

The Christmas Community Challenge 2019

This PDF provides some information about the steps done to calculate the pressure within the Pitot tube and how Holzmann CFD does set-up the case. Thanks to József Nagy, who invented this challenge in 2017.

Prologue

Contributor: Tobias Holzmann

Dedicated to the OpenFOAM® community for further improvement in the field of numerical simulations by using open source toolboxes such as OpenFOAM®, Blender®, ParaView® and Salome®.

The case files will be provided on [Holzmann CFD](#)'s website. The description given in the next pages are kept short. The set-up of the cases are provided within the case files.

Enjoy and keep foaming

Generated by L^AT_EX on October 26, 2019

The Pitot Tube Case - 2019

The following document provides basic information of the Pitot tube case built by Tobias Holzmann. As already mentioned, the description is kept short based on the available case.

The Challenge

The competition was invented by József Nagy before christmas 2019. The challenge is to calculate the pressure difference of an arbitrary Pitot tube using OpenFOAM® while comparing the numerical results with the analytic ones at five defined velocity points: 0.1 m/s, 1 m/s, 67 m/s, 100 m/s, and 240 m/s. The Pitot tube design is arbitrary as well as the analysis type such as 2D or 3D and the fluid in use.

The Geometry - The Pitot Tube

The Pitot tube used by Holzmann CFD was a 2D model built arbitrarily out of his mind. The tube was placed inside a pipe. The geometry dimensions can be evaluated within the case files. The CAD data are provided too.

The Numerical Mesh

The numerical mesh was generated by snappyHexMesh while extruded for a 2D mesh. Note, no rotational-symmetric mesh was generated. Thus, the geometry does not represent a pipe because of the missing axis-symmetry. The mesh density was set-up arbitrarily based on Tobias Holzmann's feeling. Furthermore, no layers were generated nor a investigation into the viscous sublayer was performed.

The Numerical Set-up

The set-up of the numerical simulation is as follows. Holzmann CFD's performed four simulations. The first simulation was a transient one with a rising velocity profile (linear increase with respect to the time). As the velocity at the boundary increases, the time step of the simulation decreases based on the Courant number. Thus, the calculation time increases and therefore, after around 50 m/s, the simulation was stopped while the remaining points namely 67 m/s, 100 m/s and 240 m/s were performed within single cases. It is worth to mention that all calculations were performed in a transient mode while using the PIMPLE algorithm with a Courant number of 15. The PIMPLE algorithm is explained in the book of Tobias Holzmann named: *Mathematics, Numerics, Derivations and OpenFOAM®*.

The turbulence model was set to kOmegaSST. The fluid is water (incompressible) and thus the incompressible solver `pimpleFoam` was used.

Pressure Value Extraction

During the simulation, the pressure value of the Pitot tube at two patches were evaluated using function objects. At the whole patch of interest, the `areaAverage` function was used to calculate the pressure. As the Pitot tube's design evaluates the static pressure as well as the total pressure (kinematic pressure + static pressure), the pressure difference can be calculated afterwards.

The analytic relation between the velocity at the inlet and the pressures is given by:

$$p_{\text{total}} - p_{\text{static}} = \frac{1}{2}\rho U^2, \quad (0.1)$$

and is related to the Bernoulli equation. Thus, we can compare the analytical pressure difference with the numerical calculated ones.

Results

The quantitative results are given in the next two figures. The analysis was done as follows:

1. Extract the pressure values at the given patches of the CFD analysis at a corresponding velocity at the inlet,
2. Calculate the numerical pressure difference Δp
3. Plot the data; velocity vs. pressure difference

Doing the analysis steps given above, the labeled *OpenFOAM* curve (red) and points are achieved. Furthermore, the analytic result of equation (1) is presented in green. Analysing the graphics it is obvious that there is a discrepancy between the numerical and analytical curves.

