#### EDF R&D



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

6, quai Watier F-78401 Chatou Cedex

MAY 2020

Code\_Saturne documentation

Code\_Saturne version 6.0 tutorial: simple junction

contact: saturne-support@edf.fr



EDF R&D	Code_Saturne version 6.0 tutorial: simple junction	Code_Saturne documentation Page 1/44
---------	----------------------------------------------------	--------------------------------------------

#### TABLE OF CONTENTS

	I Introduction	3
1	Introduction	4
1.1	Code_Saturne SHORT PRESENTATION	4
1.2	About this document	4
1.3	Code_Saturne COPYRIGHT INFORMATIONS	4

	II Simple junction testcase	<b>5</b>
1	Study description	6
1.1	Study creation and preparation	6
1.2	Objective	7
1.3	Description of the configuration	7
1.4	Characteristics	8
1.5	Mesh characteristics	8
2	CASE 1: Basic calculation	8
2.1	CALCULATION OPTIONS	8
2.2	Initial and boundary conditions	9
2.3	Parameters and User routines	10
2.4	RESULTS	10

	III	Step by step solution	
1	Solution for CASE1		13

# 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).

## **1.2 About this document**

The present document is a tutorial for *Code\_Saturne* version 6.0. 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. Here, the study directory  $\boxdot$  simple\_junction will contain a single calculation directory  $\boxdot$  case1.

Create the study <code>msimple\_junction</code> and the <code>mcase1</code> within SALOME \_CFD module CFDStudy as explained in the Shear driven cavity tutorial.

Alternatively, this can be done by typing the following commands in your terminal:

```
$ salome shell
$ code_saturne create -s simple_junction -c case1
```

Here, the first command salome shell loads the complete SALOME environment in your terminal, and allows to use *Code\_Saturne* command and all its subcommands, as well as to launch, for example, ParaView (simply type the command paraview).

*Code\_Saturne* Graphical User Interface (GUI) can be launched by typing the command lines as below:

```
$ cd simple_junction/case1/DATA
$ ./SaturneGUI &
```

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



Figure II.1: Code\_Saturne (GUI) graphic window

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

EDF R&D

\$ cd simple\_junction/MESH/

\$ cp /data/projets/echanges.004/ITECH\_CS\_TRAINING/meshes/1-simple\_junction/downcomer.des

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  $\bigcirc$  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.2. 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.2: 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.3 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.

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.3 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.

#### $\mathbf{Type}:$ 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

"Color" or group definition: see figure II.4. To specify boundary conditions on the boundary faces of the mesh, the latter have to be identified. It was commonly done by assigning an integer to each of them, this integer was then characteristic of the boundary group they belong to. This integer is referred to as **color** or **reference**. It is more common now to assign a group name during the meshing step as done in the first tutorial (see lid-driven cavity tutorial).

## 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:



Figure II.3: Mesh of the downcomer

Outlet

- $\rightarrow\,$  Time settings: steady algorithm with constant relaxation coefficient (Velocity-Pressure algorithm is the SIMPLE one)
- $\rightarrow$  Turbulence model:  $k-\epsilon$
- $\rightarrow$  Scalar(s): 1 temperature
- $\rightarrow$  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 colors used for boundary conditions and table II.2 defines the correspondance between the colors 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).



Figure II.4: Colors of the boundary faces

Colors	Conditions
1	Inlet
5	Outlet
$2\ 3\ 4\ 6\ 7$	Wall
8 9	Symmetry

Table II.2: Boundary conditions 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. They are specified in the following table:

Calculation control parameters				
Pressure-Velocity coupling	SIMPLE algorithm			
Number of iterations	300			
Relaxation coefficient	0.9			
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  $\bigcirc$  simple\_junction and the  $\bigcirc$  case1 within SALOME \_CFD module CFDStudy as explained in the Shear driven cavity tutorial.

Alternatively, this can be done by typing the following commands in your terminal:

```
$ salome shell
$ code_saturne create -s simple_junction -c case1
```

Here, the first command salome shell loads the complete SALOME environment in your terminal, and allows to use *Code\_Saturne* command, as well as to launch ParaView (simply type the command paraview).

