Research Practicum

Simulating wind around the Flatiron building using RANS CFD simulations

Author:
Ruben Biesheuvel
Marinus Geuze

Supervisor:
Dr. S. Kenjeres

TU Delft
Faculty of Applied Sciences

October 5, 2014

Abstract

Contents

1 Introduction

The Flatiron building, as seen in figure 1, is the iconic Manhattan skyscraper shaped like a right triangle. Clasped between 5th Avenue and Broadway, with Madison Square Park just north-east of him, there is a lot of open space around this building. If the wind comes from the north, it will be forced through an “alley”, creating a windtunnel effect around the Flatiron building. Legend has it that men would hang out out at the corner to watch the wind blowing women's dresses up so that they could see their ankles [2]. This is also shown on a postcard from the early 20th century (figure 2), showing a man being blown away by the wind and a woman's skirt being blown up by the wind.

The main goal of this research is to see if it was actually the geometry of the building creating the updraft. This will be done by simulating the building and surrounding buildings in an in-house built CFD program made for CFD analysis for urban areas.

This report will first discuss the theory and numerical models of the CFD simulation, after which it describes the cases and their results, and furthermore these results will be discussed and a conclusion about the billowing of the skirts will be made.

Figure 1: A picture of the famous Flatiron building

2 Theory

2.1 Navier Stokes

The Navier-Stokes equations describe the balance of forces acting at any given region of a fluid. This set of equations is used for describing the flow

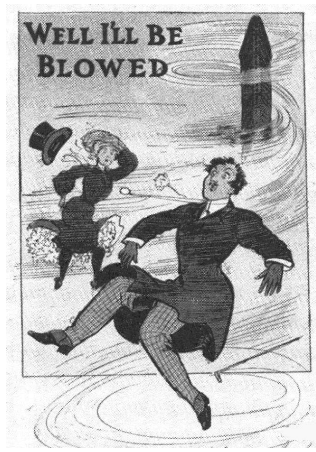

Figure 2: A postcard by an unknown artist displaying the unpredictable winds and billowing skirts around the Flatiron building [1].

of a fluid, such as environmental flow. In the case of an incompressible fluid, which air can be approximated as at low wind speeds, the Navier-Stokes equations can be simplified as:

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_j \partial x_j} + f_i. \tag{1}$$

The left hand side of this equation is the change of momentum of the fluid, where the first term is the rate of change in time, and the second term is the steady acceleration over position. The right hand side of the equation are the forces driving the change of momentum, where the first term is the pressure gradient, the second term is momentum transport due to viscosity, and the third term are other body forces such as a gravity or electromagnetic field.

There is no analytic solution for these equations governing turbulent flow, so the solutions must be found using numeric calculations on a computer. However, it would not be viable to attempt this calculation down to the smallest scale for a large scale airflow such as the flow around a building. The size of the mesh and computational power needed would mean that it would take years to complete with even the most potent supercomputers.

The question arises whether it is really needed to compute the physics down to the smallest level. Where each single small vortex is at each moment is generally not relevant information. The macroscopic effects of turbulent flow is usually what interests the engineer. Since we are only looking for the macroscopic effects of all tiny turbulent phenomena, these effects can be modeled instead of computed to greatly reduce the computational power needed.

2.2 Reynolds Averaged Navier Stokes (RANS)

A common way to simplify the turbulent flow calculations is by separating the mean value and a fluctuating value of the solution. Separating a quantity u in this way is called *Reynold decomposition*, and can be written as:

$$u(x, y, z, t) = \overline{u(x, y, z)} + u'(x, y, z, t). \tag{2}$$

Note that the mean value is not dependent on time. Averaging the fluctuating value over time will return 0: since the mean value of u is already captured in $\bar{u}$, the fluctuating part necessarily fluctuates around 0. By substituting **??** in **??** and averaging the equation, we arrive at the time independent Reynolds averaged Navier-Stokes (RANS) equations:

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial}{\partial x_i}\left[\bar{p} + \mu \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \rho \overline{u_i' u_j'}\right] + f_i. \tag{3}$$

The solution of these equations consists of the mean kinetic energy (flow) and a term indicating the mean turbulent kinetic energy. Since the effects of turbulence are now averaged, it is possible to use a coarser mesh. It is no longer needed to compute turbulence down to the smallest scale.

