraulic diameter, adapted to the current inlet (used for turbulence entry conditions).

Group name	**Conditions**
INLET	Inlet
OUTLET	Outlet
WALL1 WALL2 WALL3 WALL4 WALL5	Wall
ZMIN ZMAX	Symmetry

Table II.2: Boundary conditions and associated groups

2.3 Parameters

All parameters necessary to this study can be defined through the Graphical Interface without using any user Fortran files. They are specified in the following table:

Calculation control parameters	
Pressure-Velocity coupling	SIMPLEC algorithm
Number of iterations	300
Reference time step	0.1
Maximal CFL number	1.0
Output period for post-processing files	1

2.4 Results

Figure II.5 presents the results obtained at different iterations in the calculation. They were plotted from the post-processing files, with ParaView.

Note: since the **steady flow** option has been chosen, the evolution of the flow iteration after iteration has no physical meaning. It is merely an indication of the rapidity of convergence towards the (physical) steady state.

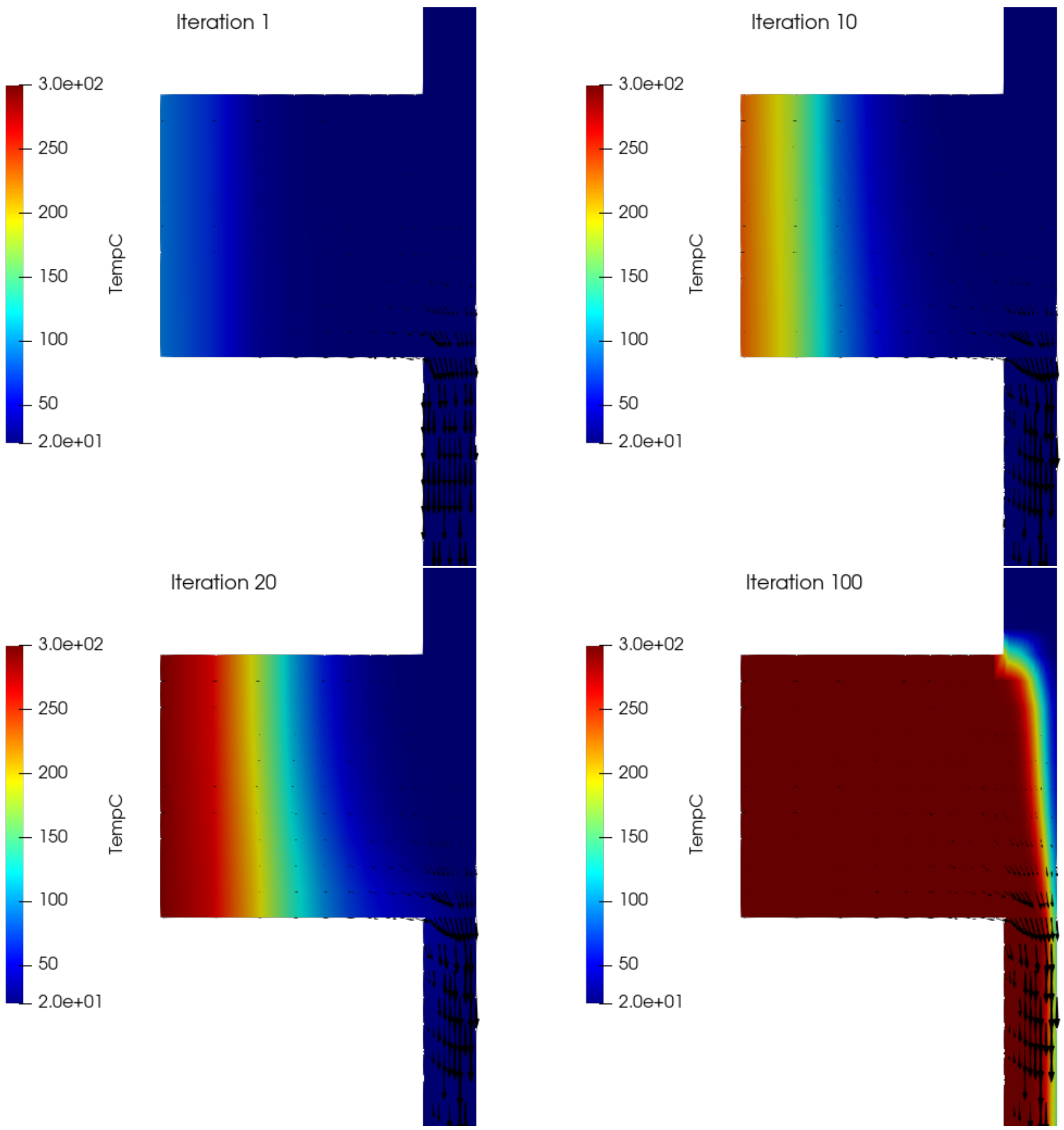


Figure II.5: Water velocity field colored by temperature at different iterations

Part III

Step by step solution

1 Solution for CASE1

The first thing to do is to prepare the computation directories. Here, the study directory 🗁`simple_junction` will contain a single calculation directory 🗁`case1`.

Create the study 🗁`simple_junction` and the 🗁`case1` using one of the three methods explained in part 2 of this document.

In this case we use the code_saturne Graphical User Interface (GUI) method.

code_saturne Graphical User Interface (GUI)can be launched by typing the command lines as below:

```
$ cd simple_junction/case1/DATA
$ ./code_saturne gui &
```

And the following window opens (fig III.1).

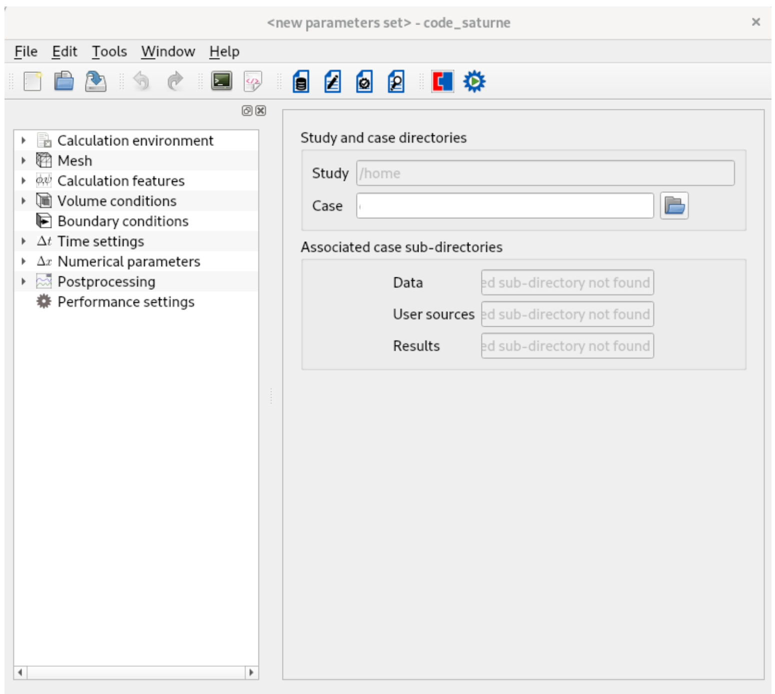


Figure III.1: code_saturne (GUI) graphic window

The mesh files should be copied in the directory 🗁`MESH/`, by the command line as follows or by your favorite explorer:

```
$ cd simple_junction/MESH/
$ cp .../examples/1-simple_junction/mesh/downcomer.med .
```

Go to the File menu and click on **New file** to open a new calculation data file. The interface automatically updates the following information:

- Study name
- Case name
- Directory of the case
- Associated sub-directories of the case

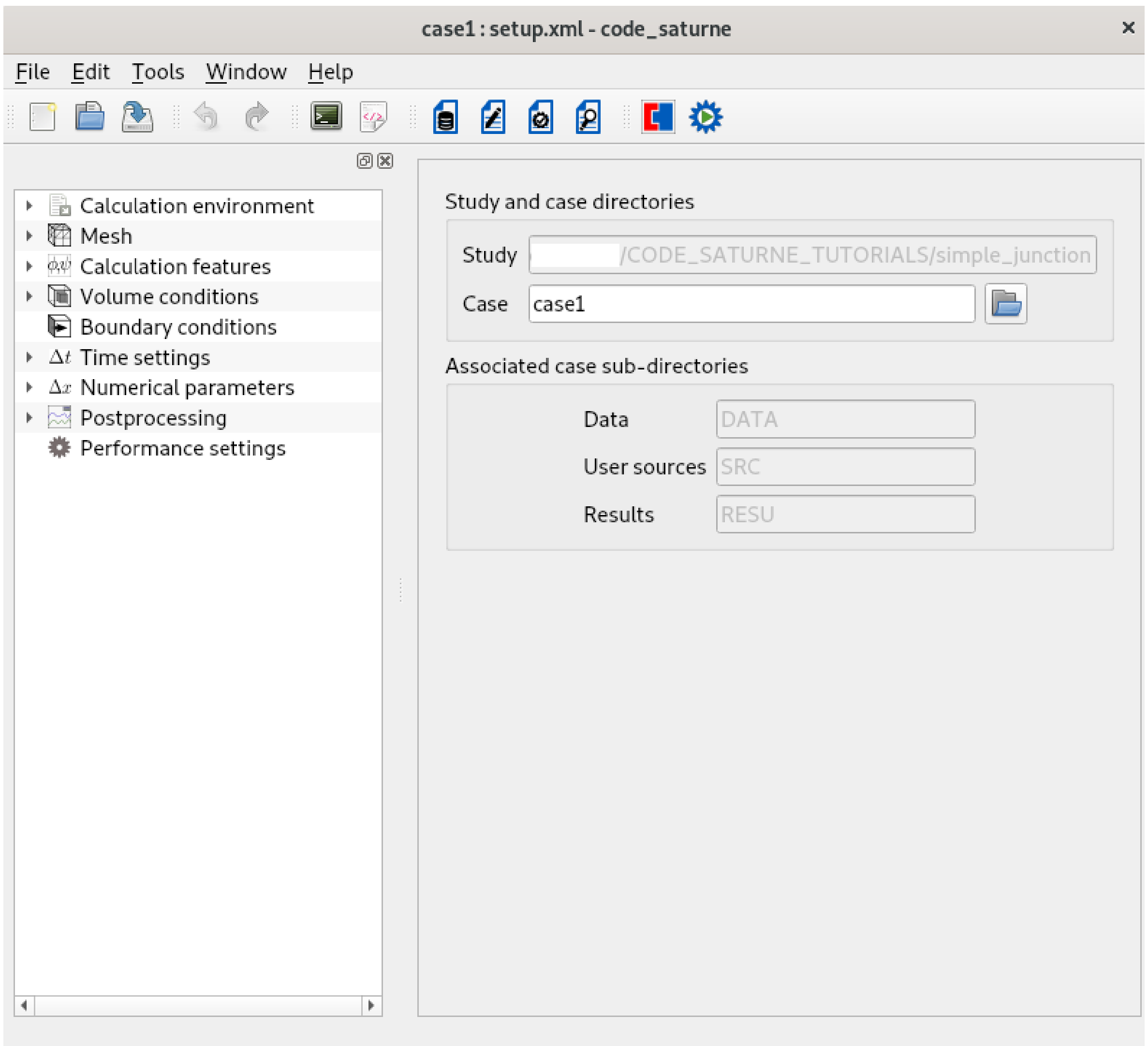


Figure III.2: Identity and paths

Don't forget to regulary save your work by clicking on File › Save.

1.1 Mesh tab

The next step is to specify the mesh(es) to be used for the calculation. Click on the **Mesh** heading. Select **Import meshes**. Then click on [+] to add meshes.

The list of meshes appears in the window **List of meshes**. In this case only the mesh `downcomer.med` is needed.

The **Periodic Boundaries** is not used in this case so **Preprocessing** page does not need to be visited. Keep the default values.

