Comparing strategies for pedestrian wind comfort and safety around high-rise buildings

Project for course DD2365

Jacob Wahlgren
jacobwah@kth.se

June 9, 2021

Abstract

High-rise buildings may have an adverse effect on the pedestrian environment by causing strong winds at ground level. We investigate what the most effective building design to ensure pedestrian wind comfort and safety is. Different scenarios are compared using a Direct Finite Element Simulation (DFS) method. The mean and max wind speeds at pedestrian height is measured as well as the wind induced drag on the building. The scenario with smaller neighboring buildings stood out by reducing both wind speeds and drag, though the method needs improvement to draw strong conclusions.

1 Introduction

High-rise buildings are prominent features of many urban centers, and they may have an adverse effect on the pedestrian environment by directing strong winds down to ground level [1]. An extreme example of this problem are the "unforseen wind effects" of the Bridgewater Place building in Leeds in the UK that lead to several injuries and one lethal accident in 2011 [2]. Many factors impact the wind effect of a building including height, shape, angle to the wind and surrounding structures. Some post-construction mitigation strategies include shrubbery, trees, fences and podiums [1].

This study aims to evaluate the effect of different strategies for pedestrian wind comfort and safety, and visualize how they impact the airflow. Apart from the pedestrian environment, the wind is also important for assessing the structural integrity of buildings, and therefore the wind induced drag on the building is also analyzed. The research question we investigate is: what are the most effective designs for high-rise buildings to ensure pedestrian wind comfort and safety?

In recent years, many studies have been conducted where real city centers are modeled and pedestrian wind effects are analyzed using computational fluid dynamics (CFD). Some examples are the financial district of Toronto [1], the London Olympic Park [3], and the Eindhoven University campus [4]. Some of these studies also validate their results using real world measurements. In many cities it has become mandatory to analyze the effect on pedestrian level wind environment as part of the design of new buildings [5].

The most common methods to analyze wind are Reynolds-averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) [6]. In a RANS method the Navier-Stokes equations are decomposed into a time-averaged and a fluctuating component. RANS methods are relatively simple and inexpensive compared to LES. In LES, low pass filtering is used to average out small scale information in the Navier-Stokes equations. LES methods achieve higher accuracy than RANS, but require significantly more computational resources and are very sensitive to inflow boundary conditions. See [7] for a comparison of RANS and LES in building simulation.

For this study we use a method based on the more recent Direct Finite Element Simulation (DFS) framework developed by Hoffman et al. [8], which we have used in the course previously.

2 Method

The airflow around a high-rise building is simulated using a DFS method in various scenarios. Assuming a reasonable case of a 100 m tall building and a wind speed of 10 m/s at 20 °C ($\nu \approx 1.5 \cdot 10^{-5}$ [9]), there will be a Reynolds number on the order of $\mathrm{Re} = 1.5 \cdot 10^{8}$. This means that complex turbulent structures develop.

Boundary	Condition	Formula
Ground	No slip	$u = 0$
Sky	Free slip	$(u, n) = 0$
Building walls	Free slip	$(u, n) = 0$
Inflow	Velocity profile	$u = 6 \cdot (\frac{y}{10})^{0.2}$
Outflow	Zero pressure	$p = 0$

Table 1: Boundary conditions in the model, where n is the surface normal and y is the height above ground.

2.1 Mathematical model

The incompressible Navier-Stokes equations are used to model the airflow. We assume the air is incompressible since normal wind speeds are much lower than the speed of sound. Below u is the velocity, p the pressure, and ν the kinematic viscosity.

$$\dot{u} + (u \cdot \nabla)u + \nabla p - \nu \Delta u = f \tag{1}$$

$$\nabla \cdot u = 0 \tag{2}$$

To model free-slip boundaries on the structures, a penalty approach using a skin friction boundary is used like in [8], where the friction parameter β is set to 0. Additionally, to model porous media we use a penalty term based on the Navier-Stokes-Brinkman equations like in [10]. The volume penalization term added in the Brinkman model is called the Darcy drag. It is scaled by a permeability matrix K which indicates how restricted the flow is by the porous medium. For this project the permeability will be the same in all directions and thus K is a scalar matrix.

