The Impact of Air Quality and Site Selection on Gas Turbine Engine Performance

David MacPhee\textsuperscript{a}, Asfaw Beyene\textsuperscript{b}

\textsuperscript{a} University of Alabama, Tuscaloosa, AL, USA, dvmacphee@ua.edu
\textsuperscript{b} San Diego State University, San Diego, CA, USA, abeyene@mail.sdsu.edu

Abstract:

Air pollution can have detrimental effects on combustion engine performance leading to blade fouling which reduces power output and requires frequent cleanings. This issue is a fairly well-known phenomenon in the power industry. However, site selection for gas turbine installation is rarely part of the decision making process, mainly due to lack of site selection options especially in an urban environment or due to a simple assumption that air quality at a local micro-level has no bearing on the performance of the engine. In this paper, we perform a CFD study on an area surrounding a Combined Heat and Power facility to assess the impact of local wind distribution on air quality and the performance of a gas turbine engine thereof. Several aerodynamic properties are suggested as possible indicators of air quality and/or high air particulate concentration. These indicators are then compared to data collected at various points in and around the plant. The results suggest that CFD analysis of surrounding terrain could provide motivation for gas turbine site selection and result in an increased efficiency.

Keywords:
Air Quality, CFD, Gas Turbine, Energy.

1. Introduction

The burning of fossil fuels has increased significantly over the past decade, and with it the presence of greenhouse gases in the atmosphere which have been linked to climate change and overall adverse health effects in human population centers. Due in large part to human health related interests, there has been a recent surge in research involving pollutant transport modeling, especially for urban areas, to predict air quality and ensure safe air and water are available to the general public.

While such pollutant transport models have garnered a significant increase in research interest in the past two decades, the majority of these micro-scale studies, as previously mentioned, involve the investigation of pollutant dispersion to ascertain impacts on human health in urban settings [1]. Although this is certainly an important application of applied CFD, other implications are possible, including for example the impact on the performance of gas turbines for power and/or heat generation in urban settings.

Gas turbine performance depends on many factors, such as inlet air temperature and humidity, and the presence of pollutants in inlet air act to degrade system performance over time. It is an inevitable consequence of the operation of gas turbines that regular cleanings are required, sometimes involving complete shut-down of the system. This can be seen clearly in Fig. 1, where actual performance data, in Million Standard Cubic Feet (MSCF) per kilowatt hour (kWh) is shown over an entire year for a 5.2MW gas turbine utilized in a CHP plant in San Diego, CA, USA. Data is taken every 15 minutes for the duration of this period, and a “saw-tooth” curve of daily efficiency maxima is quite clearly seen to decrease over time between cleansing events.
The main reason for this loss in performance over time is a result of particulate matter becoming attached to compressor and turbine blades. It is typical to schedule frequent regular cleanings based on the operational history of the turbine and its expected performance degradation which is due in part to local pollutant concentrations. These cleanings can be simple pressure washings or, in extreme circumstances, can require complete shut-down and hand-washing of all turbo-machine blades. As a result, turbine fouling is a significant factor in the economic viability of gas turbine power plants, and must be considered carefully when designing such a system.

While the effects of pollutant concentrations on gas turbine performance has been extensively studied by researchers in the commercial and academic sectors over the years (e.g., [2], [3]), surprisingly very little research has been conducted into the exact role of turbine geographic location, with respect to local ground topology, on the economic viability of a CHP plant.

Due to the nature of fluid flow, air velocity is significantly impacted by local ground curvature, and, due to the relatively close proximity of gas turbine inlets the ground, local topology should be taken into account when designing CHP sites. In fact, simple hills and valleys adjacent to the proposed CHP site could significantly impact the transport of pollutants into and/or out of the CHP area.

