École Polytechnique Fédérale de Lausanne

Plume in the atmospheric boundary layer

ENG-420: Environmental Transport Phenomena

Authors: Giulia Mescolini, Niccolò Polvani, Thomas Rimbot
Tutor: Dr. Tristan Revaz

14th January, 2022

Research Question

The aim of this project is to study the diffusion in the atmospheric boundary layer of a plume leaking from the crater of a volcano. More specifically, we are interested in studying how advection and diffusion of the plume behave with respect to varying parameters (such as exit velocity of the plume from the crater and diffusivity of the plume in the atmosphere) and in observing target variables in a point downstream the volcano.
The importance of monitoring the behavior of volcanic plumes comes from the fact that they have important implications for the chemistry and composition of the troposphere and stratosphere. They also have the ability to alter Earth's radiation budget and climate system over a range of temporal and spatial scales.
Moreover, the study of the concentration of some volcanic gases can be crucial for human life as well, since at certain concentrations some volcanic gases can be lethal. Indeed, as stated in [7], in the past deaths have been caused by sulfur dioxide and carbon dioxide, which are among the main components of volcanic plumes.
With that in mind, we proposed to study the impact of inflow velocity and plume diffusivity on the concentration of the emitted substance at a house situated downstream of the volcano, in order to evaluate a possible threat to people living there.

Setting

This case study is inspired by a real volcano: the **Puy de Dôme** (Figure 1). Situated in the central part of France, it is one of the youngest volcanoes in the Chaîne des Puys region of Massif Central.

The volcano is located on a granite plateau (crystalline base), culminating at an altitude of nearly 1000 meters. Its *lava dome*, which is the circular mound-shaped protrusion resulting from the slow extrusion of viscous lava, has the shape of a truncated cone, with a height of 500 meters and a width of 2 km at its base.

Figure 1
The Puy de Dôme

1 GEOMETRY

We considered a planar 2D geometry (only streamwise and vertical component), with upper limit that is four times the height of the volcano and horizontal extension of seven times its base. Indeed, on one hand it was necessary to allow the plume to fully develop, and this would not have been possible with a too small extension.
On the other hand, such a horizontal extension guaranteed that the volcano was placed at a sufficient distance from the left vertical limit, which represents the inlet where the wind comes from. Sticking the volcano to this boundary resulted in an undesired behavior of the solution due to the fact that the wind was not able to develop itself in the lower part of the domain to the left of the volcano.

1.1 FIRST ATTEMPT

Our first attempt was to build an accurate geometry reflecting the true dimensions of the volcano (see Figure 2). Therefore, we fixed an upper limit of 2 km and an horizontal extension of the domain of 14 km. The dimensions of the truncated cone representing the volcano are the ones mentioned in the previous paragraph.

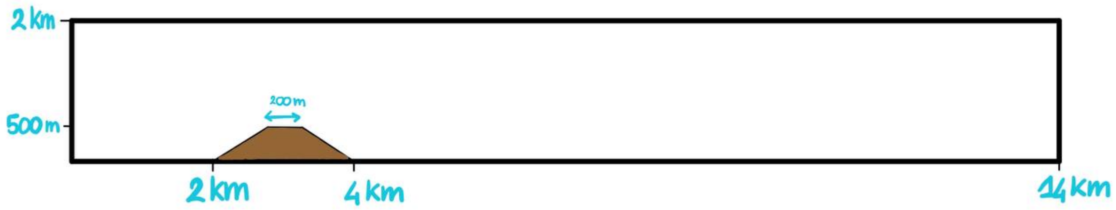


FIGURE 2
Sktech of the geometry, with the real-based dimensions.

By running a first wind simulation without considering the plume, we encountered two main issues:

- lack of convergence of the simulation
- lack of physical meaning of the solution

We managed to solve the first problem with some modifications to the mesh, which will be discussed in subsection 2.1.
The second and more critical issue consisted in the fact that, as shown by Figure 3, the wake after the volcano was too wide, and it did not seem to fade away even very far from the volcano.
The reason behind this is that FLUENT is mainly designed for mechanical or aerospatial applications, where the scale of the geometry is a lot smaller than in our test case. For this reason, for example, the default limits (in the panel `Solution > Controls > Limits...`) are not suitable to work with such a big scale.

