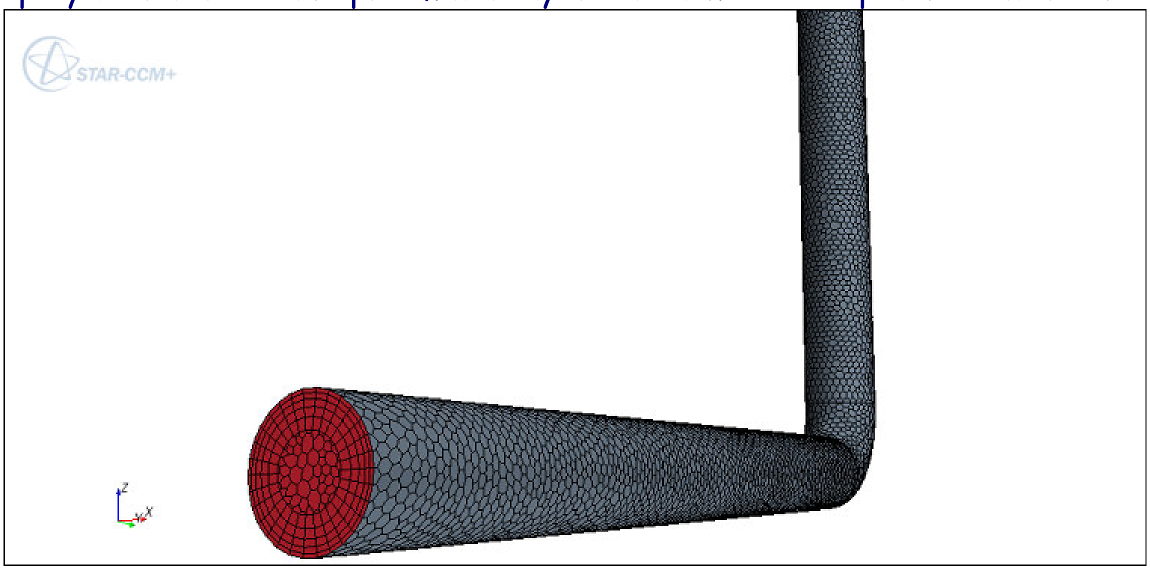
Tutorial for the problem of the upward bend with STAR-CCM+

Geometry definition

The geometry consists of an horizontal and a vertical pipe connected by an upward directed bend.

The figure below shows that the geometry was meshed making use of polyhedral cells and prismatic layers as it will be explained later on

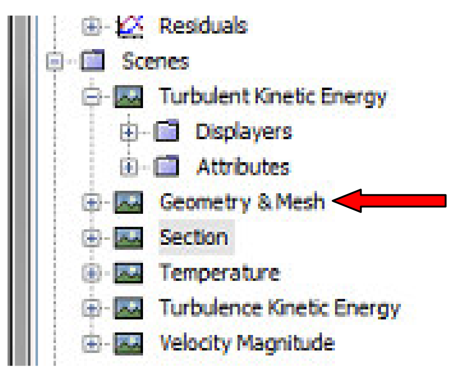


The "dragon-like" skin is the result of the prismatic layering close to the surface (needed to evaluate in detail the trends of variables close to the surface) and of the polyhedral prisms inside the "core" of the pipe.

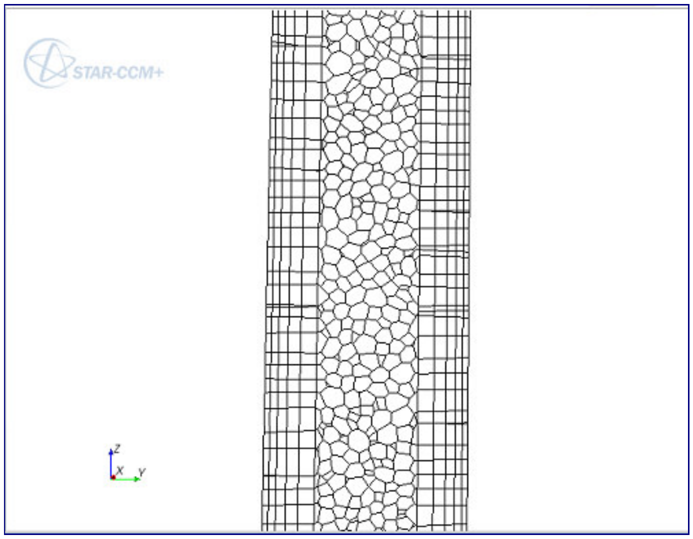
The size of the system is as follows:

- pipe diameter: 0.1 m
- horizontal and vertical pipe length: 1 m.

