International Journal of Application on Sciences, Technology and Engineering
(IJASTE)
Volume 1, Issue 2,2023.ISSN:2987-2499

COMPARATIVE ANALYSIS OF WIND PRESSURE OF UNTAR MAIN BUILDING

Vincent[1, a) and Sunarjo Leman[2,b)]

[1]Civil Engineering Department, Faculty of Engineering, Tarumanagara University, Jakarta, Indonesia 11440

[a)] Corresponding author: *vincent.325180051@stu.untar.ac.id*
[b)]Another author: *sunarjo@ft.untar.ac.id*
Submitted: March 2023, Revised: April 23 2023, Accepted: May 22, 2023

Abstract.

Wind flows from high to low pressure due to presence of solar heat. The movement of wind in air is freely exposed to obstacles such as buildings, especially in large cities. Buildings that are exposed to wind create pressure for building which can be compressive or suction. In this study, wind pressure analysis was used in three ways, that is with MIDAS GEN software, Autodesk CFD, and with manual calculation analysis referring to Indonesia National Standard (SNI) 1727:2020. The building modeling used as test is main building of UNTAR campus 1 with total height 89 m. The tested wind direction moved from south to north and from west to east at a wind speed 32 m/s. The results of analysis showed that percentage of wind pressure results between MIDAS GEN with manuals was at least 5.66% for both wind directions and between CFD and manuals at least 0.7% for wind directions from south to north. The results from CFD also show that result the main building surface wind pressure for wind flows moving from south to north results in wind load in the edge area building smaller than in the central area building. The amount of wind pressure in center of surface can be 4 to 5 times greater than wind pressure at the edge of surface.

INTRODUCTION

The analysis for wind loading basically uses the regulations that apply in each country to design a building structure. Analysis can also be modeled using computer modeling software for structures, such as ETABS, MIDAS, etc. Analysis using building structure modeling is also based in general regulations in force. Another method that can be used to analyze wind loads is to use CFD (*Computational Fluid Dynamic*). This method is based on the calculation of finite elements so that CFD analysis is numerical in mathematics and is not bound by applicable wind load regulations. In CFD, it can be included modeling a wide variety of buildings up to a large scale. CFD works in these mathematical equations help in solving mathematical equations that are so complex. CFD is a branch of science of fluid mechanics that uses numerical analysis and structured data to analyze and solve problems involving fluids. CFD are methods that use numbers, algorithms and computer assistance to perform the analysis of their calculations. The calculation in this CFD is based on the finite element method because the element to be analyzed is required to meshing. [1]

For this study, a comparative analysis was carried out using manual analysis of calculations by referring to calculations to SNI 1727:2020, which this regulation refers to ASCE 7-16 and as a comparison using MIDAS Gen structure modeling software, the building frame structure is made in form of a *frame structure*. Wind load analysis using MIDAS Gen refers to 2012 IBC loading regulation (ASCE 7-10) and compared to also using CFD from Autodesk output with building modeling only the outer geometry. This comparative research on the results of wind pressure and load was carried out for main building of UNTAR as a modeling test object. The main building of UNTAR has a height of 89 m so that it becomes the tallest building in UNTAR campus 1 complex itself.

https://doi.org/10.24912/ijaste.v1.i2.723-730

GENERAL CONCEPT

The Effects of Wind Pressure in Building

In general, the application of wind force in buildings is in form of pressure to side of the wall on surface of the building. The positive wind pressure side is the one that can push the building surface and the negative side of wind pressure is the one that can pull the building surface. The positive wind side is commonly called windward wall and the negative wind side is commonly called leeward wall. For single and simple buildings as shown in figure 1 below, for example, the wind blows from the westward, it can be seen that which side is windward and leeward. Walls parallel to blowing wind direction will be attracted as well. For a gable-shaped roof, the left side can be depressed or attracted depending on the magnitude of the wind speed and the right side is attracted. [2].

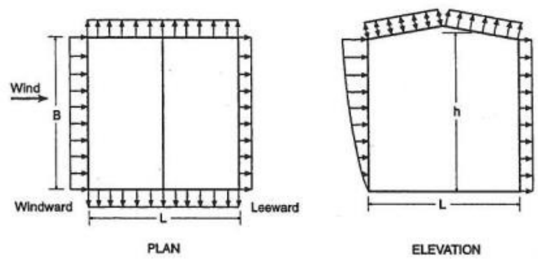

FIGURE 1 Wind Load That Occurs Happen in Single Building [2]

Basically, there are general regulations that regulate the wind load that enters the building. The regulation applies in Indonesia today, namely SNI 1727 regulates the loading system. The rules follow ASCE 7-16 regulations in force in America. The wind load that enters the building according to applicable rules is calculated statically and simplified as an even load, but in fact the wind does not apply like that, the wind moves dynamically where the situation is always changing so that the pressure that hits the building changes. [2]

