#### EDF R&D



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: simple junction

contact: saturne-support@edf.fr



EDF R&D	<i>Code_Saturne</i> version 5.0 tutorial: simple junction	$\begin{array}{c} Code\_Saturne\\ documentation\\ Page 1/51 \end{array}$
	simple junction	r age 1/01

#### TABLE OF CONTENTS

I Introduction					
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	6
1.3	Description of the configuration	6
1.4	Characteristics	6
1.5	Mesh characteristics	7
2	CASE 1: Basic calculation	8
2.1	CALCULATION OPTIONS	8
2.2	Initial and boundary conditions	8
2.3	Parameters and User routines	9
2.4	RESULTS	10

	III	Step by step solution	12
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 5.0. 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 before running *Code\_Saturne* is to prepare the computation directories. In this first example, the study directory  $\boxdot$  simple\_junction will be created, containing a single calculation directory case1.

Create the study cisimple\_junction and the case1 with SALOME module CFDSTUDY.

Alternatively, you can use the command line and some *Code\_Saturne* commands. Create first an **alias**<sup>1</sup> to the *Code\_Saturne* command and then type the command:

\$ code\_saturne create -s simple\_junction -c case1

The mesh files should be then copied in the directory  $\boxdot$  MESH/.

Finally, if you have set up the study with the command line, the *Code\_Saturne* Graphical User Interface (GUI) is launched by typing the command lines as below:

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

#### 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.1. 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.

#### 1.4 Characteristics

Characteristics of the geometry and the flow:

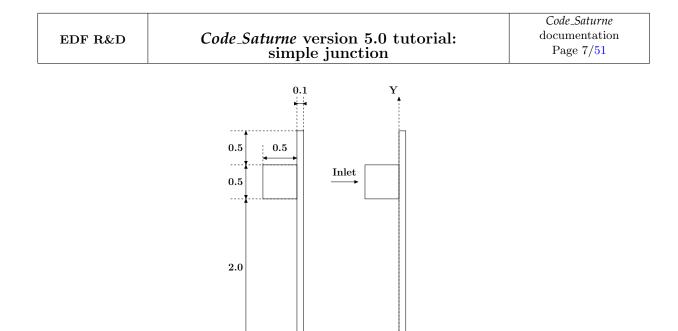
Height of downcomer	$H = 3.00 \ m$
Thickness of downcomer	$E_d = 0.10 \ m$
Diameter of the cold branch	$D_b = 0.50 \ m$
Inlet velocity of fluid	$V = 1 \ m.s^{-1}$

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  $150 \times 10^5 Pa$ :

<sup>&</sup>lt;sup>1</sup>Create a file .bash\_aliases in your home directory and add the line "alias code\_saturne='PATH\_TO\_CS/code\_saturne'." PATH\_TO\_CS should be *Code\_Saturne* installation bin directory.



Downcomer Frame Outlet

Ô

x

Figure II.1: Geometry of the downcomer

- 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.2 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.2 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.1)

#### Mesh generator used: SIMAIL

**Color definition**: see figure II.3. To specify boundary conditions on the boundary faces of the mesh, the latter have to be identified. It is commonly done by assigning an integer to each of them, characteristic of the boundary region they belong to. This integer is referred to as **color** or **reference**.

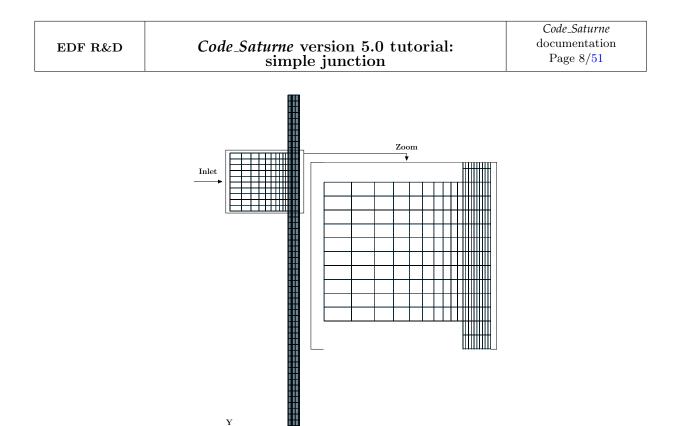


Figure II.2: Mesh of the downcomer

## 2 CASE 1: Basic calculation

#### 2.1 Calculation options

Most of the options used in this calculation are default options of Code\_Saturne.

Outlet

- $\rightarrow\,$  Flow type: steady flow
- $\rightarrow$  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

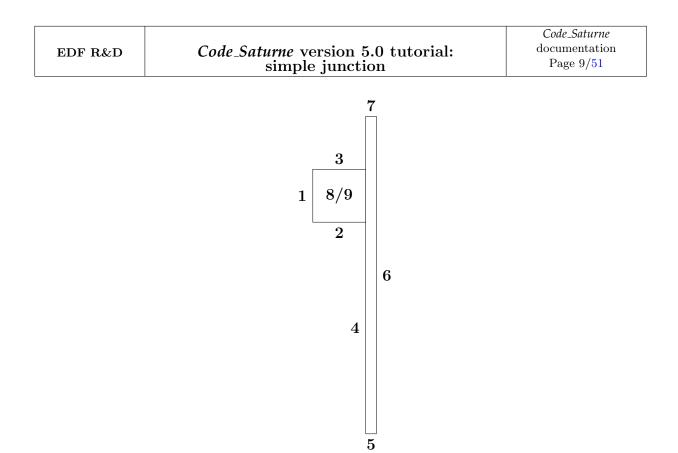


Figure II.3: Colors of the boundary faces

Figure II.3 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).

Colors	Conditions
1	Inlet
5	Outlet
$2\ 3\ 4\ 6\ 7$	Wall
89	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.4 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.

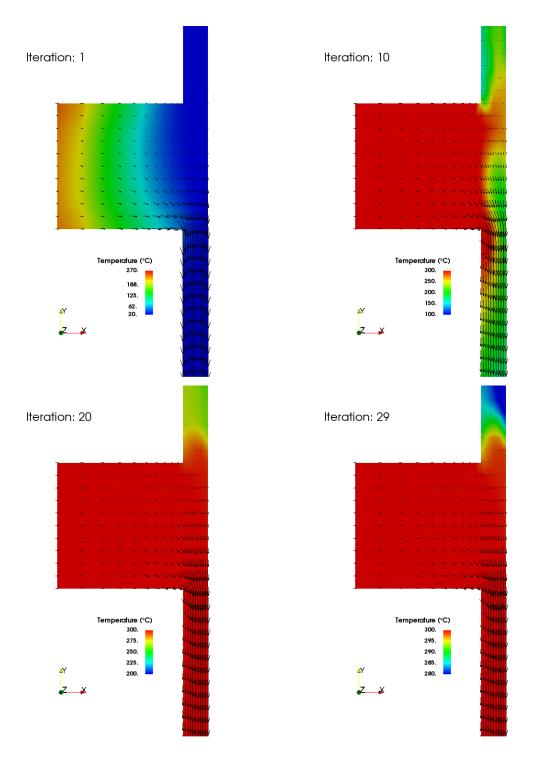


Figure II.4: Water velocity field colored by temperature at different iterations

## Part III

# Step by step solution

## **1** Solution for CASE1

The first thing to do before running *Code\_Saturne* is to prepare the computation directories. In this first example, the study directory  $\boxdot$  simple\_junction will be created, containing a single calculation directory  $\boxdot$  case1. This is done by typing the command:

\$ code\_saturne create -s simple\_junction -c case1

The mesh files should be copied in the directory  $\bigcirc$  MESH/, as follow:

\$ cd simple\_junction/MESH/

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

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

\$ cd simple\_junction/case1/DATA

\$ ./SaturneGUI &

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

<u>F</u> ile <u>E</u> d	dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	
<b></b>	🖥 🖎 🥝 🎼 👘 👘 📰 🖉	S
		ØX
Study:		
Case:		
XML file:		

Welcome to Code\_Saturne GUI.

