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 2013

documentation

version 3.0 tutorial - Turbulent mixing in a T-junction with CFDSTUDY in SALOME

contact: saturne-support@edf.fr

CONTENTS

CHAPTER

ONE

GENERALITIES ABOUT CFDSTUDY

1.1 Introduction

The **CFDSTUDY** is a component for the SALOME platform. The purpose of this software 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 `http://code-saturne.org`.

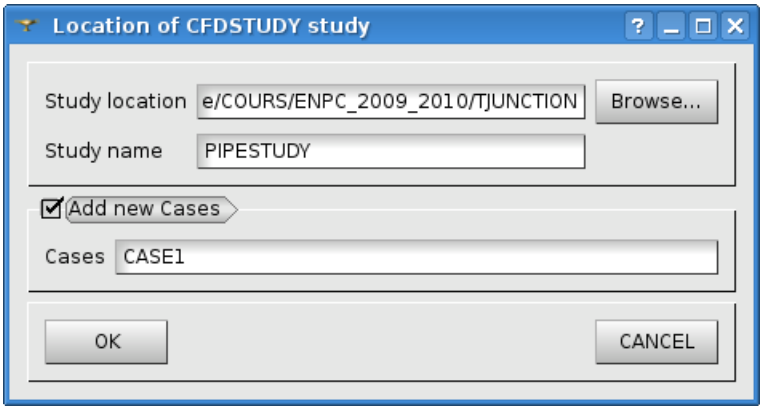
This document provides a tutorial for the use of CFDSTUDY with *Code_Saturne*. For a *Code_Saturne* tutorial itself and more information about user issues, please consult the software documentation.

Note: CFDSTUDY is a pure Python module of SALOME. Only the GUI part of the module is implemented without its engine counterpart. Therefore, the dump functionality is not available.

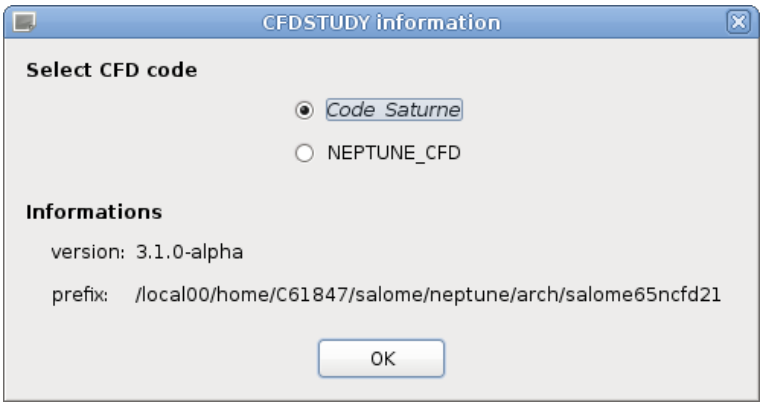
1.2 Reference functionalities

The main purpose of **CFDSTUDY** is to embed the GUI of *Code_Saturne* inside the SALOME desktop and to make easier the setup of a case. For that, when the module is loaded, several hooks are available:

- Menubar
  - **File > CFD code** menu:
    - Save CFD Data file
    - Save as CFD Data file
  - **CFDSTUDY** menu:
    - Set CFD study Location: allow to chose an existing study, or to create a new one.



* Update Object Browser: refresh the directories list in the Object Browser.
* CFDSTUDY information: display information about the code.



* Tools: display the file of parameter, open an xterm.

– **Help** menu:

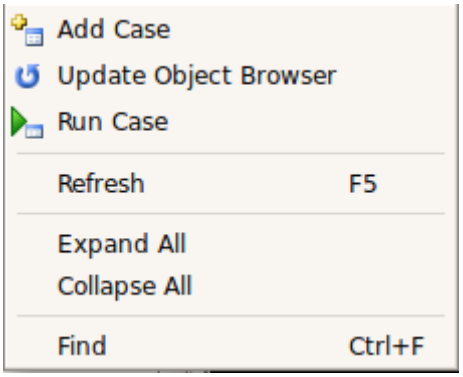
* **CFDSTUDY module User's guide** menu: display this document in html format.
* **Code_Saturne NEPTUNE_CFD** menu: several guides are available through this menu. These guides are enable only if the GUI of the code is displayed.

- Toolbar
  – From the left to the right:

* Set CFD study Location
* Add a new case in a study
* Launch the GUI
* Run a case
* Save CFD Data file
* Save as CFD Data file
* Close a GUI
* Undo

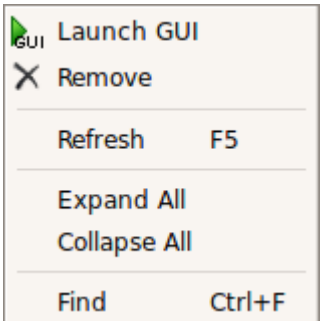
        * Redo

- GUI: additional functionalities are available:
  - Groups of boundary faces can be selected in the Object Browser or graphically,
  - Groups of cells can be selected in the Object Browser or graphically,
  - Monitoring points can be displayed in the VTK viewver.
- Object Browser: several actions are available through a specific contextual menu (open by *Right click*)
  - Study directory:



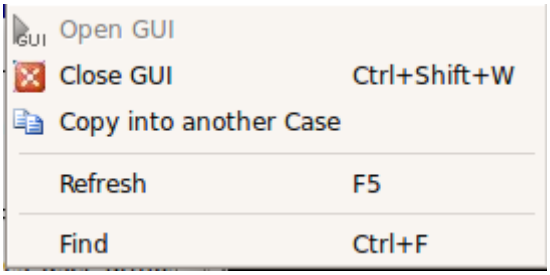
  - Mesh file:

  - Case directory:

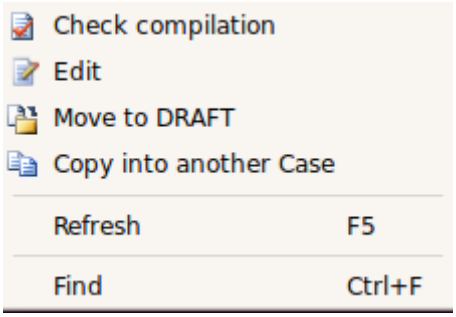


  - *SaturneGUI* file:

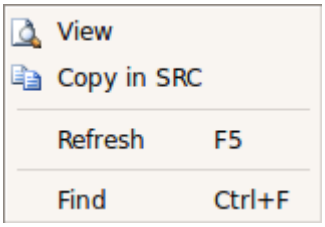
  - File of parameters:



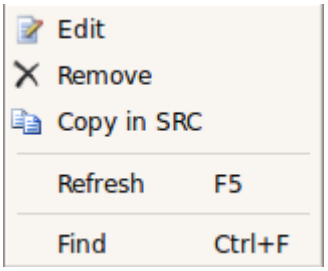
– File of functions in the *SRC* directory:



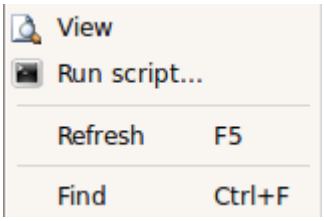
– File of functions in the *REFERENCE* directory:



– File of functions in the *DRAFT* directory:



– Script of *runcase* file:



– Results directories in the *RESU* directory:


Remove
Refresh F5
Collapse All
Find Ctrl+F

CHAPTER

TWO

TUTORIALS

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

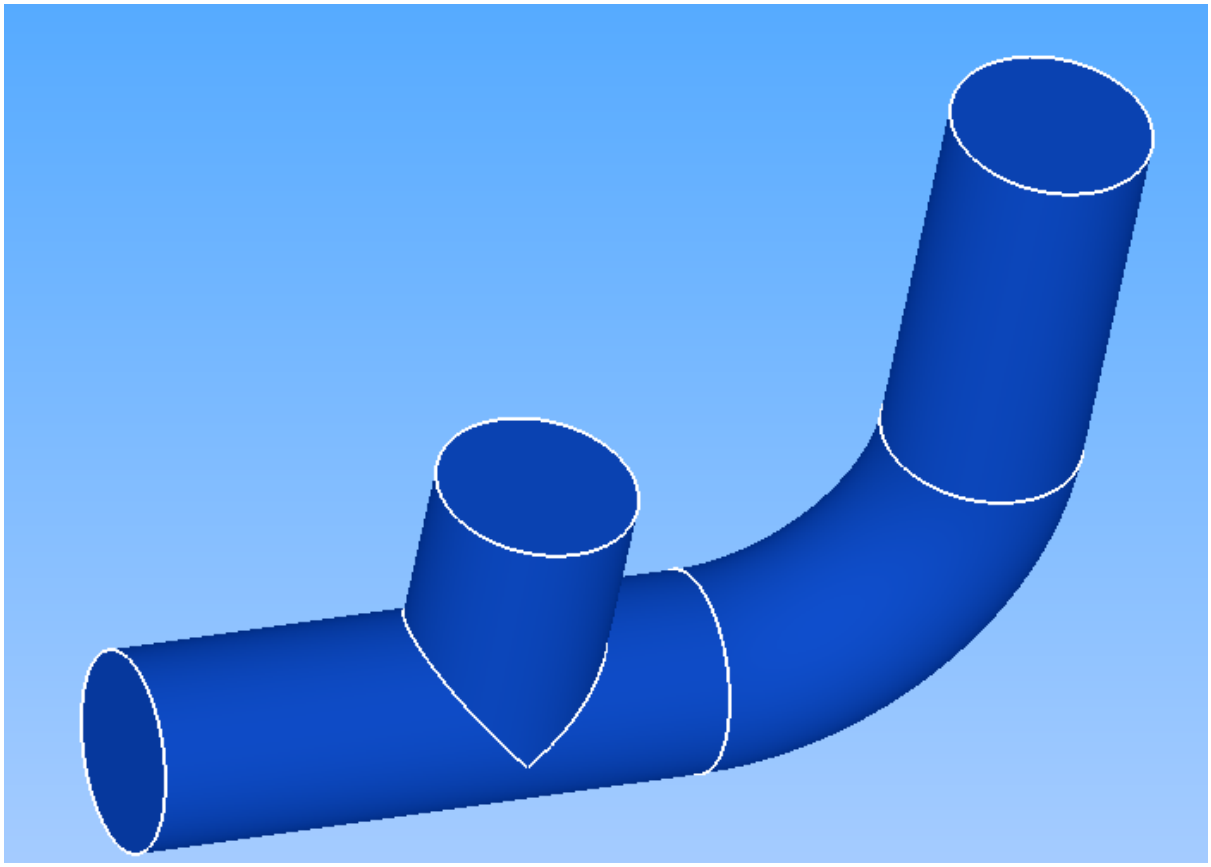
2.1.1 Introduction

This tutorial provides a 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, do a setup of *Code_Saturne* through the module **CFDSTUDY**
- at last, some elements for the post processing of the results with the module **Paravis**

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.



2.1.2 Open SALOME

The command to open salome is `./code_saturne salome`. For information about installation of *Code_Saturne* with SALOME, please consult the installation guide of *Code_Saturne*.

2.1.3 CAD design with the module Geometry

The CAD model 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.

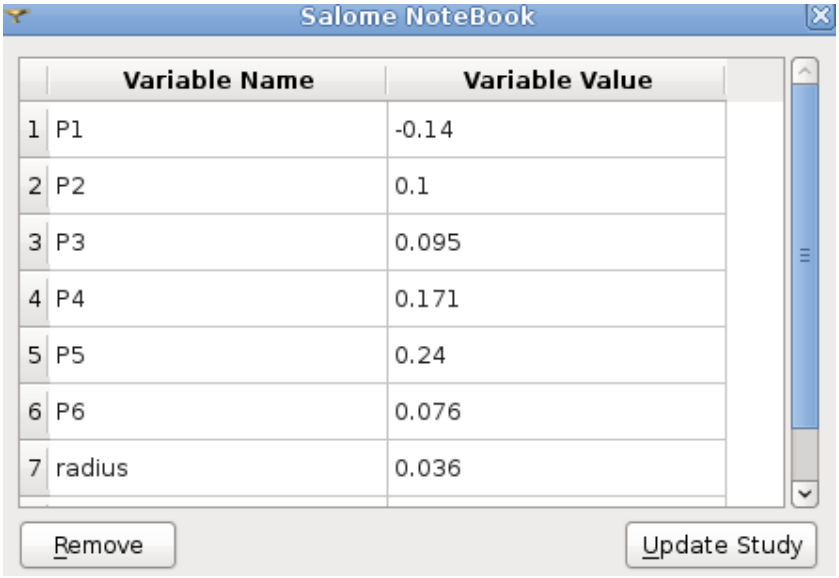
- **Note**: objects graphical manipulation in the 3D view (rotation, zoom, translation) can be done with <*Ctrl*> + *mouse buttons*.

Activate the module **Geometry**.

