



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

6, quai Watier F-78401 Chatou Cedex

MARCH 2024

code_saturne documentation

code_saturne version 8.0 tutorial: T-junction Flow

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



		code_saturne
EDF R&D	code_saturne version 8.0 tutorial: T-junction Flow	documentation Page $1/29$

TABLE OF CONTENTS

	I Introduction	3
1	Introduction	4
1.1	TUTORIAL COMPONENTS	4
1.2	TUTORIAL STRUCTURE	4

	II Setting up	5
1	Creating the geometry and the mesh	6
1.1	CREATION OF THE GEOMETRY	6
1.2	GENERATION OF THE MESH	12
2	Setting up	15
2.1	Mesh pre-processing	15
2.2	Computation setup	16
3	Advanced setting up	25
3.1	Boundary layer insertion	25
3.2	Restarting from the previous computation	27
3.3	Local mesh refinement	28
3.4	RESTARTING FROM THE PREVIOUS COMPUTATION (AGAIN)	29

Part I

Introduction

1 Introduction

1.1 Tutorial Components

This tutorial makes use of:

- The SALOME [?] platform for geometry generation, meshing, and post-processing
- code_saturne [?], [?] for CFD calculations

1.2 Tutorial Structure

This tutorial is made of two complementary sections:

- Section 1 describes all the procedures required to get create the geometry and the mesh for the present case using SALOME
- Section 2 illustrates setting up, running, and analysing a T-junction CFD simulation entirely with code_saturne
- Section 3 illustrates the setting up of an advanced T-junction CFD simulation, restarting from the previous computation with additional advanced code_saturne user sources in C.

Part II

Setting up

1 Creating the geometry and the mesh

In this section, we will see how to generate a geometry and a mesh with the SALOME platform.

We invite you to save regularly your work.

1.1 Creation of the geometry

First of all, you need to activate SHAPER by clicking on the "SHAPER" icon.

A part named "Part_1" was automatically generated. We can now create a sketch with a size view of 0.3 on the plane "XOZ". You can select the plane in the viewer or in the "Constructions" folder in the "Object browser".



Figure II.1: Creation of a sketch in SHAPER



Figure II.2: Select sketch plane in SHAPER

In this sketch, we create a circle and set a constraint on the radius of r1 = 0.1m. Please note that drawings appear in green when the sketch is fully fixed (zero degree of freedom). Once it is done, we can validate the sketch.



Figure II.3: First sketch

With this sketch, we are going to create a cylinder with the extrusion function. Please note that you need to select "Sketch_1" in "Construction" folder to do so.



Figure II.4: Selection of extrusion function

In order to generate the first cylinder, use a length of l1 = 0.25m in both directions along the Y axis.



τn

Object browser 🌯 Part set \$

0x

Figure II.5: Extrusion of cylinder 1

Now, we create a second sketch in the plane XOY. It consists of a circle with a radius r2 = 0.025m and an offset of l2 = 0.05m from the Y axis. They are several ways to apply these constraints, for instance one can use "coincident" constraint to fix the center of the circle on the a X axis and then use "distance" constraint to set the offset.



Figure II.6: Second sketch

In order to create the second cylinder, extrude "Sketch_2" along the Z axis with a length l1.





Figure II.7: Extrusion of cylinder 2

The final geometry is generated as the fusion of both extrusions. Please note that you need to select them in "Results" folder and then click on "Fuse" icon.



Figure II.8: Selection of fuse function

Now, we can create groups of faces that will be used to defined boundary conditions in the data settings of the code_saturne computation.

To do so, we select "Fuse_1_1" in the "Results" folder, click on "Feature" tab and select "Group".