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

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

File Edit Tools Window Help				<b>S</b>
Study: simple_junction				
Case: case1				
XML file:				
<ul> <li>Identity and paths</li> <li>Calculation environment</li> </ul>	Directory of the case	Code_Saturne/Tut	corials/simple_junction/case1 盾	9
Calculation control      Calculation management	Associated sub-directories of t	the case		
		Data	DATA	
		Results	RESU	
		User subroutines	SRC	
		Running scripts	SCRIPTS	
<	3			

Figure III.2: Identity and paths

Save the case to give a name to the new xml file (such as case1.xml) by opening the File menu and clicking on Save as....

A new window will appear, enter the name of the case in File name then click on Save.

Remember to save the case regularly throughout the preparation of the calculation.

: simple_j		Save File As	
case1 Nom:	case1.xml		
file: Enregistre	r dans le dossier : < Code_Saturne Tuto	orials simple_junction case1 DAT	A Créer un dossier
Raccourc	s Nom		✓ Taille Modifié
Galculation	cher EFERENCE		Hier à 16:41
🕒 Calculation 🛛 🕙 Récem	ment u		
Calculation DATA			
🔂 TM178	BON		
🐻 Bureau			
🖾 Systèr	ne de fi		
🛅 Docum	ents		
🐻 Musiqu	ie in the second se		
🛅 Image:			
🛅 Vidéos			
🔯 Téléch	argeme		
			Code_Saturne GUI files 🛛 🗸
			Annuler Enregistrer

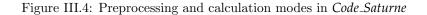
Figure III.3: Saving the xml file

In Code\_Saturne, two modes are available:

- A preprocessing mode, used for mesh preprocessing, and mesh quality checking;
- A calculation mode, used to run computations on the preprocessed mesh.

The mesh selection can still be done in both mode.

Calculati	on Mo	$\mathbf{d}\mathbf{e}$							 		
	File	Edit	Too	ls \	<u>N</u> indo	w	He	lp			
		e		$\bigotimes$	Þ		<b>今</b>	Ċ		Ċ	
Preproces	sing M	ode									



<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
📄 🖆 🖄 🔕 📴 🕱 🖄				S
				0 🛛
Study: simple_junction				
Case: case1				
XML file: /home/TM178B0N/Code_Saturn	e/Tutorials/simple_junction/case1/	DATA/case1.xml		
8				
ldentity and paths	Directory of the case			
🕀 🎦 Calculation environment		Code_Saturne/Tut	torials/simple_junction/case1	<b></b>
🕀 📔 Thermophysical models				
🕀 🎦 Physical properties	Associated sub-directories of t	the case		
🕀 🔛 Volume conditions		Data	DATA	
Boundary conditions				
Dumerical parameters		Results	RESU	
Calculation control     Calculation management		User subroutines	SRC	
				$\overline{}$
		Running scripts	SCRIPTS	

Figure III.5: Calculation mode in Code\_Saturne

The next step is to specify the mesh(es) to be used for the calculation. Click on the **Meshes selection** item under the heading **Calculation environment**. Click on + to add meshes.

The list of meshes in the folder **Meshes** appears in the window **List of meshes**. In this case only the mesh downcomer.des is needed.

<u>File Edit Tools Window H</u> elp				ć
Study: simple_junction				
Case: casel				
XML file: /home/TM178B0N/Code_Saturne	/Tutorials/simple_junction/case	e1/DATA/case1.xml		
o x				
<ul> <li>Identity and paths</li> <li>Calculation environment</li> </ul>		Periodic Boundaries		
<ul> <li>Meshes selection</li> <li>Thermophysical models</li> </ul>	Mesh import <ul> <li>Import meshes</li> </ul>	Use existing mesh inj	out	
Physical properties     Volume conditions	Local mesh directory (	optional)		
Boundary conditions     Dumerical parameters	/MESH			<b>b</b>
Calculation control	List of meshes			
🕀 🛅 Calculation management	File name	Format	Reorient	Path
	downcomer.des	Simail/NOPO		
			÷ -	
	Face joining (optional)			
	Fraction Plane Verbos	ity Visualization		Selection criteria
			4	

Figure III.6: Meshes: list of meshes

The **Periodic Boundaries** is not used in this case. Keep the default values.

The **Calculation features** item under the heading **Thermophysical models** allows to define the type of flow to be simulated. In this case, a steady flow will be chosen.

dy: simple_junction		
e: casel		
	rne/Tutorials/simple_junction/case1/DATA/case1	
Inte: //nome/IM1/6B0N/Code_Sat	rne/Tuconals/simple_junction/case1/DATA/case1	xmt
6		
🗈 Identity and paths	Steady/Unsteady flow algorithm	
Calculation environment	steady flow	×
🖿 Thermophysical models		
📄 Calculation features	Eulerian-Lagrangian multi-phase treatmen	nt
📄 Deformable mesh	off	×
Turbulence models		
📄 Thermal model	Atmospheric flows	
Radiative transfers	off	▼
📄 Conjugate heat transfer		
Species transport	Gas combustion	
Turbomachinery	off	✓
📄 Fans		
Physical properties	Pulverized fuel combustion	
Volume conditions	off	~
Boundary conditions		
🔁 Numerical parameters	Electrical models	
Calculation control	off	~
Calculation management		
	Compressible model	
	off	v
	groundwater flows	
	off	*
	UII	· · · · · · · · · · · · · · · · · · ·

Figure III.7: Flow type

The turbulence model is selected in the following list:

- laminar flow (no model) - mixing length -  $k-\varepsilon$ -  $k-\varepsilon$  Linear Production - Rij- $\varepsilon$  LLR - Rij- $\varepsilon$  SSG - v2f BL-v2/k -  $k-\omega$  SST - Spalart-Allmaras - LES (Smagorinsky) - LES (classical dynamic model)
- LES (WALE)

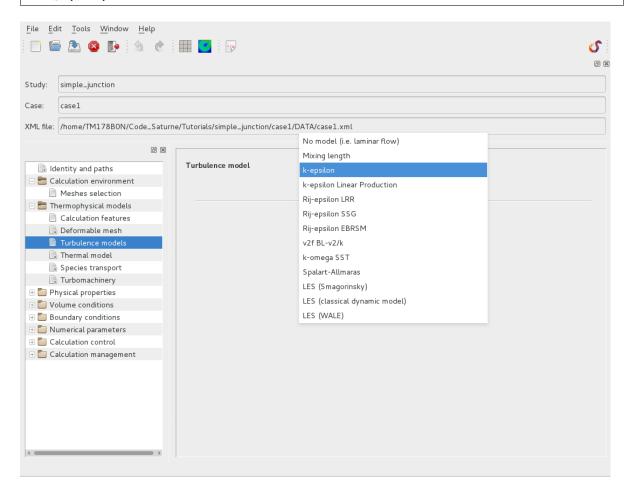


Figure III.8: Turbulence model: list of models

EDF R&D

In this case, the k- $\varepsilon$  model is used.

