#### EDF R&D



FLUID DYNAMICS, POWER GENERATION AND ENVIRONMENT DEPARTMENT SINGLE PHASE THERMAL-HYDRAULICS GROUP

6, quai Watier F-78401 Chatou Cedex

MAY 2013

 $\operatorname{documentation}$ 

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

contact: saturne-support@edf.fr



# CONTENTS

1	Gene	ralities about CFDSTUDY	1
	1.1	Introduction	1
	1.2	Reference functionalities	1
2 Tutorials			
	2.1	<i>Code</i> Saturne tutorial: <b>turbulent mixing in a T-junction</b>	7

# **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 NEP-TUNE\_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.

👻 Location of (	CFDSTUDY study	? _ 🗆 X
Study location Study name	e/COURS/ENPC_2009_2010/TJUNCTION PIPESTUDY	Browse
✓ Add new Cas Cases CASE1	ses	
ОК		CANCEL

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

CFDSTUDY information
Select CFD code
<ul> <li>Code Saturne</li> </ul>
O NEPTUNE_CFD
Informations
version: 3.1.0-alpha
prefix: /local00/home/C61847/salome/neptune/arch/salome65ncfd21
ОК

- \* 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:



- \* 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:

•	Add Case	
U	Update Object Browser	
	Run Case	
	Refresh	F5
	Expand All	
	Collapse All	
	Find	Ctrl+F

– Mesh file:

-@	Export in SMESH	
	Refresh	F5
	Find	Ctrl+F

- Case directory:



GUI	Launch GUI	
	Refresh	F5
	Find	Ctrl+F

– File of parameters:



- File of functions in the *SRC* directory:

2	Check compilation	
2	Edit	
2	Move to DRAFT	
Þ	Copy into another Case	
	Refresh	F5
	Find	Ctrl+F

- File of functions in the *REFERENCE* directory:

F5
Ctrl+F

- File of functions in the *DRAFT* directory:

2	Edit	
$\times$	Remove	
C)	Copy in SRC	
	Refresh	F5
	Find	Ctrl+F

- Script of *runcase* file:

4	View	
	Run script	
	Refresh	F5
	Find	Ctrl+F

- Results directories in the *RESU* directory:



# 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

۴	Salom	e NoteBook	×
	Variable Name	Variable Value	_
1	P1	-0.14	
2	P2	0.1	
3	Р3	0.095	Ξ
4	P4	0.171	
5	P5	0.24	
6	P6	0.076	
7	radius	0.036	~
	Remove	Update Study	ā

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	Ζ
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.

Object Browser		00	C scene:1 - view	er:1			1						×
Image         Name           Image         Geometry           Image         + Vertex, 1           Image         + Vertex, 2           Image         + Vertex, 4           Image         + Vertex, 5           Image         + Vertex_6	<b>\$</b>	4 q C	Ø~ ♣~ 3	P 33	<b>.</b>	5 C	C.	¶   <b>⊫</b>	<b>0</b>	lo é	1 2	B	
🔫 Poin	t Construction		<u>x</u>										
Points													
• • •	o 🖾 🛛 🗙	0 🜌	)8										
Result name Name Vertex_7													
Coordinates													
X: P4		<b>^</b>											
Y: P5		<b>^</b>											
Z : 0													
Apply and Close	Apply <u>C</u> lose	<u>H</u> elp											

• 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>H</u> elp

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).

Arc			
03	•	0 📥	

Then the arc must be defined:

	Name	Center Point	Start Point	End Point		
	Arc_1	Vertex_5	Vertex_3	Vertex_6		
@ ~ \$	Name Geometry + Vertex_1	5	• <b>*</b> • <b>?</b> • 4	<i>\$</i> ° ⊅	C 🚱 🏝	🖻 🕪 🖻 🎄 🗞
@ @ @ @	+ Vertex_2 + Vertex_3 + Vertex_4 + Vertex_5 + Vertex_6					
•	+ Vertex_7 / Line_1 / Line_2 / Line_3					
*		Arc Construction				
Arc	Ĵ	ାସି ୍≽				
Result	name					
Nam	e Arc_1					
Points						
Cent	er Point 🕜	Vertex_5				
Poin	t Start 🛛 🍖	Vertex_3				
Poin	t End  🥐	Vertex_6				
🗆 F	Reverse					

