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

MAY 2020

Code_Saturne documentation

Code_Saturne version 6.0 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 6.0. 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

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.

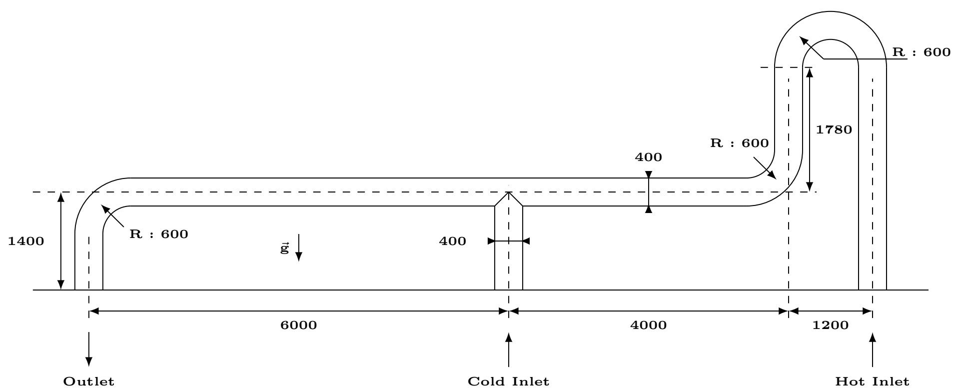

Figure II.1: Geometry of the case, with dimensions in mm

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	$T_{cb} = 18.6$°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 38.5°C and $10^5\ Pa$:

- heat capacity: $C_p = 4{,}178\ J.kg^{-1}.°\mathrm{C}^{-1}$
- thermal conductivity: $\lambda = 0.628\ W.m^{-1}.°\mathrm{C}^{-1}$

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

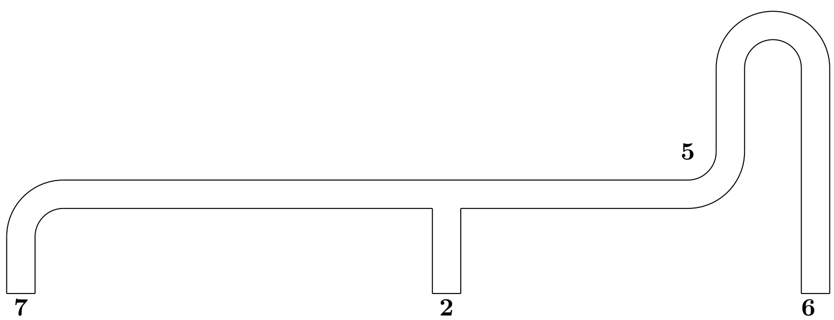

Figure II.2: References of the boundary faces

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
Global parameters	Improved pressure interpolation for stratified flows

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 the 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 $\frac{dt_{min}}{dt_{ref}}$	0.01
Maximal time step factor $\frac{dt_{max}}{dt_{ref}}$	70
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	0.725

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 file must to be copied from the folder 🗀`SRC/EXAMPLES` into the folder 🗀`SRC`[1]:

- `cs_user_postprocess.c`;

In this test case, advanced post-processing features will be used. An additional writer will 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.

  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

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

to **false** here, and we will use a user-defined output with **cs_user_postprocess_values**. 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.

3.7 Results

Figure II.3 shows the evolution of temperature in a clip plane created along the symmetry plane of the domain. 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

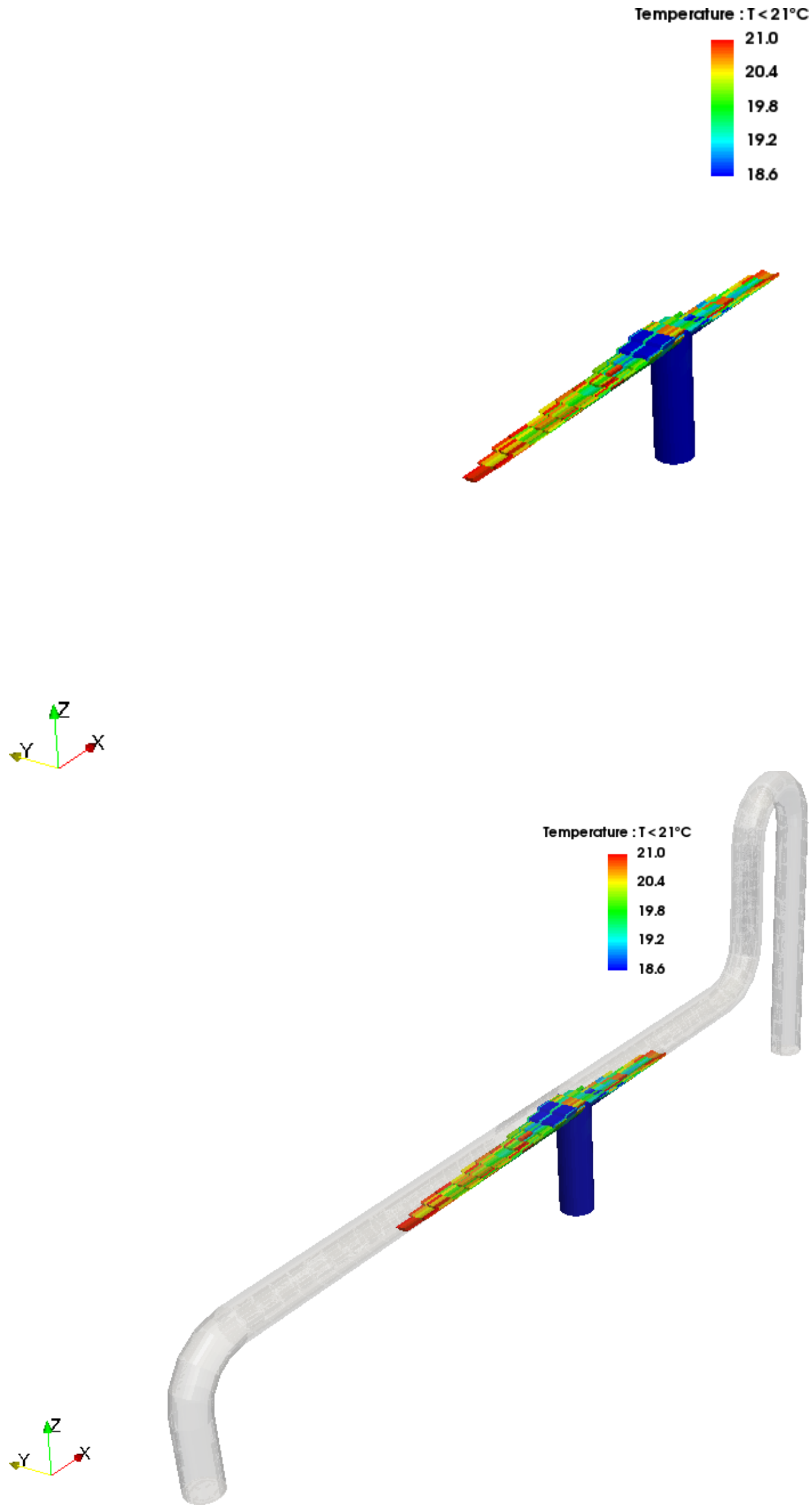

Figure II.4: Sub-domain where the temperature is lower than 21°C (upper figure) and localization in the full domain (lower figure)

Part III

Step by step solution

1 Detailed tutorial step by step

1.1 Creation of the study in a terminal

This tutorial will be set up within *SALOME* using the CFDSTUDY module (*Code_Saturne*). The first thing to do is to prepare the computation directories. In this example, the study directory 🗀 `T_junction` will be created, containing a single calculation directory 🗀 `case1`. It can be directly done in the terminal using the SALOME shell with the following commands:

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

Then, the mesh of the tutorial (sn_total.des) can be moved into the directory 🗀 `MESH` of the study in order to be used later.

