Wydział Mechaniczny
Energetyki i Lotnictwa
POLITECHNIKA WARSZAWSKA

Institute of Faculty of Power and Aeronautical Engineering

Bachelor's diploma thesis

in the field of study Aerospace Engineering
and specialisation Aerospace Engineering

Aerodynamics of Urban structures

Kacper Frelek
student record book number 313668

thesis supervisor
Ph. D. Marta Poćwierz

WARSAW 2023

Contents

1. Introduction . . . 4
1.1. Urban structures Aerodynamics Overview . . . 4
2. Motivation . . . 5
2.1. Purpose of the work . . . 5
3. Model . . . 6
3.1. Real-life problem . . . 6
3.2. Scaled model . . . 6
4. Aerodynamics . . . 7
4.1. Navier-Stokes equations . . . 7
5. Methodology . . . 8
5.1. Finite Volume Method (FVM) . . . 8
5.2. Model . . . 8
6. Solution . . . 9
6.1. Velocity Inlet profile . . . 9
6.2. Turbulence . . . 9
6.3. Geometry . . . 9
6.4. Mesh . . . 10
6.4.1. Coarse Mesh . . . 11
6.4.2. Fine Mesh . . . 11
7. Solver . . . 13
7.1. Solver Settings . . . 13
8. Results . . . 14
8.1. Coarse Mesh vs. Fine Mesh . . . 14
8.2. Fine Mesh . . . 15
8.2.1. velocity and pressure contours for 0 degree flow . . . 15
8.2.2. velocity and pressure contours for 45-degree flow . . . 16
8.2.3. velocity and pressure contours for 90-degree flow . . . 17
9. Conclusion and Comparison . . . 19
9.1. Wind Tunnel Results . . . 19
9.1.1. AOA 0 degree . . . 19
9.1.2. AOA 45 degree . . . 21
9.1.3. AOA 90 degree . . . 23
9.1.4. Summing up . . . 24
9.2. Conclusions . . . 24
References . . . 25
List of Figures . . . 26

Nomenclature:

- U_∞ – Velocity at infinity
- TI - Turbulent intensity
- v_m - velocity at 10 meters
- $\frac{\partial}{\partial t}$ - partial derivative in respect to time
- $\overrightarrow{\nabla}$ - nabla operator
- $\overrightarrow{u}$ - velocity vector
- $\overline{\overline{\tau}}$ - stress tensor
- e - internal energy
- $\overline{\overline{u \otimes u}}$ - tensor of velocities
- s_{ij} - mean strain-rate tensor
- ρ - density
- ν - kinematic viscosity
- H_max - maximum height of the buildings

1. Introduction

1.1. Urban structures Aerodynamics Overview

Accurate determination of local conditions is a significant factor in building aerodynamics. It is recommended, or sometimes it is even a must when changing the infrastructure of a city of big size. Airflow around and between buildings is crucial, as an exchange between old and fresh air controls temperature and humidity and ensures thermal comfort for pedestrians and Noise pollution. Moreover, if the flow is uncontrolled, it can cause wind Loads and structural failure and create a wind tunnel effect, which is essential in terms of structures. Knowing that one can magnify that good aerodynamic properties are crucial factors in urban systems and have to be cautiously analyzed.
This thesis presents a numerical study of some urban area and how creating a high skyscraper would influence aerodynamic conditions in the close neighborhood of other buildings. It could be done using Computational Fluid Dynamics (CFD) method. Due to the improvement in computational power that people can use, CFD is a powerful tool that allows engineers to optimize designs more and more accurately in many fields. Unfortunately, experimental methods, like a wind tunnel, are still better and more worth to be done, so sometimes, to compare results from the CFD model, one can conduct an experiment to notice differences between them.

2. Motivation

2.1. Purpose of the work

The work aims to compare results and find possible differences between the CfD solution and the one obtained before from the wind tunnel. It is important to obtain similar results, an done should aim to get them. Both methods are burdened with different errors, which may cause differences in results.
There is a need to simulate similar conditions as the ones during the test and finally testify whether obtained results were correct or not. The main problem of the work is to create a good enough model and mesh, which with the proper choice of conditions, would give accurate results. The physical value that is going to be compared is a wind velocity and how and where local regions of acceleration occur.

3. Model

3.1. Real-life problem

The described problem is a set of buildings between which one would like to create another much higher than others. The question is, how that would change the conditions of the local environment.

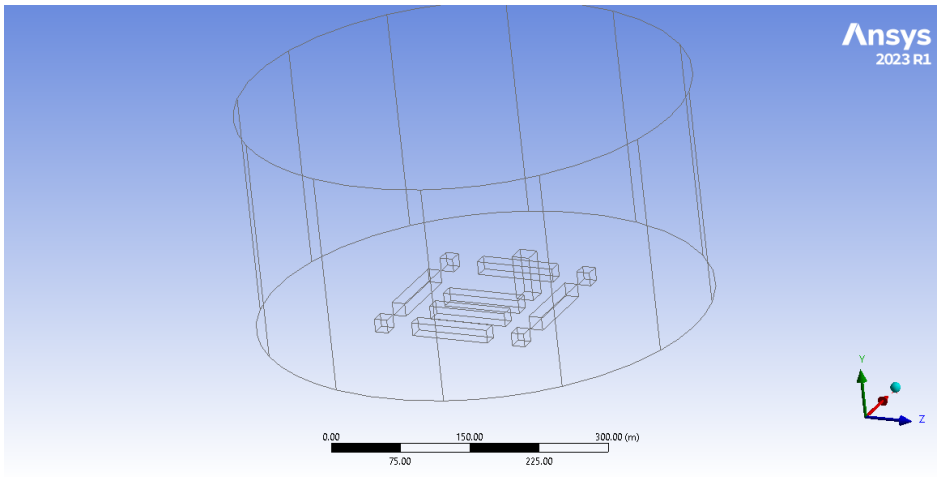


Figure 3.1. Real size geometry

3.2. Scaled model

To perform a comparability test, one has to change conditions so they would match ones in a wind tunnel. The easiest way is to scale the buildings' geometry to the pattern size used during tests. To do that, one had to use a 1:500 scale, so the maximum height of buildings would be 0.6m.