• 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				
Name				_			
🔻 😫 Geometry			~ 💠 🖋 🚿	🗇 - 🏷	C 🕼 🤇	🌯 - 🛛 🗖	🅪 🛄 🔎 🤌
b + Vertex_1		-					
+ Vertex_2							
I≥ + Vertex_3							
▶ + Vertex_4							
P + Vertex_5							
# Vertex_6							
■ N / Line 1							
■ p/Line_1 ■ b/Line 2							
D D ∠Line 3							
D / Arc 1				/			
D >> Wire_1							
			$\sim$ $\sim$				
-	Diele Comptending		$\sim$				
	Disk Construction	×					
Disk						/	
- +							
0 🆕	🔍 🎃 🛛 🗢						
Result name							
Nomo Dick 1							
Arguments							
Center Point 🛛 🍖	[Vertex_1						
Vector 🍖	Line_1						
Radius : radius		÷					

۲	Name		t [0]	.aa	~ _		¢1.	69		-	Den	*	
⊽ 😡	Geometry	1 🛛 🖤 🕒 📭	<u>,</u>	🚓 🔨 🔪	🥒 🛄 🗸	5 6	<b>S</b>	4 <u>1</u> ~	124			<u> </u>	<b>60</b>
۲	+ Vertex_1												
۲	+ Vertex_2												
۲	+ Vertex_3												
۲	+ Vertex_4												
۲	+ Vertex_5												
۲	+ Vertex_6												
۲	+ Vertex_7				-								
	/ Line_1												
A 1	/ Line_2												
(i)	/ Line_3												
(a) p	/ Arc_1												
	> wire_i	$\smile$		_									
و مع	Disk_1			$\smallsetminus$								/	
Ý	Disk Construct	tion	×	$\sim$									
Disk									/				
0 📥	•	0 🦛											
		· •											
Result n	ame												
Name	Disk_2												
			_										
Argumer	nts												
Center	Point 🕐 Vertex_4												
Vector	Line_2												
Vector	Line_2												
Vector Radius	Line_2												

• 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	1						
	Pipe_2	Disk_2	Line_2	1						
	-			1						
	Name       Geometry       + Vertex_1       + Vertex_2       + Vertex_4       + Vertex_5       + Vertex_6       + Vertex_7       > / Line_1       > / Line_3       > / Arc_1       > Wire_1       > Disk_1       > Disk_2		<b>क क २० क</b>	9 D	5	¢ 🚱	Q	•	<u>a</u>	R
Ŷ		Pipe Construction	×							
Pipe										
۲	10	o 🗶								
Result	t name ne Pipe_1				_	_	_			
Argum	e Object 🔗	Disk 1								
Path	n Object 🕜	Wire_1								
	Select unpublish	ned edges								
Advan	ced options									
<b>V</b>	Preview									



#### 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			
	• Fuse Two (	Obiects	-	2 🗆	×
	Fuse	>			
	Arguments Object 1 🕐 Object 2 🔗	Pipe_1 Pipe_2			
	(Advanced op Set preser Apply and Cl	tions ntation parar ose <u>A</u> pp	meters and subshapes from argum ply <u>C</u> lose <u>H</u> el	ents Ip	

Name           Geometry           + Vertex_1           + Vertex_2           + Vertex_4           + Vertex_5           + Vertex_6           + Vertex_7	4 व <b>ं</b> ©	· P &- J	⊅r <b>⊡</b> ~ 5	C & @_~	<b>6</b>   40 <b>  2</b>	<u>j</u> ∎ n2g → >>
	tion					
Object And its Topological Informat Object Pipe Number of sub-shapes : VERTEX: 8 EOGE : 13 WIRE : 8 FACE : 7 SHELL: 1 SOUD : 1 COMPSOLID : 0 COMPSOLID : 0 SHAPE : 38	:ion	Check Shape Check Shape Object And Its Object	eck Shape Inform	rmation X		
Kind of Shape : Solid	Basic Properties	This Shap	e seems to be v	alid.		

• 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):

Shape Type			
○ +	0 /	•	0 🔎

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.

