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

SEPTEMBER 2017

Code_Saturne documentation

Code_Saturne version 5.0 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 axisymmetric 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).

1.2 About this document

The present document is a tutorial for *Code_Saturne* version 5.0. 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 before running *Code_Saturne* is to prepare the computation directories. In this first example, the study directory `simple_junction` will be created, containing a single calculation directory `case1`.

Create the study `simple_junction` and the `case1` with SALOME module CFDSTUDY.

Alternatively, you can use the command line and some *Code_Saturne* commands. Create first an `alias`[1] to the *Code_Saturne* command and then type the command:

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

The mesh files should be then copied in the directory `MESH/`.

Finally, if you have set up the study with the command line, the *Code_Saturne* Graphical User Interface (GUI) is launched by typing the command lines as below:

```
$ cd simple_junction/case1/DATA
$ ./SaturneGUI &
```

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.1. 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.

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 $150 \times 10^5\ Pa$:

[1] Create a file `.bash_aliases` in your home directory and add the line "`alias code_saturne='PATH_TO_CS/code_saturne'`". `PATH_TO_CS` should be *Code_Saturne* installation bin directory.

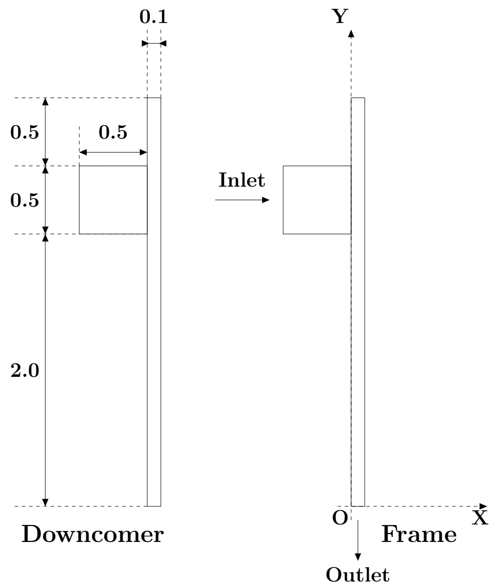

Figure II.1: Geometry of the downcomer

- 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.2 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.

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.2 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.1)

Mesh generator used: SIMAIL

Color definition: see figure II.3. To specifiy boundary conditions on the boundary faces of the mesh, the latter have to be identified. It is commonly done by assigning an integer to each of them, characteristic of the boundary region they belong to. This integer is refered to as **color** or **reference**.

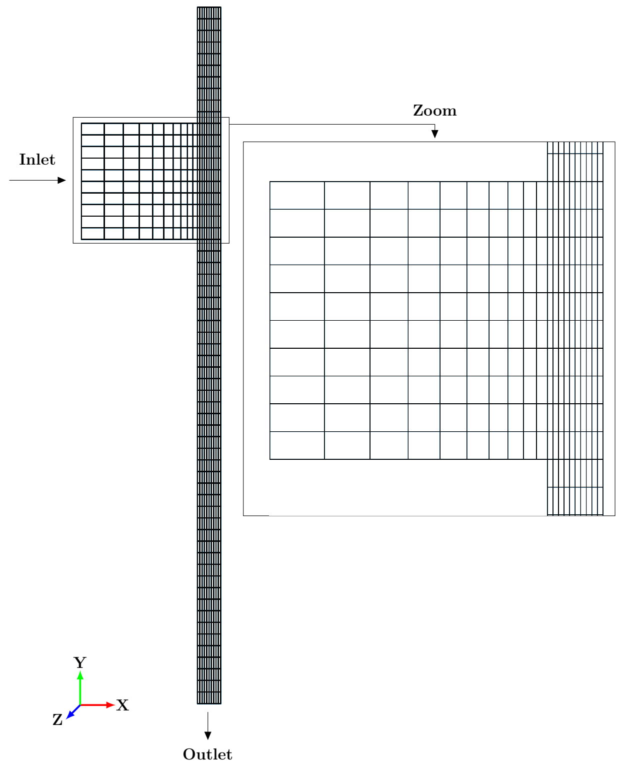

Figure II.2: Mesh of the downcomer

2 CASE 1: Basic calculation

2.1 Calculation options

Most of the options used in this calculation are default options of *Code_Saturne*.

→ Flow type: steady flow

→ Turbulence model: $k - \epsilon$

→ Scalar(s): 1 - temperature

→ 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

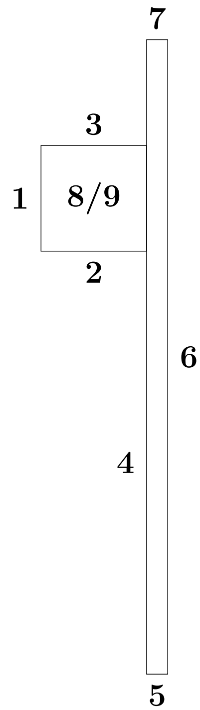

Figure II.3: Colors of the boundary faces

Figure II.3 shows the colors used for boundary conditions and table II.2 defines the correspondance between the colors 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).

Colors	**Conditions**
1	Inlet
5	Outlet
2 3 4 6 7	Wall
8 9	Symmetry

Table II.2: Boundary conditions and associated references

2.3 Parameters and User routines

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	SIMPLE algorithm
Number of iterations	300
Relaxation coefficient	0.9
Output period for post-processing files	1

2.4 Results

Figure II.4 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.

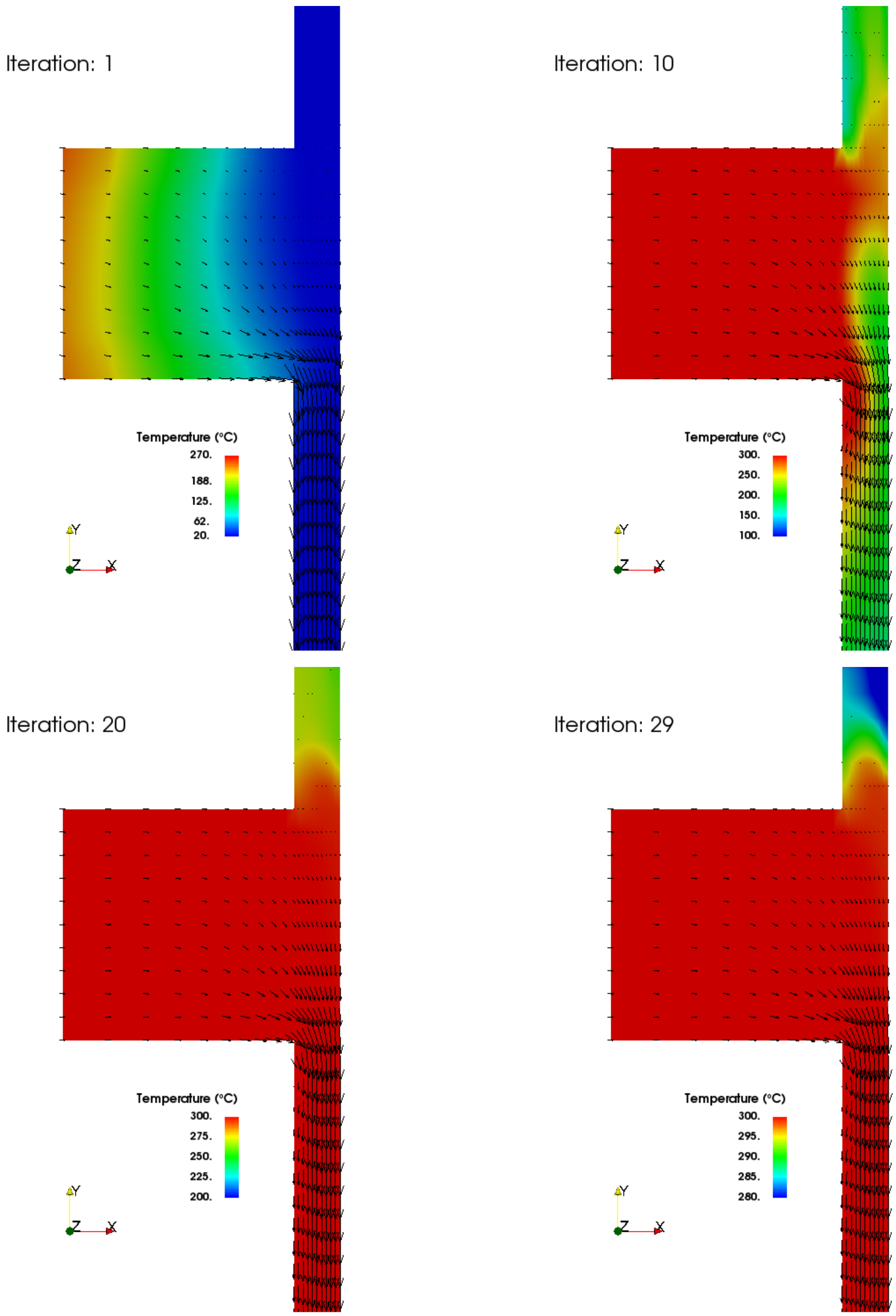

Figure II.4: Water velocity field colored by temperature at different iterations

Part III

Step by step solution

1 Solution for CASE1

The first thing to do before running *Code_Saturne* is to prepare the computation directories. In this first example, the study directory 🗁`simple_junction` will be created, containing a single calculation directory 🗁`case1`. This is done by typing the command:

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

The mesh files should be copied in the directory 🗁`MESH/`, as follow:

```
$ cd simple_junction/MESH/
$ cp /projets/echanges.004/ITECH_CS_TRAINING/meshes/1-simple_junction/downcomer.des .
```

The *Code_Saturne* Graphical User Interface (GUI) is launched by typing the command lines as below:

```
$ cd simple_junction/case1/DATA
$ ./SaturneGUI &
```

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

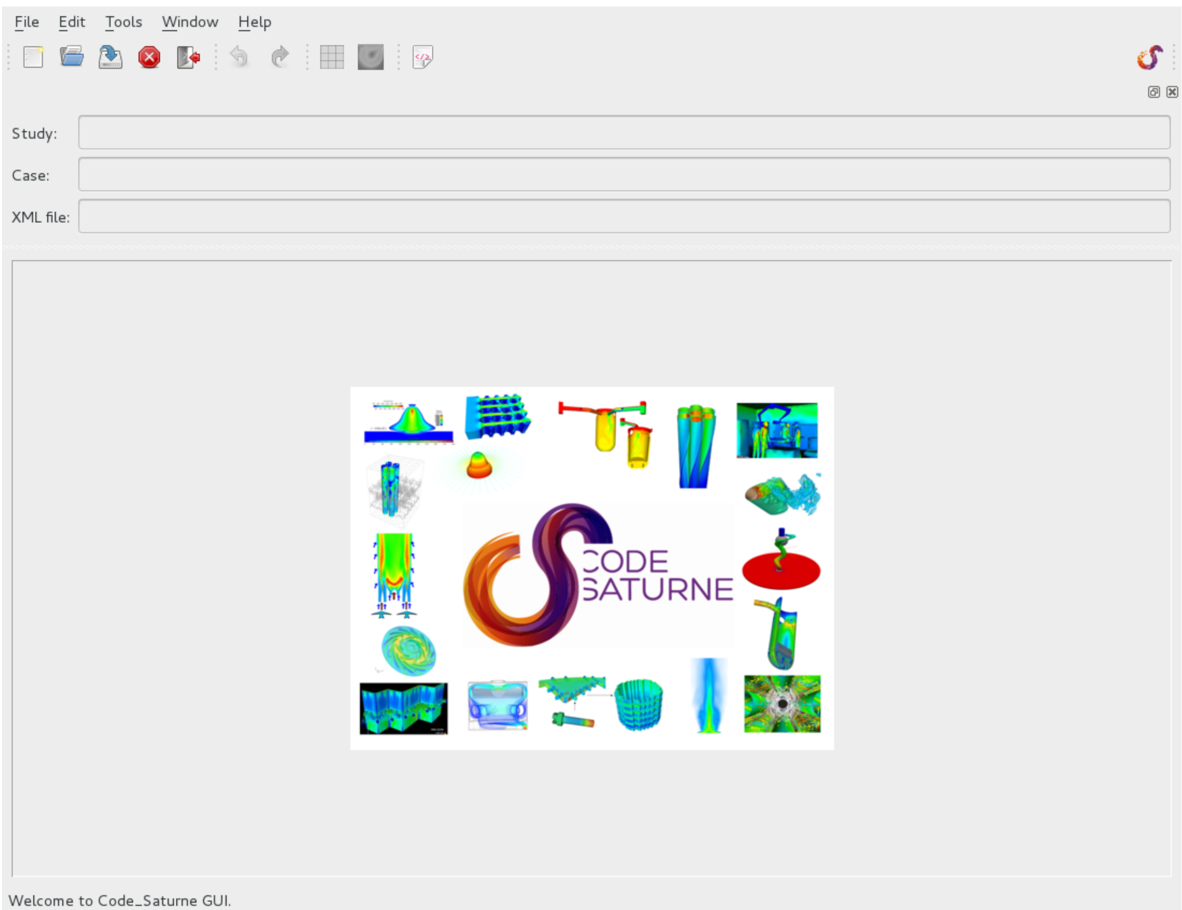

Figure III.1: *Code_Saturne* (GUI) graphic window

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

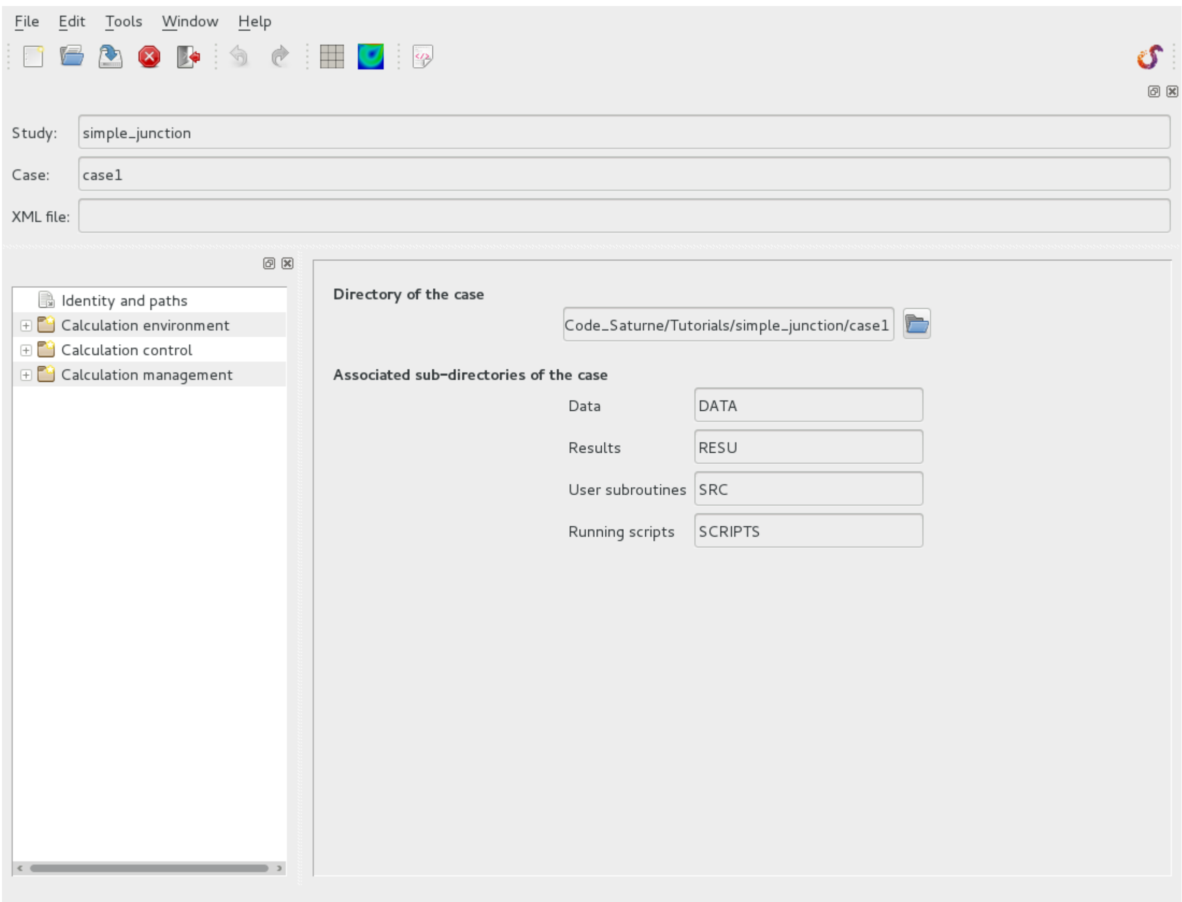

Figure III.2: Identity and paths

Save the case to give a name to the new `xml` file (such as `case1.xml`) by opening the File menu and clicking on **Save as...**.

A new window will appear, enter the name of the case in **File name** then click on **Save**.

Remember to save the case regularly throughout the preparation of the calculation.

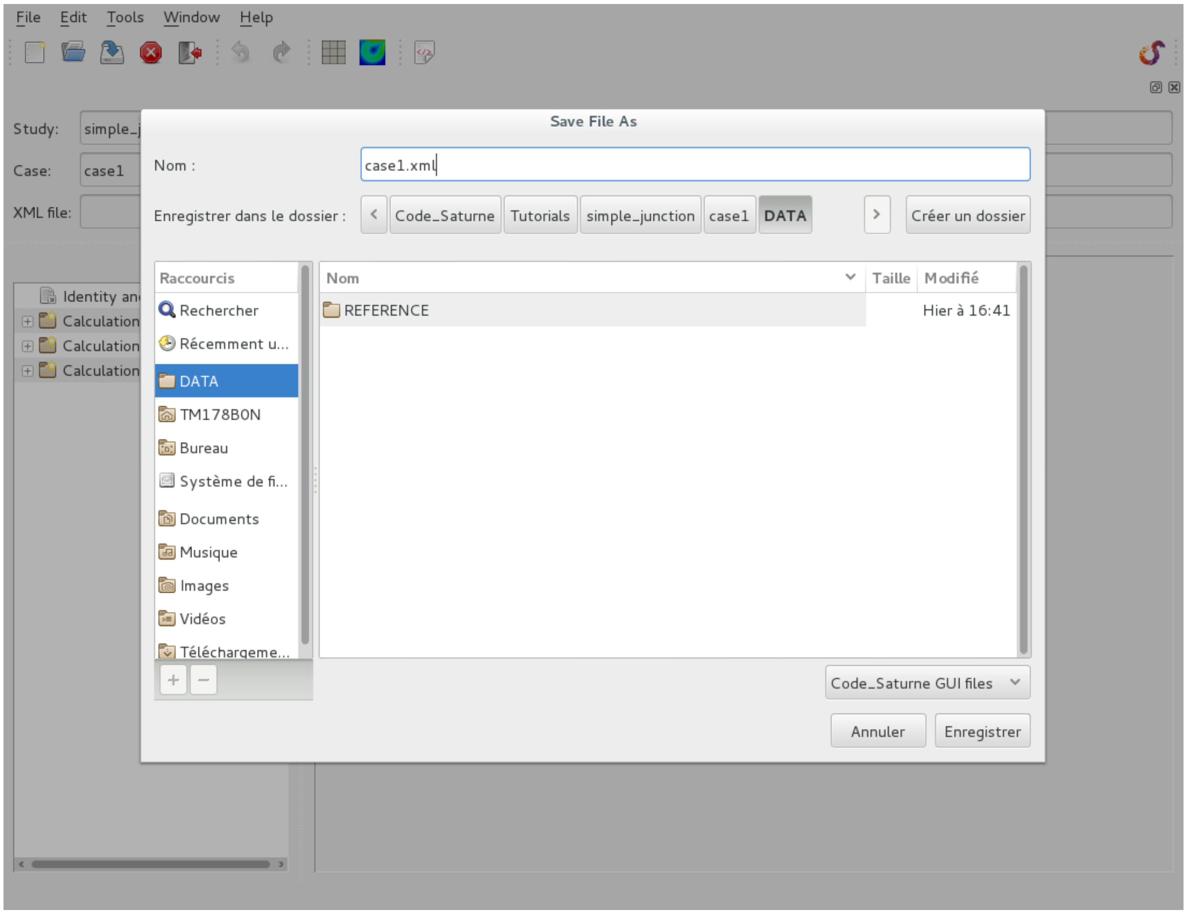

Figure III.3: Saving the `xml` file

In *Code_Saturne*, two modes are available:

- A preprocessing mode, used for mesh preprocessing, and mesh quality checking;
- A calculation mode, used to run computations on the preprocessed mesh.

