

Fluid Dynamics, Power Generation and Environment Department Single Phase Thermal-Hydraulics Group

6, quai Watier F-78401 Chatou Cedex

SEPTEMBER 2017

Code_Saturne documentation

Code_Saturne version 5.0 tutorial: full domain

contact: saturne-support@edf.fr



EDF R&D	<i>Code_Saturne</i> version 5.0 tutorial: full domain	Code_Saturne documentation Page 1/73
---------	---	--

TABLE OF CONTENTS

	I Introduction	5
1	Introduction	6
1.1	Code_Saturne SHORT PRESENTATION	6
1.2	About this document	6
1.3	Code_Saturne COPYRIGHT INFORMATIONS	6

 $\mathbf{7}$

1	Study description	8
1.1	Objective	8
1.2	Description of the configuration	8
1.3	CHARACTERISTICS	8
1.4	Mesh characteristics	9
1.5	SUMMARY OF THE DIFFERENT CALCULATIONS	9
2	CASE 1: Passive scalar with various boundary conditions and output management	10
2.1	CALCULATION OPTIONS	10
2.2	Initial and boundary conditions	11
2.3	PARAMETERS AND USER ROUTINES	12
2.4	OUTPUT MANAGEMENT	12
2.5	Results	13
3	CASE 2: Time dependent boundary conditions and variable fluid density $\ .$	16
3.1	CALCULATION OPTIONS	16
3.2	Initial and boundary conditions	16
3.3	VARIABLE DENSITY	17
3.4	Parameters	17
3.5	User routine	17
3.6	OUTPUT MANAGEMENT	18
3.7	CALCULATION RESTART	19
3.8	Results	19

4	CASE 3: Head losses, parallelism and spatial average 22	2
4.1	CALCULATION OPTIONS	2
4.2	Initial and boundary conditions	2
4.3	VARIABLE DENSITY	3
4.4	HEAD LOSSES	3
4.5	PARAMETERS	3
4.6	USER ROUTINES	4
4.7	Output management	5
4.8	Results	5

	III Step by step solution	28
1	Solution for CASE1	29
2	Solution for CASE2	60
3	Solution for CASE3	68

EDF R&D	<i>Code_Saturne</i> version 5.0 tutorial: full domain	$\begin{array}{c} Code_Saturne\\ documentation\\ Page 4/73 \end{array}$

Part I

Introduction

1 Introduction

1.1 *Code_Saturne* **short presentation**

Code_Saturne is a system designed to solve the Navier-Stokes equations in the cases of 2D, 2D axisymmetric or 3D flows. Its main module is designed for the simulation of flows which may be steady or unsteady, laminar or turbulent, incompressible or potentially dilatable, isothermal or not. Scalars and turbulent fluctuations of scalars can be taken into account. The code includes specific modules, referred to as "specific physics", for the treatment of lagrangian particle tracking, semi-transparent radiative transfer, gas, pulverized coal and heavy fuel oil combustion, electricity effects (Joule effect and electric arcs) and compressible flows. *Code_Saturne* relies on a finite volume discretization and allows the use of various mesh types which may be hybrid (containing several kinds of elements) and may have structural non-conformities (hanging nodes).

1.2 About this document

The present document is a tutorial for *Code_Saturne* version 5.0. It presents three simple test cases and guides the future *Code_Saturne* user step by step into the preparation and the computation of the cases.

The test case directories, containing the necessary meshes and data are available in the examples directory.

This tutorial focuses on the procedure and the preparation of the *Code_Saturne* computations. For more elements on the structure of the code and the definition of the different variables, it is higly recommended to refer to the user manual.

1.3 *Code_Saturne* copyright informations

Code_Saturne is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version. *Code_Saturne* is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

Part II

Full domain

1 Study description

1.1 Objective

The aim of this case is to tackle the merging of initially separate meshes into a single fluid domain. The questions of mesh joining and hanging nodes will be addressed. The test case will then be used to present more complex calculations, with time dependent variables and Fortran user routines.

1.2 Description of the configuration

The fluid domain is composed of three separate meshes, very roughly representing elements of a nuclear pressurized water reactor vessel:

- the downcomer
- $\bullet\,$ the vessel's bottom
- the lower core plate and core

Figure II.1 represents the complete domain. The flow circulates from the top left horizontal junction to the right vertical outlet.



Figure II.1: Geometry of the complete domain

1.3 Characteristics

Characteristics of the geometry and the flow:

Physical characteristics of fluid:

The initial water temperature in the domain is equal to 20°C. The inlet temperature of water in the

Height of downcomer	$H = 3.00 \ m$
Thickness of downcomer	$E_d = 0.10 \ m$
Diameter of the inlet cold branch	$D_b = 0.50 \ m$
Height of vessel's bottom	$H_{fc} = 1.00 \ m$
Width of vessel's bottom	$l_{fc} = 1.00 \ m$
Height of core above the lower core plate	$H_{pic} = 1.00 \ m$
Width of core above the lower core plate	$l_{pic} = 0.50 \ m$
Inlet velocity of fluid	$V = 1 \ m.s^{-1}$

Table II.1: Characteristics of the geometry and the flow

cold branch is 300°C. Water characteristics are considered constant¹ and their values taken at 300°C and $150 \times 10^5 Pa$, except density which is considered variable in case2 and case3:

- density: $\rho = 725.735 \ kg.m^{-3}$
- dynamic viscosity: $\mu = 0.895 \times 10^{-4} \ kg.m^{-1}.s^{-1} = 8.951 \times 10^{-5} \ Pa.s$
- heat capacity: $C_p = 5\,483 \; J.kg^{-1}.^{\circ}C^{-1}$
- thermal conductivity = $0.02495 \ W.m^{-1}.K^{-1}$

1.4 Mesh characteristics

Figure II.2 shows a global view of the mesh and some details of the joining zones, to show that *Code_Saturne* can deal with hanging nodes. This mesh is composed of 1650 cells, which is very small compared to those used in real studies. This is a deliberate choice so that tutorial calculations run fast.

Type: block structured mesh

Coordinates system: cartesian, origin on the edge of the main pipe at the outlet level, on the nozzle side (figure II.2)

Mesh generator used: SIMAIL and mesh joining with the Preprocessor of *Code_Saturne* (in order to deal with hanging nodes)

Color definition: see figure II.3

1.5 Summary of the different calculations

Three cases will be studied with this geometry. The following table gives a summary of their different characteristics.

CASE	Characteristics	
CASE 1	Unsteady flow, additionnal passive scalar, output management	
CASE 2	Same as case 1 with time dependent boundary conditions,	
CASE 2	fluid density depending on the temperature and calculation restart	
CASE 3	Same as case 2 with head losses, parallelism and spatial average	

Table II.2: Summary of the different calculations

Remark: In this case, you must add three meshes which have to be joined. In order to join the three

¹Which makes temperature a passive scalar ... but it is only for simplification purposes.



Figure II.2: View of the full domain mesh with zoom on the joining regions

meshes, you must add a selection criteria in the box Selection criteria. In this case, only faces of colors 5, 24 and 32 are liable to be joined (different colors can be entered on a single line, separated by comma).

You can verify the quality of your mesh by running a **Mesh quality criteria** computation, which you can access through **Run type** in the **Prepare batch calculation** heading.

2 CASE 1: Passive scalar with various boundary conditions and output management

2.1 Calculation options

Some options are similar to those of the simple_junction tutorial:

- \rightarrow Turbulence model: $k-\epsilon$
- $\rightarrow~{\rm Scalar(s):}$ 1 temperature
- $\rightarrow\,$ Physical properties: uniform and constant

The new options are:



Figure II.3: Colors of the boundary faces

- \rightarrow Flow type: unsteady flow
- $\rightarrow\,$ Time step: uniform and constant
- \rightarrow Scalar(s): 2 passive scalar², with diffusion coefficient 8.55 (×10⁻⁵ m².s⁻¹)
- \rightarrow Management of monitoring points

2.2 Initial and boundary conditions

 \rightarrow Initialization: 20°C for temperature 10 for the passive scalar

The boundary conditions are defined in the user interface and depend on the boundary zone.

- Flow inlet: Dirichlet condition, an inlet velocity of $1 m \cdot s^{-1}$, an inlet temperature of 300°C and an inlet value of 200 for the passive scalar are imposed
- **Outlet**: default value
- Walls: velocity, pressure and thermal scalar: default value passive scalar: different conditions depending on the color and geometric parameters

In order to test the ability to specify boundary condition regions in the Graphical Interface, various conditions will be imposed for the passive scalar, as specified in the following table:

²It could correspond to a tracer concentration for instance.

Wall	Nature	Value
wall_1	Imposed value (Dirichlet)	0
wall_2	Imposed value (Dirichlet)	5
wall_3	Imposed value (Dirichlet)	0
wall_4	Imposed value (Dirichlet)	25
wall_5	Imposed value (Dirichlet)	320
wall_6	Imposed value (Dirichlet)	40

The wall_1 to wall_6 regions are defined as follows, through color references and geometric localization:

Label	Color and geometric parameters
wall_1	24 and $0.1 \leq x$ and $x \leq 0.5$
wall_2	2 or 3
wall_3	4 or 7 or 21 or 22 or 23
wall_4	6 and y > 1
wall_5	6 and $y \leq 1$
wall_6	31 or 33

Figure II.3 shows the colors used for boundary conditions and table II.3 defines the correspondance between the colors and the type of boundary condition to use.

Colors	Conditions
1	Inlet
34	Outlet
2 3 4 6 7 21 22 23 24 31 33	Wall
8 9 28 29 38 39	Symmetry

Table II.3: Boundary faces colors and associated references

2.3 Parameters and User routines

All parameters necessary to this study can be defined through the Graphical Interface without using any user Fortran files.

Calculation control par	ameters
Pressure-Velocity coupling	SIMPLEC algorithm
Number of iterations	300
Reference time step	0.05
Output period for post-processing files	2

In order to join the separate meshes into a single domain, colors 5, 24 and 32 will have to be joined through the Graphical Interface.

2.4 Output management

In this case, different aspects of output management will be addressed.

By default, in the Graphical Interface, all variables are set to appear in the listing, the post-processing and the chronological records. This default choice can be modified by the user.

In this case, the **Pressure**, the **Turbulent energy** and the **Dissipation** will be removed from the listing file.

The **Courant number** (CFL) and **Fourier number** will be removed from the post-processing results³.

Eventually, probes will be defined for chronological records, following the data given in figure II.4. Then the **total pressure** will be deactivated for all probes.



Probe n^o .	x (m)	y (m)	z (m)
1	-0.25	2.25	0
2	0.05	2.25	0
3	0.05	2.75	0
4	0.05	0.5	0
5	0.05	-0.25	0
6	0.75	-0.25	0
7	0.75	0.25	0
8	0.75	0.75	0

Figure II.4: Position and coordinates of probes in the full domain

In addition the domain boundary will be post-processed. This allows to check the boundary conditions, and especially that of the passive scalar.

