

The Application of CFD to IUS Buildings Analysis

Enes Akca

International University of Sarajevo, Faculty of Engineering and Natural Sciences, Mechanical Engineering Program
Sarajevo, Bosnia and Herzegovina

enesacka@hotmail.com.tr, eakca@ius.edu.ba

Abstract

In this study, wind characteristics and energy potential of IUS Campus region is investigated. Assessment of the site is carried out by using turbulent flow models. Commercial software STAR CCM+ was used to solve turbulent flow equations. STAR CCM+ is a computational fluid dynamics (CFD) software package that solves Navier-Stokes equations that define fluid flow, by using numerical techniques.

Keywords: computational fluid dynamics, wall y^+ , heat transfer coefficient, stream lines

1 Introduction

Computational Fluid Dynamics (CFD) can play an important role in building design. For all aspects and stages of building design, CFD can be used to provide more accurate and rapid predictions of building performance with regard to air flow, pressure, temperature, and similar parameters.

Computational Fluid Dynamics (CFD) can play an important role in building design. For all aspects and stages of building design, CFD can be used to provide more accurate and rapid predictions of building performance with regard to air flow, pressure, temperature, and similar parameters.

Computational fluid dynamics (CFD) is used in science and engineering for numerical simulation of steady or unsteady flowfields. At Fraunhofer IBP we provide CFD competences in terms of modelling, simulation and analysis of complex phenomena related to indoor environmental quality. CFD enables in depth analysis of thermodynamic systems in all relevant application domains like buildings and construction, as well as the automotive and aviation sector. The CFD simulations consider heat transfer modes of

convection, radiation and conduction in combination with multi-physics approaches. The CFD results are used to evaluate thermal comfort of humans, quality of trading goods, ventilation efficiency in buildings or thermal management strategies in vehicles.

Also the purpose of the case study is to compute flow around A and B buildings of IUS campus using CDF. The main focus of the case study is how heat is transferred by the forced convection which is natural wind. The computation is used to compute a heat transfer coefficient at the buildings wall for the winter season. For that purpose, velocity field, heat transfer coefficient distribution along the buildings, pressure distribution, wall y^+ , turbulent kinetic energy, and temperature distribution will be shown in scalar, vector and graph forms. The temperature of outside the buildings is declared as -2°C , and the wind flows at 21°C with the velocity 5 m/s.

Figure 1 describes the wind direction in the region in the winter season as defined. The wind direction

is based on real observations from the weather station at Sarajevo/Butmir [1].

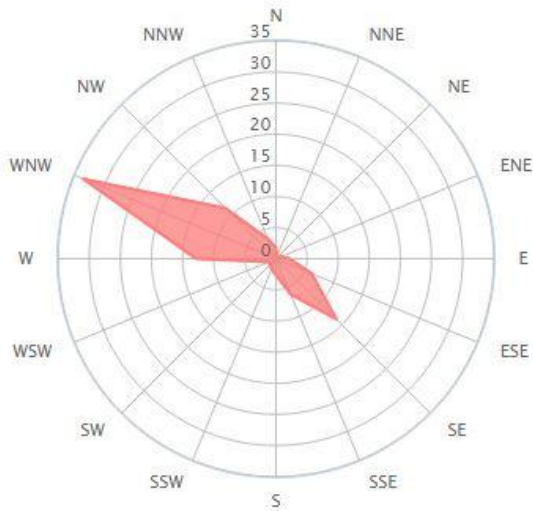


Figure 1: Direction of wind

2 Star CCM+

STAR-CCM+ is one of the most modern in comparison with competitors software package designed to solve the problems of continuum mechanics. It is characterized by extraordinary ease of use.

STAR-CCM+ includes the latest numerical algorithms such as advanced solvers, *raspredelemnny* (segregated solver) and connected (coupled solver), moreover, it is created using the most advanced programming techniques. All this allows STAR-CCM+ provide unprecedented accuracy, reliability and flexibility in solving problems of computational continuum mechanics. Powerful new tools for building grids: from restoring the integrity of the surface (surface wrapping) to create grids of polyhedral cells. Using these new tools for building grids can reduce the clock time for solving the problem. A large set of physical models: laminar and turbulent flow of Newtonian and non-Newtonian fluids, multiphase flow, cavitation, radiation, combustion, development of boundary layer flows with high Mach numbers, the conjugate heat transfer, as well as new models for the calculation of heat exchangers and fans.