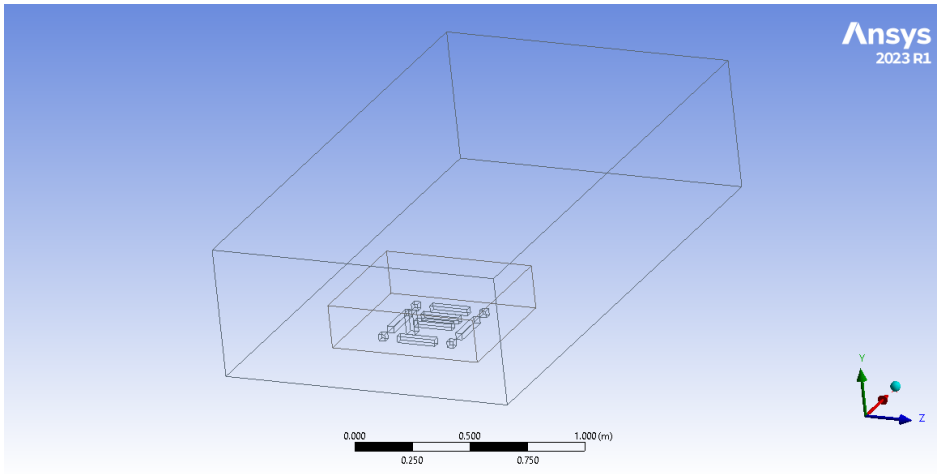


Figure 3.2. Scaled geometry

4. Aerodynamics

4.1. Navier-Stokes equations

Governing equations are three main equations describing conservation laws of mass (continuity), momentum (Second Newton's law), and energy (first law of thermodynamics).

$$\frac{\partial \rho}{\partial t} + \overrightarrow{\nabla} \cdot (\rho \overrightarrow{u}) = 0 \tag{1}$$

$$\frac{\partial (\rho \overrightarrow{u})}{\partial t} + \overrightarrow{\nabla} \cdot [\rho \overrightarrow{\overline{u \otimes u}}] = -\overrightarrow{\nabla p} + \overrightarrow{\nabla} \cdot \overline{\overline{\tau}} + \rho \overrightarrow{f} \tag{2}$$

$$\frac{\partial (\rho e)}{\partial t} + \overrightarrow{\nabla} \cdot ((\rho e + p) \overrightarrow{u}) = \overrightarrow{\nabla} \cdot (\overline{\overline{\tau}} \cdot \overrightarrow{u}) + \rho \overrightarrow{f} \overrightarrow{u} + \overrightarrow{\nabla} \cdot (\overrightarrow{\dot{q}}) + r \tag{3}$$

Because velocity in analyzed conditions will always be less than 0.3 Mach, one can ignore the Energy equation (assume incompressibility of fluid). Moreover, flow is assumed to be viscous, so the above equations would look as follows:

$$\frac{\partial \mathbf{u_i}}{\partial \mathbf{x}_i} = 0 \tag{4}$$

Newton's Second Law (Momentum Equation):

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (2 \nu s_{ij})$$

Where s_{ij} is the strain-rate tensor described by:

$$s_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{5}$$

5. Methodology

5.1. Finite Volume Method (FVM)

At its core, CFD involves the discretization of the fluid domain into a computational mesh. This mesh is composed of small control volumes/cells, and each cell contains properties such as velocity, pressure, and temperature. By applying Navier-Stokes equations, to these control volumes, we can simulate and analyze the fluid behavior.

5.2. Model

Analyzing official papers, works, etc., the RANS model $k-\varepsilon$ is chosen for this simulation. RANS models aim to provide averaged flow properties by considering turbulence as a fluctuating quantity. Assumptions of $k-\varepsilon$ model are that:
Turbulence can be represented by a single scalar variable - turbulent kinetic energy (k), which reflects the random motion and fluctuations of fluid particles within a turbulent flow.
There exists a characteristic rate at which the turbulent kinetic energy dissipates - The Turbulent dissipation rate (ε). It signifies the conversion of turbulent energy into internal energy due to viscous effects.

6. Solution

6.1. Velocity Inlet profile

In real conditions, velocity is not uniform over the whole altitude. For convenience of this project, it is assumed that velocity at the inlet is a profile:

$$V_{\infty}(z) = v_m * (z/10)^{\alpha} \tag{6}$$

The same profile was used in the wind tunnel test. Values of V_m depends on external conditions, and the value of α depends on the category of a terrain. During the project, two categories of terrain will be tested: III and IV. For terrain category III, α coefficient is equal to 0.19, and for category IV it is 0.24.

6.2. Turbulence

For perfect simulation of conditions, one would have to describe Turbulent Intensity (TI), also as a function of 'z'. A good approximation is to describe TI as:

$$TI_v(z) = 1/ln(z/z_0) \tag{7}$$

Parameter z_0, which occurred in the above equation, is also dependent on terrain category. Due to the complexity of describing TI in Ansys Fluent program, the value of it is to be estimated by trial and error. Thank to careful analysis of many cases, with TI varying from 15 to 35, it was established that the closest results are obtained when using TI about 25

6.3. Geometry

Geometry which was analyzed, consists of two cuboids, where one is much smaller than another and buildings inside of small cuboid.

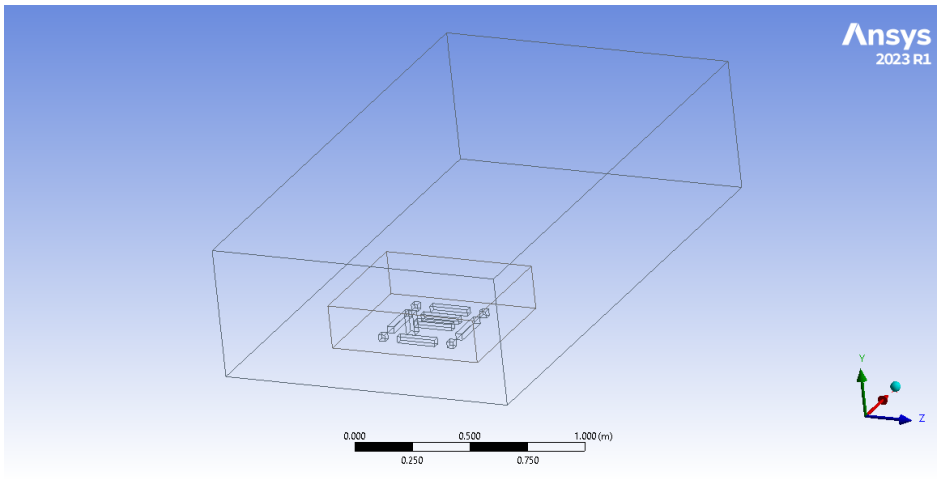


Figure 6.1. Analyzed geometry

From the literature, one can state what are minimum values of such domain.

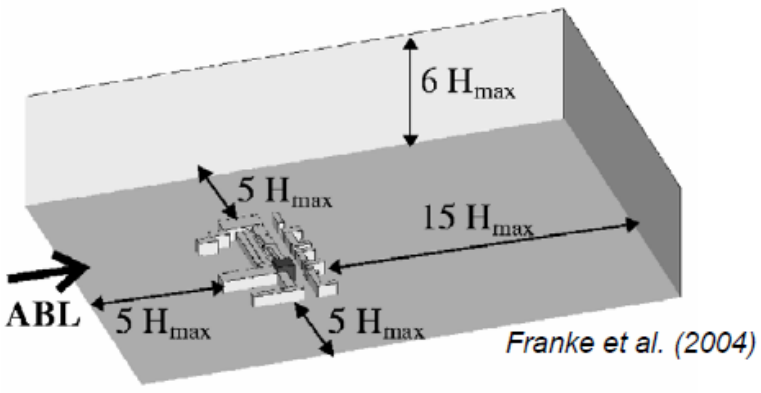


Figure 6.2. Minimum domain size
Hall 1997, Franke et al. 2004, 2007, Tominaga et al. 2008

As the maximum height H_max in this project is 0.1m, one can easily create a proper domain.