File Edit Tools Window Help			
📄 🖆 🔕 💽 🖄 🖄			Q
			0
Study: simple_junction			
Case: casel			
XML file: /home/TM178B0N/Code_Saturn	e/Tutorials/simple_junction/cas	e1/DATA/case1.xml	
o x			
ldentity and paths	Turbulence model		
Calculation environment		k-epsilon 🗸	
Meshes selection		k-epsiton	
🖃 🛅 Thermophysical models			
Calculation features		Advanced options 🛛 🛞	
Deformable mesh			
Turbulence models			
🖫 Thermal model			
Species transport			
🗟 Turbomachinery			
🕀 🛅 Physical properties			
🕀 🛅 Volume conditions			
🕀 🛅 Boundary conditions			
🕀 🛅 Numerical parameters			
🕀 🛅 Calculation control			
🕀 🛅 Calculation management			
<			

Figure III.9: 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:

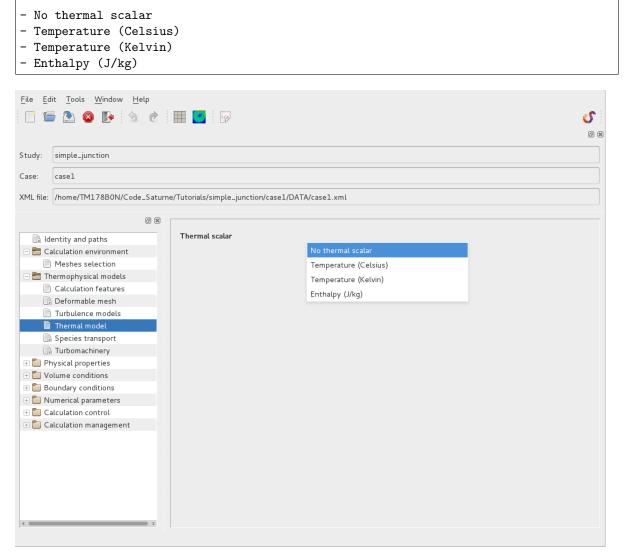


Figure III.10: Thermal scalar conservation: list of models

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

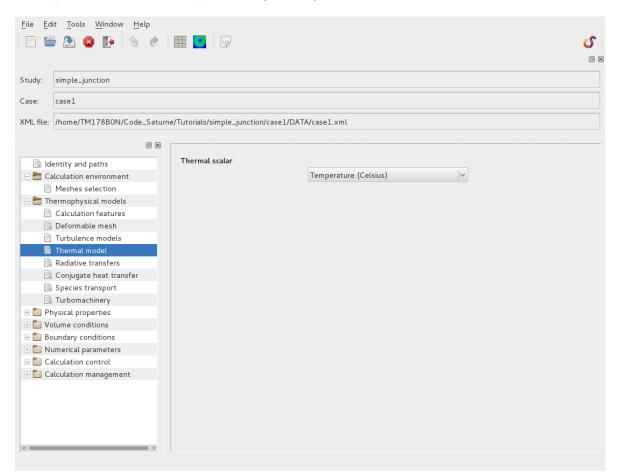


Figure III.11: Thermal scalar conservation: choice of a model

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

#### 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 values te be set are: zero velocity (default), an initial temperature of  $20^{\circ}$ C and a turbulence level based on a reference velocity of  $1 \ (m.s^{-1})$  (default). Specific zones can be defined with different initializations. In this case, only the default all cells is used.

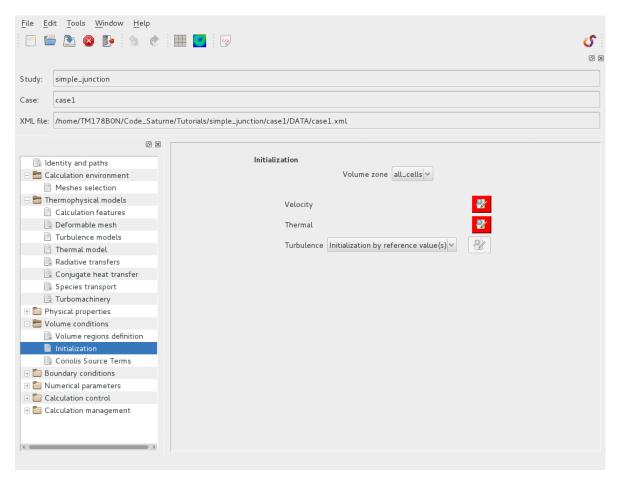


Figure III.12: Initialization of the scalar, velocity and turbulence

• 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

<u>F</u> ile <u>E</u> c	it <u>T</u> ools <u>W</u> indow <u>H</u> elp				
	3 🖎 🔕 📭 🗟 🕐 🗄	📕 🔝 : 🖗			6
					0 8
					D K
Study:	simple_junction				
Case:	case1		Mathematical expre	ession editor	
XML file:	/home/TM178B0N/Code_Saturne,	User expression	Predefined symbols	Examples	
	8 x -	temperature	= 20.;		
_			,		
📑 Id	entity and paths				
🗆 🚍 Ca	alculation environment				
	Meshes selection				
🗆 🚍 Tł	nermophysical models				2/
	Calculation features				
	Deformable mesh				
	Turbulence models				
	Thermal model				
	Radiative transfers				
	Conjugate heat transfer				
R	Species transport				
	Turbomachinery				
🕀 🗂 Pl	nysical properties				
🗆 🚍 Va	olume conditions				
R	Volume regions definition				
	Initialization				
	Coriolis Source Terms			● <u>A</u> nnuler	
🕀 🛅 Bo	oundary conditions				_
🕀 🗂 🛛	umerical parameters				
🕀 🗄 Ca	alculation control				
🗄 🗄 Ca	alculation management				
<	3				

Figure III.13: Initialization of the scalar

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

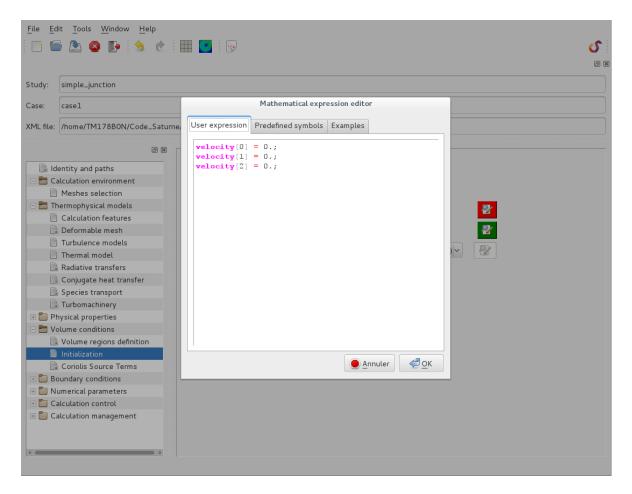


Figure III.14: Initialization of the velocity

Under the heading **Physical properties** in the main list, the **Reference values** item allows to set the reference pressure, the reference velocity and the reference length.

