International Journal of GEOMATE, July 2023, Vol. 25, Issue 107, pp.191-203
ISSN: 2186-2982 (P), 2186-2990 (O), Japan, DOI: https://doi.org/10.21660/2023.107.g12243
Geotechnique, Construction Materials and Environment

SIMULATION OF SMOKE DISPERSION AND TEMPERATURE DISTRIBUTION ON KEBON MELATI SUB-DISTRICT FIRE USING COMPUTATIONAL FLUID DYNAMICS

*Deffi Ayu Puspito Sari[1], Agnes Setioningrum[1],and Dani Harmanto[2]

[1]Faculty of Engineering and Computer Science, Bakrie University, Indonesia;
[2]Aeronautical Engineering, De Montfort University, United Kingdom;

*Corresponding Author, Received: 15 April 2022; Revised: 16 March 2023; Accepted: 02 April 2023

ABSTRACT: Kebon Melati sub-district is one of the densely populated areas in the capital city of Jakarta which is prone to fires. In the last twenty years, there have been three cases of fire mostly caused by electrical short circuits. Those fire cause fire smoke dispersion and fire propagation as secondary impact. The aim of this research is to analyze fire smoke dispersion pattern and temperature distribution pattern around hotspots, also to analyze potential area that has to encounter those secondary impacts, henceforth the side that can be used as an evacuation route can be determined. Google Maps, AutoCAD 2015, Revit 2015, Autodesk Flow Design 2015, and Fire Dynamics Simulator are used in this research. Autodesk Flow Design 2015 simulation result wind flow in the west side of Kebon Melati sub-district is more stable than in the east and middle side which have wind flow turbulence as effect of wind flow separation by skyscraper buildings. On the outer west side of Kebon Melati sub-district, wind velocity increase up to 9 m/s as effect of wind flow separation by skyscraper buildings. Fire Dynamics Simulator simulation result, in 30 s simulation with 10.000 kW/m^2 Heat Release Rate per unit Area, fire smoke disperse up to 1.440 m in South direction. The South Side has the greatest potential for reduced visibility, fire smoke pollution, and fire propagation. The sides that can be used as evacuation route is the Westside and the Middle side which has more stable wind flow than the East side.

Keywords: CFD, Fire, Fire Dynamics Simulator, Smoke Dispersion, Temperature Distribution

1. INTRODUCTION

The vast growth in air pollution emissions, particularly in metropolitan areas, has made the world's air quality a major environmental issue. Due to rising pollution and its detrimental effects on human health, the quantity and distribution of pollutants within metropolitan areas are receiving a lot of attention [21].

Contrary to the research on the distribution of fire in peatlands [23], this research focuses on fires in urban areas. Previous research on fire cases in urban areas has been carried out with fire sources originating from fuel tanker trucks using Computational Fluid Dynamics (CFD) method [22]. The results show that the buoyancy of the fire pollutant and the intensity of the perpendicular ambient wind flow control its dispersion. The buoyancy of the fire pollutants controls their dispersal in the low wind velocity case. However, in higher wind velocity cases, the dispersions of fire pollutants are controlled by their intensities.

The densities of the fire products and the ambient air change due to temperature, and as a result, fresh air from the surroundings enters the product stream. When there is wind at the street canyon's roof level, the ambient air that the wind force is pursuing combines with the fire byproducts. The wind speed has a direct correlation with the mixing intensity. Therefore, the fire products dispersions in the presence of wind at the street canyon roof level are caused by turbulence mixing.

When urban fires occur, the ambient air quality in street canyons will deteriorate if the pollutant emissions from the fires are not adequately ventilated. In this emergency condition, the flow pattern inside the road canyon will experience a significant difference from normal conditions due to the strong thermal buoyancy effect caused by the fire [6].

Under perpendicular crosswind conditions, an unanticipated fire occurs on the ground of the street canyon, releasing smoke and hazardous gases with varying buoyancy strengths, and as wind speed increases to a certain degree, the fire-induced pollutant plume will re-circulate and build inside the roadway canyon. The smoke which recirculated back into the street canyon after the wind velocity exceeds the critical re-circulation velocity, putting both vehicles and pedestrians in grave risk [8].

CFD also can be used to analyze the efficiency between piano key weir and rectangular labyrinth weir to handle increased flood flows [12], gas turbines [28], photo bioreactors [14], metallurgy, biomedicine, and general industrial process [4]. In this study, CFD is being used to analyze fire smoke dispersion patterns and temperature distribution patterns around the fire building

Previous study has shown that the shape of the roof could greatly alter the parameters of the flow pattern and pollution distribution, it demonstrated how the height and design of buildings' roofs could drastically alter how street canyons' flow patterns behaved, affecting how pollutants were distributed [9]. The analysis in this study was conducted on a high-density city scale with various types of buildings (slums, simple house buildings, and high-rise buildings), complementing earlier research that only examined dispersion patterns based on the type of roof and the height of the building, or focused on the street canyon.

Jakarta, the Capital of Indonesia, has dense population that increase every year. One of problems appear from densely populated settlements is fire case. Cases of fire that occur on densely populated settlements can come from human error factors (human activities) [10].

Based on the fire history data by DKI Jakarta Province Fire and Rescue Service, many cases of fire in Jakarta caused by short-circuit electricity. One of the most densely populated areas that are prone to fire is Kebon Melati Sub-district, located in Tanah Abang District, Central Jakarta. In the last twenty years, there have been three cases of fire in Kebon Melati Sub-district.

This study describes the method needed to analyze smoke distribution if a fire occurs in an area with the characteristics of the Kebon Melati Sub-district, a densely populated area with different types of buildings, this analysis method can be used in future research for other similar areas.

In DKI Jakarta 2030 Regional Spatial Plan (RTRW), DKI Jakarta government maps Kebon Melati area as a middle-class vertical residence that combines inspection roads, green open spaces, and adequate transportation rates to Tanah Abang Market. That plan was formed because the government's target to make Tanah Abang a center for world trade [25].

The purpose of this study was to determine and explain smoke dispersion patterns and temperature distribution patterns in Kebon Melati Sub-district fire simulations using Computational Fluid Dynamics (CFD). In addition to knowing the smoke dispersion patterns and temperature distribution patterns in fire simulation on city scale, also determined factors related to smoke dispersion patterns and temperature distribution patterns. So that it can be predicted in areas that will experience air pollution due to fire smoke dispersion and areas prone to fire propagation in Kebon Melati Sub-district.

2. RESEARCH SIGNIFICANCE

This study can determine and explain smoke dispersion patterns, temperature distribution patterns, and wind flow patterns in Kebon Melati Sub-district fire case. So that it can be predicted which areas that will experience air pollution due to fire smoke dispersion and areas prone to fire propagation. Implicate with the evacuation process can determine the side that safest and can be used as an evacuation route of fire cases. This study can be used as a basis for application to other similar fire cases.

3. METHOD

In analyzing Kebon Melati fire case, the CFD method can be used to produce information about fluid flow and heat transfer patterns on fire conditions in Kebon Melati Sub-district. CFD is one of the numerical methods in fluid mechanics which basically uses the philosophy of discretization as an approach in solving mathematical equations. The use of CFD is very useful in virtual prototype analysis of a system or tool that will be analyzed by applying real conditions in the field [20].

Computational Fluid Dynamics/ CFD is the study of how to predict fluid flow, heat transfer, chemical reactions, and other phenomena by solving mathematical equations (mathematical models) [26]. The simulation process in CFD in general is divided into three stages, preprocessing, solving, and post processing [15]. The previous step calculation results are interpreted into several forms including graphs, curves, animations, 2D and 3D images.

Preprocessing is the first step in a simulation that is making a preparation of building's geometry model in the form of CAD (Computer Aided Design), creating a grid or mesh, and determining the boundary conditions of the geometry. Solving is the calculation process and the equations contained in the CFD model are solved using a computer program in accordance with the conditions that have been determined during preprocessing before. Postprocessor displays visualizations including:

1. The results of geometry and cells that have been formed;
2. Plots based on vectors;
3. Contour based plots; and
4. Plot based on the surface (two dimensions or three dimensions) [16].

