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: stratified 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 of a stratified flow in a T-junction and guides the future *Code_Saturne* user step by step into the preparation and the computation of the case.

The test case directories, containing the necessary meshes and data are available in the `examples/3-stratified_junctio` directory in *Code_Saturne* source directory.

This tutorial focuses on the procedure and the preparation of the *Code_Saturne* computations with or without SALOME. 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

Code_Saturne is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version. *Code_Saturne* is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

Part II

Stratified junction

1 Study description

1.1 Objective

The aim of this case is to train the *Code_Saturne* user on a simplified but real 3D computation. It corresponds to a stratified flow in a T-junction. The test case will be used to present some advanced post-processing techniques.

1.2 Description of the configuration

The configuration is based on a real mock-up designed to characterize thermal stratification phenomena and associated fluctuations. The geometry is shown on figure II.1.

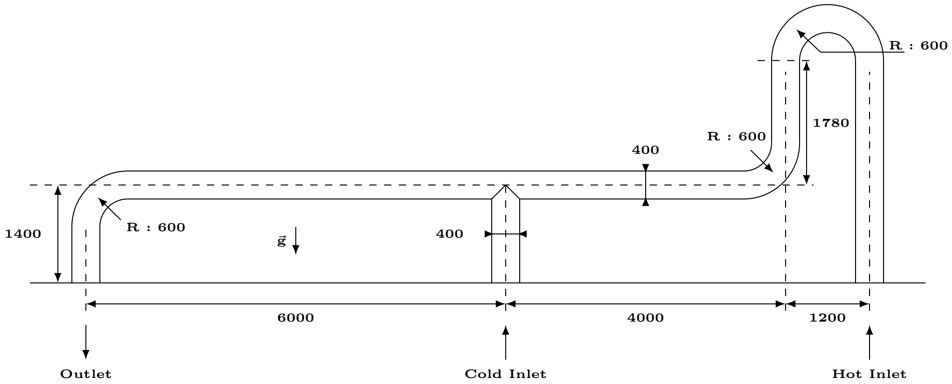


Figure II.1: Geometry of the case, with dimensions in mm

There are two inlets, a hot one in the main pipe and a cold one in the vertical nozzle. The volumic flow rate is identical in both inlets. It is chosen small enough so that gravity effects are important with respect to inertia forces. Therefore cold water creeps backwards from the junction towards the elbow until the flow reaches a stable stratified state.

1.3 Geometry

Characteristics of the geometry:

Diameter of the pipe	$D_b = 0.40\ m$

1.4 Data settings

The boundary conditions of the flow are as follows:

Cold branch volume flow rate	$Dv_{cb} = 4\ l.s^{-1}$
Hot branch volume flow rate	$Dv_{hb} = 4\ l.s^{-1}$
Cold branch temperature	$T_{cb} = 18.26$°C
Hot branch temperature	$T_{hb} = 38.5$°C

The initial water temperature in the domain is equal to 38.5°C.

Water specific heat and thermal conductivity are considered constant and calculated at 18.26°C and $10^5\ Pa$:

- heat capacity: $C_p = 4{,}182.88\ J.kg^{-1}.°\mathrm{C}^{-1}$
- thermal conductivity: $\lambda = 0.601498\ W.m^{-1}.°\mathrm{C}^{-1}$

The water density and dynamic viscosity are variable with the temperature. The functions are given below.

2 Mesh characteristics

The mesh used in the actual study had 125 000 elements. It has been coarsened for this example in order for calculations to run faster. The mesh used here contains 16 320 elements.

Type: unstructured mesh

Coordinates system: cartesian, origin on the middle of the horizontal pipe at the intersection with the nozzle.

Mesh generator used: SIMAIL

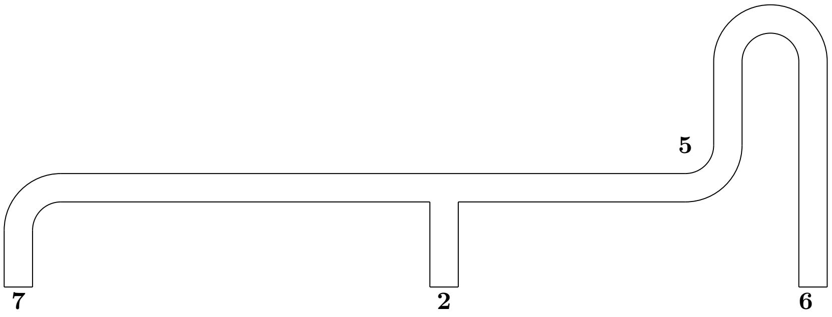


Figure II.2: References of the boundary faces

3 Computation of the Stratified junction configuration

In this case, advanced post-processing features will be used. A specific post-processing sub-mesh will be created, containing all the cells with a temperature lower than 21°C, so that it can be visualized (with ParaView for instance). The variable **temperature** will be post-processed on this sub-mesh. A 2D clip plane will also be extracted along the symmetry plane of the domain and the temperature will be written on it.

3.1 Options and models

The following options are considered for the case:

Modeling feature	**choice**
Flow type	unsteady flow
Time step	variable in time and uniform in space
Turbulence model	$k - \varepsilon$ LP
Thermal model	Temperature (°C)
Physical properties	uniform and constant for specific heat and thermal conductivity and variable for density and dynamic viscosity

References	Type of boundary conditions
2	Cold inlet
6	Hot inlet
7	Outlet
5	Wall

Table II.1: Boundary faces colors and associated references

3.2 Initial and boundary conditions

The temperature should be initialized at 38.5°C in the whole domain.

The boundary conditions are defined as follows:

- **Flow inlet**: Dirichlet condition
  - Velocity of 0.03183 $m.s^{-1}$ for both inlets
  - Temperature of 38.5°C for the hot inlet
  - Temperature of 18.6°C for the cold inlet
- **Outlet**: default value
- **Walls**: default value

Figure II.2 shows the references used for boundary conditions and table II.1 defines the which type of boundary conditions is imposed for each reference.

3.3 Physical properties

In this case the density and the dynamic viscosity are functions of the temperature.

The following variation law for te density needs to be specified in the Graphical User Interface:

$$\rho = T(AT + B) + C \tag{II.1}$$

where ρ is the density, T is the temperature, $A = -4.0668 \times 10^{-3}$, $B = -5.0754 \times 10^{-2}$ and $C = 1\,000.9$.

For the dynamic viscosity, the variation law is:

$$\mu = T(T(AMT + BM) + CM) + DM \tag{II.2}$$

where μ is the dynamic viscosity, T is the temperature, $AM = -3.4016 \times 10^{-9}$, $BM = 6.2332 \times 10^{-7}$, $CM = -4.5577 \times 10^{-5}$ and $DM = 1.6935 \times 10^{-3}$.

In order for the variable density to have an effect on the flow, gravity must be set to a non-zero value. $\underline{g} = -9.81\underline{e}_z$ will be specified in the Graphical Interface.

3.4 Time stepping parameters

All the parameters necessary to this study can be defined through the Graphical Interface, except the advanced post-processing features, that have to be specified in user routines.

time stepping parameters	
Reference time step	0.1 s
Number of iterations	100
Maximal CFL number	20
Maximal Fourier number	60
Minimal time step factor	0.01 s
Maximal time step factor	70 s
Time step maximal variation	0.1

The time step limitation by gravity effects will also be enabled.

3.5 Output management

In a first step, standard options for output management will be used. Four monitoring points will be created at the following coordinates:

Probe	**x(m)**	**y(m)**	**z(m)**
1	0.010025	0.01534	-0.011765
2	1.625	0.01534	-0.031652
3	3.225	0.01534	-0.031652
4	3.8726	0.047481	7.25

Two vertical temperature profiles will be extracted, at the following locations:

Profile	**x(m)**	**y(m)**	**z(m)**
`profil16`	1.6	0	$-0.2 \leqslant z \leqslant 0.2$
`profil32`	3.2	0	$-0.2 \leqslant z \leqslant 0.2$

A period of 10 will be associated to the output writer.

3.6 User routines for advanced post-processing

The following files must to be copied from the folder 🗀`SRC/REFERENCE` into the folder 🗀`SRC`[1]:

- `cs_user_postprocess.c`;
- `cs_user_postprocess_var.f90`.

In this test case, advanced post-processing features will be used. A clip plane will be created, along the symmetry plane of the domain, on which the temperature will be written. This plane will be added to the standard **writer** (*i.e.* it will be an extra part in the standard `RESULTS.case` output). The periodicity of output on the standard writer will be 10 iterations.

An additional writer will also be created, with a periodicity of 5 iterations. It will only contain one part (*i.e.* one sub-mesh): the set of cells where the temperature is lower than 21°C. The temperature will be written on this part. The interest of this part is that it is time dependent as for the cells it contains.

The following user functions and subroutines will be used:

- **cs_user_postprocess_meshes** (in `cs_user_postprocess.c`)
  This function is called only once, at the beginning of the calculation. It allows to define the different writers and parts.

---

[1] Only when they appear in the 🗀`SRC` directory will they be taken into account by the code.

