The Application of Numerical Modeling in Efficiency Driven Designs – Ventilation Problem

Eric Chen, P.Eng., M.Eng., PMP,
Lyne Ricard, P.Eng., PMP
& Alexia Grenier, P.Eng., Ph.D.

February 2019

Disclaimer

This document is provided for informational purposes only. Anyone who uses this document is responsible for determining whether or not it applies to his or her activities. At no time will BBA or the author be held liable for any error or omission that the document may contain or that may result from its use.

The information contained in this document is the proprietary and exclusive property of BBA Inc. except as otherwise indicated. No part of this document may be reproduced, stored, transmitted, or used for design purposes without the prior written consent of BBA Inc.

The information contained in this document is subject to change without notice.

Executive Summary

The concept of Computational Fluid Dynamics (CFD) and its applications can be dated back as early as the 1960s. Over the years, due to the development of both theoretical research and computer technologies, CFD has become a very useful tool for design engineers and researchers for complex problems where manual calculation is neither sufficient nor possible.

This paper demonstrates a design example of a CFD application for ventilation problems in a greenhouse. Due to the forced air inflow and outflow by fans, turbulence was expected to form inside the greenhouse, which will affect the CO2, H2O and temperature distribution and ultimately impact the plant growth in certain areas of the greenhouse.

A 3D numerical model was developed in Ansys CFX to analyze this problem. This model is a multi-physics model that includes the simulation of dynamic flow distribution, heat transference and gas mixing. For accurate calculation of these three variables in the domain, three submodels were created within the overall model and were two-way coupled to achieve integration.

Various methodologies and techniques where applied to verify the model input and output, and different model sensitivity analysis scenarios were also discussed. Based on the simulation results, optimizations were implemented, which significantly improved the air circulation inside the greenhouse.

Although this paper is an example of a CFD application in the greenhouse, a similar methodology and concept have been applied in many other applications and other industries, which are listed at the end of this paper.

Table of Contents

Project Challenge

The greenhouse is a controlled environment adjusted to the optimal conditions for the plant growth. To increase efficiency, some greenhouses are designed with an additional supply of CO_2 gas through piping. The temperature and humidity inside the greenhouse will also be controlled according to the plant being grown. One of the most common challenges of operating a greenhouse in cold weather is air ventilation. Due to the cold ambient temperature outside, the air has to be heated before entering the greenhouse by being directed to a heat source by fans before returning to the atmosphere to reduce heat loss. The forced airflow (rather than natural airflow) will usually cause turbulence or eddies, where relative humidity, CO_2 concentration and temperature become the potential problem for plant growth efficiency.

Since this is a multi-physics problem that includes dynamic flow distribution, heat transference and gas mixing, it is impossible to solve all the equations by hand. On the other hand, traditional physical modeling is costly and requires a lot of effort to adjust the model for iterations.

Numerical Model Simulation

Due to the above-mentioned factors, a 3D numerical model will be the most efficient solution for the greenhouse ventilation problems, because it can produce a detailed and reliable result while allowing for iterations of the model in a much faster timeline and at a much lower cost than physical modeling.

The proposed footprint of the greenhouse model is 60m (L) x 40m (W) with 12 air inlets on three walls and 16 air outlets on the roof (see Figure 1). The three walls with ventilation are exposed to the outside atmosphere. The last wall is a separation with another cell of the greenhouse. To properly simulate this problem, Ansys CFX has been chosen for the numerical modeling. Ansys CFX is the state-of-art CFD simulation software that has been used for over a decade with proven accuracy in many industries, including aircraft, car, hydroelectric, mining, etc.

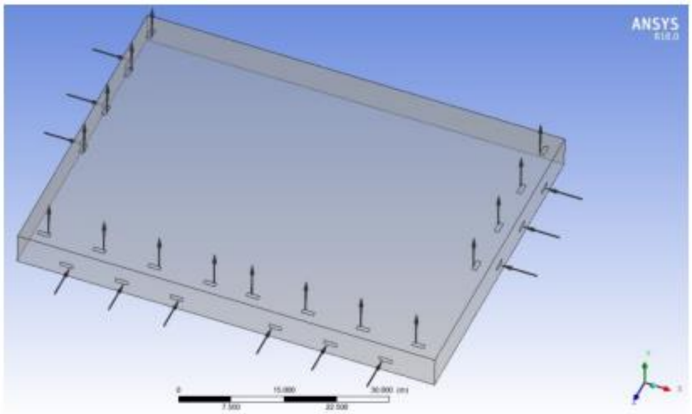

Figure 1 Model Geometry

Model Set Up

To start the project, a simplified 3D geometry representing the interior volume of the greenhouse was created and imported to Ansys. The geometry was then meshed inside of Ansys, which created over 5 million high-quality elements. A mesh inflation technique was also applied to the model near the walls, which generated 10+ thin layers of elements within the boundary layer region to accurately capture the performance near the walls. The simulation set up mainly includes the set up of three sub-models:

1. General air flow simulation

The Standard k-e turbulence model is employed to capture the turbulence & eddy formation and associated kinetic energy. This model is the industrial standard model and has been proven to be stable and robust for this type of analysis. A scalable wall function was added to this turbulence model to increase model accuracy for the turbulence behavior near the wall. The inlet and outlet were assigned with the associated flow rate, turbulence intensity and length scale.

2. Air mixing simulation

Aside from the inlet, additional H2O gas will also enter the greenhouse through plant transpiration. To properly simulate the H2O gas concentration inside the greenhouse, a Variable Composition Mixture material (which was defined to contain air and H2O) was used in the model instead of Pure Air. This material has the feature of capturing the volume percentage of each component, which is partially based on the flow streamline and distribution obtained from the flow simulation. Transpiration was modelled for various types of plants and growth stages. CO2 gas was not modeled in this simulation, as it is not of major concern for this project; however, the concept is the same as the H2O gas and a good prediction of CO2 gas distribution can be obtained based on the flow field and distribution of H2O gas.

3. Sun radiation and thermal convection simulation

A static temperature was defined at the inlet boundary for gas entering the greenhouse. In addition, a Monte Carlo Radiation model was employed to simulate the energy the greenhouse will gain from the sunshine. The greenhouse wall was assigned a proper heat transfer coefficient based on its material to represent heat loss or gain.

The heat will in general enter the greenhouse via the inlets and radiation from the sun. It will then transfer freely inside the greenhouse based on the thermal energy difference and based on the airflow field. It will eventually exit the greenhouse via outlets and walls.

The above three sub-models were all two-way coupled (e.g. the overall flow distribution induced by the fans at the inlet and outlet will influence heat transference and the convection induced by the thermal energy difference will also impact the overall flow distribution (although it is small in this project)). Due to the complexity of the problem, a high-resolution model solver was chosen for computing, and a small time step was applied to accurately capture the interactions.

After the model is set up, it was sent to a 24-core super computer for processing.

Model Results

The initial model results showed large turbulence formation due to the forced airflow from the inlet and outlet fans, which created “dead zones” where air is not well circulated. As a result, the temperature is lower than average in that region and the humidity is higher (see Figures 2 to 5)