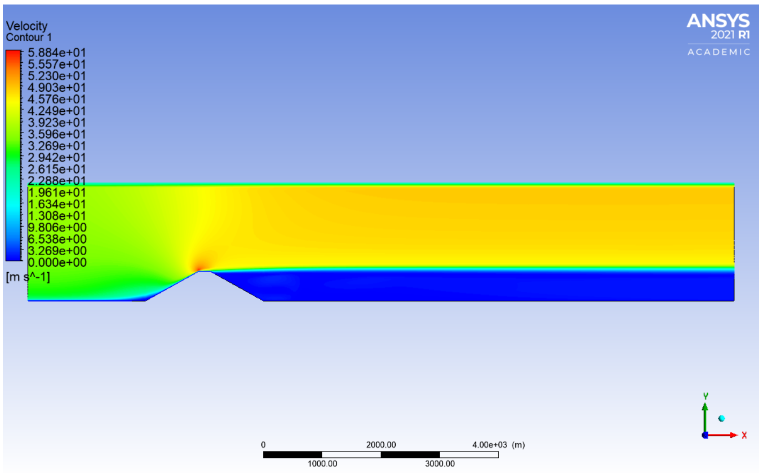


FIGURE 3
Unphysical solution obtained with initial geometry

1.2 CORRECTION OF THE GEOMETRY SCALE

Several fix can be done to solve these issues. For example, it is possible to change the default limits. The solution adopted in this project consisted of a rescaling of the whole geometry: each length has been divided by 100, and we ended up working with scales of the order of meters.
In this case, the result of the exploratory, wind-only simulation, turned out to be coherent with physics, showing an expiring wake.

2 MESH

2.1 STRUCTURE

The mesh that we used for the domain discretization consists of a *structured mesh* made in principle of rectangles.
In order to get a mesh with a resolution that varies with the location on the domain, we have added some lines to the geometry, as shown in Figure 4, which have clearly no physical meaning, but are necessary to separate regions in which generate meshes with different resolutions.

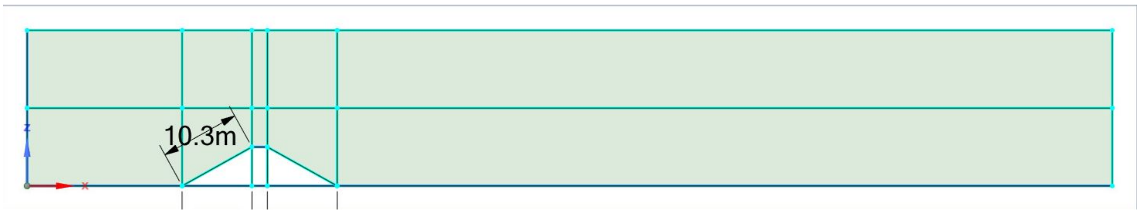


FIGURE 4
Geometry built in ANSYS.

The areas where we are interested in a higher resolution are the ones closer to the volcano, specifically downstream to it. The adopted mesh was obtained with the following choices:

VERTICAL REFINEMENTS

- 30 divisions in the half upper part of the domain, with a bias of 1.05 that makes the central divisions thinner and enables a continuity in cell size with the lower part.
- 60 divisions, with no bias, in the half lower part; note that the higher resolution on the volcano crater is granted by the fact that we still use 60 divisions, but on a shorter edge in that region.

HORIZONTAL REFINEMENTS

- 30 divisions and no bias in the left part of the domain, before the volcano and on its slopes.
- 5 divisions and no bias on the volcano crater.
- 150 divisions with a 1.01 bias in the downstream region to the volcano; the bias ensures a slightly higher resolution close to the plume inlet, where we expect to have more turbulence.

