

Study of the flow of the pollutant emitted by cruise ships in Venice

Francesco Sala, Leonardo Trentini, Nicolò Viscusi

Supervisor: Arslan Salim Dar

EPFL, Switzerland

Abstract

The aim of this project is to assess if the pollutant produced by the funnels of cruise ships may reach (and hence potentially harm) people in an inhabited maritime city. In particular, we focused on a 2D adaption of a real scenario, the S. Marco square in Venice, which is subjected to the frequent transit of medium and large-sized cruise ships. We studied two different heights of the source above the ground, and two different geometries were used to model the source of the pollutant. Results show that, for the considered geometry, the square will be affected by the presence of the pollutant only if the source is located at ground level. We also found out that a zone of recirculation takes place over the square, showing that a harmful species could potentially remain among inhabitants for a significant amount of time. However, such results are strongly affected by the limits of the 2D simulation and cannot be considered reliable results in the 3D world.

I. INTRODUCTION

Cruise ships in Venice have been constituting the cause of debate because of the pollution and damage that they may cause to the local ecosystem. In this paper, a simplified 2D cross-section of such topography was considered, as shown in fig. 1. In the adopted geometry, only three buildings were considered to study the flow of pollutants toward S. Marco's square.

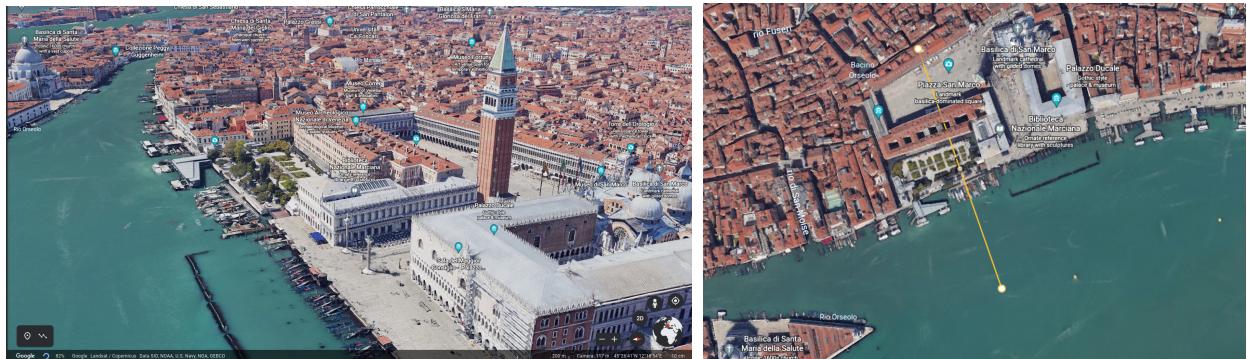


Fig. 1: View of the considered topography. The right image shows the line that defines the cross-section used for the simulation. Snapshots taken from Google Earth.

An unstructured mesh was employed to correctly capture the behavior of the emitted pollutant. The source of the pollutant was first modeled as a line source, and subsequently as a circular source, in order to find the most realistic model. In this project, after a brief presentation of the parameters of the employed CFD simulation, we present the most relevant results.

II. MODELS AND METHODS

The geometry of the problem is shown in fig. 2 as implemented in Ansys SpaceClaim. The adopted lengths were taken from Google Earth, although roundings were added. In particular, the domain is 150 m

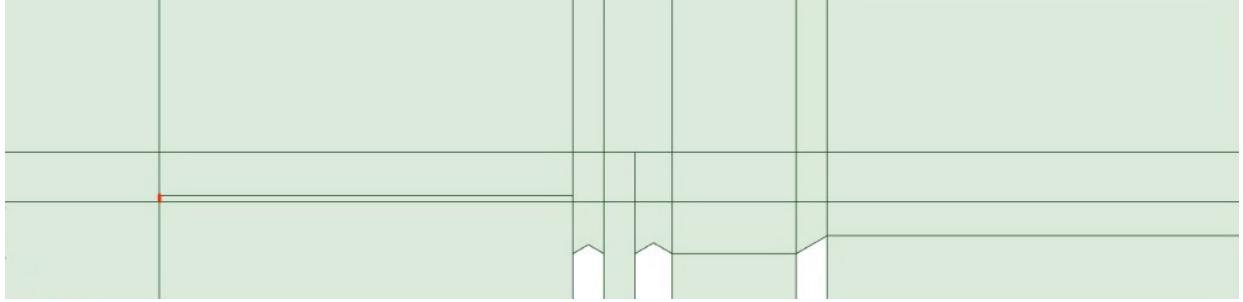


Fig. 2: Geometry of our case study. Note the source modeled as a line 3 m long (highlighted in red).

high and 600 m wide. Buildings were modeled as polygons of height 25 m (roof excluded), and the width fixed at 15 m, 18 m and 15 m (moving from the left to the right).

