Testing an aerodynamic braking system of a ground vehicle on OpenFOAM

ED DAHHAOUI Abdelilah 28600673

December 9, 2023

Abstract

This study aims to show the advantages and the flaws of an aerodynamic braking system, for a moving ground vehicle (such as a car). To understand the problem, the simulations will be done around a simple geometry first (in this case, around a 2D square). In a more advanced part, we will simulate a flow around an Ahmed Body, with some modifications to the boundary conditions or the body itself.

1 Introduction

Nowadays, the most commonly used braking system on a ground vehicle is the "friction brake". This braking system uses the pressing of a brake pad against a moving part (such as a rotating disk) to generate a friction force against the movement of the object. This will result in the progressive slowing down of any vehicle that uses this system. While being pretty straight forward and effective, this system leads to two major problems : first of all, the generating of heat from the kinetic energy of the moving disk that comes as a price for the friction generated. And second, the emission of "microplastics" from this friction, that leads to a serious environmental problem. So the main idea behind this project will be to study the effectiveness of an aerodynamic braking system on a moving car, that will use the incoming air flow to its advantage, and slow down the vehicle. More precisely, we'll look on how to increase effectively the drag of our moving object, by adding a geometry to our system (a spoiler for example) or by changing the boundary conditions (by blowing air in the wake for example). To first understand the problem, we will study 3 different cases of a flow around a 2D square, looking for the values of the drag coefficient for a specific Reynolds number. We will then study a 3D case of an "Ahmed Body", a well know test case used to study the flow around a car.

2 Flow around an isolated 2D square

2.1 The steps

In this first case scenario, we will test an incoming flow on a 2D square on OpenFOAM. By setting a non-slip boundary condition on the square walls, we will try to simulate an incoming air flow from the far left, with a given initial speed. For a given Reynolds number, we will plot our lift and drag coefficients, knowing that for a symmetrical square, we expect a lift equal to 0 (no pressure difference because of the symmetry). We will then try to validate our simulation by comparing our numerical results to the experimental results found in the bibliography.
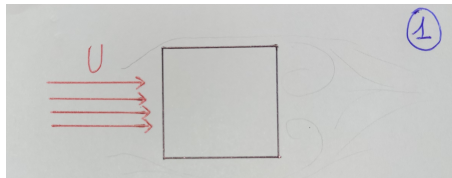

Figure 1: Flow around a 2D square

2.2 The problem parameters and resolution

2.2.1 Geometry and Mesh

Let's start by creating the domain that we will study in this section. We first define a square cylinder of 1 by 1 (in the X and Y direction) centered at the position (X = 0,Y = 0,Z = 0). The characteristic length of this problem L_{ch} is equal to the surface of the square cylinder. Starting

from the center, the bounding box expands by $25L_{ch}$ to the right, $8L_{ch}$ to the left, $8L_{ch}$ to the bottom and $8L_{ch}$ to the top (see Figure 2).
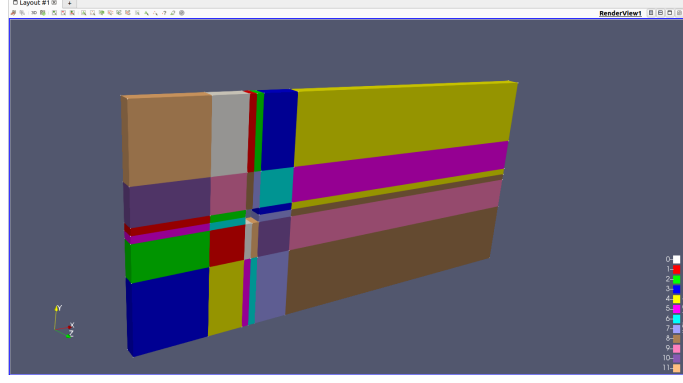

Figure 2: Geometry of the first case

As it can be seen in this figure, multiple blocks have been defined around our square cylinder. In fact, as a fine mesh is required near the periphery of the cylinder (as well as the wake region), defining multiple blocks will give us more control over the general meshing of this system. So, for a more precise calculation of the coefficients, the mesh elements in these locations have to be small enough. To lower the simulation time, a gradual mesh grading is done to get a coarse mesh at the boundaries. As the problem is in 2D, the Z direction is neglected in this case and set to 1. The final meshing is achieved with the help of blockMesh and the number of mesh cells produced in (X ∗ Y) is around 63,000 (see Figure 3).
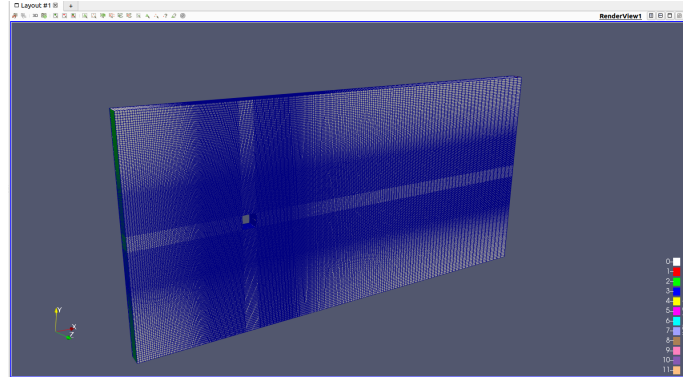