The purpose of this study is to investigate the possibility of using a form of a traditional meteorological micro-scale [4] method in predicting the pollutant concentration in and around a gas turbine power plant. More specifically, the aim is to assess the role of traditional fluid dynamics variables (e.g., velocity, pressure, etc.) as opposed to adding extra transport equations (besides the fluid momentum and any other turbulence transport equations) which serves to computationally increase the complexity of simulations.

In order to compare these variables with pollutant concentrations, particulate data is gathered using a P-Trak Particle Counter at strategic points in the area surrounding the gas turbine power plant. These concentrations, combined with simulation data, could give considerable insight into possible locations of future gas turbine or Combined Heat and Power (CHP) plants, increasing power production and economic viability when compared to those sites selected without such measures.

**1.0 – CFD Modeling**

To simplify the modeling procedure, consider a rectangular prism, whose six sides correspond to the ground, sky (or, top), sides, inlet and outlet, Fig. 2.
For the time-being, the exact dimensions of the simulation domain seen in Fig. 2 need not be of concern, as the governing equations and boundary conditions are of sole importance. Also, note that the origin is marked as “0” and the orientation of the Cartesian coordinate system $x_i$ is also included for simplicity in the following section.

The scale of the problem will be on the micro-scale order, and as a result the Coriolis force effects have been neglected. Similar studies (e.g. [5]) of similar scales have made the same assumption without problem.

The equations of motion in the domain will be governed by the steady state $k$-$\varepsilon$ RANS turbulence equations. Assuming an incompressible fluid, neglecting gravity and other thermo-physical effects the mass conservation and momentum equations are written as:

1. \[ \frac{\partial u_i}{\partial x_i} = 0 \]
2. \[ \rho u_j \frac{\partial u_i}{\partial x_j} = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( 2\mu S_{ij} + \rho \tau_{ij} \right) \]

where $u_i$ is the fluid velocity, $S_{ii}$ is the mean strain rate tensor, and $\tau_{ij}$ is the specific Reynolds stress tensor resulting from applying the foundations of Reynolds averaging. These tensors can be described as follows:

3. \[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \]
4. \[ \tau_{ij} = -u_i u_j \]

Here, the over-bar in Eqn. (4) indicates a time average, while the prime symbol indicates a fluctuating quantity. It should be noted that in Eqns. 1-4, the pressure $p$ and velocity $u_i$ are already time-averaged.

The goal of all turbulence modeling is to approximate the specific Reynolds tensor, $\tau_{ij}$. This has been done a variety of ways over the years, however, in this paper the k-$\varepsilon$ turbulence model is used. This model makes use of the Boussinesq hypothesis, which relates the Reynolds stress to two other quantities: one, an intuitive variable in turbulence modeling, is the turbulence kinetic energy, $k$, and the other, is the kinematic eddy viscosity, $\nu_T$.
\[ k = u_i u_i \]  
\[ \tau_{ij} = 2\nu_t S_{ij} - \frac{2}{3}k\delta_{ij} \]

The standard k-\( \varepsilon \) turbulence model relates the kinematic eddy viscosity to the turbulence kinetic energy and another variable, the turbulence dissipation, \( \varepsilon \). This is termed a two-equation model, since closure can be achieved through the addition of two more transport equations. First, the relationship between the eddy viscosity and the two new turbulence quantities is expressed as follows:

\[ \nu_t = C_\mu k^2 / \varepsilon \]

Here, \( C_\mu = 0.09 \) is a constant. The two transport equations for a steady-state solution can be written as:

\[ u_i \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} + \nu \frac{\partial}{\partial x_j} \left[ \nu + \nu_t \right] \left[ \frac{\partial}{\partial x_j} \right] \]

\[ u_i \frac{\partial \varepsilon}{\partial x_j} = C_{e1} \varepsilon \frac{\partial u_i}{\partial x_j} - C_{e2} \frac{\varepsilon^2}{k} \]

\[ + \frac{\partial}{\partial x_j} \left[ \nu + \nu_t \right] \frac{\partial \varepsilon}{\partial x_j} \]

Finally, to completely describe the system of equations, the closure coefficients must be defined: \( C_{e1} = 1.44, C_{e2} = 1.92, \sigma_k = 1.0 \) and \( \sigma_\varepsilon = 1.3 \).