Name     Geometry     Hotor 1	<b>9</b> C	\$	1 2	~ 🂠~	S	Ø	<b>0</b> ~	5	C 🕃	۹ <u>·</u> -	æ	•	0	b. 💫
Create Gro	up			×	1									
Shape Type														
○ + ○ / ●		0												
Group Name														
Name Injet]														
Main Shape And Sub-shapes														
Main Shape 🕜 Pipe														
Main Shape Selection restriction		Show	only sele	cted										
<ul> <li>No restriction</li> </ul>		Hide	e selecte											
Geometrical parts of the Second S	Shape													
<ul> <li>Only Sub-shapes of the Second Sh</li> </ul>	nape	Show a	ill sub-sh	apes										
Second Shape						1								
3		s	elect All								-			
			Add											
		F	Remove											

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.

۲	Name マ 😂 Geometry		5	र्ष 🕐	<u>الل</u>	ρ.		SP	۲	<b>0</b> ~	5	¢ (	<u>i</u>	<u>ب</u> آ	F	•	<u>þ</u> .	æ
۲	+ Vertex_1																	
~		Create	Group				$(\mathbf{X})$											
s	hape Type																	
	○ +	0 /	۲	(	0 🔎													
G	roup Name																	
	Name Inlet2																	
м	lain Shape And Su	ub-shapes																
	Main Shape 🕜	Pipe																
	Main Shape Sele	ection restriction	n	Show	v only s	electe	d											
	No restriction	on		Hi	de sele	ected							-	-1				
	<ul> <li>Geometrica</li> </ul>	l parts of the Seco	ond Shape				5											
	<ul> <li>Only Sub-sh</li> </ul>	apes of the Secor	nd Shape	Show	all sub	o-shape	es											
	Second Shape	P		)														
	35				Select	All								_				
					Add													
					Remov	ve												

