Université de Strasbourg

UFR de Mathématiques et Informatique

Calcul Scientifique et Mathématique de l'Information

Project

Laminar, Isothermal Backward Facing Step Benchmark

Présenté par:
 Lilya Nesrine **FEZANI**
 Antoine Schuffenecker
 Bryan Masson
 Mohamado Faye

22 May 2020

Contents

1   Presentation of feel-pp

Feel++ is environment which allows users to modelize and resolve different physics problems in the case of scientific studies or in more factual situations of engineering. This tool can solve complex cases by starting with monophysic and all the way to Multiphysics problems.

The use of these tools does not require any profound knowledge nor a complete one of the equations and the theory behind all the calculus. But a solid knowledge of the situation and physics parameters which allow users to adapt the simulation to their particular case. With all this, and given the right simulation is chosen, the simulation done will be realistic and will illustrate as best as one can, the situation.

To add to diversity of the equations available, the users can use different method to obtain the results. By default, the method uses the finite element method with a lot of numerical solvers but, if required, some more advanced methods are implemented and can be use as well.

2 Presentation of the project: contribute to the feel-pp documentation

The project which was given to our group is to complete the documentation on several cases on the feel-pp website. The documentation is mainly about the fluid flows in given geometries and the effort reactions on particular solid. The documentation is enhanced by simulations where users can change some physicals parameters. This provides a large choice of possibilities that can be created by the same equations. The main methodology behind any resolution is:

1. Definition of a geometry and a mesh which change the problem to a discrete one as well as the domain and the numerical method used.

2. The actual solving of the problem.

3. The exploitation of the results.

2.1 Fluid :

The fluid part allows to modelize a bunch of situations in 2d or 3d:

1. Stokes flow

2. Laminar incompressible flow

3. Steady and unsteady simulations

4. Bdf time schemes up to order 4

5. Moving domain support using Arbitrary Lagrangian Eulerian (ALE) formulation

6. Support high order geometry including in context of moving domain

7. Stabilization pressure and advection dominated using Galerking Least Square

8. Boundary conditions: no-slip, slip(symmetry), inflow (not necessarily aligned with an axis), pressure, outflow

2.2 Solid :

A solid is, in opposition to a fluid (gaz or liquid), a substance which cannot flow and, in a way, has a high resistance to deformation. It can be defined as well as an entity which has an integrity. The solid part allows to modelize the effects on effort on solids in 2d or 3d. In feel++ the main interest is to study the axisymmetric reduced model. Every solid has a different geometry and different physical parameters by changing them anyone can simulate with any material and can see how they will react with given efforts. The most common pair of parameters which characterize the elasticity of a solid are the Poisson coefficient and the Young-Modulus.

3 Presentation of the problem

3.1 Problem

More precisely this project consists in carrying out a simulation of internal, laminar, incompressible flow in a channel on a backward facing stage. Flow separation is a common and interesting phenomenon in fluid mechanics with significant effects in practical application. Among them the backward-facing step flow which represents a very popular reference and validation test for computational fluid dynamics (CFD) simulations due to the availability of a good number of good experimental data.

3.2 Simulation Model

Among the representative models of separation flows is **the backward-Facing Step Flow** (BFS), which can be seen in :

1. Aerodynamic flows

2. Engine flows

3. Engine flows

4. Heat transfer systems, and even

5. he flow around buildings

The BFS is a very popular reference and validation model for Computational Fluid Dynamics (CFD) simulations because of the availability of a good number of experimental data. Flow separation depends on several parameters such as BFS geometry, inlet and outlet conditions, turbulent intensity, as well as heat transfer conditions. Although the geometry is very simple, the flow may have interesting separation regions, which also makes it an ideal candidate for testing numerical boundary conditions.

3.3 Objectives

the different objectives which lead the project are :

1. Test the coupling between pressure field and flow velocity

2. Compare the results of the velocity profile with numerical results from the literature

3. Prescribe input condition data from a file.

For CFD, we find that it supports both the incompressible Navier-Stokes equations and the Stokes equations which will be defined in the following section.

4 The incompressible equations of Navier-Stokes

4.1 Definition

The Navier-Stokes equations are a system of nonlinear partial differential equations which describe the movement of fluids in a continuous medium, it is demonstrated from a balance of momentum per unit volume for an incompressible fluid it is given in the form:

$$\overrightarrow{\nabla}.\overrightarrow{v} = 0 \tag{1}$$

$$\rho\frac{\partial \overrightarrow{v}}{\partial t} + \rho(\overrightarrow{v}.\overrightarrow{\nabla})\overrightarrow{v} = -\overrightarrow{\nabla}p + \rho\overrightarrow{f}_{ext} + \mu\Delta\overrightarrow{v} \tag{2}$$

The first equation (1) is zero divergence equation of the velocity field $\overrightarrow{v}(\overrightarrow{r},t)$, it ensures the incompressibility of the fluid. Each term of the second equation (2) represents a force per unit of volume such as:

1. ρ: fluid density

2. $p(\overrightarrow{r},t)$: pressure, it is the isotropic part of the stress tensor.

3. $-\overrightarrow{\nabla}p$: normal stresses related to pressure forces

4. $\overrightarrow{f}_{ext}(\overrightarrow{r},t)$: force per unit mass

5. p_0: the value taken by the pressure in the absence of flow

when we divide the balance equation (2) of forces per unit volume ρ, we get a balance of forces per unit mass, which can also be interpreted as a velocity transport equation:

$$\frac{\partial \overrightarrow{v}}{\partial t} + (\overrightarrow{v}.\overrightarrow{\nabla})\overrightarrow{v} = -\frac{1}{\rho}\overrightarrow{\nabla}(p - p_0) + v\Delta\overrightarrow{v} \tag{3}$$

Such as, $v = \frac{\mu}{\rho}$ represents the kinematic viscosity of the fluid end $v\nabla\overrightarrow{v}$ a linear term represents the movement quantity transport.