2.5 Results

Figure II.5 shows the boundary domain colored by the passive scalar boundary conditions. The different regions of boundary conditions defined earlier can be checked.

Figure II.6 presents results obtained at different times of the calculation. They were plotted from the post-processing files, with ParaView.

 $^{^{3}}$ This can be very useful to save some disk space if some variables are of no interest, as post-processing files can be large.



Figure II.5: View of the boundary domain colored by the scalar2 variable - Case 1



Figure II.6: Water velocity field colored by temperature at different time steps - Case 1

3 CASE 2: Time dependent boundary conditions and variable fluid density

In this case some boundary conditions will be time dependent and some physical characteristics of the fluid will be dependent on the temperature.

Remark: You can copy your case1 in order to make the case2:

\$ code_saturne create --copy-from case1 case2

3.1 Calculation options

The options for this case are the same as in **case1**, except for the variable fluid density:

- \rightarrow Flow type: unsteady flow
- $\rightarrow\,$ Time step: uniform and constant
- \rightarrow Turbulence model: $k \epsilon$
- \rightarrow Scalar(s): 1 temperature
 - 2 passive scalar
- \rightarrow Physical properties: uniform and constant (except density)
- \rightarrow Management of monitoring points

3.2 Initial and boundary conditions

 \rightarrow Initialization: 20°C for temperature 10 for the passive scalar

The boundary conditions are defined in the user interface and depend on the boundary zone. The time dependence of the temperature boundary condition implies the use of a Fortran user routine (see below).

- Flow inlet: Dirichlet condition, an inlet velocity of $1 m.s^{-1}$, a time dependent inlet temperature and a value of 200 for the passive scalar are imposed;
- **Outlet**: default value;
- Walls: velocity, pressure and thermal scalar: default value passive scalar: different conditions depending on the color and geometric parameters.

The boundary conditions for the passive scalar are identical as those in **case1**, as specified in the following table:

Wall	Nature	Value
wall_1	Imposed value (Dirichlet)	0
wall_2	Imposed value (Dirichlet)	5
wall_3	Imposed value (Dirichlet)	0
wall_4	Imposed value (Dirichlet)	25
wall_5	Imposed value (Dirichlet)	320
wall_6	Imposed value (Dirichlet)	40

The wall_1 to wall_6 regions are defined as follows, through color references and geometric localization:

Figure II.3 shows the colors used for boundary conditions and table II.4 defines the correspondance between the colors and the type of boundary condition to use.

Label	Color and geometric parameters
$wall_1$	24 and $0.1 \leq x$ and $x \leq 0.5$
wall_2	2 or 3
wall_3	4 or 7 or 21 or 22 or 23
wall_4	6 and y > 1
wall_ 5	6 and $y \leq 1$
wall_6	31 or 33

Colors	Conditions
1	Inlet
34	Outlet
2 3 4 6 7 21 22 23 31 33	Wall
24 for $0.1 \le x \le 0.5$	Wall
8 9 28 29 38 39	Symmetry

Table II.4: Boundary faces colors and associated references

3.3 Variable Density

In this case the density is a function of the temperature. The variation law is defined in the Graphical User Interface, although it can also be defined in a Fortran user routine. The expression is:

$$o = T(AT + B) + C \tag{II.1}$$

where ρ is the density, T is the temperature, $A = -4.0668 \times 10^{-3}$, $B = -5.0754 \times 10^{-2}$ and C = 1000.9.

In order for the variable density to have an effect on the flow, gravity must be set to a non-zero value. $g = -9.81 \underline{e}_{\mu}$ will be specified in the Graphical Interface.

Remark:

The temperature is temperature in the user expression. Don't forget ; at the end of the expression.

3.4 Parameters

The calculation parameters are identical as those in case1.

All the parameters necessary to this study can be defined through the Graphical Interface, except the time dependent boundary conditions that have to be specified in user routines.

In order to join the separate meshes into a single domain, colors 5, 24 and 32 will have to be joined through the Graphical Interface.

3.5 User routine

The routine cs_user_boundary_conditions.f90 has to be copied from the folder \boxdot SRC/REFERENCE into the folder \boxdot SRC⁴.

⁴Only when it appears in the \boxdot SRC directory will it be taken into account by the code.

Parameters of calculation control	ol
Number of iterations	300
Reference time step	0.05
Output period for post-processing files	2

• cs_user_boundary_conditions.f90

This routine allows to define advanced boundary conditions on the boundary faces.

Even if cs_user_boundary_conditions.f90 is used, all boundary conditions have to be defined in the Graphical User Interface (GUI). Only the conditions that differ from this first definition need to appear in cs_user_boundary_conditions.f90. The boundary conditions defined in cs_user_boundary_conditions.f90 will replace those specified in the Graphical Interface.

In this case, the temperature at entry is supposed variable in time, following the law:

$$\begin{cases} T = 20 + 100t & \text{for } 0 \le t \le 3.8\\ T = 400 & \text{for } t > 3.8 \end{cases}$$
(II.2)

where T is the temperature in $^{\circ}$ C and t is the time in seconds (s).



Figure II.7: Time evolution of the temperature at inlet.

Remark:

ttcabs is the current physical time. See the example file in the subdirectory \bigcirc SRC/EXAMPLES for the complete cs_user_boundary_conditions.f90 file.

3.6 Output management

The output management is the same as in **case1**, except that a nineth monitoring point will be added, just at the entry, to monitor the temperature evolution at inlet.

In this case, the **Pressure**, the **Tubulent Energy** and the **Dissipation** will be removed from the listing file.

The **Courant number** (CFL) and **Fourier number** will be removed from the post-processing results⁵.

Eventually, probes will be defined for chronological records, following the data given in figure II.8. Then the **total_pressure** will be deactivated from all probes.

In addition the domain boundary will be post-processed. This allows to check the boundary conditions, and especially that of the temperature and passive scalar.

 $^{^{5}}$ This can be very useful to save some disk space if some variables are of no interest, as post-processing files can be large.



Figure II.8: Position and coordinates of probes in the full domain

3.7 Calculation restart

After the first run, the calculation will be continued for another 400 time steps. The calculation restart is managed through the Graphical Interface.

3.8 Results

Figure II.9 shows the time evolution of temperature recorded on each monitoring probe.



Figure II.9: Time evolution of temperature at monitoring probes - Case 2

Figure II.10 shows the velocity fields colored by temperature in the first run of calculation. Figure II.11 shows the velocity fields in the second calculation (restart of the first one).



Figure II.10: Water velocity field colored by temperature and inlet temperature value at different time steps (first calculation) - Case 2



Figure II.11: Water velocity field colored by temperature at different time steps (second calculation) - Case 2

4 CASE 3: Head losses, parallelism and spatial average

This case will be run in parallel on two processors. Head losses will be used to simulate the presence of an obstacle in the flow and the spatial average of the temperature will be calculated at each time step.

4.1 Calculation options

The options for this case are the same as in case2:

- \rightarrow Flow type: unsteady flow
- $\rightarrow\,$ Time step: uniform and constant
- \rightarrow Turbulence model: $k \epsilon$
- → Scalar(s): 1 temperature 2 - passive scalar, with diffusion coefficient 8.55 (×10⁻⁵ $m^2.s^{-1}$)
- \rightarrow Physical properties: uniform and constant (except density)
- \rightarrow Management of monitoring points

4.2 Initial and boundary conditions

 \rightarrow Initialization: 20°C for temperature

10 for the passive scalar

The boundary conditions are defined in the user interface and depend on the boundary zone.

- Flow inlet: Dirichlet condition, an inlet velocity of $1 m \cdot s^{-1}$ and a time dependent inlet temperature and a value of 200 for the passive scalar are imposed
- Outlet: default value
- Walls: velocity, pressure and thermal scalar: default value passive scalar: different conditions depending on the color and geometric parameters

The boundary conditions for the passive scalar are identical as those in **case2**, as specified in the following table:

Wall	Nature	Value
wall_1	Imposed value (Dirichlet)	0
wall_2	Imposed value (Dirichlet)	5
wall_3	Imposed value (Dirichlet)	0
wall_4	Imposed value (Dirichlet)	25
wall_5	Imposed value (Dirichlet)	320
wall_6	Imposed value (Dirichlet)	40

The **wall_1** to **wall_6** regions are defined as follows, through color references and geometric localization:

Label	Color and geometric parameters
wall_1	24 and $0.1 \leq x$ and $x \leq 0.5$
wall_2	2 or 3
wall_3	4 or 7 or 21 or 22 or 23
wall_4	6 and y > 1
wall_5	6 and $y \leq 1$
wall_6	31 or 33

Figure II.3 shows the colors used for boundary conditions and table II.5 defines the correspondance between the colors and the type of boundary condition to use.

Colors	Conditions
1	Inlet
34	Outlet
2 3 4 6 7 21 22 23 31 33	Wall
24 for $0.1 \le x \le 0.5$	Wall
8 9 28 29 38 39	Symmetry

Table II.5: Boundary faces colors and associated references

4.3 Variable Density

The law for the variable density is identical as that in case2.

In this case the density is a function of temperature, the variation law is defined in the Graphical User Interface although it can also be defined in a Fortran user routine. The expression is:

$$\rho = T(AT + B) + C \tag{II.3}$$

where ρ is the density, T is the temperature, $A = -4.0668 \times 10^{-3}$, $B = -5.0754 \times 10^{-2}$ and $C = 1\,000.9$.

In order for the variable density to have an effect on the flow, gravity must be set to a non-zero value. $g = -9.81 \underline{e}_{y}$ will be specified in the Graphical Interface.

4.4 Head losses

To simulate the presence of an obstacle $0.20 \ (m)$ large and $0.5 \ (m)$ high in the vessel, a zone of head losses will be created in the domain (fig II.12).

The head losses zone is located between the coordinates x = 0.2 (m) and x = 0.4 (m), and y = -0.75 (m) and y = -0.25 (m).

The head losses coefficient to apply is $K_{11} = K_{22} = K_{33} = 10^4 = \frac{1}{2} \alpha_{11} = \frac{1}{2} \alpha_{22} = \frac{1}{2} \alpha_{33}$ and is isotropic.

4.5 Parameters

All the parameters necessary to this study can be defined through the Graphical Interface. However, the calculation of the spatial average is defined by a user routine.

Parameters of calculation cont	trol
Number of iterations	900
Reference time step	0.01
Output period for post-processing files	2
The calculation will be run in parallel	2 procs.

In order to join the separate meshes into a single domain, colors 5, 24 and 32 will have to be joined through the Graphical Interface.

Note that the time step has been reduced because of the head losses: the pressure step is more difficult to be solved in presence of head losses.



