## 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

MARCH 2013

Code\_Saturne documentation

*Code\_Saturne* version 2.0 tutorial -Fluid-structure interaction

contact: saturne-support@edf.fr



## **Executive Summary**

The present document aims at describing the use of the main features of *Code\_Saturne* available for fluid-structure interaction calculations. The key points adressed here concern:

- performing a calculation on a mobile mesh using the ALE (Arbitrary Lagrangian-Eulerian) frame- work;
- how to parameter the mesh deformation and properly define the parameters of the boundary conditions for a mobile mesh;
- how to impose the mesh deformations or how to link a solid body in the fluid domain to a mass-spring system (with fluid-structure coupling);
- how to perform some basic post-processing related to such calculations.

The tutorial has been created based on *Code\_Saturne 2.0*. Adjustments might be needed for other versions.

## Sommaire / Summary

1	l Objectives					
2	Descri	ption of the test case	5			
	3.1	How to setup a case involving fluid-structure interactions (GUI)	7			
	3.2	How to setup a case involving fluid-structure interactions with user sub- routines	47			
	3.3	How to impose the displacement of the structure (GUI)	51			
	3.4	How to impose the displacement of the structure (user subroutines)				
	3.5	How to impose the velocity of the structure (GUI)				
	3.6	How to impose the velocity of the structure (user subroutines)				
	3.7	How to compute the force acting on the structure (usersubroutines)	67			
<ul> <li>2 Description of the test case</li></ul>						

EDF R&D	A tutorial on fluid-structure interaction in <i>Code_Saturne</i> 2.0	Code_Saturne Documentation Page 4/68
---------	--	--

## **1** Objectives

The present tutorial aims at describing the use of the main features of *Code\_Saturne* available for fluid- structure interaction calculations. The key points adressed here concern:

- performing a calculation on a mobile mesh using the ALE (Arbitrary Lagrangian-Eulerian) frame- work;
- how to parameter the mesh deformation and properly define the parameters of the boundary condi- tions for a mobile mesh;
- how to impose the mesh deformations or how to link a solid body in the fluid domain to a mass-spring system (with fluid-structure coupling);
- how to perform some basic post-processing related to such calculations.

The tutorial has been created based on *Code\_Saturne 2.0*. Adjustments might be needed for other versions.

### **2** Description of the test case

The present test case focuses on the numerical simulation of the transverse response of an elastically mounted cylinder subjected to vortex-induced vibrations (VIV). A sketch of the flow setup is provided in figure 1: the flow is uniform and the mechanical dynamics of the cylinder is modeled by a simple mass-spring system.

The physical parameters are chosen so that the system configuration may be simulated with a reasonable computational cost. The Reynolds number based on the cylinder diameter is taken to be 100 so that the flow is laminar and a 2D calculation may then be able to capture the whole features of the phenomenon. The parameters of the simulation are given throughout the tutorial but a sum-up is provided in table 1.

As shown in figure 2 the calculation domain is 2D with only one cell in spanwise direction.

Parameter	Description	
D	cylinder diameter	0.025 m
L	cylinder spanwise length	0.005 m
$U_\infty$	inflow velocity	$0.004 \ m.s^{-1}$
ρ	density	$1000 \text{ kg.m}^{-3}$
μ	dynamic viscosity	0.001 Pa.s
m	cylinder mass (kg)	see m*
k	cylinder stiffness (N.m <sup>-1</sup> )	see k*
$m * = \frac{m}{1/2\rho D^2 L}$	normalized mass	3.3
$\mathbf{K}^* = \frac{k}{1/2\rho U_{\infty}^2 L}$	normalized stiffness	12.02

Table 1: Simulation parameters of the vortex-induced vibration test case.





Bottom

Figure 2: Overview of the computational domain.



Figure 3: View of the mesh.

#### **3.1** How to setup a case involving fluid-structure interactions (GUI)

#### 3.1.1 Setting up and running the test case

The first step is to create the *Code\_Saturne* case. The following command creates a study directory, named Tuto\_VIV, and a new case, referred to as VIV :

```
code_saturne create -s Tuto_VIV -c VIV
```

It is then necessary to copy the mesh file mesh\_viv. des into the mesh directory of the study whose path is Tuto\_VIV/MESH (the mesh file is located into the tutorial directory Tutorial\_Files). Note that the mesh mesh\_viv. des is rather coarse so that it will lead to short calculations. Another mesh, referred to as mesh\_viv\_file. des, is also available. It will give more accurate results but will requier more computational time. One may first start with the coarse mesh to setup the test case and then switch to the fine one.

The graphical user interface (GUI) can now be started. Just go into the DATA directory of the case (with the path Tuto\_VIV/VIV/DATA/) and type in the command:

./SaturneGUI

in order to run the GUI. The next steps and commentaries are provided in figures 4 to 41.

#### 3.1.2 Post-processing

As concern calculations with a mobile mesh, basic flow visualization can be performed. As pointed out during the tutorial (in figure 37), as long as the outputs are based on a deformable mesh one may visualize the mesh deformations.

Some others valuable outputs related to the structure motions are also available in the working directory of the run. These data are copied into the RESU directory of the case, in the HIST directory corresponding to the run:

- go into the history directory of the run which should be Tuto\_VIV/VIV/RESU/HIST. 010100. The last eight digits corresponds to the date and time at which the calculation has started;
- a series of data files are available on the time history of the acceleration (str\_acceleration\_x. dat), velocity (str\_vitesse\_x. dat), displacement (str\_deplacement\_x. dat) of the structure and on the fluid forces acting on it (str\_force\_x. dat). Note that each direction, x, y and z, are treated in separated files;
- these data can easily be plotted using standard plotting software. The following command shows how to plot the displacement of the structure in the *y*-direction as a function of time, using the plotting tool Xmgrace:

xmgrace -block str\_deplacement\_y.dat -bxy 2:3

which plots the displacement in the *y*-direction as a function of time. The figure 42 illustrates the result that should be obtained. Remark that during the initialization period, for t < 500 s, the structure does not move. See figure 10 for the definition of the initialization period in the GUI.

	Code_Saturne GUI	_ o ×
Elle Tools Window Help		
I 🗖 🚰 🙆 🥵 📭 I 🗷		
New file		8
Study:		
Case:	Open a new calculation data file	
XML file:	open a new calculation data me	
1		
Open a new case.		



New parameter	s set • Code_Saturne	9 GUI	_ = ×
<u>File Tools Window H</u> elp			
- 🗇 🖄 🔕 📭 🖪			
New file			3
Stude Tido VIV			
Case: VIV			
XML file:			
Directory of the case			
Calculation environment	uments/CAC14/Tuto	riel_CS/Test_Tuto/Tuto_VIV/VIV	
Contract of the second se	the second second		
Physical properties     Associated sub-directorie	s of the case		
Volume conditions     Boundary conditions	Data	DATA	
Numerical parameters	Results	RESU	
E- Calculation control      Calculation management	l lear subroutines	SBC	
	Running scripts	SCRIPTS	
	Directory of meshes	MESH	
Once the update a name, the	new data file i few information case directory a	s created, the interfa is such as the study and sub-directories.	ace automatically name, the case
Open a new case.			

Figure 5: VIV test case. First steps of the case opening.

	New parameter	s set - Code_Saturne	a GUI	_ = ×
<u>File</u> Tools <u>Window</u> <u>Help</u>				
I 📑 🔛 🕭 🚯 I 🗷				
				(3 K)
Study Tuto VIV				
	and the second se			
	Right	t click and ex	(pand the treeview.	
XML file:				
0	/			
Identity - Expand	Directory of the case			7
Calculati     Collapse		uments/CAC14/Tuto	riel_CS/Test_Tuto/Tuto_VIV/VIV	
Additional scalars	Associated sub-directorie	s of the case		
Physical properties     Volume conditions		Data	DATA	
Boundary conditions     Dumerical parameters		Back		
Calculation control		Results	RESU	
Calculation management		User subroutines	SRC	
		Running scripts	SCRIPTS	
		Directory of meshes	MESH	

## Figure 6: VIV test case. First steps of the case opening.

			New parameters set - Code_Saturn	e GUI 💶 🗖	×
File Tool	s <u>W</u> indow <u>H</u> elp				
100	🖹 🕭 🔕 🃭 🖪 👘				
			1. Select «Meshe	s selection» in the treeview	X
Study	Tuto VIV				1
occuj.			/		4
Case:	VIV				1
XML file:		/			
	Ø	$/ \_$	souther		
	leptity and paths	Me	shes Periodic Boundaries		
6 6 C	alculation environment	PL.	st of meshes		
	Meshes selection		Meshes	Format	
	Mesh quality criteria		mask studes		
	Calculation features		mesn_viv.des	Simaii (NOPO) ".des"	
	Mobile mesh				
	Turbulence models		R I		
	Thermal model		A A A A A A A A A A A A A A A A A A A		
E- 6 A	dditional scalars		Λ.		
	Definition and initialization		1		
	Physicals properties		1		
E P	hysical properties		1		
	Reference values		1		
	Fluid properties		1		
	Gravity, hydrostatic pressure		1		
	Volume regions definition		\		
	Initialization			Vdd Delete	
	Head losses				
E B B	oundary conditions		Join mochor		
THE	Definition of boundary regi	-	Join mesnes		
	Boundary conditions		1		
E D N	umerical parameters	- D	Subdivide warped faces		
1 3++ L	Time step			2. Verity mesh selection	
	Equation parameters		Correct cell and face orientations	The list of all the meshes available in the	
	Global parameters		correct cell and face offentiations	directory MESH appears in this window.	
18-0 <u>6</u>	alculation control				
	Cuteut control			The mesh needed in the present case is	
	Volume solution control			named «mesh viv des» Any other mesh	
	Profiles			should be removed from the list using the	
0.000	alculation management			"Deleter butten	
1-6	User arrays			«Delete» button.	
-	Memory management				
6	Start/Restart				
	Prepare batch calculation				
4	110				1
alt d					
Click right	for context menu				

Figure 7: VIV test case. Meshes selection.

		New parameters set - Code_Saturne GUI	
File Tools	s <u>W</u> indow <u>H</u> elp		
1	🖹 🕭 🔕 📭 i 🗷		3
		1. Select «Mesh quality criteria» in the treeview	(10)
Study:	Tuto_VIV		-
Case:	VIV		=
XML file:			=
			=
-	0	Mesh quality criteria	
	ientity and paths alculation environment	Post-processing format EnSight Gold	
	Mesh quality criteria	Options	
	hermophysical models Calculation features	format binary 🗢	
	Turbulence models	polygons display 🗢	
	Thermal model	polyhedra display 🚖	
	dditional scalars	bio-endian	
	Physicals properties		
E D P	hysical properties	Interior faces selection for post-processing	
	Eluid properties		
	Gravity, hydrostatic pressure	Check mesh	
0 D V	blume conditions		-
	Volume regions definition	4	
	Initialization		
E B B	oundary conditions		
1 HG	Definition of boundary regi		
-6	Boundary conditions		
E D N	umerical parameters		
	Equation parameters		
	Global parameters	2. Click on «Check mesh»	
8 6 C	alculation control	This action launches Code Saturne	
	Time averages	preprocessor. In our case we will use the	
	Output control	preprocessor to get the group listing of the	
	Profiles	mesh in order to enforce the boundary	
e e c	alculation management	conditions later on.	
	User arrays		
	Memory management		
	Prepare batch calculation		
4	•		

Figure 8: VIV test case. Mesh quality criteria.



Figure 9: VIV test case. Mesh quality criteria.



Figure 10: VIV test case. Mobile mesh.



Figure 11: VIV test case. Mobile mesh.



Figure 12: VIV test case. Turbulence models.

	New parameters set - Code_Saturne GUI	
<u>He Tools Window Help</u>		
	1. Select «Fluid properties» in the treeview	ତ କ
Study: Tuto_VIV		
Case: VIV		
XML file:		
Identity and paths     Calculation environment     Meshes selection     Mesh quality criteria     Thermophysical models	Density constant Reference value p 1000 kg/m <sup>3</sup>	
Calculation features Mobile mesh Turbulence models Thermal model Additional scalars Definition and initialization Physicals properties	Viscosity constant Reference value  µ 0.001 Pa.s	
Privata properties     Privata properties     Privata properties     Fluid properties     Gravity, hydrostatic pressure     Volume conditions     Volume regions definition	Constant Reference value Cp 1017.24 J/kg/K	
2. Set density to 1000 k Water density.	<mark>g/m<sup>3</sup> / /</mark>	
3. Set viscosity to 0.001 The value of the viscosity is set w respect to the Reynolds number.	Pa.s /	
Calculation management Calculation management Calculation management Calculation management Calculation Memory management Calculation		
4		

Figure 13: VIV test case. Fluid properties.