4.2 Reynolds number

We will write the Navier-Stokes equation using dimensionless combinations (which will be noted by premiums) of the different sizes involved. Let L and U be the scales respective size and flow velocity, we have:

$$\overrightarrow{r} = L\overrightarrow{r}'; \overrightarrow{v} = L\overrightarrow{v}'; p - p_0 = (\rho U^2)p'; t = \frac{L}{U}t'$$

So the Navier-Stokes equation becomes:

$$\frac{\partial \overrightarrow{v}'}{\partial t'} + (\overrightarrow{v}'.\overrightarrow{\nabla}')\overrightarrow{v}' = -\overrightarrow{\nabla}'p' + \frac{1}{Re}\Delta'\overrightarrow{v}' \tag{4}$$

5

$$Re = \frac{UL}{v} \tag{5}$$

There appears a dimensionless number, Re which is a combination of L, U and v: the number of Reynolds. It weighs the term viscous diffusion against the other terms in the equation. We recognize

the two characteristic times necessary to transport the momentum over a length L by diffusion and by convection. Transportation will be the shortest time thus dominating the Reynolds number is the relationship between convective and diffusive effects:

$$Re = \frac{effects_{convective}}{effects_{diffssive}} \tag{6}$$

It is also often useful to understand the Reynolds number as the relationship between the terms of inertia and viscous forces of the Navier-Stokes (2) equation such as:

$$Re = \frac{inertia_{forces}}{viscous_{forces}} \tag{7}$$

5 Boundary condition

To solve a fluid mechanics problem, we use physical laws to find the mathematical equations which describe the physical properties of the fluid, such as velocity, temperature, pressure, density and viscosity. It is assumed that the fluid is incompressible, it means that the volume of the fluid cannot be reduced by an increase of pressure. Theses equations are called the governing equations of CFD:

- Continuity equation

Consider the fundamental principle of physics proposed by Antoine Lavoisier, which gives the intergral form of the mass conservation equation :

$$\frac{\partial}{\partial t} \int_{\Omega} \rho d\omega + \int_{\partial \Omega} \rho v \cdot \vec{n} d\sigma = 0 \qquad \forall v \in \mathbb{R}$$

where ρ (kg/m³) is the density in the domain Ω, v (m/s) is the velocity of the fluid and $\vec{n}$ is the unit normal vector to the boundary $\partial \Omega$.
Consider also Gauss' divergence theorem :

$$\int_{\Omega} \nabla \cdot v d\omega = \int_{\partial \Omega} v \cdot \vec{n} d\sigma \qquad \forall v \in \mathbb{R}$$

Using Gauss' divergence theorem in the mass conservation equation, we obtain the differential form, which is the continuity equation :

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0$$

- Momentum equation

Newton's second law states that the force of a moving object is equivalent to its rate of change of momentum. In fluid mechanics, the momentum theorem is :

$$\sum \vec{F}_{ext} = \int_{\Omega} \frac{\partial (\rho v)}{\partial t} d\omega + \int_{\partial \Omega} (\rho \vec{v}) \cdot (\vec{v} \cdot \vec{n}) d\sigma$$

and it was converted to differential form by the french mathematician Cauchy, with the application of the divergence theorem.
The result is the momentum equation :

$$\rho \left(\frac{\partial v}{\partial t} + v \cdot \nabla v \right) = -\nabla p + \mu \Delta v + \rho g$$

where p (Pa) is the pressure, μ is the dynamic viscosity and g is the external forces acting on the fluid, such as gravity.

- Energy equation

$$\frac{\partial (\rho h - p)}{\partial t} + \nabla (\rho v h) = \nabla \left((\mu + \frac{\mu_t}{\sigma_t}) \nabla h \right) - S_h$$

where $h = U + pV$ (J) is the enthalpy, U is the internal energy of the system, V is the volume, μ_t is the turbulence viscosity, σ_t is a constant and S_h is the volumetric heat source.

6 Presentation of the test case

6.1 Presentation

this study will be based on laminar flow around a step laid in a flat channel. The fluid is subjected to a sudden widening that causes an inverse pressure gradient, where the flow separates into several zones, among which a recirculation zone is formed, noted x_r, where the flow closes to return to the step. The Reynolds number denoted Re for this flow is calculated from the channel height S, the average flow velocity U_{ave} and the kinematic viscosity v, and is defined by:

$$Re = \frac{SU_{ave}}{v}$$

and as $v = \frac{\mu}{\rho}$ so :

$$Re = \frac{S\rho U_{ave}}{\mu}$$

When the flow has low Reynolds number values, it is said to be stationary, while flows with higher Reynolds number values become turbulent and the average length of the recirculation zone decreases until a constant saturation value is reached. In this project, we are only interested in two different Reynolds number values: $Re = 389$ and $Re = 1095$.

6.2 Geometry and data

The computational domain Ω is a channel with a descending step as shown in the figure below.
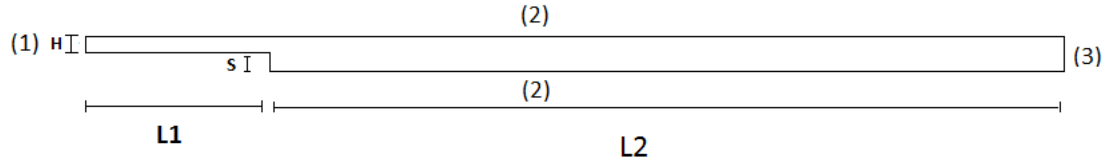

Figure 1: Calculation area.