Name     Geometry     Hyptox 1	<b>\$</b>	٢	ক	Ļ	₽.	<b>⊷</b> •	۶	۵	<b>0</b> ~	5	¢	6	¶	F	•		þ.	R
Create	Group					×												
Shape Type O + O /	•		0														$\subset$	
Group Name Name Outlet																		
Main Shape And Sub-shapes						_												
Main Shape																		
Main Shape Selection restriction <ul> <li>No restriction</li> </ul>	1	9	Show ( Hide	only s e sele	electe ected	d												
<ul> <li>Geometrical parts of the Seco</li> </ul>	nd Shape	,, 	how a	all sub	-shap	es												
Only Sub-shapes of the Secon	d Shape	]		in out	, and p													
37			S	elect	All									_				
			F	Add	ve													
		_																
<ul> <li>P Name</li> <li>         √ (a) Geometry         <ul> <li>             + Vertex 1         </li> </ul> </li> </ul>	5	0	ক	<u>الل</u>	<b>P</b> ~	<u>م</u>	S	Ø	<b>0</b> ~	5	¢	G	¶.~	E	•	10	þ.	R
Name     Seometry     + Vertex_1     Create	Group	٢	ব	JL.	<b>P</b> ~		S	Ŋ	<b>0</b> ~	5	¢	¢}	¶_~	E	• •	0	<u>þ</u> .	R
Name     Seometry     + Vertex_1     Create Shape Type     +	Group	٢	<b>व</b> † 0	ł	<b>P</b> ~	** ~	59	ð	<b>•</b> ~	S	¢	CF.	۹ <u>۰</u> ۰	F	•		<u>þ</u>	R
Name     Secondary     Vertex_1     Create     Shape Type     +     O +     Group Name     O	Group	٢	<b>व</b> 0	1	<b>P</b> ~	*••	59	¢	<b>O</b> ~	5	¢	C.	<b>Q</b> _~	ø	•		<b>卢</b>	
Name     Geometry     + Vertex_1      Create  Shape Type     +      Group Name     Name     Name     Wall	Group	٥	<b>ৰা</b> ০		<b>P</b> ~	◆ →	59	ð	<b>0</b> ~	5	¢	<b>(</b> }	<b>Q</b> _~		• •		<u>a</u>	~
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> </ul> Shape Type <ul> <li>+</li> <li>✓</li> </ul> Group Name <ul> <li>Name</li> <li>Wall</li> </ul> Main Shape And Sub-shapes	Group	٥	<b>4</b>		2.		59	ð	<b>0</b> ~	3	¢	<b>(</b>	<b>A</b>		•		<u>a</u>	
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> </ul> Shape Type <ul> <li>+</li> <li>✓</li> </ul> Group Name <ul> <li>Name Wall</li> </ul> Main Shape And Sub-shapes <ul> <li>Main Shape Yeipe</li> </ul>	Group	٢	<b>et</b> 0		2.	 ☆ • • 	59	3	<b>0</b> ~	3	¢	8	۹ <u>۰</u> - ۲		• •			<b>R</b>
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> </ul> Shape Type <ul> <li>+</li> <li>/</li> </ul> Group Name <ul> <li>Name Wall</li> </ul> Main Shape And Sub-shapes <ul> <li>Main Shape Pipe</li> <li>Main Shape Selection restriction</li> </ul>	Group		tp O	یک only s	P ↓		<i>5</i> °	ð	<b>0</b> ~	5	¢	¢.	Q		• •			
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> <li>Shape Type</li> <li>+</li> <li>/</li> <li>Group Name</li> <li>Name [Wall]</li> <li>Main Shape And Sub-shapes</li> <li>Main Shape Selection restriction</li> <li>No restriction</li> </ul>	Image: Second	0	5how «	only s	₽ ~		5	ð,	<b>•</b>	5	¢	¢.	Q		• •			
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> </ul> Shape Type <ul> <li>+</li> <li>/</li> </ul> Group Name <ul> <li>Name [Wall]</li> </ul> Main Shape And Sub-shapes <ul> <li>Main Shape And Sub-shapes</li> </ul> Main Shape [Image] <ul> <li>Pipe</li> </ul> Main Shape Selection restriction <ul> <li>Geometrical parts of the Second</li> <li>On the Sub-shape set the Second</li> </ul>	Group     Oroup     oroup     oroup		Image: Show of Hide       Hide	only s e sele	electe	**************************************	<i>3</i> °	X	<b>7</b> ~	5	¢	6	<b>₩</b> .		•••		<u>a</u>	<b>R</b>
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> </ul> Shape Type <ul> <li>+</li> <li>/</li> </ul> Group Name <ul> <li>Name Wall</li> </ul> Main Shape And Sub-shapes <ul> <li>Main Shape And Sub-shapes</li> </ul> Main Shape Pipe <ul> <li>Main Shape Selection restriction</li> <li>Geometrical parts of the Second</li> <li>Only Sub-shapes of the Second</li> <li>Second Shape ()</li> </ul>	Group     Group     ond Shape     d Shape	©	to O Show o Hide	only s e sele	electe ected -shape	** ~ (X) (X) (A) (A) (A) (A) (A) (A) (A) (A) (A) (A	5	x	0.	3	c	6			• •			
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> </ul> Shape Type <ul> <li>+</li> <li>Group Name</li> <li>Name</li> <li>Wall</li> </ul> Main Shape And Sub-shapes <ul> <li>Main Shape And Sub-shapes</li> <li>Main Shape Image Pipe</li> <li>Main Shape Selection restriction</li> <li>Geometrical parts of the Second</li> <li>Only Sub-shapes of the Second</li> <li>Second Shape</li> </ul>	Group     Group     ond Shape     d Shape		tp O Show o Hidd	only s e sele all sub	electe ected -shape		59	ð.	0.	5	¢	8						
<ul> <li>Name</li> <li>Geometry</li> <li>+ Vertex_1</li> <li>Create</li> <li>Shape Type</li> <li>+</li> <li>Group Name</li> <li>Name Wall</li> <li>Main Shape And Sub-shapes</li> <li>Main Shape  Pipe</li> <li>Main Shape Selection restriction</li> <li>Geometrical parts of the Secon</li> <li>Only Sub-shapes of the Secon</li> <li>Second Shape  </li> <li>30</li> <li>20</li> <li>7</li> </ul>	Group Group ond Shape d Shape		5how o Hido	only s e sele all sub	electe ected All		58	×	•••	5	¢	8						

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.

v / Line_s b / Arc 1	*	Hypothesis Construction	×
▷ >> Wire_1	💥 Viscous Layers		
Disk_1 Disk_2	Arguments		
▷	Total thickness	0.01	-
✓ I Pipe	Number of layers	2	~
* Pipe_2	Stretch factor	1	
Dilet1		3	bbA
Dutlet	Faces without layers	35 37	
✓ Wesh N the set of th	(inlets and oulets)		Remove
D ≥ Algorithms			
● ▷ ☞ Mesh_1	<u>O</u> K <u>C</u> ancel		Help

- 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.