The difference is related to the geometry and boundary conditions in use. As the Pitot tube is placed inside a closed channel, the velocity around the Pitot tube is higher than at the inlet

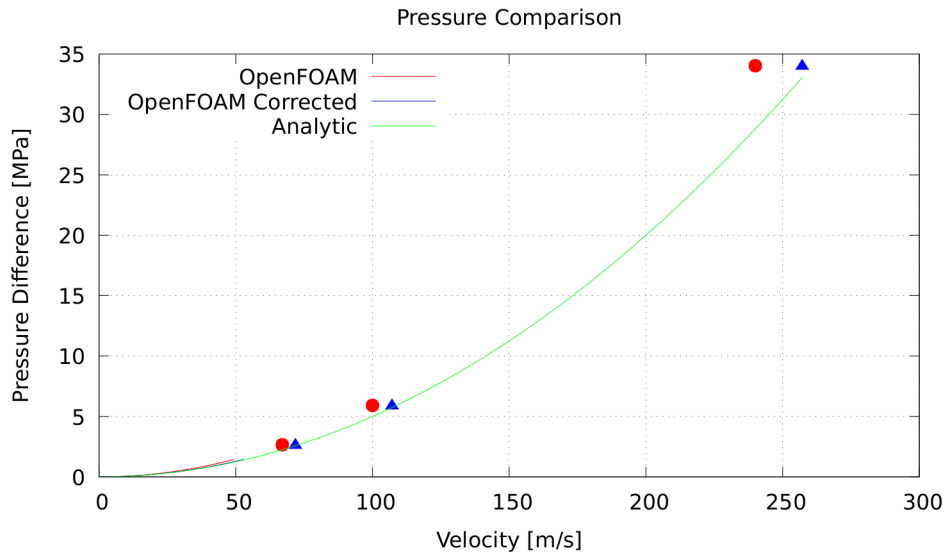


Figure 0.1: Comparison between the numerical and analytical results (1)

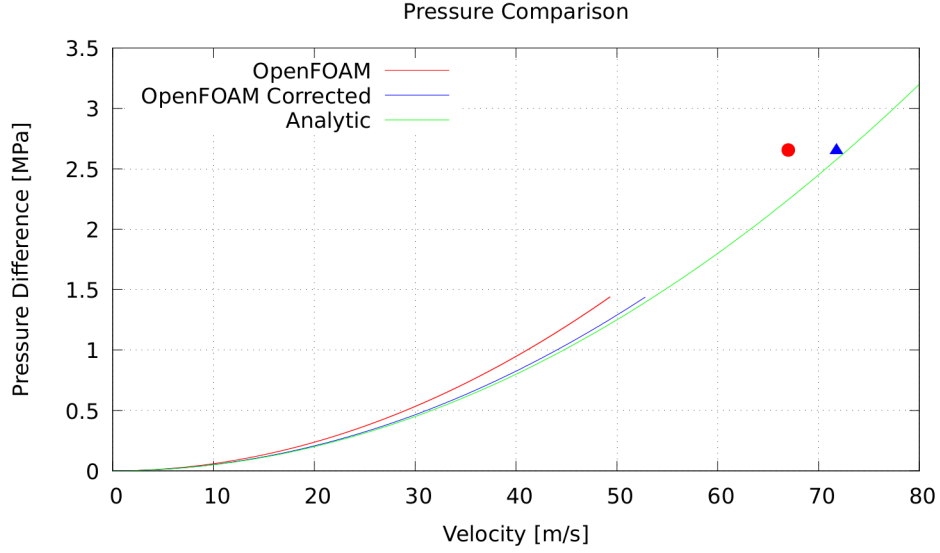


Figure 0.2: Comparison between the numerical and analytical results (2)

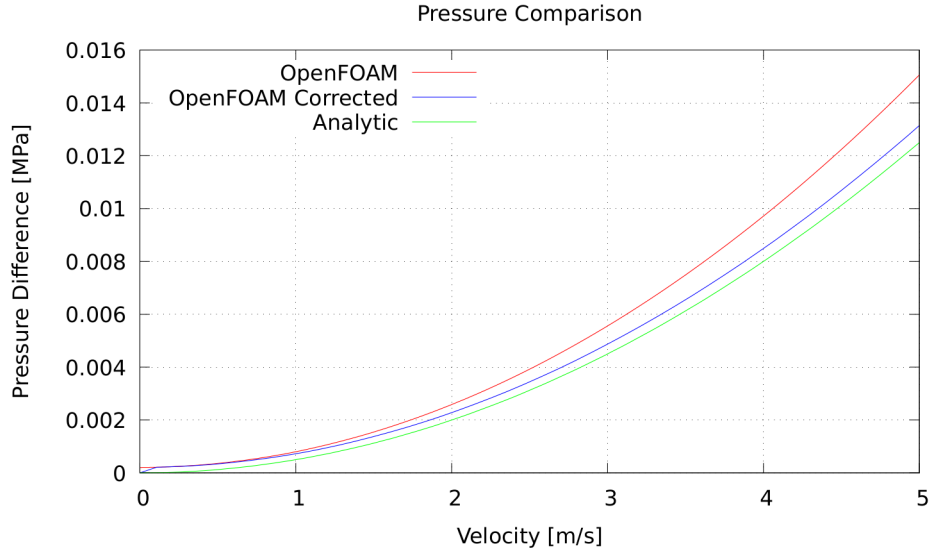


Figure 0.3: Comparison between the numerical and analytical results (3)