The Data provided on the whole Ω domain allows us to have conditions at the specified limits, the tables below summarize these data.

Name	Description	Nominal Value	Units
L1	Length of the upstream section	2e-1	m
L2	Length of the downstream section	5e-2	m
S	Step height	4.9e-3	m
H	Inlet channel height	5.2e-3	m
U_{int}	Initial velocity	-	m/s
U_{ave}	Average velocity	-	m/s

Table 1: Geometric data

Name	Description	Nominal Value	Units
ρ	density	1.23	Kg/m^3
μ	dynamic viscosity	1.79e-5	$Pa.s$
v	kinematic viscosity	1.4553e-5	m^2/s

Table 2: Physical data

6.3 Boundary conditions

In this study 3 boundary conditions are imposed:

1. Inlet condition : On boundary (1) a Poiseuille profile is placed as an entry condition, it is defined by:

$$u = U_{int} = 6U_{ave}(\frac{y_1}{H})(1 - \frac{y_1}{H}) \tag{8}$$

such as:

$$y1 = y - S$$

and U_{ave} are derived from the selected Reynolds number as:

$$U_{ave} = \frac{vRe}{S} = \frac{\mu Re}{\rho S}$$

9

The profile of Poiseuille at the entrance is represented by the graphs below:
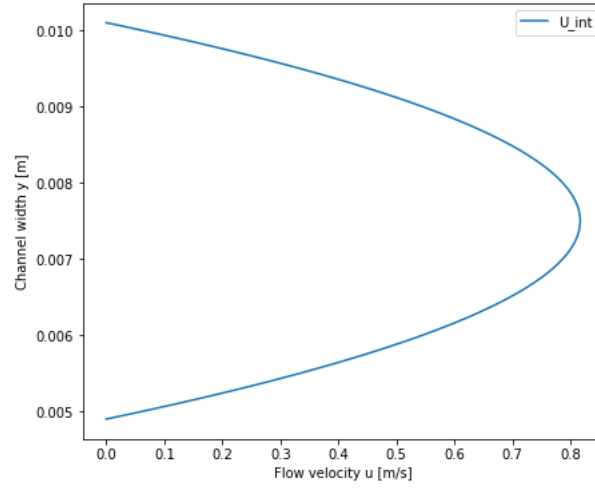
(a) For $Re = 389$:

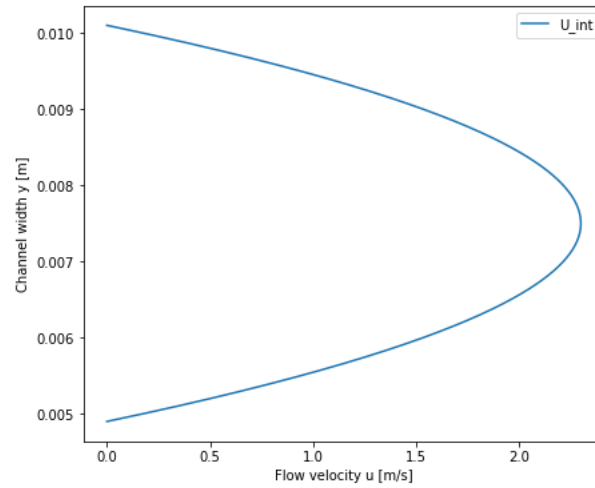Figure 2: Profile of Poiseuille at the entrance for Re=389

(b) For $Re = 1095$:

Figure 3: Profile of Poiseuille at the entrance for Re=1095

2. Wall condition : On the limits (2), i.e. on the upper and lower wall we have

$$u = 0 \qquad (9)$$

3. Outlet condition : On boundary (3) the exit boundary condition is free, which means that no constraint is imposed on the exit boundary.

6.4 Conformal blocks division

To study the laminar flow around a backward Facing Step we have devised the geometry [Figure 1] in conformal blocks as the figure below illustrates :
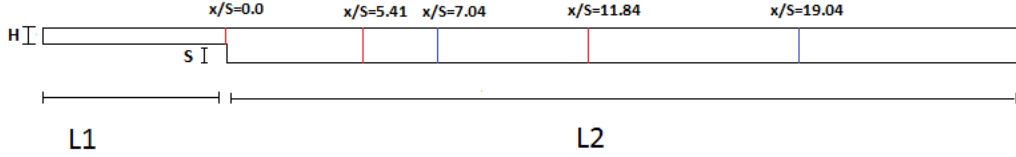

Figure 4: Splitting the Ω domain into conformal blocks

6.5 Mesh

After cutting our domain, we move on to meshing to generate fully structured uniform triangular cells, [Figure 5] shows a part of the generated mesh
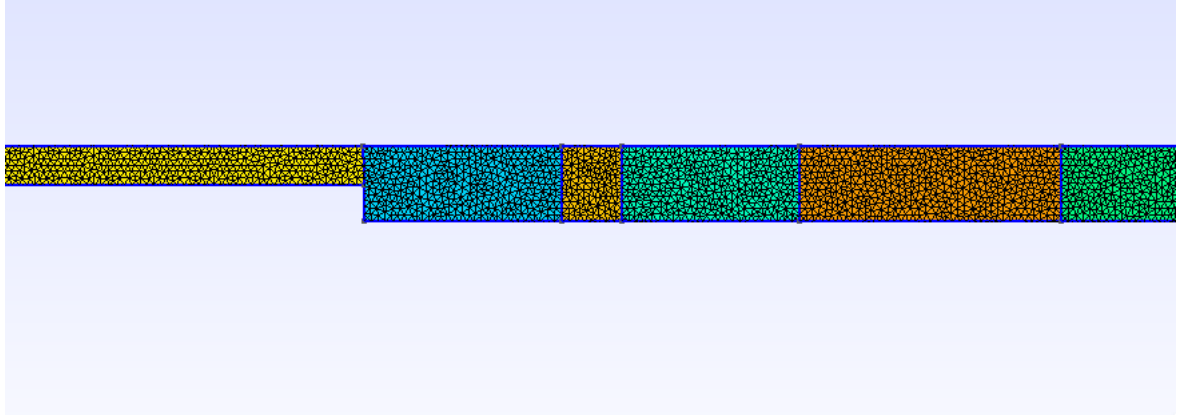

Figure 5: Generated mesh

The study will be based on velocity profiles at different downstream locations, represented by the different vertical lines x_r/S shown in [Figure 4] for $Re = 389$ and $Re = 1095$.