In order to display the system geometry, open the related node in the lateral tree structure



Clicking on "Section", a diametral section of the meshing is displayed



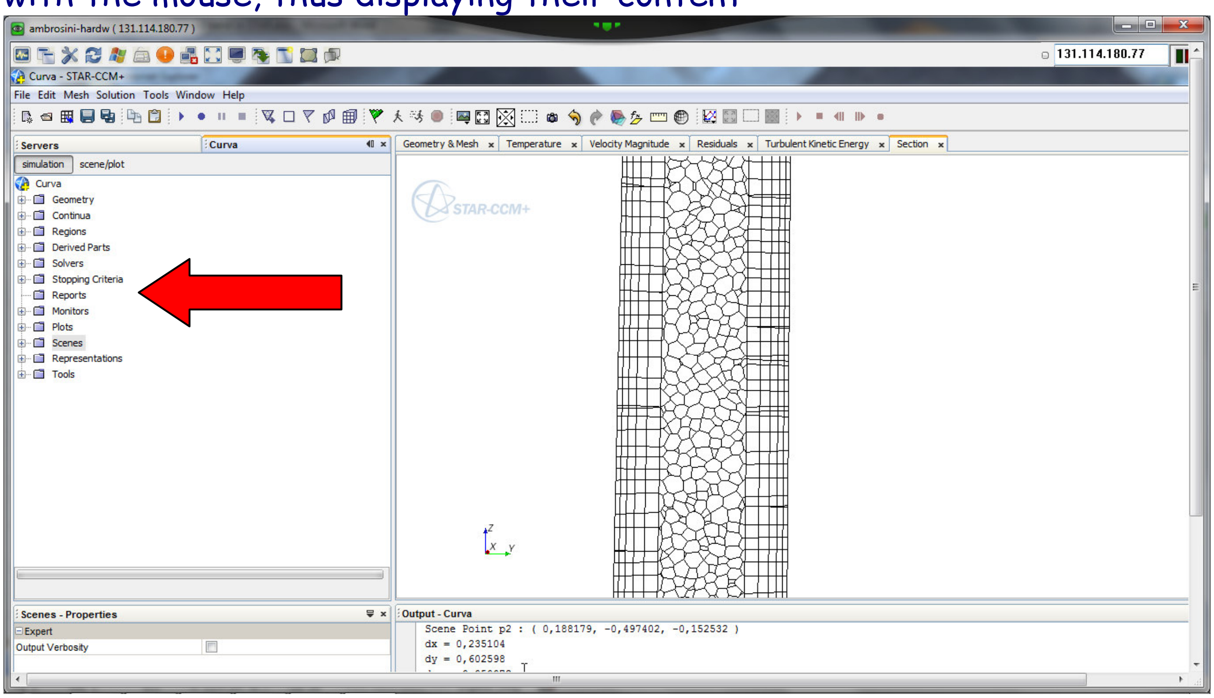
showing the structure of the core polyhedrals and of the prismatic layers.

These "Scenes" have been prepared by the "user" in previous sessions.