		N	ew parameters set - Code_Sal	ume GUI		= = ×
File Tools	<u>W</u> indow <u>H</u> elp					
	🖢 🕭 🔕 📭 🖪					
						(B) (S)
Study	Tuto_VIV					
Case:	VIV	1	1. Select «Initial	zation» in the	treeview	
XML file:		- /				
	(a)		30001000			
1200022	191 1	elect vo	lume zone for initialization			
	entity and paths alculation environment	/	Ň	/olume zone all_cells 💠		
	Mesh quality criteria	- Antonitus				
B B Th	nermophysical models	/ Velocity i	nitialization			
	Calculation features	/	VitesseX	0.004	m/s	
	Turbulence models	/	VitesseY	0.0	mis	
	Thermal model					
	Definition and initialization		Vitessez	0.0	m/s	
	Physicals properties					
1 -0	Reference values			<b>∔</b>		
	Fluid properties					
E E V	olume conditions					
1 O	Volume regions definition					
	Initialization					
	Head losses					
	Definition of boundary regi.					
	Boundary conditions		1000			
	Fluid structure interaction		2. Set av	ial velocity to	0.004 m/s	
	umerical parameters					
	Equation parameters					
	Global parameters					
🗄 🗁 Ca	alculation control					
	Time averages					
	Volume solution control					
	Profiles					
🖻 🗁 Ca	alculation management					
- E	User arrays					
	Memory management					
	Prepare batch calculation					
4						

Figure 14: VIV test case. Initialization.

	New parameters set - Code_Saturne GUI	_ 0
le <u>T</u> aols <u>W</u> indow <u>H</u> elp		
🗂 📾 🕭 👩 📭 🗐		
		8
itudy Tuto VIV		
Case: VIV		
(MI file:		
2 10		
1 D Identify and asthe	Definition of boundary regions	3
Gentity and paths	Lobal Zana Natura	
Heshes selection	Laber Zone Wature Selection Chitema	_
Mesh quality criteria		
🖻 🛅 Thermophysical models		
Calculation features	1 Coloris Definition of home home sizes in	
Mobile mesh     Turbulence models	1. Select «Definition of boundary regions» in	
Thermal model	/ the treeview	
E B Additional scalars		
Definition and initialization		
Physicals properties		
Physical properties		
- Reference values		
Gravity, bydrostatic process		
B Volume conditions		
Volume regions definition	Add Delete	
Initialization	- Add from Preporessor listing	
Head losses		
🖻 🛅 Boundary conditions 🖉 🖉	Import groups and references from Preprocessor listing 🛛 🍯	
Definition of boundary regi		
Boundary conditions		
Pluid structure interaction	· · · · · · · · · · · · · · · · · · ·	
Time sten		
- A Equation parameters		
📑 Global parameters		
E 🛅 Calculation control		
🔝 Time averages		
🛃 Output control		
- 🛃 Volume solution control		
Profiles	/	
Calculation management		
Memony management	2 Click on «Import groups and »	
A Start/Restart	2. Click on «Import groups and»	
A Prepare batch calculation	Instead of manually entering every colors of the	
	mesh, the boundary regions can be directly taken	
	from the Preprocessor listing.	
•		

Figure 15: VIV test case. Definition of boundary regions.



Figure 16: VIV test case. Definition of boundary regions.

			tuto	_viv.xml -	Code_Saturne GUI	
File Tools	s <u>W</u> indow <u>H</u> elp					
-						(79, 67)
Study:	Tuto_VIV					
Case:	VIV					
XML file:	/home/berland/Documents/CAC14/Ti	utoriel_CS/1	Test_Tut	o/Tuto_VIV/V	//V/DATA/tuto_viv.xml	
	antha and eather	Definitio	n of bou	undary regio	ons	
	alculation environment	Label	Zone	Nature	Selection criteria	
I C	Meshes selection	BC 1	1	Wall	1	
	Mesh quality criteria	BC 2	2	Symmetry	2	
THC	Calculation features	BC 2	2	Inlot	- -	
	Mobile mesh Turbulence models	BC 4	4	Outlot	4	
	Thermal model	DC_4	4	Contractor	4	
	ditional scalars	BC_5	5	Symmetry	10	
	Physicals properties	BC_6	6	Symmetry		
E D P	hysical properties					
	Reference values					
	Gravity, hydrostatic pressure					
e e v	olume conditions				Add Delete	
	Initialization	Add fro	om Pren	ocessor list	tina	
	Head losses					
E B	oundary conditions			Imp	port groups and references from Preprocessor listing	
1	Boundary conditions	2				
	Fluid structure interaction					
	Time step	-				
	Equation parameters		Defi	ne the	e nature of boundary regions	
	Global parameters		The d	efault va	alue of this field is «Wall». You can change	
	Time averages		the va	alue by s	simply double-clicking on it. The nature of	
	Output control	1	the bo	oundary	conditions must be set according to the	
	Volume solution control		select	tion crite	eria (i.e. the «color» of the B.C.).	
B B C	alculation management		Costh		at case the patture of the D.C. must be	
+C	User arrays		Hor the	e preser	lows:	
	Memory management		uenne	Color	Nature	
	Prepare batch calculation			1	Wall	
				2	Symmetry	
				3	Inlet	
				4	Outlet	
				10	Symmetry	
				11	Symmetry	
4						

Figure 17: VIV test case. Definition of boundary regions.

		New	parameters :	set - Code_Satu	me GUI	= = ×
File Tool	s <u>W</u> indow <u>H</u> elp					
- HH -						(70 GR)
						(2) (3)
Study:	Tuto_VIV					
Case:	VIV					
VMI files	[					
Are the						
	0 8					
	entity and paths	Boundary co	onditions			
E C C	alculation environment	Label	Zone	Nature	Selection criteria	
I I In	Meshes selection	PC 1	1	wall	1	
1 4	Mesh quality criteria	BC 2	2	nan	2	
G-BT	hermophysical models	BC_2	2	symmetry	2	
1 4	Calculation features	BC_3	3	niet	3	
	Mobile mesh	BC_4	-	outlet	4	
	Turbulence models	80_5	5	symmetry	10	
1 14	Thermal model	ac_0	0	symmetry	11	
E- C A	dditional scalars					
1 1-1	Definition and initialization					
+	Physicals properties					
E P	hysical properties					
1 4 5	Reference values					
1.4	Fluid properties					
1.4.3	Gravity, hydrostatic pressure					
B.B.V	olume conditions					
1. 1. [	Volume regions definition					
1 1 1	Initialization					
	Head losses					
A DB	oundary conditions					
	Definition of boundary regi					
	Boundany conditions					
	Eluid structure interaction					
in the N	umerical parameters		2			
	Time step			100	1.11.1 I.	
	Equation parameters	1. C.	Select	«Bounda	ry conditions» in the treeview	
	Clabel exemptors		Once the	e boundary re	gions have been defined, the features of	the
	Global parameters		boundar	v conditions r	you be given by the user	1
E-D C	alculation control		Doundar	y conditions i	low be given by the user.	
	Time averages					
	output control					
1 1 1	Volume solution control					
	Profiles					
660	alculation management					
1.4	User arrays					
21	Memory management					
-	Start/Restart					
++	Prepare batch calculation					
•						

Figure 18: VIV test case. Boundary conditions.

		New	parameters :	set - Code_Satur	ne GUI	
File Tools	s <u>W</u> indow <u>H</u> elp					
1	🖻 🕭 🔕 🗈 🖪					
Study:	Tuto_VV	t «BC_1»	prresponds	to the cylinde	r wall	88
Case:		ary region co	responds	to the cynnic		
XML file:			1			
				2000.000 S		
	leatth: and notice	- Boundary co	nditions 🔶 🗕			
	alculation environment	Label	Zone	Nature	Selection criteria	
1-1-0	Meshes selection	BC 1	1	wall	16	
C	Mesh quality criteria	BC 2	2	symmetry	2	
0 D T	hermophysical models	BC_3	3	inlet	3	
	Calculation features	BC_4	4	outlet	4	
	Turbulance models	BC_5	5	symmetry	10	
	Thermal model	BC_6	6	symmetry	11	
	dditional scalars	1.				
	Definition and initialization					
	Physicals properties					
E B P	hysical properties					
	Beference values	- Smooth o	r rough wall			
	Fluid properties					
	Gravity, hydrostatic pressure			🖲 sm	ooth wall 🔘 rough wall	
6 BV	olume conditions					
I I I	Volume regions definition					
	initialization	Mobile me	sh			
	Head losses		100			
6 B B	oundary conditions			Interna	coupling 👌 🔍	
1 1-6	Definition of boundary regi			[meenie		
1 44	Boundary conditions				20	
	Fluid structure interaction					
E D N	lumerical parameters				1	
	Time step				1	
	Equation parameters				1	
	Global parameters				1	
E C C	alculation control				1	
	Time averages					
-	Output control			2 Select	Internal counling»	
	Volume solution control			This select of	anternar couping/	
1.0	Profiles			inis paramete	ers permits to define now the	
BBC	aiculation management			boundary con	ditions is treated by the ALE.	
1	a user arrays					
	a Memory management			Using «interna	al coupling», a mass-spring	
	Start/Restart			system is atta	ched to the boundary region	
	A Prepare batch calculation			The displacer	ant of the mosh for this P.C.	
				me uspiacen	the dial constant of the	
				will be given i	by the displacement of the	
				mass-spring s	ystem.	
4						
cinit	1.52					

Figure 19: VIV test case. Boundary conditions.

	New	parameters se	et - Code_Satur	me GUI	
File Tools Window Help					
i 🖂 🔛 🙆 🚺 🗷					
					2 8
Study Tto M	«BC_2»			10	
This boundar	y region co	orresponds t	o the upper	and	
Case: VIV lower sides of	the mesh	N2			
XML file:		<u>۱</u>			
		$\backslash$			
8 B	- Boundary co	nditions			
identity and paths		- \	Denter		
Meshes selection	Label	Zone	Nature	Selection criteria	
Mesh quality criteria	BC_1	1	wall	1	
🗄 🛅 Thermophysical models	BC 3	3	inlet	3	
Calculation features	BC 4	4	outlet	4	
Mobile mesh	BC_5	5	symmetry	10	
- D Thermal model	BC_6	6	symmetry	11	
E B Additional scalars	and a second sec				
E Definition and initialization					
Physicals properties					
🖻 🛅 Physical properties					
- D Reference values					
Gravicy, hydrostatic pressure					
Volume regions definition					
Initialization	Mobile me	esh			
Head losses			Fixed b	aundani 🔺 💷	
🖻 🛅 Boundary conditions			[Hked b		
Definition of boundary regi				1	
Boundary conditions				T	
E B Numerical parameters				1	
Time step				1	
- 🔂 Equation parameters				1	
- 🔂 Global parameters				1	
🖻 🛅 Calculation control				E.	
Time averages					
Output control		2. Sele	ct «Fixed	boundary»	
Profiles		The uppe	r and lowers	ides of the mesh are	
A B Calculation management		fixed from	tions discreti	and by a fixed mach	
H User arrays		fixed from	uers discreti	zed by a fixed mesh.	
- 🚺 Memory management		68 			
- 📑 Start/Restart					
- 📑 Prepare batch calculation					

Figure 20: VIV test case. Boundary conditions.

