Tutorial for the heated pipe with constant fluid properties in STAR-CCM+

For performing this tutorial, it is necessary to have already studied the tutorial on the upward bend. In fact, after getting abilities with that case, many concepts will turn out relatively straightforward.

Problem definition

A circular pipe having an heated length of 1 m with imposed heat flux. The inner pipe diameter is 6.26 mm.

The contained fluid is water at relatively low temperature (around 25 °C) and 0.1 MPa. The code assigns water properties by default; they are:

- Density: 997.561 kg/m^3
- Dynamic viscosity: 8.8871E-4 Pa.s
- Specific heat at constant pressure: 4181.72 J/(kgK)
- Thermal conductivity: 0.620271 W/(mK)

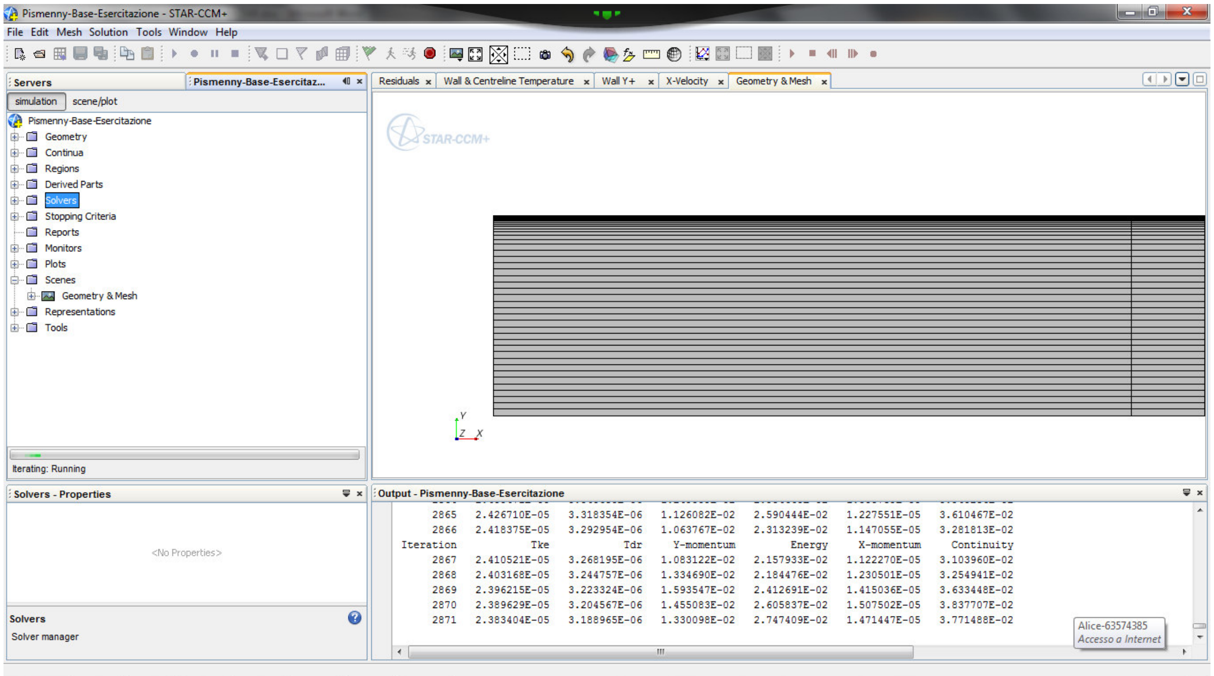
We will accept these default values for our purposes of making an exercise on variable distribution in the pipe.

The boundary conditions adopted for this case are:

- Inlet velocity: 2 m/s
- Inlet temperature: 293.15 K

In this problem, it was chosen to use "a low-Reynolds number model", requiring a very fine discretisation close to the wall. The resulting radial discretisation is much larger than the axial one.