2.2 CONTROL OF THE BASIC REQUIREMENTS FOR ABL FLOW SIMULATION

Since our case study falls into the category of CFD simulations in the lower part of the *Atmospheric Boundary Layer* (ABL), inspired by [1], we have checked that an accurate description of the flow near the ground surface is guaranteed by our mesh.
In particular, we built a mesh that respects the following two requirements:

1. the mesh resolution in the vertical dimension close to the bottom of the computational domain is sufficiently high. Indeed, the height of the cells adjacent to the terrain is $0.17m$, and we managed to get a higher resolution close to the ground than in the upper part with the above choice of refinements.
2. the distance of the centre point P of the wall-adjacent cell to the bottom wall (y_P) is larger than the physical roughness height of the terrain ($k_{S,ABL}$).
   In our case:
   $$y_P = \frac{1}{2} \cdot 0.17m = 0.083m$$
   and the value of $k_{S,ABL}$ was computed as reported in [1], using the formula which holds for Fluent:
   $$k_{S,ABL} = \frac{9.793 y_0}{0.5} = 0.02m$$
   We can see that the condition $y_P > k_{S,ABL}$ is fulfilled.
   The value of $k_{S,ABL}$ was specified in the *wall conditions*, which are described in section 3.2, and for the volcano a slightly different value was chosen, in order to represent a change in the roughness of the terrain.

2.3 ASSESSMENT OF MESH QUALITY

We evaluated the quality of the above described mesh using ANSYS built-in tools for its study. In particular way, we focused on the *cells aspect ratio*, the *orthogonal quality* and the *skewness*.

- **Aspect Ratio** measures the stretching of cells, and it is better to avoid values higher than 5 far for walls and 10 in the boundary layer.
  In our case, the maximum aspect ratio is 8.49, and the few cells with values higher than 4.5 are far from the volcano.

- **Orthogonal Quality** relates to how close the angles between adjacent element faces (or adjacent element edges) are to the optimal angle of $90°$, and 1 is the optimal value; in our case, we are extremely close to it as the average is 0.99 and the minimum is 0.87.
- **Skewness**, defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume, must be analyzed because highly skewed cells can decrease accuracy and destabilize the solution. In our case, the maximum value is 0.323 and the average is $2.80 \cdot 10^{-2}$, therefore the general rule of keeping it below 0.95 is respected with large margin.

3 MODEL

3.1 TURBULENCE MODEL

This case study cannot correspond to a laminar flow, due to the high Reynolds numbers typical of flows in the Atmospheric Boundary Layer (ranging in the scale $10^8 - 10^9$).
Hence, we chose to deal with turbulence with the $k - \varepsilon$ turbulence model, and to solve the equations using the *Coupled* scheme, which ensured a faster convergence.

3.2 BOUNDARY CONDITIONS

INLETS

Our case study is characterized by two inlets:

1. the left vertical boundary, where the **wind** comes from.
2. the **volcano crater**, where the plume comes from.

For the wind inflow, we imposed the fully developed **logarithmic profile** known from theory, following the $k - \epsilon$ model proposed in [1, 5]:

$$U(y) = \frac{u^*_{ABL}}{\kappa} ln\left(\frac{y}{y_0}\right)$$

$$k(y) = \frac{u^{*2}_{ABL}}{\sqrt{C_\mu}}$$

$$\varepsilon(y) = \frac{u^{*3}_{ABL}}{\kappa(y + y_0)}$$

where:

- k is the turbulent kinetic energy.
- ε is the turbulence dissipation rate
- $\kappa \approx 0.42$ is the Von-Karman constant
- y_0 is the aerodynamic roughness length; from [2] we have chosen a value of $y_0 = 0.1m$ (corresponding to a terrain with few trees), and of $0.15m$ for the volcano, then we rescaled it by dividing by 100.
- u^*_{ABL} is the ABL friction velocity; in order to obtain it, we imposed the measured velocity at the top of the volcano:

$$U(y_{top}) = 21.5m/s \quad \text{at} \quad y_{top} = 5m \text{ in the rescaled geometry}$$

and then computed it as

$$u^*_{ABL} = \frac{\kappa U(y_{top})}{ln\left(\frac{y_{top}}{y_0}\right)} = 1.060m/s$$

- C_μ is a model constant, with standard value equal to 0.09 (see [3]).