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

\$ cd simple\_junction/case1/DATA
\$ ./SaturneGUI &

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



Figure III.1: Code\_Saturne (GUI) graphic window

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

```
$ cd simple_junction/MESH/
$ cp /data/projets/echanges.004/ITECH_CS_TRAINING/meshes/1-simple_junction/downcomer.des
.
```

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

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
📑 🖆 🏝 🕱 🔄 🖬 🖗	2	2 🌣		
0 × 1				
= 📄 Calculation environment	Study and	d case directories		
🔒 Notebook	Study	/home/jo680ben/simple-ji	unction	
🕀 🔜 Mesh				
$\oplus \phi \psi$ Calculation features	Case	casel		
🕀 🖻 Closure modeling				
ρμ Fluid properties	Associate	d case sub-directories		
🛨 📄 Particles and droplets tracking		Data	DATA	
🛨 🍡 Volume zones		Desults	DECH	
🕀 👥 Boundary zones		Results	RESU	
$\pm \Delta t$ Time settings		User sources	SRC	
⊕ Δx Numerical parameters				
🕀 🖂 Postprocessing		Run scripts	SCRIPTS	
Performance settings				
📑 OpenTurns study				
C				

Figure III.2: Identity and paths

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

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

The list of meshes appears in the window List of meshes. In this case only the mesh downcomer.des is needed. The **Periodic Boundaries** is not used in this case so **Preprocessing** page does not need

📑 🖿 🏝 🔄 🖻 🖗	22			
Calculation environment Calculation environment  Mesh Preprocessing	Mesh input Import meshes	Use existing	) mesh inp	ut
+ AW Calculation features       AH Fluid properties       + Outme zones	/MESH	ry (optional)		<b>D</b>
At Time settings	List of meshes			
$\pm \Delta x$ nime settings $\pm \Delta x$ Numerical parameters	File name	Format	Reorient	Path
	downcomer.des	Simail/NOPO		
Performance settings			4	Þ —
	Execution mode			
	Standard Computat	ion		<b>v</b>
	✓ Use unmodified of ✓ Save mesh if mo	checkpoint mesh dified by preprod	i in case of essing	restart

Figure III.3: Meshes: list of meshes

to be visited. Keep the default values.

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.

📑 🖻 🕭 🔞 🖻 📴 🚱	22					
Calculation environment  Mesh  Calculation features  H Fluid properties  Calculation seatures  Calculation features  Calculation fe	Mesh input <ul> <li>Import meshes</li> </ul> Local mesh directo/MESH List of meshes	○ Use existing ry (optional)	g mesh inp	ut		
$\oplus \Delta t$ Time settings	File name	Format	Reorient	Path		
+ Δx Numerical parameters	downcomer des	Simail/NOPO				
<ul> <li>Performance settings</li> </ul>						
	Import mesh only		í.	3		
	Mesh preprocessing	g only				
	Ex Mesh quality criteri	a only				
	Standard Computation					
	✓ Use unmodified checkpoint mesh in case of restart					
	🗹 Save mesh if mo	dified by prepro	cessing			

Figure III.4: Preprocessing and calculation modes in Code\_Saturne

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. Now, let's choose a turbulence model for our simulation. To do so, go **Turbulence models** sub-folder

📄 💼 🚵 🔄 🙋 🔜 🖗	22		
) X			
🕀 📄 Calculation environment	Flow Models		
🛨 🛄 Mesh	<ul> <li>Standard Eulerian single phase</li> </ul>	Incompressible	~
$= \phi \psi$ Calculation features	Atmospheric		
Turbulence models	0		
📑 Thermal model	<ul> <li>Electric arcs</li> </ul>		
📑 Body forces	Groundwater		
🗒 Conjugate heat transfer	Departive flows (combustion)		
🗟 Species transport			
📑 Fans	🔘 Homogeneous Eulerian - VoF model		
PH Fluid properties	Eulerian multiphase (NEPTUNE_CFD)		
🕀 🍡 Volume zones	- , , , , , , , , , , , , , , , , , , ,		
🕀 👥 Boundary zones	Additional Features		
$\pm \Delta t$ Time settings	Eulerian-Lagrangian model	off	~
$\pm \Delta x$ Numerical parameters			
🕀 🖂 Postprocessing	Turbomachinery model	None	~
Performance settings	Deformable mesh (ALE method)		
	🗹 Fans (source-term model)		

Figure III.5: Flow modelling

and open **Turbulence model** drop-down menu.