6.4. Mesh

To obtain credible results, one has to create a fine enough mesh. From sources available on the web:

- Average Skewness about: 0.60
- Maximum Skewness less than: 0.95 (Preferably less than 0.90)
- Average Orthogonal Quality: 0.5
- Minimum Orthogonal Quality: 0.05 (Preferably more than 0.15)
- Average Aspect Ratio: 10:1

Mentioned above manual states that *'With regard to the stability of the flow solution, it can go as high as possible; however, with regard to the stability of the energy solution, the maximum aspect ratio should be kept below 35:1.'* Due to that fact, one can neglect AR values in this project (incompressibility of the flow).

6.4.1. Coarse Mesh

The first attempt at calculations will be done on the randomly chosen quality mesh.

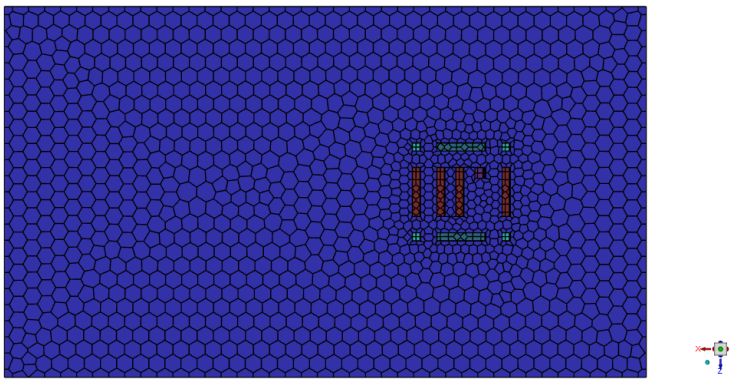


Figure 6.3. Created mesh (coarse)

After obtaining mesh with using the above settings, one can obtain mesh and check, as mentioned in 6.4, some of the important properties:

- number of cells = 23469
- Skewness Maximum = 0.58027814
- Skewness Average = 0.021423092.
- Orthogonal Value Minimum = 0.20509661
- Orthogonal Value Average = 0.92090019
- Average Aspect Ratio = 1.97736
- Maximum Aspect Ratio = 11.8665

6.4.2. Fine Mesh

In performing simulations like this one, it is a need to check for results for better quality mesh. It is done to confirm or deny the correctness of obtained results in coarse mesh simulation. If obtained results would differ, it would be a must to check the mesh quality limit, where results are no longer changing.

By using better options than in the first approach (like the body of influence, denser mesh in regions where one could expect accelerations of air, etc.) said options, one obtains really dense and really precise mesh with the following values:

- number of cells = 12654509

- Skewness Maximum = 0.69987
- Skewness Average = 0.00609
- Orthogonal Value Minimum = 0.30013
- Orthogonal Value Average = 0.99137
- Average Aspect Ratio = 1.25968
- Maximum Aspect Ratio = 22.2445

Comparing obtained quantities with quantities of coarse mesh, one can state that this mesh provides better possibilities. Next, it has to be checked according to the results of simulations with aerodynamic tunnel ones. An important aspect to mention is that in $k-\epsilon$ model values of y+ should be varying from 30 to 300. From literature, it can be said that this mesh meets the requirements.

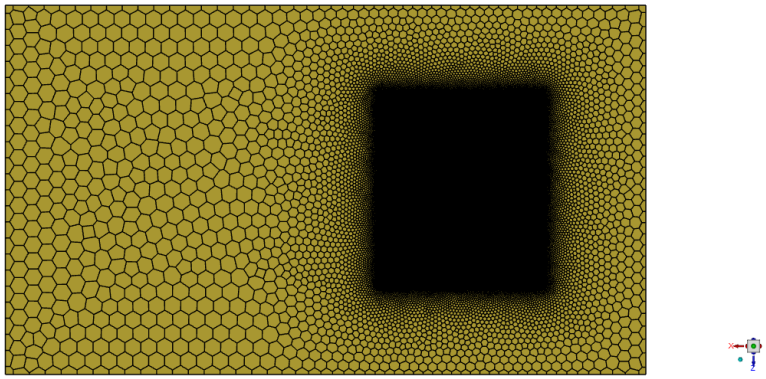


Figure 6.4. Created mesh overview (fine)

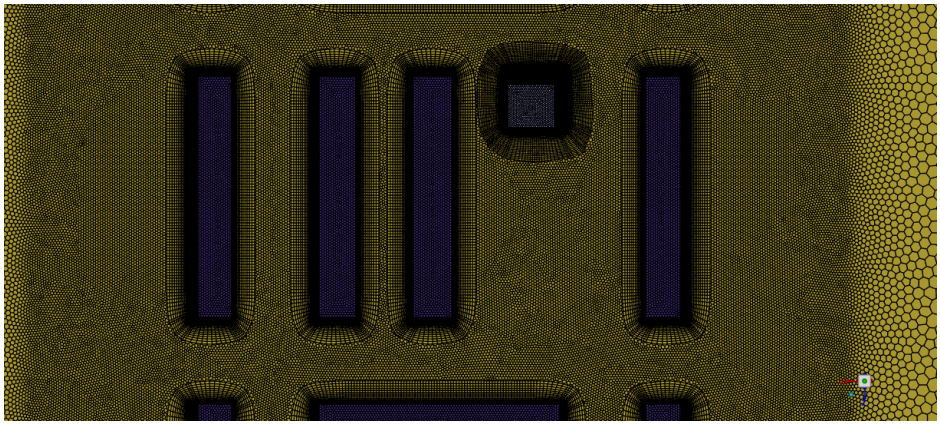


Figure 6.5. Created mesh closeup (fine)

7. Solver

7.1. Solver Settings

c

Doing some research on Urban Flow through aerodynamic structures, one can select settings that simulate as actual conditions as possible. Many kinds of literature say that such flow's behavior is most similar when using *SST k-ϵ realizable* solver. Fluid is assumed to be constant-density and constant-viscosity air. The velocity profile on the inlet is assumed to be (6), where operating pressure was equal to 101325Pa and:

- V_m = 10 m/s
- α=0.19

Nextly it is a must to define walls conditions in Fluent. Buildings walls and lower wall are treated as *no slip* condition, but upper and side walls are defined as specified shear.

Moreover, as mentioned in the (**??**), TI is assumed to be 25. The analyzed solution method is set to SIMPLE, and all discretization models are of Second Order.

8. Results

The velocity results were analyzed at the pedestrian level, which is assumed to be about 2 meters, which was 0.004m in scaled geometry. There is a tendency that simulations are usually done between 1 and 2 meters above the ground.