In this function, adapt the block using the **cs_post_define_volume_mesh_by_func**, replacing `He_fraction_05` with `T_lt_21` (do not forget to set the enclosing test to `true`). If the argument matching **the automatic variables output** is set to **true**, all variables (including temperature) postprocessed on the main output will be added to this one. For finer control, we set it to **false** here, and we will use a user-defined output with **cs_user_postprocess_var**. The associated writer list should contain writer 1, which may be created either using the GUI, or the **cs_user_postprocess_writers** (in the same file). Make sure this writers allows for **transient connectivity**. The **_he_fraction_05_select** near the beginning of the file must also be adapted, renaming it to **_t_lt_21_select**, and adapting its contents (mainly calling **cs_field_by_name** on **temperature** instead of **He_fraction**, and replacing **$>$ 5.e-2** with **$<$ 21**). This selection function is called automatically at each output time step so as to update the selected sub-mesh.

- `cs_user_postprocess_var.f90`
  This routine is called at each time step. It allows to specify which variable will be written on which part (in this case, temperature).

3.7 Results

Figure II.3 shows the evolution of the temperature in the domain at different time steps. The evolution of the stratification is clearly visible.

Figure II.4 shows the cells where the temperature is lower than 21°C. It is not an isosurface created from the full domain, but a visualization of the full sub-domain created through the post-processing routines.

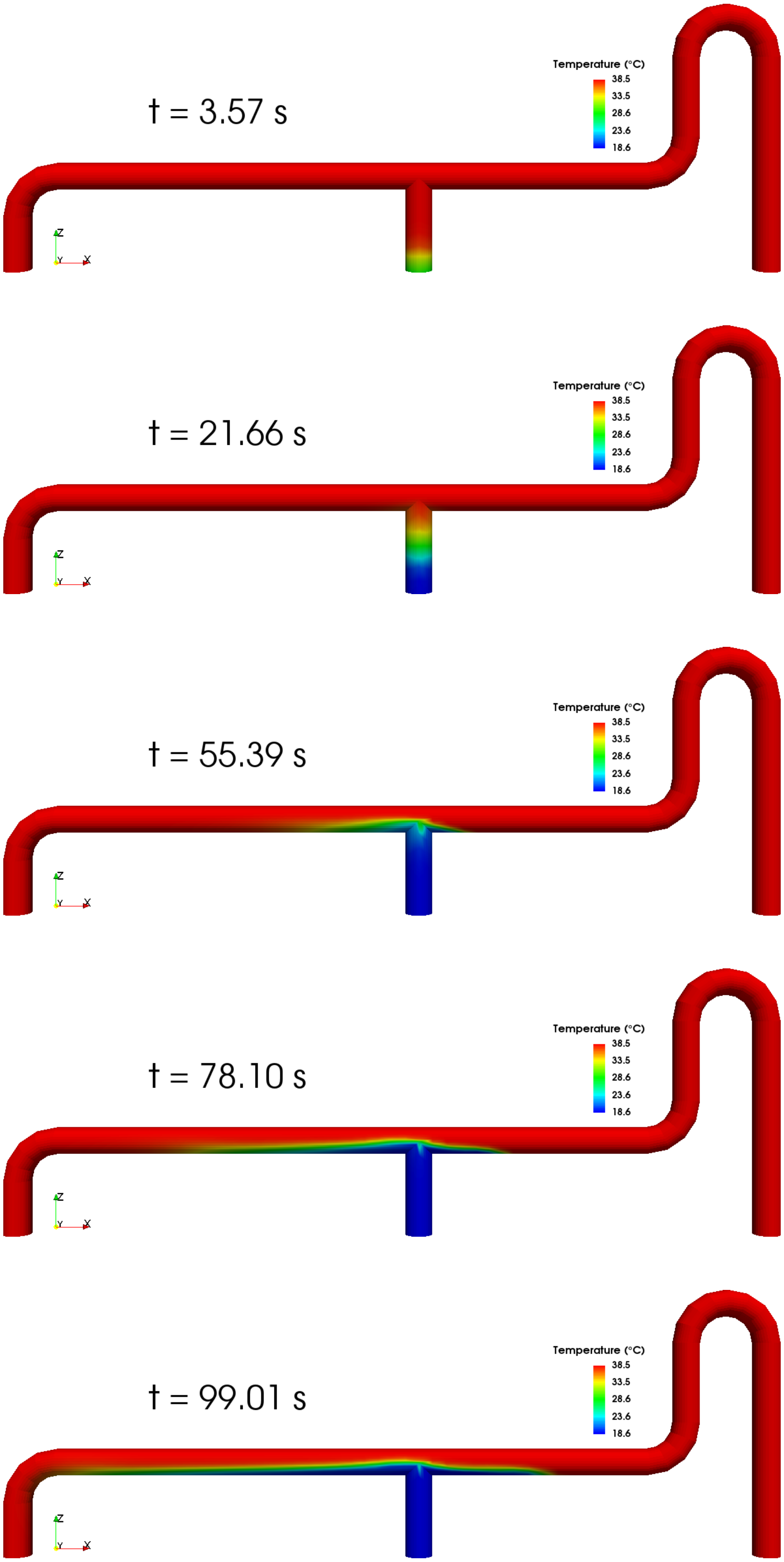


Figure II.3: Evolution of the temperature

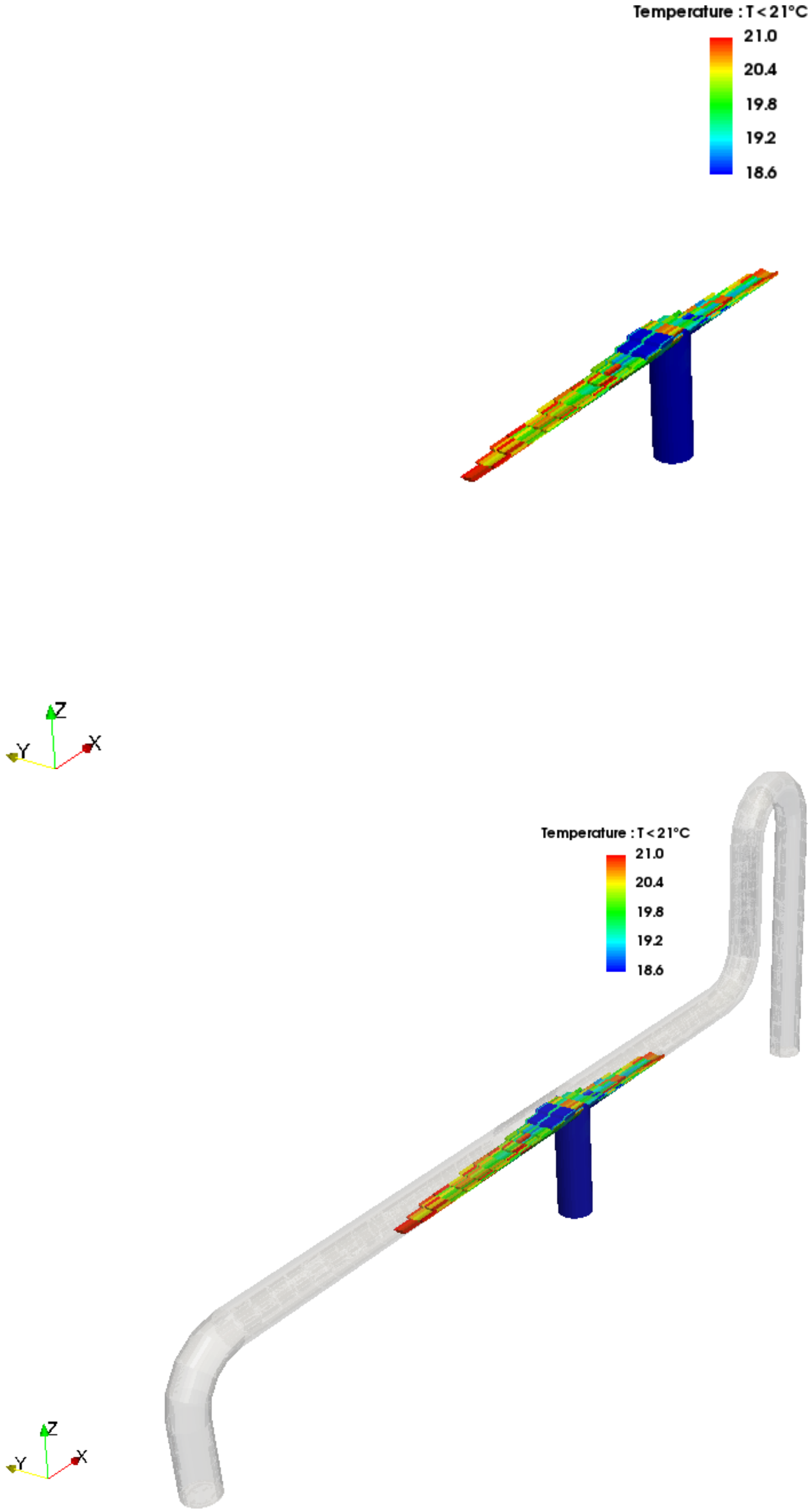


Figure II.4: Sub-domain where the temperature is lower than 21°C (upper figure) and localization in the full domain (lower figure)