The mesh selection can still be done in both mode.

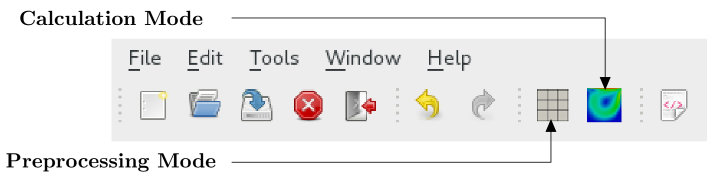

Figure III.4: Preprocessing and calculation modes in *Code_Saturne*

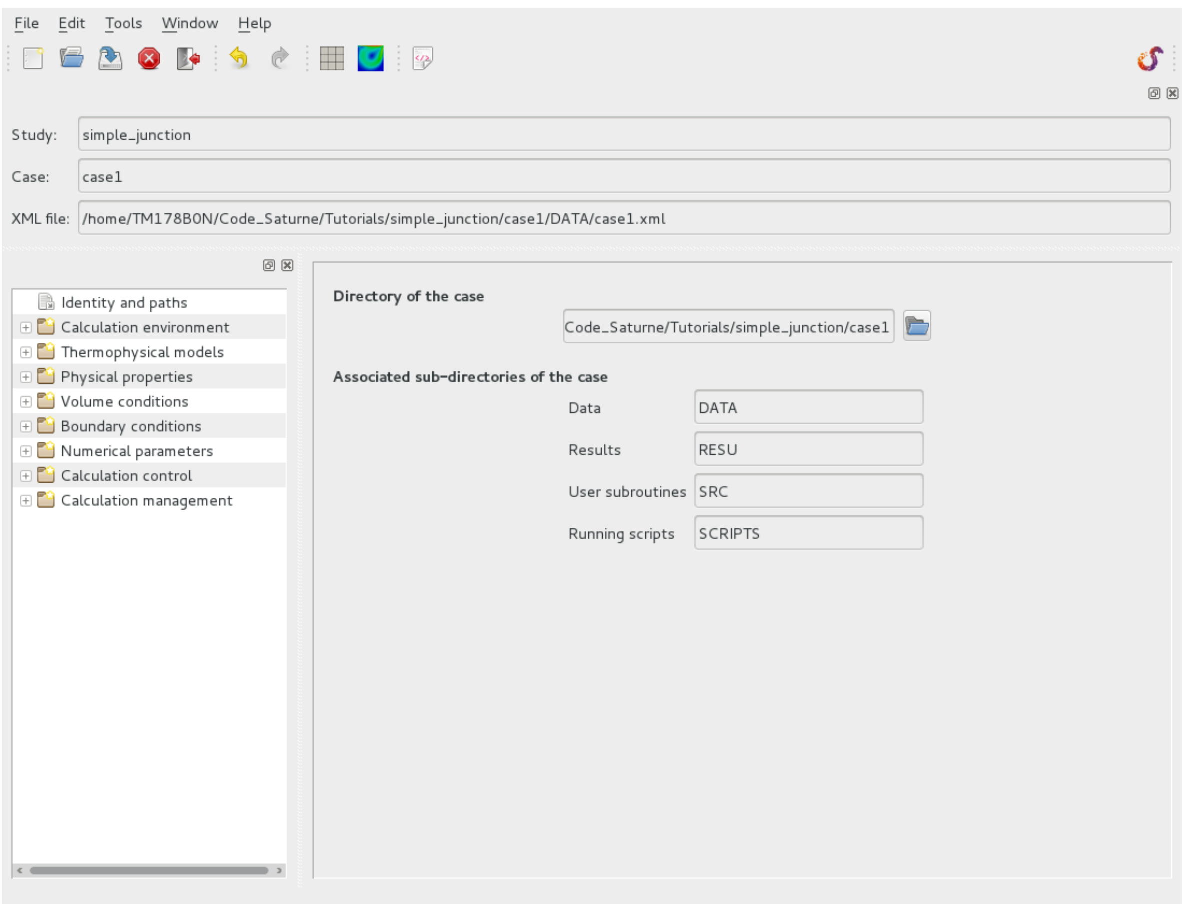

Figure III.5: Calculation mode in *Code_Saturne*

The next step is to specify the mesh(es) to be used for the calculation. Click on the **Meshes selection** item under the heading **Calculation environment**. Click on [+] to add meshes.

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

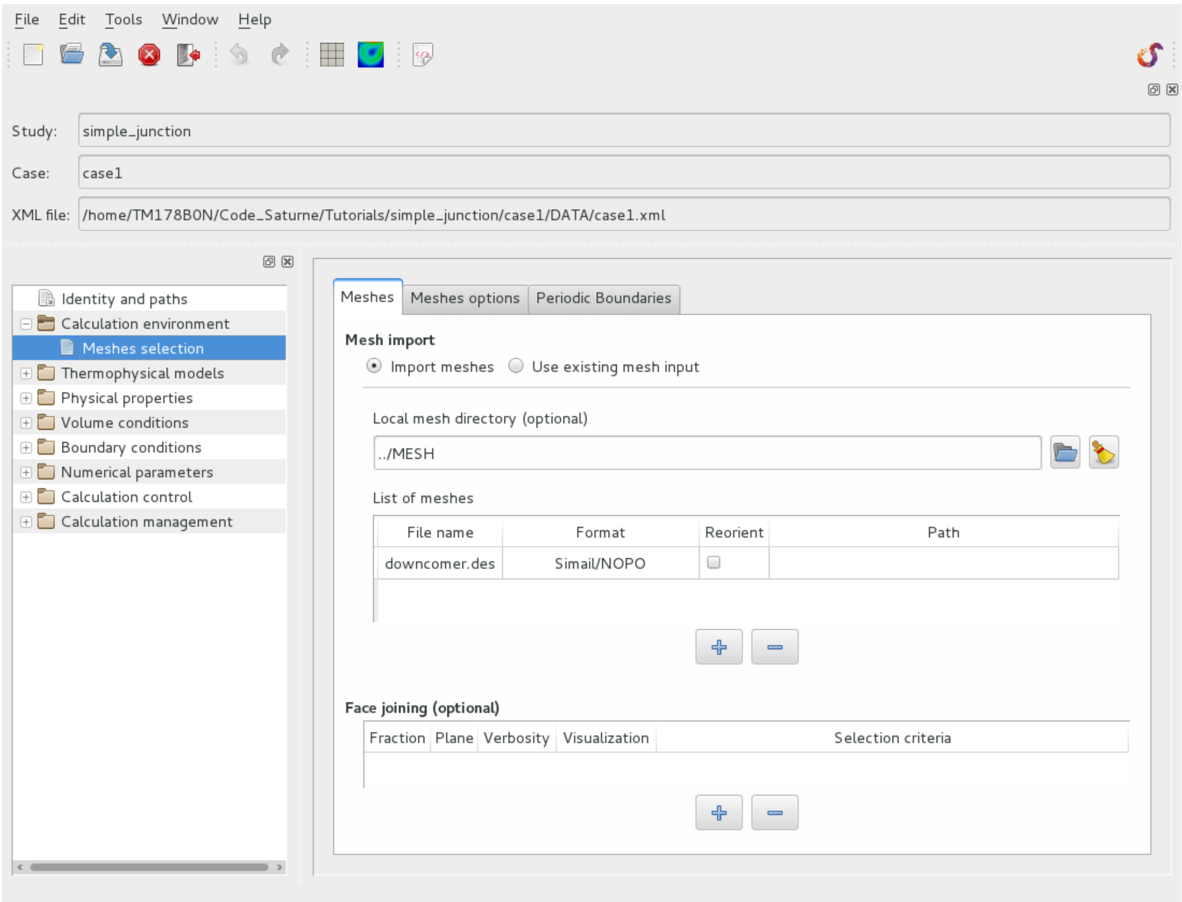

Figure III.6: Meshes: list of meshes

The **Periodic Boundaries** is not used in this case. Keep the default values.

The **Calculation features** item under the heading **Thermophysical models** allows to define the type of flow to be simulated. In this case, a steady flow will be chosen.

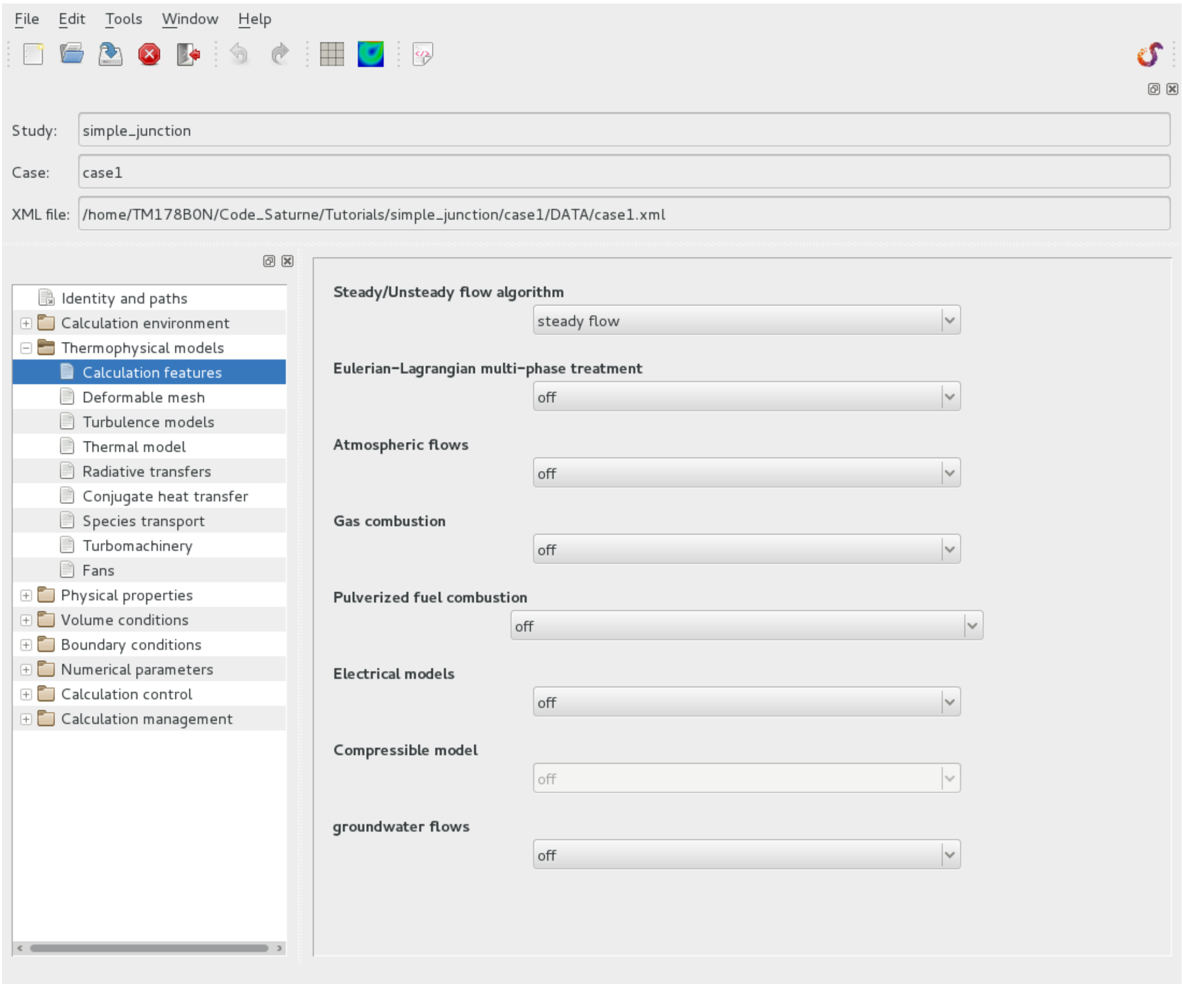

Figure III.7: Flow type

The turbulence model is selected in the following list:

- laminar flow (no model)
- mixing length
- k-ε
- k-ε Linear Production
- Rij-ε LLR
- Rij-ε SSG
- v2f BL-v2/k
- k-ω SST
- Spalart-Allmaras
- LES (Smagorinsky)
- LES (classical dynamic model)
- LES (WALE)

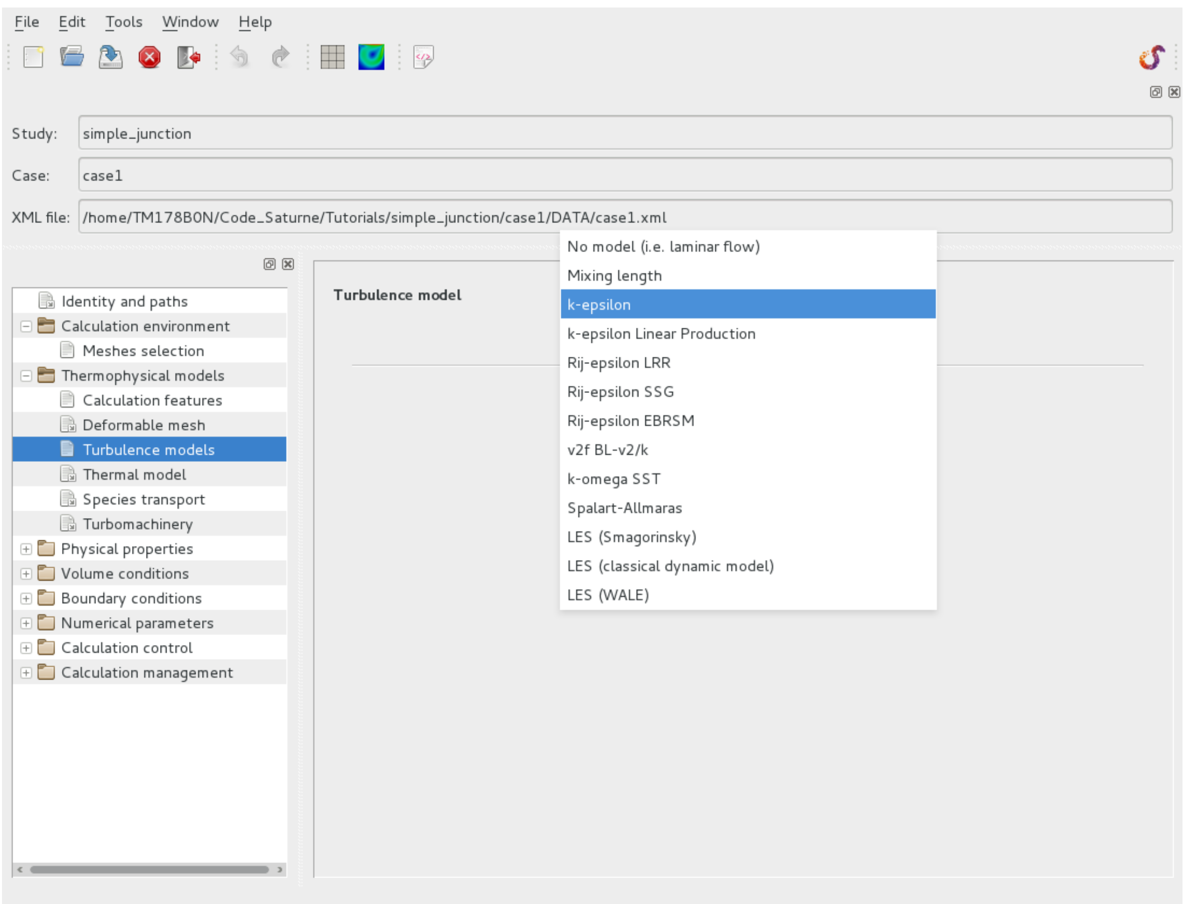

Figure III.8: Turbulence model: list of models

In this case, the k-ε model is used.