Figure II.12: Full domain geometry with the obstacle

4.6 User routines

The following routines have to be copied from the folder \boxdot SRC/REFERENCE/ into the folder \boxdot SRC/⁶: cs_user_boundary_conditions.f90 and cs_user_extra_operations.f90. We can find and copy some basic and specific boundary conditions examples in the folder \boxdot SRC/EXAMPLES/ to correctly impose the *Code_Saturne* boundary conditions.

$\bullet \ cs_user_boundary_conditions.f90$

This routine allows to define advanced boundary conditions on the boundary faces.

Even if cs_user_boundary_conditions.f90 is used, all boundary conditions have to be defined in the Graphical User Interface (GUI). Only the conditions that differ from this first definition need to appear in cs_user_boundary_conditions.f90. The boundary conditions defined in cs_user_boundary_conditions.f90 will replace those specified in the Graphical Interface.

In this case, the temperature at entry is supposed variable in time, following the law:

$$\begin{cases} T = 20 + 100t & \text{for } 0 \le t \le 3.8\\ T = 400 & \text{for } t > 3.8 \end{cases}$$
(II.4)

where T is the temperature in $^{\circ}$ C and t is the time in seconds (s).

• cs_user_extra_operations.c

This routine is called at the end of each time step and has access to the whole set of variables of the code. It is therefore useful for many user-specific post-processing, including the calculation of a spatial average in the present case.

The spatial average of the temperature will be calculated at each time step and the result wrote in a file named moy.dat. The values are saved in order to draw the time evolution of the average temperature.

Beware when calculating the average. Since the calculation is running in parallel, computing the sum of the temperatures on **all the cells** will only yield for each processor to the sum on the cells managed by this processor. In order to obtain the full sum, the parallelism routine cs_parall_sum must be used (see example in the cs_user_extra_operations-scalar_balance.c routine).

⁶Only when they appear in the SRC directory will they be taken into account by the code.

Remark: $cs_user_extra_operations_xxx.c$ are different example routines present in the subdirectory \boxdot SRC/EXAMPLES. They should be removed from the \boxdot SRC/ before running the case.

4.7 Output management

The output management is the same as in case2.



Probe n^o .	\mathbf{x} (m)	\mathbf{y} (m)	\mathbf{z} (m)
1	-0.25	2.25	0
2	0.05	2.25	0
3	0.05	2.75	0
4	0.05	0.5	0
5	0.05	-0.25	0
6	0.75	-0.25	0
7	0.75	0.25	0
8	0.75	0.75	0
9	-0.5	2.25	0

Figure II.13: Position and coordinates of probes in the full domain

In this case, the **Pressure**, the **Tubulent Energy** and the **Dissipation** will be removed from the listing file.

The **Courant number** (CFL) and **Fourier number** will be removed from the post-processing results⁷.

Eventually, probes will be defined for chronological records, following the data given in Figure II.4. Then the **total pressure** will be deactivated from all probes.

In addition the domain boundary will be post-processed. This allows to check the boundary conditions, and especially that of the temperature and passive scalar.

4.8 Results

Figure II.14 shows the evolution of the spatial average of the temperature.

Figure II.15 shows velocity fields colored by temperature. The effect of the head loss modeling the obstacle is clearly visible.

 $^{^{7}}$ This can be very useful to save some disk space if some variables are of no interest, as post-processing files can be large.



Figure II.14: Evolution of the spatial average of the temperature as a function of time - Case 3



Figure II.15: Water velocity field colored by temperature - Case 3 $\,$

Part III

Step by step solution

1 Solution for CASE1

This case corresponds to a new study, in which there will be three calculation cases (cases 1, 2 and 3). We can create one case in a single code_saturne create command and additional cases can be added later. To test this functionality, first create the study directory, with case subdirectory \bigcirc case1, as below:

\$ code_saturne create -s full_domain -c case1
\$ cd full_domain

Go to the \boxdot DATA directory in \boxdot case1, open a new case and select the meshes to use.

In the preprocessing mode, click on the heading **Calculation environment** then on the **Meshes** selection item. In this case, you must add three meshes which have to be joined.

In order to join the three meshes, you must add a selection criteria in the box Selection criteria of the Face joining (optional) item. In this case, only faces of colors 5, 24 and 32 can be joined (different colors can be entered on a single line, separated by comma).

Click on the + icon to enter the list of colors to be joined in the Face joining (optional) item.

dy: full_domain					
					_
casei					
file: /home/TM178B0N/Code_Satu	rne/Tutorials/full_domain/cas	se1/DATA/case1.xml			
Ø 🛛					
ldentity and paths	Meshes Meshes opti	ons Periodic Boundaries			
Calculation environment					
Meshes selection	Mesh import	A			
Calculation control	 Import meshes 	Use existing mesh in	put		
Calculation management	Local mesh direct	ory (optional)			
	/MESH				
					9
	List of meshes				
	File name	Format	Reorient	Path	
	downcomer.des	Simail/NOPO			
	fdc.des	Simail/NOPO			
	pic.des	Simail/NOPO			
		4			
	Face joining (optional)			
	Fraction Plane V	erbosity Visualization	Sele	ction criteria	_
		croosicy risualization	bette		

Figure III.1: Meshes: list of meshes

You can now verify the quality of your mesh by running a Mesh quality criteria calculation.

dy: full_domain						
e: casel						
L file: /home/TM178B0N/Code_Satu	ırne/Tutorials/fi	ull_domain/case1,	/DATA/case1.xml			
		·	[]			
Identity and paths	Meshes	Meshes options	Periodic Boundaries			
Meshes selection	Mesh ir	nport				
Calculation control	• Ir	nport meshes 🤇	Use existing mesh inp	ut		
🛅 Calculation management			(
	Loc	al mesn directory	(optional)			.
	/N	1ESH				١
	List	of meshes				
		File name	Format	Reorient	Path	
	do	wncomer.des	Simail/NOPO			
		fdc.des	Simail/NOPO			
		pic.des	Simail/NOPO			
			-t-	_		
	Face joi	ning (optional)				
	Fra	ction Plane Verb	osity Visualization		Selection criteria	
	0,1	25 1	1 5,2	4,32		

Figure III.2: Join a mesh

Select the calculation mode, and click on the **Calculation features** item in the **Thermophysical models**. In this case **unsteady flow** must be selected.

The rest of the heading **Thermophysical models** is identical to **simple_junction**.

Study: full_domain Case: case1 XML file: /home/TM178B0N/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml
Study: full_domain Case: case1 XML file: /home/TM178BON/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml
Study: full_domain Case: case1 XML file: /home/TM178BON/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml
Study: Introductionality Case: case1 XML file: /home/TM178BON/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml
Case: case1 XML file: /home/TM178BON/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml
XML file: /home/TM178BON/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml
Script file
Calculation environment
Calculation control
Calculation management Calculation script parameters
B Performance tuning Run type Mesh quality criteria
Prepare batch calculation
Runia
Number of processes 1
Threads per task 0
Advanced options
Calculation start
Start calculation

Figure III.3: Mesh quality criteria

To add an additional scalar, click on the **Species transport** item under the **Thermophysical models** heading.

<u>File E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
📄 🗁 🙆 💽 🤌 🙋	🔲 🗾 🦗			S
				0 *
Study: full_domain				
Case: casel				
XML file: /home/TM178B0N/Code_Saturne	/Tutorials/full_domain/			
Ane me. (mome) mail obord/code_batame,	, raconats, rate_domain,	case1/DATA/case1.Am		
6 X (
Identity and paths	Steady/Unsteady flo	w algorithm		
Calculation environment		unsteady flow	~	
Thermophysical models		/		
Calculation features	Eulerian-Lagrangian	multi-phase treatment		
🗒 Deformable mesh		off	~	
🗒 Turbulence models				
📑 Thermal model	Atmospheric flows			
🗟 Radiative transfers		off	V	
🗎 Conjugate heat transfer				
📄 Species transport	Gas combustion			
📑 Turbomachinery		off	~	
📑 Fans				
🕀 🛅 Physical properties	Pulverized fuel com	bustion		
🕀 🛅 Volume conditions	of	f		~
🕀 🛅 Boundary conditions				
🕀 🛅 Numerical parameters	Electrical models			
Calculation control		off	~	
🕀 🛄 Calculation management				
	Compressible model			
		off	~	
	groundwater flows			
		off	~	
· · · · · · · · · · · · · · · · · · ·				
Click right for context menu				

Figure III.4: Thermophysical models/Calculation features: unsteady flow

The heading **Physical properties** is identical to **simple_junction**, except for the new scalar.

In the **Fluid properties** item, still under the heading **Physical properties**, specify the diffusion coefficient of this new scalar.

Click on the scalar name to highlight it, then enter the value in the box. In this case, the species diffusion coefficient value is 8.55 (×10⁻⁵ $m^2 s^{-1}$) for the scalar2 scalar to solve.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📑 📾 🖎 🔕 📭 👆 🕐 🗄		5
Study: full_domain		
Case: casel		
XML file: /home/TM178B0N/Code_Saturne	/Tutorials/full_domain/case1/DATA/case1.xml	
ð X		
ldentity and paths	Species transport	
+ Calculation environment	Name	Turbulent flux model
Thermophysical models	temperatura	S CDH
Calculation features	temperature	SGDH
Deformable mesh	scalar2	SGDH
📑 Turbulence models		
📑 Thermal model		
Radiative transfers		
🕞 Conjugate heat transfer		
Species transport	,	
📑 Turbomachinery	5°	
📑 Fans	N	
🕀 🛅 Physical properties	Variance of Species	
🕀 🛅 Volume conditions	Variance	Species_Name
🕀 🛅 Boundary conditions		
🕀 🛅 Numerical parameters		
🕀 🛅 Calculation control		
🕀 🛄 Calculation management		
	수 ·	-
· · · · · · · · · · · · · · · · · · ·		
Click right for context menu		

Figure III.5: Additional scalar

Initialization:

To initialize variables at the instant t = 0 s, go to the **Initialization** item under the heading **Volume** conditions.

Here the velocity, the thermal scalar and the turbulence can be initialized. In this case, the default values to be set are: zero velocity (default), an initial temperature of 20° C and a turbulence level based on a reference velocity of $1 \ (m.s^{-1})$ (default). You must also initialize the scalar2 species at 10° C.

Specific zones can be defined with different initializations. In this case, only the default **all_cells** is used.

		만 1
Study: full_domain		
Case: case1		
KML file: /home/TM178B0N/Code_Satur	rne/Tutorials/full_domain/case1/DATA/case1.xml	
đ		۸
🗎 Identity and paths		
🕀 🛅 Calculation environment	Density	
🕀 🛅 Thermophysical models	constant 🗸 🖓	
🗆 🛅 Physical properties	Deference unlug 0 725 725	
Reference values	Reference value p 723.735 kg/m	
Fluid properties	Viscosity	
Gravity		
Volume conditions	constant 🛛	
Boundary conditions	Reference value µ 8.951e-05 Pa.s	
Numerical parameters		
	Specific heat	
	constant 🗸 🕅	
	Reference value Cp 5483.0 J/kg/K	
	Thermal conductivity	
	constant 🗸 🖓	
	Diffusion coefficient of species	
	Name scalar?	
	Name scatarz *	
	constant 💌 🖉	
	reference 8 550-05 m ² /s	

Figure III.6: Fluid properties

Create the boundary zones: The procedure is the same as in simple_junction, but the colors are different. Note that colors 5 and 32 have completely disappeared in the joining process (they are now internal faces and are not considered as boundaries), while some boundary faces of color 24 remain.

