COMPUTATIONAL FLUID DYNAMICS MODELING FOR WIND RESOURCE ASSESSMENT IN URBAN AREAS USING REMOTE SENSING

David Jess Ecal[1], Jara Kaye Villanueva[1], Ma. Rosario Concepcion O. Ang[1], Loureal Camille V. Inocencio[1], Ma. Victoria D. Rejuso[1] and Jerome T. Tolentino[1]

[1]Training Center for Applied Geodesy and Photogrammetry, University of the Philippines, Diliman, Quezon City, Metro Manila, Philippines

Email: ecaldavid@gmail.com, jarakayevillanueva@gmail.com, moang@up.edu.ph, lcvinocencio@gmail.com, victoria.rejuso@gmail.com, tolentinojeromet@gmail.com

Keywords: CFD, LiDAR, Wind flow, Wind Power Density

ABSTRACT: In the advent of growing interest in renewable energy, particularly wind energy, there has been an increase in research regarding wind flow and behavior across desired areas. Due to the intricacies of wind flow and the complexity of urban structures, Computational Fluid Dynamics (CFD) is used to simulate the wind flow. CFD simulation can be a valuable tool in assessing the wind conditions on candidate sites as it accounts for the physical properties of the fluid and the domain being simulated.

This study aims to simulate a detailed model for an urban setting, in particular, the Academic Oval of the University of the Philippines Diliman. This will take into account the velocity streamline model and wind power density across the area using ANSYS Fluent. With the availability of Light Detection and Ranging (LiDAR) derived products such as Digital Elevation Models (DEM) and Digital Terrain Models (DTM) is used for the urban wind simulation.

The entire schema consists of two integral parts: pre-processing using Geographic Information System (GIS) tools on remotely-sensed data and simulation using CFD on the LiDAR extracted building geometries. Using a high-resolution DSM and DTM data, building features are delineated by optimizing GIS tools. The three-dimensional model generated from this is essential to the CFD wind modelling. For the wind simulation, the k–ώ Shear Stress Transport (SST) is used as a turbulence model as it gives a relatively more accurate result. Steady State Condition is assumed. Results are verified and wind flow is simulated using a post processor. Results are visualized as velocity streamlines and its animation, velocity and pressure contour map, and wind power density map. Based on the results, places of high elevation have a higher wind power density. These places have low presence of turbulence as it is distant from obstructions. This simulation provides a basic visualization on wind behavior around an urban area and may be used as an assessment tool for wind resource evaluation.

For this study, the use of high-resolution data images provides a more accurate identification of urban geometries due to high-detailed image information. Based from the simulation results, incorporating it with a numerical model for wind simulation and analysis proves to be a promising tool for an accurate wind flow modelling.

1. INTRODUCTION

With the increasing knowledge in Computational Fluid Dynamics (CFD) across the globe, more engineers and designers are incorporating CFD simulations in their designs. The same can be said on the study of wind by researchers who focus on wind behavior and its potential to generate power. With the use of Light Detection and Ranging (LiDAR) technology, an accurate representation of the geometries and terrain on a specific area of interest. The data collected from LiDAR specifically the digital elevation model (DEM) and the digital terrain model (DTM) are processed and used to generate and extract the urban features of the area.

The selected domain of the building extraction is in UP Diliman, Quezon City, Philippines. This study used Geographic Information System (GIS) tools to extract and generate 3D geometry which can then be used in the CFD modelling.

2. OBJECTIVE OF THE STUDY

This study aims to simulate urban wind flow using the extracted geometry of the University of the Philippines Diliman from LiDAR and export it on a CFD modeling software. The extraction will use GIS tools and a Computer Aided Design

(CAD) program to generate the 3D geometry. The geometry is then processed and simulated using the kappa – omega Shear Stress Transport model (k-ω SST) as the turbulence model. A steady state condition is also assumed.