Figure 3: Meshing of the first case

2.2.2 Boundary Conditions

Concerning the velocity, let's first simulate an incoming flow from the left surface of the domain (we will call this face "inlet") with a normalised velocity U_x of 1 m/s. Let's note that this speed is only horizontal as U : $(U_x = 1, U_y = 0, U_z = 0)$. Regarding the far right plane (defined as "outlet"), let's set a Neumann condition ("zeroGradient" on OpenFOAM, meaning that there is no value gradient along the flow direction on this surface). On the top and bottom walls, a slip condition is applied (meaning that the flow velocity at the walls is equal to the incoming flow velocity), with planes defined as symmetry planes. Finally, on the surface of the square cylinder, let's set a no slip condition, meaning that the flow reaches 0 velocity at the walls of the object. Concerning the pressure, let's set a Neumann zero gradient condition everywhere except at the outlet, where a fixed pressure equal to 0 will be set.
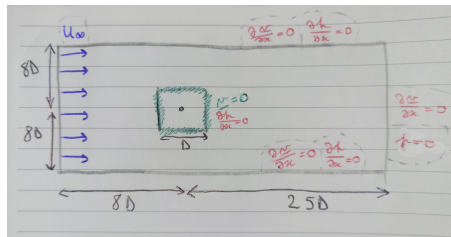

Figure 4: Geometry, boundary condition and initial conditions of the problem

2.2.3 The simulation

After having our geometry, mesh and boundary conditions set, let's proceed to the simulation parameters. The solver that will be used is icoFOAM, for incompressible laminar Navier-Stokes equations. The Reynolds number Re will range from 10 to 200 for a total of 5 different case scenarios, one for each Re : (10,50,100,150,200). On OpenFOAM, this can be achieved by changing the ν coefficient in the "constant/transportProperties" file following this formula :

$$Re = \frac{U_x * L_{ch}}{\nu} = \frac{1}{\nu}$$

So :

$$\nu = \frac{1}{Re}$$

Depending on the desired Reynolds of one simulation, changing the only value of ν will be sufficient to achieve the desired type of flow. Next, to calculate the various coefficients of drag and lift, let's set a function named "forceCoeffs" on the "system/controlDict" file. By setting all the different parameters of our problem (density, surface of reference, velocity at inlet), the system will calculate and save the values of the coefficients in a "postProcessing" folder. Finally, the time step is set to an arbitrary 0.05 for a total of 400 iterations per simulation. For a sake of efficiency, the simulation will always be run on parallel using the "mpirun" tool of OpenFOAM, and the convergence will always be checked with the "pyFoamPlotWatcher.py" tool.

2.3 Results and discussions

At this point, all that is left to do is to run the simulations for various Re numbers. Two main examples will be studied in detail : $Re = 10$ and $Re = 150$. We will explain later the main goal behind the study of these two examples precisely.
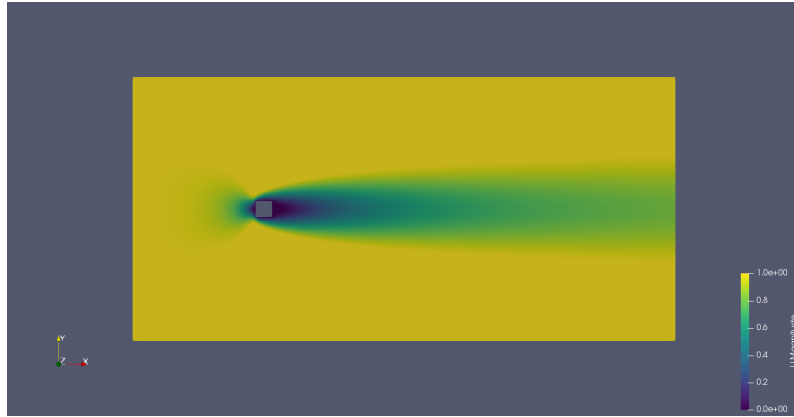
2.3.1 First example : $Re = 10$

Figure 5: Flow in 2D around a square cylinde with $Re = 10$

This first simulation was successfully completed for a CPU time of about 573 seconds. Let's display the residuals and the drag/lift coefficients versus the iteration time.
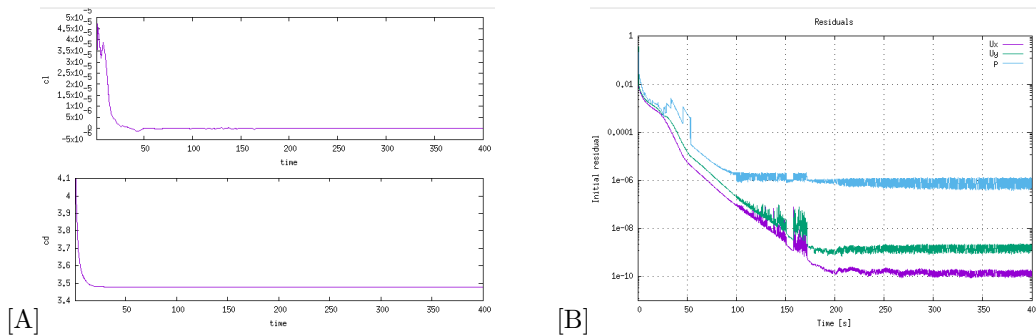

Figure 6: $Re = 10$; [A] : At the top, C_L VS Iterations, at the bottom, C_D VS Iterations ; [B] : Residuals VS iterations

First, let's denote a convergence for this simulation around 200 iterations, as shown from the Figure 6 [B]. Second, regarding the coefficients, we observe for the lift a value that stays close to

