



Application of open-source CFD to engineering problems

CFD Analysis: Incompressible Injector Pipe

Authors:

Cristian Asensio García
Alexis Leon Delgado
Santiago Villarroya Calavia

Professor:

Dr. Robert Castilla

Contents

1	Introduction	1
2	Simulation	1
2.1	Description of mesh	1
2.2	Boundary conditions	1
2.3	Models used	2
3	Results	3
3.1	Calculation of flow rate	3
3.2	Pressure	4
3.3	Streamlines	5
4	Conclusions	5

List of Figures

1	Mesh created with SnappyHexMesh	1
2	Velocity integrated distribution for different patches at $t = 1000s$	4
3	Pressure profiles obtained with ParaView for different times.	4
4	Streamlines obtained with ParaView for different times.	5

List of Tables

1	Snapped mesh data	1
2	Inlet 1 integrated values	3
3	Inlet 2 integrated values	3
4	Outlet integrated values	3

1 Introduction

This assignment focuses on the study of incompressible flow inside an injector pump by means of OpenFoam software. The geometry of study consists of an Y pipe formed by two inlets and one outlet. The entering flow will mix together under turbulent conditions, and its behaviour regarding velocity and pressure will be analysed.

2 Simulation

2.1 Description of mesh

As previously explained, the geometry of study is a Y pipe. Since the tube is a relatively complex geometry to mesh, the SnappyHexMesh mesher has been used in order to obtain a hexdominant mesh.

Briefly, three main steps have been followed in order to create the mesh. Firstly, the background mesh has been prepared using *blockMesh*, obviously having previously verified that the bounding box of the geometry, in our case $(-3.88578e-16 \ -2.5 \ -0.5)(8 \ 7.5 \ 0.5)$, fits inside the base mesh of size $(-1 \ -3 \ -1)(9 \ 8 \ 1)$. Then, the Castellated mesh has been obtained by removing the unused cells positioned outside the pipe's internal flow. Once completed this step, the final snapped mesh (showed in Figure 1) has been obtained and checked. As a summary, Table 1 shows the main characteristics of the snapped mesh used, being all cells classified as refinement level 2.

Cells	Faces	Points
48763	148963	52310

Table 1: Snapped mesh data

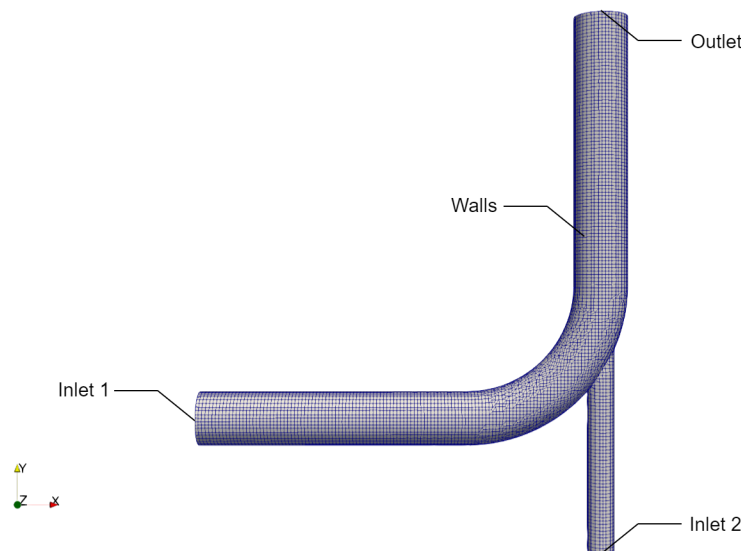


Figure 1: Mesh created with SnappyHexMesh

2.2 Boundary conditions

Regarding the boundary conditions for that case of study, it is necessary to consider and understand how the flow inlets and outlets of an injector pump work. In this case, the geometry of the pipe has two inlets and one outlet but with different conditions. As it can be observed in the attached .zip with the case, these sections have been named inlet1, inlet2 and outlet, respectively. Therefore, the pressure and velocity initial conditions defined for each of these regions are the following.