Figure II.9: Selection of group function

The four groups of type "Face" should be created with the following instructions:

Group name	Position
------------	----------

-	
inlet1	disk of cylinder 1 along plane XOZ with minimum Y coordinates
inlet2	disk of cylinder 2
outlet	disk of cylinder 1 along plane XOZ with maximum Y coordinates
walls	remaining boundary faces

Table II.1: Group definition



Figure II.10: Group of face inlet1



Figure II.11: Group of face inlet2



Figure II.12: Group of face outlet



Figure II.13: Group of face walls

1.2 Generation of the mesh

We are going to generate a mesh based on the geometry we just created. We first have to activate SMESH by clicking on the "SMESH" icon.

After that, we select "Fuse_1_1" in "ShaperResults" folder, click on "Mesh" tab and select "Create Mesh".



Figure II.14: Selection of function create mesh

In the new window, we can select the type of element and the corresponding algorithm that will be used to generate the mesh. We invite you to use "MG-Tetra" and "MG-CADSurf" as 3D and 2D algorithms with the default configuration.

	Create mesh	×					
Name	Mesh_1						
Geometry 🦿	Fuse_1_1						
Mesh type	Mesh type Any						
✔ Create all Gro	ups on Geometry						
3D 2D 1	D OD						
Algorithm	MG-CADSurf 🗸						
Hypothesis	<default></default>						
Add. Hypothes	is <none> 🔪 🌌</none>	2					
		12					
	Assign a set of automatic hypotheses						
Apply and Close	Apply Close Help						

Figure II.15: Selection of meshing algorithms

It is now possible to generate the mesh with a right click on "Mesh_1" followed by the selection of "Compute".



Figure II.16: Selection compute mesh function

At the end of the computation of the mesh, a new window will appear. It helps to verify the type and the number of elements.

	Me	esh computation	succeed	
compute mesh				
•				
lame				
Mesh_1				
1esh Infos				
	Total	Linear	Quadratic	Bi-Quadratic
Nodes:	2780			
0D Elements :	0			
Balls :	0			
Edges :	140	140	0	
Faces :	2346	2346	0	0
Triangles :	2346	2346	0	0
Quadrangles :	0	0	0	0
Polygons :	0	0	0	
Volumes :	12875	12875	0	0
Tetrahedrons :	12875	12875	0	
Hexahedrons :	0	0	0	0
Pyramids :	0	0	0	
Prisms :	0	0	0	0
Hexagonal prisms :	0			
Polyhedrons :	0			
				Class

Figure II.17: Mesh information

It is now necessary to save the mesh to be able to use it in the code_saturne data settings. Use export function to do so. We invite you to select the med format as it is the native format of the SALOME platform and can be read by code_saturne.



Figure II.18: Save mesh in med format

EDF R&D

2 Setting up

2.1 Mesh pre-processing

It is always advise not to keep tetrahedral cells at the inlet/outlet of a CFD mesh. The mesh made in the previous section is fully tetrahedral. To add some layers of prisms at the two inlets and outlets, we will perform an extrusion on those faces directly in code_saturne. Also, the outlet might be located a little close to the T junction, so extruding the outlet will also allow to make the pipe after the T longer.

First, select the mesh from previous section in the **Mesh** item and make sure to change the "Execution mode" from "Standard Computation" to "Mesh preprocessing Only". This execution mode, will only perform the preprocessing operations on the mesh, but do not require any CFD setup for now.

	c	ASE1 : setup.xm	l - code_sat	urne		×
Eile Edit Tools Window Help						
📄 🖹 鸟 👌 国 🖗 🧯	🔅 📘 😫 🍐					
Calculation environment Calculation environment Calculation environment Calculation environment Calculations	Mesh input Import meshes U Local mesh directory (o/MESH Ust of meshes File name mesh_tjunction.med Execution mode	se existing mesh i ptional) Format MED	Numbers	Reorient	tesian mesh Path	
	Mesh preprocessing on	v	*			
	 Use unmodified check Save mesh if modified 	point mesh in cas by preprocessing	se of restart			

Figure II.19: Select lesh and switch the "Execution mode" to "Mesh preprocessing Only".

