

Ecoulement dans une conduite

X. Fischer
ESTIA

Study Case

- **Flow simulations in a T-Junction**

- Laminar flow

- Turbulent flow

- Without viscosity AND No-Slip-Wall

- **Fluid Characteristics**

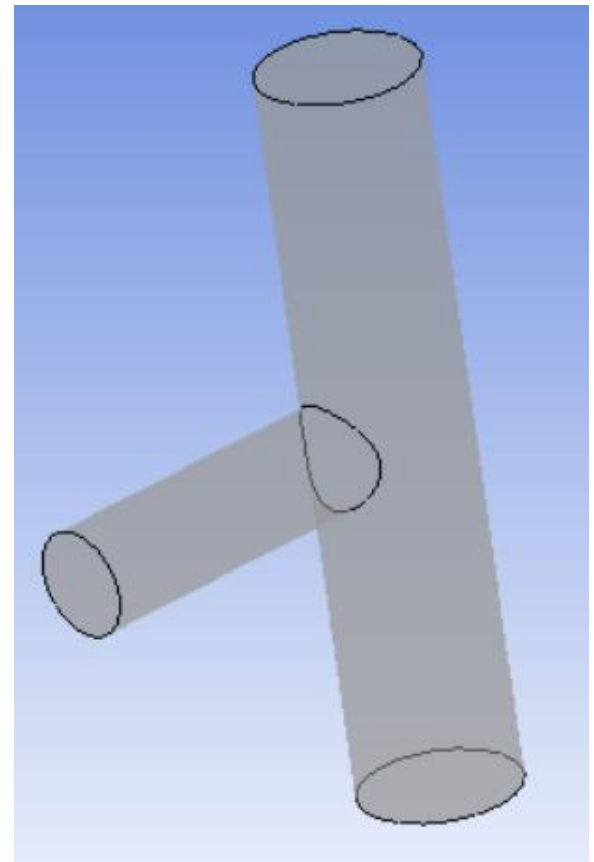
- Water

- **Flow characteristics**

- Isotherm

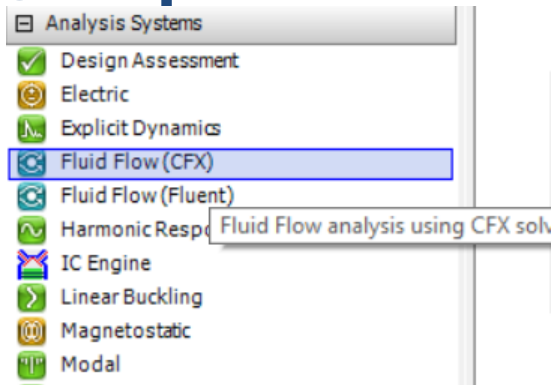
- Not considering Heat Transfer

Mono-Physic Simulation



1- Preparing Simulation

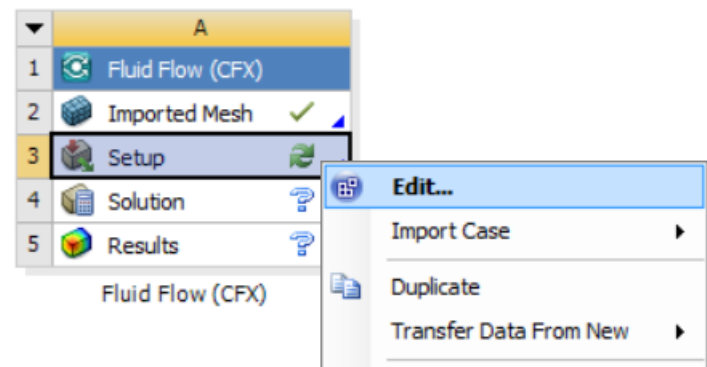
- Open workbench and choose the CFX Component