3. METHODOLOGY

The process is divided in to two major parts namely the GIS processing and the CFD modelling.

3.1 GIS Processing

For the extraction of the building geometries from LiDAR data, Geographic Information System is used. GIS is a set of tools which helps in the analysis and interpretation of geospatial information. The GIS processing is subdivided into three parts namely: pre-processing, delineation, and 3D building generation.

3.1.1 Pre-Processing

The dataset used is a LiDAR-derived product which includes Digital Surface Model (DSM) and Digital Terrain Model (DTM) with an image resolution of 1x1 m. Study area covers the academic oval of the University of the Philippines Diliman, Quezon City with an area of 0.78 square kilometers.

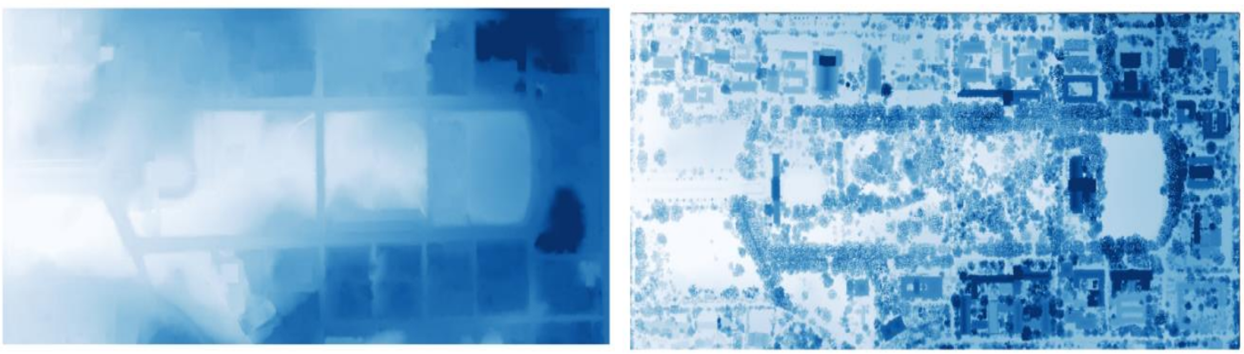

Figure 1. (L)Digital Terrain Model (DTM) and (R) Digital Surface Model (DSM) of UP Diliman Acad Oval

Steps in pre-processing consists of normalization, threshold setting, and feature extraction. Normalization is the subtraction of the digital surface model from the digital terrain model. Since the DSM contains elevation information of the surface features and the DTM contains the ground data or terrain elevation model, the difference between the two would be normalized digital surface model or nDSM. In this case, the surface features could be vegetation, roads, buildings or other structures.

Since this study will simulate wind flow using building geometries, threshold parameters are set to separate building geometries from other surface features. Parameters include height, slope, and area. It is important to note, however, that parameter values may vary depending on the quality of the LiDAR data used.

This study uses elevation information to separate high features from ground features. Through trial and error, the height value is determined and value used is 2.5 meters. In theory, slope is defined as the change in elevation which can be expressed either in percent or in grade. Since the buildings are assumed to be planar, they have generally lower slope than trees. Polygonal areas are just used to separate the building polygons from small ungrouped polygons. A comprehensive visual analysis is needed to determine the area value to be used.

3.1.2 Delineation

Building geometry delineation is the process of masking, aggregating, and smoothing the building polygons. Using tools in ArcGIS, the polygons that are considered to be within the threshold values are extracted. These polygons are classified as the building polygons. Since the edges from the extraction are still rough from the threshold setting results, a tool for smoothing and simplification of the edges is used.

This delineates the building boundaries and generates smoother edges which are deemed necessary for three-dimensional building generation and visualization.

3.1.3 3D Building Generation

The generation of the buildings into three-dimensional space uses the concept of Triangulated Irregular Networks (TIN). TIN represents a surface morphology and is considered to be in 2.5 D. This can be produced from raster data through triangulating a set of vertices. This makes use of the elevation information from the LiDAR data and is necessary for the extrusion of the polygons intro three-dimensional space.