Create the inlet, outlet and symmetry boundary zones with the following colors:

- Inlet : ''1'' - Outlet : ''34'' - Symmetry: ''8 or 9 or 28 or 29 or 38 or 39''
| <u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp | | |
|---|---|-----|
| i 🗂 🚍 🕭 🔕 📭 i 🥱 🕐 i | III 🗾 😳 | 5 |
| | | |
| | | e & |
| Study: full_domain | | |
| Case: | | |
| | | |
| XML file: /home/TM178B0N/Code_Saturne | /Tutorials/full_domain/case1/DATA/case1.xml | |
| | | |
| | | |
| ldentity and paths | Initialization | |
| 🕀 🛅 Calculation environment | Volume zone all_cells 🗸 | |
| 🕀 🛅 Thermophysical models | | |
| 🕀 🛅 Physical properties | Velocity 🥺 | |
| 🗆 🛅 Volume conditions | | |
| Volume regions definition | Thermal 🔧 | |
| Initialization | Turbulance Initialization to enforce on the (a) and | |
| 📑 Coriolis Source Terms | Iurbulence Initialization by reference value(s) * | |
| 🕀 🛅 Boundary conditions | Species scalar2 🗸 | |
| 🕀 🛅 Numerical parameters | | |
| 🕀 🛅 Calculation control | | |
| 🕀 🛅 Calculation management | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| < | | |
| | | |

Figure III.7: Initialization

In this case, different conditions are applied for the walls. Separate corresponding wall boundary regions must therefore be created, following the data in the following table.

Label	Zone	Nature	Selection criteria
Wall_1	4	Wall	24 and $0.1 \leq x$ and $x \leq 0.5$
Wall_2	5	Wall	2 or 3
Wall_3	6	Wall	4 or 7 or 21 or 22 or 23
Wall_4	7	Wall	6 and y > 1
Wall_5	8	Wall	6 and $y \leq 1$
Wall_6	9	Wall	31 or 33

The Wall_1 region combines color and geometrical criteria. The associated character string to enter in the Selection criteria box 1 is as follows:

¹Note that, due to the joining process, there are in fact no boundary faces of color 24 with x coordinate outside the [0.1;0.5] interval. The geometrical criterium is therefore not necessary. It is presented here to show the capabilities of the face selection module.

File Edit Tools Window Help	H H H G H H G H H G H	S N
Study: full_domain		
Case: casel		
XML file: /home/TM178B0N/Code_S	Mathematical expression editor	
œ	User expression Predefined symbols Examples	
 Identity and paths Calculation environment Thermophysical models Physical properties Volume conditions Volume regions definition Initialization Coriolis Source Terms Boundary conditions Numerical parameters Calculation control Calculation management 	scalar2 = 10.;	

Figure III.8: Initialization: species

''24 and 0.1 <= x and 0.5 >= x''

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp)			0 8
Study:	full_domain					
Case:	casel					
XML file:	/home/TM178B0N/Code_Sat	urne/Tutorials/full_d	omain/ca	ase1/DATA/	case1.xml	
	0 🗵	6				
📑 Ide	entity and paths	Definition of	boundar	y regions		
🕂 🛅 Ca	lculation environment	Label	Zone	Nature	Selection criteria	
🕀 🛅 Th	ermophysical models	Inlet	1	Inlet	1	
	lysical properties	Outlet	2	Outlet	34	
E Bo	undary conditions	Symmetry	, 3	Svmmetrv	8 or 9 or 28 or 29 or 38 or 39	
	Definition of boundary regi			-,,		
	Boundary conditions					
🕀 🛅 Nu	Imerical parameters					
	loulation control					
	actuation management					
					Add Delete	
		Add from P	Febroce	ssor listing		
		Add Hollin	Iepioce	mport group	os and references from Preprocessor listing	
<						

Figure III.9: Creation of the boundary zones

Define the other wall boundary zones. The faces of color 6 have to be divided in two separate zones, based on a geometrical criterium on y.

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp					
	/ 🔛 🥹 📴 : 🍤 (° : :	🖿 💟 : 🖤				. 🔾 😧
Study:	full_domain					
Case:	casel					
XML file:	/home/TM178B0N/Code_Saturne	/Tutorials/full_do	main/ca	ase1/DATA/0	casel.xml	
	0 8					
📑 Ide	entity and paths	Definition of b	oundar	y regions		
🗄 🎦 Ca	lculation environment	Label	Zone	Nature	Selection criteria	
🕀 📔 Th	ermophysical models	Inlet	1	Inlet	1	
🕀 🔛 Ph	ysical properties	Outlet	2	Outlet	34	
	undary conditions	Symmetry	3	Symmetry	8 or 9 or 28 or 29 or 38 or 39	
	Definition of boundary regi	Wall_1	4	Wall	24 and 0.1 <= x and 0.5 >= x	
	Boundary conditions				'	
🕀 🎦 Nu	umerical parameters					
🕀 🔛 Ca	Iculation control					
	liculation management					
					Add Delete	
		Add from Pr		ssor listing		
		Additioniti	-proce	mport grour	os and references from Preprocessor listing	
				inport group	s and references from r reprocessor as ang	
	,					

Figure III.10: Creation of a wall boundary region

The dynamic boundary conditions are the same as in simple_junction for the inlet, and there are still no sliding walls.

- Inlet:

ıdy: full_domain					
se: casel					
L file: /home/TM178B0N/Code_Sat	turne/Tutorials/full_d	omain/c	ase1/DATA/	'casel.xml	
0	X				
ldentity and paths	Definition of I	boundar	ry regions		
Calculation environment	Label	Zone	Nature	Selection criteria	^
Thermophysical models	- ,	-	- ,,		
Physical properties Volume conditions	Wall_1	4	Wall	24 and 0.1 <= x and 0.5 >= x	
Boundary conditions	Wall_2	5	Wall	2 or 3	
Definition of boundary regi	Wall_3	6	Wall	4 or 7 or 21 or 22 or 23	
Boundary conditions	Mall 4	7	14/-11	6 and up 1	
Calculation control	Wall_4	/	wall		
Calculation management	Wall_5	•	wall	0 and y <- 1	
	vvall_0	9	vvall	31 of 33	v
				Add Delete	
	Add from P	гергосе	essor listing		
			Import grou	ps and references from Preprocessor listing 🛛 🔚	

Figure III.11: Creation of wall boundary regions

- Outlet:

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📄 🗁 🖄 🔇 📭 🅱 🖻		S
		0 ×
Study: full_domain		
C		
Case: Casei		
XML file: /home/TM178BON/Code_Saturn	e/Tutorials/full_domain/case1/DATA/case1.xml	
0 ×		
A Identity and paths	Velocity	^
+ Calculation environment	norm v 10 m/s	
🕀 🔛 Physical properties	Direction specified coordinates 🗸 🖓	
🕀 📔 Volume conditions		
🗆 🛅 Boundary conditions	X 1.0 Y 0.0 Z 0.0	
Definition of boundary regi		
Boundary conditions	Turbulance	
🕀 🛅 Numerical parameters	Calculation by bydraulic diameter V	
Calculation control		
Calculation management		
	Hydraulic diameter 0.5 m	
	I hermal	
	Type Prescribed value	
	temperature Value 300.0	
	Species	
	Type Prescribed value 🗸 🖅	
	scalar2 V	
	value 200.0	
c >		v

Figure III.12: Dynamic variables boundary: inlet

To configure the scalar boundary conditions on the walls, select individually each wall in the **Boundary conditions** item.

On all the walls, a default homogeneous **Prescribed flux** is set for temperature, and **Prescribed value** is specified for the passive scalar for each wall, named **scalar2**, according to the following table:

Wall	Nature	Scalar2 value
Wall_1	Prescribed value (Dirichlet)	0
Wall_2	Prescribed value (Dirichlet)	5
Wall_3	Prescribed value (Dirichlet)	0
Wall_4	Prescribed value (Dirichlet)	25
Wall_5	Prescribed value (Dirichlet)	320
Wall_6	Prescribed value (Dirichlet)	40

- Tutte domain						
casel						
ile: /home/TM178B0N/Code_Satur	ne/Tutorials/full_d	omain/case1/DATA	/case1.xm	ıl		
0 🗙			199999999999999			
Identity and paths	Boundary cone	litions				
Calculation environment	Label	Zone	Nature		Selection criteria	-
Thermophysical models	Inlet	1	inlet		1	-1
Physical properties	Outlet	2	outlet		34	
Volume conditions	Wall_1	4	wall		24 and 0.1 <= x and 0.5 >= x	
Boundary conditions	Wall_2	5	wall		2 or 3	
Definition of boundary regi	Wall_3	б	wall		4 or 7 or 21 or 22 or 23	
Boundary conditions	Wall_4	7	wall		б and y > 1	
Numerical parameters	Wall_5	8	wall		б and у <= 1	
Calculation control						
Calculation management	Thermal f	or backflow				
				Туре	Prescribed flux	
		temperature 🗸		Flux	0.0	
				T LUX	0.0	
	Species fo	r backflow				
				Type	Prescribed flux	
				· / F = [
		scatarz 👻		Flux	0.0	

Figure III.13: Dynamic variables boundary: outlet

Some calculation parameters need to be defined now. Go to the **Global parameters** item under the heading **Numerical parameters**. In our case the **Velocity-Pressure algorithm** is **SIMPLEC**.

se: case1				
se. cuber				
L file: /home/TM178B0N/Cod	e_Saturne/Tutorials/full_do	omain/case1/DA7	A/case1.xml	
	0 ×			
	Label	Zone	Nature	Selection criteria
Calculation environment	Inlet	1	inlet	1
Thermophysical models	Outlet	2	outlet	34
Physical properties	Wall_1	4	wall	24 and 0.1 <= x and 0.5 >= x
Volume conditions	Wall_2	5	wall	2 or 3
Boundary conditions	Wall_3	6	wall	4 or 7 or 21 or 22 or 23
Definition of boundary r	egi Wall_4	7	wall	6 and y > 1
Boundary conditions	Wall_5	8	wall	6 and y <= 1
Calculation control Calculation management	Smooth or	rough wall	⊙ smoot	n wall 🔘 rough wall
	Sliding	wall		
	Thermal			
			Tvi	e Prescribed flux
			. / 1	
		temperature		
		temperature	Flu	x 0.0
	Species	temperature	Flu	x 0.0
	Species	temperature	Flu	
	Species	temperature	Flu	x 0.0

Figure III.14: Scalars boundaries: wall_5

Go to the **Equation parameters** item under the heading **Numerical parameters**. You can define the maximum and minimum value for the **temperature** and for the **scalar2** scalars.

ly: full_domain		
casel		
file: /home/TM178B0N/Code_Sat	urne/Tutorials/full_domain/case1/DATA/case1.xml	
0 8		
ldentity and paths	Global parameters	
Calculation environment	Gradient calculation method:	
Thermophysical models	Iterative handling of non-orthogonalities	
Physical properties	Pseudo-coupled velocity-pressure solver	
Boundary conditions		
Numerical parameters	Handling of transposed gradient and divergence	
📄 Global parameters		
Equation parameters	Extrapolation of pressure gradient Neumann 1st order	
📑 Time step		
Calculation control		
Catculation management	Relaxation of pressure increase 1.0	
	Improved pressure interpolation in stratified flow $\hfill \square$	
	Velocity-Pressure algorithm SIMPLEC V	

Figure III.15: Time step setting

Go to the **Time step** item under the heading **Numerical parameters**. In our case the time step is **Constant**. Set the number of iterations to **300** and the reference time step to **0.05** (s).