- Next, Import the mesh file: *T pipe.msh*

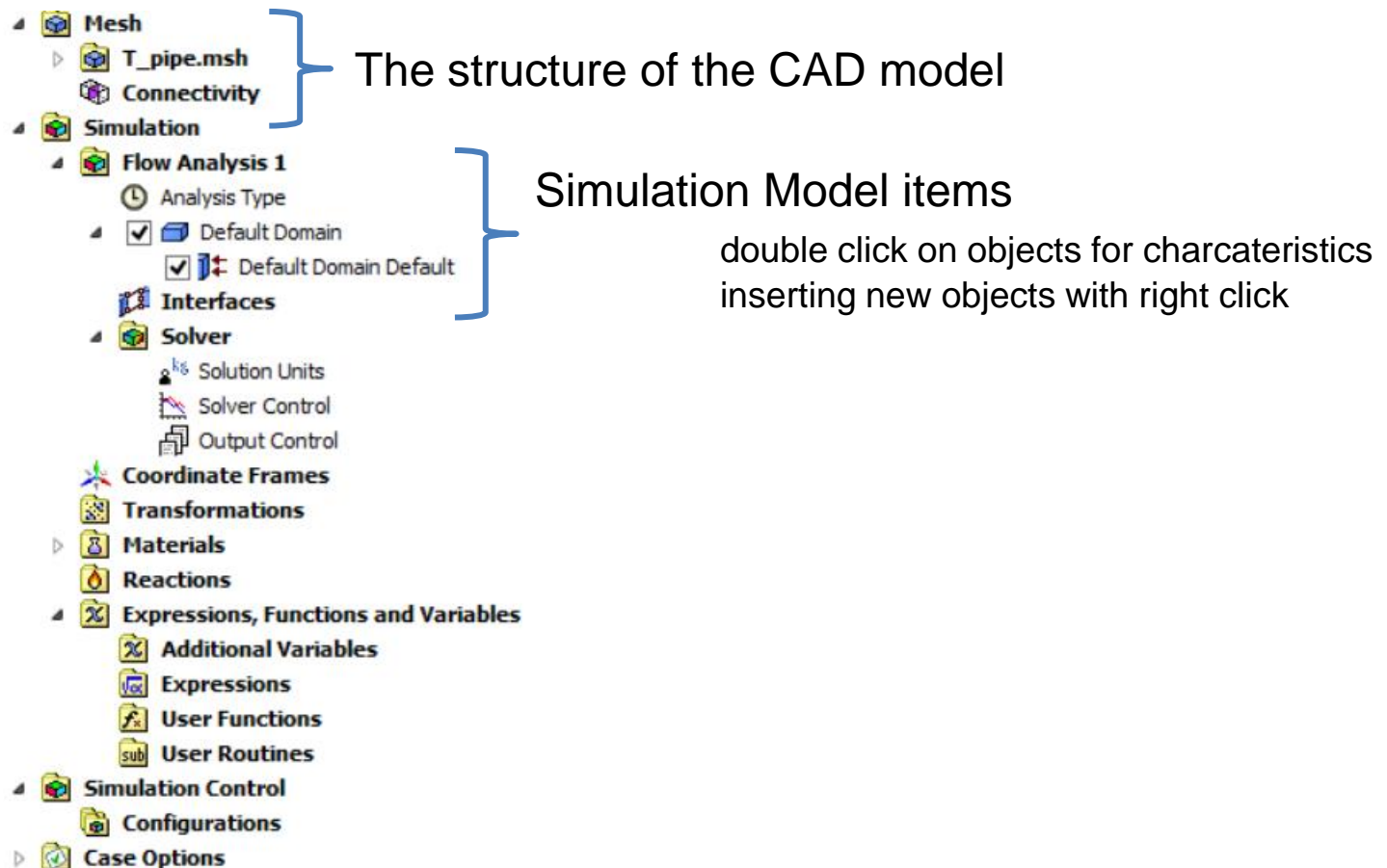


- Edit the Mesh with CFX



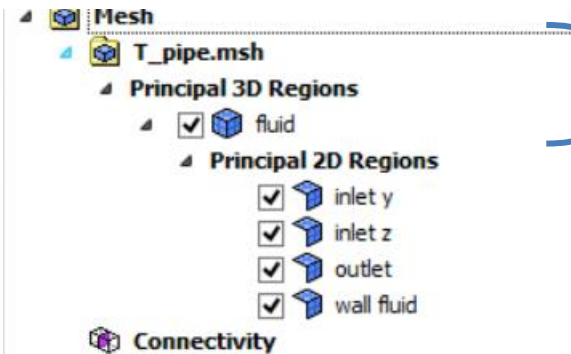
1- Preparing Simulation

- CFX is based on a Tree Architecture



1- Preparing Simulation

- **Mesh Structure**



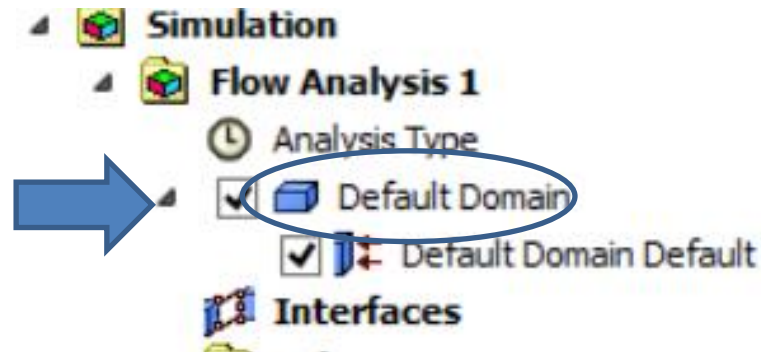