3 Theoretical Background

The flow module determines the velocity and pressure fields by solving the three-dimensional

momentum equations and the pressure correction equations, respectively. These equations are guided by the laws of conservation of mass and momentum, which lead to the use of the Navier-Stokes equations to iteratively resolve the flow solutions. The following sections describe the governing flow equations[2].

Applying the mass, momentum and energy conservation, the continuity equation, momentum equation and energy equation can be derived as follows.

3.1 Continuity Equation

$$\frac{d\rho}{dt} + \rho \frac{\partial U_i}{\partial x_i} = 0$$

Momentum Equation

$$\underbrace{\rho \frac{\partial U_j}{\partial t}}_I + \underbrace{\rho U_i \frac{\partial U_j}{\partial x_i}}_{II} = \underbrace{-\frac{\partial P}{\partial x_j}}_{III} - \underbrace{\frac{\partial \tau_{ij}}{\partial x_i}}_{IV} + \underbrace{\rho g_j}_V$$

Where

$$\tau_{ij} = -\mu \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \frac{2}{3} \delta_{ij} \mu \frac{\partial U_k}{\partial x_k}$$

- I : Local change with time
- II : Momentum convection
- III : Surface force
- IV : Molecular-dependent momentum exchange (diffusion)
- V : Mass force

3.2 Energy Equation

$$\underbrace{\rho C_\mu \frac{\partial T}{\partial t}}_I + \underbrace{\rho C_\mu U_i \frac{\partial T}{\partial x_i}}_{II} = \underbrace{-P \frac{\partial U_i}{\partial x_i}}_{III} + \underbrace{\lambda \frac{\partial^2 T}{\partial x_i^2}}_{IV} - \underbrace{\tau_{ij} \frac{\partial U_j}{\partial x_i}}_V$$

- I : Local energy change with time
- II : Convective term
- III : Pressure work
- IV : Heat flux (diffusion)
- V : Irreversible transfer of mechanical energy into heat

If the fluid is compressible, the continuity equation and momentum equation can be simplified as follows.

Continuity Equation

$$\frac{\partial U_i}{\partial x_i} = 0$$

Momentum Equation

$$\rho \frac{\partial U_j}{\partial t} + \rho U_i \frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} - \mu \frac{\partial^2 U_j}{\partial x_i^2} + \rho g_j$$

3.3 General Form of Navier-Stokes Equation

To simplify the Navier-Stokes equations, they can be rewritten as the general form.

$$\frac{\partial(\rho\Phi)}{\partial t} + \frac{\partial}{\partial x_i} \left(\rho U_i \Phi - \Gamma_\Phi \frac{\partial \Phi}{\partial x_i} \right) = q_\Phi$$

When $\Phi=1$, U_j , T , and they can be respectively get as continuity equation, momentum equation and energy equation.

3.4 Finite Volume Method

The Navier-Stokes equations are analytical equations. Human can understand and solve them, but if we want to solve them by computer, we have to transfer them into discretized form. This process is discretization. The typical discretization methods are finite difference, finite element and finite volume methods. Here we introduce finite volume method, because finite volume methods are generally used in CFD [3].

The Approach of Finite Volume Method

Integrate the general form of Navier-Stokes equation over a control volume and apply Gauss Theory,

$$\int_V \frac{\partial}{\partial x_i} \Phi dV = \int_S \Phi \cdot n_i dS$$

The integral form of Navier-Stokes equation can be written as

$$\int_V \frac{\partial(\rho\Phi)}{\partial t} dV + \int_S \left(\rho U_i \Phi - \Gamma \frac{\partial \Phi}{\partial x_i} \right) \cdot n_i dS = \int_V q_\Phi dV$$

3.5 $k - \epsilon$ Turbulent Model

In the $k - \epsilon$ turbulent model, the notation of k is the kinetic energy and ϵ is the turbulence dissipation rate. The turbulent dynamic viscosity is expressed as

$$\frac{\mu_t}{\mu} = 20$$

$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon}$$

$$\epsilon = \frac{C_\mu \rho k^2}{20 \cdot \mu} \quad , \quad \frac{\sqrt{k}}{V_b} = 10\% = 0.1$$

In the above equations, V_b is the bulk velocity, ρ is the fluid density, μ and μ_t are the laminar and turbulent dynamic viscosities respectively. The turbulence model constant used in the above equations are $C_\mu = 0.09$ [4].