The solution to this system of equations, given sufficient boundary and initial conditions (to be explained shortly) is accomplished using the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm. It is a pressure based solver, consistent with most incompressible simulations, which, explained simply, takes an initial guess and refines this based on subsequent substitutions into a pressure Poisson equation, which in turn is derived from the momentum equation.

To completely explain the boundary value problem in question, all that remains is to describe the boundary and initial conditions. First, the velocity boundary is required to mimic a natural average wind speed for the selected site. The boundary in question is oriented to have wind velocity in the \( x_1 \) direction, Fig. 2. Using a reference wind speed \( u_r \), and the height at which it was measured at, \( h \), a power-law relationship is described, which models the incoming air as a function of the height above ground, or \( (x3 - e) \), where \( e \) is the elevation in m, as:

\[ u_1 = u_r \left( \frac{x_3 - e}{h} \right)^\alpha \]

\[ u_2 = u_3 = 0 \]
Here, $\alpha$ is a constant set to 1/7 or 0.143. This type of velocity profile is often used to calculate power densities of wind, and especially for description of macroscopic wind velocities [6,7].

At all other boundaries, a zero gradient boundary condition is applied normal to the surface:

$$\frac{\partial u_i}{\partial x_j} n_j = 0$$  \hspace{1cm} (11)

where $n_j$ is the outward pointing surface normal. Simply speaking, this means that the flow is not changing in the direction of the surface and can be loosely expressed as “fully developed”. As long as boundaries are far enough away from the surface, this boundary condition does not affect validity of results.

Concerning initial conditions, the velocity is set to a value of $(4.5,0,0)$ m/s everywhere in the domain for the sake of simplicity.

The pressure boundaries have similar constraints as the velocity; on the outlet a pressure of zero is enforced:

$$p = 0$$  \hspace{1cm} (12)

And, on all other boundaries, the zero gradient condition normal to the wall is again invoked:

$$\frac{\partial p}{\partial x_j} n_j = 0$$  \hspace{1cm} (13)

Due to the nature of the incompressible solver, the units of pressure herein are $m^2/s^2$, due to the division by density to facilitate the solution of governing equations. Finally, the pressure is initialized by letting $p = 0$ everywhere in the domain. Note that since this solver takes into account only relative pressure differences, this zero pressure was chosen arbitrarily.

For the turbulence kinetic energy $k$, the inlet boundary will have a constant value according to the following:

$$k = \frac{3}{2} (u_r I)^2$$  \hspace{1cm} (14)

where $I$ is the turbulence intensity. The remaining boundaries have the same zero-gradient boundary conditions as the pressure:

$$\frac{\partial k}{\partial x_j} n_j = 0$$  \hspace{1cm} (15)

Finally, the turbulence dissipation $\varepsilon$ at the inlet is assumed constant and defined as follows:

$$\varepsilon = C_\mu \frac{k^{3/4}}{l}$$  \hspace{1cm} (16)

where $l$ is the turbulence length scale, taken as the mean side length for the ground discretization. Other boundaries once again enforce the zero-gradient condition.
\[
\frac{\partial \varepsilon}{\partial x_j} n_j = 0
\]  

(17)

The boundary value problem is now sufficiently described, and is solved using OpenFOAM [8], a finite volume CFD toolbox written in C++ and capable of a wide array of multi-physics simulations, including CFD. The domain discretization is completed using blockMesh, a native meshing tool which creates volumes based on the subdivision of blocks, created from vertices.