Expanding the mesh object allows to have details on:

- **3D object** : here the fluid domain
- **2D objects**: here the wall of the pipe and the inlet/outlet of the pipe

- **By default, the 3D domain is named in the model « *Default Domain* »**

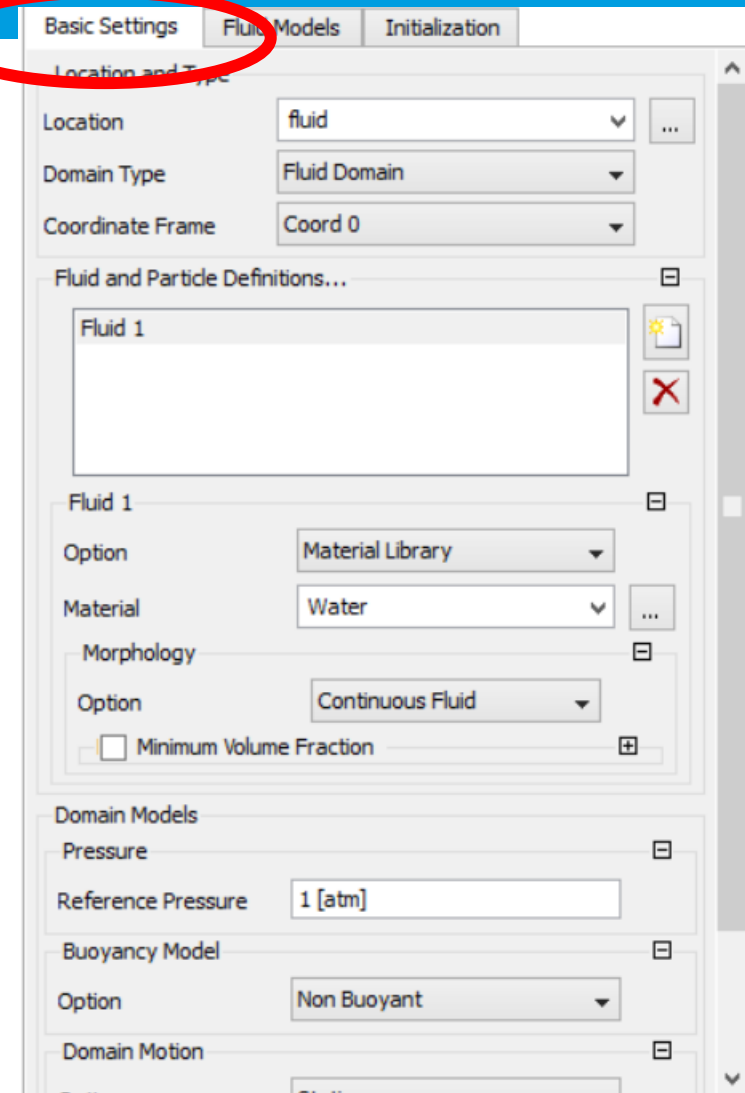
Here the fluid domain is named the « default domain »