11

7 Results

For the readability of the results, we have separated the calculations for Re= 389 and those for Re=1095.

7.1 Root Mean Square Error(RMSE)

$$RMSE = \sqrt{\frac{\sum_{i=0}^{n}(\widehat{y_i} - y_i)^2}{n}}$$

The RMSE is an index that provides an indication of the dispersion or variability in the quality of the prediction. The RMSE can be related to the variance of the model.
Often, RMSE values are difficult to interpret because it is not possible to tell whether a variance value is low or high. To overcome this, it is more interesting to normalize the RMSE so that this indicator is expressed as a percentage of the mean value of the observations. This can be used to make the indicator more meaningful.

7.2 Results for Re=389

In this part the problem is dealt with in a stationary frame, i.e. the solution does not depend on the time, more precisely the flow profile is to long at the points where the flow variation is not visible.

```
mpirun -np 4 feelpp_toolbox_fluid--config-file laminar.cfg--fluid.snes-monitor=1
                              --fluid.gmsh.hsize=3e-4
```

Once the command is executed the results are exported to ParaView for viewing the flow by executing:

```
rsync -avz atlas.math.unistra.fr:~/feel\\
```

The solution obtained by ParaView is represented in the figure below,it is a final solution obtained in a stationary state:
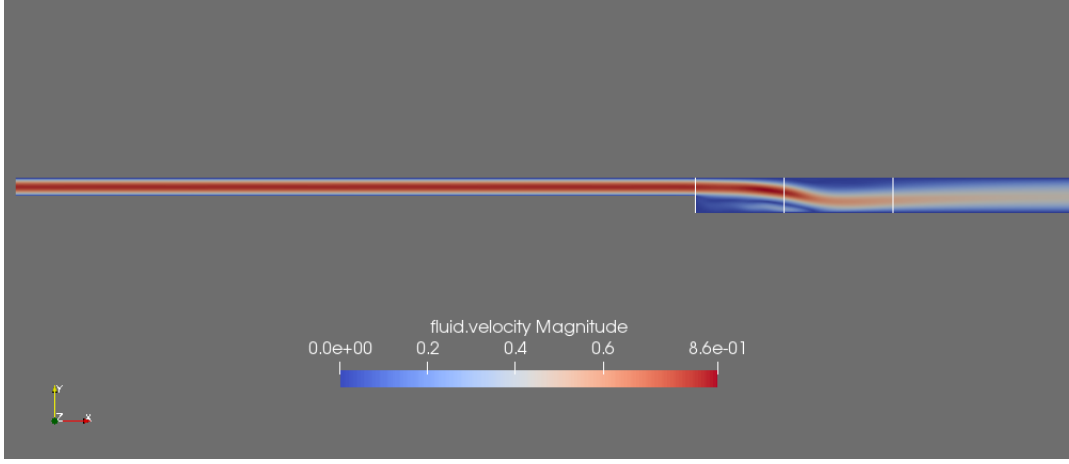
Figure 6: flow velocity profile visualized by ParaView

Now we are going to study the flow profile on the different vertical lines x_r/S, using the filter **PlotOverLine** which allows us to specify these lines and extract the velocity profile on them.

1. x_r/S=0.0:

fluid.velocity	arc-length
0	4.90
20	5.2
30.1	5.39
42	5.65
51.1	5.87
58.5	6.07
72.1	6.57
76.9	6.85
81.2	7.53
76.4	8.09
67.6	8.51
54.2	8.96
41.8	9.27
32.7	9.47
22.3	9.69
0.00	10.1

Table 3: Flow variable profiles at x_r/S=0.0 downstream location, for Re= 389

13

2. x_r/S=5.41:

fluid.velocity	arc-length
0.0	0.0
-8.26	0.172
-18.9	0.444
-24.1	0.646
-27.4	1.07
-21.7	1.52
-1.05	2.44
11.6	2.96
26.3	3.48
41.2	3.97
56.7	4.46
70.2	4.96
79.7	5.48
81.9	5.97
77.7	6.44
66.8	7.01
41.6	7.91
26.1	8.44
18.8	8.73
12.8	8.99
4.96	9.45
1.91	9.74
0.0	10.10

Table 4: Flow variable profiles at x_r/S=5.41 downstream location, for Re= 389

3. x_r/S=11.84:

fluid.velocity	arc-length
0	0
12.1	0.525
23	1.01
33.4	1.52
42	2.01
48.8	2.52
52.5	2.96
54.3	3.48
53.4	3.99
50.5	4.45
45.3	4.97
39	5.46
32	5.98
25.3	6.47
19.2	6.98
14.7	7.41
10.4	7.93
7.42	8.39
5.87	8.68
4.44	8.97
2.81	9.34
1.5	9.68
0.45	9.97
0.0	10.10

Table 5: Flow variable profiles at x_r/S=11.84 downstream location, for Re=389

7.2.1 Comparison

To visualize the flow profile we pass to the graphical representation of the simulated results [ta-ble 3,4,5] using the plotly bibliography, then we will compare the results obtained in the simulation for Re=389 with the results of the literature.