Using DSM, DTM, and the building footprint delineated from the previous process, the three-dimensional building geometries are generated. This is then used for the wind simulation and computational fluid dynamics modelling.

3.2 CFD Modelling

The CFD process is done on ANSYS. It is subdivided in to four parts namely: Geometry Creation, Meshing, Solver, and Post-Processing. The post-processing is can also be done on a separate platform or program called TecPlot360. It is used to further obtain selections for visual outputs.

3.2.1 Geometry Creation: The 3D generated model serves as the subject to be analyzed and this will basically serve as the core of the whole study. After creating the 3D features in AutoCAD, it is exported as a Standard ACIS Text (SAT) file which the Design Modeler (DM) in ANSYS supports. SAT files store three-dimensional geometry information in a standard text file format.

Inside the DM, a computational domain is created which encloses the buildings and serves as the boundaries of the process. This is basically a box in which its dimensions rely on the tallest building in the geometry. The distance of the inlet, top and side boundaries are set to five times the tallest building height (h_b) and the outlet boundary is set to $10h_b$ (Mohamed 2015). The Carillon tower with a height of 40 m is used as the h_b. A Boolean subtract was performed to specify the building as a solid and that the boundary volume is liquid.

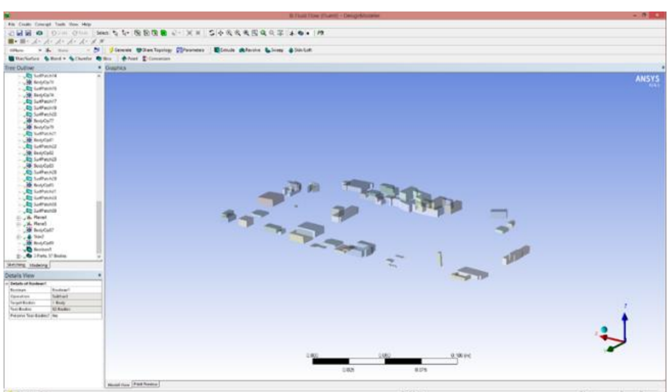

Figure2. UP Diliman Geometry

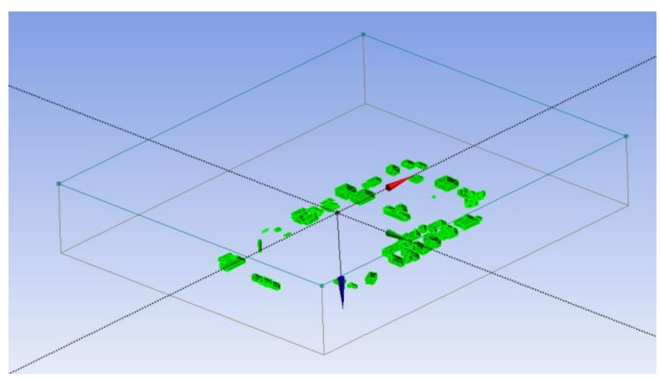

Figure 3. Domain Creation of Geometry

3.2.2 Meshing: The mesh divides the domain into cells where the governing equations are computed. The Grid or Mesh is the sub-division of the domain into smaller, non-overlapping sub-domains. The sub-domains are also known as cells which act as the control volumes.

Pre-processing also selects the physical and chemical phenomena of the subject being modeled. Versteeg and Malalasekera stated that fluid and material properties must be defined along with the boundary conditions at cells within the boundary. The solution to every flow problem is defined at the nodes inside each cell. The larger the number of cells, the more accurate the solution is. Optimal meshes are meshes that are non-uniform or that has varying cell shapes depending on the point inside the domain (Versteeg & Malalasekera, 1995). For example, inserting inflation to a target body like buildings generate hexahedral cells instead of tetrahedrons.