	New	parameters s	et - Code_Satu	me GUI	_ = ×
File Tools Window Help					
1 🥅 📾 🕭 👩 📭 1 🗟					
					(79.67)
					13 (2)
Study: Tuto_VIV					
Case: 1 Select #BC 3			6		
This boundary region corre	sponds to t	the inlet			
XML fi This boundary region corre	sponds to i	the milet.			
0.0	1				
But	- Boundary co	nditions			
Identity and paths     Calculation environment	Inhel	7000	Natura	Selection criteria	
Meshes selection	BC 1	1	wall	1	
Mesh quality criteria	BC_2	2	symmetry	2	
Calculation features	BC_3	3	inlet	3	
- 🖸 Mobile mesh	BC_4 BC_5	4	symmetry	4	
- ] Turbulence models	BC_6	6	symmetry	11	
E Additional scalars	10000				
Definition and initialization					
Physical properties					
Reference values	Velocity				
- D Fluid properties			24		
2 Cat in lat wal a site	1	nor	m	€ 0.004 m/s	
2. Set lillet velocity				Direction	
to 0.004 m/s			normal dir	ection to the inlet 🛊 🛛 🖓	
🗄 💼 Boundary conditions					
Definition of boundary regi	1000000000				
Boundary conditions	Mobile m	esh			
E D Numerical parameters			Fixed b	oundary 🛊 🕅	
🔂 Time step					
- Global parameters					
E Calculation control			/		
- 🔂 Time averages					
- A Volume solution control					
Profiles					
🖻 🛅 Calculation management		/			
- A Memory mana	enne Aerites				
- G Start/Restart 3. Select <	Fixed b	oundary	>>		
- A Prepare batch The inlet is a	fixed fronti	er discretiz	ed by a		
fixed mesh.					

Figure 21: VIV test case. Boundary conditions.

	New parameters set - Code_Saturne GUI	
<u>File Tools Window Help</u>		
🛅 🗁 🙆 🕒 🗷		
Study. Tuto_VIV This bound Case: VIV	<b>*BC_4</b> » ary region corresponds to the outlet.	
XML file:		
	Boundary conditions         Label       Zone       Nature       Selection criteria         BC_1       1       wall       1         BC_2       2       symmetry       2         BC_3       3       inlet       3         BC_4       4       outlet       4         BC_5       5       symmetry       10         BC_6       6       symmetry       11	
Fluid structure interaction     Numerical parameters     Global parameters     Global parameters     Calculation control     Output control     Volume solution	2. Select «Fixed boundary» The outlet is a fixed frontier discretized by a fixed mesh.	

Figure 22: VIV test case. Boundary conditions.

	New parameters set - Code_Saturne GUI	
<u>File T</u> ools <u>Wi</u> ndow <u>H</u> elp		
I 🗋 🖀 🕭 🚳 📴 🗷		
study: Tuto_VIV This bour	ct «BC_5» dary regions front side of the domain.	88
Case: VIV		
XML file:		
(B) (B)		_
Li D Identify and noths	Boundary conditions	
E- Calculation environment	Label Zone Nature Selection criteria	
Meshes selection		4
- 🗋 Mesh quality criteria	BC 2 2 Symmetry 2	
🗄 🗁 Thermophysical models	BC 3 3 inlet 3	
Calculation features	BC 4 4 outlet 4	
Mobile mesh	BC 5 5 symmetry 10	
Turbulence models	BC 6 6 symmetry 11	•
Thermal model		
E C Additional scalars		
Definition and initialization		
Physicals properties		
Beference values		
Eluid properties		
- C Gravity, bydrostatic pressure		
- D Volume conditions		
Volume regions definition		
	Mobile mesh	
Head losses		
Boundary conditions	Sliding boundary 💠 🛃	
Definition of boundary regi		
Boundary conditions	E   L <sup>**</sup>	
Fluid structure interaction		
🖻 🛅 Numerical parameters		
Time step	T T	
🕂 📑 Equation parameters	1	
- 📑 Global parameters		
🖻 🇁 Calculation control	1	
- 📑 Time averages		
🛃 Output control	2 Select Sliding houndary	
- 🛃 Volume solution control	2. Select «Sliding boundary»	
Profiles	The front side is a fixed frontier but the mesh	
E Calculation management	may be deformable on this boundary region.	
User arrays		
Memory management		
Branava batab calculation		

Figure 23: VIV test case. Boundary conditions.

		New parameters set - Code_Saturne GUI	
File Tools V	<u>V</u> indow <u>H</u> elp		
1	2 🔕 🗈 🗉		
Study: Tut	to_VV This bounda	<b>«BC_6»</b>	88
Case: VI	/	y regions back side of the domain.	
XML file:			
	0 0	annun :	
Li. D. Ident	ity and nathe	Boundary conditions	
E Calcu	ulation environment	Label Zone Nature Selection criteria	
I M	eshes selection		
М []	esh quality criteria	BC 2 2 symmetry 2	
🖻 🛅 Ther	mophysical models	BC 3 3 inlet 3	
-100	alculation features	BC_4 4 outlet 4	
	obile mesh	BC_5 5 symmetry 10	
	permai model	BC_6 6 symmetry 11	
E-D Addit	tional scalars		
	efinition and initialization		
- D PI	hysicals properties		
E E Physi	ical properties		
- Re	eference values		
- 🗋 FI	uid properties		
G G	ravity, hydrostatic pressure		
🖻 🗁 Volur	me conditions		
	olume regions definition	Mobile mesh	
	itialization		
	ead losses	Sliding boundary 🔹 🕺	
Boun	afinition of houndary ragi		
	oundary conditions		
- 13 FI	uid structure interaction		
E B Num	erical parameters	<u>†</u> 1	
I - D T	ime step	1	
🔒 E	quation parameters	1	
- 🔂 G	lobal parameters		
🖻 🗁 Calcu	ulation control	I I	
П 🔤 П	me averages		
- 🔂 o	utput control	2 Select «Sliding boundary»	
- 1 v	olume solution control	The back side is a fixed fraction but the mach	
- La Pi	rotiles	The back side is a fixed frontier but the mesh	
	liation management	may be deformable on this boundary region.	
	ser arrays		
	tart/Restart		
	renare batch calculation		
	The second s		
4			

Figure 24: VIV test case. Boundary conditions.

	New parameters set - Code_Saturne GUI	_ = ×
<u>File Tools Window Help</u>		
1 🗂 🗁 🕭 🔕 📭 🔍 👔		
	Select «Fluid structure interaction» in the treeview	ଜନ
	This interface permits to link the cylinder to a mass-spring system.	69 (8)
Study: Tuto_VIV	Since in the boundary conditions tab we have defined a wall B.C. with	
	the option «internal coupling», one structure, linked to BC_1, is visible	
Case: VIV	in the structure definition window.	
XML file:	The second secon	
	The parameters of the structure now need to be defined by the user.	
2 12		1
Identity and paths	Internal coupling with a simplified structure model External coupling with Code_Aster	
🖻 🛅 Calculation environment	- Internal coupling	
Meshes selection	1	
- 🗋 Mesh quality criteria	Maximum number of sub-iterations for implicit	
🖻 🛅 Thermophysical models	coupling with internal structures	
Calculation features	Belative precision for implicit coupling	
Mobile mesh	with internal structures 1e-05	
Turbulence models		
	Advanced options	
Additional scalars	Christian definition	
Physicals properties	Structures delinition	
P Physical properties	Fructure number Label Location	
Reference values		
- Fluid properties		
Gravity, hydrostatic pressure		
🖻 🛅 Volume conditions		
Volume regions definition		
- Initialization	Structure linked to BC_1	L
- Head losses		
Boundary conditions		
Boundary conditions		
Fluid structure interaction	Initial position	
🖻 🖻 Numerical parameters		
Time step	X m Y m Z m	
Equation parameters	Position of equilibrium	
- 📑 Global parameters		
🖻 🛅 Calculation control	X m Y m Z m	
- 📑 Time averages	Initial velocity	
- Output control		
- Rectiles	V <sub>X</sub> m/s V <sub>Y</sub> m/s V <sub>z</sub> m/s	
Calculation management	Caracteristics of the structure	
User arrays		
A Memory management	Mass matrix 🛛 🖓	
- A Start/Restart		
- 🔂 Prepare batch calculation	Damping matrix 🛛 🖓	
	Talle and and a local state	
	SUITINESS MALTIX	
	Force applied to the structure	
4		
10 L		
Click right for context menu		

Figure 25: VIV test case. Fluid structure interaction.

		tuto_viv.xml - Code_Saturne GUI	
File Tool	s Window Help	Set the sub-iterations number to 20 and the	
	🖉 🔝 🔯 🚺 🖾	precision to 10 <sup>-6</sup>	
		These parameters control the iterative process used to converge the	(X) (S)
		solution between the fluid and the structure.	
Study:	Tuto_VIV		
Case.	VIV	Increasing the maximum number of sub-iterations or decreasing the	
XML file:	/home/berland/Documents/0	precision increase the computational cost but in the other hand it may prevent the emergence of non-physical solutions.	
	(		
		Internal coupling with a simplified structure model. External coupling with Code Actes	
10	ientity and paths	Internal coupling with a simplified structure model External coupling with code_Aster	
	alculation environment	Internal coupling	
	Meshes selection		
	J Mesh quality criteria	Maximum number of sub-iterations for implicit	
	Colculation features	coupling with internal structures	
	Mobile mash	Relative precision for implicit coupling	
	Turbulence models	with internal structures 1e-08	
	Thermal model		
E-P-A	dditional scalars	Advanced options	
	Definition and initialization	Structures definition	
	Physicals properties		
B-B-P	hysical properties	Structure number Label Location	
THE	Reference values		
	Fluid properties	1 BC_1 1	the second s
	Gravity, hydrostatic pressur	e	
0 D V	blume conditions		
	Volume regions definition		
	) Initialization		
	Head losses		
🖻 🖻 B	oundary conditions		
	Definition of boundary regi.		
1 20	Boundary conditions	Telefol exclusion	
	Fluid structure interaction		
E D N	umerical parameters	X 0 m Y 0 m Z 0 r	n
	Fruction parameters		
	Global parameters	Position of equilibrium	
a.e.c	alculation control	X 0 m Y 0 m Z 0 r	n
	Time averages		Ma
	Output control	Initial velocity	
	Volume solution control		mie
	Profiles		mrs
E D C	alculation management	Caracteristics of the structure	
14	User arrays		
	Memory management	Mass matrix	
+	Start/Restart		
	Prepare batch calculation	Damping matrix	
		Stiffness matrix	
		Force applied to the structure	
		roice applied to the structure	
4	11		
finish			

Figure 26: VIV test case. Fluid structure interaction.



Figure 27: VIV test case. Fluid structure interaction.

		tuto_viv.xml - Code_Saturne GUI	
File Tools	<u>Window H</u> elp		
-			
= L W	= 🖾 🥯 📭 🗉		
			(S) (S)
Study	Tido VIV		
Study.	lato_ww		
Case:	VIV		
XML file:	/home/berland/Documents/CAC14/	Tutoriel CS/Test Tuto/Tuto VIV/VIV/DATA/tuto viv.xml	
0.000			
	0 8		
	entity and notice	Internal coupling with a simplified structure model External coupling with Code_Aster	
	alculation environment	Internal coupling	
	Meshes selection	- incernal coupling	
	Mesh quality criteria	Maximum number of sub-iterations for implicit	
B-B T	nermophysical models	coupling with internal structures 20	
	Calculation features		
	Mobile mesh	Relative precision for implicit coupling 1e-08	
-	Turbulence models	with internal structures	
	Thermal model	Advanced entions	
B-BA	dditional scalars		
	Definition and initialization	Structures definition	
	Physicals properties		
E D P	hysical properties	Structure number Label Location	
1 + 0	Reference values		
-	Fluid properties		
	Gravity, hydrostatic pressure		
0 D V	olume conditions		
	Volume regions definition		
	Initialization		
	Head losses	Click on the «Mass matrix» button	
🖻 🗁 B	oundary conditions	In order to define the mass matrix using the formula edit	or.
	Definition of boundary regi		
	Boundary conditions		
- 14 H	Fluid structure interaction	Initial position	
E D N	umerical parameters		
	Time step		
-	Equation parameters	Position of equilibrium	
+	Global parameters		
⊡ 🗁 C	alculation control	x 0 m y 0 m z 0 m	
1 4	Time averages	Initial velocity	
	Output control		
	Volume solution control	V <sub>x</sub> 0 m/s V <sub>y</sub> 0 m/s V <sub>y</sub> 0 m/s	
	Profiles		
BBC	alculation management	Caracteristics of the structure	
1	User arrays	Mass matrix	
	Memory management		
	Start/Restart	Damping matrix	
100	Prepare batch calculation	Damping matrix	
		Stillness matrix	
		Force applied to the structure	
		· · · · · · · · · · · · · · · · · · ·	
4			
finish	1.1		22

Figure 28: VIV test case. Fluid structure interaction.