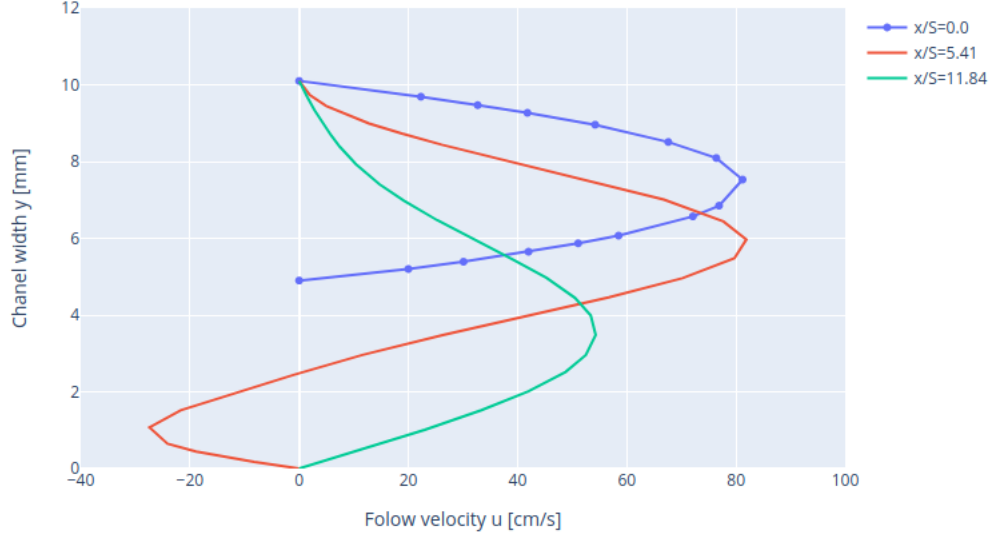The graph below shows the flow profile simulated at the different x_r/S locations:

Figure 7: Flow velocity profile simulate for Re=389 at x/S=0.0, x/S=5.41 and x/S=11.84

Now we move on to the comparison of the results using the root mean square (or RMS) of the deviation as an indicator to quantify the level of agreement between the experimental results and to simulate.

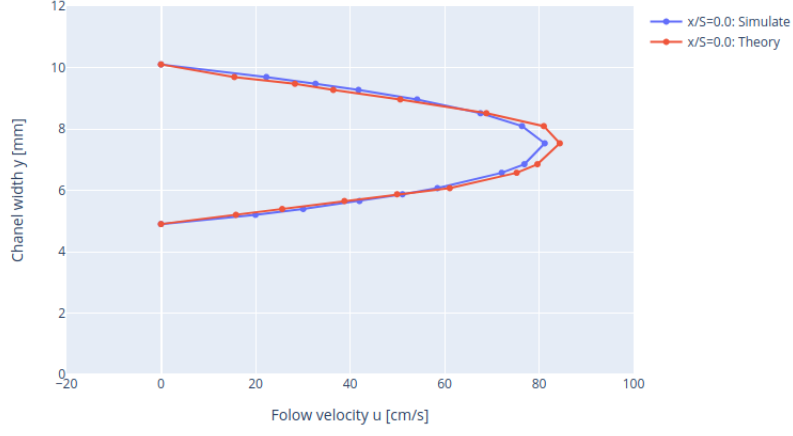The graph below show the comparison of the theorical solution with simulate flow solution for $x_r/S{=}0.0$.

Figure 8: Comparison of the theorical solution with simulate flow solution at $x_r/S{=}0.0$

The graph below show the comparison of the theorical solution with simulate flow solution for $x_r/S{=}5.41$.
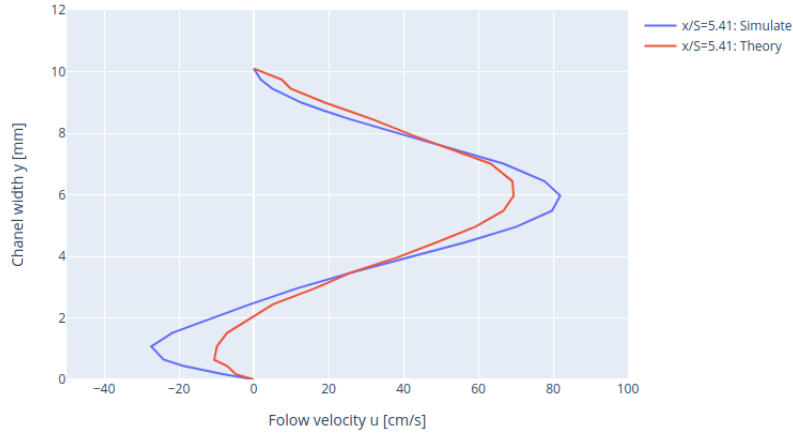

Figure 9: Comparison of the theorical solution with simulate flow solution at $x_r/S{=}5.41$

17

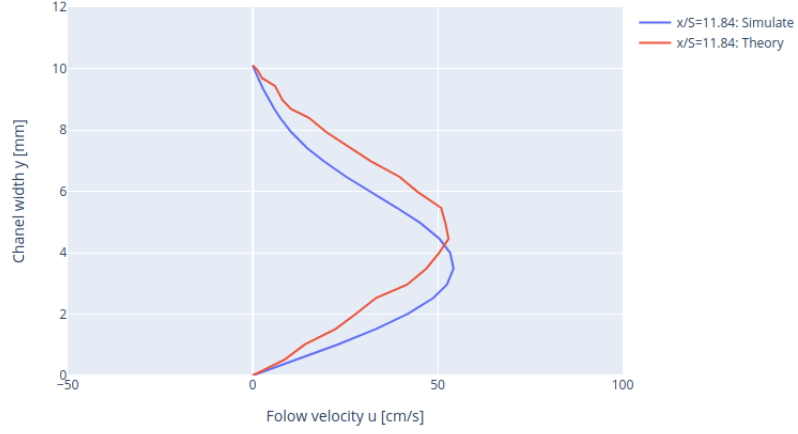The graph below show the comparison of the theorical solution with simulate flow solution for x_r/S=11.84.

