VALIDATION OF ATMOSPHERIC BOUNDARY LAYER CFD SIMULATION OF A GENERIC ISOLATED CUBE: BASIC SETTINGS FOR URBAN FLOWS.

Rawand Khasraw Bani
University of Sulaimani, Faculty of Engineering, Department of Architectural Engineering
Sulaimaniayh, Iraq

ABSTRACT

Computational Fluid Dynamics (CFD) is a very useful and inexpensive method to study wind flow in and around buildings, but its accuracy is a subject of concern. Many commercial codes are available each with different levels of complexity and accuracy. Autodesk Simulation CFD (ASCFD) is a product that directly integrates complex Autodesk Revit modes. It is a very appealing option for architectural and urban studies. This research is a validation process to detect the best settings for ASCFD; it will be a base for further complicated urban studies in lack of access of a wind tunnel. The research conducted a systematic sensitivity analysis to examine the impact of many variables that affect Atmospheric Boundary Layer simulations. Generally, the code produced good results compared to a wind tunnel test in the stream wise velocity field, in a combined case the averaged deviation reached 7.5% and R^2=0.95. Nevertheless, a notable deviation persisted in the wake region.

INTRODUCTION

Wind flow studies traditionally depended on physical tools to study buildings i.e., wind tunnels and water tanks. Wind tunnels are very useful and precise in studying wind flow, so they are very common in research and practice (Wu & Stathopoulos 1993, Moon et al., 2007 & Blocken et al., 2011). Recently the numerical tool Computational Fluid Dynamics (CFD) has gained traction. It has been widely used in wind flow studies because it is cheaper compared to wind tunnel experiments. In addition, it gives complete control over each point in the entire study field, a feature that omits from wind tunnels. Nevertheless, these benefits come with a price, which is accuracy (Stathopoulos 1997, Blocken 2014). Autodesk Simulation CFD (ASCFD) is one of the large tech company's recent releases among its simulation packages. It directly integrates Autodesk Revit models, which is a very handy feature as architectural and urban studies often times require complex models that are not logical to rebuild in other software for further analysis. Furthermore, the growing field of sustainable design now requires architects to test their preliminary designs with CFD tools. The ASCFD is an appealing option for architectural applications because of its direct integration of Autodesk Revit models, but to the author's knowledge, there has not been any reports on its accuracy in Atmospheric Boundary Layer (ABL) applications. Validation of results is very crucial for reliability (Stathopoulos 1997, Casey & Wintergerste 2000), but wind tunnels are not accessible every time. Therefore, (Blocken et al., 2012) proposed that for urban studies the model can be broken into basic parts and be compared to available wind tunnel tests, then the study can adopt the settings obtained from the most accurate simulation. This research is a validation process, it investigates the accuracy of the ASCFD code in ABL applications. The validation is based on comparing results from ASCFD to a wind tunnel experiments.

The following sections provide a literature review, outline of a wind tunnel experiment by CEDVAL, the baseline CFD simulation, sensitivity analysis, discussion and conclusion.

