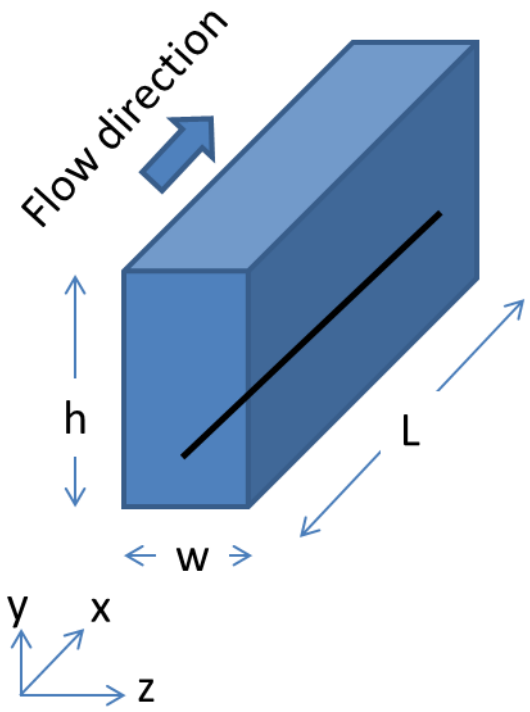
Test case: 3D Flat plate, L = 2m, w = 0.2m, h = 0.5m



2 Cases were investigated:

- Case A: Code_Saturne was used for simulation, Paraview used for post-processing
- Case B: Ansys Fluent was used for simulation, Paraview used for post-processing

Steady State simulation using k-epsilon turbulence model, with standard wall function.

Boundary conditions:

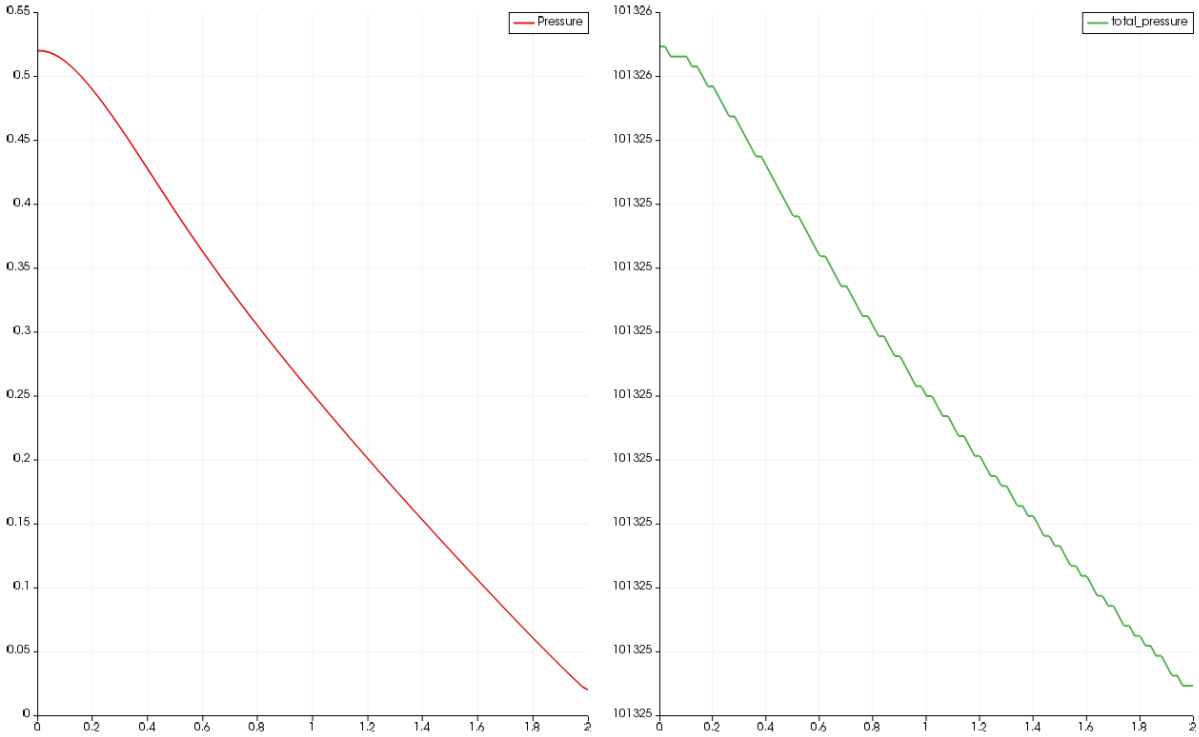
- Inlet: velocity 0.2m/s
- Outlet: pressure 101325 Pa (imposed_p_outlet)
- Walls: no-slip wall