(due to the fact that the Pitot tube reduces the cross section; cf. 2D pictures given in the next pages).

Therefore, the velocity at the cross section was evaluated in addition. While plotting the pressure differences with respect to the new evaluated velocity, the *OpenFOAM Corrected* (blue) data are achieved. Thus, the accuracy of the data increases and the analytical and numerical results are almost identical in the plots. Nevertheless, comparing the quantitative data, a discrepancy of around 18 % (for the red) and 3 % (for the blue) within the high velocity range can be observed.

However, for small velocities there is a larger discrepancy. This might be related to the set-up used for the simulation. As already mentioned before, the velocity analysis between 0 ms^{-1} and 50 ms^{-1} was done in a transient (single) simulation, while the inlet velocity profile was increased with respect to the time. The two quantitative values for 0.1 ms^{-1} and 1 ms^{-1} for the transient simulation is given in the table at the bottom (last two values marked **red**). As the differences

are very high, these points were re-investigated with single calculation cases.

Thus, in the tables, the single calculation analysis are presented in the top of the table. Analysing the data, it follows:

- For low velocities, the discrepancy between the analytical and numerical result is larger compared to higher velocities.
- Using the transient increasing velocity profile, the quantities do have a much larger discrepancy compared to the analytical data (marked in red) and the single analysis cases.

Further investigations to that particular topic were not performed. However, a first guess would be a non-converged result, to high Courant number, or maybe the wrong velocity; **The velocity profile at the cross section was not analyzed for that cases.**

Another interesting fact is the convergence behavior of the algorithm itself. During low velocities the PIMPLE algorithm took around 40 to 60 outer corrections while these correction steps reduce with increasing velocity values (around 14 outer loops at 50 ms⁻¹).

The following table summarizes the quantitative results of the five points (the density was assumed to be 1000 kgm⁻³). The second section in the table refers to transient increased velocity profile simulation, while the values for 0.1 ms⁻¹ and 1 ms⁻¹ point were extracted. It is worth to mention that the corrected velocity profile do correspond to the single calculations. That means, that for the transient analysis (**red data**) the velocity at the cross section were not recalculated.

U_{inlet} ms ⁻¹	p_{total} Pa m ³ kg ⁻¹	p_{static} Pa m ³ kg ⁻¹	Δp_{FOAM} Pa	$\Delta p_{\text{Eqn(1)}}$ Pa	Difference %
0.1	1.00000564e3	0.999999306e3	6.33	5	26
1.0	1.00054682e3	0.999935358e3	611.42	500	22
67.0	3.35501067e3	6.99523019e2	2655487.65	2244500	18
100.0	6.24551626e3	332.879487e2	5912636.77	5000000	18
240.0	3.12071873e4	-2.81786943e3	34025056.73	28800000	18
0.1	1.00073685e3	1.00052301e3	213.84	5	4176
1.0	1.00125244e3	1.00045364e3	798.80	500	59

Analyzing the same data while having the increased velocities at the reduced cross section, it follows:

U_{inlet} ms ⁻¹	U_{cross} ms ⁻¹	p_{total} Pa m ³ kg ⁻¹	p_{static} Pa m ³ kg ⁻¹	Δp_{FOAM} Pa	$\Delta p_{\text{Eqn(1)}}$ Pa	Difference %
0.1	0.107	1.00000564e3	0.999999306e3	6.33	5.72	10
1.0	1.0715	1.00054682e3	0.999935358e3	611.42	574	6
67.0	71.780	3.35501067e3	6.99523019e2	2655487.65	2576184.2	3
100.0	107.146	6.24551626e3	332.879487e2	5912636.77	5740132.65	3
240.0	257.150	3.12071873e4	-2.81786943e3	34025056.73	28800000	3
0.1	0.107	1.00073685e3	1.00052301e3	213.84	5.72	3638
1.0	1.0715	1.00125244e3	1.00045364e3	798.80	574	39

The quantitative data shows the big discrepancy of the numerical and analytical results for small velocities while using a transient solver with Co = 15 and an time depended velocity inlet; as already mentioned twice, this might be due to the wrong used velocity. For higher velocities,

the error is around 3 %, if the correct velocity value is used. Summing up: The single transient calculations result in a quantitative good agreement to the analytical solution. The transient run from 0 ms^{-1} to 50 ms^{-1} result in less accuracy; probably related to wrong velocities.