LITERATURE REVIEW

A wide range of variables affects the accuracy of a CFD simulation depending on the type of the flow such as steady-unsteady and compressible-incompressible, and/or within the same type of flow as well. ABL simulations are generally high Reynolds number and incompressible flows. ABL simulations are unsteady in nature, but they can be simulated as steady with a slight compromise of accuracy (Franke 2007). The inlet boundary condition parameters such as velocity profile, turbulent kinetic energy and turbulence dissipation rate, are very important to report as they impact the development of horizontal inhomogeneity (stream wise gradient) in the computational domain (Blocken et al., 2007, Yang et al., 2008, Yang & Zhang 2009). Another parameter is the roughness height, which depends on the aerodynamic roughness length (Z_0) of the ground plane (Laporte 2010, Blocken et al., 2007). The roughness height is the limit of the laminar sublayer up to which the wall function replaces the turbulence model in commercial CFD codes to reduce computational cost (See Figure 1). It also contributes to the development of horizontal inhomogeneity in the vertical mean wind speed profile in the computational domain. It is recommended to run a simulation in an empty domain to compare the consistency of the inlet and incident velocity profiles (Blocken et al., 2007). Previous literature shows that ABL simulations are very sensitive to the set of equations to solve i.e., Reynolds Averaged Naiver-Stock equations (RANS) or Large Eddy Simulations (LES), type and resolution of grid, computational domain and turbulence models (Franke et al., 2004, 2007). The steady RANS model is less accurate than LES but it requires much less computational time, therefore it is a common choice for ABL simulations (Bitsuamlak & Dagnewa & Choudhury 2010, Defraeye et al., 2010, Blocken et al., 2011). The type of grid or the grid generation technique is proven to be an important factor (Van Hoof & Blocken 2010), this is also proven by this research. The grid resolution must be analysed in any case even if it is not a validation study. The resolution near the wall or the wall adjacent cell pertains to the separation of the laminar sublayer, wall roughness and the wall function. Grid sensitivity analysis by comparing results from different grids has been used, and reported to be satisfactory (Montazeri & Blocken 2012). Various turbulence models are available in each code such as Standard (k-ε) which is a general-purpose model and its variant Renormalized Group (RNG). Studies showed that RNG produced favourable results against other models in ABL flows (Franke et al., 2004, Bitsuamlak, Dagnewa & Choudhury 2010, Blocken et al., 2011b). It is more accurate than Standard k-ε but computationally more intensive. The (k-ω) models such as Shear Stress Transport (SST k-ω) does not use wall function but it requires finer grid close to the wall. There is a function for the SST k-ω in the flow separation region between the laminar sublayer and the fully turbulence region called Intelligent Wall Formulation (IWF), when IWF is on the SST model does not account for wall roughness. The SST model also offers hybrid models i.e., SST Detached Eddy Simulations (SST k-ω DES) for very high Reynolds number external flows and SST Scale Adaptive Simulations (SST k-ω SAS) purposed for transient flows. The Mixing Length (ML) and Eddy Viscosity (EV) models are both less intensive in computational time and work best with buoyancy-induced flows.

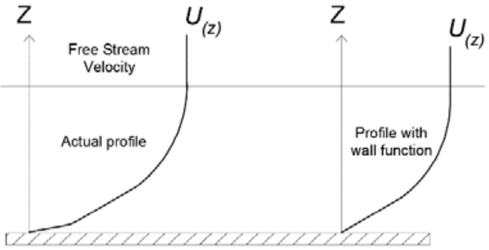

Figure 1 Actual profile versus wall function

DESCRIPTION OF THE EXPERIMENT

The wind tunnel test was extracted from the CEDVAL database of university of Hamburg. Many tests are available, for the purpose of this research a generic test of flow around an isolated cube was selected (category A). The test was conducted on a reduce scale model (1:200). The test area dimensions are (600mm x 1100mm x 500mm) which amounts to (120m x 220m x 100m) actual dimensions. The cube dimension is (125^3) mm placed at 287.5mm from the inlet boundary of the test area. The stratification is neutral, Reynolds number of the test is 37250, friction (shear stress) velocity (U_{ABL}) of the model boundary layer is 0.35 and the aerodynamic roughness length is (Z_0) 0.6mm.

CFD SIMULATION BASELINE

Computational Domain and Grid

The AAM generates fine prismatic tetrahedral cells near corners and solid surfaces, it also automatically applies enhancement layers of hexahedral cells in the same region which satisfies recommendations from (Tominga et al., 2008). The initial grid was composed of 483,830 compound tetrahedral and hexahedral cells.

Boundary Conditions

The inlet vertical velocity boundary condition was defined according to the logarithmic law (1). The inlet turbulent kinetic energy is defined based on the vertical velocity profile and the stream wise turbulent intensity according to (2). In (2) a is taken as 0.5 by assuming that the square of the standard deviation of turbulent fluctuation in the longitudinal direction ($\sigma^2 u$) to be much greater than the value for the other two directions ($\sigma^2 v$ and $\sigma^2 w$) which are assumed to be approximately equal ($\sigma^2 u >> \sigma^2 v \approx \sigma^2 w$) (Ramponi & Blocken 2012). The turbulence energy dissipation is defined based on the turbulent kinetic energy and the length scale of the model in (3).

$$Uz = \frac{U_{ABL}}{\kappa} \,.\, Ln(\frac{Z}{Z_0}) \qquad (1)$$

$$K_z = a((I_{U.Z})\,(U_z))^2 \qquad (2)$$

$$\varepsilon_z = C_\mu \,.\, \frac{{K_z}^{1.5}}{\delta_s} \qquad (3)$$

Where Z is the height coordinate, U_{ABL} is friction velocity, κ is Von Karman constant taken as 0.4, Z_0 is the aerodynamic roughness length equal to 0.6mm, I_u is the stream wise turbulence intensity, C_μ is an empirical constant taken as 0.09 and δs is the length scale of the model taken equal to the height of the cube (125mm).