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
i 📑 🔚 🏊 🔕 📭 i 🕤 🦿 i 🛛			5
			0 X
Study: full_domain			
Case: case1			
VML 5lay (bomo/TM178BON/Code, Saturno/	Tutorials/full_domain/case1/DATA/ca		
Arie lite. Minine/ Ini / Obola/Code_Saturle/		Set.XIIIt	
@ X			
Identity and paths	Solver Scheme Clipping		
🕀 🛅 Calculation environment			
🕀 🖺 Thermophysical models	Name	Minimal	Maximal
🕀 🛅 Physical properties		value	value
🕀 🔛 Volume conditions	temperature	0	400
Boundary conditions	scalar2	0	400
E Mumerical parameters			
Global parameters			
+ Calculation control			
Calculation management			
<	·		

Figure III.16: Clipping

Go to the **Output control** item under the heading **Calculation control** to set the output parameters. In the **Output Control** tab, keep the default value for the output listing frequency.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
- 🖻 🚘 🔊 🚳 📭 - 🗮			6
			V :
			ð 🗶
Study: full_domain			
Case: case1			
XML file: /home/TM178B0N/Code_Saturne/Tuto	rials/full_domain/case1/DATA/case1.xml		
0 ×			
A Identity and paths			
Calculation environment	Time step option Constant	~	
Thermophysical models			
+ Physical properties	Reference time step	0.05 s	
🕀 🔛 Volume conditions			
🕀 🛅 Boundary conditions	Number of iterations (restart included)	300	
🖃 🛅 Numerical parameters			
Global parameters	Option zero time step		
Equation parameters	1 1		
📄 Time step			
🕀 🔛 Calculation control			
🕂 🔛 Calculation management			

Figure III.17: Time step setting

For the post-processing, go to the **Writer** item and click on **results**.



Figure III.18: Output control: log frequency

Now you can select the third option in the **Frequency** (output every 'n' time steps) item and set the value of n to 2. By default, the boundary faces are selected.

udy: full_domain				
se: casel				
file: /home/TM178B0N/Code_Satur	ne/Tutorials/full_domain/cas	e1/DATA/case1.xml		
0 X				***************************************
ldentity and paths	Output Control Write	Mesh Particles mesh	Monitoring Points	
Calculation environment Thermophysical models	Name	Id	Format	Directory
Physical properties	results	-1	EnSight	postprocessing
Volume solution control Surface solution control Profiles				
Balance by zone Calculation management				
Balance by zone				

Figure III.19: Output control: writer

You can also choose the format. In this case, you will choose the EnSight format.

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp					
	i 🖄 🔕 📭 i 🖄 🖒 i i					S
						0
Study:	full_domain					
Case:	casel					
XML file:	/home/TM178B0N/Code_Saturne/	Tutorials/full_do	main/case1	/DATA/case1.xml		
		Output Contro	Writer	Mach Bartislas m	ash Manitaring Painta	
	entity and paths	Output Contro	vviitei	Particles m		
	ermophysical models	Nar	ne	ld	Format	Directory
🕀 🎦 PH	vsical properties	resu	lts	-1	EnSight	postprocessing
🕀 🔛 Va	Jume conditions					
🕀 🛅 Bo	undary conditions					
🕀 🛅 Ni	umerical parameters					
🗆 🛅 Ca	lculation control					
	Time averages					
	Output control					
	Volume solution control					
	Surface solution control				4	
B	Profiles				u	
B	Balance by zone	Frequency				
🕂 🗎 Ca	lculation management		Output e	very 'n' time steps	✓ 1	
			Cutpu	t at end of calculatio	n	
			_ outpu			
		Time-depend	ency			
		· · · ·		Fixed	mesh 🗸	
		Options				
				format	binary (native)	
				polygopa	dicalay	
				potygons	disptay 🖓	
				polyhedra	display 🗸	
<						

Figure III.20: Output control: results

Go to the **Mesh** tab.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	-				S 0 5
Study: full_domain					
Case: case1					
XML file: /home/TM178B0N/Code_Saturne,	Tutorials/full_d	omain/case1	l/DATA/case1.xml		
0 8 [
ldentity and paths	Output Conti	ol Writer	Mesh Particles m	esh Monitoring Points	
+ Calculation environment	output cont				
Thermophysical models	Na	me	ld	Format	Directory
Physical properties	res	ults	-1	EnSight	postprocessing
🕀 📔 Volume conditions					
🕀 🛅 Boundary conditions					
🕀 🛅 Numerical parameters					
🖃 🛅 Calculation control					
Time averages					
Output control					
Volume solution control					
Surface solution control				1 L	
🗎 Profiles				T	
Balance by zone	Frequency	No period	dic output		
🕀 🖺 Calculation management		Output e	very 'n' time steps	1	
		Output e	verv 'x' seconds		
		Output u	sing a formula		
	Timordopon	doncy	sing a ronnuta		
	rime-depen	uency			
			Fixed	mesh 🗡	
	Options				
	Options		r .		
			format	binary (native)	
			polygons	display 🗸	
			polytr-du-	dicelari	
			potynedra	disptay	
,,					

Figure III.21: Output control: frequency

You can click on the **Fluid domain** mesh **Name** item and new options will appear.

Study: full_domain Case: case1 XML file: /nome/TM178BON/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xmt	File Edit Tools <u>W</u> indow <u>H</u> elp	· · · · · · · · · · · · · · · · · · ·
Case: case1 XML file: /home/TM178B0N/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml Calculation environment Calculation environment Calculation environment Dutput Control Writer Mesh Particles mesh Monitoring Points Calculation environment Name Id Format Directory results 1 EnSight postprocessing MED CGNS Calculation control Coltput control Coltput control Volume solution control Surface solution control Profiles Balance by zone Calculation management Output every 'n' time steps 2 Control Time-dependency Fixed mesh v polygons display v	Study: full_domain	
XML file: [home/TM178B0N/Code_Saturee/Tutorials/full_domain/case1/DATA/case1.xml	Case: case1	
Identity and paths Calculation environment Thermophysical models Physical properties Volume conditions Boundary conditions Calculation control Calculation control Strate solution control Profiles Balance by zone Calculation management Output every 'n' time steps Output at end of calculation Time-dependency Fixed mesh Fixed mesh	XML file: /home/TM178B0N/Code_Saturn	ne/Tutorials/full_domain/case1/DATA/case1.xml
Identity and paths Calculation environment Thermophysical models Physical properties Volume conditions Boundary conditions Name Id Format Directory Calculation environment Id Format Directory Coutput Control MED Calculation control Catalyst Cotyput control CCM-IO Volume solution control Surface solution control Surface solution control Calculation management Profiles Frequency Output every 'n' time steps 2 Output at end of calculation Time-dependency Fixed mesh Fixed mesh Options	(X)	
Calculation environment Calculation environment Physical properties Volume conditions Boundary conditions Calculation control Calculation control Cottput control Volume solution control Surface solution control Balance by zone Calculation management Calculation management Cottput every 'n' time steps Cottput at end of calculation Time-dependency Frequency Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	ldentity and paths	Output Control Writer Mesh Particles mesh Monitoring Points
Image: Thermophysical models Image: Physical properties Image: Physical P	🕀 🔛 Calculation environment	
Physical properties Image: Physical properties Image: Physical properties Image: Physical parameters Image: Physical parameters Image: Physical	🗄 🖺 Thermophysical models	Name Id Format Directory
Volume conditions Wether MED Boundary conditions CGNS Numerical parameters Calculation control Time averages CCM-IO Output control CCM-IO Surface solution control Image: Control Profiles Image: Control Balance by zone Image: Control Output every 'n' time steps 2 Frequency Image: Control Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	🕀 🎦 Physical properties	results -1 EnSight postprocessing
Boundary conditions Boundary conditions Numerical parameters Calculation control Time averages Output control Volume solution control Surface solution control Profiles Balance by zone Calculation management Output every 'n' time steps Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	🕀 📔 Volume conditions	MED
Numerical parameters Calculation control Time averages Output control Volume solution control Surface solution control Balance by zone Frequency Output every 'n' time steps Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	🕀 🛅 Boundary conditions	CGNS
Calculation control CCM-IO CCM-II CCM	🕀 🛅 Numerical parameters	Catalyst
Time averages Output control Volume solution control Surface solution control Profiles Balance by zone Calculation management Output every 'n' time steps Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons	🖃 🛅 Calculation control	Catatyst
Output control Nolume solution control Surface solution control Profiles Balance by zone Calculation management Output every 'n' time steps Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	📄 Time averages	CCM-IO
Nolume solution control Surface solution control Profiles Balance by zone Calculation management Output every 'n' time steps v 2 Output at end of calculation Time-dependency Fixed mesh v Options format binary (native) polygons	📄 Output control	
Surface solution control Profiles Balance by zone Calculation management Output every 'n' time steps Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	🕞 Volume solution control	
Profiles Balance by zone Calculation management Output every 'n' time steps Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons	Surface solution control	+ -
Balance by zone Frequency	Profiles	
Calculation management Output every 'n' time steps 2 Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	Balance by zone	Frequency
✓ Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display	🕀 🎦 Calculation management	Output every 'n' time steps 🛛 🗸 2
Time-dependency Fixed mesh v Options format binary (native) v polygons display v		✓ Output at end of calculation
Time-dependency Fixed mesh v Options format binary (native) v polygons display v		
Fixed mesh V Options format binary (native) V polygons display V		Time-dependency
Options format binary (native) v polygons display v		Fixed mesh 🗸
Options format binary (native) polygons display		
format binary (native) V polygons display V		Options
polygons display 🗸		format binary (native) 🗸
porygons display		ashirana dialay
		potygons display
polyhedra display 🗸		polyhedra display 🗸
	<	

Figure III.22: Output control: format

You can associate a mesh with several writers (when created in the Writer tab).