8.1. Coarse Mesh vs. Fine Mesh

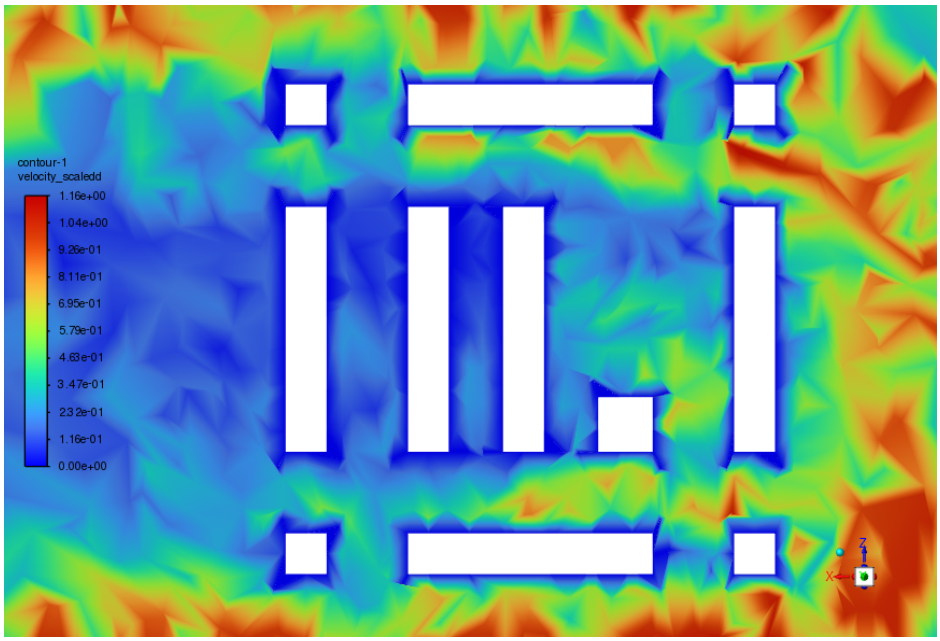


Figure 8.1. Velocity distribution for AOA 0 degree (coarse mesh)

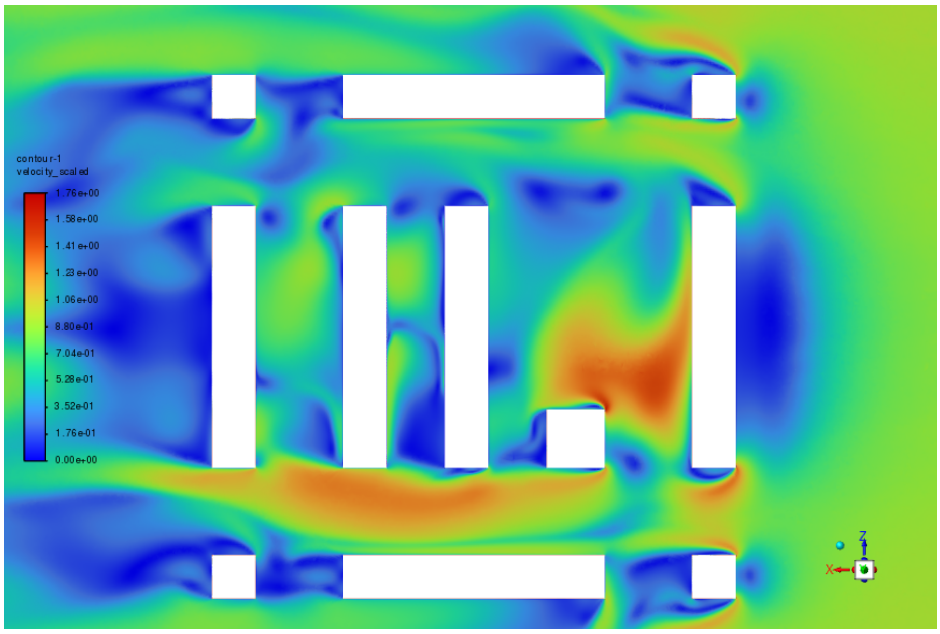


Figure 8.2. Velocity distribution for AOA 0 degree (fine mesh)

One can easily notice that the solution for better mesh differs from the coarse one. First iteration of mesh has visible elements, which is not desired as the smoothness of results is violated. Also, it is visible how results totally differ in both of these cases, which is not desired at all. Noticing all that aspects results for fine mesh will be compared in the next comparisons.

8.2. Fine Mesh

8.2.1. velocity and pressure contours for 0 degree flow

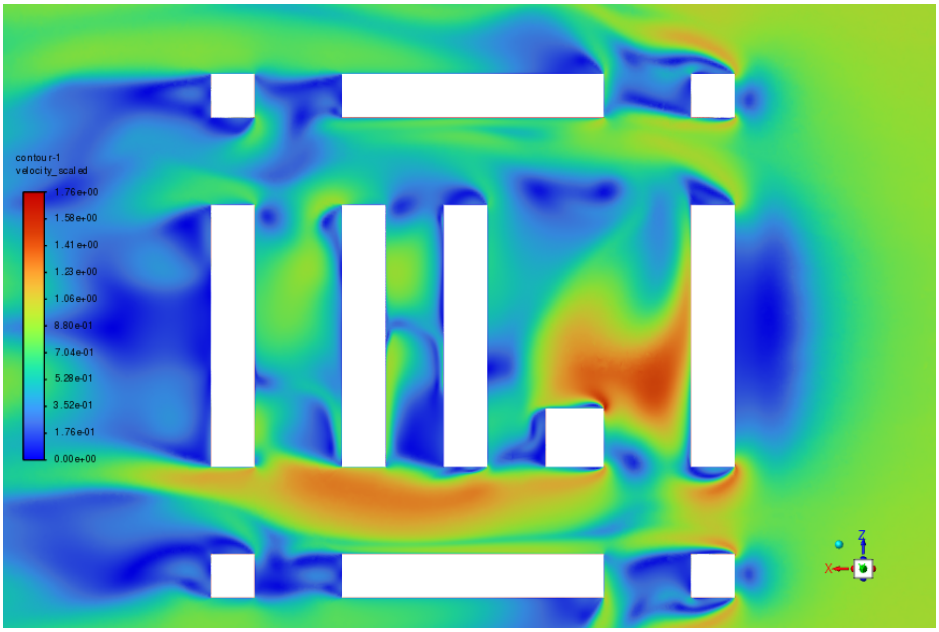


Figure 8.3. Velocity distribution for AOA 0 degree (fine mesh)

To validate results and correctness of displayed contours, one can view pressure contours of the same results:

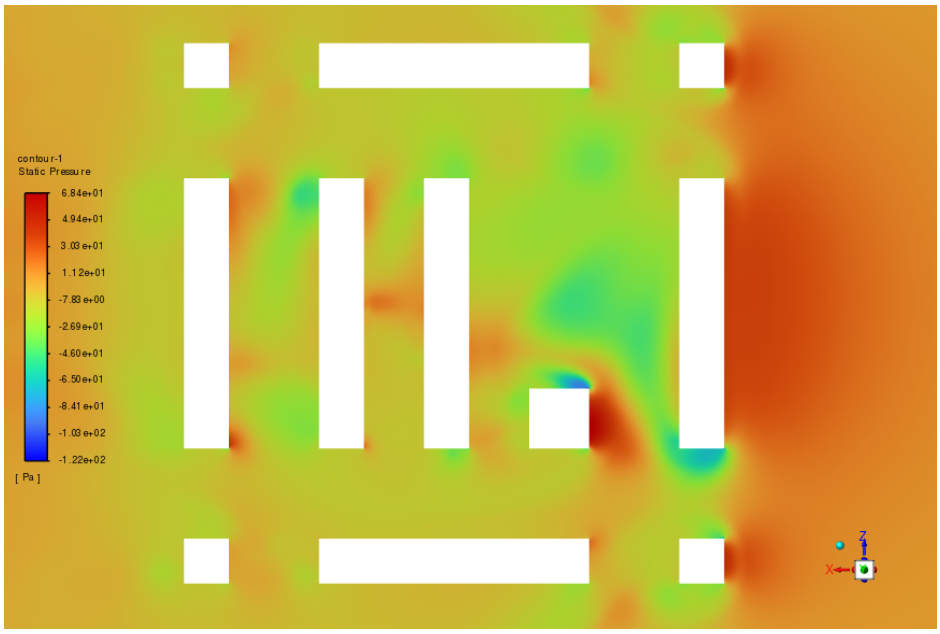


Figure 8.4. Pressure distribution for AOA 0 degree

As expected, pressure contours confirmed the results of velocity ones. Pressure dropped in regions where velocity got greater, and it raised in regions where velocity got smaller. It was done for every choice of angle of attack so to confirm definitely results that one obtained.

8.2.2. velocity and pressure contours for 45-degree flow

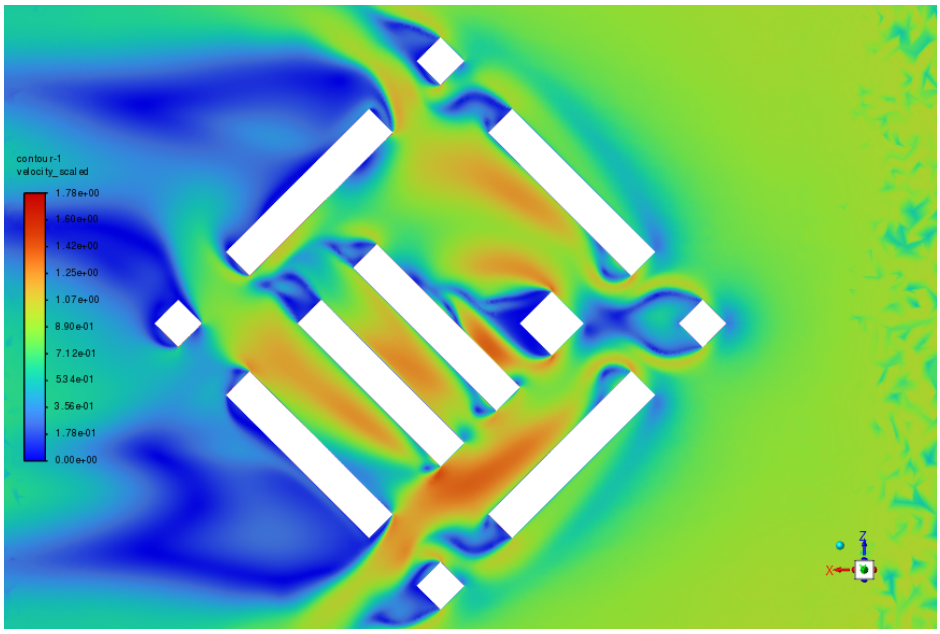


Figure 8.5. Velocity distribution for AOA 45 degree

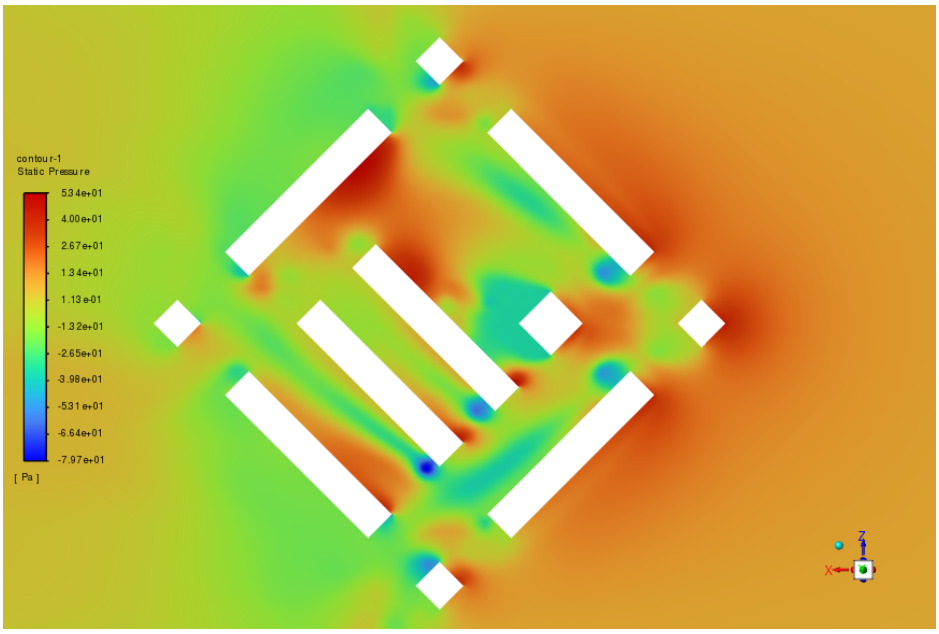


Figure 8.6. Pressure distribution for AOA 45 degree

Results of this case are also looking promising, as they present similar relations and characteristics as in the first case.