For the baseline simulation the turbulence model (SST k-ω) was selected which does not use wall function but it accounts for wall roughness effects (Autodesk Inc. 2015). The ground plane is a rough surface of roughness (ks) defined by (4).

$$k_s = Z_0 \,.\, e^{kB} \qquad (4)$$

Where k_s is the roughness height, κ is Von Karman constant (0.4) and B is a constant taken equal to 8.5.

A simulation was performed in an empty computational domain to evaluate horizontal inhomogeneity in the stream wise extension of vertical velocity profile and turbulence terms. The comparison showed that the developed inhomogeneity is not substantial (See Figure 2). A velocity reference (U_{ref} =4.81m/s) was taken in the approaching flow (X_{ref}=218.75mm) at height (H_{ref} =150mm) in this simulation to formulate the velocity ratio (U/U_{ref} CFD) to compare normalized results with the empirical data from the wind tunnel test (U/U_{ref} Emp). At the outlet surface zero gauge static pressure was applied and zero normal velocity and gradient, was applied at the top and side surfaces.

At the outlet surface zero gauge static pressure was applied and no slip/symmetry condition, which means zero normal velocity and gradient, was applied at the top and side surfaces.

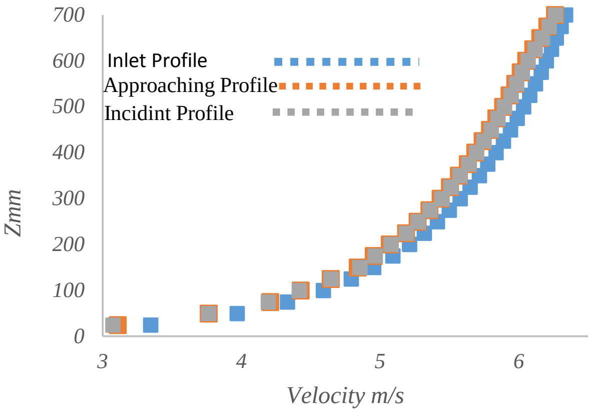

Figure 2 Velocity Profile Comparison in the empty domain

Solving the Model

The simulation was carried out using ASCFD 2015 with 3D steady RANS and the SST k-ω turbulence model. The turbulence model for the initial simulation was based on literature reports that it produced favourable results in comparison to other options (Ramponi & Blocken 2012). The advection scheme 5 (Modified Petrov-Galerkin) was selected, in which the discretization scheme is second order for all terms. ASCFD automatically assumes convergence when the residuals level off. After the examination of Averaged Residual Out (ARO) values it was found that the average error per node (a method recommended by Autodesk Inc.) is sufficiently small. The residual out values are the residual vectors over the whole field after the last iteration. The AROs are averaged by (5) per node. The AROs for u velocity field was (5×10^{-5}) m/s, for v velocity field was (5×10^{-6}) m/s and for pressure equaled (2×10^{-11}) Pa.

$$ARO = \frac{\sqrt{\sum_{i=1}^{N} (r_i)^2}}{N} \qquad (5)$$

Where r_i =residual out (residual vector), N= number of terms = number of nodes.

Results and Comparison

The numerical (CFD) results for the baseline case were compared to the empirical data through the velocity ratio U/U_{ref} in a longitudinal profiled that extends from the stream wise side to the wake region over the cube (See Figure 3 & 4). The velocity ratio showed less than satisfactory agreement in the stream wise field and on the top of the cube, 26% Averaged Absolute Deviation (AAD). The wake region demonstrates dissatisfactory deviation from the empirical results. An unrealistic vortex is created in the wake region that could be attributed to vast underestimation of turbulence intensity in that region (See Figure 5). In later sensitivity analysis the deviation in the stream wise side was reduced to a satisfactory agreement with the empirical data.

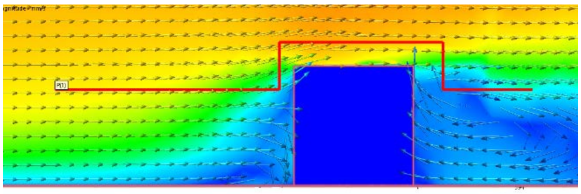

Figure 3 stream wise velocity measurement profile.