Part III

Step by step solution

1 Detailed tutorial step by step

1.1 Creation of the study in a terminal

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

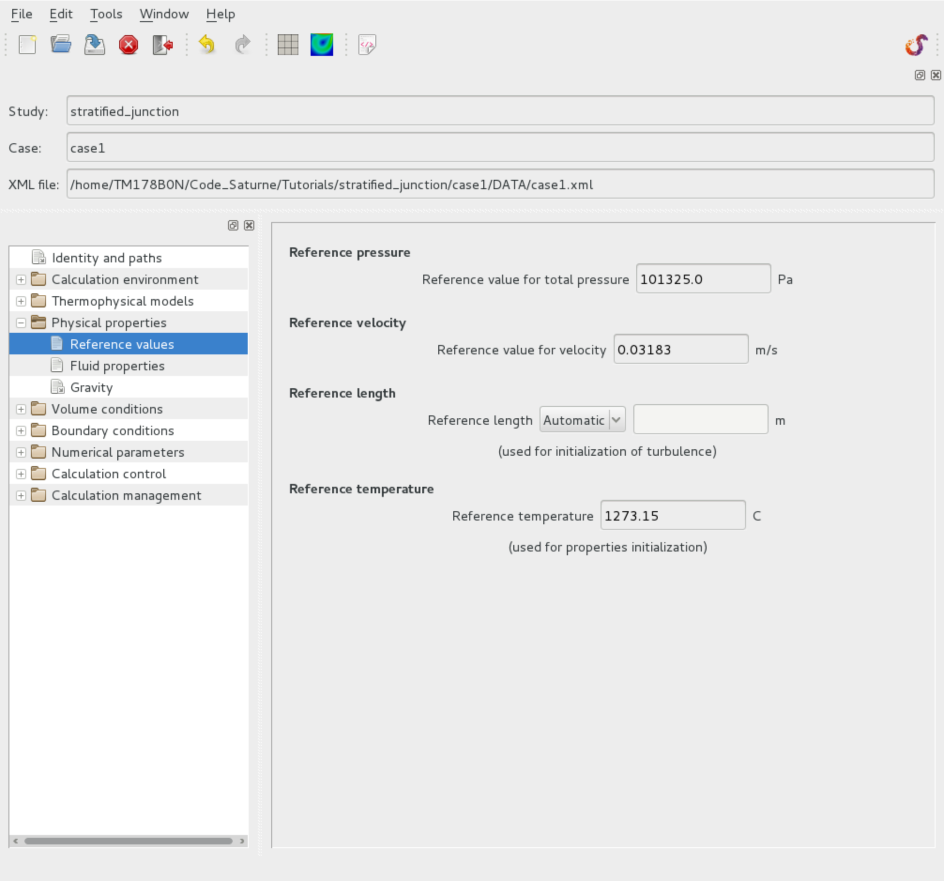
```
$ code_saturne create -s T_junction -c case1
```

1.2 Preparing and launching *Code_Saturne* computation

After that, the next steps are:

- Open the *Code_Saturne* interface;
- Create a new file;
- Select the mesh that will be used;
- Select the **unsteady flow** item under the **Calculation features** heading;
- Select a k-ε LP turbulence model;
- Add a thermal scalar in Celsius degrees.

In the item **Reference values**, under the heading **Physical properties**, set the reference value for velocity to 0.03183 $m.s^{-1}$.



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

Reference pressure

Reference value for total pressure 101325.0 Pa

Reference velocity

Reference value for velocity 0.03183 m/s

Reference length

Reference length Automatic m

(used for initialization of turbulence)

Reference temperature

Reference temperature 1273.15 C

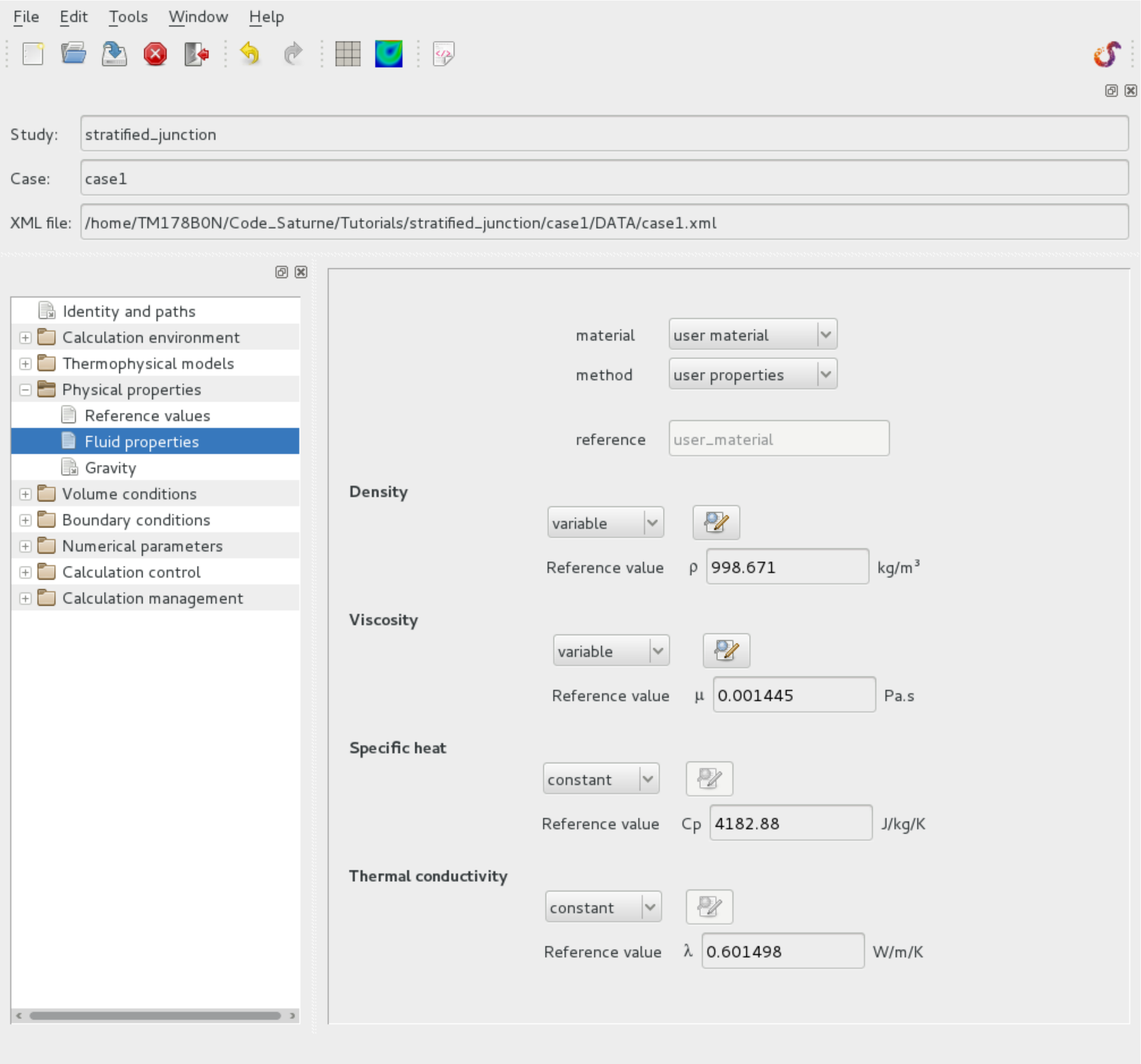
(used for properties initialization)

Figure III.1: Physical properties: reference values

In the item **Fluid properties**, under the heading **Physical properties**, enter the following information:

Variable	**Type**	**Value**
Density	User law	$998.671\ kg.m^{-3}$
Viscosity	User law	$0.445 \times 10^{-4}\ kg.m^{-1}.s^{-1}$
Specific Heat	Constant	$4\,182.88\ J.kg^{-1}.{}^{\circ}\mathrm{C}^{-1}$
Thermal Conductivity	Constant	$0.601498\ W.m^{-1}.K^{-1}$

For density and viscosity, the value given here will serve as a reference value (see user manual for details).