0, as expected. In fact, knowing that our square cylinder is strictly symmetrical and that there isn't any influence from a close wall (as we will see in the next part), our object will not experience any difference of pressure between its high and low parts. So, our object will not experience any lift force and, as a consequence, its lift coefficient is strictly equal to zero. Now, this case doesn't apply to the drag coefficient because of the presence of a wake behind the square cylinder. This wake will generate a drag force on the object, and consequently the drag coefficient is not equal to zero. Here, we observe a convergence of the drag coefficient C_L to a value around 3.47.
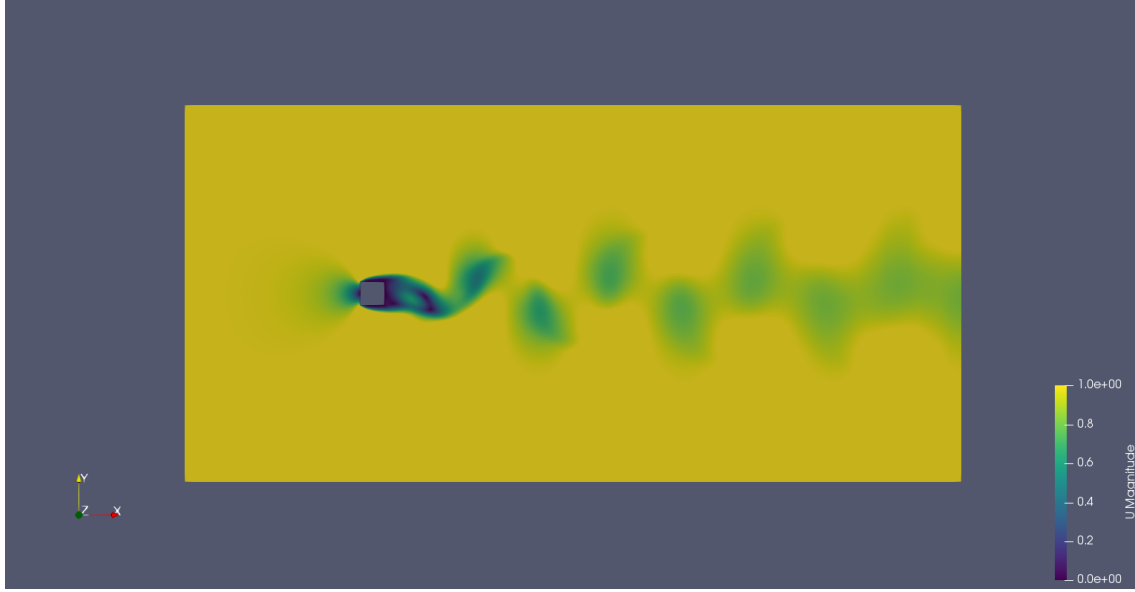
2.3.2 Second example : $Re = 150$

Figure 7: Flow in 2D around a square cylinde with $Re = 10$

This second simulation was successfully completed for a CPU time of about 1900 seconds. For the same time step, this simulation took subsequently more time on the CPU than the first case. Let's display the residuals and the drag/lift coefficients versus the iteration time for this case.
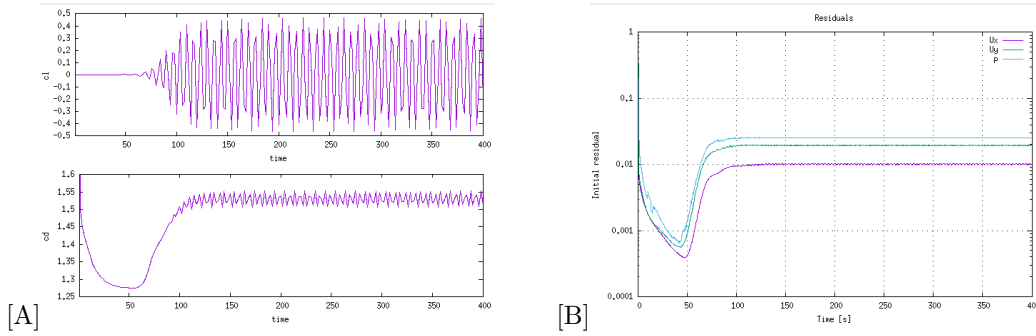

Figure 8: $Re = 150$; [A] : At the top, C_L VS Iterations, at the bottom, C_D VS Iterations ; [B] : Residuals VS iterations

This second example seems to develop what is called a "Laminar Von Kármán Vortex Street". In fact, for a sufficiently large domain and for a Re number ranging from 40 to around 200, swirling vortices can be observed behind the object. These are mainly caused of what is called a "Vortex Shedding", which is a phenomenon where vortices are shed alternately from one side to the other, and where alternating low-pressure zones are generated on the downwind side of the structure (here, the square cylinder). Our study will not necessarily focus on this phenomenon, but it is important to denote its effects on the residuals and the coefficients convergence. Instead of a direct convergence, here the values of both jump when the Vortex Street appears, converging elsewhere. For the drag coefficient, we will look into the smallest value observed (here, about 1.27 for the 60th iteration), not counting for our comparisons the effects of the Votex Shedding.

2.3.3 How to verify the accuracy of the results?

After studying different cases for different Re numbers, how can we be sure about the actual accuracy of our values? Let's compare our results to the results issued from the paper "Accurate

4

computations of the laminar flow past a square cylinder", extracted from the work "International Journal of Heat and Fluid Flow" by M. Breuer. Hereby, the results and their comparison :
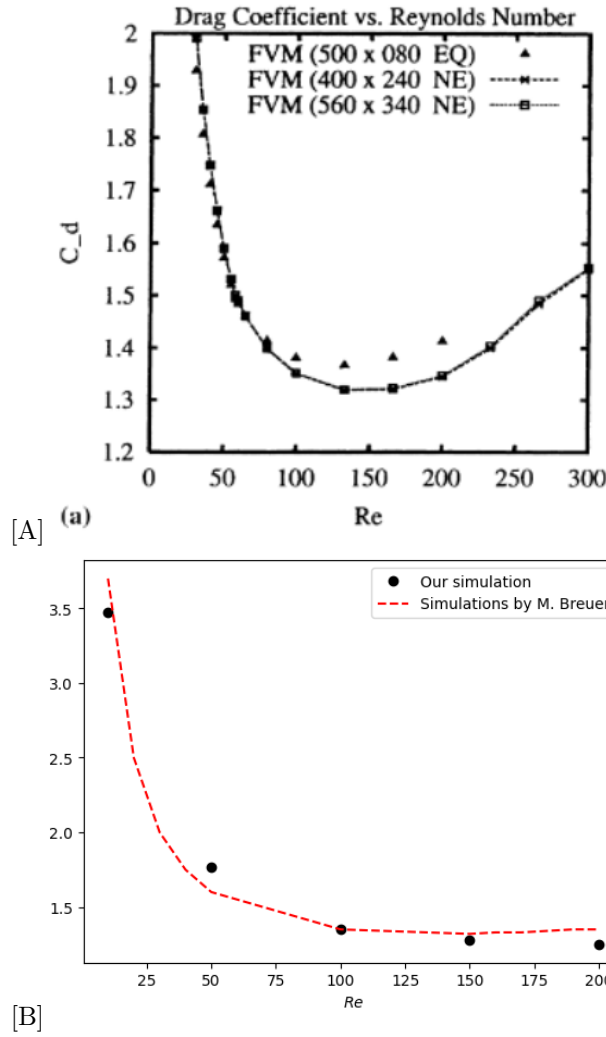

