



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

6, quai Watier F-78401 Chatou Cedex

MARCH 2024

code_saturne documentation

code_saturne version 8.0 tutorial: simple junction

 $contact: \ saturne-support@edf.fr$



		code_saturne
EDF R&D	code_saturne version 8.0 tutorial: simple junction	documentation Page $1/42$

TABLE OF CONTENTS

	I Introduction	3
1	Introduction	4
1.1	CODE_SATURNE SHORT PRESENTATION	4
1.2	About this document	5
1.3	CODE_SATURNE COPYRIGHT INFORMATIONS	5
	II Simple junction testcase	6
1	Study description	7
1.1	Study creation and preparation	7
1.2	Objective	8

1.4	
1.3	Description of the configuration
1.4	CHARACTERISTICS
1.5	Mesh characteristics
2	CASE 1: Basic calculation
2.1	CALCULATION OPTIONS
2.2	INITIAL AND BOUNDARY CONDITIONS
2.3	PARAMETERS
2.4	Results

	III Step by step solution	14
1	Solution for CASE1	15
1.1	Mesh tab	. 17
1.2	CALCULATION FEATURES TAB	. 21
1.3	Volume conditions tab	. 26
1.4	Boundary conditions tab	. 30
1.5	TIME SETTINGS TAB	. 34
1.6	NUMERICAL PARAMETERS TAB	. 35
1.7	Postprocessing tab	. 37
1.8	RUN COMPUTATION	. 41

Part I

Introduction

EDF R&D

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). This code_saturne GUI version is architectured to provide users a logical approach to process CFD simulation. The following figure I.1 code_saturne GUI. You will find 3 main zones :

<u>File E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📄 🚔 🖄 🥱 🕐 🔳 🖗	📵 🖉 🗟 😰 🎑 🐯 Upper icons
© ₪ Calculation environment Mesh Mic Calculation features Mic Volume conditions E Boundary conditions	Study and case directories Study /home Case /home/ /code_saturne
 Δ<i>x</i> Time settings Δ<i>x</i> Numerical parameters M Postprocessing Performance settings 	Associated case sub-directories Data Data Ed sub-directory not found User sources Ed sub-directory not found Results Ed sub-directory not found
Headers	Main window

Figure I.1: code_saturne GUI

- 1. Upper icons green zone : User can manage case file (from the creation to the computation)
- 2. Header tabs window orange zone : User can access and define all mandatory settings to perform CFD analyzis
- 3. Main window red zone : User can set parameters for every selected tab

code_saturne tutorials follow a logical process for every analyzis. User should begin by **Calculation envrionement** tab and finish by **Performance settings** tab before running computation.

1.2 About this document

The present document is a tutorial for code_saturne version 8.0. It presents a simple test case and guides the future code_saturne user step by step into the preparation and the computation of that case.

The test case directory, containing the necessary meshes and data is 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

Simple junction testcase

1 Study description

1.1 Study creation and preparation

The first thing to do is to prepare the computation directories. You will find all tutorial folders in the examples directory \boxdot examples. Here, the study directory \boxdot simple_junction will contain a single calculation directory \boxdot case1.

Create the study \boxdot simple_junction and the \boxdot case1. There are three ways to create the study :

- 1. Within SALOME module CFDStudy -as explained in the Shear driven cavity tutorial-
- 2. With code_saturne-via the terminal-
- 3. With code_saturne-via the GUI (Graphic User Interface)-

The second option can be done by typing the following commands in your terminal:

\$ code_saturne create -s simple_junction -c case1

Then code_saturne Graphical User Interface (GUI) can be launched by typing the command lines as below:

```
$ cd simple_junction/case1/DATA
```

```
$ ./code_saturne gui &
```

And the following window opens (fig II.1).



Figure II.1: code_saturne (GUI) graphic window

EDF R&D

The third option can be done by first launching code_saturne GUI then creating and opening new cases and or meshes by clicking on \boxed{File} New Case as follow (fig II.2):

<new para<="" th=""><th>rameters set> - code_saturne ×</th><th>New case</th><th>×</th></new>	rameters set> - code_saturne ×	New case	×
File Edit Tools Window Help			
New case	262 5	Current path /	
New case Image: Crit-N Image: Crit-N	Image: Contract of the second seco	Current path /	
Open a new case.	47.01		

Figure II.2: code_saturne (GUI) graphic window - case creation

The mesh files, available in the examples directory, should be copied in the directory \cong MESH/, by the command line as follows or by your favorite explorer:

```
$ cd simple_junction/MESH/
```

```
$ cp .../examples/1-simple_junction/mesh/downcomer.med .
```

If you do use SALOME here is a helping note for you :

SALOME interface helping note: Once the mesh is copied in the directory \boxdot MESH, you can update the object browser (open a contextual menu by a right-click on the study name or the case name in the object browser, and left-clik on the entry [Update Object Browser]).