Points, lines and wire

- Creation of points: open the Notebook: **"File > Notebook"** and add the following variables:

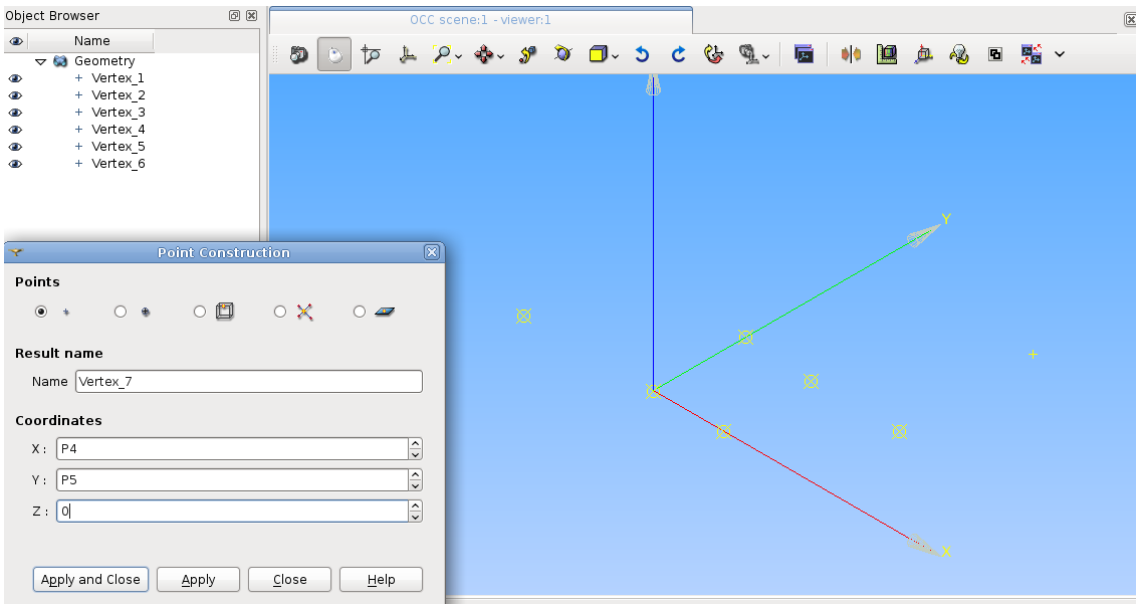
Variable_Name	Variable_Value
P1	-0.14
P2	0.1
P3	0.095
P4	0.171
P5	0.24
P6	0.076
radius	0.036



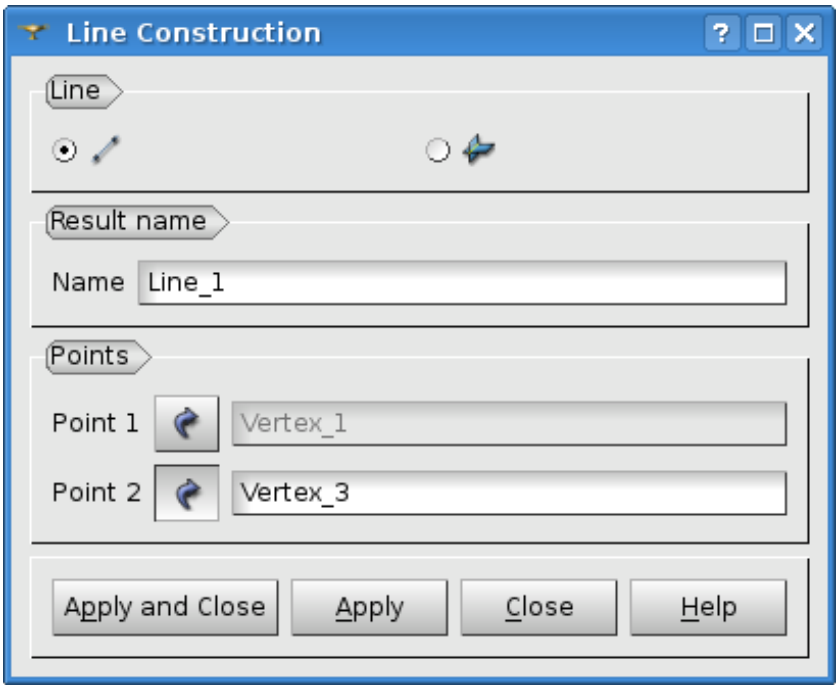
Select the menu **"New Entity > Basic > Point"** or click the toolbar button **"Create a Point"**. In the dialog window for the creation of the points create the following entities:

Name	X	Y	Z
Vertex_1	P1	0	0
Vertex_2	0	0	0
Vertex_3	P6	0	0
Vertex_4	0	P2	0
Vertex_5	P6	P3	0
Vertex_6	P4	P3	0
Vertex_7	P4	P5	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.



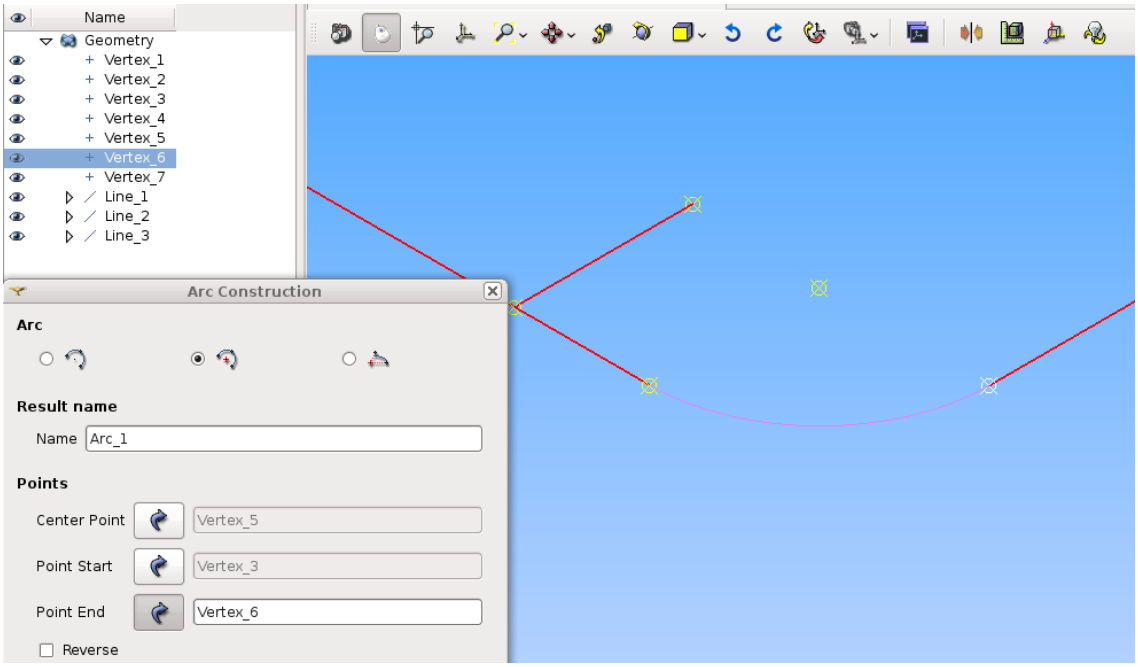
Three lines must be defined:

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

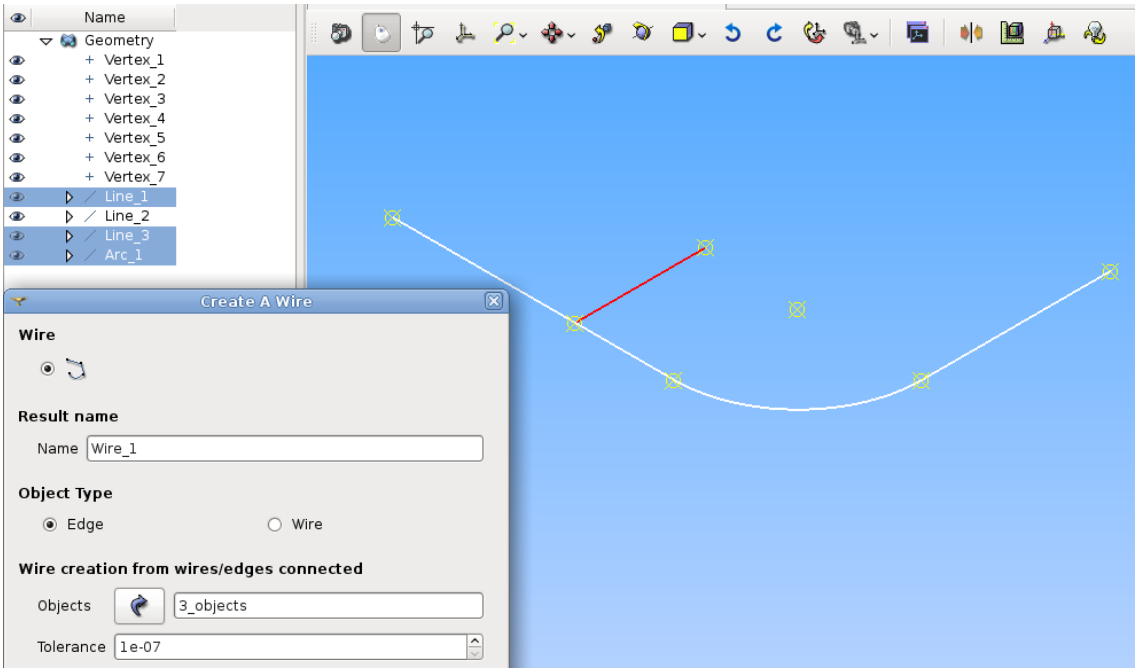
- Creation of the arc (a 1/4 of circle): select the menu **"New Entity > Basic > Arc"** (or click the equivalent toolbar button). Then, in the dialog window, select the second mode of creation (i.e. with a center point, and two points).

Then the arc must be defined:

Name	Center Point	Start Point	End Point
Arc_1	Vertex_5	Vertex_3	Vertex_6



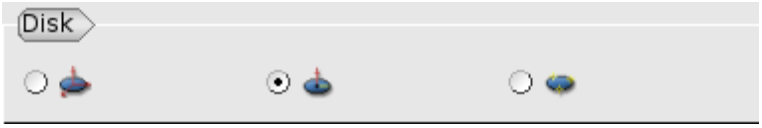
- Creation of the wire: select the menu **"New Entity > Build > Wire"**. To select together *Line_1*, *Arc_1* and *Line_3*, use *<Ctrl> + left click* in the **Object Browser**.



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

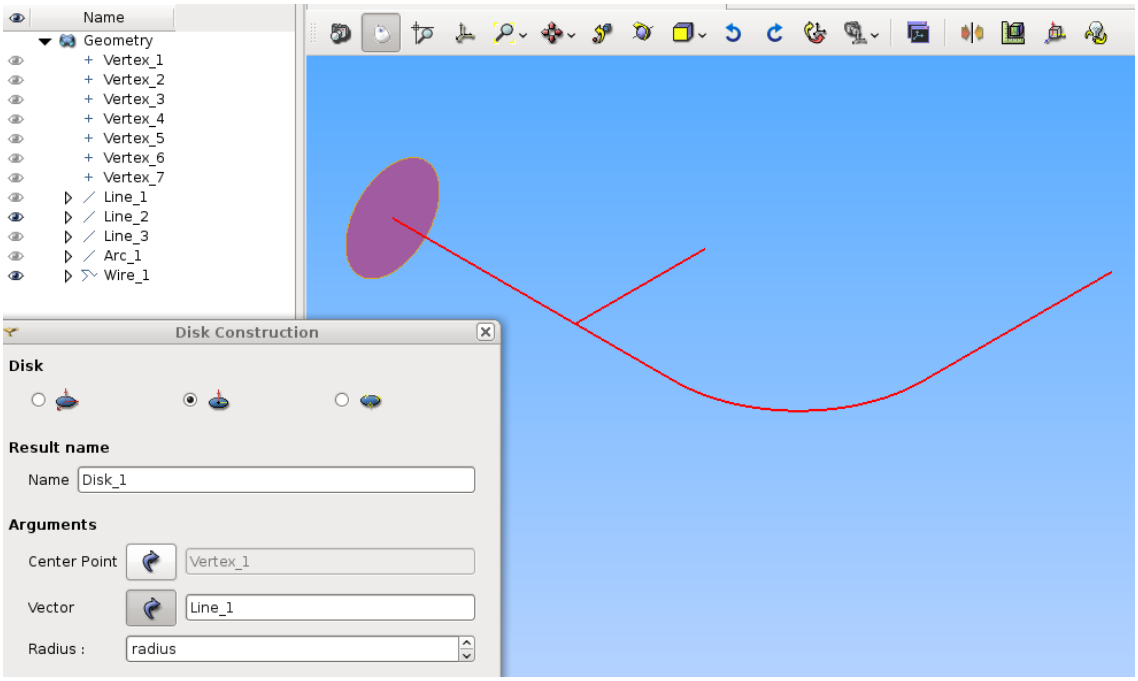
Faces and pipes

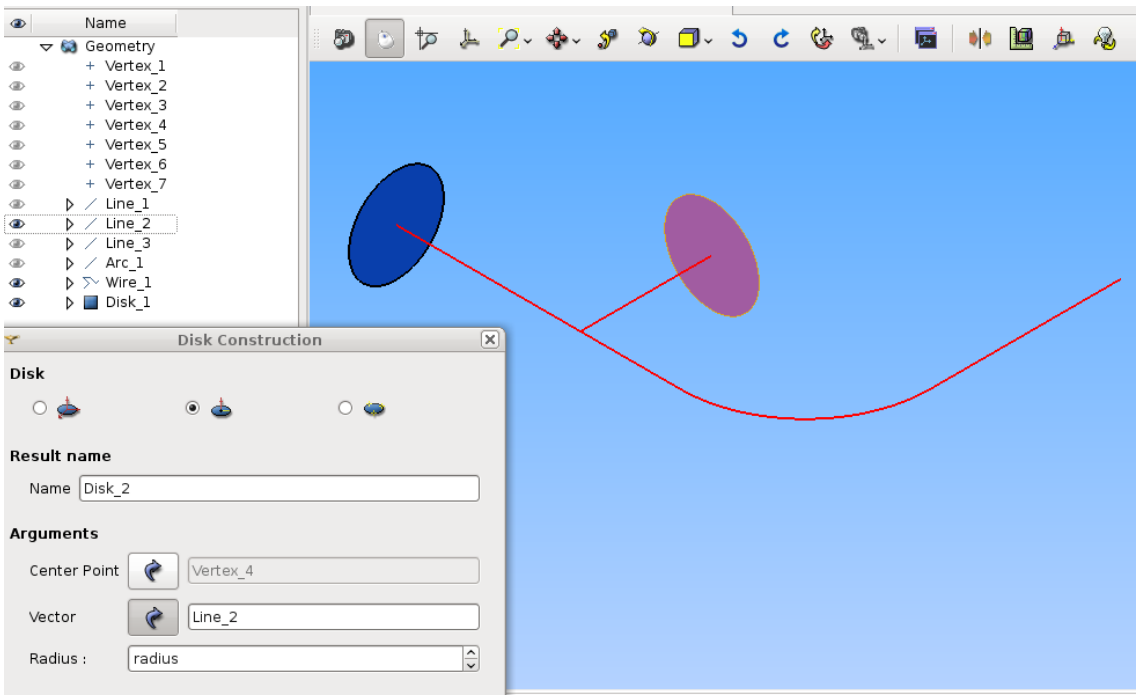
- Creation of the two disks: open the dialog window with the menu **"New Entity > Primitive > Disk"**. For each disk, in the dialog window, select the second mode of creation (i.e. with a center point, a vector and a radius).