For the second inflow, i.e. the volcano, we chose a **constant inflow** with varying velocities in the range $30 - 50m/s$.
At first, we were tempted to use a bell-shape inlet profile, but in the end we discarded this option, since the bell-shape profile could be suitable immediately after the exit from the crater and if there were not other emissions from the volcano. But due to the presence of other ejected products together with the plume and subject to the wind velocity, the original profile really does not matter because its shape is quickly "destroyed".

OUTLET

The outlet is a *pressure outlet*: this means that the value of pressure is imposed. We set this value to atmospheric pressure by imposing a Gauge Pressure equal to 0 Pa.

VERTICAL BOUNDARIES

We imposed a *symmetry boundary* on the upper part of the domain, because it enabled us to easily set zero vertical velocity. After all, we are not interested in what happens above that line, so we should not worry about the lack of sense in the physical meaning of symmetry.

On the bottom part of the domain, we imposed a *non-slip condition*, together with the aforementioned roughness of the terrain and of the volcano (section 2.2).

3.3 PLUME MODELLING

After a research on the composition of volcanic plumes, we chose to simulate in this project the **water vapour**, which turns out to be the principal component of plumes being present at even 90% of the total composition in some cases. [4]
In order to simulate a real plume, we activated the `Gravity` option to make it undergo Earth's gravitational attraction as well as the `Energy Equations`, which allowed to take into account the temperature's effect for which a hotter fluid would rise above a colder one.
We also added a `Thermal Expansion Coefficient` for the ambient air (with a value of 0.0034, which is usually set in these temperature conditions), and enabled the `Boussinesq Assumption` for density. Then, we used Fluent's `Scalar` feature to define a *scalar* that would represent the above chosen plume's substance. Its behaviour in the ambient fluid was determined mainly by two parameters:

- `Temperature`: this plays a role in the `Energy Equations` previously mentioned, and influences how high the substance will rise after exiting the volcano. The *scalar*'s temperature was set to 1000K, inspired by [6]. Note that the surrounding air's temperature was set to 300K.
- `Diffusivity`: this indicates how much the substance diffuses in the surrounding medium. Different values were considered for that. The results can be found in subsection 4.1.

The quantity of the *scalar* is defined arbitrarily at the volcano's inlet. The scale is then automatically set within the simulation and on the final contour lines. The actual concentration of the substance can then be derived from the original density and by proportionality.

4 SIMULATIONS

4.1 RESULTS

We report in this section the results of simulations performed by varying the velocity and the diffusivity. We showcase some contour plots with specific values, as well as some trends of the target variables at a particular position, corresponding to an imaginary house close to the volcano. The range of velocities was from 30 m/s to 50 m/s. The diffusivity coefficient ranged from 10^{-1} to 10^{-5}. This range was set by doing a sort of *reverse engineering*: since we did not have a direct way to compute it as the parameter in ANSYS does not correspond to a physical variable, we chose the one providing reasonable results by visual inspection.

VELOCITY

Figure 5 shows the contour plot of velocity magnitude, setting an inlet velocity of 40 m/s at the volcano. On the left boundary, we recover the logarithmic profile, and after the volcano a recirculation bubble with some eddies is generated. Changing the value of the inlet velocities obviously changed the actual scale, but the general profile was similar for the different situations.

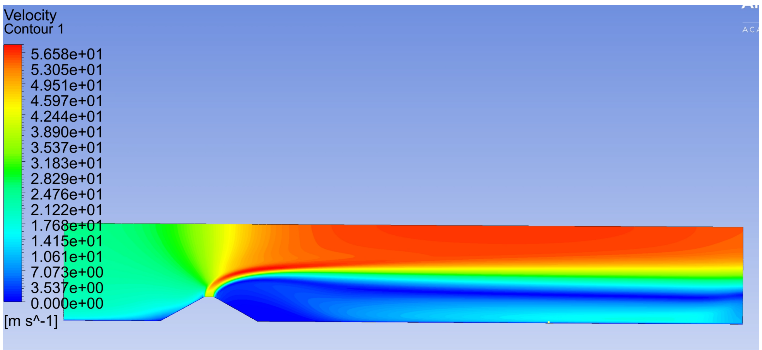
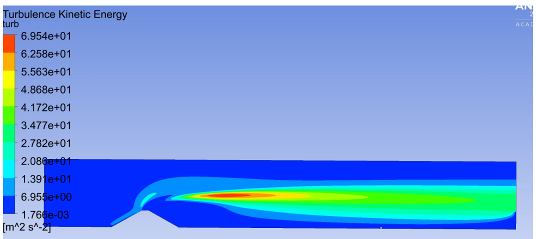