Go to the **Preprocessing** sub item and click on the **Other** tab. Under the mesh extrusion menu, add the following extrusion :

n layers	thickness	expansion factor	selector
4	0.05	1	inlet1
8	0.025	1	inlet2
40	1.0	1.03	outlet

Table II.2: Extrusion parameters

		$code_saturne$
EDF R&D	code_saturne version 8.0 tutorial: T-junction Flow	documentation Page $16/29$

CASE1 : setup.xml - code_saturne					
ile <u>E</u> dit <u>T</u> ools <u>W</u> indow <u>H</u> elp					
📄 🖹 🔦 🕐 🔳 🖗 🧧 🖉	2 😰 🚺	•			
© ∞ Galculation environment ♥ ↑	Face joining	g Perio	dic Bounda	ries Other	
Preprocessing	Interior to b	oundary	faces (boun	dary insertion)	
Volume zones Boundary conditions Conditions Postprocesing Performance settings	zone id selector				
	Mesh extru	sion			
	zone id	n layers	thickness	expansion facto	tor selector
	0				inlet1
	1	8	0,025	1	inlet2
	2	40	1	1,03	oulet
					Add Delete

Figure II.20: Define the three extrusion.

Run the case. You can then visualize the resulting mesh by opening Paraview. You should ended with the following mesh :



Figure II.21: The final mesh with extrusions.

2.2 Computation setup

First, in the **Mesh** item switch back the "Execution mode" to "Standard Computation".

Notebook variables

It is possible in code_saturne to create variables that can be used throuhgout the whole setup. We will here create a variable to define the inlet velocity in the main and side branch of the T junction. In the **Notebook** item located under the **Calculation environment** menu simply add the variables :

variable name	value

main_inlet_velocity	0.5
<pre>side_inlet_velocity</pre>	2.0

Table II.3: Notebook variables

		CASE1 : se	tup.xml - code_saturne			×
<u>File Edit T</u> ools <u>W</u> indow <u>H</u> elp						
📄 🖻 🕭 🐀 🕐 🔳 🖗 🧯 💋	o 🛛 🖸 🙆					
ØX						
 Calculation environment 	Notebook variables					
Notebook	variable name	value	OpenTurns Variable	Editable	Description	
▼ (Mesh	main inlet velocity	0.5	No	No	beschption	
Preprocessing	side_inlet_velocity	2.0	No	No		
Volume zones						
Boundary zones						
▼ dit Calculation features						
Turbulence models						
Thermal model						
Body forces						
Species transport						
 Volume conditions 						
all_cells						
 Boundary conditions 						
📑 main_inlet						
🐘 side_inlet						
outlet						
a wall						
Coupling parameters			4	-	import	
$\overline{}$ Δt Time settings						

Figure II.22: Notebook variables definition

Boundary zones

In order to define the boundary zones, go the **Boundary zones** item located under the **Mesh** menu. From here you can manually add the different boundary zones, but since we already performed a preprocessing of our mesh, we can simply click on **Import groups and references from preprocessor log**. Here, you should be able to select the file **preprocessort.log**, located in the results folder from the previous section.

		CASE1 : setup.xr	nl - code_saturne		×
<u>File Edit Tools Window H</u> elp					
🖺 🗎 🧆 🤌 🥐 🔳 🖗 🧯	6	E 🔅			
Calculation environment Calculation environment Notebook Calculation eavies Calculation features	Boundary	regions definition Zone Select a pre	: processor log	Selection criteria	
 Wolume conditions Au Time settings Au Time rectal parameters Postprocessing Additional user arrays Time averages Volume solution control Surface solution control Profiles Balance by zone Performance settings 	Look in:	/home/F75673/CodeSU/20 Name checkpoint postprocessing performance.log preprocessoniog run_solver.log setup.log	2220301-1415_1 ♥ ♥ Size Type Folder 2 KB log File \$ KB log File 7 KB log File 17 bytes log File	> > •	9
		[0	
	Files of two	Proprocessoring			
6		rreprocessor log (*log)		- JI gancer	

Figure II.23: Boundary zones definition from existing preprocessing file

Four different boundary zones should appear. Simply change the label so that they could be easily identified.

