# **CFDSTUDY Documentation**

Release 1

EDF R&D

January 12, 2010

## CONTENTS

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

▼ Line Construction? □ ×				
Line	1			
• / •				
Result name				
Name Line_1				
Points				
Point 1 🕐 Vertex_1				
Point 2 🕜 Vertex_3				
Apply and Close <u>Apply</u> <u>C</u> lose <u>B</u>	Help			

Three lines must be defined:

Name	Name Point1	
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

▼ SALOME 5.1	.2ax - [Study1]	
<u>File</u> <u>E</u> dit <u>V</u> iew	New Entity Ope	erations Repair Measures <u>T</u> ools <u>W</u> indow <u>H</u> elp
	🗈 🛍 😡 Ge	ometry 🔄 🗟 😡 🛎 🏁 🕾 ≓ 🌿 🐐 🔸 🗸 💿 🕁 🖓 ∿ 🖌 🕁 🗐 🗐 🗐 🥥 🕹 🖬 🖉
Object Browser	<b>5</b> 23	OCC scene:1 - viewer:1
Name	Entry	20 上 22 会 ダ 20 回 G G 国 He 国
🗄 📾 Geometry	0:1:1	
+ Vertex_1	0:1:1:1	
+ Vertex_2	0:1:1:2	Arc Construction 2 E X
+ Vertex_3	0:1:1:3	
- + Vertex_4	0:1:1:4	Arc
+ Vertex_5	0:1:1:5	01 01 04
+ Vertex_6	0:1:1:6	Result name
+ Vertex_7	0:1:1:7	
□ □· / Line_1	0:1:1:8	Name Arc_2
E-/Line_2	0:1:1:10	Points
Arc 1	0:1:1:12	Center Paint
		Point Start
		Point End 🕐
		Apply and Close Apply Close Help
		Z
		L Y

• 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.

SALOME 5.1.2a)	x - [Study1	
<u>File Edit View Nev</u>	w Entity Op	erations Repair Measures <u>T</u> ools <u>W</u> indow <u>H</u> elp
🗋 📔 😹 🗶 🗈	📓 🛤 Ge	ometry 🔄 💀 🛤 📚 🗚 🛸 🏓 🖌 🔹 🗸 🕐 💩 🥱 A 🖉 🕹 🛣 🗐 🖬 🗐 🕲 🚳 🍐 🖬 👄 🖣
Object Browser	<b>5</b> 23	OCC scene:1 - viewer:1
Name	Entry	8 x 2 & 2 x 3 & 4 1 a + 10
🗄 📾 Geometry	0:1:1	
-+ Vertex_1	0:1:1:1	
- + Vertex_2	0:1:1:2	
- + Vertex_3	0:1:1:3	
- + Vertex_4	0:1:1:4	
+ Vertex_5	0:1:1:5	Y Create A Wire
- + Vertex_6	0:1:1:6	
- + Vertex_7	0:1:1:7	Wire
⊛ ∠ Line_1	0:1:1:8	03 <sup>+</sup>
⊪ ∕ Line_2	0:1:1:10	Besult name
i⊪ ∕ Line_3	0:1:1:11	
iii ∕ Arc_1	0:1:1:12	Name Wire_1
		Wire exection from wires/admos connected
		Whe creation norm whesredges connected
		Objects 🕐 3_objects
		Tolerance 1e-07
		Apply and Close <u>Apply</u> <u>Close</u> <u>Help</u>

• 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.