In the hierarchical geometric entities, these disks are faces.

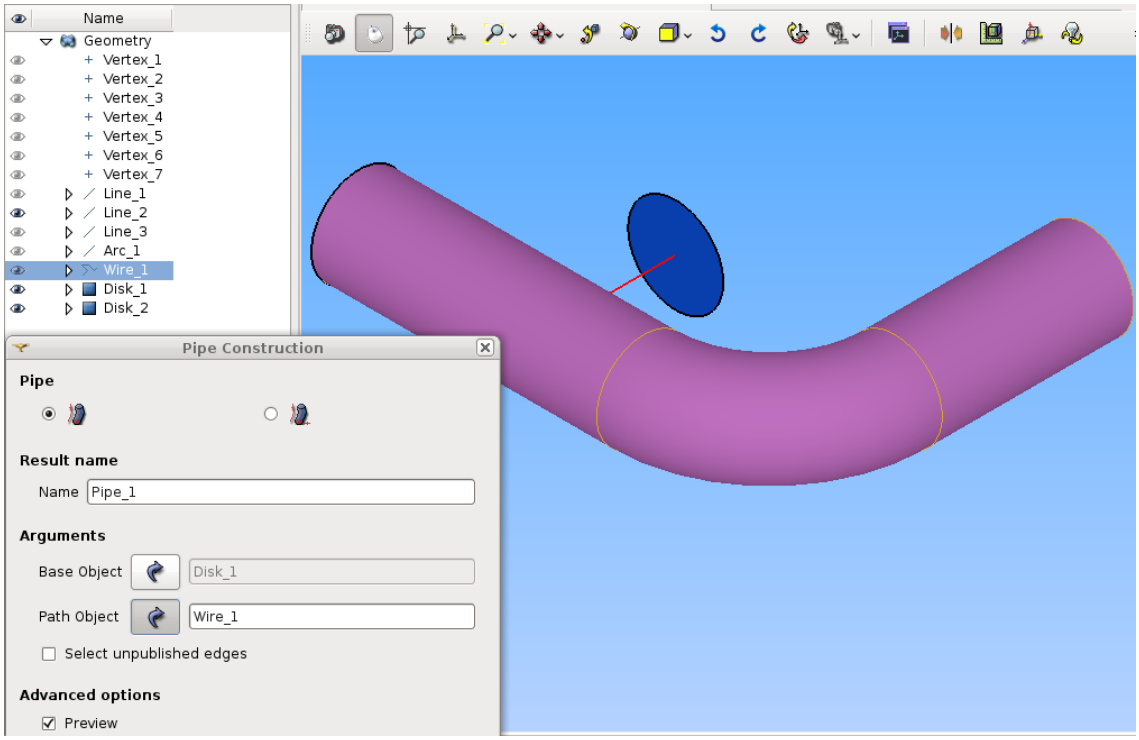
Name	Center Point	Vector	Radius
Disk_1	Vertex_1	Line_1	radius
Disk_2	Vertex_4	Line_2	radius

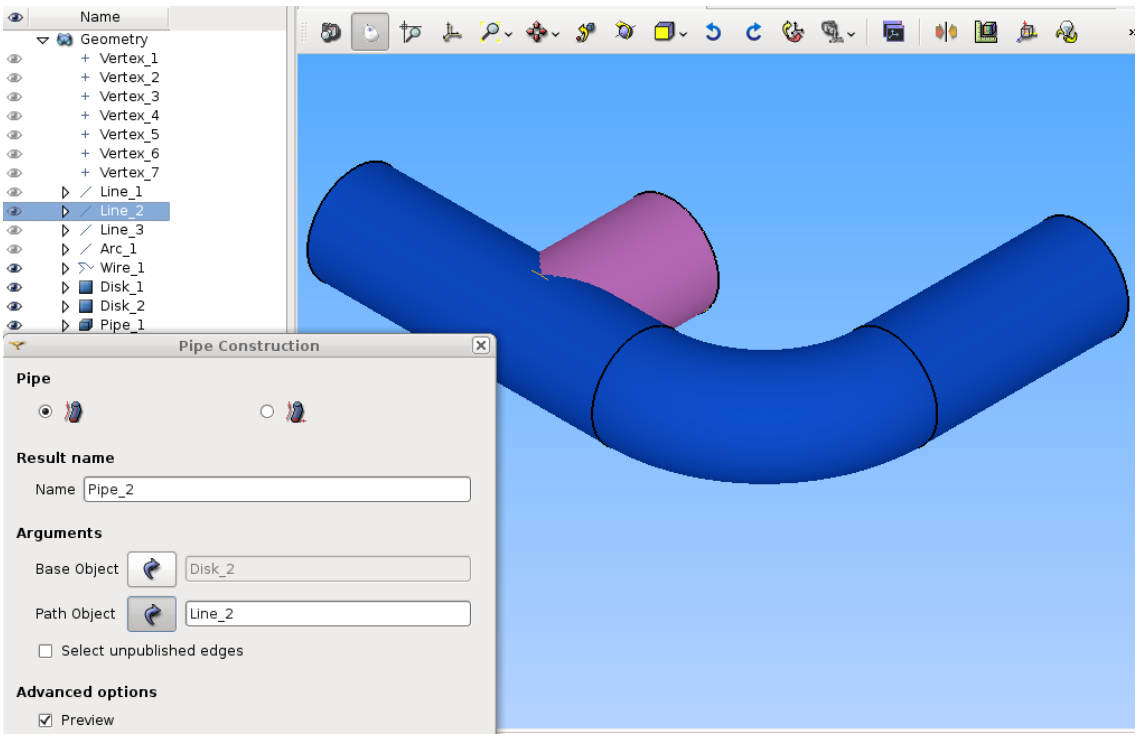




- Creation of the two pipes: select the menu **"New Entity > Generation > Extrusion Along a Path"**. In our case the two paths are respectively: *Wire_1* and *Line_2*. In the hierarchical geometric entities, these pipes are solids.

Name	Base Object	Path Object
Pipe_1	Disk_1	Wire_1
Pipe_2	Disk_2	Line_2

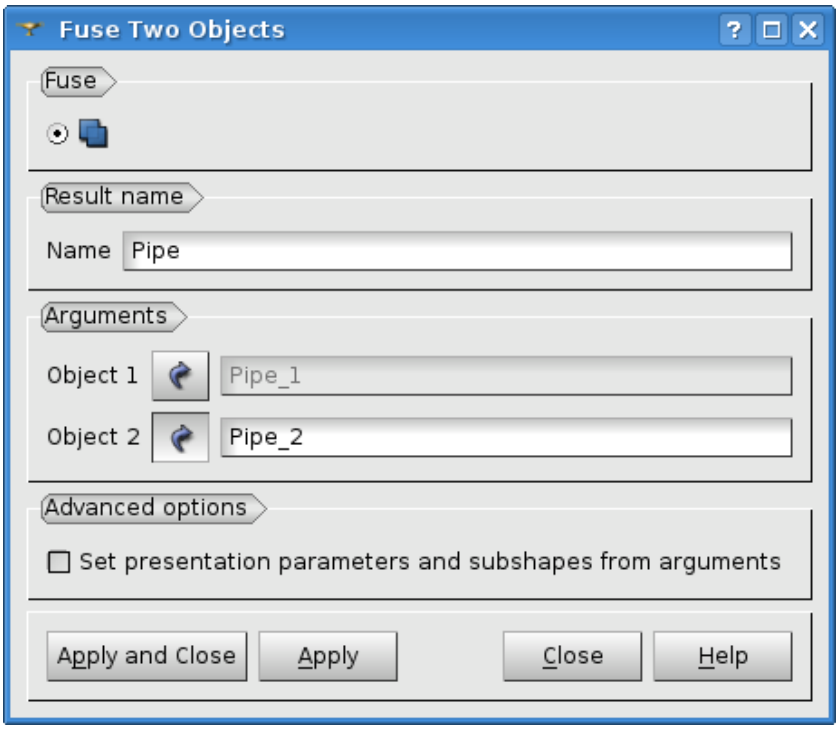




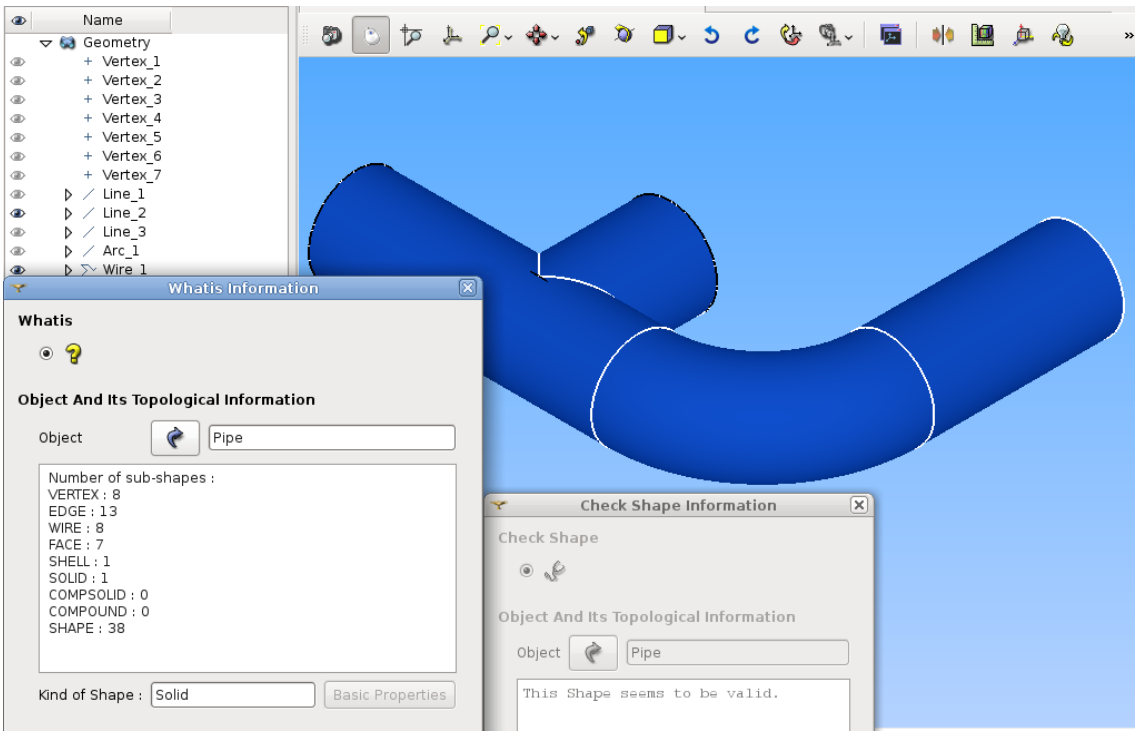
Fusion of the two pipes

- 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



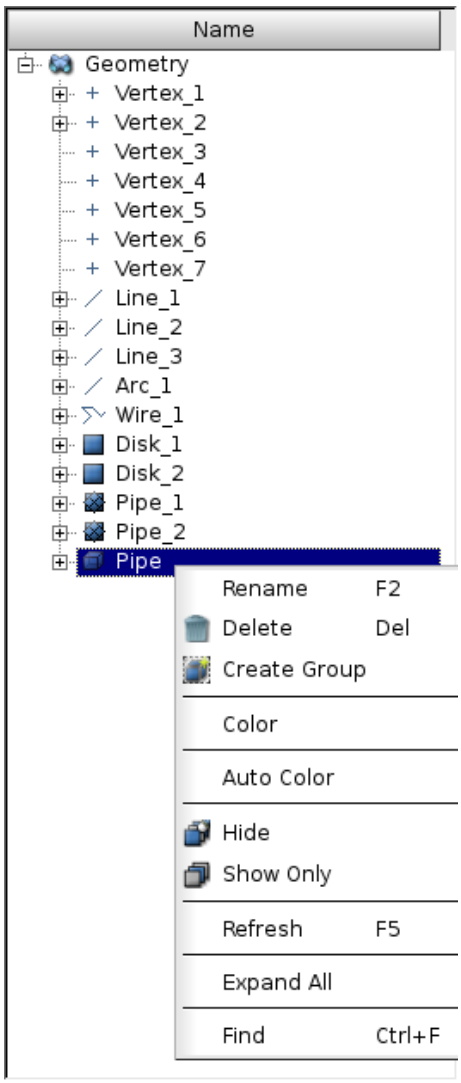
- Use the menus **"Measures > Check shape"** and **"Measures > What is"** to verify the object *Pipe*. It must be constituted of a single solid.