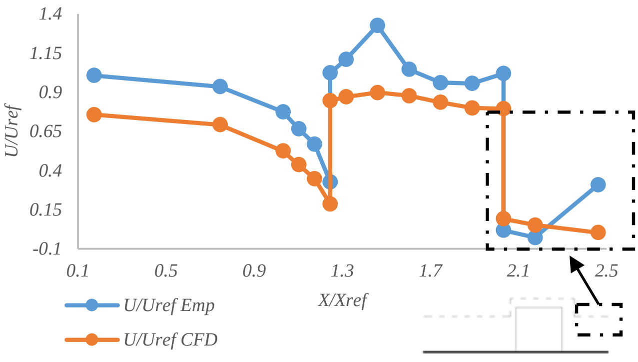

Figure 4 normalized stream wise velocity data comparison between empirical and numerical data.

SENSITIVITY ANALYSIS

Impact of Computational Domain

Three computational domains were modelled for analysis, exact replica of the Wind Tunnel Dimensions (WTD), Best Practice Guidelines (BPG) case and a BPG with a shorter frontal extension. The WTD was considered because in (Franke et al., 2007) it is also recommended to use the wind tunnel test area dimensions as an alternative. The shorter BPG case was based on the fact that stream wise gradient develops consistently throughout the extension. The BPG case originally was extended by only 3H in the stream wise side but for the BPG shorter the frontal extension was decreased to 2H. The analysis showed that the results are only slightly sensitive between the three cases. The WTD showed increased deviation in the stream wise extension from the BPG by 6.3% of

AAD and 1.7% from the empirical data. Meanwhile the BPG shorter case is slightly better than the WTD but overall the AAD increased by 5% from the BPG and 0.9% from empirical data.

Impact of IWF

The SST k-ω computes the dynamics down to the wall without using a wall function, and it employs IWF function for better performance adjacent to the wall.

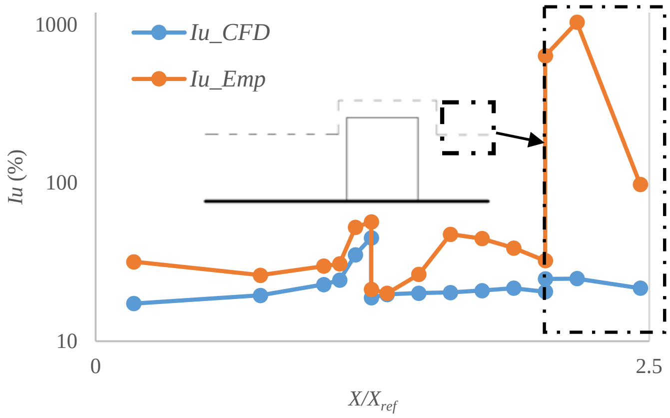

Figure 5 stream wise turbulence intensity (Iu) comparison on a logarithmic scale.

Also, if the IWF is on the model will not account for roughness effects. The baseline case was simulated with IWF-off, therefore an additional case was simulated with IWF-on for sensitivity. The comparison showed that the IWF-on reduced deviation in the stream wise velocity ratio, and the AAD was decreased to 17% compared to 26% with IWF-off. The IWF function is clearly advantageous for the SST k-ω without additional computational cost.

Impact of Grid Resolution

Two more grids were modelled in addition to the baseline case. Using a resolution factor of 1.5 a coarser grid with 336,705 cells and a finer grid with 747,384 cells were simulated. The results showed notable sensitivity to the grid resolution from coarse grid to the baseline grid by 3.8% (from 20.8% to 17%). Meanwhile the results did not show sensitivity from the baseline to the finer grid, the difference is less than 1% (17% to 16.7%). This shows that the initial grid resolution is in the acceptable range and hence was retained for further analysis.

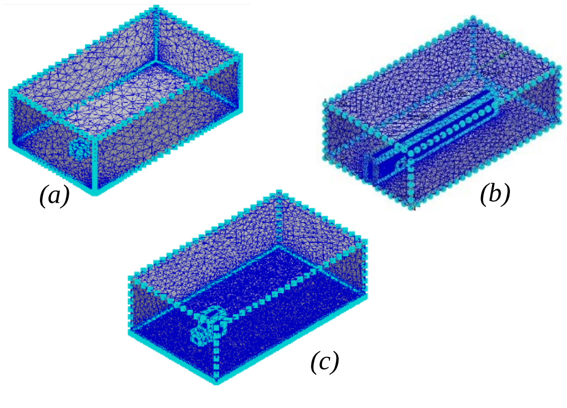

Figure 6 Grid types; (a) auto size, (b) manual (RR) and (c) manual (RR2).