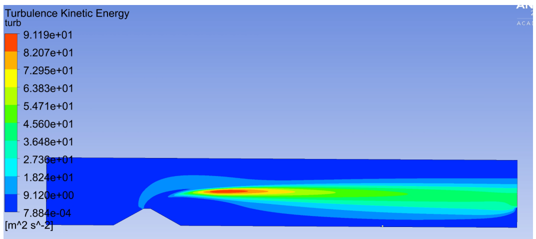
FIGURE 5
Velocity contour plot for inlet velocity of $40m/s$.

TURBULENT KINETIC ENERGY

The behaviour of the turbulent kinetic energy, computed as $\frac{1}{2}(< u'u' > + < v', v' >)$, is shown in Figure 6a and Figure 6b. In particular, we see a peak in a point that is close to the volcano.
We note that, from the comparison between its contour plot for different inlet velocities at the volcano, the bigger the velocity, the further its peak is from the crater. We concluded that it is due to the fact that the position of the biggest clash between wind and plume gets further from the crater as the velocity increases.



(A) Inlet velocity of $30m/s$.



(B) Inlet velocity of $40m/s$.

FIGURE 6
Turbulent Kinetic Energy contour plot for different inlet velocities.

CONCENTRATION

We report in Figure 7 the contour plot of the concentration for an inlet velocity of 40 m/s and a diffusivity coefficient of 10^{-3}. The reference concentration at the inlet is around 0.75 kg/m^3; this value should in fact be the water vapour density (0.804 kg/m^3.), but numerical imprecisions slightly transform it. Negative values are also present but should be considered as 0 (note in fact that their order is negligible compared to the rest).
We also noted that higher values of the diffusion coefficient result in a more important diffusion of the plume; however, as soon as the value becomes very small (as is usually the case in reality), the dispersion is almost nonexistent, and advection clearly prevails over diffusion.

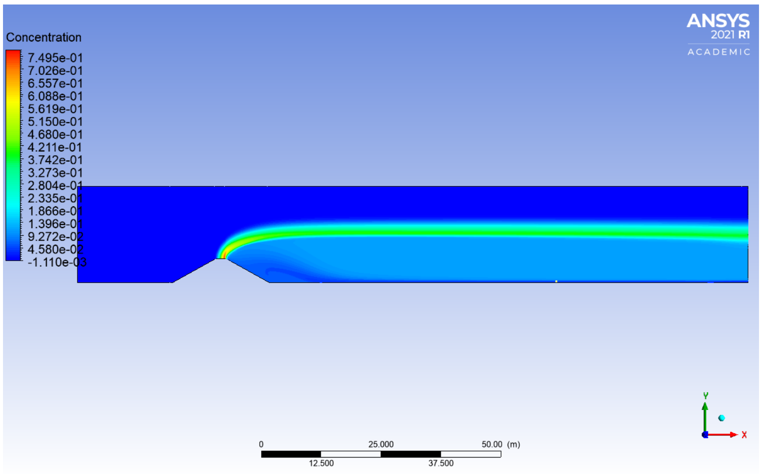


FIGURE 7
Concentration contour plot for inlet velocity of $40m/s$.

TEMPERATURE

We report in Figure 8 the contour plot of the temperature; coherently with intuition, the hot plume has an effect of warming the area downstream the volcano. On the other hand, varying the diffusivity coefficient does not influence the temperature contour plots: we supposed that this is due to the fact that the diffusivity does not appear among the constants of the heat equation, which regulates the heat flux.