[A]

[B]

Figure 9: [A] : C_D VS Re issued from "International Journal of Heat and Fluid Flow" ; [B] : Comparison of the results for the drag coefficient VS Re : (10,50,100,150,200)

Hence, our simulation values are concordant with the experience of M. Breuer, without taking into account the effects of laminar Von Kármán vortex street. For the general error, we calculate the RMS error between our values and the values issued from M. Breuer's paper.

$$RMSE = \sqrt{\frac{\sum_{i=1}^{N} \|y(i) - \hat{y}(i)\|^2}{N}},$$

Figure 10: RMSE calculation

As a result, we find 13 percent of error between our values and the values issued from M. Breuer's paper. Knowing that this error is between 10 and 20 percent, our expectation is fair compared to the bibliography results.

3 Flow around a 2D square close to a ground

In this second case scenario, we will study the impact of a close ground (at height h) on the coefficients calculated by the program. More precisely, we want to investigate the impact of this ground proximity on the drag coefficient, and how this impacts the lift coefficient (because of the breaking of the symmetry, the lift should not be equal to zero). This part will mainly be based of the paper "Boundary shear flow past a cylinder near a wall" by Lin Feng Chena and Guo Xiong Wu.
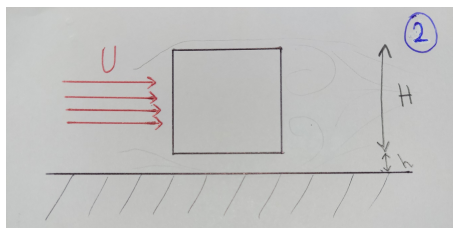
Figure 11: Flow around a 2D square close to the ground

3.1 A new approach

To study conveniently this problem, let's first study a case of a circular cylinder close to the ground. This will be done in order to validate our numerical model, by comparing our results with the work of Lin Feng Chena and Guo Xiong Wu. The domain general dimensions are unchanged, but the blocks are rearranged in order to move the object near the ground.
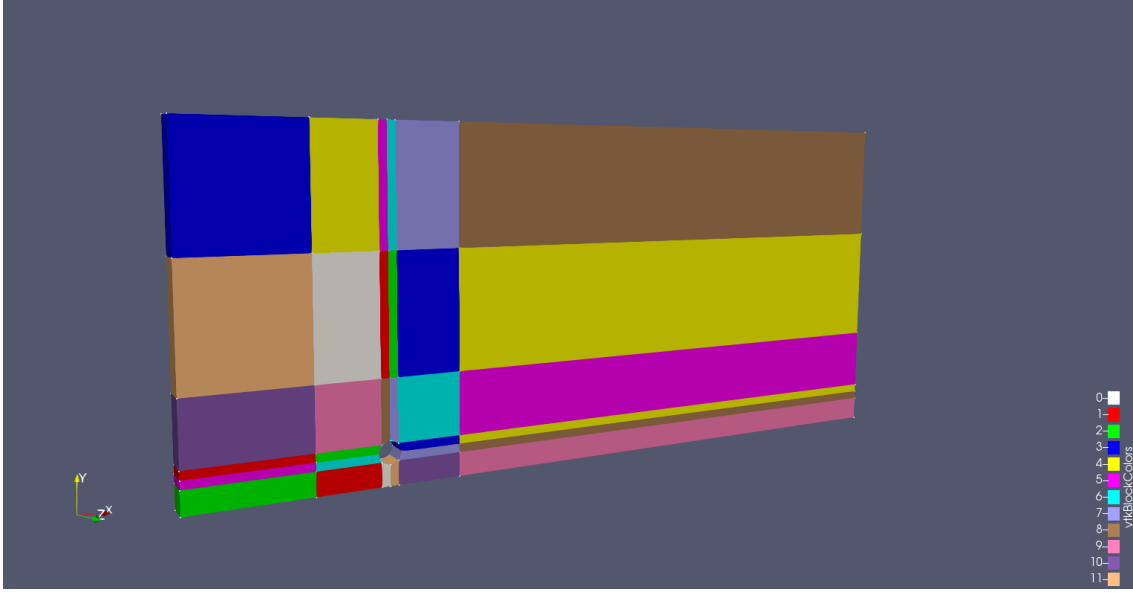

Figure 12: 2D circular cylinder close to the ground on Paraview

The various parameters of the problem remain unmodified and the study will be done for one unique Re_{fixed} equal to 100. What remains to do in this case is setting the boundary condition at the ground. In fact, we cannot set a zero gradient condition as before because we want to observe the ground effects on our cylinder. So, to achieve such a condition, we will present our new initial condition and boundary condition at the ground in the next section.