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

material: user material

method: user properties

reference: user_material

Density

variable

Reference value ρ 998.671 kg/m³

Viscosity

variable

Reference value μ 0.001445 Pa.s

Specific heat

constant

Reference value Cp 4182.88 J/kg/K

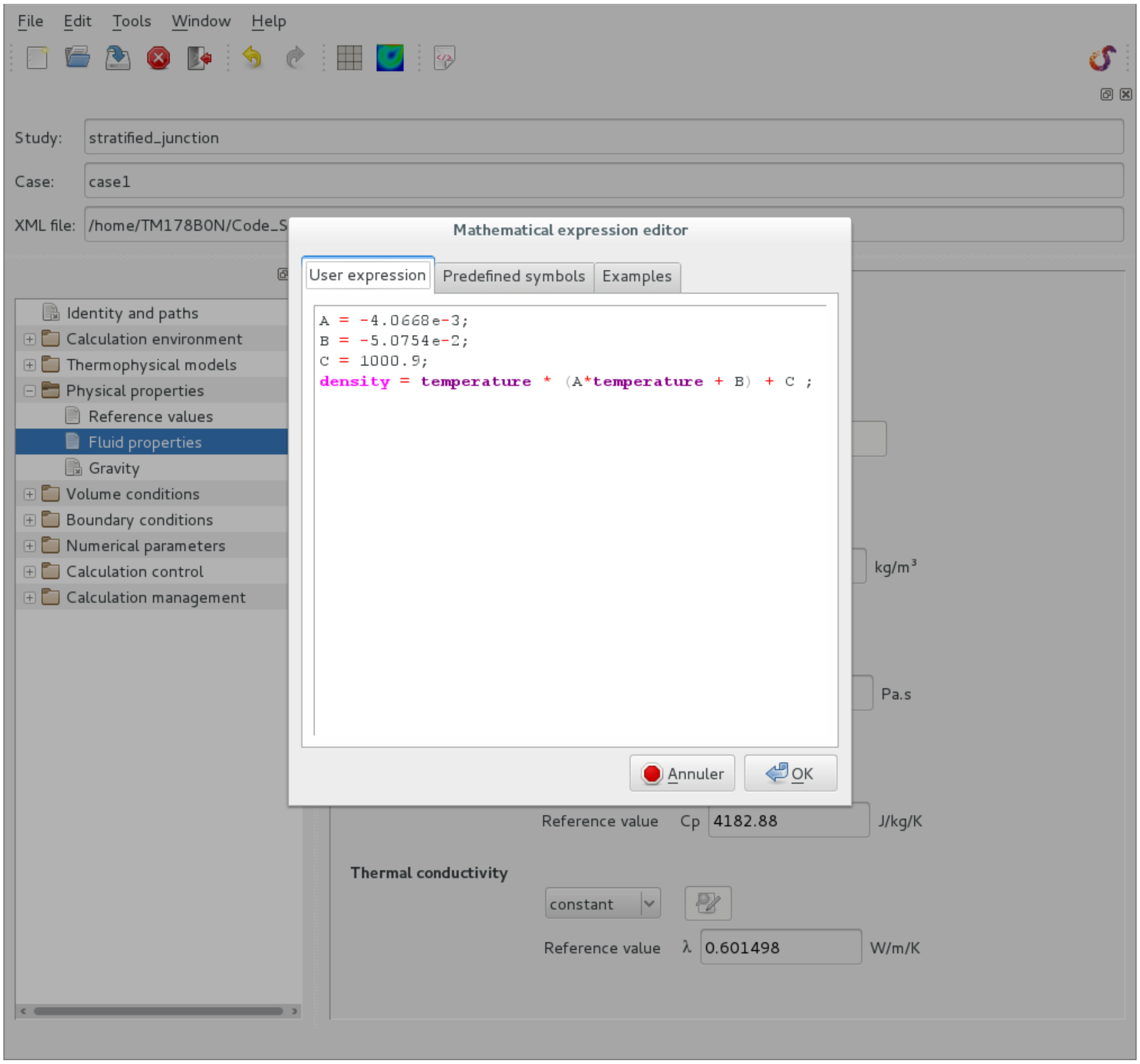
Thermal conductivity

constant

Reference value λ 0.601498 W/m/K

Figure III.2: Physical properties: fluid properties

For the density and viscosity, enter the expressions of the user laws as showed in figures III.3 and III.4, in the windows poping while clicking on the highlighted boxes.



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_S

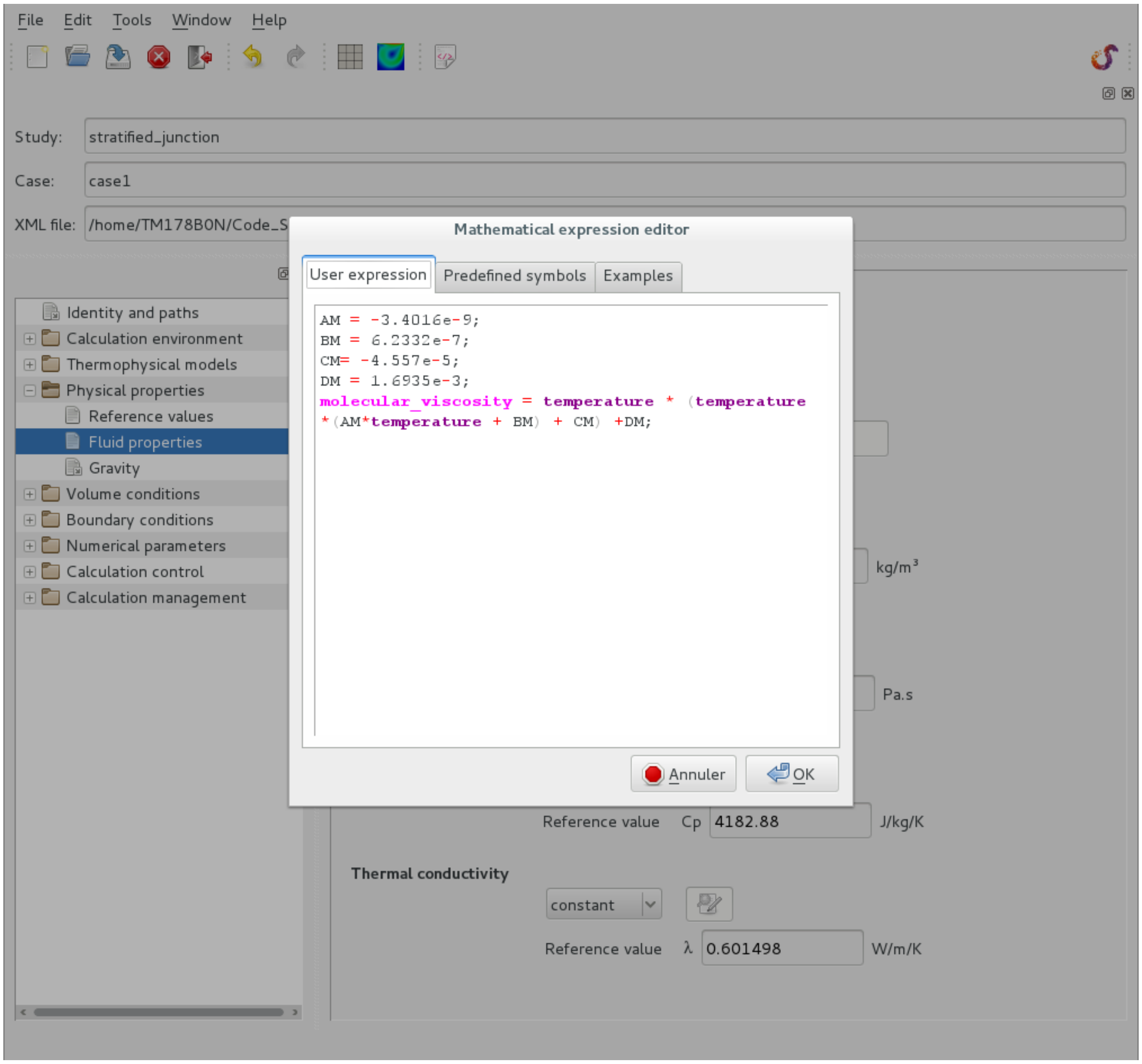
```
A = -4.0668e-3;
B = -5.0754e-2;
C = 1000.9;
density = temperature * (A*temperature + B) + C ;
```

Reference value Cp 4182.88 J/kg/K

Thermal conductivity: constant

Reference value λ 0.601498 W/m/K

Figure III.3: Variable density



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_S

```
AM = -3.4016e-9;
BM = 6.2332e-7;
CM= -4.557e-5;
DM = 1.6935e-3;
molecular_viscosity = temperature * (temperature
*(AM*temperature + BM) + CM) +DM;
```