Groups for boundary conditions definition

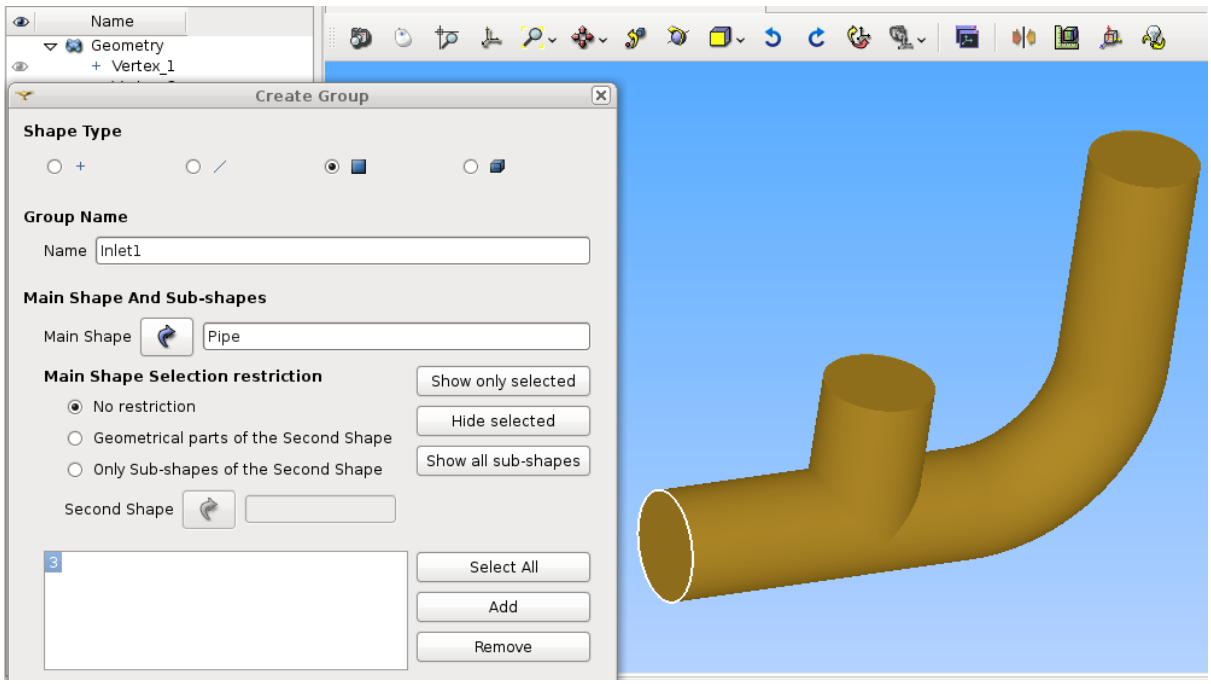
Fisrt, choose the *shading* view mode instead of the *wireframe* view mode, in order to select faces in the menu **"View > Display Mode"**.

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

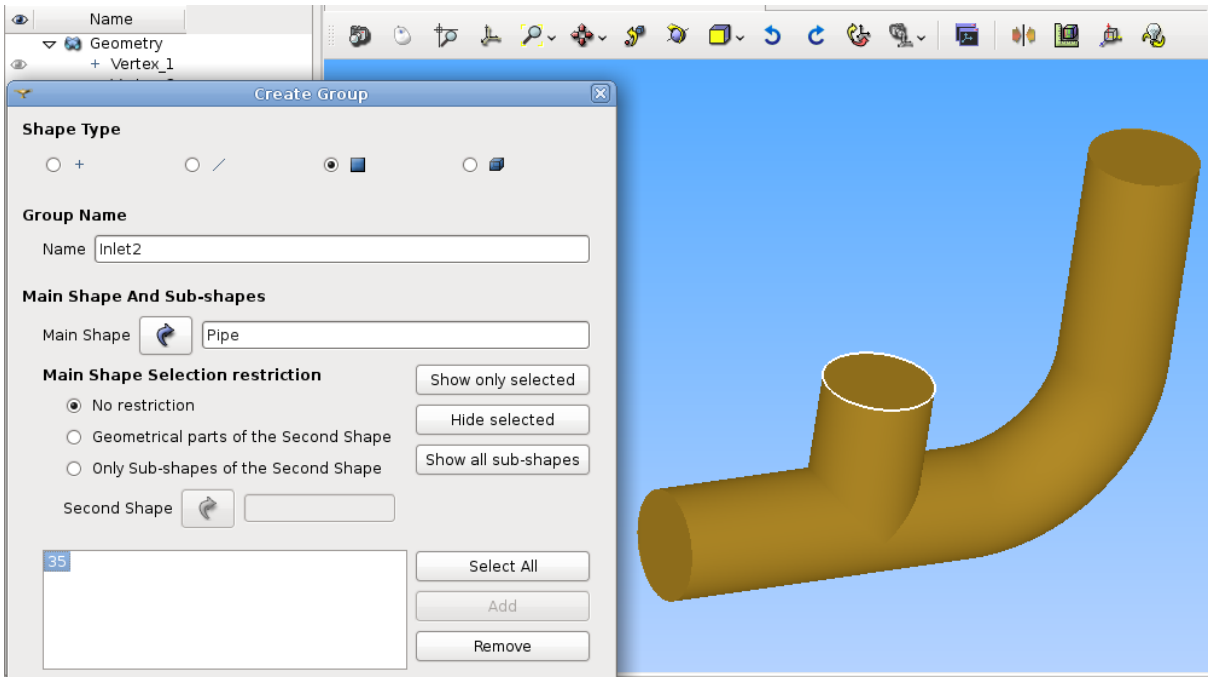


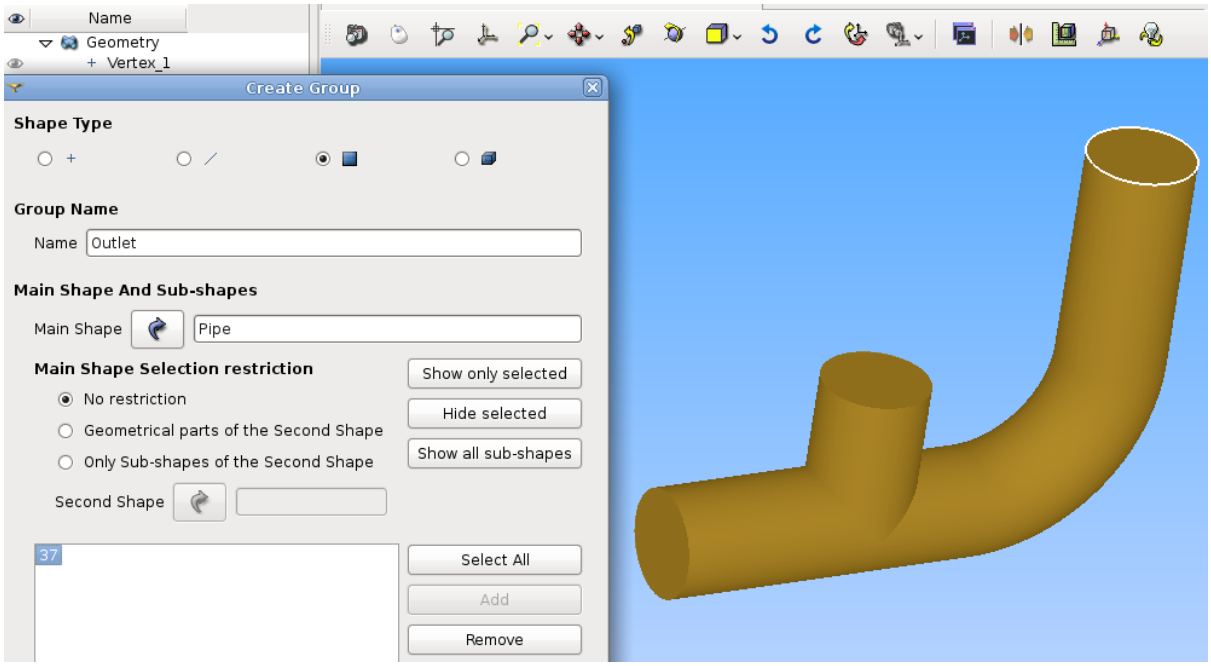
Select faces as shape type (3rd choice under **Shape Type** header: one can select Vertices, Edges, Faces or Solids on a shape):

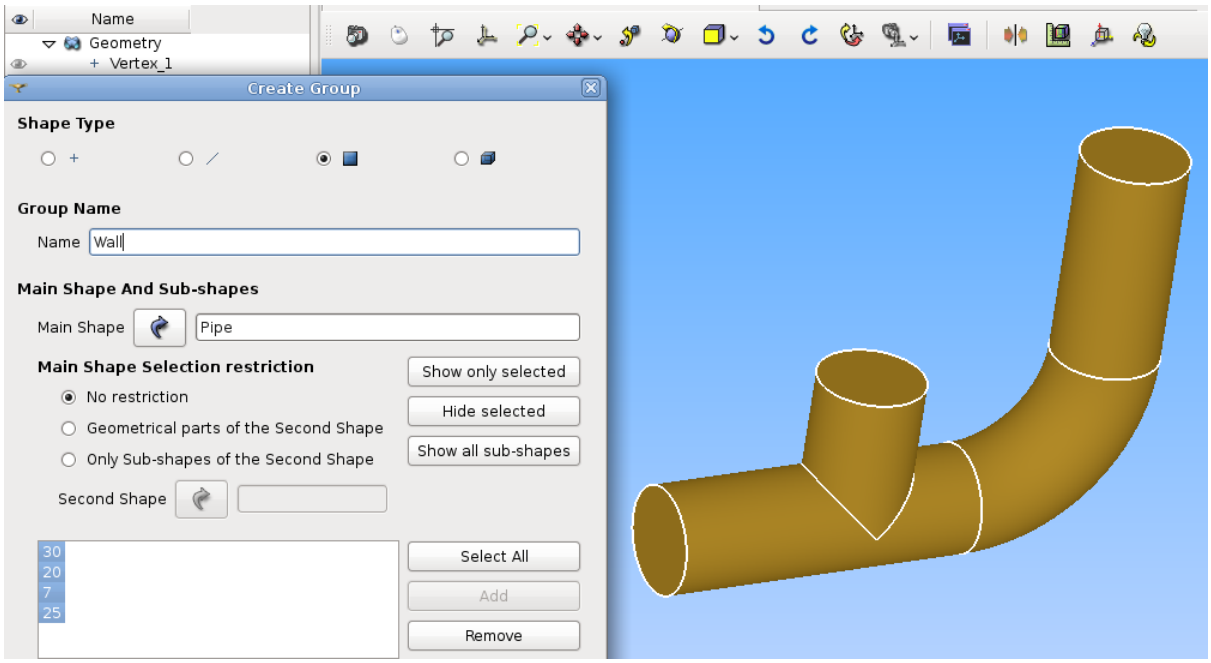
Give the name *Inlet1* to the new group and highlight (right click in the 3D view) the face corresponding to *Inlet1* on the *Pipe*. Then, push button *"Add"* (the number below identifies the face into the main shape), and apply. To be able to select a face, you may have to rotate the shape: *<Ctrl>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



Proceed as above for the 3 other groups: *Inlet2*, *Outlet* and *Wall*. For faces selection of "Wall", use the *<Shift> + left 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 Meshing

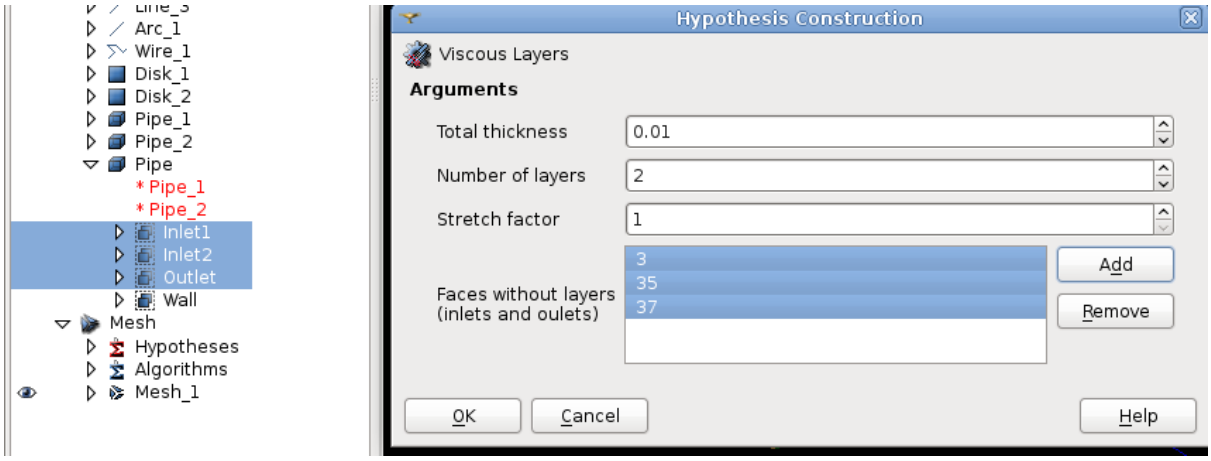
In the scope of this tutorial, only the simplest way to mesh a CAD model is shown.

Activate the module **Mesh**.