Label	Zone	e Selection criteria	
main_inlet		inlet1	
side_inlet		inlet2	
outlet		oulet	
wall		walls	

Figure II.24: Final boundary zones with explicit labels

Calculation features

Let's now go in the **Calculation features** top menu.

- In the **Turbulence models** item, select the $k \omega$ SST model. Let the velocity scale to 1, it is only be used for initilization purpose.
- In the Thermal model item, switch from "No thermal scalar" to "Temperature (Celsius)".

Volume conditions

Let's now move in the **Volume conditions** top menu. By default, there is only a default volume zone containing all the domain. If needed, some additional volume zones can be defined under the **Mesh/Volume zones** item.

For the default all_cells volume zones, activate the "Initialization".

		CAS	5E1 : setup.xml - code	e_satu	rne				
<u>File Edit Tools Window H</u> elp									
📄 🖆 🥱 👌 🙋 🖬 🐼 👩 💋	🖢 🖻 🚺	Ø							
e x									
 Calculation environment 	Volume zone t	/pes							
Notebook									
🔻 🕅 Mesh	Zone label	Initialization	Physical properties	Solid	Porosity	Head losses	ferm	Thermal source term	
Preprocessing	all cells	7	7						
Volume zones									-
Boundary zones									
▼ dw Calculation features									
Turbulence models									
Thermal model									
Body forces									
Species transport									
💌 📔 Volume conditions									
📑 all_cells									

Figure II.25: Volume zone types

Now, under the tab **all_cells**, set the physical properties as follows (water at $20^{\circ} C$) :

property	value
Density	$ ho = 1000 \ kg/m^3$
Viscosity	$\mu = 0.001 \ Pa.s$
Specific heat	$C_p = 4190 \ J/kg/K$
Thermal conductivity	$\lambda = 0.604 \ W/m/K$

Table II.4: Property values

In the **Initialization** tab of the same page, set the velocity to 0. For the thermal scalar, switch from "Automatic" to "Initilization by formula". Click then on the red button and set the temperature to $20^{\circ} C$.

Physical properties initialization
General parameters
Mathematical expression editor ×
User expression Predefined symbols Examples
temperature = 20.;

Figure II.26: Initialization of the temperature

Boundary conditions

Move to the **Boundary condition** menu. Set the boundary conditions according to the following table :

Label	\mathbf{type}
$main_inlet$	Inlet
$\mathtt{side_inlet}$	Inlet
outlet	Outlet
wall	Wall

Table II.5: Type of boundary conditions

Label	Nature
main_inlet	Inlet
side_inlet	Inlet
outlet	Outlet
wall	Wall

Figure II.27: Type of boundary conditions in the GUI

Now there should be one item in the left menu for each of your boundary zone.

• For the **main inlet** item, we want to prescribe a bulk velocity equal to the variable "main_inlet_velocity" that was previsouly added in the "Notebook".

In the **Velocity** part, switch from "norm" to "norm (user law)". Click on the red button on the right, and simply indicate that u_norm = main_inlet_velocity.

	✓ Map	bed Inlet						
	Trans	lation vector	X 0.0		Y 0.05		Z 0.0	
			Mathematical	expression editor		×		
User e	expression	Predefined sy	mbols Exam	nples				1 272
u_nor	m = main_ir	let_velocity;						
					<u>C</u> ancel	<u>o</u> ĸ		

Figure II.28: Defining the norm of the velocity

Set the thermal prescribed value to 20° C.