4 Computational Details

A 3-D finite-volume approach is adopted by the software STAR CCM+ because of its capability of conserving solution quantities. The software solves the conservation equations for continuity, momentum, and energy as well as the equations for turbulent kinetic energy and its dissipation rate. The two-equation $k - \epsilon$ turbulence model is used.

A three-dimensional model of the IUS buildings in the shape of a rectangular prism was developed. The physical dimensions were set to be 48 m wide by 51 m long by 20 m tall.

RANS solver is used to solve the governing equations of continuity and momentum in a viscous flow around the buildings model. The Navier-Stokes Equations (NSE) are solved with a segregated, algebraic multigrid solver using Gauss-Seidel iterations.

In this study case, for the three-dimensional case, implicit unsteady simulations were carried out at step intervals of 0.01 seconds for 200 time steps which is equivalent to 2 seconds. A description of the conditions and assumptions is given in the following sections.

4.1 Computational domain

Generally, the size of the entire computational domain depends on the targeted area and the boundary conditions. For a single building, the distance from the top of the building to the top

of the computational domain should be about $20H$, where H is the building height and is defined as 20m. For the lateral boundary, $20H$ is required between the buildings and the computational domain. In the figure 2, the computational domain and dimensions are shown, and dimensions' units are in cm.

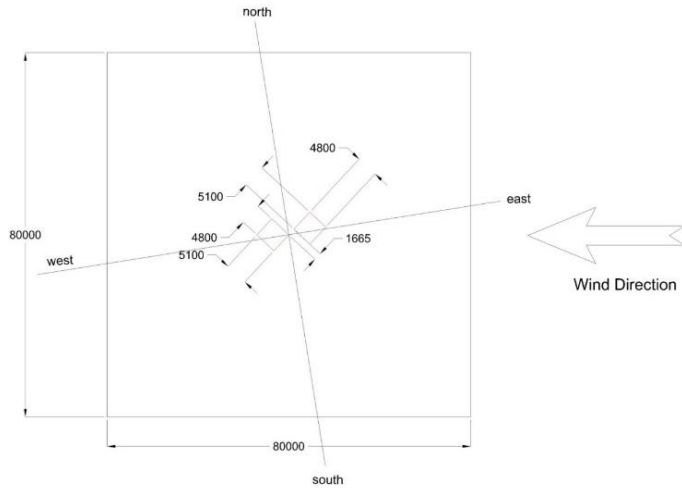


Figure 2: The Computational domain and dimensions.

4.2 Computational Grids

The hexahedra cell has been chosen for meshing purpose. Also prism layer mesher and surface remesher options were used. All computational domains of this study were undertaken for generating mesh by means of relative cell size. The base size value is defined as 5m. Boundary growth rate is selected as very slow, and default growth rate is as medium. Also the mesh values in the buildings are selected relative size is 1m. Thus, relative maximum cell size of 5m and relative minimum size of 1m were used as limits for generating local mesh of all domains and the buildings. Meshing of this computational domain was carried out with 1,243,390 cells and 3737529 faces, covering the whole volume of the domain.

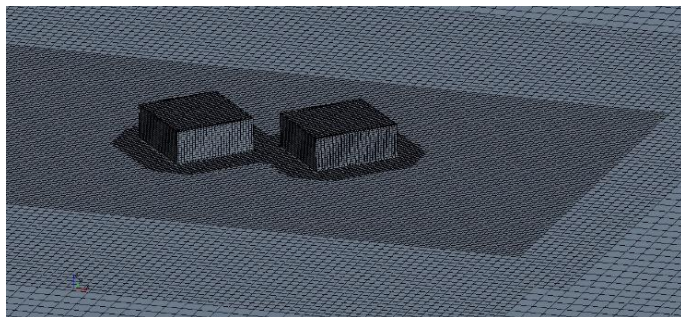


Figure 3: computational grids of the buildings.

4.3 Turbulence Models

The CFD program was set to use a standard $k-\epsilon$ model for turbulence. The required input variables were the turbulent kinetic energy, k , and the turbulent dissipation rate, ϵ . The inlet flow velocity was taken to be 5 m/s. This inlet flow velocity can be directly inserted into equations in section 2.5, to result in a turbulent kinetic energy value, k , of 0.25 J/kg. Also, the turbulent dissipation rate, ϵ , can be calculated using the equation in the section 2.5, and the result is calculated as, ϵ , of $17 \text{ m}^2/\text{s}^3$.