Figure 10: Comparison of the theorical solution with simulate flow solution at x_r/S=11.84

Based on the results obtained during the simulation and the theoretical results we calculate the RMSE for each case, the results are grouped in the table below:

x_r/S	RMSE	observation average	model variance
0.0	3.63	44.47	8.16 %
5.41	8.54	24.34	35.08 %
11.84	8.86	25.31	35 %

Table 6: Root Mean Square Error,Observation average and Model variance

Indeed, according to [Table 6] the variance of the model for x_r/S=0.0, x_r/S=5.41 and x_r/S=11.84 corresponds to only 8.16%, 35.08% and 35.00% of the mean of the observations respectively, we can therefore say that the model has a high variance at x_r/S=5.41 and x_r/S=11.84 contrary at x/S=0.0 the variance is low.

7.2.2 Recirculation zone

The recirculation zone is defined as the zone of negative horizontal velocities, n case re=389 the recirculation zone is defined in the graph below.
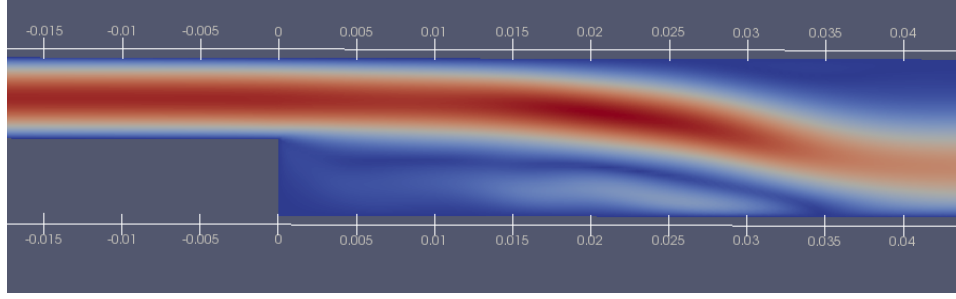
Figure 11: Recirculation zone

From the graph above, we deduce that the recirculation zone when Re=389 has as an reattachment length: x_r=0.035.

7.3   Results for Re=1095

To have a stationary solution when Re=1095 the problem has to be treated in an unstationary frame, where the solution will depend on time.
The command line to run this case is:

```
mpirun -np 4 feelpp_toolbox_fluid--config-file laminar.cfg--fluid.snes-monitor=1
                        --fluid.gmsh.hsize=5e-4
```

Once the command is executed the results are exported to Paraview for viewing the flow by executing:

```
rsync -avz atlas.math.unistra.fr:~/feel\\
```

The solution obtained by ParaView is represented in the figure below, but as the solution depends on time a video is made to better visualize the flow, we can see that for $x_r/S = 0.0$ the solution is stationary because the flow is so long and there is not really a variation in the flow as a function of time that we can say that it is stationary, but for $x_r/S = 7.04$ and $x_r/S = 19.04$ the solution is clearly unstationary because the flow of the fluid depends directly on time.

Once the results are present we will proceed to the study of the velocity profile on the different vertical lines x_r/S using the filter **PlotOverLine** which allows us to specify these lines and extract the velocity profile on them.
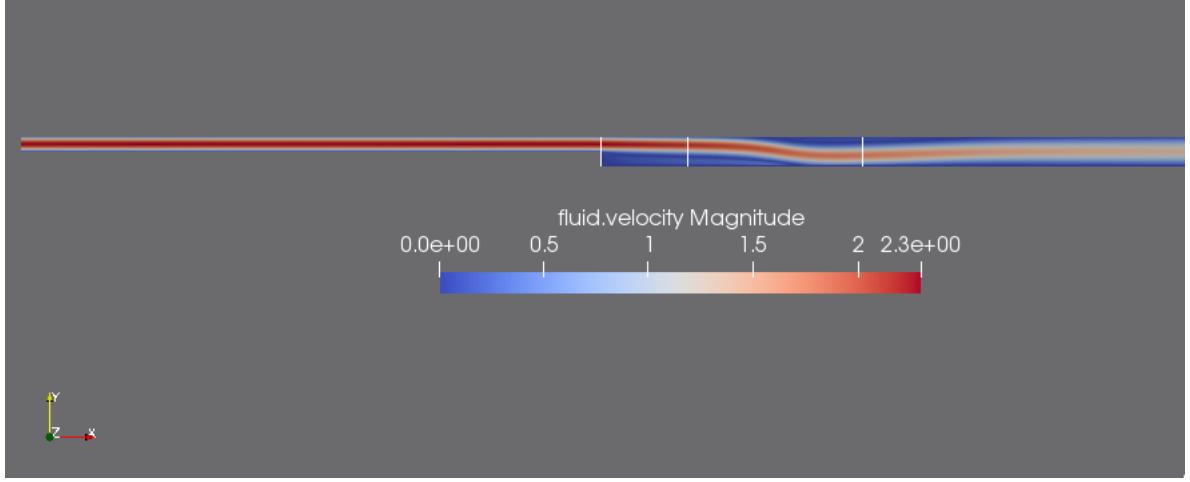The extracted results are saved in csv files for each vertical line:

Figure 12: flow velocity profile visualized by ParaView

1. x_r/S=0.0:

fluid.velocity	arc-length
0.0	4.9
16.9	5.0
34	5.1
71.8	5.32
110	5.61
137	5.82
185	6.33
214	6.81
221	7.03
228	7.27
228	7.52
224	7.83
204	8.31
160	8.89
126	9.21
102	9.4
74.3	9.62
48.2	9.79
0.00	10.10