3.1.1 How to mimic a No-Slip condition at the ground?

To visualize the ground effects on our object, let's try to set a No-Slip condition first in the "0/U" file and see what happens to our simulation. As a reminder, a No-Slip condition means that the fluid cannot slip at the surface, so the velocity will be equal to zero at this same surface. This also means that there will be a boundary layer developing.
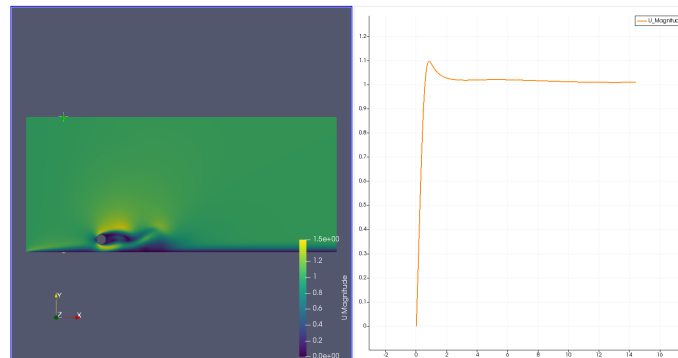

Figure 13: Plot over line on Paraview with a No-Slip condition at the ground

Surprisingly, there seems to be an "overshoot" of the velocity, reaching levels equal to 1.1 m/s. This answer is not physical and can be explained by the non conservation of the flow rate in this set. The rest of the boundary conditions here are set to zero gradients. Thus, instead of setting and actual boundary condition that mimics the No-Slip condition in OpenFOAM at the ground, let's try to set an initial velocity at the inlet based on the Blasius laminar boundary layer theory. This new velocity profile is directly issued from the work of Lin Feng Chena and Guo Xiong Wu as :

$$\bar{u}(y) = \begin{cases} 2\frac{(y+e+0.5)}{\delta} - \frac{(y+e+0.5)^2}{\delta^2} & \text{if} \quad (y+e+0.5) \leq \delta \\ 1 & \text{otherwise} \end{cases}$$

where y is perpendicular to the wall and $y = -e - 0.5$ is the wall surface,

$$\delta = 4.92\sqrt{\frac{(x-x_s)}{Re}}$$

Figure 14: Blasius profile issued from "Boundary shear flow past a cylinder near a wall" by Lin Feng Chena and Guo Xiong Wu

In this case, e is the distance between the ground and the object and x the position of our object. We estimate the δ at 2.329 for $Re = 100$. Setting this profile as an inlet coded profile will mimic the boundary layer impact on the object that we are studying, at a given position. What is left is to set a boundary condition at the ground that reflects the flow without altering the flow parameters. This condition is known as a "Slip" condition.

3.2 Simulation and results

Having all the parameters and boundary conditions for this problem, let's run the simulations for various e distances and compare our results with the bibliography. The Re is set at 100 for all the simulations.
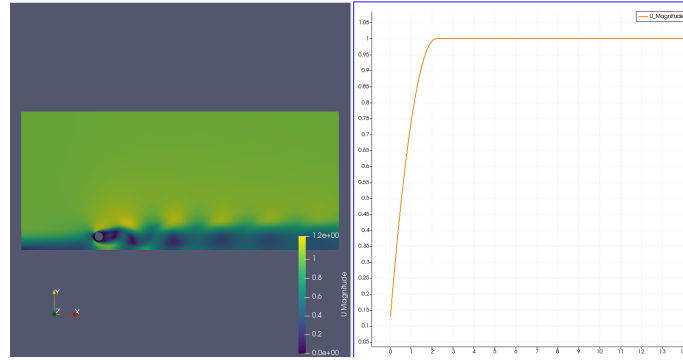

Figure 15: Plot over line on Paraview with the Blasius velocity profile condition at the inlet for $e=1$

As expected, in this example we can observe that the Blasius profile is well depicted. This time, the "overshoot" in velocity is absent and the boundary layer is well modeled. As before, let's now compare the drag and lift coefficients values with the ones issued from the bibliography.
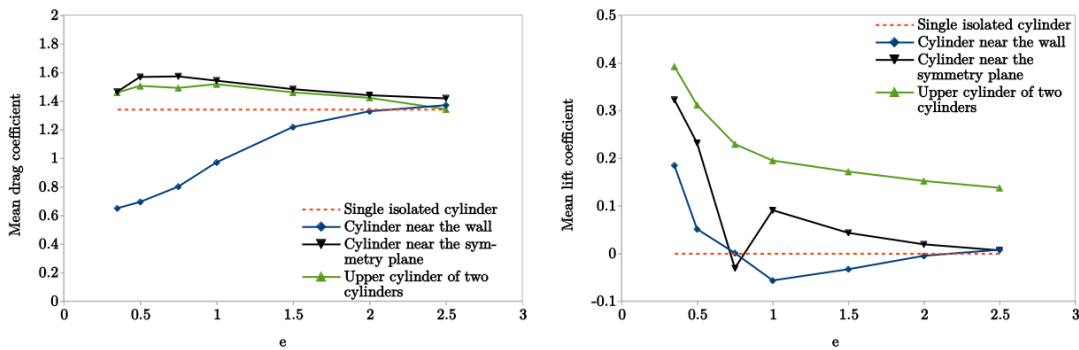

Figure 16: Mean drag and lift coefficients issued from "Boundary shear flow past a cylinder near a wall" by Lin Feng Chena and Guo Xiong Wu