4.4 Initial Conditions

Initial conditions describe the state of the system at the initial instant of time through the values of a solution variable, and have to be defined in the whole domain: $f_0 = f(t_0)$. If the equations to be solved include high-order time derivatives, the initial condition include also the specification of the correspondingly lower derivatives of solution variables at the initial instant of time. The initial parameters are set as calculated before, shown in table 1.

Table 1: Initial conditions.

Pressure	Static Temperature	Turbulent Dissipation Rate	Turbulent Kinetic Energy	Velocity
101,325 Pa	-2°C	$17 \text{ m}^2/\text{s}^3$	0.25 J/kg	5 m/s

4.5 Boundary Conditions

A boundary condition describes the state of a system, expressed through conditions on the solution variables holding on all boundaries of the system.

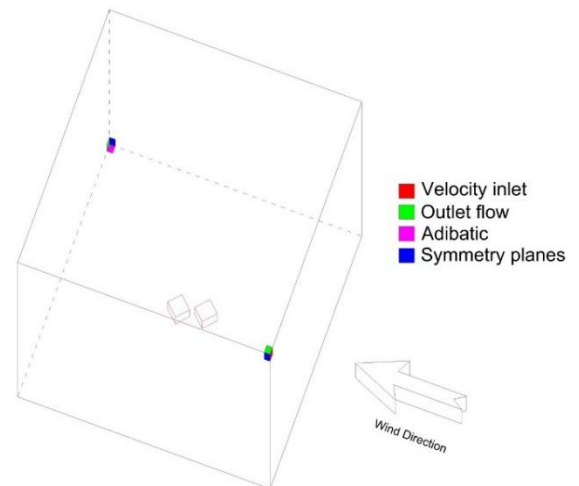


Figure 4: Boundaries of all faces

In figure 4, each surface is described as boundary conditions, and their types of boundaries are defined. Also the physical values of boundaries are given in table 2. Heat transfer coefficient and thermal resistance were as constant typical values [5].

Table 2: Boundary conditions and physical values.

Velocity inlet	Outlet flow	Adiabatic	Symmetry planes	Building Boundaries
Mag. : 5 m/s	Flow- Split Outlet	No Heat Transfer		Environment
Temp. : -2 °C				Amb.Tem.: 21°C
Turb. Diss. Rate: 17 m ² /s ³				HeatTransfer Coefficient: 2.18 W/m ² -K
Turb.Kinetic Energy : 0.25 J/kg				Thermal Resistance: 0.13 m ² -K/W

Asymmetry plane boundary represents an imaginary plane of symmetry in the simulation. The solution that is obtained with asymmetry plane boundary is identical to the solution that would be obtained by mirroring the mesh about this symmetry plane.

5 Results

The results are shown in figures below. In Figure 5, the velocity magnitude of streamlines around the building for turbulence model is presented.

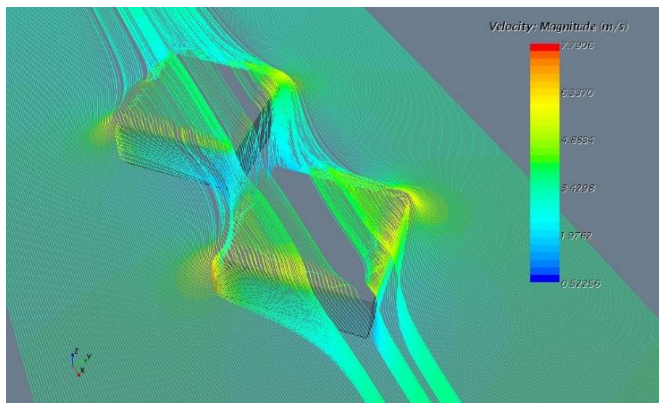


Figure 5: Streamlines with velocity magnitude.