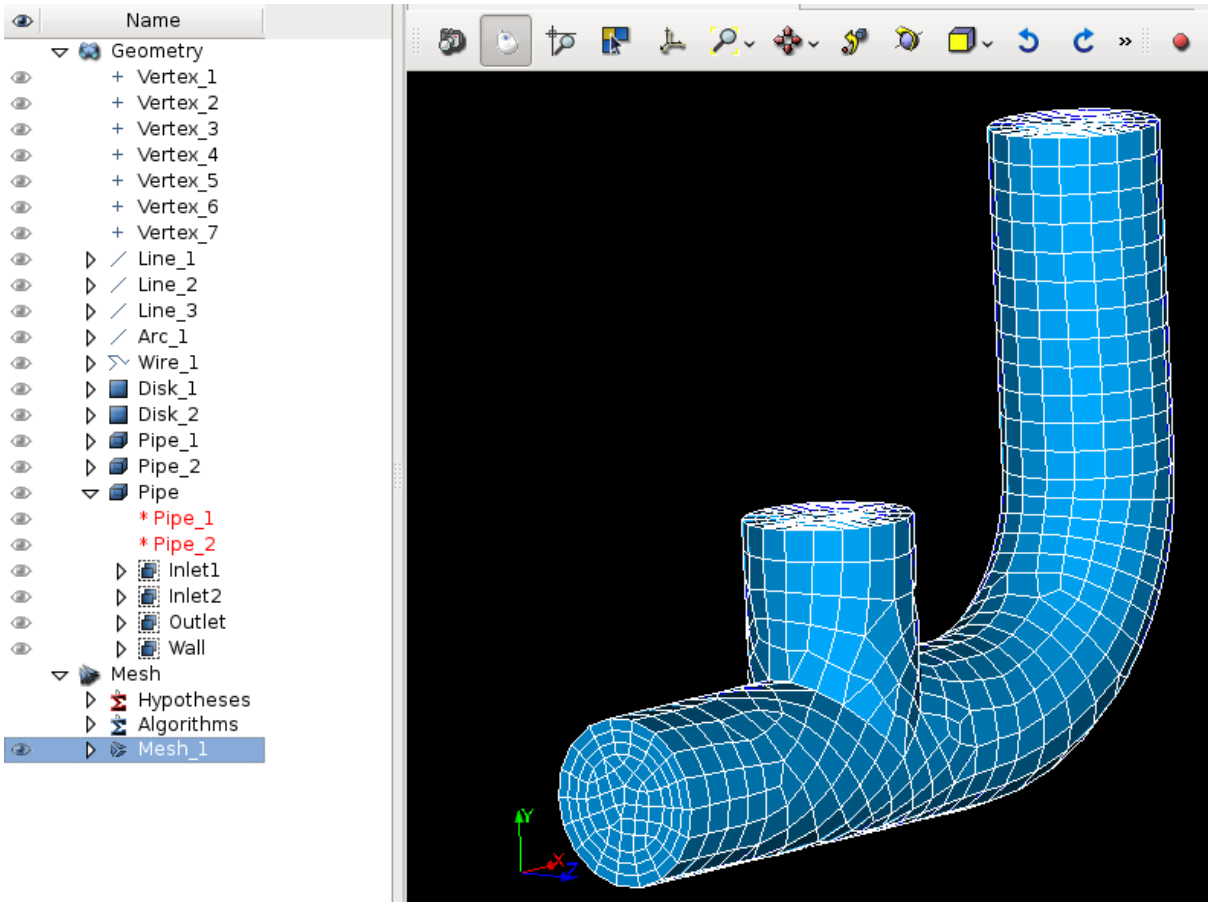
Mesh with a layer of prisms on *Wall*

- Select the *Pipe* object in **Object Browser**, then select menu **"Mesh > Create Mesh"**.
- In **"3D"** tab, select option **"(Tetrahedron) Netgen"**.

- Click on the only active button on **"Add Hypothesis"** line, and select **"Viscous Layer"**.
- Click on the only active button on **"Add Hypothesis"** line, and select **"Viscous Layer"**. Set the **"Total thickness"** to 0.015 and the **"Number of layers"** to 3. Then add the Faces without layers: select in the Object Browser the groups *Inlet1*, *Inlet2* and *Outlet* in *Geometry* and click on the **Add** button.



- In **"2D"** tab, select option **"Netgen 1D-2D"** (nothing to do in the other tabs 1D and 0D).
- Click on the only active button on **"Hypothesis"** line, and select **"NETGEN 2D Parameters"**.
- The **"Max. size"** and the **"Min. size"** correspond to the maximal and minimal edge length of the tetrahedrons. Set the sizes to 0.025 and 0.012. The **"Fineness"** governs the curves meshing: set fineness equal to **"Very fine"**, and finally select **"Allow Quadrangles"**.
- 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.

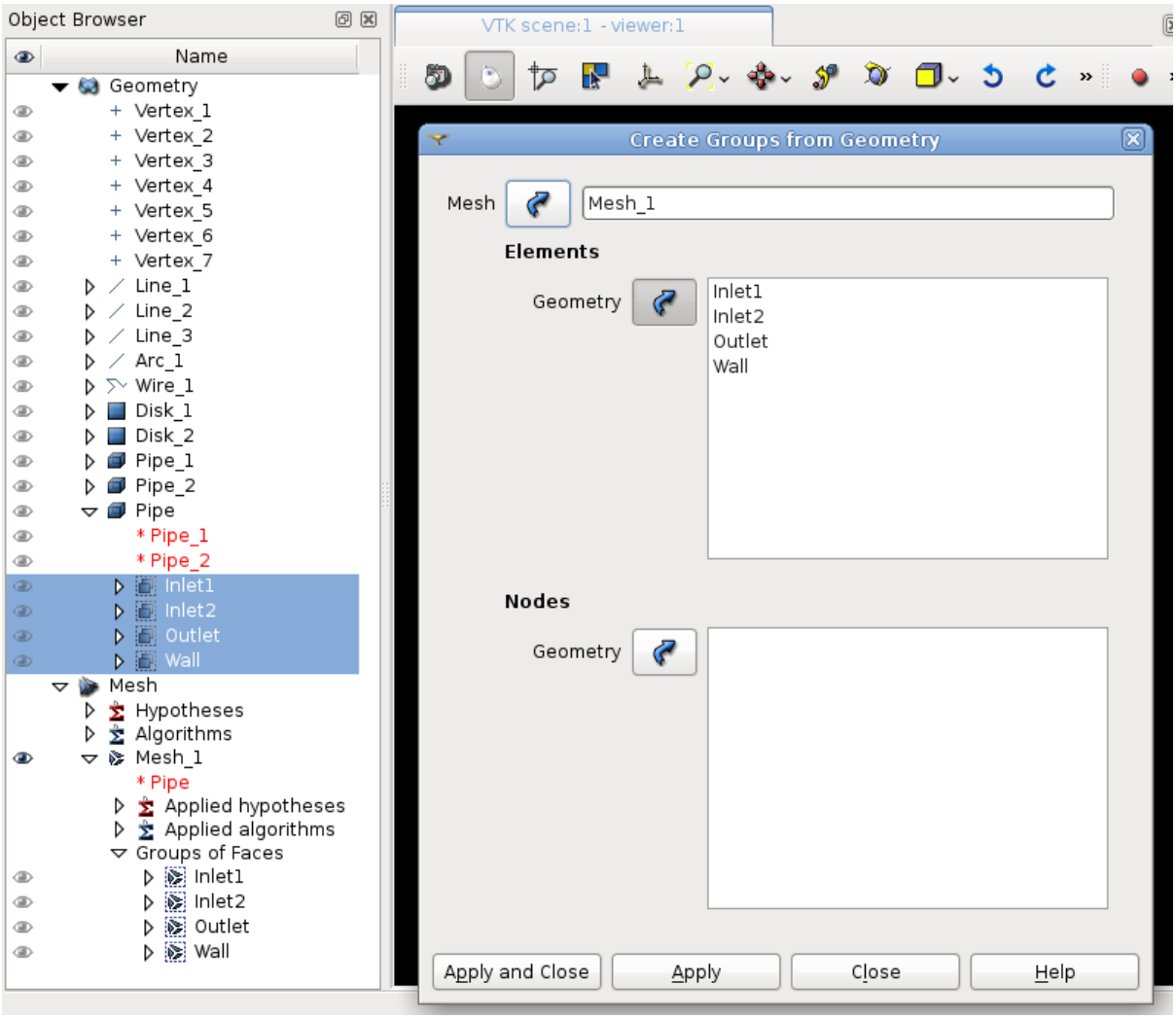


- **Note**: for a full tetrahedrons mesh, in **"3D"** tab just select option **"Netgen 1D-2D-3D"** (nothing to do in the other tabs), and fit hypothesis by clicking on the only active button on **"Hypothesis"** line, and select **"NETGEN 3D Parameters"**.

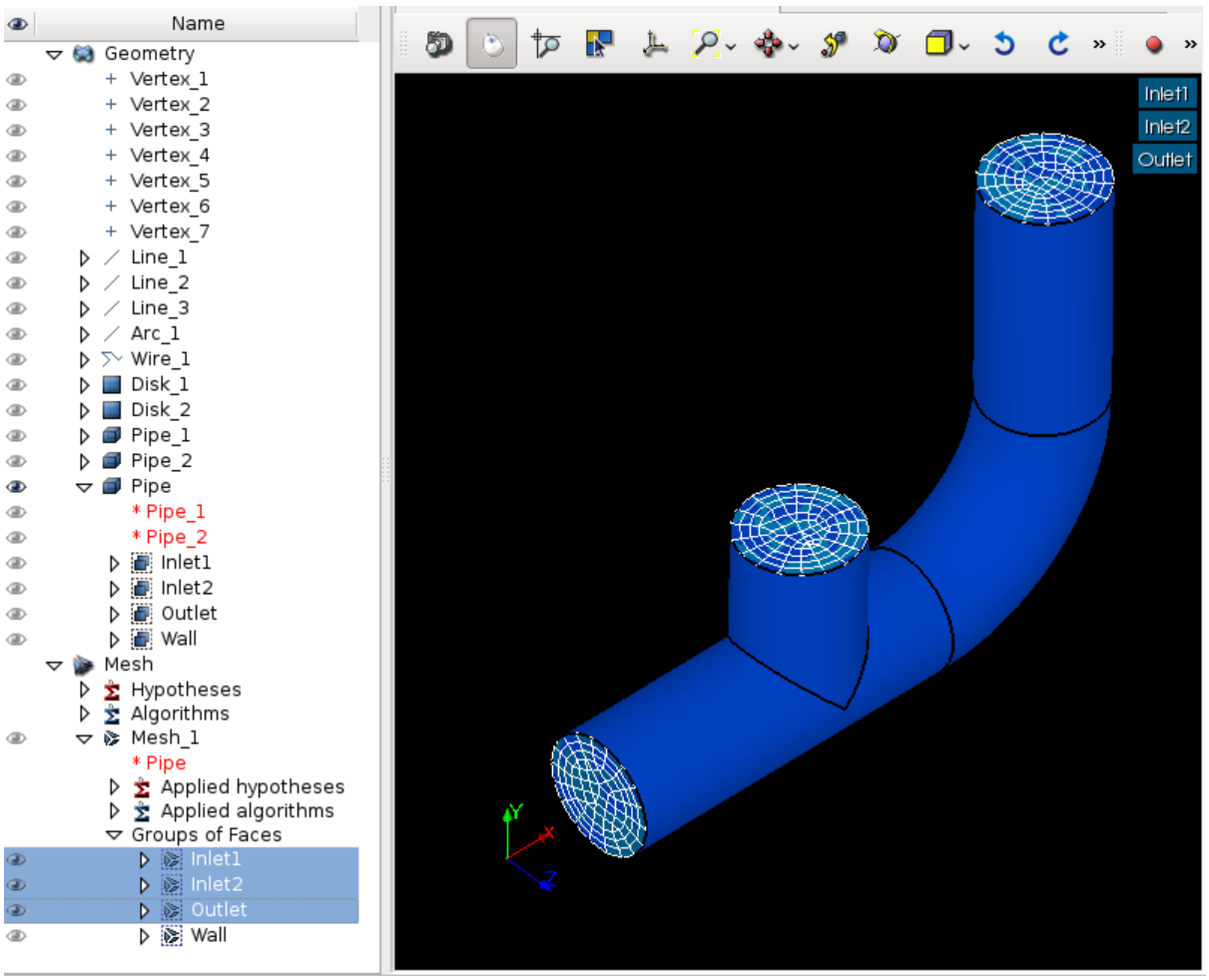
Groups on the mesh for boundary conditions definition

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"**.
- With the mesh still 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:



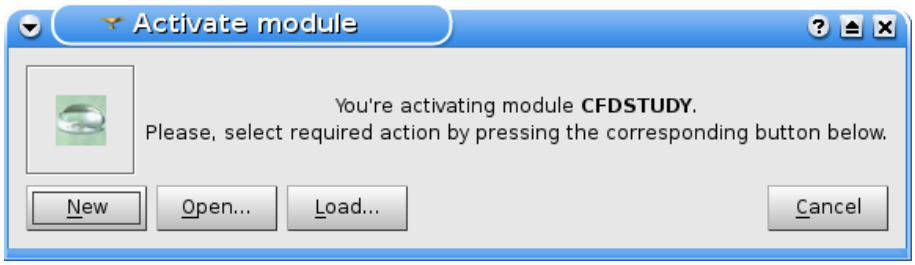
- Save the mesh in a MED file. Click left on mesh *Pipe* in **Object Browser** and select **"Export to MED File"**, and use the name *Pipe.med*.

Warning: verify that all faces belong to a single group.

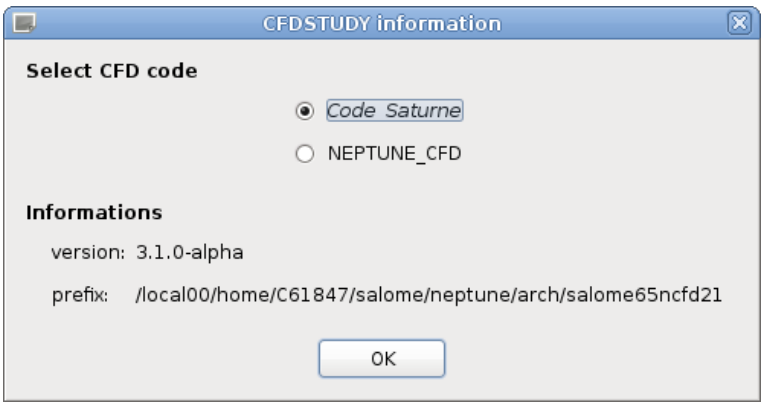
The mesh *Pipe* is ready for a CFD calculation. Save your study (**"File > Save"** or <*Ctrl*> + *S*).

2.1.5 CFD calculation with *Code_Saturne*

Activate the module **CFDSTUDY**.



Click on **"New"**. A dialog window displays information about *Code_Saturne* installation.