It is possible to change the name with a right click



2- Flow Characteristics

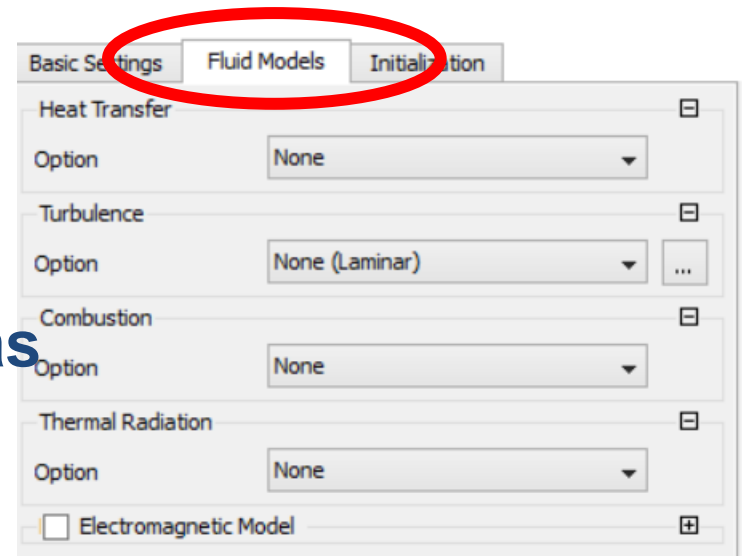
- **Modelling of Flow Characteristics**
 - Double click on « default domain » in order to define the characteristics of the **FLUID DOMAIN**
- Define the fluid domain as being **WATER**
- The pressure of reference is 101 325 Pa (1 atm)
- The flow is stationary (permanentà)



2- Flow Characteristics

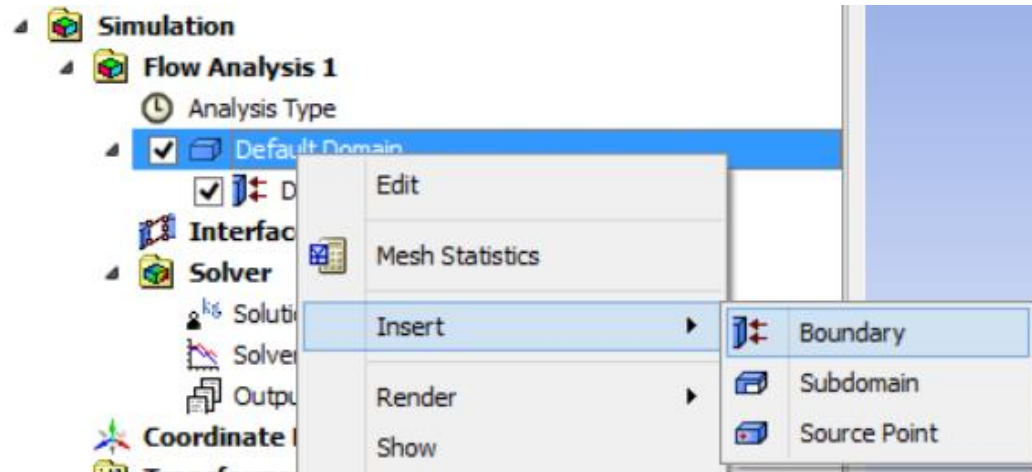
- **Modelling of Flow Characteristics**

- We are not considering the heat transfer
- The flow is, first, considered as being laminar



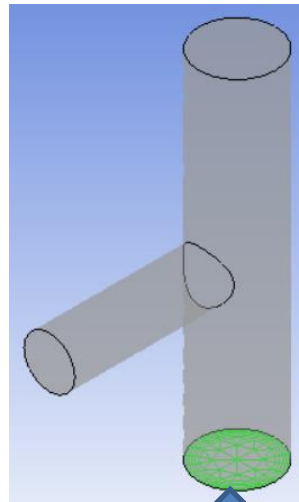
3- Boundary Conditions

- The boundary conditions are linked to the « *default domain* »
 - Right click on the « *default domain* » object on the tree
 - Select *insert Boundary*
 - Give a name to the BC

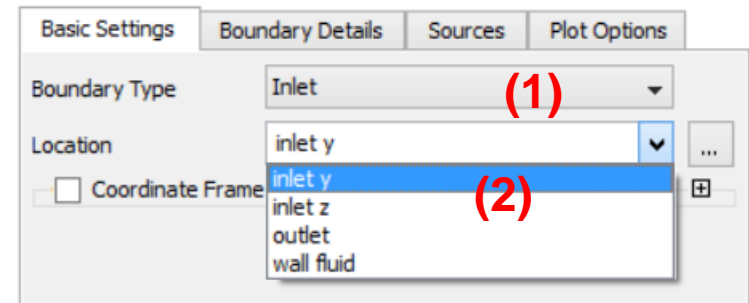


3- Boundary Conditions

- We define the BC on the inlet face of the pipe
 - (1) Precise that the BC is related to an inlet condition
 - (2) Choose the *inlet y* CAD object to support the BC (location of the BC)