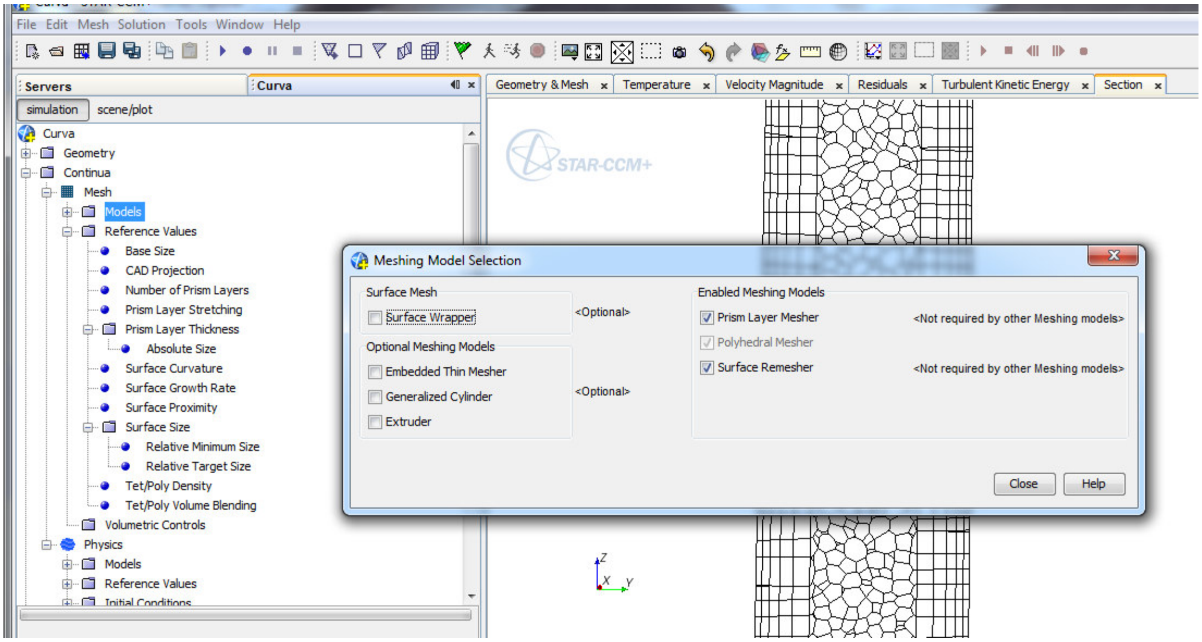
Let's understand what was made by the user.

The tree structure

The lateral "tree structure" has "nodes" that can be opened by clicking with the mouse, thus displaying their content



<u>Continua</u>



By right-clicking on the node Mesh > Models, the pop-up menu for selecting the meshing models is displayed. In this case, it shows that the Polyhedral Mesher, the Prism Layer Mesher and the Surface Remesher models were selected.

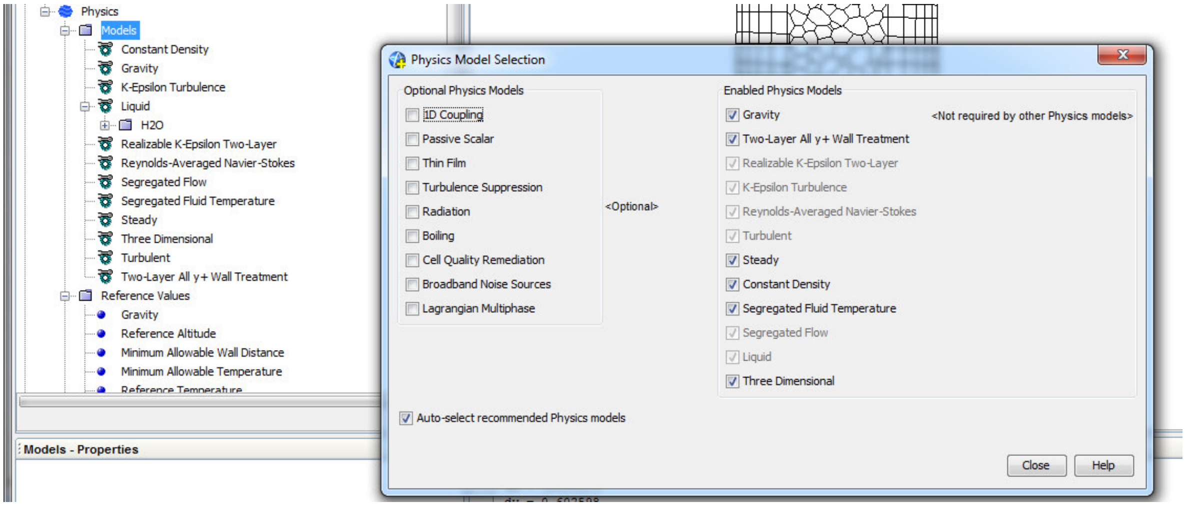
By clicking on each node in the tree, discover which values have been selected for the "base size" (the reference mesh size), the "number of prism layers", the "prism layer stretching" (the ratio between the size of two adjoining meshes in the prism layer), the prism layer thickness and try to compare with the figure to understand the consequence of each selection.

