
CFDSTUDY Documentation

Release 1

EDF R&D

January 12, 2010

CONTENTS

1	Introduction	1
2	Tutorials	3
2.1	<i>Code_Saturne</i> tutorial : turbulent mixing in a T-junction	3

INTRODUCTION

The **CFDSTUDY** is a component for the Salome platform. The purpose of this program is to provide an interface between CFD (Computational Fluid Dynamics) softwares *Code_Saturne* and NEPTUNE_CFD with other modules of the platform.

Code_Saturne and NEPTUNE_CFD are CFD softwares from EDF R&D. *Code_Saturne* could be freely downloaded from www.code-saturne.org

This document provides a tutorial for the use of CFDSTUDY with *Code_Saturne*. For a *Code_Saturne* tutorial itself, please consult the software documentation.

TUTORIALS

2.1 *Code_Saturne* tutorial : turbulent mixing in a T-junction

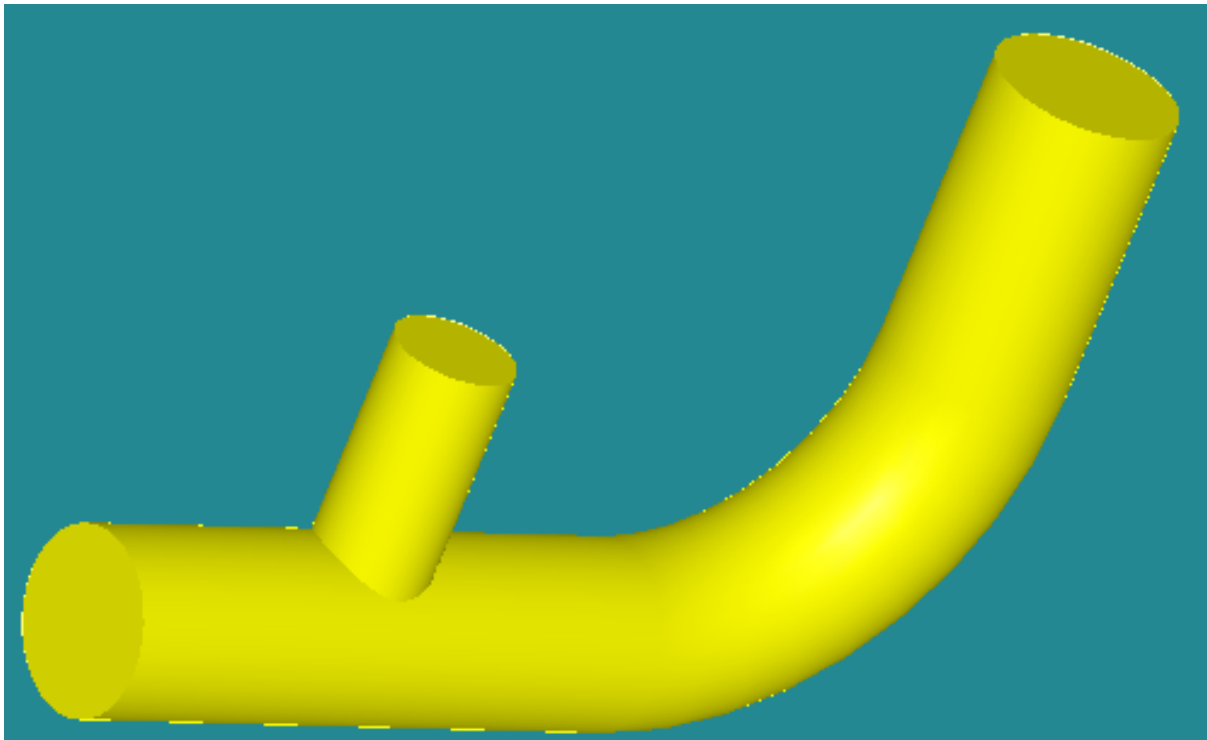
2.1.1 1. Introduction

This tutorial provides a complete complete course with *Code_Saturne*.

This tutorial is covering the following items :

- first, creation of the CAD design with the module **Geometry**
- then the meshing step with the module **Mesh**
- in order to do a CFD calculation, use of *Code_Saturne* through the module **CFDSTUDY**
- at last, post processing of the results with the module **Post-Pro**

The proposed case is on turbulent mixing between cold and hot water inside a pipe. The pipe is composed with a T-junction and an elbow. This exercise is inspired from a more complex study of thermal fatigue caused by the turbulent mixing of hot and cold flows just upstream of the elbow. Of course, the case is very simplified here.

**Main tube:**

- internal diameter $d = 0.3$ m
- first section: length = 1,0 m
- second section: elbow, rayon = 0,5 m
- third section: length = 0,5 m

Hot inlet:

- internal diameter $d = 0,2$ m
- section: length = 0,5 m

2.1.2 2. Prerequisites

Before starting the Salomé platform, it is necessary to update the environment variable **PYTHONPATH** so that the module **CFDSTUDY** knows the details of the installation of *Code_Saturne*. In order to do that, one should indicate in the variable PYTHONPATH where are the additional Python modules related to *Code_Saturne*. For example (sh):

```
export PYTHONPATH=/home/login/Code_Saturne/2.0/lib/python2.4/site-packages:$PYTHONPATH
```

If you want to put mathematical formula in the GUI of *Code_Saturne* the PYTHONPATH variable should be updating once again, in order to indicate the Python API of the **MEI** librarie of *Code_Saturne*. For example:

```
export PYTHONPATH=/home/login/Code_Saturne/2.0/lib/python2.4/site-packages/mei:$PYTHONPATH
```

- **Note:** the version of python must be the same between Salome and *Code_Saturne*.

2.1.3 3. CAD design with Geometry

The geometry is built by extrusion of disks along paths (i.e. lines and wires). We need to define two paths for the two tubes, and two disks which are faces built on circles. The two volumes obtained are regrouped into one volume (fusion).

After the construction of the solid, we have to define the **boundary conditions zones** for the CFD calculation: that is to say two inlet faces, the outlet face, and the internal wall of the tubes.

- **Note:** objects graphical manipulation in the 3D view (rotation, zoom, translation) can be done with `<Ctrl> + mouse buttons`.

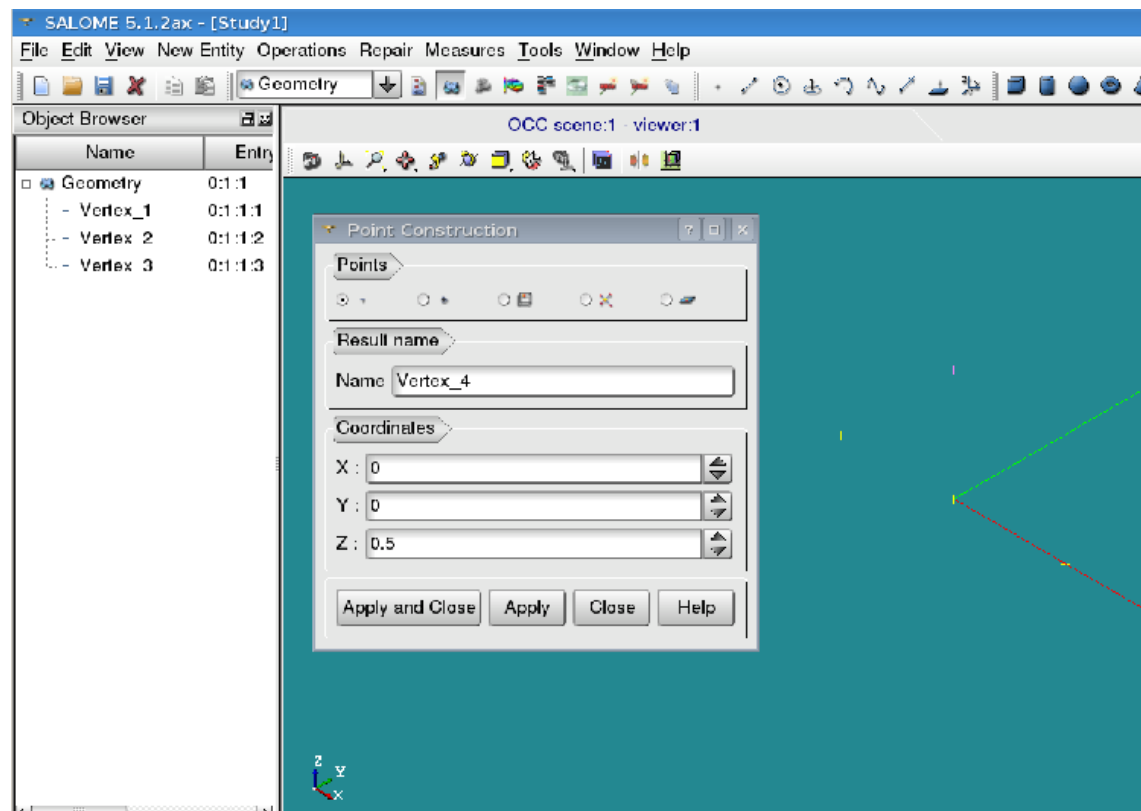
Activate the module **Geometry**.

3.1. Points, lines and wire

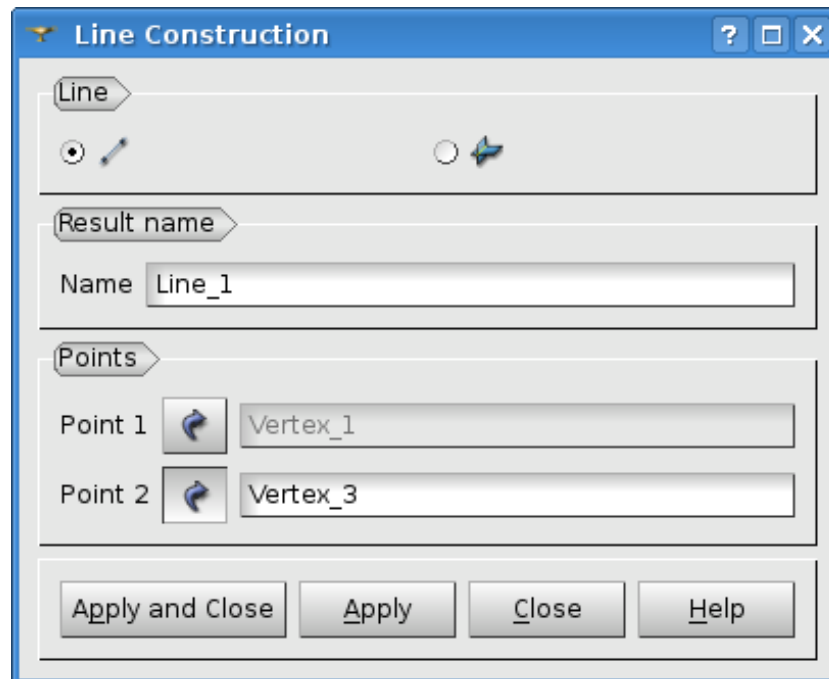
- Creation of points: select the menu “**New Entity > Basic > Point**” or click the toolbar button “**Create a Point**”. In the dialog window for the creation of the points create the following entities:

Name	X	Y	Z
Vertex_1	-0.14	0	0
Vertex_2	0	0	0
Vertex_3	0.076	0	0
Vertex_4	0	0.1	0
Vertex_5	0.076	0.095	0
Vertex_6	0.171	0.095	0
Vertex_7	0.171	0.24	0

The points are not visible without a zoom. After 3 or 4 new points, use the mouse wheel to zoom in.



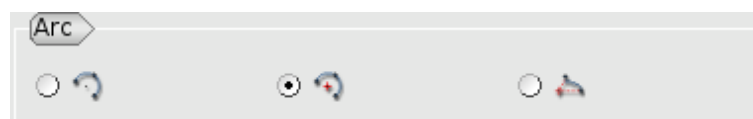
- Creation of the lines: select the menu “**New Entity > Basic > Line**” (or click the equivalent toolbar button). To define a line, select successively the begin and end point, either in **Object Browser** or in the 3D view.



Three lines must be defined:

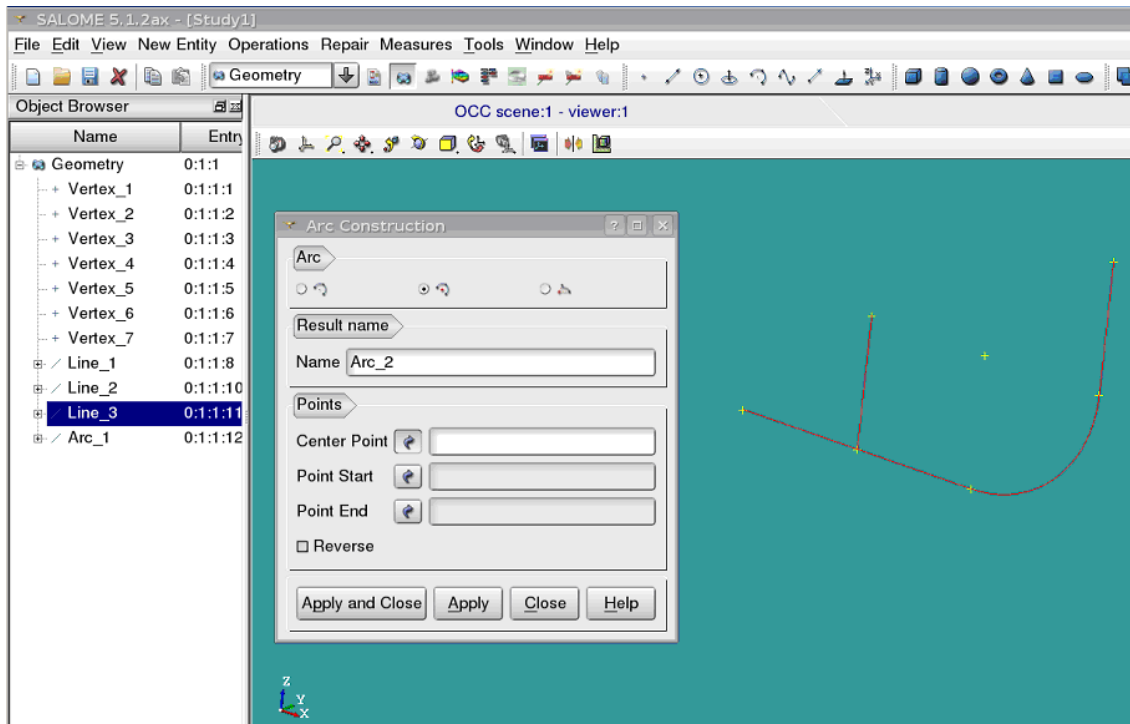
Name	Point1	Point2
Line_1	Vertex_1	Vertex_3
Line_2	Vertex_2	Vertex_4
Line_3	Vertex_6	Vertex_7

- Creation of the arc (a 1/4 of circle): select the menu “**New Entity > Basic > Arc**” (or click the equivalent toolbar button). Then, in the dialog window, select the second mode of creation (i.e. with a center point, and two points).