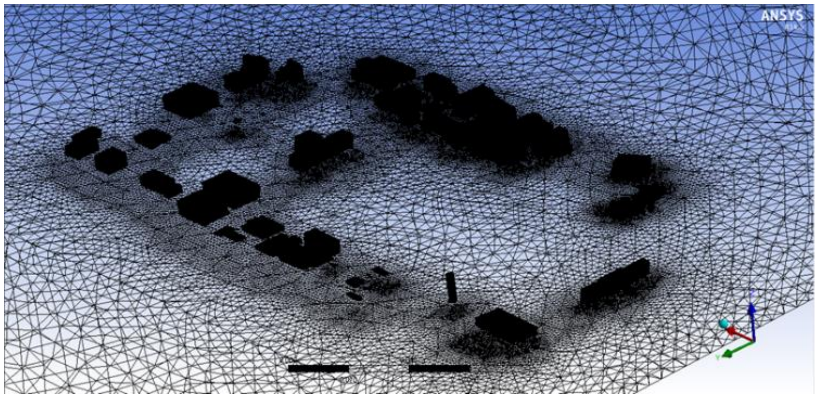

Figure 4. UP Diliman Generated Mesh

3.2.3 Solver: The solver handles all the algebraic and numerical solutions involving the flow. It performs three steps: the approximation of the unknown flow variables by using simple functions, the discretization by substitution of the approximations into the governing flow equations, and lastly the solution of algebraic expressions.

In the case of Fluent and other CFD codes, it uses a special finite difference formulation known as Finite Volume Method. It starts with the formal integration of the governing equations of fluid flow over all the control volumes of the solution domain. Next is the discretization, that involves the substitution of a variety of finite-difference-type approximations for the terms in the integrated equation representing flow processes. This converts the integrals into a system of algebraic equations. Finally the solution of the algebraic equations are solved by an iterative method.

This simulation used the Reynolds-Averaged Navier-Stokes Equations to serve as the governing equations and solve for the mean motion. Among the RANS model, SST k-omega *(k-* ω) is chosen because it has both the strong points of k-epsilon and k-omega. It is both dependent on wall distance and turbulence properties. It simulates the k-ε behavior in the free-stream and avoids the common problem of the k-ω model with the inlet free-stream turbulence properties. On the other hand, the strengths of the k- ω model is utilized on near wall points. (Tryggvason, 2011)

In this simulation, the input velocity of the wind to be analyzed is taken from actual results of monthly outputs of wind sensors installed inside the UP Diliman Campus. The sensor outputs include two measurements of wind speeds (in m/s) from the same elevation but different distance and wind direction in degrees. The wind speed used has a magnitude of 1.27 m/s and is blowing from the East. These are set in the boundary conditions for velocity inlet, pressure outlet and walls. The trees and other smaller geometries are taken into account by the surface roughness that can be set in the boundary conditions of the ground which is considered as wall. The surface roughness can be set to 0.75 which presents moderate roughness because according to A.M. Endalew et al. / (International Journal of Heat and Fluid Flow 30 (2009) 356–368), these tree canopies does not present a significant change in the airflow and may be considered as smooth surfaces or porous sub-domain.

4. RESULTS AND DISCUSSIONS

Results from the building extraction from LiDAR surveys using Geographic Information Systems (GIS) and wind simulation using Computational Fluid Dynamics (CFD) are discussed below.

4.1 GIS Processing Results

As can be seen in the image below, the result of pre-processing still shows small, ungrouped polygons which are considered to be features other than building geometries.