Use the default value of 101 325 (Pa) for the pressure and 1 (m/s) for the velocity.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp		
📄 🖆 🚵 🔕 💽 🖄 👌		S
		Ø×
Study: simple_junction		
Case: casel		
XML file: /home/TM178B0N/Code_Satu	rrne/Tutorials/simple_junction/case1/DATA/case1.xml	
Ø Ø		
ldentity and paths	Reference pressure	
Calculation environment	Reference value for total pressure 101325.0 Pa	
Meshes selection		
Thermophysical models	Reference velocity	
Calculation features	· · · · · · · · · · · · · · · · · · ·	
Deformable mesh	Reference value for velocity 1.0 m/s	
Turbulence models	Defense v laarde	
Thermal model	Reference length	
Radiative transfers	Reference length Automatic 💙 m	
🗒 Conjugate heat transfer	(used for initialization of turbulence)	
Species transport		
Turbomachinery	Reference temperature	
Physical properties	Reference temperature 20 C	
Reference values		
🕞 Fluid properties	(used for properties initialization)	
Gravity		
Volume conditions		
🕀 🛅 Boundary conditions		
🕀 🛅 Numerical parameters		
🕀 🛅 Calculation control		
① Calculation management		
< • • • • • • • • • • • • • • • • • • •		

Figure III.15: 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>
- $C_p = 5\,483 \; J.kg^{-1}.K^{-1}$
- $(\lambda/C_p) = 0.02495 \ W.m^{-1}.K^{-1}$

ly: simple_junction			
simple_junction			
casel			
file: /home/TM178B0N/Code_Sat	urne/Tutorials/simple_junctior	n/case1/DATA/case1.xml	
0	a		
ldentity and paths			
Calculation environment		material user material 🗸	
Meshes selection		method user properties 🗸	
Thermophysical models			
Calculation features			
Deformable mesh		reference user_material	
Turbulence models			
Thermal model	Density		
Radiative transfers		constant 🗸 🖓	
🕞 Conjugate heat transfer		Reference value ρ 725.735 kg/m³	
Species transport		Reference value p 725.735 kg/m	
Turbomachinery	Viscosity		
Physical properties	VISCOSICY		
Reference values		constant 🗸 🖓	
Fluid properties		Reference value µ 8.951e-05 Pa.s	
Gravity			
Volume conditions	Specific heat		
Boundary conditions	Specific field		
Numerical parameters		constant 🗸 🕎	
Calculation control		Reference value Cp 5483.0 J/kg/K	
Calculation management			
	Thermal conductivity		
		constant 🗸 🕎	
		constant 👻 🔟	
		Reference value λ 0.02495 W/m/K	

Figure III.16: Physical properties: fluid properties

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.

As for the pressure interpolation, the interpolation method keeps the standard default value.

<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp			
📄 🗁 📐 🔕 📭 🖄 🤞	E 🗾 🖗		S
			6
Study: simple_junction			
Case: case1			
XML file: /home/TM178B0N/Code_Sat	arne/Tutorials/simple_junctio	on/case1/DATA/case1.xml	
0			
Identity and paths	Gravity		
🗆 🛅 Calculation environment		g <sub>x</sub> 0.0 m/s <sup>2</sup>	
Meshes selection			
🗆 🛅 Thermophysical models		g <sub>y</sub> 0.0 m/s <sup>2</sup>	
Calculation features		g <sub>z</sub> 0.0 m/s <sup>2</sup>	
🔄 Deformable mesh		52 T.	
Turbulence models			
Thermal model			
Radiative transfers			
🕞 Conjugate heat transfer			
Species transport			
🕞 Turbomachinery			
🗆 🛅 Physical properties			
Reference values			
Fluid properties			
📄 Gravity			
🗄 🛅 Volume conditions			
🕀 🛅 Boundary conditions			
🕀 🛅 Numerical parameters			
🕀 🛅 Calculation control			
🕀 🛅 Calculation management			
· · · · · · · · · · · · · · · · · · ·			

Figure III.17: Physical properties: gravity and hydrostatic pressure

Boundary conditions now need to be defined. Go to the **Definition of boundary regions** item under the heading **Boundary conditions**. The following window opens (fig III.20).

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp		
<b>1</b>	i 🖎 🔕 📴 🖄 👌		S
			0 8
Study:	simple_junction		
Case:	casel		
XML file:	/home/TM178B0N/Code_Saturn	ne/Tutorials/simple_junction/case1/DATA/case1.xml	
	0 🕱		
_ ■ Id	entity and paths	Definition of boundary regions	
	alculation environment	Label Zone Nature Selection criteria	
	Meshes selection		
🕀 🛅 Th	ermophysical models		
🕀 🛄 Ph	ysical properties		
🕀 🛅 Va	olume conditions		
🗆 🛅 Bo	oundary conditions		
	Definition of boundary regi		
	Boundary conditions		
	umerical parameters	Add Delete	
	alculation control		
🕀 🛅 Ca	alculation management	Add from Preprocessor listing	
		Import groups and references from Preprocessor listing 🛛 🔚	