- **Inlet 1:** It is no more considered as an inlet velocity, hence the total pressure (P) value is fixed but the velocity is unknown. Therefore, the initial conditions for velocity and pressure are defined such that:
 - Velocity initial condition:
 - * Type: *pressureInletOutletVelocity*;
 - * Value: *NoUniform List Vector*;
 - Pressure initial condition:
 - * Type: *totalPressure*;
 - * Value: *uniform 0*;
- **Inlet 2 :** This inlet defines the case's main velocity, which is given by the value of our group number (1m/s). The direction of this velocity vector must be in the Y coordinate. Consequently, the value and direction of the velocity vector is defined as an input fixed value of the case and the pressure distribution is unknown. Despite not being able to know the pressure value as initial condition, it is known for sure that the pressure distribution should have a gradient equal to 0 at time 0.
 - Velocity initial condition:
 - * Type: *fixedValue*;
 - * Value: *uniform (0, 1, 0)*;
 - Pressure initial condition:
 - * Type: *zeroGradient*;
- **Outlet:** This patch has a behaviour much more similar to that showed in Inlet 1 since the pressure in it must be known and equal to the external pressure. Then, after running the case it will be possible to know the velocity profile, which is the main goal of the study.
 - Velocity initial condition:
 - * Type: *pressureInletOutletVelocity*;
 - * Value: *Nouniform List Vector*;
 - Pressure initial condition:
 - * Type: *fixedValue*;
 - * Value: *Uniform 0*;
- **Walls:** For the walls, which are the surfaces that are going to define the domain inside the injector, the boundary conditions are defined according the relative behaviour of the fluid with respect to these surfaces. A condition of *no-slip* is set for the velocity and a *zeroGradient* for the pressure.
 - Velocity initial condition:
 - * Type: *noslip*;
 - Pressure initial condition:
 - * Type: *zeroGradient*;

2.3 Models used

Before running the simulation, the turbulence model shall be determined because, although a laminar model could be used, it does not predict the behaviour of the fluid in turbulent regime. Therefore, on the “momentumTransport” file, the simulation type is defined as “RAS: Reynolds Average Simulation” and not “Laminar”. This simulation type can work properly with incompressible flows, such as the one studied in this simulation.

However, in order to get proper initial conditions values, which will be defined as internal initial field vectors in the U folder, the *PotentialFoam* solver might be executed before running the *simpleFoam* with the turbulence model enabled.

The model chosen for the “RAS” simulation type is the “kOmegaSST”, a two-equation model from the $k-\omega$ family. These models are robust and easy to integrate, they perform better for flows with weak adverse

pressure gradient [1] and have improved accuracy for internal flows [2]. In this case the “SST: Shear Stress Transport” type model is chosen, which has an improved adverse pressure gradient performance.

As additional information, the option of solving the case dividing the simulation in two parts and using a laminar model in the first 500 steps and a turbulent model in the last 500 has been evaluated but dismissed. Although this would allow to get a faster convergence of the simulation results, these results would not be totally accurate due to the fact that the first 500 steps of the simulation would not be simulated considering the turbulence effects.

3 Results

Once the simulation has been executed, it is highly important to analyse and study the results obtained. This process is carried out by using some of the paraFoam tools presented hereunder.

3.1 Calculation of flow rate

For calculating the flow rate through both inlets and the outlet of the injector pump, the integrate variables command from paraFoam is used. Thanks to this tool it is possible to compute the magnitude and direction of the flow rate for each one of the surfaces of interest inside the injector. Additionally, the integral plots of the velocity magnitude have been obtained and presented in Figure 2.

Is is important to remark that all the values and plots represent the integrated values of the velocity and pressure magnitudes at the end time of the simulation (t= 1000 s).

- **Inlet 1:**

The values obtained for the flow rate, and the pressure integral all over the inlet 1 surface are:

Area (m^2)	Flow rate ($\frac{m^3}{s}$)	Static Pressure $\frac{p}{\rho}$ ($\frac{m^4}{s^2}$)
0.7799	(0.1493,0,0)	-0.0144

Table 2: Inlet 1 integrated values

- **Inlet 2:**

For the inlet 2, where the initial boundary condition is set as 1 m/s as inlet constant speed, the following integrated values are obtained:

Area (m^2)	Flow rate ($\frac{m^3}{s}$)	Static Pressure $\frac{p}{\rho}$ ($\frac{m^4}{s^2}$)
0.1885	(0,0.1885,0)	0.0144

Table 3: Inlet 2 integrated values

- **Outlet:**

Regarding the surface that has been defined as the injector outlet, the pressure integral and the total flow rate are:

Area (m^2)	Flow rate ($\frac{m^3}{s}$)	Static Pressure $\frac{p}{\rho}$ ($\frac{m^4}{s^2}$)
0.7799	(-0.0061, 0.3379 , -0.0011)	0

Table 4: Outlet integrated values

With this integrated data, the conservation of mass and total pressure will be verified in the Conclusions section 4.