<u>F</u> il	e <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
	] 🗎 🕭 🔄 🖉 🖗	]	No model (i.e. laminar flow)	
	\$		Mixing length	
+	Calculation environment	Tur	k-epsilon	
+	Mesh		k-epsilon Linear Production	
Ξ	φψ Calculation features		Rii-epsilon LRR	
	Turbulence models		Rii-epsilon SSG	
	📑 Thermal model			
	Body forces			
	Conjugate heat transfer	Ref	v2fBL-v2/k	ence)
	Species transport		k-omega SST	
	ρμ Fluid properties		Spalart-Allmaras	
+	Volume zones	L	LIES (Smagorinsky)	
+	🐏 Boundary zones		LLS (Shiagoniisky)	
+	$\Delta t$ Time settings		LES (classical dynamic model)	
+	$\Delta x$ Numerical parameters		LES (WALE)	
+	🖾 Postprocessing			
÷	🌞 Calculation management			

Figure III.6: Turbulence model: list of models

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

🗋 💼 🏝 🚳 🕐 🖬 🖗	
o x	
🕀 📄 Calculation environment	Turbulence model
🕀 🛄 Mesh	k-ensilon
□ φψ Calculation features	
Turbulence models	
📑 Thermal model	Advanced options  🛞
Body forces	
🗎 Conjugate heat transfer	Peteronee values (used for initialization of turbulence)
Species transport	
📑 Fans	Velocity scale 1.0 m/s
ρμ Fluid properties	length scale Automatic V
🕀 🍡 Volume zones	
🕀 👥 Boundary zones	
$\pm \Delta t$ Time settings	
$\pm \Delta x$ Numerical parameters	
🕀 🖂 Postprocessing	
Performance settings	
<	

Figure III.7: Turbulence model: choice of a model

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



- Enthalpy (J/kg)

<ul> <li>Calculation environment</li> <li>Mesh</li> <li>Calculation features</li> <li>Turbulence models</li> <li>Thermal model</li> <li>Body forces</li> <li>Conjugate heat transfer</li> <li>Species transport</li> <li>Species transport</li> <li>Fans</li> <li>Volume zones</li> <li>Au Time settings</li> <li>Au Numerical parameters</li> <li>Postprocessing</li> <li>Performance settings</li> </ul>	📄 🖹 👌 👌 📓 🖗	
* Performance settings	Image: Conjugate heat transfer         Image: Conjugate heat transfer <td< th=""><th>Image: Second state of the second s</th></td<>	Image: Second state of the second s
	<ul> <li></li></ul>	

Figure III.8: Thermal scalar conservation: list of models

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

📑 🖹 🏂 🧔 🖬 👰	2 🖻 🖉 🔅
0 🗙	
🕀 📄 Calculation environment	Thermal scalar
🕂 🔜 Mesh	Temperature (Celsius) 🗸
$= \phi \psi$ Calculation features	
Turbulence models	Thermal radiative transfers
📄 Thermal model	No radiative transfers 🗸
Body forces	
🗒 Conjugate heat transfer	
Species transport	
🕞 Fans	
P.H. Fluid properties	
🗄 📕 Volume zones	
🕀 👥 Boundary zones	
$\pm \Delta t$ Time settings	
$\pm \Delta x$ Numerical parameters	
🕀 🖂 Postprocessing	
Performance settings	
«	

Figure III.9: Thermal scalar conservation: choice of a model

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

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).

🗈 🖹 🏂 👌 🙋 🗖 🖗		2 🖄	
ØX			
🕀 📄 Calculation environment	Gravi	ty	
🛨 🔜 Mesh	g×	0.0	m/s <sup>2</sup>
□ φψ Calculation features			
Turbulence models	9 <sub>y</sub>	0.0	m/s²
📄 Thermal model		0.0	m/c <sup>2</sup>
Body forces	9z	0.0	
🗎 Conjugate heat transfer	Corio	lis source terms (rotat	tion vector)
📑 Species transport			
📑 Fans	$\Omega_{x}$	0.0	S.,
P. Fluid properties	O.,	0.0	s-1
🕀 🍡 Volume zones	y		
🕀 👥 Boundary zones	Ω"	0.0	S <sup>-1</sup>
$\pm \Delta t$ Time settings			
$\pm \Delta x$ Numerical parameters			
🕀 🖂 Postprocessing			
Performance settings			
< > >			

Figure III.10: Body forces

#### Initialization:

To initialize variables at the instant t = 0 (s), go to the **Initialization** item under the heading **Volume zones**. Here the velocity, the thermal scalar and the turbulence can be initialized.