Figure III.18: 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
```

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.

🖺 🗁 🙆 📴 🤌 🦿		4			<b>5</b> 0
Study: simple_junction					
Case: case1					
XML file: /home/TM178B0N/Code_Saturn	e/Tutorials/sim	nple_junct	on/case1/DATA/	case1.xml	
0 8					
ldentity and paths	Definition	of bounda	ry regions		
Calculation environment		Zone Nat		Selection criteria	
Meshes selection	BC_1	1 Wa	all		
🕀 🛅 Thermophysical models	bell		e inlet/outlet		
🕀 🛅 Physical properties					
🕀 🛅 Volume conditions		Inl			
🖃 🛅 Boundary conditions			mmetry		
Definition of boundary regi		Ou	tlet		
Boundary conditions					
🕀 🛅 Numerical parameters				Add Delete	
Calculation control		_			
🕀 🛅 Calculation management	Add froi	т Ргергос	essor listing		
			Import gr	oups and references from Preprocessor listing	
<					

Figure III.19: 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.20

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp					
<b></b>	i 🖎 🔕 📭 🕱 👌		4			S
						ð 🕱
Study:	simple_junction					
Study.	Simple_junction					
Case:	casel					
XML file:	/home/TM178B0N/Code_Saturn	e/Tutorials/si	mple_junction/case1	/DATA/case1.xml		
	0 8					
📑 Ide	entity and paths	Definitio	of boundary region	S		
	lculation environment	Label	Zone Nature	Select	tion criteria	
Đ	Meshes selection	Inlet	1 Inlet 1			
🕀 🛅 Th	ermophysical models					
	iysical properties					
	lume conditions					
	oundary conditions					
	Definition of boundary regi					
	Boundary conditions					
	umerical parameters			Add Dele	te	
	lculation control					
+ <b>C</b> a	lculation management		om Preprocessor list	•		<b>F</b> =
			In	port groups and references from Pre	processor listing	
<	3					

Figure III.20: Creation of a boundary region

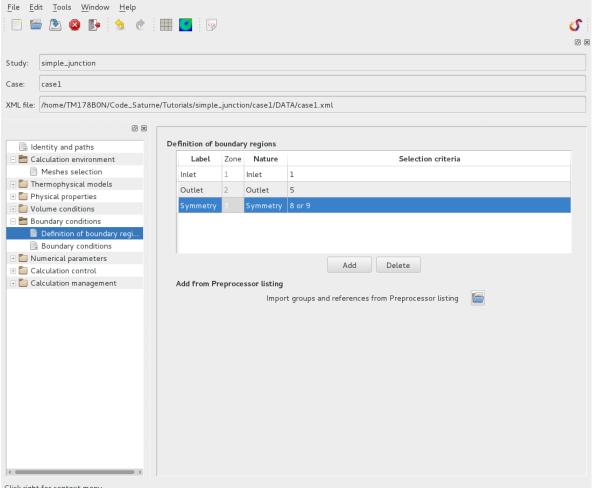
Remember to save the xml file regularly!

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

But it is usually faster to regroup the different colors in one single zone, as shown on figure III.22. In our case, the localization for this zone is the string '8 or 9''.



Click right for context menu

Figure III.22: 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.23.

<u>F</u> ile <u>E</u> di	it <u>T</u> ools <u>W</u> indow <u>H</u> elp									
i 📑 🗲	🖥 🖎 💽 🐏 😒 🖗	🔲 🗾 🖗				S				
						ð	×			
Study:	simple_junction						٦			
Study.	Simple_junction						5			
Case:	casel									
XML file:	/home/TM178B0N/Code_Saturn	e/Tutorials/simple_	juncti	on/case1/DA	TA/casel.xml		٦			
	0 8									
		Definition of b	oundar							
	entity and paths	Definition of boundary regions								
Calculation environment     Meshes selection		Label	Zone		Selection criteria					
	ermophysical models	Inlet	1	Inlet	1					
	ivsical properties	Outlet	2	Outlet	5					
	plume conditions	Symmetry	3	Symmetry	8 or 9					
Boundary conditions		Walls	4	Wall	2 or 3 or 4 or 6 or 7					
	Definition of boundary regi									
	Boundary conditions									
	umerical parameters									
	alculation control				Add Delete					
🕀 🛅 Ca	alculation management	Add from Pro	ергосе	ssor listing						
	-			-	t groups and references from Preprocessor listing 🛛 📔					
			import groups and references from Preprocessor usting							
د 💷	3									

Figure III.23: 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** item to set the inlet boundary conditions for velocity, turbulence and themal scalar.

As shown on figure III.26, outlet and wall boundary zones also appear in the window.

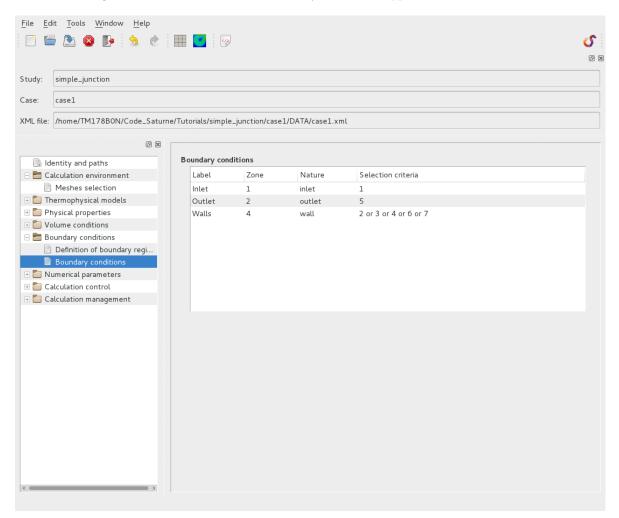


Figure III.24: Dynamic variables boundary conditions

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)

dy: simple_junction					
se: casel					
IL file: /home/TM178B0N/Code_Saturr	ne/Tutorials/simple	_junction/case	1/DATA/case1.xm	l	
ð X					
📑 Identity and paths	Boundary cond	litions			
🛅 Calculation environment	Label	Zone	Nature	Selection criteria	
Meshes selection	Inlet	1	inlet	1	
🛅 Thermophysical models	Outlet	2	outlet	5	
Dhysical properties	Walls	4	wall	2 or 3 or 4 or 6 or 7	
Uolume conditions					
Boundary conditions					
Definition of boundary regi					
Boundary conditions					
🛅 Numerical parameters					
Calculation control	Velocity				
🛅 Calculation management		norm	~	1.0 m/s	
		Direction		specified coordinates	
		Direction		specified coordinates 🗸 🗸	
		× 1.0		x [0.0]	
		X 1.0		Y 0.0 Z 0.0	
	Turbulence				
	Turbutence		Calculatio	n by hydraulic diameter 🔽 🔣	
			Calculatio		
			Hydraulic	diameter 0.5 m	

Figure III.25: Dynamic variables boundary conditions: inlet

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

📄 🗁 🖄 🔕 📭 🖄 🖑		<b>U</b> 0 x
Study: simple_junction		
Case: casel		
XML file: /home/TM178B0N/Code_Saturne	/Tutorials/simple_junction/case1/DATA/case1.xml	
0 8 [		
ldentity and paths		<u>^</u>
Calculation environment		
Meshes selection		
Thermophysical models	Velocity	
Physical properties		
+ 🔁 Volume conditions	norm 💙 1.0 m/s	
🗆 🛅 Boundary conditions	Direction specified coordinates 🗸	
Definition of boundary regi		
Boundary conditions	X 1.0 Y 0.0 Z 0.0	
🕀 🛅 Numerical parameters		
🕀 🛅 Calculation control		
🕀 🛅 Calculation management	Turbulence	
	Calculation by hydraulic diameter 👻 🛛 🖉	
	Hydraulic diameter 0.5 m	
	Thermal	
	Type Prescribed value 💙 🛃	
	temperature Value 300	

Figure III.26: 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.

dy: simple_junction					
e: casel					
. file: /home/TM178B0N/Code_Satu	rne/Tutorials/simp	le junction/case	1/DATA/case1 xr	nl	
	inie, raconaco, oinip				
8					
ldentity and paths	Boundary co	nditions			
Calculation environment	Label	Zone	Nature	Selection criteria	
Meshes selection	Inlet	1	inlet	1	
Thermophysical models	Outlet	2	outlet	5	
Physical properties	Walls	4	wall	2 or 3 or 4 or 6 or 7	
Volume conditions					
Boundary conditions					
Definition of boundary regi					
Boundary conditions					
Numerical parameters	Smooth	or rough wall			
Calculation control	Sinoocii (	or rough wat		smooth wall 🔘 rough wall	
Calculation management			G	sinootn watt 🕤 rough watt	
		a wall			
	Slidin 🗌 Slidin	ig watt			
	Thermal				
				Type Prescribed flux	► 2
		tempe	rature 🗸	Flux 0.0	

Figure III.27: 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**.

Study: simple_junction					
Case: case1					
XML file: /home/TM178B0N/Code_Satur	ne/Tutorials/simple	e_junction/case	1/DATA/case1.xi	ml	
0 8					
ldentity and paths	Boundary con	ditions			
Calculation environment	Label	Zone	Nature	Selection criteria	
Meshes selection	Inlet	1	inlet	1	
🕀 🛅 Thermophysical models	Outlet	2	outlet	5	
🕀 🛅 Physical properties	Walls	4	wall	2 or 3 or 4 or 6 or 7	
🕀 🛅 Volume conditions					
🖃 🛅 Boundary conditions					
Definition of boundary regi Boundary conditions					
Boundary conditions     The second seco					
Calculation control	Smooth o	r rough wall			
🗄 🛅 Calculation management		-		) smooth wall 🔘 rough wall	
	Sliding				
		g wau			
				Prescribed value	
	Thermal			Prescribed value (user law)	
				Type Prescribed flux	2
		temper	ature 🗸	Prescribed flux (user law)	Ē.
				Exchange coefficient	
				Exchange coefficient (user law)	