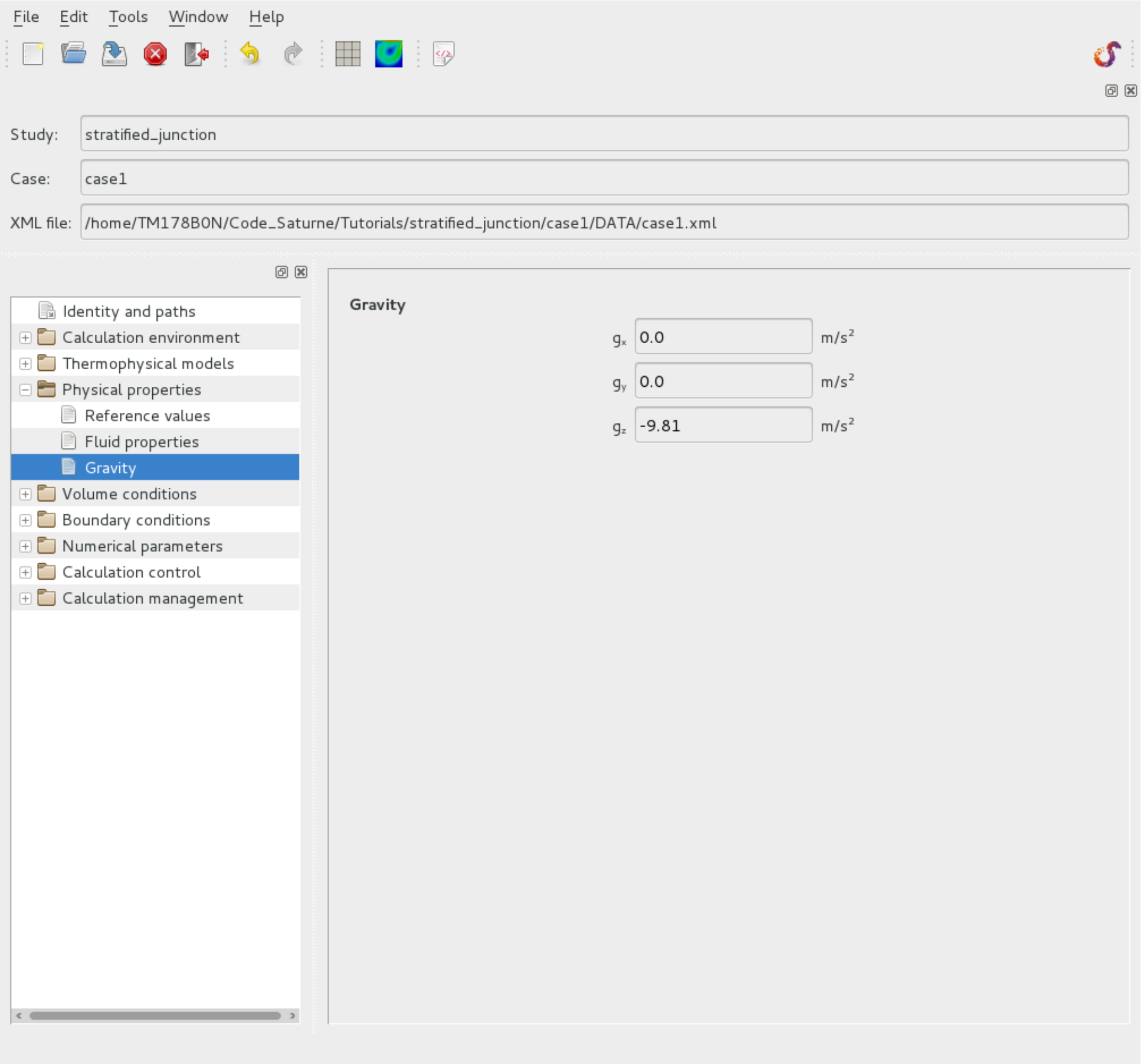
Reference value Cp 4182.88 J/kg/K

Thermal conductivity: constant

Reference value λ 0.601498 W/m/K

Figure III.4: Variable viscosity

The aim of the calculation is to simulate a stratified flow. It is therefore necessary to have gravity. Set it to the right value in the item **Gravity** under **Physical properties**.



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

Gravity

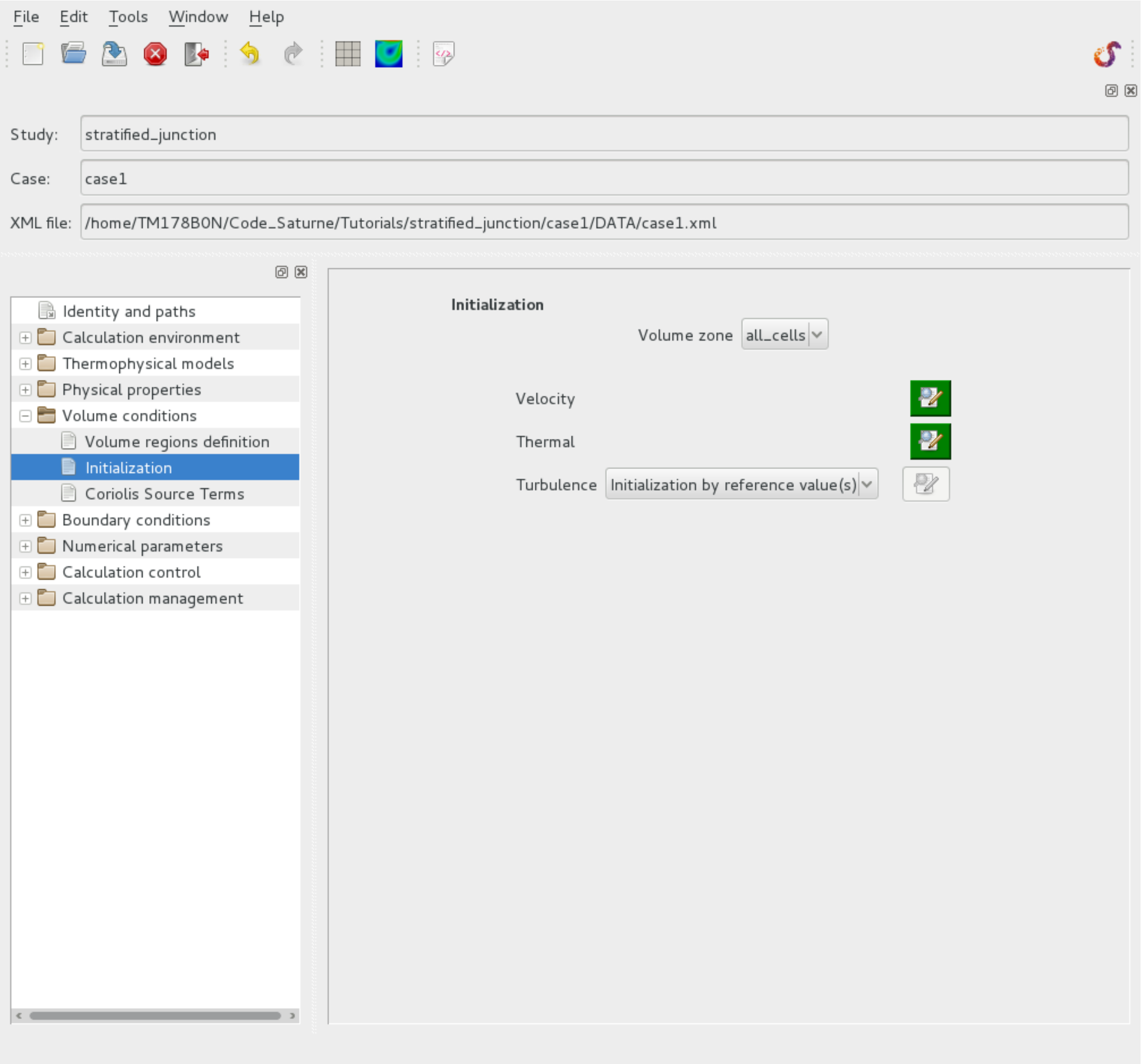
g_x 0.0 m/s^2

g_y 0.0 m/s^2

g_z -9.81 m/s^2

Figure III.5: Fluid properties: gravity

In the item **Initialization** under the heading **Volume conditions**, set the initial value of the temperature in the domain to 38.5°C. Initialize the turbulence with the reference velocity 0.03183 $m.s^{-1}$.



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

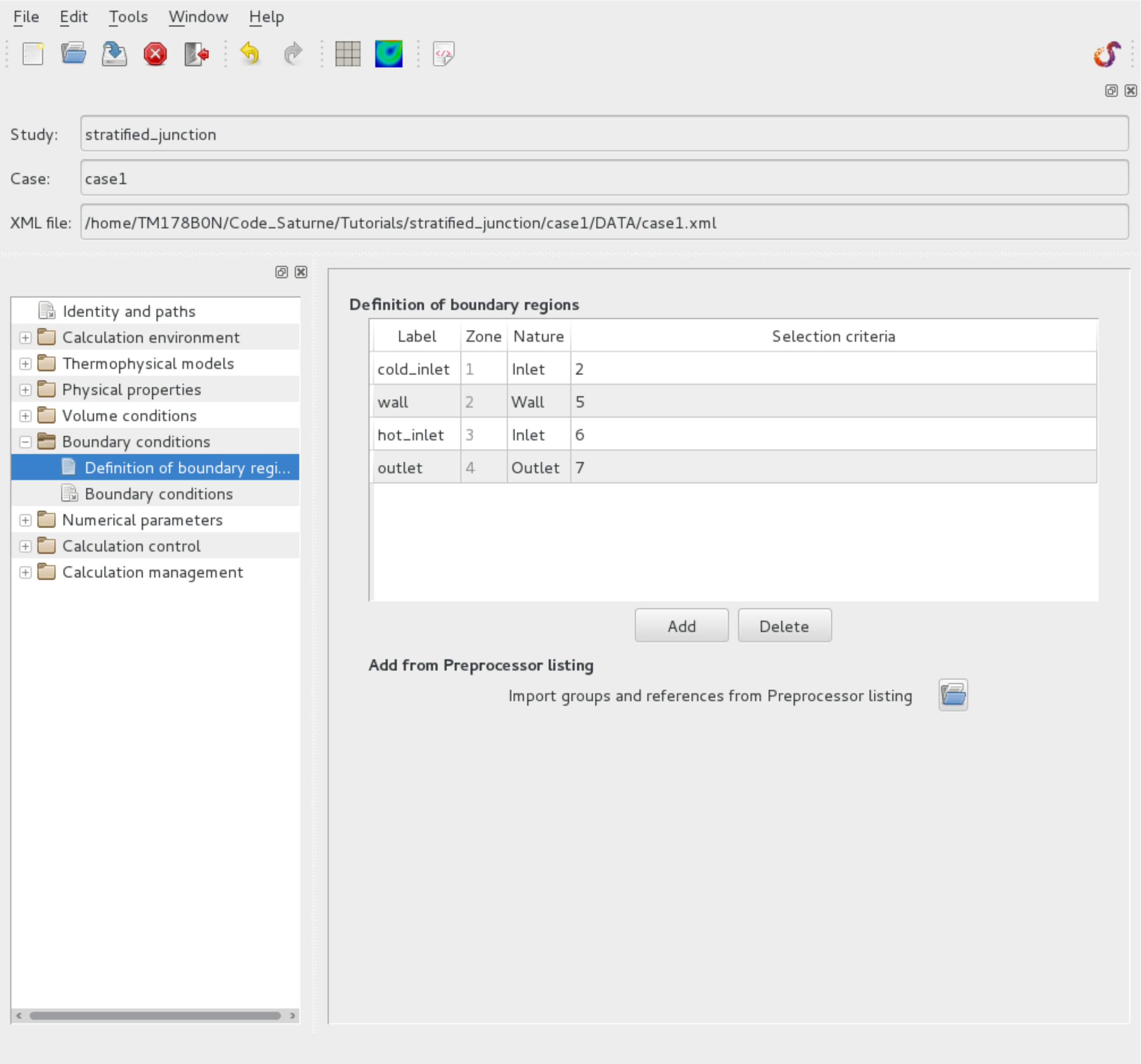
Volume zone: all_cells

Turbulence: Initialization by reference value(s)

Figure III.6: Thermophysical models: initialization

Create the boundary regions.