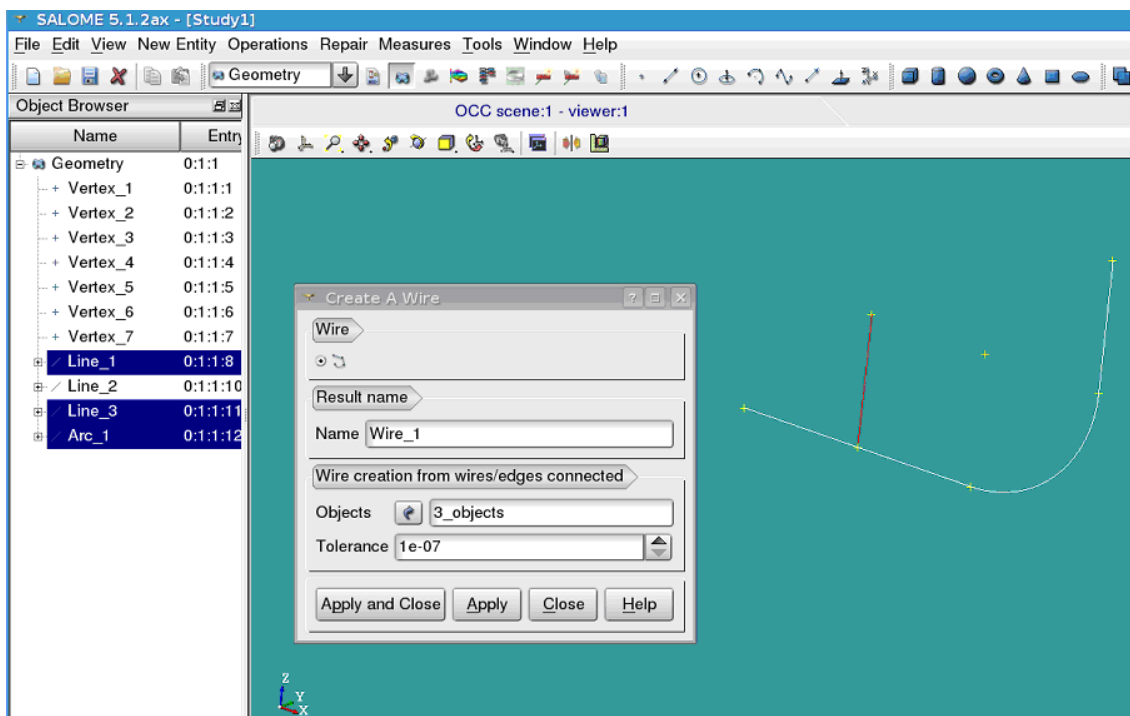


Then the arc must be defined:

Name	Center Point	Start Point	End Point
Arc_1	Vertex_5	Vertex_3	Vertex_6



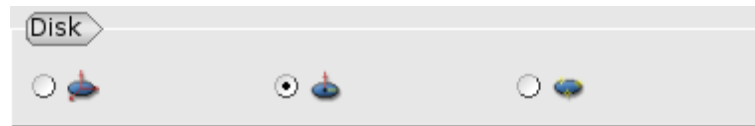
- Creation of the wire: select the menu “**New Entity > Build > Wire**”. To select together *Line_1*, *Arc_1* and *Line_3*, use <Ctrl> + right click in the **Object Browser**.



- **Note:** in order to create this wire, we could use also the menu “**New Entity > Sketch**”.

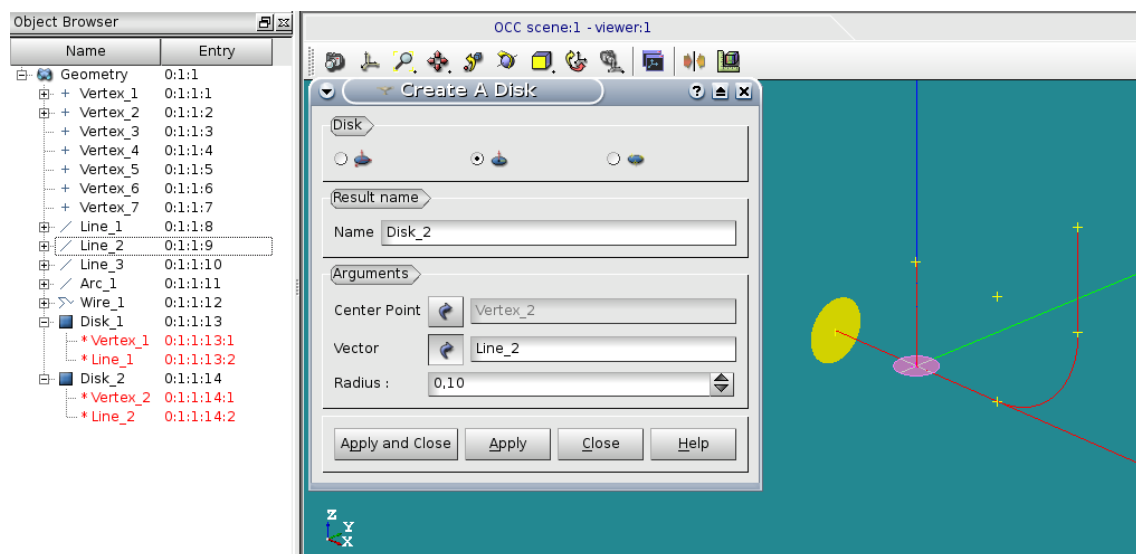
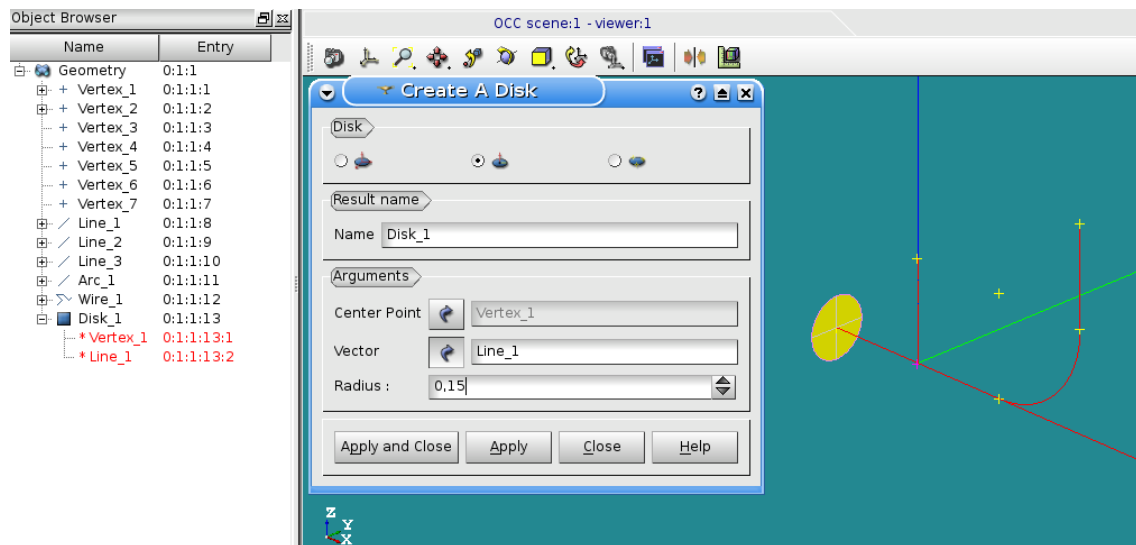
3.2. Faces and pipes

- Creation of the two disks: open the dialog window with the menu “**New Entity > Primitive > Disk**”. For each disk, in the dialog window, select the second mode of creation (i.e. with a center point, a vector and a radius).



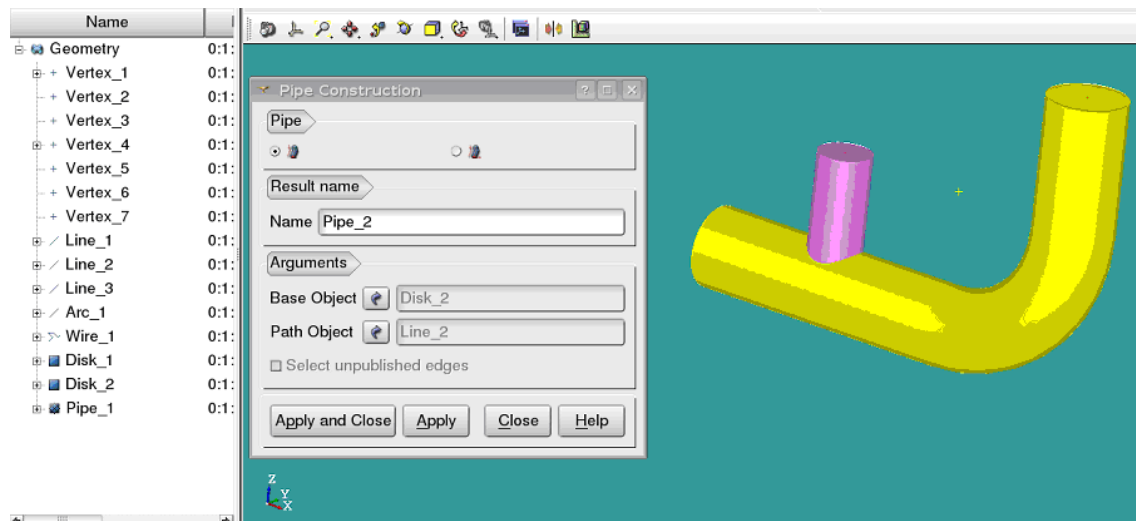
In the hierarchical geometric entities, these disks are faces.

Name	Center Point	Vector	Radius
Disk_1	Vertex_1	Line_1	0.036
Disk_1	Vertex_4	Line_2	0.036



- Creation of the two pipes: select the menu “**New Entity > Generation > Extrusion Along a Path**”. In our case the two paths are respectively: *Wire_1* and *Line_2*. In the hierarchical geometric entities, these pipes are solids.

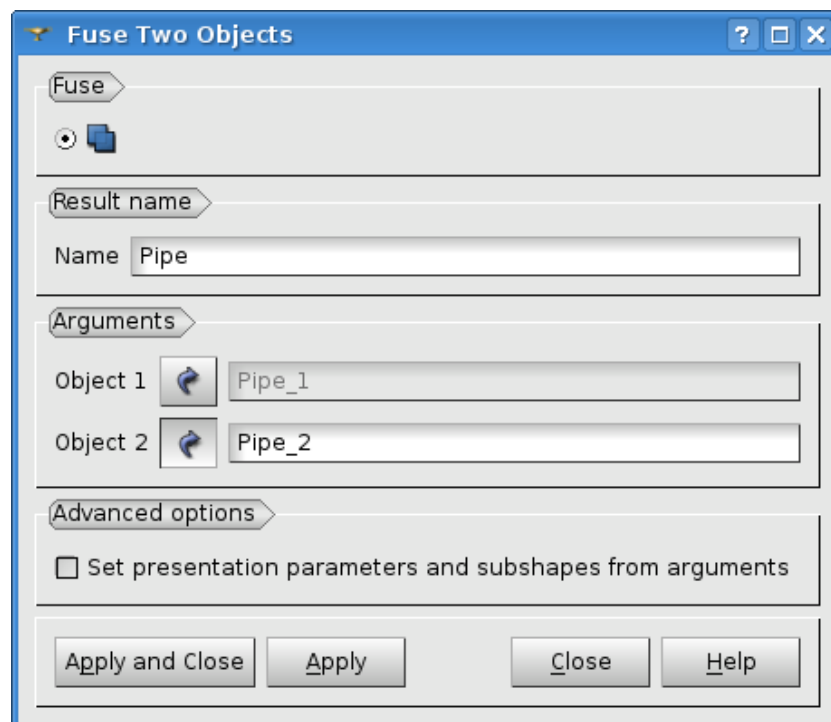
Name	Base Object	Path Object
Pipe_1	Disk_1	Wire_1
Pipe_2	Disk_2	Line_2