CASE1 : setup.xml - code_saturne ×							
<u>File Edit Tools Window H</u> elp							
📄 🚔 👲 👌 🖉 📓 🖗 📔 💋	o 2 🖪 🐯						
	main_inlet [Inlet] Convective Inlet Mapped Inlet Velocity Direction Turbulence Calculation by hydraulic diameter Hydraulic diameter 0.2 m						
	Thermal						
	Type Prescribed value • Value 20.0	Ţ					
•							

Figure II.29: Boundary conditions for the main inlet

• For the side inlet item, we want to prescribe a bulk velocity equal to the variable "side_inlet_velocity". In the Velocity part, simply switch from "norm" to "norm (user law)". Click on the red button on the right, and simply indicate that u_norm = side_inlet_velocity.



Figure II.30: Defining the norm of the velocity

Set the hydraulic diameter to 0.05.

Set the thermal prescribed value to 30° C.

EDI	F R&D	code_saturne version 8.0 tutorial: T-junction Flow	code_saturne documentation Page 21/29
		CASE1 : setup.xml - code_saturne	×

<u>File Edit T</u> ools <u>W</u> indow <u>H</u> elp	
📄 🚔 👌 👌 🔳 🖗 🧯	e e 🧧 🖬 🔅
68	
Calculation environment Notebook	side_inlet (inlet)
Mesh Preprocessing Volume zones	Convective Inlet
Boundary zones Calculation features Turbulence models Thermal model	Aapped Inlet
Body forces Species transport Volume conditions	Velocity norm (user law) m/s
Boundary conditions main_inlet side inlet	Direction normal direction to the inlet 👻
outlet	Turbulence
wall Coupling parameters ∴ Av Unerrical parameters Ar Numerical parameters Postprocessing Additional user arrays Time averages Volume solution control Surface solution control Profiles	Calculation by hydraulic diameter 🔹 💱
Balance by zone Performance settings	Thermal Type Prescribed value Value 30.0
۲. (

Figure II.31: Boundary conditions for the side inlet

• Leave the outlet and the wall boundary zones as they are defined by default.

Time settings

Move to the Time settings menu. Set the time step option to "Steady", the reference time step to 0.1 s and the number of time steps to 600.

	CASE1	: setup.xml - code_saturne	×
<u>File Edit Tools Window Help</u>			
📄 🖻 🖄 🚖 🗖 🐼 🌔	a 😰 🖪 🌣		
ØX			
 Calculation environment 			
Notebook			
🔻 🕅 Mesh	Time step option Steady (loca	ai time step) 🔹	
Preprocessing	Velocity-Pressure algorithm SI	IMPLEC +	
Volume zones			
Boundary zones			
▼ av Calculation features			
Turbulence models	Reference time step 0.	.1 s	
Thermal model	Movimal CEL pumber		
Body forces	Maximal CPE number 1.		
Species transport	Maximal Fourier number 10	0.0	
 Volume conditions 	Marine al Marine and an Anatoma		
📄 all_cells	Minimal time step factor 0.	.1	
 Boundary conditions 	Maximal time step factor	000.0	
📑 main_inlet			
📑 side_inlet	Time step maximal variation 0.	.1	
📄 outlet			
🐘 wall			
Coupling parameters	Champing aritarian blumbar of	time stone 600	
All Time settings	stopping criterion Number of	time steps + buu	
Start/Restart			



Numerical parameters

Move to the **Numerical parameters** menu.

For stability issues, it can be interesting to switch the gradient reconstruction mode to "Least Squares" with "Full (all vertex adjacent)" extended cell neighbors. This option is more

robust on tetrahedral cells, but is more expensive to compute.

radient reconstruction type	Least squares	•
xtended cell neighbors	Full (all vertex adjacent)	*

Figure II.33: Numerical parameters for gradient computations

Move to the **Clipping** tab in the **Equation parameters** item. Set the minimum for the temperature to 20 and the maximum to 30. This will enforce the min/max principle to be respected.