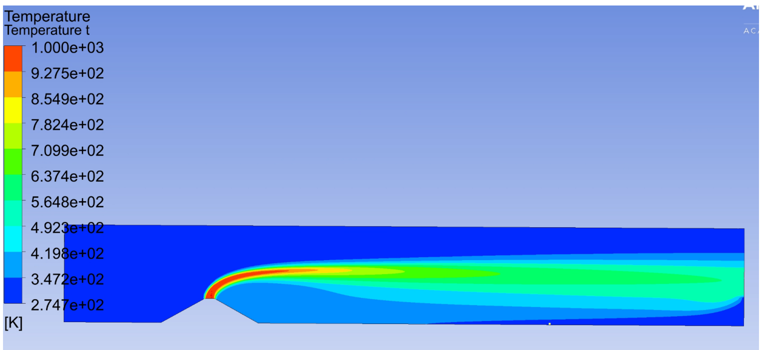
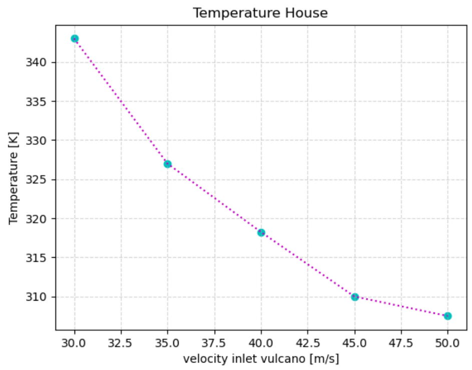
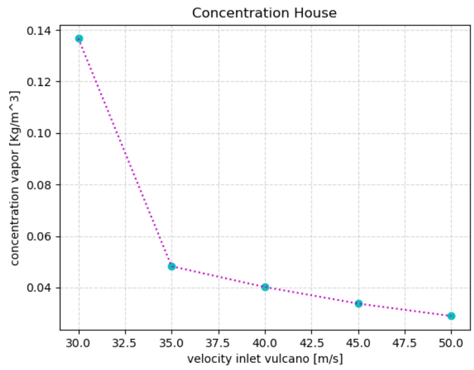


FIGURE 8
Temperature contour plot for inlet velocity of $40m/s$.

TEMPERATURE AND CONCENTRATION AT HOUSE POSITION



(A) Temperature at X=10 km, Z=20 m for different volcano's inlet speed



(B) Concentration of water vapor at X=10 km, Z=20 m for different volcano's inlet speed

VARYING VOLCANO'S INLET SPEED: we inspected the values of temperature and concentration of water vapor at a point where the roof of a hypothetical building could reside. The location of this point is $10km$ and $20m$ far away from the origin of the geometry in the X and Z direction, respectively, (100 and 0.2 m after rescaling). By varying the velocity of the plume at the volcano's inlet from 30 m/s to 50 m/s, we can see from Figure 9a that at lower velocities the temperature increases significantly (43 degrees Kelvin hotter than air), while at higher speed the influence of the plume is not as relevant at such a low altitude ($T = 307.5K$ with $v = 50m/s$, recalling that $T_{air} = 300K$). Similarly, the concentration of water vapor decreases rapidly from $v_{inlet} = 30m/s$ to $v_{inlet} = 35m/s$, and continues to diminish at a lower rate for higher speeds.

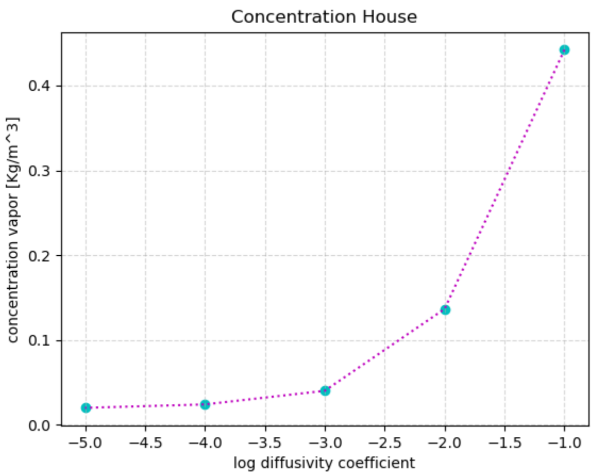


FIGURE 10
Concentration of water vapor at X=10 km, Z=20 m for different diffusivity coefficients, volcano's inlet speed = 40 m/s.