The wind flow passing around the rectangle will be separated towards left and right as in figure 2 below (in the picture moving up and down). The separate wind flow will form a vortex due to its instability, and then it will reunite after passing through the building. The wind pressure point also changed to become larger because separate of wind speeds formed a vortex greater than the wind speed before hitting the building itself. The wind only affects the outer frame of the building, has no inward effect, except that no one opens the surfaces. [3]

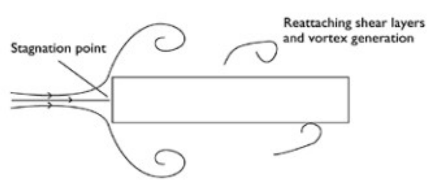

FIGURE 2 Wind Behavior Around Building [3]

RESEARCH METHODOLOGY

Modelling the Main Building of UNTAR Campus 1

With Software MIDAS GEN

Midas Gen is a *software* for modeling structure of the civil field (usually multi-storey buildings) to facilitate structural planning. In this study, this *software* was used to analyze the wind load profile that occurred in the main building of UNTAR. Basically, the calculation of wind loads using MIDAS Gen follows several regulations that apply in this world. These regulations generally apply in respective countries that issued the regulations, but the commonly used charging regulations for whole are ASCE or IBC. In this study, regulations from the 2012 IBC were used which were compared to SNI 1727:2020 from Indonesia. Modeling the main building presented in figure 3 below:

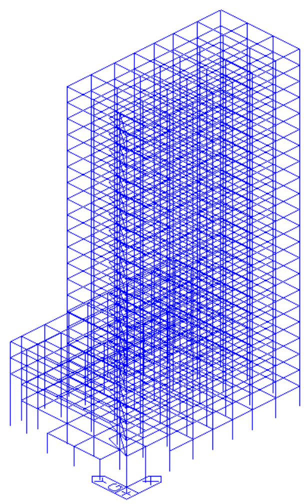

FIGURE 3 Main Building of UNTAR Structure Modeling in MIDAS Gen Software

With Manual Calculation of SNI 1727:2020

Manual calculations using SNI 1727:2020 do not need to be used modeling, this calculation refers to the provisions that have been regulated in SNI. Various parameters must be obtained in accordance with SNI to get the load and wind pressure that occurs.

With Software Autodesk CFD

CFD is special *software* to account for fluids such as wind flow. Modeling in CFD is different from MIDAS Gen, if in MIDAS Gen the structure of the building is modeled, in CFD it is modeled a real whole building modeled in Revit first. After that, it can only be imported into CFD as a test object or which will be researched. Modeling view of the test object in the CFD as in figure 4 below.