1.2 Preparing and launching *Code_Saturne* computation

After that, the next steps are:

- Open the *SALOME* graphical interface;
- Select the *CFDSTUDY* module;
- Load the study previously created with the option *'Choose an existing CFD study or create'*. A window as in figure III.1 should be obtained;

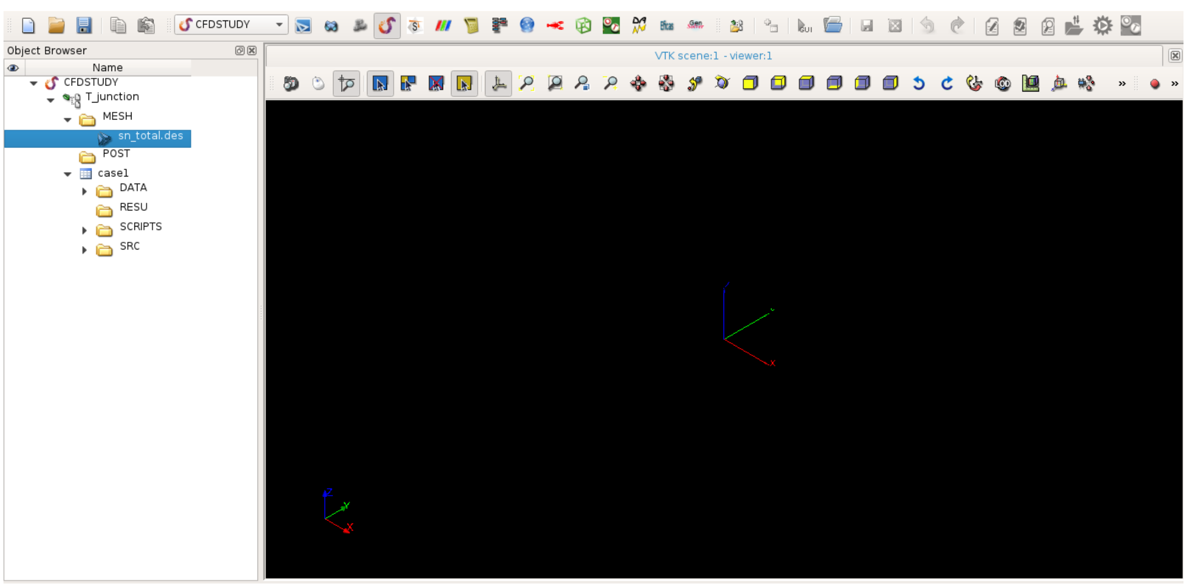

Figure III.1: Graphical user interface of the *SALOME*

The mesh can be directly displayed in the VTK viewer. 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 *fluid_domain* and then *'Display mesh'*;

The window should be like in figure III.2.

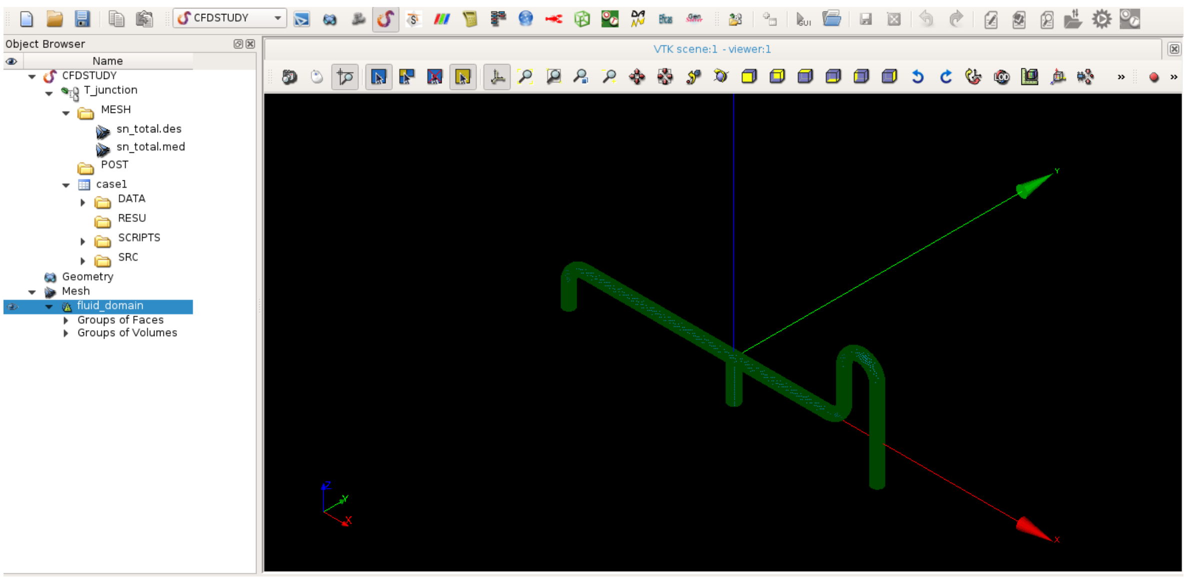

Figure III.2: Display of the mesh in *SALOME*

In order to set up the case using the graphical user interface of *Code_Saturne*, the GUI can be directly launched by right-clicking on *case1* in the object browser under the heading **CFDSTUDY**, and then selecting *'Launch GUI'*. The graphical interface of *Code_Saturne* appears within *SALOME* as shown in III.3.

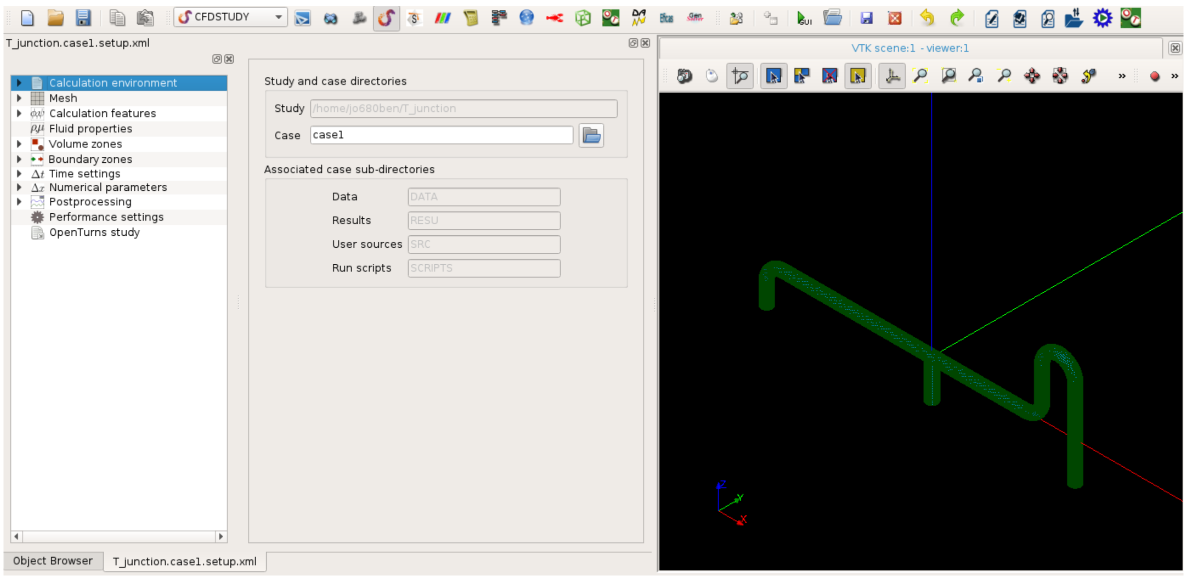

Figure III.3: Graphical user interface of *Code_Saturne* in *SALOME*

Under the heading **Mesh**, the med mesh can be added to the list of meshes.
Then in the item **Turbulence models** under the heading **Calculation features**, select k-ε *Linear Production* as turbulence model and set the velocity scale to 0.03183 $m.s^{-1}$ as shown in figure III.6.
Under the same heading, in the item **Thermal model**, add a thermal scalar in Celsius degrees.