	New parameters set - Code_Saturne GUI	
Tools Window Help		
] 🔤 🕭 🚳 🃭 🔍		
		ß
dy: Tuto_VIV		
e: VIV		
1 #Ita		
Ør	Mathematical expression editor	
Identity and naths	User expression Predefined symbols Examples Ith Code_Aster	
Calculation environment	petar = 3.3	
Meshes selection	E3(4) - 3.3;	
Mesh quality criteria	zho = 1000.;	
Thermophysical models	d = 0.025; 1	
Calculation features	L = 0.005;	
Mobile mesh	n = nstar*0.5*rho*d*2*L; le-05	
- U Turbulence models		
	n11-1.;	
Additional scalars	m22=1; m33=1.*	
Definition and initialization	n12=0;	
Physical properties	m13-0;	
Reference values	m23-0;	
- C1 Eluid properties	m21-0;	
Gravity, hydrostatic pressure	m31=0;	
Volume conditions	m32=0;	
Volume regions definition		
Initialization		
- Head losses		
Boundary conditions		
- Definition of boundary regi	A	
- 🗋 Boundary conditions		
👾 📓 Fluid structure interaction	A A A A A A A A A A A A A A A A A A A	
Numerical parameters		m
- 📑 Time step		
Equation parameters	K Cancel Cancel	
- 🔝 Global parameters		
Calculation control		m
Ime averages	Initial velocity	
Utput control		in the second
Profiles	$V_X   0 $ $m/s V_Y   0 $ $m/s V_Z   0$	m/s
Calculation management	Caracteristics of the structure	
Liser arrays		
A Memory management	Fill in the editor with the following t	formula
- A Start/Restart		ormana
- A Prepare batch calculation	for the mass matrix	
	There is no motion of the cylinder in the x and z o	direction.
	The mass is nonetheless still defined for those dir	rections
	but the force acting on the cylinder following x ar	nd z will be
	set to zero later on	IC 2 WIII DC
	Set to zero later on.	
	1.46	

Figure 29: VIV test case. Fluid structure interaction.

		tuto_viv.xml - Code_Saturne GUI	
File Tools	s <u>W</u> indow <u>H</u> elp		
	🖥 🧥 👩 📭 🗐		
			C70 (62)
			69.69
Study	Tuto_VIV		
Case:	VIV		
VMI files	/home/horland/Documents/CAC140	Starial CETTart TitaTita \@/ATI/DATAButa \www.	
AME HIG:	/none/benand/bocuments/cacia/		
	0 8		
	entity and paths	Internal coupling with a simplified structure model External coupling with Code_Aster	11
D D C	alculation environment	- Internal coupling	
	Meshes selection		
A B T	armonbusical models	Maximum number of sub-iterations for implicit	
1 1 0	Calculation features	coupling with internal structures	
	Mobile mesh	Relative precision for implicit coupling	
	Turbulence models	with internal structures	
	Thermal model		
E P A	ditional scalars	Advanced options	
	Definition and initialization	Structures definition	
	Physicals properties		
P-PP	hysical properties	Structure number Label Location	
THO	Reference values		
	Fluid properties	1 8C_1 1	
	Gravity, hydrostatic pressure		
B B V	olume conditions		
1 4 1	Volume regions definition		
	Initialization		
	Head losses		
6 B 8	oundary conditions		
	Definition of boundary regi		
	Boundary conditions	Click on the «Damping matrix» button	
- 14 H	Fluid structure interaction	In order to define the damping matrix using the formula	a
E D N	umerical parameters	editor	
	] Time step	Calcol.	111
	Equation parameters	Position of equilibrium	
+	Global parameters		
B B C	alculation control		
	Time averages	Initial velocity	
	Output control		
	Volume solution control	V <sub>x</sub> 0 m/s V <sub>y</sub> 0 m/s V <sub>z</sub> 0 m/s	
	Profiles	Constantially states at mature	
	alculation management	Caracteristics of the structure	
	User arrays	Mass matrix	
	Stort Postort		
	Branara batch calculation	Damping matrix	
	repare bacch calculation		
		Stiffness matrix	
		Force applied to the structure	
4			

Figure 30: VIV test case. Fluid structure interaction.



Figure 31: VIV test case. Fluid structure interaction.

		tuto_viv.xml - Code_Saturne GUI	
File Tools	s <u>W</u> indow <u>H</u> elp		
	🖥 🧥 👩 📭 🗐		
- L.			CT (T)
			125 (25)
Study:	Tuto_VIV		
Case:	VIV		
cosci			
XML file:	/home/berland/Documents/CAC14/	Tutoriel_CS/Test_Tuto/Tuto_VIV/VIV/DATA/tuto_viv.xml	
	(R)	2000000 S	
		Internal coupling with a simplified structure model External coupling with Code Aster	
10	entity and paths	Internal equiling	
	Meshes selection	- Internal coupling	
	Mesh quality criteria	Maximum number of sub-iterations for implicit	
B-B T	nermophysical models	coupling with internal structures	
	Calculation features		
-	Mobile mesh	Relative precision for implicit coupling 1e-08	
	Turbulence models	with internal structures	
	Thermal model	Advanced entions	
E-BA	dditional scalars		
	Definition and initialization	Structures definition	
	Physicals properties		
E D P	hysical properties	Structure number Label Location	
	Reference values	RC 10 1 Million and an and	
	Fluid properties		
	Gravity, hydrostatic pressure		
ė 🖨 V	olume conditions		
	Volume regions definition		
	Initialization		
	Head losses		
🖻 🗁 B	oundary conditions		
1 1 1 1 1	Definition of boundary regi		
	Boundary conditions		
11.44	Fluid structure interaction	Click on the «Stiffness matrix» button	
E D N	umerical parameters	X D In order to define the stiffness matrix using the formul	
	) Time step	in order to denne the sumess matrix using the formula	a
1 -	Equation parameters	editor.	
+	Global parameters		
D D D	alculation control	x o m y o m z o m	
	Time averages	Initial velocity	
	Output control		
	Volume solution control	Vx 0 m/s Vy 0 m/s Vz 0 m/s	
	Profiles		
	alculation management	Caracteristics of the structure	
	User arrays	Mass matrix	
	Memory management		
	Start/Restart	Damping matrix	
100	Prepare batch calculation	Damping matrix 2	
		Chilfmans matrix	
		SUITIESS THALTK	
		Force applied to the structure	
		2	
4			
1 - MAR			

Figure 32: VIV test case. Fluid structure interaction.



Figure 33: VIV test case. Fluid structure interaction.