The mesh can then be directly displayed in the VTK viewer (the open viewer when module CFDStudy is active). To do so, follow these steps:

- In the object browser of *SALOME*, right-click on the mesh of the study (in the directory \boxdot MESH of the study), then select '*Convert to MED*'. A med file should be generated in the same directory;
- Right-click on this med file, then select '*Export in SMESH*'. A heading **Mesh** should appear in the object browser;
- Under this heading, right-click on the mesh name and then 'Display mesh';

1.2 Objective

The aim of this case is to train the user of code_saturne on an oversimplified 2D junction including an inlet, an outlet, walls and symmetries.

1.3 Description of the configuration

The configuration is two-dimensional.

It consists of a simple junction as shown on figure II.3. The flow enters through a hot inlet into a cold environment and exits as indicated on the same figure. This geometry can be considered as a very rough approximation of the cold branch and the downcomer of the vessel in a nuclear pressurized water reactor. The effect of temperature on the fluid density is not taken into account in this first example.



Figure II.3: Geometry of the downcomer

1.4 Characteristics

Characteristics of the geometry and the flow:

Height of downcomer	$H = 3.00 \ m$
Thickness of downcomer	$E_d = 0.10 \ m$
Diameter of the cold branch	$D_b = 0.50 \ m$
Inlet velocity of fluid	$V = 1 \ m.s^{-1}$

Table II.1: Characteristics of the geometry

Physical characteristics of fluid:

The initial water temperature in the domain is equal to 20°C. The inlet temperature of water in the cold branch is 300°C. Water characteristics are considered constant and their values taken at 300°C and $150 \times 10^5 Pa$:

- Density: $\rho = 725.735 \ kg.m^{-3}$
- Dynamic viscosity: $\mu=0.895\times 10^{-4}~kg.m^{-1}.s^{-1}=8.951\times 10^{-5}~Pa.s$
- Specific heat: $C_p = 5\,483 \ J.kg^{-1}.K^{-1}$
- Thermal conductivity = $0.02495 W.m^{-1}.K^{-1}$

1.5 Mesh characteristics

Figure II.4 shows a global view of the downcomer mesh. This two-dimensional mesh is composed of 700 cells, which is very small compared to those used in real studies. This is a deliberate choice so that tutorial calculations run fast.



Figure II.4: Mesh and groups of boundary faces

Note that here the case is two-dimensional but code_saturne always operates on three-dimensional mesh elements (cells). The present mesh is composed of a layer of hexahedrons created from the 2D mesh shown on figure II.4 by extrusion (elevation) in the z direction. The virtual planes parallel to Oxy will have slipping (symmetry) conditions to account for the two-dimensional character of the configuration.

Type: structured mesh

Coordinates system: cartesian, origin on the edge of the main pipe at the outlet level, on the nozzle side (figure II.3)

Group definition: see figure II.4. To specify boundary conditions on the boundary faces of the mesh, the **groups** have to be identified. To do so, a name is first assign to a group of boundary faces during the definition of the geometry. This group is reused during the generation of the mesh to associate the list of boundary elements to the given name.

2 CASE 1: Basic calculation

2.1 Calculation options

Most of the options used in this calculation are default options of code_saturne. Some none default options are listed below:

- $\rightarrow\,$ Time settings: steady algorithm (local time step) (Velocity-Pressure algorithm is the SIMPLEC one)
- \rightarrow Turbulence model: $k \varepsilon$ LP (Linear Production)
- \rightarrow Temperature activated with no gravity (acts like a passive scalar)
- $\rightarrow\,$ Physical properties: uniform and constant

2.2 Initial and boundary conditions

 \rightarrow Initialization: none (default values)

The boundary conditions are defined as follows:

- Flow inlet: Dirichlet condition, an inlet velocity of 1 $m.s^{-1}$ and an inlet temperature of 300°C are imposed
- **Outlet**: default values
- Walls: default values

Figure II.4 shows the groups used for boundary conditions and table II.2 defines the correspondance between the group names and the type of boundary condition to use.

Do not forget to enter the value of the hydraulic diameter, adapted to the current inlet (used for turbulence entry conditions).

Group name	Conditions
INLET	Inlet
OUTLET	Outlet
WALL1 WALL2 WALL3 WALL4 WALL5	Wall
ZMIN ZMAX	Symmetry

Table II.2: Boundary conditions and associated groups

2.3 Parameters

All parameters necessary to this study can be defined through the Graphical Interface without using any user Fortran files. They are specified in the following table:

Calculation control parameters				
Pressure-Velocity coupling	SIMPLEC algorithm			
Number of iterations	300			
Reference time step	0.1			
Maximal CFL number	1.0			
Output period for post-processing files	1			

2.4 Results

Figure II.5 presents the results obtained at different iterations in the calculation. They were plotted from the post-processing files, with ParaView.

Note: since the **steady flow** option has been chosen, the evolution of the flow iteration after iteration has no physical meaning. It is merely an indication of the rapidity of convergence towards the (physical) steady state.