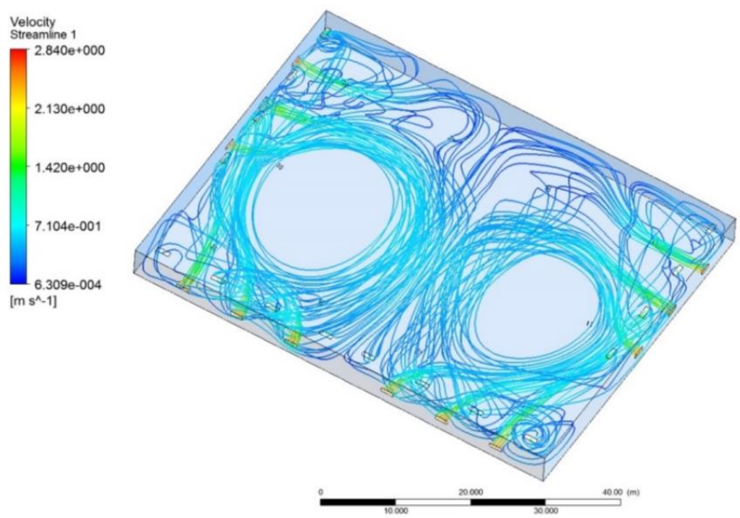

Figure 2 Flow Streamline inside the Greenhouse

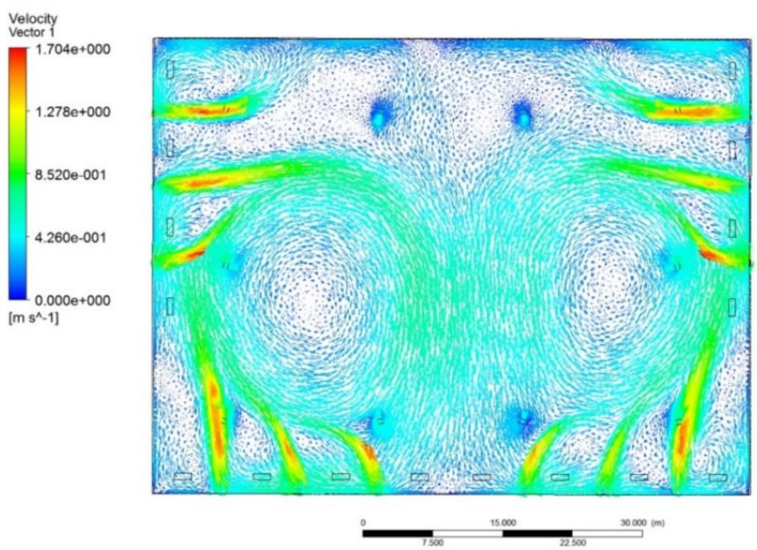

Figure 3 Flow Field Velocity Vector

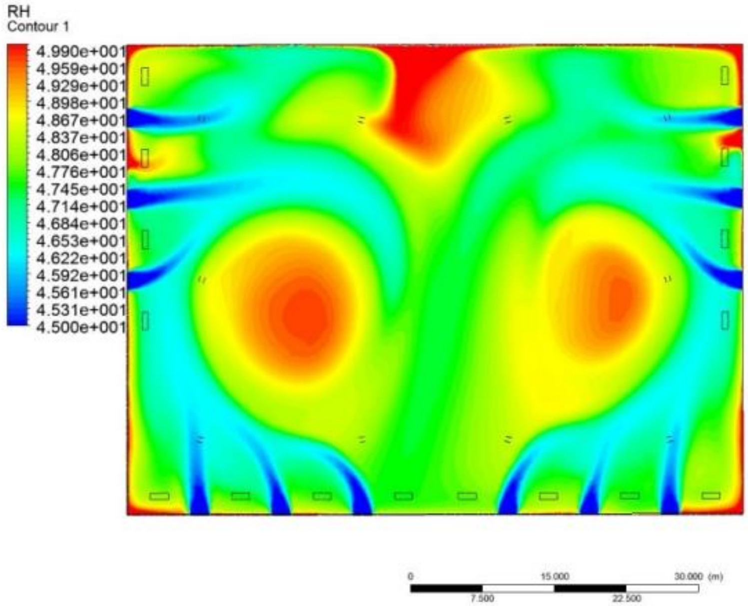

Figure 4: Relative Humidity Distribution

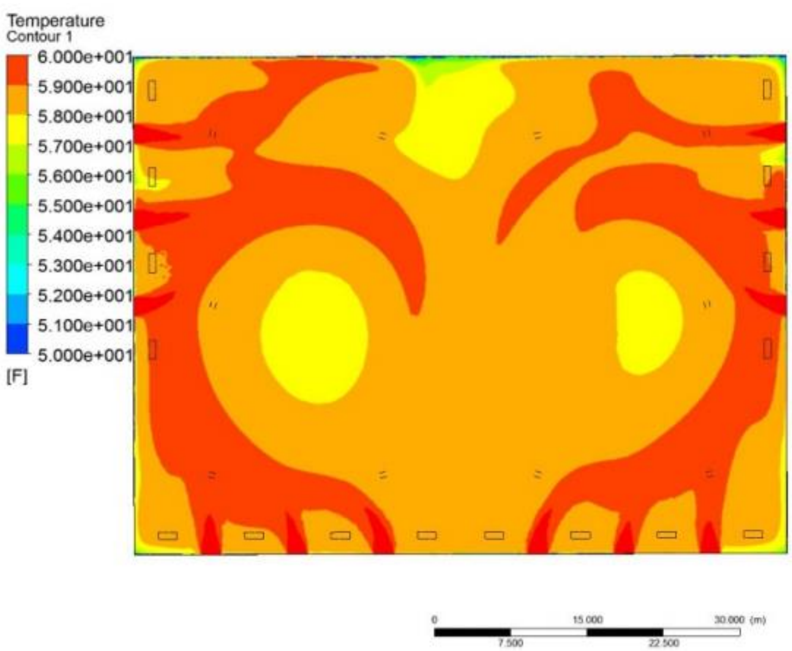

Figure 5 Temperature Distribution

Model Optimization

To improve the flow performance, a new model was proposed with 18 fans inside the greenhouse (see Figure 6). The inlets and outlets were placed side by side on one wall of the greenhouse, minimizing installation cost. The fans were placed in 4 rows and each row has the airflow direction opposite to adjacent rows. It is to be noted that this configuration is different from the traditional linear fan arrangement.