$$\dot{u} + (u \cdot \nabla)u + \nabla p - \nu \Delta u + \frac{\nu}{K} u = f \tag{3}$$

These differential equations are formulated as a boundary value problem. The boundary conditions are presented in table 1. Generally, the wind is strong at higher altitudes, and this is commonly modeled with a power law with exponent around 0.2 [11]. We assume a moderate wind of 6 m/s (at 10 m height). A plot of the inflow profile is shown in figure 1.

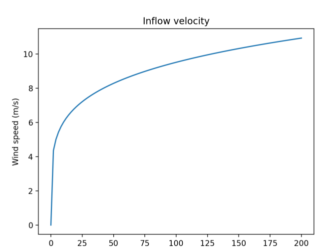


Figure 1: Inflow velocity.

At the Reynolds numbers relevant to this project, small scale vortices have an important effect on the flow. Simulating these directly is intractable in practice due to resource constraints. Instead, we approximate the smallest scales using a turbulence model. Close to walls and the ground a turbulent boundary layer develops. This is approximated by a no-slip condition for the ground and a free-slip condition for building walls. This corresponds to infinite and zero friction respectively, which means that the velocity may be underestimated close to the ground and overestimated close to the walls. For small friction at large Reynolds numbers, the solution is not sensitive to the exact friction constant so a free slip condition is a good model as long as transition to turbulence is not required to be modeled [8]. A free-slip condition was not used for the ground due to issues of stability in the solution. The turbulence in the interior flow is implicitly modeled by the stabilization term SD, which has a dissipative effect [8].

2.2 Numerical method

The DFS method is implemented in a Jupyter notebook using Fencis, based on the lab 2, turbulence and Brinkman template files. Piecewise linear polynomial approximation spaces are used with the stabilization term SD with parameters $\kappa_1 = 4$ and $\kappa_2 = 2$, as recommended by Johan Hoffman in the seminars. The base mesh resolution is 64, with one level of refinement in the area of interest close to the building. The timestep is defined as

$$\Delta t = \frac{0.5h}{15}$$

where h is the minimum mesh size and 15 a characteristic velocity chosen experimentally to achieve stability. With the meshes used in the project we get $\Delta t \approx 0.09$.

Mesh functions in Fencis are used to mark porous zones and building walls. Measures based on these mesh functions are then used in the definition of the residual based variational form, so that the Darcy drag term and the skin friction term are only applied in their intended areas.

During the execution, solutions at different timesteps are sampled and stored in pvd files, and measurements of drag and pedestrian wind speeds are saved in npy files. These files are later visualized using ParaView and Matplotlib respectively.

2.3 Experiments

Experiments are run in several scenarios modeled in 2D. High-rise buildings are modeled as a 100 m tall rectangle with a 25 m base. To allow space for flow structures to develop the domain extends 200 m upstream and 500 m downstream from the building and is 300 m tall. Measurements of wind speed are made in the refined area of interest defined as the rectangle with the vertices $(100, 0)$ and $(450, 200)$. Descriptions of the scenarios are presented in table 2 and the meshes are shown in figure 3. Each mesh has around 18 000 cells.

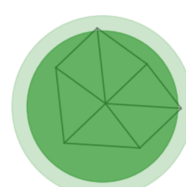
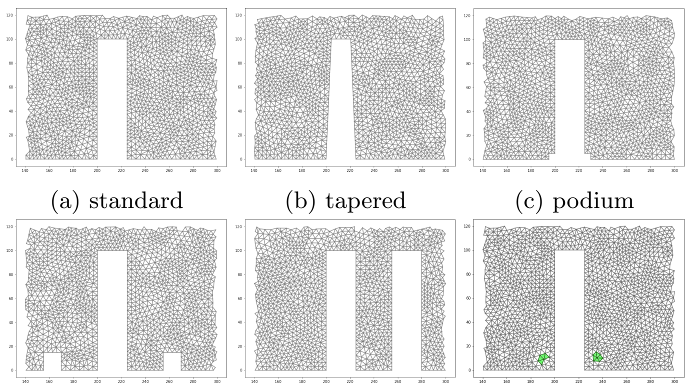


Figure 2: Mesh for the right tree.