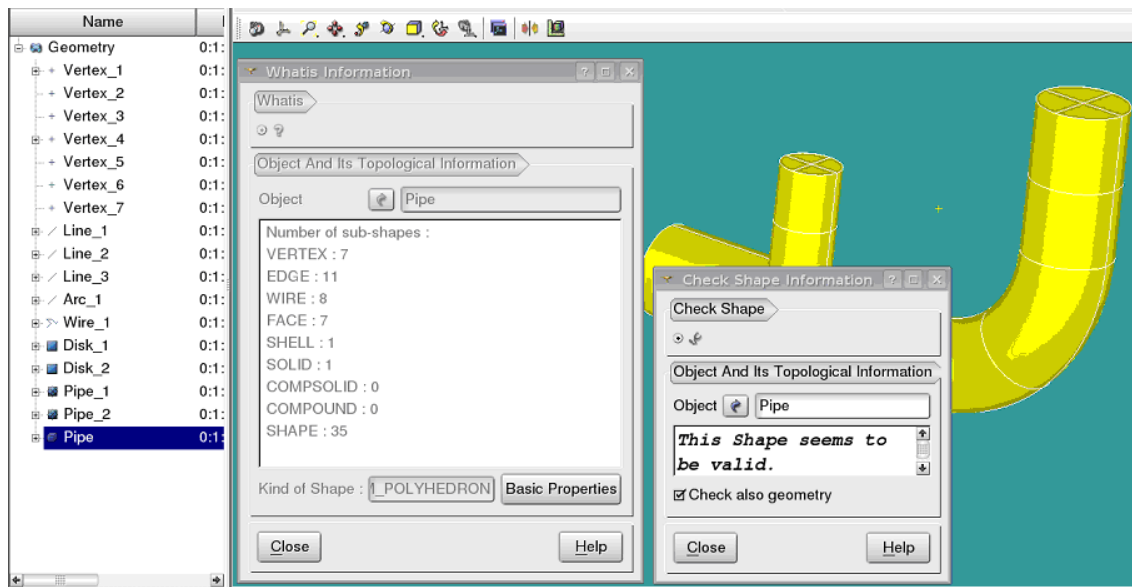
3.3. Fusion of the two pipes

- At that stage, we have build two separate solids. We must fuse these two solids into a single one. In order to do this fusion, select the menu “**Operations > Boolean > Fuse**”. Then rename the new object as *Pipe* (by default, is name is *Fuse_1*).

Name	Object 1	Object 2
Pipe	Pipe_1	Pipe_2

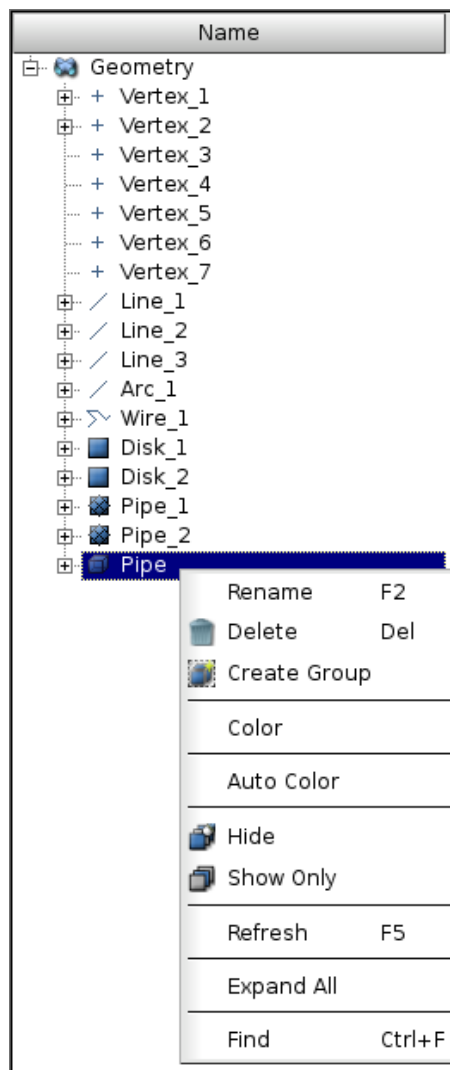


- Use the menus “**Measures > Check**” and “**Measures > What is**” to verify the object *Pipe*. It must be constituted of a single solid.



3.4. Groups for boundary conditions definition

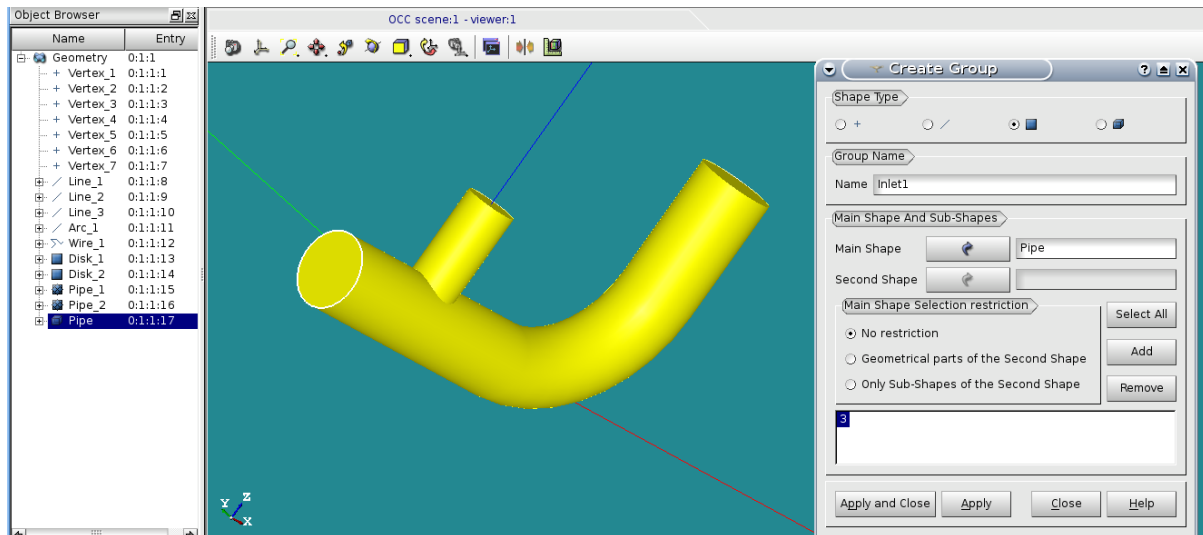
In the **Object Browser**, select the *Pipe* object, use popup menus “**Show only**” and “**Create group**”.



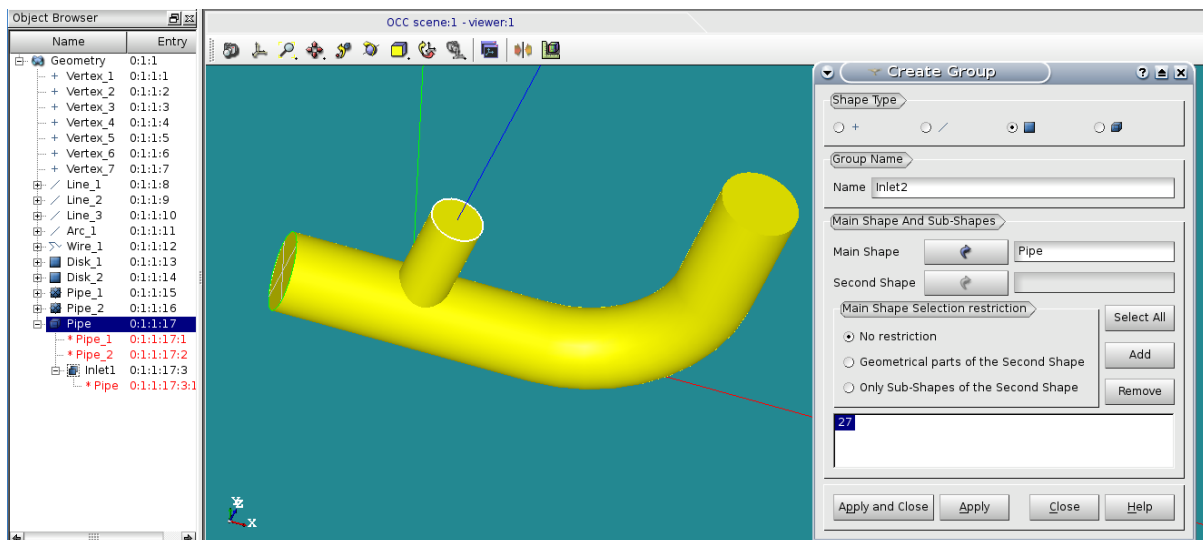
Select faces as shape (3rd button under Shape Type: one can select Vertices, Edges, Faces or Solids on a shape):

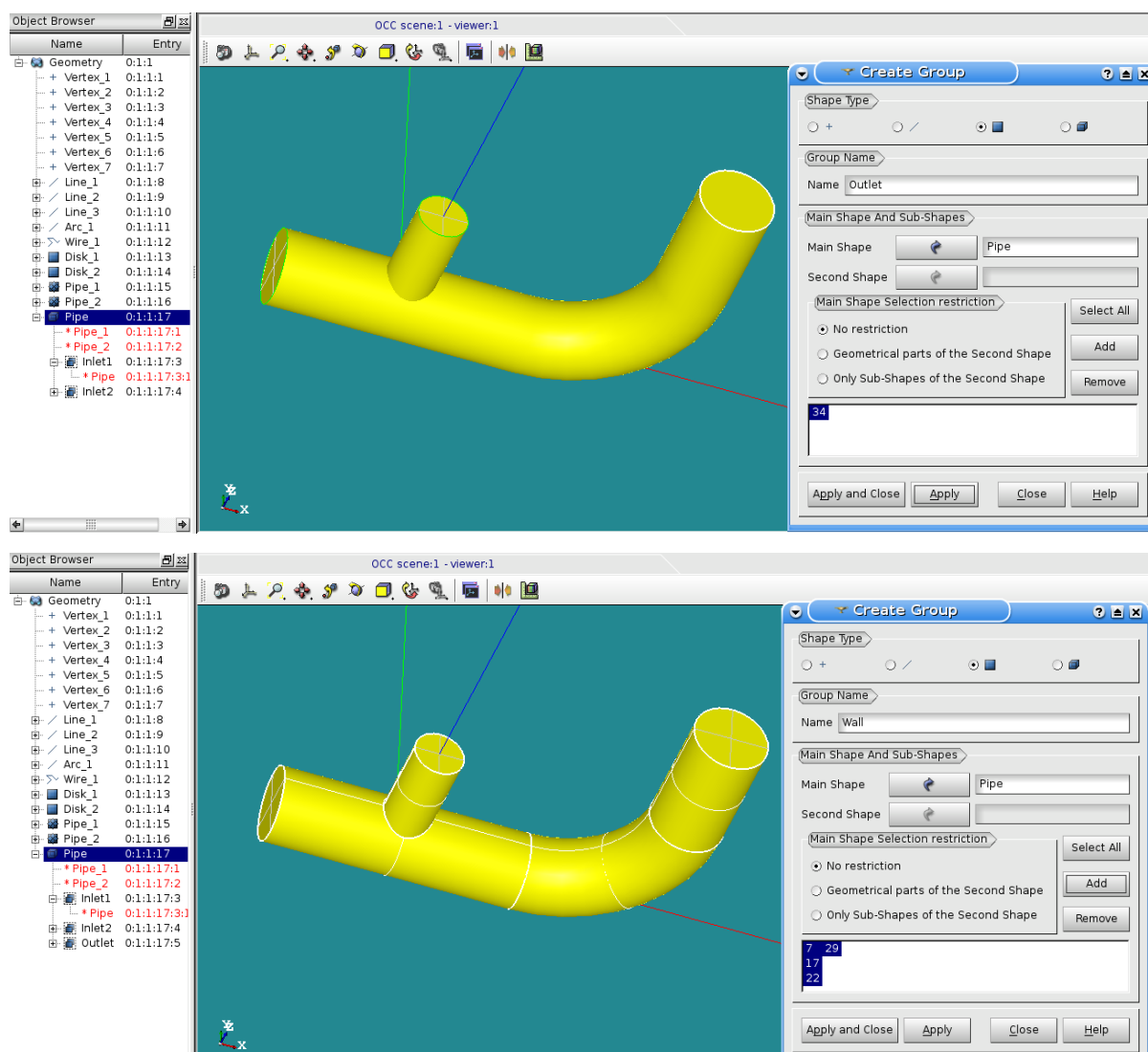


Give the name *Inlet1* to the new group and highlight (right click in the 3D view) the face corresponding to *Inlet1* on the *Pipe*. Then, push button “Add” (the number below identifies the face into the main shape), and apply. To be able to select a face, you may have to rotate the shape: **<Ctrl> + right click**.



Proceed as above for the 3 other groups: *Inlet2*, *Outlet* and *Wall*. For faces selection of “Wall”, use the **<Shift> + right click** to make a multiple selection: the wall is constituted with 4 faces.





The CAD model (i.e. *Pipe*) is ready for meshing. Save your study (“**File > Save**” or **<Ctrl> + S**).