Impact of Grid Generation Technique

For an easy control over the resolution of any region of interest in the grid its volume can be added in the BIM environment. The ASCFD directly recognizes the volumes and adds separate regions at the contact plane with the ground plane which is very handy to assign different resolutions to volumes and surfaces of contact. The auto size option assigns fine cells to corners neglecting the necessity of high resolution along the ground plane, at least in the aligned region with the obstacle. To address this issue two manual grids were created for analysis both with all volumes added in the BIM environment. The first grid Refinement Region (RR) has total of 463,964 cells, it is composed of three major volumes and two surfaces:

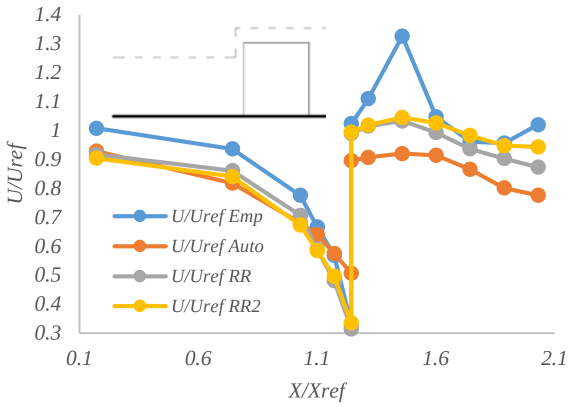

Figure 7 Grid type comparison, manual grid with fine ground plane is superior to the auto size option.

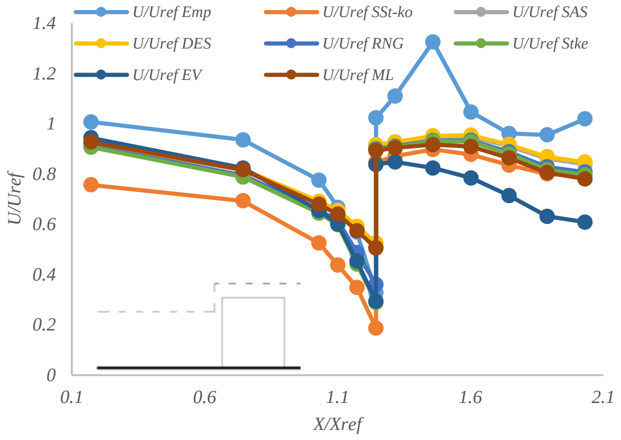

Figure 8 Turbulence model comparison.

ground plane 100mm cell height, strip aligned with the cube 5mm cell height (this insures 3 cells under k_s) the cube 50mm cell height, a refinement region across the domain aligned to the cube 30mm cell height, and the air volume of 100mm. The second grid Refinement Region (RR2) has total of 377,897 total cells is composed of three major volumes and a surface: ground plane 20mm cell height (this insures that the first cell is aligned to k_s), the cube 30mm cell height, a refinement volume right in the wake region (an additional reason for this is to address the drastic deviation in the in this region observed earlier) and the air volume 100mm cell height, with surface growth rate for both grids default at 1.2 (See Figure 6).

The analysis showed that the results are substantially sensitive to the type of grid. The stream wise velocity ratio demonstrated vast improvement compared to the baseline (auto size) (See Figure 7), the AAD is reduced by 8% for the RR grid and 9% for the RR2 grid. This proves that better accuracy is attainable with fewer cells if cells are properly distributed along the domain. Refining the corners, as in auto size, is very wasteful and unsuitable for ABL simulations. Although the RR2 grid had only 8% deviation in the stream wise velocity ratio but the improvement was not satisfactory in the wake region.

Impact of Turbulence Model

Six additional models were investigated in the analysis beside the initial SST k-ω. The choices were based on suitability for the ABL flows. The choices included the general purpose Standard k-ε, RNG k-ε, Mixing Length, Eddy Viscosity, the hybrid SST k-ω DES and SSTk-ω SAS. All models demonstrated improvements from the base line, the RNG showed the highest improvement with 14% AAD which is 3% lower than the baseline. The variants of SST k-ω DES and SAS both showed better agreement with the empirical results, 14.7% and 14.5% AAD which amounts to 2.3% and 2.5% improvement, respectively. The Eddy Viscosity also demonstrated similar improvement (14.5%), meanwhile the Mixing Length had similar results to the SST k-ω.