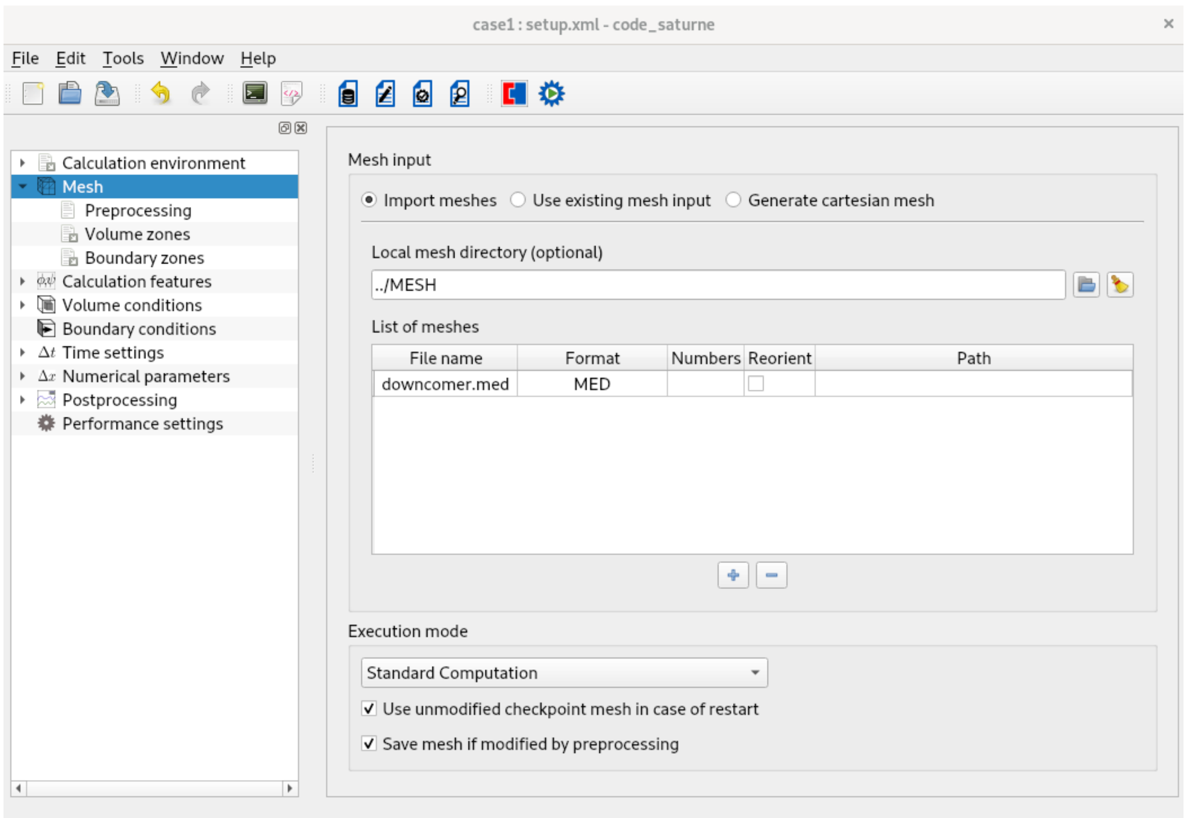


Figure III.3: Meshes: list of meshes

Preprocessing By default, the execution mode is set to standard computation i.e. a flow computation. It can be set in the **Mesh** menu.

Several other execution modes are available. They allow to perform operations linked to the mesh:

- Import mesh only: code_saturne reads the specified mesh files, convert them to code_saturne internal format and save them in a mesh_input with this format.
- Mesh preprocessing only: code_saturne imports the mesh and performs preprocessing tasks (joining, boundary insertion, extrusion, boundary layer meshing, ...) specified in the GUI or in user source file cs_user_mesh.
- Mesh quality criteria only: code_saturne imports the mesh, performs preprocessing tasks and computes quality criteria of the resulting mesh.

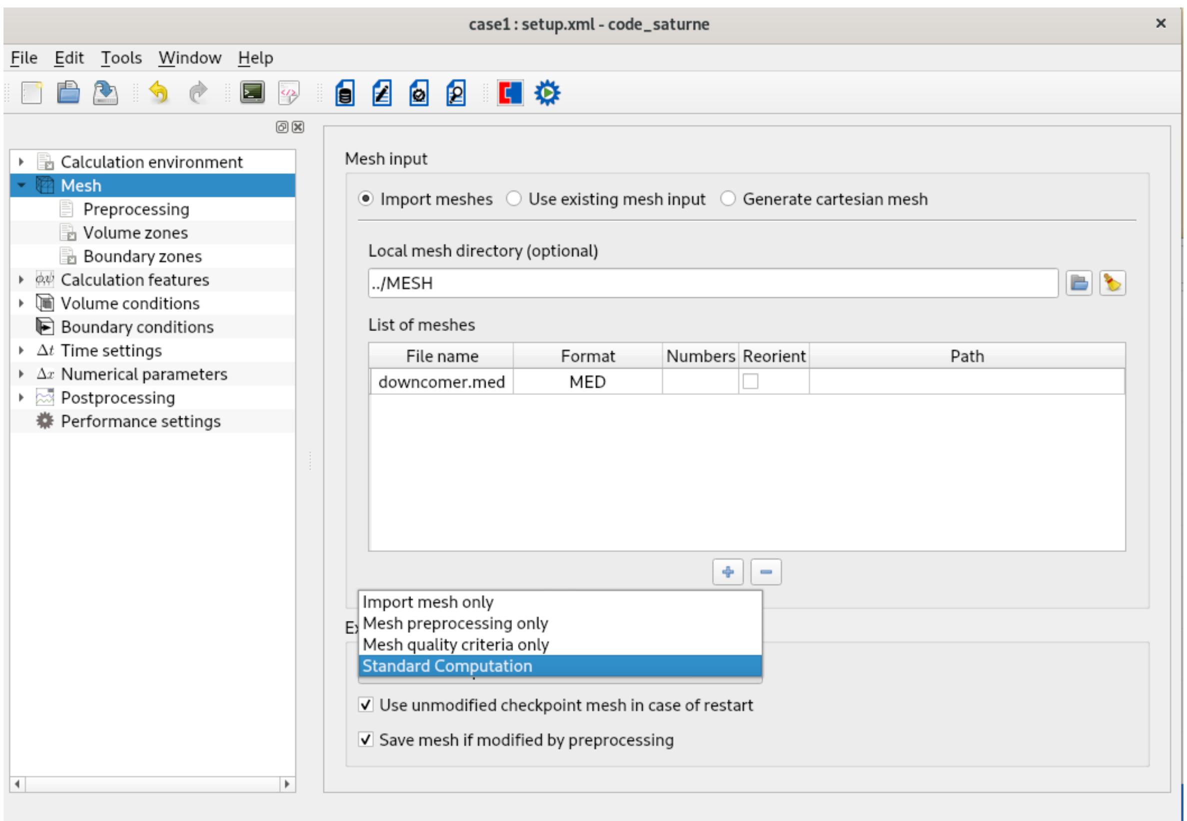


Figure III.4: Preprocessing and calculation modes in code_saturne

Note: If you need to run one of this exectution modes, you just need to select the one you need then to click on icon in the menu bar. You will learn more about this modes toward all code_saturne tutorials.

For this case, select Standard Computation.

Boundary zones Boundary conditions now need to be defined. Go to the **Boundary zones** under Mesh heading. The following window opens (fig III.5).

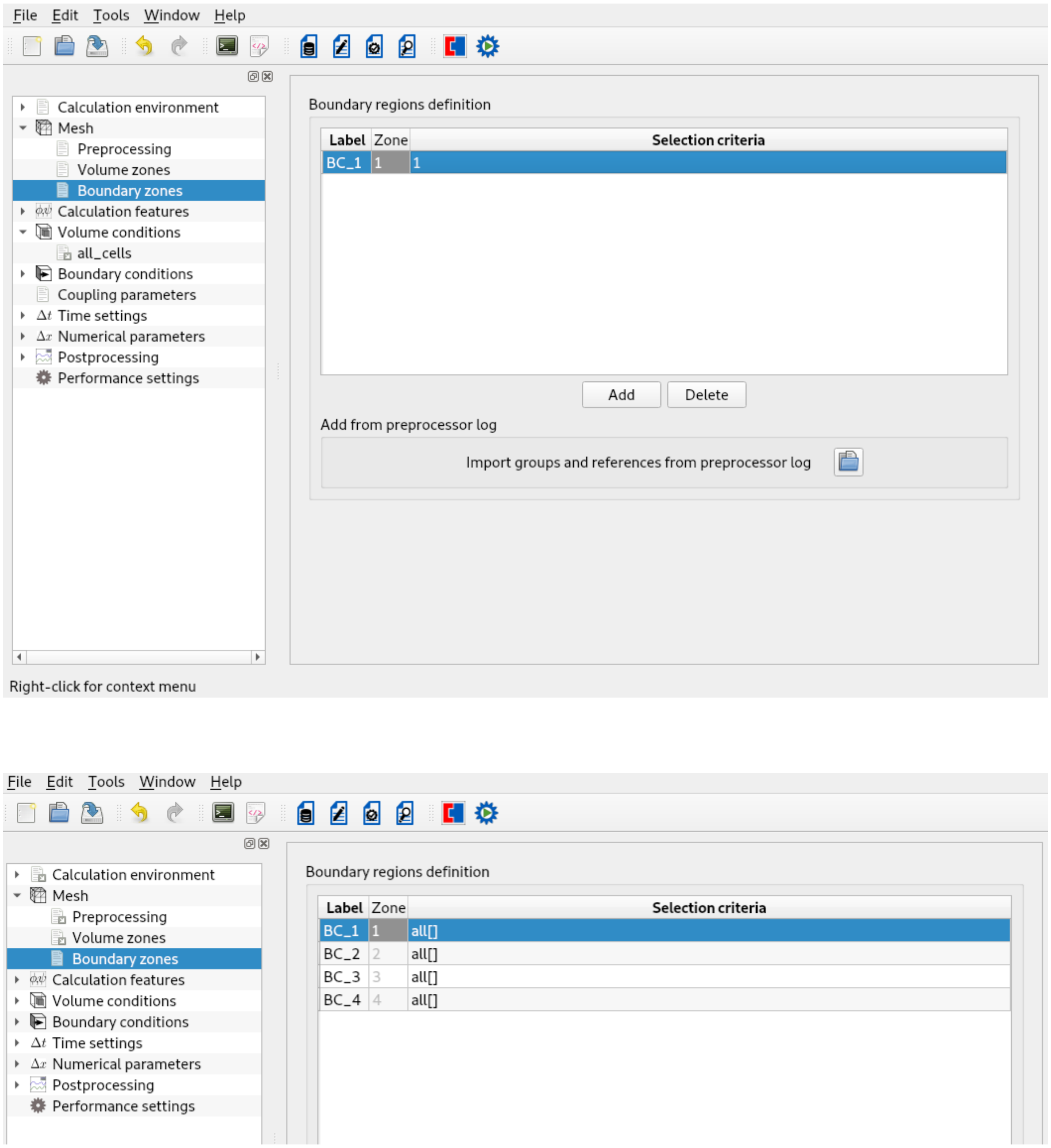


Figure III.5: Creation of a boundary region

Each boundary must be defined. Click on Add to edit a new boundary. The boundary faces will be grouped in user-defined zones, based on the name of the group or on geometrical conditions. For each zone, a label and a selection criteria must be assigned by double clicking on the field you wish to set.

The **Label** can be any character string. It is used to identify the zone more easily. It usually corresponds to the nature of the zone.

The **Selection criteria** is used to define the faces that belong to the zone. It can be a group name, geometrical conditions, or a combination of them, related by **or** or **and** keywords.

Note: A zone number is used by the code to identify each zone.

The table III.6 is a short boundary conditions reminder for our case. Set boundary regions as follows.