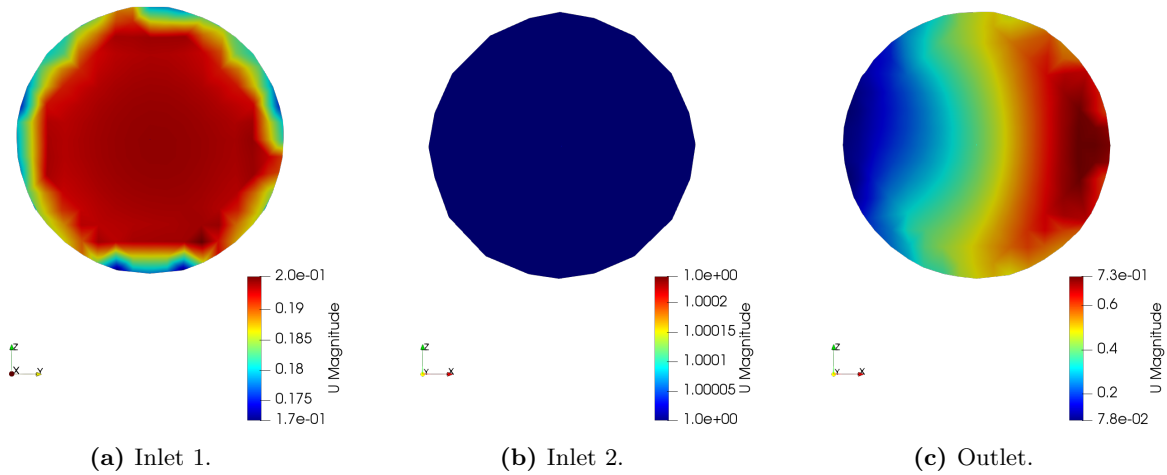


Figure 2: Velocity integrated distribution for different patches at $t = 1000s$.

3.2 Pressure

Regarding the visualisation of the pressure profile on the inlets and outlets, the following three figures represent the pressure plots for the times $t = 0 s$, $t = 500 s$ and $t = 1000 s$, respectively, performed with ParaView.

In Figure 3a, it can be observed that initially the pressure is the same on the whole domain with a value of $0 m^2/s^2$, since no iteration has been performed. As the time goes by, a pressure difference is generated achieving a pressure maximum at the inlet on the lower right corner, the Inlet 2, while the pressure on the outlet is fixed at $0 m^2/s^2$. These results are stabilised at the time $t = 500 s$, and as a result, the pressure profile for $t = 500 s$ (Figure 3b) and $t = 1000 s$ (Figure 3c) are nearly identical.