e Tools Window Help	21		viv.xml • Code_Saturne GU	tuto_viv		
With the solution of the s					<u>Window H</u> elp	Tools
udy:       Tuto_VIV         see       VV         utline       Internal coupling with a simplified structure model         External coupling with a simplified structure model       External coupling with Code_Aster         Internal coupling       Internal coupling         Maximum number of sub-terations for implicit       20         Calculation newtronment       Maximum number of sub-terations for implicit       20         Calculation textures       20         Calculation textures       20         Calculation textures       20         Calculation textures       20         Definition and initialization       Physical properties         Fluid structure regions definition       Structures definition         Physical properties       Structure number Label       Location         Structure number Label       Location         Mumerical parameters       0       m         Outing v conditions       Wolf m       m         Definition of boundary regiment       No m       0       m         Structure solution       m       m       m       m         Boundary conditions       Wolf m       2       m       m         Calculation parameters       Global parameters       m       m </th <th></th> <th></th> <th></th> <th></th> <th>🖥 🖎 🔕 🃭 🖪</th> <th>6</th>					🖥 🖎 🔕 🃭 🖪	6
udy       Luto_VIV         see:       V/         At file:       (home/berland/Documents/CAC14/Tutoriel_CS/Test_Tuto/Tuto_VIV/V/DATAtuto_viv.xml)         Image: transmitted in the internal structure internal coupling with a simplified structure model       External coupling with a simplified structure model         Image: transmitter internal coupling with a simplified structure model       External coupling with a simplified structures         Image: transmitter internal coupling with a simplified structures       20         Image: transmitter internal coupling with a simplified structures       20         Image: transmitter internal coupling with a simplified structures       20         Image: transmitter internal structures       1         Image: transmitter internal structures       1         Image: transmotes       Struc						
uoys luto_vvv  se: Vv  t. tile: (home/berland/Documents/CAC14/Tutoriel_CS/Test_Tuto/Tuto_VV/VV/DATAtuto_vixxml  Calculation environment Calculation and initialization Physical properties Physical properties Calculation engions definition Calculation of bounday regu. Boundary conditions Calculation environment Calculation control Calculation control Calculation control Calculation management Calculation management Calculation environment Calculation environm					massar	
ise:       V/         4L file:       /home/berland/Documents/CAC14/Tutoriel_CS/Test_Tuto/Tuto_V/V/V/DATAtuto_v/wml         Image: the image:					luto_viv	uay:
At. Hie: /home/berland/Documents/CAC14/Tutoriel_CS/Test_Tuto/Tuto_VMV/MIATA/tuto_VMVV/MIATA/tuto_MIATA/tuto_MIATA/tuto_VMV/MIATA/tuto_VMV/MIATA/tuto_VM					VIV	se:
Identity and paths   Calculation environment   Meshes selection   Meshes selection   Meshes selection   Maximum number of sub-iterations for implicit   Calculation features   Mobile mesh   Thermal model   Additional scalars   Definition and initialization   Physicals properties   Physicals properties   Physicals properties   Public regions definition   Internal coupling with a simplified structures   Additional scalars   Definition and initialization   Physicals properties   Physicals properties   Position of boundary regluine   Boundary conditions   Definition in theat coupling with a simplified structure number i Label   Location   Internal coupling with internal structures   Position of boundary regluine   Boundary conditions   Definition of boundary regluine   Boundary conditions   Definition of boundary regluine   Boundary conditions   Definition of boundary regluine   Boundary conditions   Calculation control   Position of equilibrium   X 0 m Y 0 m Z 0 m   Numerical parameters   Calculation control   Position of equilibrium   X 0 m Y 0 m Z 0 m   Calculation management   Volume solution control   Position of equilibrium   X 0 m Y 0 m Z 0 m   Structure services of the structure   Mass matrix			Tuto_VIV/VIV/DATA/tuto_viv.xml	//Tutoriel_CS/Test_Tuto/Tut	/home/berland/Documents/CAC14	4L file:
Identity and paths   Calculation environment   Meshes selection   Meshes selection   Meshes selection   Meshed wality citeria   Calculation features   Mobile mesh   Thermophysical models   Calculation scalars   Definition and initialization   Physical properties   Physical properties   Physical properties   Calculation features   Additional scalars   Definition and initialization   Physical properties   Structure number Label   Location   Peristion of boundary regl   Boundary conditions   Definition of boundary regl   Boundary conditions   Definition of requilibrium   Calculation control   Profiles   Calculation control   Profiles   Calculation management   Vie O   Memor			2000.000 :			
Internal coupling with a singulated discrete models Internal coupling with internal structures Relative precision for implicit Calculation features Relative precision for implicit Calculation as calars Physical properties Reference values Fluid properties Boundary conditions Initial position Initial position Initial position Initial position X 0 m Y 0 m Z 0 m Initial velocity Vs 0 m/s Vr 0 m/s Vr 0 m/s Vr 0 m/s Vr 0 m/s Vs 0 m/s Vr 0 m/s Vr 0 m/s Vr 0 m/s Vr 0 m/s Initial velocity Vs 0 m/s Vr 0 m/s Vr 0 m/s Vr 0 m/s Memory management Memory management Memory management	n -	External coupling with Code Actor	with a simplified structure more	Internal coupling wi	() ()	
Internal coupling   Internal coupling      Internal coupling Internal coupling Internal coupling Internal coupling Internal coupling Internal scalars Calculation features Mobile mesh Thermodels Additional scalars Definition and initialization Physical properties Fluid sproperties Fluid sproperties Fluid structure interaction Initial position Initial position X 0 m Y 0 m Z 0 m Position of equilibrium X 0 m Y 0 m Z 0 m Initial velocity Volume solution control Providue sarays Output control Volume solution control Profiles Calculation management Use management Internal coupling		External coupling with code_Aster	with a simplified structure more	incental coupling wi	entity and paths	D Id
Meshes subtiction   Meshes subtiction   Meshes subtiction   Meshes subtiction   Mobile mesh   Calculation features   Mobile mesh   Multiculation features			g	Internal coupling	Alculation environment	D Ca
Thermophysical models 20   Calculation features 20   Mobile mesh 120   Multiple mesh 120   Mul				Lange to the	Mesh quality criteria	H
Calculation features   Mobile mesh   Multiplence models   Thermal model   Additional scalars   Definition and initialization   Physical properties   Physical properties   Physical properties   Fluid properties   Gravity, hydrostatic pressure   Volume conditions   Definition of boundary regi   Boundary conditions   Values explores   Initial position   X 0 m Y 0 m Z 0 m   Y 0 m/s Vx 0 m/s Vx 0 m/s Vx 0 m/s Vx 0 m/s   Value solution control   Profiles   Calculation management   Defined to formation   Memory management   Defined to formation		20	um number of sub-iterations fo	Maximum	nermophysical models	D Th
Mobile mesh 1=-08   Turbulence models additional scalars   Additional scalars Advanced options   Physicals properties Structures definition   Physical properties Structure number Label   Caracteristics of the structure   Mumerical parameters   Global parameters   Global parameters   Global parameters   Global parameters   Output control   Vulue solution control   Vitue solution control   Wemory management		L	ig with internal structures	coupling	Calculation features	-p
Turbulence models Turbulence models Thermal model Additional scalars Oblightion and initialization Physicals properties Physicals properties Physicals properties Fluid properties Gravity, hydrostatic pressure Volume regions definition Initialization Head losses Boundary conditions Boundary conditions Fluid structure interaction Numerical parameters Global parameters Galculation control Volume solution control Wemory management Galculation management Memory management<	08	10.08	e precision for implicit coupling	Relative p	Mobile mesh	-6
Additional scalars   Additional scalars   Physicals properties   Physical properties   Physical properties   Physical properties   Structure number Label   Location   Perference values   Structure number Label   Location   Parameters   Volume regions definition   Initial position   Numerical parameters   Boundary conditions   Boundary conditions   Boundary conditions   Parameters   Calculation parameters   Calculation control   Value solution control   Wemory management   User arrays   Memory management		16-00	ternal structures	with inter	Turbulence models	-n
Additional scalars  Definition and initialization Physical properties Fixid properties Fixid properties Gravity, hydrostatic pressure Volume conditions Fixid structure interaction Head losses Boundary conditions Fixid structure interaction Fixid structur	ก	20	and entione	Advances	Thermal model	-6
Definition and initialization   Physicals properties   Physicals properties   Gravity, hydrostatic pressure   Volume regions definition   Initialization   Head losses   Boundary conditions   Definition of boundary regi   Boundary conditions   Definition of boundary regi   Boundary conditions   Calculation parameters   Global parameters   Global parameters   Global parameters   Calculation control   Profiles   Calculation management   User arrays   Memory management			ced options	Advanced	dditional scalars	P Ac
Physicals properties Physical properties Reference values Fluid properties Gravity, hydrostatic pressure Volume conditions Volume regions definition Initialization Head losses Boundary conditions Boundary conditions Boundary conditions Boundary conditions Boundary conditions Boundary conditions Calculation parameters Global parameters Global parameters Calculation control Profiles Calculation management Volume solution control Profiles Calculation management Calculation management Memory management Memory management Calculation boundary control Value solution control Memory management Calculation management Memory management Calculation boundary control Mass matrix			tures definition	Structur	Definition and initialization	-0
Physical properties   Reference values   Fluid properties   Gravity, hydrostatic pressure   Volume conditions   Volume conditions   Numerical parameters   Boundary conditions   Fluid structure interaction   Numerical parameters   Global parameters   Global parameters   Gaculation control   Time averages   Output control   Volume solution control   Profiles   Calculation management   User arrays   Memory management					Physicals properties	-0
Reference values Fluid properties Gravity, hydrostatic pressure Volume conditions Volume regions definition Initial ration Head losses Boundary conditions Boundary conditions Boundary conditions Head losses Boundary conditions Head losses Boundary conditions Fuid structure interaction Numerical parameters Filid structure interaction Numerical parameters Global parameters Global parameters Galculation control Time averages Output control Volume solution control Profiles Calculation management Calculation management Memory management Memory management Experiment Mass matrix		Location	ucture number Label	Struc	hysical properties	Ph Ph
Fluid properties Gravity, hydrostatic pressure Volume regions definition Initialization Head losses Boundary conditions Boundary conditions Fluid structure interaction Numerical parameters Global parameters Global parameters Calculation control Volume solution control Profiles Calculation management Calculation manag					Reference values	-D
Gravity, hydrostatic pressure     Volume conditions     Volume regions definition     Initialization     Head losses     Boundary conditions     Boundary conditions     Definition of boundary regi     Boundary conditions     Initial position     Numerical parameters     Global	ann an		BC_1 1	1	Fluid properties	-0
<ul> <li>Volume conditions</li> <li>Volume regions definition</li> <li>Initialization</li> <li>Head losses</li> <li>Boundary conditions</li> <li>Definition of boundary regi</li> <li>Boundary conditions</li> <li>Fluid structure interaction</li> <li>Numerical parameters</li> <li>Fluid structure interaction</li> <li>Numerical parameters</li> <li>Global parameters</li> <li>Global parameters</li> <li>Calculation control</li> <li>Time averages</li> <li>Output control</li> <li>Volume solution control</li> <li>Profiles</li> <li>Calculation management</li> <li>Global paramagement</li> <li>Memory management</li> <li>Mass matrix</li> </ul>					Gravity, hydrostatic pressure	-0
<ul> <li>Volume regions definition initialization Head losses Boundary conditions Definition of boundary regi Boundary conditions Huid structure interaction Numerical parameters Calculation control Calculation control Volume solution control Value solution control Value solution control Memory management Mass matrix</li> </ul>					olume conditions	D Vo
Initialization Head losses          Boundary conditions       Initial position         Definition of boundary regi       Boundary conditions         Imitial parameters       Initial position         Numerical parameters       Position of equilibrium         Equation parameters       Position of equilibrium         Calculation control       X 0 m Y 0 m Z 0 m         Output control       X 0 m Y 0 m Z 0 m         Output control       Initial velocity         Volume solution control       Vx 0 m/s Vr 0 m/s V2 0 m/s         Profiles       Caracteristics of the structure         User arrays       Memory management         Memory management       Mass matrix					Volume regions definition	
<ul> <li>Head losses</li> <li>Boundary conditions</li> <li>Definition of boundary regi</li> <li>Boundary conditions</li> <li>Initial position</li> <li>Initial position</li> <li>Initial position</li> <li>Numerical parameters</li> <li>Equation parameters</li> <li>Global parameters</li> <li>Global parameters</li> <li>Calculation control</li> <li>Time averages</li> <li>Output control</li> <li>Volume solution control</li> <li>Profiles</li> <li>Calculation management</li> <li>Calculation management</li> <li>Calculation management</li> <li>Calculation management</li> </ul>					Initialization	- 0
Boundary conditions   Definition of boundary regi   Boundary conditions   Boundary conditions   Initial position   Numerical parameters   Image: Sequence of the structure interaction   Soloal parameters   Soloal parameters   Global parameters   Calculation control   Image: Sequence of the structure   Volume solution control   Volume solution control   Volume solution control   Value solution control   Calculation management   Calculation management   Memory management   Memory management					Head losses	- 0
Definition of boundary regi     Boundary conditions     Boundary conditions     Initial position     Initial position     Initial position     Initial position     X 0 m Y 0 m Z 0 m      Position of equilibrium     X 0 m Y 0 m Z 0 m      Initial velocity     Volume solution control     Volume solution management     Galculation management     Memory management					oundary conditions	B BC
Boundary conditions     Fluid structure interaction     Numerical parameters     Global parameters     Global parameters     Global parameters     Calculation control     Vulture solution control     Vulture solution control     Volume solution cont					Definition of boundary regi	-0
Fluid structure interaction       Initial position         Numerical parameters       Initial position         Equation parameters       Position of equilibrium         Global parameters       Position of equilibrium         Calculation control       X       0       m       Y       0       m       Z       0       m         Output control       Imitial velocity       Imitial velocity       Imitial velocity       Imitial velocity       Imitial velocity         Volume solution control       Vx       0       m/s       Vr       0       m/s       V2       0       m/s         Calculation management       User arrays       Mass matrix       Imitial position					Boundary conditions	
Numerical parameters       X       0       m       Y       0       m       Z       0       m         Image: Sequence       Global parameters       X       0       m       Y       0       m       Z       0       m         Global parameters       Calculation control       X       0       m       Y       0       m       Z       0       m         Time averages       Initial velocity       Initial velocity       Initial velocity       0       m/s       Vz       0       m		tial position			Fluid structure interaction	- Hi 📑
Time step     Equation parameters     Global parameters     Calculation control     Volume solution control     Volume solution control     Vrx     O     m/s     Vry     O     m/s     Vry     O     m/s     Vry     O     m/s     Vry     Mass matrix	m	m 7 0	X 0 m V	×	umerical parameters	D No
Equation parameters     Global parameters     Calculation control     Vx     O     m     Vx     O     m/s     m/s     O     m/s     O     m/s     Mass matrix     Mass     matrix					Time step	
Global parameters     Global parameters     Galculation control     Galculation control     Output control     Output control     Profiles     Galculation management     Galculation management     Galculation management     Mass matrix		on of equilibrium	Pos		Equation parameters	
Calculation control     X 0     m Y 0     m Z 0     m       Time averages     Initial velocity       Output control     Vx 0     m/s Vr 0     m/s Vz 0       Profiles     Caracteristics of the structure       Calculation management     Mass matrix	<b>n</b>				Global parameters	
Time averages     Output control     Output control     Volume solution control     Volume solution control     Vx     O     m/s     Vr     O     m/s     Vr     O     m/s     Vz     O     m/s     Vz     Mass matrix	m	m Z 0	x o m y	×	alculation control	Ca
Output control     Volume solution control     Vx     O     M/s     Mass matrix     O		itial velocity			Time averages	
Volume solution control     Vx     0     m/s     Vr     0     m/s     Vz     0     m/s       Profiles     Calculation management     Caracteristics of the structure       User arrays     Mass matrix     Image: Caracteristics of the structure		icial verocity			Output control	
Profiles     Calculation management     User arrays     Memory management     Mass matrix	m/s	m/s ∨ <sub>z</sub> 0	0 m/s V <sub>Y</sub> 0	V <sub>x</sub> 0	Volume solution control	U U
Calculation management Caracteristics of the structure User arrays Memory management Mass matrix					Profiles	
User arrays Mass matrix		stics of the structure	Caract		alculation management	D Ca
Memory management		atriv 🔊	Mae		User arrays	
1 Etact Dectact			Plas		Memory management	
Damping matrix		a matrix 🔊	Dem		Start/Restart	-U
Prepare batch calculation		g matrix	Dan		Prepare batch calculation	
Stimess matrix		s macrix	Stiff	a number of	Silah an Alea France	
Click on the «Force applied Force applied to the structure		lied to the structure	Force	applied	lick on the «Force	C C
to the structure» button				outton	o the structure» b	t
		2		and the second		1

Figure 34: VIV test case. Fluid structure interaction.



Figure 35: VIV test case. Fluid structure interaction.

		New parameters set - Code_Saturne GUI	
File Tools	s <u>Window H</u> elp		
	🖢 🖎 🔕 📭 🗷 🚺 1. Sele	ct «Time step» in the treeview	
			8
Study:	Tuto_VIV		
Case:	VIV		
XML file:			
	Ø 8		
1010	lentity and paths	eady flow algorithm management	7
E E C	alculation environment	Time step option Uniform and constant	
	Mesh quality criteria	Defense the star	
B B T	hermophysical models	Vereience une step	
	Mobile mesh	Number of iterations (restart included) 15000	
	Turbulence models		
	) Thermal model		
	Definition and initialization	Option zero time step	
1 - 0	Physicals properties	1	
E D P	hysical properties	/	
	Fluid properties	/	
	Gravity, hydrostatic pressure	/	
0 0 V	blume conditions	/	
	Initialization	/	
	Head losses	/	
E B B	oundary conditions	2. Set time step to 0.1 s and number of	
	Boundary conditions	2. Set time step to 0.1 s and number of	
	Fluid structure interaction	iterations to 15,000	
B-B N	umerical parameters		
	Equation parameters		
	Global parameters		
E D C	alculation control		
	Time averages		
	Volume solution control		
G	Profiles		
E E C	alculation management		
-	Memory management		
- FG	Start/Restart		
	Prepare batch calculation		
<u></u>			

Figure 36: VIV test case. Time step.