<u>Physics</u>

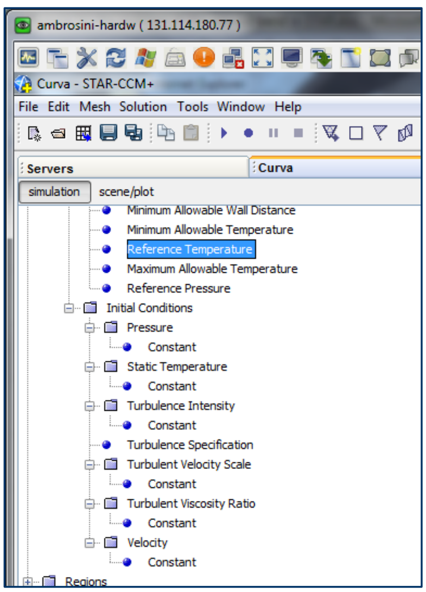
By right clicking on the Physics node, the pop-up menu for selecting the physics models is displayed.

Try understanding which models have been selected. For instance:

- gravity is activated
- a two-layer wall treatment with all y+ (wall functions) is used
- the turbulent, RANS, realizable k-ε models are selected
- a steady algorithm (no time derivatives) is used with constant density for a liquid (H2O, see in the tree)

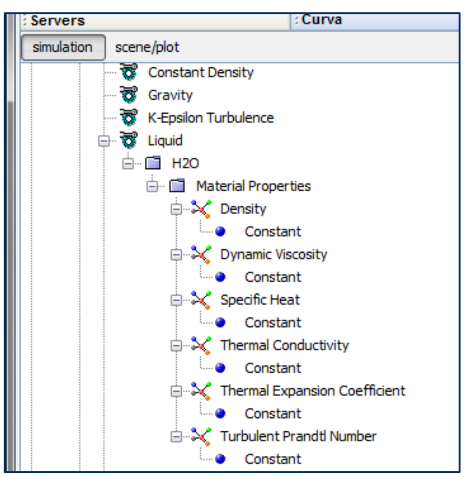


Try discovering the "Reference values" defined for gravity, altitude, etc. appearing in the related branch of the tree.



Initial values of the different variables (pressure, temperature, turbulence intensity, etc.) are also provided in the tree for purpose of initialization of the steady-state calculation.

Try understanding the format in which these data are assigned.



Expanding the node of H2O the properties assigned to water for the calculation can be considered.

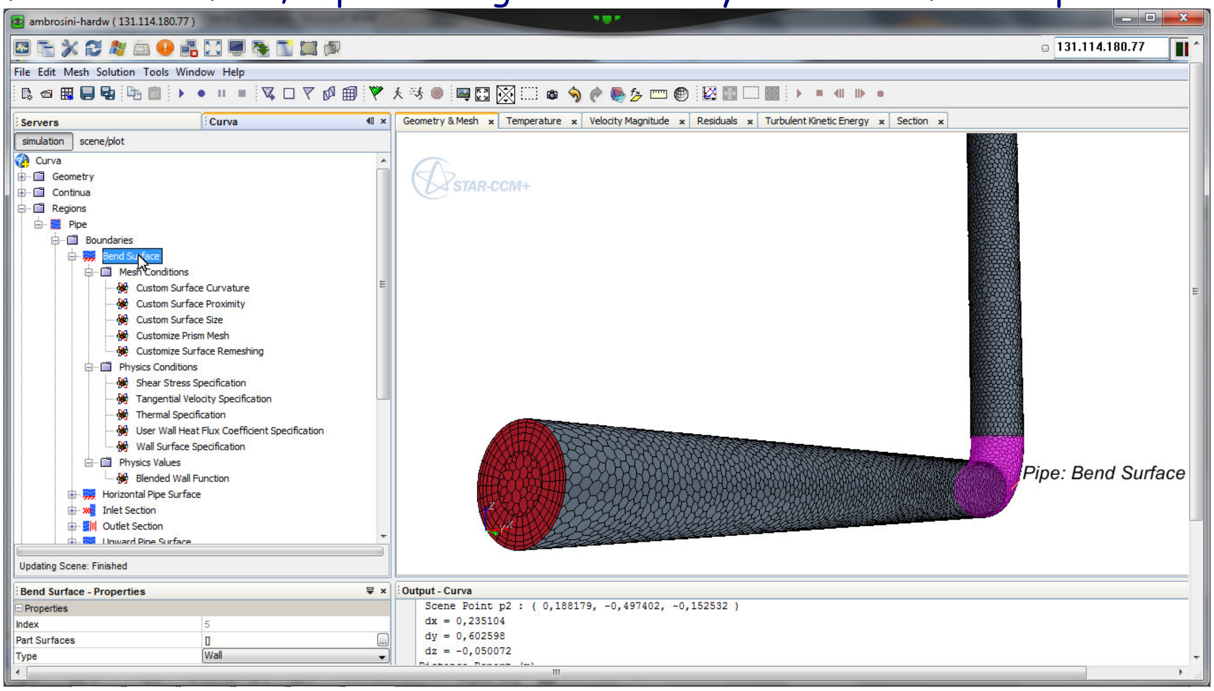
Each value can be changed at ease.