One of the CFD method modeling programs in fire simulation is Fire Dynamic Simulator (FDS). FDS is a CFD method modeling program for fluid flow with buoyancy style [17].

FDS was developed by the National Institute of Standards and Technology (NIST) in collaboration with the VIT Technical Research Center of Finland, and other communities engaged in the field of fire safety. In visualizing the results of programming, FDS is continuous with the Smokeview program. Both programs are free and open-source accessible on the official NIST website.

FDS completes basic mass conservation, momentum equations, and energy for thermal expansion, as well as ideal gases with various mixture components. FDS applies the Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) turbulent models [7].

In its program, FDS requires a basic input line including building geometry, computational cell size, the location of the fire source, heat release rate (HRR) of the fire source, physical properties and thermal properties of the wall, as well as the size, location and time of ventilation will affect the growth and spread of fire [2].

The study area in this simulation is Kebon Melati Sub-district, Tanah Abang District, Central Jakarta. The research methodology can be observed in Figure 1. The retrieval of meteorological data in the form of dominant wind direction and velocity data was carried out in November 2017. Three weeks of data collection was the dry season, and the last one week of data collection was the rainy season. The meteorological data obtained was validated with BMKG meteorological data (the Meteorology, Climatology and Geophysics Agency) in November 2017 which can be accessed through the official BMKG website. Data analysis techniques used are quantitative data analysis techniques by Computational Fluid Dynamics (CFD) simulation method.

The software used in this study includes Microsoft Office 2010, Microsoft Excel 2010, AutoCAD 2015, Revit 2015, Autodesk Flow Design, Fire Dynamics Simulator. In this study, the AutoCAD 2015 license, Revit 2015, and Autodesk Flow Design were obtained by educational licenses that have 3 years of validity period.

Meteorological data analysis techniques were obtained using Microsoft Excel 2010 to determine the distribution of dominant wind direction and velocity in the form of tables and charts of Wind Rose. The analysis phase was also carried out to determine the flow pattern and temperature distribution and air rate around the fire simulation location. Analysis of flow patterns and temperature distribution and air velocity are obtained through CFD simulation methods.

The first stage in CFD simulation is the creation of location geometry obtained by Google Maps and converted in the form of two-dimensional (2D) line sketches using AutoCAD 2015 with 1:26 scale. The sketch of 2D lines was converted back into three-dimensional (3D) sketches using Revit 2015. The types of buildings in Revit 2015 modeling were divided into three types of building models, slums, simple houses, and high-rise buildings. The determination of the three types of buildings is based on field observations that have been carried out. Slum houses concentrated on the West side of Kebon Melati Sub-district. Multi floor buildings are concentrated in parts of the North and East sides, while simple houses are spread on the North, Central and South sides of Kebon Melati Sub-district.

At Revit 2015, building lines are converted into walls of varying types and heights. The walls of the slums are made of wood or plywood, while the walls of simple houses are made of brick material, and the walls of high-rise buildings are made of concrete.

The wall thickness of slums and simple houses in Revit 2015 software modeling is 0.1016 meters and the wall thickness of high rise buildings is 0.2032 meters. Variations in the type and thickness of the walls of the building can be arranged in the Properties column of each wall. In the case of slums, there is no variation in the number of floors of buildings, slum houses only have one floor of the building. Whereas a simple house building has a variety of building floors, one-floor building, and two-floors building. Multi floors building have variations in the number of floors, namely three floors to twenty floors. The height of each floor of the building is 3 meters and is set on the Revit 2015 leveling process.

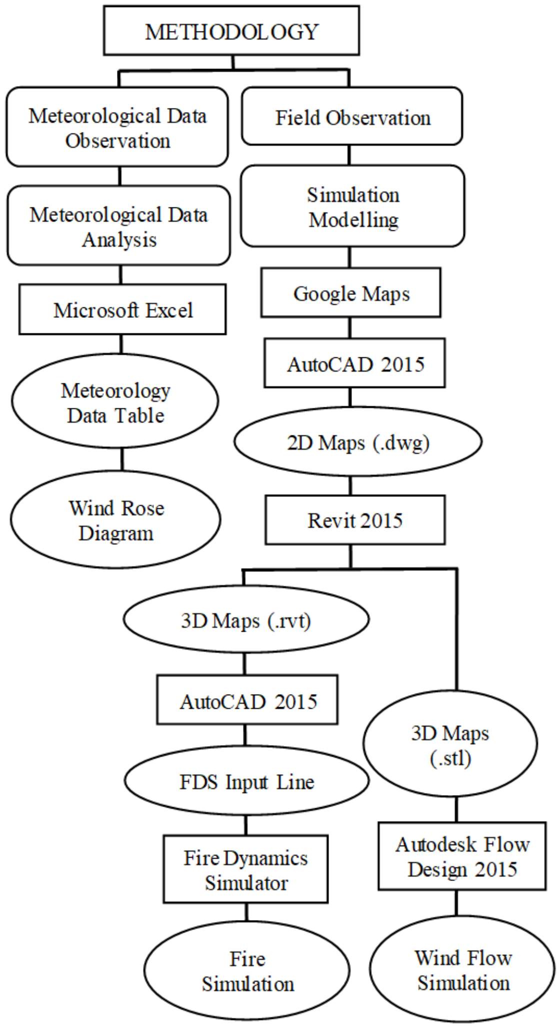


Fig. 1 Research methodology

After finishing 3D geometry, it exported into Fire Dynamics Simulator. In modeling FDS software, there are several input data that need to be prepared. The input data includes building property data in the form of building geometry and its constituent material, meteorological data in the form of wind direction and wind velocity, desired output parameters, ambient state of the environment in the simulation domain.

Setting the ambient state in the simulation domain includes mesh settings, wind velocity and wind direction, combustion reactions that will cause sparks in the simulation, starting point of fire and fire strength, and the desired simulation Mesh used has an X-axis that extends from the North side to the South side by 76 units, the Y-axis that extends from the Westside to the Eastside extends for 72 units, and 4 units of Z-axis. X and Y mesh axes have a unit value of 0.25 m, while the Z-axis has a unit value of 0.1667 m. So that the total dimensions of the mesh in this simulation are 18 m x 19 m x 4 m.

FDS operations require a very large computer storage capacity in large-scale geometry modeling. Therefore the 3D sketches that have been formed are scaled down to a scale of 1: 12,000 using Revit 2015. Fire Dynamics Simulator will also obtained room scope fire simulation with fire-triggering specifications, the type and utilization of burned buildings.

Through Fire Dynamics Simulator, we can obtain temperature distribution patterns on burned objects and in the surrounding objects. In determining the fire smoke dispersion pattern, 3D geometry is exported to the Autodesk Flow Design program for further modeling with specifications of wind direction and velocity.

4. RESULTS

Fire simulations were carried out at five different points in Kebon Melati Sub-district, as in Figure 2. The four points are determined with the assumption that they represent the four cardinal directions, namely North, East, South, West and one point in the middle of Kebon Melati Sub-district. Based on the field observations that have been carried out, the types and shapes of the buildings at the five points are divided into three types of buildings namely slum houses, simple houses, and high-rise buildings.

Point 1 on the north side and Point 3 on the south side of Kebon Melati Sub-district represents simple house buildings. Point 2 on the middle side and Point 4 on the east side of Kebon Melati Sub-district represents high-rise buildings, according to the observations that have been made. While Point 5 on the west side of Kebon Melati Sub-district represents slums.

Observation of meteorological data in the form of measurements of velocity and wind direction observations carried out for one month in November 2017, the first three weeks are the dry season and the last one week is the rainy season. Wind measurements carried out at five measurement points were analyzed using Microsoft Excel in the form of simple Wind Rose tables and diagrams formed from Radar type diagrams. Observations of meteorology data can be observed in Table 1.