Finally, pictures of the geometry at different velocity points are given on the next pages.

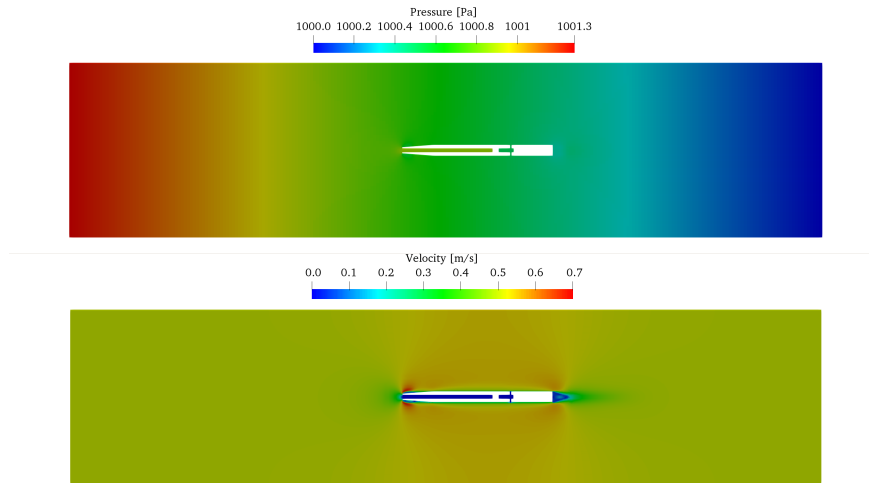


Figure 0.4: Velocity and pressure plots at 0.5 ms^{-1} .

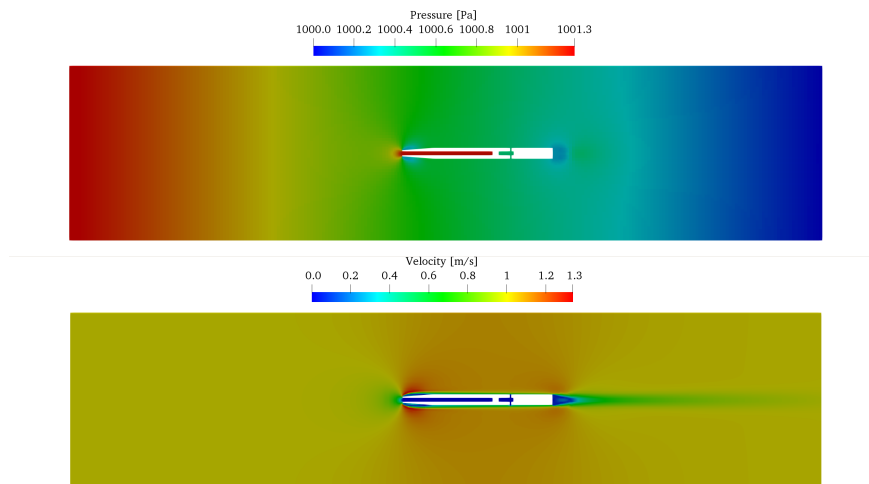


Figure 0.5: Velocity and pressure plots at 1 ms^{-1} .

Epilogue

The project was done without investigating to much time. However, I hope you enjoyed the small report and the results that were achieved here. Looking forward to the achieved results from others and maybe some fancy two-phase calculations.