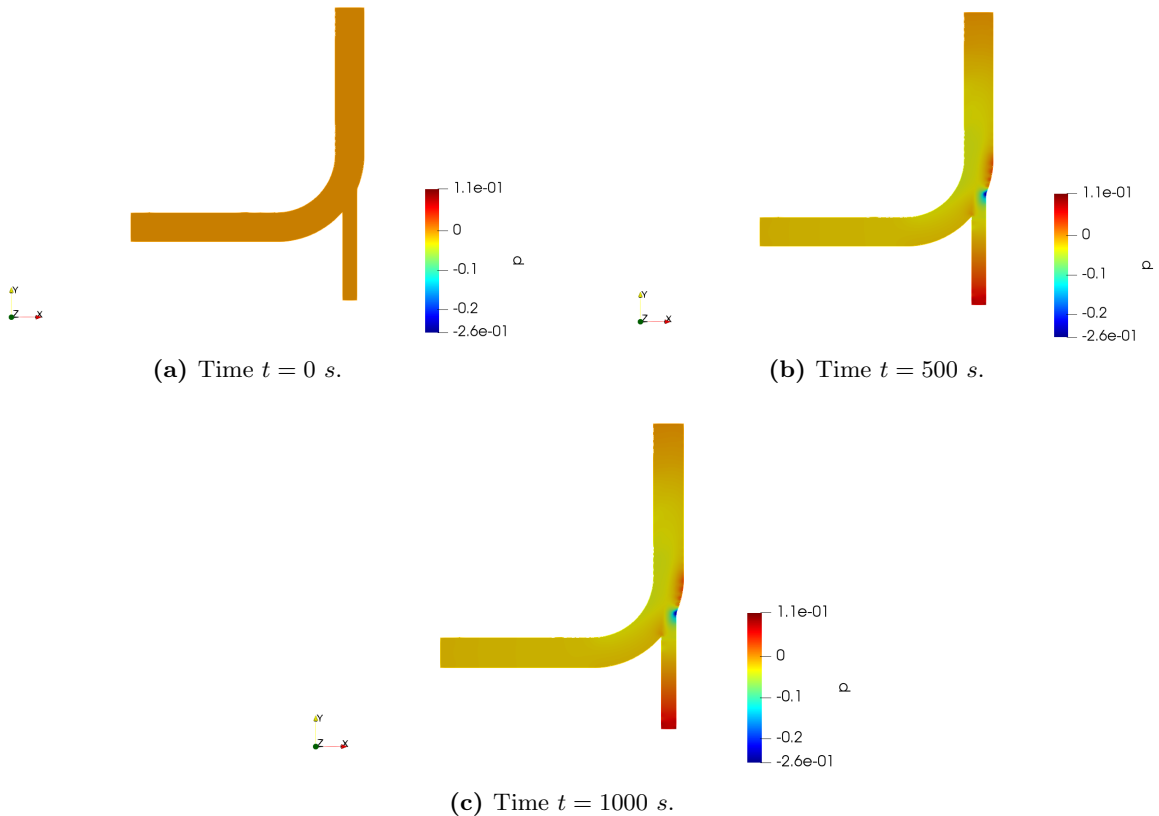


Figure 3: Pressure profiles obtained with ParaView for different times.

3.3 Streamlines

In order to illustrate the streamlines, it has been used the Stream Tracer tool of ParaView with the Tube filter enabled. The results are presented in Figure 4 for the times $t = 0$ s, $t = 500$ s and $t = 1000$ s with streamlines of 0.01 m of radius.

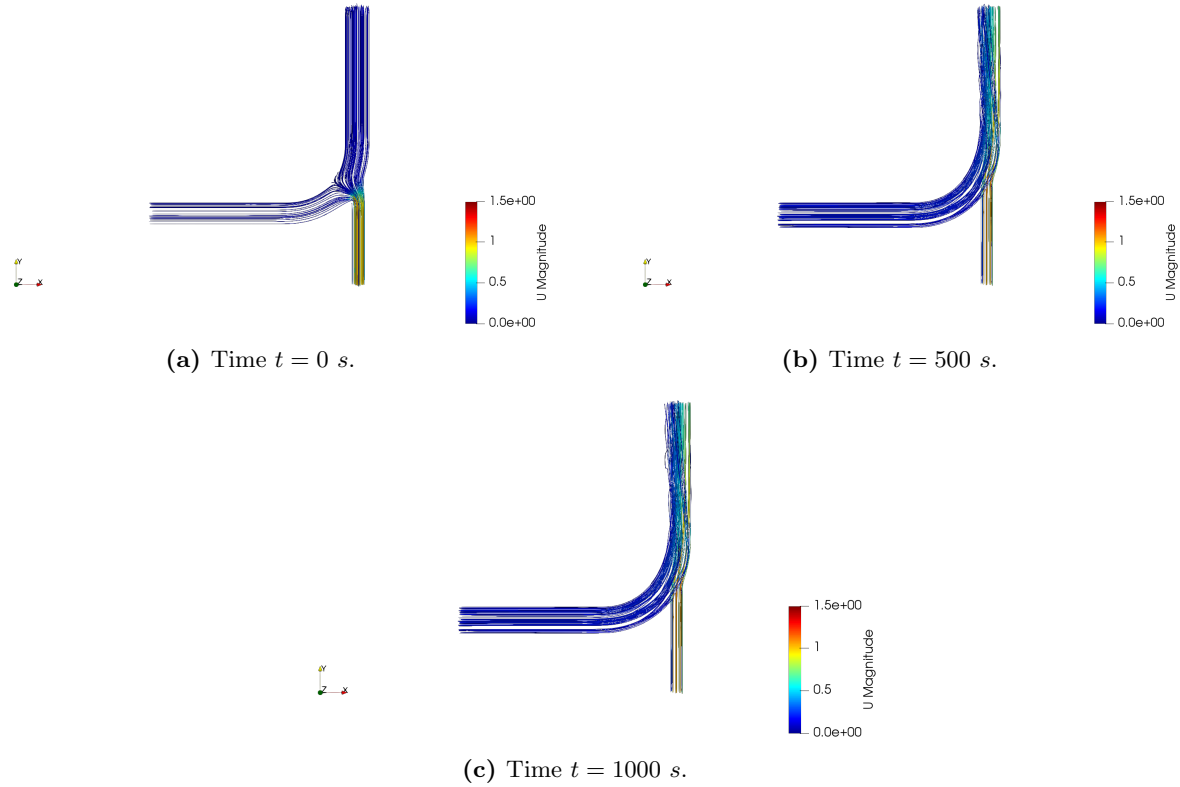


Figure 4: Streamlines obtained with ParaView for different times.