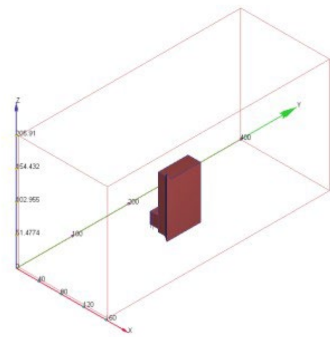

FIGURE 4 Main Building of UNTAR Geometry Modeling in Autodesk CFD

Wind Direction and Speed Test Data

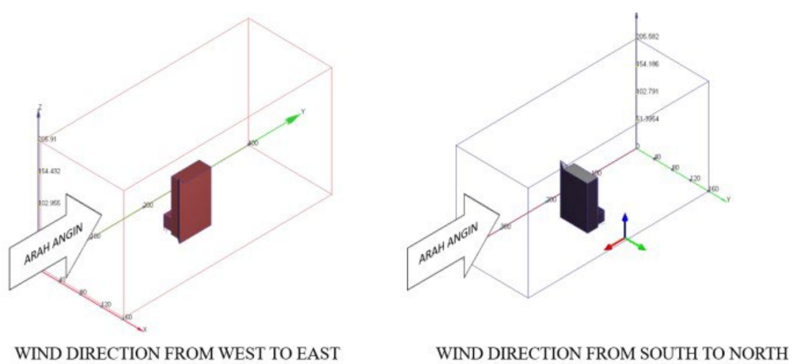

FIGURE 5 Wind Flow Direction in Autodesk CFD

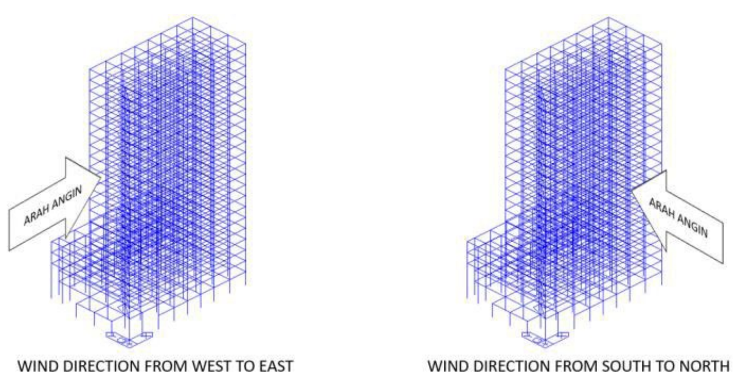

FIGURE 6 Wind Flow Direction in MIDAS Gen

FIGURE 7 Wind Flow Direction Seen From the Top View of Google Earth

The Wind Direction to be tested in condition of 90° axis of wind from west to east and wind from south to north as presented in figure 5, figure 6, and figure 7 that for the better seen its look from the top of Google Earth. The wind speed used is the base wind speed with a repeat period of 50 years in Indonesia, which is 32 m/s.

RESULTS AND DISCUSSION

Comparison of Wind Pressure Results With MIDAS Gen Software, Autodesk CFD, and Manual Analysis Referring to SNI 1727:2020

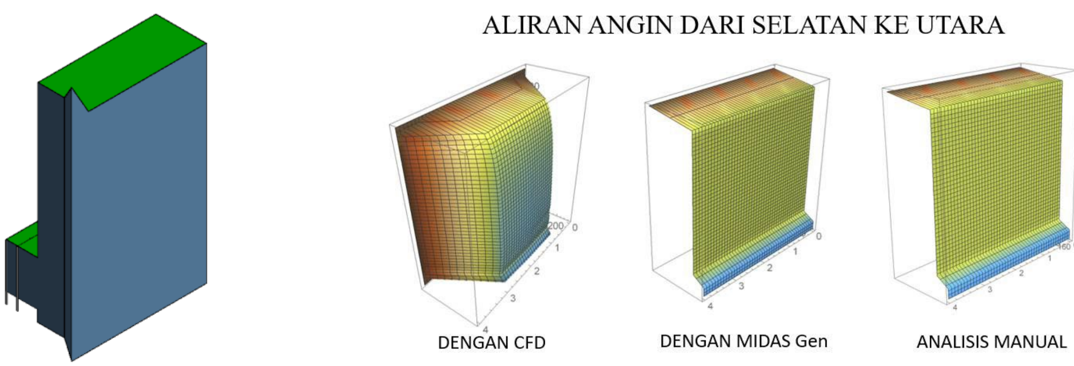

FIGURE 8 Comparative Results of Wind Load Calculation from South to North with CFD Software , MIDAS Gen, and with Manual (SNI 1727:2020)

FIGURE 9 Comparative Results of Wind Load Calculation from West to East with CFD Software , MIDAS Gen, and with Manual (SNI 1727:2020)

It can be seen for the wind load profile of manual analysis and MIDAS Gen approximately are the same shape. For the direction of wind gusts from south to north is compared in a 3-dimensional graph profile because it is to show the magnitude of wind load at the edge and in the middle. Therefore, the analysis has an axis that takes into account the width of building that appears in the CFD for show the difference in pressure at the edges and in the center of building. This is different from the gusts of wind from west to east which have little effect on the width of the building. In the calculation of analysis using manual (SNI) and MIDAS Gen or using the loading regulation only takes into account for the height of the building, but this is quite conservative because this regulation always applies to the overall design of buildings or other infrastructure buildings.

Windflow Trase Profile

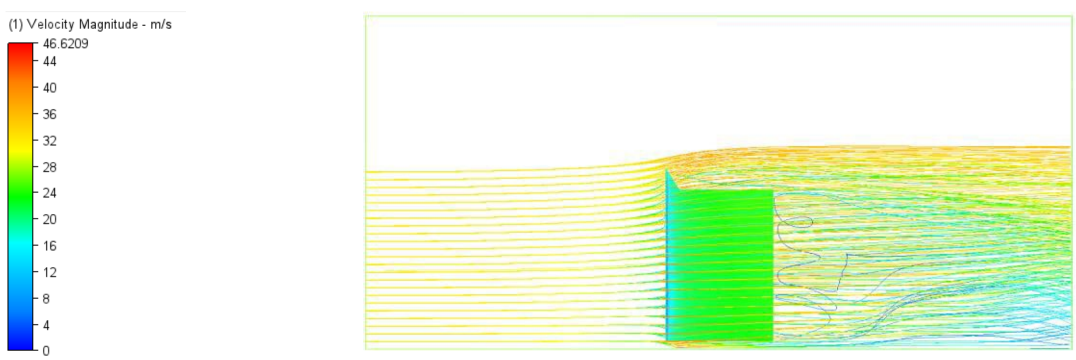

FIGURE 10 Wind Flow Trace Profile from West to East Passing through the UNTAR Main Building From The Side View