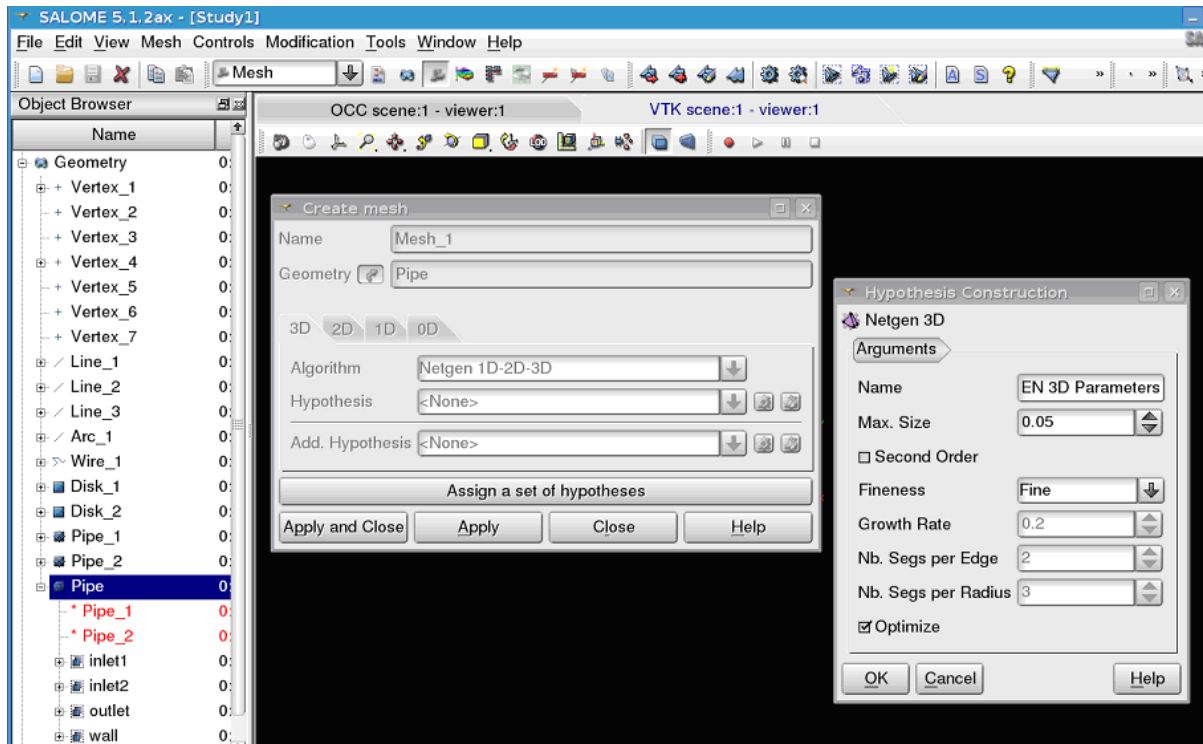
2.1.4 4. Meshing

In the scope of this tutorial, only the simplest way to mesh a CAD model is shown.

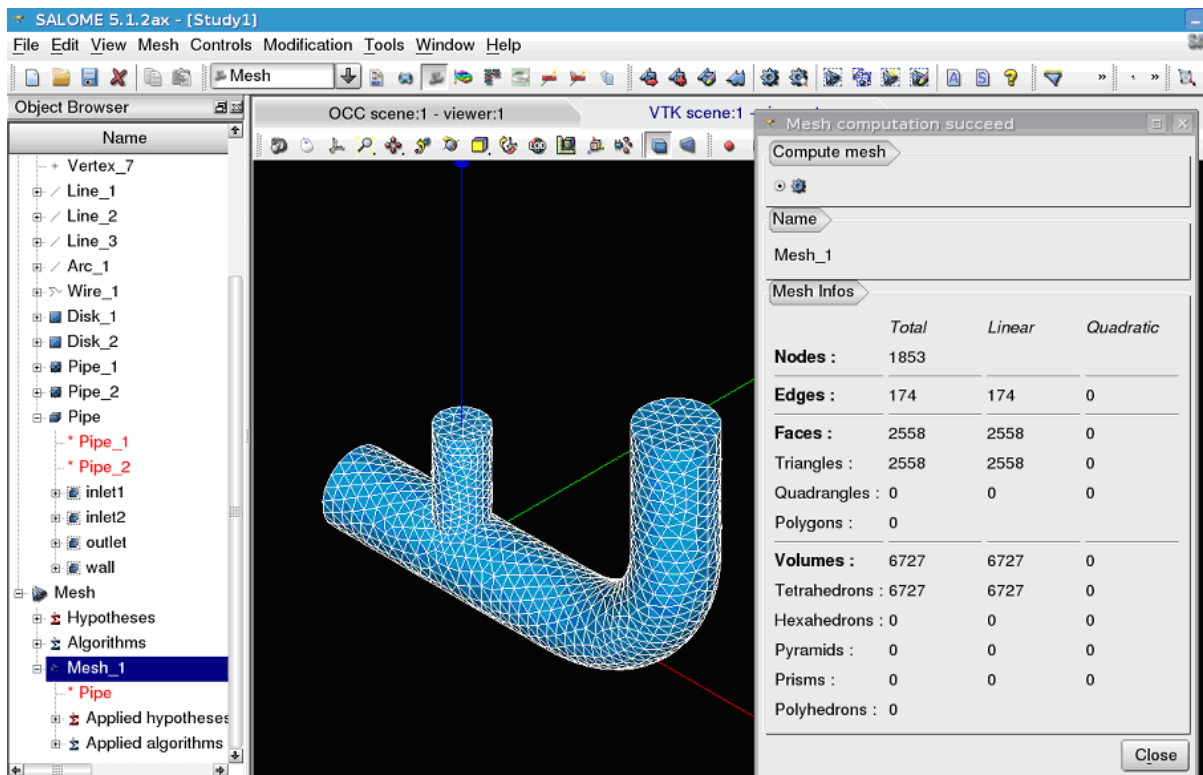
Activate the module **Mesh**.

4.1. Meshing with tetrahedrons, Netgen algorithm

- Select the *Pipe* object in **Object Browser**, then select menu “**Mesh > Create Mesh**”.
- In “**3D**” tab, select option “**Netgen 1D-2D-3D**” (nothing to do in the other tabs).
- Click on the only active button on “**Hypothesis**” line, and select “**NETGEN 3D Parameters**”.
- The “**Max. size**” corresponds to the maximal edge length of the tetrahedrons. Set the size is to 0.05. The “**Fineness**” governs the curves meshing. A fineness equal to “**fine**” will give approximately 6000 tetrahedrons, which is fine for this exercise.



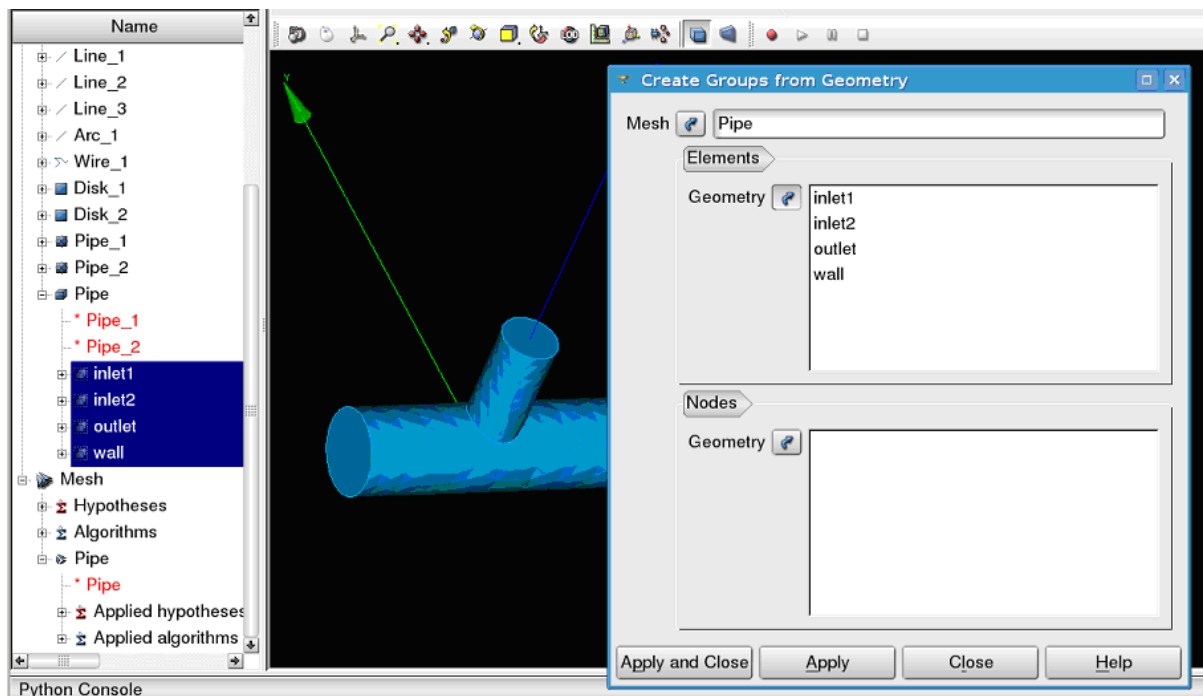
- After accepting the dialogs, select the new mesh in the **Object Browser** *Mesh_1*, and compute it by selecting the popup menu “**Compute**” or the toolbar button “**Compute**”.
- After a few seconds, the mesh is displayed, with an information dialog.



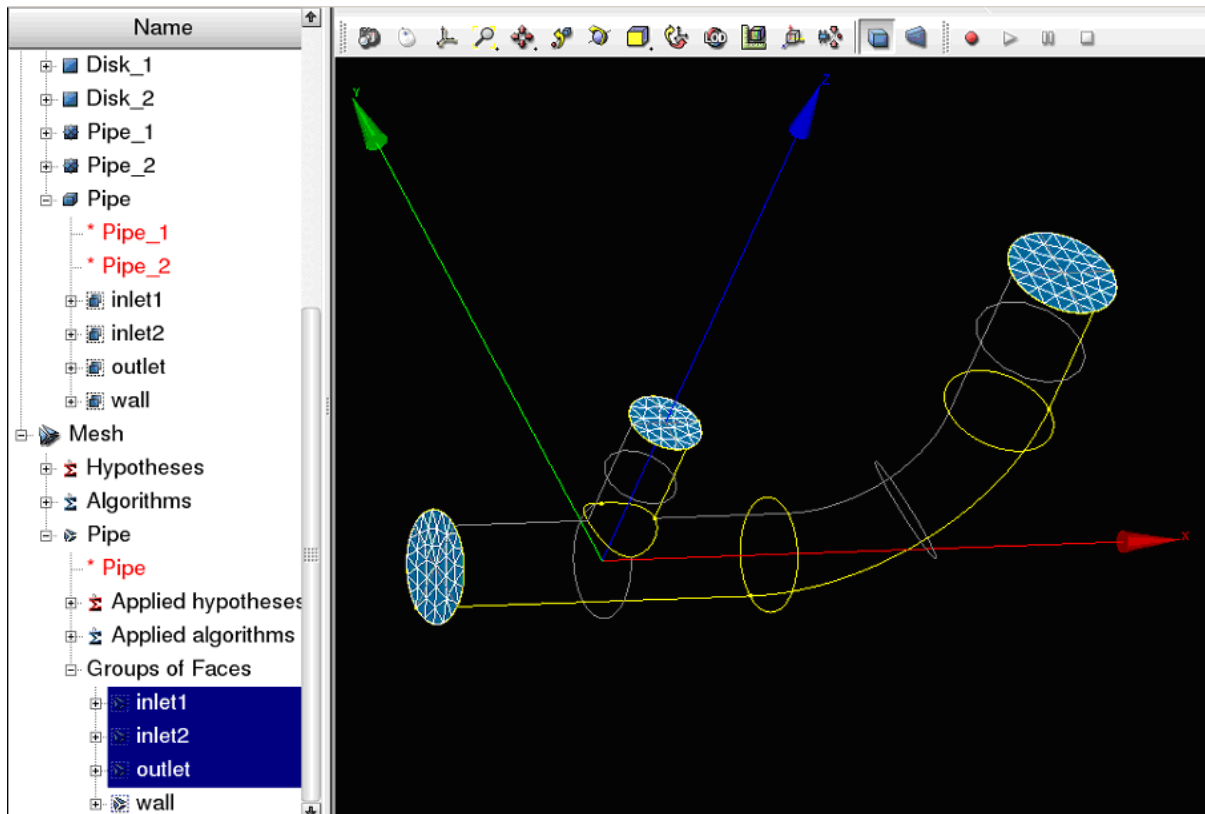
4.2. Groups on the mesh for boundary conditions definition

The groups defined on the CAD model for the boundary condition zones must have their counterparts in the mesh.

- Select the mesh *Mesh_1* in **Object Browser**, rename the mesh as *Pipe* with the popup menu “**Rename**”.
- Always with the mesh selected, create groups from Geometry (popup menu “**Create Groups from Geometry**”). In the **Object Browser** select the 4 groups defined on the CAD model. They appear in the dialog window. Apply.



- Display only the 3 groups corresponding to inlets and outlet, with the geometry in wireframe:

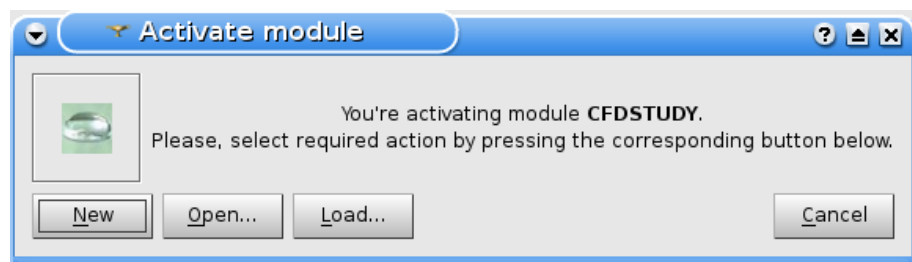


- Save the mesh in a MED file. Click left on mesh *Pipe* in **Object Browser** and select “**Export to MED File**”, and use the name *Pipe.med*.