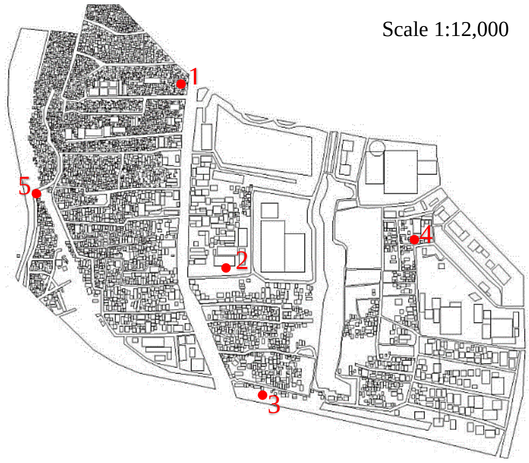


Fig. 2 Meteorological data observation points and fire simulation points

In the last week of measurement, there was an increase in wind velocity blowing in Kebon Melati Sub-district. This is caused by the transition of the season between the dry season to the rainy season the following month. The measured wind velocity converted in the form of a simple Wind Rose diagram. The Wind Rose diagram in this study is a diagram that illustrates the distribution of wind velocity and wind direction distribution based on the number of events.

In the wind velocity conversion process of Wind Rose diagram data in Figure 3, wind velocity grouped and the frequency of occurrence calculated based on sixteen wind directions. The grouping of wind velocity is divided into six groups of velocity with a velocity range of each group of 1 m/s.

The wind group consists of groups with wind velocity range of 0 m/s to 1 m/s, groups with wind velocity range of 1 m/s to 2 m/s, groups with wind velocity range of 2 m/s to 3 m/s, groups with wind velocity range of 3 m/s to 4 m/s, groups with wind velocity range of 4 m/s to 5 m/s, and groups with wind velocity of more than 5 m/s.

Combined Wind Rose Diagram of five observation points in Figure 3 shows that in November 2017 the average wind observed at five observation points tends to blow towards the South with wind velocity of 1 m/s to 2 m/s. This trend is obtained from the observation of the number of events during November 2017, twenty incidents of wind gusts towards the South with wind velocity of 1 m/s to 2 m/s. So that is determined the wind flow modeling in Autodesk Flow Design is carried out with wind direction control to the South with 2 m/s wind velocity.

Kebon Melati Sub-district area modeling begins with a depiction of a line sketch map of Kebon Melati Sub-district area obtained from Google Maps. The map of area was redrawn with the Rectangle and Polyline commands in the AutoCAD 2015 so that the final results in the form of a line sketch map of the Kebon Melati Sub-district area, as in Figure 4. The sketch is then processed into a three-dimensional sketch with the help of Revit 2015.

Table 1 Meteorological data of five observation points

Date	Wind Velocity (m/s) Location (Points) 1	2	3	4	5	Wind Direction
1	2.1	2.6	2.6	2.2	2.0	SBD
2	1.0	1.3	1.1	0.7	0.8	S
3	2.1	2.4	2.2	1.9	1.9	S
4	0.8	1.5	1.4	0.9	0.8	Ti
5	1.9	1.7	2.1	2.4	1.9	SBD
6	1.9	2.5	2.0	1.7	1.1	S
7	3.5	2.8	2.8	3.1	3.4	SBD
8	1.5	1.3	0.9	1.5	1.1	S
9	0.8	1.7	1.4	1.2	0.9	Te
10	2.4	2.2	2.4	1.9	1.9	S
11	1.5	1.8	2.0	1.7	1.5	S
12	2.0	1.9	2.1	2.4	1.9	ST
13	2.2	2.6	2.6	2.2	2.0	ST
14	1.5	2.5	2.1	1.7	1.1	Te
15	2.3	2.8	2.8	2.2	2.4	Ti
16	1.1	1.7	1.5	1.1	1.5	BD
17	2.2	2.3	2.1	1.7	1.8	TL
18	1.1	0.7	1.1	1.4	0.9	TT
19	0.8	0.8	0.8	1.1	1.0	SBD
20	2.2	2.2	2.5	1.9	1.9	ST
21	2.5	1.9	2.1	2.4	1.9	TTL
22	1.9	1.7	1.4	1.8	1.8	Ti
23	2.1	2.5	2.5	2.1	2.4	Te
24	3.6	3.6	3.2	2.6	2.9	TL
25	5.1	4.3	4.3	3.9	4.6	TTL
26	5.4	5.0	4.4	4.4	4.5	TTL
27	6.1	6.2	5.9	5.3	6.2	TL
28	5.5	5.5	4.8	5.1	5.4	TL
29	4.3	4.5	4.6	4.2	4.2	TTL
30	3.8	4.0	3.8	3.5	3.5	TTL
Av.	2.51	2.62	2.52	2.34	2.31	

Note:
BD= Southwest; S= South; SBD= South Southwest; S = South Southeast; Te = Southeast; Ti= East; TL= Northeast; TT=East Southeast; TTL= East- Northeast

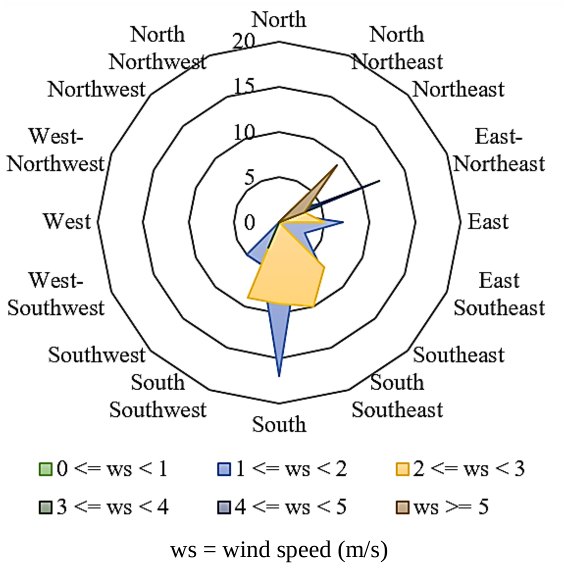


Fig. 3 Wind rose diagram of five observation points

In Revit 2015, building lines are converted into building's wall with varying types and heights. The walls of the slums are made of wood or plywood, while the walls of simple houses are made of brick material, and the walls of high-rise buildings are made of concrete. The height of each floor of the building is 3 meters and is set in Revit 2015 leveling process. The three-dimensional conversion results can be observed in Figure 5, as follows.

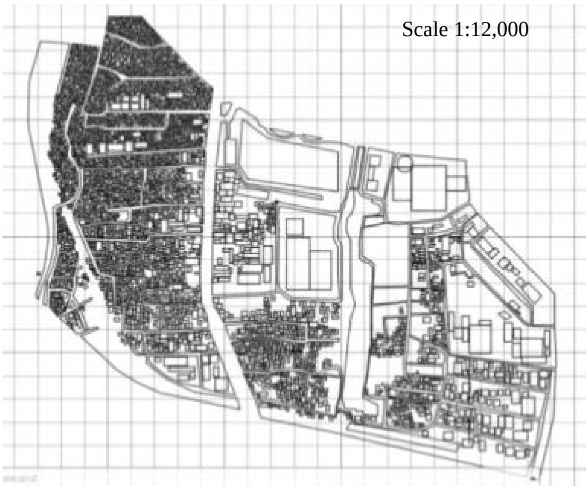


Fig. 4 AutoCAD 2015 results

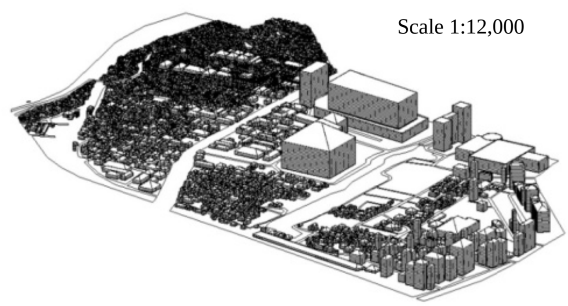


Fig. 5 Revit 2015 modeling results

Autodesk Flow Design's modeling produces wind flow model with 2 m/s of velocity to the South. Modeling is limited to the observation of velocity and wind flow patterns. The fluid modeling used is the Flow Line model with fluid lines and tubes line type to facilitate the observation of the fluid flows pattern in the form of elongated lines.

Top side observation aims to observe the pattern of wind flow horizontally at different heights. Top side observation results can be observed in Figure 6 and Figure 7. Observation variations of height include wind flow at altitudes below 3 meters (low) and wind flow with altitudes more than 3 meters (high).