The trees are modeled as a porous circular canopy. The trunk is not modeled as it would be relatively thin and have a

Name	Description
none	Empty domain
standard	Rectangular building
twins	Two rectangular buildings spaced 30 m apart
tapered	Building with tapered walls (2.5°)
podium	Building with a 5×5 m podium
neighbors	Building with smaller 15 m buildings at 30 m distance
trees	On either side of building 15 m tall tree with 5 m radius canopy

Table 2: Scenario descriptions.



(a) standard (b) tapered (c) podium

(d) neighbors (e) twins (f) trees

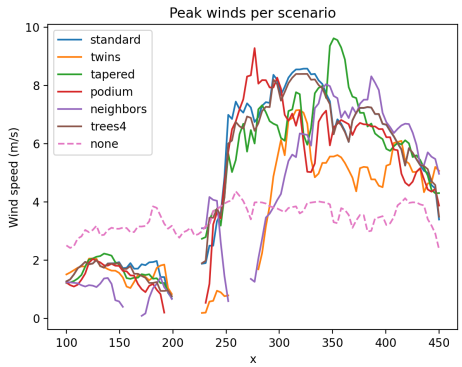
Figure 3: Meshes in each scenario, cropped to region with structures. Green color marks porous regions.

small impact on the flow. The permeability 10^{-4} was chosen experimentally by trying different values such that the flow is not completely restricted, while the trees still have an impact on the results. To mark which elements belong to the canopy, those with all points within the desired radius plus 1 m where chosen, as shown in figure 2. The dark green shows the desired radius and the light green shows the extended radius.

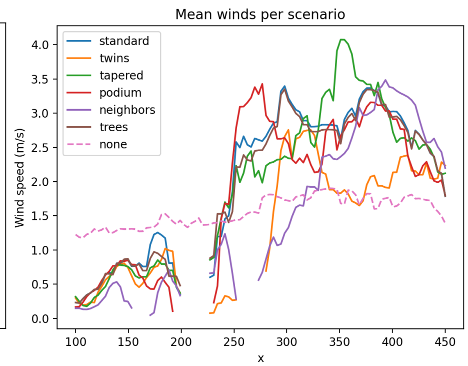
The wind speed is measured at 1.5 m height. Each scenario is simulated for 120 s in simulation time. Longer time periods were attempted to see whether the flow had completely developed at this point, however this was not successful due to divergence issues, even when decreasing the timestep length.

3 Results

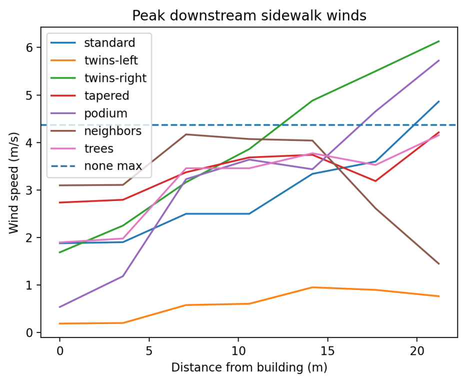
The main results are the plots of pedestrian wind speed in figure 4 and the drag force on the building in figure 5. Measurements of the first 4 s are not presented as they fluctuate wildly with large amplitudes. Each simulation took around 45