		New parameters	set - Code_S	aturne GUI 💶 🗶
Elle Tool:		ect «Output cont	rol» in	the treeview
	🖥 🖭 🔕 🚺 💶 Sei	ect «Output cont	roi» ili	
		1		8 K
Study	Tito VIV	/		
orday.		/		2 Choose «Post-processing every »
Case:	VIV	/		and set the period to 500 iterations
XML file:				· · · · · · · · · · · · · · · · · · ·
	20			
	fentity and paths	Output Control Monitorin	g Points Coord	linates
E C C	alculation environment	Outputs listings		
	Meshes selection		Output listin	n at each time sten
	] Mesh quality criteria		- output institu	
	Calculation features	Pest-processing		
	Mobile mesh	1 Processing		
	Turbulence models		Post-proces	sing every 'n' time steps 💠 500
C	) Thermal model	1		
E D A	dditional scalars	1	Fluid	domain post processing 👻
	Physicals properties	/	Doma	ain boundary post processing 🗌
B B P	hysical properties	/		
TE	Reference values	Туре о	f post-processi	ing for mesh deformable (with displacement)
	Fluid properties	Post-o	rocessing form	sat Sight Gold
	Gravity, hydrostatic pressure	1	locossing form	
	olume conditions	Options		
	j volume regions definition		4	ormat Innan
	Head losses		S.	ormat and a second seco
B B B	oundary conditions		F	olygons display 🗢
1 10	Definition of boundary regi.			
	Boundary conditions		F	oolyhedra display
	Fluid structure interaction		b	oig-endian
	Time step			
	Equation parameters			
	Global parameters			
Ė € C	alculation control			1
	Time averages			
	Output control			1
	Profiles			/
E B C	alculation management			,
	User arrays		12.22	
-6	A Memory management	3. Choose «defo	rmable	(with displacement)»
	Start/Restart	This option must be ac	tivated to	see the mesh deformation
	Prepare batch calculation	during the post-proces	sing.	
	L			
4				
Canal	1.11			

Figure 37: VIV test case. Output control.

		New parameters set - Code_Saturne GUI	_ = ×
Elle Tool	s <u>W</u> indow <u>H</u> elp		
	🖶 🕭 🙆 📭 🔍		
	1 6	elect "Benare batch " in the treeview	20
	I <sup>1.3</sup>	elect «repare batch» In the treeview	6.6
Study	Tuto_VIV		
Case:			
XML file:			
	2 2		
	dentity and naths	Computer selection	
E-6 9	alculation environment	Workstation	
	Mesh quality criteria	Select the batch script file	
	Calculation features	Prenare batch calculation	
	Turbulence models		
	Thermal model	Number of processors	
	Definition and initialization	User files	
E-B F	Physicals properties	Advanced options	
ŢŦŗ	Reference values		
	Gravity, hydrostatic pressure	Code_Saturne	
e e V	olume conditions	running	
	Initialization		
	) Head losses	2. Set the number	
	Definition of boundary regi	of processors you	
	Boundary conditions	want to use	
e e h	Iumerical parameters		
	Time step		
	Global parameters		
	alculation control		
	Output control		
	Volume solution control		
	alculation management		
- F	User arrays		
	Start/Restart		
144	Prepare batch calculation		
<u></u>			

Figure 38: VIV test case. Prepare batch calculation.

		New parameters set - Code_Saturne GUI	- <b>-</b> ×
File Tools Window Help			
🗂 📾 💽 🔕 📭 💷			
Sav			ලා ගැ
study: Tuto_VIV			
Case: VIV			
XML file:	1	Save the calculation data file	
	-71		
0	×	Computer selection	
- 🗋 Identity and paths	1		
E Calculation environment		Workstation 🗢	
Meshes selection			
Thermophysical models		Select the batch script file 🛛 🔚 runcase	
Calculation features			
Mobile mesh		Prepare batch calculation	
Turbulence models			
🔄 🗋 Thermal model		Number of processors 3	
🖻 🛅 Additional scalars			
- Definition and initialization		User hies	
Physicals properties		Advanced entires	
E- Physical properties		Advanced options	
Hererence values			
Gravity bydrostatic pressure		Code_Saturne	
B Volume conditions		batch	
Volume regions definition		running	
Initialization			
Head losses			
🗄 🛅 Boundary conditions			
Definition of boundary regi	-		
- D Boundary conditions	100		
- Fluid structure interaction	4		
E- D Numerical parameters			
Ime step     Gruption parameters			
Global parameters			
Calculation control			
Time averages			
- C Output control			
Volume solution control			
Profiles			
🖻 🛅 Calculation management			
User arrays			
Memory management			
- Start/Restart			
Prepare batch calculation			
r 21	-		
	1		
Save the current case.			

Figure 39: VIV test case. Prepare batch calculation.

		New parameters set - Code_Saturne GUI	_ 0 ×
File Tool	s <u>W</u> indow <u>H</u> elp		
1	🖻 🕭 🔕 📭 🗄		
			88
Study:	Tuto_VIV		
Case:			
XML file-			
And the	ļ	annun -	]
		Save File As	
	dentity and paths		
	Meshe selection Meshe selection Mesh quality criteri hermophysical model Calculation feature Mobile mesh Turbulence models Thermal model didtional scalars Definition and initia Physicals properties Definition and initia Physicals properties Statuty, hydrostatic folume conditions Volume regions del Initialization Head losses gundary conditions	Compu berland	
	Definition of bound	File <u>n</u> ame: tuto_viv.xml	
	Fluid structure inte	Files of type: Code Saturne GUI file (mil)	
	umerical parameters		
	Equation parameters Global parameters alculation control Time averages Output control Volume solution cont Profiles alculation managemen User arrays Memory managemen Start/Restart Prepare batch calcul	ntrol ent ulation	<mark>id save it</mark>
•			

Figure 40: VIV test case. Prepare batch calculation.

	tuto_viv.xml - Code_Saturne GUI
Tools <u>Window H</u> elp	
🗁 🕭 🔕 📭 🔍	
Tuto VIV	
file: //home/berland/Documents/CAC14	/Tutoriel_CS/Test_Tuto/Tuto_VIV/VIV/DATA/tuto_viv.xml
69 69	- Computer selection
] Identity and paths	
Calculation environment	Workstation 🗢
Meshes selection	
Mesh quality criteria	Select the batch script file
Inermophysical models	
Calculation features	Prepare batch calculation
D Turbulance medala	riepare pater calculation
Thermal models	Number of processors
Additional scalars	
Additional scalars     Definition and initialization	User files
Physicals properties	
Physical properties	Advanced options
Reference values	
- D Fluid properties	
Gravity, hydrostatic pressure	Code saturne
Volume conditions	Daton
Volume regions definition	Landrung
Initialization	
- 🗋 Head losses	7
Boundary conditions	
Definition of boundary regi	
Boundary conditions	
Fluid structure interaction	
Numerical parameters	
I lime step	
Equation parameters	/
- Global para	ada Caturna hatah
Time avera	.ode_Saturne patch» /
Output con to run the	calculation
- Volume sol	
Profiles	
Calculation management	
User arrays	
Memory management	
Start/Restart	
Prepare batch calculation	
11	

Figure 41: VIV test case. Prepare batch calculation.



Figure 42: VIV test case. Time history of the structure displacement in the y direction.

## 3.2 How to setup a case involving fluid-structure interactions with user sub- routines

In order to fully define the present test case the following source files are required and must be copied in the SRC directory:

- usini1.f90: definition of the standard calculation parameters;
- usclim.f90: definition of the nature of the boundary conditions;
- usalin.f90: definition of the specific parameters of the ALE (to turn it on and provide the parameters of the internal fluid-structure coupling);
- usalcl.f90: definition of the behavior of the boundary regions when the mesh is mobile (fixed b.c., sliding b.c.,...);
- usstru.f90: to link a mass-spring structure to a boundary condition ("internal" fluid-structure coupling);
- usvima.f90: definition of the mesh viscosity.

#### 3.2.1 Standard parameters definition

The standard calculation parameters are defined in the user source file usinil. f90. Nonetheless, these variables will not be described here since they fall outside the scope of a tutorial on fluid-structure interaction in *Code\_Saturne*. However, as concern the outputs, one should not forget to set the keyword ichrmd to 1:

ichrmd = 1

in order to obtain output data on a deformable mesh. This feature makes possible to observe the mesh motions with a visualization software (Ensight or Paraview for instance).

#### **3.2.2** Boundary conditions

In a similar manner, even though a mobile mesh is used, the setup of the boundary conditions does not require any specific treatment in the file usclim. f90. This source file will not be described here.

#### **3.2.3** Definitions specific to the ALE

The features of the mobile mesh procedure, as well as those of the fluid-structure internal coupling algorithm, are defined in the user source file usalin.f90. For the present case the various parameters of the ALE should be set to the following values:

```
iale = 1  ! turn on the ALE
nalinf = 5000! number of iterations for fluid initialization
nalimx = 20  ! number of sub-cycling iteration for fluid-structure
interaction epalim = 1.d-8 ! relative precision of sub-cycling fluid-
structure coupling iortvm = 0  ! isotropic mesh viscosity
```

As pointed out earlier in section 3.1 the key parameters are:

- iale: equals 0 when the mobile mesh feature is deactivated and 1 when it is activated;
- nalinf: is the number of iteration before the fluid-structure coupling is actually performed. Some iterations might indeed be necessary in order to have a well-established flow before allowing any motion of the structure;
- nalimx and epalim: control the convergence of the iterative resolution of the fluid-structure coupling. These parameters should be chosen carefully to avoid any unphysical behavior of the solution (see for instance section 3.8);
- iortvm: defined whether the mesh viscosity is orthotropic (the mesh viscosity is a vector) or isotropic (the mesh viscosity is the same in all three directions).

#### 3.2.4 Mobile mesh boundary conditions

Since the mesh is mobile one can define if a boundary region is fixed or mobile. These definitions can be done in the source file usalcl.f90. For each face ifac of the mesh, the keyword ialtyb(ifac) can be set to igliss if the mesh is sliding, or to ibfixe if the mesh is fixed. In our present case, only boundary regions 10 and 11, which corresponds to the front and back frontiers of the mesh, are sliding:

The final source file is provided in the tutorial directory  $Tutorial_Files/User_Sources$ .

#### 3.2.5 Structure definition

The mechanical properties of the structure and the definition of the forces acting on it can be provided by the user in the file usstru. f90.

The first subroutine usstr1 is called at the beginning of the calculation. It first permits to link a structure to a boundary region. For this structure a few parameters mau be defined:

- xstr0(idim,istructure): initial position of the structure (idim=1,2,3 is the index of the dimension and istructure the number of thestructure);
- vstr0(idim, istructure): initial velocity of the structure;
- xstreq(idim, istructure): equilibrium position of the structure;
- aexxst, bexxst and cfopre: advanced users may also change some parameters of the internal fluidstructure coupling procedure.

For the present test case the source code for this subroutine should read:

```
! --- Assign faces of boundary region 1 to a structure
call getfbr('1', nlelt, lstelt)
do ilelt = 1, nlelt
  ifac = lstelt(ilelt)
  idfstr(ifac) = 1
enddo
! --- Some structure parameters
xstr0(2,1) = 0.d0
xstreq(2,1) = 0.d0
vstr0(3, 2) = 0.d0
! --- Here one can modify the values of the prediction coefficients for
  displacements and fluid forces in internal FSI coupled algorithm.
1
aexxst = 0.5d0
bexxst = 0.0d0
cfopre = 2.d0
! --- Activation of structural history output
ihistr = 1
```

Then, in subroutine usstr2 which is called at each time step the mass, stiffness and damping of the structure, as well as the forces acting on the structure can de defined:

- xmstru(i,j,istr), xcstru(i,j,istr) and xkstru(i,j,istr) correspond to the mass, damping and stiffness matrices of the structure number istru;
- forstr(idim, istr) is the force acting on the structure. By default this vector is filled in with the fluid-forces but the user can modify these values if needed.