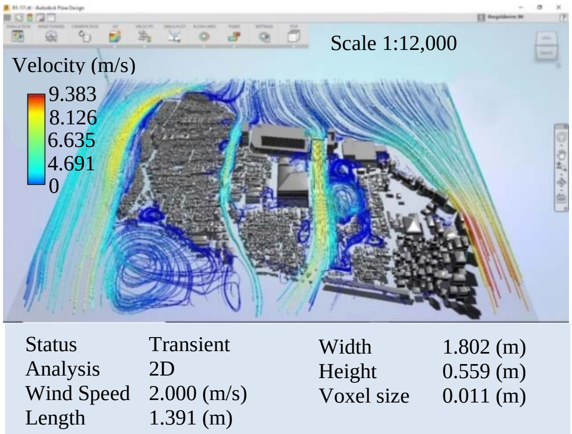


Fig. 6 Low wind flow modeling

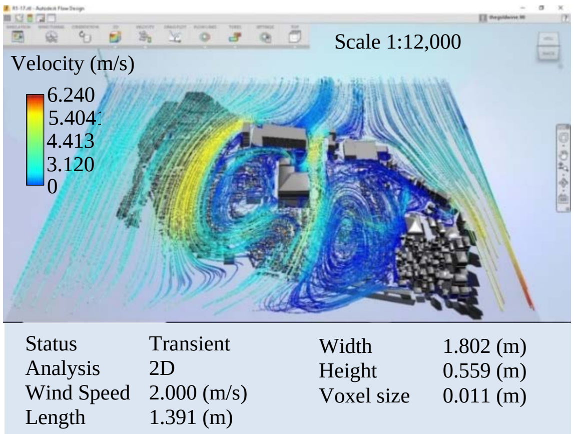


Fig. 7 High wind flow modeling

Based on these two modeling, several color lines can be observed. The difference in the color of the line illustrates the different of wind velocity. The red line shows the highest wind velocity while the dark blue line shows the lowest wind velocity, which is close to 0 m/s on the Southwest side of Kebon Melati Sub-district. Modeling with wind flow altitude variations caused differences of wind velocity range despite of equal initial wind velocity, which is 2 m/s. In modeling with low altitude, the largest wind velocity reached 9 m/s, which is on the eastern side of the Kebon Melati Sub-district. This velocity occurs as a result of the wind flow deflection in the area that has tight distance between buildings.

The largest wind velocity in modeling with high wind flow altitudes only reach 6 m/s, with the same flow location as low wind flow altitude modeling. The deceleration of wind velocity is due to the difference in building height in the area with the distance between buildings that are quite tight. The difference in height of the building causes a gap that allowed the wind to flow so that the wind flow hits the area is not completely deflected. The lowest wind velocity can be observed in the area surrounded by high-rise buildings on the Middle side of Kebon Melati Sub-district, with the turbulence of wind flow forming a circular flow pattern.

In accordance with Hosker analysis [3], buildings can change wind flow fields and ward off wind flow, causing wind flow to rise and wind flow to fall, horizontal and vertical vortices between buildings, on street corners, and other places on the Urban Canopy Layer. The wind flow pattern produced is also in accordance with the research conducted by [18] and [15].

The wind flow around the building is broken down at the side of the building. In both models, turbulence can also be observed in the form of a circular flow pattern in an area that is lower than the height of the building. This turbulence occurs with lower wind velocity compared to deflected wind velocity. In accordance with the experiments conducted by Hoydysh in [3] wind flow on the street canyon forms a vertical vortex pattern paralleled with the axis to the street canyon and affects pollutant concentration on street canyon and ventilation outside the street canyon.

Side viewpoint modeling, the East side of Kebon Melati Sub-district, aims to observe the pattern of wind flow vertically in different building types. Variations in this modeling are carried out with variants of high-rise buildings, slums and simple buildings.

The largest wind velocity generated from two models has a slight difference. The largest wind velocity range in variants of high-rise buildings is higher than the variants of the slums and simple buildings. The largest wind velocity occurs when the coming wind hits the building so that the wind flow deflected toward the building.

In contrast to the research on low-rise buildings [19], in this FDS simulation, the roofs of slums and simple buildings that have low building height are planar shaped so that the recirculation area produced due to the sharpness of the roof cannot be observed clearly.

Figure 8 shows modeling with variants of building heights. In this modeling, turbulence can be observed in areas covered by buildings from wind exposure. Turbulence is in the form of a flow line that rolls up at a velocity lower than the velocity of the wind that hits building. This phenomenon is in accordance with the research conducted by [5], the wind flow pattern in the Urban Canopy Layer (low altitude area with several buildings as a barrier), will experience turbulence in the form of circular flow patterns due to interaction with the barrier.

The circular flow pattern can increase pollutant concentration on the street canyon. The flow will be stable in the Inertial Sublayer that is above the Urban Canopy Layer area due to the absence of buildings that cause deflection interaction of wind flow.

In the high-rise buildings modeling, turbulence can be observed between its gaps. Turbulence forms a circular flow pattern with wind flow velocity up to 1 m/s. The turbulence of air flow can increase pollutant concentrations in street canyon. Increased flow velocities in streams with altitudes exceeding the height of high rise buildings occur due to the upward flow of wind flow by high rise buildings. The highest wind flow velocity reaches 2.8 m/s which is at the point of wind flow deflection by high-rise buildings, illustrated by the red line

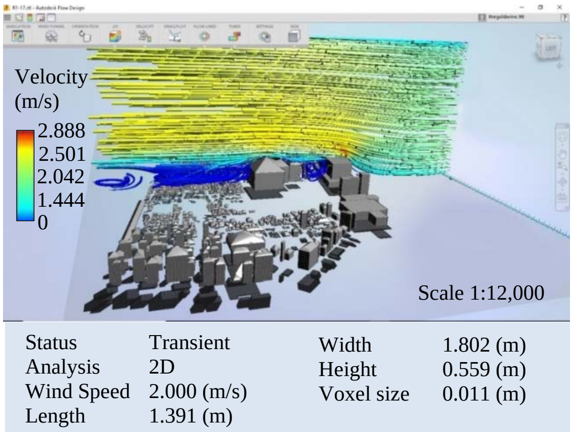


Fig. 8 Wind flow modeling on different height of building

Wind flow velocity and flow pattern in slums and simple house buildings modeling are more stable and constant than high-rise buildings modeling. This can be observed in Figure 9. The biggest velocity reaches 2.4 m/s on deflected wind flow after hit the outermost building from the direction of the wind coming. The velocity is represented by a red line. Wind velocity between slums and simple buildings is only between 0 m/s to 1.2 m/s. While the wind velocity at high altitude (more than 3 m) reaches 2.2 m/s which is depicted with a yellow line.

After obtaining Autodesk Flow Design modeling, the three-dimensional sketches generated from Revit 2015 software are also stored in drawing format (.dwg) that supported by AutoCAD 2015 software. These three-dimensional sketches are converted in the form of input lines that correspond to the Fire Dynamic Simulator (FDS) format. The conversion is done with the 3dsolid2fds command line in the AutoCAD 2015 software which was developed separately from the software.

In FDS modeling, there are several input data that need to be prepared. These input data includes building properties in the form of building geometry and its constituent material, meteorological data in the form of wind direction and wind velocity data, desired output parameters, ambient state of the environment in the simulation domain.

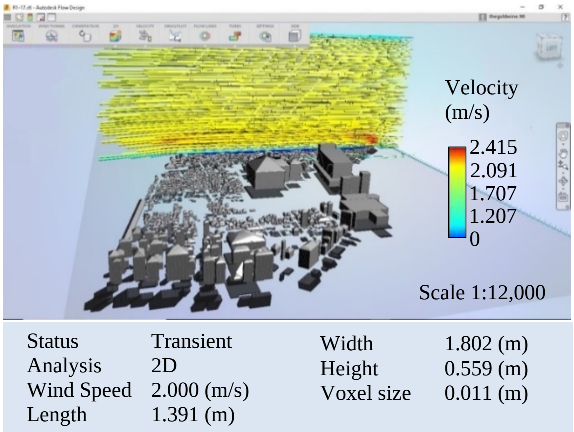


Fig. 9 Slums and simple buildings wind flow modeling

The mesh used has X dimension that extends from the North side to the South side for 72 units, the Y dimension extending from the Westside to the East side which extends along 76 units, and the Z dimension representing mesh height for 4 units. The geometry of the building and its constituent material has been determined in the previous 3dsolid2fds conversion. The geometry of the simulation area can be observed in Figure 10.