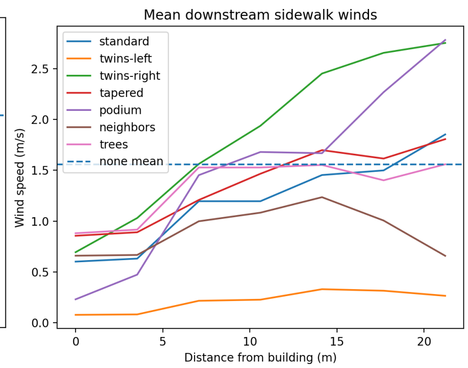
(a) Peak wind overview



(b) Mean wind overview



(c) Peak wind on sidewalk



(d) Mean wind on sidewalk

Figure 4: Measurements of wind speeds at pedestrian height. Discontinuities in (a) and (b) show the location of the buildings. In (c) and (d) the wind just downstream of the building is shown. Note the different scales on the y-axis.

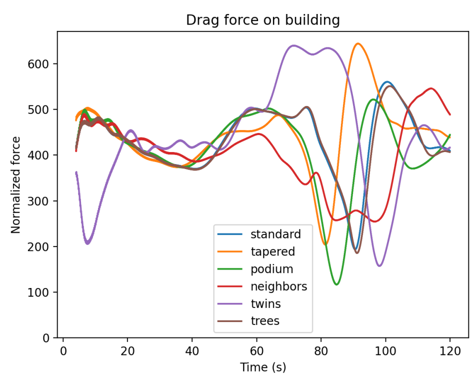
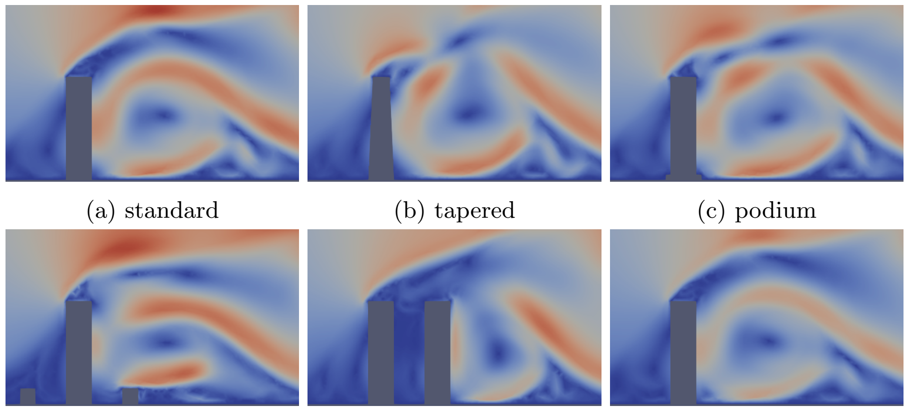


Figure 5: Measurements of the drag force on the building. For the twins scenario the force on both buildings is measured.



(a) standard (b) tapered (c) podium

(d) neighbors (e) twins (f) trees

Figure 6: Visualization of the velocity at $t = 80$ in each scenario.

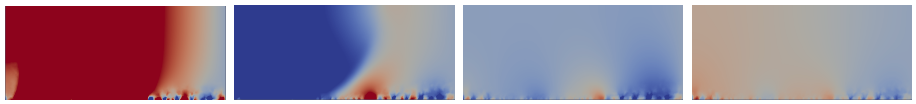
min on a Macbook laptop.

Generally the building causes the wind to slow down upstream and accelerate downstream compared to the none scenario. In the wake of the building, vortices form that move downstream with the flow. The flow pattern is illustrated in figure 6. Subfigures 6a, 6b and 6c show that tapering or adding a podium does not have a large effect on the flow near ground level. From subfigure 6d it is clear that the neighboring buildings limit the vortex so that the high velocity air does not reach ground level. There is almost no flow in between the twin buildings in subfigure 6e, though the wake to the left is similar to the standard scenario. The trees in subfigure 6f only seem to have a local effect on the flow.