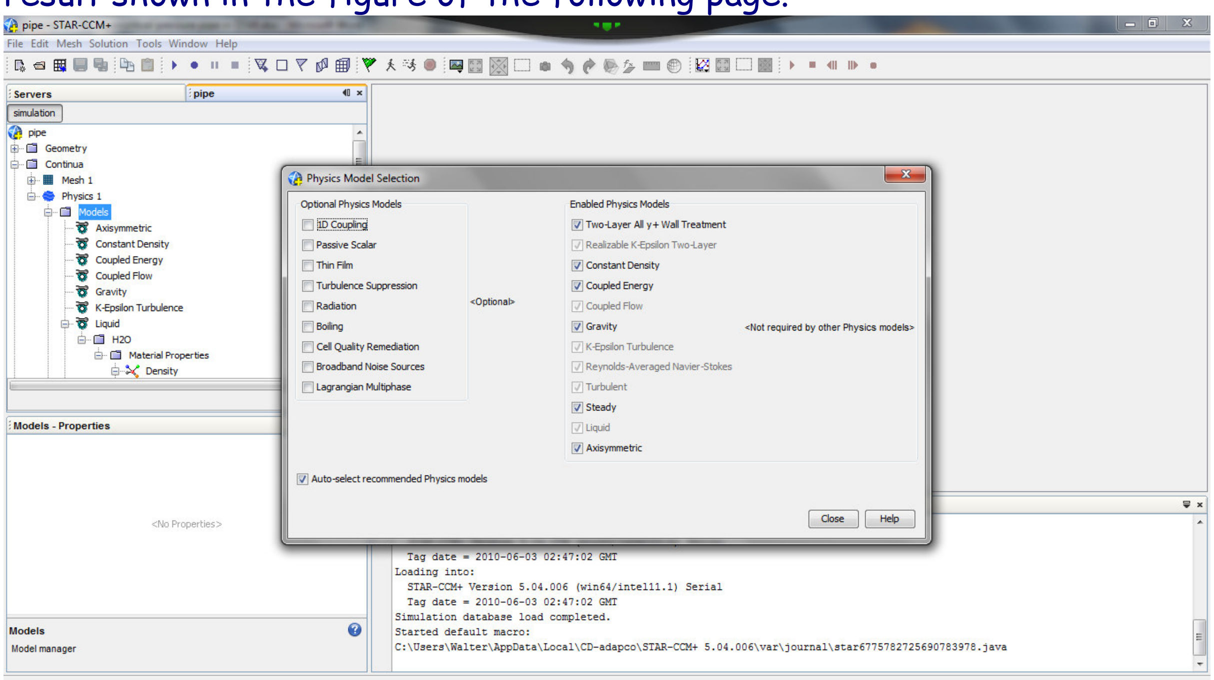
Expand the related "Scene" to understand how the mesh looks like. Make use of the mouse for displacing the geometry in the view.



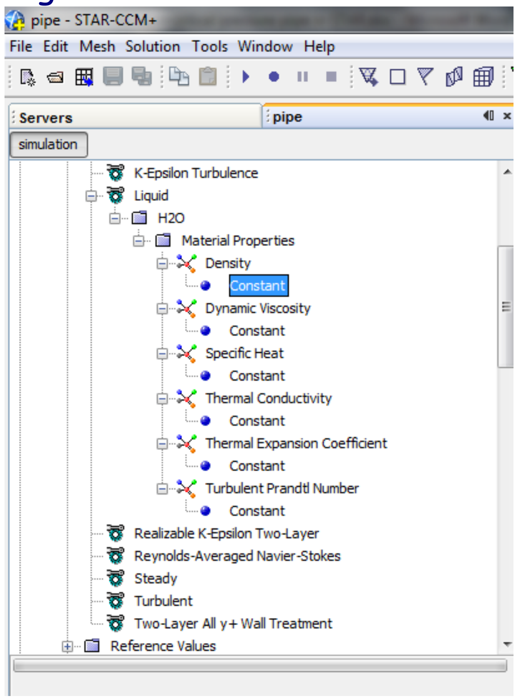
In the following, suggestions are given to get through the different sections of the tree structure.

Continua

Try understanding which models have been selected for this case. Open the related node in the tree structure by right-clicking to obtain the result shown in the figure of the following page.



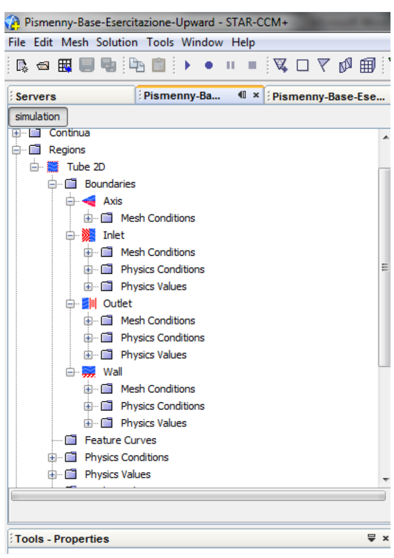
The properties are assigned as constants:

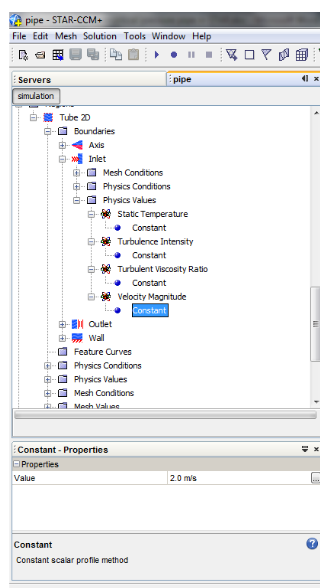


Regions

Try understanding the nature of the boundary conditions applied to the four different boundaries.