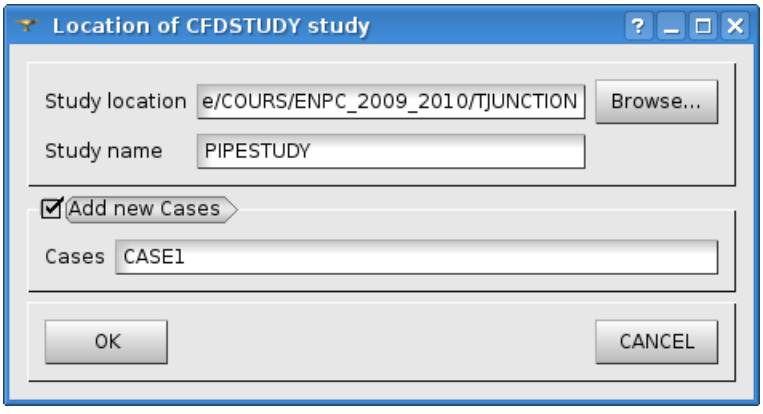


If the installation of *Code_Saturne* is not found the following error message is displayed:

CFD study and case creation

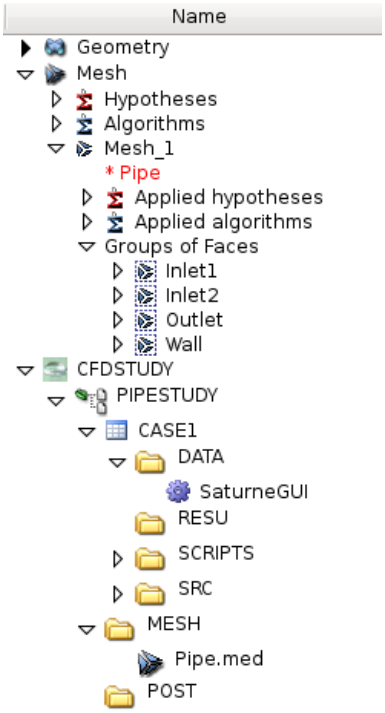
By convention, CFD calculations with *Code_Saturne* are organized in studies and cases. Several calculations that share the same meshes and data sets, define a study for *Code_Saturne*. Each data set defined in a case.

- 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*.



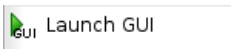
  The new study directory with the new case is created with its sub directories and files. Move the mesh file *Pipe.med* in the directoty MESH of the sudy.

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

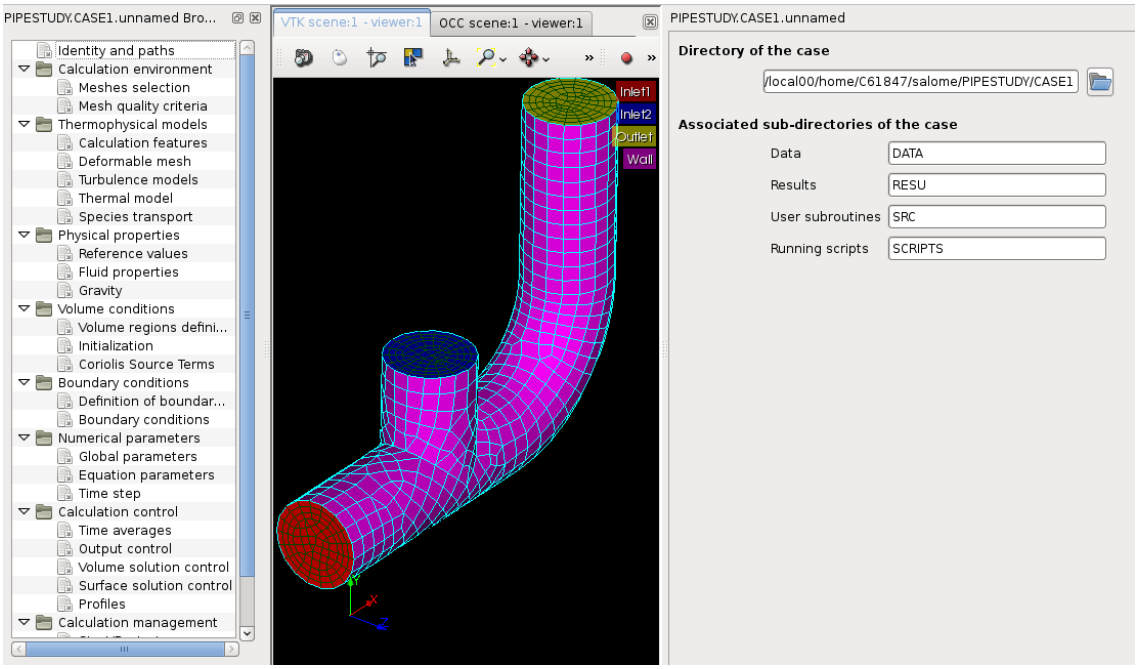


Open the *Code_Saturne* GUI

- 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 dekstop.



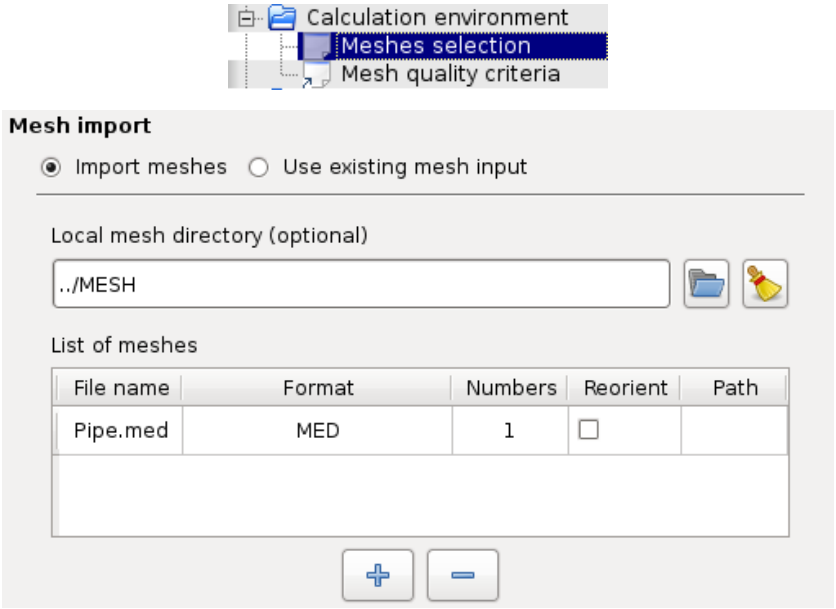
On the left dockWidget, the salome **Object Browser** and the navigation tree of the GUI are grouped on tabs. When an item of the tree is selected, the corresponding panel raises in the GUI.

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 adapted to this case, so we just have to fill a few panels.

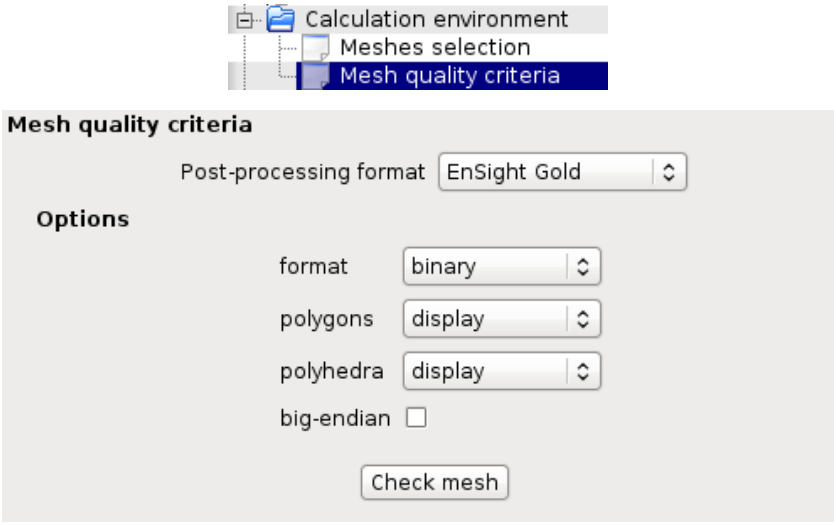
Location of the mesh file

Open **"Meshes selection"**. Use **"Add"** button to open a file dialog, and select the MED file previously saved.



Mesh quality criteria

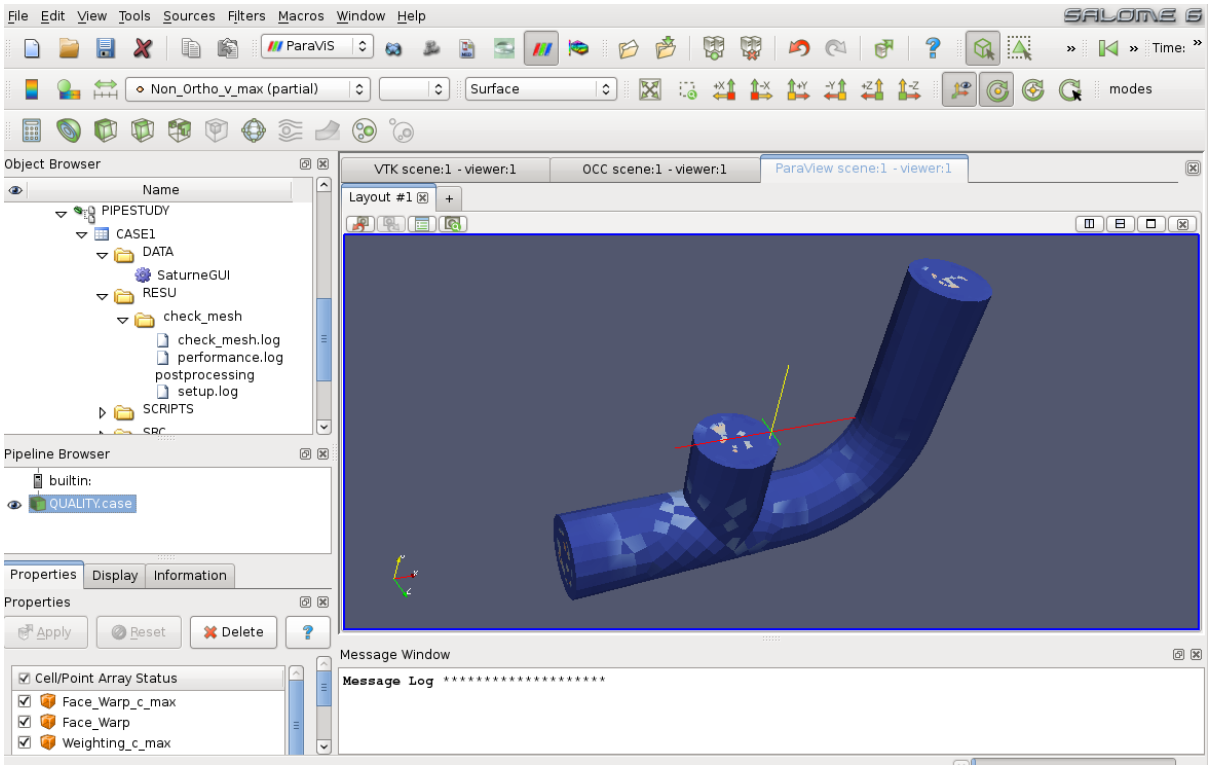
Open **"Mesh quality criteria"**. Verify that the **"Post-processing format"** is choosen to Ensight Gold. Click on **"Check mesh"** button.



The GUI displays a listing with information about quality. Then, refresh the **Object Browser** with the toolbar button **"Updating Object browser"**. There are new directories *check_mesh/postprocessing* in

the *RESU* directory.

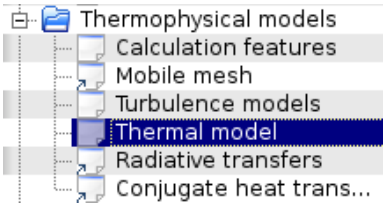
The file *BOUNDARY_GROUPS.case* and *MESH_GROUPS.case* contain information on groups location. The file *QUALITY.case* contains quality criteria as fields. In order to visualize these quality criteria, we have to open the **Paravis** module and open the *QUALITY.case* file from the *postprocessing* directory.

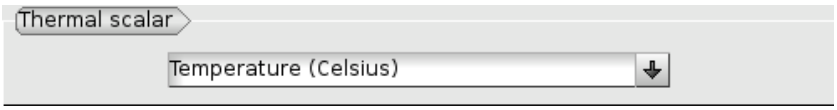


After exploring mesh quality criteria, re-activate the module **CFDSTUDY** in order to continue the data input.

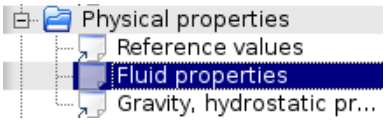
Thermophysical models

Open **"Thermal model"** and choose *Tempreature (Celsius)*.