In order to subdivide the domain into blocks usable by blockMesh, a program was written that projects terrain data, which contains geocentric latitude \( \psi \), longitude \( \theta \) and elevation \( e \) from sea level, into data of the form \( \{x_1,x_2,x_3\} \). If the sample spaces of latitude and longitude are contained in the intervals \([\psi_{\text{min}},\psi_{\text{max}}]\) and \([\theta_{\text{min}},\theta_{\text{max}}]\), respectively, then the coordinates of each point in the domain can be constructed according to the arc-length experienced at the average latitude and longitude. To adhere to the origin seen in Fig. 2, the mapping of points is as follows:

\[
x_1 = R|\theta - \theta_{\text{max}}| \cos \left( \frac{\psi_{\text{max}} - \psi_{\text{min}}}{2} \right) \\
x_2 = R|\psi - \psi_{\text{min}}| \\
x_3 = e
\]

(18)  
(19)  
(20)

Here, \( R \) is taken as the radius of the earth, \( 6.371 \times 10^6 \text{m} \). As the domain is assumed rectangular, the maximum height in the \( x_3 \) direction must also be described, and is taken as a large number, \( H \), which is chosen to be far enough away from the ground so that the upper boundary does not affect the solution.

\[
x_{3\text{max}} = H
\]

(21)

As a final note on geometric considerations, it should be pointed out that the previous discussion has assumed increases in elevation do not affect arc-lengths due to their small values when compared to \( R \).

Blocks were then created for each square mapped to the terrain, extending from \( x_3 = e \) to \( x_3 = H \). The subdivision of these volumes is accomplished according to an assumed expansion ratio, \( \Delta_r \), as follows:

\[
\Delta_r = \frac{\Delta_e}{\Delta_s}
\]

(22)

where \( \Delta_e \) is the end cell length in the vertical direction and \( \Delta_s \) is the beginning cell length. Intermediate cell lengths are interpolated linearly.
2.0 – Case Study

The area to be considered herein is taken as the region lying within coordinates \( \psi \in [32.735^\circ, 32.81^\circ] \) and \( \theta \in [-117.273^\circ, -117.05^\circ] \), Fig. 3. This region represents the area surrounding the aforementioned gas turbines at San Diego State University, shown as the red dot in Fig. 3. This area is chosen due to the frequent cleanings required for the gas turbines providing power and waste heat, and in fact the performance data from Fig. 1 are taken directly from measurements from one of the two gas turbines at this particular CHP site.

For the simulation of this area, the air kinematic viscosity and density are taken as \( \mu = 1.8 \times 10^{-5} \) kg/(ms) and \( \rho = 1.2 \) kg/m\(^3\), respectively, the turbulence intensity is taken as 1% or \( I = 0.01 \), and the reference velocity (see Eqn. 10) is taken as 4.5 m/s, measured at a height of 80m [10]. The grid discretization results in a projected area of 8.34km (north-south) by 20.85km (west-east), and, with 91×269 nodes, the average side length gives an estimated turbulence length scale of \( l = 42.6 \)m. The ceiling or “top” height in Fig. 2 is chosen to be \( H = 1000 \)m, and elevation data is extracted from Google Maps [11].

After importing into OpenFOAM, the entire domain is seen in Fig. 4, along with a visual representation of the ground elevation, associated contours and block grid discretization. Each block is meshed to have 80 volumes in the \( x_3 \) direction with a cell expansion ratio of \( \Delta r = 500 \), as well as 2 volumes in each of the \( x_2 \) and \( x_1 \) directions. This results in a total grid size of just over \( 7.7 \times 10^6 \) cells, with the first cell center located around 2cm above ground level.

The simulation is then run using traditional linear 2nd order finite volume spatial discretization until relative residual errors for the \( u_1, u_2, u_3, p, e, \) and \( k \) solvers is less than \( 1 \times 10^{-5} \). After reaching these tolerances all flow variables were saved and are analyzed herein. The simulation itself was conducted using a six-core Intel Xeon X5680 processor running at 3.33 GHz with adequate (24GB) high speed RAM, and required just over 3 days to complete in parallel, splitting the region up into 3, 2, and 1 decomposed sub-domains in the \( x_1, x_2 \) and \( x_3 \) regions, respectively.
Figure 4: Discretized domain, showing a) entire domain, b) elevation, c) elevation contours, and d) ground grid spacing.