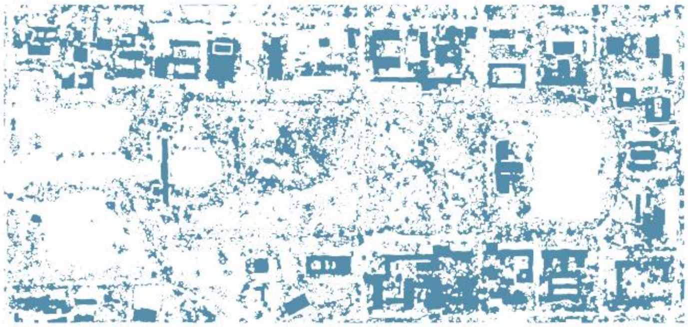

Figure 5. Initial results from pre-processed building polygons

Some polygon edges are still jagged and building outlines are not that smooth. Hence, there is a need for the aggregation and smoothing of polygons. This completely excludes all the unnecessary polygons and cleans the edges of the building footprint.

Figure 6. Building footprint with smoothed edges

The generated three-dimensional building polygons are shown below.

Figure 7. Extruded building polygons in three-dimensional space

4.2 CFD Post-processing

In the post-processing tool of ANSYS, output types like Velocity Streamlines and Pressure Contours are readily available. Wind Power Density can also be manually set as a variable and it will generate a contour plot across a plane of reference. Three planes were selected with varying 'z' values.

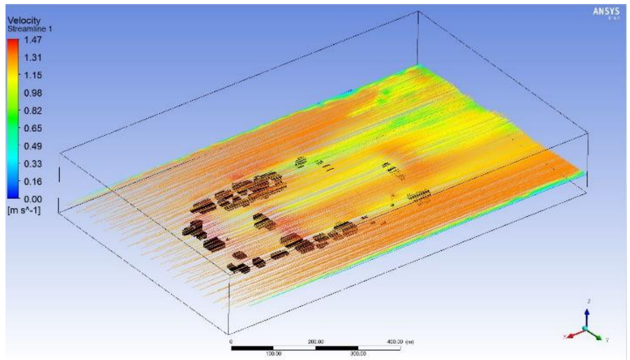

Figure 4. Velocity Steamlines at 20 m elevation

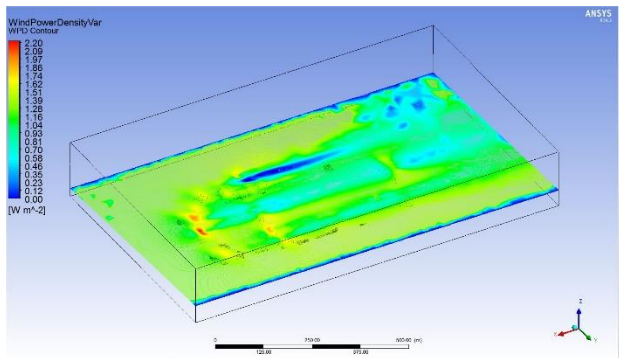

Figure 5. Wind Power Density Contour at 20 m elevation

5. CONCLUSIONS AND RECOMMENDATIONS

This study aims to simulate a detailed wind model for an urban setting, in particular, the Academic Oval of the University of the Philippines Diliman. Particularly, this study implements Geographic Information System tools for the building extraction from LiDAR data products and Computational Fluid Dynamics in the wind simulation and computation.

For the GIS processing, the values for the threshold parameter is based on the researchers' visual judgment and interpretation. Numerical values were generated from trial and error. Hence, these values are not constant. Every area is considered to have a different threshold value for the parameters used.

From the results of the simulation, we can confirm that the buildings and structures have a significant effect on Wind Power Density as the reference plane or 'z' increases. Based on the WP D contour plots, certain locations on the domain have a significantly higher WPD value. These values prominently occur at areas dense with buildings and structures.

Approximately a difference of 1.1 W/m^2 of WPD is present between the highest and the average value. These values can be explained by the "Channeling Effect" (Blocken & Stathopoulos, 2007). This phenomena occurs when wind passes through narrow channels. The channeling effect revolves around Bernoulli's Theorem where it states that higher velocities occur at low pressure areas.