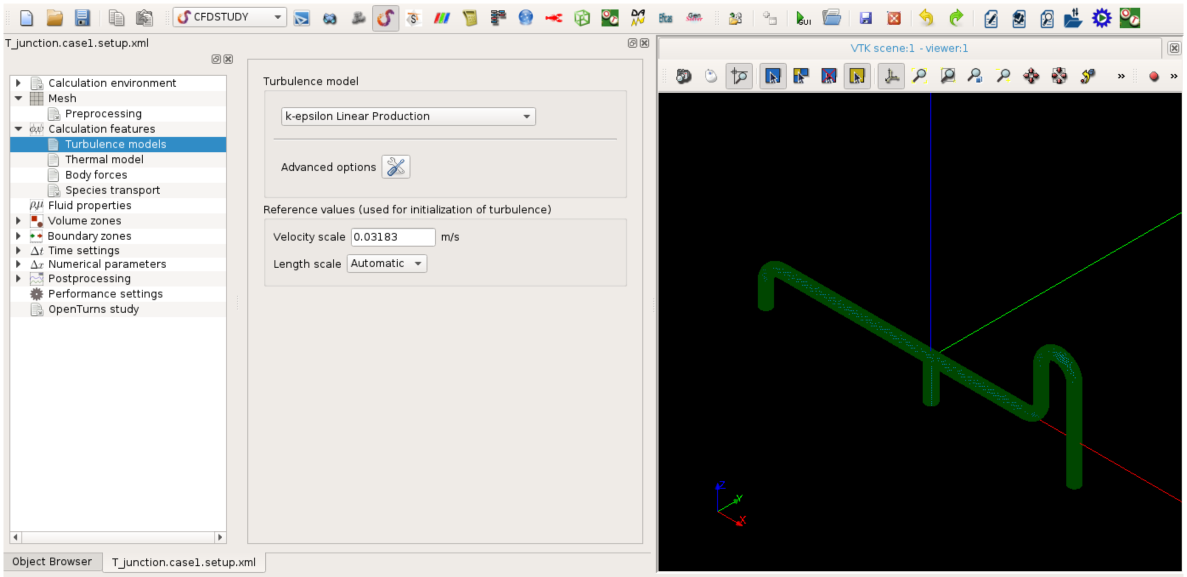

Figure III.4: Calculation feature : Turbulence model

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 **Body forces** under **Calculation features**.

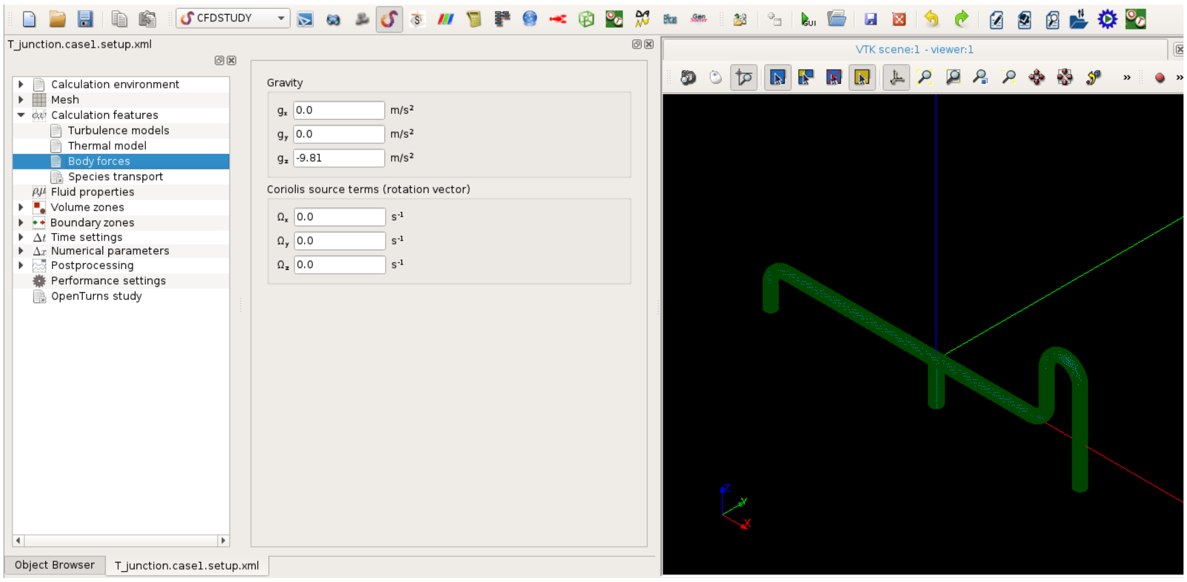

Figure III.5: Calculation features: Body forces

Under the heading **Fluid properties**, enter the following information:

Variable	Type	Reference value
Density	User law	$992.91\ kg.m^{-3}$
Viscosity	User law	$6.68 \times 10^{-4}\ kg.m^{-1}.s^{-1}$
Specific Heat	Constant	$4\,178\ J.kg^{-1}.°\mathrm{C}^{-1}$
Thermal Conductivity	Constant	$0.628\ W.m^{-1}.K^{-1}$

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

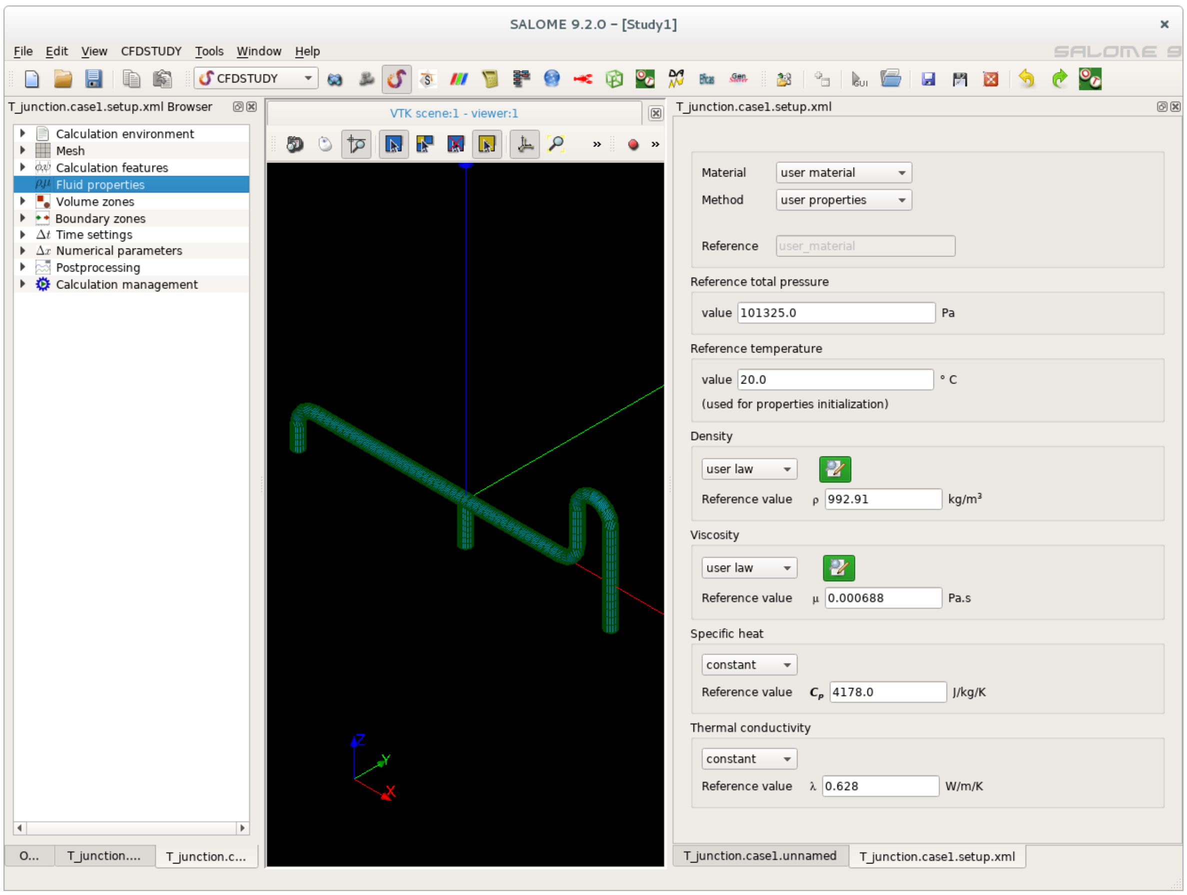

Figure III.6: Fluid properties

For the density and viscosity, enter the expressions of the user laws as shown in figures III.7 and III.8, in the pop-up window while clicking on the green highlighted boxes.