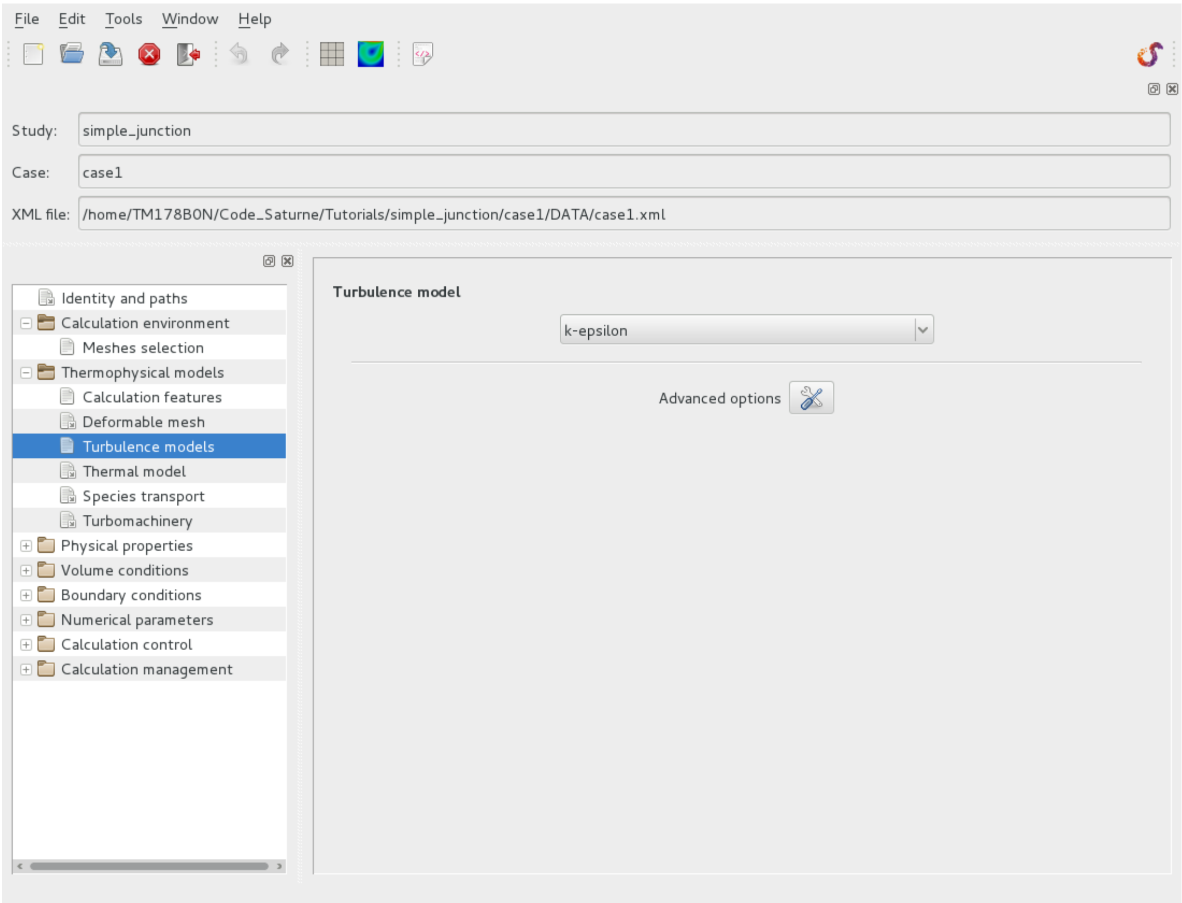

Figure III.9: Turbulence model: choice of a model

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)
```

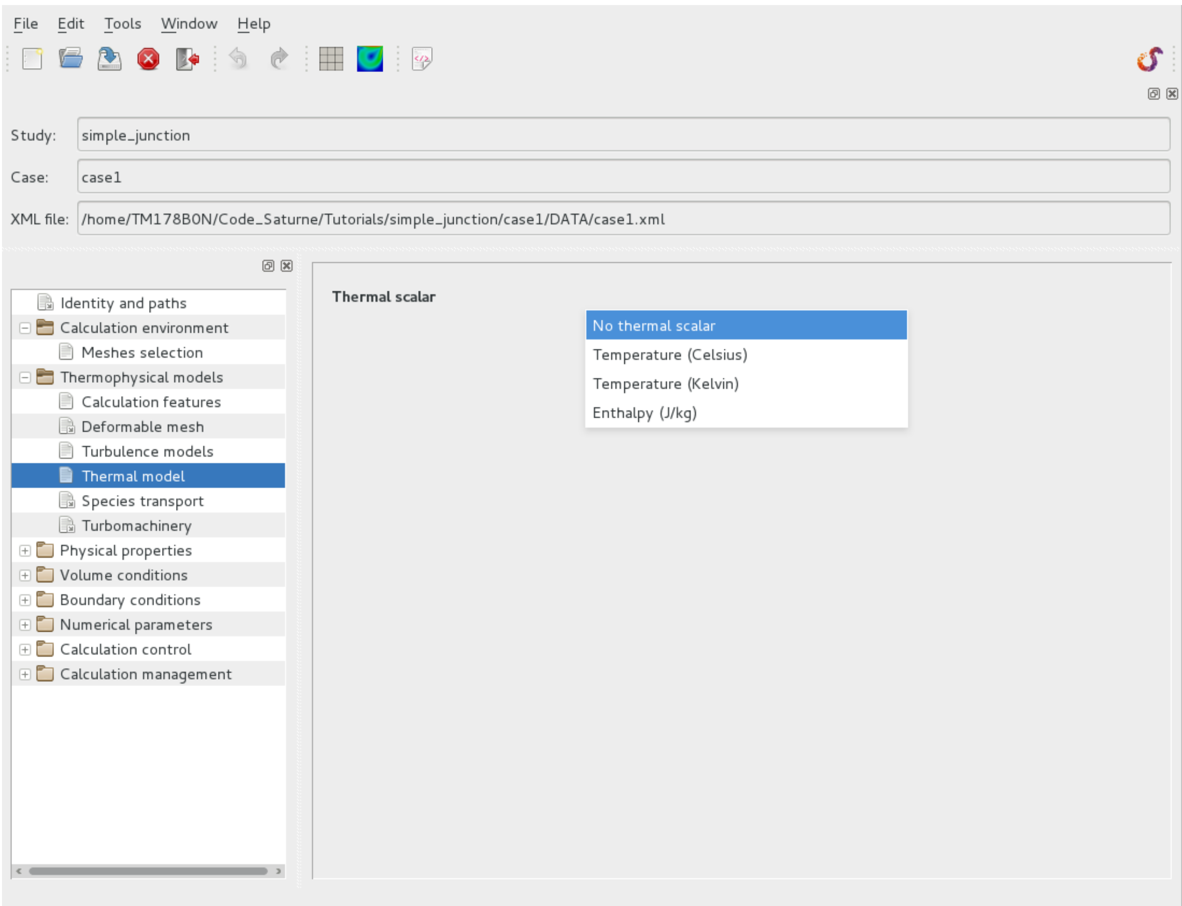

Figure III.10: Thermal scalar conservation: list of models

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

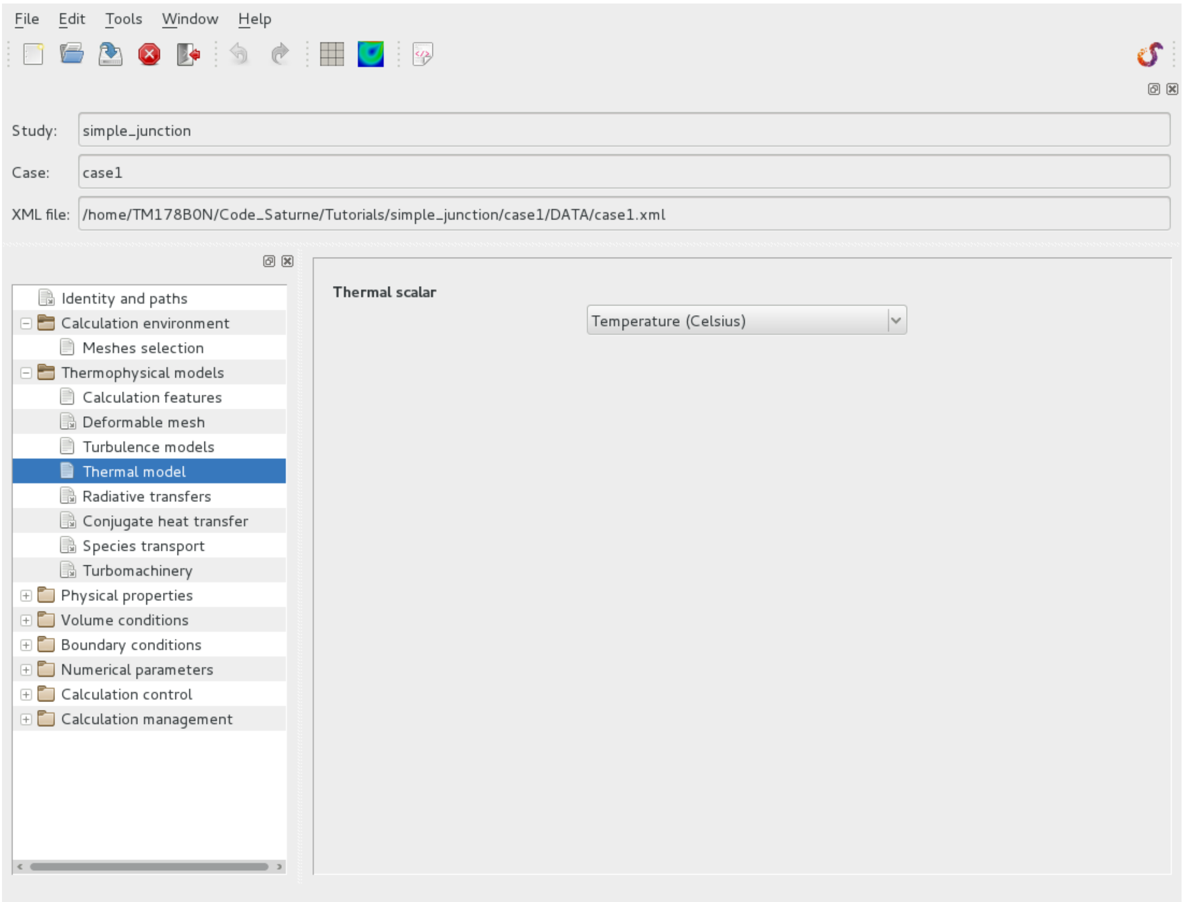

Figure III.11: Thermal scalar conservation: choice of a model

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

Initialization:

To initialize variables at the instant $t = 0$ (s), go to the **Initialization** item under the heading **Volume conditions**. Here the velocity, the thermal scalar and the turbulence can be initialized.

In this case, the values te be set are: zero velocity (default), an initial temperature of **20**°C and a turbulence level based on a reference velocity of **1** $(m.s^{-1})$ (default). Specific zones can be defined with different initializations. In this case, only the default **all cells** is used.

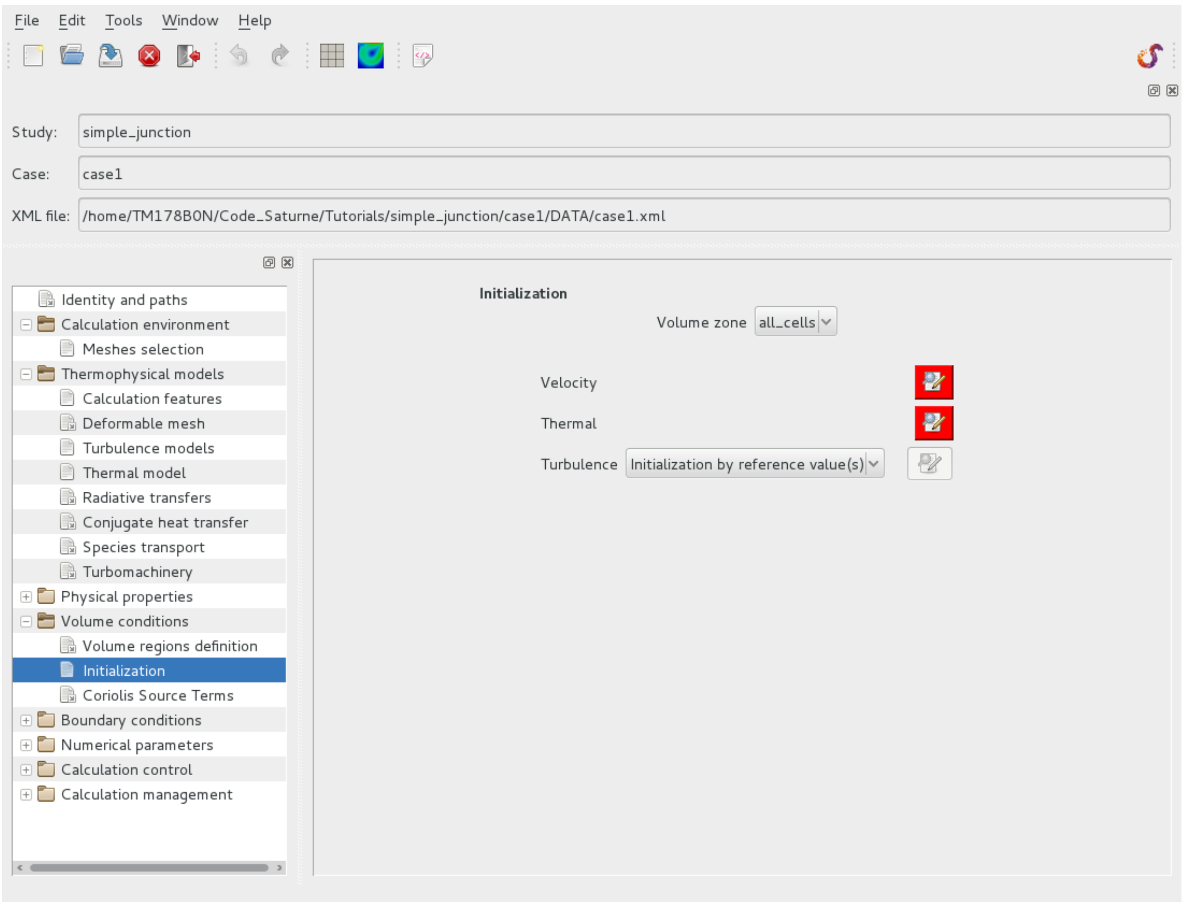

Figure III.12: Initialization of the scalar, velocity and turbulence

- Click on the icon near **Thermal** in order to specify the initial value of the thermal scalar. It can be a value or a user expression.

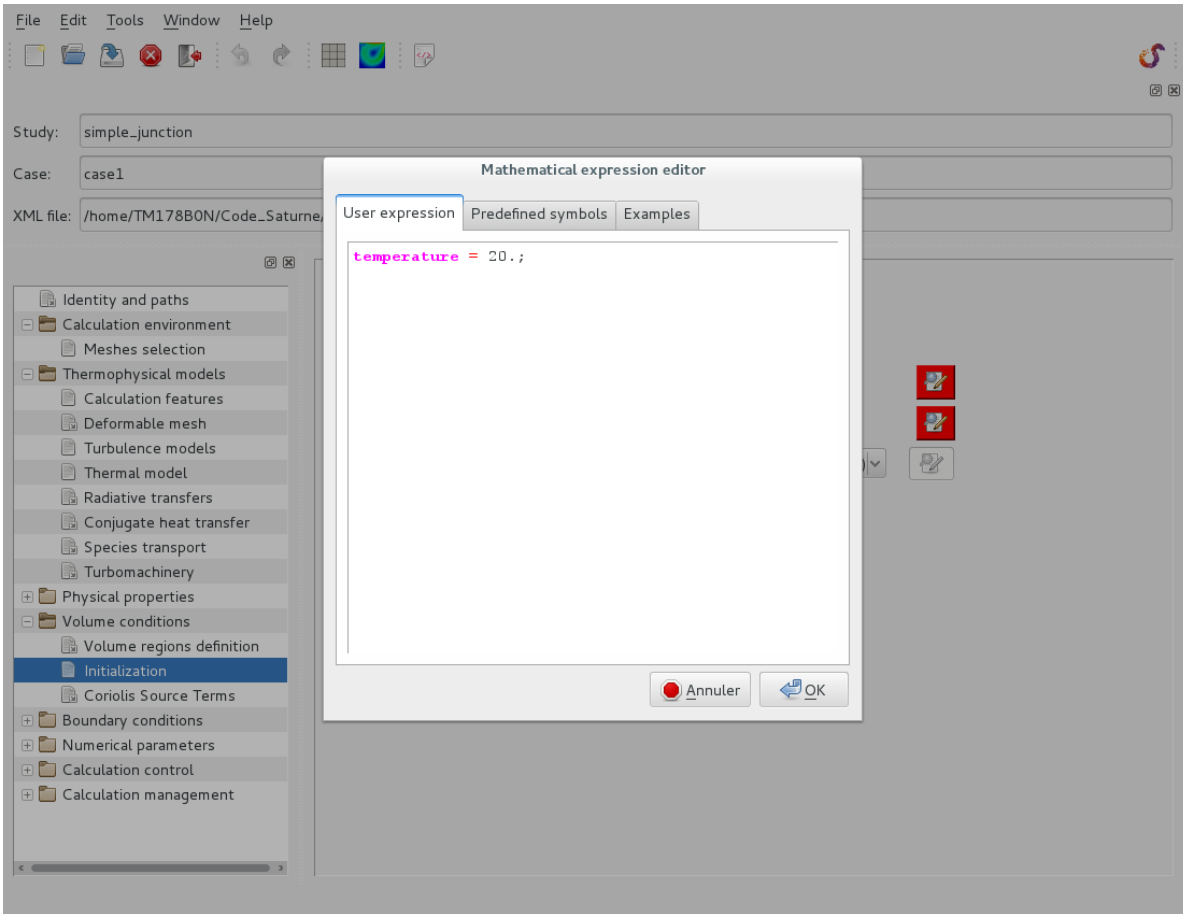

Figure III.13: Initialization of the scalar

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

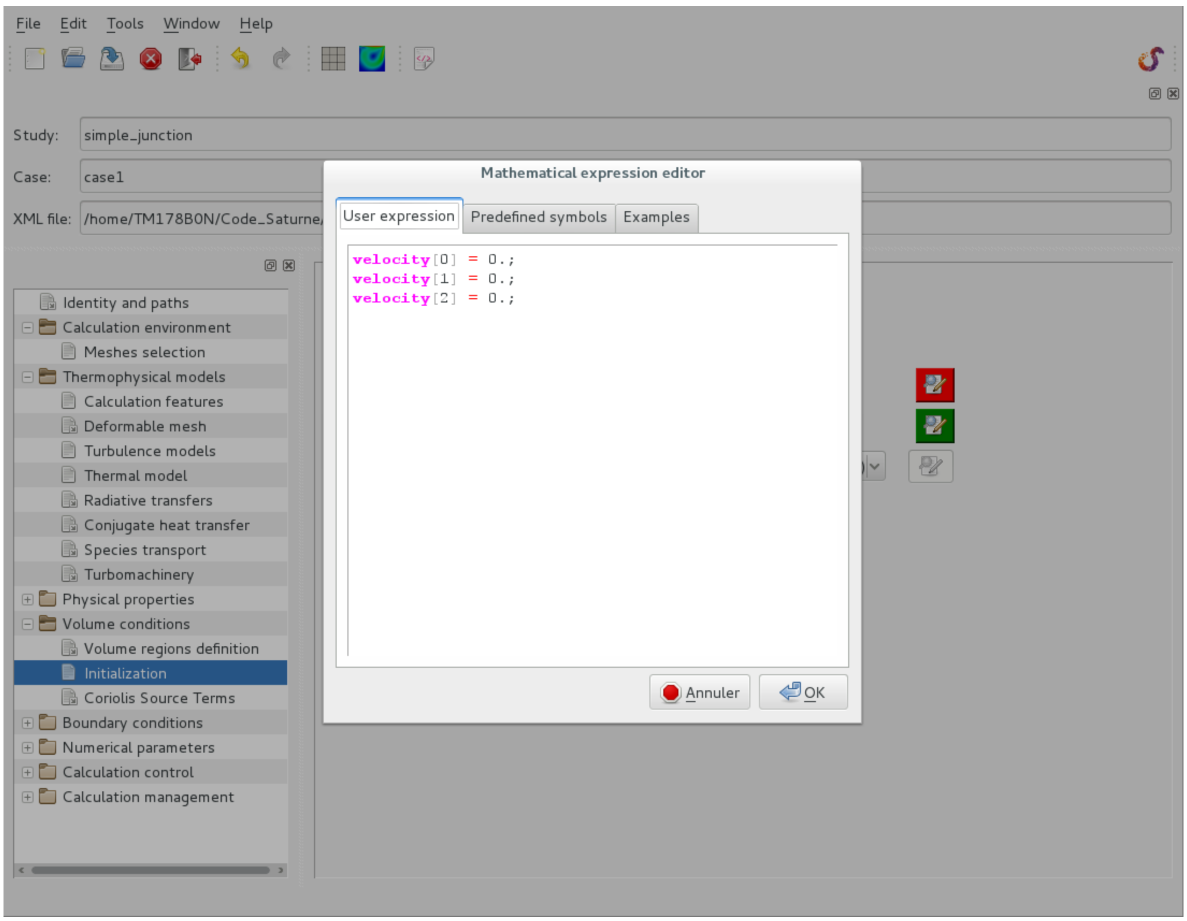

Figure III.14: Initialization of the velocity

Under the heading **Physical properties** in the main list, the **Reference values** item allows to set the reference pressure, the reference velocity and the reference length.

Use the default value of **101 325** (Pa) for the pressure and **1** (m/s) for the velocity.

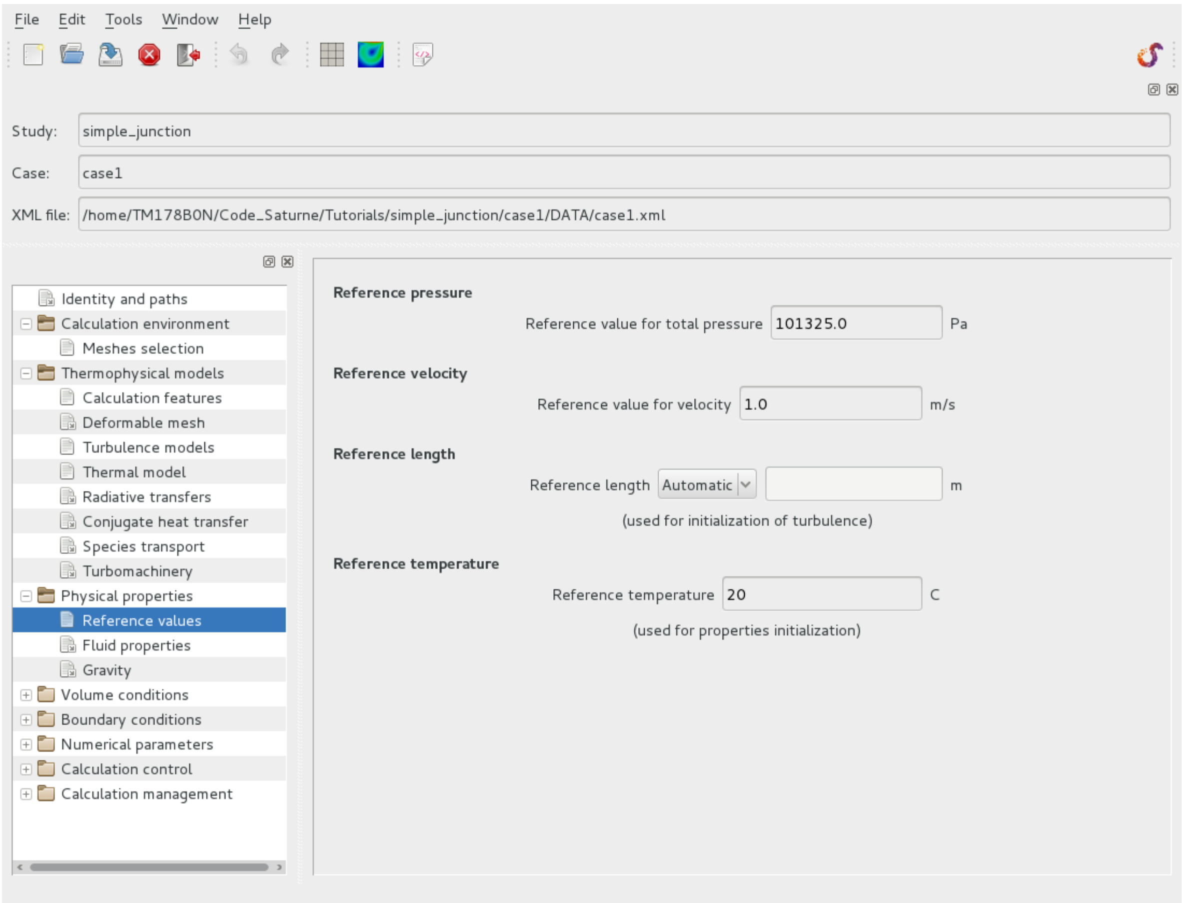

Figure III.15: Physical properties: reference pressure

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

- Density
- Viscosity
- Specific Heat
- Thermal Conductivity

In this case they are all constant.

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

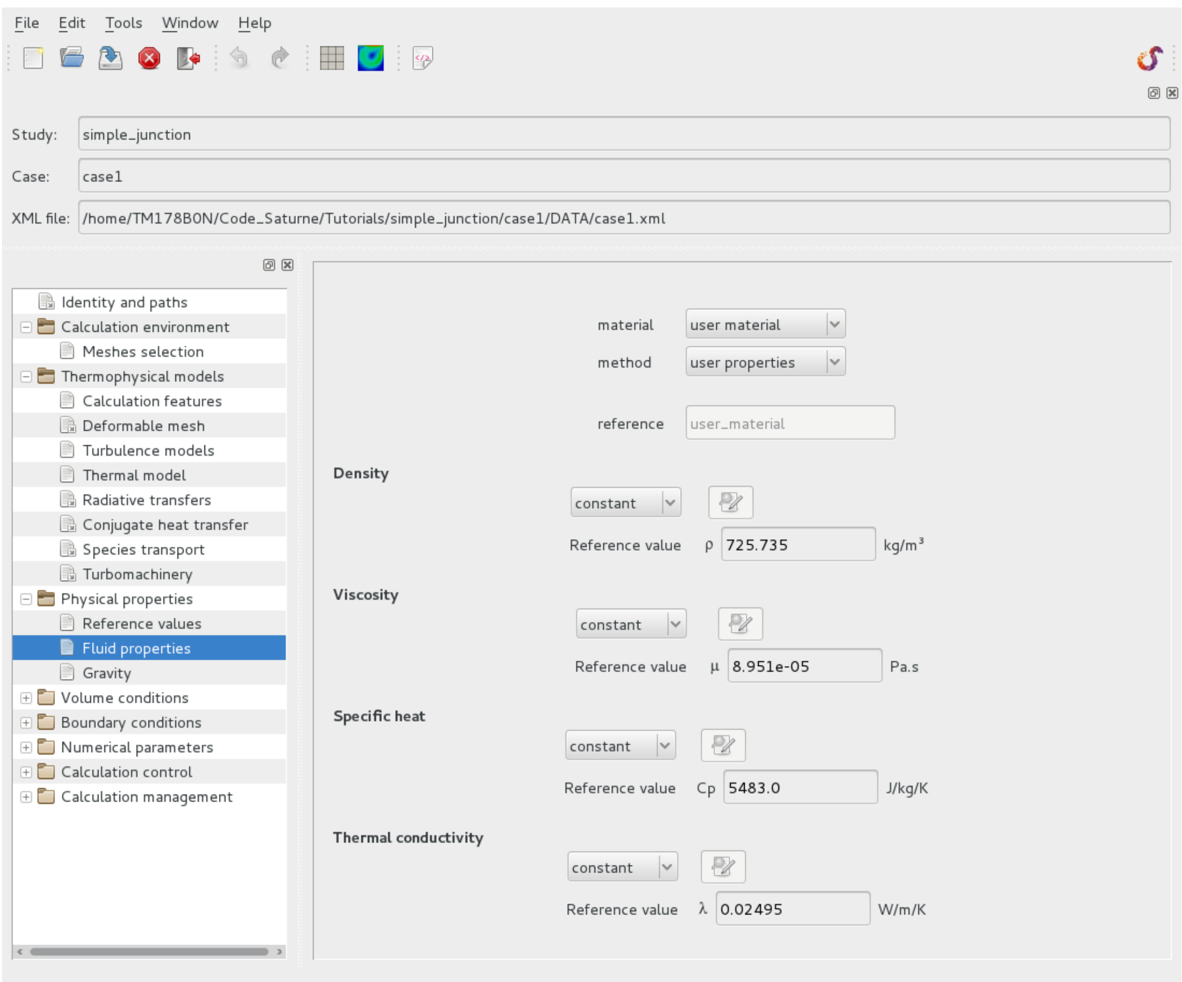

Figure III.16: Physical properties: fluid properties

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**.

As for the pressure interpolation, the interpolation method keeps the standard default value.

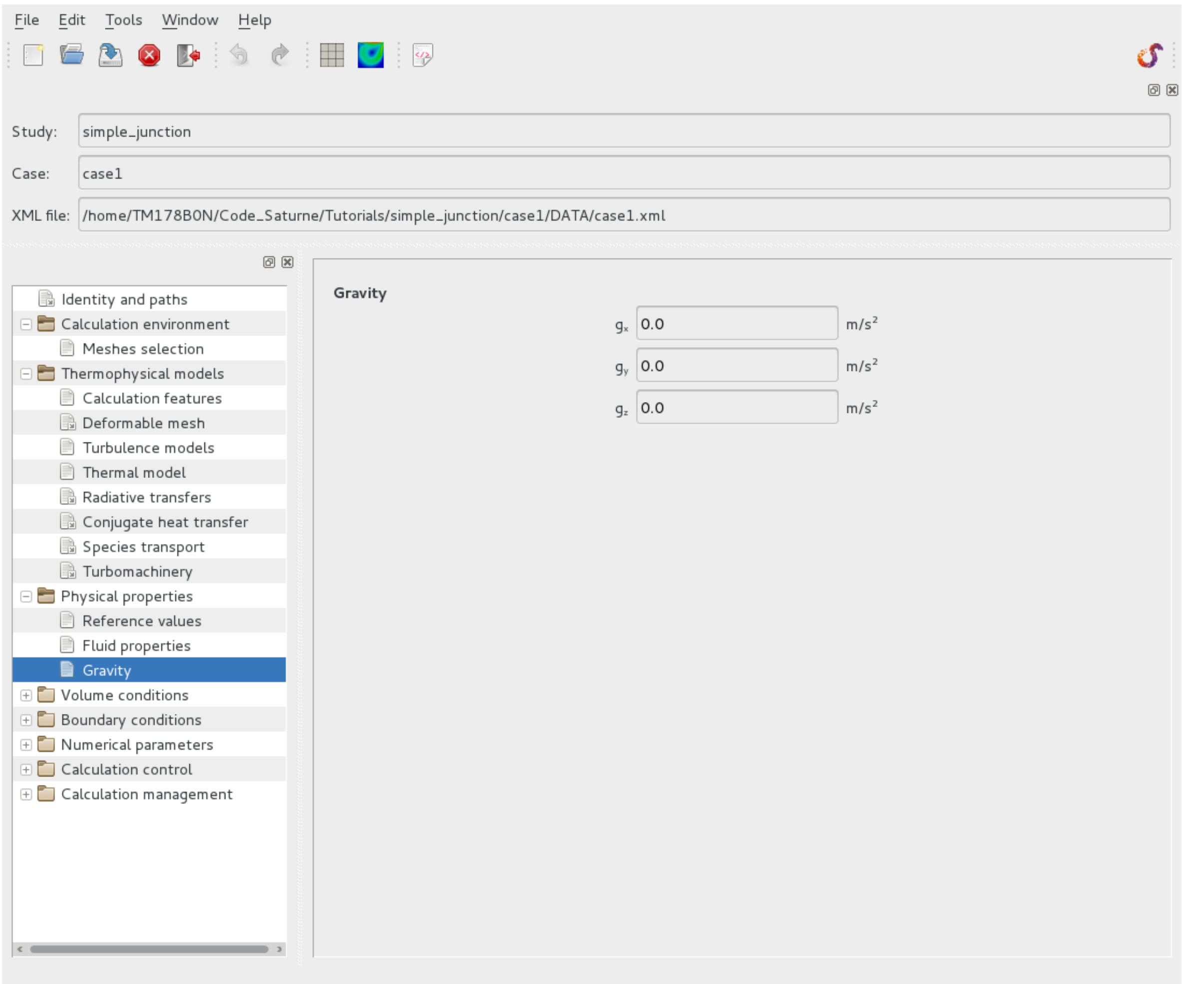

Figure III.17: Physical properties: gravity and hydrostatic pressure

Boundary conditions now need to be defined. Go to the **Definition of boundary regions** item under the heading **Boundary conditions**. The following window opens (fig III.20).

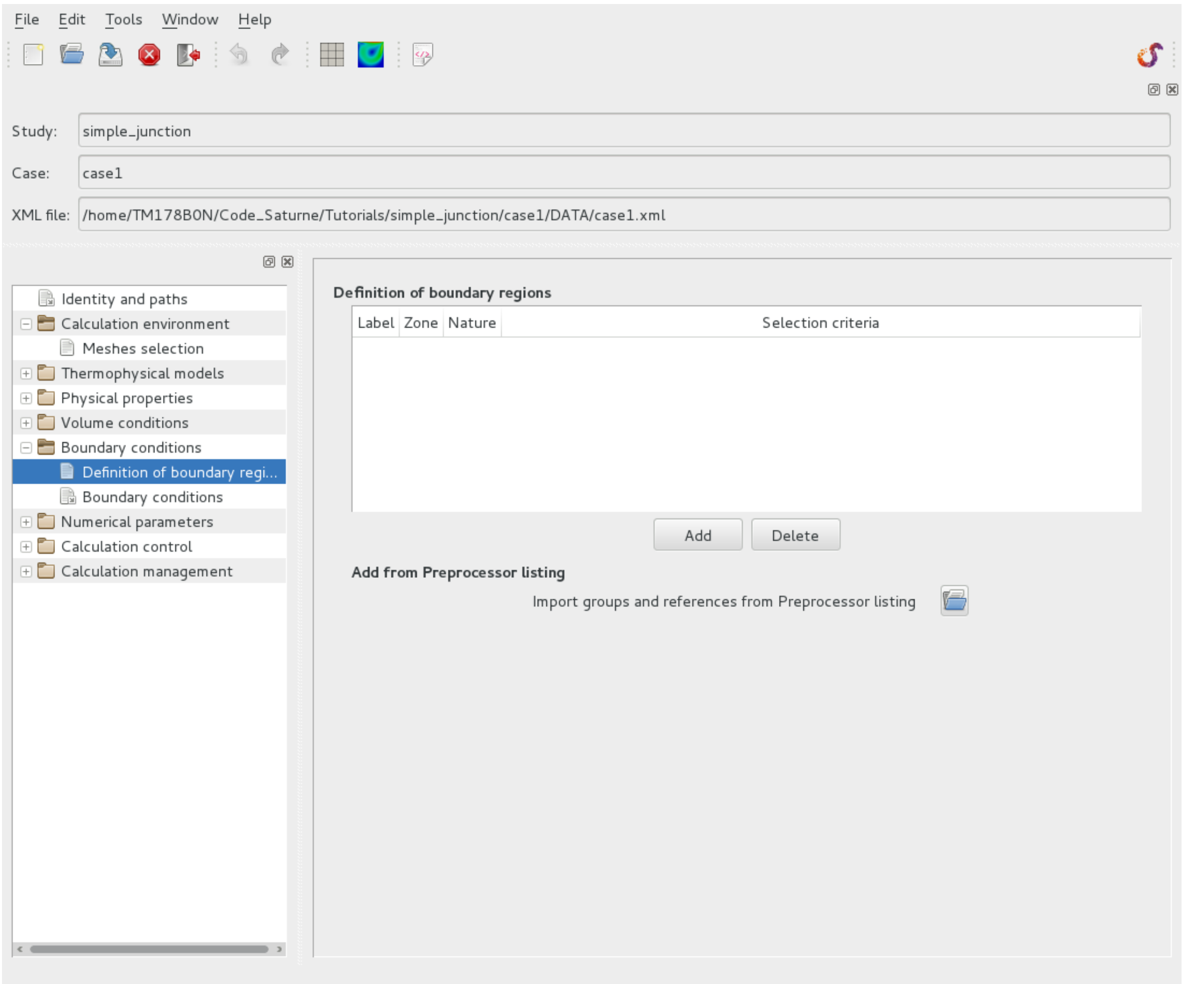

Figure III.18: Creation of a boundary region

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 their color or on geometrical conditions. For each zone, a reference number, a label, a nature and a selection criteria must be assigned. The different natures that can be assigned are:

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

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 **Zone** number can be any integer. It will be used by the code to identify the zone. No specific order or continuity in the numbering is needed.

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

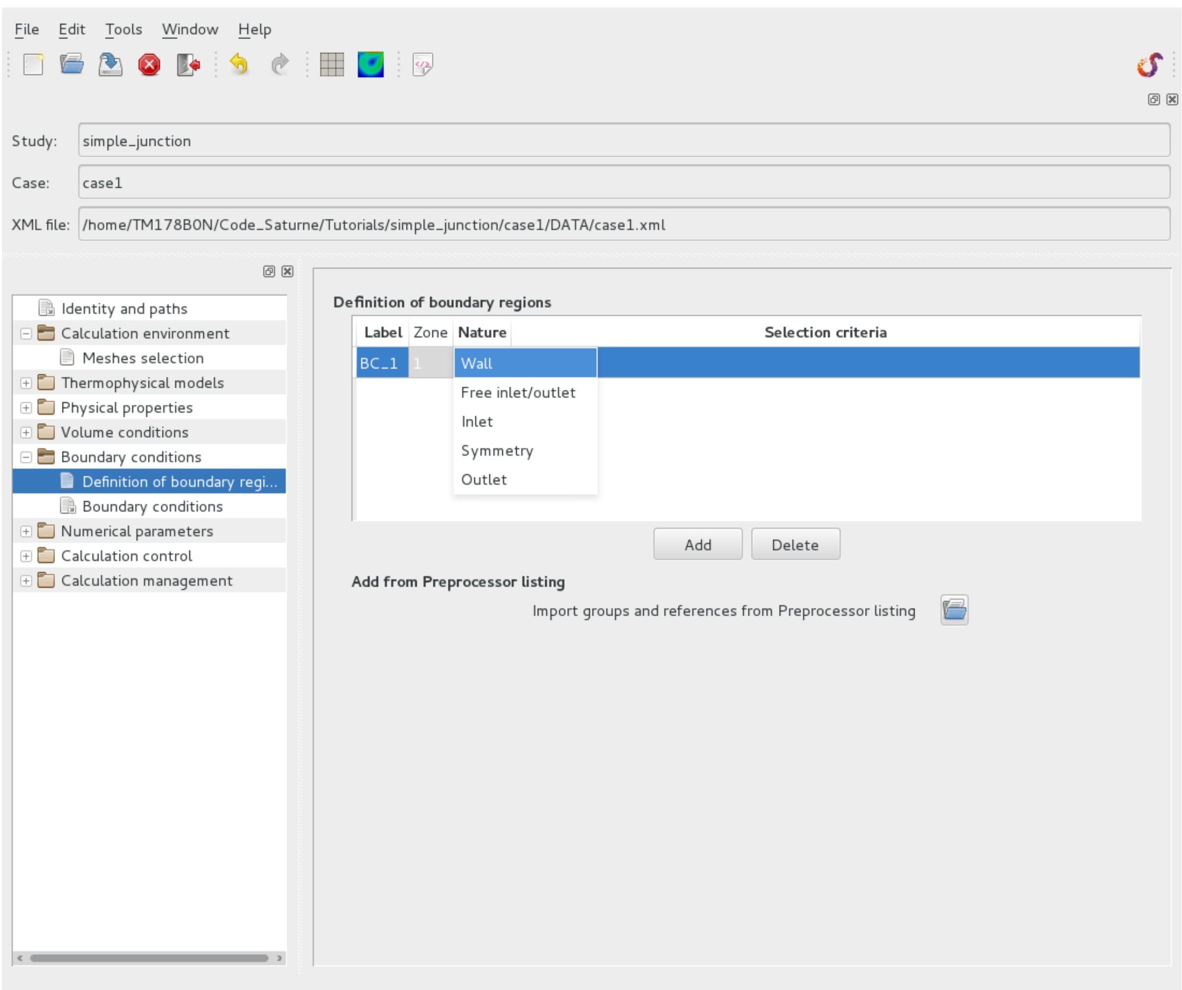

Figure III.19: Creation of a boundary region

The specification of the inlet condition is detailed in the following pages. The settings will be as follows:

```
Label:  Inlet,
Zone:  1,
Nature:  Inlet,
Selection criteria:  1
```

Type all the information in the fields, the result diplays as figure III.20

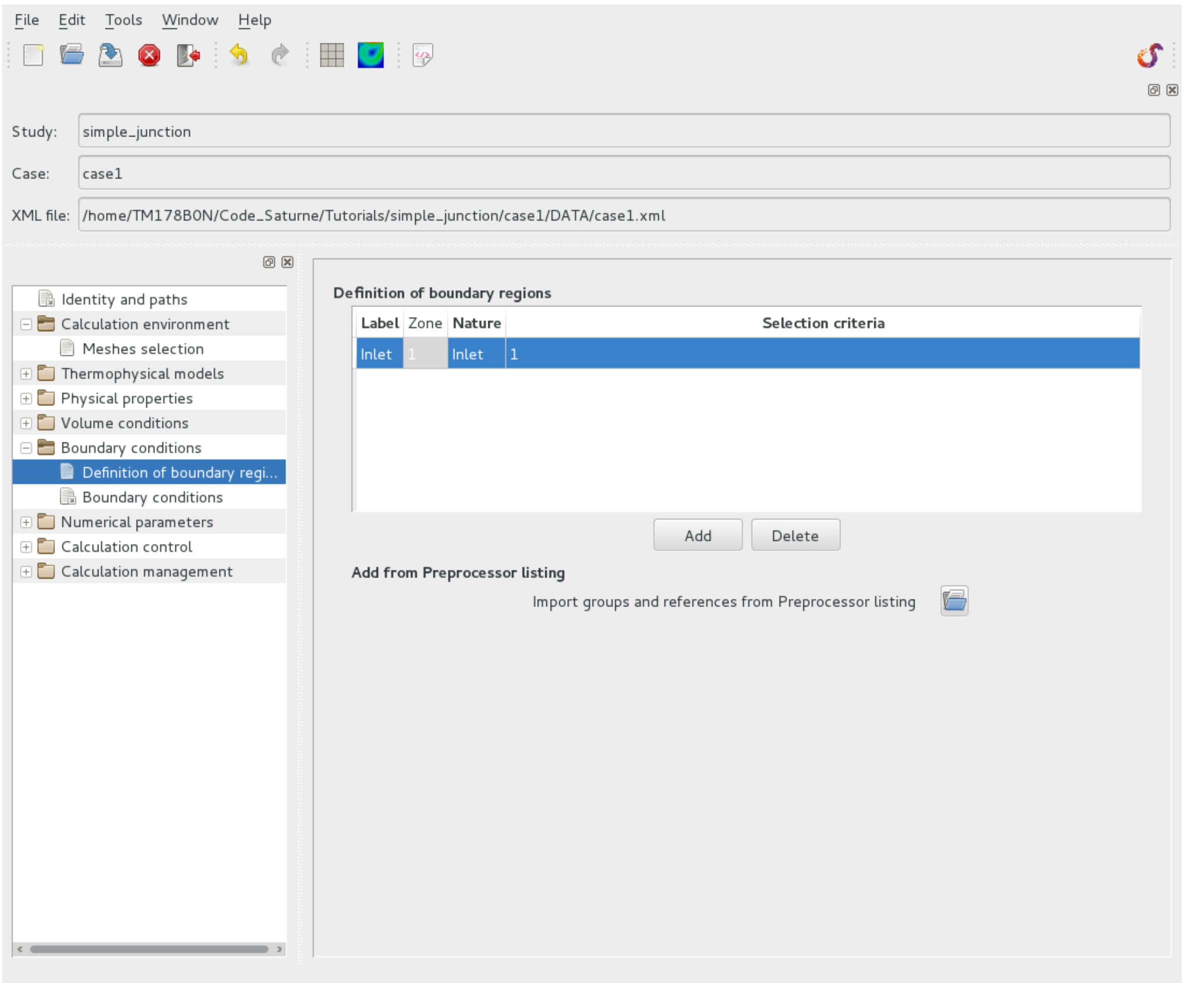

Figure III.20: Creation of a boundary region

Remember to save the `xml` file regularly!

Do the same thing for the other boundaries.

In our case, colors 8 and 9 are symmetry boundaries. One option can be to define a separate zone for each color, as follows:

Label	**symmetry_1**	**symmetry_2**
Zone	3	4
Nature	symmetry	symmetry
Localization	8	9

Figure III.21: Symmetric boundary conditions

But it is usually faster to regroup the different colors in one single zone, as shown on figure III.22. In our case, the localization for this zone is the string `‘‘8 or 9’’`.

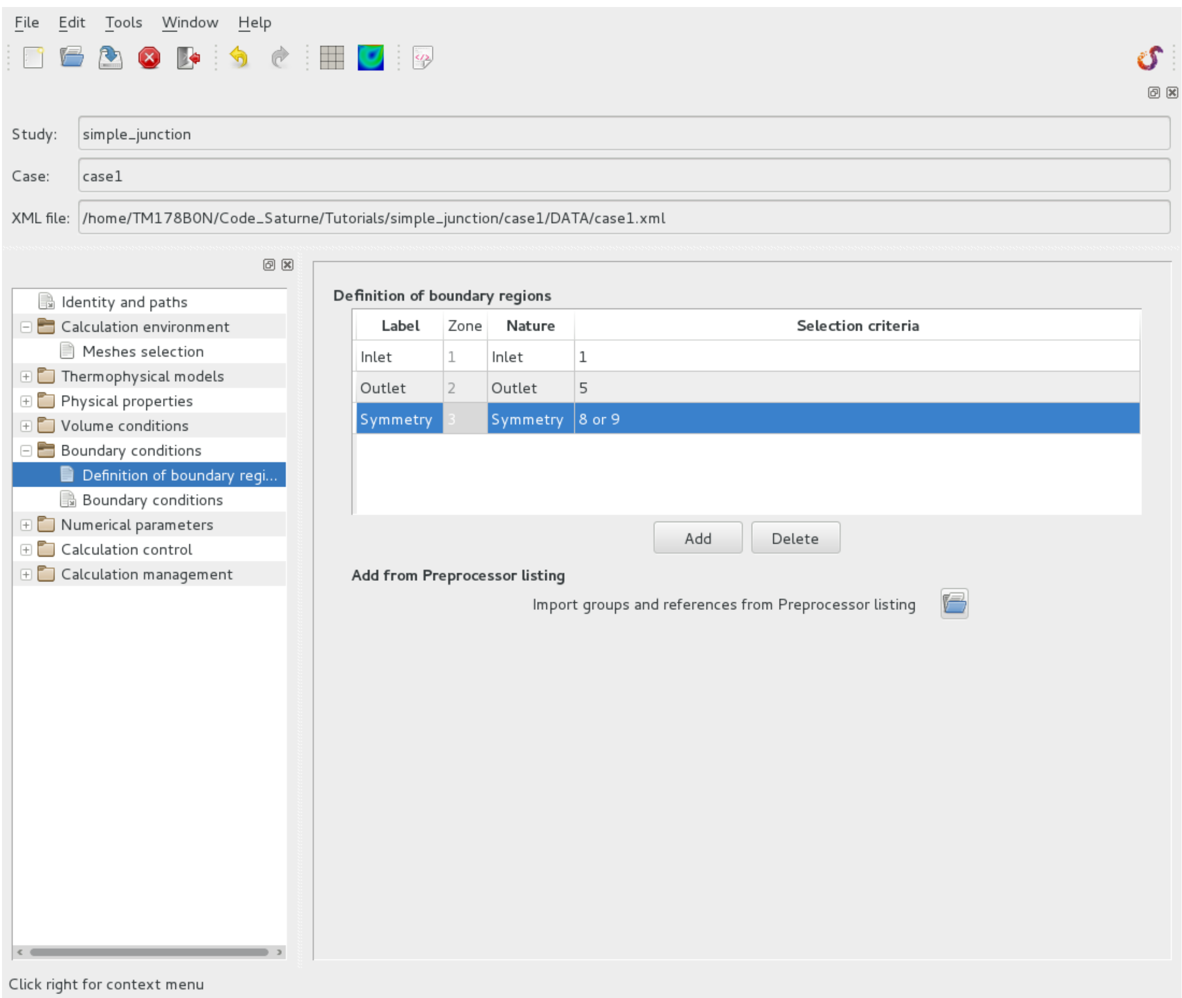

Figure III.22: Creation of boundary regions: symmetry region

The same treatment must be done for the wall conditions. All colors 2, 3, 4, 6 and 7 can be grouped in a single boundary zone.

After defining all the boundary zones, the Interface window will look as in figure III.23.

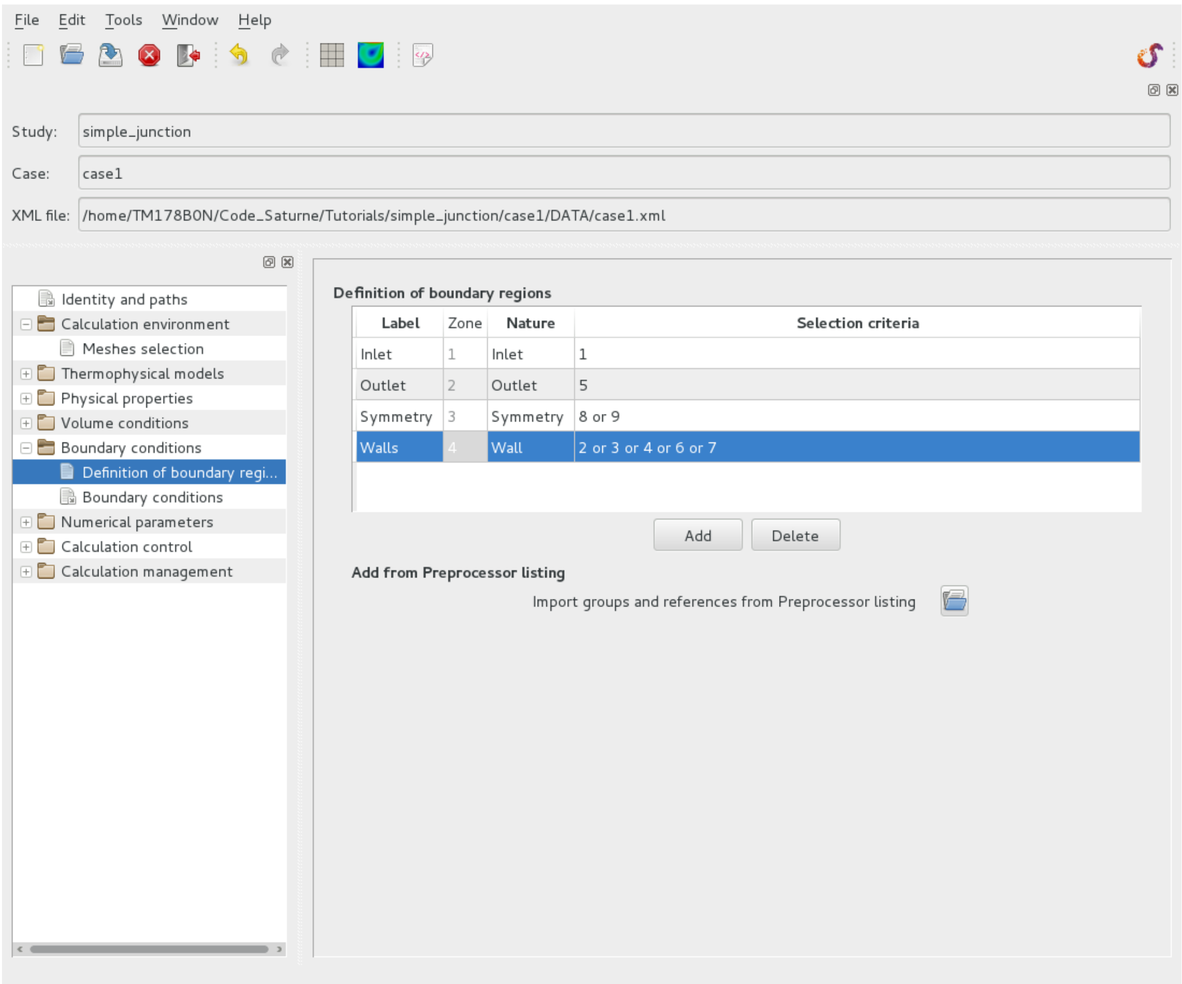

Figure III.23: Creation of boundary regions

Now that the boundary zones are defined, the boundary conditions assigned to them will be specified. Click on the **Boundary conditions** item to set the inlet boundary conditions for velocity, turbulence and themal scalar.

As shown on figure III.26, outlet and wall boundary zones also appear in the window.

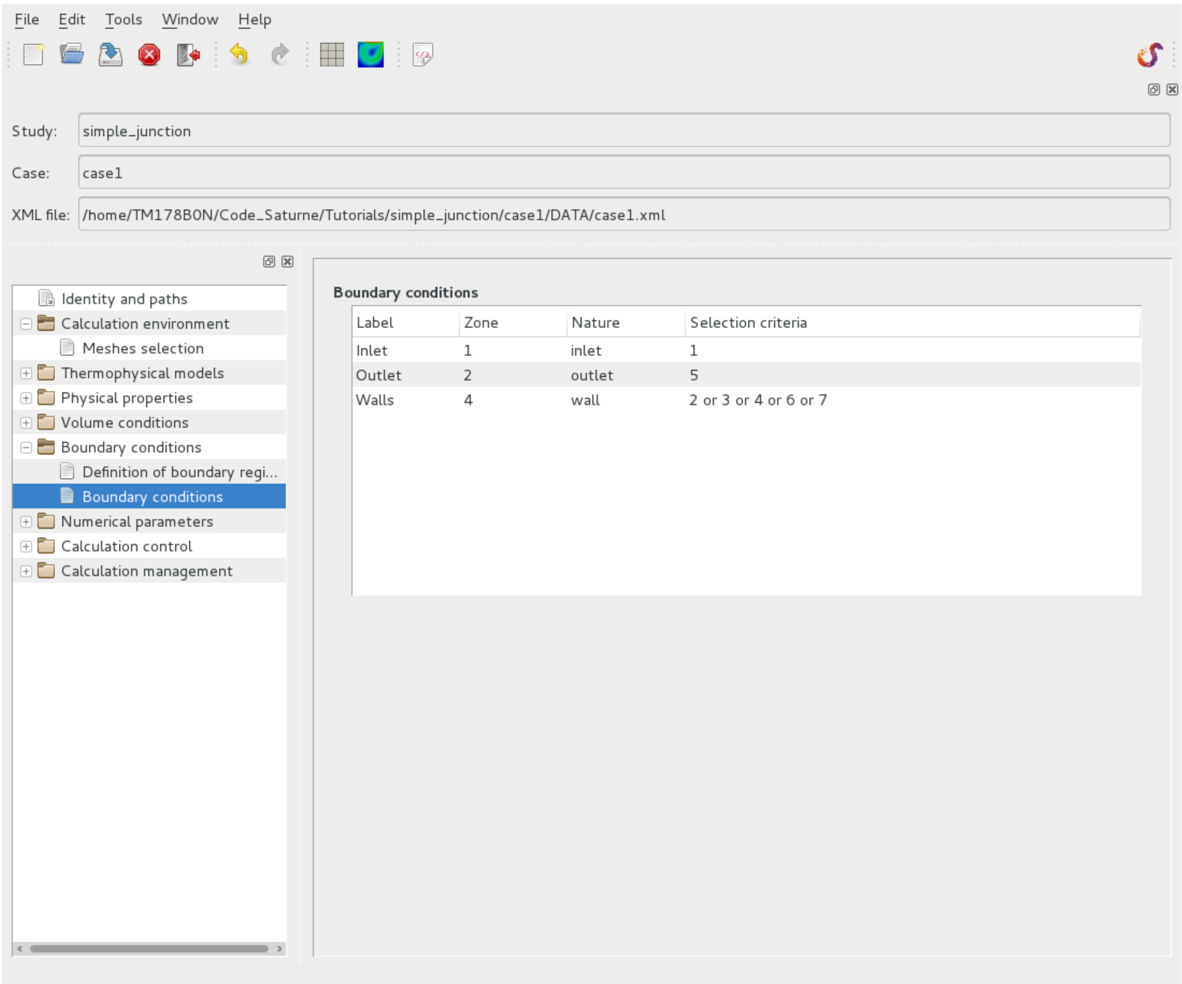

Figure III.24: Dynamic variables boundary conditions

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)
```