Object Browser		ð 🗙 🦷	VT	K scene	:1 - viev	wer:1							(
۲	Name		60 L		_	* 50		48	~				
🛨 😫 Geom	netry		S 🔊	12	<b>N</b> .	🗸 🚽	'~ 💎	~ 🌮	N)	<b>_</b> ~	Э	C »	
	rtex_1												
I → Ve	rtex_2		*		C	reate	Groups	from	Geom	etrv			
	rtex_3												
Image: A text and	rtex_4				<b></b>	-							_
Image: A text and	rtex_5		Mesh	<b>(</b>	Mesh_	_1							
Image: A text of the second	rtex_6				+-								
Image: A text of the second	rtex_7			Elemen	its								
@	ne_l			Goor	notn/	2	Inlet1						
@ ♪ / Lin	ne_2			Geor	neuy	¢.	Inlet2						
@ ♪ / Lin	ne_3						Outlet						
@ ♪ ∕ Ar	c_1						Wall						
Image: Image	re_l												
🐵 👂 🔲 Dis	sk_1												
🐵 👂 📕 Dis	sk_2												
🐵 D 💷 Pip	pe_1												
🐵 D 💷 Pip	be_2												
	De l												
	ripe_1												
	npe_z	- 12											
	Inlet2			Nodes									
	Outlot				-								_
	Wall			Geon	netry	~							
🗸 🖉 🖉	Wall					•							
V ₩ Mesh	notheses												
	porithms												
	esh 1												
* F	Pipe												
≥ \$	Applied hypothese	s											
⊳ ≿	Applied algorithms												
⊽ Gr	oups of Faces												
I ■	🔉 Inlet1												
I ■	lnlet2												
⊕ ♪	🔉 Outlet												
●	📚 Wall		Apply a	nd Clea		Ann	ly.		Close			Holp	
			Apply a	ind clos		Abb	iy		ciose			Help	
· ·			A <u>p</u> ply a	nd Clos	se	App	ly		C <u>l</u> ose			<u>H</u> elp	

• 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.

ج 🗨	Activate module	? 🛋 🗙
0	You're acti Please, select required actior	vating module <b>CFDSTUDY</b> . I by pressing the corresponding button below.
New	Open Load	<u>C</u> ancel

Click on "New". A dialog window displays information about Code\_Saturne installation.

	CFDSTUDY information	3
Select CFD	) code	
	Code Saturne	
	O NEPTUNE_CFD	
Informatio	ns	
version:	3.1.0-alpha	
prefix:	/local00/home/C61847/salome/neptune/arch/salome65ncfd21	
	ок	

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*.

▼ Location of CFDSTUDY study	? _ 🗆 X
Study location e/COURS/ENPC_2009_2010/TJUNG Study name PIPESTUDY	CTION Browse
Add new Cases	
ок	CANCEL

The new study directory with the new case is created with its sub directories and files. Move the mesh file *Pipe.med* in the directory 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"**:

🗞 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

I

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

	🖻 🔁 Calculation er	nvironment election ity criteria		
lesh import				
Import mes	hes 🔘 Use existing mes	sh input		
Local mesh di	rectory (optional)			
/MESH				ک 🖻
List of meshe	s			
File name	Format	Numbers	Reorient	Path
Pipe.med	MED	1		
	<b>+</b>	-		

#### Mesh quality criteria

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

E Calculat Mest	ion environment nes selection n quality criteria
Mesh quality criteria	
Post-processing for	mat EnSight Gold 🛛 😂
Options	
format	binary 😂
polygons	display 😂
polyhedra	display 😂
big-endian	
Ch	neck mesh

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

#### Density Ľ user law 0 Reference value ρ 998.0 kg/m³ Viscosity user law \$ 2 Reference value μ 0.001 Pa.s Specific heat constant \$ Cp 4181.0 Reference value J/kg/K Thermal conductivity user law \$ Reference value λ 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;

TempC^2;

User expression Predefined symbols Examples mu = [0.0015452 - 3.2212e-5 \* TempC + 2.45422 \* TempC^2;

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

User expression Predefined symbols Examples

 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.