In this case, the values te be set are: zero velocity (default) and an initial temperature of  $20^{\circ}$ C. Specific zones can be defined with different initializations. In this case, only the default **all cells** is used.

📑 눹 📐 🥠 👌 国 🖗	2 2 2
Calculation environment	Initialization Volume zone all_cells
⊕ ₩ Calculation features	
<ul> <li>Pluid properties</li> <li>Volume zones</li> </ul>	Velocity 🛃
■ Initialization	Thermal 🛃
$\oplus \Delta t$ Time settings	Turbulence Initialization by reference value(s)
Performance settings	

Figure III.11: Initialization of the scalar, velocity and turbulence

• Click on the icon near **Thermal** in order to specify the initial value of the thermal scalar. It can be a value or a user expression.

EDF R&D	<i>Code_Saturne</i> version 6.0 tutorial: simple junction	Code_Saturne documentation Page 23/44
<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>V</u>	Mathematical expression editor	×
	temperature = 20.;	
H Mesh	ironment	
$\oplus \phi \psi$ Calculation feat	ures	
$\rho\mu$ Fluid properties		
🗆 📕 Volume zones		
Initialization		
🕀 🐏 Boundary zones		
$\oplus \Delta t$ Time settings		
$\oplus \Delta x$ Numerical paran	neters	
<ul> <li>Postprocessing</li> <li>Performance se</li> </ul>	ttings	

Figure III.12: Initialization of the scalar

<u> 🖉 о</u>к

e <u>A</u>nnuler

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

	Mathematical expression editor	×
File       Edit       Tools       Window       H         Image: State of the stat	Mathematical expression editor User expression Predefined symbols Examples temperature = 20.;	*
<ul> <li>Polume zones</li> <li>Initialization</li> <li>Boundary zones</li> <li>∆t Time settings</li> <li>∆x Numerical parameters</li> <li>Mostprocessing</li> <li>Performance settings</li> </ul>		
<	<u>Annuler</u>	

Figure III.13: Initialization of the velocity

Under the heading **Fluid properties** in the main list, we can specify reference values of some physical quantities and the physical properties of the fluid.

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

:	🖹 🏝 🔙	2 🛛 🦻	2	2 🌣		
		ð 🗶				^
+	Calculation envir	onment	Referen	e total pressure		- I.
+	Mesh		value	101325.0	Pa	
$+\phi,\psi$	Calculation featu	ires				

Figure III.14: Physical properties: reference pressure

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

- Density
- Viscosity
- Specific Heat
- Thermal Conductivity

In this case they are all constant.

- $\rho$  = 725.735 kg.m<sup>-3</sup>
- $\mu$  = 0.895 × 10<sup>-4</sup> kg.m<sup>-1</sup>.s<sup>-1</sup>
- $C_p = 5\,483 \; J.kg^{-1}.K^{-1}$
- $(\lambda/C_p) = 0.02495 \ W.m^{-1}.K^{-1}$

📑 🖆 🏝 🥱 🙋 国 🖗	2 🖻 😰 🌞
Ø×	
🕀 📄 Calculation environment	Reference total pressure
+ Mesh	value 101325.0 Pa
+ 👾 Calculation features	
PP Fluid properties	Reference temperature
🖃 🌄 Volume zones	value 20.0 °C
Initialization	
🕀 👥 Boundary zones	(used for properties initialization)
$\pm \Delta t$ Time settings	Density
$\pm \Delta x$ Numerical parameters	
🕀 🖾 Postprocessing	constant V
Performance settings	Reference value p 725.735 kg/m <sup>3</sup>
	Viscosity constant
	Reference value µ 8.951e-05 Pa.s
	Specific heat
	constant 🗸
	Reference value $C_{\rho}$ 5483.0 J/kg/K
	Constant     V
	Reference value λ 0.02495 W/m/K
c 2	

Figure III.15: Physical properties: fluid properties

Boundary conditions now need to be defined. Go to the **Boundary zones** heading. The following window opens (fig III.16).

