CFD MODELING OF WIND-DRIVEN RAIN WETTING OF BUILDING FAÇADE

Ali Khalilzadeh1,*, Hua Ge2, Hoi Dick Ng1
1 Department of Mechanical and Industrial Engineering, Concordia University, Montreal, Québec, Canada
2 Department of Building, Civil, and Environmental Engineering, Concordia University, Montreal, Québec, Canada

ABSTRACT
To investigate the wind-driven rain wetting of façade, several Computational Fluid Dynamics (CFD) models using the traditional Lagrangian Particle Tracking method are developed and validated with wind tunnel measurements for wind flow around the building and field wind-driven rain measurements on a mid-rise six-story building. The commercial CFD package is used to solve the multiphase flow of wind and rain around the building. A MATLAB code is developed to post-process the CFD results and calculate the spatial distribution of Wind-Driven Rain (WDR) on façade. This paper investigates the solutions to different turbulence models, k-ε method with wall treatment proposed in the literature, and Standard k-ω, a better turbulent model in terms of near wall wind prediction, and k-ω SST. The effect of upstream mean horizontal velocity models, Log-Law and Power-Law, on the predicted data has also been studied. The comparison between the turbulence models, and inlet velocity profiles, shows a significant reduction in errors favored to Log-Law and Standard k-ω.

INTRODUCTION
WDR study is important in a variety of areas including earth sciences, meteorology and building science. It is the critical source of moisture deposition on buildings (Foroushani Mohaddes, et al., 2014). There are three methods typically used to quantify WDR, i.e. measurements, semi-empirical methods and numerical simulations based on CFD (Blocken, 2014). The importance of the latter is the convenience and its low cost as well as its capacity in investigating various geometries and details.

There are two approaches to simulate WDR using CFD, the traditional Lagrangian particle tracking method and Eulerian-Eulerian multi-phase flow method. In both Lagrangian and Eulerian approaches of WDR simulation using CFD, the accuracy in wind flow prediction around the building is important. Turbulence modeling scheme, wall boundary treatment, domain inlet and outlet profile functions are some of the few that are studied in this paper. Among the listed, wall boundary treatment plays a major role. The problem with the simulation of WDR is the incompatibility of atmospheric-boundary-layer (ABL) flow and the standard wall functions. It rises from the fact that for urban and suburban areas, the roughness height is greater than the $y_p$ value – the center point of the wall-adjacent cell, to achieve the desired $y^+$ for the validity of the wall treatment. $y_p$ has to be at least equal to the roughness height, $K_s$, for it to hold its physical meaning, and that obviously does not result in accurate solutions. Blocken (Blocken, 2004) proposed a couple of solutions to this classical problem of ABL simulation in commercial software packages. The effectiveness of such modifications has not been transparently studied. In this paper, we discuss how the value of $y_p$ makes a difference in the calculations and accuracy of CFD predicted data.

The pioneering work of Choi (1991) has been the starting point of the numerical research on WDR conducted in the past two decades. In Choi’s method (Choi, 1991, 1993), the rain drop trajectories are calculated based on a 3D steady-state RANS solution of the wind flow around the building using the standard k-ε turbulence model. The spatial distribution of WDR on buildings is then determined based on the computed raindrop trajectories. Recently, an averaging technique has been introduced and validated by Blocken and Carmeliet (2002) that incorporates the temporal component of WDR into the CFD-based models.

Blocken and Carmeliet (Blocken & Carmeliet, 2002) introduced an averaging technique that incorporated temporal components of WDR into the CFD-based models. Various studies such as (Blocken & Carmeliet, 2005; 2007), (Chang & Wu, 2003) and (Hangan, 1999), established new numerical techniques as a tool to validated numerical CFD simulations which are based on Reynolds Averaged Navier-Stokes (RANS) equations.