VARYING DIFFUSIVITY COEFFICIENT: we performed a similar analysis to the previous section, considering always the point at X=$10km$, Z=$20m$, varying this time the diffusivity coefficient from 10^{-1} to 10^{-5} and keeping the volcano's inlet speed constant at 40 m/s. As expected, the concentration decreases when lowering the diffusivity coefficient (Figure 10), and remains almost constant for values lower than 10^{-3}.

4.2 LIMITS

Several points can be discussed regarding the accuracy of the performed simulations.
First of all, the case study does not correspond to a real one: the height of the considered volcano is relatively small ($500m$), and such volcanos are usually not active. Note also that the Puy de Dôme is actually located on an elevated plateau (about $1000m$ high), so it is in fact much farther from potential houses; it has simply been chosen because we were looking for a real counterpart of an ideal small volcano.
The geometry itself is also a bit inaccurate, as the volcano is modeled as a perfect cone, with its basis making a sharp angle with the ground. This can impact the behaviour of the flow in an unphysical way, which can differ from reality where the surface change is smoother and more rounded.
Secondly, using Fluent's `Scalar` can be a good first approximation but better models could be considered to improve accuracy. In particular, we only studied a plume made of water vapour, while volcano gases are usually composed of several different substances with different densities and diffusivities in the surrounding air.
Thirdly, the enforcement of boundary conditions can yield inaccurate results for the plume's concentration at ground level. More precisely, by looking at the concentration contour lines, we can see a fine band (the first grid cells) where the concentration is completely 0, while it is non negative (sometimes very important) directly above. To cope with that, we measured the quantities at a point higher than the ground. A finer mesh resolution or a better dealing of boundary conditions could solve this problem.
Fourtly, the behaviour of the plume is a bit different from what we could expect. In fact, it does not rise as it travels, while this is usually the observed behaviour in reality. This can come from the fact that the wind velocity is too important. Alternatively, this could also depend on the considered substance.

5 CONCLUSION

In this project, we analyzed the advection and diffusion of a volcanic plume in the atmospheric boundary layer, using as a test case an idealized volcano. After a first attempt at recreating an accurate geometry, we rescaled it to get a ground simulation without the plume with physical sense. We then built a high-quality structured mesh, and performed simulations aimed at studying the plume behavior varying inlet velocity at the volcano crater and diffusivity of the plume substance. Finally, we analyzed what happens in terms of concentration and temperature at a point close to the surface, representing the position of a house. Coherently with intuition, concentration increases with diffusivity and decreases with exit velocity, while we noted that temperature is mainly affected by exit velocity. In most cases it can be concluded that advection greatly prevails over diffusion in this configuration: the plume is mainly transported away and will most likely dissipate. Turbulence kinetic energy and velocity profiles do not change much and their contour plots are basically the same in all cases. It should be noted, however, that the limited physical accuracy of these simulation does not allow to realistically transfer the results to a real situation. We then discussed several possible fixes to consider in order to make this study more relevant.

BIBLIOGRAPHY

[1] B. Blocken, T. Stathopoulos, and J. Carmeliet. CFD simulation of the atmospheric boundary layer: wall function problems. *Atmospheric Environment*, 41(2):238–252, 2007.

[2] Saheb D., Koussa M., and Hadji S. Technical and economic study of a stand-alone wind energy system for remote rural area electrification in Algeria. *Renewable Energy and Power Quality Journal*, 10(12), 2014.

[3] BE. Launder and DB. Spalding. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, 3:269–289, 1974.

[4] Hawaiian Volcano Observatory. Volcano watch: can we see volcanic gases?, 2018. https://www.usgs.gov/observatories/hvo/news/volcano-watch-colorful-plumes-can-we-see-volcanic-gases.

[5] P.J. Richards. Computational modelling of wind flows around low rise buildings using phoenix. 1989.

[6] T. Roberts, G. Dayma, and C. Oppenheimer. Reaction rates control high-temperature chemistry of volcanic gases in air. *Frontiers in Earth Science*, 7:154, 2019.

[7] T. Simkin and L. Siebert. *Volcanoes of the World*. 1994.