In order to compare the CFD results to real data, a series of measurements were taken at specified locations using a particle counter, described previously. At each location, the counter was allowed
to measure concentrations for a full day, and the resulting measurements, in particles per cubic centimeter, or pt/cc, are shown in Tab. 1, along with geographic data for each locale.

Table 1: Measurement locations and measured particle concentrations.

<table>
<thead>
<tr>
<th>Location</th>
<th>Latitude [°]</th>
<th>Longitude [°]</th>
<th>Particle Conc. [pt/cc]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-117.07003</td>
<td>32.78090</td>
<td>5797</td>
</tr>
<tr>
<td>2</td>
<td>-117.09395</td>
<td>32.79015</td>
<td>7462</td>
</tr>
<tr>
<td>3</td>
<td>-117.12061</td>
<td>32.78437</td>
<td>4886</td>
</tr>
<tr>
<td>4</td>
<td>-117.14043</td>
<td>32.77397</td>
<td>3874</td>
</tr>
<tr>
<td>5</td>
<td>-117.14521</td>
<td>32.78668</td>
<td>2429</td>
</tr>
<tr>
<td>6</td>
<td>-117.16640</td>
<td>32.76935</td>
<td>3821</td>
</tr>
<tr>
<td>7</td>
<td>-117.24773</td>
<td>32.75085</td>
<td>3995</td>
</tr>
</tbody>
</table>

Along with the saved variables in this case, i.e., velocity, pressure, turbulence kinetic energy and turbulence dissipation, the wall shear traction vector was also calculated as follows:

$$\tau_w = \mu n_i \frac{\partial u_j}{\partial x_i}$$

(23)

This variable serves as yet another to compare particle concentrations with the output of a simplified CFD simulation, which will be discussed shortly.

After tolerances were reached and field variables recorded, data was sampled at heights of 0m, 2.5m, 5m, 10m, 20m and 40m above ground level, since CHP gas turbine intakes are generally within this region of elevation above ground. To begin with, the thermodynamic pressure is seen plotted against particle concentration in Fig. 5.

**Figure 5**: Pressure at various specified heights above ground, compared to particle concentration.
As can be seen in Fig. 5, a general relationship between decreased pressure and decreased concentration is observed. However, this result should be taken lightly, since the pressure gradient is of primary importance in describing fluid flow, not the absolute value. It is highly likely that this pressure gradient is more indicative of particulate concentration, which will be discussed further when discussing wall shear traction vector magnitudes at the ground.

Another quantity readily available in CFD simulations is the air velocity. However, since this is a vector quantity it is much easier to simplify this by calculating the velocity magnitude, or $|u_i|$, which is defined as follows:

$$|u_i| = \sqrt{u_i^2}$$  \hspace{1cm} (24)

The velocity magnitude can be loosely related to the mass flux, which should be indicative of the amount of particulate transported through a given area. The relationship between velocity magnitude and particle concentration in this simulation is shown in Fig. 6.

![Figure 6](image)

**Figure 6:** Velocity magnitude at various specified heights above ground, compared to particle concentration.

Unfortunately, no discernible relationship between the velocity magnitude data of Fig. 6 and the particle concentration can be drawn.

Due to the nature of the turbulence model used, it is also possible to plot the sampled data of turbulence kinetic energy $k$ and turbulence dissipation $\varepsilon$, which are shown in Figs. 7 and 8.
Obviously, the turbulence kinetic energy at the ground should in theory be zero, however the sampled data has been interpolated to the surface and is a result of logarithmic near-wall modeling, explaining the non-zero values close to the wall.