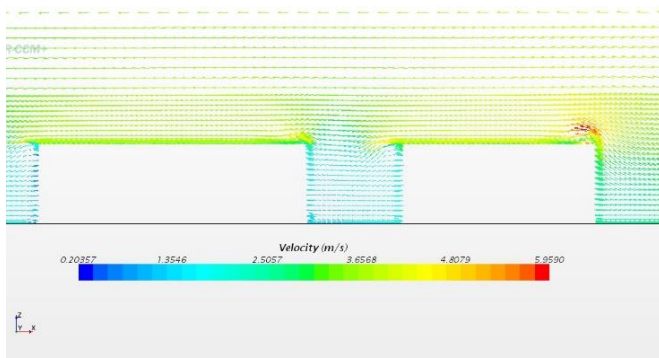


Figure 6: Velocity vector

In figure 6, velocity is shown as vector of a created plane from the middle of the computational domain, and as it is seen in the figure, when wind direction is defined from left to right, there is impinging on the up side of the wall where the wall has the first impact of the wind.

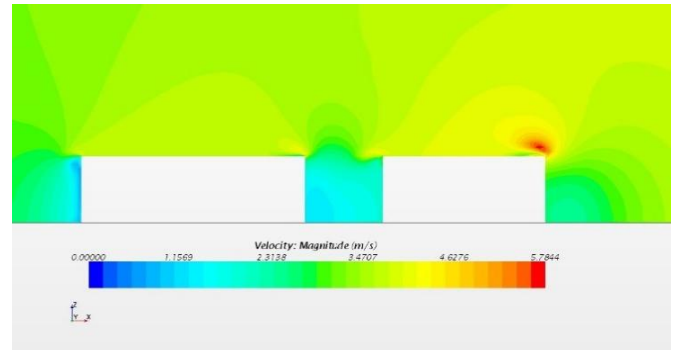


Figure 7: velocity contour

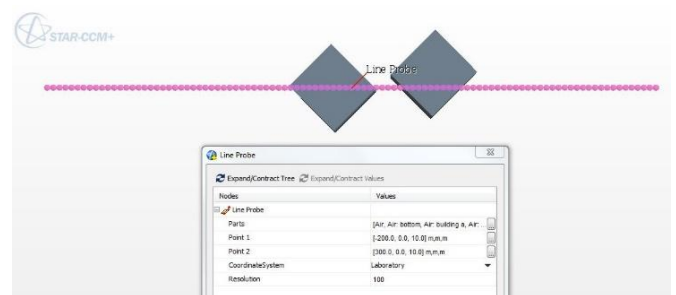


Figure 8: Probe line shown

Monitoring the vector and contour plots of velocity during calculation of the flow analysis are shown in figure 6 and 7. Also while figure 6 shows velocity vector and figure 7 shows velocity scalar, they have the same sense of velocity as expected.

In figure 8, a probe line was created along the flow direction in the middle of the computational domain. The purpose of the probe line is to plot some graphs along it. Also the probe line was located in 10 m height.

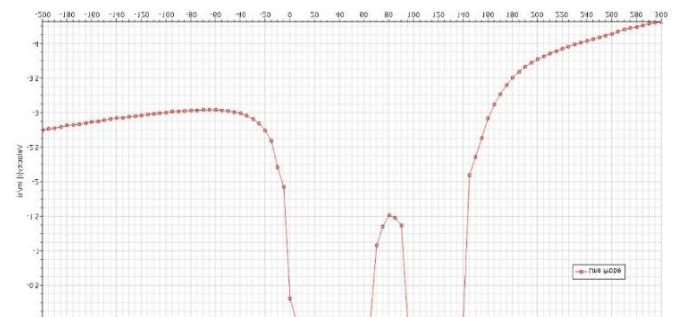
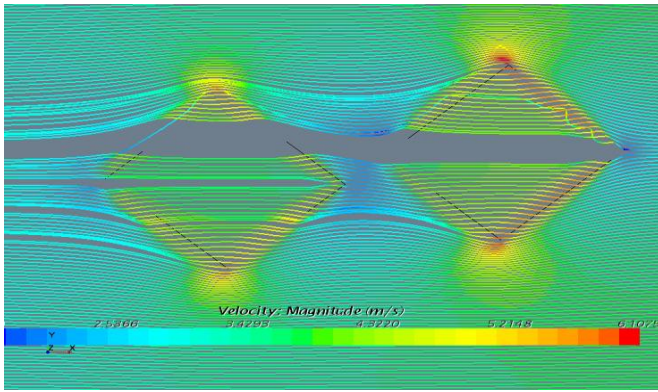
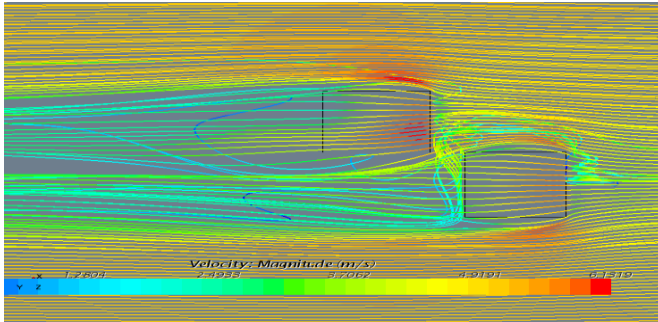