2.3 The closure problem

When evaluating **??**, the term $\overline{u_i' u_j'}$ governing the turbulent energy is still an unknown. At this point we have three equations, but an extra variables: u_i and $\overline{u_i' u_j'}$.

2.3.1 k-ϵ model

2.4 Finite volume method

The control volume is discretized into finite volume cells. In each of these cells the average intensive properties are computed. The cell size should be fine enough that this assumption is valid. A good balance should be found between higher accuracy with a high cell density, and a higher computation speed with lower cell density.

With every computational step the new values in the equation are computed. The fraction of the difference between the two sides of the equation in the last computational step and the initial values is called the residual. On the long term the residual should become smaller until the calculation converges to the solution.

3 Case and Results

In this section the setup of the CFD analysis will be discussed. This includes the generation of the obstacles for the CFD analysis, a discussion on the Reynolds number associated with this geometry and the fluid properties used during the simulation. Our goal is to see if there is any updraft around 1 meter height, because if there is, that would mean skirts will be blown away in the wind.

3.1 Obstacle creation

To be as true to reality as possible (within the scope of the research practicum), not only the Flatiron building was simulated, but also 4 of the surrounding buildings, creating the "alley" of wind talked about in the introduction and [2]. The dimensions of the Flatiron building itself were found in Bradford and Condit's book "Rise of the New York Skyscraper 1865-1913" [3], while the other dimensions were roughly estimated from Google Maps (see appendix A for a clarification of the method and a complete list of dimensions used).

The program used makes use of an "obstacle generator", in which you specify the start and end of the obstacle in the x, y and z direction, creating rectengular cuboids as obstacles. By combining multiple obstacles together, more complex shapes can be created. By discretizing a right angled triangle, the Flatiron building has been built up of X amount of 0.25 metre wide blocks of varying lenth of the building. The height of the building is uniform.

3.2 Mesh

While the mesh is automatically created by the program used in this experiment, there are some parameters that influence the amount of control volumes. To avoid long computation times and the inability of the computer to post-process the results, it was recommended that the total amount of control volumes should not exceed 2 million.

In order for the ground effect to not affect our results, it has been decided that the minimum height of the control volumes at walls should be 0.1 meter. To keep the amount of control volumes around 2 million, the cell width and length of the cells at the walls were set at 0.5 meters. Due to the linear nature of the buildings in question, the cell expansion factor has been set at 1.25.

The last parameter that was set was the parameter that determines the space in front and after buildings. This has been set at 50 meters for both x, y and z directions. This was done in order to give enough space for the wind to develop and ensure as close to reality results as possible.

This way of obstacle creation and parameters for the mesh results in the following wind tunnel model. Notice the high density of control volumes around the Flatiron building in the x direction, this is to ensure that the effects of the shape of the building are well taken into account. At the base and top of the buildings (both surrounding and the Flatiron) there is also a high density of cells in the z direction, this is caused by the 0.1 meter cell height that was imposed on the mesh generator.

3.3 Reynolds Number

3.4 Fluid properties

4 Results

Because the men would hang around the corner on windy days [2], a windy day was simulated with a wind speed of 9 ms^-1. To get a relatively accurate simulation, a iteration convergence of $1 \cdot 10^-4$ was recommended to us.

In order to adequately judge the wind around the building, an overview is created by plotting the streamlines at 3 different heights. These are shown at the bottom(figure 8), around the middle(figure 9) and at the top(figure 10) of the Flatiron building and can be found in appendix B
To check if women's ankles would really show, a plot of the velocity in the z-direction, the pressure and the turbulent kinetic energy were made at a height of 1.37 meter (this was the lowest the program could go). These a shown in figures x y z.