At Point 1, Point 3, and Point 5, the starting point of the fire (hotspots) is simulated on a flat plane located at the top of each building. Unlike the research conducted by [13], hotspots in this simulation building are carried out on the roof of a building that has direct contact with the wind flow. Complex physical and chemical modeling in electrical short circuit case occurs on smaller modeling scale than the computational cell size used in FDS simulations [2]. So in this simulation, the source of the fire is from the planar plane on the top side of the building.

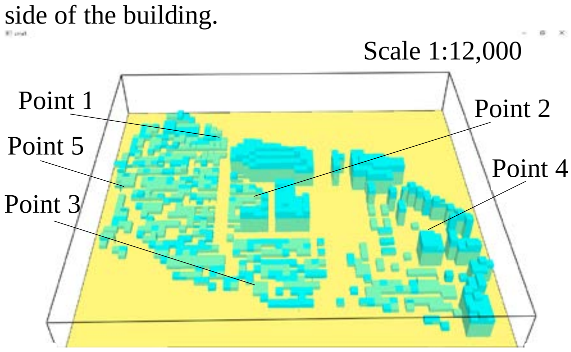


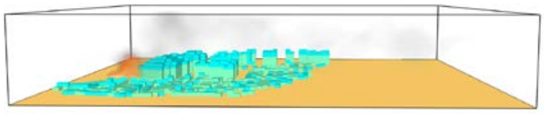
Fig. 10 Five observation points geometry

Fire simulations in FDS software can only be run on fire generated by a chemical reaction in each simulation. The reaction can only be processed on a flat plane that has been determined its surface constituent material. Results of five fire simulation points can be observed in Figure 11. Fire simulations are carried out within a period of 30 seconds. In this period of time, the smoke flow and radiation patterns around the simulation point can be clearly observed.

Fire and fire smoke parameters from time to time are then studied by FDS. In contrast a study conducted by [27], with 700 kW/m^2 of Heat Release Rate per small area unit (HRRPUA) and ignition time of 35 seconds. The fire simulation in this study uses 10,000 kW/m^2 of HRRPUA. The amount of HRRPUA determines the magnitude of the starting point (hot point) of the simulated fire, and the period of time determining the duration of the burning time. The use of large HRRPUA is intended to facilitate visualization of the dispersion of smoke produced.

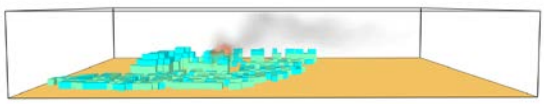
Based on the visualization of the five fire points, it can be observed that the resulting fire and the smoke produced are quite large. The resulting smoke interacts with the force produced by the wind flow that blows from the North to the South by 2 m/s. This is in accordance with previous research conducted by [24], the continuity of combustion is determined by the availability of oxygen which is affected by openings or ventilation around the fire point. These interactions cause fire smoke dispersions that are more likely disperse to the south side of the fire point. In accordance with the wind flow modeling in Autodesk Flow Design, the resulting fire smoke dispersion flow pattern is quite stable in areas with slums and simple houses.

a)



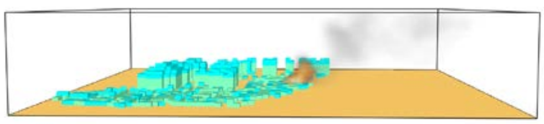
Scale 1:12,000

b)



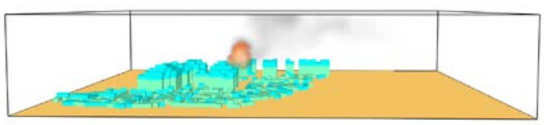
Scale 1:12,000

c)



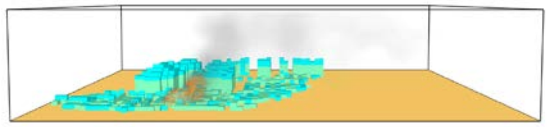
Scale 1:12,000

d)



Scale 1:12,000

e)



Scale 1:12,000

Fig. 11 Fire simulation visualization at a) north point, b) middle point, c) south point, d) east point, and e) west point

Fire smoke also tends to move upward, with altitude exceeding the height of the fire point. The fire smoke dispersion of the five simulation points reached a distance of 1,440 m to the south within 30 seconds of simulation. This was also observed in the simulation of Point 2 and Point 4 above the high-rise building. Fire smoke does not experience turbulence as in the Urban Canopy Layer, smoke tends to move up towards the south. So the potential dispersion of fire smoke concentrated in areas with high building density area on the south side of the fire point is not too significant.

In research conducted by [7] scenario of fire with critical wind velocity, fire smoke will be circulated back to street canyon so that it can endanger pedestrians and potentially cause traffic accidents due to reduced visibility by smoke soot. However, these pollutant fumes can generally be released from street canyon with the help of buoyancy of smoke with strong fire source's buoyancy or at relatively weak wind velocity.

Visualizing the temperature distribution around the fire point, in Figure 12, it can be observed that the increase in temperature does not only occur at its hotspot, but also around the point of fire. The highest temperature in buildings in the North Point and the South Point reaches a temperature of 307°C and the wind that interacts with the fire produces a temperature distribution up to 125°C to the south of the fire point. The fire simulation of high-rise building in the Middle Point produced the highest temperature of 570°C and the wind that interacted with the fire caused a temperature distribution up to 130°C to the south of the fire point.

The pattern of fire smoke dispersion at the Middle Point is more likely to move upwards so that the distribution of air temperature towards the south does not take a long period of time. While the fire simulation at the East Point produces the highest temperature of 770°C with a temperature distribution up to 170°C to the south of the fire point. That was the highest temperature produced in all five fire simulations.

The achievement of the temperature distribution is caused by the location of the hotspots above the upper floor of a high-rise building in high-rise buildings area.