Moreover, instead of constant values, variable ones as a function of temperature or other variables can be defined (e.g., through "field functions")

Region

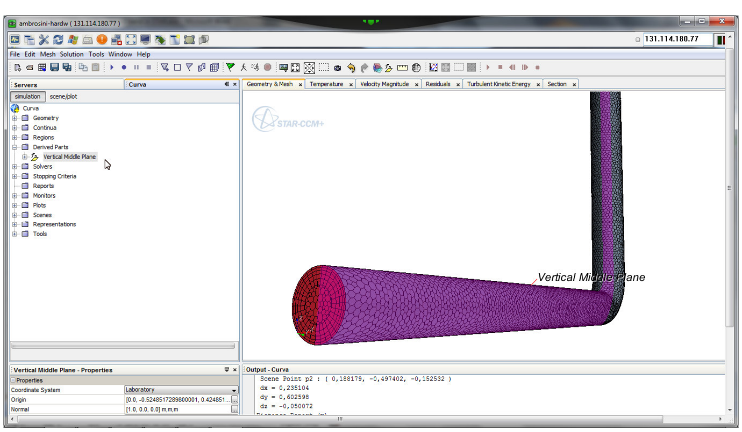
A single region is present in this problem called “Pipe”. The Pipe has several surfaces (boundaries) that can be highlighted in magenta colour when selected in the tree

It is quite interesting to look at the “physics conditions “ selected for each surface, implementing the boundary conditions for the problem.



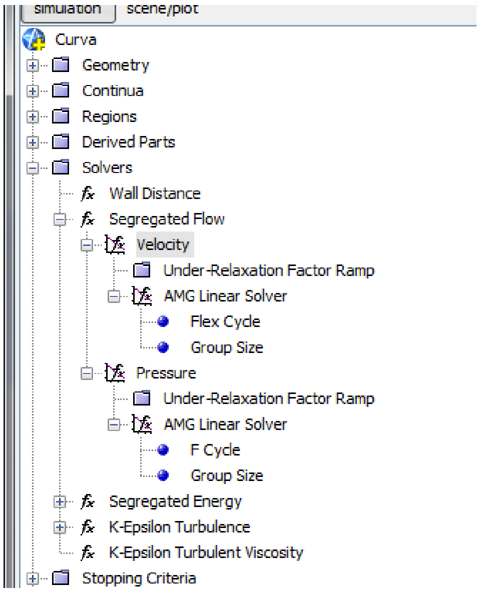
Try discovering the specifications assigned to the different boundaries, being “walls” with different shear stress and thermal specifications, inlet and outlet. Try to use the window menus to discover the different available options that the user can select and try understanding which ones have been selected for this particular case, together with the adopted numerical values.