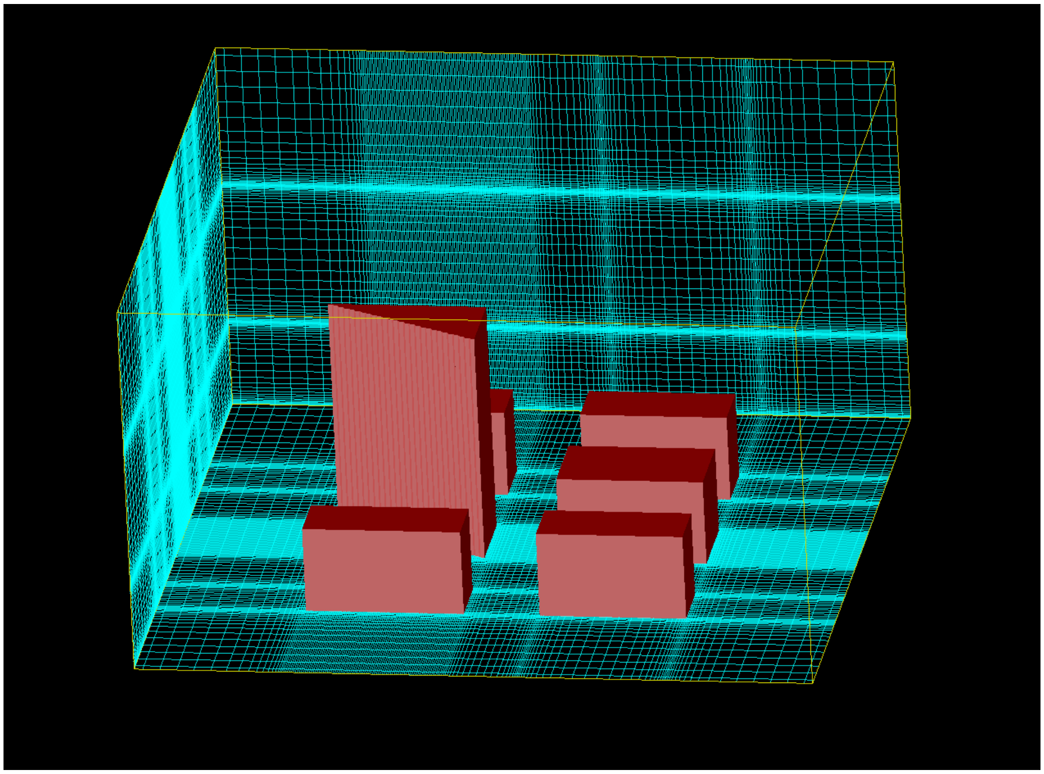

Figure 3: The obstacles and mesh used in the analysis

5 Conclusion and Discussion

5.1 Grid convergence

5.2 Uncertainty

References

[1] Alice Sparberg Alexiou. *The Flatiron*. St. Martin's Griffin, 2010.