On the initial time, Figure 4a shows a high density of streamlines in the inlet2 tube, which translates into a velocity gradient between the lower inlet 2 and the tube intersection.

For the time $t = 500$ s, represented in Figure 4b, it is seen that the entering flow through inlet 2 causes some irregularities in the leaving streamlines through the outlet. This turbulence phenomenon is clearly observed since the streamlines near the top-right wall have velocities in the range $\sim 0.5 - 1$ m/s (green-yellow in the colour map), whereas the ones near the top-left wall have low velocities around $\sim 0 - 0.5$ m/s (blue in the colour map).

Finally, the velocity streamlines for $t = 1000$ s are observed in Figure 4c. Since stability has been previously reached, the results are practically identical to the ones analysed in $t = 500$ s.

Another interesting phenomenon is the fact that pressure minimum, represented in blue in Figure 3c, is reached at the right intersection between the inlet 2 tube and the outlet tube, whereas the maximum is obtained at the inlet 2. In terms of velocity streamlines, it can be observed that the entering flow through the inlet 2 must drastically reduce its velocity when mixed with the inlet 1 flow, which causes a high pressure gradient that leads to a maximum pressure in the inlet 2.

4 Conclusions

In the light of all the results presented above, some interesting conclusions are reached.

Regarding the flow rate values obtained, it is easy to check that the sum of the flow rates through inlet 1 and inlet 2 is equal to the flow rate value at the outlet. This is mainly due to the fact that the mass conservation equation must be respected. This means that the total amount of mass fluid entering the

pipe (inlets) must be identical to the total amount leaving it (outlet).

Therefore, considering the flow rates of each surface we have:

- Inlet 1: $\dot{V}_{\text{inlet1}} = 0.1493 \frac{m^3}{s}$
- Inlet 2: $\dot{V}_{\text{inlet2}} = 0.1885 \frac{m^3}{s}$
- Outlet: $\dot{V}_{\text{outlet}} = \sqrt{(-0.0061)^2 + 0.3379^2 + (-0.0011)^2} = 0.338 \frac{m^3}{s}$

Thus, mass conservation is verified:

$$\dot{V}_{\text{inlet1}} + \dot{V}_{\text{inlet2}} - \dot{V}_{\text{outlet}} = 0.1885 + 0.1493 - 0.338 = 0$$

Finally, another interesting approach is the study of the pressure conditions in the inlet 1 patch. As mentioned, the inlet 1 is set with a total pressure equal to 0 as boundary condition. Therefore, the combination of the static pressure and the kinetic energy of the flow rate through its surface must be equal to 0. With the values presented in the results section, it can be checked if the integrated values of flow rate and static pressure allow obtaining a total pressure equal to 0 in inlet 1.

It must be remarked that specific values are used, which means that all the pressure values are divided by the fluid density. Hence, the total density is defined as:

$$P = \frac{U^2}{2} + p$$

For the inlet 1 the values for each term of the equation above are:

- P: Total pressure = 0
- U: Total average speed = Flow rate / Area = $0.1493/0.7799 = 0.1916 \frac{m}{s}$
- p: Static pressure = Integrated value / Area = $-0.0144/0.7799 = -0.0184 \frac{m^2}{s^2}$

Therefore:

$$P = \frac{0.1916^2}{2} - 0.0184 = 0$$

Newly, the results obtained make sense due to the fact that the total pressure boundary condition is respected during all the simulation.

Bearing in mind all the performed analysis, it can be considered that the simulation results are validated.

References

- [1] Moukalled, F., Mangani, L., & Darwish, M. (2016). Turbulence modeling. *The finite volume method in computational fluid dynamics: An advanced introduction with openfoam® and matlab* (pp. 693–744). Springer International Publishing. https://doi.org/10.1007/978-3-319-16874-6_17
- [2] Sen, B., Yuksel, U., & Kirkkopru, K. (2004). Comparison of turbulence models for an internal flow with side wall mass injection, 31.