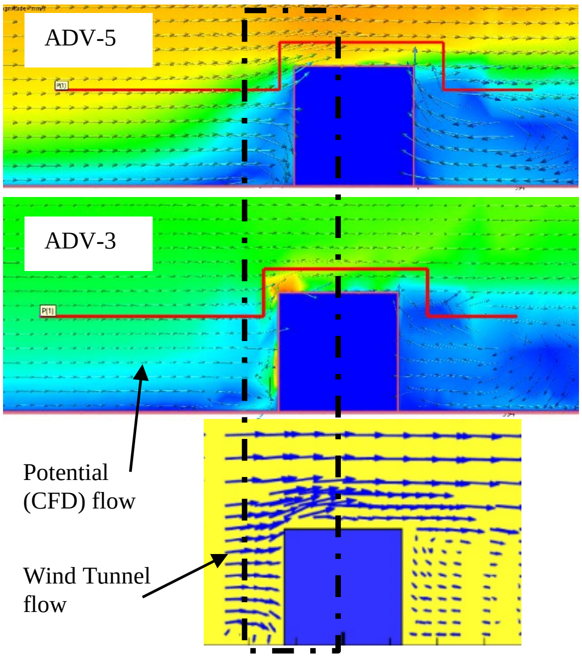

Figure 9 Impact of order of discretization on flow separation regions.

Impact of Order of Discretization

It is a known fact that the accuracy of discretization is affected by it is order (Sayma 2009). There are five discretization schemes available in ASCFD, the ADV-1 (Monotone streamline upwind) is first order which is not recommended (Franke et al., 2007) the ADV-2 is similar to ADV-5 which has been explored, only numerically less stable and ADV 4 is also a Petrov-Galerkin variant of second order. The Flux based scheme ADV-3 in the code provides fourth order discretization for all terms. A simulation was conducted with this scheme and an important improvement was observed. Naturally at the top of the cube there should be a notable acceleration developing, as noted in the wind tunnel. The only simulation that predicted this phenomenon accurately was the ADV-3 (See Figure 9 & 10). Also better results were produced in the flow separation regions. Unfortunately the scheme is numerically unstable and two out of three attempts failed into divergence even with fine grids.

DISCUSSION

The sensitivity analysis showed that the CFD results from ASCFD are most sensitive to type of grid, turbulence model and order of discretization, provided that the domain adheres to the BPGs. The grid for ABL simulation should always be modelled manually for proper distribution of fine cells. The ASCFD code

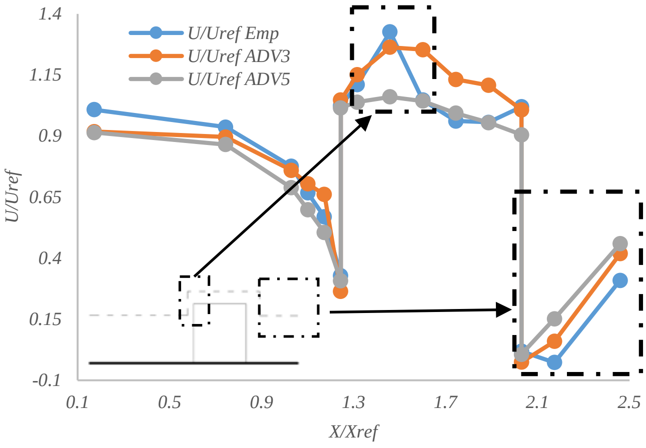

Figure 10 Impact of order of discretization, note that ADV-3 recorded better results both at incident profile and wake region.

gives best results with the turbulence models that use wall functions and the RNG model is favourable for ABL simulations. Also within the SST k-ω model both variants, DES and SAS, gave better results. In previous literature the SST k-ω showed favorability with different codes, this discrepancy can be attributed to the different programing of these codes. The ADV-3 accurately predicts flow separation regions. To achieve the best results a simulation was conducted using the best outcomes from the sensitivity analysis; BPG dimensions, manual grid (RR2) and RNG turbulence model. The results showed good agreement with the empirical data, an AAD of 7.5% in the stream wise velocity ratio which is considered a substantial improvement from the baseline case. Also a linear regression analysis (See Figure 11) showed a least square error (R^2) value of 0.95 for the entire range of points, (6) shows the calibration relation from the regression analysis. These are preferable settings that can be utilized by ASCFD users for ABL simulations. An extended vortex in the wake region persisted throughout the various simulations, which implies that velocity data taken in the immediate wake region are less than reliable.

$$U_{Emp} = 1.1457U_{CFD} - 0.0722 \qquad (6)$$

Where U_{Emp} is the *u* velocity field of the wind tunnel and U_{CFD} is the *u* velocity field for the CFD simulation. The equation applies to the entire point range.