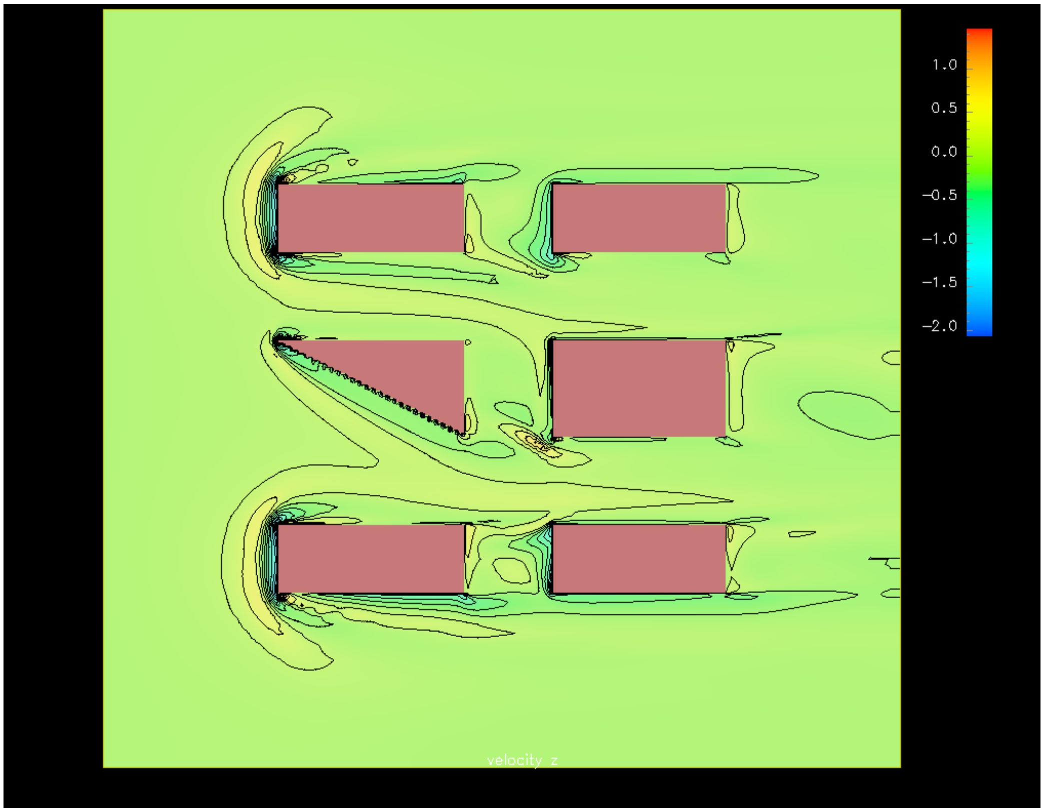

Figure 4: Velocity in the z-direction around the Flatiron building at z=1.37 m

[2] Andrew S. Dolkart. *The Architecture and Development of New York City*. Columbia University, 2014.

[3] Carl W. Condit Sarah Bradford Landau. *Rise of the New York Skyscraper 1865-1913*. 1999.

A Dimensions of the obstacles

As discussed in section 3.1, the obstacles used in the CFD analysis were estimated using Google Maps. In figure 7 it is shown how these distances were estimated. From this figure and the measurement tool on the Google Maps program, we found the following dimensions for the Flatiron building and

Figure 5: Pressure distribution(in Pa) around the Flatiron building at z=1.37 m

surroundings. The numbers correspond with the building numbers in table 1.

Table 1: Parameters used in the generation of the obstacles that are not the Flatiron building, in meters

Building No.	x-length	y-length
1	30	140
2	30	60
3	30	140
Space between	25	25

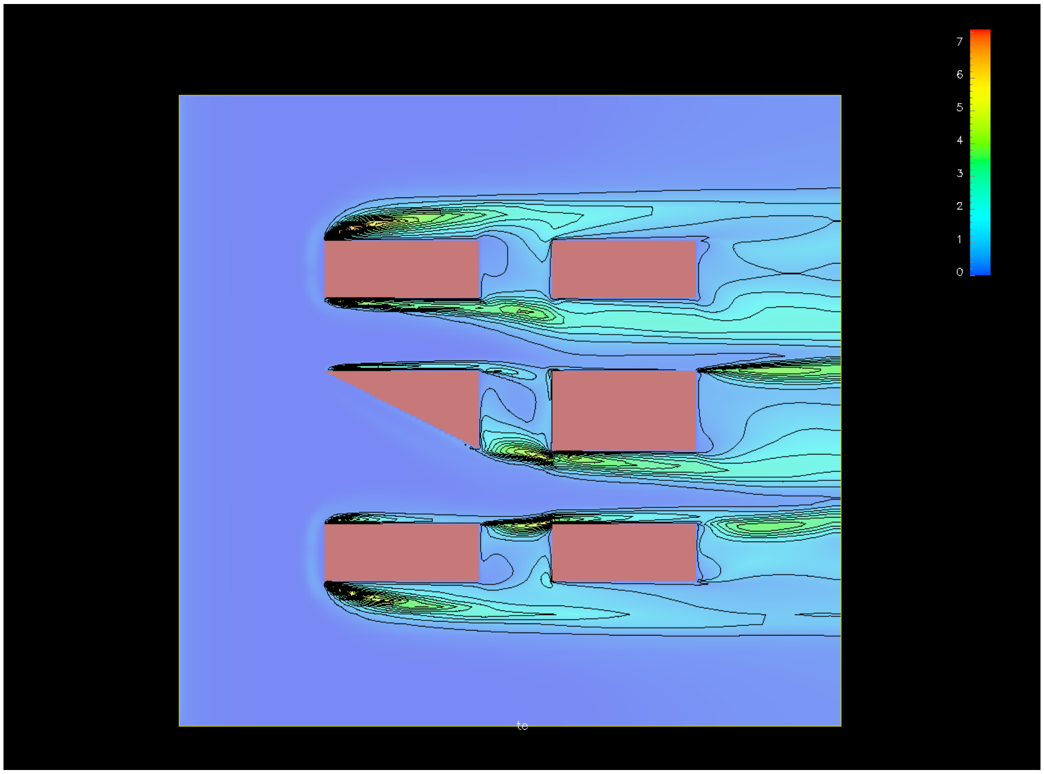

Figure 6: Turbulent kinetic energy(in m^2s^{-2}) around the Flatiron building at z=1.37 m

B Streamlines

In this section the streamlines to get a general overview.