In conclusion, the use of high-resolution data for the building extraction provides a more detailed identification of building polygons. For considerably large areas, the method presented can be a useful tool for a more time efficient processing. As evident in the simulation results, computational wind simulation using the output from the GIS processed building polygons can be a promising tool for an accurate microscale urban wind flow modelling

6. REFERENCES

References from Journals:

Jiang, N., & Lin, X. G. (n.d.). Object-Oriented Buinding Extraction By Dsm and Very High-Resolution Orthoimages. *Archives*.

Varshney, A., Rajesh, E., Sarabandi, P., & Solutions, R. M. (1962). Advanced region – based pattern recognition algorithms for building footprint extraction- an automatic approach to generate building inventory.

Shen, W. (2008) Building Boundary Extraction Based on LiDAR Point Clouds Data. The /International Archives of the Photogrammetry, Remote Sensing and Spatial Information Sciences. Vol. XXXVII. Part B3b. Beijing

Demir, N., and Baltsavias E., (2012) Automated Modeling of 3D Building Roofs Using Image and LiDAR Data. ISPRS Annals of Photogrammetry, Remote Sensing and Spatial Information Sciences.

Shiravi S., Zhong M., Beykaei S., (2012) Accuracy Assessment of Building Extraction using LiDAR Data for Urban Planning/Transportartion Applications. 2012 Conference of the Transportation Association of Canada Fredericton, New Brunswick.

Blocken, B., Carmeliet, J., & Stathopoulos, T. (2007). CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow. Elsevier, pp. 958-960.

Mohamed, M. A. (2015). Computational Modelling of the Wind Flow over the University of Calgary Campus. University of Calgary, pp. 36.

A.M. Endalew et al. (2009). International Journal of Heat and Fluid Flow 30, Elsevier, pp. 356–368

References from Books:

Versteeg, H. K., & Malalasekera, W. (1995). An Introduction to Computational Fluid Dynamics. Essex, England: Longman Scientific and Technical, pp. 100-110.

Vosselman G., and Dijkman, S., (2001) 3D Building Model Reconstruction from Point Clouds and Ground Plans. IAPRS, Vol. XXXIV, Part 3/W4, pp.37-43

References from Other Literature:

Wei, W. W. W., Chen, X. C. X., & Ma, A. M. A. (2005). Object-oriented information extraction and application in high resolution remote sensing image. *Proceedings. 2005 IEEE International Geoscience and Remote Sensing Symposium, 2005. IGARSS '05.*, *6*(C), 5–8. http://doi.org/10.1109/IGARSS.2005.1525737

Dy, E. (2013) Automatic Generation of 3D Buildings and Trees from DSM and DEM Data using GIS. Print.

Teo, T., Rau, J., Chen, L., Liu, J., & Hsu, W. (n.d.). Reconstruction of Complex Buildings using LIDAR and 2D Maps. *Lecture Notes in Geoinformation and Cartography Innovations in 3D Geo Information Systems*, 345-354.

Tryggvason, G. (2011). Classical Turbulence Modeling. University of Notre Dame, slides 26-30.

Ming, D. M. D., Luo, J. L. J., Shen, Z. S. Z., & Li, J. L. J. (2005). Features based parcel unit extraction from high resolution image. *Proceedings. 2005 IEEE International Geoscience and Remote Sensing Symposium, 2005. IGARSS '05.*, *3*(section 3), 5–8. http://doi.org/10.1109/IGARSS.2005.1526374

Shridhar J., Panditrao S., and Luis A., "Airborne LiDAR and High Resolution Satellite Data for Rapid 3D Feature Extraction." National Remote Sensing Centre Indian Space Research Organisation. Web. 4 May 2015.

7. ACKNOWLEDGEMENTS

This research is funded by the Department of Science and Technology (DOST) of the Republic of the Philippines under the Phil-LiDAR 2 Program: Nationwide Detailed Resource Assessment using LiDAR.