Figure II.5: Water velocity field colored by temperature at different iterations

Part III

Step by step solution

1 Solution for CASE1

The first thing to do is to prepare the computation directories. Here, the study directory \boxdot simple_junction will contain a single calculation directory \boxdot case1.

Create the study \boxdot simple_junction and the \boxdot case1 using one of the three methods explained in part 2 of this document.

In this case we use the code_saturne Graphical User Interface (GUI) method.

code_saturne Graphical User Interface (GUI)can be launched by typing the command lines as below:

\$ cd simple_junction/case1/DATA

\$./code_saturne gui &

And the following window opens (fig III.1).

<new parameters="" set=""> - code_saturne</new>					
<u>File Edit T</u> ools <u>W</u> indow <u>H</u> elp					
📑 🗎 🏝 🔄 🥐 🔳 [🖗 🍙 🖉 🖪 🛄 🌣				
 Calculation environment Mesh Colculation features Volume conditions Ar Time settings Ar Numerical parameters Postprocessing Performance settings 	Study and case directories Study /home Case Case Data ed sub-directory not found User sources ed sub-directory not found Results ed sub-directory not found				

Figure III.1: code_saturne (GUI) graphic window

The mesh files should be copied in the directory $\bigcirc MESH/$, by the command line as follows or by your favorite explorer:

```
$ cd simple_junction/MESH/
$ cp .../examples/1-simple_junction/mesh/downcomer.med .
```

Go to the File menu and click on New file to open a new calculation data file. The interface automatically updates the following information:

- Study name
- Case name
- Directory of the case
- Associated sub-directories of the case

case1:setup.xml-code_saturne ×				
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
📑 🖹 🖄 🚸 🖻 📴 🤪 🛛	2 0 2	G 🔅		
 Calculation environment Mesh Mosh Calculation features Nource conditions Boundary conditions ∆t Time settings ∆x Numerical parameters Postprocessing Performance settings 	Study and case dir Study Case case1 Associated case si Dat Use Res	ectories /CODE_SATURNE_TUTORIALS/simple_junction ub-directories a DATA r sources SRC ults RESU		

Figure III.2: Identity and paths

Don't forget to regulary save your work by clicking on File Save.

1.1 Mesh tab

The next step is to specify the mesh(es) to be used for the calculation. Click on the **Mesh** heading. Select **Import meshes**. Then click on + to add meshes.

The list of meshes appears in the window List of meshes. In this case only the mesh downcomer.med is needed.

The **Periodic Boundaries** is not used in this case so **Preprocessing** page does not need to be visited. Keep the default values.



Figure III.3: Meshes: list of meshes

Preprocessing By default, the execution mode is set to standard computation i.e. a flow computation. It can be set in the **Mesh** menu.

Several other execution modes are available. They allow to perform operations linked to the mesh:

- Import mesh only: code_saturne reads the specified mesh files, convert them to code_saturne internal format and save them in a mesh_input with this format.
- Mesh preprocessing only: code_saturne imports the mesh and performs preprocessing tasks (joining, boundary insertion, extrusion, boundary layer meshing, ...) specified in the GUI or in user source file cs_user_mesh.
- Mesh quality criteria only: code_saturne imports the mesh, performs preprocessing tasks and computes quality criteria of the resulting mesh.

	case1:s	etup.xml - code	_saturne			×
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp						
📑 🗎 🕭 👌 👌 🗾 🖗 🗌	6262 🕻	\$				
Ø 🗵						
	Mesh input Import meshes Local mesh directory (Use existing me (optional)	sh input (Generate cartesia	n mesh	
► av Calculation features	"/MESH					
 The second tions The second tions 	List of meshes					
→ Δt Time settings	File name	Format	Numbers	Reorient	Path	
► Δx Numerical parameters	downcomer.med	MED				
Performance settings						
	Import mesh only E Mesh preprocessing or Mesh quality criteria or Standard Computation Use unmodified chec Save mesh if modifie	nly nly ckpoint mesh in ed by preproces:	case of resta] -		

Figure III.4: Preprocessing and calculation modes in code_saturne

Note: If you need to run one of this exectution modes, you just need to select the one you need then to click on icon in the menu bar. You will learn more about this modes toward all code_saturne tutorials.

For this case, select Standard Computation.

Boundary zones Boundary conditions now need to be defined. Go to the **Boundary zones** under Mesh heading. The following window opens (fig III.5).