📄 💼 🏝 🚳 👌 🔳 🖗	
	Boundary regions definition
Calculation environment	
+ Mesh	Label Zone Nature Selection criteria
+ ev Calculation features	
PP4 Fluid properties	
🕂 🛃 Volume zones	
🖃 🔛 Boundary zones	
Boundary conditions	
$\pm \Delta t$ Time settings	
$\pm \Delta x$ Numerical parameters	
🕀 🖾 Postprocessing	
Performance settings	
	Add Delete
	Add from preprocessor log
	import groups and references from preprocessor tog
<	

Figure III.16: 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 their color or on geometrical conditions. For each zone, a reference number, a label, a nature and a selection criteria must be assigned. The different natures that can be assigned are:

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

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 **Zone** number can be any integer. It will be used by the code to identify the zone. No specific order or continuity in the numbering is needed.

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

📄 🖹 🏂 👌 🙋 🖬 🖗	2 2 2 🌣	
<ul> <li>② ∑</li> <li>① Calculation environment</li> <li>① Mesh</li> <li>① Mc Calculation features</li> <li>② Fluid properties</li> <li>② Fluid properties</li> <li>③ Volume zones</li> <li>③ Boundary zones</li> <li>③ Boundary conditions</li> <li>① Δt Time settings</li> <li>① Δx Numerical parameters</li> <li>④ Postprocessing</li> <li>③ Performance settings</li> </ul>	Boundary regions definition         Label       Zone       Free inlet/outlet         BC_1       Inlet       Symmetry         Vall       Imposed P Outlet       Outlet	Selection criteria
	Add Add from preprocessor log Import groups and references	Delete

Figure III.17: Creation of a boundary region

The specification of the inlet condition is detailed in the following pages. The settings will be as follows:

```
Label: Inlet,
Zone: 1,
Nature: Inlet,
Selection criteria: 1
```

Type all the information in the fields, the result diplays as figure III.18

6 x 1		
🕀 🗒 Calculation environment	Boundary regions definition	
🛨 🔜 Mesh	Label Zone Nature	Selection criteria
⊕      ψ     Calculation features	BC_1 1 Inlet 1	
ρμ Fluid properties		
🕀 🍡 Volume zones		
🖃 🐏 Boundary zones		
Boundary conditions		
$\pm \Delta t$ Time settings		
$\pm$ $\Delta x$ Numerical parameters		
🕀 🖂 Postprocessing		
🏶 Performance settings		
	A	dd Delete
	Add from preprocessor log	
	Import groups and refe	erences from preprocessor log 🛛 📔

Figure III.18: Creation of a boundary region

Remember to save the xml file regularly!

Do the same thing for the other boundaries.

In our case, colors 8 and 9 are symmetry boundaries. One option can be to define a separate zone for each color, as follows:

Label	$symmetry_1$	$symmetry_2$
Zone	3	4
Nature	symmetry	symmetry
Localization	8	9

Figure III.19	: Syn	imetric	boundary	conditions
()	•/		•/	

But it is usually faster to regroup the different colors in one single zone, as shown on figure III.20. In our case, the localization for this zone is the string ''8 or 9''.

📄 🖆 🕭 🤌 🖉 📓 🔤	2 🛛 🖓	٥	ŧ	
Bi Calculation environment	oundary regio	ns def	inition	
🗄 🔜 Mesh	Label	Zone	Nature	Selection criteria
$\oplus \phi \psi$ Calculation features	Inlet	1	Inlet	1
ρμ Fluid properties	Outlet	2	Outlet	5
🕀 🍡 Volume zones	Conter	-	Conter	
😑 🔜 Boundary zones	Symmetry	3	Symmetry	8 or 9
Boundary conditions				
$\pm \Delta t$ Time settings				
$\pm \Delta x$ Numerical parameters				
🕀 🖂 Postprocessing				
Performance settings				
	Add from pre	e <b>proce</b> ort gro	Ad ssor log ups and refe	d Delete

Figure III.20: Creation of boundary regions: symmetry region

The same treatment must be done for the wall conditions. All colors 2, 3, 4, 6 and 7 can be grouped in a single boundary zone.

After defining all the boundary zones, the Interface window will look as in figure III.21.

📄 💼 🕭 🥱 🖻 🔳 🖗	22	$\mathbf{x}$		
	Boundary regio	ns defi	nition	
	Label	Zone	Nature	Selection criteria
± ₩ Calculation features	Inlet	1	Inlet	1
<u>P</u> Fluid properties	a di di	-		-
🗄 🎩 Volume zones	Outlet	2	Outlet	5
🗆 🔜 Boundary zones	Symmetry	3	Symmetry	8 or 9
Boundary conditions	Walls	4	Wall	2 or 3 or 4 or 6 or 7
${}_{\mathbb{H}} \Delta t$ Time settings				1
${ar ar D} \Delta x$ Numerical parameters				
🗉 🖂 Postprocessing				
Performance settings				
	Add from pre	proces	Ad ssor log ups and refe	d Delete rences from preprocessor log

Figure III.21: Creation of boundary regions

Now that the boundary zones are defined, the boundary conditions assigned to them will be specified. Click on the **Boundary conditions** sub-folder to set the inlet boundary conditions for velocity, turbulence and themal scalar.