Most scenarios cause similar pedestrian winds. The left twin buildings has significantly lower sidewalk winds, however the right twin has among the highest. The building in the neighbors scenario stands out by having low mean winds and a declining wind after circa 15 m. None of the tapered, podium and trees scenarios managed to reduce sidewalk winds, in fact they are comparable or even higher than in the standard scenario.

In the first half of the simulation, the drag forces are similar for all scenarios since vortices have not yet developed. After the first minute, the drag starts to oscillate. Over a period of 10 s, the force changes by a factor of 3 in the tapered, podium and twins scenarios. In the standard and trees scenarios the change is slower and the amplitude slightly lower. The neighbors case has the slowest change and also the lowest amplitude.

Snapshots of the pressure in the none scenario are shown in figure 7. In the first half of the simulation the pressure changes more, while it is more stationary in the latter half. Small points of high and low pressure develop along the ground.



(a) $t = 4$ (b) $t = 40$ (c) $t = 80$ (d) $t = 119$

Figure 7: Snapshots of the pressure in the none scenario.

4 Discussion

Several of the scenarios had similar results, namely standard, tapered, podium and trees. The twins scenario is also similar when considering that the wake is just offset to the right. The differences between the geometry in these scenarios are on a small scale, and there might have been larger differences with a higher resolution mesh. The scenario that stands out from the others is neighbors, where both the wind speeds and drag were reduced compared to the other scenarios, especially close to the building. The neighbors scenario is also the scenario with larger scale additions.

The wind speed was reduced between the buildings in the twins and neighbors scenarios, however this 2D simulation did not take into account street canyon effect. That is, strong winds can also occur due to the "canyon" formed between buildings in an urban environment.

There seems to be a correlation between the drag and the mean and max pedestrian winds. High wind speeds lead to higher drag. All scenarios except for neighbors had large oscillations in drag, which is negative for the structural integrity.

The results for the first 60 s are probably not as useful to analyze since the flow seems to take about a minute before it settles into a pattern. This is evident both when looking at the pressure plot of the none scenario, and the drag force plot. Therefore it would have been good to run the simulation for longer and skip the first minute of the solution.

To ensure that the simulation had time to reach a stable state, it would have been necessary to run it for longer and observe a periodic behavior. The fact that the solver diverged after the presented time period suggests that the flow had in fact not reached a stable state yet. Tentative results indicate that the simulation could be run for at least 200 s by significantly reducing the timestep length to $\Delta t = 0.5h/100$, though this means that the simulation takes over 24 h to run which was too long to finish the project in time.

5 Conclusion

Based on the results of the experiments, the best strategy for pedestrian wind comfort and safety was constructing low-rise buildings near a high-rise building, or several high-rise buildings close together. This reduced wind speeds around the building and oscillations in drag on the structure. However, refined experiments are required to support such a conclusion.

5.1 Future work

Several shortcomings of the simulation limit the conclusions that can be drawn from the experiments. To improve the accuracy of the simulation a higher mesh resolution should be used. Adaptive mesh refinement might be useful for this. The simulation interval should also be longer so as to see if the observed pattern is stable or not over time. Finding a stable model with free-slip boundary conditions for the ground could also improve the accuracy of the method. The accuracy could also be improved by a more physically correct inflow model, i.e. atmospheric boundary layer.

To gain more knowledge about wind strategies, more scenarios could be tried. The results suggest that larger structures have a bigger impact. Additionally, variations of the investigated scenarios could have different outcomes, for instance smaller trees or a larger podium.

Extending the model to three dimensions would allow deeper analysis of the behavior of the wind. At high Reynolds numbers, three dimensional structures develop which cannot be represented in 2D. Additionally, this would allow analysis of factors such as the angle of the wind and street canyon effects.

To evaluate the results and make them more understandable, a real wind comfort model may be used, for instance one of the models in [12]. These models are combine the mean and peak speeds over some time interval.

To investigate the accuracy of the method, the results could be compared to those from other methods. This could be either measurements from wind tunnel experiments or measurements from buildings in the real world.