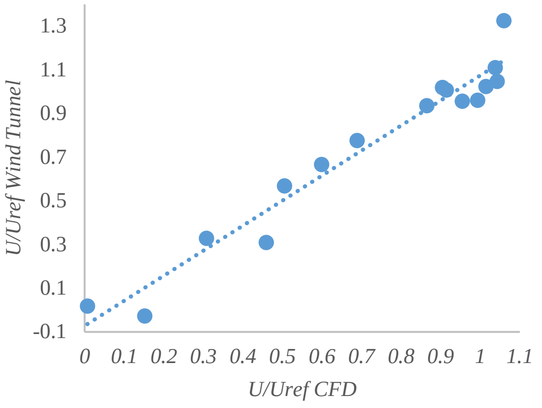

Figure 11 Relationship between the best attained results from the CFD simulation (combined case) and the wind tunnel data.

CONCLUSIONS

A systematic sensitivity analysis was performed to detect the level of accuracy and reliability of Autodesk Simulation CFD (ASCFD) code in Atmospheric Boundary Layer (ABL) simulations. The analysis was conducted by comparing normalized velocity data to a wind tunnel test and calculating Averaged Absolute Deviation (AAD). Various variables were analyzed; computational domain, grid resolution/technique, turbulence models and order of discretization. The analysis showed that auto size mesh generation technique is wasteful and unsuitable for ABL simulation (17% AAD), rather better accuracy was attained with a smaller number of cells when they were manually and properly distributed in the domain (8% AAD). The domain had a small impact on the results, given that the BPG dimensions is the best option (variations in ADD of 1%). The turbulence model SST k-ω (17% AAD) did not work as well as the RNG k-ε with the code, the latter showed the best results among all the available options (14% AAD). The advection scheme also had substantial impact on the results. The flux based scheme with fourth order discretization for all terms showed very good abilities to calculate the dynamics at the flow separation regions. Unfortunately it is numerically unstable. A final combined case with all best options was simulated and the ADD in the stream wise gradient was reduced to 7.5%, also an R^2 value of 0.95 was achieved for the entire point range. Nevertheless the wake region demonstrated high disagreement with experimental results throughout the sensitivity analysis in the ADDs. Based on the ADDs, the research recommends avoiding velocity field analysis in wake regions, rather studies should rely on pressure coefficient analysis in the immediate vicinity of such regions when using ASCFD for ABL simulations. The wind tunnel test did not provide data on pressure terms to conduct comparative analysis.

NOMENCLATURE

Z: height coordinate

Z_0: aerodynamic roughness length

U_{ABL}: friction velocity

κ: Von Karman constant

I_u: stream wise turbulence intensity

C_μ: an empirical constant taken as 0.09

δs: length scale of the model

k_s: roughness height

U_{Emp}: stream wise (*u*) velocity field of the wind tunnel

U_{CFD}: stream wise (*u*) velocity field of the CFD model

U_{ref}: reference velocity (4.81 m/s) taken in the approaching flow X_{ref} at the H_{ref}

X_{ref}: approaching flow reference (218.75 mm)

H_{ref}: reference height (150 mm)

REFERENCES

Autodesk Inc. 2015. Autodesk Knowledge Network, AutodeskCFD,Turbulence.http://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/SimCFD-UsersGuide/files/GUID-E9E8ACA1-8D49-4A49-8A35-52DB1A2C3E5F-htm.html.

Bitsuamlak, G., Dagnewa, A. & Chowdhury, A. 2010. Computational assessment of blockage and wind simulator proximity effects for a new full-scale testing facility. Wind and Structures. Vol 13; 21-36. doi: 10.12989/was.2010.13.1.02.

Blocken, B. 2014. 50 years of Computational Wind Engineering: Past, present and future. Journal of Wind Engineering and Industrial Aerodynamics; Vol 129; 69-102. doi:10.1016/j.jweia.2014.03.008.

Blocken, B., Janssen, W. & Van Hoof, T. 2012. CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus. Vol 30; 15-34. doi:10.1016/j.envsoft.2011.11.009.

Blocken, B., Stathopoulos, T., Carmeliet, J. & Hensen, J. 2011. Application of CFD in building performance simulation for the outdoor environment: an overview. Journal of Building Performance Simulation. Vol 4; 157-184.