Blocken (Blocken, 2014) and Blocken and Carmeliet (Blocken & Carmeliet, 2010) provide an extensive overview of methods used to simulate WDR over a few decades. The effect of turbulence modeling on
WDR modeling has been investigated. In most of the cases above, Realizable k-ε with a Log law inlet velocity profile has been chosen as the turbulence modeling scheme. Foroushani Mohaddes et al. (2014) used Realizable k-ε with Power law inlet velocity profile and uses unmodified inlet turbulence kinetic energy and dissipation rate. Tominaga (Tominaga, 2015) studied the performance of the unsteady Reynolds-averaged Navier–Stokes (URANS) turbulence modeling of the flow field around a high-rise building and compared the use of k-ω shear stress transport (SST), Realizable k-ε, k-ε, and Standard k-ω models. They concluded that the RNG k-ε model with the modified ε-equation exhibited the best agreement with experimental results. Abadie and Mendes (Abadie & Mendes, 2008) studied turbulence modeling effects on WDR catch ratio on an isolated building façades. This is done for a single droplet diameter, which is called specific catch ratio for fixed wind speed and angle only. Abuku et al. (Abuku, et al., 2009) studied a low-rise rectangular building under various oblique wind conditions. Full-scale measurements were used for validation of their results.

The objective of this study is to find the best turbulence model fit to predict wind flow around a mid-rise building, as well as to predict the distribution of catch ratio on wind-ward building façade. For the terrain under study, different CFD models are set up and the solutions are compared and validated by wind tunnel measurements to assess the effectiveness of each modeling technique. Having established the right value of y_p, the comparison of Lagrangian WDR catch ratio results to the field measurements on a building in Vancouver, British Columbia named Cassier is presented. The comparison is made for four different schemes and the scheme that works the best is identified.

**METHODOLOGY**

This section presents the experimental setup including wind tunnel and field measurements. Turbulence models studied are briefly discussed followed by the details of numerical solution and modifications made to traditional boundary condition values to overcome the compatibility problem of ABL flow, and method of simulating the deposition of raindrops on the building. The section ends with listed boundary condition values.

**Wind Tunnel and Field Measurements**

The test building, known as the Cassier Building, is a six-story rectangular residential building with a flat roof located in Vancouver (near Burnaby) in British Columbia. The building is located within a suburban location. The test building is 39.2 m long, 15.2 m wide, and 19.8 m high. The prevailing wind direction during rain is predominately from the East (Chiu, 2016). A wind monitor measuring the wind speed and the wind direction is mounted on a tripod, which is 4.6 meters above the mechanical room located on the main roof of the building. The location of gauges mounted on the building is given in Figure 1.

The Atmospheric Boundary Layer (ABL) wind tunnel has a test section of 12.20 m long and 1.80 m wide, and has an adjustable suspended roof with a minimum and maximum height of 1.40 m and 1.80 m, respectively. A 1:400 scale model of the test building have been placed in the ABL wind tunnel. The 1:400 scale was selected based on the surroundings and successful simulations at this scale of the most important variables of the atmospheric boundary layer under strong wind conditions, carried out in this wind tunnel. The test building model is 98 mm long, 38 mm wide, and 50 mm high. There is a mechanical room located on the center of the roof measuring 15 mm long, 13 mm wide, and 6 mm high (Chiu, 2016).

The test building in the field is located within a suburban environment, therefore, a similar exposure is simulated in the wind tunnel. To obtain a suburban wind profile, a mixture of roughness elements have been placed along the length of the test section of the tunnel. The roof of the wind tunnel was adjusted along the length of the test section to satisfy the condition of zero longitudinal pressure gradient for a suburban exposure (Chiu, 2016).
A Series 100 Cobra Probe was used to measure the velocities. The Cobra Probe is a multi-hole pressure probe that provides dynamic, 3-component velocity and local static pressure measurements in real-time. The Probe is capable of a linear frequency response from 0 Hz to more than 2 kHz and is available in various ranges for use between 2 m/s and 100 m/s. Although the probe comes pre-configured, the accuracy was verified by comparing the mean values measured by the Cobra Probe with the measurements of a pitot static tube mounted at the same location. In addition, the measurements were checked for repeatability for the wind profile above the mechanical room roof and in front of the east facade. The average percent difference between the two tests were 1% and 6% for the wind profile above the roof and the East facade, respectively (Chiu, 2016).

Vertical measurement profiles used in this study are depicted in Figures 2 and 3.