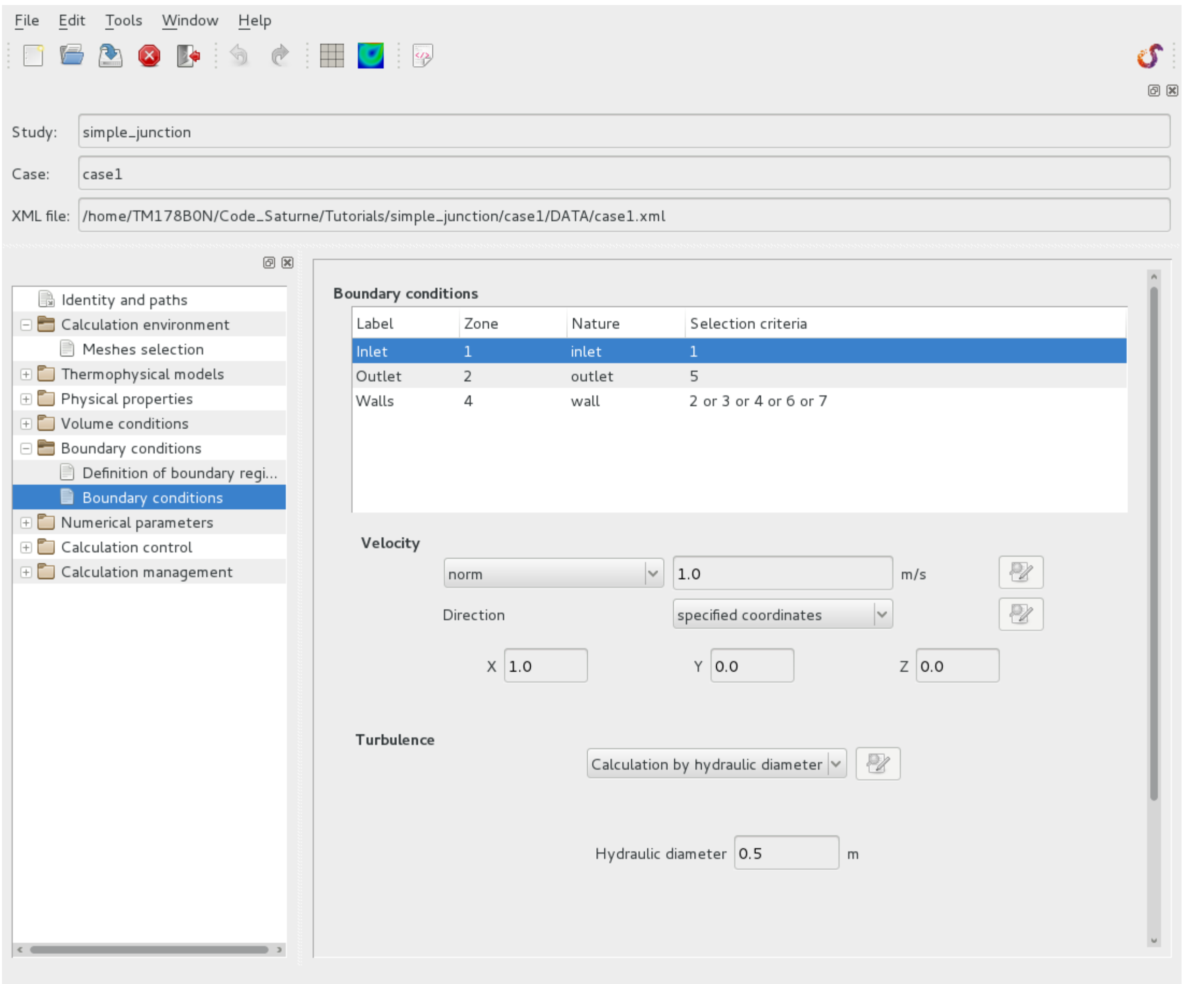

Figure III.25: Dynamic variables boundary conditions: inlet

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

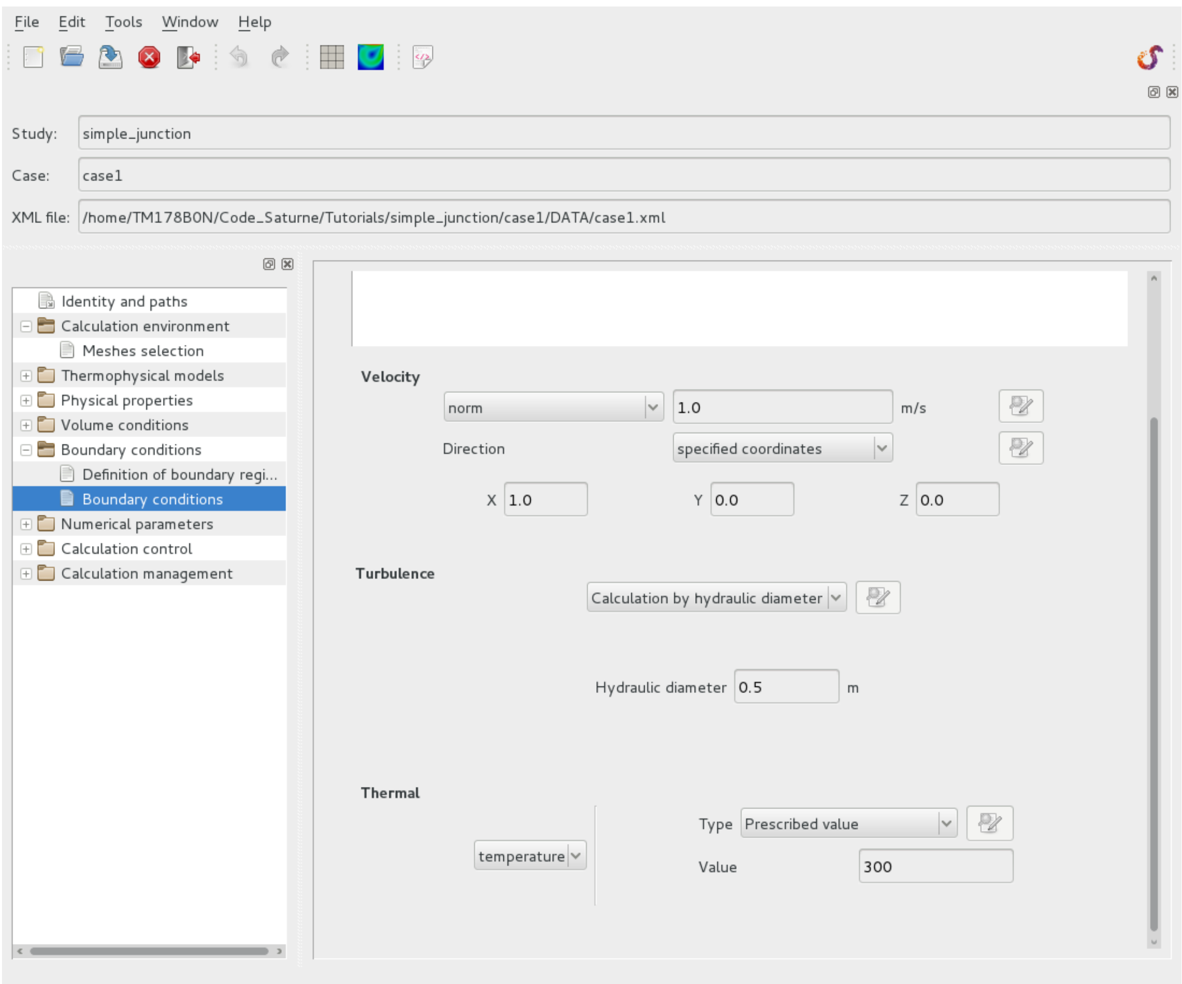

Figure III.26: Dynamic variables boundary conditions: inlet

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.