In both cases, the pressure was plotted along a line as in the figure above, which goes on the X axis, from the start to the end of the flat plate.
The line is positioned in the centre of the geometry

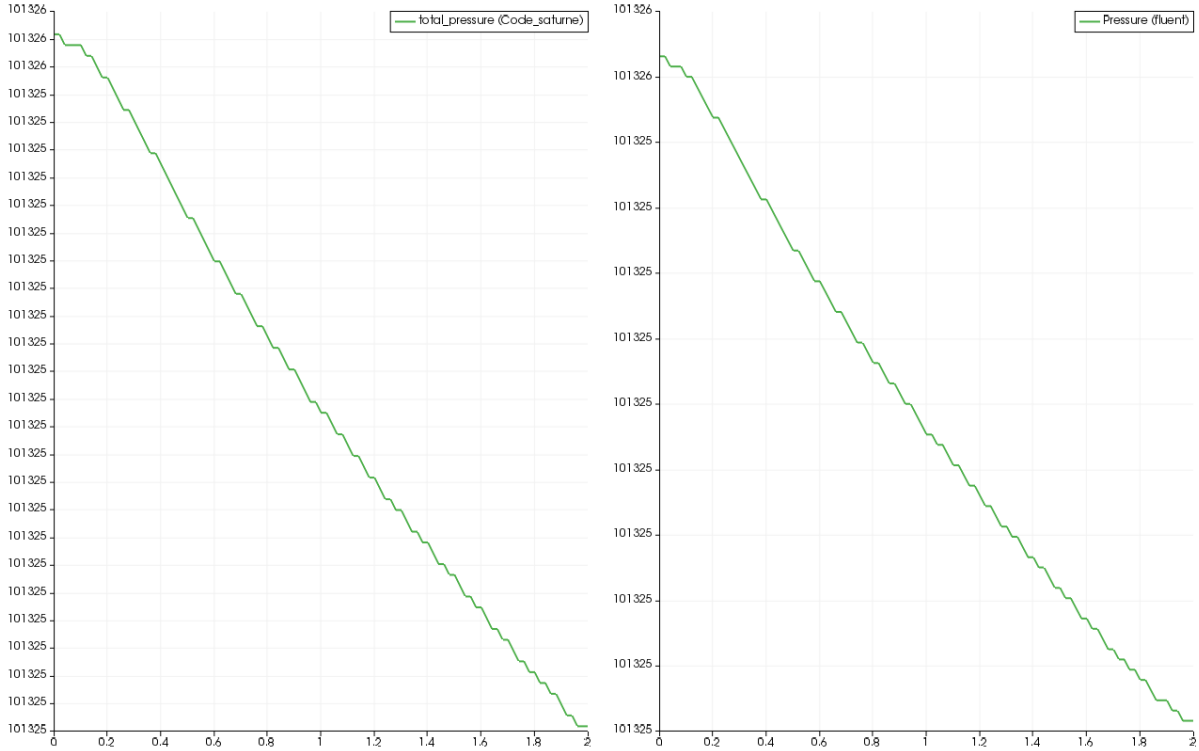
- x1=0m, y1=0.25m, z1=0.1m;
- x2=2m, y2=0.25m, z3=0.1m.

The reference pressure was set to 101325 Pascals

Why the “Pressure” variable has so small values (max 0.52)?
In the figures below, I plotted the Pressure and the Total Pressure along the line.
I made a comparison between Code_saturne and Fluent, it seems that the Total Pressure from Code_saturne is similar with the Static Pressure from Fluent.



Pressure and Total Pressure from Code_saturne, plotted along the line



Total Pressure (Code_saturne) vs Static Pressure (Ansys Fluent)