![Figure 2 - Position of wind tunnel measurements upstream and around the building; values are in meters (Field Scale)](image)

![Figure 3 - Position of wind tunnel measurements close to the building; values are in meters (Field Scale)](image)

**Closure problem and RANS**

Two-equation turbulent modeling methods to closure problem of Reynolds Average Navier-Stokes equations (RANS) equations used in this paper are $k-\epsilon$, Realizable, Standard $k-\omega$ and $k-\omega$ SST. Standard Wall functions for Realizable $k-\epsilon$ turbulent model in a dimensionless form is $U^* = \frac{1}{k} \ln E y^*$.

**Matching the wall-function-modified-for-roughness and ABL flow**

The modifications proposed to boundary conditions based on 7-step procedure, are listed in Table 1. (Blocken, 2004)

For $k-\omega$ turbulence model, we suggest the use of $k-\epsilon$ model modification and finding $k-\omega$ model parameters based on them and using the following equation for $\omega$ calculation on the inlet boundary (Wilcox, 2006).

$$\omega = \frac{\epsilon}{C_\mu k}$$

(1)

**Implementation and Numerical Calculation Details**

The computational domain chosen is depicted in Figure 4. The building is located at the center of the domain on the bottom boundary, i.e. the ground. The information regarding the computational domain size and discretization is summarized in Table 2. A grid convergence study has been suggested in (Foroushani Mohaddes, et al., 2014), using the Richardson extrapolation-based scheme proposed by Roy (Roy, 2004). Wind flow grid convergence index is 2.5%. The area-averaged pressure coefficient of the building has been used as the main field variable for calculating this index.

![Figure 4 - Computational schematics with boundary conditions and domain size.](image)

The commercial CFD code ANSYS Fluent is used to find the wind solution around the building.

Pressure-based steady-state solver is chosen. As of the turbulence model, each scheme has its own viscous model. SIMPLE scheme is chosen for the pressure-velocity coupling. As for the Spatial Discretization, standard pressure, second order upwind for momentum and turbulent kinetic energy and first order upwind for turbulent dissipation rate is chosen. At convergence, residuals of continuity as the largest residuals of all, reach a limit of $10^{-7}$ for Realizable $k-$
ε and 10^{-11} for Standard k-ε and k-ω SST. Moreover, pressure coefficient of the building, which is the only wall boundary condition other than the ground, is monitored to perceive convergence.

<table>
<thead>
<tr>
<th>Log Law</th>
<th>Power Law</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_{ABL}(y) = U_{ref} \ln \left( \frac{y + y_0}{y_0} \right)$</td>
<td>$U_{ABL}(y) = U_{ref} \frac{y}{y_{ref}}$</td>
</tr>
<tr>
<td>$k = \left( \frac{u'<em>{ref}^2}{\sqrt{\frac{15}{2}} \frac{y}{y</em>{ref}}} \right)^2$</td>
<td>$k = \left( \frac{\mu u'<em>{ref} y}{y</em>{ref}} \right)^2$</td>
</tr>
<tr>
<td>$\epsilon(y) = \left( \frac{u'<em>{ref}^2 y}{y</em>{ref}^2} \right)^3$</td>
<td>$\epsilon(y) = \left( \frac{u'<em>{ref} y}{y</em>{ref}} \right)^3$</td>
</tr>
</tbody>
</table>

Table 2 - Computational domain size and discretizations.

<table>
<thead>
<tr>
<th>Type</th>
<th>Size</th>
<th>Number of CV's</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain sides</td>
<td>0.5 m - 3.99m</td>
<td>~1.56 x 10^9</td>
</tr>
<tr>
<td>Building</td>
<td>0.5 m</td>
<td></td>
</tr>
<tr>
<td>Surface mesh</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Volume mesh</td>
<td>Quadrangle</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Hexahedron</td>
<td></td>
</tr>
</tbody>
</table>

Discrete Phase Model (DPM) module of ANSYS Fluent is used to find the trajectory of particles injected in the domain. A fourth-order Runge-Kutta integration scheme has been used. A fourth-order curve is fit to the drag coefficients for falling raindrops measured (Gunn & Kinzer, 1949). The curve is as follows

$$\log(C_D) = 0.0358 z^4 - 0.225 z^3 + 0.5731 z^2 - 1.2466 z + 1.4633$$

With $z = \log(Re_d)$. A User-Defined Function is used to hook this to the solver.