8.2.3. velocity and pressure contours for 90-degree flow

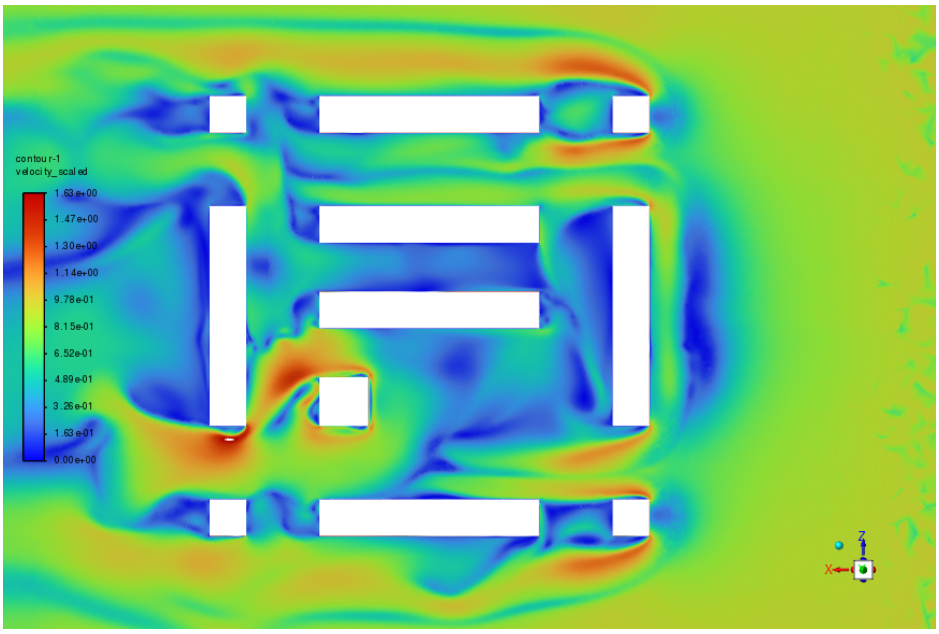


Figure 8.7. Velocity distribution for AOA 90 degree

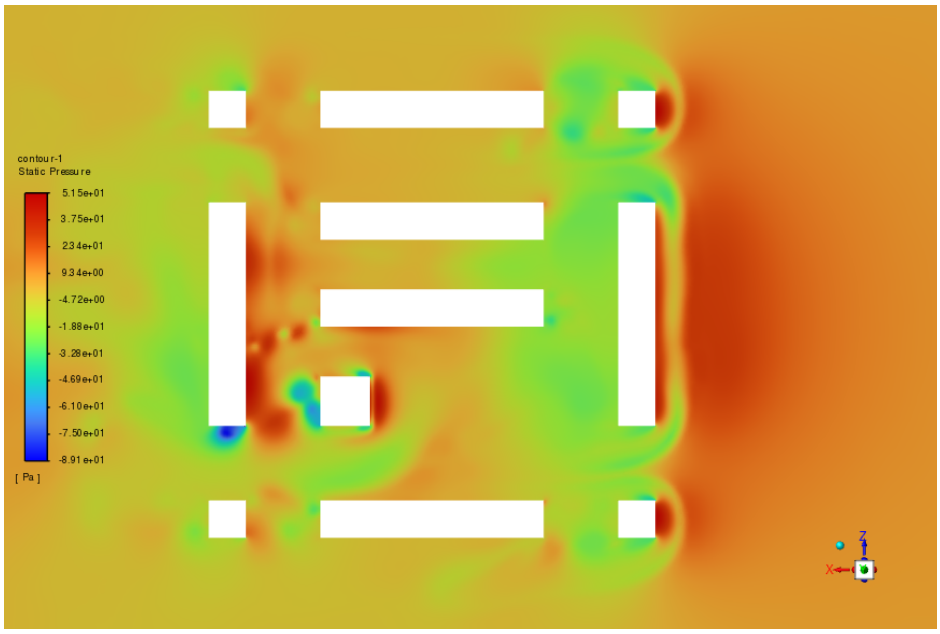


Figure 8.8. Pressure distribution for AOA 90 degree

After checking previous cases, one can expect this one to look similar. Pressure contours are of higher value in places where velocity slows down, and they are of greater magnitude when velocity is higher.

9. Conclusion and Comparison

9.1. Wind Tunnel Results

Below are presented results obtained in wind tunnel tests. Results might be burdened with errors coming from:

- not-ideal velocity at inlet
- non-uniform rising of sand grains on two sides of the geometry