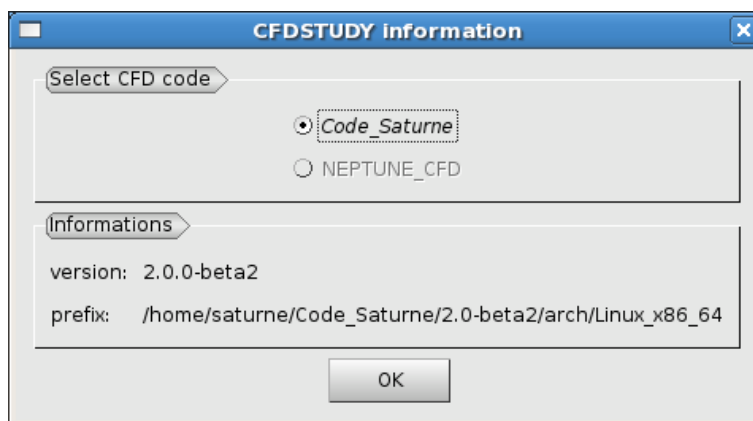
The mesh *Pipe* is ready for a CFD calculation. Save your study (“**File > Save**” or <Ctrl> + S).

2.1.5 5. CFD calculation with *Code_Saturne*

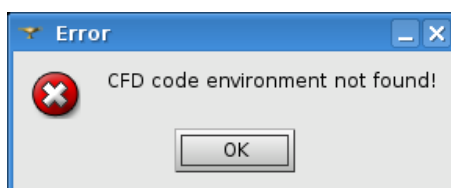
Activate the module **CFDSTUDY**.



- Click on “**New**”. A dialog window displays information about *Code_Saturne* installation.



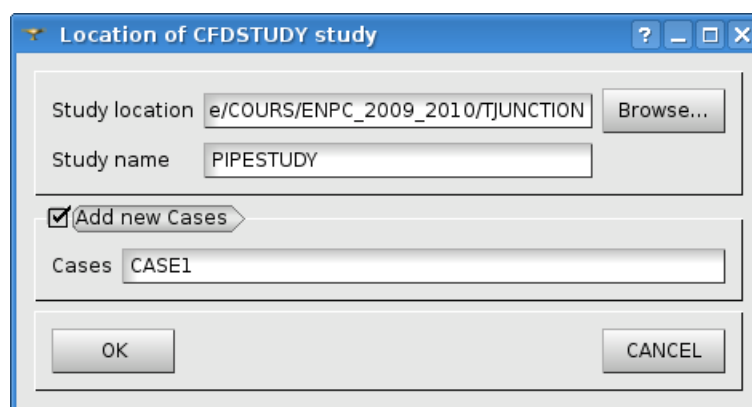
If the installation of *Code_Saturne* is not found (see section 2) the following error message is displayed:



5.1. CFD study and case creation

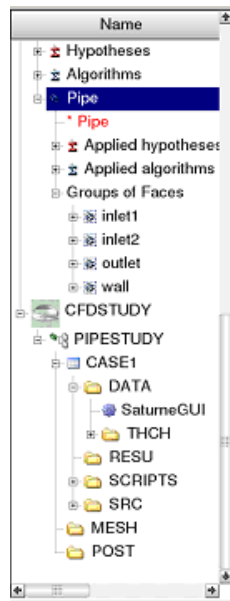
By convention, CFD calculations with *Code_Saturne* are organized in studies. Several calculations that share the same meshes and data sets, define a study for *Code_Saturne*. Each data set defined in a case.

- Create a CFD study and a case by selecting the menu “**CFDSTUDY > Set CFD study location**” (or the equivalent button in the toolbar).
- Use “**Browse**” button to select the directory which will contain the study directory. In our scope, the study will be named *PIPESTUDY*, and the case *CASE1*.



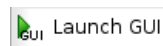
The new study directory with the new case is created with its sub directories and files.

- The **Object Browser** reflects the study structure on the directory :

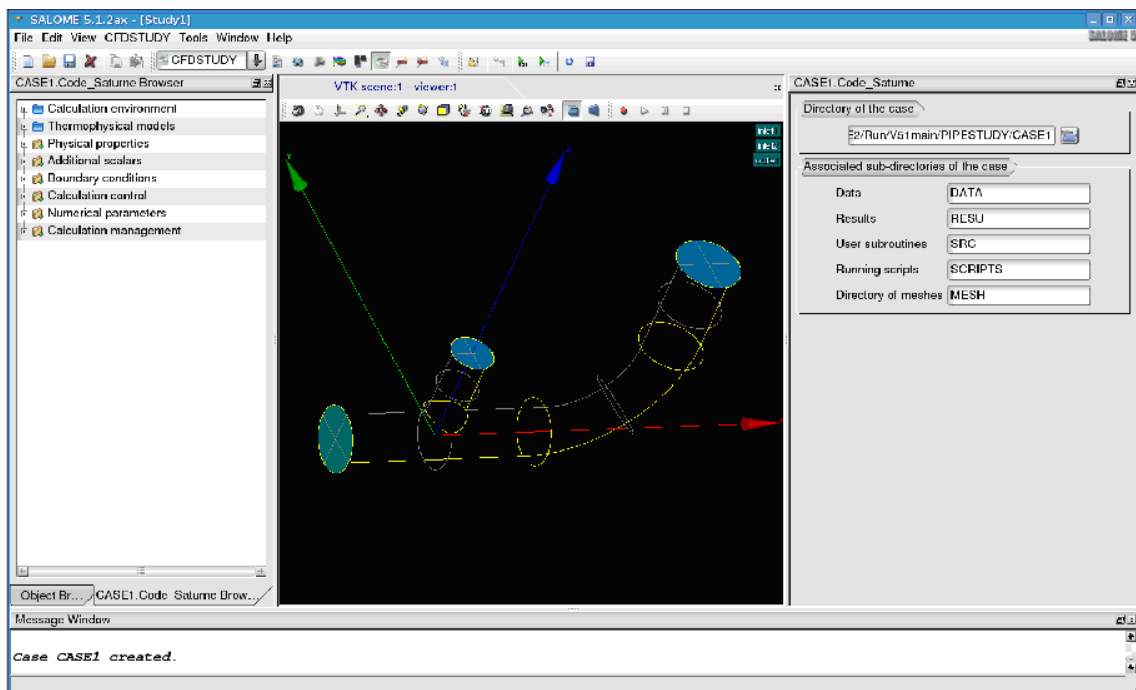


5.2. Open the *Code_Saturne* GUI

- Open the *Code_Saturne* GUI by selecting *CASE1* or *SaturneGUI* with the left mouse button in **Object Browser** and click right on menu “**Launch GUI**”:



- Then a window dialog appear, click on “**Activate**”. The *Code_Saturne* GUI open itself in the Salome desktop.



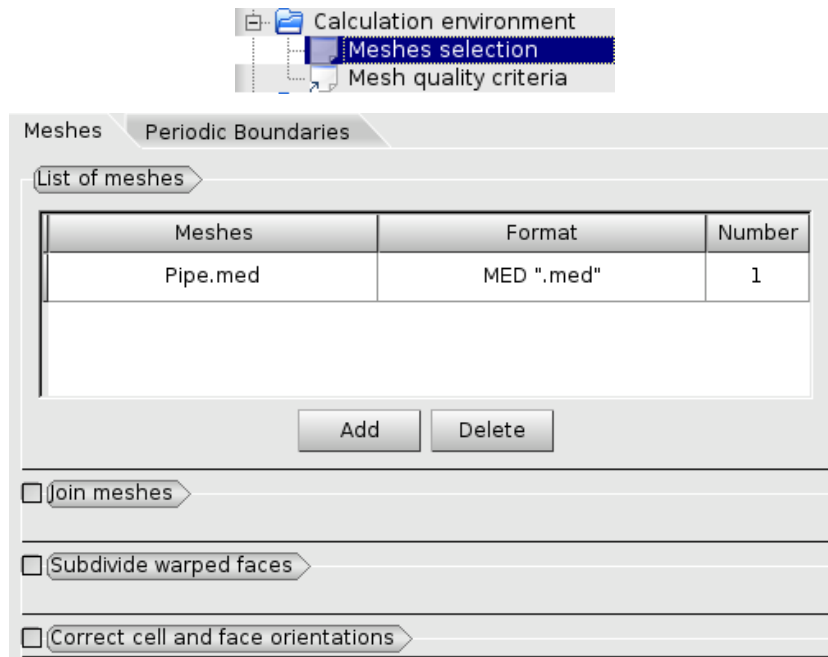
On the left dockWidget, the salome **Object Browser** and the navigation tree of the GUI are grouped on tabs. When an item of the tree is selected, the corresponding panel raises in the GUI.

5.3. Define the CFD calculation

Now we start to input data for the CFD calculation definition. In the scope of this tutorial, we do not have to explore all the panels of the tree (from top to bottom), because lot of default values are good, so we just have to fill a few panels.

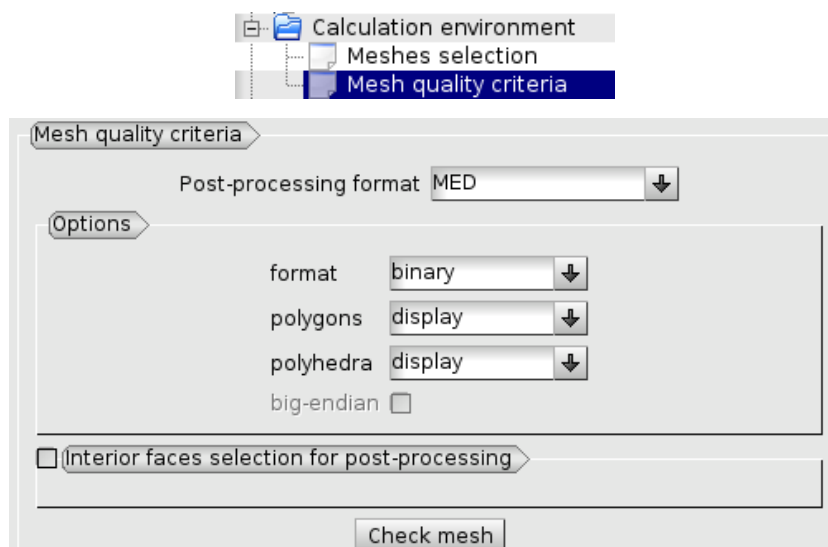
5.3.1 Location of the mesh file

Open “**Mesher selection**”. Use “**Add**” button to open a file dialog, and select the MED file previously saved.

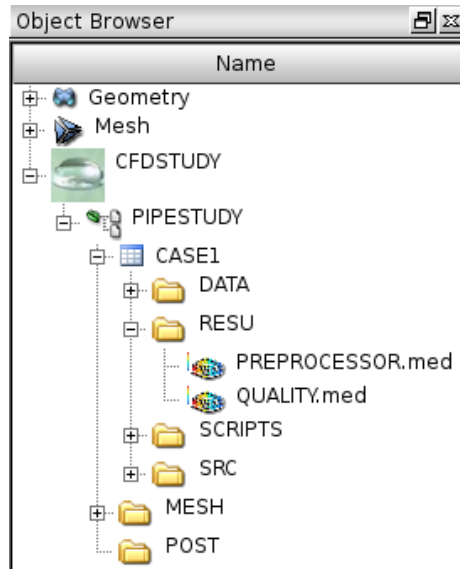


5.3.2 Mesh quality criteria

Open “**Mesh quality criteria**”. Verify that the “**Post-processing format**” is chosen to MED. Click on “**Check mesh**” button.



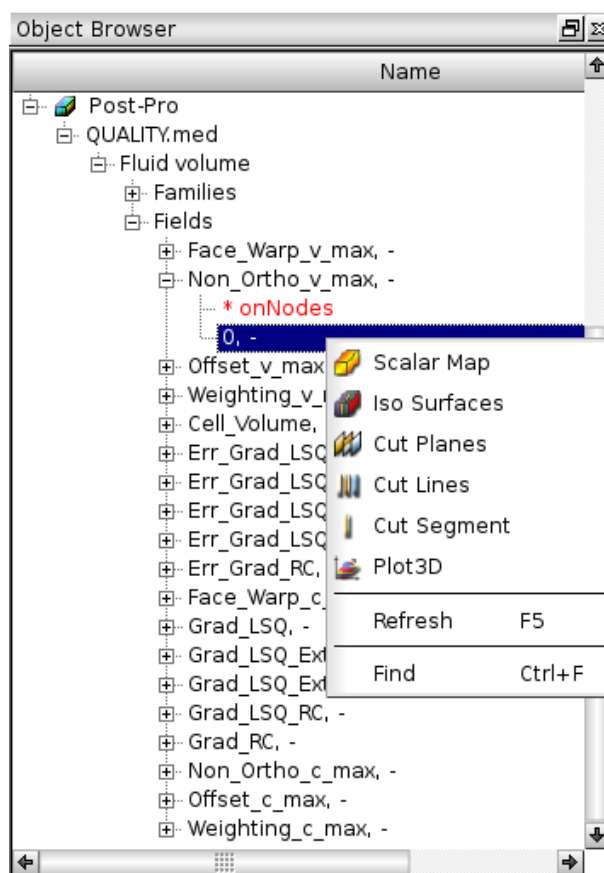
The GUI displays a listing with information about quality. Then, refresh the **Object Browser** with the toolbar button “**Updating Object browser**”. There are two new MED file in the directory *RESU*: *PREPROCESSOR.med* and *QUALITY.med*.



The file *PREPROCESSOR.med* contains information on groups location. The file *QUALITY.med* contains quality criteria as fields. In order to visualize these quality criteria, export *QUALITY.med* in the **Post-Pro** module (click left and select “**Export in Post-Pro**”).