	Nan	ne	Center	• Point	Vector	Radius		
	Disk	1	Vertex_	1	Line_1	0.036		
	Disk	1	Vertex	4	Line_2	0.036		
Ohio at Da								
Object Br	owser	1		5	9 3 3 0 C 8	scene:1 - viewer:1		
⊡- 🤮 Ge	ometry	0:1:1	Entry	50 2 2	ମ୍ 💠 🖋 🔉	<mark>-) 🔮 <u>9</u> 🗖</mark>	i 🕪 🛄	1
±+ +	Vertex_1 Vertex_2	0:1:1 0:1:1	:1 :2	• •	Create A Disl	<	? 🖻 🗙	
+	Vertex_3	0:1:1	:3	Disk			1	
+	Vertex_4 Vertex_5	0:1:1	:4 :5	0 🆕	ی 💿	0 🧼		
+	Vertex_6 Vertex 7	0:1:1 0:1:1	:6 :7	(Result na	me			
<u></u> <u>−</u> /	Line_1	0:1:1	:8	Name Di	sk 1			†
₽ / ₽ /	Line_3	0:1:1	:10		-			+
⊡-∑~	Arc_l Wire l	0:1:1 0:1:1	:11 :12	Argument	s			+
ė- 🗖	Disk_1	0:1:1	:13	Center Po	oint 🕐 Vertex_3	1		
	* Line_1	0:1:1	:13:2	Vector	🕐 Line_1			
				Radius :	0,15		\$	
				Apply an	d Close Apply	<u>C</u> lose	<u>H</u> elp	
				Z Y				
				~x				
Object Br	rowser	1			000 9	scene:1 - viewer:1		
N ⊡ 😂 Ge	ame eometry	0:1:1	Entry	5 k /	P. 💠 🖋 🔉	🗖 🚱 🍕 🗖	i 🕪 🛄	
±. +	Vertex_1	0:1:1	:1	◄ (◄	Create A Disl	<	? 🛋 🗙	
+	Vertex_3	0:1:1	.:3	Disk			1	
+	Vertex_4 Vertex_5	0:1:1 0:1:1	::4 ::5	0 🆕	ی 🕑	0 🧇		
+	Vertex_6 Vertex_7	0:1:1	::6 ·7	Result na	me			
<u></u> <u> </u>	Line_1	0:1:1	:8	Name Di	sk_2			†
₽1 <u>/</u> ₽-/	Line_3	0:1:1	:10	Argument	5			
⊡- ∕ ⊕- ∑~	Arc_1 Wire_1	0:1:1 0:1:1	:11 :12	Contor Po	int 🔿 Vortov 1	2		
<u>-</u>	Disk_1	0:1:1	:13	Center Po	Int rentex_	2		
	*Line_1	0:1:1	:13:2	Vector	¢ Line_2			
B- <b>u</b>	Vertex_2	0:1:1	:14 :14:1	Radius :	0,10			
ι	* Line_2	0:1:1	:14:2	Annhuan	d Classa Annhu	Class		
				Apply an		Liose	Help	
				z Y				
				<b>A</b>				

• 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

Name		D 上 2 会 2 D 目 4 風 画 4 風
🗄 📾 Geometry	0:1:	
+ Vertex_1	0:1:	
+ Vertex_2	0:1:	Pipe Construction ? 🗉 X
-+ Vertex_3	0:1:	Pipe
⊕ + Vertex_4	0:1:	
- + Vertex_5	0:1:	
+ Vertex_6	0:1:	Result name
+ Vertex_7	0:1:	Name Pipe 2
	0:1:	
	0:1:	Arguments
⊪- ∕ Line_3	0:1:	Base Object 🖉 Disk 2
	0:1:	
i⊧-≫ Wire_1	0:1:	Path Object 🕐 Line_2
⊪ 🔳 Disk_1	0:1:	Select unpublished edges
🐵 🔳 Disk_2	0:1:	
è ≇ Pipe_1	0:1:	Apply and Close Apply Close Help
<b>•</b>	ф	

#### 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		
_	1		-	
-	ኛ Fuse Two	Objects	3	
	Fuse			
	•			
	Result name	>		- 1
	Name Pipe			
	Arguments			- 1
	Object 1	Pipe_1		
	Object 2 🌾	Pipe_2		
	Advanced op	tions		
	🗌 Set prese	ntation parar	meters and subshapes from argume	nts
	A <u>p</u> ply and C	lose <u>A</u> pp	oly <u>C</u> lose <u>H</u> elp	