	CASE1 : set	tup.xml - code_saturne	
<u>File Edit T</u> ools <u>W</u> indow <u>H</u> elp			
📄 🖹 👌 👌 国 🖗	6262 🕻		
	0 8		
 Calculation environment 	Solver Scheme Clipping		
Notebook			
🔻 🕅 Mesh	Name	Minimal	Maximal
Preprocessing	Nume	value	value
Volume zones	temperature	20	30
Boundary zones			
▼			
Turbulence models			
Thermal model			
Body forces			
Species transport			
 Volume conditions 			
🕞 all_cells			
🔻 🕞 Boundary conditions			
🕞 main_inlet			
📄 side_inlet			
📄 outlet			
🕞 wall			
Coupling parameters			
▼ ∆t Time settings			
📄 Start/Restart			
▼ Δx Numerical parameters			
Equation parameters			

Figure II.34: Numerical parameters the temperature clippings

Postprocessing

Move to the **Postprocessing** menu. Define monitoring point according to the following locations :

n	х	У	\mathbf{Z}
1	0	0	0
2	0	0.1	0
3	0	0.4	0
4	0	1	0
5	0.04	0.2	0.06

Table II.6: Monitoring points

Running the computation

Run the computation on 4/5/6 processes. It should take around 15/20 minutes.

• First check the convergence of the case by plotting the monitoring points in Paraview (or any other plotting tool). The computation should be converged.



Figure II.35: Temporal evolution of monitoring points (left y-velocity, right temperature)

• You can now visualise your results in Paraview. Check the streamlines :)



Figure II.36: Results



Figure II.38: Results

3 Advanced setting up

3.1 Boundary layer insertion

It is always recommended to have a layer of prisms close to walls so that the boundary layer could be more precisely reproduced.



Figure II.39: Schematic view of boundary layer insertion

code_saturne offers the possibility to insert boundary layer in your mesh. This functionnality is not available through the GUI (for now). It is necessary to add a **user source file** in the SRC folder of your case.

To do so :

- Click on the icon "Manage the SRC foler"
- Double click on the "REFERENCE" file and right click on the file cs_user_mesh.c file.
- Click on "Copy to SRC file"

You should now have the following (empty) file in the SRC folder of your case :

	CASE1 : setup.xml - code_saturne		
<u>File Edit Tools Window H</u> elp			
	Editor: cs_user_mesh.c	×	
File • Calculat • Wesh • Perform • Body • Wolume • Wolume • Body • Perform • Perform	 Perinition of the calculation mesh. Definition of the calculation mesh. Reah-related user functions (called in this order); 1) Manage the exchange of data between Code_saturne and the pre-processor 2) Define thin walls: 3) Define thin walls: 4) Define thin walls: 5) Modify the geometry and mesh. 4) Bestime version 7.2-alpha */ ************************************		
4			

Figure II.40: Open the reference cs_user_mesh.c file

You can then open the cs_user_mesh-modify.c file in the EXAMPLES file and look for the snippet of code that will insert the boundary layer. We propose here the following code to add in the the cs_user_mesh_modify subroutine :

```
int n_zones = 1;
       const char *sel_criteria [] = {"walls"};
2
       const int zone_layers [] = \{4\};
3
       const double zone_thickness [] = \{-1\};
4
       const float zone_expansion [] = \{0.8\};
5
6
       cs_mesh_extrude_face_info_t *efi = cs_mesh_extrude_face_info_create(mesh);
7
8
       cs_lnum_t n_faces;
9
       cs_lnum_t *face_list;
10
      BFT_MALLOC(face_list, mesh->n_b_faces, cs_lnum_t);
12
13
       for (int z_id = 0; z_id < n_zones; z_id++) {
14
15
16
         cs_selector_get_b_face_list(sel_criteria[z_id], &n_faces, face_list);
17
18
         cs_mesh_extrude_set_info_by_zone ( efi ,
                                             zone_layers [z_id],
19
                                             zone_thickness [z_id],
20
                                             zone_expansion [z_id],
21
22
                                             n_faces ,
23
                                             face_list);
24
25
       }
26
      BFT_FREE(face_list);
27
28
       /* Determine vertex values for extrusion */
29
30
       cs_mesh_extrude_vectors_t *e = cs_mesh_extrude_vectors_create(efi);
31
       /* Insert boundary layer */
33
34
       cs_mesh_extrude_face_info_destroy(& efi);
35
36
37
       cs_mesh_boundary_layer_insert (mesh, e, 0.2, false, 0, NULL);
38
39
       cs_mesh_extrude_vectors_destroy(&e);
```