Figure 7: A snapshot of the method used to estimate the dimensions using Google Maps. The red lines indicated the measured distances.

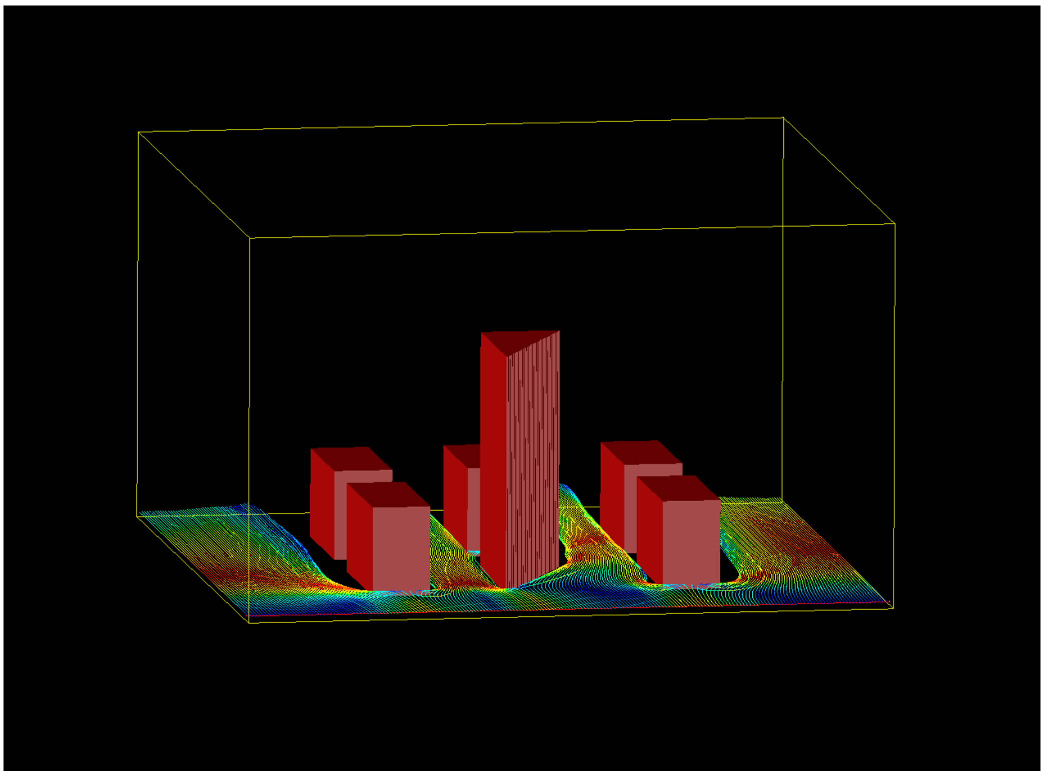

Figure 8: Streamlines around the Flatiron and surrounding buildings just above ground level