Derived parts



A single derived part was defined for this case, as a plane cutting the solid in the middle in order to display relevant variables (temperature, velocity, etc.) in form of contour plots (this will be seen in “Scenes”)

Solvers and Stopping Criteria

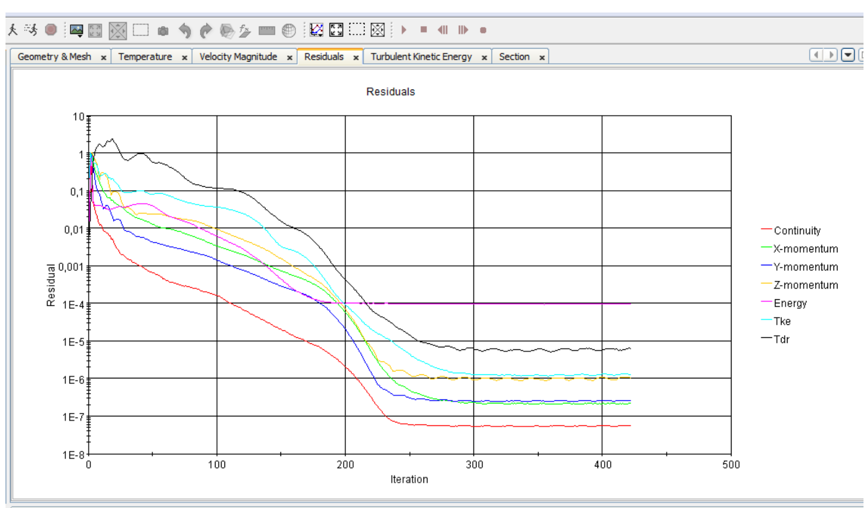


Under the "solvers" section, the different options for the solvers selected in the physics section are displayed. In particular, the under-relaxation factors used for "velocity", "pressure", "energy", k and ε can be displayed and changed at ease.

Different stopping criteria can be selected for controlling the iterations.