A MATLAB code has been developed to calculate WDR catch ratio (Foroushani Mohaddes, et al., 2014). Catch ratio is the main parameter used to compare WDR simulation to field measurements. Specific catch ratio is defined by this

$$\eta_d = \frac{A_h(d)}{A_f(d)} \quad (3)$$

Where $A_h$ is the area of a reference horizontal surface and $A_f$ is the wetted area on the building façade (Blocken, 2004). Catch ratio is found by integration over the range of raindrop diameters. The raindrops size distribution is given by Best, (Best, 1950) as a weighting function.

$$\eta = \int \eta_d f(d) \, d\, d \quad (4)$$

The weighting function $f(d)$ is calculated based on the “flux-based” modification suggested by (Blocken, 2004).

Raindrops are injected into the domain in a specific area that is upstream of the building. The vertical velocity of injection is equal to the free falling terminal velocity (Gunn & Kinzer, 1949). The horizontal velocity of the rain drops are equal to that of the undisturbed wind at the same height above the ground as the injection plane. Trial and error settles the position of the injection planes. As long as the landing drops around the façade are conservatively low, and raindrops cover the façade entirely. No dissipation modeling is included in these simulations.

Table 3 - Summary of boundary conditions

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Inlet</th>
<th>Sides/Top</th>
<th>Bottom</th>
<th>Outlet</th>
<th>Building</th>
</tr>
</thead>
<tbody>
<tr>
<td>Condition</td>
<td>Momentum</td>
<td>Table 1, Direction: Normal to boundary, $U_{ref} = 2.30$ and 6.7 m/s, $\alpha = 0.22$</td>
<td>Symmetry</td>
<td>No slip, Backflow pressure ($\rho_0 = 0$, gauge)</td>
<td>No slip, impermeable</td>
</tr>
<tr>
<td>Turbulence</td>
<td>Table 1</td>
<td>Symmetry</td>
<td>Roughness height ($K_r = 0.152 m$)</td>
<td>$k_{backflow} = 1 m^2/s^2$, $\epsilon_{backflow} = 1 m^2/s^3$</td>
<td>Roughness height ($K_r = 0$), smooth wall</td>
</tr>
<tr>
<td>DPM</td>
<td>-</td>
<td>-</td>
<td>Trap</td>
<td>-</td>
<td>Trap</td>
</tr>
</tbody>
</table>

Boundary Conditions

Von Karman constant, $\kappa$ is fixed in the code to be 0.4187. $\mu = 0.09$ is a constant used in the modifications as well as a constant in Realizable k-ε (Wilcox, 2006). $y_p = 0.25$ is the distance of the center of the wall adjacent cell to the ground. The summary of boundary conditions are given in Table 3.
Near-wall-cell center-point distance to the wall
The modifications mentioned in Table 1 for inlet boundary parameters are proposed to tackle the inhomogeneity imposed to adaptation of ABL to wall-functions. To study the effectiveness, three empty 2D 2000 m × 500 m domains with \( y_p \) values of 0.0025, 0.025 and 0.25 m are solved with Realizable \( k-\varepsilon \) and standard wall-functions. Inlet and outlet horizontal velocity comparison is provided for each case, shown in Figure 5a, b and c. The residuals of continuity as the largest of all are let to a maximum of \( 10^{-12} \). This justified the choice on the value of \( y_p \). For that matter, \( y_p \) is chosen to be 0.25 m so that the homogeneity and the desired grid spacing on the building are met.

RESULTS AND DISCUSSIONS
The variations made to \( \varepsilon \) (turbulent dissipation rate) and \( k \) (turbulence kinetic energy) and \( \omega \) (specific dissipation rate) in for \( k-\varepsilon \) and \( k-\omega \) models, will cause in a change in these variables as the wind approaches the building and this results in a change in eddy prediction, separation and other turbulence related phenomena, not to mention velocity prediction on the roof and downstream of the building.

Comparison of CFD solutions to wind tunnel measurements
In this section, the comparison of different CFD modeling results to wind tunnel measurements is presented. All the CFD models feature treated boundary as discussed. Negative values of velocity correspond to back flow which is a result of turbulent eddies near the building.