The simulation results for the updated model revealed a smoothly distributed pattern for flow, temperature and humidity. The 4 rows of fans act as air barriers to cut the turbulences and no dead zones were observed (see Figure 7 ~9). Despite the significant reduction in the number of inlets and outlets (i.e. from 12 inlets and 16 outlets in the first model to 4 inlets and 4 outlets in the latter model), the temperature and humidity remained at the same level as the first model.

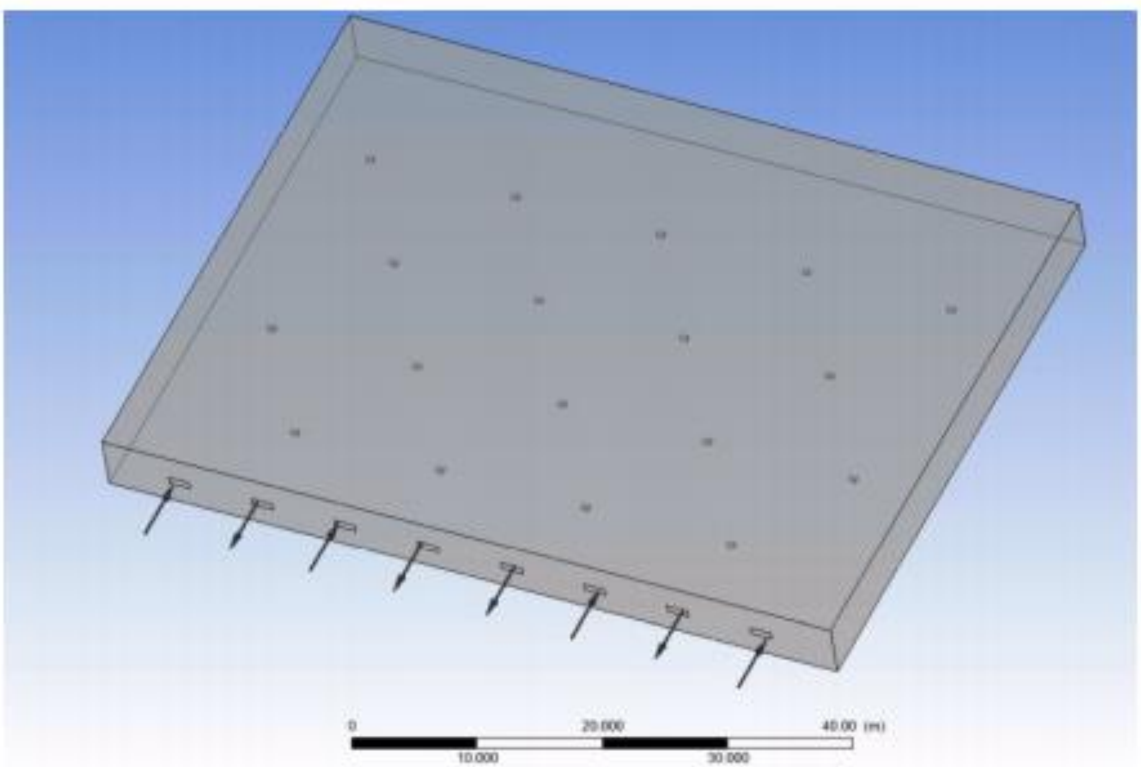

Figure 6: Improved Model Geometry

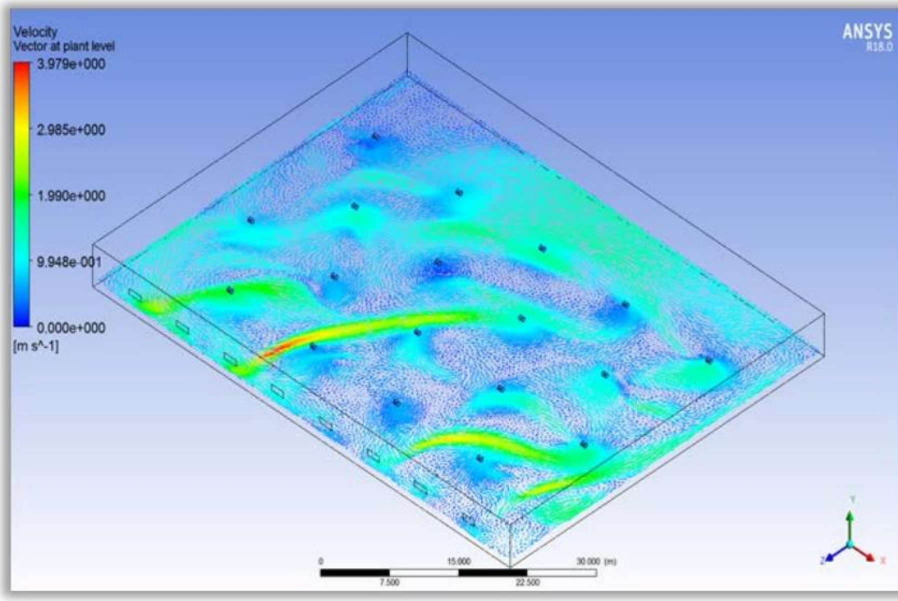

Figure 7: Flow Field Velocity Vector (Improved Model)

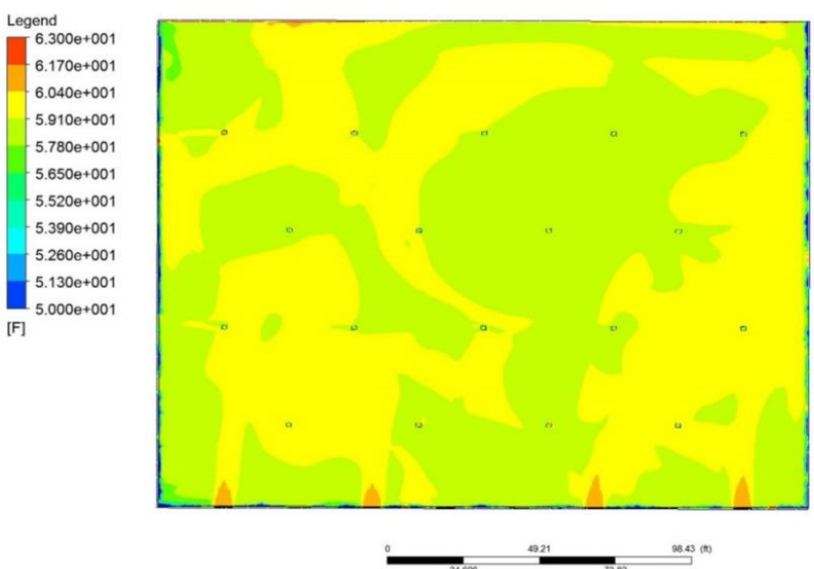

Figure 8: Temperature Distribution (Improved Model)