As shown on figure III.22, outlet and wall boundary zones also appear in the window.

📄 🖹 🍐 👌 🖉 🔄 🖗	22	\$		
0 2				
🕀 📑 Calculation environment	Boundary cond	ditions		
🕂 🔜 Mesh	Label	Zone	Nature	Selection criteria
$\pm$ $\phi\psi$ Calculation features	Inlet	1	inlet	1
ρμ Fluid properties	Outlet	2	outlet	5
🕀 🍡 Volume zones	Walls	4	wall	2 or 3 or 4 or 6 or 7
🗆 👥 Boundary zones				
Boundary conditions				
$\pm \Delta t$ Time settings				
$\pm \Delta x$ Numerical parameters				
🕀 🖂 Postprocessing				
Performance settings				
<				

Figure III.22: Dynamic variables boundary conditions

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)

📑 🖹 🤔 🥱 🖻 🔄	2 2	ø		
8				
🕀 📄 Calculation environment	Boundary cond	itions		
+ Mesh	Label	Zone	Nature	Selection criteria
$+ \phi \psi$ Calculation features	Inlet	1	inlet	1
ρμ Fluid properties	Outlet	2	outlet	5
🧧 Volume zones	Walls	4	wall	2 or 3 or 4 or 6 or 7
Boundary zones				
Boundary conditions				
$\Delta t$ Time settings				
Δx Numerical parameters				
≥ Postprocessing	0.5			
	Mapper Velocity norm Directio	<b>i Inlet</b>	♥ 1.0 speci	m/s 🕎
	Turbulence	X 1.0	Y 0.0	Z 0.0
<		Hyd	draulic diameter	0.5 m

Figure III.23: Dynamic variables boundary conditions: inlet

۲ P 2 R R 6 X 🕀 📑 Calculation environment Velocity 🗆 📕 Mesh ~ 1.0 P2 norm m/s Preprocessing 2 Direction specified coordinates V  $-\phi\psi$  Calculation features 🗟 Turbulence models Y 0.0 X 1.0 Z 0.0 📄 Thermal model 🗟 Body forces 📑 Conjugate heat transfer Turbulence Species transport 2 Calculation by hydraulic diameter 🗸 ρμ Fluid properties 🗄 📕 Volume zones 🖃 🐏 Boundary zones Boundary conditions Hydraulic diameter 0.5 m  $\pm \Delta t$  Time settings  $\pm \Delta x$  Numerical parameters 🕀 🔜 Postprocessing Performance settings Thermal Type Prescribed value 2 temperature 🗸 Value 300.0

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

Figure III.24: Dynamic variables boundary conditions: inlet

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.

A      A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A  A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A     A	2 2 2	٥		
🕀 🗟 Calculation environment	Boundary cond	litions		
Mesh	Label	Zone	Nature	Selection criteria
Preprocessing	Inlet	1	inlet	1
$-\phi\psi$ Calculation features	Outlet	2	outlet	5
🗎 Turbulence models	Walls	4	wall	2 or 3 or 4 or 6 or 7
📄 Thermal model				
🗟 Body forces				
🗟 Conjugate heat transfer				
🗎 Species transport				
ρμ Fluid properties				
📕 Volume zones	Smooth or	rough wall		
🗈 👥 Boundary zones		0	<ul> <li>Smooth wall</li> </ul>	Rough wall
Boundary conditions				
$\Delta t$ Time settings				
Ξ Δx Numerical parameters	Suaing	watt		
🗈 🖂 Postprocessing				
Performance settings	Thermal			
	ten	nperature	Type Prescr Flux	ibed (outgoing) flux 🔽 🖉

Figure III.25: Dynamic variables boundary: walls

The boundary conditions on the temperature are only applied on inlets, outlets and walls.

For the walls, three conditions are available:

```
Prescribed value
-
- Prescribed flux
```

```
- Exchange Coefficient
```

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.