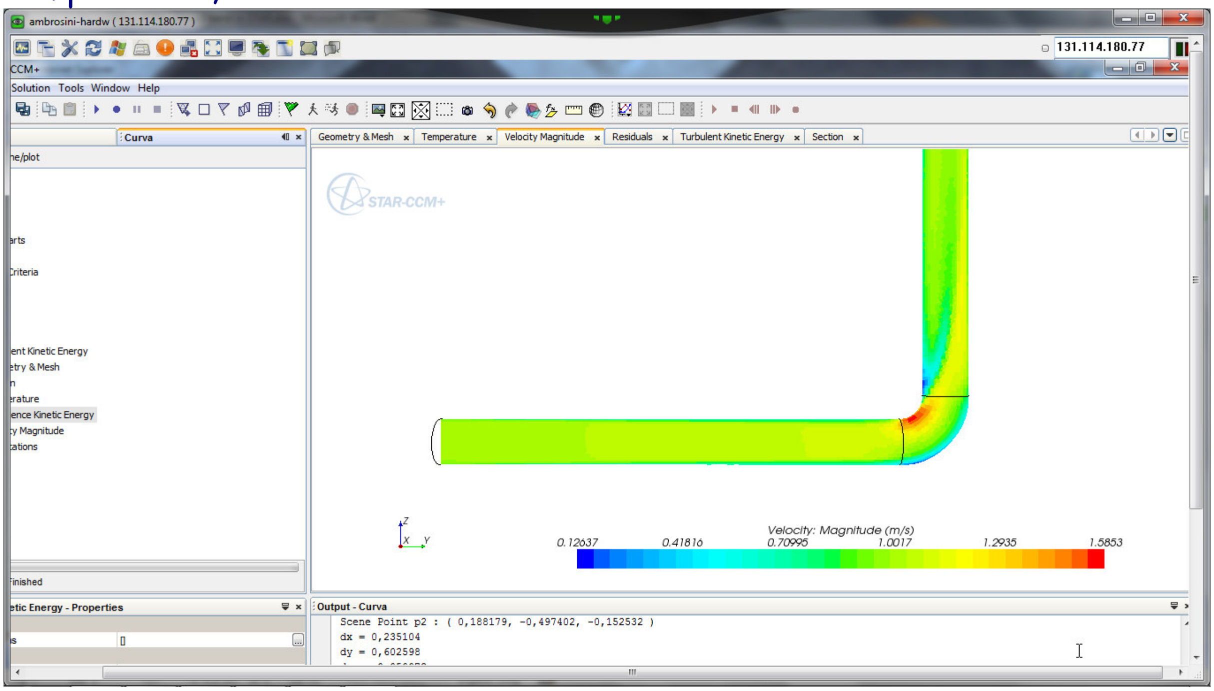
Reports, Monitors and Plots

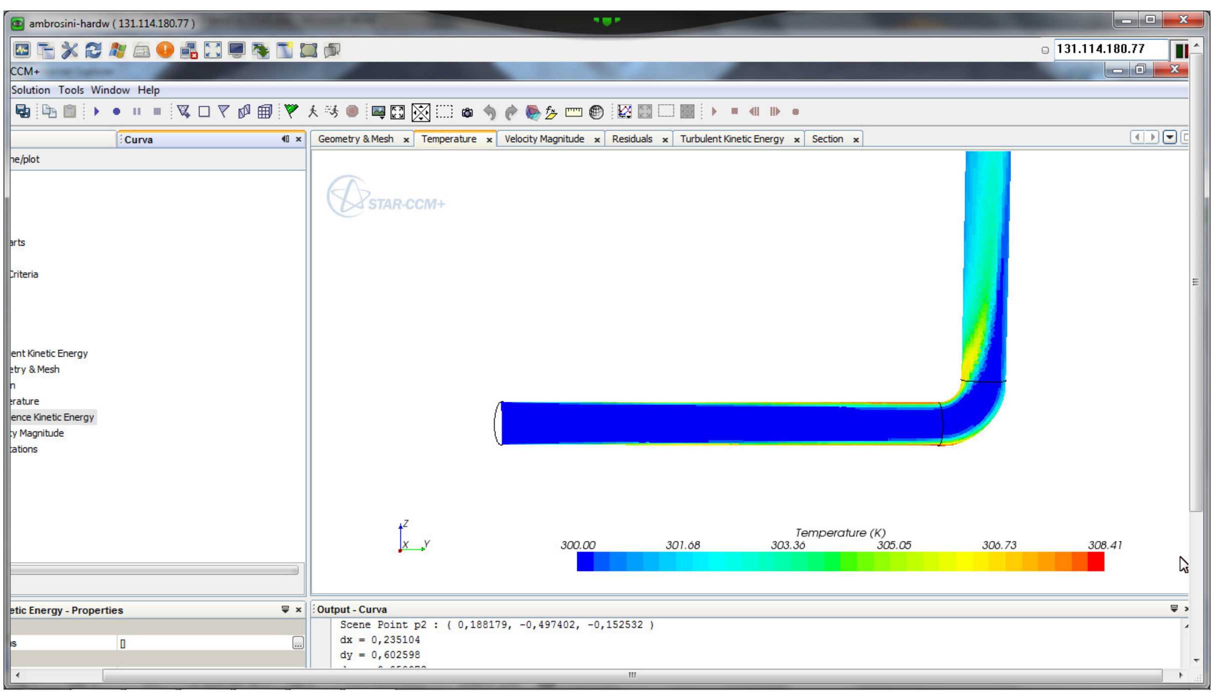
They represent variables that can be defined in different ways in order to "report" and "monitor" relevant variables to be considered during iterations. Among them, of overwhelming importance are the "residuals" of the different balance equations, that must be monitored to judge about the convergence of the process. They are displayed in a Plot.



Scenes

In addition to the Geometry and Mesh Scene that we have already considered, other "Scenes" can be generated in order to display different interesting variables, as the velocity magnitude or the temperature, etc.





Tools

Among the different tools, the Field Functions represent variables defined throughout the whole computational domain that can be used to represent variables in plots or scenes or to make computations by defining further field functions.