Figure 6 shows the measurements and CFD predictions done above the Mechanical Room, located on top of the building. Quantitative error analysis shows that \( k-\omega \) SST has the lowest maximum error that happens at the lowest point of measurement, 4m above the roof. Moreover, \( k-\omega \) SST features the lowest mean value of errors and lowest absolute deviation from the mean.

Figure 7 to 9 show the comparison between wind tunnel measurements and CFD modeling results on a profile at 160 m, 80m and 40m upstream of the façade’s center point, respectively. Quantitative error analysis shows that Standard \( k-\omega \) has the lowest maximum error for points below 20 m, which is at the lowest point of measurement, 4m above the ground. Moreover, Standard \( k-\omega \) features the lowest mean value of errors. For the points above 20 m, Realizable \( k-\varepsilon \) with Power law inlet velocity profile features the lowest maximum error, which is at the lowest point above 20 m; as well as the lowest mean value of errors.
Figure 8 - Comparison between CFD modeling results and wind tunnel measurements: 80m Upstream of the East Facade

Figure 9 - Comparison between CFD modeling results and wind tunnel measurements: 40m Upstream of the East Facade

Figure 10 - Comparison between CFD modeling results and wind tunnel measurements: above rooftop, North-East Corner

Figure 11 - Comparison between CFD modeling results and wind tunnel measurements: above rooftop, Middle of North on the Edge

Figure 12 - Comparison between CFD modeling results and wind tunnel measurements: above rooftop, North-West Corner

Figure 10 to 12 show the comparison between wind tunnel measurements and the CFD modeling results at a projection point above the roof that is 0.6 m away from both North and East façade, 0.6m away from North placed in the middle between East and West façade, and 0.6m away from both North and West façade, respectively. Quantitative error analysis shows that for all three locations, the standard k-ω has the lowest maximum error, which is at the lowest point of measurement. For the location at the North east corner (Figure 10), the realizable k-ε with Power law inlet velocity profile features the lowest mean value of errors. Standard k-ω has the lowest absolute deviation from the mean. For the location at middle of the North (Figure 11), the standard k-ω features the lowest mean value of errors and lowest absolute deviation from the mean. For the location at the North West corner, Realizable k-ε with Power law inlet velocity profile features the lowest mean value of errors and lowest absolute deviation from the mean.
Figure 13 - Comparison between CFD modeling results and wind tunnel measurements: 3.6m away from the East Facade, ES1

Figure 13 to 15 show the comparison between wind tunnel measurements and the CFD modeling results at points 3.6 m away from the East façade upstream, at point ES1, EN1, and EC1 (shown in Figure 1), respectively. Quantitative error analysis shows that Standard k-ω has the lowest maximum error, which at the lowest point of measurement, for all three locations. For ES1, k-ω SST features the lowest mean value of errors and Standard k-ω has the lowest absolute deviation from the mean. For EN1, k-ω SST features the lowest mean value of errors and the lowest absolute deviation from the mean. For EC1, Standard k-ω features the lowest mean value of errors and k-ω SST has the lowest absolute deviation from the mean.

Figure 16 - Comparison between CFD modeling results and wind tunnel measurements: 3.6m away from the East Facade, ES2

Figures 6 to 18 show the comparison between wind tunnel measurements and the CFD modeling results at points 3.6 m away from the East façade upstream, at point ES2, EN2, and EN3 (shown in Figure 1), respectively. Quantitative error analysis shows that for ES2 and EN2, Standard k-ω has the lowest maximum error, which is at the lowest point of measurement, and Standard k-ω features the lowest mean value of errors and the lowest absolute deviation from the mean. For EN3, Realizable k-ε with Log law inlet velocity profile has the lowest maximum error, which is at the lowest point of measurement, and has the lowest mean value of errors and the lowest absolute deviation from the mean.
Catch Ratio Comparison

Catch ratio for four modeling schemes, Realizable k-ε with Log law, Realizable k-ε with Power law, k-ω SST with Log law, and Standard k-ω with Log law are compared to measurements and the results are presented in Figures 14 to 17. The cases modeled are as follows:

- $U_{10} = 2.42 \text{ m/s}$
- $R_h = 3.16 \text{ mm/hr}$
- Wind angle is approximately 90 degrees on the field
- Number of particles injected is 250×400
- Number of consecutive hours of measurement is 15 done in January 10, 2014

Using quantitative error analysis, the following observations are made for the catch ratio comparison.