	E Physical	properties ence values properties ty, hydrostatic p	
Gravity			
G	ravity along X	-9.81	m/s²
G	ravity along Y	0.0	m/s²
G	ravity along Z	0.0	m/s²

#### Initialization

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

<ul> <li>Volume conditions</li> <li>Volume regions definition</li> <li>Initialization</li> <li>Head losses</li> </ul>	
Initialization	
Volume zone all_cells	
Velocity	2
Thermal	2
Turbulence Initialization by reference value(s) <	2
Y Mathematical expression editor	×
User expression Predefined symbols Examples	
TempC = 19;	
<u>C</u> ancel	<u>о</u> к

#### **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.

	80	٢	Þ	P	JL.	2.	🚸 - 🔗	» 🥥 »	De	finition	ofbou	undary r	egions
1		-					-	lolo#1		Label	Zone	Nature	Selection criteria
								Inlet?		BC_1	1	Wall	Inlet1
										BC_2	2	Wall	Inlet2
								Wall		BC_3	3	Wall	Outlet
								$\mp$		BC_4	4	Wall	Wall
										Add fro In	m Prep	Add processo roups and	Delete Add from Salome r listing d 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.

C	Definition of boundary regions					
	Label	Zone	Nature	Selection criteria		
ľ	cold	1	Inlet	Inlet1		
ľ	hot	2	Inlet	Inlet2		
ľ	outlet	3	Outlet	Outlet		
ľ	wall	4	Wall	Wall		
			Add	Delete Add from Salome		
	Add from Prepocessor listing					
	Import groups and references from Preprocessor listing					

#### **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.



unuary conur	lions		
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		norma	al direction to the inlet 🗘 🛛 🖉
Turbulence	Calcula	ition by hydraul	ic diameter 🗘 🔣
	Hydra	aulic diameter	0.072 m
Thermal	ſ		
			rihad yalya

# Chapter 2. Tutorials

Bounda	ary condit	ions		
Labe		Zone	Nature	Selection criteria
cold		1	inlet	Inlet1
hot		2	inlet	Inlet2
outle	et	3	outlet	Outlet
wall		4	wall	Wall
Velo	ocity			
	norm		\$ 4.18	m/s 🛛 🖉
	Direction		norma	al direction to the inlet 🗘 🛛 🖉
Turb	ulence	Calcul	ation by hydraul	lic diameter 🗘 🕎
		Hydr	aulic diameter	0.072 m
The	rmal Ten	npC  🗢	Type Presc	ribed value

#### **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).



Solver	Scheme	Clipping			
	Name	Solver Choice	Maximum Iteration Number	Solver Precision	Time Step Factor
P	ressure	Multigrid	10000	1e-05	
V	/elocityX	Automatic	10000	1e-05	
V	/elocityY	Automatic	10000	1e-05	
V	/elocityZ	Automatic	10000	1e-05	
Т	urbEner	Automatic	10000	1e-05	
	Dissip	Automatic	10000	1e-05	
	TempC	Automatic	10000	1e-05	1

• 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*.

Solve	er Scheme	Clipping				
	Name	Scheme	Blending Factor	Slope Test	Flux Reconstructior	RHS Sweep Reconstructior
E F	Pressure					2
1	VelocityX	SOLU	1	$\checkmark$	$\checkmark$	1
1	VelocityY	SOLU	1	$\checkmark$	$\checkmark$	1
1	/elocityZ	SOLU	1	$\checkmark$	$\checkmark$	1
	TurbEner	Upwind	0		$\checkmark$	1
	Dissip	Upwind	0		$\checkmark$	1
	TempC	Upwind	0		$\checkmark$	1

#### Time step

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

<ul> <li>E Parameters</li> <li>Time step</li> <li>Equation parameters</li> <li>Global parameters</li> </ul>	
Time step option Constant	<b>\$</b>
Reference time step 0.0001	s
Number of iterations (restart included) 2000	

#### **Calculation control**

ė- 📄	Calculation control
	🗐 Time averages
	🗾 Output control
	Volume solution control
	💭 Profiles

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

Fr	requency	
	Output every 'n' time steps	\$ 20
	✓ Output at end of 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

**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.

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

Ou	Output Control Writer Mesh Monitoring Points					
<sub>E</sub> M	Monitoring points output					
	Monitoring	points files at each t	time step 💲 Fo	ormat .dat	÷	
					· )	
_ [ M	lonitoring point	s coordinates				
	n	Х	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		
	٥	0.135	0.113	n	•	
	□ Isplay monitoring points on SALOME VTK viewver					
	Probe's radius (m) 0.005					



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.

🖻 🚰 Calculation management
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

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 could be ommited). 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**". 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.