Label	**Inlet**	**Outlet**	**Symmetry**	**Wall**
Selection criteria	INLET	OUTLET	ZMIN or ZMAX	WALL1 or WALL2 or WALL3 or WALL4 or WALL5

Figure III.6: Boundary conditions

It is usually faster to regroup the different groups in one single zone, as shown on figure III.7. For instance, the localization of the `Symmetry` zone is the string `''ZMIN or ZMAX''`. The same treatment is done for the wall conditions.

After defining all the boundary zones, the Interface window will look as in figure III.7.

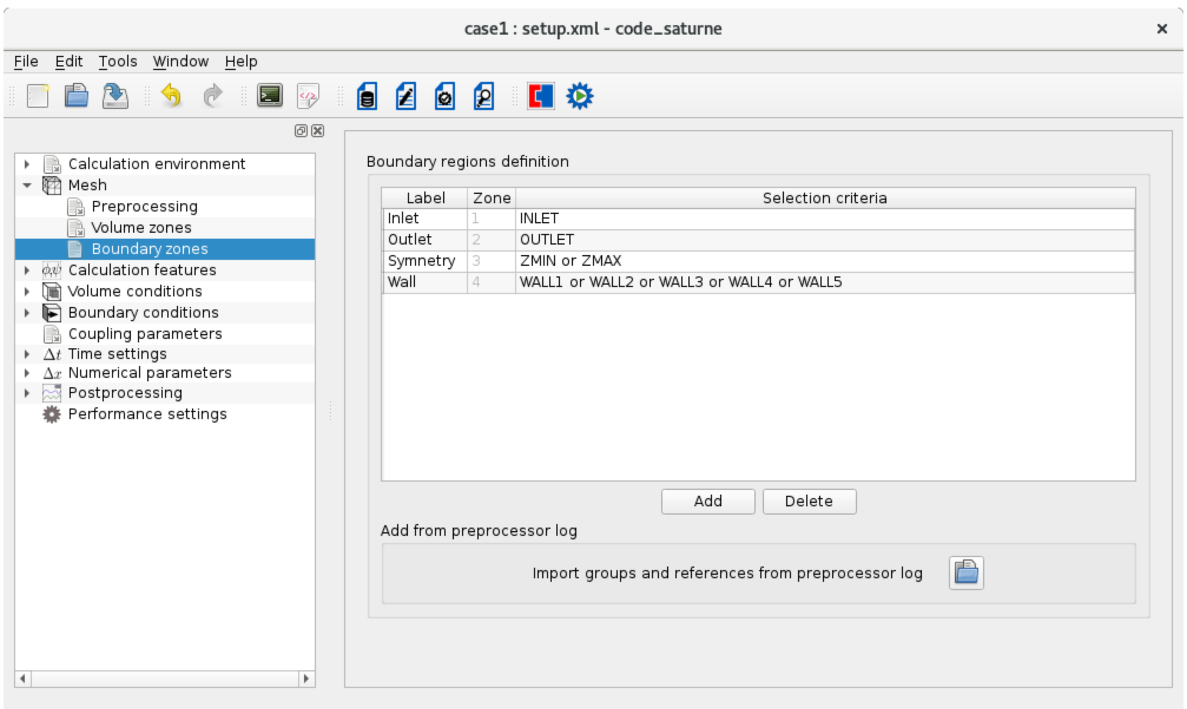


Figure III.7: Boundary zones label renamed

Remember to save the `xml` file regularly!

1.2 Calculation features tab

The **Calculation features** menu allows to choose the flow model. In this case, all default values are left unchanged, i.e. we choose to simulate an incompressible single phase with an eulerian approch.

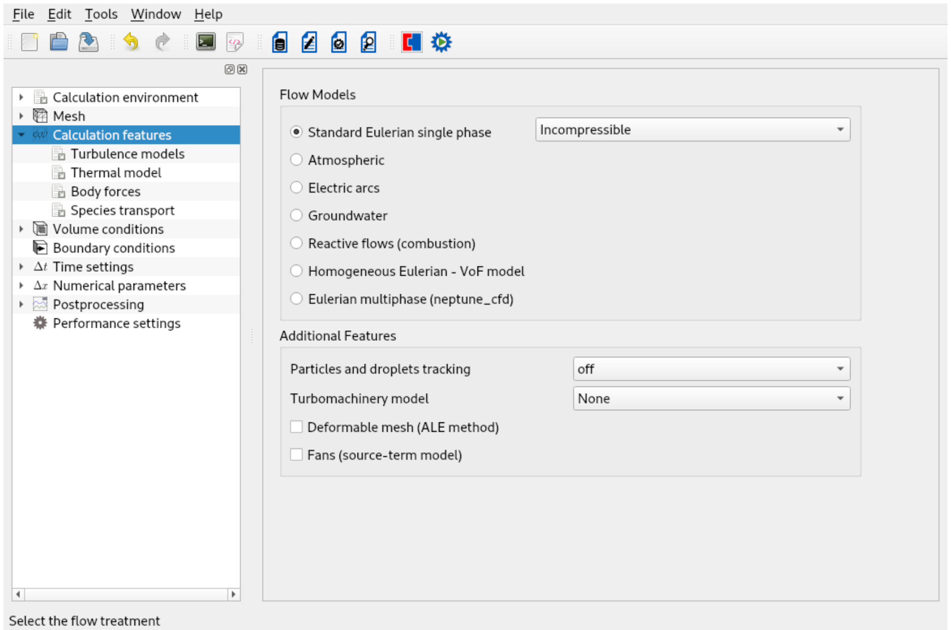


Figure III.8: Flow modelling

Turbulence Model Now, let's choose a turbulence model for our simulation. To do so, go **Turbulence models** sub-folder and open **Turbulence model** drop-down menu.

In this case, the k-ε linear production model is used. Here, you can also specify a turbulence level based on a reference velocity. Leave the default values unchanged ($\mathbf{1}\ m.s^{-1}$).

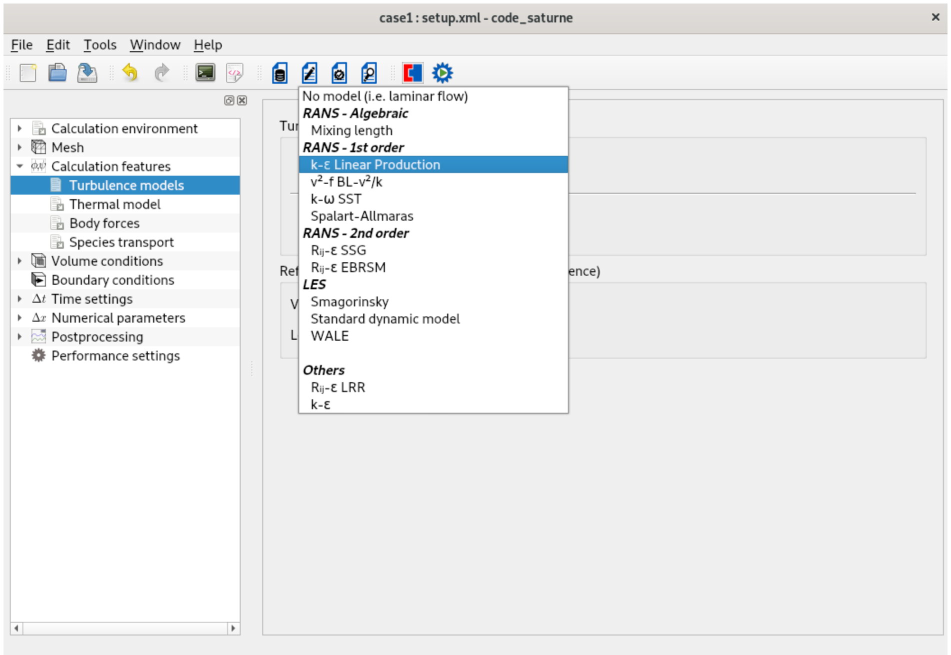


Figure III.9: Turbulence model: list of models

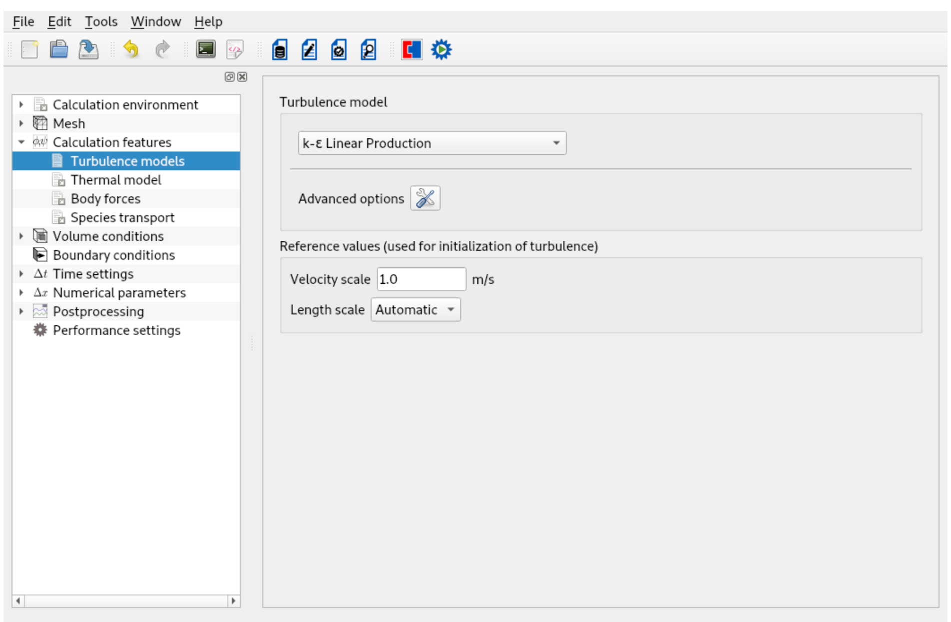


Figure III.10: Turbulence model: choice of a model

Thermal Model For this study the equation for temperature must be solved. Click on the **Thermal model** item to choose between:

```
- No thermal scalar
- Temperature (Celsius)
- Temperature (Kelvin)
- Enthalpy (J/kg)
```

In the present case, select **Temperature (Celsius)**.

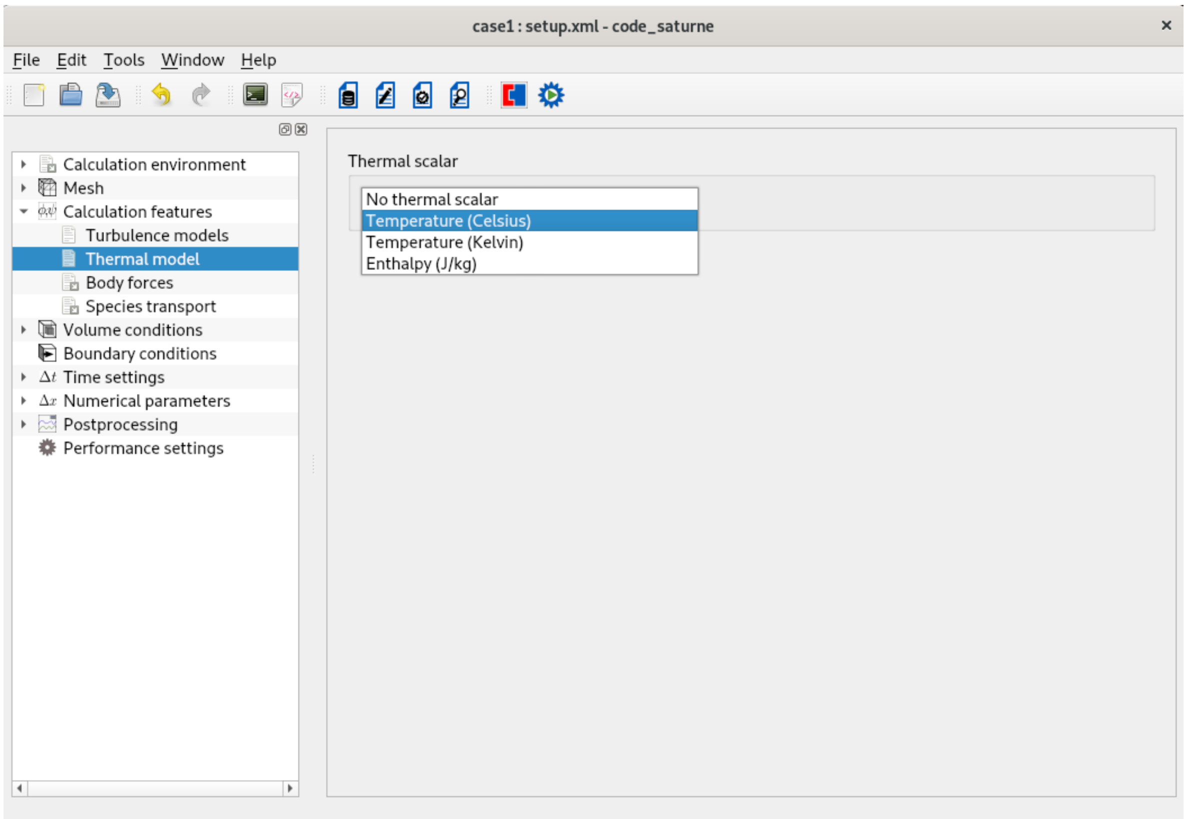


Figure III.11: Thermal scalar conservation: list of models

Once the thermal scalar selected, additional items appear. There are no radiative transfers in our case, so this item can be ignored.