The initial mesh will have 4 layers inserted close to the "walls" boundary faces. We let code_saturne compute its thickness by setting the thickness to -1. If you want to visualize your mesh, you can change the execution mode to "Mesh preprocessing only" and run your case.



Figure II.41: Mesh with boundary layers

3.2 Restarting from the previous computation

Move to the Time settings menu and switch the stopping criterion from "Number of time steps" to "Additional time steps" and set it to 200.

Time step option Steady (lo	cal time step)
Velocity-Pressure algorithm	SIMPLEC +
Reference time step	0.1 s
Maximal CFL number	1.0
Maximal Fourier number	10.0
Minimal time step factor	0.1
Maximal time step factor	1000.0
Time step maximal variation	0.1
Stopping criterion Additiona	l time steps 🔹 200

Figure II.42: Set 200 additional time steps

Under the **Time settings** menu, you will find the **Start/Restart** item. Switch the restart to **On** and in the "Checkpoint directory" item, select the **checkpoint** folder from your previous computation.

Click on "Different mesh" and select the file mesh_input.csm which is located inside the checkpoint folder from your previous computation.

Note that we make here a restart from a mesh which is different from the current mesh (since we added boundary layer). code_saturne offers this possibility.

Checknoint directory	RESU/20220301-1901/checkpoint	
encerpoint directory		
Different mesh	RESU/20220301-1901/checkpoint/mesh_input.csm	
Different mesh	RESU/20220301-1901/checkpoint/mesh_input.csm dvnamic (velocity, pressure and turbulence)	

Figure II.43: Select the checkpoint folder from your previous computation

Running the computation

Run the computation on 4 to 6 processes.





Figure II.45: Results

3.3 Local mesh refinement

We now want to refine the mesh around the T junction. This can be done through the use of user sources. You can then open the cs_user_mesh-modify.c file in the EXAMPLES file and look for the snippet of code that will locally refine the mesh. We copy here the example code to add in the the cs_user_mesh_modify subroutine **AFTER** the boundary layer insertion from the previous subsection:

```
.... HERE IS THE BOUNDARY LAYER INSERTION
    //
    // FROM PREVIOUS SUBSECTION
4
5
       \dot{//} Here we specify the box in which we want to perform a refinement
6
7
       const char criteria [] = "box[-0.2, -0.1, -0.2, 0.2, 0.2, 0.15]";
8
                   n_selected_cells = 0;
       cs_lnum_t
9
10
       cs_lnum_t *selected_cells = NULL;
      BFT_MALLOC(selected_cells, mesh->n_cells, cs_lnum_t);
12
13
       cs_selector_get_cell_list (criteria,
14
                                  &n_selected_cells ,
15
                                   selected_cells);
16
17
       cs_mesh_refine_simple_selected (mesh,
18
                                                             /* conforming or not */
19
                                        true,
20
                                        n_selected_cells ,
21
                                        selected_cells);
22
       BFT_FREE(selected_cells);
23
24
    }
```

If you want to visulaize your mesh, you can change the execution mode to "Mesh preprocessing only" and run your case.





3.4 Restarting from the previous computation (again)

Move to the **Time settings** menu and in the **Start/Restart** item, make the computation restart from the previous computation (with boundary layers).

Run the computation on 4 to 6 processes.