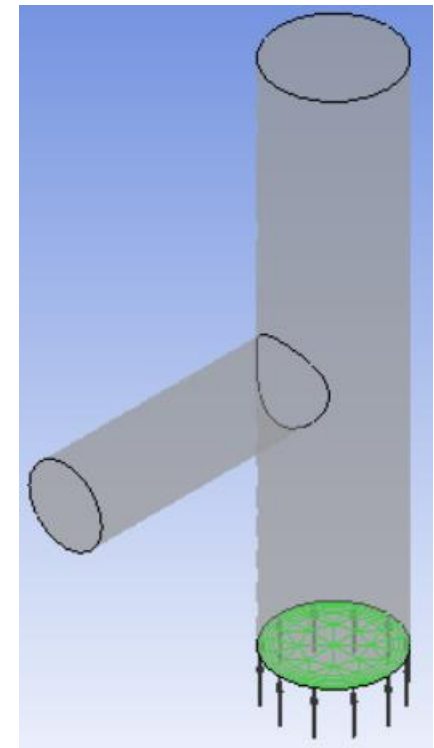
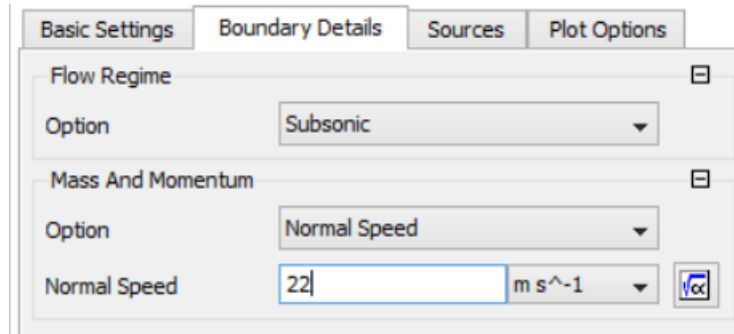


Inlet of the Fluid



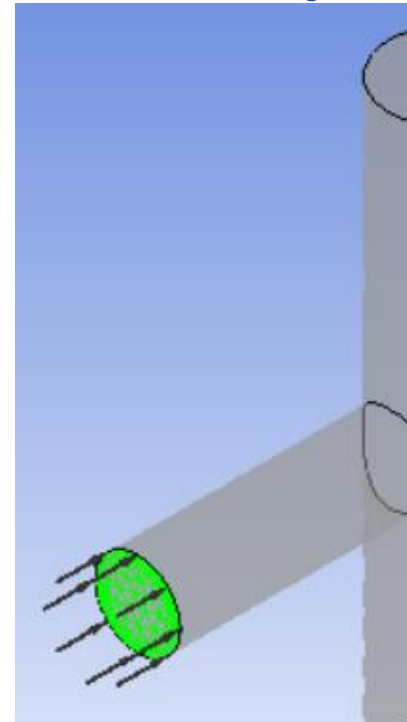
3- Boundary Conditions

- We define the BC on the inlet face of the pipe
 - The inlet speed of the fluid is 3 m/s



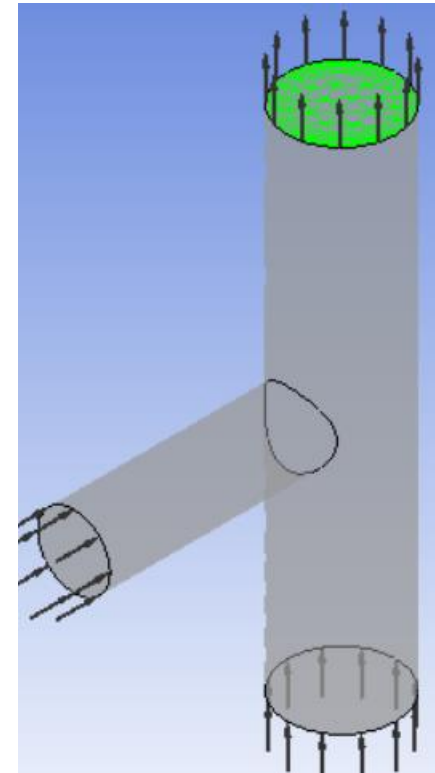
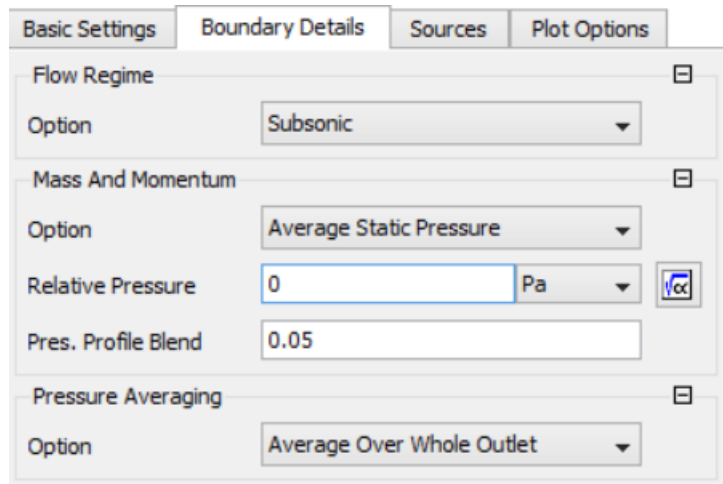
3- Boundary Conditions