Figure 9: velocity graph along the probe line.



(a)



(b)

Figure 10: Flow direction with velocity magnitude; (a) Real direction (b) Imaginary direction.

In figure 9, velocity changes are plotted along the probe line which was created in figure 8. Also in figure 10, real flow direction with velocity magnitude is shown on the part A, and imaginary flow direction is shown on the part B. when they are compared according to flow directions. In part B, there is more and more turbulent and it needs to less and less. Thus the real flow can be considered as the optimum flow direction.

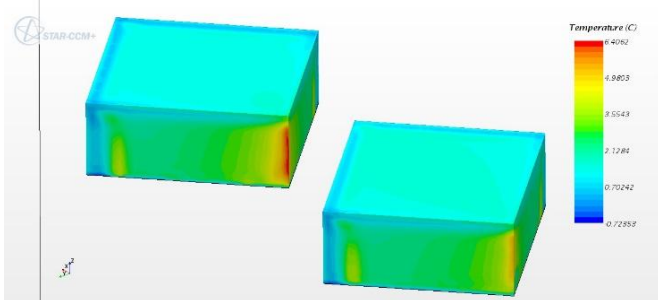


Figure 11: temperature field around the buildings.

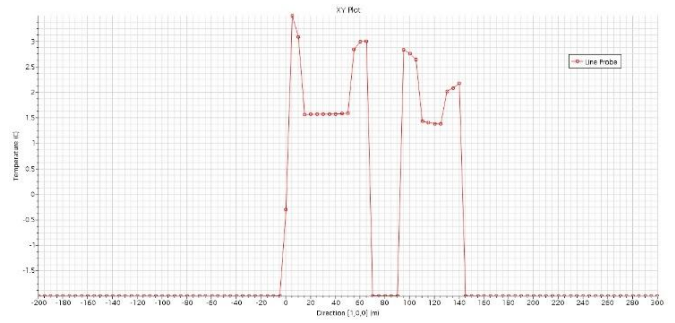


Figure 12: plotted graph of temperature along the probe line.

Temperature field around the buildings are represented in figure 11, and the graph of temperature along the probe line is plotted in figure 12. Also in figure 11, the highest temperature appears at back side of the first building when the flow is on $-x$ direction.

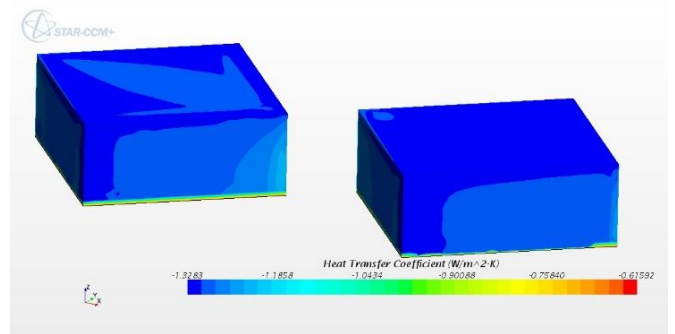


Figure 13: Heat transfer coefficient around the buildings.

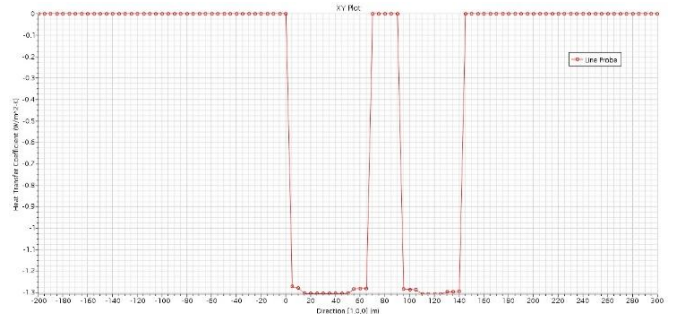


Figure 14: graph of heat transfer coefficient around the buildings.