Rethinking to the overall picture

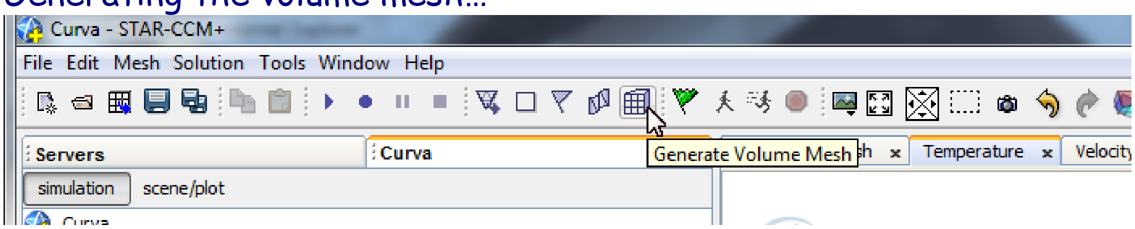
After this description of the specific nodes in the tree, now try to go back asking yourself what to do in case you would like to change a model, a value or such…; for instance, try answering the following questions:

- how to change the model from turbulent to laminar?
- how to change the thermal boundary condition of a surface (making it adiabatic or imposing a temperature or a heat flux…)?
- how to change the mesh parameters?
- etc.

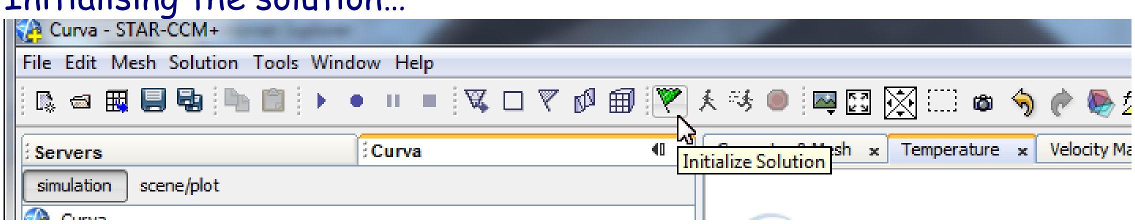
Relevant buttons

There is a number of buttons on the screen that can be pushed to activate a function. Try discovering them by the pointing device (the mouse arrow) as below.

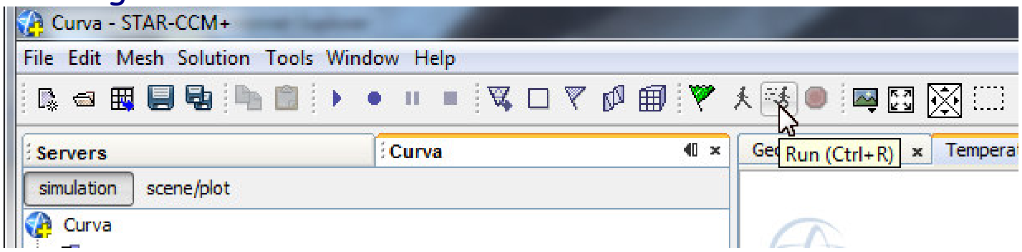
Generating the volume mesh…



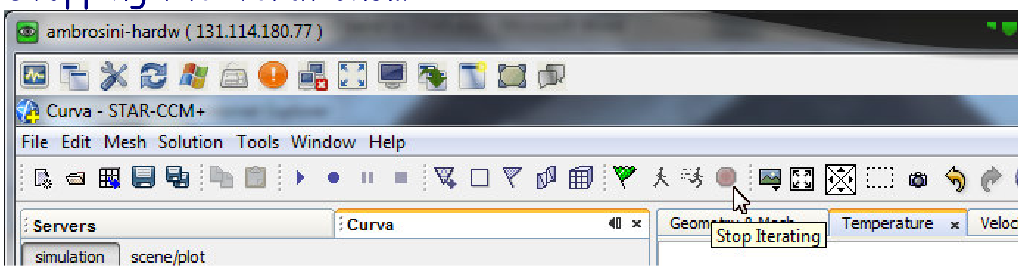
Initialising the solution…



Running the solution…



Stopping the iterations…



For instance, after saving the file with another name (to preserve the old one), try doing the following:

- re-initialise the solution and start iterating while monitoring the residuals… (WOW!!! You are running your problem!!!)
- while you are running, look at the different Scenes to see how the displayed variables change during iterations
- make a little change to the mesh parameters (e.g., use a 9 mm base size instead of the original 1.0 cm) and generate a new mesh, running again the problem
- make a change in the boundary conditions, reinitialise and run again, considering how the Scenes modify during iterations… you can really monitor what's happening during the progress of the calculation…

GOOD LUCK !