Blocken, B., Stathopoulos, T. & Carmeliet, J. 2007. CFD simulation of the atmospheric boundary layer:wall function problems. Atmospheric Environment Vol41;238-252. doi:10.1016/j.atmosenv.2006.08.019.

Blocken, B., Van Hooff, T., Aanen, L. & Bronsema, B. 2011b. Computational analysis of the performance of a venturi-shaped roof for natural ventilation: venturi-effect versus wind-blocking effect. Computers & Fluids. Vol 48; 202-213. doi:10.1016/j.compfluid.2011.04.012.

Casey, M. & Wintergerste, T. 2000. Best Practice Guidelines, ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD, ERCOFTAC, Brussels.

Defraeye, T., Blocken B., Koninckx E., Hespel, P. & Carmeliet, J. 2010. CFD analysis of cyclist aerodynamics: Performance of different turbulence modelling and boundary-layer modelling approaches. Journal of Biomechanics. Vol 43; 2281-2287. doi:10.1016/j.jbiomech.2010.04.038.

Frnake, J., Hellsten, A., Schlünzen, H. & Carissimo, B. 2007. Best practice guideline for the CFD simulation of flows in the urban environment. Cost Action 732, Quality assurance and improvement of microscale meteorological models. COST Office Brussels, ISBN: 3-00-018312-4.

Frnake, J., Hirsch, I., Jensen, A., Krus, H., Schatzman, M., Westbury, P., Miles, S., Wisse, J. & Wright, N. 2004. Recommendations on the use of CFD in wind engineering. Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics. COST Action C14, Impact of Wind and Storm on City Life Built Environment. Von Karman Institute, Sint-Genesius-Rode, Belgium, 5–7 May 2004.

Laporte, D. 2010. A surface roughness parameterization study near two proposed windfarm locations in Southern Ontario. Unpublished master's thesis. University of Victoria. British Columbia, Canada.

Montazeri, H. & Blocken, B. 2012. CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: validation and sensitivity analysis. Building and Environment. Vol 60; 137-149. doi:10.1016/j.buildenv.2012.11.012.

Moonen, P., Blocken, B. & Carmeliet, J. 2007. Indicators for the evaluation of wind tunnel testsection flow quality and application to a numericalclosed-circuit wind tunnel. Journal of Wind Engineering and Industrial Aerodynamics. Vol 95; 1289-1314. doi:10.1016/j.jweia.2007.02.027.

Ramponi, R. & Blocken, B. 2012. CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters. Building and Environment. Vol 53;34-48. doi:10.1016/j.buildenv.2012.01.004.

Sayma, A. 2009. Computational Fluid Dynamics. Abdulnaser Sayma & Ventus Publishing ApS. ISBN: 978-87-7681-430-4. https://kosalmath.files.wordpress.com/2010/08/computational-fluid-dynamics.pdf

Stathopoulos, T. 1997. Computational wind engineering: Past achievements and future challenges. Journal of Wind Engineering and Industrial Aerodynamics. Vol 67-68; 509-532. doi:10.1016/S0167-6105(97)00097-4.

Tominaga, Y., Mochidab, A., Yoshiec, R., Kataokad, H., Nozue, T., Yoshikawaf, M. & Shirasawa, T. 2008. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. Journal of Wind Engineering and Industrial Aerodynamics. Vol 96; 1749-1761. doi:10.1016/j.jweia.2008.02.058.

Van Hooff, T., & Blocken, B. 2010. On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium. Computers & Fluids. Vol 39; 1146-1155. doi:10.1016/j.compfluid.2010.02.004.

Wu, H. & Stathopoulos, T. 1993. Wind Tunnel Techniques for Assessments of pedestrian Level Winds. Journal of Engineering Mechanics Vol 119; 1920-1936. doi: 10.1061/(ASCE) 0733-9399(1993)119:10(1920).

Yang, W., Quan, Y., Jin, X., Tamura, Y. & Gu, M. 2008. Influences of equilibrium atmosphere boundary layer and turbulence parameter on wind loads of low-rise buildings. Journal of Wind Engineering and Industrial Aerodynamics. Vol 96; 2080-2092. doi:10.1016/j.jweia.2008.02.014.

Yang, Q. and Zhang, J. 2009. Simulation of horizontally homogeneous atmosphere boundary layer based on k-e variant models combined with modified wall function. The proceedings of The Seventh Asia-Pacific Conference on Wind Engineering, November 8-12, 2009, Taipei, Taiwan.