Colors	Conditions
2	inlet
6	inlet
7	outlet
5	wall



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

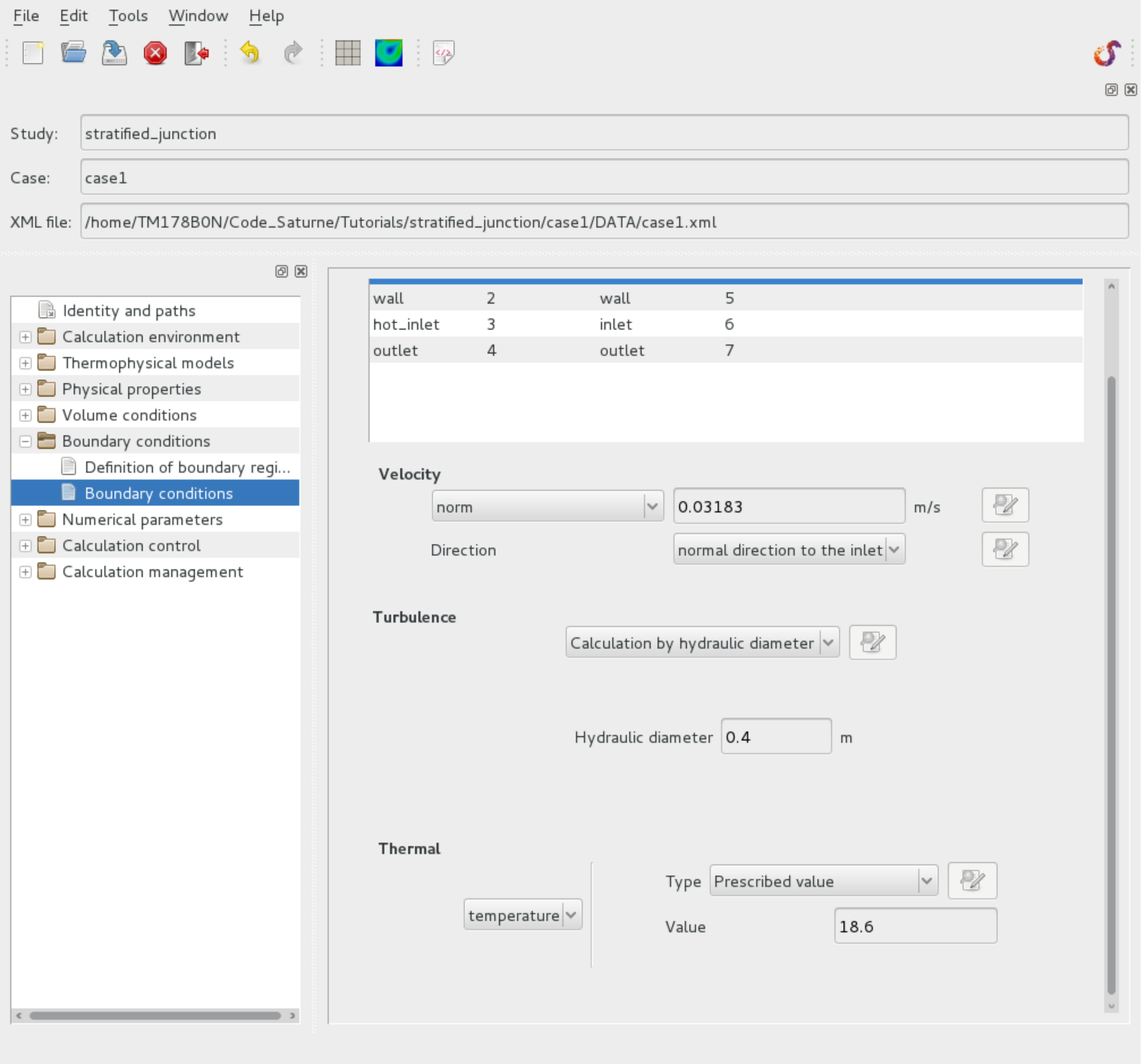
Definition of boundary regions

Label	Zone	Nature	Selection criteria
cold_inlet	1	Inlet	2
wall	2	Wall	5
hot_inlet	3	Inlet	6
outlet	4	Outlet	7

Figure III.7: Boundary regions

For the inlet boundary conditions, the velocity is 0.03183 $m.s^{-1}$ in the z direction and the hydraulic diameter 0.4 m for both inlets. For the scalar boundary conditions, the temperature of the cold inlet is 18.6°C and that of the hot inlet is 38.5°C. The outlet and wall boundary conditions remain with their default values.

– Cold inlet:



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

wall	2	wall	5
hot_inlet	3	inlet	6
outlet	4	outlet	7

Velocity

norm 0.03183 m/s

Direction normal direction to the inlet

Turbulence

Calculation by hydraulic diameter

Hydraulic diameter 0.4 m

Thermal

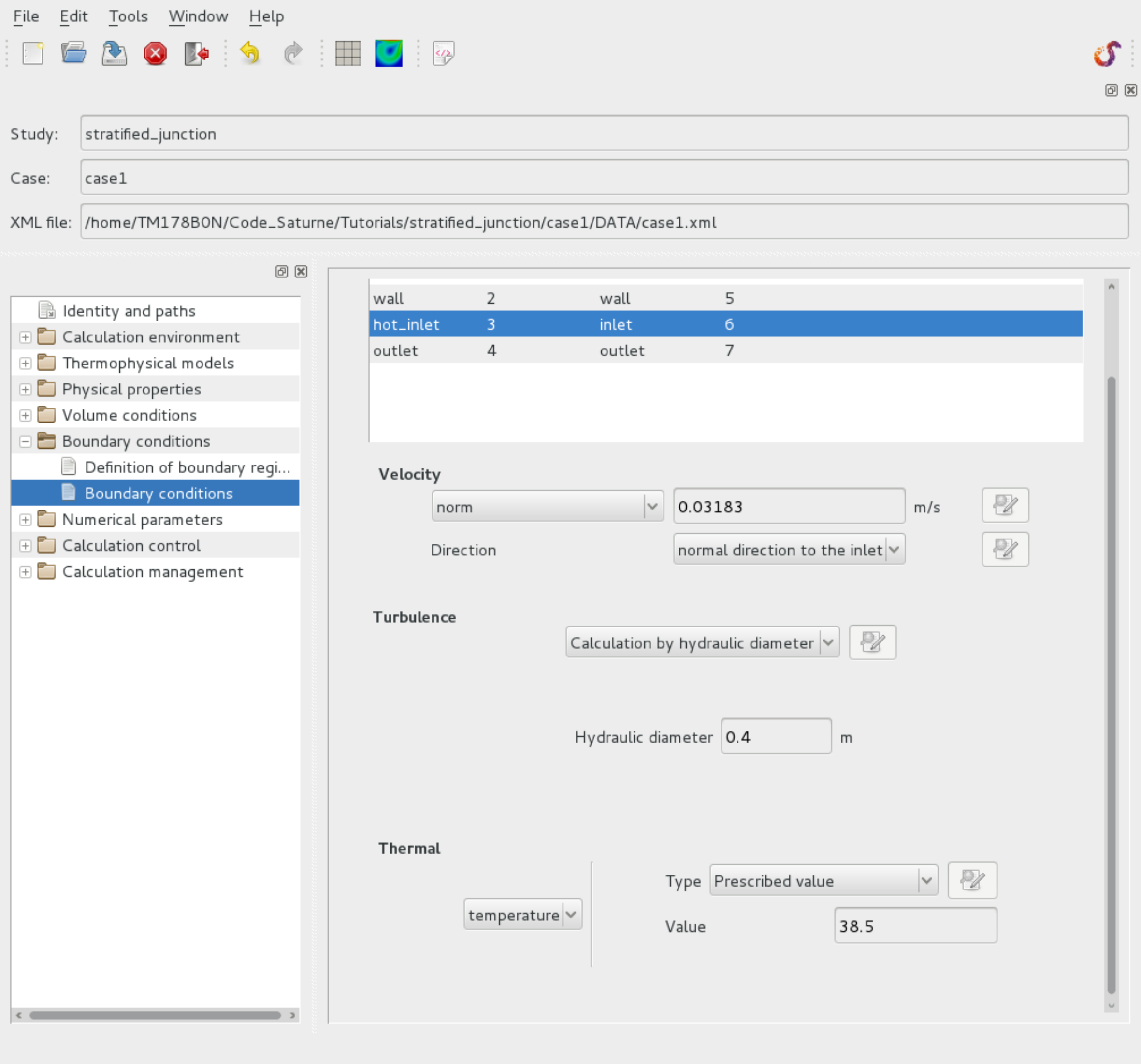
temperature

Type Prescribed value

Value 18.6

Figure III.8: Cold inlet boundary condition

– Hot inlet:



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

wall	2	wall	5
hot_inlet	3	inlet	6
outlet	4	outlet	7

Velocity

norm 0.03183 m/s

Direction normal direction to the inlet

Turbulence

Calculation by hydraulic diameter

Hydraulic diameter 0.4 m

Thermal

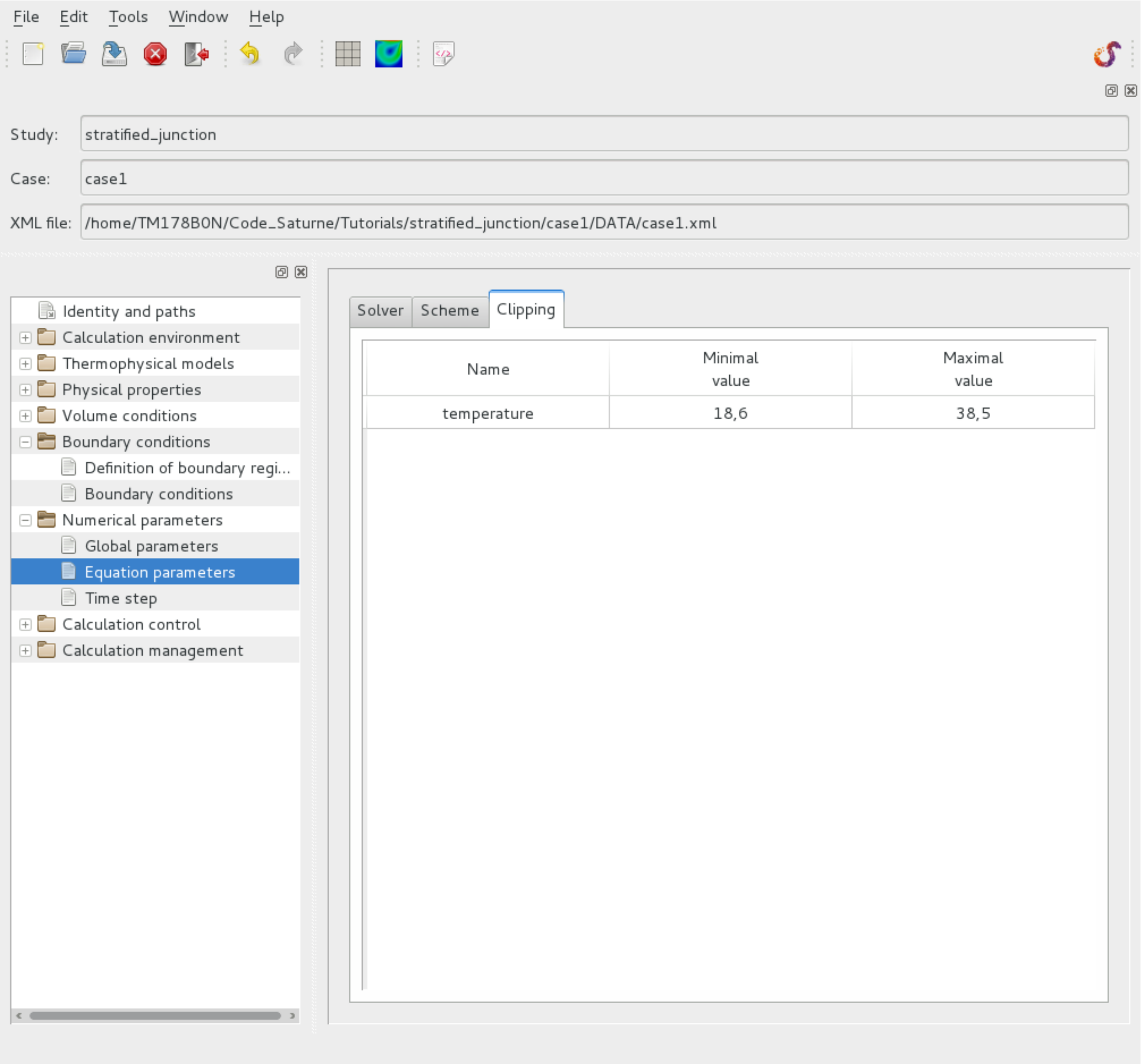
temperature

Type Prescribed value

Value 38.5

Figure III.9: Hot inlet boundary condition

Go to the item **Equation parameters** under the heading **Numerical parameters** to specify the minimal and maximal values for the temperature: 18.26°C and 38.5°C. Note that the initial value of 38.5°C set earlier is properly taken into account.



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

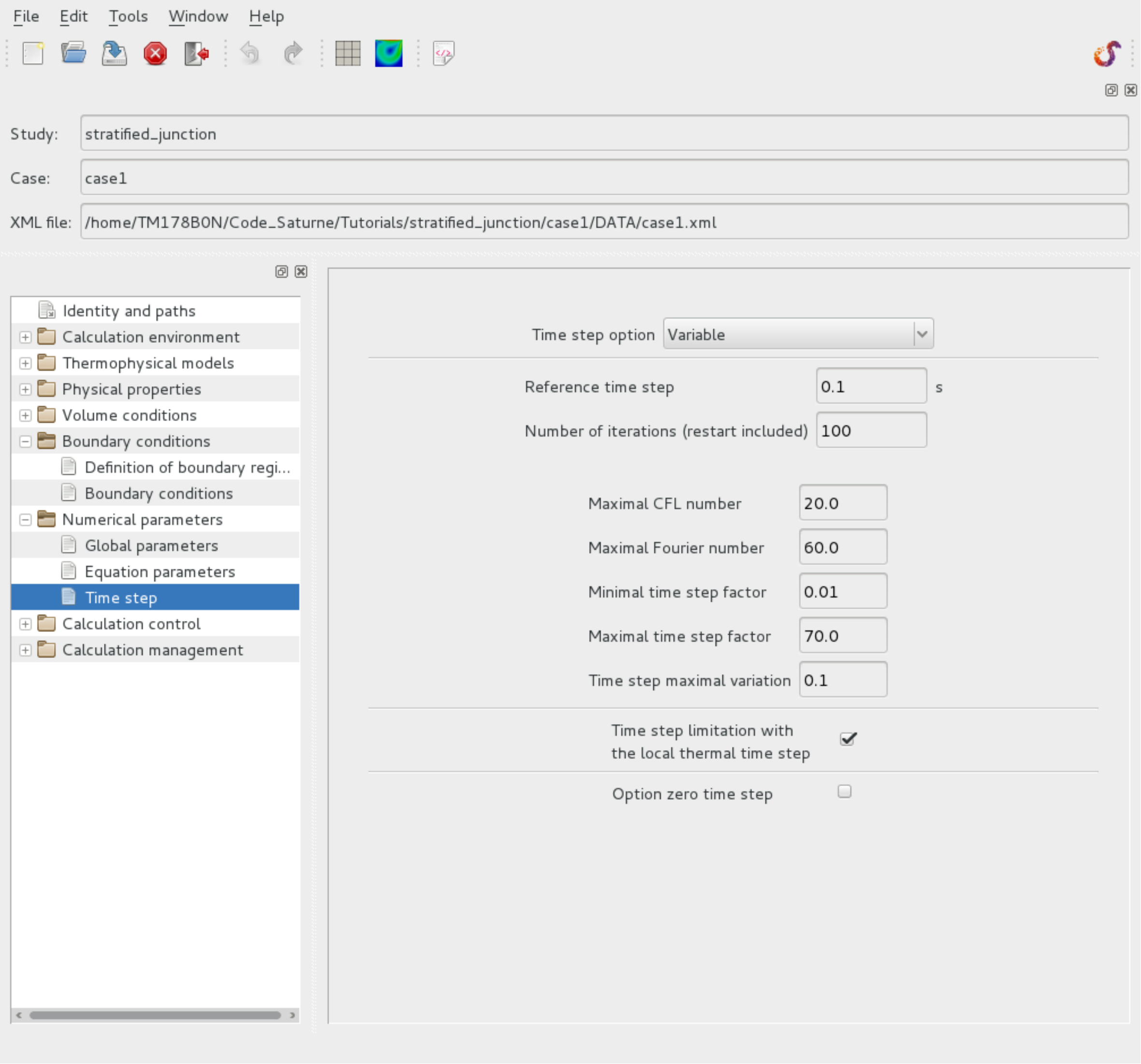
Name	Minimal value	Maximal value
temperature	18,6	38,5

Figure III.10: Scalar initialization

Tick the appropriate box for the time step to be variable in time and uniform in space. In the boxes below, enter the following parameters:

Parameters of calculation control	
Number of iterations	100
Reference time step	0.1 s
Maximal CFL number	20
Maximal Fourier number	60
Minimal time step	0.01 s
Maximal time step	70 s
Time step maximal variation	0.1

Then, activate the option **Time step limitation with the local thermal time step**