References

[1] K. Adamek, N. Vasan, A. Elshaer, E. English, and G. Bitsuamlak, "Pedestrian level wind assessment through city development: A study of the financial district in Toronto," *Sustainable Cities and Society*, vol. 35, pp. 178–190, Nov. 1, 2017, ISSN: 2210-6707. DOI: 10.1016/j.scs.2017.06.004.

[2] "Bridgewater Place 'wind tunnel caused Leeds injuries'," *BBC NewsLeeds & West Yorkshire*, Mar. 1, 2013. [Online]. Available: https://www.bbc.com/news/uk-england-leeds-21633206 (visited on 04/20/2021).

[3] A. Hosseinzadeh and A. Keshmiri, "Computational Simulation of Wind Microclimate in Complex Urban Models and Mitigation Using Trees," *Buildings*, vol. 11, no. 3, p. 112, 3 Mar. 2021. DOI: 10.3390/buildings11030112.

[4] B. Blocken, W. D. Janssen, and T. van Hooff, "CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus," *Environmental Modelling & Software*, vol. 30, pp. 15–34, Apr. 1, 2012, ISSN: 1364-8152. DOI: 10.1016/j.envsoft.2011.11.009.

[5] H. Mittal, A. Sharma, and A. Gairola, "A review on the study of urban wind at the pedestrian level around buildings," *Journal of Building Engineering*, vol. 18, pp. 154–163, Jul. 1, 2018, ISSN: 2352-7102. DOI: 10.1016/j.jobe.2018.03.006.

[6] Y. Toparlar, B. Blocken, B. Maiheu, and G. J. F. van Heijst, "A review on the CFD analysis of urban microclimate," *Renewable and Sustainable Energy Reviews*, vol. 80, pp. 1613–1640, Dec. 1, 2017, ISSN: 1364-0321. DOI: 10.1016/j.rser.2017.05.248.

[7] B. Blocken, "LES over RANS in building simulation for outdoor and indoor applications: A foregone conclusion?" *Building Simulation*, vol. 11, no. 5, pp. 821–870, 2018. DOI: 10.1007/s12273-018-0459-3.

[8] J. Hoffman, J. Jansson, N. Jansson, and R. V. De Abreu, "Towards a parameter-free method for high Reynolds number turbulent flow simulation based on adaptive finite element approximation," *Computer Methods in Applied Mechanics and Engineering*, Error Estimation and Adaptivity for Nonlinear and Time-Dependent Problems, vol. 288, pp. 60–74, May 1, 2015, ISSN: 0045-7825. DOI: 10.1016/j.cma.2014.12.004.

[9] Engineers Edge. "Viscosity of Air, Dynamic and Kinematic." (2021), [Online]. Available: https://www.engineersedge.com/physics/viscosity_of_air_dynamic_and_kinematic_14483.htm (visited on 04/23/2021).

[10] J. Fuchsberger, P. Aigner, G. Haase, *et al.* "On the Incorporation of Obstacles in a Fluid Flow Problem Using a Navier-Stokes-Brinkman Penalization Approach." arXiv: 2009.03113 [physics]. (Mar. 15, 2021), [Online]. Available: http://arxiv.org/abs/2009.03113 (visited on 06/04/2021).

[11] H. Mittal, A. Sharma, and A. Gairola, "Numerical simulation of pedestrian level wind conditions: Effect of building shape and orientation," *Environmental Fluid Mechanics*, vol. 20, no. 4, pp. 663–688, 2020. DOI: 10.1007/s10652-019-09716-7.

[12] H. Holger Koss, "On differences and similarities of applied wind comfort criteria," *Journal of Wind Engineering and Industrial Aerodynamics*, Urban Civil Engineering (UCE), Impact of Wind and Storms on City Life and Built Environment. Vol. 94, no. 11, pp. 781–797, Nov. 1, 2006, ISSN: 0167-6105. DOI: 10.1016/j.jweia.2006.06.005.