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📄 눹 📐 🥱 🙋 国 🖗	
Image: Antiperformance of the settings Image: Antiperformance of the settings	Boundary regions definition Label Zone BC_1 1 Add Delete Add from preprocessor log
٩	Import groups and references from preprocessor log

<u>File E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
📑 💼 🏝 👌 🍖 🔳 🖗	i 🛛 🕯	2	G 🔅
Ø 🛙			
Calculation environment	Boundary	regions de	finition
👻 🕅 Mesh	Label	7	Colortian mitania
Preprocessing	Label	Zone	Selection criteria
Volume zones	BC_1	1 all[]	
Boundary zones	BC_2	2 all[]	
▶ @₩ Calculation features	BC_3	3 all[]	
 Volume conditions 	BC_4	4 all[]	
Image: Boundary conditions			
▶ ∆t Time settings			
▶ ∆x Numerical parameters			
Postprocessing			
Performance settings			

Figure III.5: Creation of a boundary region

Each boundary must be defined. Click on Add to edit a new boundary. The boundary faces will be grouped in user-defined zones, based on the name of the group or on geometrical conditions. For each zone, a label and a selection criteria must be assigned by double clicking on the field you wish to set.

The Label can be any character string. It is used to identify the zone more easily. It usually corresponds to the nature of the zone.

The **Selection criteria** is used to define the faces that belong to the zone. It can be a group name, geometrical conditions, or a combination of them, related by **or** or **and** keywords.

Note: A zone number is used by the code to identify each zone.

The table III.6 is a short boundary conditions reminder for our case. Set boundary regions as follows.

Label	Inlet	Outlet	Symmetry	Wall
Selection criteria	INLET	OUTLET	ZMIN or ZMAX	WALL1 or WALL2 or WALL3
Selection criteria			or WALL4 or WALL5	

Figure III.6: Boundary conditions

It is usually faster to regroup the different groups in one single zone, as shown on figure III.7. For instance, the localization of the Symmetry zone is the string 'ZMIN or ZMAX''. The same treatment is done for the wall conditions.

After defining all the boundary zones, the Interface window will look as in figure III.7.

File Edit Tools Window Help Image: Second Sec		case1:	setup.xml - code_saturne	×
 Calculation environment <l< td=""><td><u>F</u>ile <u>E</u>dit <u>T</u>ools <u>W</u>indow <u>H</u>elp</td><td></td><td></td><td></td></l<>	<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
Calculation environment Calculation environment Preprocessing Volume zones Boundary zones Boundary zones Zone Zone Zone Selection criteria Couldet Z OUTLET Sempetry Z MUN or ZMAY	📑 💼 🕭 👌 👌 🖬 🖗 🧯	2 0 2	C 🔅	
 Add I Delete Add from preprocessor log Import groups and references from preprocessor log 	 Calculation environment Calculation environment Preprocessing Volume zones Boundary zones Coupling parameters ∆t Time settings ∆t Time settings ∆r Numerical parameters Performance settings At data 	ndary regions de Label Zone nlet 1 Symnetry 3 Vall 4	finition Selection criteria INLET OUTLET ZMIN or ZMAX WALL1 or WALL2 or WALL3 or WALL4 or WALL5 Add Delete ssor log Import groups and references from preprocessor log	

Figure III.7: Boundary zones label renamed

Remember to save the xml file regularly!

1.2 Calculation features tab

The **Calculation features** menu allows to choose the flow model. In this case, all default values are left unchanged, i.e. we choose to simulate an incompressible single phase with an eulerian approch.

Figure III.8: Flow modelling

Turbulence Model Now, let's choose a turbulence model for our simulation. To do so, go **Turbulence models** sub-folder and open **Turbulence model** drop-down menu.

In this case, the k- ε linear production model is used. Here, you can also specify a turbulence level based on a reference velocity. Leave the default values unchanged $(\mathbf{1} \ m.s^{-1})$.

File Edit Tools Window Help Image: Solution features Image:		case1:setup.xml-code_saturn	e ×
Image: Solution of the set of the s	<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
Image: Second State St	📄 💼 🏝 🥱 👌 💽 📝	2 2 2 🖬	
A state of the		No model (i.e. laminar flow) RANS - Algebraic Mixing length RANS - 1st order k-£ Linear Production v ² -7 BL-v ² /k k-w SST Spalart-Allmaras RANS - 2nd order R _i -t SSG Ref LES V Smagorinsky Standard dynamic model WALE Others R _i -£ LRR k-£	ence)

Figure III.9: Turbulence model: list of models

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📄 🚔 🧐 🥐 🔳 🖗	🖻 🛛 🙆 📕 🌣
6 X	
Calculation environment	Turbulence model
🕨 🕅 Mesh	
 Ø	k-ε Linear Production 👻
Turbulence models	
🗄 Thermal model	
Body forces	Advanced options 🐹
Species transport	
Volume conditions	Reference values (used for initialization of turbulence)
Boundary conditions	
 Ar Numerical parameters 	Velocity scale 1.0 m/s
Destprocessing	Length scale Automatic *
A Performance settings	
· · · · · · · · · · · · · · · · · · ·	
4	

Figure III.10: Turbulence model: choice of a model

Thermal Model For this study the equation for temperature must be solved. Click on the **Thermal model** item to choose between:

- No thermal scalar
- Temperature (Celsius)
- Temperature (Kelvin)
- Enthalpy (J/kg)

In the present case, select **Temperature** (Celsius).