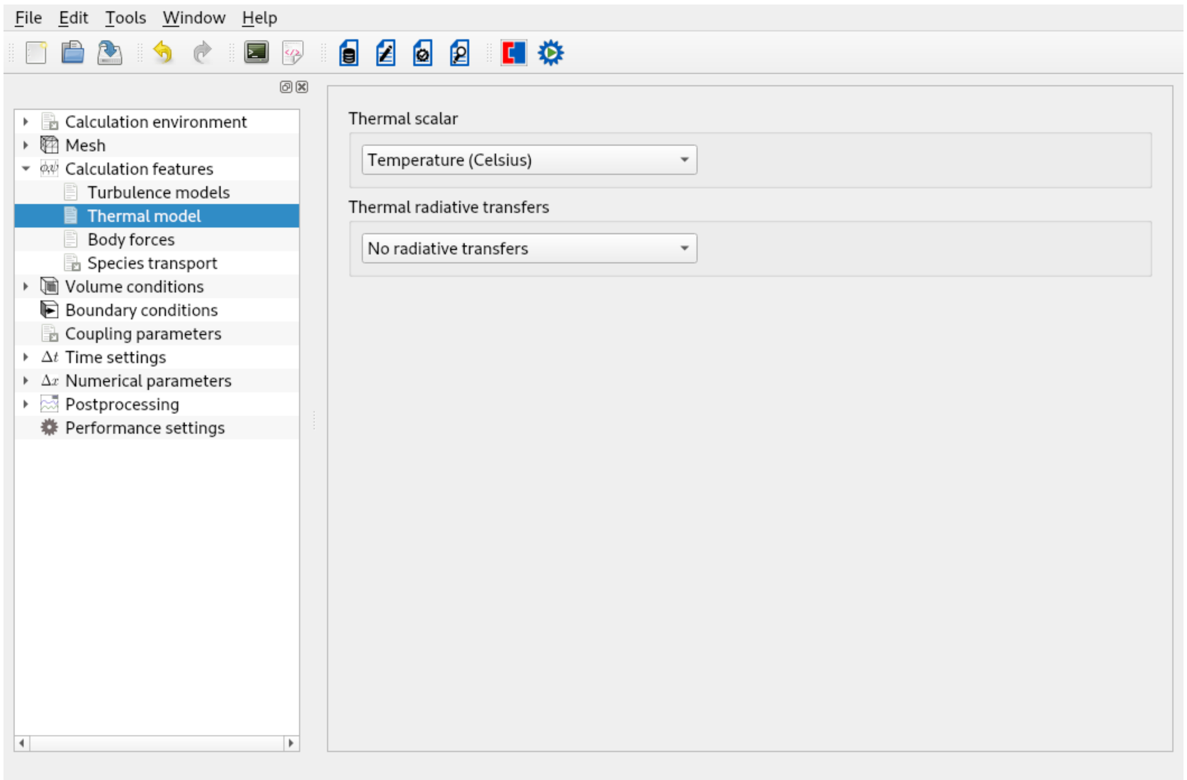


Figure III.12: Thermal scalar conservation: choice of a model

Body forces In **Body forces** heading set the three components of gravity in the **Gravity** item. In this case, since the gravity doesn't have any influence on the flow, gravity can be set to **0**. Same thing for the **Coriolis source terms (rotation vector)**.

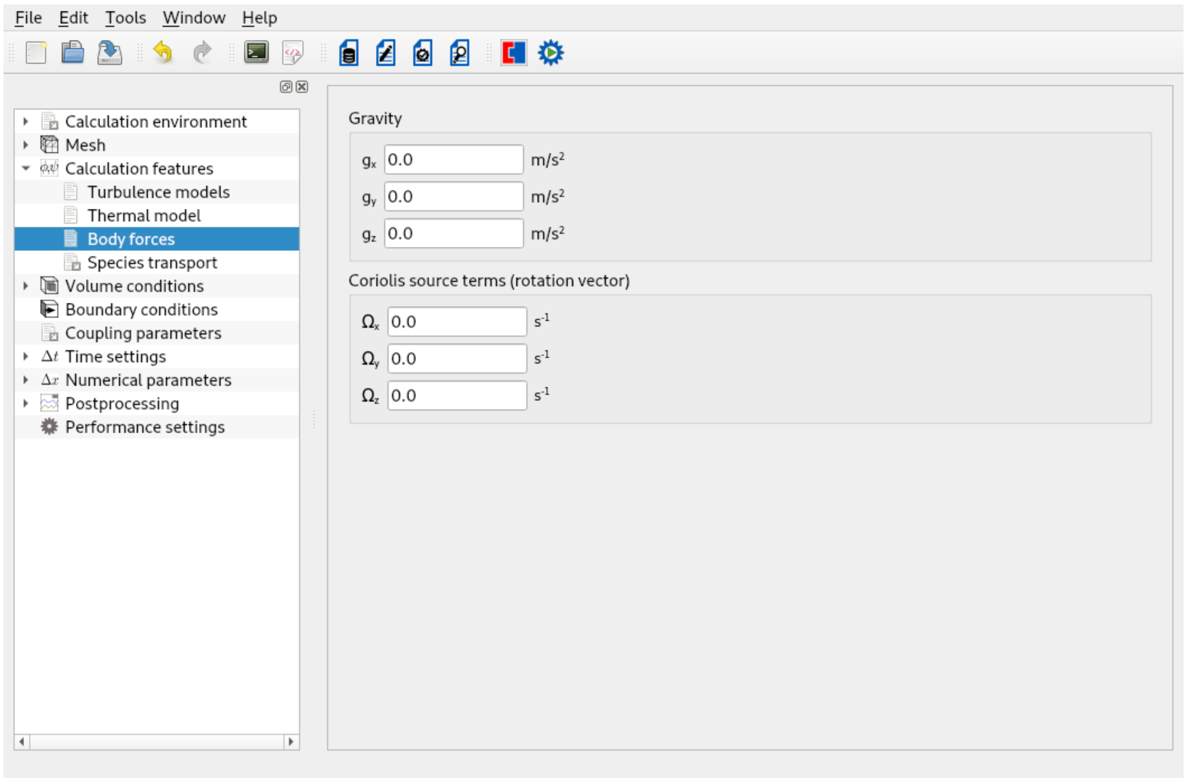


Figure III.13: Body forces

1.3 Volume conditions tab

Initialization To initialize variables at the instant $t = 0$ (s), you first need to tick **Initialization** under **Volume conditions** heading then you can select the **Initialization** tab located in all cells **all cells**. See III.14 .

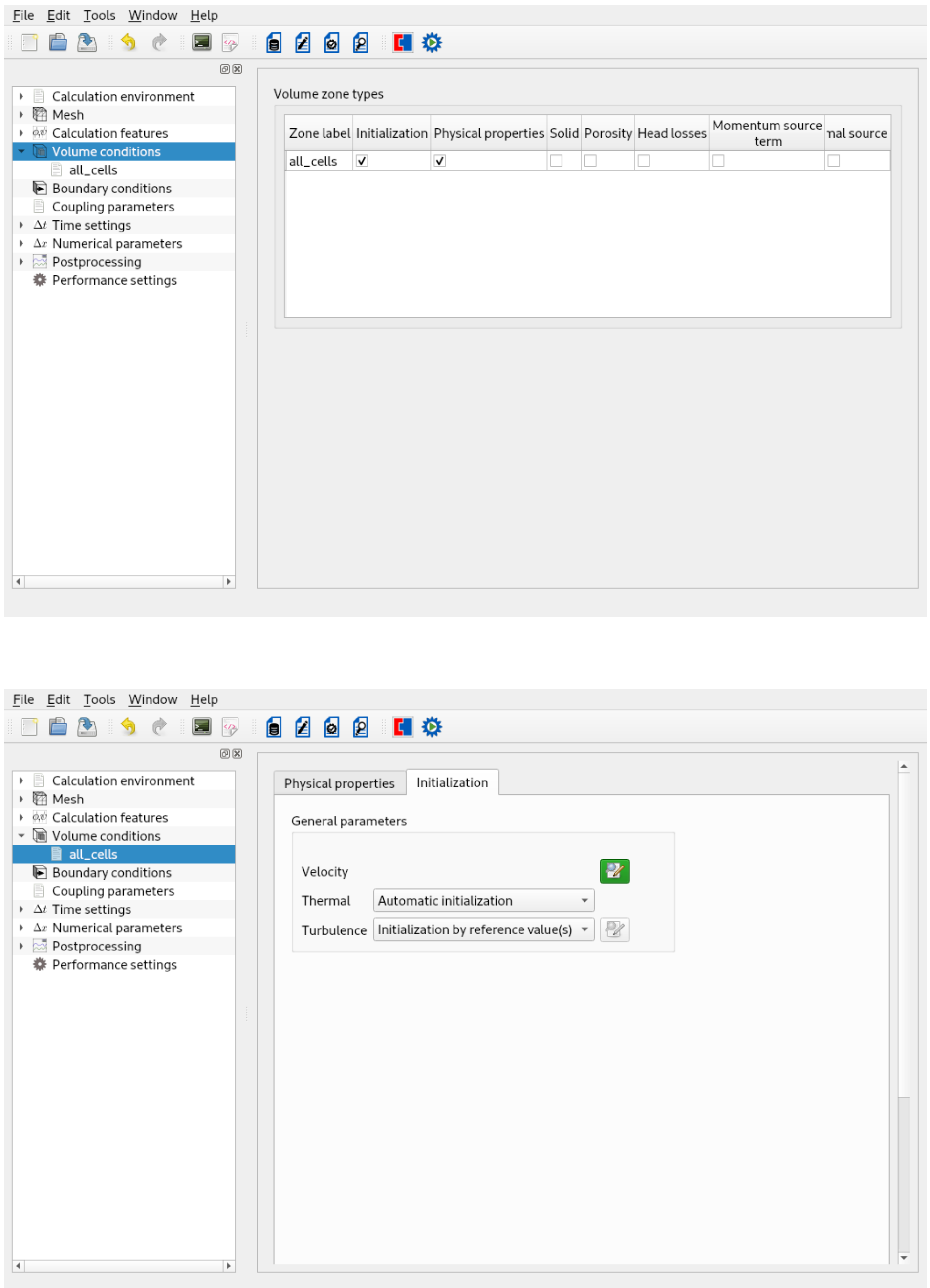


Figure III.14: Volume conditions and Initialization

Velocity, thermal scalar and the turbulence can be here initialized. In this case, the values te be set are: zero velocity (default) and an initial temperature of **20**°C. Specific zones can be defined with different initializations. In this case, only the default **all cells** is used.

- Click on **Thermal**, select **Initialization by formula** and click on the opposite icon to specify the initial value of the thermal scalar. It can be a value or a user expression.