<u>F</u> ile Edit <u>T</u> ools <u>W</u> indow <u>H</u> elp ☐ ☐ ☐ № ⊗ № 5 @							S :
Study: full_domain							
Case: case1							
XML file: /home/TM178B0N/Code_Saturne	/Tutorials/full_dom	ain/case	1/DATA/cas	e1.xml			
Ø X [
ldentity and paths	Output Control	Writer	Mesh P	articles mesh	Monitoring Points		
Calculation environment	Name	Id	Туре		Selection	Criteria	
+ Physical properties	Fluid domain	-1	cells	all[]	bettettion		
🕀 📔 Volume conditions	Boundary	-2 bou	ndarv faces	all[]			
+ 🛅 Boundary conditions			,				
Coloulation control							
Volume solution control							
Surface solution control							
🕒 Profiles							
Balance by zone							
Calculation management				4	-		

Figure III.23: Output control: mesh

In this case, chronological records on specified monitoring probes are needed. To define the probes, click on the **Monitoring Points** tab.

<u>F</u> ile <u>E</u> d	lit <u>T</u> ools <u>W</u> indow <u>H</u> elp								
	🖥 🖎 🐼 🃭 🦂 🖗								S:
									ର ହ ଜନ
Study:	full_domain								
Caraa	[]								
Case:	Casel								
XML file:	/home/TM178B0N/Code_Saturne/	Tutorials/full_dom	ain/	case1	/DATA/cas	se1.xml			
	ð X								
📑 Id	entity and paths	Output Control	W	/riter	Mesh P	articles mesh	Monitoring Points		
🕀 🔛 Ca	alculation environment						3		-
H 📔 Tł	nermophysical models	Name	ld		Туре		Selection	Criteria	
🕀 🔛 Pł	nysical properties	Fluid domain	-1		cells	all[]			
🕀 📔 Va	olume conditions	Boundary	-2	boun	darv faces	all[]			
🕀 🛅 Bo	oundary conditions	Doundary	-	boun	adiy races				(
🕀 🛅 N	umerical parameters								
🗆 🛅 Ca	alculation control								
	Time averages								
	Output control								
	Volume solution control					÷	-		
	Surface solution control								
	Profiles	Variables				_			
	Balance by zone						Auto		
± 🗖 Ci	accutation management	Associated Wri	tor	c					
		Associated With	cer	3		14	luit - a		
						V	Vriter		
					res	sults			
						n			
<	, (

Figure III.24: Output control: post-processing

Click on + and enter the coordinates of the monitoring points you want to define.

For the first probe:

Probe (1) : x = -0.25 (m) ; y = 2.25 (m) ; z = 0.0 (m)

Study: full_domain Case: case1 XML file: /nome/TM178B0N/Code_Satume/Tutorials/full_domain/case1/DATA/case1.xml	<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp ☐ ☐ ☐ 2							S
Case: case1 XML file: /home/TM178BON/Code_Saturne/Tutorials/full_domain/case1/DATA/case1.xml	Study: full_domain							
XML file: (home/TM178B0N/Code_Saturee/Tutorials/full_domain/case1/DATA/case1.xml Image: Calculation environment Image: Calculation environmen	Case: casel							
Image: Interview of the inter	XML file: /home/TM178B0N/Code_Saturne/	Tutorials/full_doma	ain/case1	/DATA/case	e1.xml			
Identity and paths Calculation environment Thermophysical models Physical properties Soundary conditions Numerical parameters Calculation control Numerical parameters Calculation control Volume solution control Variables Variables Variables Writer Mane Identity and paths Name Identity and paths Identity and paths Variables Identity and paths Identity and paths Variables Identity and paths Identity and paths Variables Identity and paths Variables Identity and paths Variables Identity and paths Identity and paths Variables Identity and paths	Ø X _							
Calculation environment Calculation environment Calculation environment Calculation environment Calculation control Calculation control Calculation control Calculation control Calculation management Calcula	ldentity and paths	Output Control	Writer	Mesh Pa	articles mesh	Monitoring Points		
Image: Thermophysical models Physical properties Volume conditions Boundary conditions Calculation control Time averages Output control Volume solution control Surface solution control Balance by zone Calculation management Variables Variables Vriter results_MED	🕀 🎦 Calculation environment							
Physical properties Volume conditions Boundary conditions Numerical parameters Calculation control Time averages Output control Surface solution control Surface solution control Balance by zone Calculation management Variables Variables Variables Function management Variables Variable Variables Variable Varia	🕀 🖺 Thermophysical models	Name	ld	Туре		Selection	Criteria	
 Volume conditions Boundary conditions Numerical parameters Calculation control Surface solution control Surface solution control Balance by zone Calculation management Auto Associated Writers Writer results_MED	🕀 🎦 Physical properties	Fluid domain	-1	cells	all[]			
 Boundary conditions Numerical parameters Calculation control Volume solution control Volume solution control Surface solution control Profiles Balance by zone Calculation management Variables V	🕀 🖺 Volume conditions	Boundary	-2 boun	dary faces	all[]			
 Numerical parameters Calculation control Time averages Output control Volume solution control Surface solution control Profiles Balance by zone Calculation management Variables Variables Variables Variables Fesults results_MED	🕀 🛅 Boundary conditions			,				-
Calculation control Time averages Output control Surface solution management Variables Variab	🕀 🛅 Numerical parameters							
 Time averages Output control Surface solution control Profiles Balance by zone Calculation management Auto Associated Writers Writer results_MED	🖃 🛅 Calculation control							
Output control Notice solution control Profiles Balance by zone Calculation management Auto Associated Writers Writer results_MED	Time averages							
Volume solution control Surface solution control Profiles Balance by zone Calculation management Associated Writers Writer results_MED	Output control							
Surface solution control Profiles Balance by zone Calculation management	Volume solution control					_		
Profiles Balance by zone Auto Auto Associated Writers Writer results results_MED	Surface solution control							
Balance by zone ✓ Auto Auto Auto Mriter results results_MED	Profiles	Variables						
Calculation management Associated Writers Writer results results_MED	Balance by zone					Auto		
Associated Writers Writer results results_MED	🕀 🛅 Calculation management							
Writer results results_MED		Associated Wri	ters					
results_MED					W	riter		
results_MED				res	ults			
				res	ults_MED			
					-1-			
								_

Figure III.25: Output control: associated writers

Repeat the procedure for the other probes. Their coordinates are indicated in the following table (the z coordinate is always 0):

Probe n^o .	x (m)	y (m)
2	0.05	2.25
3	0.05	2.75
4	0.05	0.50
5	0.05	-0.25
6	0.75	-0.25
7	0.75	0.25
8	0.75	0.75

Remember to save the xml file regularly.

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp							
	🖟 🖎 👩 📭 🗄							5
Study:	full_domain							
Casai	[case]							
Case.	Casel							
XML file:	/home/TM178B0N/Code_Saturne/	Tutorials/full_doma	in/case1	/DATA/o	ase1.xml			
	Ø 🗙							
📑 Ide	entity and paths	Output Control	Writer	Mesh	Particles mesh	Monitoring Points		
🕀 🎦 Ca	lculation environment							
🕀 🔛 Th	ermophysical models	Monitoring poin	ts outpu	t				
🕀 🎦 Ph	ysical properties		Monitorin	g points	file every 'x' tim	e_value(s) 🗸 0.1	Format .dat	: ~
🕀 🎦 Va	lume conditions							
🕀 🛅 Bo	undary conditions	Monitoring poin	ts coord	inates				
🕀 🛅 Nu	umerical parameters	n			Х	Y		Z
🗆 🛅 Ca	lculation control							
	Time averages							
	Output control							
B	Volume solution control							
	Surface solution control							
	Profiles							
	Balance by zone			÷	-	Duplicate import	from CSV	
🕀 🔛 Ca	lculation management							
		Display mon	itoring p	oints or	SALOME VTK v	viewver	-	
				Pre	obe's radius (m)			
<								

Figure III.26: Output control: monitoring points

Go to the **Volume solution control** item to define which variables will appear in the listing, the post-processing and the chronological records.

Uncheck the boxes in front of the **Pressure**, **k** and **epsilon** variables, in the **Print in listing** column. Information on these three variables will not appear in the output listing anymore.

Uncheck the boxes in front of the **CourantNb** and **FourierNb** variables in the **Post-processing** column. These variables will be removed from the post-processing results.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp							
i 📄 🗁 🖎 🔕 🃭 i 🥎 🥐 i i							S
							8 R
Study: full_domain							
Case: case1							
XML file: /home/TM178B0N/Code_Saturne,	/Tutorials/full_doma	ain/case1/	DATA/o	ase1.xml			
						1	
ldentity and paths	Output Control	Writer	Mesh	Particles mesh	Monitoring Points		
🕀 🖺 Calculation environment	Maaibaaiaa aaia						
🕀 📔 Thermophysical models	Monitoring poir	its output					
🕀 🖺 Physical properties		Monitoring	g points	s file every 'x' tim	e_value(s) V 0.1	Format .dat 🗸	
🕀 🖺 Volume conditions							
🕀 🛅 Boundary conditions	Monitoring poir	ts coordi	nates				
🕀 🛅 Numerical parameters	n			х	Y	Z	
🗆 🛅 Calculation control	1			0.25	2 2 5	0	
Time averages				0,20	_,		_
Output control							
🕞 Volume solution control							
Surface solution control							
🗎 Profiles							
🗟 Balance by zone			÷	-	Duplicate import	from CSV	
🕀 🎦 Calculation management							
	Display mon	itoring po	oints or	SALOME VTK	viewver		
			Pn	obe's radius (m)			
<							

Figure III.27: Output controls: monitoring points - 1^{st} point

Uncheck the box in front of the **total_pressure** variable in the **Monitoring** column. No chronological record will be created for this variable.

Switch to the **Calculation management** heading to prepare the launch script and run the calculation.