- We define the BC on the second inlet face of the T pipe
 - This time, The inlet is located on the *inletz* CAD object
 - The inlet normal speed is 1 m/s



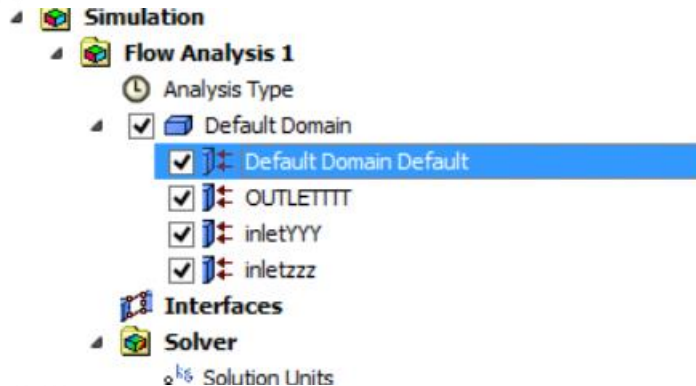
3- Boundary Conditions

- We define the BC on the Outlet face of the T pipe
 - The outlet is located on the *outlet* CAD object
 - The relative pressure is 0 Pa



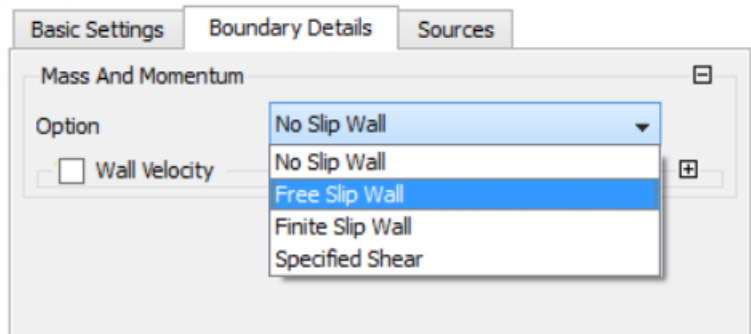
3- Boundary Conditions

- The 3 BCs are integrated in the tree



- The BC named « Default Domain Default » is related to the detail on the wall behavior

- We consider the water not having viscosity :