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

Name		つ よ ど 会 ダ な 回 後 気 届 49 個
🗄 📾 Geometry	0:1:	
⊕ + Vertex_1	0:1:	Vhatis Information 7. E X
- + Vertex_2	0:1:	Whatis
- + Vertex_3	0:1:	
	0:1:	
-+ Vertex_5	0:1:	Object And Its Topological Information
- + Vertex_6	0:1:	
+ Vertex_7	0:1:	Object Pipe
⊪ ∕ Line_1	0:1:	Number of sub-shapes :
⊪ ∕ Line_2	0:1:	VERTEX : 7
⊪ ∕ Line_3	0:1:	EDGE : 11 Check Shape Information 2 C X
⊪ ∕ Arc_1	0:1:	WIRE : 8 Check Shape
⊪-≫ Wire_1	0:1:	FACE : 7
⊛ 🔳 Disk_1	0:1:	SHELL : 1
⊛ 🔳 Disk_2	0:1:	SOLID : 1 Object And Its Topological Information
⊪ 🛱 Pipe_1	0:1:	
⊪ # Pipe_2	0:1:	
🖮 🖉 Pipe	0:1:	SHAPE:35 This Shape seems to 1
		be valid.
		Kind of Shape : 1_POLYHEDRON Basic Properties
		Close Help Close Help
4	*	

## 3.4. Groups for boundary conditions definition

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

Na	ame						
🖻 😂 Geometry							
+ Vertex_1							
⊕ + Vertex_2							
+ Vertex_3							
+ Vertex_4							
+ Vertex_5							
+ Vertex 7							
⊕ ∕ Line_1							
≞. ∕ Line_2							
🖶 / Line_3							
⊡ ∕ Arc_1							
⊕ ∑ Wire_1							
⊞ ∎ Disk_1							
⊕ Bine 1							
⊕ war Pipe_1							
⊡ <mark>©</mark> Pipe							
⊕ and Pipe_2	Rename	F2					
₽ Pipe	Rename Delete	F2 Del					
Pipe	Rename Delete Create Group	F2 Del					
Pipe	Rename Delete Create Group Color	F2 Del					
Pipe	Rename Delete Create Group Color Auto Color	F2 Del					
Pipe	Rename Delete Create Group Color Auto Color Hide	F2 Del					
Pipe	Rename Delete Create Group Color Auto Color Hide Show Only	F2 Del					
Pipe	Rename Delete Create Group Color Auto Color Hide Show Only Refresh	F2 Del					
Pipe	Rename Delete Create Group Color Auto Color Hide Show Only Refresh Expand All	F2 Del					

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  $\langle Shift \rangle + right click$  to make a multiple selection: the wall is constituted with 4 faces.