First, let's remark that for a certain distance e from the ground, the C_D and C_L coefficients converge to a certain value (zero for C_L and around 1.35 for C_D). These values are, in fact, the
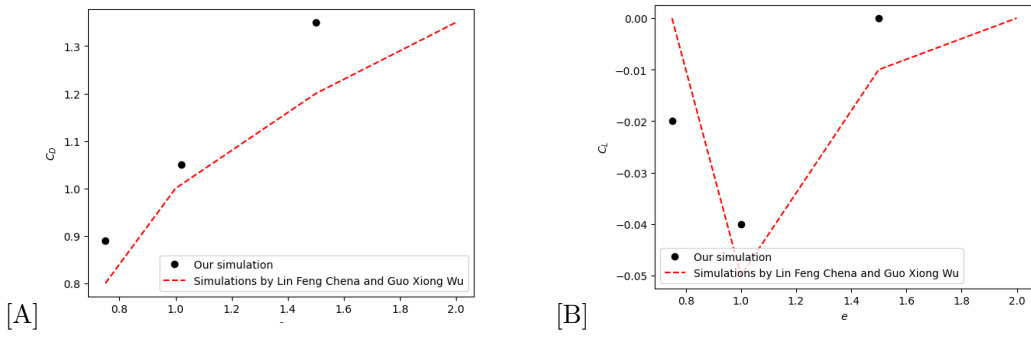
Figure 17: $Re = 100$; [A] : Comparison of the mean drag coefficients for different distances e; [B] : Comparison of the mean lift coefficients for different distances e

ones for a circular cylinder far from any walls. This first conclusion is normal because the further the object is from the ground, the less impact this same ground will have on it. Second, the values between our simulation and the simulation done in the bibliography seem to be fairly accurate for the drag coefficient but lack on precision on the lift coefficient. A better mesh or a more precise calculation of the Blasius profile (by using a higher order equation) could lead to a better approximation of these coefficients.

3.3   Simulation for a square cylinder

After validating our numerical model, let's switch back to our case scenario of a square cylinder affected by the same flow parameters than before. Hereby, the visualisation of the flow, the residuals and the convergence of the drag coefficient for a fixed distance $e = 1$ and a $Re = 100$.
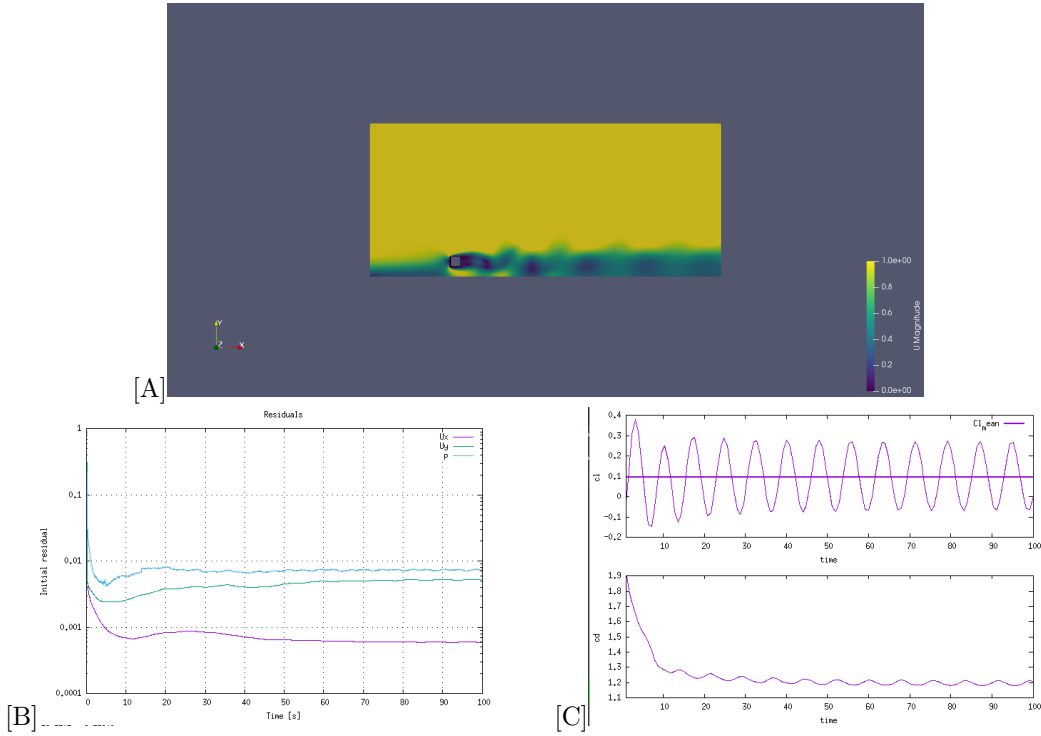

Figure 18: $Re = 100$; [A] : Flow visualisation at convergence; [B] : Residuals VS Iterations; [C] : Convergence of the lift coefficient (top) and drag coefficient (bottom)

As expected, the drag coefficient is lower than what we observed for the square cylinder, far from any walls. Indeed, it can be estimated in this case at around 1.2. The ground effects have a major impact on the object by, first, inducing a lift force on the object and by, second, lowering the drag force. Now, our main goal will be to vary the drag force undergone by the object by impacting the object wake directly. We will study the impact of an additional velocity condition set in the right face of the square cylinder. This is done in order to see if the generation of a flow (brought by a fan for example) can have a consequent impact on the wake structure.

8

4 Flow around a 2D square close to the ground with an additional blowing condition

This last case scenario will be similar to the second one, but this time we are adding a boundary condition on the back of our square, to simulate an air blow. The idea behind this addition is to understand what impact can we have on the wake and, subsequently, on the drag coefficient by changing its environment.
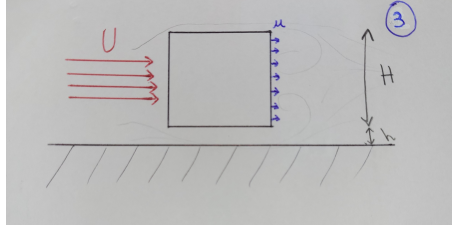

Figure 19: Flow around a 2D square close to the ground with an additional blowing condition