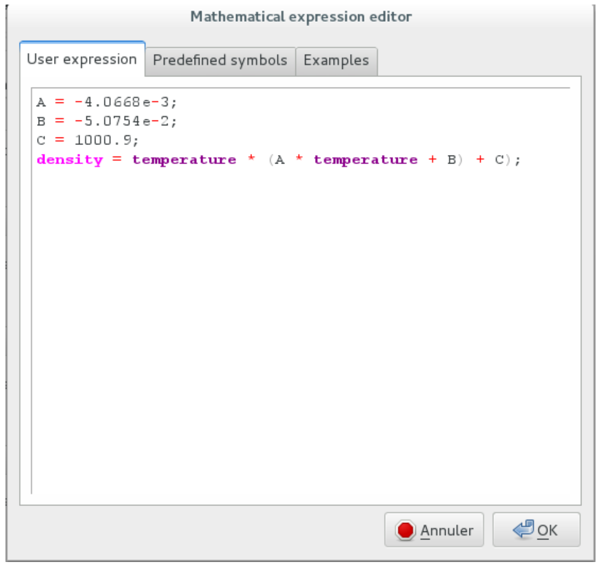

Figure III.7: Variable density

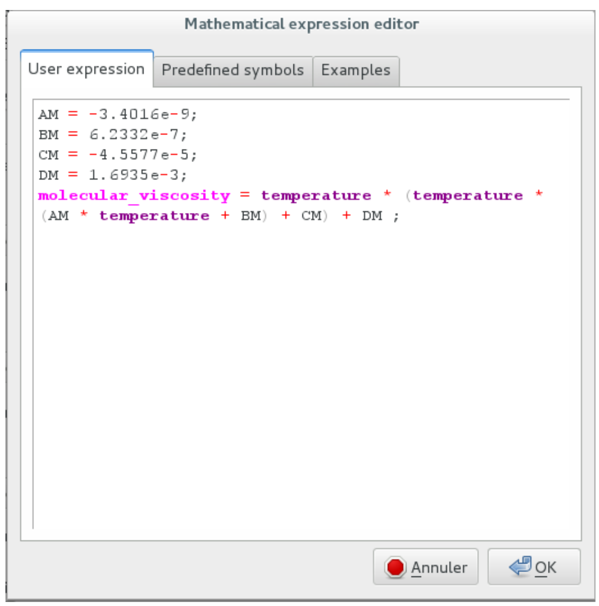

Figure III.8: Variable viscosity

In the item **Initialization** under the heading **Volume zones**, set the initial value of the temperature in the domain to 38.5°C. Initialize the turbulence with the reference velocity previously defined.

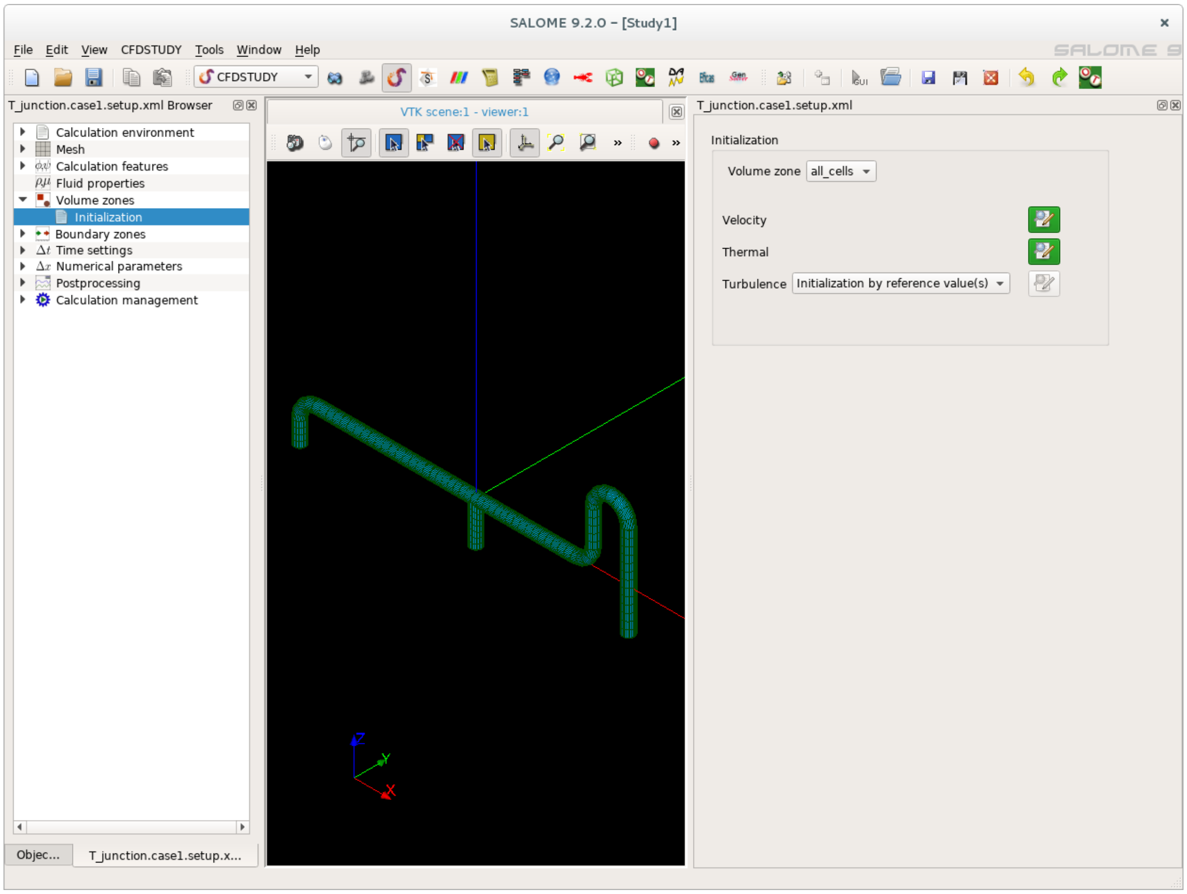

Figure III.9: Volume zones: Initialization

The boundary regions can be directly defined from the mesh by using *SALOME*. To do so, first click on the heading **Boundary Zones**. Then open the object browser of *SALOME* and click on the group of faces *'5'* for instance.