## 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.

🔻 SALOME 5,1,2ax - [Stu	udy1]													_
<u>F</u> ile <u>E</u> dit <u>V</u> iew Mesh Cor	ntrols	Modification Tool	ls <u>W</u> indow <u>H</u> elp											SA
🗋 🗎 😹 🗶 🗎 🛍 🕼	⊫ Mes	h 🕹 🖹	و 😰 😕 📚 و	差 🎽 🐚 😽	4 4 4 2 2			<b>X</b>	AS	9	4	»	« ,	а, н
Object Browser	53	OCC sce	ne:1 - viewer:1	VT	K scene:1 - viewer	:1								
Name	Ŷ	80 1. 24	. 🤊 🗴 🗖 🐫 🚳	i 🗖 💩 💀 🕞										
🗄 📾 Geometry	0:													
e⊢+ Vertex_1	0:					_								
- + Vertex_2	0:	🕆 Create mes	ih											
- + Vertex_3	0:	Name	Mesh_1											
⊕ + Vertex_4	0:	Geometry 🔗	Pipe			i II								
- + Vertex_5	0:		ipo			╹	👻 Hy	pothe	sis Cor	nstru				$\times$
-+ Vertex_6	0:	3D 2D 1D	00				🚳 Ne	tgen 3	D					
- + Vertex_7	0:	30 20 10	UD				Arq	uments	D					
i≞ ∕ Line_1	0:	Algorithm	Netgen 1D-2D-3D	0	+					6				
⊕ / Line_2	0:	Hypothesis	<none></none>				Nar	ne		E	EN 3D F	aram	eters	
⊕ ∕ Line_3	0:						Max	x. Size		C	0.05			
Arc_1     Arc_1	0	Add. Hypothes	sis <none></none>		+ 🗟 🖉			econd	Order	_				
	0.					51		ooona	oraoi	F				
Disk_1	0.		Assign a set o	of hypotheses			Fine	eness		E	ine		*	
B Bine 1	0	Apply and Close	e <u>A</u> pply	Close	Help		Gro	wth Ra	ite	C	).2			
e a Pipe_1	0			(		_	Nb.	Segs (	oer Edg	e	2			
e Ø Pipe	0						NIL				)			
-* Pipe 1	0:						IND.	Segs	ber Had	ius [-	)		$\overline{\nabla}$	
-* Pipe 2	0:						ସ C	ptimize	9					
🗉 🗃 inlet1	o:						_							41.
🕸 🖉 inlet2	0:						OK	< C	ancel				Help	
🗉 🗃 outlet	o:													
⊡ 🗿 wall	0:													

- 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.

Name 🍨	5 C L Z & F & T & C & C		
I III / Line_1			_
	ζ.	Create Groups from Geometry	×
i⊪ ∕ Line_3		Mash @ Dire	
⊪ ∕ Arc_1		Mesh r Pipe	
i⊪ ≫ Wire_1		Elements	
🗉 🖬 Disk_1		Geometry 🥜 inlet1	-
⊕ <b>⊒</b> Disk_2		inlet2	
⊪ i Pipe_1		outlet	
⊪ ₩ Pipe_2		wall	
i∃ a Pipe	$\langle \rangle$		
- * Pipe_1			
* Pipe_2			
inlet1		,	
⊕ inlet2		Nodes	
eren and the second se		Geometry 🥜	-
🗈 📷 Wall			
🖻 🥨 Miesn	And and the test of the		
* Pipe			
Applied hypotheses			
4 II 4		Apply and Close Apply Close Help	
Python Console			

• 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  $\langle Ctrl \rangle + S$ ).

## 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.

CFDSTUDY information				
Select CFD code				
⊙ Code_Saturne				
O NEPTUNE_CFD				
Informations				
version: 2.0.0-beta2				
prefix: /home/saturne/Code_Saturne/2.0-beta2/arch/Linux_x86_64				
ОК				

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*.

🝸 Location of (	CFDSTUDY study	? _ 🗆 X
Study location Study name	e/COURS/ENPC_2009_2010/TJUNCTION PIPESTUDY	Browse
Add new Cas	ses	
ОК		CANCEL

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"**:

🍋 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.

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

E Calculation environment					
Meshes Periodic Boundaries					
List of meshes					
Meshes	Format	Number			
Pipe.med	MED ".med"	1			
Add	Add Delete				
□ (oin meshes )					
Subdivide warped faces					
Correct cell and face orientation	15				

## 5.3.2 Mesh quality criteria

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

<ul> <li>Calculation environment</li> <li>Meshes selection</li> <li>Mesh quality criteria</li> </ul>					
Mesh quality criteria					
Post-processing for	rmat MED 🔸				
Options					
format	binary 🔸				
polygons	display 🕹				
polyhedra	display 🔸				
big-endian					
Interior faces selection for post-processing					
Check mesh					

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**").

🗠 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 Tempreature (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.