4.1 Flow around a 2D square close to the ground with an additional blowing condition

To study the impact of this additional velocity, let's run a simulation for four main cases, one for each additional velocity at the back of the square cylinder. These velocities will be equal to 10, 40, 110 and 150 percent of the incoming velocity. The main goal here is to see the impact of these velocities on the calculated drag coefficient.
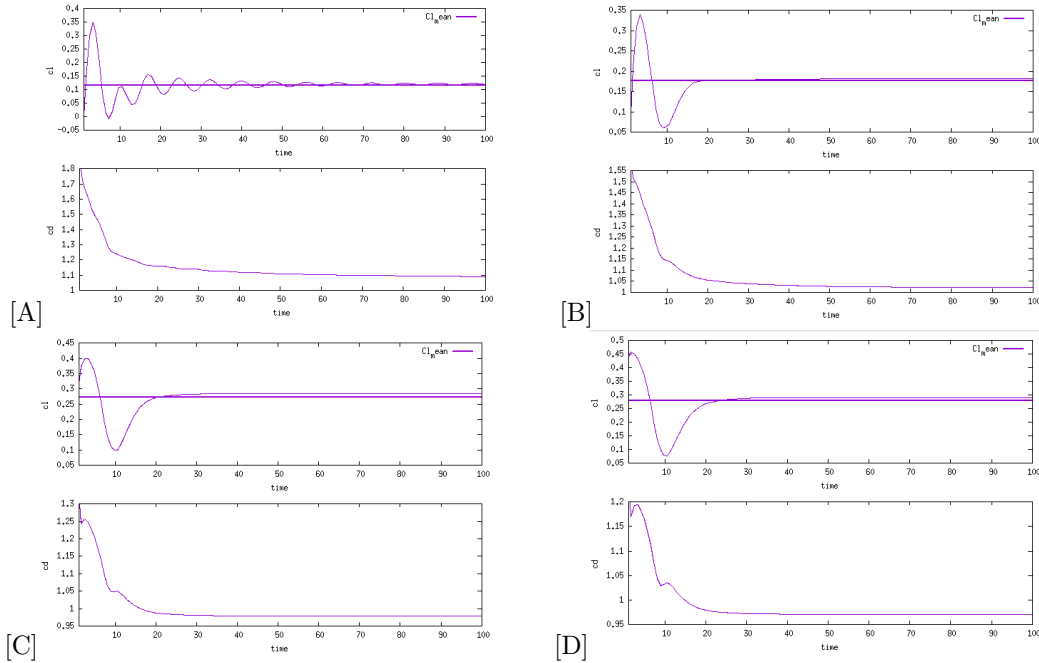

Figure 20: $Re = 100$, $e = 1$; [A] : C_D VS Iteration with additional 0.1 m/s; [B] : C_D VS Iteration with additional 0.4 m/s; [C] : C_D VS Iteration with additional 1.1 m/s; [D] : C_D VS Iteration with additional 1.5 m/s

As a reminder, we estimated the drag coefficient for a square cylinder at a distance $e = 1$ and for a $Re = 100$ at about 1.2. Surprisingly, the results here demonstrate that adding a velocity behind the object does not, in fact, increase the drag coefficient but the lift coefficient. The drag coefficient, in the contrary, decreases with the additional velocity. So, the idea of fueling the wake and increase the drag force is, hereby, not valid. Let's remind the main formula of the drag :

$$D = C_d \cdot \frac{1}{2}\rho v^2 A \tag{1}$$

The drag is proportional to the square of the incoming velocity and directly proportional to the drag coefficient of the system. During this report, we fixed the incoming velocity to strictly study the effects on the drag coefficient. What seems to be clear at this point is that this coefficient directly depends on its environment and directly depends on how the flow circulates around our

object. One last try to increase this coefficient will be done in the next and last section, studying one case of Ahmed Body with and without a spoiler.

5   Flow around an Ahmed Body

The final goal of this project will be to simulate a flow on a 3D object (the "Ahmed body" in this case). We will first be trying to validate our numerical model with the analytical solutions for this test case. As a heads up, this 3D model will take in count the turbulence effects on the object, making the simulations some more complicated from all the cases that we had until know. The main goal here will be to increase the drag of our object by increasing its wake size. The precedent case in 2D showed how an additional velocity at the back of the object does not increase the drag. So, in this case, we will test the effects of a spoiler on the drag coefficient.
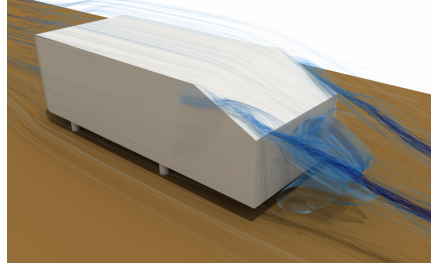