In particular, look at the Physics Conditions and the Physics Values. E.g., the velocity at the inlet is 2 m/s…



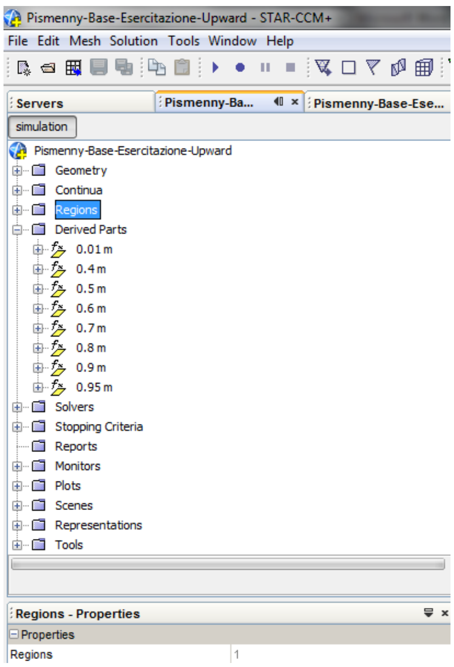


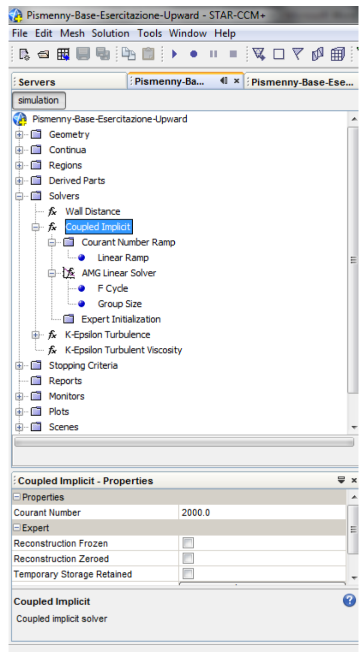
Derived parts

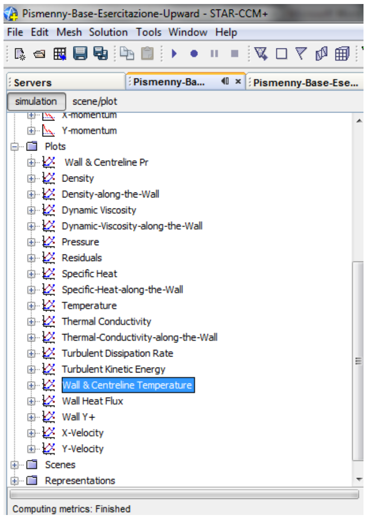
They have been introduced as "sections" to be used in setting up plots of radial distributions at different distances along the pipe. Look at how they are defined.

Solvers

In this case a "Coupled" flow and energy numerical scheme (as opposed to the "segregated" one) was used. This option is preferable in cases in which the flow is affected by strong buoyancy forces. A large "Courant number" is assigned in a "pseudo-transient" iteration scheme.



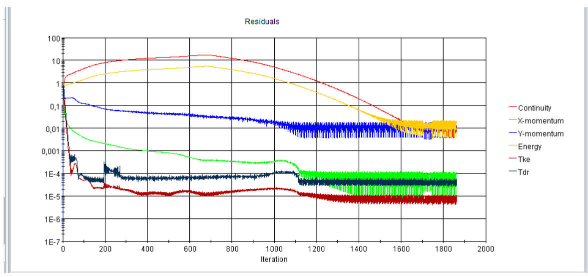




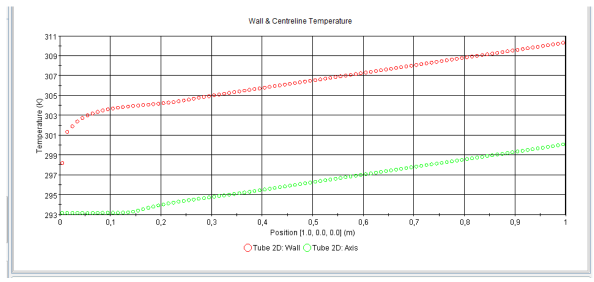
Plots

In this case several 2D plots have been defined in order to monitor axial and radial distributions of different quantities

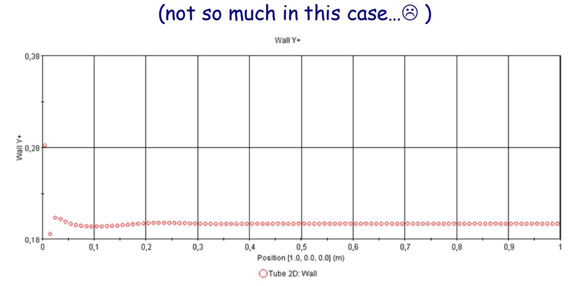
Some of them are presented hereafter.