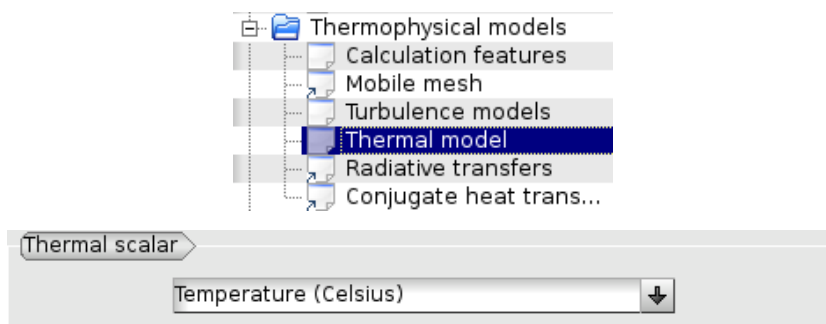
Then activate the module “**Post-Pro**”, select the criteria to display (for example click left and select “**Scalar Map**”):



After exploring mesh quality criteria, re-activate the module **CFDSTUDY** in order to continue the data input.

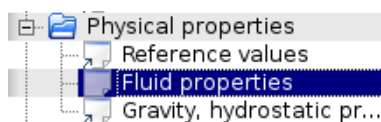
5.3.3 Thermophysical models

Open “**Thermal model**” and choose *Temperature (Celsius)*.



5.3.4 Fluid properties

Open “**Fluid properties**”.



Here the tutorial proposes two options:

5.3.4.1 Constant properties

- Use constants for water at 19 degrees Celsius.

The screenshot shows four panels for setting material properties to constant values:

- Density:** Set to 'constant' with a reference value of $\rho = 998$ kg/m³.
- Viscosity:** Set to 'constant' with a reference value of $\mu = 0.001$ Pa.s.
- Specific heat:** Set to 'constant' with a reference value of $C_p = 4181$ J/kg/K.
- Thermal conductivity:** Set to 'constant' with a reference value of $\lambda = 0.6$ W/m/K.

5.3.4.2 Variable properties

- *This section is optional.* User laws are proposed for density, viscosity and thermal conductivity. First, fill all properties like the section above, and then for density, viscosity and thermal conductivity, select “user law”, and open the window dialog in order to give the associated formula:

- density: $\rho = 1000.94843 - 0.049388484 * \text{TempC} - 0.000415645022 * \text{TempC}^2;$

The 'User expression' dialog box shows the formula: $\rho = 1000.94843 - 0.049388484 * \text{TempC} - 0.000415645022 * \text{TempC}^2;$

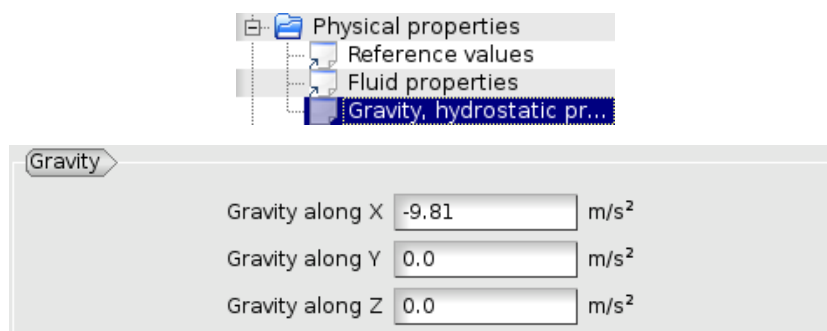
- viscosity: $\mu = 0.0015452 - 3.2212e-5 * \text{TempC} + 2.45422 * \text{TempC}^2;$

The 'User expression' dialog box shows the formula: $\mu = 0.0015452 - 3.2212e-5 * \text{TempC} + 2.45422 * \text{TempC}^2;$

- thermal conductivity: $\lambda = 0.57423867 + 0.01443305 * \text{TempC} - 9.01853355e-7 * \text{TempC}^2;$

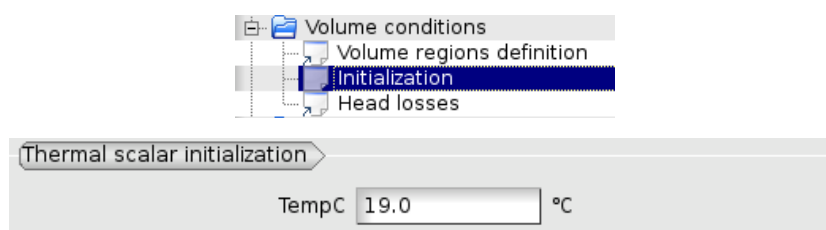
The 'User expression' dialog box shows the formula: $\lambda = 0.57423867 + 0.01443305 * \text{TempC} - 9.01853355e-7 * \text{TempC}^2;$

To take into account the effects of buoyancy, we have to impose a non-zero gravity.



5.3.5 Initialization

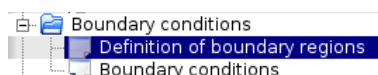
The initial temperature of the water in the pipe is set to 19 degrees.



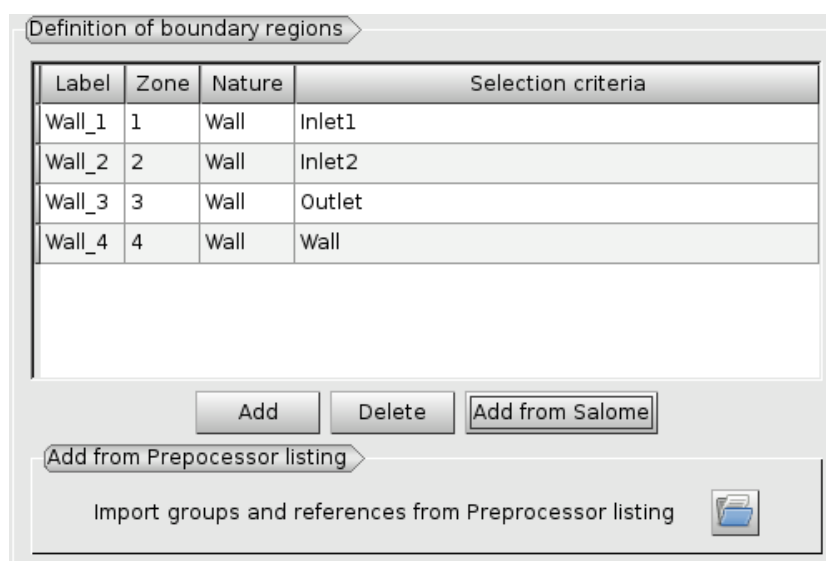
5.3.6 Boundary conditions

5.3.6.1 Define locations graphically

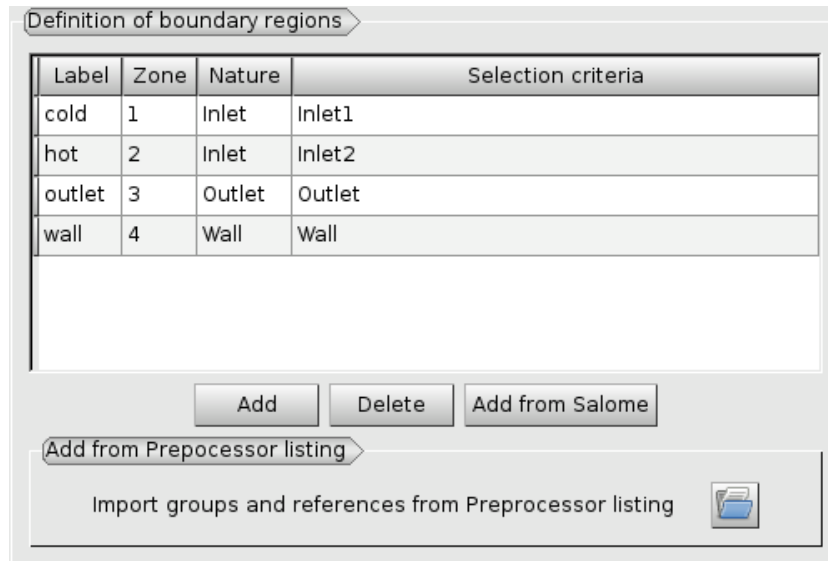
- Open “Definition of boundary regions”.



Highlight successively each group of the mesh *Pipe*, by selecting the name of the group in the **Object Browser** or by clicking the group in the VTK scene. When the group is highlighted, click on the “Add from Salome” button.

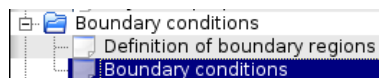


By default the nature of each new imported group is *Wall*. Double click in the cell of the nature in order to edit it. In the same way, edit the label of the boundary condition zone.



5.3.6.2 Boundary conditions values

- Open “**Boundary conditions**”. For each inlet, give norm for the velocity, the hydraulic diameter for the turbulence, and the prescribed value for the temperature.



Boundary conditions

Label	Zone	Nature	Selection criteria
cold	1	inlet	Inlet1
hot	2	inlet	Inlet2
outlet	3	outlet	Outlet
wall	4	wall	Wall

Velocity

norm
0.617 m/s

Direction

normal direction to the inlet

Turbulence

Calculation by hydraulic diameter

Hydraulic diameter 0.072 m

Scalars

Scalar Name	Type	Value	Exchange Coefficient
TempC	Prescribed v...	19	

Boundary conditions

Label	Zone	Nature	Selection criteria
cold	1	inlet	Inlet1
hot	2	inlet	Inlet2
outlet	3	outlet	Outlet
wall	4	wall	Wall

Velocity

norm m/s

Direction

normal direction to the inlet

Turbulence

Calculation by hydraulic diameter

Hydraulic diameter m

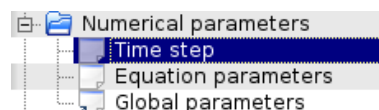
Scalars

Scalar Name	Type	Value	Exchange Coefficient
TempC	Prescribed v...	52	

5.3.7 Numerical parameters

5.3.7.1 Time step

- In the “**Time step**” heading, set 0.0002 s for the time step. The number of iterations is set to 1000.



Unsteady flow algorithm management

Time step option

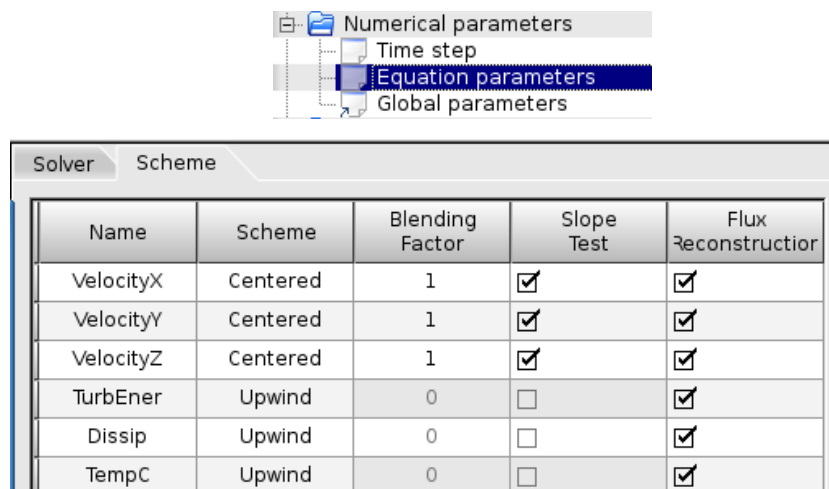
Reference time step s

Number of iterations (restart included)

Option zero time step ☐

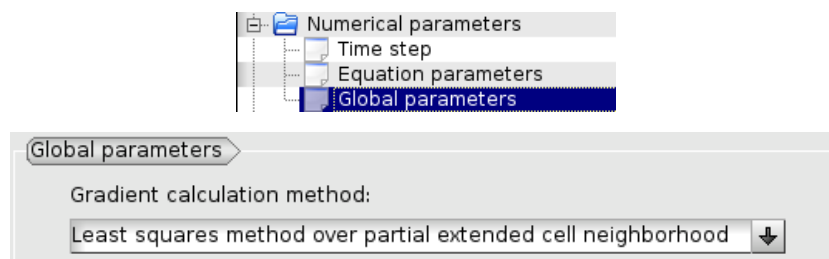
5.3.7.2 Equation parameters

- In order to save computation time, in the “**Solver**” tab, the precision is increase to 0.00001 (select all the concerned cells, and <Shift> + *double right click* to edit all cells in a single time).