Figure 21: Flow around an Ahmed Body on SimFlow

5.1   Running an "Ahmed Body" case

Before the addition of a spoiler, let's run ourselves a test case of a flow around an Ahmed Body. The flow velocity is set to 25m/s and the turbulence model used is k-omega. The OpenFOAM solver used is SimpleFOAM for a total of 3000 iterations.
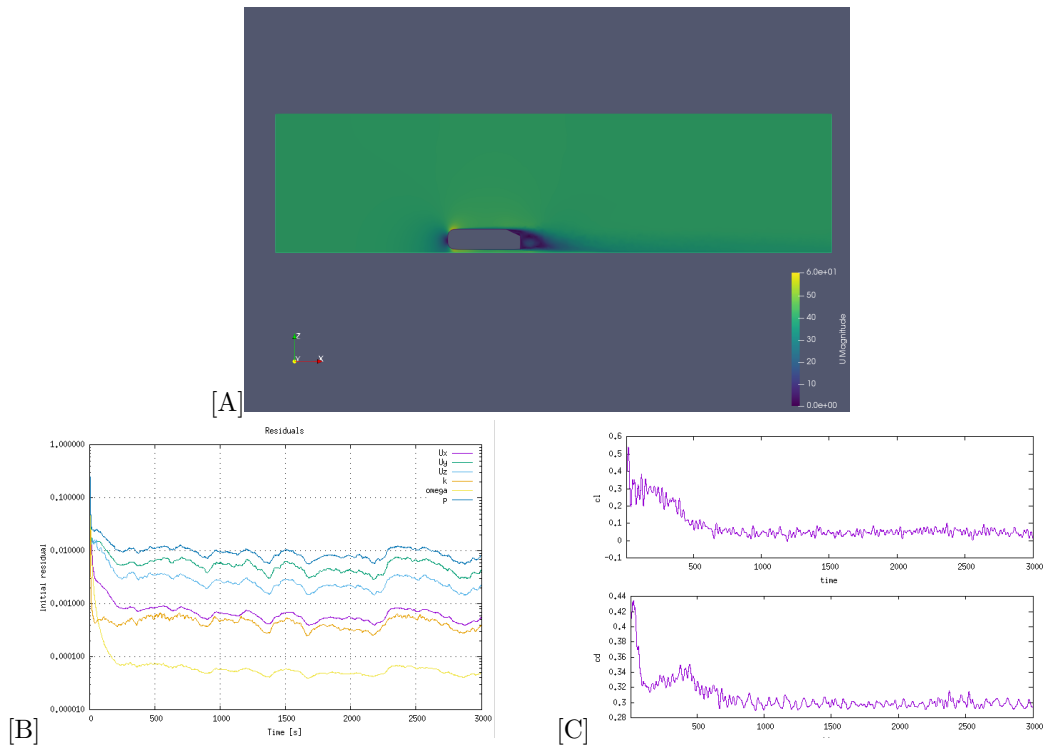

Figure 22: [A] : Flow around an Ahmed Body in 3D with $v = 25$ m/s; [B] : Residuals VS Iterations; [C] : C_L VS Iteration (top) and C_D VS Iteration (bottom)

The drag coefficient found around the body for this type of flow is around 0.3. The total CPU time for this simulation was around 4h30, for a developed mesh.

5.2 Comparing with an "Ahmed Body" case that has an additional spoiler

For this section, we use the study done on the report of Skill Lync (www.skill-lync.com/student-projects/flow-over-ahmed-body-with-spoiler) by Tribhuvan Batish. In the study, an Ahmed Body case with a 15° spoiler was studied. The results demonstrate how this addition increases by almost 80 percent the value of the drag coefficient. Indeed, if we compare our results with the ones issued from the report, we get a bound from 0.3 to almost 0.5 for the drag coefficient.
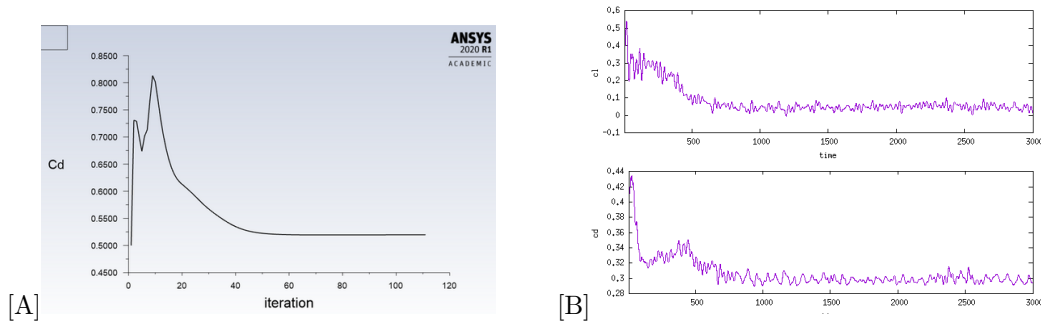

Figure 23: [A] : C_D VS Iteration issued from the report of Tribhuvan Batish on : www.skill-lync.com/student-projects/flow-over-ahmed-body-with-spoiler; [B] : C_L VS Iteration (top) and C_D VS Iteration (bottom) for our simulation without spoiler

6 Conclusion

In conclusion, this report presents a comprehensive study on the aerodynamic braking system of a ground vehicle using OpenFOAM. The investigation focused on understanding the advantages and drawbacks of such a system, aiming to address issues associated with traditional friction brakes, such as heat generation and environmental issues.

The study involved simulations around a 2D square to establish a baseline and validate the numerical model against known results. Then, a following analysis included the impact of ground proximity on the drag and lift coefficients, demonstrating the influence of the ground on these aerodynamic forces. The introduction of a laminar Blasius profile and slip condition allowed for accurate modeling of the boundary layer effects near the ground.

Furthermore, the report explored the effects of an additional blowing condition at the back of a square cylinder. Surprisingly, the results showed that while the lift coefficient increased with the additional velocity, the drag coefficient decreased.

Finally, the study extended to a 3D simulation around an Ahmed Body, incorporating turbulence effects. Our goal was to increase the drag force by introducing a spoiler.

In summary, this report showed how it is important to understand the behaviour of the flow around a 2D object before developing any models in 3D. Unfortunately, we could not prove the efficiency of a blowing condition behind an object and its effects on the drag coefficient. To further develop this case, more simulations should be done in 3D around an Ahmed Body to confirm the 2D assumptions.

7 References

https://help.sim-flow.com/validation/ahmed-body
https://www.linkedin.com/pulse/alternative-method-how-improve-vehicles-braking-jaroslav-kme%C5%A5/
https://www.sciencedirect.com/science/article/pii/S0142727X99000818
https://github.com/nithinadidela/circular-cylinder/tree/master
https://discovery.ucl.ac.uk/id/eprint/10084187/1/elsarticle-revised.pdf
https://www.sciencedirect.com/science/article/pii/S2215098615000786#tbl3
www.skill-lync.com/student-projects/flow-over-ahmed-body-with-spoiler