Figure III.11: Thermal scalar conservation: list of models

Once the thermal scalar selected, additional items appear. There are no radiative transfers in our case, so this item can be ignored.

Figure III.12: Thermal scalar conservation: choice of a model

Body forces In **Body forces** heading set the three components of gravity in the **Gravity** item. In this case, since the gravity doesn't have any influence on the flow, gravity can be set to **0**. Same thing for the **Coriolis source terms (rotation vector)**.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📑 🗎 🏝 👌 👌 🗾 🖗	i 🖉 🧕 🖻 💶 🌣	
File Edit Tools Window Help Image: Second state of the seco	Gravity g_x 0.0 m/s^2 g_y 0.0 m/s^2 g_z 0.0 m/s^2 Coriolis source terms (rotation vector) Ω_x 0.0 s^{-1} Ω_z 0.0 s^{-1}	
4		

Figure III.13: Body forces

1.3 Volume conditions tab

Initialization To initialize variables at the instant t = 0 (s), you first need to tick **Initialization** under **Volume conditions** heading then you can select the **Initialization** tab located in all cells all cells. See III.14.

<u>File Edit Tools Window Hel</u>	p								
📄 🛅 🚵 😏 🦿 🔳	🖗 🛛 [2 🙆	2 🖪 🕻	*					
Calculation environment	G X	/olume zone	types						
Mesh Mesh Galculation features		Zone label	Initialization	Physical properties	Solid	Porosity	Head losses	Momentum source term	mal source
all_cells		all_cells	v	v					
 Coupling parameters At Time settings Ax Numerical parameters 									
Postprocessing									
₩ Performance settings									

<u>File E</u> dit <u>T</u> ools <u>Window H</u> elp	
📄 🖆 🏂 🔶 🔳 🖗 🟮 🖉 🧕 😰	
Ø Ø	
 Calculation environment Mesh Calculation features Volume conditions General parameters Coupling parameters Ar Time settings Ar Numerical parameters Postprocessing Performance settings 	
())) () () () () () () () ()	•

Figure III.14: Volume conditions and Initialization

Velocity, thermal scalar and the turbulence can be here initialized. In this case, the values te be set are: zero velocity (default) and an initial temperature of 20° C. Specific zones can be defined with different initializations. In this case, only the default all cells is used.

• Click on **Thermal**, select **Initialization by formula** and click on the opposite icon to specify the initial value of the thermal scalar. It can be a value or a user expression.

User expression	Predefined symbols	Examples	
temperature =	20.;		
		● <u>C</u> ancel	<u>₽</u> 0К

Figure III.15: Initialization of the scalar

• To initialize the velocity, click also on the icon near **Velocity**.

User expression	Predefined symbols	Examples	
<pre>velocity[0] = velocity[1] = velocity[2] =</pre>	0.; 0.; 0.:		
	,		
		Cong	

Figure III.16: Initialization of the velocity

Physical properties Under the heading **Volume conditions** we can also specify reference values of some physical quantities and the physical properties of the fluid in **Physical properties** tab.

Use the default value of **101 325** (*Pa*) for the pressure and **20** ($^{\circ}$ C) for the temperature.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📄 🚔 🅭 👌 🙋 🗖 🖗	2 2 2 1
g X	
B Calculation environment M Mesh M Calculation features	Physical properties Initialization
✓ Wolume conditions ✓ all_cells ✓ Boundary conditions ✓ Coupling parameters ✓ Δt Time settings	Material user_material * Method user_properties *
 → Δx Numerical parameters → □ Postprocessing 	Reference total pressure
Performance settings	value 101325.0 Pa
	Reference temperature
	value 20.0 ° C (used for properties initialization)
	Density
	constant Tensor Reference value p 725.735 kg/m ³
	Viscosity
	constant -
<u> (</u>	Reference value µ 8.95e-05 Pa.s

<u>File E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📄 🚇 👌 👌 🔲	2 🙆 2 💶 🌣
 Calculation environment Calculation features Colculation features Coupling parameters Coupling parameters Coupling parameters Ar Time settings Ar Time settings Postprocessing Performance settings 	
4	, v

Figure III.17: Physical and fluid properties

Specify the fluid physical characteristics in the **Fluid properties** item:

- Density
- $\bullet~{\rm Viscosity}$
- Specific Heat
- Thermal Conductivity

In this case they are all constant.