Figure III.28: Scalars boundaries: walls

The **Global parameters** need then to be specified, under the header **Numerical parameters**. In this case, the **SIMPLE** algorithm must be chosen.

<u>File E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp	·			
: 🖸 🖆 🏝 🔕 ┣ : 😏 🖒	🔛 💟 🤪			S
				0
Study: simple_junction				
Case: case1				
XML file: /home/TM178B0N/Code_Satur	ne/Tutorials/simple_junction/	case1/DATA/case1.xml		
ØX				
ldentity and paths	Global parameters			
Calculation environment		Gradient calculation method:		
Meshes selection		Iterative handling of non-orthogonalitie	es 🗸	
🕀 🛅 Thermophysical models				
🕀 🛅 Physical properties		Pseudo-coupled velocity-pressure s	olver 🗌	
🕀 🛅 Volume conditions		Handling of transposed gradient and	divergence	
🖃 🛅 Boundary conditions		source terms in momentum equation		
Definition of boundary regi				
Boundary conditions		Extrapolation of pressure gradient on domain boundary	Neumann 1st order 🗸	
🖃 🛅 Numerical parameters		on domain boundary		
Global parameters				
Equation parameters		Relaxation of pressure increase	1.0	
Steady flow management				
Calculation control				
🕀 🛅 Calculation management		Improved pressure interpolation in st	tratified flow	
		Velocity-Pressure algorithm	SIMPLE	
		, ,	SIMPLEC	
			PISO	
			1150	

Figure III.29: Steady flow management

After selecting the **Equation parameters** item, the tab **Scheme** allows to change different more advanced numerical parameters.

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

udy: simple_junction						
se: casel						
IL file: /home/TM178B0N/Code_Saturne/	'Tutorials/simple_juncti	on/case1/DATA/cas	sel.xml			
0 x _						
ldentity and paths	Solver Scheme C	lipping				
Calculation environment						
Meshes selection	Name	Scheme	Blending Factor	Slope Test	Flux Reconstruction	RHS Sweep Reconstruction
Thermophysical models			ractor	Test	Reconstruction	2
<ul> <li>Physical properties</li> <li>Volume conditions</li> </ul>	pressure			~	-	
Boundary conditions	velocity	Automatic	1			1
Definition of boundary regi	k	Automatic	0	2	2	1
Boundary conditions	epsilon	Automatic	0	<b>Z</b>		1
Numerical parameters	temperature	Automatic	1	2	<b>a</b>	1
Global parameters						
Equation parameters						
🗟 Steady flow management						
Calculation control						
Calculation management						

## Figure III.30: Numerical parameters

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

idy: simple_junction			
se: casel			
L file: /home/TM178B0N/Code_Saturne	e/Tutorials/simple_junction/case1/DATA/case	e1.xml	
ldentity and paths	Solver Scheme Clipping		
Calculation environment			
Meshes selection	Name	Minimal	Maximal
🛅 Thermophysical models		value	value
Physical properties	temperature	20	400
Volume conditions			
Boundary conditions			
Definition of boundary regi			
Boundary conditions			
Numerical parameters			
📄 Global parameters			
Equation parameters			
Steady flow management			
Calculation control			
Calculation management			

Figure III.31: Clipping

Go to the **Steady flow management** item to specify the number of iterations, **300** in this case. The relaxation coefficient is equal to **0.9** and the **Zero iterations option** will not be activated.

<u>F</u> ile <u>E</u> di	it <u>T</u> ools <u>W</u> indow <u>H</u> elp				
	i 🖎 😣 📭 🖄 👌	III 🔽 🖗			S
					0 8
Study:	simple_junction				
Case:	casel				
XMI file	/home/TM178B0N/Code_Saturn	e/Tutorials/simple_junction/case1			
ATTE ITC.	monie, mier oboin, code_satam	ie, raconaci, simple_janeton, cases			
	0 ×				
📑 Ide	entity and paths	Steady flow algorithm manag	ement		
	lculation environment		Relaxation coefficient	0.9	
	Meshes selection				
🕀 🛅 Th	ermophysical models		Number of iterations (restart included)	300	
	ysical properties		Zero iterations option		
	lume conditions				
	undary conditions				
	Definition of boundary regi				
	Boundary conditions				
	imerical parameters				
	Global parameters				
	Equation parameters				
	Steady flow management lculation control				
	lculation control				
± 🗖 Ca	iculation management				
<					

Figure III.32: Steady flow management

Under the heading **Calculation control**, click on the **Output control** item to 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							<b>S</b> 0 X
Study: simple_junction							
Case: case1							
XML file: /home/TM178B0N/Code_Saturne	/Tutorials/simple_iu	Inction/cas	e1/D/	ATA/case1.xml			
Ø 🗙		<u></u>					
ldentity and paths	Output Control	Writer	Mesh	Particles mesh	Monitoring Points		
Calculation environment	Log frequency			No output			
Thermophysical models				Output listing at	each time step	1	
Physical properties     Olume conditions				Output using at Output every 'n'			
+ D Boundary conditions				Output every n	time steps		
Image: Second and							
Calculation control							
Time averages							
Output control							
Volume solution control							
Surface solution control							
Profiles							
Balance by zone							
🗄 🫅 Calculation management							
< >							

Figure III.33: 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 n set to 1.

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

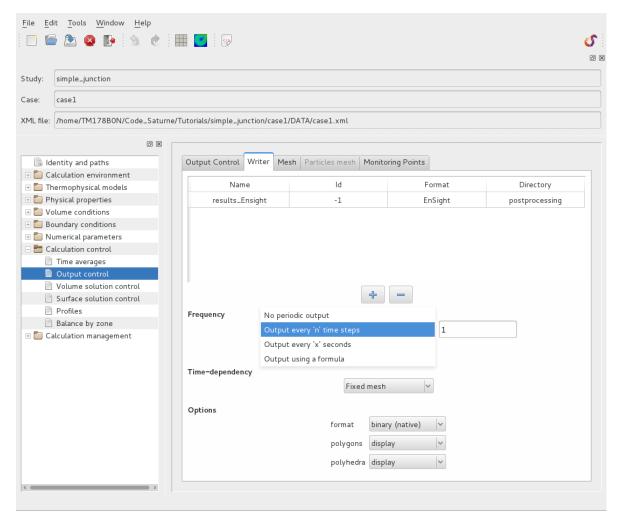


Figure III.34: Output control: post-processing

The other options are kept to their default value.

Image: Solution:       Image: Solution: <td< th=""><th><u>F</u>ile <u>E</u>di</th><th>it <u>T</u>ools <u>W</u>indow <u>H</u>elp</th><th></th><th></th><th></th><th></th><th></th></td<>	<u>F</u> ile <u>E</u> di	it <u>T</u> ools <u>W</u> indow <u>H</u> elp					
Case: case1 XML file: //nome/TM178B0N/Code_Saturme/Tutorials/simple_junction/case1/DATA/case1.xml  Catculation environment Catculation Catculation environment Catculation Catculation environment Catculation Cat		i 🖭 🔕 📭 👌 👌	🗮 🗾 🧐				S
Case: case1 XML file: //nome/TM178B0N/Code_Saturne/Tutorials/simple_junction/case1/DATA/case1.xml  Catculation environment Catculation Catculation environment Catculation Catculation environment Catculation Cat							0(
XML file:       /home/TM17880N/Code_Saturme/Tutorials/simple_junction/case1/DATA/case1.xml         Image: Control Contrel Contrecontrol Control Contrecon Control Control Co	Study:	simple_junction					
Identity and paths         Calculation environment         Thermophysical models         Physical properties         Volume conditions         Budany conditions         Numerical parameters         Calculation control         Profiles         Balance by zone         Calculation management         Image: Calculation management         Image: Calculation management	Case:	case1					
Identity and paths         Calculation environment         Thermophysical models         Physical properties         Volume conditions         Budany conditions         Numerical parameters         Calculation control         Profiles         Balance by zone         Calculation management         Image: Calculation management         Image: Calculation management		[					