As concerns the case treated in this tutorial the source code of the subroutine is given by:

```
! --- Define matrices istr = 1
cyl m = 5.16e-3
cyl k = 4.81e-4
xmstru(1, 1, istr) = 1.
xmstru(2,2,istr) = cyl_m
xmstru(3,3,istr) = 1.
xcstru(1, 1, istr) = 0.
xcstru(2, 2, istr) = 0.
xcstru(3,3,istr) = 0.
xkstru(1, 1, istr) = 1.
xkstru(2,2,istr) = cyl_k
xkstru(3,3,istr) = 1.
! --- Define forces acting on the structure
! --- only the force in the lift direction (2)
! --- is kept. The others are set to zero istr = 1
forstr(1, istr) = 0.
forstr(3, istr) = 0.
! --- Structural time step istr = 1
dtstr(istr) = dtcel(1)
```

The final source file is provided in the tutorial directory Tutorial\_Files/User\_Sources.

#### 3.2.6 Mesh viscosity definition

A shown in figure 11, the mesh viscosity is set to a very high value close to the cylinder whereas it is equal to 1 away from the surface of the cylinder. This can be done using the source file usvima. f90 with the following code: in the declarations add the line,

double precision rad, xr2, xcen, ycen

and the subroutine body should contain:

```
if (ntcabs.eq.0) then
    rad = (0.025)**2
    xcen = 0.d0
    ycen = 0.d0
    do iel = 1, ncel
        xr2 = (xyzcen(1,iel)-xcen)**2 + (xyzcen(2,iel)-ycen)**2
        if (xr2.lt.rad) viscmx(iel) = 1.d10
    enddo
endif
```

The aim here is to loop over all the cells (do iel = 1, ncel) and apply a given mesh viscosity (viscmx(iel)) according to the location of the center of the cell (xyzcen(1,iel) for x location and xyzcen(2,iel) for y location).

The viscosity is isotropic so that only viscmx(iel) needs to be defined. In the general case the three viscosity components, viscmx(iel), viscmy(iel) and viscmz(iel) have to be provided by the user.

#### 3.2.7 Runcase script

No special step need to be performed for the runcase when the mobile mesh features is turned on. Simply define the mesh used, and the number of processors required:

```
MESH='mesh_viv.des'
NUMBER_OF_PROCESSORS=3
```

#### **3.3** How to impose the displacement of the structure (GUI)

Instead of using the internal coupling to obtain the motions of the structure one may want to impose the displacement of the cylinder. The steps required in the GUI to perform such an operation are explained in figures 43 to 48. Note that the steps described in section 3.1 also need to be performed to setup the calculation.

		tuto_viv.xml -	Code_Saturne (	iui	_ = ×	
File Tools Window Help						
1 Select "Boundary conditions"						
I. Select «Boulidal	y condit	ions#				
					ØX	
Study: Tuto VIV						
2 Select #	PC 1.					
Case: VIV Z. Select «	DC_1»	accorde to				
XML file: /home/berla the cylinder cy	region con	esponds to	DATA/tuto viv.xm	£		
the cylinder su	nace.					
(2 X	1	e Source and the second				
D. D. Montilu and nothe	Boundary c	nditions				
R- Calculation environment	Label	7000	Natura	Selection criteria		
Heshes selection	PC 1	Zone	wall	Selection criceria		
🕂 🗋 Mesh quality criteria	BC 2	2	symmetry	2		
🖻 🗁 Thermophysical models	BC_3	3	inlet	3		
Calculation reatures	BC_4	4	outlet	4		
Turbulence models	BC_5	5	symmetry	10		
Thermal model	BC_6	6	symmetry	11		
🖻 🛅 Additional scalars						
Definition and initialization						
Physicals properties						
E- Physical properties	Smooth	or rough woll				
	sinooun	or rough wan				
Gravity hydrostatic pressure			🖲 sr	nooth wall 🔘 rough wall		
🖻 🗁 Volume conditions						
Volume regions definition						
Initialization	Mobile m	iesh			-	
Head Idsses			(m. 1)			
Boundary conditions			Hxed d	isplacement 🗘		
Boundary conditions			- <b>1</b>			
Fluid structure interaction			- /	▲		
🖻 🛅 Numerical parameters			/	T		
Time step			/			
Equation parameters						
Global parameters						
	d allowed as					
3. Select «Fixe	d displa	cement»				
Vol The displacement of	the mesh r	region is goi	ng			
Pro to be given by the us	ser. The for	mula editor	64			
🖶 🗁 Calcula will help us doing thi	s.					
Use						
Memory management						
Start/Restart     Prepare batch calculation						
				4. Open the formula editor		
•						

Figure 43: Imposed displacement case. Boundary conditions.

<u>File Tools Window Help</u>		
i 🛅 🚰 🕭 🥝 🃭 i 🗷		
		0 8
Study: Tuto_VIV		]
Case: VIV		
XML file: /home/berland/Documents/CAC14/T	utoriel CS/Test Tuto/Tuto VIV/VIV/DATA/tuto viv.xml	
	Mathematical expression editor	
8	Iser expression Predefined symbols Examples	
Identity and paths Calculation environment Meshes selection Meshes selection Mesh quality criteria Thermophysical models Calculation features Mobile mesh Turbulence models Thermal model Additional scalars Definition and initialization Physical properties Physical properties Physical properties Fluid properties Volume conditions Volume regions definition Initialization Head losses Boundary conditions Fluid structure interaction Numerical parameters Global parameters Calculation control Profiles Calculation management User arrays Memory management Start/Restart Prepare batch calculation	<pre>d=0.025; mesh_x=0; mesh_x=(4/4)*sin(2*pi*t/100/dt); mesh_z=0;</pre> Fill in the editor with the formula for the mesh dialacement in direction is set to zero. The cylin undergoes a transverse motion to axis. The motion is harmonic with an to d/4 (d is the cylinder diamete period is set to 100 time steps.	te following isplacement the x and z ider only following the y- amplitude equal r) and the

Figure 44: Imposed displacement case. Boundary conditions.

	tuto_viv.xml - Code_Saturne GUI	_ D X				
File Tools Window Help						
1. :	Select «Mobile mesh»					
	1	8				
Study: Tuto_VIV	/					
Case: VIV		î				
XML file: /home/berland/Documents/GAC	4/Tutoriel_C5/Test_Tuto/Tuto_VIV/VIV/DATA/tuto_viv.xml	i				
L D Identity and paths	Mobile mesh (ALE method)					
Calculation environment  Meshes selection  Mesh quality criteria  Calculation features  Calculation features  Mobile mesh	Number of iterations for fluid initialization Type of the viscosity of mesh Spatial distribution of	0 isotropic ÷				
- D Turbulence models	Formula for the viscosity of mesh	8				
Definition and initialization     Physical properties     Physical properties     Gravity, hydrostatic pressure     Volume regions definition     Initialization     Head losses     Boundary conditions     Definition of boundary regi     Definition parameters     Output control     Dutput control     Dutput control     User arrays     Memory management     Start/Restart     Prepare batch calculation	2. Set the number of iterations finitialization to 0 The flow initialization is skipped in the prese example to spare some computational time.	<b>for</b> Int				

Figure 45: Imposed displacement case. Mobile mesh.

		tuto_viv.xml - Code_Saturne GUI	_ = ×
Elle Tool	s <u>W</u> indow <u>H</u> elp		
	🖶 🕭 🙆 🃭 i 🗷 💳		
	1	Select «Time step» in the treeview	
	n		6 6
Study	Tuto_VIV		
Case:	VIV		
XML file:	/home/berland/Documents/CAC1	4/Tutoriel_CS/T4st_Tuto/Tuto_VIV/VIV/DATA/tuto_viv.xml	
	12 12	Unsteady flow algorithm management	
	dentity and paths alculation environment	Time step option Uniform and constant	
	Mesh quality criteria	Reference time step 0.1 s	
	hermophysical models		
	Mobile mesh	Number of iterations (restart included) 500	
	Turbulence models		
B B A	Additional scalars	Option zero time step	
	Definition and initialization		<u></u>
E B B	] Physicals properties	/ /	
T FC	Reference values		
	Fluid properties		
L B	Gravity, hydrostatic pressure		
	Volume regions definition		
	j Initialization		
	] Head losses	/	
	Definition of boundary real	2 Set time step to 0.1 s and number of	
	Boundary conditions	its and indicate to E00	
	Fluid structure interaction	Iterations to 500	
0.0 M	lumerical parameters	500 iterations correspond to 5 periods of the cylinder	
	Equation parameters	transverse periodic motion.	
	Global parameters		
Ė € 0	alculation control		
	] Time averages		
	Volume solution control		
	Profiles		
E 6 9	alculation management		
in the	) User arrays		
	] Memory management		
	Prepare batch calculation		
4			
120	564.		

Figure 46: Imposed displacement case. Time step.

_	🗖 tuto_viv.xml - Code_Saturne GUI 📃 🗖 🗙				
Ele Too	is <u>Window H</u> elp 2 🕲 👔 🚺 1. Selec	ct «Output control» in the treeview			
			2 29		
Study	Tuto_VIV	2. Choose «Post-processing every»	2		
Case:	VIV	and set the period to 10 iterations			
XML file:	/home/berland/Documents/CAC14/Tu	Itoriel_CS/Test_Tuto/Tuto_VIV/VIV/DATA/tu The output period must be chosen small enough to			
		ensure that the cylinder motions will be visible			
	2 A 1	Output Control Monitoring Points Co snapshots of the flow-field per period.			
	aentry and paths (alculation environment ) Meshes selection ) Mesh quality criteria hermophysical models ) Calculation features ) Mobile mesh ) Turbulenes models	Outputs listings Output listing at each time step  Pist-processing  Post processing  Dest processing  1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1			
	Thermal model				
E E 7	dditional scalars	Fluid domain post processing 🔍			
	Physicals properties	Domain boundary post processing			
	Physical properties Reference values	Type of post-processing for mesh deformable (with displacement)			
ļ-ļ	Fluid properties	Post-processing format EnSight Gold			
E D V	olume conditions	Options			
	Volume regions definition	format binary 🗢			
D-D B	] Head losses Joundary conditions	polygons display \$			
TF	Definition of boundary regi.	nokhedra disnlav			
	Fluid structure interaction	his endian			
e e r	Iumerical parameters	big-enulari			
	Equation parameters				
	Global parameters				
	Time averages				
	Output control				
I E	Profiles				
6 B 9	alculation management				
- Ch	J User arrays Memory management				
-	j Start/Restart				
1	Prepare batch calculation				
1	lif 💽				

Figure 47: Imposed displacement case. Output control

	tuto_viv.xml - Code_Saturne GUI	_ <b>=</b> ×
File Tools Window Help		
1 🖂 📾 🔕 📭 1 🗷		
		12 B
Study: Tuto_VIV		
Case: VIV		
XML file: /home/berland/Documents/CAC14/	1. Save the calculation data file	
0 R		
I D Identify and notice	Computer selection	
E Calculation environment	Workstation 😂	
Meshes selection		
Mesh quality criteria	Select the batch script file	
E E Thermophysical models	select the batch script me	
Calculation features	Pressors batch colculation	
Mobile mesh	Prepare batch calculation	
- U lurbuience models	Number of processors	
Additional scalars		
Definition and initialization	User files	
- D Physicals properties		
E Physical properties	Advanced options	
- Reference values		
- 1 Fluid properties	Cada Catuma	
Gravity, hydrostatic pressure	batch	
🖻 🛅 Volume conditions		
Click on «Code_S batch» to run the     Definition of boundary regime Boundary conditions     Boundary conditions     Fluid structure interaction     Time step     Boundary conditions     Calculation parameters     Goloal parameters	aturne calculation	
I = 100 million		

Figure 48: Imposed displacement case. Prepare batch calculation.

#### **3.4** How to impose the displacement of the structure (user subroutines)

Copy the user-source usalcl.f90 in the source directory:

cp Tuto VIV/VIV/SRC/REFERENCE/base/usalcl.f90 Tuto VIV/VIV/SRC/.

Edit the source file and add in the "local variables" declaration zone the following line to add a variable for the displacement of the mesh:

double precision delta

Then, in the body of the subroutine, add the following source code in order to impose to each node a given displacement:

```
! displacement amplitude (the diameter is 0.025)
delta = (0.025d0/4.d0)*sin(2.d0*pi*ttcabs/100.d0/dtref)
! get the cell faces corresponding to color 1
call getfbr('1',nlelt,lstelt)
!==========
! loop over these faces and impose them some
! displacement
do ilelt = 1, nlelt
   ifac = lstelt(ilelt)
   do ii = ipnfbr(ifac), pnfbr(ifac+1)-1
      inod = nodfbr(ii)
      if (impale(inod).eq.0) then
         depale(inod,1) = 0.d0 ! displacement /x
         depale(inod,2) = delta ! displacement /y
         depale(inod, 3) = 0.d0 ! displacement /z
         impale(inod) = 1
      endif
   enddo
enddo
```

Note that the displacement is defined at the nodes, whereas the mesh velocity is defined at the cell faces (see section 3.6). Finally run *Code\_Saturne* using the GUI.