- ρ = 725.735 kg.m⁻³
- μ = 0.895 × 10⁻⁴ kg.m⁻¹.s⁻¹
- $C_p = 5\,483 \; J.kg^{-1}.K^{-1}$
- λ = 0.02495 $W.m^{-1}.K^{-1}$

1.4 Boundary conditions tab

All boundary zones were defined in the mesh section but not their nature. To do it click on the **Boundary conditions** sub-folder to first set the **Nature** then start setting inlet boundary conditions for velocity, turbulence and themal scalar. The different natures that can be assigned are:

- Wall
- Free inlet/outlet
- Inlet
- Symmetry
- Outlet
- Imposed P Outlet

As shown on figure III.18, outlet and wall boundary zones also appear in the window. The thermal

		case1 : setup.xml - code_saturne ×
<u>File E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📄 🖹 👌 🥐 🖬 🖗	i 🛛 🙆 😰	I I I I I I I I I I I I I I I I I I I
	Boundary zone U Label V Inlet I Outlet C Symmetry S Wall F I	ypes Vall het Jutlet iymmetry free inlet/outlet mposed P Outlet

6			
Calculation environment	Boundary zo	ne types	
👻 🕅 Mesh	Label		Network
Preprocessing	Label		Nature
Volume zones	Inlet	Inlet	
Boundary zones	Outlet	Outlet	
 de Calculation features 	Symmetry	Symmetry	
Volume conditions	Wall	Wall	
 E Boundary conditions 			
🔁 Inlet			
Dutlet			
Symmetry			
🚡 Wall			
→ ∆t Time settings			
 \Delta x Numerical parameters 			
Postprocessing			
🎄 Derformance settings			

Figure III.18: Boundary conditions

boundary condition are only applied on inlets, outlets and walls.

- For the inlets, only **Prescribed value** is available.
- For the outlet, only **Prescribed value** and **Prescribed flux** are available, but they are taken into account only when the flow re-enters from the outlet.
- Otherwise, homogeneous **Prescribed flux** is considered by code_saturne.

- Inlet:

Click on the label Inlet. In the section Velocity, select norm, then in the sub-section Direction choose specified coordinates and enter the normal vector components of the inlet velocity.

For the turbulence, choose the inlet condition based on a hydraulic diameter and specify it as below:

x = 1.0 (m); y = 0.0 (m); z = 0.0 (m) hydraulic diameter = 0.5 (m)

Scroll down to choose the temperature inlet value. Here this value is **300**°C.

Calculation environment Calculation features Turbulence models Turbulence models Thermal model Body forces Species transport Volume conditions Velocity Intet Outlet Symmetry Wall Coupling parameters Aut Time settings Start/Restart Aut Numerical parameters Performance settings Turbulence Turbulence Intet Calculation by hydraulic diameter ▼ Hydraulic diameter 0.5 m	<u> </u>
4	•

Body forces	Turbulence	
Species transport		
 Wolume conditions 	Calculation by hydraulic diameter 👻 👘	
all_cells		
 E Boundary conditions 		
📄 Inlet		
Dutlet	Hydraulic diameter 0.5 m	
Symnetry		
🔁 Wall		
Coupling parameters		
→ ∆t Time settings	Thormal	
▶ ∆x Numerical parameters	Ineimat	
Postprocessing	Type Prescribed value	
Performance settings		
	temperature - Value 300	
4		-
,		

Figure III.19: Dynamic variables boundary conditions: inlet

- Wall:

As for the wall boundary zone, the specifications the user might have to give are if the wall is sliding, and if the wall is **smooth** or **rough**. In this case, the walls are fixed so the option is not selected, and the wall is considered as **smooth**.

Note that if one of the walls had been sliding, it would have been necessary to isolate the corresponding boundary faces in a specific boundary region.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📑 🖨 🕭 🥱 🖉 📓	2 🙆 2 💶 🌣
6	
Calculation environment	Nall [Wall]
👻 🕅 Mesh	
Preprocessing	Smooth or rough wall
Volume zones	
Boundary zones	Smooth wall O Rough wall
▼ ØØ Calculation features	
Turbulence models	Sliding wall
Thermal model	Stang wat
Body forces	
Species transport	Thermal
 Wolume conditions 	Inermat
all_cells	Type Prescribed (outgoing) flux
 Boundary conditions 	
Inlet	temperature - Flux 0.0
Dutlet	
Symnetry	
U Wall	
Coupling parameters	
• Δt Time settings	
• Δx Numerical parameters	
Postprocessing	
W Performance settings	
4	

Figure III.20: Dynamic variables boundary: walls

For the walls, seven conditions are available:

```
- Prescribed value
```

```
- Prescribed value (user law)

Prescribed (outgoing) flux
Prescribed (outgoing) flux (user law)
Exchange Coefficient
```

```
- Exchange Coefficient(user law)
```

- SYRTHES coupling

In this case all walls are adiabatic. So the boundary condition for the temperature will be a **Prescribed flux** set to **0**.

	case1:setup.xml - code_saturne	×
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📑 🖹 🕭 🥱 🔳 💀 🛛		
 Calculation environment Mesh Preprocessing Volume zones Boundary zones Calculation features Turbulence models Thermal model Body forces Species transport Wolume conditions all_cells Enlet Outlet Symetry Wall Coupling parameters Ar Numerical parameters Mostprocessing Performance settings 	Wall [Wall] Smooth or rough wall Sliding wall Thermal Thermat Type Frescribed value Prescribed value (user law) Flux Flux Frescribed (outgoing) flux (user law) SYRTHES coupling SYRTHES coupling	