The horizontal distance of the source (i.e. ship funnel) from the first building d was chosen in order to match that of the usual path followed by ships that cross the *Canal Grande*, $d = 200$ m. Note that the original topography was significantly simplified. In addition, we decided to add 200 m on the right side of the last building, to allow the plume of the pollutant to develop completely.

A. About modeling of the source of the pollutant

The point source of the pollutant was initially modeled with a line of length 3 m (as highlighted in fig. 2; this is the typical height of the upper part of a funnel), whose bottom point was positioned at a height of $h = 50$ m, which is a plausible height above the sea of the funnels of a cruise ship. At a later stage, the source was moved to the ground level. This was done in order to better understand what may happen when a small-sized ship releases pollutants into the air and to see if a pollutant released from such a position could reach the region above the square more easily. Finally, in order to overcome the singularity issues caused by the line source, we decided to remodel the source as a circular element of radius 1.5 m, once again located at a height of $h = 50$ m. As a pollutant, after a brief research on the composition of pollution from cruise funnels, CO₂ was taken into account, as it turned out to be one of the major potentially toxic components of the exhaust of a cruise ship (see [4] for insights).

B. Mesh

The fluid domain was meshed taking into consideration the importance of having a sufficiently fine mesh in the boundary layer: near all the wall constraints, inflations were used. An unstructured mesh was adopted, and face sizings were used in the regions around the three buildings (i.e. the square), exploiting the rectangles visible in fig. 2, to ensure a proper solution of the flow. The resulting mesh for the circular source at height $h = 50$ m is shown in fig. 3. The quality of the mesh was assessed by means of both the orthogonal quality and the skewness. The final mesh has skewness smaller than 0.7 for each element, and an orthogonal quality always greater than 0.4. Topologically identical meshes were adopted for the different sources of the pollutant considered. All such meshes have approximately 5×10^4 elements.

C. Turbulence model

The Atmospheric Boundary Layer (ABL) is inherently turbulent. As the Ansys 2021 guide states, the $k - \varepsilon$ model is valid “(...) for fully turbulent flows” [5]. Therefore, the CFD simulation was carried out adopting the $k - \varepsilon$ realizable turbulence model. For what concerns the aerodynamic roughness y_0 , an appropriate value has to be imposed, depending on the particular surface under consideration. For the present case, a value of $y_0 = 0.25$ was considered consistent with the morphology of the local environment¹, based on the values provided in [1] for different surfaces.

¹Having also taken into account that the aerodynamic roughness is always smaller than the height of individual roughness elements.

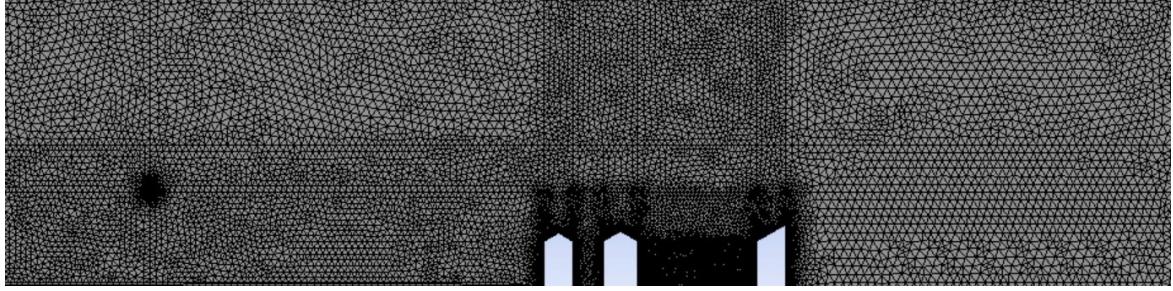


Fig. 3: Resulting mesh, circular source at $h = 50$ m. Note how the mesh was refined near the buildings and near the source.