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

🗋 ៉ 🤌 📩 👘	2 🙎 🙎	<b>\$</b>		
0 ×				
Calculation environment	Boundary cond	ditions		
Mesh	Label	Zone	Natu	re Selection criteria
Preprocessing	Inlet	1	inlet	1
$\phi \psi$ Calculation features	Outlet	2	outle	t 5
🗎 Turbulence models	Walls	4	wall	2 or 3 or 4 or 6 or 7
📄 Thermal model				
🗒 Body forces				
📄 Conjugate heat transfer				
🗒 Species transport				
<u>P</u> Fluid properties				
🔩 Volume zones	Smooth or	rough wall		
👥 Boundary zones		(	Smooth	wall 🔘 Rough wall
Boundary conditions				
$\Delta t$ Time settings		wall		
Δx Numerical parameters	Siding	watt		
2 Postprocessing				Prescribed value
🏶 Performance settings	Thermal			Prescribed value (user law)
			Туре	Prescribed (outgoing) flux
	ten	nperature 🗸	Elux	Prescribed (outgoing) flux (user law)
			rtux	Exchange coefficient
				Exchange coefficient (user law)

Figure III.26: Scalars boundaries: walls

To specify Time settings, click on the **Time settings** header. Choose a **Steady (constant relaxation coefficient)** as a **Time step option**. For **Velocity-Pressure algorithm** choose **SIMPLE**. Modify the **Relaxation coefficient** to 0.9 and the **Number of time steps** to 300.

The **Numerical parameters** need then to be specified, under the header **Numerical parameters**. In this case, the **SIMPLE** algorithm must be chosen. This choice is not available for all **Time step option** choices.

📄 💼 🏝 😏 🦿 🔄 👳	
ðx	
🕀 📄 Calculation environment	
🕂 🔜 Mesh	Time step option $\left. Steady \right.$ (constant relaxation coefficient) $\left  \lor \right.$
$\pm \phi \psi$ Calculation features	
ρμ Fluid properties	Velocity - ressure augorithm Shire LL
🗆 🍡 Volume zones	
Initialization	
🖃 👥 Boundary zones	Relaxation coefficient 0.9
Boundary conditions	
😑 🕰 Time settings	
🗎 Start/Restart	Stopping criterion Number of time steps 💙 300
$\pm \Delta x$ Numerical parameters	
🕀 🖂 Postprocessing	
Performance settings	

Figure III.27: Steady flow management

Now, select the **Equation parameters** item under the **Numerical parameters** folder. The tab **Scheme** allows to change different more advanced numerical parameters.

In this case none of them should be changed from their default value.

📄 🖹 👌 🙋 国 🖗	2	2 🌣				
@ X						
🕀 📄 Calculation environment	Solver S	cheme Clipping				
🗄 🔜 Mesh						
$\oplus \phi \psi$ Calculation features	Nam	e Scheme	Blending	Slope	Flux	RHS Sweep
ρμ Fluid properties			Factor	Test	econstructio	econstructio
🕀 🎩 Volume zones	pressu	ıre				2
🕀 🐏 Boundary zones	veloci	ty Automatic	1	<b>Z</b>	<b>Z</b>	1
$\pm \Delta t$ Time settings	k	Automatic	0	<b>Z</b>		1
$\Box \Delta x$ Numerical parameters	R III	Automatic				-
Equation parameters	epsilo	on Automatic	0		✓	1
🕀 🖂 Postprocessing	tempera	ture Automatic	1			1
🏶 Performance settings						

Figure III.28: Numerical parameters

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

📑 🖆 🖄 🔶 🖬 🖗	2 🖻 🔒 🌣		
8			
🕀 📄 Calculation environment	Solver Scheme Clipping		
🕀 🔜 Mesh			
$\pm \phi \psi$ Calculation features	Name	Minimal	Maximal
ρμ Fluid properties		value	value
🕀 🌄 Volume zones	temperature	20	400
+ 🐏 Boundary zones			
$\pm \Delta t$ Time settings			
$= \Delta x$ Numerical parameters			
📄 Equation parameters			
🗄 🖂 Postprocessing			
Performance settings			
< (			

Figure III.29: Clipping

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.



Figure III.30: 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.