The Standard k-ω model has the lowest maximum error and least mean value of errors for both top-edge gauge points and top two rows (Figure 1).

Among points ES1, ES2, EN1 and EN2 (Figure 1), For the top two points on either side edges, i.e. ES1, ES2 and EN1, EN2, the Standard k-ω has the lowest maximum error and Lowest absolute deviation from the mean. The k-ω SST has the least mean value of errors, although the difference between k-ω SST and Standard k-ω is negligible (less than 2%).

For points on the center column (Figure 1), k-ω SST has the lowest maximum error and the lowest absolute deviation from the mean. The Standard k-ω has the least mean value of errors.

For points on the edge of North corner, i.e. EN1, EN2, EN3 and EN4 (Figure 1), the Standard k-ω has the lowest maximum error and the least mean value of errors. The k-ω SST has the lowest absolute deviation from the mean although the difference between k-ω SST and Standard k-ω is negligible (less than 1%).

For the second column points on the North corner of the East façade, i.e. EN5, EN6 and EN7 (Figure 1), the Realizable k-ε with Log law inlet velocity profile has the lowest maximum error, although the difference between k-ω SST and Standard k-ω and Realizable k-ε with Log law inlet velocity profile is negligible (less than 3%). The Standard k-ω has the least mean value of errors belongs to.

Similar observations are made on the South side of the façade.
Figure 15 - Catch ratio comparison: $k$-$\epsilon$ Realizable, Log-Law inlet wind profile

Figure 16 - Catch ratio comparison: $k$-$\omega$ SST, Log-Law inlet wind profile

Figure 17 - Catch ratio comparison: Standard $k$-$\omega$, Log-Law inlet wind profile
CONCLUSIONS
Wind prediction accuracy plays a major role in the accuracy of the catch ratio prediction at any location on the windward façade. This is clearly shown in the results obtained from the comparison between measurements and numerical predictions for all three turbulence modeling schemes. Comparison with wind tunnel measurements shows a better match at points close to the building height for all modeling schemes. The closer to the ground, the higher the discrepancy. This is identically reflected in catch ratio prediction. Closer to the top edge, a better agreement is achieved with filed measurements.

Below the building height, the Standard k-ω shows a better match upstream of the building than the other three methods in maximum value of error (6% to 9% improvement over the conventional Realizable k-ε) and average value of error (nearly 2% improvement over conventional Realizable k-ε) compared to measurements. Windward façade comparison shows that Standard k-ω predicts a more accurate velocity field almost everywhere on the façade.

Above the mid-height of the building, Realizable k-ε with Power law inlet velocity profile shows better simulation results upstream of the building in maximum error and average value of errors.

Above the roof, Standard k-ω shows a better match in terms of maximum error and least average value of errors almost everywhere except for locations above the Mechanical room for which the least value of maximum error is by the k-ω SST, although all the methods including Standard k-ω on an average perform the same above the mechanical room.

Subsequently, we expected the similar level of accuracy in catch ratio modeling to wind prediction from different numerical methods employed in this paper. At near top edge and side edge gauges, catch ratio is better obtained using Standard k-ω.

As the purpose of such CFD simulations is accurate predictions near the top of the façade as that region is the most vulnerable to WDR, Standard k-ω performs better than conventional Realizable k-ε in terms of maximum error by nearly 5% and in terms of mean value of errors, by more than 5%.

On the comparison of Power law and Log law inlet velocity profiles, Log law performs better by a margin of nearly 20% on the maximum error value and 12% on mean value of errors.

More cases have to be added to these results to be able to effectively suggest the best turbulence model for this application. Building façade details and its surroundings effects as well as a transient solution to this problem will be investigated to determine which model would performs better.

ACKNOWLEDGEMENTS
Financial supports are received from NSERC strategic research Network for Engineered Wood-based Building Systems (NEWBuildS), Homeowner Protection Office branch of BC Housing, and the Faculty of Engineering and Computer Science of Concordia University.

REFERENCES


low-rise building: new validation efforts. s.l., s.n., pp. 428-435.