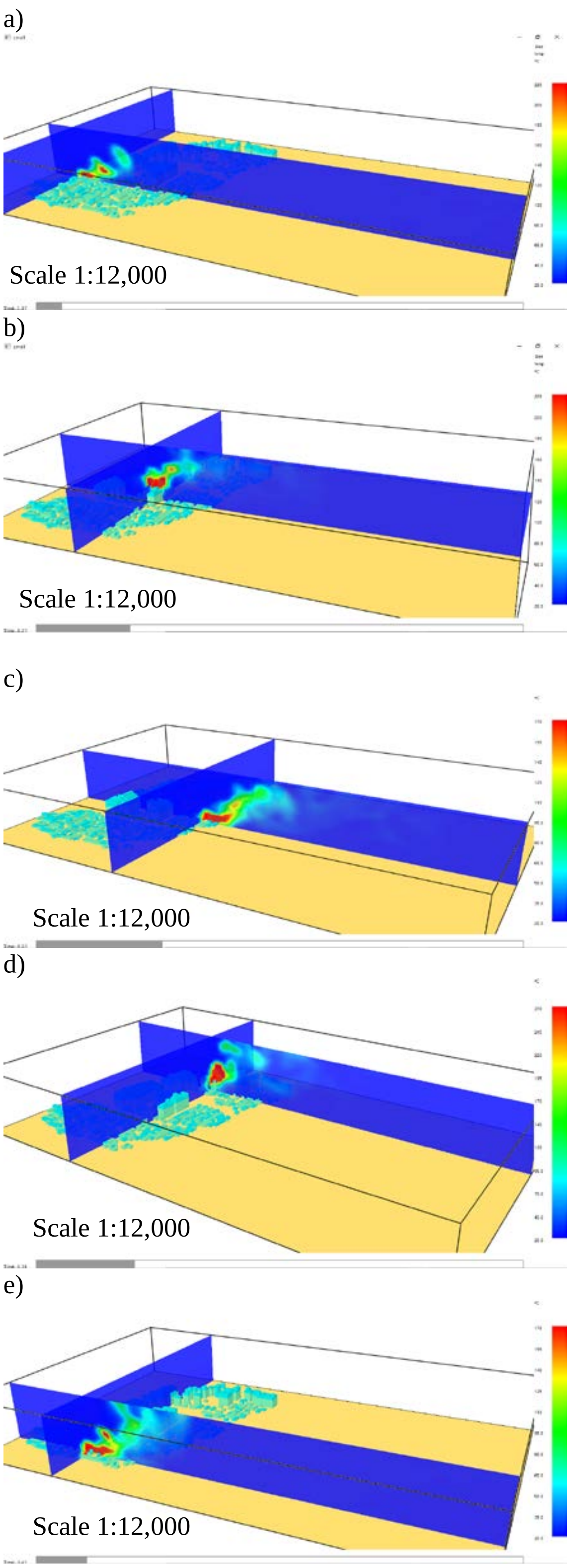
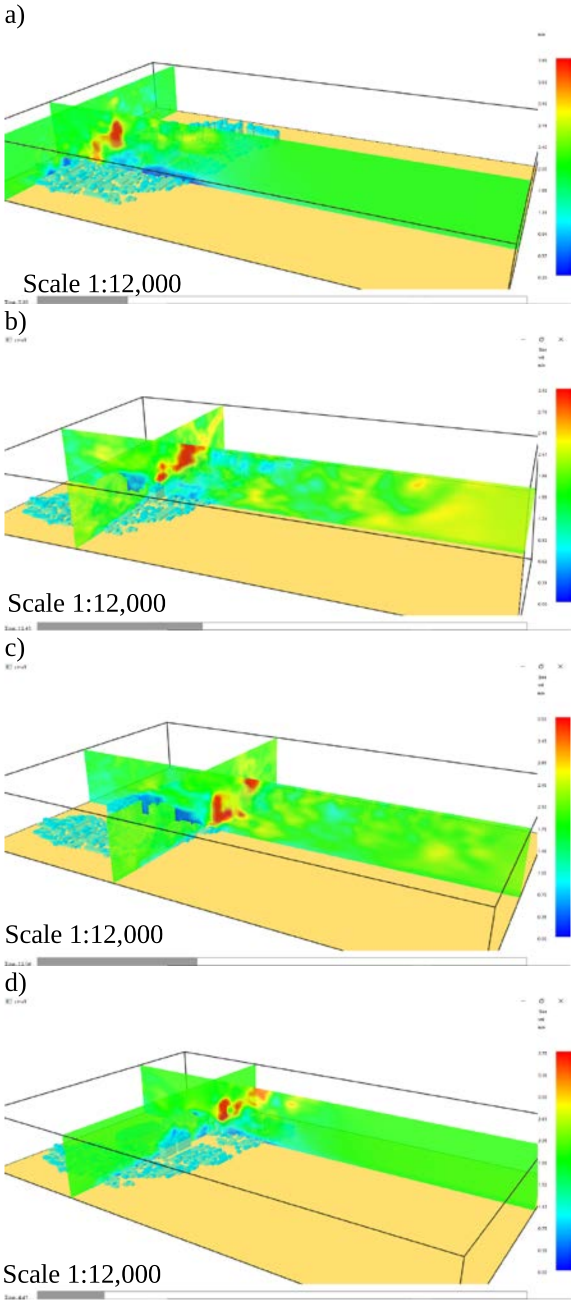


Fig. 12 Temperature distribution visualization at a) north point, b) middle point, c) south point, d) east point, and e) west point

The wind flow that hit the hotspots has a constant flow because there is no turbulence produced on the Urban Canopy Layer. The highest temperature produced in the fire simulation West Point only reaches 420 °C with 140 °C to the south of the fire point.

The high wind distribution at the beginning of the simulation is caused by the initial wind velocity of 2 m/s used in the FDS simulation blowing at the beginning of the simulation. Until a certain time the wind exhaled has decreased velocity causing an increase in wind velocity only in the area around the fire point.

Graph 1 shows the comparison of the rate of heat release at the North Point, Middle Point, South Point, East Point, and West Point of fire simulations. The graph shows the tendency of increasing the rate of heat release the same at the beginning of the fire. But after passing a certain time, the rate of combustion in the East Point increased more sharply than the other four points.



e)

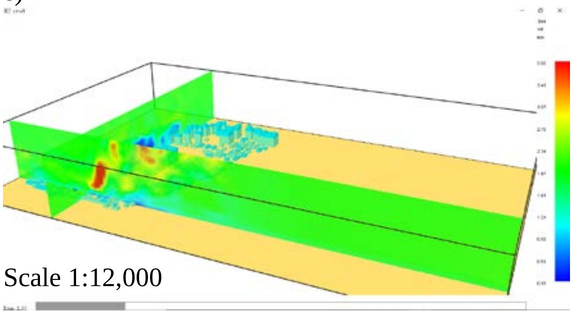
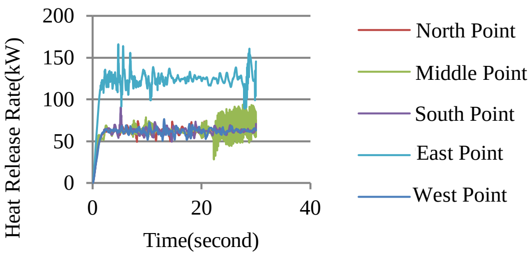


Fig. 13 Wind Velocity Visualization around a) north point, b) middle point, c) south point, d) east point, and e) west point

In the research in Rusunami buildings [24], the higher the wind blows, the higher Heat Release Rate produced. East Point with fire simulation on the top floor of high rise buildings has a higher level of exposure to wind and oxygen than the other four points.



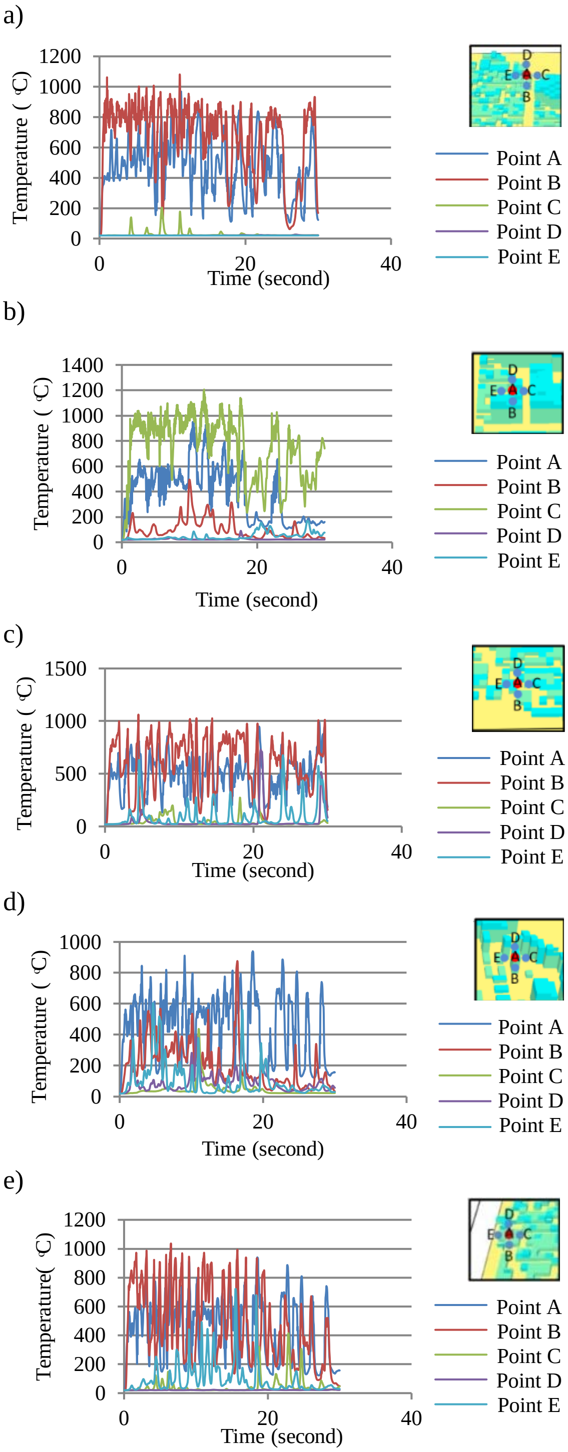
Graph 1 Heat Release Rate of Five Fire Simulations

The Middle Point is also simulated in high-rise buildings, there is a large gap in the surrounding buildings, causing a turbulence flow at the hotspots. North Point, South Point, and West Point show stable heat release rate due to the steady flow of wind in the three points, according to the modeling in Autodesk Flow Design software.