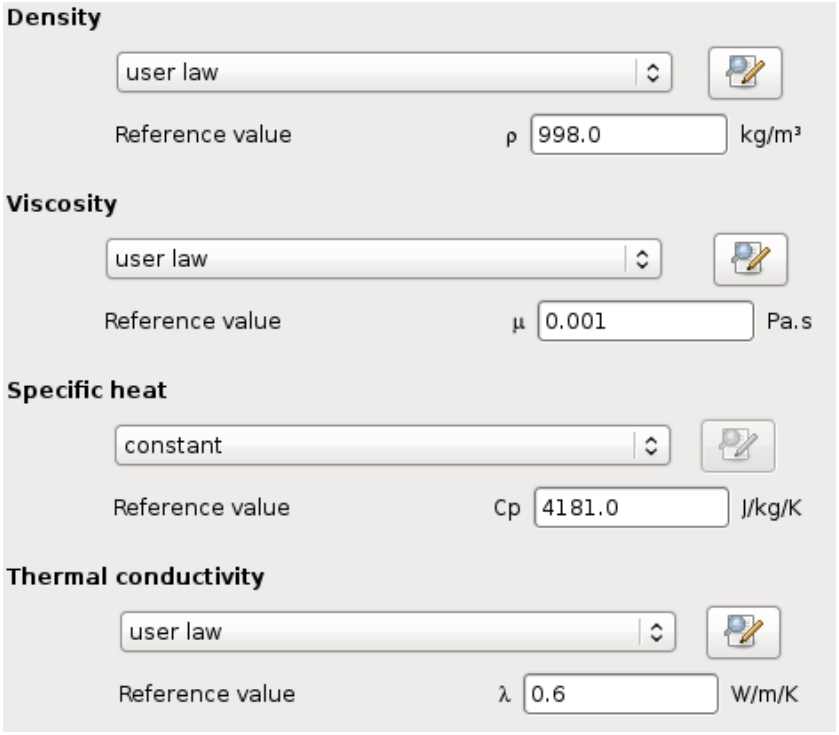


Fluid properties

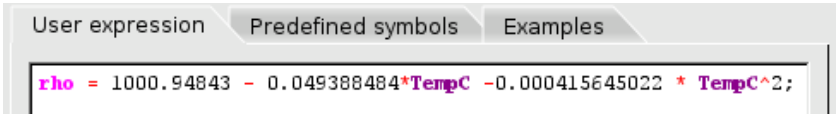


Open **"Fluid properties"**.

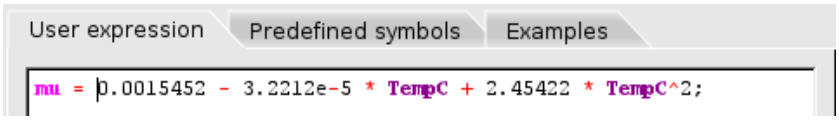
- Set reference values for water at 19 degrees Celsius:
  - density: 998 kg/m3
  - viscosity: 0.001 Pa.s
  - Specific heat: 4181 J/kg/K
  - thermal conductivity: 0.6 W/m/K



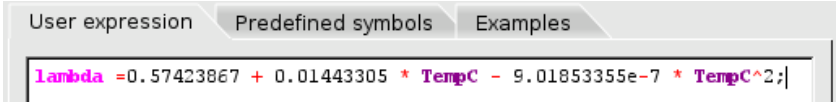
- User laws are imposed for density, viscosity and thermal conductivity. For density, viscosity and thermal conductivity, select **"user law"**, and open the window dialog in order to give the associated formula:
  - density: `rho = 1000.94843 - 0.049388484 * TempC -0.000415645022 * TempC^2;`



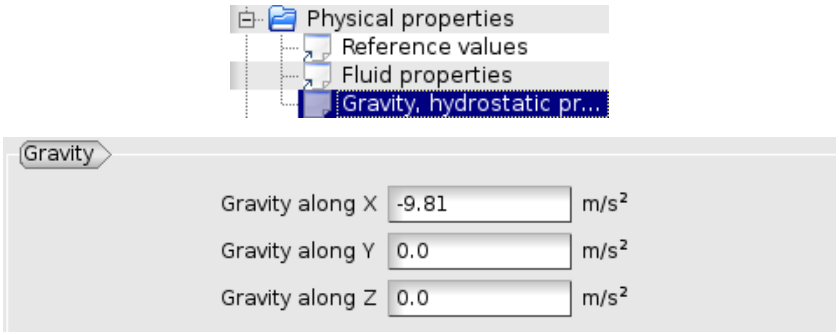
  - viscosity: `mu = 0.0015452 - 3.2212e-5 * TempC + 2.45422e-7 * TempC^2;`



  - thermal conductivity: `lambda = 0.57423867 + 0.01443305 * TempC - 9.01853355e-7 * TempC^2;`

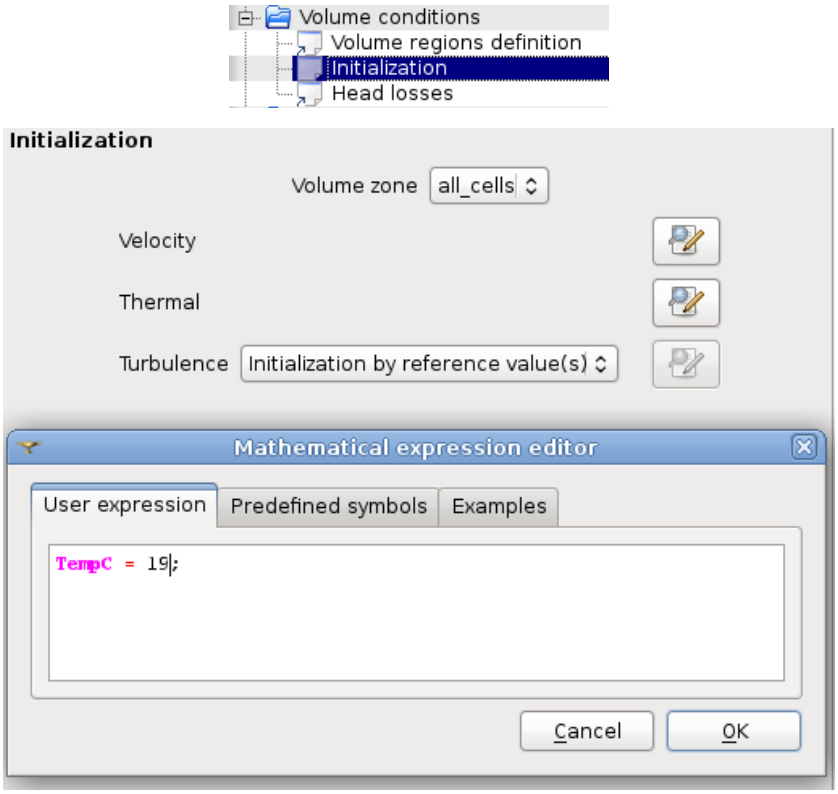


To take into account the effects of buoyancy, we have to impose a non-zero gravity.



Initialization

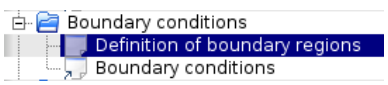
The initial temperature of the water in the pipe is set to 19 degrees.



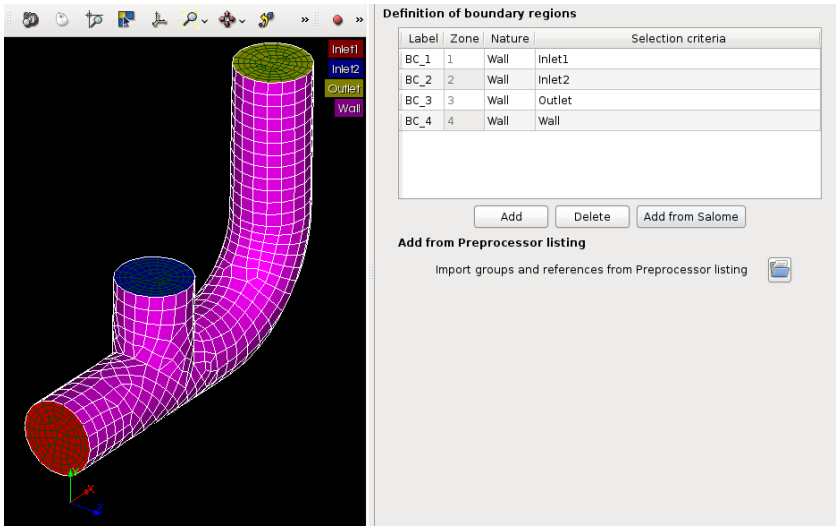
Boundary conditions

Define locations graphically

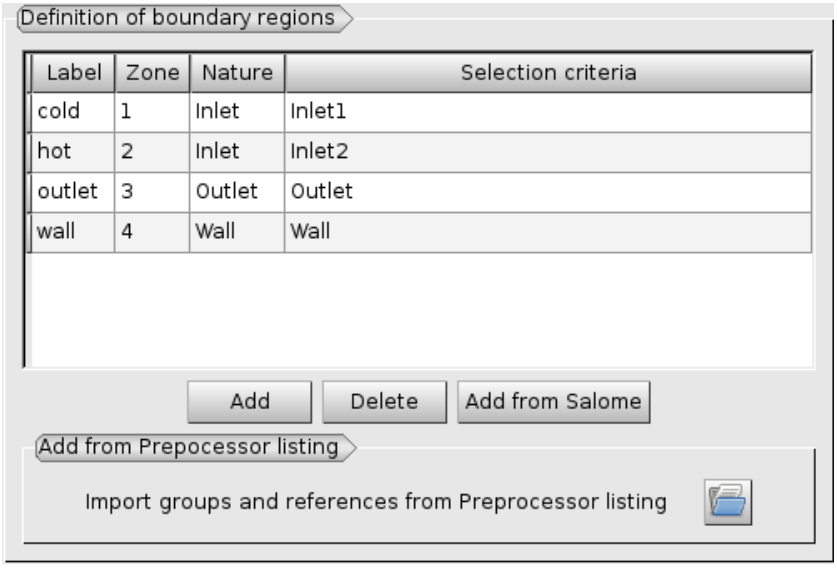
- Open **"Definition of boundary regions"**.



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.

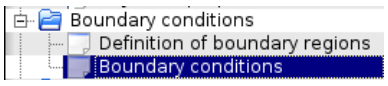


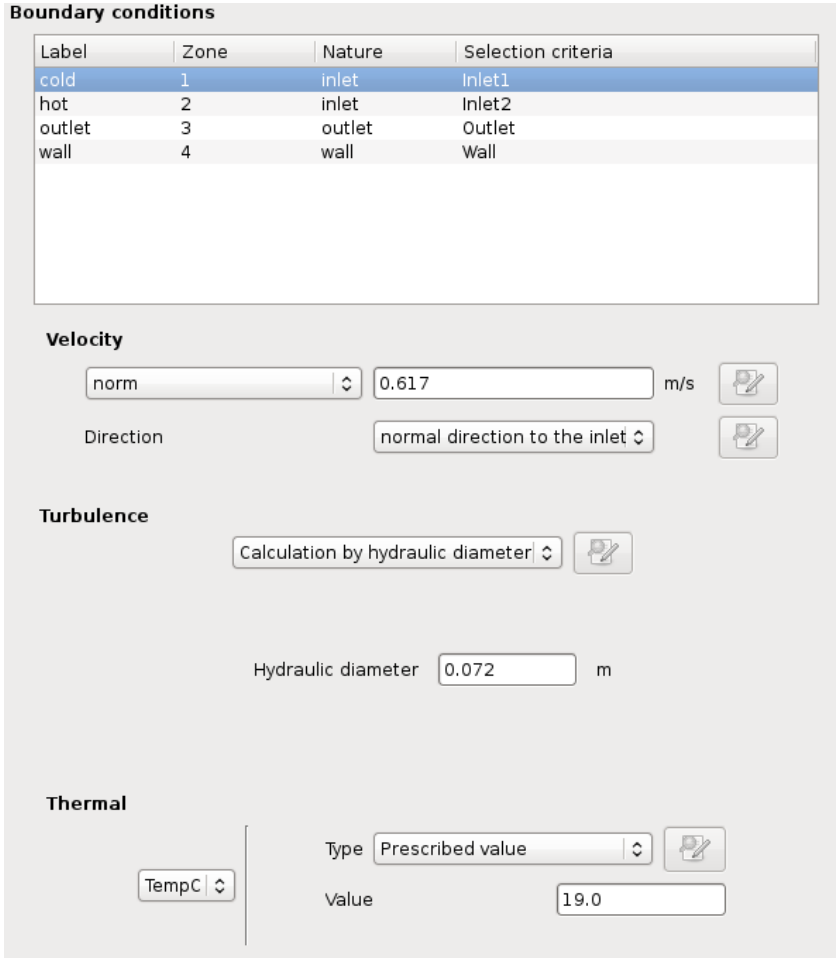
By default the nature of each new imported group is *Wall*. *Double click* in the cell of the nature in order to edit it. In the same way, edit the label of the boundary condition zone.

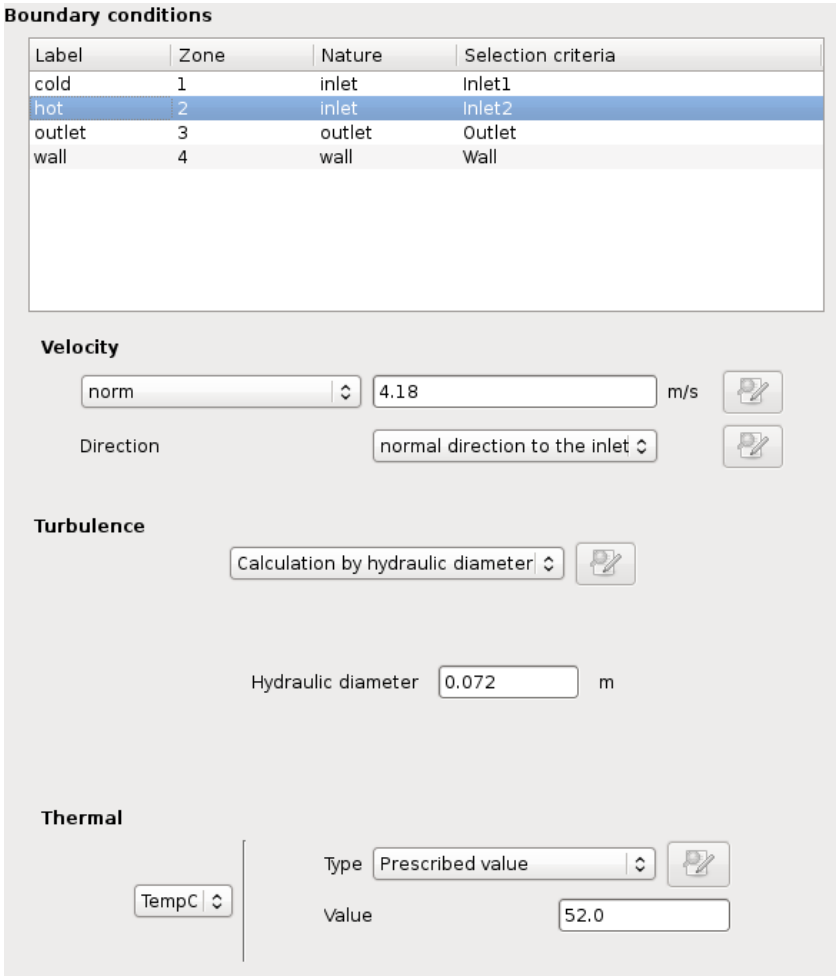


Boundary conditions values

- Open **"Boundary conditions"**. For each inlet, give norm for the velocity, the hydraulic diameter for the turbulence, and the prescribed value for the temperature, as shown on the figures below.