Table 7: Flow variable profiles at x_r/S=0.0 downstream location, for Re= 1095

2. x_r/S=7.04:

fluid.velocity	arc-length
0.00	0.00
-7.6	0.333
-13.3	0.636
-18.2	0.97
-20.8	1.23
-22	1.56
-20.6	2.04
-15.8	2.44
-11.9	2.67
-7.41	2.87
7.57	3.38
25.6	3.83
56.5	4.38
92.1	4.89
132	5.39
170	5.89
199	6.4
213	6.9
213	7.36
195	7.91
160	8.46
112	8.99
81.6	9.29
50.2	9.6
0.00	10.10

Table 8: Flow variable profiles at x_r/S=7.04 downstream location, for Re= 1095

3. x_r/S=19.04:

fluid.velocity	arc-length
0.00	0.00
36.5	0.52
70.5	1.01
103	1.52
131	2.01
155	2.54
168	2.96
177	3.48
174	3.99
162	4.45
140	4.97
114	5.46
85	5.98
58.7	6.47
35.4	6.98
19.2	7.41
4.57	7.93
-4.38	8.39
-8.38	8.68
-10	8.97
-9.07	9.43
-7.03	9.68
-2.23	9.97
0.0	10.10

Table 9: Flow variable profiles at x_r/S=19.04 downstream location, for Re= 1095

22

7.3.1 Comparison

To visualize the flow profile we pass to the graphical representation of the simulated results [table 7,8,9], then we will compare the results obtained in the simulation with the results of the literature.

The graph below shows the flow profile simulated at the different x_r/S locations:
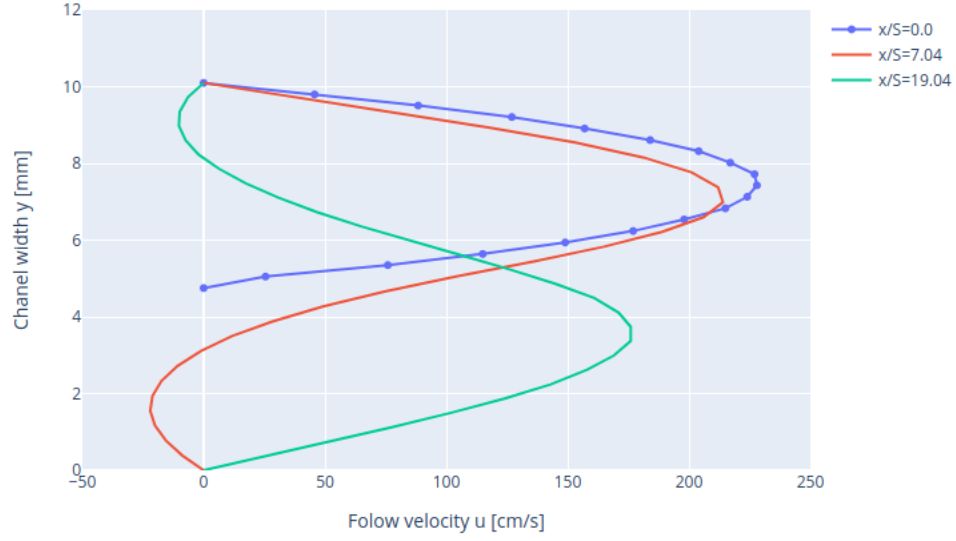

Figure 13: Flow velocity profile simulate for Re=1095 at x/S=0.0, x/S=7.04 and x/S=19.04

Now we move on to the comparison of the results using the root mean square (or RMS) of the deviation as an indicator to quantify the level of agreement between the experimental results and to simulate.

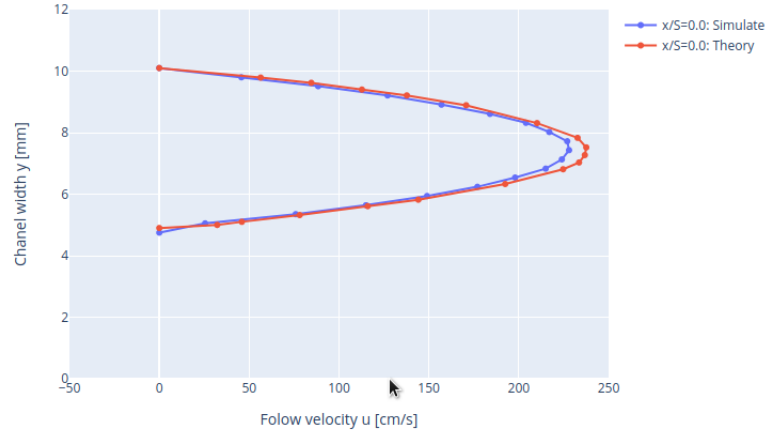The graph below shows the comparison of the theorical solution with simulate flow solution for x_r/S=0.0.

Figure 14: Comparison of the theorical solution with simulate flow solution at x_r/S=0.0

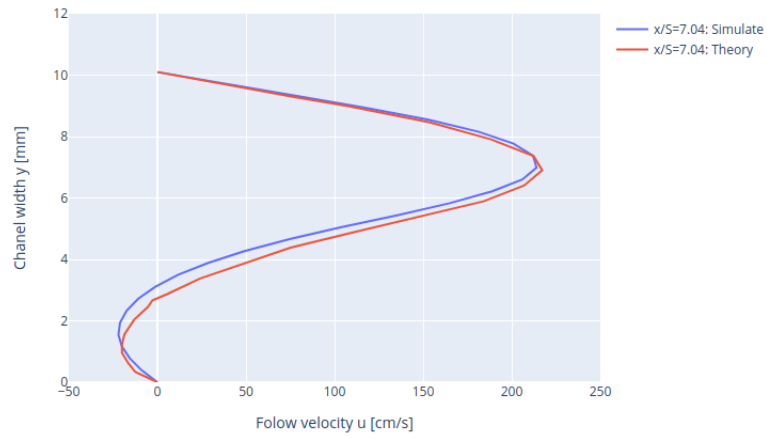The graph below shows the comparison of the theorical solution with simulate flow solution for x_r/S=7.04.