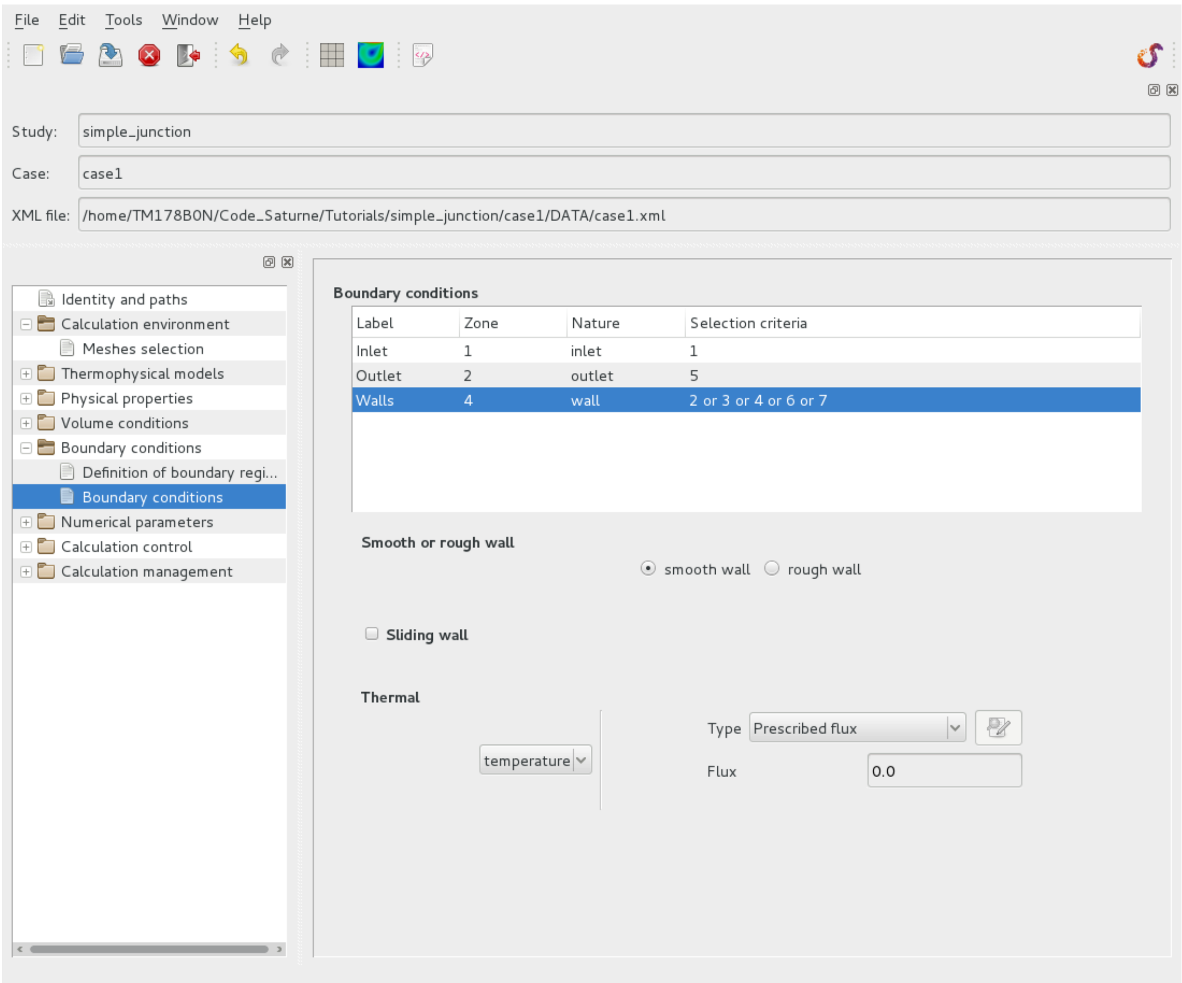

Figure III.27: Dynamic variables boundary: walls

The boundary conditions on the temperature are only applied on inlets, outlets and walls.

For the walls, three conditions are available:

```
- Prescribed value
- Prescribed flux
- Exchange Coefficient
```

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*.

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

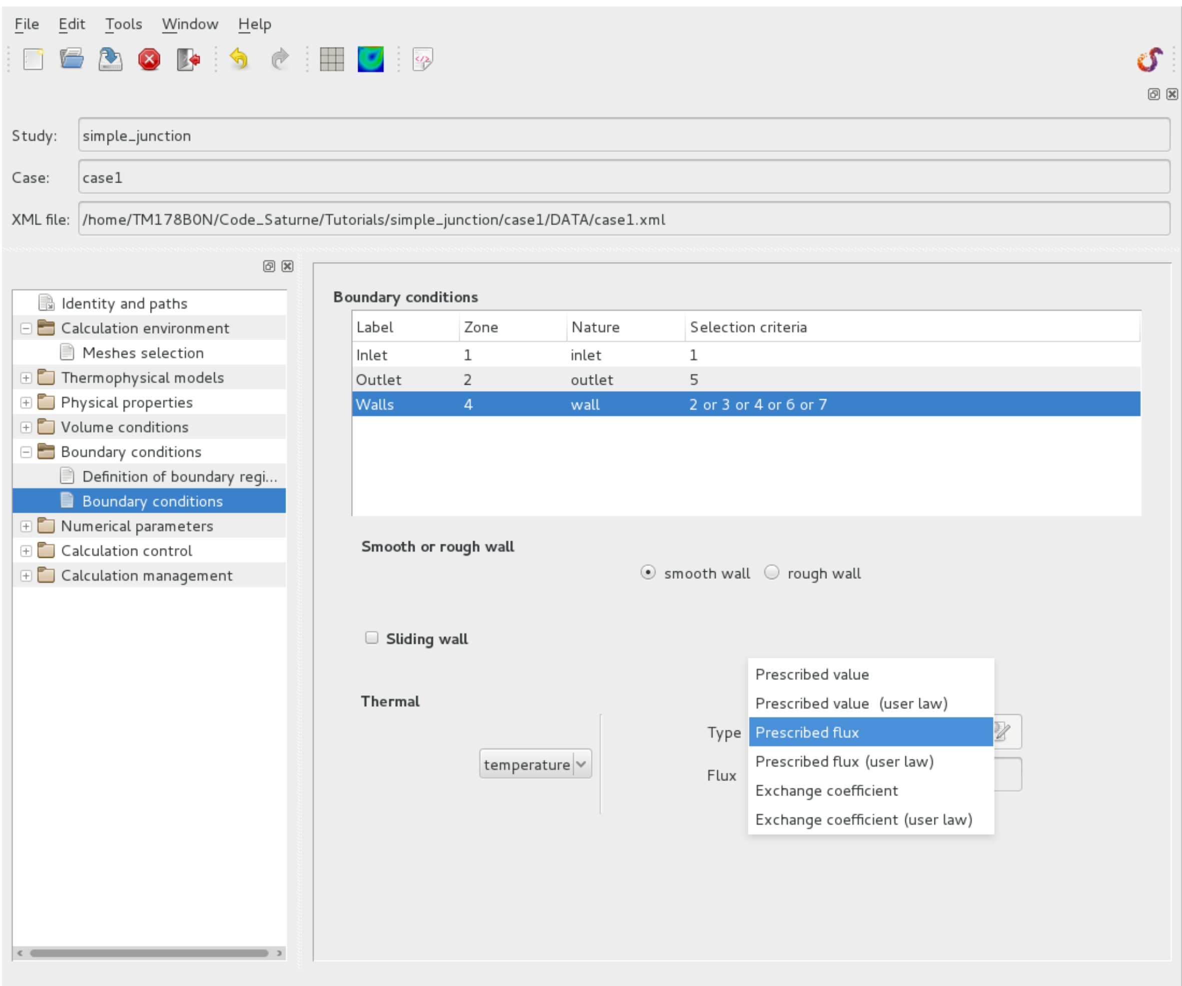

Figure III.28: Scalars boundaries: walls

The **Global parameters** need then to be specified, under the header **Numerical parameters**. In this case, the **SIMPLE** algorithm must be chosen.

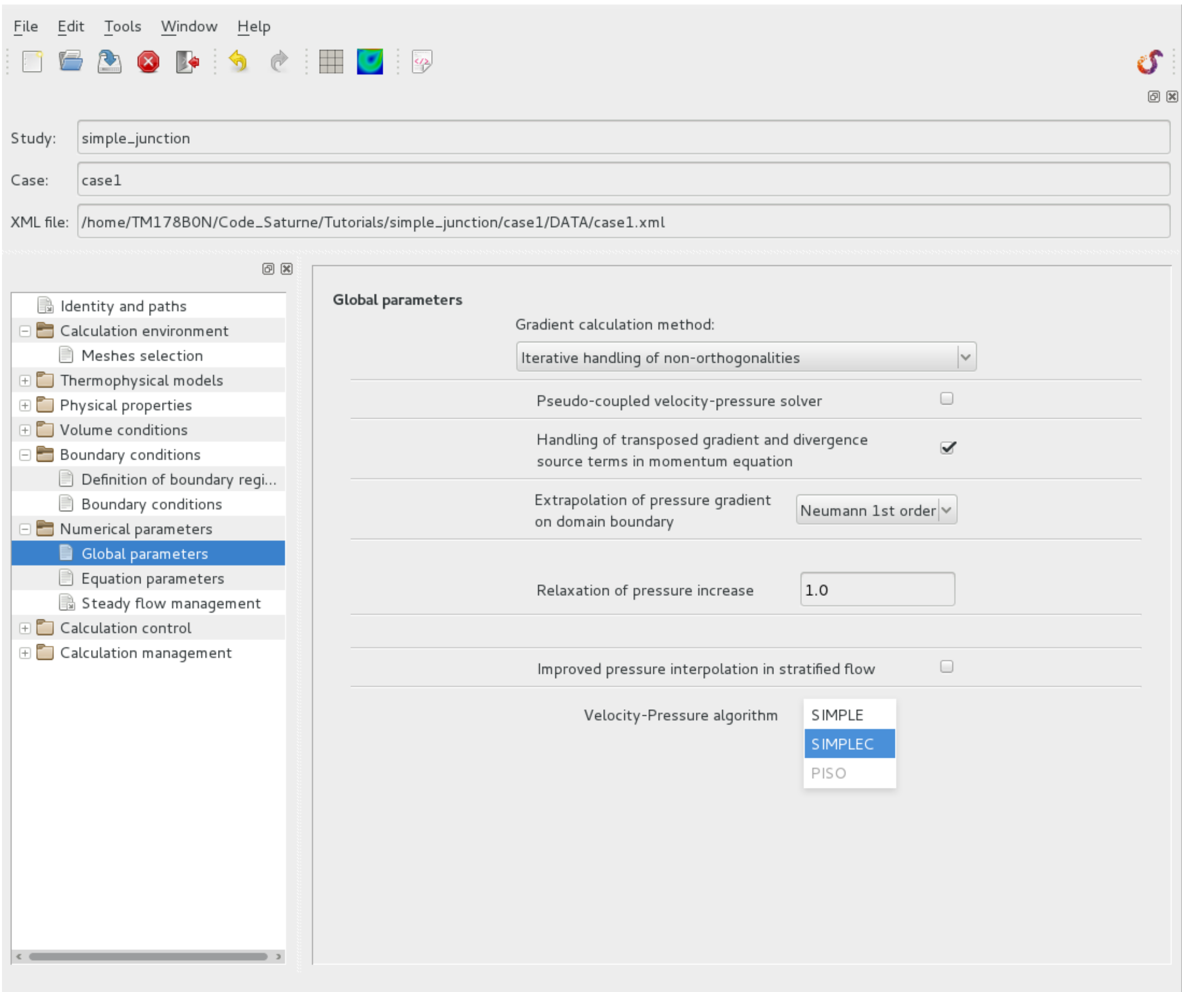

Figure III.29: Steady flow management

After selecting the **Equation parameters** item, the tab **Scheme** allows to change different more advanced numerical parameters.

In this case none of them should be changed from their default value.

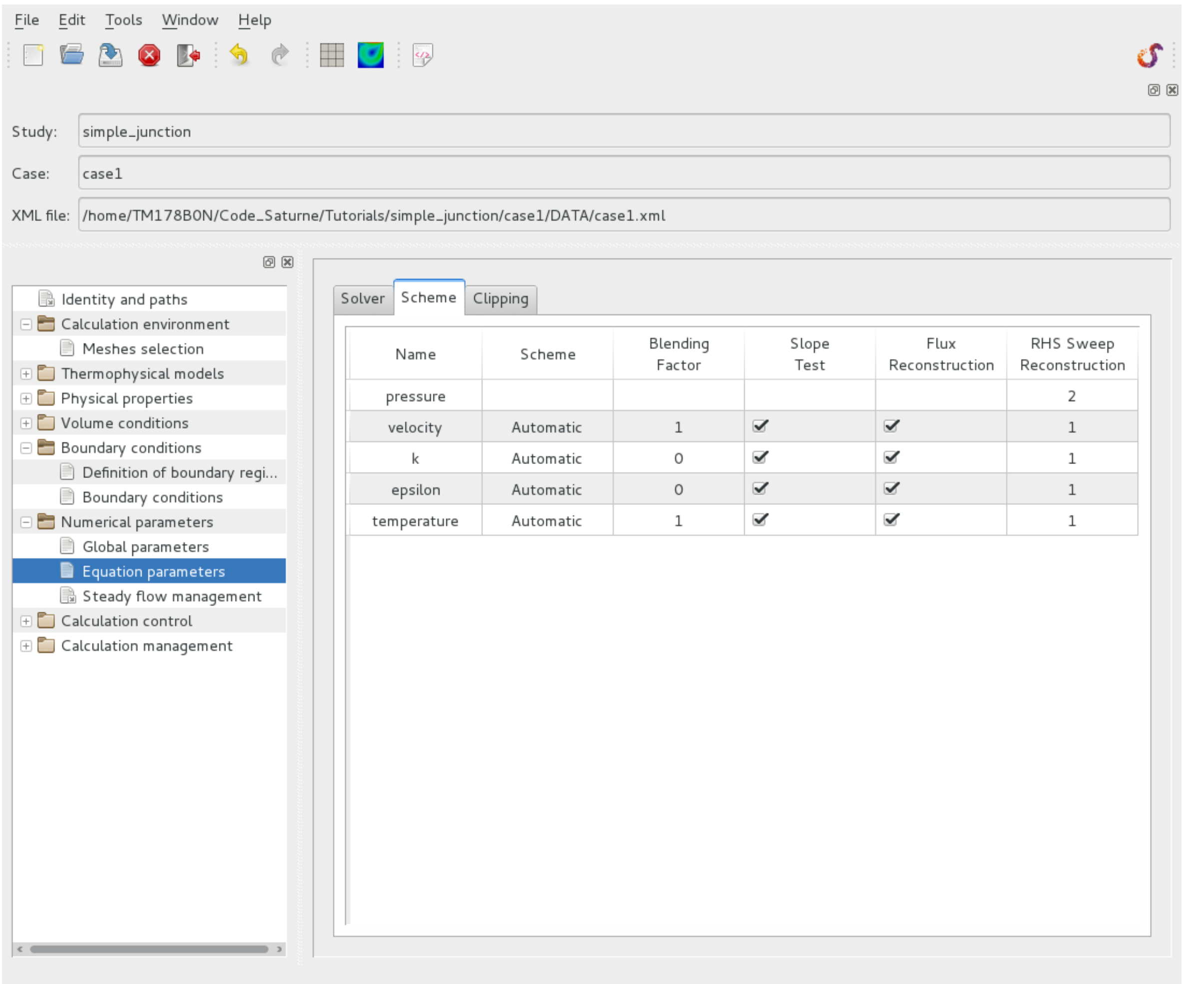

Figure III.30: Numerical parameters

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

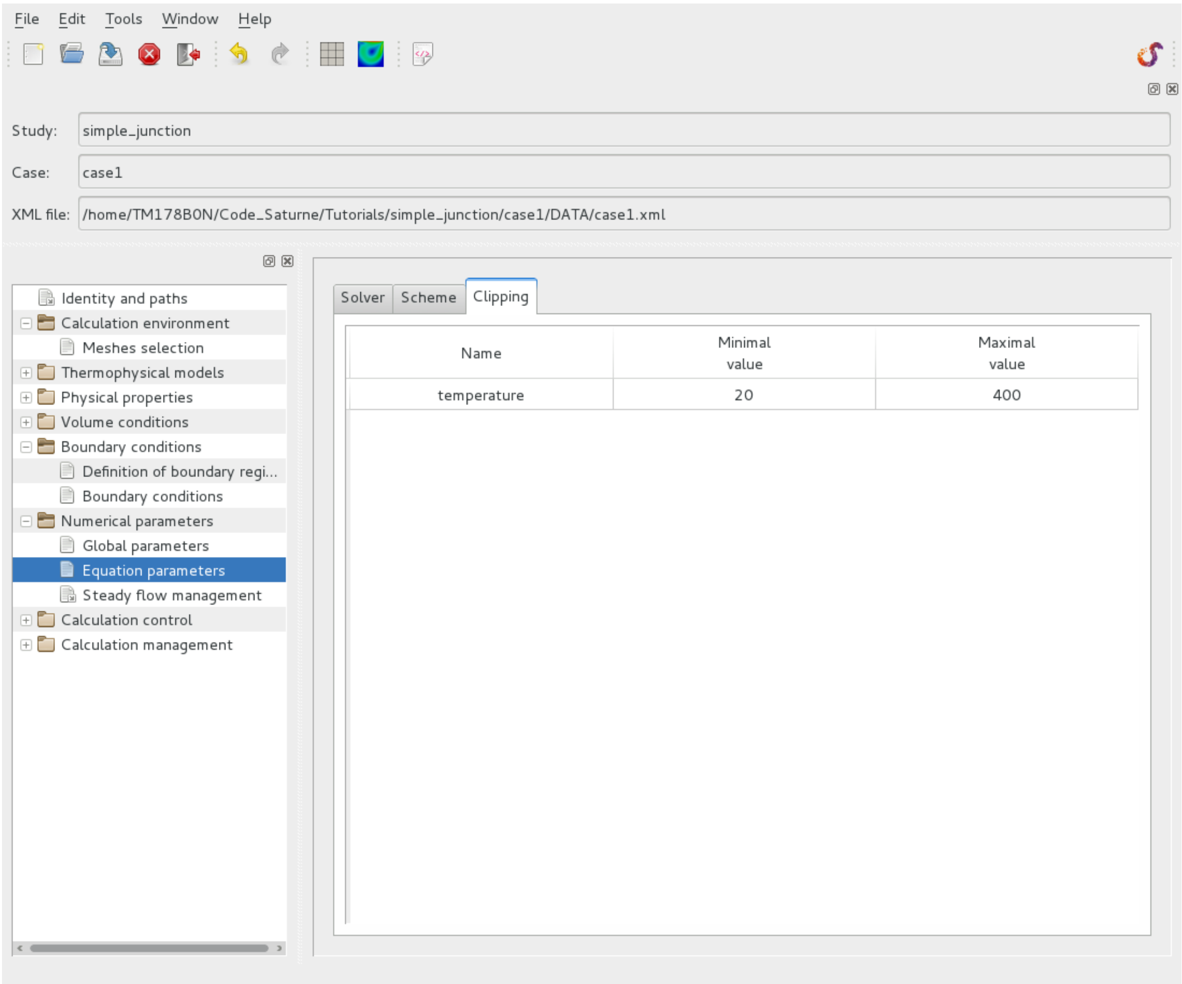

Figure III.31: Clipping

Go to the **Steady flow management** item to specify the number of iterations, **300** in this case. The relaxation coefficient is equal to **0.9** and the **Zero iterations option** will not be activated.