5.3.7.3 Global parameters

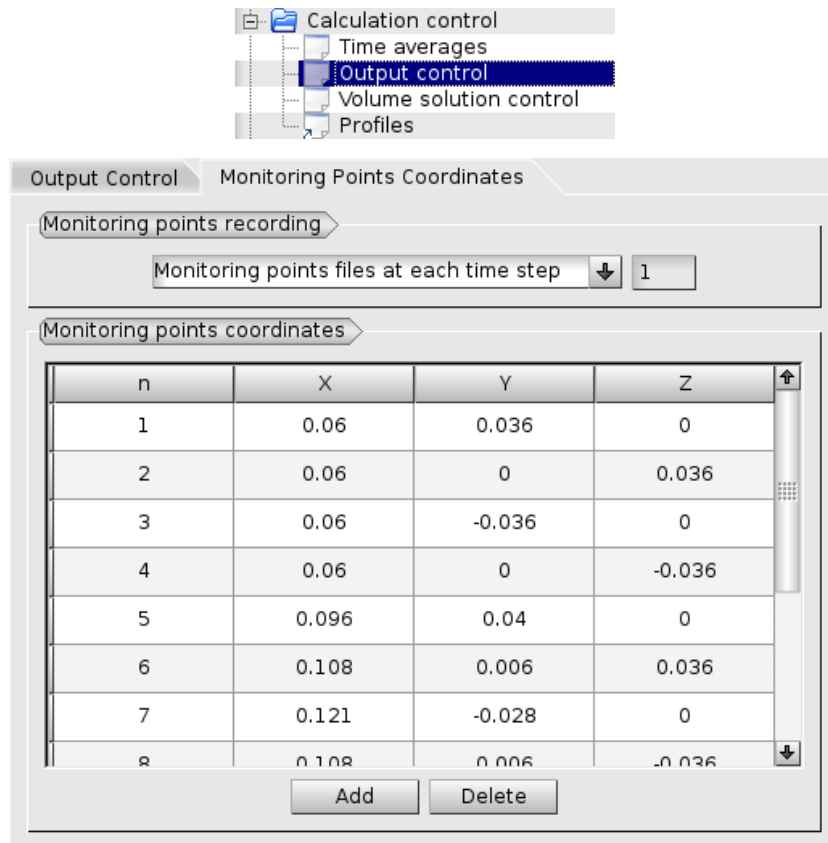
- The default gradient calculation method is changed for *Least Squares method over partial extended cell neighborhood*, which is better for full tetrahedrons mesh.



5.3.8 Calculation control: define monitoring points

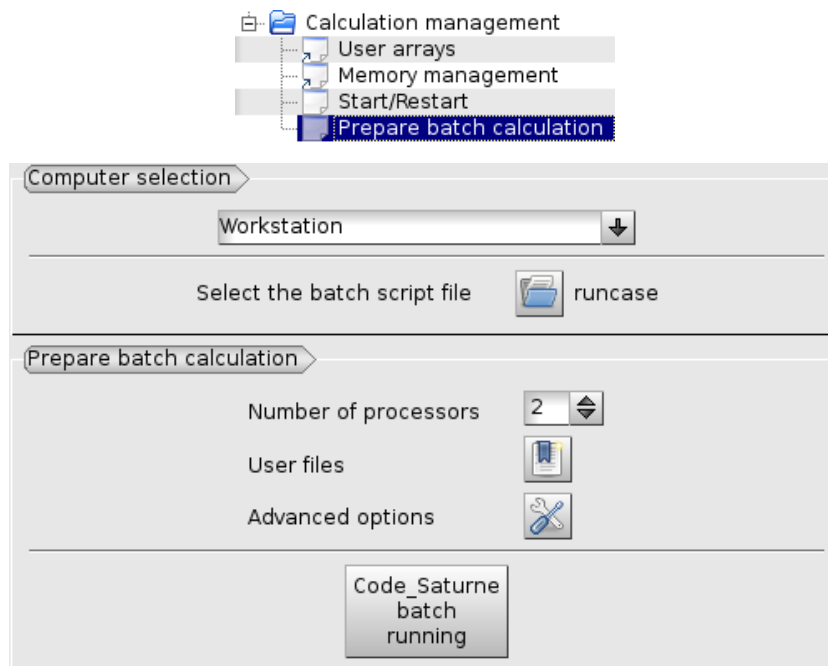
The purpose of the monitoring points is to record for each time step, the value of selected variables. It allows to control stability and convergence of the calculation.

Number	X	Y	Z
1	0.06	0.036	0
2	0.06	0	0.036
3	0.06	-0.036	0
4	0.06	0	-0.036
5	0.096	0.04	0
6	0.1	0.006	0.036
7	0.121	-0.028	0
8	0.1	0.006	-0.036
9	0.135	0.113	0
10	0.171	0.113	0.036
11	0.207	0.113	0
12	0.171	0.113	-0.036

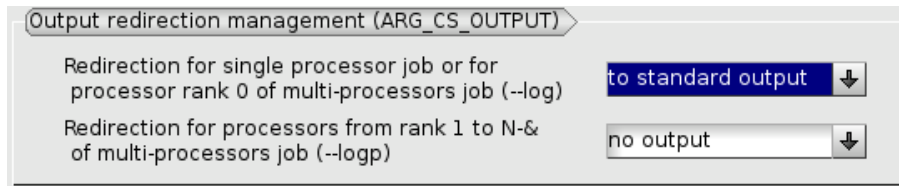


5.4 Calculation

Select “**Prepare batch calculation**”.

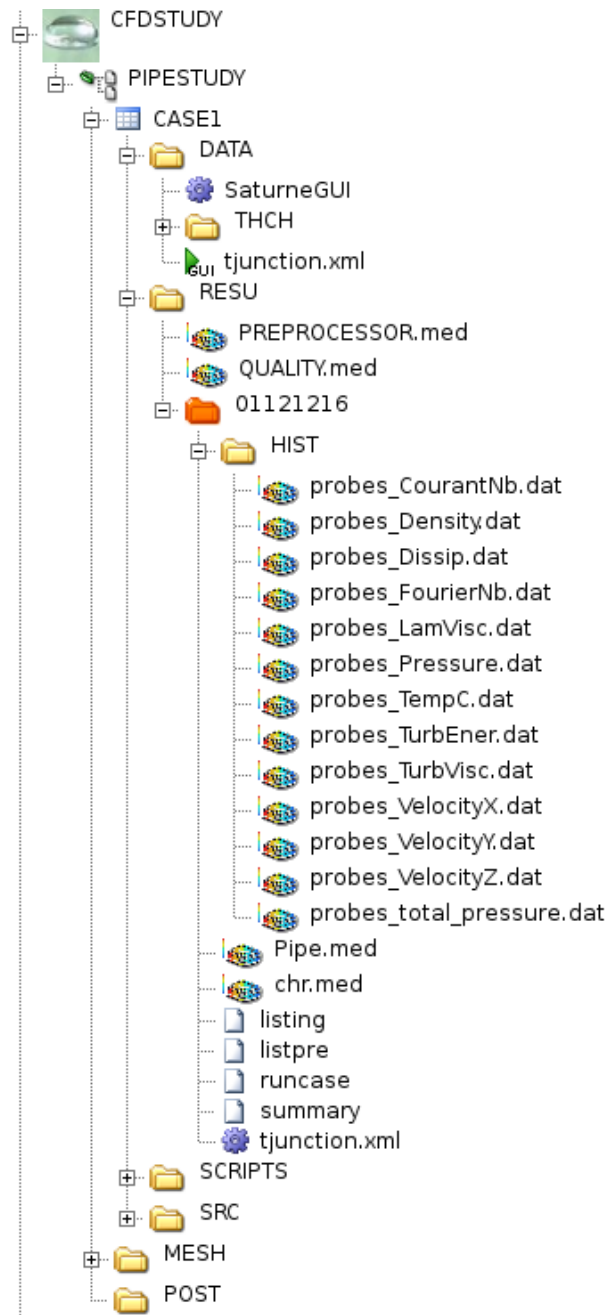


Before running *Code_Saturne*, save the case file (toolbar button or “**File > Code_Saturne > Save as data xml file**” or <Shif> + <Ctrl> + S), with the name “tjunction.xml” (extension .xml must be explicit). It is possible to see the listing in real time, in order to do that in the “**Advanced Options**” the option *to listing* must be replaced by *to standard output*.



Click on Button “Code Saturne batch running”.

When the calculation is finished (success or error), a new folder appears in the **Object Browser**, in “RESU” folder under “CASE1”. The **Object Browser** looks like:



Export the result *chr.med* and the probes files (extension *.dat*) into the **Post-Pro** module, with the popup menu “**Export in Post Pro**”.



In case of troubles, check these causes:

- the **Object Browser** does not reflect correctly the study (try the popup menu “**Update Object Browser**” on *PIPESTUDY*)
- the **Object Browser** is not correctly refreshed (popup menu *Refresh* in the **Object Browser**),
- if nothing, look at the temporary directory for the calculation, in \$HOME/tmp_Saturne. Listings of compilation and execution are here.

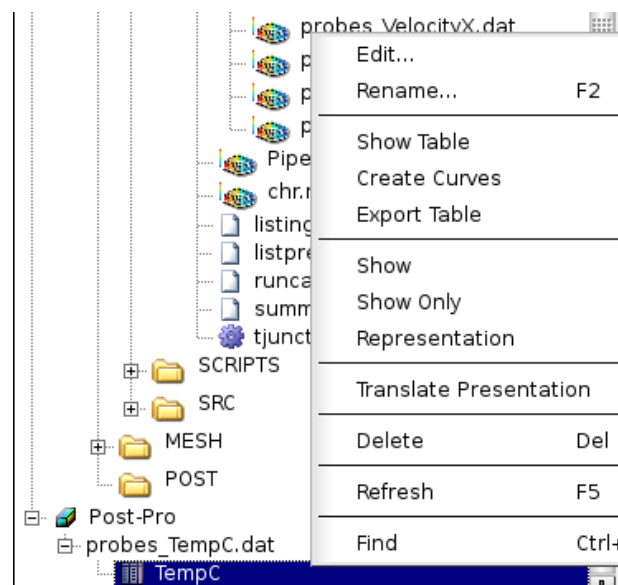
2.1.6 6. Post processing of the solution

6.1 Create curves for the monitoring points

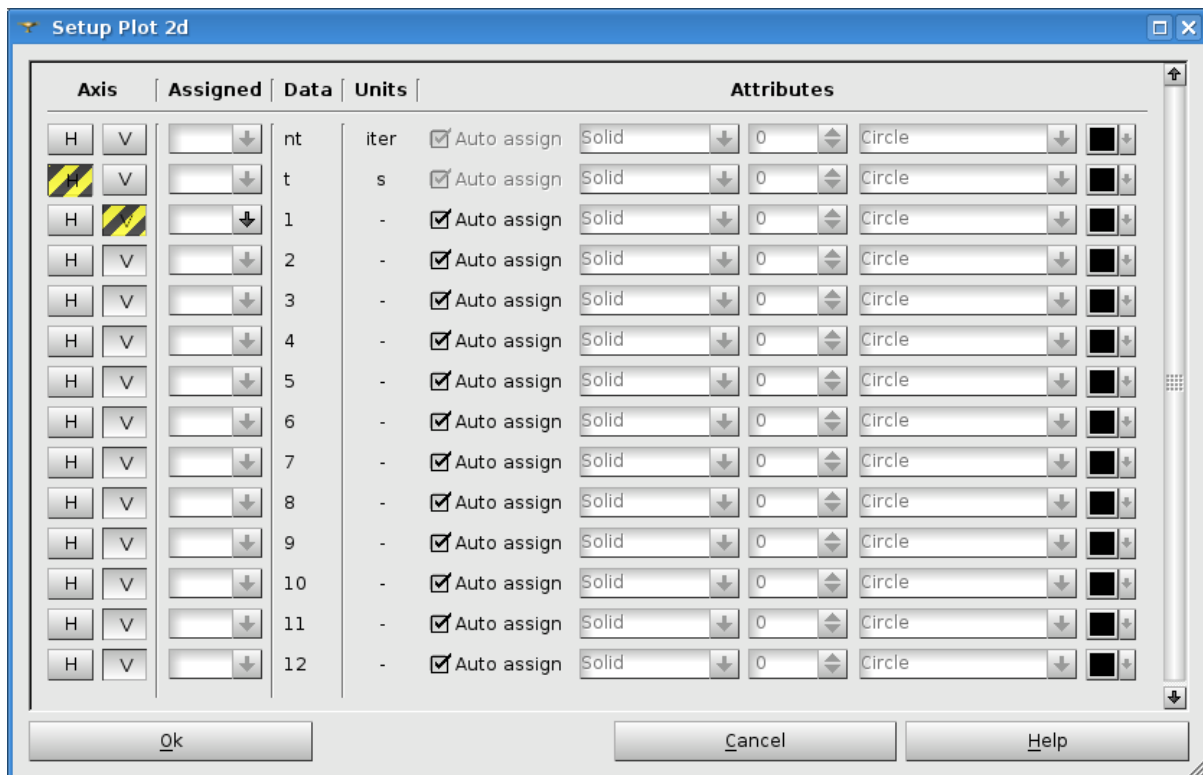
First, export in the **Post-Pro** module the files of monitoring points (extension *.dat*) to be created. For example, export the monitoring points concerning the temperature: *probes_TempC.dat* :