Heat transfer coefficient around the buildings is shown in figure 13, and it is plotted along the probe line in figure 14. Heat transfer coefficient can be shown in formula as $h=q/\Delta T$, where q is amount of heat required (heat flux), h is heat transfer coefficient, and ΔT is difference in temperature between the solid and surrounding fluid area. The temperature differences bottom side of the buildings' wall is less, and thus heat transfer coefficient is high at that region as shown in figure 13 and 14.

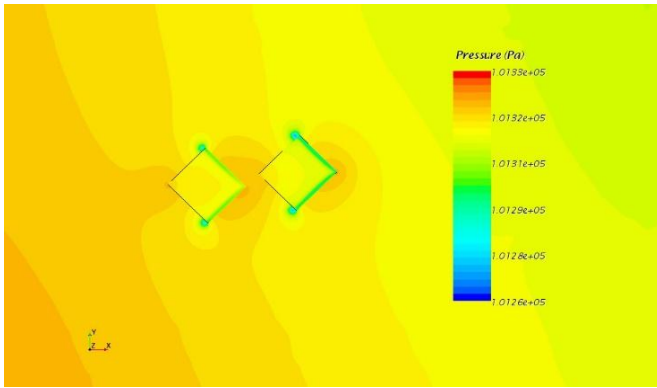


Figure 15: pressure distribution

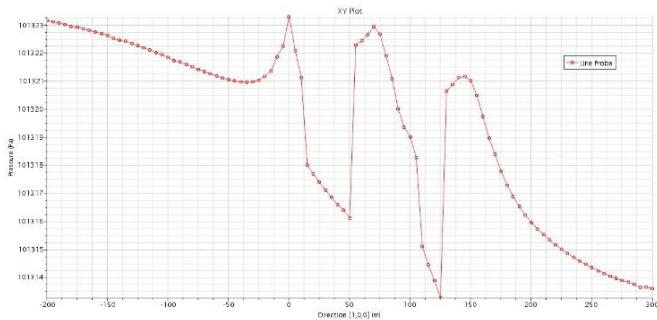


Figure 16: pressure graph along the probe line.

In addition, scalar of pressure distribution on the buildings is shown in figure 15, and also pressure drop is plotted as a graph along the probe line in figure 16.

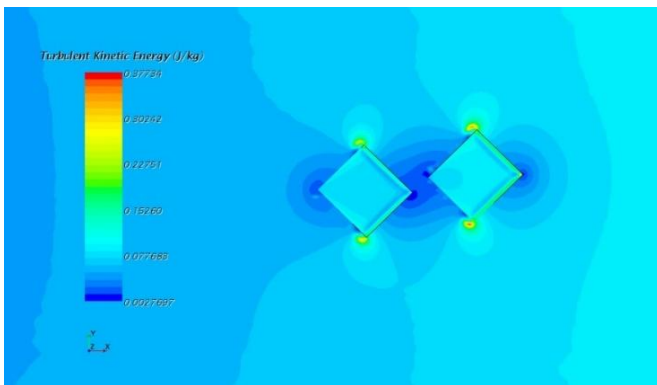


Figure 17: Turbulent kinematic energy

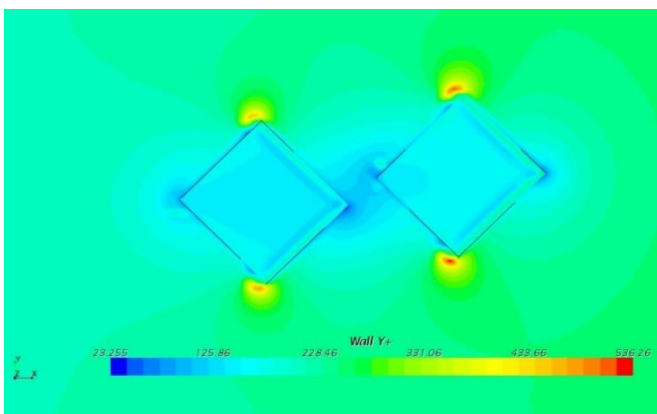


Figure 18: Wally+

A non-dimensional wall distance for a wall-bounded flow can be define as $y^+ = (u_* y) / \nu$. Where u_* is the friction velocity at the nearest wall, y is the distance to the nearest wall and ν is the local kinematic viscosity of the fluid. y^+ is often referred to simply as y plus and is commonly used in boundary layer theory and in defining the law of the wall.