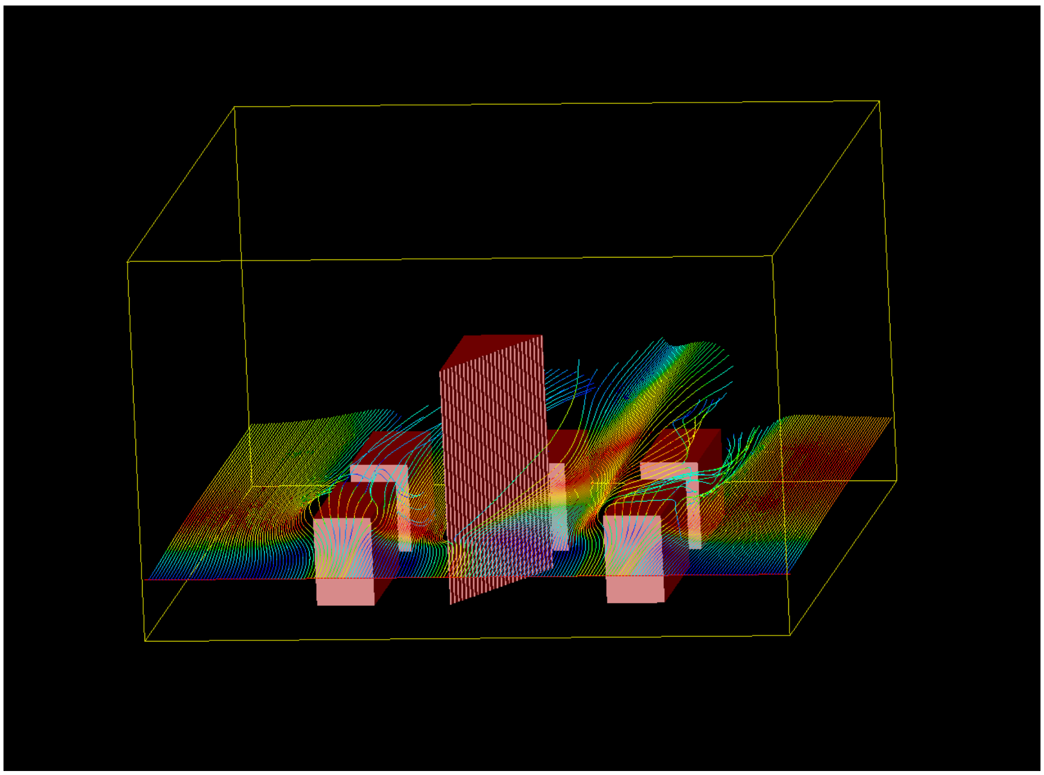

Figure 9: Streamlines around the Flatiron and surrounding buildings just below the top of the surrounding buildings

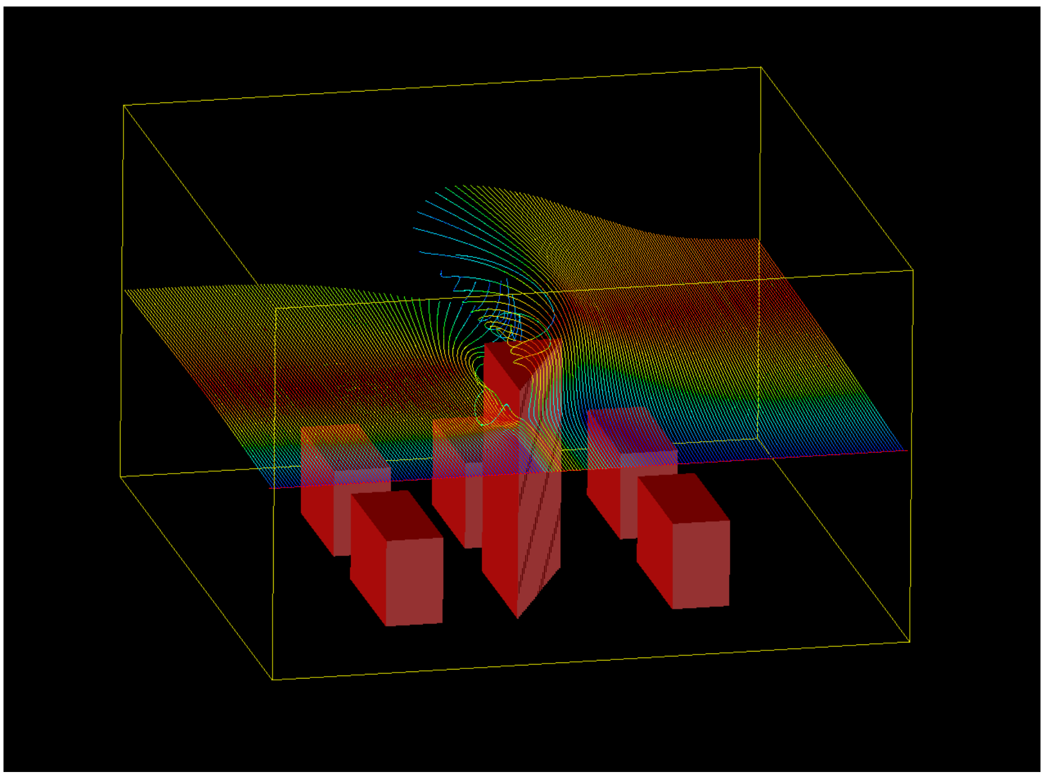

Figure 10: Streamlines around the Flatiron and surrounding buildings just below the top of the Flatiron building