9.1.1. AOA 0 degree

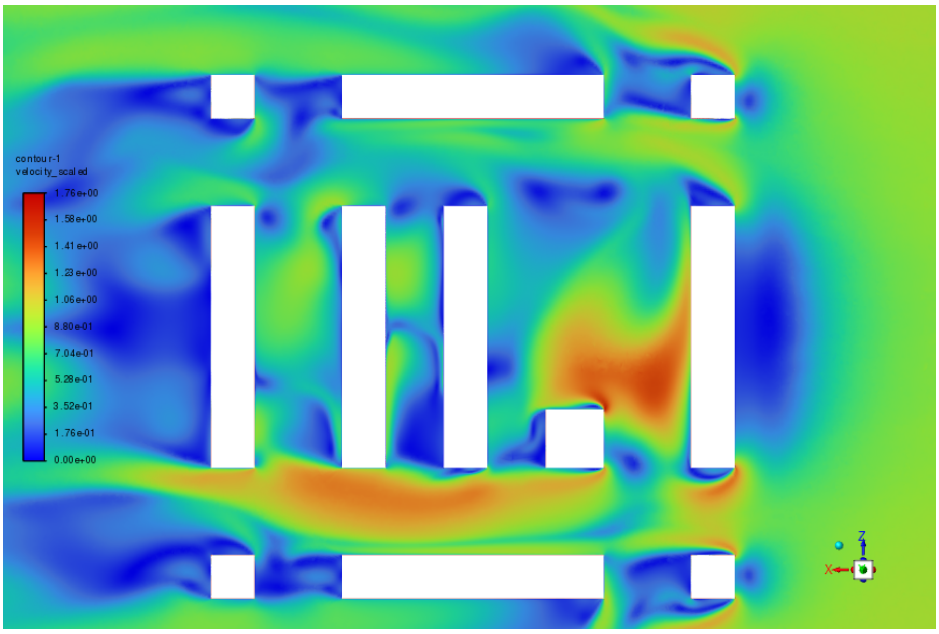


Figure 9.1. Velocity distribution for AOA 0 degree

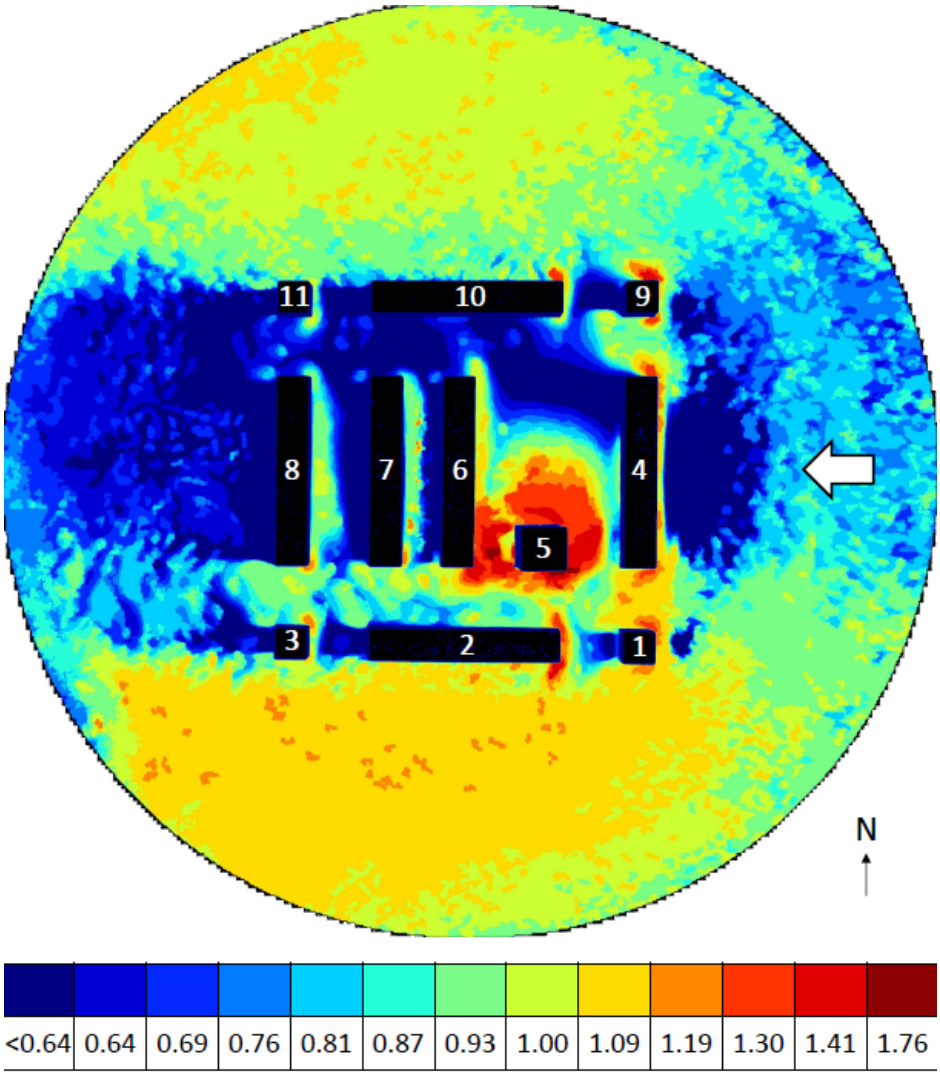


Figure 9.2. Wind tunnel results for 0 degree

One can notice that regions between buildings (**8** – **7**) and (**7** – **6**) are well air-conditioned as it is desired. Results are not perfectly the same, but the character of them is similar in this region. The next part is the region between buildings (**1** – **2** – **3**) and (**4** – **5** – **6** – **7** – **8**), which differs a bit between obtained wind tunnel results, but the character of them is also similar, and one can even state that results from ANSYS simulation are better, as it is more air-conditioned, but still not too windy. As one could expect, the close region of the tallest building **5** is a region of greatest wind velocity.

9.1.2. AOA 45 degree

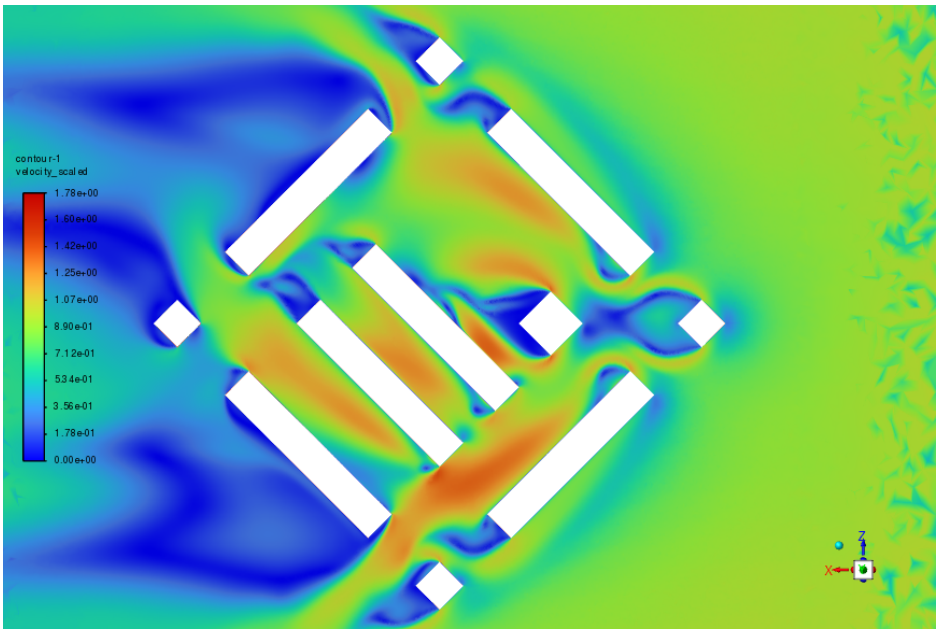


Figure 9.3. Velocity distribution for AOA 0 degree

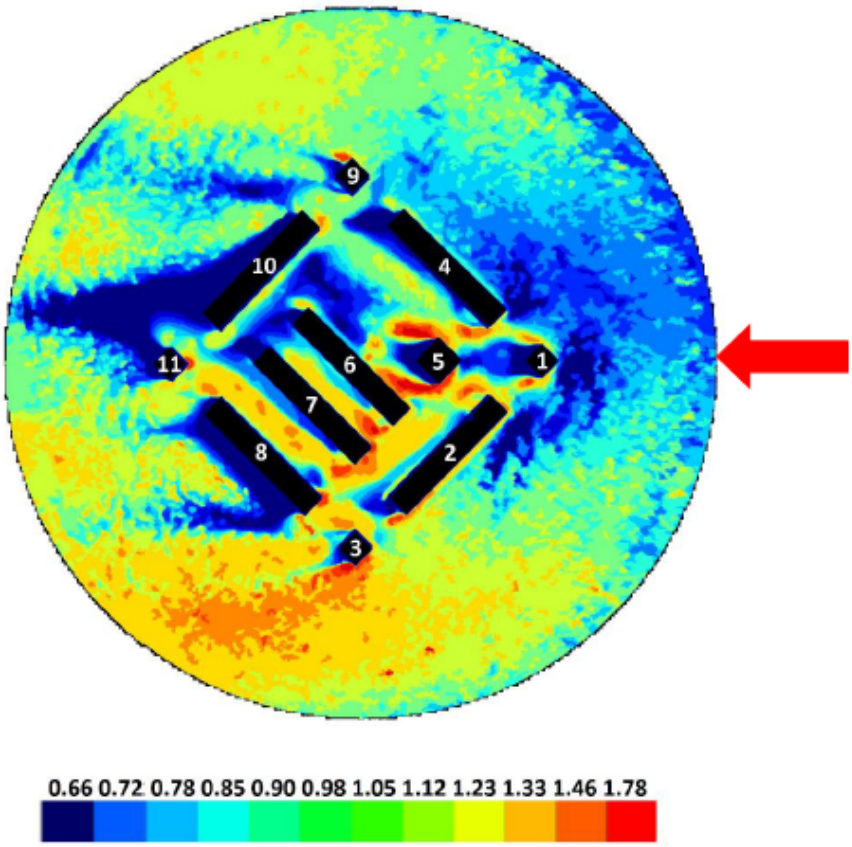


Figure 9.4. Wind tunnel results for 45deg

In this case, one can also notice easily visible similarities in the flow. Zone next to the highest building (**5**) is of great wind velocity, zones between (**6** – **7** – **8**) is well conditioned, and there is no stall of air. The only difference is that the flow between (**2** – **6** – **7**) is a bit faster than in the wind tunnel test, which can be a bit too much for pedestrians.