Both the turbulence kinetic energy and the dissipation close to the wall exhibit similar characteristics, which is not at all surprising since turbulence dissipation should in theory be higher in regions of high turbulence kinetic energy.

Lastly, the wall shear traction vector magnitude has been calculated according to the following:

$$|\tau_w| = \sqrt{\tau_{w,i} \tau_{w,i}}$$  \hspace{1cm} (25)
Low wall shear could be a general indicator of stagnant pressures areas, and, as a result, the wall shear traction vector is plotted in Fig. 9, along with its inverse, $1/|\tau_w|$, shown in Fig. 10.

**Figure 9**: Wall shear traction vector magnitude at various specified heights above ground, compared to particle concentration.

**Figure 10**: Wall shear stress inverse.

The wall shear traction magnitude inverse seems to comply loosely with the particulate concentration, especially predicting the lowest concentration at point 5.

This suggests that there may be a connection between CFD-simulated wall shear stress and particulate concentration in urban areas. Although pressure in general must decrease along the flow direction due to the nature of fluid flow, Fig. 11, nevertheless there could be further relationships between pressure gradient, which drives fluid flow, and particulate concentration.

These regions of low wall shear stress likely represent areas of stagnation, and therefore, more particulate matter is likely to coalesce. For this reason, it is suggested that a low-fidelity CFD
simulation, calculating wall shear stress at the ground, could be an excellent predictor of air pollutant concentration, especially those areas which experience low wall shear stress.

**Figure 11:** Wall shear traction magnitude (top) and pressure (bottom) at the ground.

### 3.0 – Conclusions

Site selection for Combined Heat and Power (CHP) plants is usually related mainly to factors associated with ease of power and heat distribution in a given area. However, it is well known that the presence of air pollutants can have a detrimental effect on gas turbines, causing frequent cleanings and associated down-time, which can adversely affect system performance both energetically and monetarily. This study has advocated for a simple Computational Fluid Dynamics (CFD) approach to better predict areas which experience low particulate density, better suited to CHP site selection.

The present study has indicated a general approach to modeling a micro-scale simulation using the open-source CFD software, OpenFOAM, and utilizing the standard $k$-$\varepsilon$ turbulence model. Using terrain data and an assumed velocity profile, CFD field variables are compared alongside particulate concentrations, which were collected at strategic locations around the CHP site at San Diego State University, a location where turbine fouling is a major concern and detriment to turbine performance.

Using the results from CFD simulations, it was found that generally, a high wall shear stress is indicative of low particulate density, and likely a good candidate for CHP site selection. Ongoing studies in this area are underway to investigate with much higher cell density, especially in areas directly adjacent the CHP site, the effect of wall shear stress and perhaps pressure gradient at selected levels above ground, to determine whether further correlations can be drawn.
Nomenclature

**Greek Letters**
- $\tau_{ij}$: Reynolds stress tensor
- $\nu_T$: Kinematic eddy viscosity
- $\delta_{ij}$: Kroenecker delta
- $\mu$: Dynamic viscosity
- $\varepsilon$: Turbulence Dissipation
- $\theta$: Angle of longitude
- $\psi$: Angle of latitude
- $\Delta_r$: Cell expansion ratio
- $\Delta_e$: End cell length
- $\Delta_s$: Beginning cell length
- $\tau_w$: Wall shear traction vector
- $\rho$: Air density

**Latin Characters**
- $u_i$: Air velocity
- $x_i$: Coordinate position
- $p$: Thermodynamic pressure
- $S_{ij}$: Mean strain-rate tensor
- $k$: Turbulence kinetic energy
- $e$: Elevation
- $u_r$: Reference velocity
- $h$: Reference height
- $n_i$: Surface normal vector
- $I$: Turbulence intensity
- $l$: Turbulence length scale
- $R$: Earth radius
- $H$: Maximum domain elevation

References