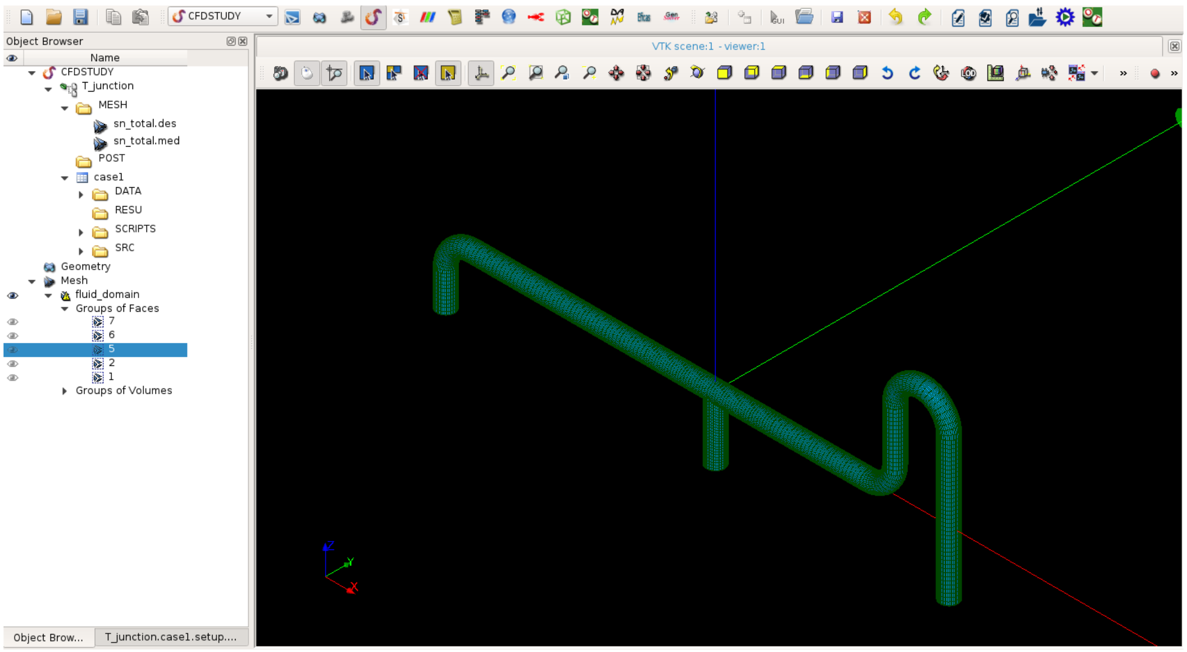

Figure III.10: Select a boundary regions from Salome

Once the group of faces is selected, go back to the **Boundary Zones** section and click on *'Add from Salome'* in the *Code_Saturne* GUI as shown in figure III.11.

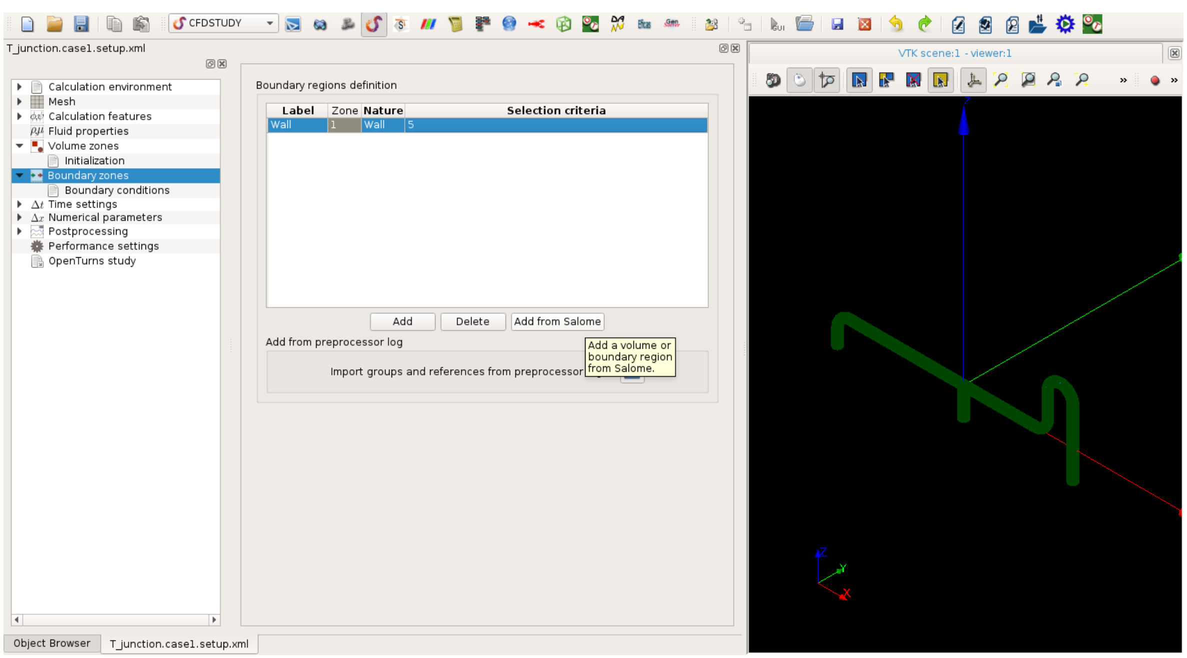

Figure III.11: Select a boundary regions from Salome

Then the type of boundary condition can be defined then with the zone *Nature*. Repeat the same process for the other boundary regions listed in the following table.

Colors	**Conditions**
2	inlet
6	inlet
7	outlet
5	wall

The boundary regions should be defined as in figure III.12.

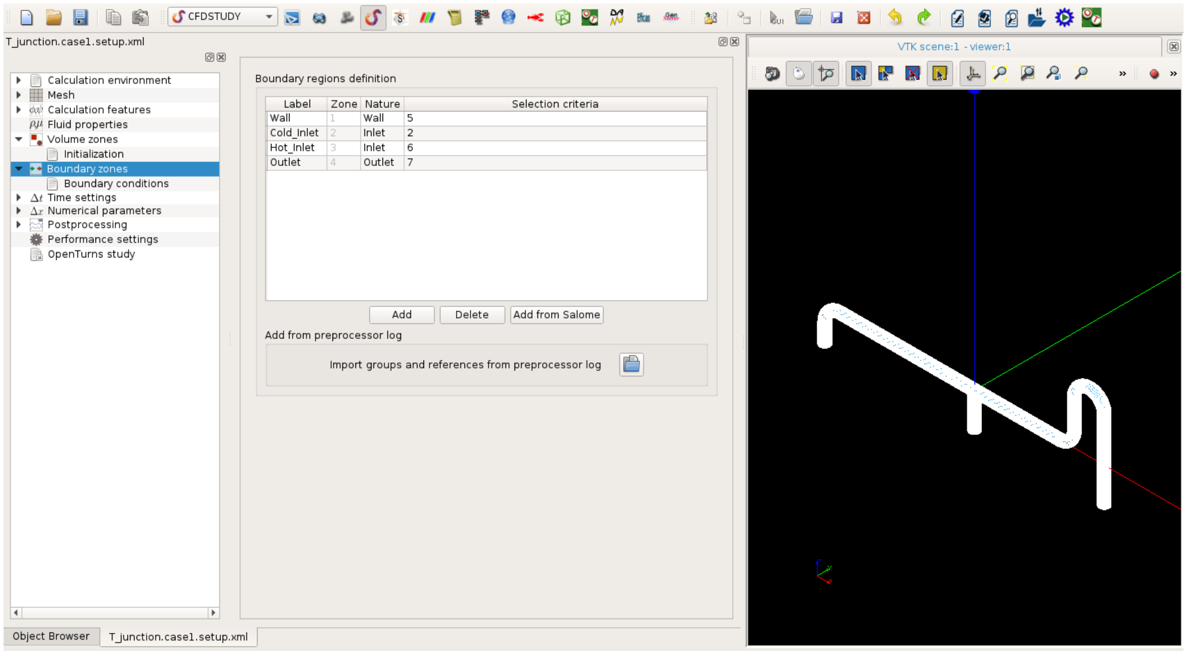

Figure III.12: Boundary regions

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

– Cold inlet:

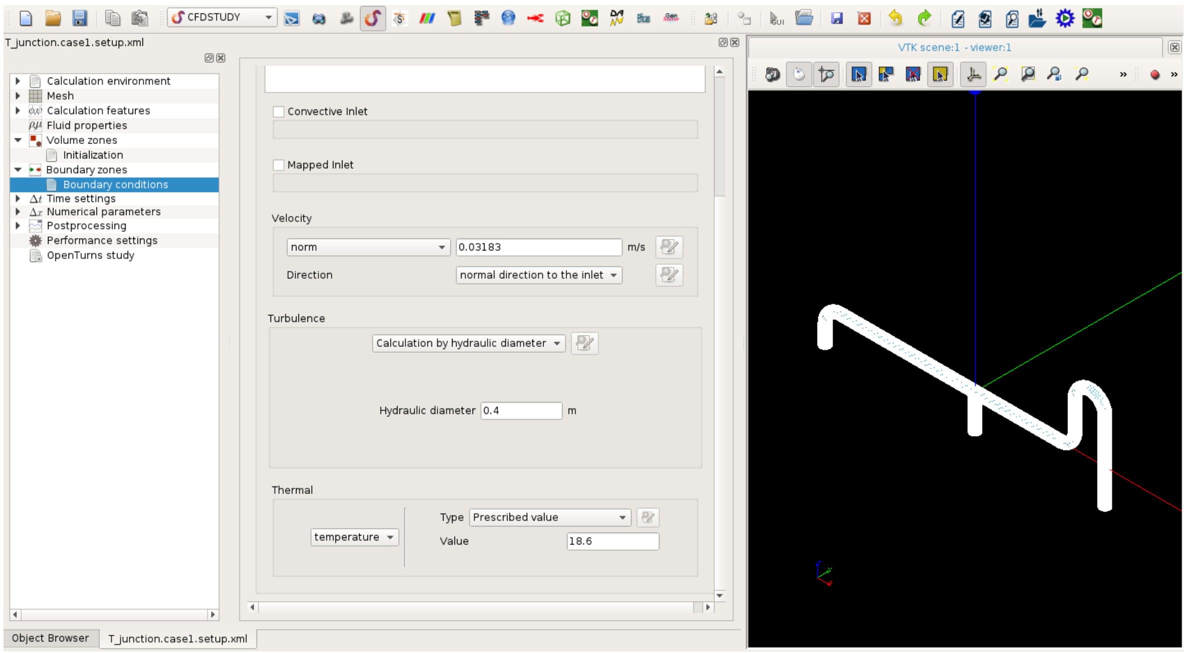

Figure III.13: Cold inlet boundary condition

– Hot inlet:

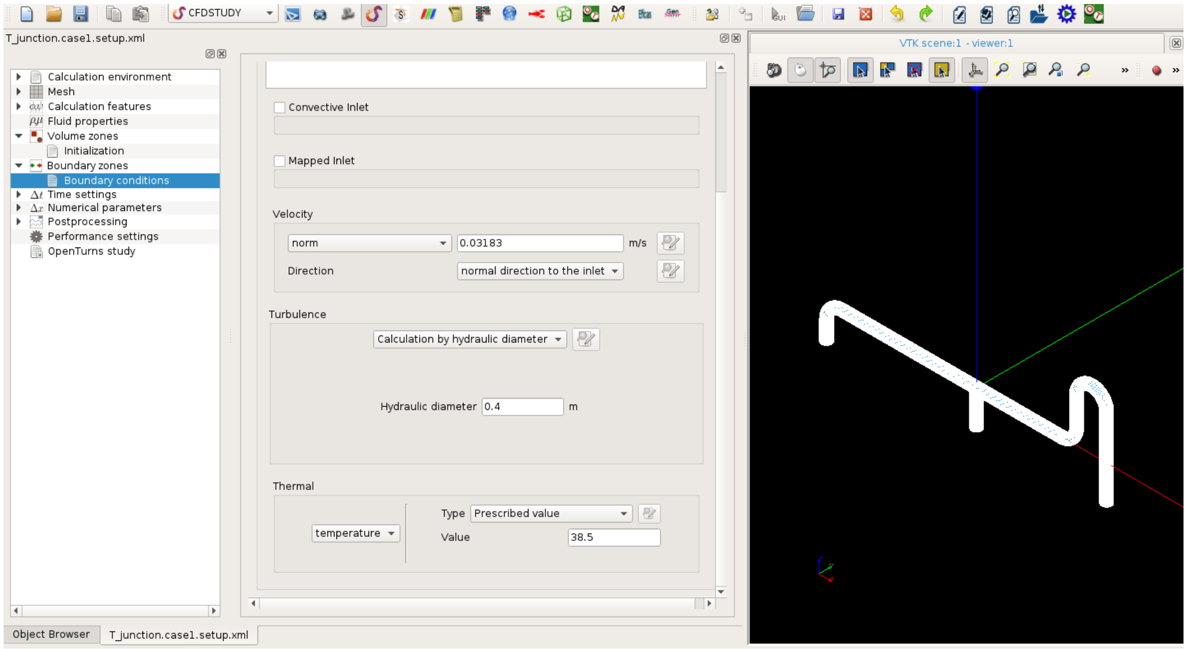

Figure III.14: Hot inlet boundary condition

Under the heading **Time settings**, 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 time steps	100
Reference time step	0.1 s
Maximal CFL number	20
Maximal Fourier number	60
Minimal time step factor	0.01
Maximal time step factor	70.0
Time step maximal variation	0.1

Then, activate the option *Limitation by local thermal time step*

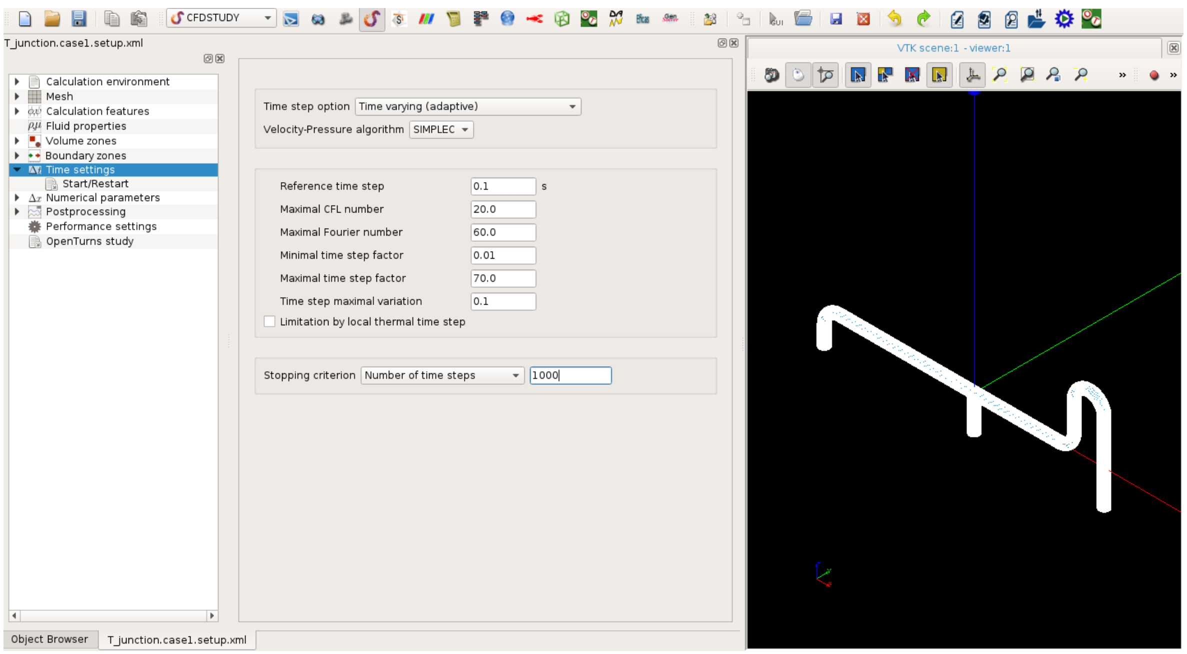

Figure III.15: Time step

Under the heading **Numerical parameters**, tick the option *Improved pressure interpolation in stratified flow.*