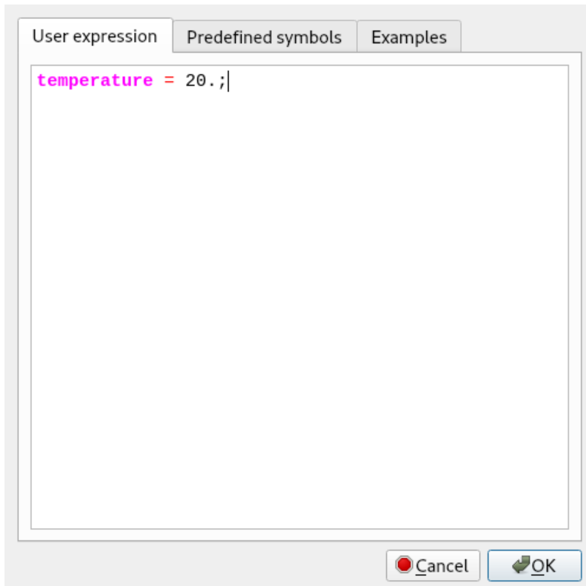


Figure III.15: Initialization of the scalar

- To initialize the velocity, click also on the icon near **Velocity**.

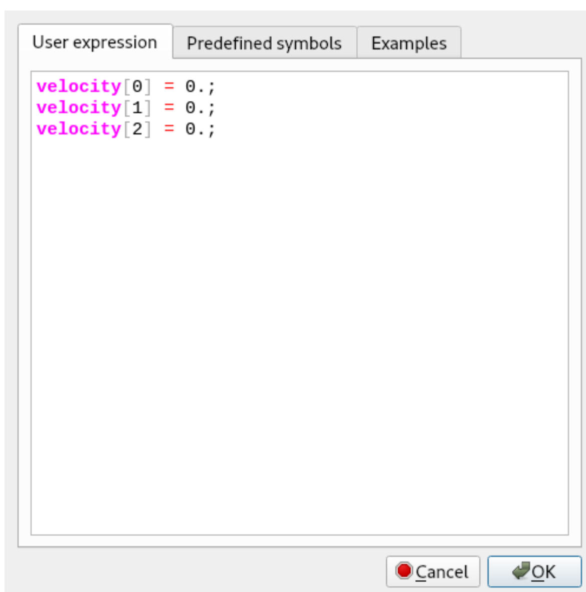
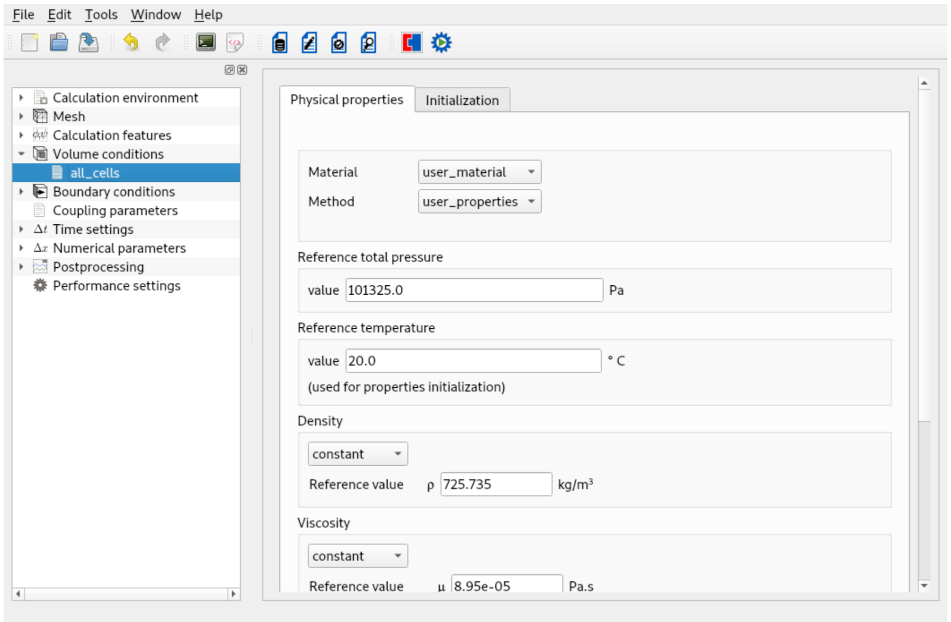


Figure III.16: Initialization of the velocity

Physical properties Under the heading **Volume condtions** we can also specify reference values of some physical quantities and the physical properties of the fluid in **Physical properties** tab.

Use the default value of **101 325** (Pa) for the pressure and **20** (°C) for the temperature.



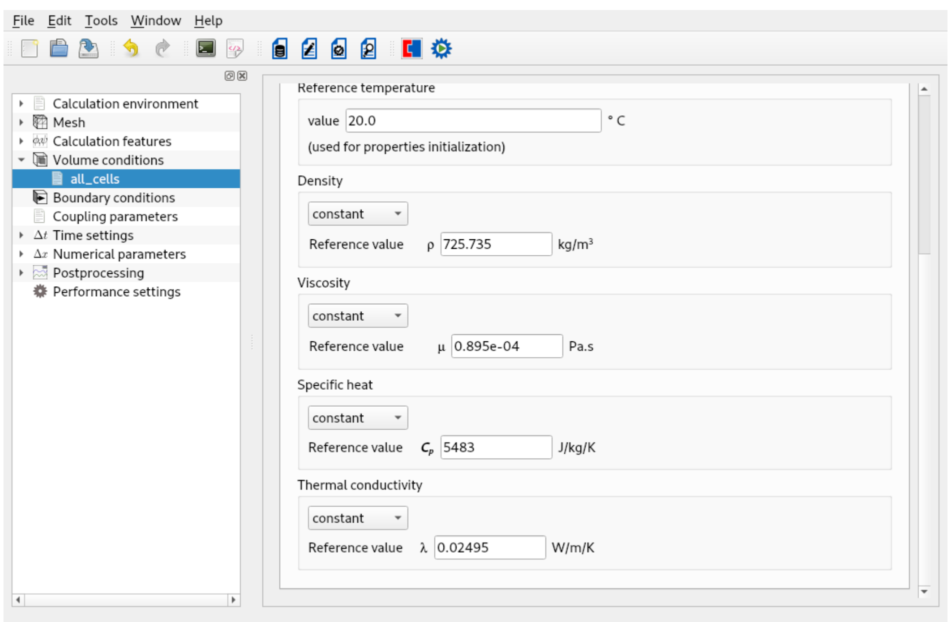


Figure III.17: Physical and fluid properties

Specify the fluid physical characteristics in the **Fluid properties** item:

- Density
- Viscosity
- Specific Heat
- Thermal Conductivity

In this case they are all constant.

- $\rho \quad = 725.735\ kg.m^{-3}$
- $\mu \quad = 0.895 \times 10^{-4}\ kg.m^{-1}.s^{-1}$
- $C_p \quad = 5\,483\ J.kg^{-1}.K^{-1}$
- $\lambda \quad = 0.02495\ W.m^{-1}.K^{-1}$

1.4 Boundary conditions tab

All boundary zones were defined in the mesh section but not their nature. To do it click on the **Boundary conditions** sub-folder to first set the **Nature** then start setting inlet boundary conditions for velocity, turbulence and themal scalar. The different natures that can be assigned are:

```
- Wall
- Free inlet/outlet
- Inlet
- Symmetry
- Outlet
- Imposed P Outlet
```

As shown on figure III.18, outlet and wall boundary zones also appear in the window. The thermal

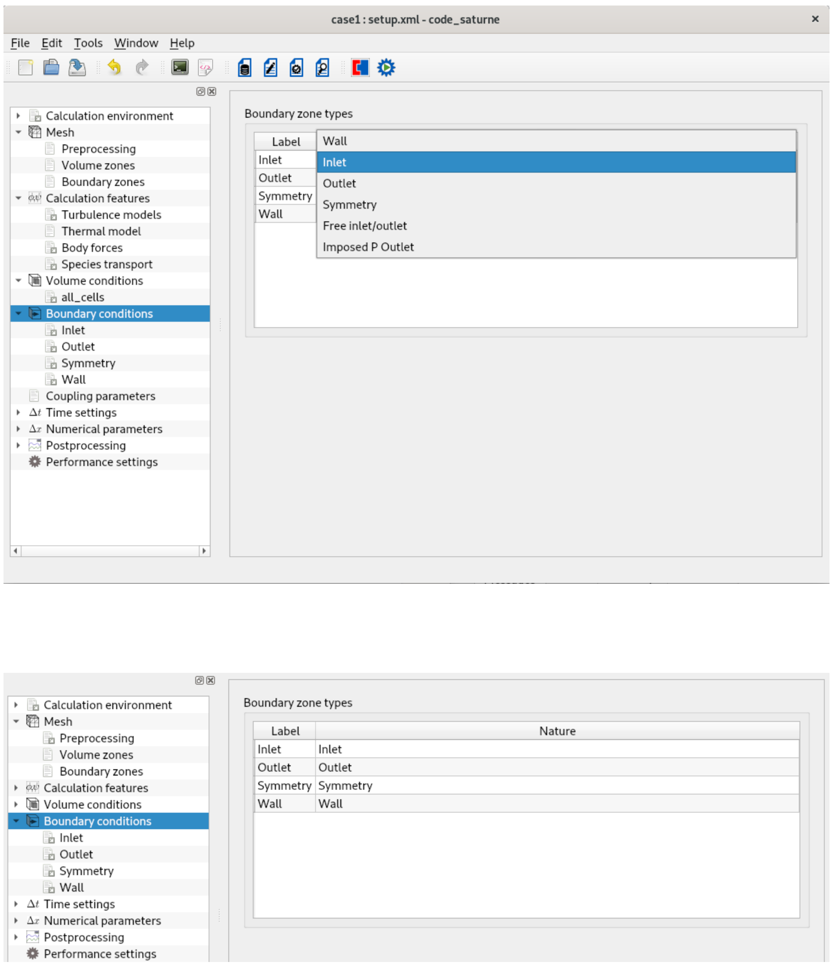


Figure III.18: Boundary conditions

boundary condition are only applied on inlets, outlets and walls.

- For the inlets, only **Prescribed value** is available.
- For the outlet, only **Prescribed value** and **Prescribed flux** are available, but they are taken into account only when the flow re-enters from the outlet.
- Otherwise, homogeneous **Prescribed flux** is considered by code_saturne.

– Inlet:

Click on the label **Inlet**. In the section **Velocity**, select **norm**, then in the sub-section **Direction** choose **specified coordinates** and enter the normal vector components of the inlet velocity.

For the turbulence, choose the inlet condition based on a hydraulic diameter and specify it as below:

```
x = 1.0 (m) ; y = 0.0 (m) ; z = 0.0 (m)
hydraulic diameter = 0.5 (m)
```

Scroll down to choose the temperature inlet value. Here this value is **300**°C.