Additionally, one of the basic requirements for ABL flow simulations (as [3] states), is a “(...) distance y_P from the center point P of the wall-adjacent cell to the wall (bottom of domain) that is larger than the physical roughness height k_S of the terrain ($y_P > k_S$)”. Furthermore, [3] also provides a relationship to estimate the equivalent sand-grain roughness from the aerodynamic roughness:

$$k_{S,ABL} \approx 30 y_0 = 7.5 \text{ m} \quad (1)$$

Therefore, such a value was considered when setting the boundary condition for the ground to correctly solve the boundary layer.

D. Boundary conditions and dummy simulation

On the inlet, i.e. left edge of the domain, the velocity profile was imposed. To do so, following the procedure described in [2], a dummy simulation was run, in order to estimate the value of y^+ of the logarithmic velocity profile. The value of y^+ is defined as follows:

$$y^+ = \frac{yu_\tau}{\nu} \quad (2)$$

where $u_\tau = \sqrt{\frac{\tau_w}{\rho}}$ is the friction velocity, y is the distance from the wall and ν is the kinematic viscosity. In the atmospheric boundary layer, the mean velocity \bar{u} follows the logarithmic law:

$$\frac{\bar{u}}{u_\tau} = \frac{1}{\kappa} \ln \left(\frac{y}{y_0} \right) \quad (3)$$

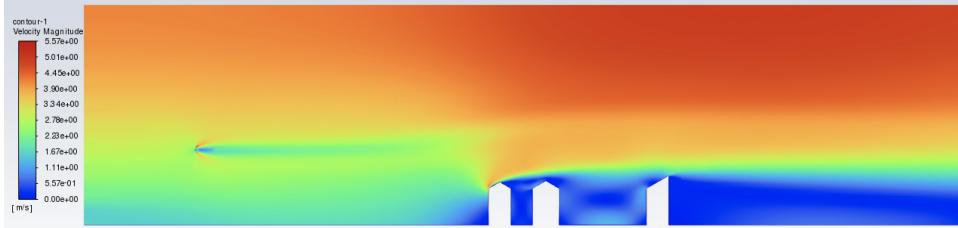
where κ is the *Von Karman* constant. For the dummy simulation, both the inlet and the outlet were made periodic, and a uniform inlet velocity profile of 5 m/s was imposed². Such a simulation was run letting a maximum number of iterations equal to 500, to ensure proper convergence of the boundary layer velocity profile. Once the solution converged, profiles of velocity, dissipation rate (ε) and turbulent kinetic energy (k) were exported and used in the subsequent simulations as inlet boundary conditions.

For what concerns the remaining boundary conditions, a condition of symmetry was imposed on the upper boundary, so as to model an infinitely high domain, while the inlet velocity for the source was set equal to 2 m/s, with a diffusivity coefficient $D = 1.6 \times 10^{-5} \text{ m}^2/\text{s}$.

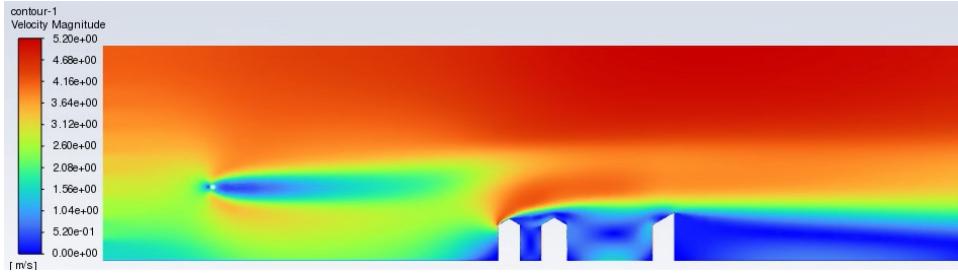
III. RESULTS

For each of the aforementioned scenarios, a CFD simulation was run. Figs. 4 show the velocity contour for the line source at $h = 50$ m, for the circular source at $h = 50$ m and for the line source at ground level.

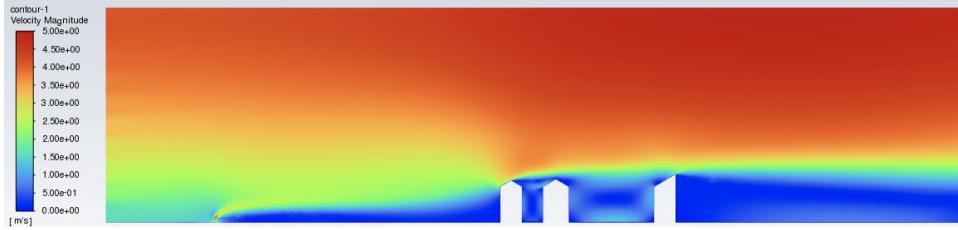
Figs. 5 show the behavior of the turbulent kinetic energy in the same settings. This highlights the singularity caused by the line source.