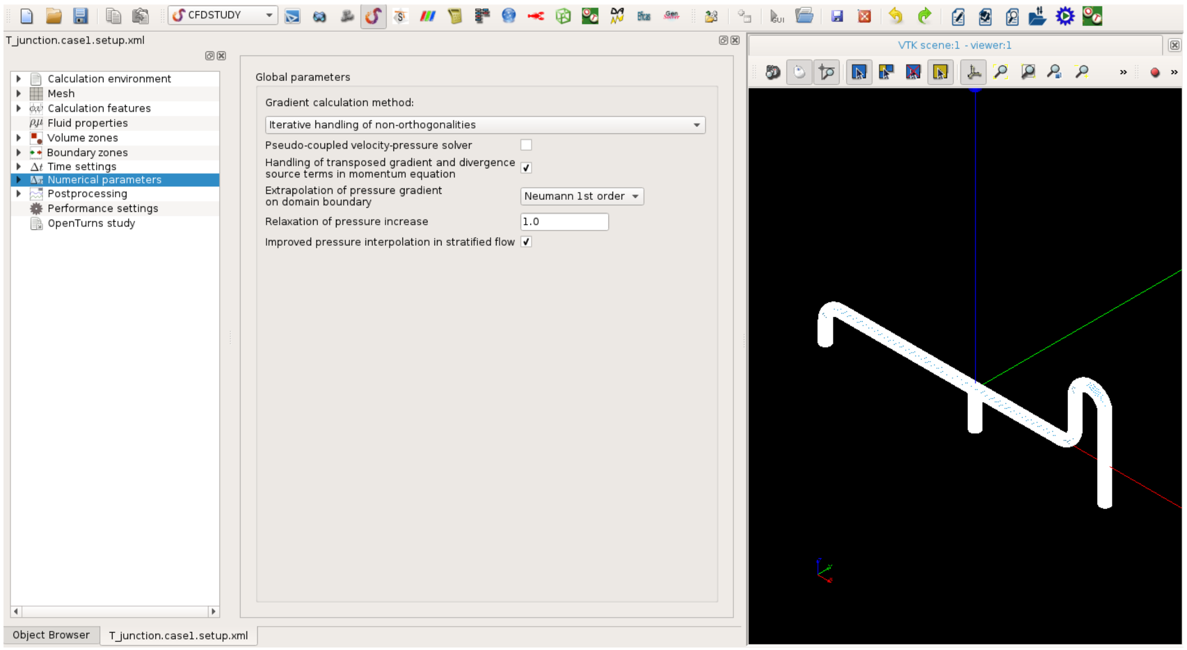

Figure III.16: Numerical parameters

Still under the same heading, go to the item **Equation parameters**, and open the *Clipping* tab to specify the minimal and maximal values for the temperature: 18.6°C and 38.5°C. Note that the initial value of 38.5°C set earlier is properly taken into account.

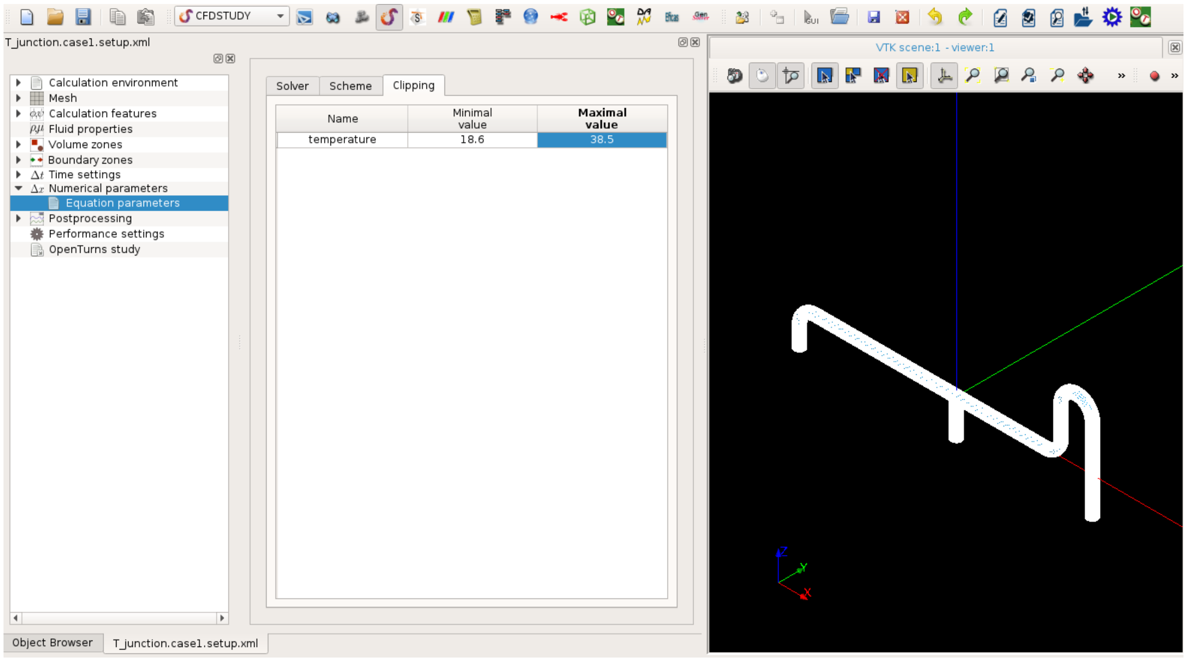

Figure III.17: Scalar clipping

Under the heading **Postprocessing**, got to the *Writer* tab and set the frequency of post-processing for the main writer `results` to 10 (time steps).

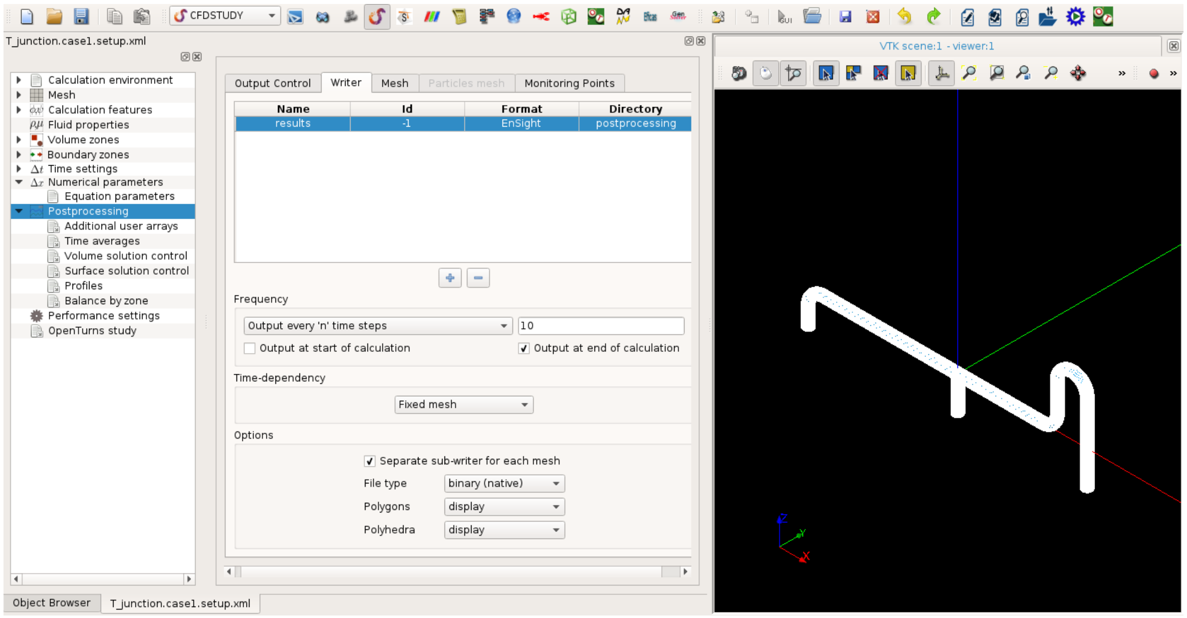

Figure III.18: Output management

Switch to the *Monitoring Points* tab and 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	0.725

Note that the monitoring points can be directly displayed in the viewer as shown below by ticking the box *Display monitoring points on SALOME viewer*.