Figure III.21: Scalars boundaries: walls

1.5 Time settings tab

To specify Time settings, click on the **Time settings** header. Choose a **Steady (local step time)** as a **Time step option**. For **Velocity-Pressure algorithm** choose **SIMPLEC**. Leave all default values except the **Number of time steps**. Modify it to 300.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📑 🗎 🏝 🥱 👌 🖉 🗐	8 2 8 2 🖪 🌣
 ⊘ ∞ Calculation environment № Mesh ₩ Calculation features ₩ Volume conditions ₩ Boundary conditions Coupling parameters 	Time step option Steady (local time step)
 ▲ Time settings Bart/Restart Δx Numerical parameters Mostprocessing Performance settings 	Reference time step0.1sMaximal CFL number1.0Maximal Fourier number10.0Minimal time step factor0.1Maximal time step factor1000.0Time step maximal variation0.1
٩	Stopping criterion Number of time steps - 300

Figure III.22: Steady flow management

As mentioned earlier in this document, be aware for a Steady analyzis, the intermediate results are not significant. Only converged results should be taken as significant values.

1.6 Numerical parameters tab

The **Numerical parameters** need then to be specified, under the header **Numerical parameters**. Now, select the **Equation parameters** item under the **Numerical parameters** folder.

Scheme The tab Scheme allows to change different more advanced numerical parameters.

In this case none of them should be changed from their default value, see III.24.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp						
📑 🗎 🕭 👌 🖻 💀		d 🔅				
ØX						
Calculation environment	Solver Scheme	Clipping				
	Name	Scheme	Blending Factor	Slope Test	Flux Reconstruction	RHS Sweep Reconstruction
Boundary conditions	pressure				\checkmark	2
Coupling parameters	velocity	Automatic	1	v	v	1
→ Δt Time settings	k	Automatic	0	\checkmark	v	1
▼ Δx Numerical parameters	epsilon	Automatic	0	V	v	1
Equation parameters	temperature	Automatic	1	\checkmark	v	1
Postprocessing						
•						

Figure III.23: Numerical parameters

Clipping The tab **Clipping** in the **Equation parameters** item allows to vanish the too small or too big value.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
📄 🚔 🕭 🥐 📓 🖗	🖉 🧕 🖻 📑 🌣		
ØX			
Calculation environment	Solver Scheme Clipping		
Mesh Add Colouidation features		Minimal	Maximal
Volume conditions	Name	value	value
Boundary conditions	temperature	20	400
Coupling parameters	· · · · ·		
∆t Time settings			
▼ Δx Numerical parameters			
Equation parameters			
Postprocessing			
Performance settings			
			,

Figure III.24: Clipping

1.7 Postprocessing tab

Click on the heading **Postprocessing**. In this folder we can change the frequency for the printing of information in the output listing.

The options are:

```
No output
Output listing at each time step
Output at every 'n' time step (the value of 'n' must then be specified)
Here and in most cases, the second option should be chosen.
```

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
📄 🖆 🖄 👌 💌 🐼 🧯 🖉) 😰 🖪 🌣
8	
Calculation environment	Output Control Writer Mesh Monitoring Points
Mesh	
▶ ØØ Calculation features	Log frequency
 Wolume conditions 	Output listing at each time step * 1
 Boundary conditions 	output using at each time step
Coupling parameters	
▶ ∆t Time settings	
▶ ∆x Numerical parameters	
Postprocessing	
Additional user arrays	
Time averages	
Volume solution control	
Surface solution control	
Profiles	
Balance by zone	
Performance settings	
4	
T	

Figure III.25: Output control: output listing

For the post-processing (by default EnSight format files), there are four options:

- No periodic output
- Output every 'n' time step
 Output every 'x' seconds
 Output using a formula

In this case, we are interested in the evolution of the variables during the calculation, so the second option is chosen, with \mathbf{n} set to 1.

In addition, in order to get the **Output at the end of calculation**, the corresponding box must be checked.

	case1:setup	o.xml - code_saturi	ne		×
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp					
📑 🖹 🕭 🥱 🖉 🗐	a 🛿 🙆 😰 🚺 🌣				
8					
Calculation environment	Output Control Writer	Mesh Monito	oring Points		
• 🖾 Mesh	Neme		F		Dimentering
Ave Calculation features	Name	Id	Form	nat	Directory
Wolume conditions	results	-1	En Si	ght po:	stprocessing
Boundary conditions					
Coupling parameters					
• Δt Time settings					
• Δx Numerical parameters			4		
Postprocessing	F				
Additional user arrays	Frequency No periodic out	tout			
Time averages	Output every 'r	n' time steps	1		
Volume solution control	Output every 'x	(' seconds	-		
Surface solution control	Output using a formula Output at end of calculation				lation
Profiles					
Balance by zone	Time-dependency				
Performance settings					
		Fixed	mesh 🔹		
	Ontions				
	Options				
	 Separate sub-writer for each mesh 				
		File type	binary (native)	•	
		Polygons	display	•	
		Polyhedra	display	•	
		yrredra	,		
4					