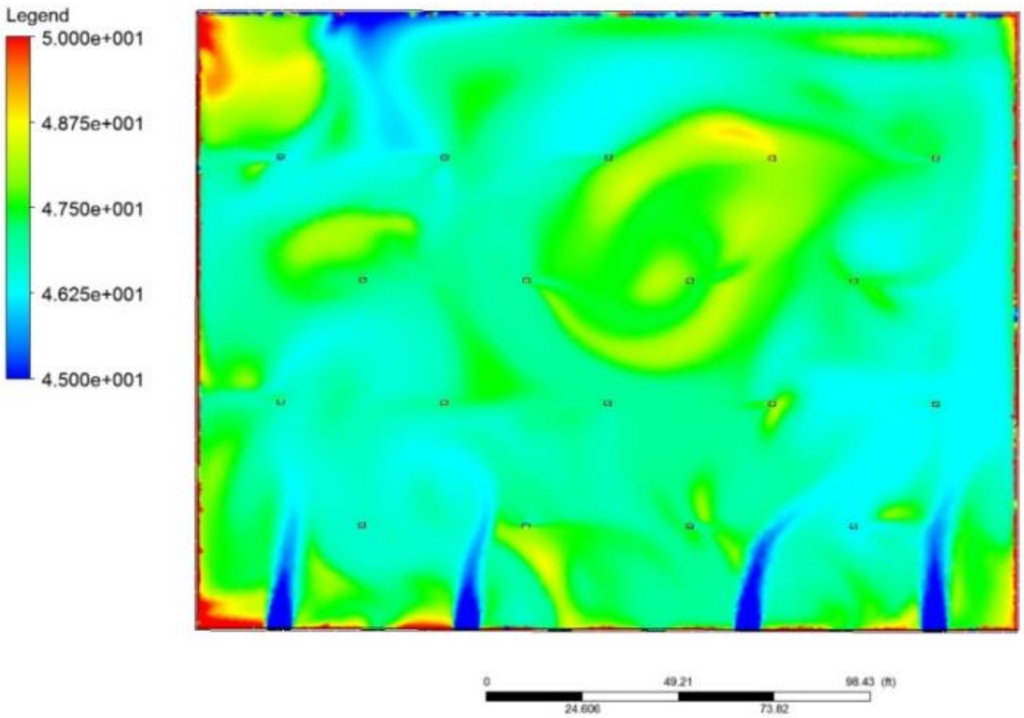

Figure 9: Relative Humidity Distribution (Improved Model)

Grid Independent Analysis

The mesh quality is important to the accuracy and convergence of the model, especially for the multi-physics model with many variables and equations. Therefore, an independent analysis of the grid was carried out by checking the model with all parameters unchanged except the meshing methodology and element size, which varied. The comparison shows consistence in the results between different mesh cases.

The mesh element quality was also checked against Orthogonality, Aspect Ratio, Expansion Rate and other criteria.

Model Sensitivity Analysis

A series of sensitivity analyses were conducted by changing one input parameter while keeping the rest of the parameters and meshing methodology unchanged (see below). The results of the sensitivity cases were compared to base model for differences.

- Turbulence models

A separate simulation with Shear Stress Transport (SST) turbulence model was created to compare with the base case (i.e. k-ε turbulence model) for pressure distribution, velocity field, turbulence kinetic energy, etc.

- Boundary Conditions

Some assumptions were made based on different operational scenarios (e.g. outside temperature, fan flow rate, wall heat transfer coefficient, sunshine intensity, etc.). These scenarios were applied to the model by changing the boundary conditions (one variable at a time) and results were also compared with the base case to see the sensitivity of the flow field with the change of boundary conditions.

- **Fan capacity and position inside the greenhouse**

  Different scenarios for the layout of the interior fans were modeled (e.g. location, number, capacity and orientation). The results were compared with the base case to find out the optimal solution.

Conclusion

Based on the simulation results and the optimization work, the atmosphere inside the greenhouse was created in such a way to ensure uniformity of air flow/distribution, temperature and CO_2 concentration, which significantly improved the conditions for plant growth in the optimum controlled space.

Figure 10: Greenhouse set-up with Rows of Fans

This greenhouse simulation is an example application of CFD. A similar methodology and concept has been used in many other applications where the problem or geometry is too complicated for manual calculation. The following are example applications for which BBA has experience and knowledge:

River Flood simulation;

- Flow simulation in hydraulic mechanical equipment (e.g. pipe, bifurcation, valve or turbine), including erosion simulation;
- Flow simulation in a hydraulic structure (e.g. intake, tailrace, spillway, etc.);
- Combustion simulation in a reactor;
- Ventilation simulation in any closed space (e.g. tunnel, mine, industrial plant etc.);
- Heat transfer simulation in a buried utility or closed space;
- Fluid mixing simulation.

This whitepaper is part of a series of whitepapers for CFD / Ansys applications in different fields.

About the Authors

Lyne Ricard, P.Eng., PMP
Director, Oil, Gaz and Biofuels
lyne.ricard@bba.ca

Lyne Ricard is a Chemical Engineer with over 30 years of experience. Known for her leadership, Ms. Ricard has wide-ranging experience in process engineering, project management, commissioning and operation, and business development. Before joining BBA in 2013, she held a number of management positions. Ms. Ricard draws on her extensive expertise to assist BBA with business development in the oil, gas, chemicals and biofuels markets. She is also responsible for the execution of all projects in these fields. Ms. Ricard is a member of BBA's Board of Directors.

Alexia Grenier, P.Eng., Ph.D.
Engineer | Oil, Gas and Biofuels
alexia.grenier@bba.ca

Alexia Grenier, Ph.D., holds over 10 years of experience as an Industrial Process Engineer. She has developed expertise in dust collection (managed 50 dust collectors for three years and performed numerous operational audits on bag filter units) and in industrial ventilation (dust control exhaust systems). She also participated in numerous solid/liquid separation projects (wastewater and industrial effluents) including sedimentation technologies, pressure filtration, vacuum filtration and membrane filtration (microfiltration, ultrafiltration).

BBA.CA