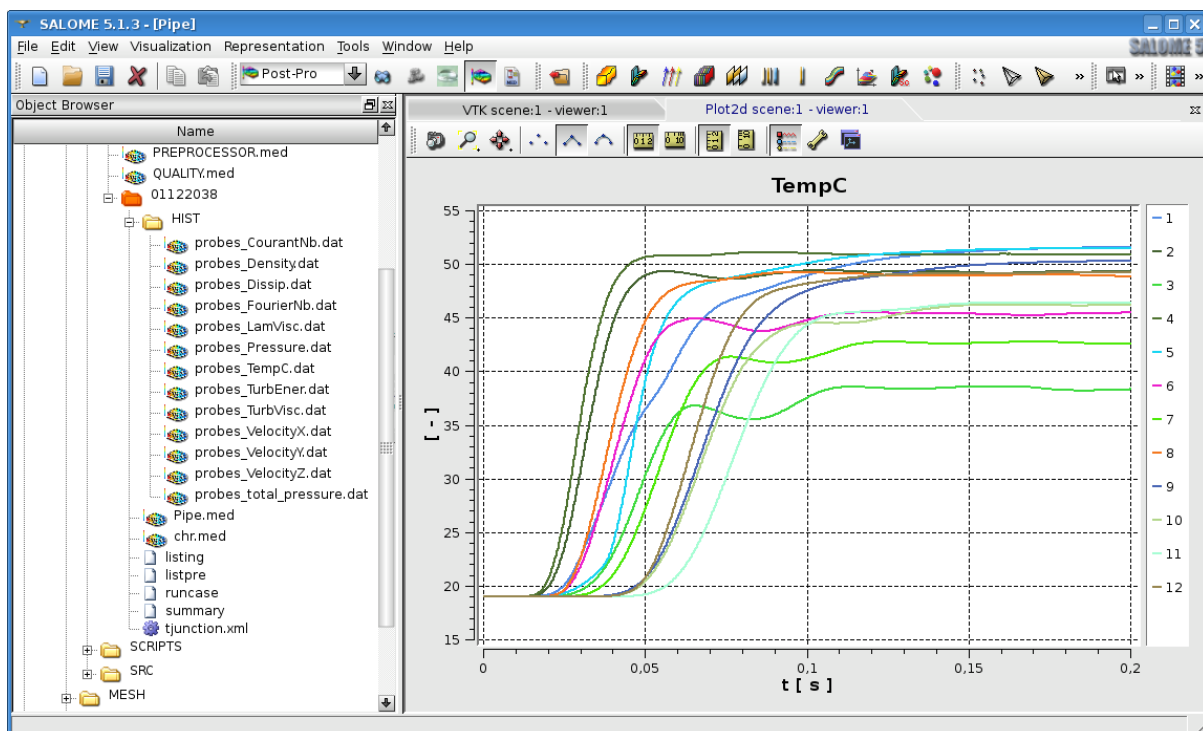
Then activate the **Post-Pro** module. Select the popup menu “**Create Curves**” (click left on *TempC*)



In the dialog window “**Setup Plot 2d**” click on the two marked buttons:

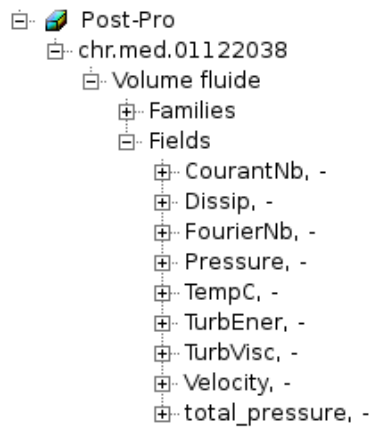


Post-Pro ask if *Do you want to choose all items with the same units for vertical axis?*. Answer *Yes* and click *Ok*.

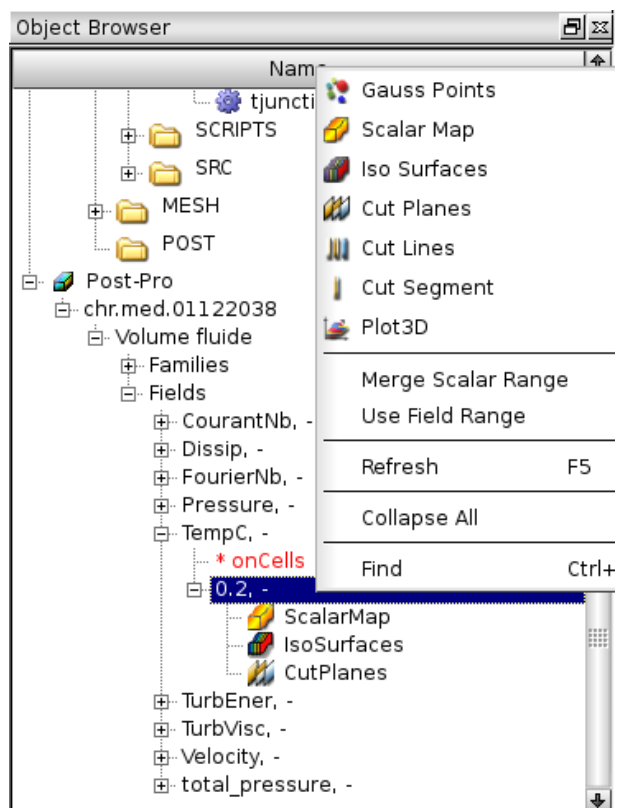


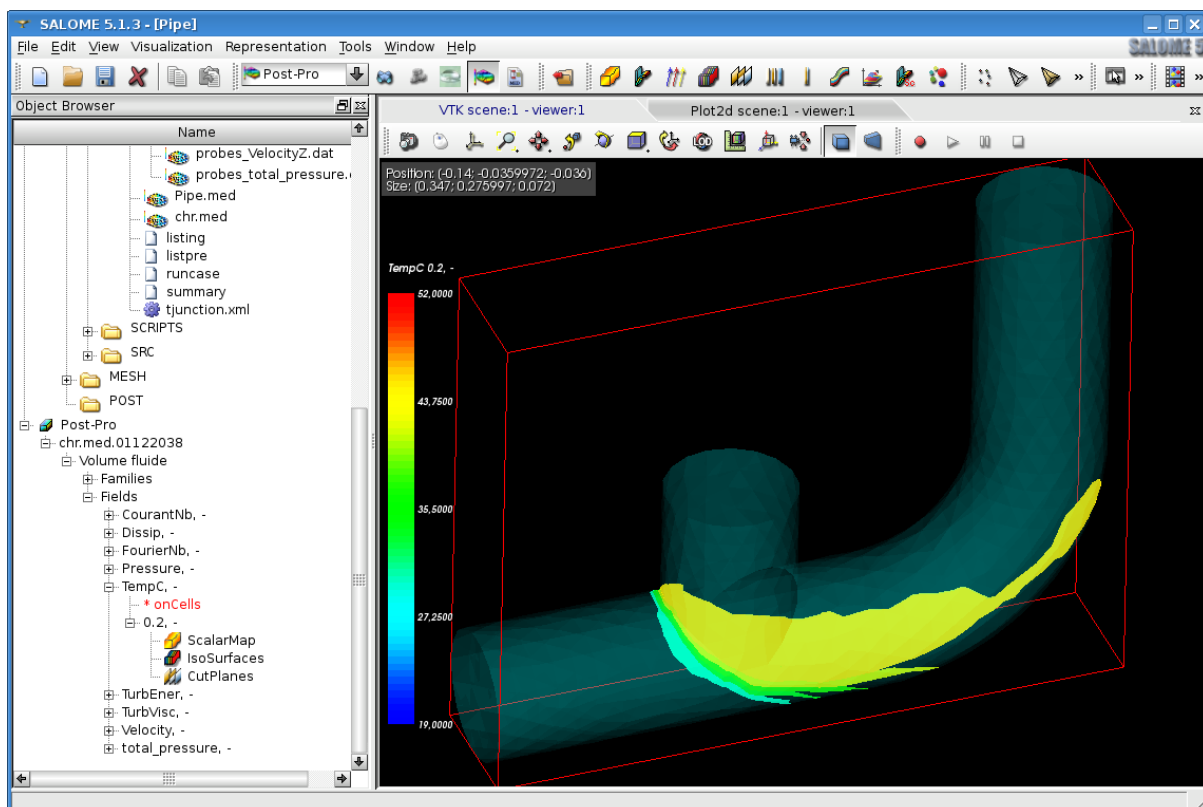
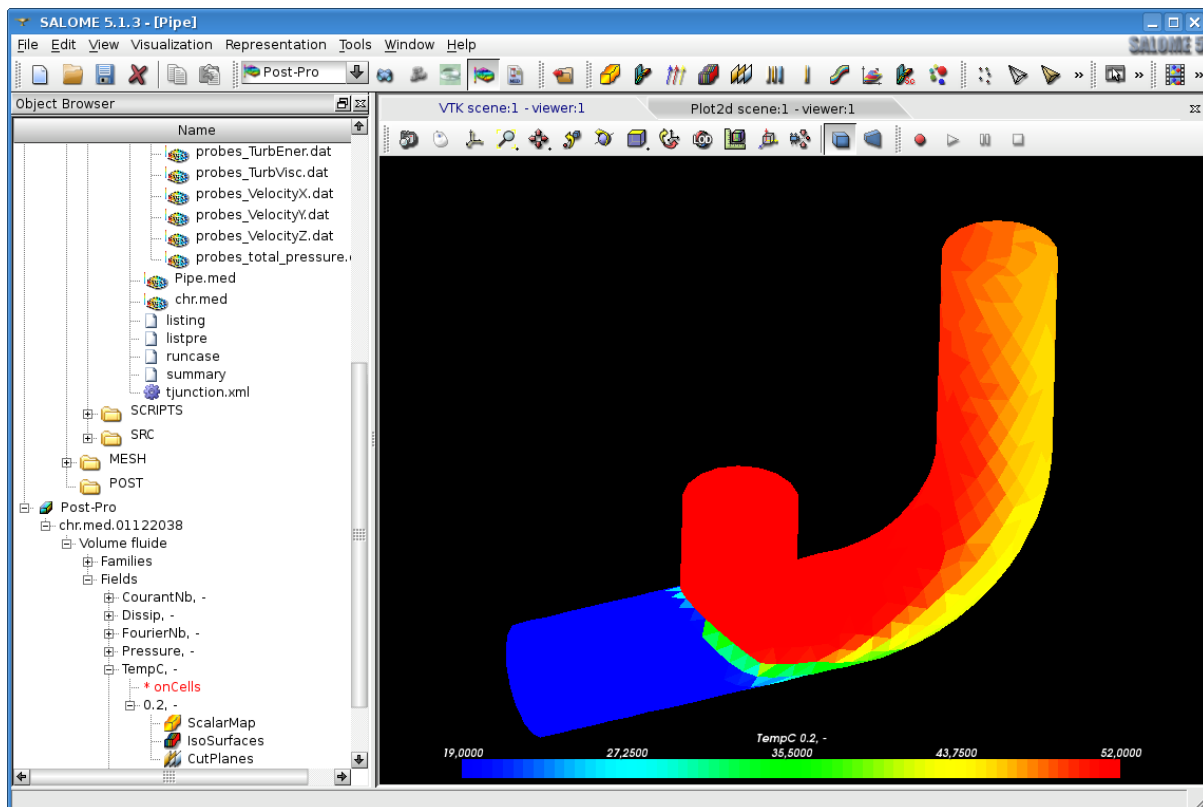
6.2 Visualisation of colored maps

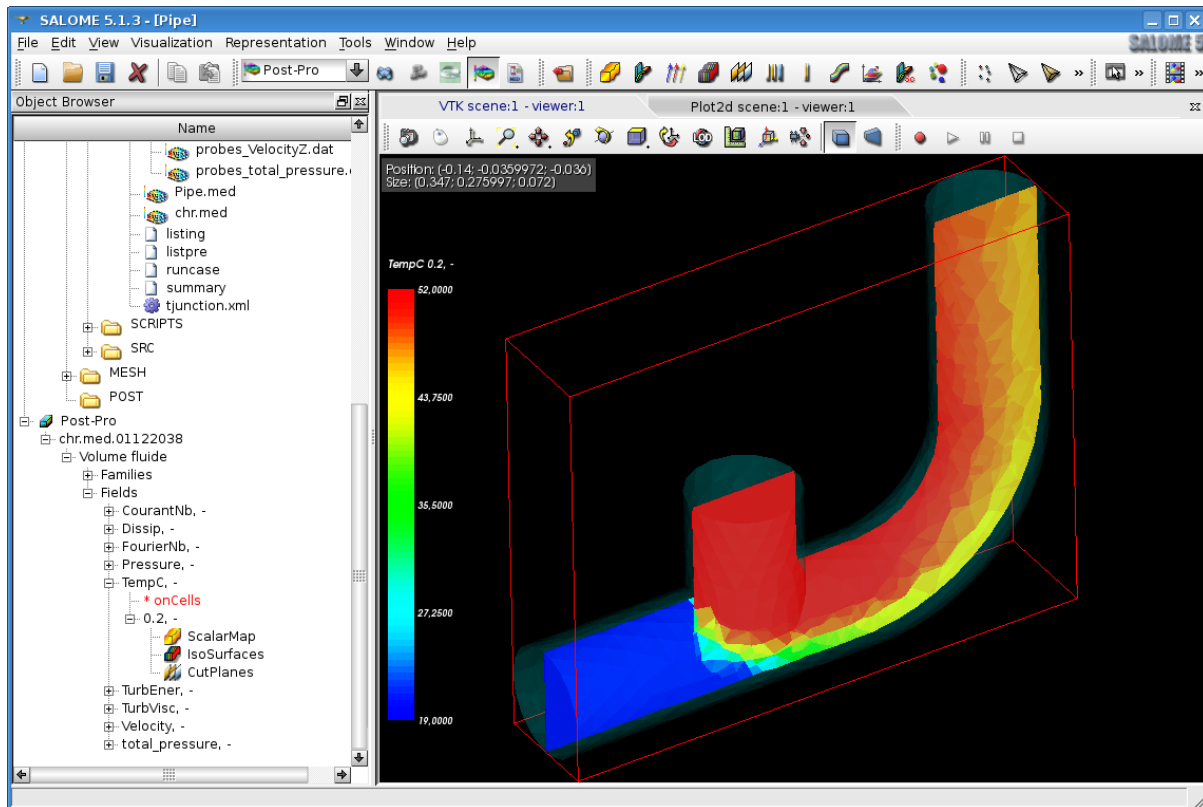
First, export in the **Post-Pro** module the results file *chr.med*.



Activate the **Post-Pro** module. Select the variable (*TempC*) and the time step (*0.2* here) to display. The select the popup menu “**ScalarMap**”, “**IsoSurfaces**” or “**CutPlanes**” (click left on *TempC*).







6.3 Velocity vector and streamlines

Select the *Velocity* and the time step (0.2 here) to display. Then select the popup menu “**Vectors**” or “**StreamLines**” (click left on *Velocity*).