Keep foaming. Tobias Holzmann

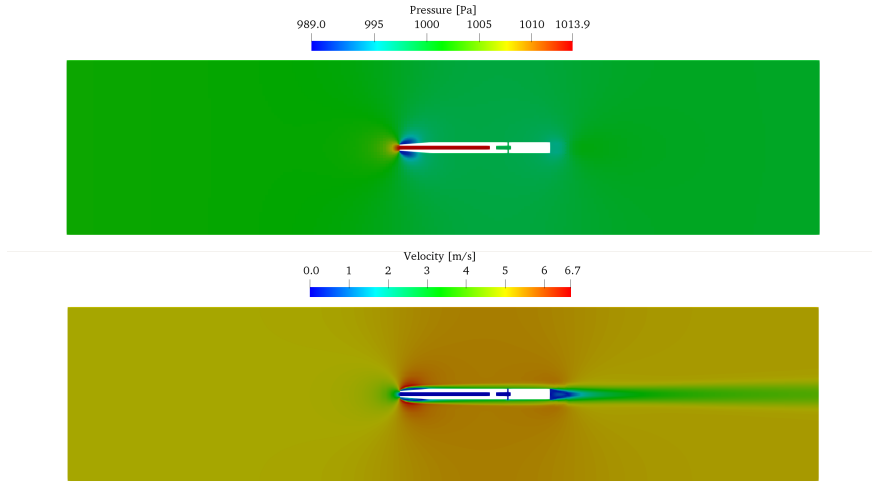


Figure 0.6: Velocity and pressure plots at 5 ms^{-1} .

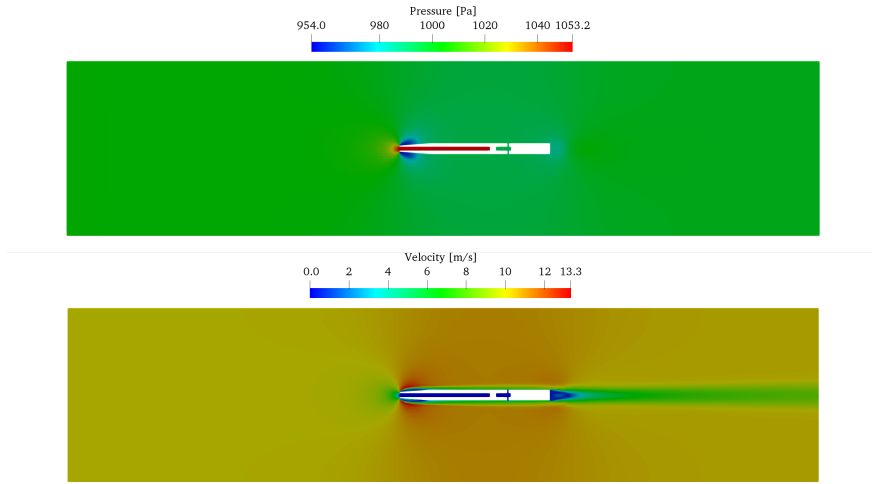


Figure 0.7: Velocity and pressure plots at 10 ms^{-1} .

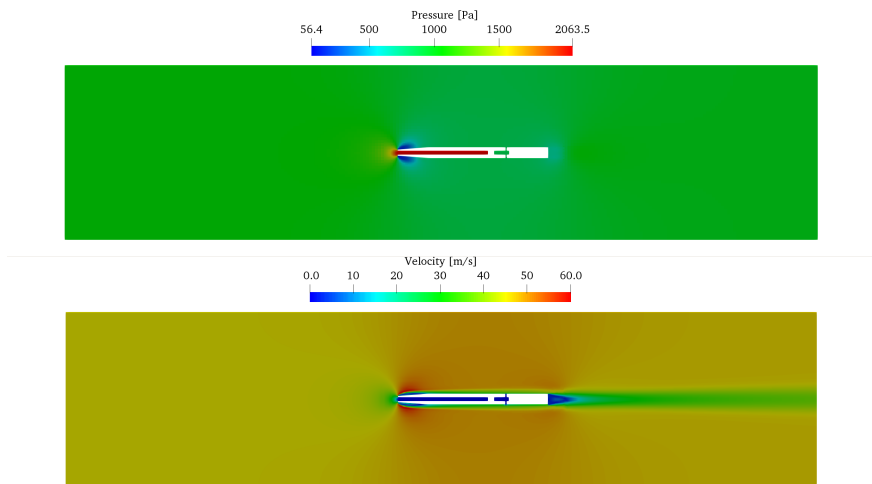


Figure 0.8: Velocity and pressure plots at 45 ms^{-1} .