Figure 18 shows the turbulent kinetic energy, and figure 17 shows the non-dimensional wall distance. Also although their scalar fields are not the same, their senses are very similar as shown in those figure. Also their graphs can be compared. Figure 19 shows the graph of turbulent kinetic energy, and figure 20 shows the graph of wall y^+ .

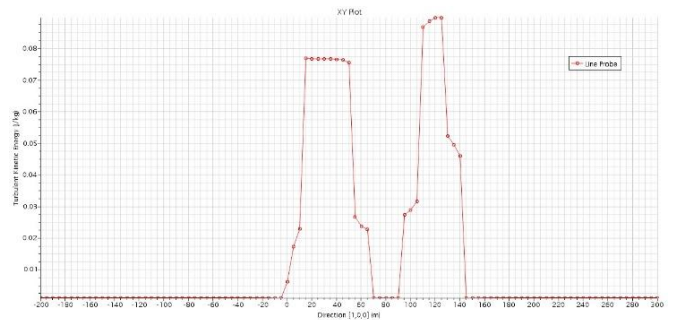


Figure 19: graph of turbulent kinetic energy

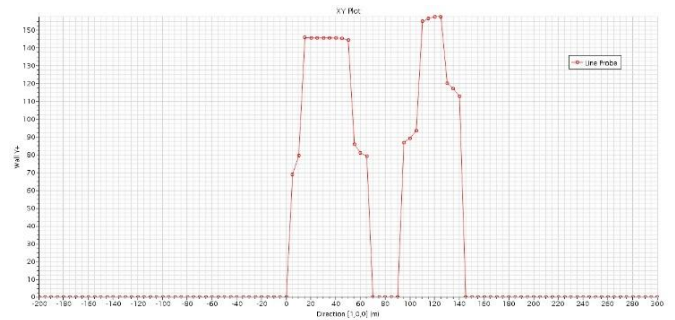


Figure 20: graph of wall y^+

6 Conclusion

Turbulence is a very important phenomenon that affects all atmospheric processes, but it is more important near the surfaces. Turbulence represents the irregularity or randomness of the flow. The k - ϵ turbulence model is used. The most unique measure for turbulence is kinetic energy of turbulent part of the flow. This variable depends on a variety of mechanisms. The most important are certainly buoyant production or consumption and shear production of turbulence kinetic energy (TKE). Both of these vary significantly in time and in space (especially in height). The third very important term is dissipation. This term provides

the reduction of turbulence with time which is the primary purpose of turbulence. This causes that the energy is dissipated from large vortices to small one.

Also the purpose of the case study was to compute flow around A and B buildings of IUS campus using CDF. The main focus of the case study was how heat is transferred by the forced convection which is natural wind. The computation was used to compute a heat transfer coefficient at the buildings wall for the winter season. For that purpose, velocity field, heat transfer coefficient distribution along the buildings, pressure distribution, wall y^+ , turbulent kinetic energy, and temperature distribution will be shown in scalar, vector and graph forms. The temperature of outside the buildings is declared as -2°C , and the wind flows at 21°C with the velocity 5 m/s. Also it can be said that as a result of the analysis, the IUS buildings are located in the optimum position, actually it was the most important information of this study case.

References

1. http://www.windfinder.com/windstatistics/saraj_evo_butmir
2. Wangda Zuo, "Introduction of Computational Fluid Dynamics", FAU Erlange-Nürnberg, St. Petersburg, 2005.
3. Bradley S. Hurak, "Computational Fluid Dynamics Analysis of Air Flow and Temperature Distribution in Buildings".
4. Chanawat Nitatwichit, Yottana Khunatorn, and Nakorn Tippayawong, "Computational analysis and visualization of wind-driven naturally ventilated flows around a school building", Maejo international journal of science and technology, 2008.
5. Francis G.N.Li, A.Z.P. Smith, and Phillip Biddulph, "Solid-wall U-values: heat flux measurements compared with standard assumptions", Routledge, London, 2015.
6. Matic Savli, "Turbulence Kinetic Energy (TKE)", University of Ljubljana, 2012.