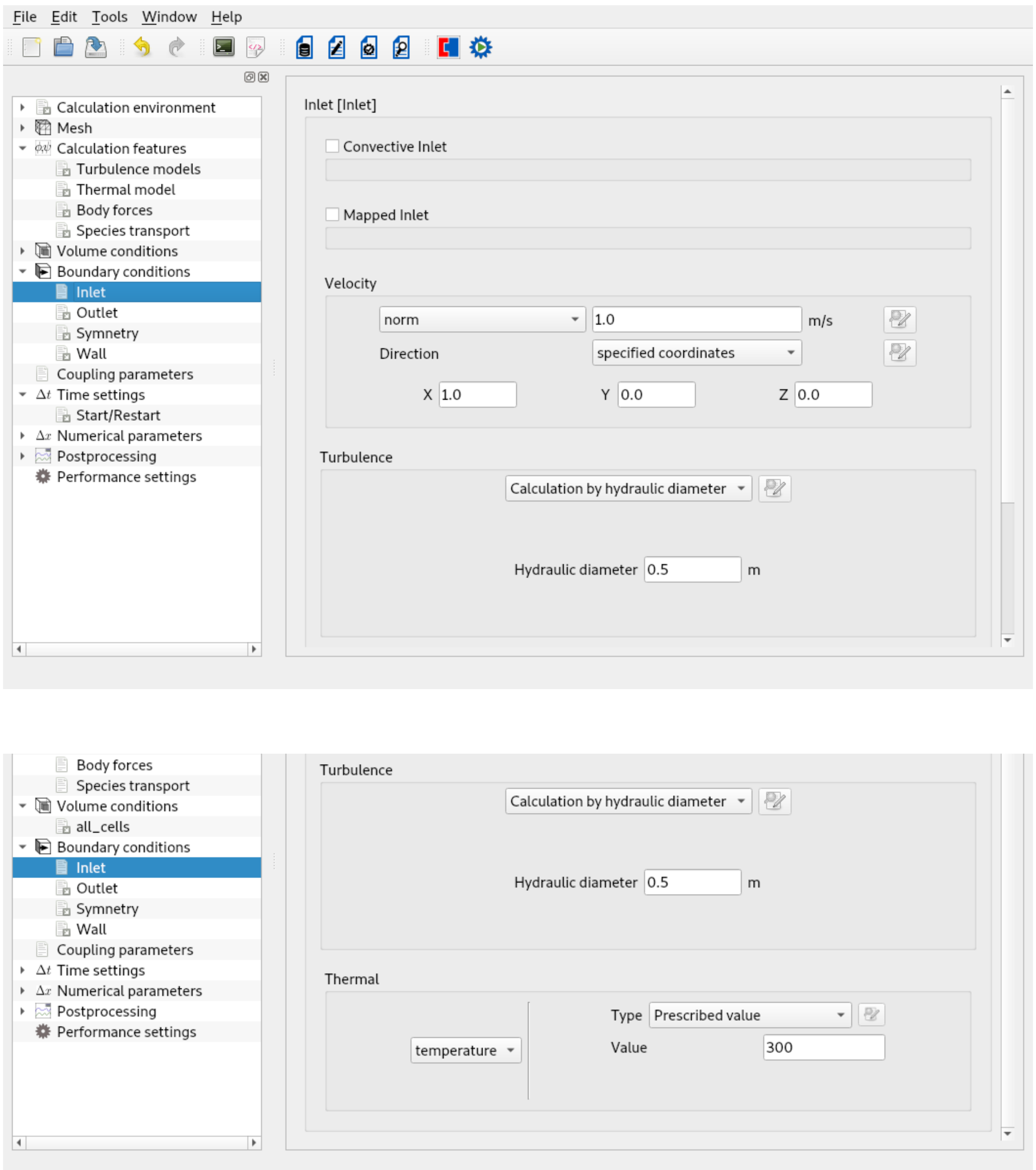


Figure III.19: Dynamic variables boundary conditions: inlet

– Wall:

As for the wall boundary zone, the specifications the user might have to give are if the wall is sliding, and if the wall is **smooth** or **rough**. In this case, the walls are fixed so the option is not selected, and the wall is considered as **smooth**.

Note that if one of the walls had been sliding, it would have been necessary to isolate the corresponding boundary faces in a specific boundary region.

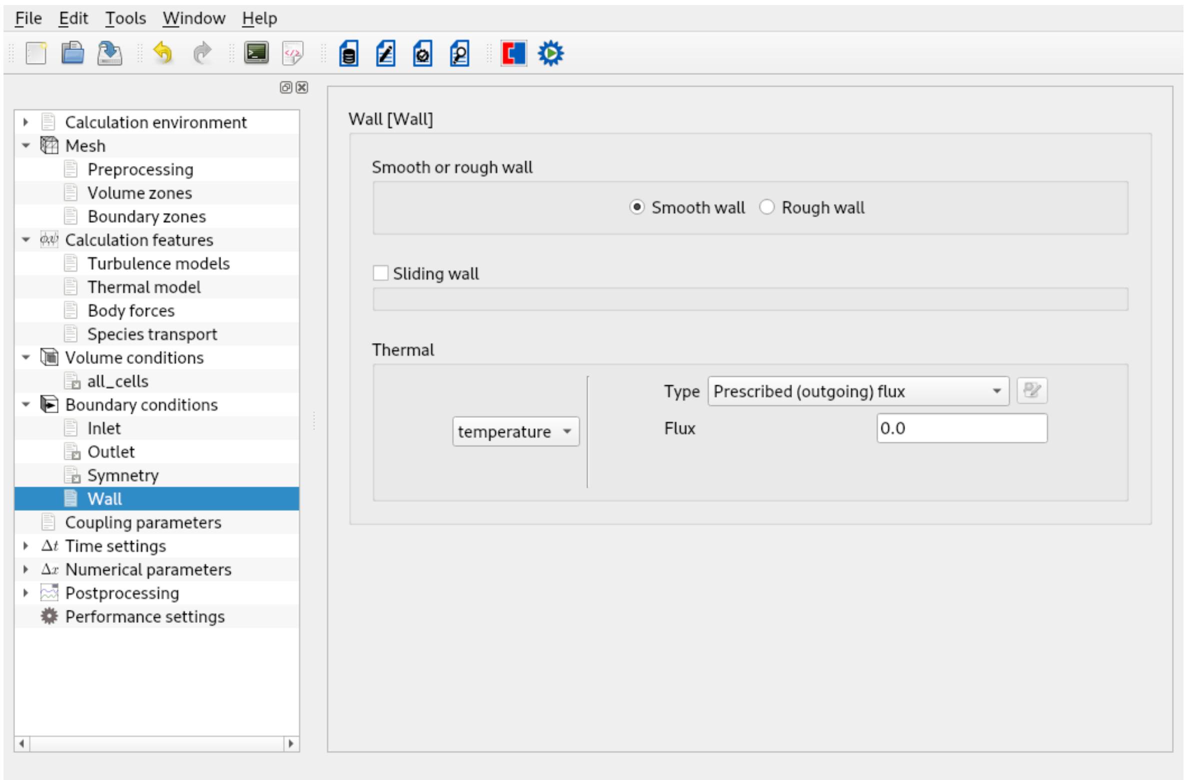


Figure III.20: Dynamic variables boundary: walls

For the walls, seven conditions are available:

```
- Prescribed value
- Prescribed value (user law)
- Prescribed (outgoing) flux
- Prescribed (outgoing) flux (user law)
- Exchange Coefficient
- Exchange Coefficient(user law)
- SYRTHES coupling
```

In this case all walls are adiabatic. So the boundary condition for the temperature will be a **Prescribed flux** set to **0**.

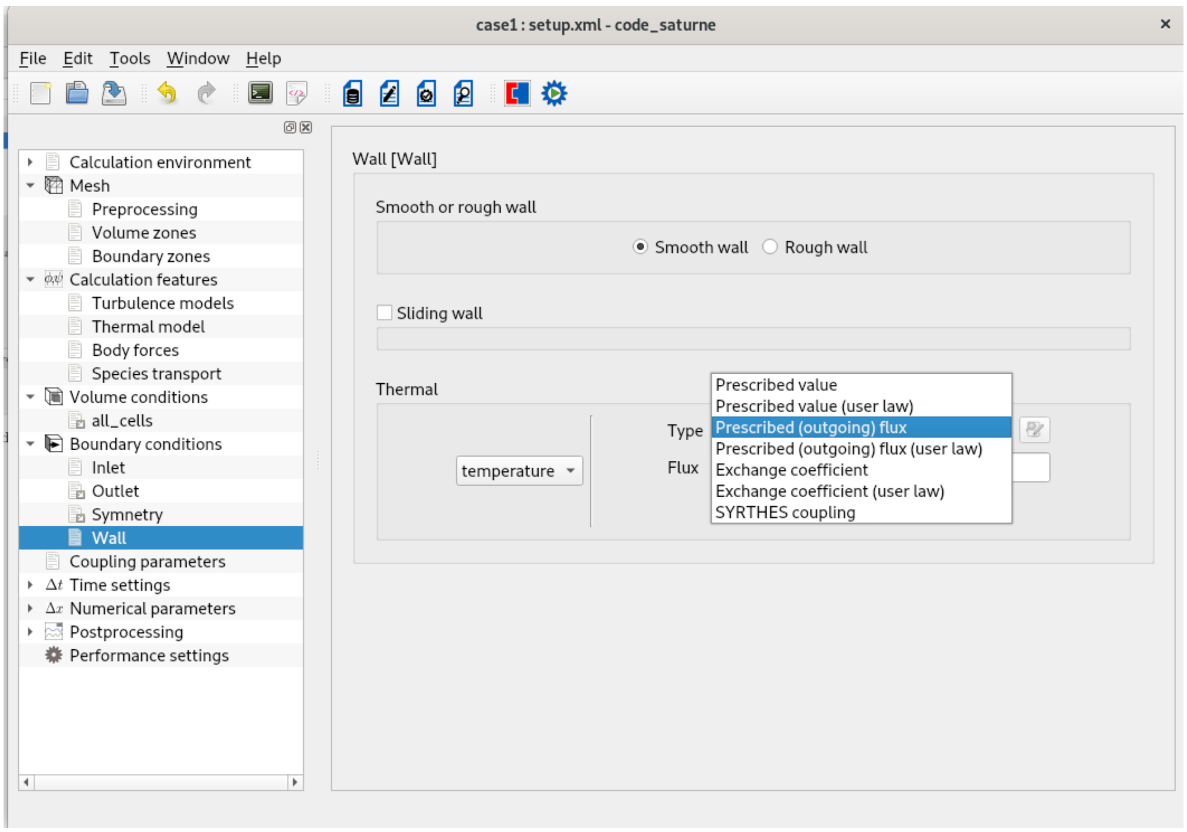


Figure III.21: Scalars boundaries: walls

1.5 Time settings tab

To specify Time settings, click on the **Time settings** header. Choose a **Steady (local step time)** as a **Time step option**. For **Velocity-Pressure algorithm** choose **SIMPLEC**. Leave all default values except the **Number of time steps**. Modify it to 300.

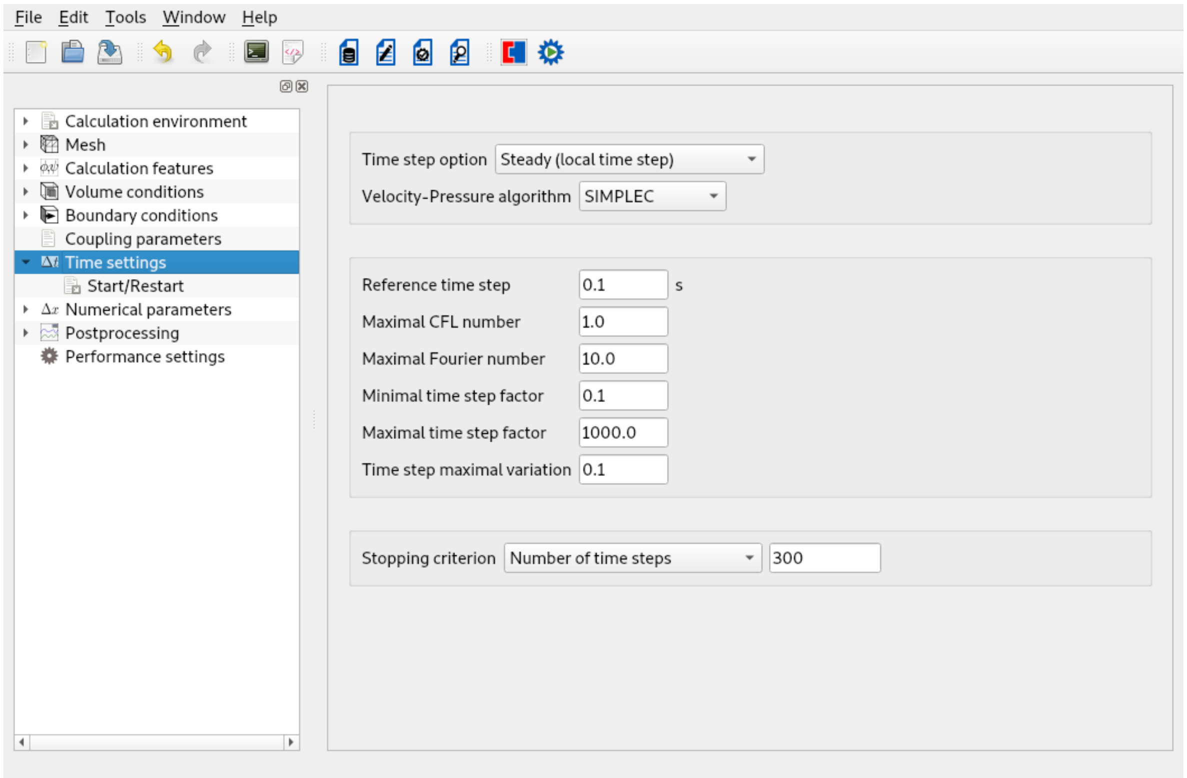


Figure III.22: Steady flow management

As mentioned earlier in this document,be aware for a Steady analyzis, the intermediate results are not significant. Only converged results should be taken as significant values.