#### 3.5 How to impose the velocity of the structure (GUI)

The velocity of the cylinder may also be imposed. The steps required in the GUI to perform such an operation are explained in figures 49 to 54. Note that the steps described in section 3.1 also need to be performed to setup the calculation.

		tuto_viv.xml -	Code_Saturne C	ווחי	2 0 ×
File Tools Window Help					
1. Select «Boundary conditions»					
					23
Study Data 10					
	00.1				
Case: VIV Z. Select «	SC_I»	ocoonds to			
XML file: /home/teria the cylinder sur	face.	esponds to	DATA/tuto_viv.xm		
Ø 8	Boundary	anditions -			
Identity and paths	boundary e				
Calculation environment	Label	Zone	Nature	Selection criteria	
Mesh quality criteria	BC 2	1	wall	2	
🖻 🛅 Thermophysical models	BC_3	3	inlet	3	
Mobile mesh	BC_4	4	outlet	4	
- D Turbulence models	BC_5	5	symmetry	10	
Thermal model	00_0	0	symmetry		
Additional scalars     Hold Definition and initialization					
- Physicals properties					
🖻 🛅 Physical properties	Creath	ar rough wall			
- I Fluid properties	Smooth	or rough wall			
Gravity hydrostatic pressure			) sm	ooth wall 🔘 rough wall	
🔁 🛅 Volume conditions					
Initialization	Mobile m	esh			
Head losses					
E Boundary Conditions			Fixed v	elocity 😫 🔡	
Boundary conditions					
- D Fluid structure interaction			/	<b>+</b>	
E D Numerical parameters			/		
- Equation parameters			/		
- 🗋 Global parameters					
	lvalacit				
Out The velocity of the m		y»	ha		
- Vol The velocity of the m	esh region	is going to	olp		
Pro given by the user. In	e ionnula i	editor win n	eib		
Memory management				1	
Prepare batch calculation				-	
				4. Open the formula edito	r 🛛
4					

Figure 49: Imposed velocity case. Boundary conditions.



Figure 50: Imposed velocity case. Boundary conditions.

		tuto_viv.xml - Code_Saturne GUI	_ = ×
<u>File Tool</u>	s <u>W</u> indow <u>H</u> elp		
1	🚔 🕭 🔕 🕪 i 🗷 🗖	Calast Time story in the transient	
		. Select «Time step» In the treeview	8 X
Etuda	Tate 10/		1
study			
Case:	VIV		
XML file:	/home/berland/Documents/CAC	4/Tutoriel_CS/Tilst_Tuto/Tuto_VV/VIV/DATA/tuto_vix.xml	
	08.6	and and a second se	
	12 K	Unsteady flow algorithm management	
E-6 0	dentity and paths Calculation environment	Time step option Uniform and constant	
	Meshes selection		
0.01	hermophysical models	Reference time step 0.1 s	
	] Calculation features ] Mobile mesh	Number of iterations (restart included) 500	
	Turbulence models		
E D A	dditional scalars	Option zero time step	
	Definition and initialization	1	
e e F	hysical properties	/ /	
	] Reference values ] Fluid properties	/	
	Gravity, hydrostatic pressure		
	Volume regions definition		
	] Initialization		
0.00	oundary conditions		
	Definition of boundary regi Boundary conditions	2. Set time step to 0.1 s and number of	
	Fluid structure interaction	500 iterations correspond to 5 periods of the cylinder	
	Time step	transverse periodic motion.	
	Equation parameters		
÷ • •	Calculation control		
	] Time averages ] Output control		
	Volume solution control		
E D C	alculation management		
t F	User arrays		
	) Start/Restart		
	Prepare batch calculation		
4			
100			

Figure 51: Imposed velocity case. Time step.

🗖 tuto_viv.xml - Code_Saturne GUI 📃 🗖 🗙					
File Tools Window Help					
🔲 🔄 🙆 🚺 🚺 1. Select «Mobile mesh»					
		1	(2) (2)		
Study:	Tuto_VIV	/			
Case:	VIV		1		
XML file:	/home/berland/Documents/GAC1	4/Tutoriel_C5/Test_Tuto/Tuto_VIV/VIV/DATA/tuto_viv.xml			
	ientity and paths	✓ Mobile mesh (ALE method)			
	alculation environment	Number of iterations for fluid initialization	0		
0.01	Mesh quality criteria hermophysical models	Type of the viscosity of mesh	isotropic 🜩		
	Mobile mesh	Spatial distribution of the viscosity of the mesh	user formula 💠		
	Turbulence models Thermal model	Formula for the viscosity of mesh	<b>R</b>		
	dditional scalars Definition and initialization				
	Physicals properties				
	Reference values				
	Gravity, hydrostatic pressure				
e e v	olume conditions	2. Set the number of iterations f	or		
	Volume regions definition	initialization to 0			
	Head losses	The flow initialization is skipped in the prese	nt		
6 D 8	oundary conditions	example to spare some computational time.			
	Definition of boundary regi				
	Fluid structure interaction				
B-B N	lumerical parameters				
	) Time step				
	Equation parameters				
	] Global parameters				
E E C	alculation control				
	Output control				
	Volume solution control				
	Profiles				
B B C	alculation management				
H	User arrays				
H-C	Memory management				
	Start/Restart				
1	Prepare batch calculation				
find	1.1				

Figure 52: Imposed velocity case. Mobile mesh.

🗖 tuto_viv.xml - Code_Satume GUI 🔤 🗖 🗙				
	s <u>Window Help</u>	ct «Output control» in the treeview		
			88	
Study	Tuto_VIV	2 Choose "Bost processing every		
Case:	VIV	and set the period to 10 iterations		
Case: XML file:	VVV //home/berland/Documents/CAC14/Tu fentity and paths alculation environment Meshes selection Mesh quality criteria hermophysical models Calculation features Mobile mesh Turbulence models Turbulence models Thermal model Definition and initialization Physicals properties Reference values Fluid properties Gravity, hydrostatic pressure folume conditions Definition of boundary regi Boundary conditions Definition features Time step Equation parameters Global parameters Calculation control Time averages Output control Profiles Calculation management User arrays Memory management	toriel_CS/Test_Tuto/Tuto_VV/VV/DATANU output/Control Monitoring Points (C Output/Control Monitoring Points (C Output Isitings Output Isiting at each time step \$1 Post-processing every 'n' time steps \$10 Fluid domain post processing Domain boundary post processing Type of post-processing for mesh deformable (with displacement) \$ Post-processing format EnSight Gold \$ Options format binary \$ polygons display \$ polydera display \$ polydera display \$ big-endian		
	Start/Restart Prepare batch calculation			

Figure 53: Imposed velocity case. Output control.

	tuto_viv.xml - Code_Saturne_GUI	_ = ×
File Tools Window Help		
1 🗖 📾 🕭 🔕 🖪		
× ×		12 B
Study: Tuto_VIV		
Case: VIV	na tan	
XML file: /home/berland/Documents/CAC14/	1. Save the calculation data file	
ØX		
L D Identity and notice	Computer selection	
E Calculation environment	Workstation	
Meshes selection		
Mesh quality criteria	Calect the batch script file	
😑 🛅 Thermophysical models	select the batch script me	
Calculation features	Province hashing and a database	
Mobile mesh	Prepare batch calculation	2
Thermal models	Number of processors	
Additional scalars		
Definition and initialization	User files	
Physicals properties		
ET Physical properties	Advanced options	
Reference values		
- Fluid properties	Code Saturne	
Gravity, hydrostatic pressure	batch	
🖻 🛅 Volume conditions	running	
Click on «Code_S batch» to run the     boundary conditions     Boundary conditions     Pluid structure interaction     Numerical parameters     Global parameters     Global parameters     Global parameters     Calculation control     Output control     Output control     Profiles     Calculation management     User arrays     Memory management     Start/Restart     Prepare batch calculation	calculation	

Figure 54: Imposed velocity case. Prepare batch calculation.

#### **3.6** How to impose the velocity of the structure (user subroutines)

Copy the user-source usalcl. f90 in the source directory:

cp Tuto\_VIV/VIV/SRC/REFERENCE/base/usalcl.f90 Tuto\_VIV/VIV/SRC/.

Edit the source file and add in the "local variables" declaration zone the following line to add a variable for the velocity of the mesh:

double precision deltav

Then, in the body of the subroutine, add the following source code in order to impose to each a given displacement:

```
! velocity magnitude (the diameter is 0.025)
          (0.025/4.)*(2.*3.141596d0/100.d0/dtref)
deltav =
                                                       &
          * cos(2.*3.141596d0*ttcabs/100.d0/dtref)
! get the cell faces corresponding to color 1
call getfbr('1', nlelt, lstelt)
!=========
! loop over these faces and impose them some
! velocity
do ilelt = 1, nlelt
 ifac = lstelt(ilelt)
 iel = ifabor(ifac)
 ialtyb(ifac) = ivimpo
 rcodcl(ifac,iuma,1) = 0.d0
 rcodcl(ifac,ivma,1) = deltav
 rcodcl(ifac,iwma,1) = 0.d0
enddo
```

Note that velocity is defined at the cell faces, whereas the displacement is defined at the nodes (see section 3.4). Finally run *Code\_Saturne* using the GUI.

#### **3.7** How to compute the force acting on the structure (usersubroutines)

As pointed out in section 3.1, some outputs specific to fluid-structure interactions, such as the forces acting on the structure, are available in the RESULTS.

However it should be noted that these quantities are calculated only when the fluid-structure coupling is activated (as shown in figure 25). When the displacement is imposed, the forces acting on the structure has to be computed by the user if needed.

To do so, copy the user-source usproj.f90 in the source directory. Edit the source file and add in the "local variables" declaration zone the following lines:

```
double precision xfor(3)
Then, in the body of the subroutine, add the following source code:
  ! set the force components to zero
 do ii = 1, ndim
    xfor(ii) = 0.d0
 enddo
 ! get the cells corresponding to the cylinder surface
 call getfbr('1', nlelt, lstelt)
  !==========
  ! loop over the cells to integrate the force
 ! over the structure surface
 do ilelt = 1, nlelt
    ifac = lstelt(ilelt)
    ! update the force
    do ii = 1, ndim
       xfor(ii) = xfor(ii) + ra(iforbr + (ifac-1)*ndim + ii-1)
    enddo
 enddo
  ! if the calculation is parallel, add the data from the
  ! other processes
 if (irangp.ge.0) then
    call
    parrsm(ndim, xfor)
 endif
```

We eventually get a variable xfor containing the three components of the force acting on the cylinder.

# **3.8** How to control the convergence of the internal fluid-structure coupling procedure (advanced user)

As shown in figure 26, the number of sub-iterations and the precision for the fluid-structure are some parameters that need to be adjusted to obtain a relevant solution. In order to control wether the iterative of the coupling has fully converged, it may be valuable to print out the number of sub-iterations that have indeed been performed.

To do so, get the source file strdep. f90 in the source themselves of *Code\_Saturne*. This file is not available in the SRC/REFERENCE directory of the case but may be found in the installation directory of *Code\_Saturne*. Edit the strdep.f90 and add at the end of the convergence test labelled "5. TEST DE CONVERGENCE" the following lines:

```
if (irangp==1) then
    if (icv==1) then
        open(unit=impusr(1),file='cv.dat',position='append')
        write(impusr(1),*) ntcabs, italim
        close(impusr(1))
    endif
endif
```

in order to create a data file named cv. dat containing the time history of the convergence of the coupling iterative process (impusr(1) is the unit id corresponding to the user file). The following *Code\_Saturne* variables are used:

- icv: when equal to 1, indicates that the algorithm has indeed converged (else equal to 0);
- ntcabs: current time-step;
- italim: sub-iteration number.

The first column of the file contains the iteration number and second column corresponds to the number of subiteration required for the fluid-structure coupling to converge. Make sure this number is always lower than the maximum number of sub-iteration defined in the GUI (see figure 26), otherwise unwanted unphysical behaviors may occur.

To allow the file cv. dat to be copied in the result directory at the end of the calculation, one also need to add its name in the runcase script located in the SCRIPTS directory of the case.