Figure III.28: Output control: monitoring points

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp											
	i 🖎 🔕 📭 😏 🦿						S					
							0 ×					
Study: full_domain												
Study:	tudy: Tull_domain											
Case:	Casel											
XML file:	/home/TM178B0N/Code_Saturne	/Tutorials/full_domain/case	1/DATA/case1.xml									
				~~~~~~~~~~~								
		Solution control										
⊕ 🔁 Ca	entity and paths alculation environment ermophysical models	Output label	Internal name	Print in listing	Post- processing	Monitoring						
🕀 🎦 Ph	vsical properties	🗆 base		-	<b>I</b>							
🕀 🎦 Vo	olume conditions	Pressure	pressure			✓						
🕀 🔛 Bo	oundary conditions	Velocity	velocity	<b>a</b>	<b>S</b>	<b>S</b>						
🕀 🔛 Nu	umerical parameters	total_pressure	total_pressure	✓	<b>S</b>	<b>S</b>						
🗆 🛅 Ca	alculation control	turbulence		-	<b>S</b>	<b>S</b>						
R	Time averages	k	k									
	Output control	epsilon	epsilon									
	Volume solution control	TurbVisc	turbulent_viscosity	<b>S</b>								
	Surface solution control	🗆 thermal										
	Profiles	TempC	temperature									
B	Balance by zone	🗆 other			-							
🕀 🎦 Ca	alculation management	scalar2	scalar2									
		CourantNb	courant_number									
		FourierNb	fourier_number	<b>Z</b>		<b>S</b>						

Figure III.29: Solution control: output configuration

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp						
📄 🗁 🕭 🔕 📭 🕱 🙋	🔛 🗾 🖗					S
						0 ×
Study: full_domain						
Case: case1						
XML file: /home/TM178B0N/Code_Satu	ırne/Tutorials/full_domain/cas	e1/DATA/case1.xml				
Ø 🛙						
ldentity and paths	Solution control					
<ul> <li></li></ul>	Output label	Internal name	Print in listing	Post- processing	Monitoring	
🕀 📔 Physical properties	⊡base		-		-	
🕀 🔛 Volume conditions	Pressure	pressure			<b>S</b>	
🕀 📔 Boundary conditions	Velocity	velocity	<b>S</b>	<b>S</b>	<b>S</b>	
🕀 📔 Numerical parameters	total_pressure	total_pressure	≤	<b>S</b>		
🗆 🛅 Calculation control	turbulence		-			
📑 Time averages	k	k				
Output control	epsilon	epsilon				
📄 Volume solution control	TurbVisc	turbulent_viscosity				
Surface solution control	🗆 thermal					
🕒 Profiles	TempC	temperature				
🗒 Balance by zone	$\Box$ other			-		
🕀 🖺 Calculation management	scalar2	scalar2				
	CourantNb	courant_number				
	FourierNb	fourier_number	≤			

Figure III.30: Solution control: probes

# 2 Solution for CASE2

Only a few elements are different from case1.

In this case the density becomes variable. Go to the **Fluid properties** item under the heading, **Physical properties** and change the nature of the density from **constant** to **variable**.

The user law of the density is defined as follows in the *Code_Saturne* (GUI):

density = temperature * ( -4.0668E-03*temperature - 5.0754E-02 ) + 1000.9;

Click on the highlighted icon and define the user law in the window that pops up. Follow the format used in the **Examples** tab.

Eile     Edit     Tools     Window     Help       Image: Ima		S
Study: full_domain		
Case: case2		
XML file: //home/TM178B0N/Code_Satu	rne/Tutorials/full_domain/case2/DATA/case2.xml	
8 B		•
ldentity and paths		
🕀 🛅 Calculation environment	Density constant	
🕀 🎦 Thermophysical models	variable	
Physical properties     Peference values	material law p 725.735 kg/m ²	8
Fluid properties		
Gravity	Viscosity	
🕀 📔 Volume conditions	constant 🗸 🖓	
🕀 🛅 Boundary conditions	Reference value II 8 951e-05 Pais	
Numerical parameters		
Calculation control	Specific heat	
	constant 🗸 🖓	
	Reference value Co. 5483.0	ĸ
	Thermal conductivity	
	constant 🔽	
	Reference value $\lambda$ 0.02495 W/m/l	¢
	Diffusion coefficient of species	
	Name scalar2 🗸	
	constant 🗸 🖉	
	reference 8.55e-05 m²/s	
· · · · · · · · · · · · · · · · · · ·		U.S.

Figure III.31: Fluid properties: variable density



Figure III.32: Fluid properties/Variable density: user expression

As the density is variable, the influence of gravity has to be considered. In the heading **Physical properties** go to **Gravity** and set the value of each component of the gravity vector.

e <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
) 🗁 🖎 🔕 🃭 😓 🧑	) III 🔽 🖓			S
				Ø
dv: full_domain				
e: case2				
file: /bome/TM178B0N/Code_Sa	turne/Tutorials/full_domain	/case2/DATA/case2 yml		
Ø	Gravity			
Identity and paths	Granicy		- 1-2	
Calculation environment		g _* 0.0	m/s²	
Ihermophysical models     Developing and estimated		g, -9.81	m/s ²	
Physical properties     Reference values				
Reference values		g _z 0.0	m/s ²	

Figure III.33: Fluid properties: gravity

Add a monitoring point close to the entry boundary condition in the **Output control** item under the **Calculation control** heading.

			Probe x (n	n) $ \mathbf{y} $ (m) $ \mathbf{z} $ (m			
			9 -0.5	5  2.25  0.0			
<u>F</u> ile <u>E</u> c	lit <u>T</u> ools <u>W</u> indow <u>H</u> el	Р					
- T	🖥 🖎 🔕 📭 🕤	2 🗐 🚺	<b>7</b>				S
							8
Studv:	full_domain						
,	[						
Case:	case2						
XML file:	/home/TM178B0N/Code_	Saturne/Tutoria	ls/full_domain/case2	/DATA/case2.xml			
📑 Id	entity and paths	Outp	out Control Writer	Mesh Particles mesh	Monitoring Points		
🕀 🎦 Ca	alculation environment						
H 📔 🕀	nermophysical models	Mo	nitoring points outpu	t			
🕀 🛄 Pi	nysical properties		Monitorin	g points file every 'x' tim	e_value(s) V 0.1 Fo	rmat .dat 🗸	
🕀 🎦 V	olume conditions						
🕀 🎦 B	oundary conditions	Mor	nitoring points coord	inates			
🕀 🎦 N	umerical parameters		n	Х	Y	Z	^
	alculation control		6	0.75	-0.25	0	
	Time averages		7	0.75	0.25	0	
	Output control		/	0,75	0,25	0	_
	Volume solution control		8	0,75	0,75	0	
	Surface solution control			-0,5	2,25		
	Profiles	-					
	Balance by zone			* -	Duplicate import from	n CSV	
+ <b>C</b>	alculation management						
			Display monitoring p	oints on SALOME VIK V	lewver		
				Probe's radius (m)			
<							



After completing the interface, before running the calculation, some Fortran user routines need to be modified.

Go to the folder  $\bigcirc$  SRC/REFERENCE/base and copy cs_user_boundary_conditions.f90 in the  $\bigcirc$  SRC directory.

• cs_user_boundary_conditions.f90:

In this case, cs_user_boundary_conditions.f90 is used to specify the time dependent boundary condition for the temperature. Refer to the comments in the routine or to the *Code_Saturne* user manual for more information on this routine.

In our case, you need to identify the boundary faces of color 1.

The command

call getfbr('1',nlelt,lstelt)

will return an integer nlelt, corresponding to the number of boundary faces of color 1, and an integer array lstelt containing the list of the nlelt boundary faces of color 1.

**Remark**: Note that the string 1 can be more complex and combine different colors, group references or geometrical criteria, with the same syntax as in the Graphical Interface.

For each boundary face ifac in the list, the Dirichlet value is given in the multi-dimension array rcodcl as follows:

```
if (ttcabs.lt.3.8d0) then
  do ielt = 1, nlelt
    ifac = lstelt(ielt)
    rcodcl(ifac,isca(2),1) = 20.d0 + 100.d0*ttcabs
  enddo
else
  do ielt = 1, nlelt
    ifac = lstelt(ielt)
    rcodcl(ifac,isca(2),1) = 400.d0
  enddo
endif
```

isca(2) refers to the first scalar and ttcabs is the current physical time.

See the example cs_user_boundary_conditions-base.f90 file in the subdirectory  $\boxdot$  SRC/EXAMPLES to complet correctly your boundary conditions for this case2.

**Remark**: Note that, although the inlet boundary conditions for temperature are specified in the cs_user_boundary_conditions.f90 file, it is necessary to specify them also in the Graphical Interface.

# The value given in the Interface can be anything, it will be overwritten by the Fortran routine.

After updating the Fortran file, run the calculation as explained in case1.

When a calculation is finished, *Code_Saturne* stores all the necessary elements to continue the computation in another execution, with total continuity. These elements are stored in several files, grouped in a  $\bigcirc$  yyyymmdd-hhmm/checkpoint subdirectory, in the  $\bigcirc$  RESU directory.

In this case, after the first calculation is finished, a second calculation will be run, starting from the results of the first one.

Go directly on the **Start/Restart** item under the heading **Calculation management**. Activate the **Calculation restart** by clicking the **On** box.



Figure III.35: Start / Restart

A window opens, with the architecture of the study sub-directories. In the **BRESU** folder, click on the folder **BYYYMMd-hhmm/checkpoint** (where **YYYMMd-hhmm** corresponds to the reference of the first calculation results). Then click on **Validate**].

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📄 🗁 🖄 🔕 📭 🕱 👌		S
		0 1
Study: full_domain		
Case: case2		
XML file: //nome/IM1/8BON/Code_Saturn	e/Tutorials/Tull_domain/case2/DATA/case2.xml	
o x		
📑 Identity and paths	Checkpoint/Restart	
🕀 📔 Calculation environment	Calculation restart (checkpoint directory): 💿 on 🔵 off	
🕀 🖺 Thermophysical models	Destart directory	
🕀 🛅 Physical properties	Restart directory	
🕀 🎦 Volume conditions	RESU/20161004-1621/checkpoint	
Boundary conditions		
🕀 🎦 Numerical parameters	Calculation on frozen velocity and pressure fields	
Calculation control		
Calculation management	Advanced options	
Start/Restart		
Performance tuning		
· · · · · · · · · · · · · · · · · · ·		

Figure III.36: Start/Restart: selection of the restart directory

Go to the **Time step** item under the heading **Numerical parameters** and change the number of iterations. It must be the total number of iterations, from the beginning of the first calculation.

The first calculation was done with 300 iterations and another 400 iterations are needed for the present case. Therefore the value 700 must be entered.