1.6 Numerical parameters tab

The **Numerical parameters** need then to be specified, under the header **Numerical parameters**. Now, select the **Equation parameters** item under the **Numerical parameters** folder.

Scheme The tab **Scheme** allows to change different more advanced numerical parameters.

In this case none of them should be changed from their default value, see III.24.

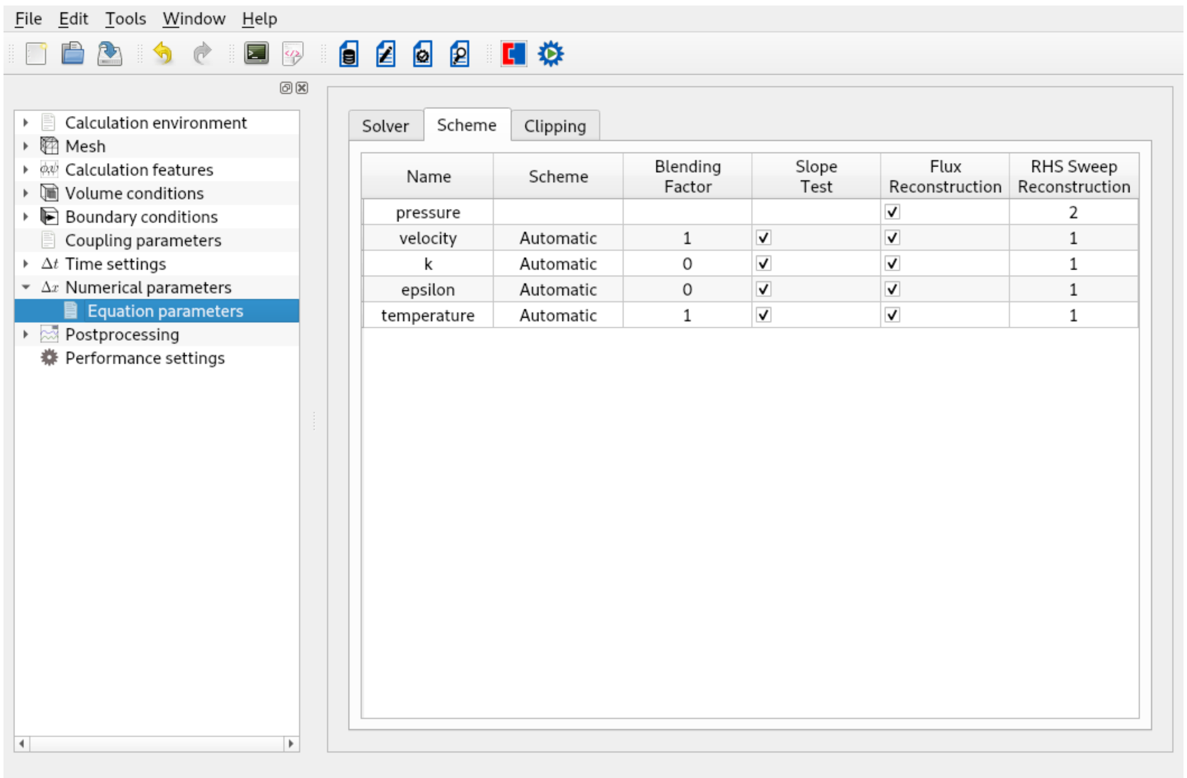


Figure III.23: Numerical parameters

Clipping The tab **Clipping** in the **Equation parameters** item allows to vanish the too small or too big value.

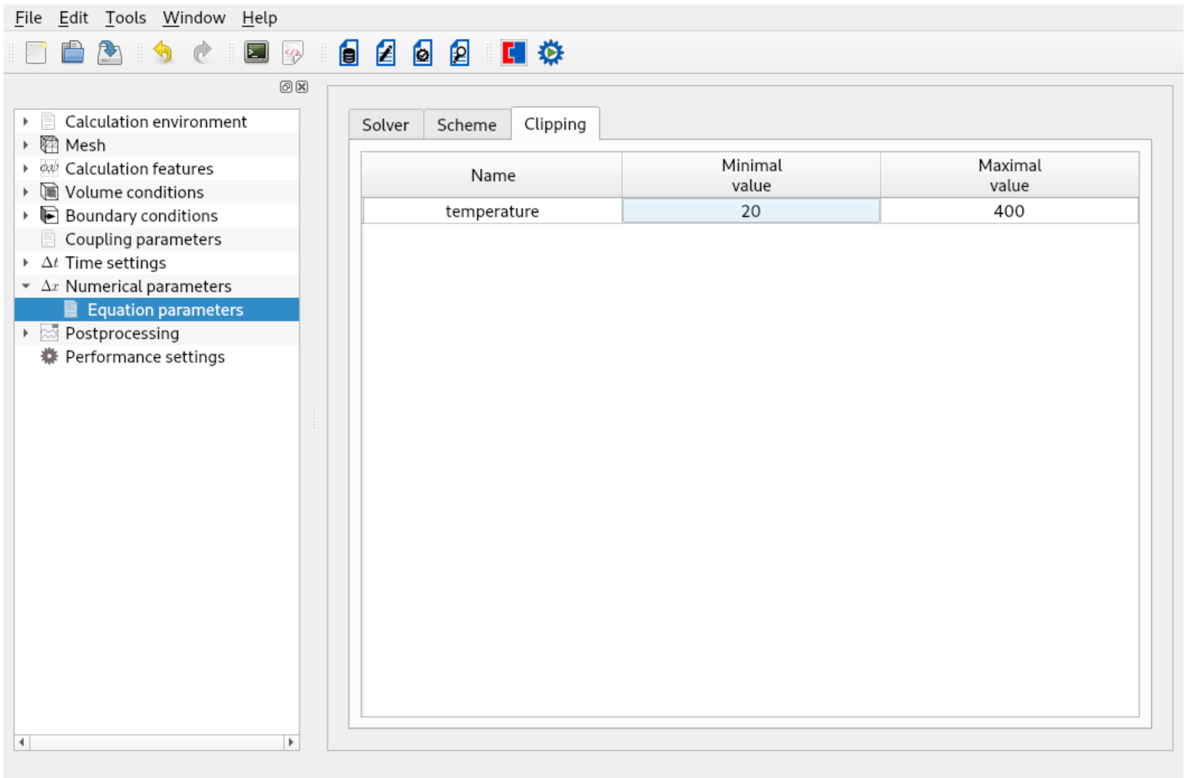


Figure III.24: Clipping

1.7 Postprocessing tab

Click on the heading **Postprocessing**. In this folder we can change the frequency for the printing of information in the output listing.

The options are:

```
- No output
- Output listing at each time step
- Output at every 'n' time step (the value of 'n' must then be specified)
```

Here and in most cases, the second option should be chosen.

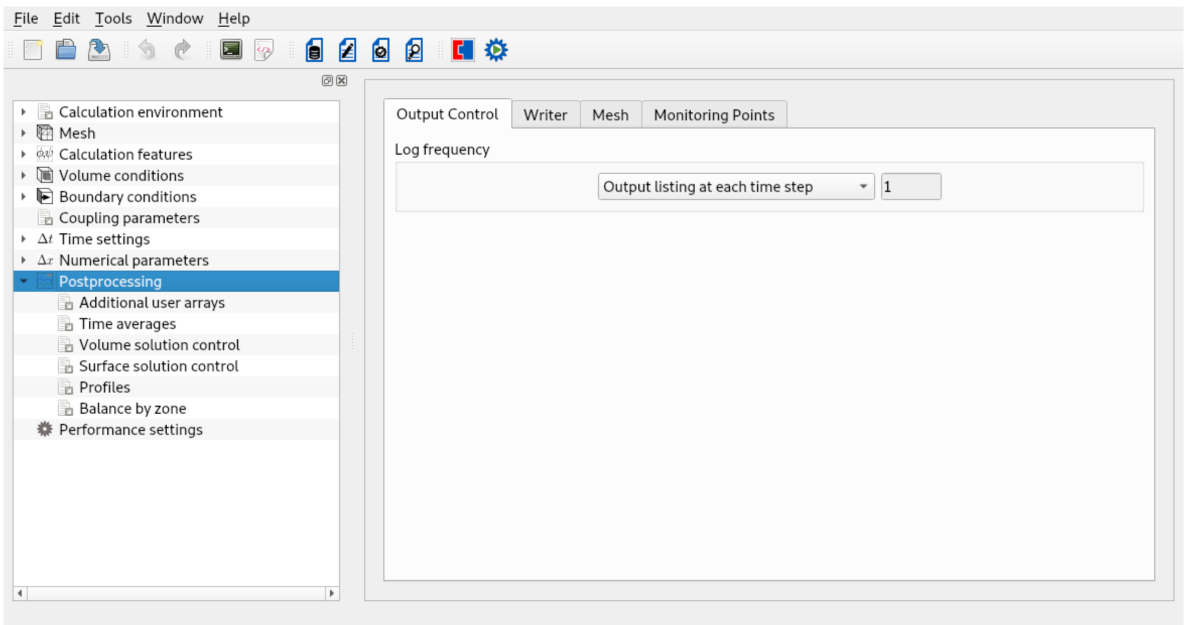


Figure III.25: Output control: output listing

For the post-processing (by default EnSight format files), there are four options:

```
- No periodic output
- Output every 'n' time step
- Output every 'x' seconds
- Output using a formula
```

In this case, we are interested in the evolution of the variables during the calculation, so the second option is chosen, with **n** set to 1.

In addition, in order to get the **Output at the end of calculation**, the corresponding box must be checked.