(a) Line source at height $h = 50$ m.

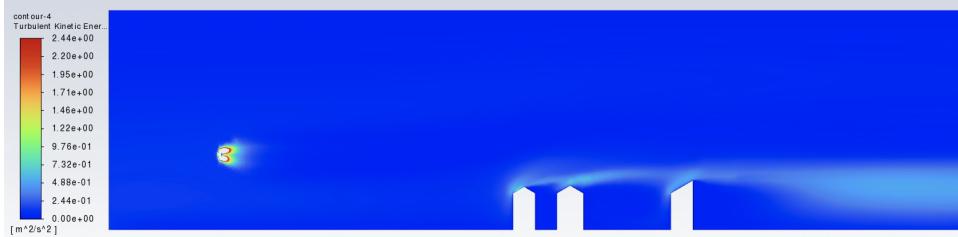


(b) Circular source at height $h = 50$ m.



(c) Line source at ground level.

Fig. 4: Velocity magnitude of the air for the three different sources taken into account.



(a) Line source at height $h = 50$ m.



(b) Circular source at height $h = 50$ m.

Fig. 5: Turbulent kinetic energy varying the geometry of the source. Note that in 5a, the value of the turbulent kinetic energy reached at the source is significantly greater than the one in 5b.

Finally, figs. 6 display the contour plot of the mass fraction of the pollutant.

²Such a value refers to the typical wind speed reached in the region during daytime.

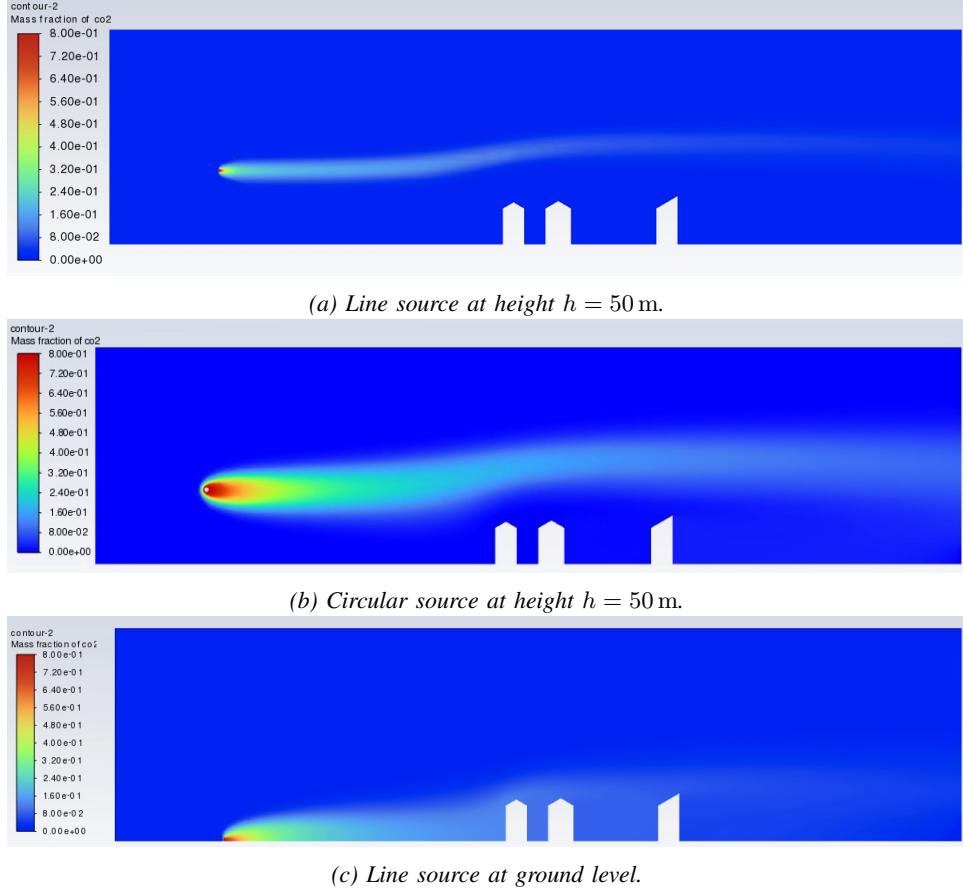


Fig. 6: Mass fraction of released CO_2 for the different scenarios.