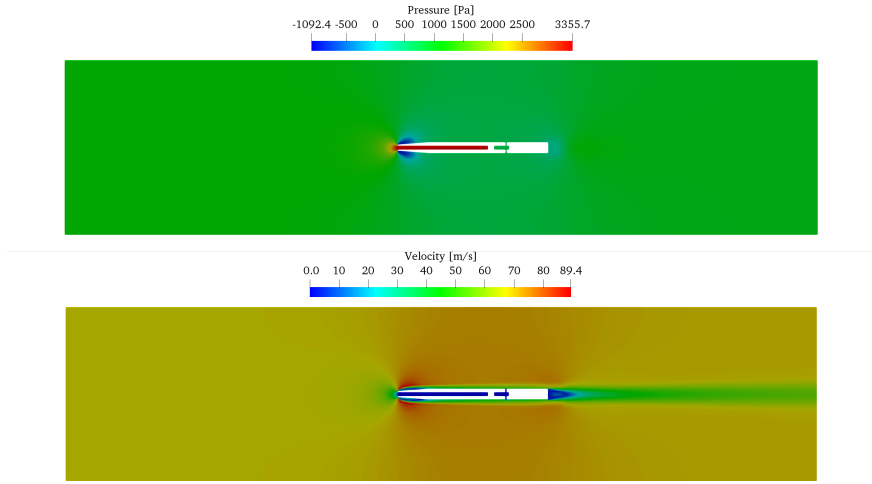


Figure 0.9: Velocity and pressure plots at 67 ms^{-1} .

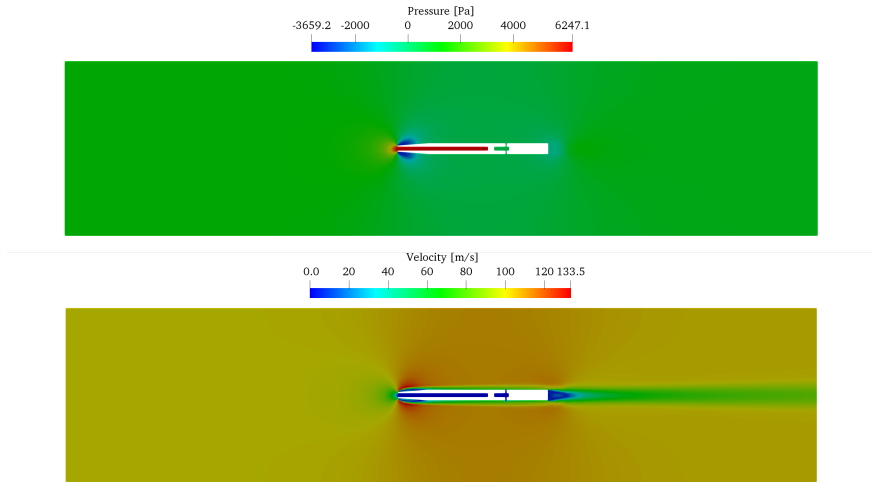


Figure 0.10: Velocity and pressure plots at 100 ms^{-1} .

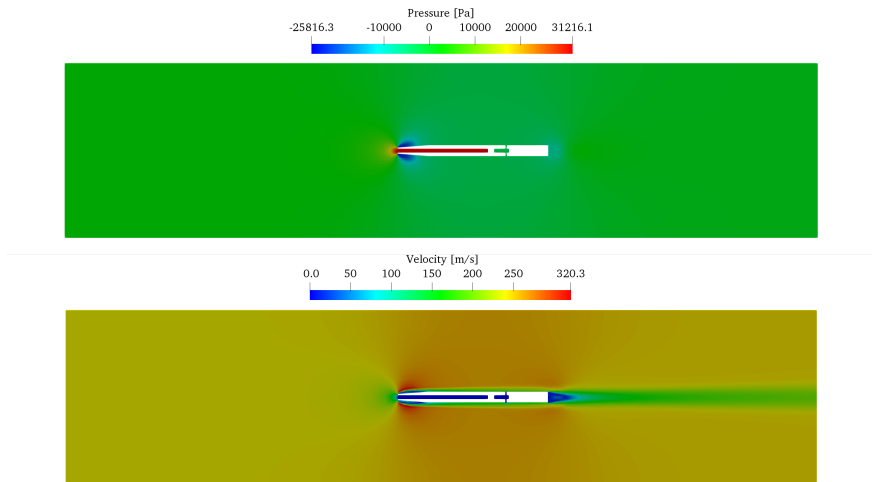


Figure 0.11: Velocity and pressure plots at 240 ms^{-1} .