Image: Study: full_domain   Case: case2   XML file: /rome/TM178B0N/Code_Saturne/Tutorials/full_domain/case2/DATA/case2.xml     Image: Study: Image: Study:   Image: Study: Image: Study:   Image: Study: Image: Study: S	<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
Study: full_domain Case: case2 XML file: //home/TM17880N/Code_Saturne/Tutorials/full_domain/case2/DATA/case2.xmL	📄 🚍 🕭 🔕 📭 🌀 👌	i 🎟 🗾 🖗 🕹 🖓	5
Study: full_domain Case: case2 XML file: /home/TM178B0N/Code_Saturne/Tutorials/full_domain/case2/DATA/case2.xml  Value: /home/TM178B0N/Code_Saturne/Tutorials/full_domain/case2/DATA/case2.xml  Value: calculation environment Case: Calculation environment Case: Calculation environment Case: Calculation environment Case: Case: Calculation environment Case: Case: Calculation environment Case:		li l	0 X
Case: case2 XML file: /home/TM178B0N//Code_Saturme/Tutorials/full_domain/case2/DATA/case2.xml	Study: full_domain		
XML file: Inome/TM178B0N/Code_Saturne/Tutorials/full_domain/case2/DATA/case2.xml       Getulation environment     Time step option Constant     Volume conditions     Number of iterations (restart included)     700     Numerical parameters     Equation parameters     Equation parameters     Calculation control     Calculation management     Option zero time step	Case: case2		
Image: Calculation environment         Image: Calculation parameters         Image: Calculation environment         Image: Calculation management	XML file: /home/TM178B0N/Code_Satur	rne/Tutorials/full_domain/case2/DATA/case2.xml	
Identity and paths   Calculation environment   Thermophysical models   Physical properties   Volume conditions   Number of iterations (restart included)   Number of iterations (restart included)   Number of iterations (restart included)   Time step   Equation parameters   Time step   Calculation control   Calculation management	o x		
Calculation environment     Thermophysical models   Physical properties   Volume conditions   Number of iterations (restart included)   Number of iterations (restart included)   Stobal parameters   Global parameters   Equation parameters   Calculation control   Option zero time step	ldentity and paths		
Thermophysical models   Physical properties   Volume conditions   Boundary conditions   Number of iterations (restart included)   Colobal parameters   Colobal parameters   Equation parameters   Equation parameters   Calculation control   Calculation management	🕀 🔛 Calculation environment	Time step option Constant	
Physical properties   Volume conditions   Boundary conditions   Number of iterations (restart included)   Clobal parameters   Clobal parameters   Equation parameters   Calculation control   Calculation management	🕀 📔 Thermophysical models		
Volume conditions   Numerical parameters   Global parameters   Equation parameters   Calculation control   Calculation management	🕀 🛅 Physical properties	Reference time step 0.05 s	
Image: Soundary conditions       Number of iterations (restart included)       700         Image: Numerical parameters       Time step limitation with the local thermal time step       Image: Soundary conditions         Image: Equation parameters       Option zero time step       Image: Soundary conditions         Image: Calculation control       Image: Soundary conditions       Image: Soundary conditions         Image: Calculation management       Option zero time step       Image: Soundary conditions	🕀 📔 Volume conditions		
Numerical parameters   Global parameters   Equation parameters   Time step   Option zero time step    Option zero time step	🕀 📔 Boundary conditions	Number of iterations (restart included) 700	
Clobal parameters   Equation parameters   Time step   Option zero time step   Calculation management	🗆 🛅 Numerical parameters	Time step limitation with	
Equation parameters   Option zero time step   Calculation control   Calculation management	📄 Global parameters	the local thermal time step	
Calculation control Calculation management	Equation parameters		
Calculation control	🔲 Time step	Option zero time step	
	Calculation control		
	Calculation management		
3			
	· • • • • •		

Figure III.37: Time step

Eventually, run the calculation.

# **3** Solution for CASE3

This case is similar to case2, with the following differences:

- Step 1: define head losses in the fluid domain,
- Step 2: compute the spatial average of temperature scalar,
- Step 3: parallel computation on 2 processors,
- Step 4: dealing with a user results file.

• Step 1-1: Define the head losses in the Graphical User Interface (GUI)

'' x >= 0.2 and x <= 0.4 and y >= -0.75 and y <= -0.25',

Go to Volume regions definition under the heading Volume conditions. Click on Add, unselect Initialization and select Head losses in the Dialog window that pops up (see Figure III.38). In the box named Label, name the head losses region.

Define the limits of the head losses region in **Selection criteria**. The associated character string to enter is as below:

udy: full_domain				
ise: case3				
1L file: /home/TM178B0N/Code_Saturne	/Tutorials/full_d	omain/case3/DATA/case3.xml		
Identity and naths	Definition of	volume regions		
<ul> <li>Thermophysical models</li> <li>Physical properties</li> <li>Volume conditions</li> <li>Initialization</li> <li>Coriolis Source Terms</li> <li>Boundary conditions</li> <li>Numerical parameters</li> <li>Calculation control</li> <li>Calculation management</li> </ul>	all_cells Zone_1	Initialization I Head losses I Momentum source term I Thermal source term Scalar source term I I Annuler I OK IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	Delete Modify rences from Preprocessor listing	

Figure III.38: Creation of the head losses region

File Edit Tools Window Help		2			<b>S</b> Ø X
Study: full_domain					
Case: case3					
XML file: /home/TM178B0N/Code_Sat	urne/Tutorials/full_	domain	/case3/DATA/	case3.xml	
Identity and paths	Definition of	• volum	e regions		
Calculation environment	Label	Zone	Nature	Selection criteria	
🕀 📔 Thermophysical models	all_cells	1	Initialization	all[]	
+ 📔 Physical properties	Zone_1	2		x >= 0.2 and x <= 0.4 and y >= -0.75 and y <= -0.25	
Volume regions definition         Initialization         Head losses         Coriolis Source Terms         Image: Source Terms </th <th>Add from</th> <th>Prepro</th> <th><b>ccessor listing</b> Import grou</th> <th>Add Delete Modify ps and references from Preprocessor listing</th> <th></th>	Add from	Prepro	<b>ccessor listing</b> Import grou	Add Delete Modify ps and references from Preprocessor listing	

Figure III.39: Selection criteria of the head losses region

• Step 1-2: Specify the head losses coefficients  $\alpha_{ii}$ 

To specify the head losses coefficients go to the **Head losses** item and select the name of the head losses volume region. In this example, the coefficient is isotropic so that we use the same value for each  $\alpha_{ii}$ . Please note that  $\alpha_{ii} = 2 \times K_{ii}$ , therefore if  $K_{ii} = 10^4$ ,  $\alpha_{ii} = 2 \times 10^4$ .

• Step 2: Compute the spatial average of temperature

The computation of the spatial average must be done in the cs_user_extra_operations.c routine.

The following code computes the spatial average temperature and writes it in a file called moy.dat.

```
/* Global declaration of the moy.dat file that will be filled with Tavg */
static FILE *file = NULL;
void
cs_user_extra_operations(void)
{
    /* Variables declaration */
```

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp					
🗋 🖆 🖄 🕼 🏂 👌	III 🗹 🖗	)		ර	
				۵	3 🗙
Study: full_domain					
Case: case3					
XML file: /home/TM178B0N/Code_Saturne	e/Tutorials/full_d	omain/case3/D	ATA/case3.xml		
ØX					
ldentity and paths	Select volume	e zone for head	losses		
🕀 🔛 Calculation environment	Label	Zone	Selection criteria		
🕀 🔛 Thermophysical models	head_loss	2	x >= 0.2 and x <= 0.4 and y >	= -0.75 and y <= -0.25	
🕀 🔛 Physical properties			,		
🖃 🛅 Volume conditions					
Volume regions definition					
Initialization	T				
📄 Head losses	l ensor coem	d lossos sooffic	ionto(mil), a dil (dt = 0.5 a millille		
Coriolis Source Terms	nea	a losses coema	$\frac{1}{2} = \frac{1}{2} = \frac{1}$		
🕀 🔛 Boundary conditions	O(XX	20000.0	αγγ 20000.0	αzz 20000.0	
🕀 🎦 Numerical parameters					
🕀 🎦 Calculation control	Reference	frame transfo	rmation matrix		
🕀 🎦 Calculation management					
· · · · · · · · · · · · · · · · · · ·					

Figure III.40: Head losses coefficients

```
/* Get pointers to the mesh and mesh quantities structures */
const cs_mesh_t *m = cs_glob_mesh;
const cs_mesh_quantities_t *fvq = cs_glob_mesh_quantities;
/* Number of cells */
const int n_cells = m->n_cells;
/* Cell volumes */
const cs_real_t *cell_vol = fvq->cell_vol;
/* Get the temperature field */
const cs_field_t *temp = cs_field_by_name_try("temperature");
/* Cell volumes sum, Temp * volumes sum and Tavg */
cs_real_t voltot = 0., temptot = 0., Tavg = 0.;
/* Compute the sum of the cell volumes */
for (int ii = 0 ; ii < n_cells ; ii++)
voltot += cell_vol[ii];</pre>
```

```
/* Compute the sum T*vol */
for (int ii = 0; ii < n_cells; ii++)
  temptot += temp->val[ii]*cell_vol[ii];
/* Parallel sums */
cs_parall_sum(1, CS_DOUBLE, &voltot);
cs_parall_sum(1, CS_DOUBLE, &temptot);
/* Compute Tavg */
Tavg = temptot / voltot;
/* Open the file moy.dat at the first iteration
   and write the first comment line only on the
   first processor (0 in parallel, -1 in serial) */
if (cs_glob_time_step \rightarrow nt_cur == 1 \&\& cs_glob_rank_id <= 0) {
  file = fopen("moy.dat", "a");
  fprintf(file, "#Time (s) Average Temperature (C)\n");
}
/* Print the averate temperature at the current time step
   on the first processor only \ast/
if (cs_glob_rank_id \ll 0)
  fprintf(file, "%.6f %.6f\n", cs_glob_time_step ->t_cur, Tavg);
/* Close the file moy.dat at the last iteration
   on the first processor only*/
if (cs_glob_time_step ->nt_cur == cs_glob_time_step ->nt_max
&& cs_glob_rank_id \ll 0)
  fclose(file);
```

• Step 3: Choose a computation with 2 processors

This modification will be done in the **Prepare batch calculation** item. To run the calculation on two processors, simply change the number of processors indicator to 2. The launch script will automatically deal with the rest.

Do not forget to set the right **Reference time step** and **Number of iterations** under the heading **Numerical parameters**.
EDF R&D

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
📄 🗁 🙆 💽 😒 🖄	🔲 🗾 🤪			S
				0 8
Study: full domain				
Case: case3				
XML file: /home/TM178B0N/Code_Satur	ne/Tutorials/full_domain/ca	se3/DATA/case3.xml		
Ø 🗙				
ldentity and paths	Script file			
🕀 📔 Calculation environment	Select the script file 🛛 🔚 runcase			
🕀 📔 Thermophysical models				
🕀 🎦 Physical properties	Calculation script para	meters		
🕀 🛅 Volume conditions		Run type	Standard 🗸	
Boundary conditions		Run id		
Numerical parameters				
		Number of processes	2	
Start/Restart		Threads per task	<b>o</b>	
Performance tuning		filledds per task	· ·	
Prepare batch calculation		Advanced options	X	
	Calculation start			
		Start c	alculation	
· · · · · · · · · · · · · · · · · · ·				

Figure III.41: Number of processors

• Step 4: Dealing with a user results file moy.dat

**Remark 3.1**: We do not have to specify the name of the new user file in the Graphical User Interface (GUI), like in previous *Code_Saturne* versions. The name of the new user file had to be identified in the launch script in order to be automatically copied in the  $\boxdot$  RESU directory; this is not requested anymore.



Figure III.42: User results files