Figure 15: Comparison of the theorical solution with simulate flow solution at x/S=7.04

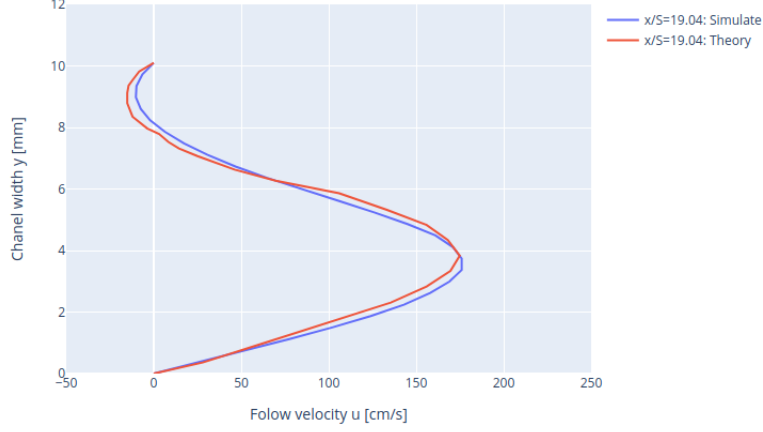The graph below shows the comparison of the theorical solution with simulate flow solution at x_r/S=19.04.

Figure 16: Comparison of the theorical solution with simulate flow solution at x/S=19.04

Based on the results obtained during the simulation and the theoretical results we calculate the RMSE for each case, the results are grouped in the table below:

x_r/S	RMSE	observation average	model variance
0.0	9.33	134.03	6.96 %
7.04	10.05	67.86	14.80 %
19.04	12.03	57.79	20.81 %

Table 10: Root Mean Square Error,Observation average and Model variance

Indeed, according to [Table 6] the variance of the model for x_r/S=0.0, x_r/S=7.04 and x_r/S=19.04 corresponds to only 6.96%, 14.80% and 20.81% of the mean of the observations respectively, so we can say that the model has a high variance.

7.3.2   Recirculation zone

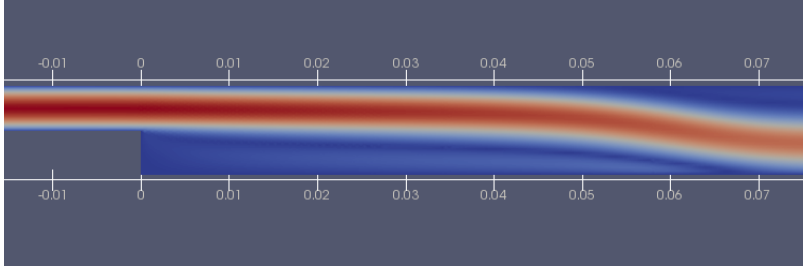In case re=1095 the recirculation zone is defined in the graph below.

Figure 17: Recirculation zone

From the graph above, we deduce that the recirculation zone when Re=1095 has as an reattachment length x_r=0.07.

7.4   Reattachment length in the laminar flow regime

To illustrate the relation between x_r/S and the Rynolde number we have performed several simulations at different Re.

x_r/S	Re
3.06	100
6.12	250
7.14	398
14.28	1095

Table 11: Reattachment length and Rynolde number

8　Conclusion

In this project a direct simulation method has been validated for two-dimensional laminar flow in a downslope channel.

Then, a comparison approach based on the results of the literature was introduced to compare the theoretical and simulated results.

Finally, from the results obtained in the simulation it is clear that there are some calculation errors which are not too large but need to be taken into account in the further study.

References

[1] Haque, A., Ahmad, F., Yamada, S. and Chaudhary,S. (2007). Assessment of Turbulence Models for Turbulent Flow over Backward Facing Step. Assessment of Turbulence Models for Turbulent Flow over Backward Facing Step, 2(4),pp.1-6

[2] G.Stèphan, Contribution a la modelisation de la population atmosphérique dans les villes, Bordeaux i , 2000

[3] C. Christopher, Numerical solution of a 2D flow over a backward facing step, May 2018

[4] R. Christoph, méthode de raccordement de maillages non-conformes pour des equations de Navier-Stokes, Université de Bordeaux I, 2006

[5] M. Himdi,Contribution à la simulation numérique des écoulements de fluides compressibles etpeu compressibles par le code de calcul KIVA-II, PH-D Thesis, Laboratoire de mécanique de Lille,France, (1993)

[6] C.H. Bruneau and E. Creuse,Towards a transparent boundary conditions for the compressible-Navier–Stokes Equations, Int. J. Num. Meth. Fluids,36, 807-840 (2001) (1993)

[7] https://en.wikipedia.org/wiki/Computational-fluid-dynamics

[8] http://docs.feelpp.org/toolboxes/0.108

[9] https://en.wikipedia.org/wiki/Navier-Stokesequations Incompressibleflow/bigskip

[10] https://en.wikipedia.org/wiki/$Divergence_t heorem$

[11] https://en.wikipedia.org/wiki/Momentum

[12] https://en.wikipedia.org/wiki/$Continuity_e quation$

[13] https://en.wikipedia.org/wiki/$Conservation_o f_m ass$