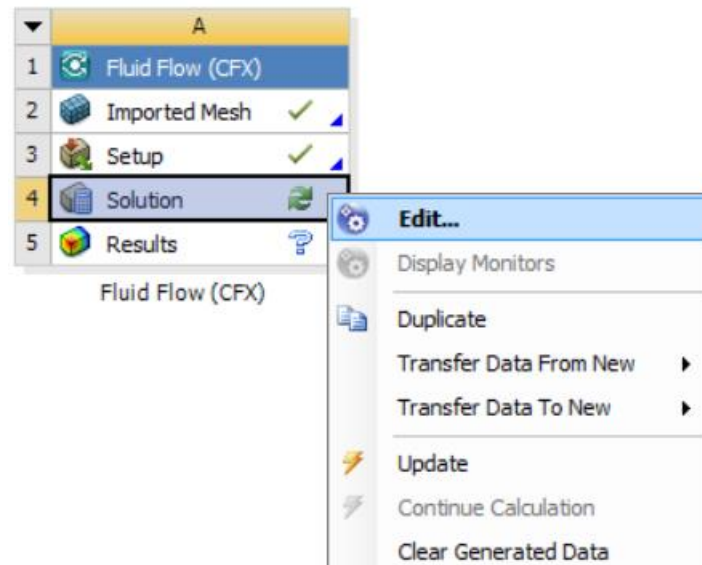


1- Double clic on the wall BC object

2- choose the right option for avoiding viscosity

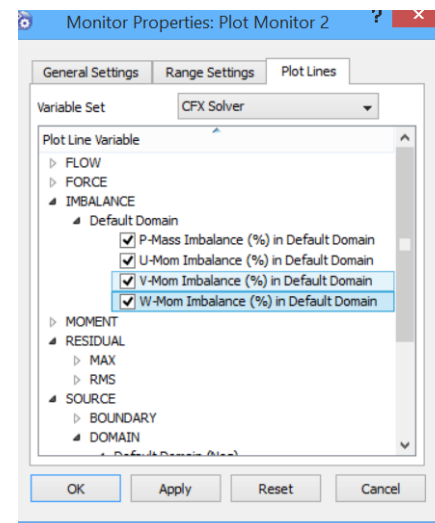
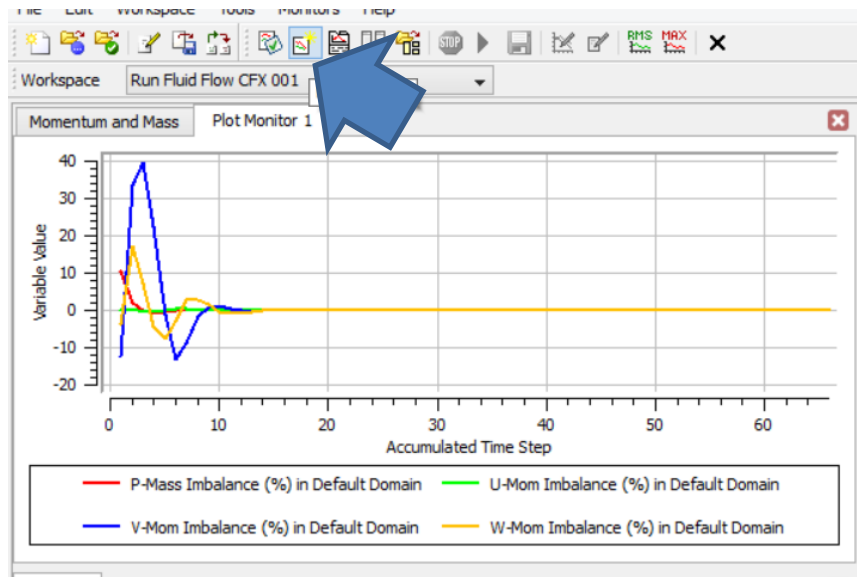
4- Processing

- For launching the numerical process, go back to workbench
- Edit the SOLUTION object



4- Processing

- You can analyze the convergence by clicking on monitor editing icon and choosing other elements



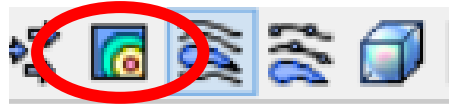
5- Post Processing

- The kind of results is defining in the head menu of the Post Processing Tool



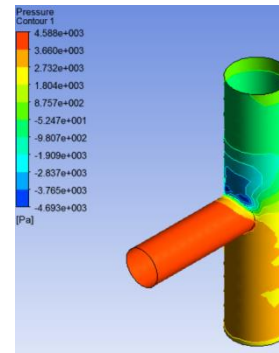
- Display the contour of pressure on the wall:

1. Clic on contour icon
2. Choose the wall domain (here default domain)



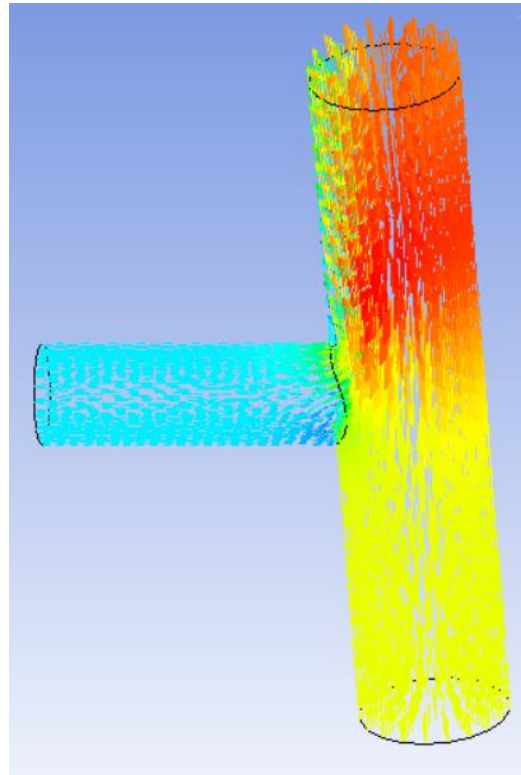
Details of **Contour 1**

Geometry	Labels	Render	View
Domains	All Domains		...
Locations	Default Domain Default		...
Variable	Pressure		...
Range	Global		
Min	-4693.38 [Pa]		
Max	4588.44 [Pa]		
Boundary Data	<input checked="" type="radio"/> Hybrid <input type="radio"/> Conservative		
Color Scale	Linear		
Color Map	Default (Rainbow)		
# of Contours	11		
<input type="checkbox"/> Clip to Range			
<div>Apply Reset Defaults</div>			



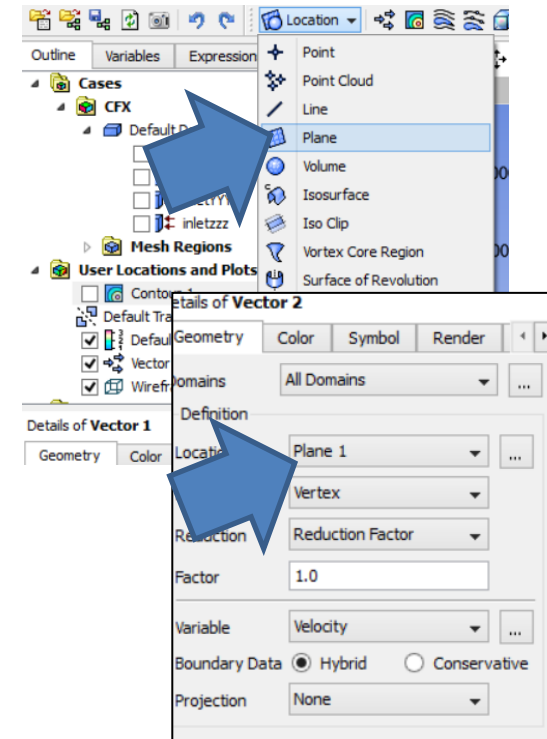
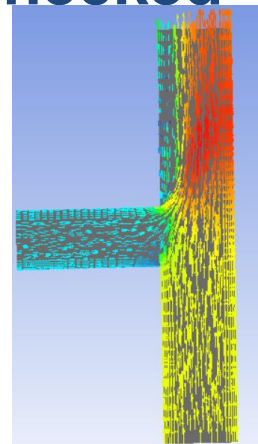
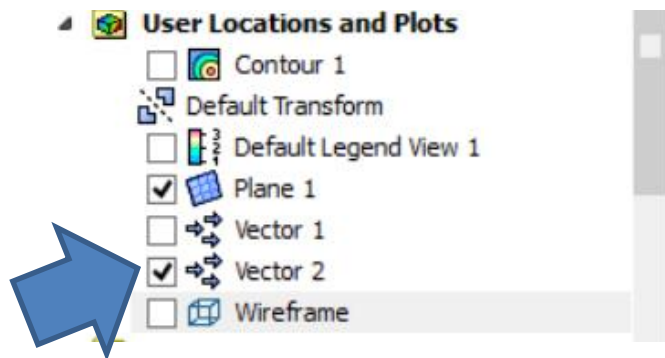
5- Post Processing

- **Next display the speed field in the fluid domain**
 - Confirm that the water is non-viscous
 - Note the profile



5- Post Processing

- Next display the speed field in the fluid domain
 1. Create a mid plane in the YZ plane
 2. Create the speed profile in this plane
- In order to have a good view only have the BC on the plan being checked





OTHER SIMULATIONS

1. The speed at inlet is 22m/s and 12m/s at inletz

Implement a laminar simulation

next implement a turbulent simulation by using k & model

2. Realize the simulation considering the water being Viscous

What are your conclusions on the speed?