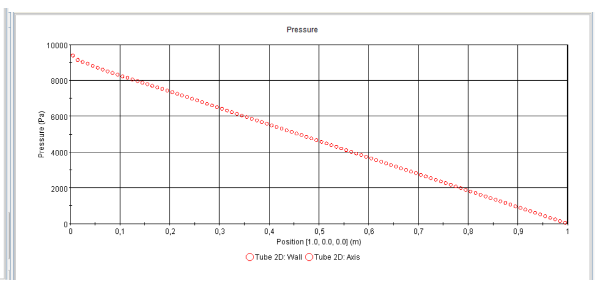
The residuals decrease and stabilise (not so much in this case…☹)



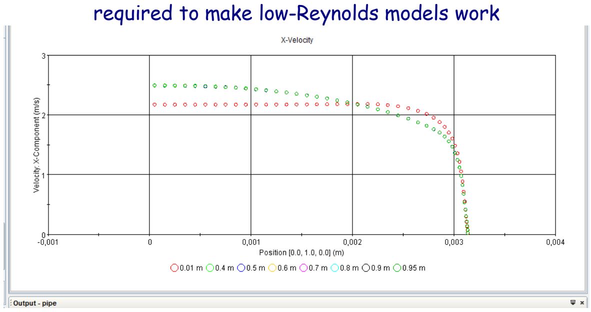
The wall temperature increases smoothly



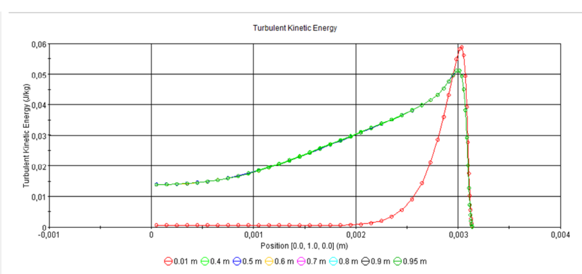
The y+ at the centroid of the first node is less than unity, as required to make low-Reynolds models work



The relative pressure decreases almost linearly along the pipe



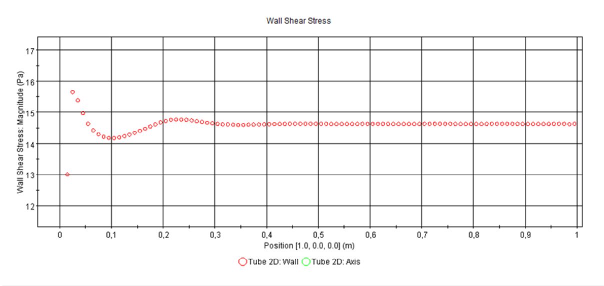
The radial velocity profiles at different locations are typical of a flow that is not affected by buoyancy forces (i.e., it is of the power law type)



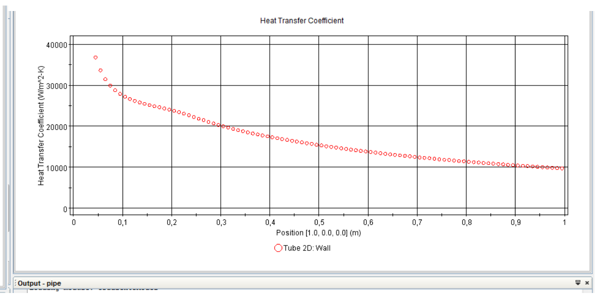
The turbulent kinetic energy has the classical sharp peak close to the wall. Note the large number of nodes needed to describe the near wall region with a reasonable detail in the boundary layer

Suggestions to make exercises.

- Compare the plots obtained for the shear stress at the wall and the heat transfer coefficient with the asymptotic values of the Blasius relationship and the Dittus-Boelter correlation.
- In making the comparison with the heat transfer coefficient, consider that the CFD code calculates it on the basis of a reference temperature as the bulk fluid one, since the bulk temperature is defined nowhere; so this comparison can be only parametrical



Shear stress at the wall



Heat Transfer Coefficients

- Check the coherence of the shear stress with the predicted pressure drop (that does not contain the gravity term: so, it is only friction driven).
- Run the code after making changes (e.g., inlet flow rate, different turbulence model). You will see the residuals jumping (the equations are no more satisfied with the new parameter value) and then, hopefully, decrease while a new steady-state is approached.

AGAIN, GOOD LUCK !