Density			
cons	stant	Ŧ	22
Refe	erence value	ρ 998	kg/m³
Viscosity			
cor	nstant	4	2
Ref	erence value	μ 0.001	Pa.s
Specific heat			
cons	stant	4	2/
Refe	rence value	Cp 4181	J/kg/K
Thermal conductivit	у>		
con	stant	Ŷ	2
Refe	erence value	λ 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. Fisrt, 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 * TempC -0.000415645022
 * TempC^2;
```

	User expression Predefined symbols Examples
	<pre>rho = 1000.94843 - 0.049388484*TempC -0.000415645022 * TempC^2;</pre>
<pre>- viscosity: TempC^2;</pre>	mu = 0.0015452 - 3.2212e-5 * TempC + 2.45422 *
	User expression Predefined symbols Examples
	<pre>mu = 0.0015452 - 3.2212e-5 * TempC + 2.45422 * TempC^2;</pre>
- thermal conductiv 9.01853355e-7	ity: lambda = 0.57423867 + 0.01443305 * TempC - * TempC^2;
User express	sion Predefined symbols Examples
lambda =0.5	7423867 + 0.01443305 * TempC - 9.01853355e-7 * TempC^2;

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

	Physical properties     Reference values     Fluid properties     Gravity, hydrostatic pr	
Gravity		
	Gravity along X -9.81 m/s²	
	Gravity along Y 0.0 m/s <sup>2</sup>	
	Gravity along Z 0.0 m/s <sup>2</sup>	

## 5.3.5 Initialization

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

🖻 🗁 🔁 Voli	ume conditions
	Volume regions definition
	Initialization
···· > > >	Head losses
Thermal scalar initialization	
Temp(	C 19.0 °C

## 5.3.6 Boundary conditions

#### **5.3.6.1 Define locations graphicaly**

• 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.

0	Definition of boundary regions						
	Label	Zone	Nature	Selection criteria			
	Wall_1	1	Wall	Inlet1			
ľ	Wall_2	2	Wall	Inlet2			
ľ	Wall_3	3	Wall	Outlet			
ľ	Wall_4	4	Wall	Wall			
	Add Delete Add from Salome						
	Add from Prepocessor listing						
	Import groups and references from Preprocessor listing						

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.

pefinition of boundary regions								
Label	Zone	Nature	Selection criteria					
cold	1	Inlet	etl					
hot	2	Inlet	Inlet2					
outlet	3	Outlet	Outlet					
wall	4	Wall	Wall					
(Add fro	Add Delete Add from Salome (Add from Prepocessor listing) Import groups and references from Preprocessor listing							

#### 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.



oundary co	nditions	>				
Label	Zor	ne Na	ature	Sele	ction criteria	
cold	1	inl	et	Inlet:	l	
hot	2	inl	et	Inlet	2	
outlet	3	OL	itlet	Outle	et	
wall	4	Wa	all	Wall		
Velocity						
norm			♦ 0.617		m/s 🕅	
Direction						
normal direction to the inlet						
Turbulenc	e					
	Calc	ulation by hydr	aulic diamete	er	4	
Hydraulic diameter 0.072 m						
Scalars						
Scalar	Name	Туре	Value		Exchange Coefficient	
TempC		Prescribed v	19			

abel.	Zor	ie Nat	ure	Selection criteria		
old	1	inle	t	Inlet1		
not	2	inle	t	Inlet2		
outlet	3	out	let	Outlet		
vall	4	wal		Wall		
Velocity	,					
norm		4	4.18	m/s 🕅		
		Dire	ction >			
normal direction to the inlet 🔸 📝						
Turbulen	ce					
	Calc	ulation by hydra	ulic diamete	er 🕹		
Hydraulic diameter 0.072 m						
Scalars						
Scalar	Name	Туре	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.

🖻 🔁 Numerical parameters

📰 Time step
Equation parameters
🦾 💭 Global parameters
(Insteady flow algorithm management
(onseeddy new algorienni managemenie
Time step option Uniform and constant