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

Time step option: Variable

Reference time step: 0.1 s

Number of iterations (restart included): 100

Maximal CFL number: 20.0

Maximal Fourier number: 60.0

Minimal time step factor: 0.01

Maximal time step factor: 70.0

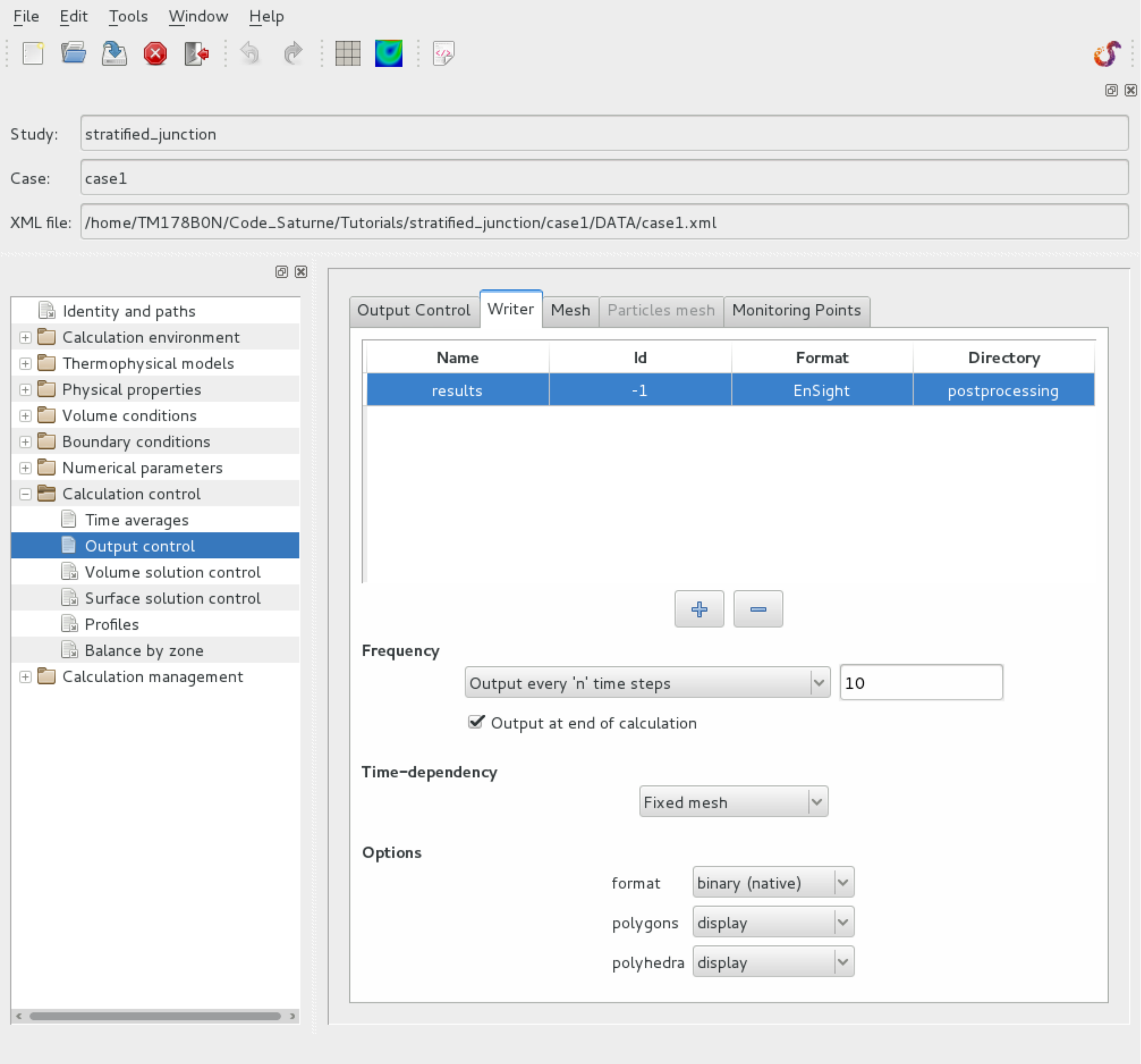
Time step maximal variation: 0.1

Time step limitation with the local thermal time step ☑

Option zero time step ☐

Figure III.11: Time step

Set the frequency of post-processing for the main writer `results` to 10.



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

Name	Id	Format	Directory
results	-1	EnSight	postprocessing

Frequency

Output every 'n' time steps: 10

☑ Output at end of calculation

Time-dependency

Fixed mesh

Options

format: binary (native)

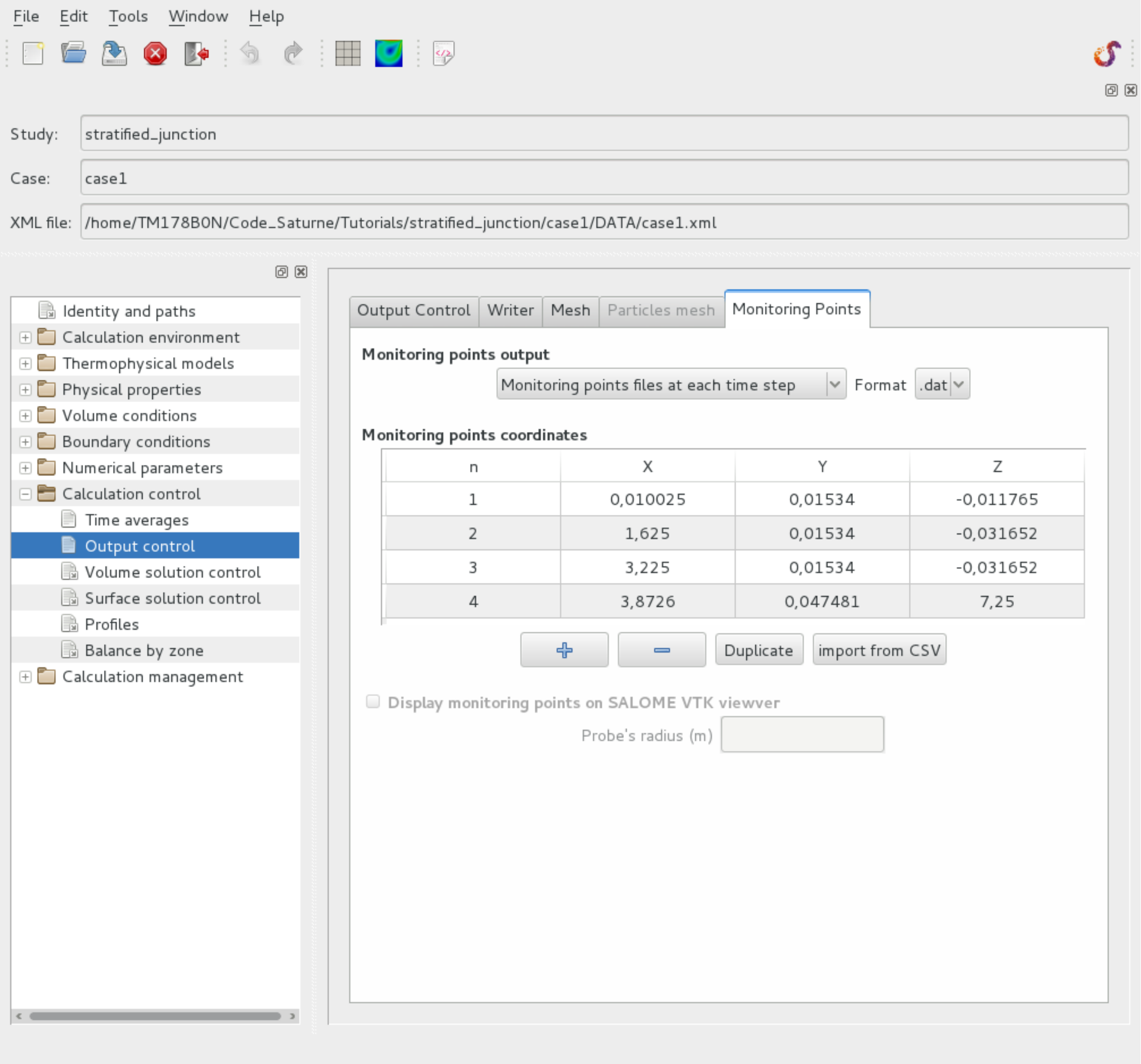
polygons: display

polyhedra: display

Figure III.12: Output management

Create four monitoring probes at the following coordinates:

Probes	**x(m)**	**y(m)**	**z(m)**
1	0.010025	0.01534	-0.011765
2	1.625	0.01534	-0.031652
3	3.225	0.01534	-0.031652
4	3.8726	0.047481	7.25



Study: stratified_junction

Case: case1

XML file: /home/TM178B0N/Code_Saturne/Tutorials/stratified_junction/case1/DATA/case1.xml

Monitoring points output

Monitoring points files at each time step   Format .dat

Monitoring points coordinates

n	X	Y	Z
1	0,010025	0,01534	-0,011765
2	1,625	0,01534	-0,031652
3	3,225	0,01534	-0,031652
4	3,8726	0,047481	7,25

☐ Display monitoring points on SALOME VTK viewver

Probe's radius (m)

Figure III.13: Monitoring points

For the advanced post-processing features, copy to the 🗀`SRC` directory the files `cs_user_postprocess.c` and `cs_user_postprocess_var.f90`. The general content of these routines is described in the user manual or in the examples available in the directory 🗀`SRC/REFERENCE`. The modified routines adapted to this test case are available in the `examples` directory. Only the main elements are mentioned here.

- cs_user_postprocess_meshes (in `cs_user_postprocess.c`):
  This is called only once, at the beginning of the calculation. It allows to define the different writers and parts.
- `cs_user_postprocess_var.f90`:
  This routine is called at each time step. It allows to specify which variable will be written on which part.