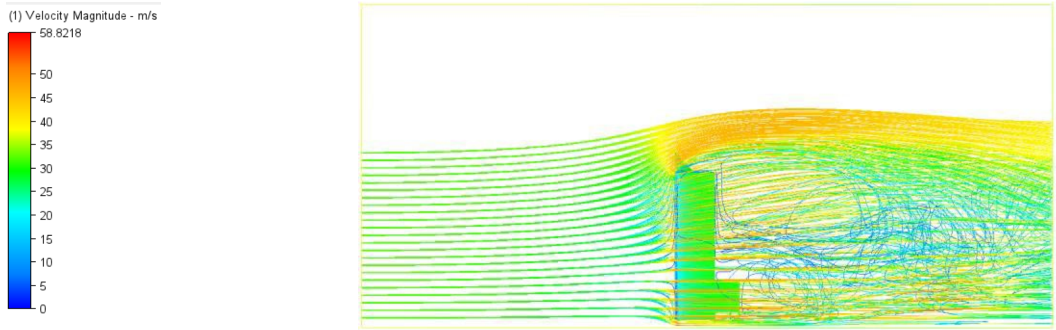

FIGURE 11 Wind Flow Trace Profile from South to North Passing through the UNTAR Main Building From The Side View

From trace profile of the wind flow, it can be seen when wind flow enters, the flow is laminar or parallel. This is because wind has not been exposed to test object. After being hit by building, the wind moved around the building by moving left, right and up. The building here is an obstacle to wind, so after the wind passes through building, the wind flow will be separated and not aligned anymore. There was an irregular wind flow behind Building that swirled around like turbulence, but the turbulence produced a small wind speed so that it did not have much effect on the humans behind the Building.

The amount of wind speed that occurs can be seen in contours of color that occurs. The greatest wind speed can be seen at time when the wind flow on the edge and above building and the wind speed is the smallest when the wind flow hits the middle side of the building and behind the building. From figure 9 and figure 10 of the trace profile of the wind flow, it can be proved that the wind speed that hits the Building as an obstacle is not always constant. This increase in wind speed has many causes and of course according to numerical and fluid analysis and the law of fluid contuinity. When the wind passes through the side edges of the building and the top of the building there is a narrowing of the surface area where the wind moves so that the wind speed at which the wind moves will be large than before. This phenomenon actually often occurs in wind. The wind that blows through a barrier will increase in speed. So that if there is a building behind it, the impact of the wind flow caused becomes greater.

CONCLUSION

Based on the results of analysis, the conclusions obtained for this study are as follows

1. From analysis using CFD for main building of UNTAR, it is obtained that the pressure that is on edge surface is smaller than the pressure that is on middle surface.
2. The amount of wind pressure that is in the center of surface can be 4 to 5 times greater than wind pressure that is at the edge of surface.
3. The peak of wind flow that occurs when the wind from south to north is higher than the wind direction towards west to east.
4. The maximum increase in wind speed that occurs when wind moves from south to north is 83.81% of the speed of wind coming and when wind moves from west to east by 45.6875% of the speed of wind coming.

ACKNOWLEDGEMENT

Thank you to Tarumanagara University Research Institutions and Community Service because funding this study. Therefore, the authors acknowledge the assistance and cooperation of the Tarumanagara University Research Institutions and Community Service.

REFERENCES

[1] Wendt, J. F. (Ed.). (2008). Computational fluid dynamics: an introduction. Springer Science & Business Media.

[2] Ghosh, S. K., Fanella, D. A., & Liang, X. (2003). Seismic and wind design of concrete buildings. International Code Council, Inc., Country Club Hills, IL.

[3] Holmes, J. D. (2010). Rational wind-load design and wind-load factors for locations affected by tropical cyclones, hurricanes, and typhoons. In Structures Congress 2010 (pp. 2151-2163).

[4] Iqbal, A., Rehman, H. S. U., Munir, M., Ashiq, M., Omar, A., Haider, Z., ... & Jahanzaib, M. (2019). Time and cost overrun in construction projects of Pakistan. Pakistan Journal of Engineering and Technology, 2(2), 22-29.

[5] Waty, M., & Sulistio, H. (2022). CAUSES FOR THE CHANGE ORDERS IN ROAD CONSTRUCTION: REVIEWED FROM OWNER. Komunikácie, 24(2).

[6] Simiu, E., & Yeo, D. (2019). Wind effects on structures: Modern structural design for wind. John Wiley & Sons.