Calculation environment	Output Control	Writer	Mesh	Particles mes	Monit	toring Points		
Mesh							France	Discutores
φψ Calculation features	Na	ame		DI			Format	Directory
ρμ Fluid properties	res	ults		-1			EnSight	postprocessing
Volume zones								
👥 Boundary zones								
$\Delta t$ Time settings								
Δx Numerical parameters						, _		
🛛 🖾 Postprocessing	Frequency	No peri	iodic ou	tout				
Additional user arrays		Output		'n' time stons			1	
Time averages		Output	every	in time steps			L	
Volume solution control		Output every 'x' seconds					🗹 Output at end	of calculation
Surface solution control		Output	using a	a formula				
Profiles	Time-depende	ncy						
Balance by zone					Fixed me	esh	~	
🏶 Performance settings								
	Options							
				🗹 Sep	arate sul	b-writer for e	each mesh	
				File typ	e	binary (nat	ive) 🗸	
				Polygo	ns	display	~	
						[		

Figure III.31: Output control: post-processing

The other options are kept to their default value.

	22	٥						
× 6								
Calculation environment	Output Contro	ol Writer M	esh   Particles mesh   M	onitoring Points				
+ Mesh	Na		ц	Earmat	Directory			
⊕ ♠ψ Calculation features	Na	ame	DI	Format	Directory			
βμ Fluid properties	res	ults		EnSight	postprocessing			
🕀 🍡 Volume zones								
🗄 👥 Boundary zones								
$\pm \Delta t$ Time settings								
$\pm \Delta x$ Numerical parameters	Frequency							
🖃 📨 Postprocessing		Output every 'n' time steps						
📑 Additional user arrays								
📑 Time averages		Output at	start of calculation	S Output at er	nd of calculation			
📄 Volume solution control								
🗟 Surface solution control	Time-depend	ency						
📄 Profiles			Fixed me	sh 🗸				
🕞 Balance by zone								
Performance settings	Options							
			🗹 Separate sub	-writer for each mesh				
			File type	binary (native)				
			Polygons	display 🗸				
			Polyhedra	display 🗸				
,								

Figure III.32: Output control

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.

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.

📄 🖹 🖄 👌 🖻 🔄	2 2 🕸				
Ø 🗙					
🕀 📄 Calculation environment	Solution control				
Mesh     Mesh     Acculation features	Output label	Internal name	Print in listing	Post- processing	Monitoring
ρμ Fluid properties	Base		<b>I</b>	<b>Z</b>	
• Volume zones	Pressure	pressure	<b>Z</b>		2
🗄 👥 Boundary zones	Velocity	velocity	<b>S</b>	<b>S</b>	<b>Z</b>
$\pm \Delta t$ Time settings	total_pressure	total_pressure	<b>I</b>	<b>S</b>	<b>a</b>
+ $\Delta x$ Numerical parameters	<ul> <li>Turbulence</li> </ul>		<b>S</b>	<b>Z</b>	
🛛 🔜 Postprocessing	epsilon	epsilon			
🗎 Additional user arrays	k	k		$\checkmark$	
📑 Time averages	TurbVisc	turbulent_viscosity	<b>Z</b>	$\checkmark$	
Volume solution control	<ul> <li>Thermal</li> </ul>				<b>S</b>
🕞 Surface solution control	TempC	temperature		<b>S</b>	
Profiles					
🗎 Balance by zone	Iterative process error es	timators			
🏶 Performance settings	Prediction reconstruction	on off		~	
	Mass conservation	off		~	
	Correction reconstruction	on off		~	
	Navier-Stokes sub-itera	ations		~	
< >	Navier-Stokes sub-itera	ations off		~	

Figure III.33: Solution control

To prepare the launch script and, on certain architectures, launch the calculation, click on the **prepare** icon in the menu bar and a new window will appear as shown below:

	Run computation					
🗄 📄 Calculation environment						
🗄 🔜 Mesh	Computation Advanced					
🕂 👾 Calculation features						
ρμ Fluid properties	Script parameters					
🗄 📕 Volume zones						
🗄 🐏 Boundary zones						
$\pm \Delta t$ Time settings	Result subdirectory name					
$\pm \Delta x$ Numerical parameters						
+ 🔜 Postprocessing	Number of processes					
🌞 Performance settings	Threads per process 1					
	Computation start					
	Cancel Apply Save of	ase and run calculation				

Figure III.34: Prepare batch calculation: computer selection

On this calculation, the number of processors used will be left to 1.

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  $\Box$  RESU/ directory.



Figure III.35: Run