9.1.3. AOA 90 degree

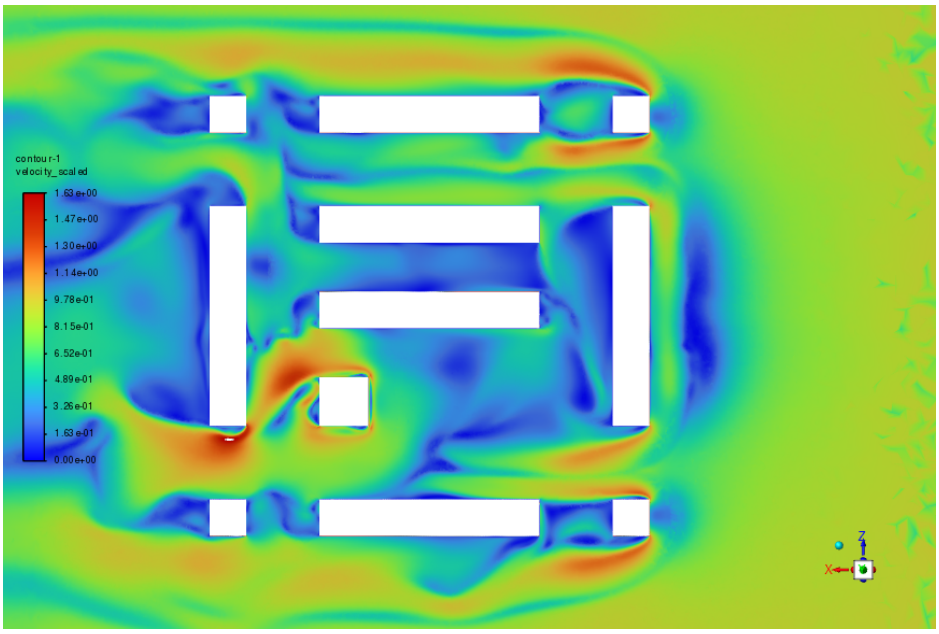


Figure 9.5. Velocity distribution for AOA 0 degree

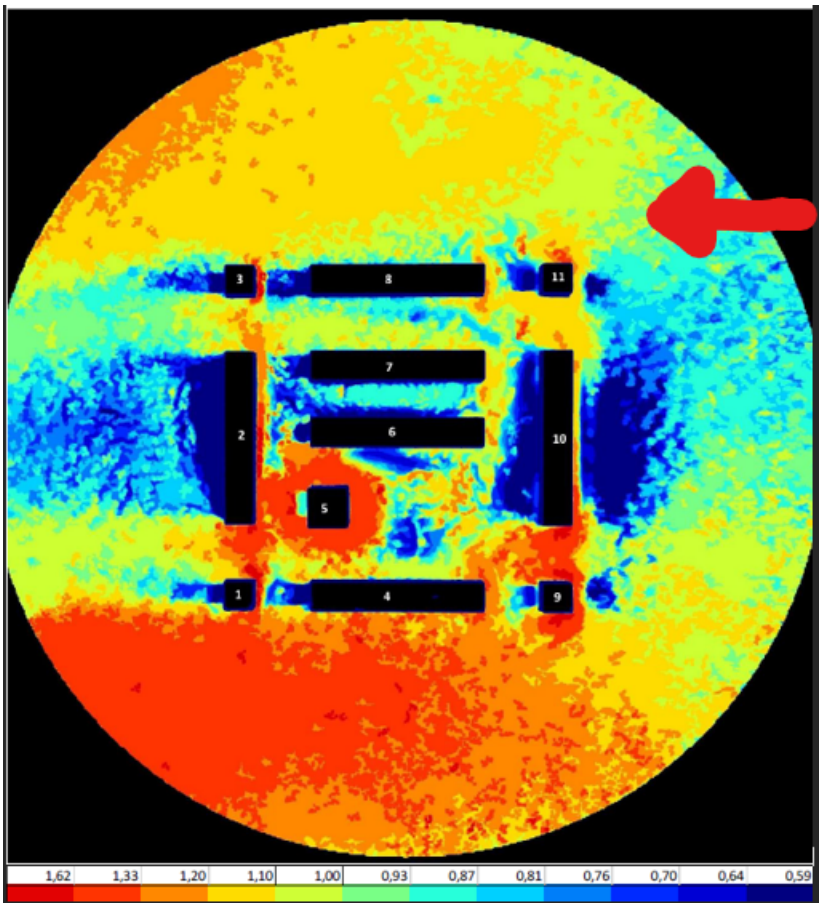


Figure 9.6. Wind tunnel results for 90deg

One can notice the similarity to previous results - flow next to the highest building (**5**) is of great amplitude. Overall characteristics of the flow are similar, but the region between (**9** – **10**) is not as windy as it came from wind tunnel analyses. Like previous cases were fine and did not have any issues with building a new high building, this one has some objections. One can notice that in ANSYS simulation, flow between (**6** – **7**) is almost in stall, which is definitely not desired.

9.1.4. Summing up

Comparing these results, one can say that the ones obtained in the ANSYS simulation are similar to the ones in the wind tunnel. Regions of wind acceleration are in similar places.
Some of the differences in results may also come from the following:

- not fine enough mesh
- not perfectly well-selected values in k-ϵ model
- differences on inlet profile used in simulation and wind tunnel
- assumption of Turbulence Intensity

9.2. Conclusions

With the above steps, it was possible to obtain proper and similar results as in wind tunnel. It is now possible to look for differences and make sure that conditions in wind tunnel, in places where results differ, are correct. To make sure that obtained in analysys results are correct, one would have to make corrections in wind tunnel, where they are needed and perform more tests to collect more data.

References

[1] [] *Pedestrian Wind Environment around Buildings: Literature Review and Practical Examples*, Bert Blocken and Jan Carmeliet

[2] *Pedestrian-level wind conditions around buildings: Review of wind-tunnel and CFD techniques and their accuracy for wind comfort assessment*, B.Blocken, T.Stathopoulos, J.P.A.J. van Beeck

[3] *Environmental Wind Engineering and Design of Wind Energy Structures*, C. Baniotopoulus, C. Borri, T. Stathopoulos

List of Figures

3.1 Real size geometry 6
3.2 Scaled geometry 6
6.1 Analyzed geometry 10
6.2 Minimum domain size
Hall 1997, Franke et al. 2004, 2007, Tominaga et al. 2008 10
6.3 Created mesh (coarse) 11
6.4 Created mesh overview (fine) 12
6.5 Created mesh closeup (fine) 12
8.1 Velocity distribution for AOA 0 degree (coarse mesh) 14
8.2 Velocity distribution for AOA 0 degree (fine mesh) 14
8.3 Velocity distribution for AOA 0 degree (fine mesh) 15
8.4 Pressure distribution for AOA 0 degree 16
8.5 Velocity distribution for AOA 45 degree 16
8.6 Pressure distribution for AOA 45 degree 17
8.7 Velocity distribution for AOA 90 degree 17
8.8 Pressure distribution for AOA 90 degree 18
9.1 Velocity distribution for AOA 0 degree 19
9.2 Wind tunnel results for 0 degree 20
9.3 Velocity distribution for AOA 0 degree 21
9.4 Wind tunnel results for 45deg 21
9.5 Velocity distribution for AOA 0 degree 23
9.6 Wind tunnel results for 90deg 23