Graph 2 showed the temperature distribution around the fire point at the five fire simulations. There was a sharp increase in temperature at Point A in the middle of the fire point and Point B which was on the south side of the fire point. The sharp increase in temperature at Point B shows the influence of wind power and wind direction that blows around the point of fire.

The high temperature produced by the fire will be distributed in the direction of the wind. In the Middle Point simulation, the highest temperature increase occurred at Point C on the East side of the fire point. This shows that the fire propagation potential due to the air temperature distribution around the hotspot is not only influenced by the direction of the wind that blows but also the turbulence of the wind flow that occurs around high-rise buildings.

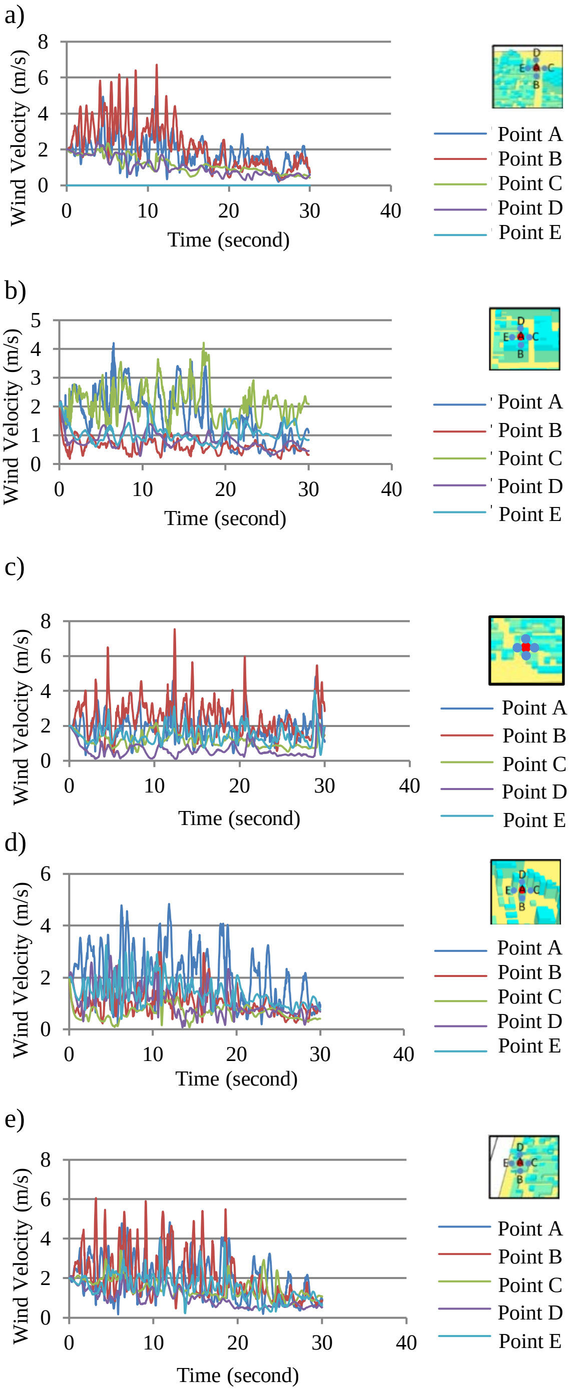
The distribution of wind velocity that occurs around each five fire points can be observed in Graph 3. At the beginning of the simulation, the highest increase rate of wind velocity occurs at the south side of the fire point, namely Point B. Increased wind velocity also occurs at Point A which is in the middle of the fire point. High wind velocity at Point B is caused by a wind flow deflection from a high rise building on the east side of the fire point.



Graph 2 Temperature distribution around hotspot on a) north point, b) middle point, c) south point, d) east point, and e) west point

Whereas in the South Point simulation, the increase in wind velocity at Point B is caused by the accumulation of wind flow velocities which are deflected by the building on the North side of the fire point with a constant velocity of wind flow on the West side of the fire point.

In the Middle Point simulation, an increase in wind velocity occurs at Points A and Point C. High wind velocity at Point A and Point C are affected by turbulence and deflection of air flow by high rise buildings on the North and East sides of the fire point.



Graph 3 Wind velocity distribution on a) north point, b) middle point, c) south point, d) east point, and e) west point

Implicate with the evacuation process of the community around the fire location, the evacuation route used should not be on the south side of the fire point. The southern side of the fire point in the fire simulation Kebon Melati Sub-district has the greatest potential in reducing visibility and increasing the concentration of fire smoke pollution as a result of the phenomenon of smoke dispersion by the wind that is blowing dominantly towards the South. Increase of temperature on the south side of the fire point is also dangerous because it potentially cause fire propagation.

Decreasing visibility can jeopardize the activity of transportation users because it can increase the risk of accidents on the main road or small roads around the fire point. Fire smoke pollution concentrated due to air flow turbulence on street canyon can endanger the health of the respiratory and visual systems of the surrounding community. In condition nitrogen-containing materials such as nylon and woolen cloth are burned, the smoke from the fire will contain hydrogen cyanide (HCN) [1]. Hydrogen cyanide can cause several health problems such as narrowing of the respiratory tract, headaches, vomiting, even death at high doses of exposure [11].

The side that can be used as an evacuation route in this fire simulation is the Westside and the Middle side of Kebon Melati Sub-district. The Eastside which is dominated by high rise buildings has greater potential for the flow of fire smoke recirculation on street canyon due to turbulence flow by high rise buildings, compared to the West side which is dominated by slums and simple houses. Middle Side Kebon Melati Sub-district which is traversed by the main road can also reduce the concentration of fire smoke with high and stable wind flow velocity due to no buildings or objects that brush off the wind flow along the main road.

5. CONCLUSIONS

Wind velocity and wind direction of Kebon Melati Sub-district that used in this study was obtained from meteorological observations in November 2017, with an average wind velocity of 2 m/s heading south.

Wind flow on the West side Kebon Melati Sub-district is more stable than the East side and Middle side which has turbulence in the flow of wind due to the bending of wind flow by high-rise buildings.

In Autodesk Flow Design wind flow simulation, wind velocity increase up to 9 m/s on the eastern side of the Kebon Melati Sub-district due to a wind flow deflection by high-rise buildings.

Wind flow velocities that experience turbulence on the East side and Middle side of Kebon Melati Sub-district only reach velocity of 2 m/s, according to the initial wind velocity.

Fire simulation using Fire Dynamics Simulator results five-point fire smoke dispersion pattern spreads towards the South side of the simulation point, at an altitude of 3 m to 60 m according to the height of the fire point, until it reaches a certain time the smoke tends to move upwards.

Within 30 seconds, the dispersion of fire smoke with 10,000 kW/m^2 of Heat Release Rate per unit Area, reaches 1,440 m to the South of the five fire simulations.

The highest temperature generated in the five fire simulations reached 770° C in the East Point simulation with a temperature distribution of up to 170°C to the south of the fire point.

The highest wind velocity at five fire simulations reaches 3.6 m/s at North Point and West Point with the distribution of wind velocity reaching 2.2 m/s around hotspot.

The highest heat release rate produced in East Point simulation, reaches 160 kW due to the level of oxygen exposure that is greater than oxygen exposure at the North Point, Middle Point, South Point, and West Point.

Temperature distribution around hot point tends to increase at Point B which is on the South side of fires due to the influence of wind flow strength.

The distribution of wind velocity around hot point affected by turbulence of wind flow due to deflection of wind flow by high rise buildings on the North side and the East side of Kebon Melati Sub-district.

The South Side of Kebon Melati Sub-district has the greatest potential in reducing visibility and increasing the concentration of fire smoke pollution, as well as fire propagation potential.