Output Control   Profiles   Balance by zone   Calculation management   Frequency Output a everapes     Output a everapes   Output a everapes   Output a everapes   Frequency Output a everapes     Output a everapes   Output a everapes   Output a everapes    Frequency   Output a everapes  Image: Calculation management  Frequency Output a everapes  Output a everapes  Image: Calculation management  Frequency Output a everapes  Image: Calculation management  Output a everapes  Image: Calculation management  Frequency Output a everapes  Image: Calculation management  Image: Calcu	XML file:	/home/IM1/8B0N/Code_Satur	ne/Tutorials/simple_junct	ion/case1/DA1	A/case1.xml		
Calculation environment   Thermophysical models   Physical properties   Volume conditions   Numerical parameters   Calculation control   Numerical parameters   Calculation control   Surface solution control   Surface solution control   Surface solution management   Frequency Output every 'n' time steps     Image: Calculation management   Frequency Output at end of calculation Time-dependency Fixed mesh Options format binary (native) polygons display polyhedra display		0 8					
Image: Thermophysical models   Physical properties   Physical properties   Volume conditions   Boundary conditions   Calculation control   Time averages   Output control   Surface solution control   Surface solution control   Profiles   Balance by zone   Calculation management   Frequency   Output at end of calculation   Time-dependency   Fixed mesh   Options   format binary (native) v polyhedra display v polyhedra display v	📑 Ide	entity and paths	Output Control W	riter Mesh	Particles mesh 🖡	1onitoring Points	
Intermining informations Provide results_Ensight -1 EnSight postprocessing Intermining informations Volume conditions Intermining informations Intermining informations Intermining information Inte							
Volume conditions          Boundary conditions         Numerical parameters         Calculation control         Output control         Volume solution control         Profiles         Balance by zone         Calculation management         Calculation management         Output every 'n' time steps         Image: Calculation management         Image: Calculation management <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>Directory</td>							Directory
Boundary conditions   Boundary conditions   Numerical parameters   Calculation control   Time averages   Output control   Surface solution control   Profiles   Balance by zone   Calculation management   Frequency   Output at end of calculation   Time-dependency Fixed mesh Options format binary (native) polygons display olyhedra display olyhedra			results_Ens	ight	-1	EnSight	postprocessing
Numerical parameters   Calculation control   Time averages   Output control   Surface solution control   Surface solution control   Profiles   Balance by zone   Calculation management   Frequency   Output every 'n' time steps   Image: Calculation management   Frequency   Fixed mesh   Options   format   binary (native)   polydedra   display   polyhedra display polyhedra display polyhedra							
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   polyhedra   display							
Image: 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) \v<							
Output control         Volume solution control         Surface solution control         Profiles         Balance by zone         Calculation management         Calculation management         Image: Calculation management </td <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td>							
Volume solution control Surface solution control Profiles Balance by zone Calculation management Calculation management Image: Calculation management Frequency Output every 'n' time steps Image: Calculation Time-dependency Fixed mesh Options format binary (native) polygons display olyhedra display V							
Surface solution control   Profiles   Balance by zone   Calculation management					_		
Profiles   Balance by zone   Calculation management     Output every 'n' time steps     Ime-dependency     Fixed mesh     Options     format   polygons   display						÷ –	
Balance by zone   Calculation management     Output every 'n' time steps     Image: Output at end of calculation     Time-dependency     Fixed mesh     Options     format   binary (native)   polygons   display			Frequency				
Calculation management  Calculation management  Cutput at end of calculation  Time-dependency  Fixed mesh  Options  format binary (native)  polygons display  polyhedra display		Balance by zone	Frequency	-			
Time-dependency Fixed mesh v Options format binary (native) v polygons display v polyhedra display v	🕀 🛅 Ca	lculation management		Output ever	y 'n' time steps	♥ 1	
Fixed mesh       Options       format     binary (native)       polygons     display       polyhedra     display				🗹 Output a	t end of calculatio	n	
Fixed mesh       Options       format     binary (native)       polygons     display       polyhedra     display			Time-dependency				
Options format binary (native) v polygons display v polyhedra display v			Time-dependency				
format binary (native) v polygons display v polyhedra display v					Fixed	mesn 🗸	
format binary (native) v polygons display v polyhedra display v			Options				
polygons display v polyhedra display v			options		format	binary (native)	
polyhedra display V					Tormat		
					polygons	display 🗸	
					polyhedra	display 🗸	
C							
	۰	3					

Figure III.35: 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.

dy: simple_junction						
e: casel						
file: /home/TM178B0N/Code_Saturne	/Tutorials/simple_junction/	case1/DATA/case1.x	ml			
0 8 (						
ldentity and paths	Solution control					
Calculation environment	Output label	Internal name	Print in listing	Post- processing	Monitoring	
Physical properties	🖃 base			<b>S</b>		
Volume conditions	Pressure	pressure				
Boundary conditions	Velocity	velocity		<b>Z</b>		
🔁 Numerical parameters	total_pressure	total_pressure		<b>S</b>		
🛅 Calculation control	turbulence					
Time averages	k	k	<b>S</b>	<b>S</b>		
Output control	epsilon	epsilon				
Volume solution control	TurbVisc	turbulent_viscosity		$\checkmark$		
Surface solution control	Ithermal					
Profiles	TempC	temperature	<b>S</b>	$\checkmark$		
Balance by zone						
Calculation management						

Figure III.36: Solution control

EDF R&D

The **Start/Restart** item allows to start a new calculation from the results of a former one. It is not the case in the present calculation so nothing has to be modified.

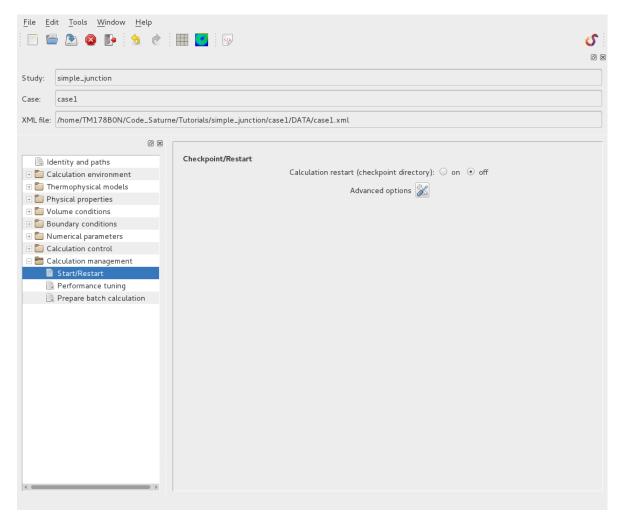


Figure III.37: Start/Restart

The final item, **Prepare batch calculation**, is used to prepare the launch script and, on certain architectures, launch the calculation.

Image: Study:     Study:   Simple_junction   Case:	<u>F</u> ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp				
Study: simple_junction Case: case1 XML file: /home/TM178B0N/Code_Saturme/Tutorials/simple_junction/case1/DATA/case1.xml  Script file Calculation environment Calculation environment Calculation environment Calculation script parameters Calculation script parameters Calculation script parameters Calculation script parameters Calculation management Calculation start Calculation start	📑 🗁 🖄 🔕 🚺	🔜 🗾 🤪			S