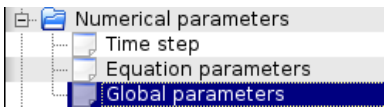

Boundary conditions
Label
Zone
Nature
Selection criteria
cold
1
inlet
Inlet1
hot
2
inlet
Inlet2
outlet
3
outlet
Outlet
wall
4
wall
Wall
Velocity
norm
0.617
m/s
Direction
normal direction to the inlet
Turbulence
Calculation by hydraulic diameter
Hydraulic diameter
0.072
m
Thermal
TempC
Type
Prescribed value
Value
19.0

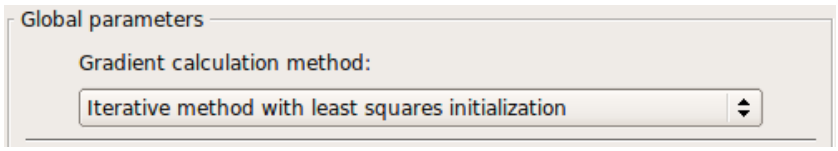


Numerical parameters

Global parameters

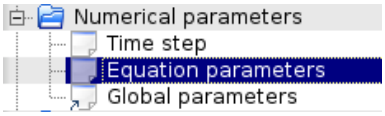
- The default gradient calculation method is changed to *Iterative method with Least Squares initialization*.

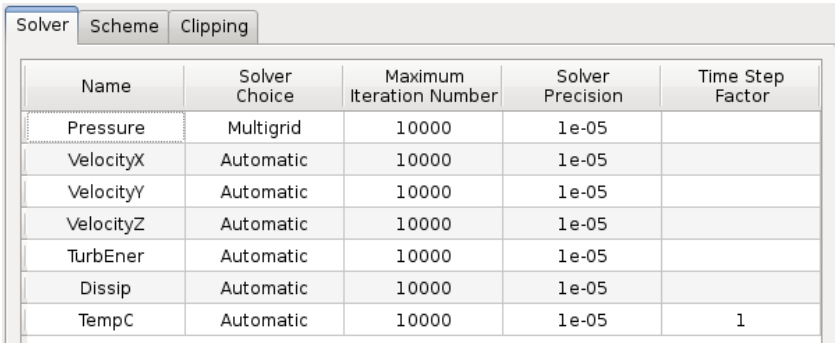




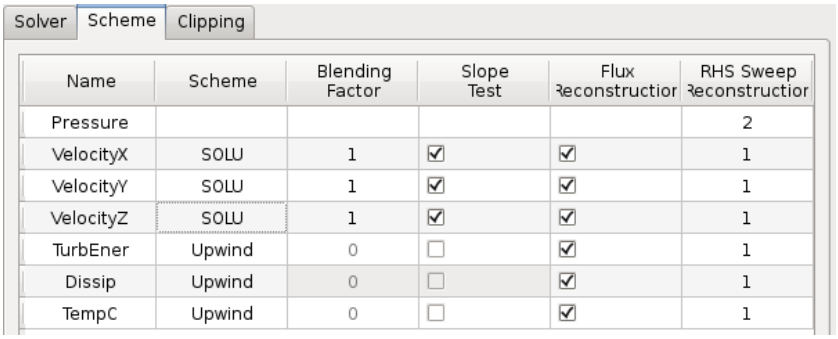
Equation parameters

- In order to save computation time, in the **"Solver"** tab, the precision is increase to 1.e-5 (select all the concerned cells, and <*Shift*> + *double right click* to edit all cells in a single time).



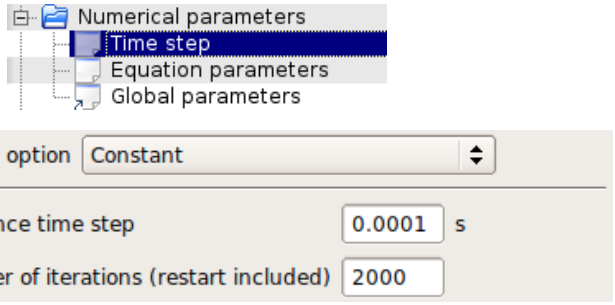


- In the **"Scheme"** tab, the convective scheme for the velocity is set to *SOLU* and for the turbulent variables and the temperature is set to *Upwind*.

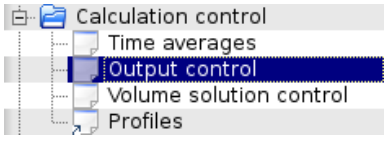


Time step

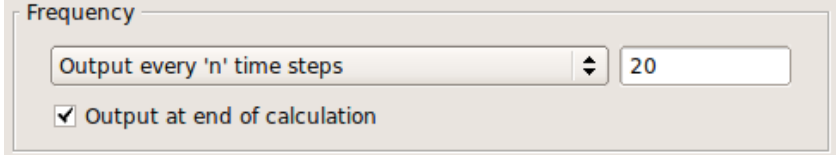
- In the **"Time step"** heading, set 0.0001 s for the time step. The number of iterations is set to 2000.



Calculation control



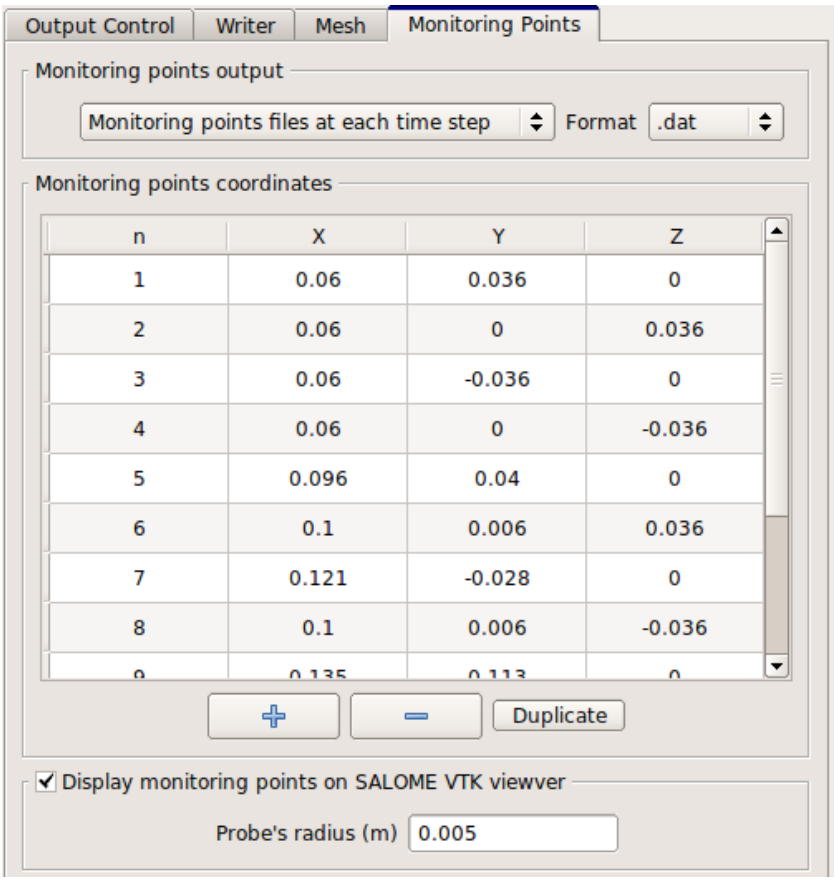
Writer In the **"Output control"** heading, tab **"Writer"**, define a frequency for the post-processing output, in order to do temporal animation with results.

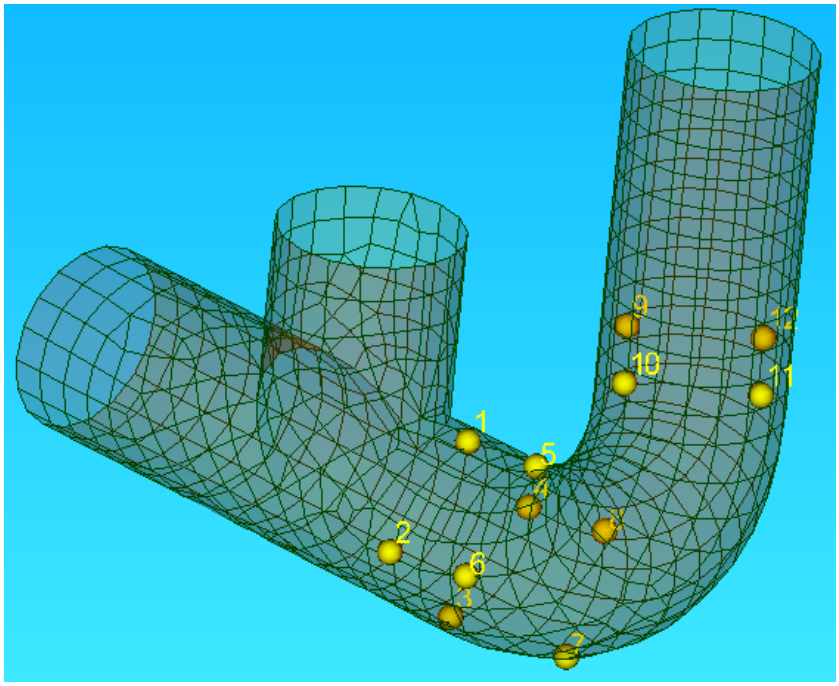


Define monitoring points The purpose of the monitoring points is to record for each time step, the value of selected variables. It allows to control stability and convergence of the calculation.

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

The positions of the monitoring points are displayed on the SALOME view. The probes radius is set to 0.005 m.

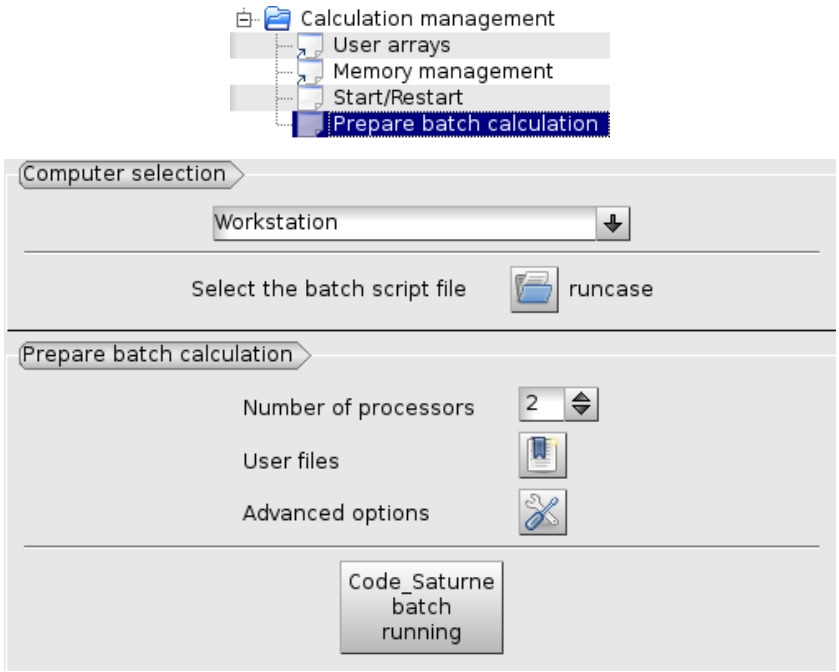




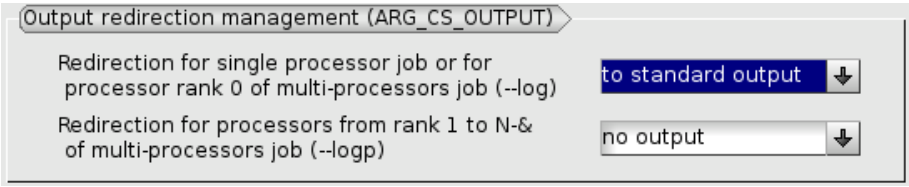
The format to be choosen (*dat* or *csv*) depends of the software which will plot the curves. For **Paravis**, *csv* must be selected.

Calculation

Select **"Prepare batch calculation"** heading.



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 could be omitted). 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"**. A popup window raises during the computation. When the computation is completed, click on **OK** to close the window.

2.1.6 Post processing of results

In this section only the loading of the data in **Paravis** and the first steps are covered.

Activate the module **Paravis**, then load the RESULTS.case by clicking the menu **File > Open ParaView file**. Click on the green button *Apply*. Now the data are loaded.

If more than a single mesh is present in the data (aka Part with the Ensight vocabulary), the filter *Extract Block* should be apply; select: **Filters > Alphabetical > Extract Block**. Then, in the *Propeties* tab, select the checkbox corresponding to the mesh to display, and click on the green button *Apply*.

It is possible to project cell data to the vertex; select **Filters > Alphabetical > Cell Data to Point Data**, and click on the green button *Apply*.

Finally, select in the *Display* tab the variable to color the mesh.