Figure III.26: Output control: post-processing

The other options are kept to their default value.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
📑 🖹 🕭 🥱 🖉 🗐	s 🖉 🙆 😰 🚺 🔅			
Ø 🗙				
Calculation environment	Output Control Writer	Mesh Monito	pring Points	
Mesh				-
▶ ⊕ψ Calculation features	Name	Id	Format	Directory
Image: Volume conditions	results	-1	EnSight	postprocessing
 E Boundary conditions 				
Coupling parameters				
• Δt Time settings				
• Δx Numerical parameters			4 –	
Postprocessing	Frequency			
Volumo colution control	Output every '	n' time steps	- 1	
Surface solution control	Output at str	art of colculation	Cutput at an	d of colculation
				or calculation
Balance by zone	Time-dependency			
Performance settings	Time-dependency			
* Ferrormance settings		Fixed	mesh 🔹	
	Options			
		✓ Separate :	sub-writer for each mesh	
		File type	binary (native)	
		Polygons	display 👻	
		Polyhedra	display 👻	
		. etyneara	aispia;	
4				



The **Monitoring Points** tab allows to define specific points in the domain (monitoring probes) where the time evolution of the different variables will be stored in historic files. In this case no monitoring points are defined.

Volume solution control The **Volume solution control** item allows to specify which variable will appear in the output listing, in the post-processing files or on the monitoring probes. In this case, the default value is kept, where every variable is activated.

Ø 🗙						
Calculation environment	Solution control					
Mesh Add Calculation features	Output label	Internal name	Print in listing	Post- processing	Monitoring	
Volume conditions	▼ Base		✓	v	v	
Boundary conditions	Pressure	pressure	V	V	\checkmark	
Coupling parameters	Velocity	velocity	v	V	✓	
 	total_pressure	total_pressure	V	\checkmark	\checkmark	
 	 Turbulence 		V	\checkmark	\checkmark	
 Postprocessing 	epsilon	epsilon	V	\checkmark	v	
Additional user arrays	k	k	v	✓	\checkmark	
Time averages	TurbVisc	turbulent_viscosity	\checkmark	\checkmark	V	
Volume solution control	 Thermal 		\checkmark	V	V	
Surface solution control	TempC	temperature	V	V	\checkmark	
- Drofilos	✓ Other		\checkmark	\checkmark	v	
Pelenes humana	LocalTime	local_time_step	✓	\checkmark	\checkmark	
Balance by zone	CourantNb	courant_number	\checkmark	\checkmark	v	
Performance settings	FourierNb	fourier_number	v	V	\checkmark	
	Iterative process error es	stimators				
	Prediction reconstruction off			•		
	Mass conservation	Mass conservation off		·		
	Correction reconstruct	tion	off 👻			
	Navier-Stokes sub-iterations off					

Figure III.28: Solution control

1.8 Run computation

To prepare the launch script and, on certain architectures, launch the calculation, click on the icon in the menu bar and a new window will appear as shown below: On this calculation, the number of processors used will be left to 1.

	case1:setup.xml-code_saturne	×
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📑 💼 🕭 🥱 🍖 国 🖗	6262	
 Calculation environment Calculation features Mesh Calculation features Coupling parameters Coupling parameters Coupling parameters Coupling parameters At Time settings At Numerical parameters Postprocessing Additional user arrays Time averages Volume solution control Surface solution control Pofiles Balance by zone Performance settings 	Run computation Computation Advanced Script parameters Image: Script parameters Number of processes 1 Number of processes 1 Threads per process 1 Computation start Image: Save case and run calculation	×
4		

Figure III.29: Prepare batch calculation: computer selection

Finally, the **Advanced options** icon allows to change some more advanced parameters that will not be needed in this simple case.

Eventually, save the xml file and execute it by clicking on Save case and run calculation. The results will be copied in the \square RESU/ directory.

Once you run the calculation process the following III.30 window appears. This window allows you either to access to multiple instantaneous calculation information either to save, to stop or to kill the process. You can also access to the convergence tool providing scalars such as residuals, etc.

Number of lines	Save as	Stop now	Stop at	Kill	convergence tool	
runber of tilles	Jure us		Stop at	Ritt	convergence toot	
	cod	e_saturne ======				
Version: 7.0. Path: /hom	.1-patch ne/					
Result director /home/	ry: /CODE_SAT	JRNE_TUTORIA	LS/simple_j	unction/cas	e1/RESU/20210923	-1536
Copying base se	etup data					
Compiling and 1	linking use	r-defined fu	Inctions			
Preparing calcu	ulation data	a -				
Single process	sor code_sa	turne simula	tion.			
Preprocessing o	calculation					
Starting calcul	lation					
Post-calculatio	on operatio	ıs				
			ОК			

Figure III.30: Run