IV. DISCUSSION

Figs. 4 and 6 show that the pollutant will reach the region between the two buildings to the right (i.e. the square) depending on the position of the source: as it is visible in fig. 6c, only in the case of a ground-level source of pollutant a small mass fraction of CO_2 will reach the square. This is because, for $h = 50$ m, the pollutant will simply pass above the square, and its flow will be also deflected by the presence of the buildings. All figs. 4 show that the streamlines of the air are deflected upwards by the first building to the left, thus preventing the pollutant from reaching the square, and shielding all the downstream regions. In such contours is also clearly visible the imposed logarithmic profile on the inlet boundary. It must be added that the 2D simulation is not able to fully capture the real 3D phenomenon: the real geometry will likely allow the pollutant to reach the people in the square, especially in a windy and more turbulent scenario³, even from a height of 50 m. Another interesting result is that, as shown in figs. 4, a region of recirculation takes place in the volume between the second and third building: the corresponding eddy can be seen in the contour plot. A pollutant may thus be initially transported by the air and remain in that region for a certain amount of time, thus becoming potentially harmful.

The modeling of the source of the pollutant led us to understand that a circular source will yield a more realistic result: as long as the source was a line, a singularity was introduced with regard to the turbulent kinetic energy. This is highlighted in fig. 5a, where the range of the contour had to be clipped to be equal to the one in fig. 5b, despite reaching values of $46 \text{ m}^2/\text{s}^2$ at the source. In general, the contours of the

³Turbulence is an inherently 3D phenomenon: the tilting and stretching term of the vorticity equation is equal to 0 in 2D, and thus the phenomenon of turbulence cannot be thoroughly captured in 2D.

turbulent kinetic energy show that the irregularities introduced by the buildings are a major cause of the production of turbulent kinetic energy. Moreover, in figs. 6 it is clear how the flow of the pollutant is strongly affected by advection over diffusion: the pathlines of the particles of CO₂ are mainly generated by the local air velocity and the CO₂ will only partially diffuse in the environment as a result of molecular or turbulent diffusion.

Finally, a few points can be made regarding the accuracy of the performed simulations. The 2D geometry is slightly inaccurate, as the buildings are modeled as polygons, with sharp angles. Furthermore, different physical models could be considered to improve accuracy. In particular, we studied a pollutant made only of CO₂, while the exhaust of a cruise ship is usually composed of several distinct toxic components, with different densities and diffusivity coefficients. A study of the local wind direction and strength could further improve the implemented model.

V. CONCLUSION

We analyzed the behavior of the pollutant released by cruise funnels close to an inhabited maritime place: San Marco square in Venice. As predictable, 2D simulations show that in most cases, the pollutant will not reach the region taken into account. That said, a region of recirculation is generated above the square, and pollutants may thus remain above the square for a long time due to this phenomenon.

Further work could focus on a 3D geometry, to better understand the potential harm caused by the transit of cruise ships in such a scenario. Additionally, a more realistic geometry could be implemented, maybe considering the presence of several trees in front of the first building (as can be seen from fig. 1). Different species of pollutants may also be considered, and further studies on the source may be realized. In light of the given results, we suggest the usage of a circular (spherical) source to model the ship funnel.

REFERENCES

- [1] R. Stull, *An Introduction to Boundary Layer Meteorology* (Atmospheric and Oceanographic Sciences Library). Springer Netherlands, 1988, ISBN: 9789027727688. [Online]. Available: <https://books.google.com.cu/books?id=BK8P9rl-CB4C>.
- [2] P. Richards, “Computational modelling of wind flows around low rise buildings using phoenix,” vol. 9, Dec. 1989.
- [3] B. Blocken, T. Stathopoulos, and J. Carmeliet, “Cfd simulation of the atmospheric boundary layer: Wall function problems,” *Atmospheric Environment*, vol. 41, no. 2, pp. 238–252, 2007, ISSN: 1352-2310. DOI: <https://doi.org/10.1016/j.atmosenv.2006.08.019>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S135223100600834X>.
- [4] I. M. Organization, *Third imo ghg study*, C. Delft, Ed., Dec. 2014. [Online]. Available: <https://www.imo.org/en/ourwork/environment/pages/greenhouse-gas-studies-2014.aspx>.
- [5] *Ansys fluent theory guide*, Ansys Inc., Jul. 2021, p. 50.