Case: case1 XML file: /home/TM178B0N/Code_Saturne/Tutorials/simple_junction/case1/DATA/case1.xml  Script file Calculation environment Calculation environment Calculation script parameters Calculation script parameters Calculation script parameters Calculation script parameters Calculation control Calculation management Calculation management Calculation management Calculation management Calculation management Calculation start Calculation start					ð×
Case: case1 XML file: /home/TM178B0N/Code_Saturne/Tutorials/simple_junction/case1/DATA/case1.xml  Script file Calculation environment Calculation environment Calculation script parameters Calculation script parameters Calculation script parameters Calculation script parameters Calculation control Calculation management Calculation management Calculation management Calculation management Calculation management Calculation start Calculation start	Study: simple_junction				
XML file: /home/TM178B0N/Code_Saturne/Tutorials/simple_junction/case1/DATA/case1.xml     Image: Standard     Image: Standard <td></td> <td></td> <td></td> <td></td> <td>_</td>					_
<ul> <li>Script file</li> <li>Calculation environment</li> <li>Calculation environment</li> <li>Thermophysical models</li> <li>Physical properties</li> <li>Volume conditions</li> <li>Numerical parameters</li> <li>Calculation script parameters</li> <li>Calculation nanagement</li> <li>Start/Restart</li> <li>Performance tuning</li> <li>Performance tuning</li> <li>Advanced options</li> <li>Calculation start</li> </ul>	Case: case1				
Script file   Script file   Calculation environment   Thermophysical models   Physical properties   Volume conditions   Source conditions   Numerical parameters   Calculation control   Calculation management   Start/Restart   Performance tuning   Prepare batch calculation   Calculation start	XML file: /home/TM178B0N/Code_Satur	ne/Tutorials/simple_junction/case1/[	DATA/case1.xml		
Script file   Script file   Calculation environment   Thermophysical models   Physical properties   Volume conditions   Source conditions   Numerical parameters   Calculation control   Calculation management   Start/Restart   Performance tuning   Prepare batch calculation   Calculation start	0 X				
Select the script file   Select the script file Calculation environment Physical properties Calculation script parameters Numerical parameters Calculation control Calculation management Start/Restart Performance tuning Performance tuning Calculation start Calculation start		Script file			
Thermophysical models   Physical properties   Volume conditions   Boundary conditions   Boundary conditions   Numerical parameters   Calculation control   Calculation management   Start/Restart   Performance tuning   Prepare batch calculation   Calculation start			Select the script	file 🛱 runcase	
<ul> <li>Physical properties</li> <li>Physical properties</li> <li>Volume conditions</li> <li>Boundary conditions</li> <li>Numerical parameters</li> <li>Calculation control</li> <li>Calculation management</li> <li>Start/Restart</li> <li>Performance tuning</li> <li>Prepare batch calculation</li> <li>Calculation start</li> </ul>					
Volume conditions   Boundary conditions   Numerical parameters   Calculation control   Calculation management   Start/Restart   Performance tuning   Prepare batch calculation     Calculation start     Calculation start		Calculation script parameters			
<ul> <li>Boundary conditions</li> <li>Numerical parameters</li> <li>Calculation control</li> <li>Calculation management</li> <li>Start/Restart</li> <li>Performance tuning</li> <li>Advanced options</li> </ul>			Run type	Standard	
Image: Numerical parameters         Image: Calculation control         Image: Calculation management         Image: Calculation         Image: Calculation         Image: Calculation management         Image: Calculation         Image: Calculation management         Imagement         Imagement	🕀 🛅 Boundary conditions				
Calculation management Calculation management Start/Restart Prepare batch calculation Calculation start	🕀 🛅 Numerical parameters		Run id		
Calculation management Calculation management Start/Restart Performance tuning Advanced options Calculation Calculation start	🕀 🛅 Calculation control		Number of processes		
Start/Restart     Ihreads per task     0       Performance tuning     Advanced options       Calculation start			ridiniber of processes	<b>.</b>	
Prepare batch calculation     Advanced options       Calculation start			Threads per task		
Calculation start			Advanced entions	2	
	Prepare batch calculation		Advanced options		
		Calculation start			
			Start	calculation	
			Start	calculation	
x (	< ( ) ) )				

Figure III.38: Prepare batch calculation: computer selection

The default launch script is named runcase and is located in the  $\boxdot$  SCRIPTS/ directory. It is already loaded.

If a different launch script needed to be selected, this can be done by clicking on the icon next to **Select the script file**. Select the desired script and click on **Open**.

Remember to save the xml file before opening the launch script.

Study: simple_jur		2					: <b>°</b> 20
Case: case1			Select the batch scr	pt			
XML file: /home/T	TM	1178BON Code_Saturne	Tutorials simple_junction	case1 SCRIPTS			
	Raccourcis	Nom			Ƴ Taille	Modifié	
ldentity and	🔍 Rechercher	🙆 runcase			185 octets	17:42	
<ul> <li>① Calculation</li> <li>① Thermophy</li> </ul>	🕙 Récemmen						
Thermophy     Thermophy     Thermophy     Thermophy	🛅 DATA						
🕀 🛅 Volume cor	🗟 TM178BON						
🕀 🛅 Boundary c 🕀 🛅 Numerical p	🐻 Bureau						
Calculation	Système d						
🖃 🛅 Calculation	Documents						
🔒 Perform	词 Musique 🛅 Images						
Prepare	🔤 Vidéos						
	🔄 Télécharge						
					All	l Files 🗡	
					Annuler	Ouvrir	

Figure III.39: Prepare batch calculation: batch script file selection

When the script is selected, new options will appear. 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 Start calculation. The results will be copied in the  $\Box$  RESU/ directory.

<u>F</u> ile <u>E</u> d	it <u>T</u> ools <u>W</u> indow <u>H</u> elp					
	i 🖎 🔕 📭 🖄 👌				S	
					ð	×
Study:	simple_junction					1
brady.	Study. Simple-Junction					1
Case: case1						
XML file: /home/TM178B0N/Code_Saturne/Tutorials/simple_junction/case1/DATA/case1.xml						1
	0 🕷					
ldentity and paths		Script file				
🕀 🛅 Calculation environment			Select the script	file 📁 runcase		
	ermophysical models					
Physical properties		Calculation script parameters				
	lume conditions		Run type	Standard 🗸		
Boundary conditions			Run id			
🕀 🛅 Numerical parameters			- Contribution			
Calculation control			Number of processes	1		
🖃 🛅 Calculation management						
Start/Restart			Threads per task	0		
Performance tuning Prepare batch calculation			Advanced options	*		
		Calculation start				
			Start	calculation		
<						

Figure III.40: Prepare batch calculation: execution

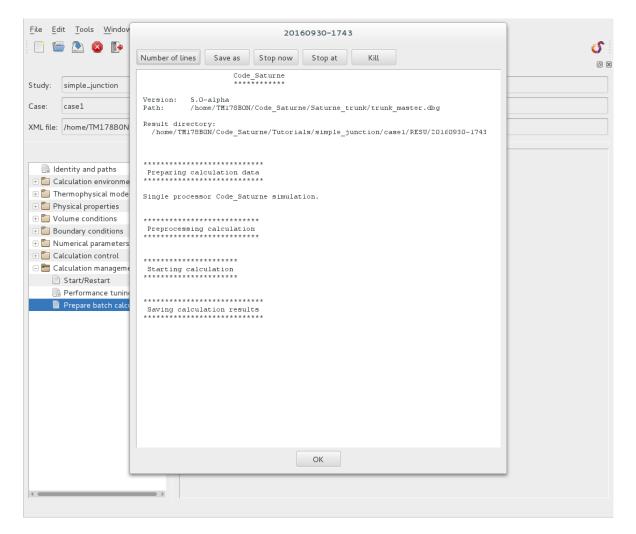


Figure III.41: Run