The side that can be used as an evacuation route is the Westside and the Middle side of Kebon Melati Sub-district which has more stable wind flow than East side.

6. RECOMMENDATION

Measurement of meteorological data in a longer period of observation so there will be comparison of simulation on dry season and rainy season. Development of FDS software to be able to simulate the starting point of a fire that is caused by an electrical short circuit. Simulation with other CFD software, such as Fluent and OpenFOAM to get a more accurate comparison of results. Observations can be focused on the three types of buildings (slums, simple houses, and high-rise buildings) and street canyons for further research. It can also be outfitted with mathematical calculations and pollutant concentration distribution. Therefore, it's possible that it will someday serve as a reference for urban planning or emergency evacuation plans.

7. ACKNOWLEDGEMENT

The authors would like to thank Bakrie University for providing support, guidance and opportunities for this research. The author would also like to express high appreciation to the referees of the paper for their critical review and valuable comments which improved the quality of the paper.

8. REFERENCES

[1] Alarie Y., Toxicity of Fire Smoke. Critical Reviews in Toxicology, Vol. 32, No. 4, 2002, pp. 259–289.

[2] Barowy A. and Madrzykowski D.M., Simulation of the Dynamics of a Wind-Driven Fire in a Ranch-Style House – Texas, NIST Technical Note 1729, 2012, pp. 1-86.

[3] Brown M. J., Urban Dispersion-Challenges for Fast Response Modeling, 5th AMS Symposium on the Urban Environment, Vancouver, B.C., 2004, pp. 1-13.

[4] Einarsrud K.E., Loomba V., and Olsen J.E., Special Issue: Applied Computational Fluid Dynamics (CFD), Processes, 2023, Vol. 11, Issue 2, 461, pp. 1-3.

[5] Grimmond C.S.B. and Oke T.R., Turbulent Heat Fluxes in Urban Areas: Observations and a Local-Scale Urban Meteorological Parameterization Scheme (LUMPS), Journal of Applied Meteorology, Vol. 41, Issue 7, 2002, pp. 792-810.

[6] Hu, L.H., Huo, R., Yang, D., Large Eddy Simulation of Fire-Induced Buoyancy Driven Plume Dispersion in an Urban Street Canyon Under Perpendicular Wind Flow. Journal of Hazardous Material, Vol. 166, Issue 1, 2009, pp. 394–406.

[7] Hu, L.H., Xu Y., Zhu W., Wu L., Tang F., and Lu K.H., Large Eddy Simulation of Pollutant Gas Dispersion with Buoyancy Ejected from Building into an Urban Street Canyon, Journal of Hazardous Materials, Vol. 192, Issue 3, 2011, pp. 940–948.

[8] Hua L., Zhang X., Zhu W., Ning Z., and Tang F., A Global Relation of Fire Smoke Re-Circulation Behaviour in Urban Street Canyons, Journal of Civil Engineering and Management, Vol. 21, Issue 4, 2015, pp. 459–469.

[9] Huang Y.D., He W.R., and Kim C.N., Impacts of Shape and Height of Upstream Roof on Airflow and Pollutant Dispersion Inside an Urban Street Canyon, Environmental Science and Pollution Research, Vol. 22, Issue 3, 2015, pp. 2117–2137.

[10] Ismawan D.A., Kajian Kerentanan Kawasan Pemukiman Padat terhadap Bencana Kebakaran di Kecamatan Tambora - Jakarta Barat, Universitas Diponegoro, Thesis, 2008, pp. 1-5.

[11] Kompasiana, Racun Para Teroris Masa Kini (Hidrogen Sianida), June 26, 2015, retrieved from https://www.kompasiana.com/jenywidya/5500e86e813311681ffa7efe/racun-para-teroris-masa-kini-hidrogen-sianida.

[12] Le A.T., Hiramatsu K., and Nishiyama T., Hydraulic Comparison Between Piano Key Weir and Rectangular Labyrinth Weir, International Journal of GEOMATE, 2021, Vol. 20, Issue 82, pp. 153-160.

[13] Lin C.S., Chen M.Y., and Pan L.S., Field Model Simulation and Analysis for a Residential Building Fire, Applied Mechanics and Materials, Vol. 851, 2016, pp. 810–815.

[14] Luzi G. and McHardy C., Modeling and Simulation of Photobioreactors with Computational Fluid Dynamics - A Comprehensive Review, Energies, Vol. 15, No. 11, 2022, pp. 1-63.

[15] Mahan V., Simulasi Penyebaran Gas SO2 dari Emisi Cerobong Menggunakan Computational Fluid Dynamics, Department of Geophysics and Meteorology-Institute Pertanian Bogor, Thesis, 2009, pp 1-42.

[16] Marihot, L., Pola Aliran dan Sebaran Suhu Udara di dalam Rumah Tanaman Tipe Modified Arch di Leuwikopo, Babakan, Dramaga, Bogor, Institut Pertanian Bogor, Thesis, 2015, pp. 1-51.

[17] McGrattan K., McDermott R., Weinschenk C., Overholt K., Hostikka S., Floyd J., NIST Special Publication 1019: Sixth Edition Fire Dynamics Simulator User's Guide, National Institute of Standards and Technology, 2013, pp. 1-288.

[18] Oke T.R. Street Design and Urban Canopy Layer Climate, Energy and Buildings, Vol. 11, Issues 1–3, 1988, pp. 103–113.

[19] Ozmen Y., Baydar E., and Beeck J.P.A.J.V., Wind Flow over the Low-Rise Building Models with Gabled Roofs Having Different Pitch Angles, Building and Environment, Vol. 95, 2016, pp. 63–74.

[20] Pasha A., Simulasi Dispersi Gas Karbon Monoksida (CO) dalam Gardu Tol Menggunakan Computational Fluid Dynamics (CFD) Studi Kasus: Gerbang Tol Bogor, Institut Pertanian Bogor, Thesis, 2011, pp. 1-62.

[21] Pesic D.J., Blagojevic M.D.J., Zivkovic N. V., Simulation of Wind-Driven Dispersion of Fire Pollutants in a Street Canyon Using FDS, Environmental Science and Pollution Research, Vol. 21, Issues 2, 2013, pp. 1270-1284.

[22] Pesic D.J., Zigar D.N., Anghel I., and Glisovic S.M., Large Eddy Simulation of Wind Flow Impact on Fire-Induced Indoor and Outdoor Air Pollution in an Idealized Street Canyon, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 155, Issue 9, 2015, pp. 89-99.

[23] Putra R., Sutriyono E., Kadir S., and Iskandar I., Understanding of Fire Distribution in The South Sumatra Peat Area During The Last Two Decades, International Journal of GEOMATE, Vol. 16, Issue 54, 2019, pp.146–151.

[24] Sujatmiko W., Dipojono H.K., Soelami F.X.N., and Soegijanto. Study on Fire Dynamic Development in a Multistory Building Compartment, Procedia Engineering, Vol. 170, 2017, pp. 162–68.

[25] Transformai, Saatnya Merevitalisasi Tanah Abang, March 9, 2015, retrieved from http://www.transformasi.org/id/pusat-kajian/berita/umum/644-saatnya-merevitalisasi-tanah-abang.

[26] Tuakia F., Dasar-Dasar CFD Menggunakan FLUENT (Basics of CFD Using FLUENT), Informatika, 2008, pp. 1-331.

[27] Wahono E., Analisis Sistem Fire Roller Shutter terhadap Tingkat Keselamatan Bangunan Pasar dengan Simulasi Komputer, Universitas Indonesia, Thesis, 2008, pp. 1-62.

[28] Zhang C and Janeway M., Optimization of Turbine Blade Aerodynamic Designs Using CFD and Neural Network Models, International Journal of Turbomachinery, Propulsion and Power, Vol. 7, No. 3, 2022, pp. 1-19.

[29] Zhang X., Zhang Z., Su G., Tao H., Xu W., Hu L., Buoyant Wind-Driven Pollutant Dispersion and Recirculation Behavior in Wedge-Shaped Roof Urban Street Canyons, Environmental Science and Pollution Research, Vol. 26, Issue 3, 2019, pp. 8289-8301.