Reference time step 0.0002 s
Number of iterations (restart included) 1000
Ontion rare time star.
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).

	i⇒ i Parameters Time step Equation parameters Global parameters								
	Solver Schen	ne							
	Name	Scheme	Blending Factor	Slope Test	Flux Reconstructior				
ľ	VelocityX	Centered	1	<b>I</b>	<b>I</b>				
ľ	VelocityY	Centered	1	<b>I</b>	<b>I</b>				
ľ	VelocityZ	Centered	1	<b>I</b>	<b>I</b>				
	TurbEner	Upwind	0		<b>I</b>				
	Dissip	Upwind	0		4				
ľ	TempC	Upwind	0		3				

#### **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.

🖻 🤗 Numerical parameters	
Equation parameters	
🗤 🗾 Global parameters	
(Global parameters	
Gradient calculation method:	
Least squares method over partial extended cell neighborhood 🛛 🔸	

## 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

<ul> <li>Calculation control</li> <li>Time averages</li> <li>Output control</li> <li>Volume solution control</li> <li>Profiles</li> </ul>										
Output Control M	Output Control Monitoring Points Coordinates									
Monitoring points r	ecording									
Monitori	ng points files at e	each time step	➡ 1							
Monitoring points o	oordinates									
n	n X Y Z 🕈									
1	0.06	0.036	0	11						
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	0.108	0.006	0.036							
7	0.121	-0.028	0							
R	0108	0.006	-0.036	÷						
	Add	Delete								

#### 5.4 Calculation

Select "Prepare batch calculation".

Calculation management Calculation management User arrays Memory management Start/Restart Prepare batch calculation							
Computer selection							
Workstation							
Select the batch script file 🛛 🖉 runcase							
Prepare batch calculation							
Number of processors 2							
User files							
Advanced options							
Code_Saturne batch running							

Before running *Code\_Saturne*, save the case file (toolbar button or "**File** > **Code\_Saturne** > **Save as data xml file**" or  $\langle Shif \rangle + \langle Ctrl \rangle + S \rangle$ , 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*.

Output redirection management (ARG_CS_OUTPUT)	
Redirection for single processor job or for processor rank 0 of multi-processors job (log)	to standard output 🛛 🕹
Redirection for processors from rank 1 to N-& of multi-processors job (logp)	no 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:

🝸 Setup Plo	t 2d								
Axis	Assigned	Data	Units [			Attribu	tes		Ŷ
ΗV	+	nt	iter	🗹 Auto assign	Solid	♦ 0	¢ Circle	e 🔸 🔳 +	_
	+	t	s	🗹 Auto assign	Solid	<b>+</b> 0	¢ Circle	• •	
н	4	1	-	🗹 Auto assign	Solid	<b>+</b> 0	¢ Circle	• •	
ΗV	+	2	-	🗹 Auto assign	Solid	♦ 0	¢ Circle	• •	
ΗV	+	3	-	🗹 Auto assign	Solid	♦ 0	¢ Circle	•	
ΗV	+	4	-	🗹 Auto assign	Solid	♦ 0	¢ Circle	•	
ΗV	+	5	-	🗹 Auto assign	Solid	♦ 0	¢ Circle	• •	
ΗV	+	6	-	🗹 Auto assign	Solid	♦ 0	¢ Circle	•	
ΗV	+	7	-	🗹 Auto assign	Solid	↓ 0	¢ Circle	•	
ΗV	+	8	-	🗹 Auto assign	Solid	↓ 0	¢ Circle	• •	
ΗV	+	9	-	🗹 Auto assign	Solid	↓ 0	¢ Circle	•	
ΗV	+	10	-	🗹 Auto assign	Solid	♦ 0	¢ Circle	• •	
ΗV	+	11	-	🗹 Auto assign	Solid	↓ 0	¢ Circle	•	
ΗV	+	12	-	🗹 Auto assign	Solid	♦ 0	¢ Circle	•	
									\$
	<u>0</u> k					<u>C</u> ancel		Help	

**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. The select the popup menu "Vectors" or "StreamLines" (*click left* on *Velocity*).