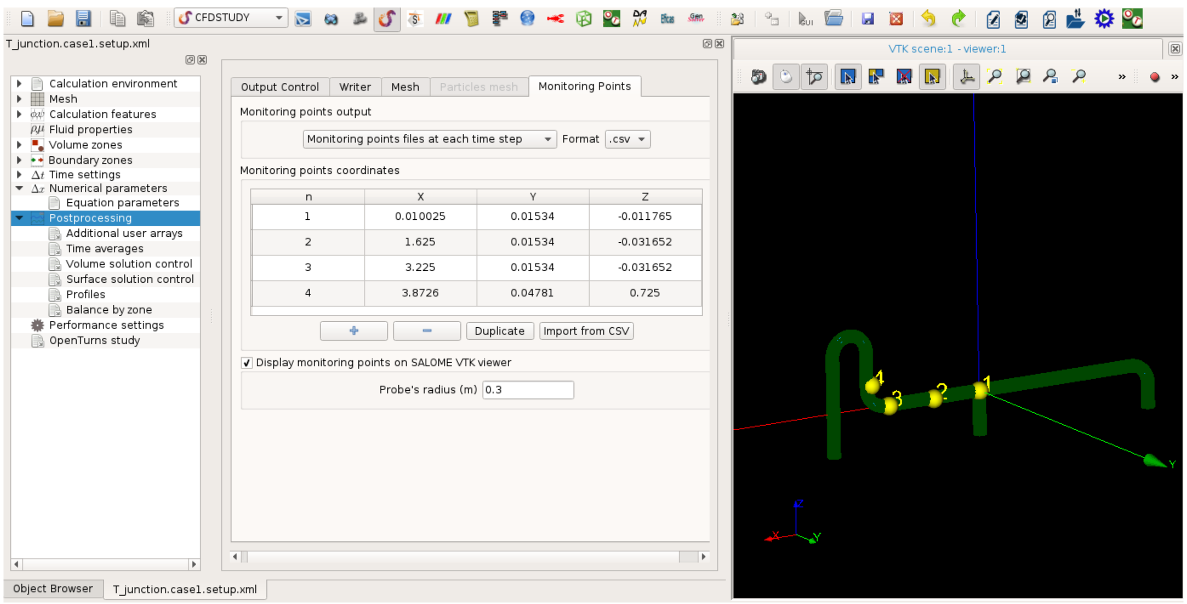

Figure III.19: Monitoring points

Still under the heading **Postprocessing**, in the item **Profiles**, create two vertical profiles at the following locations with an output frequency of 10 :

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

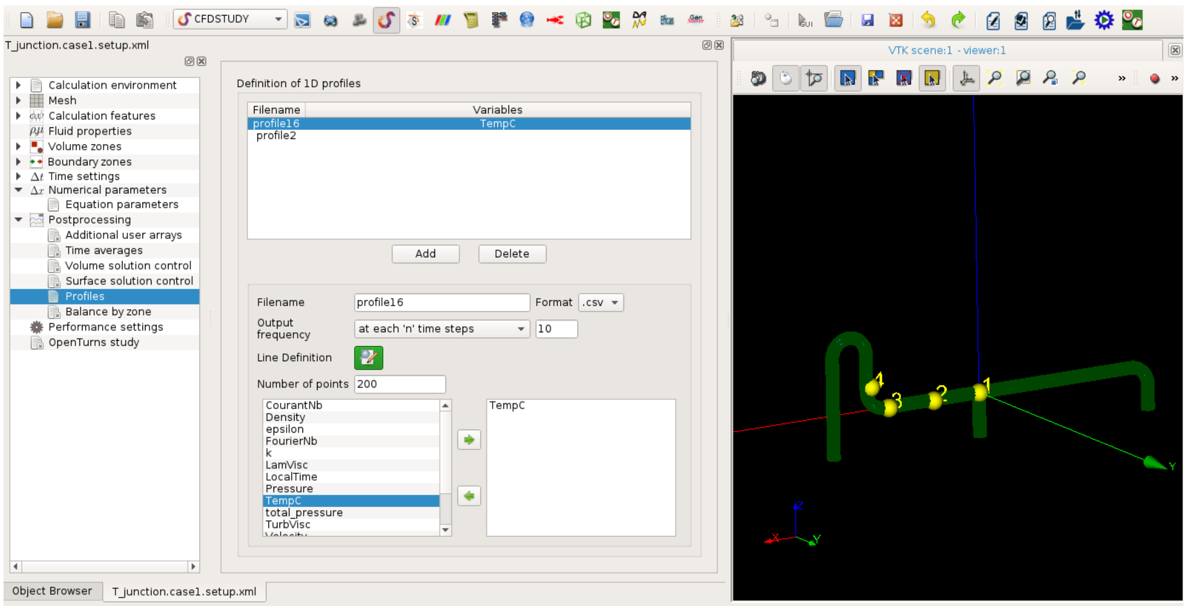

Figure III.20: Vertical profiles

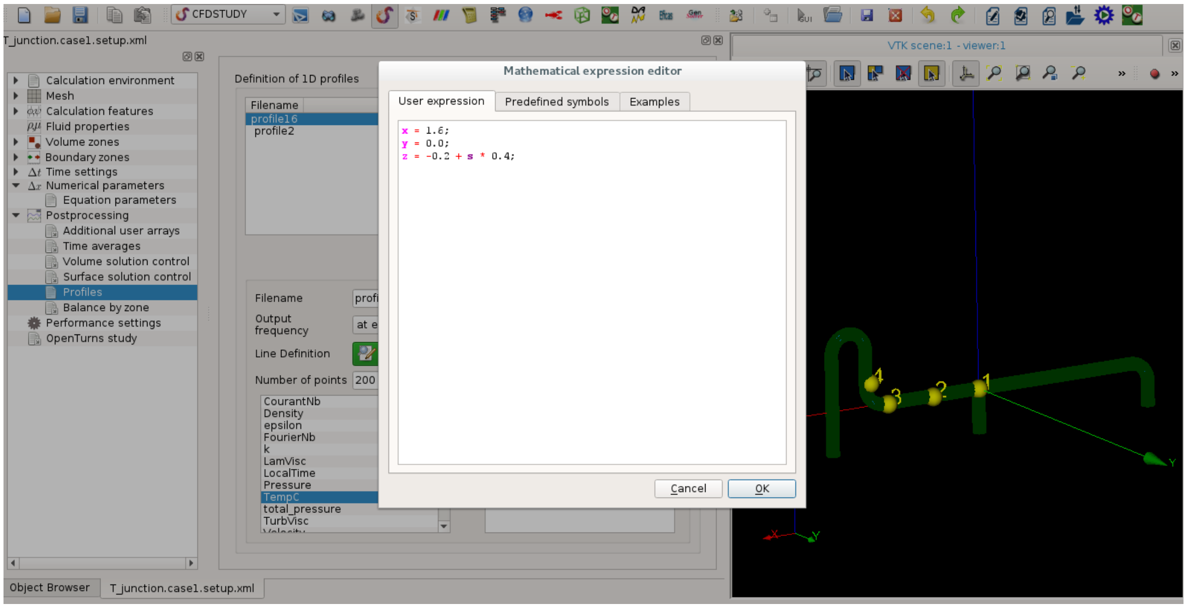

Figure III.21: Vertical profiles : Line definition

For the advanced post-processing features, copy into the SRC directory the file `cs_user_postprocess.c` from the directory `SRC/REFERENCE`. The general content of this routine is described in the user manual and some examples are available in the directory `SRC/EXAMPLES` 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_values** (in `cs_user_postprocess.c`):
  This routine is called at each time step. It allows to specify which variable will be written on which part.