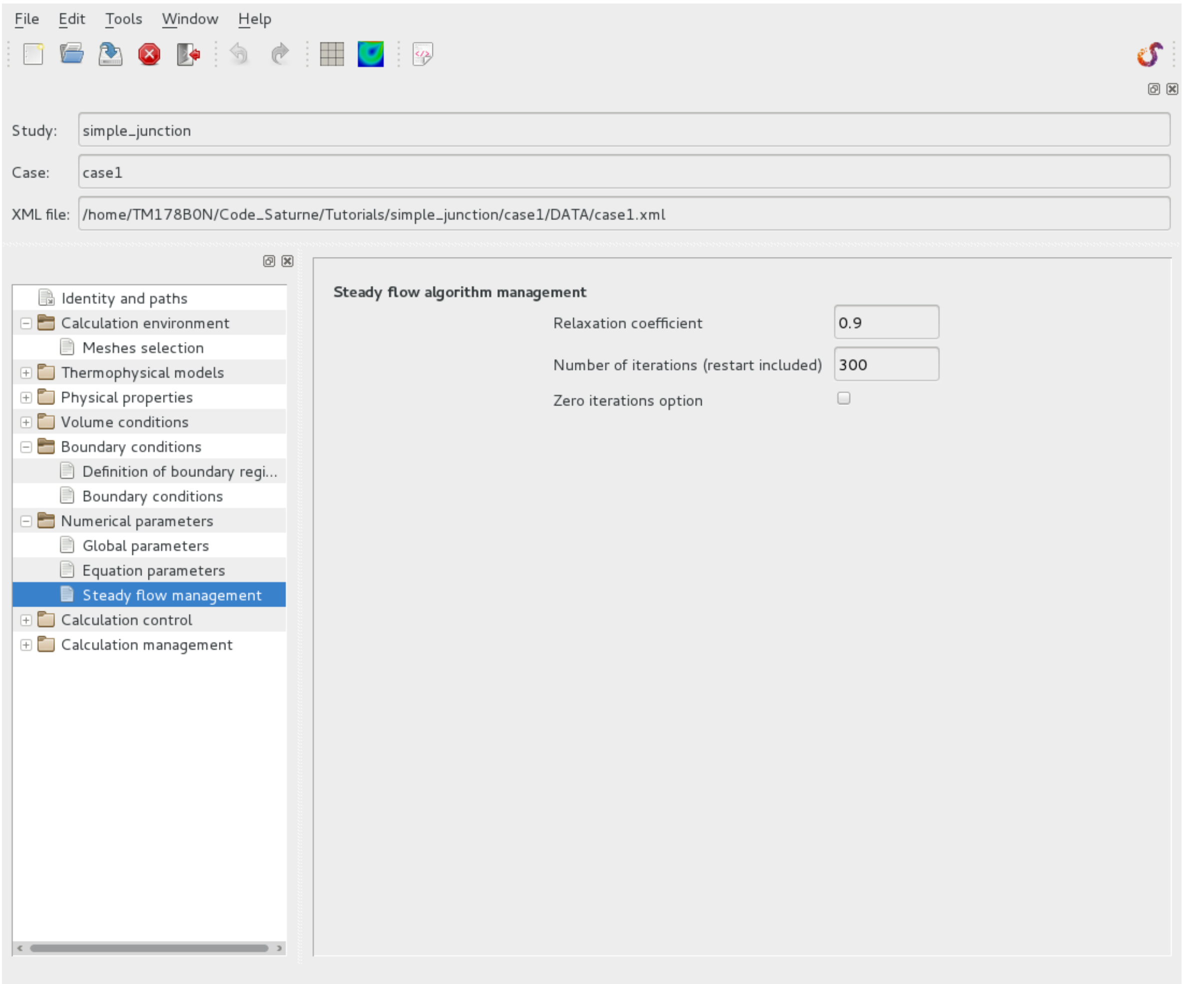

Figure III.32: Steady flow management

Under the heading **Calculation control**, click on the **Output control** item to 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.

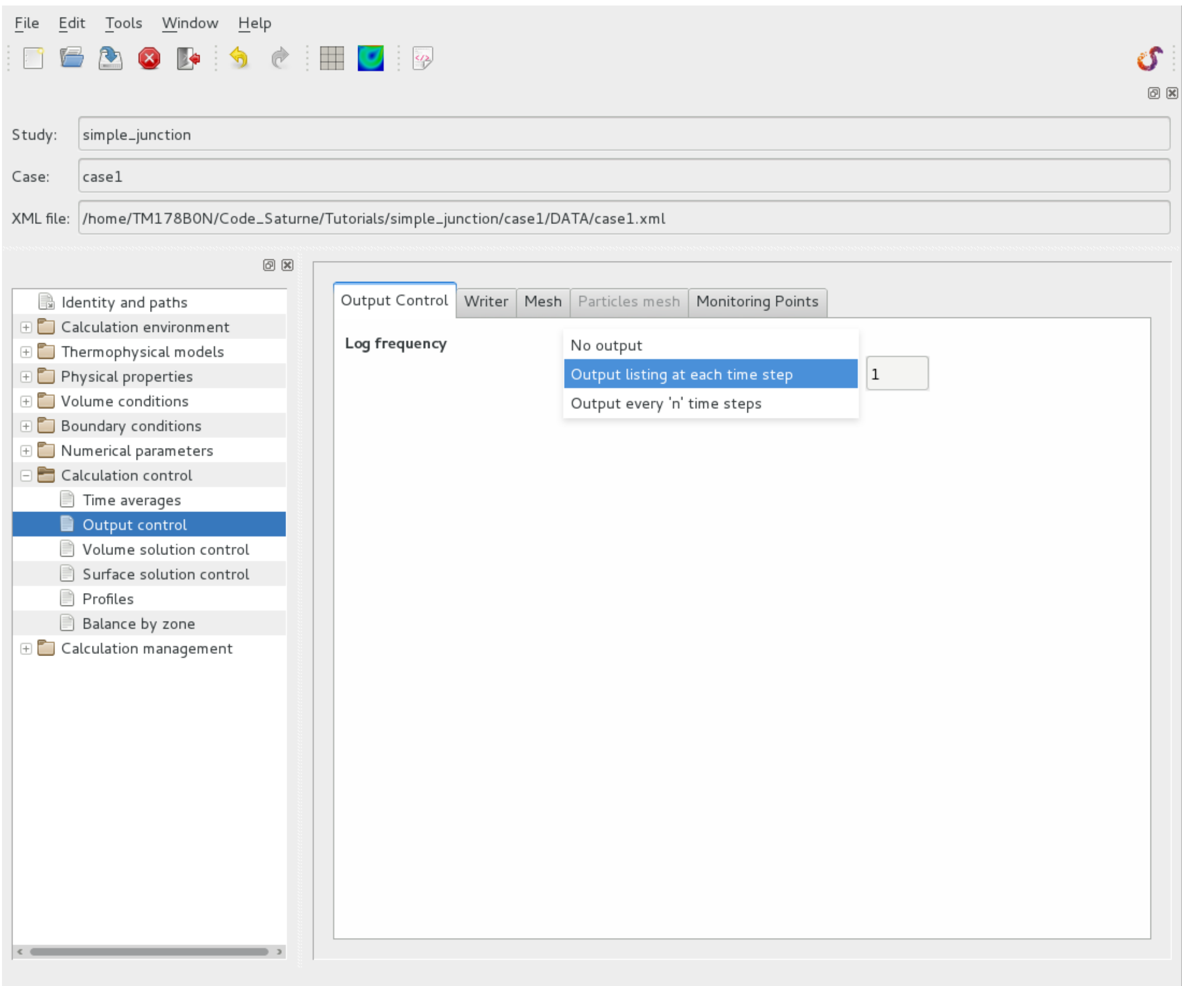

Figure III.33: Output control: output listing

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.

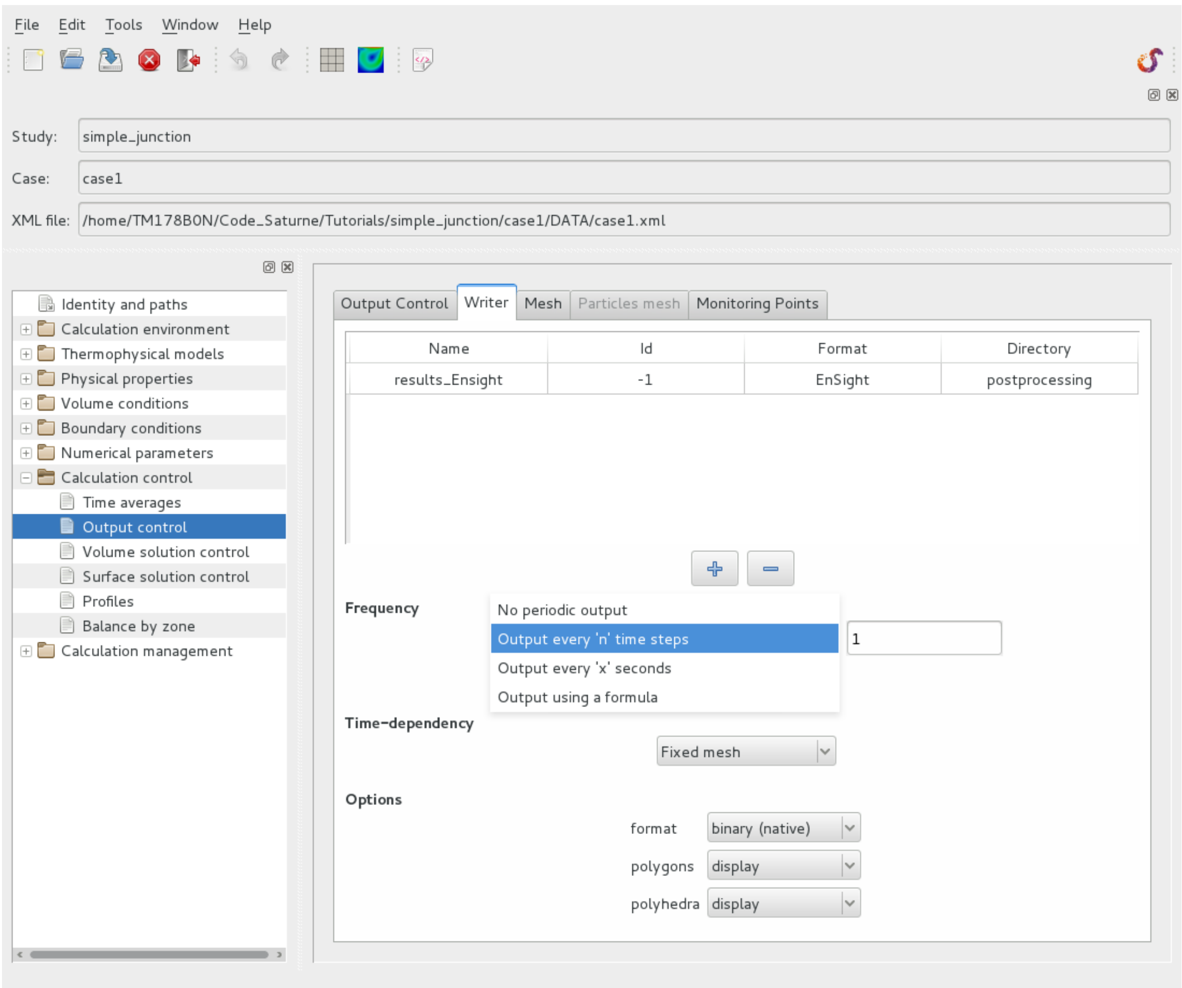

Figure III.34: Output control: post-processing

The other options are kept to their default value.

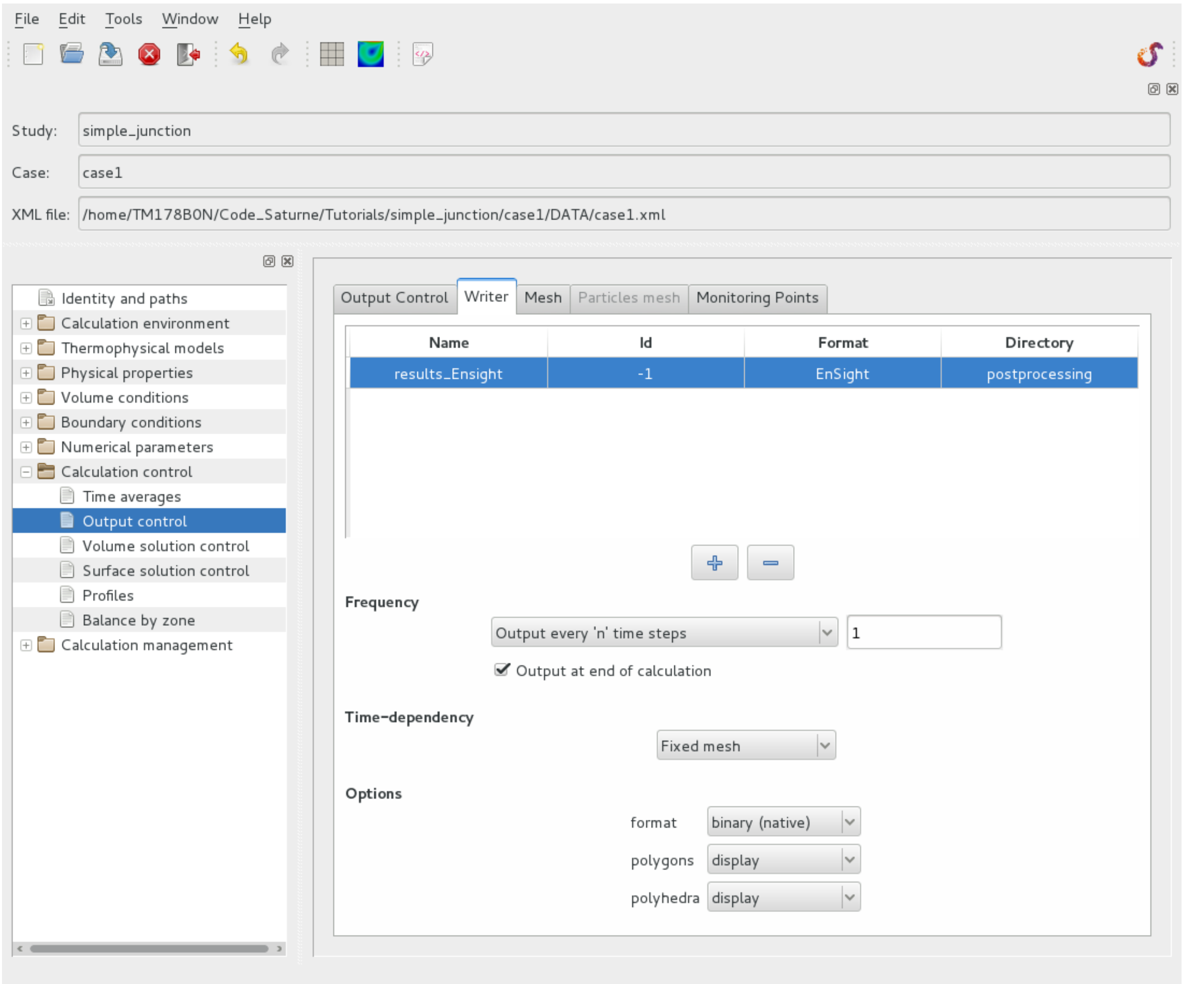

Figure III.35: Output control

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.

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.

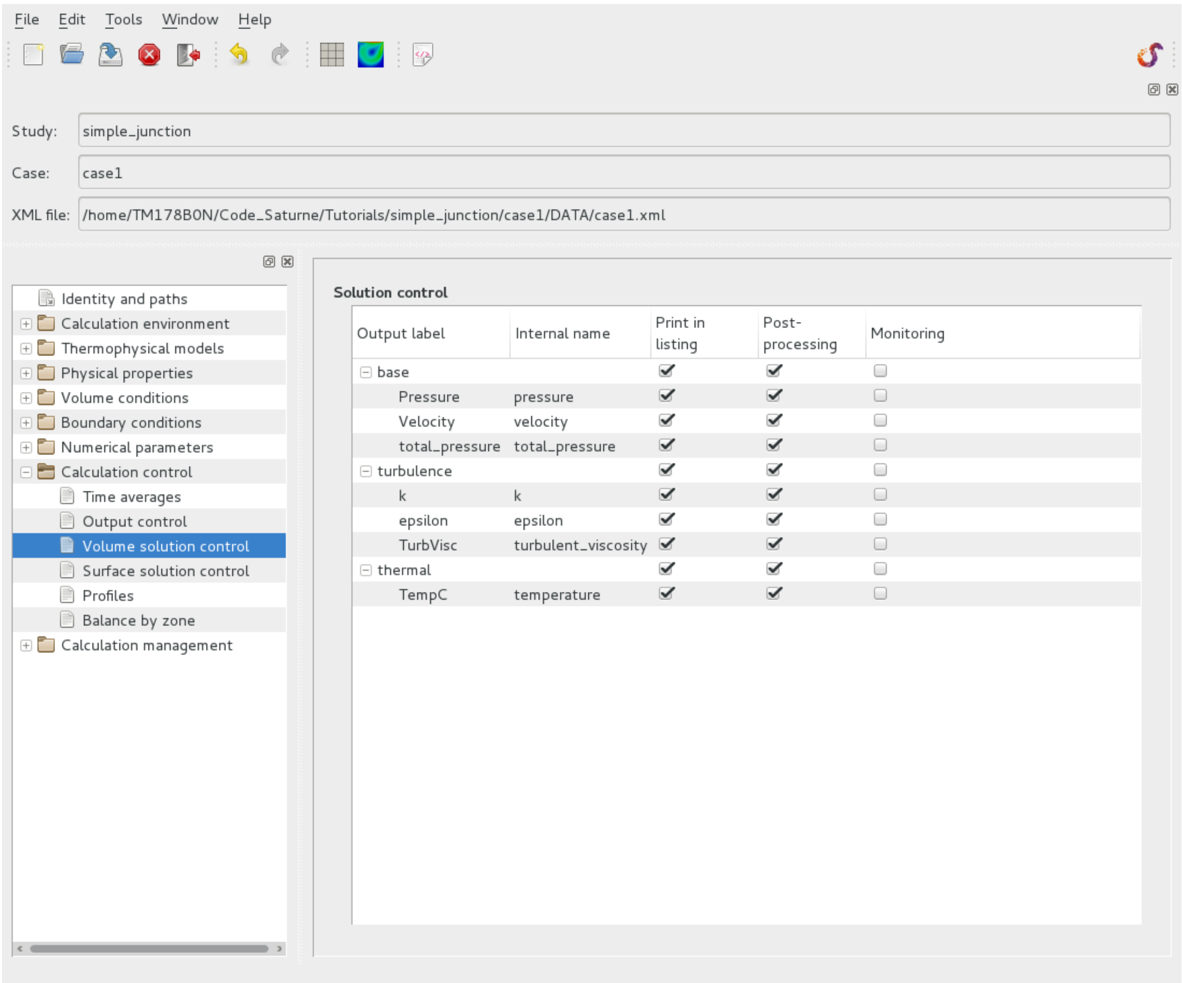

Figure III.36: Solution control

The **Start/Restart** item allows to start a new calculation from the results of a former one. It is not the case in the present calculation so nothing has to be modified.