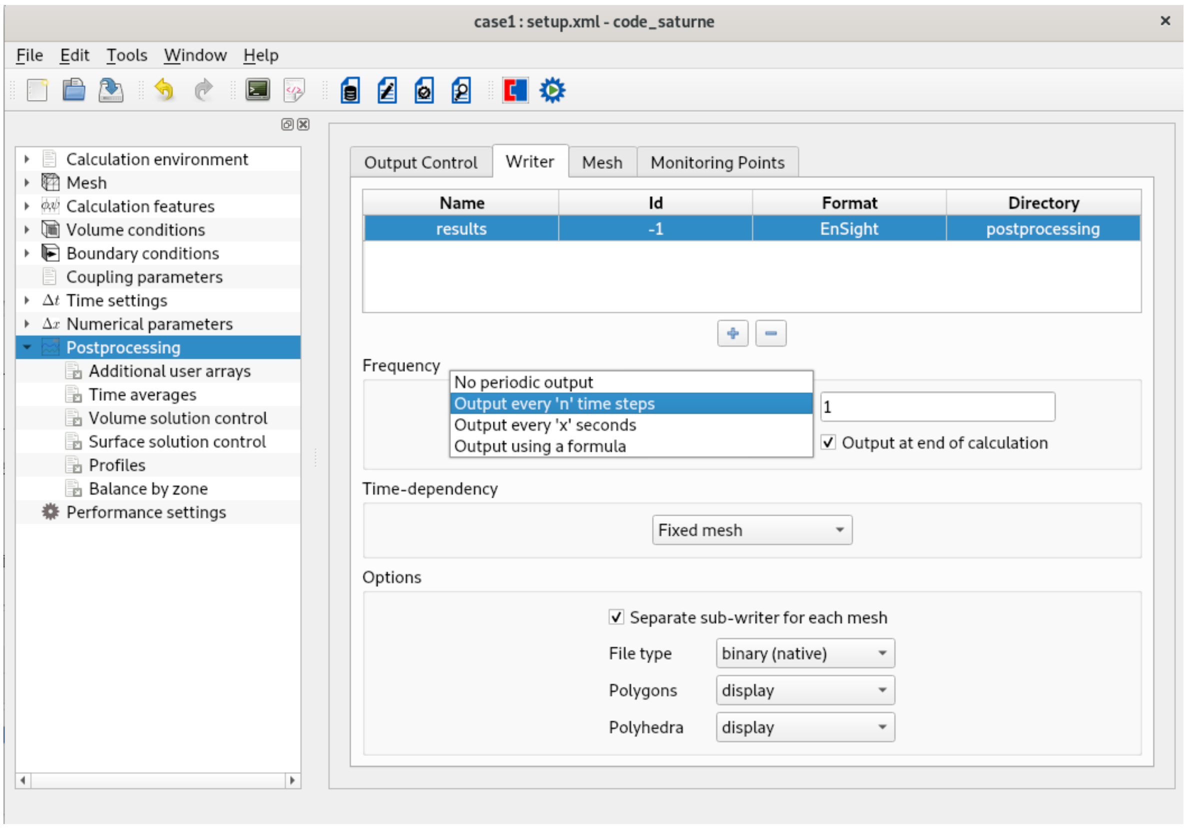


Figure III.26: Output control: post-processing

The other options are kept to their default value.

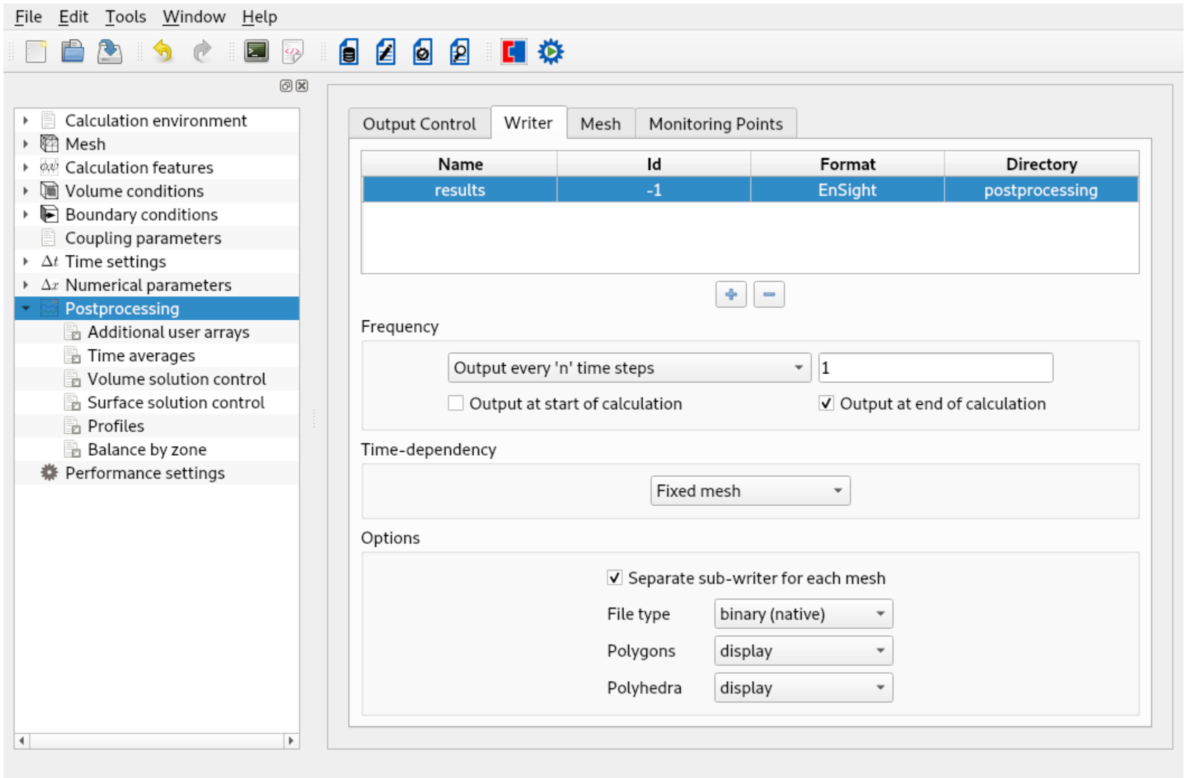


Figure III.27: Output control

The **Monitoring Points** tab allows to define specific points in the domain (monitoring probes) where the time evolution of the different variables will be stored in historic files. In this case no monitoring points are defined.

Volume solution control The **Volume solution control** item allows to specify which variable will appear in the output listing, in the post-processing files or on the monitoring probes. In this case, the default value is kept, where every variable is activated.

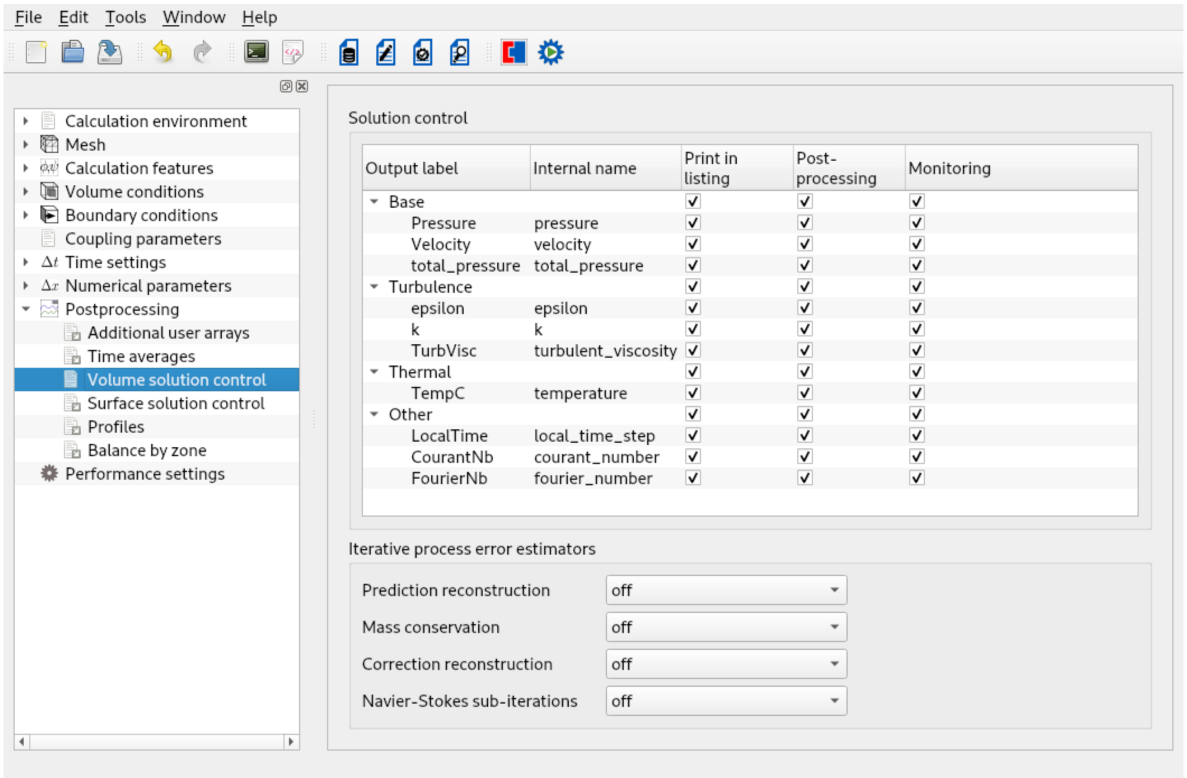


Figure III.28: Solution control

1.8 Run computation

To prepare the launch script and, on certain architectures, launch the calculation, click on the icon in the menu bar and a new window will appear as shown below: On this calculation, the number of processors used will be left to 1.

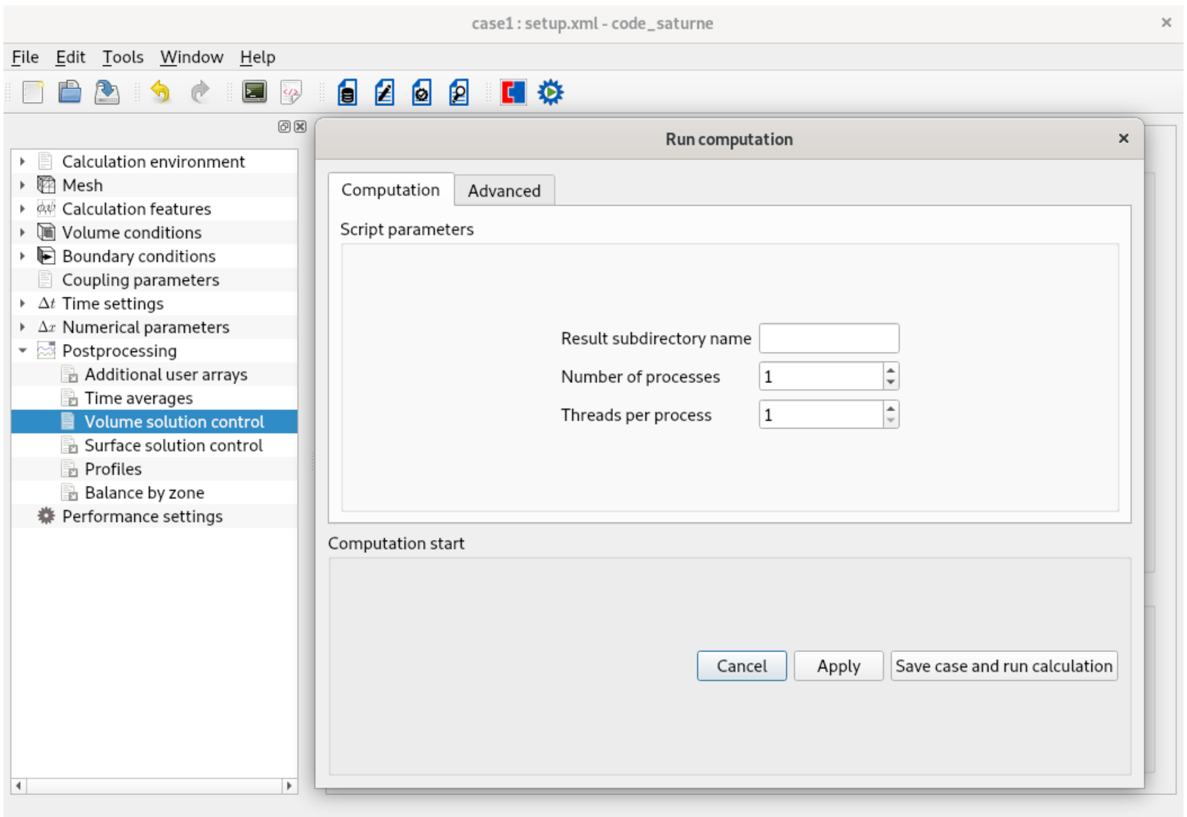


Figure III.29: Prepare batch calculation: computer selection

Finally, the **Advanced options** icon allows to change some more advanced parameters that will not be needed in this simple case.

Eventually, save the `xml` file and execute it by clicking on **Save case and run calculation**. The results will be copied in the `RESU/` directory.

Once you run the calculation process the following III.30 window appears. This window allows you either to access to multiple instantaneous calculation information either to save, to stop or to kill the process. You can also access to the convergence tool providing scalars such as residuals, etc.

Figure III.30: Run