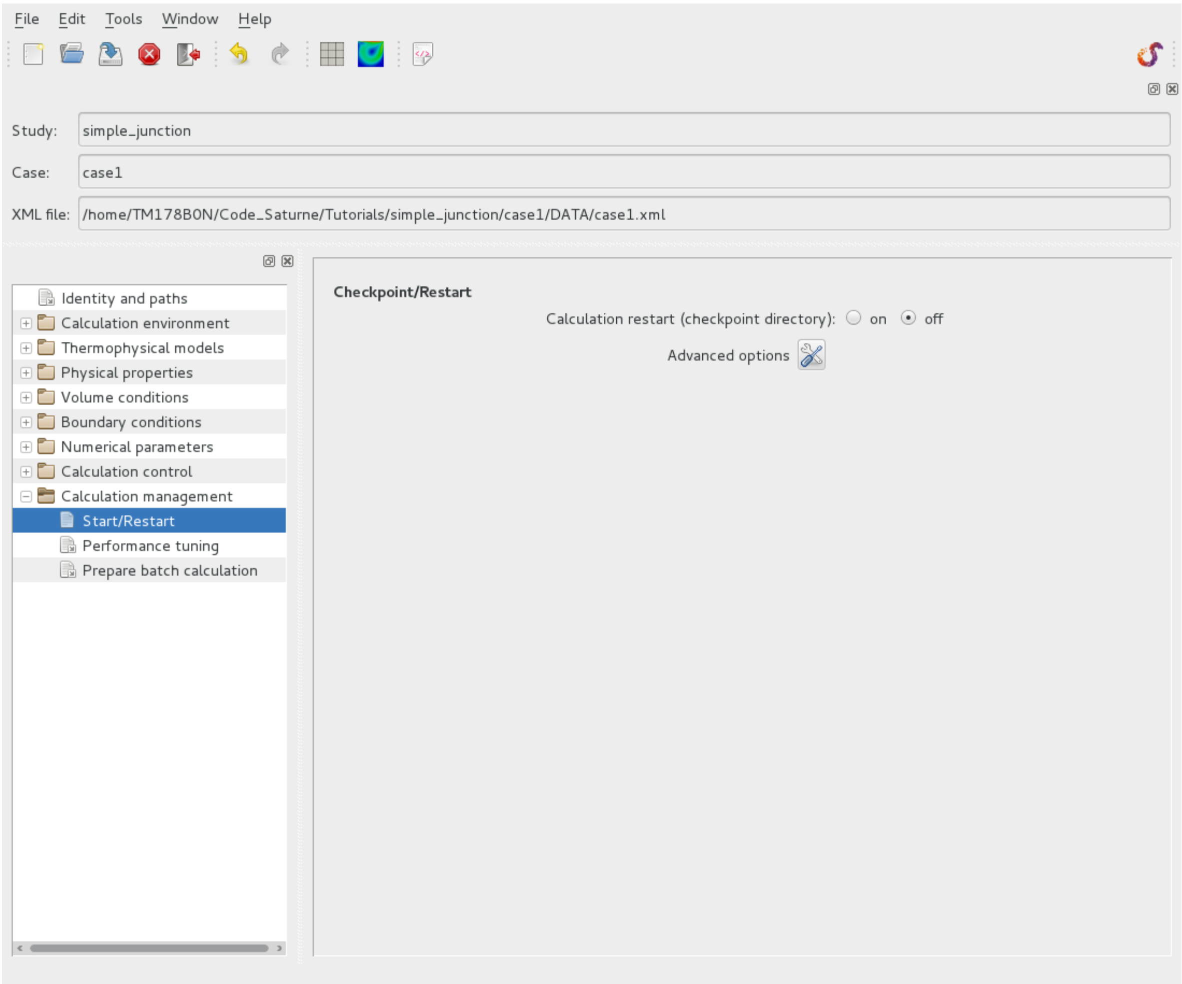

Figure III.37: Start/Restart

The final item, **Prepare batch calculation**, is used to prepare the launch script and, on certain architectures, launch the calculation.

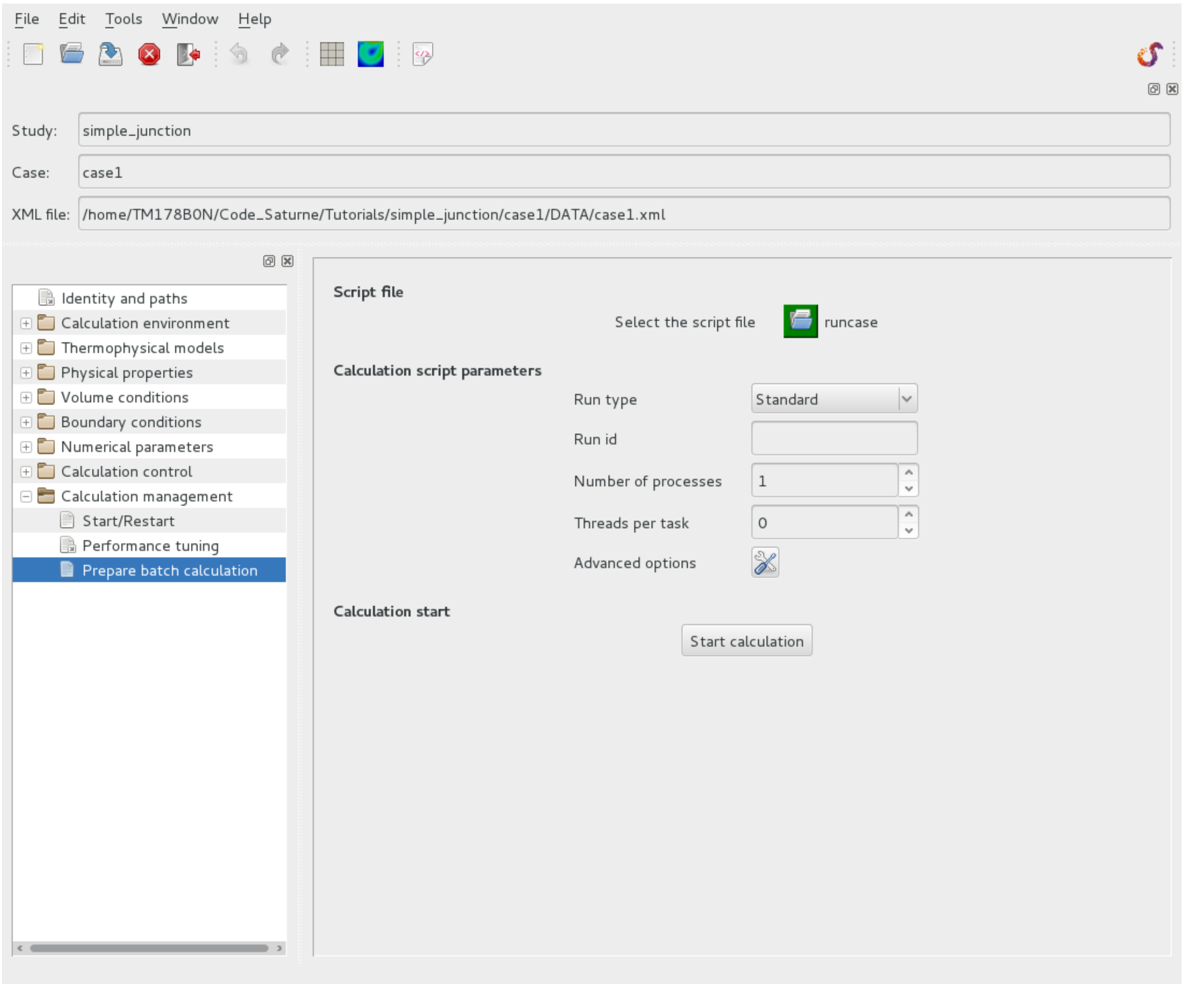

Figure III.38: Prepare batch calculation: computer selection

The default launch script is named `runcase` and is located in the `SCRIPTS/` directory. It is already loaded.

If a different launch script needed to be selected, this can be done by clicking on the icon next to **Select the script file**. Select the desired script and click on Open.

Remember to save the `xml` file before opening the launch script.

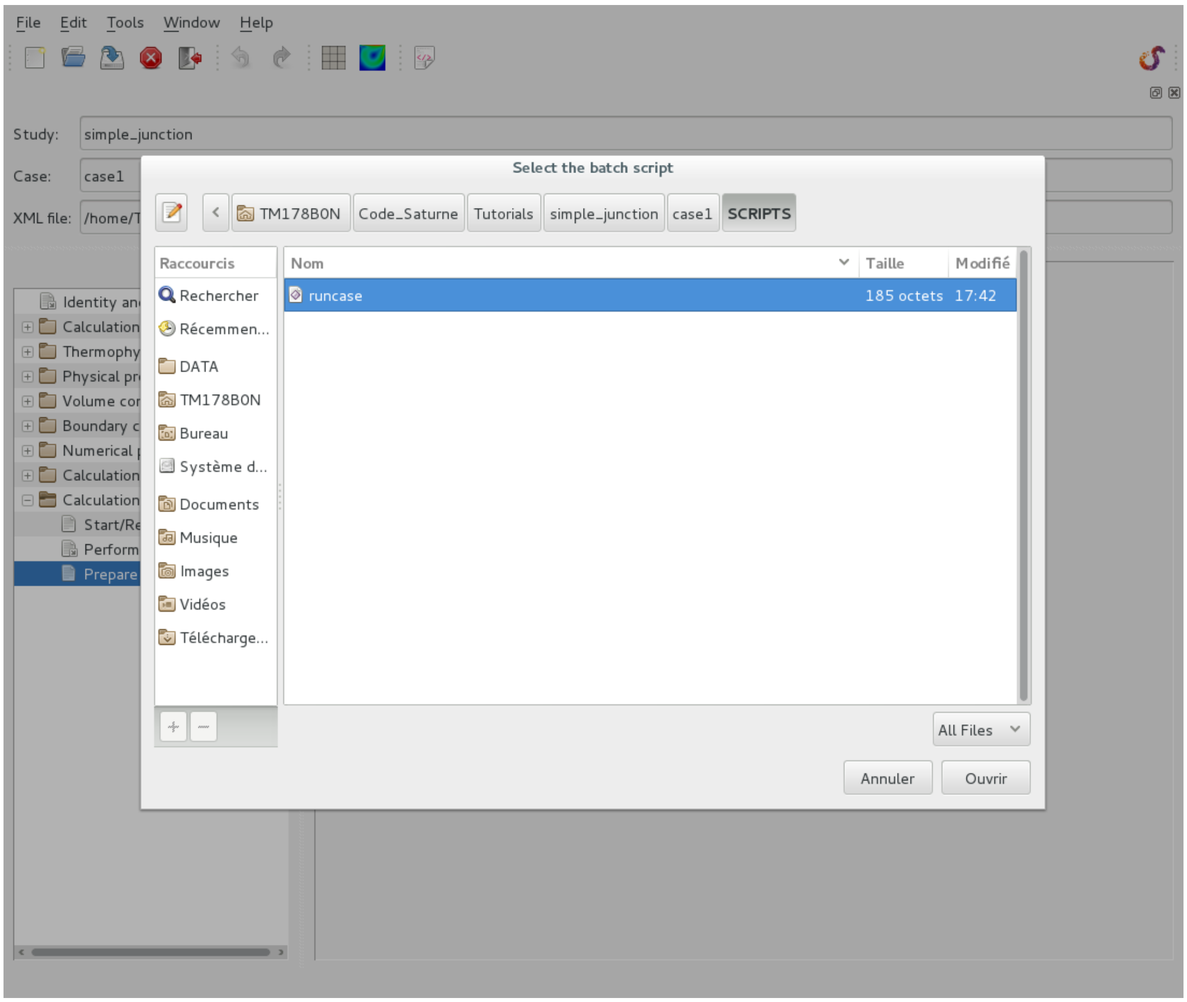

Figure III.39: Prepare batch calculation: batch script file selection

When the script is selected, new options will appear. On this calculation, the number of processors used will be left to 1.

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 **Start calculation**. The results will be copied in the `RESU/` directory.

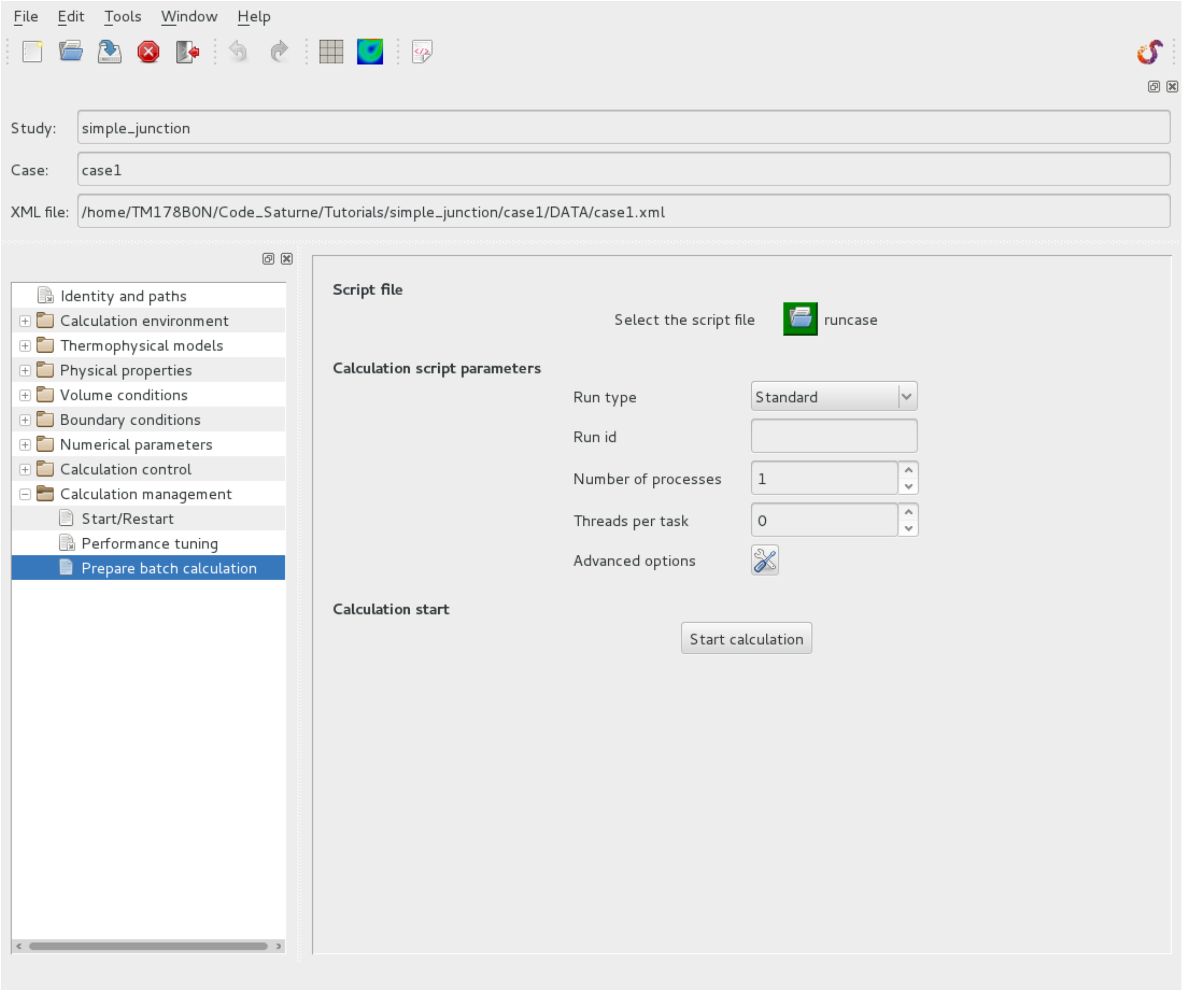

Figure III.40: Prepare batch calculation: execution

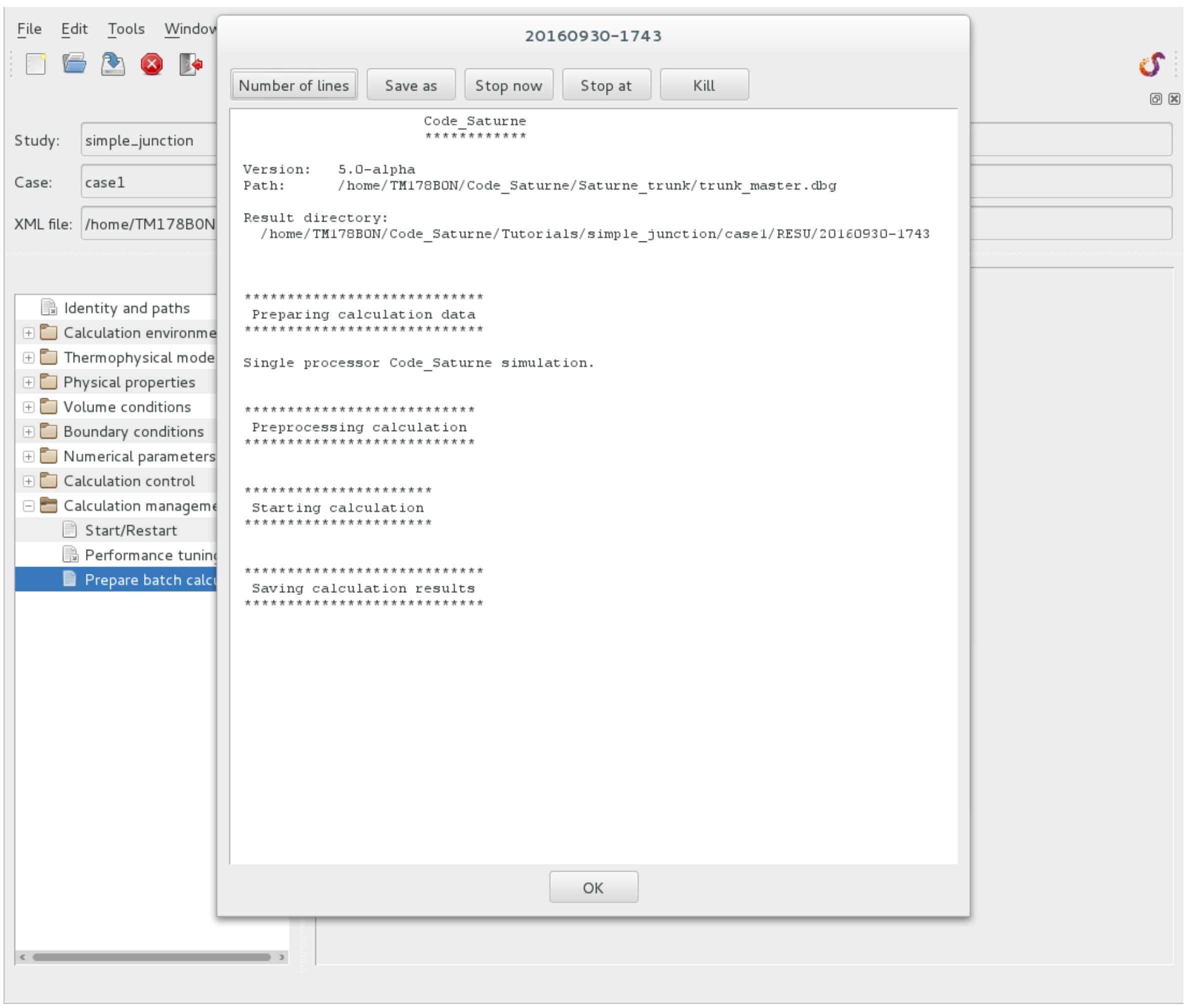

Figure III.41: Run