

Reviewed and Selected as the “Best ATMAE 2012 Conference Proceedings Paper”

Graphics

Integrated CAD/CFD Analysis of HVAC Fresh Air Intake System Design of an On-Highway Crane

Mr. Charles M. “Matt” Watson,

CTM, Link-Belt Construction Equipment Co., 2651 Palumbo Drive, Lexington, KY 40509, 859-263-5200, mwatson@linkbelt.com

Dr. Nilesh Joshi,

Morehead State University, Department of Applied Engineering and Technology, 201 Lloyd Cassity Building, Morehead, KY 40351, 606-207-0972, n.joshi@moreheadstate.edu

Abstract

There has been a consistent need regarding on-highway cranes for a means to introduce outside fresh air into the HVAC system of driver’s cab in order to better de-fog the windshield and side windows when a humid and/or wet outside environment is encountered. Existing design of the fresh air intake system was not functional because of its sub-optimal location, which resulted in the air conditioning system expelling hot air in spite of the temperature controls being set to the coldest position. Through the use of CFD analysis on 3D CAD models and graphics visualization, we were able to identify the problem with the existing design, and ultimately we were able to re-locate the fresh air intake for the system to an optimal location that yielded the best possible airflow. The integrated CAD/CFD analysis approach required minimal field testing and saved countless hours of physical modeling and mock-ups, thus minimizing the overall cost of the redesign.

Introduction

The focus of this paper is on demonstrating the use of integrated CAD (Computer Aided Design) /CFD (Computational Fluid Dynamics) analysis as an aid to mechanical design process. Computational fluid dynamics is basically the process of replacing equations that represent fluid flow with numbers and advancing those numbers in space and time to generate a final numerical description of the fluid flow of interest (Anderson et al., 2009). According to Watson (2012), so far CFD has been used to design high value-added products that depend heavily on fluid flow, such as airplanes and automobiles, but it has the potential to substantially improve performance of every product that relies upon fluid flow and heat transfer phenomena. A lot of research has been done on the development of CAD and CFD fields, but these research efforts have been mostly independent of each other. Very few research efforts can be found that captured the synergy between the two fields or used the CAD models in CFD simulation. The idea of closing the gap between CAD model and downstream applications such as CFD was first explored by Farouki (1999) at the SIAM (Society of Industrial and Applied Mathematics) Workshop on Integration of CAD and CFD, held at UC Davis. In the same year, Kellar et al. (1999) used a CAD/CFD interface to enable CFD numerical flow visualization for a generic racing car geometry as part of a detailed project investigating racing car wheel aerodynamics. More recently, Li et al. (2007) created a new innovative fully integrated CAD/CFD design environment to analyze the leakage in a twin-screw supercharger. Their integrated design environment can address all aspects of fluid and geometry requirements for the future supercharger designs. One of the important benefits of integrating CFD with CAD is that it eliminates the need to build costly physical prototypes thus saving time and resources for companies.

In this paper, we present a real world case study in which the use of integrated CAD/CFD design optimization and related graphical methods resulted in improved design of HVAC (Heating, Ventilation and Cooling) fresh air intake system of an on-highway crane. Over the course of development of an on-highway all-terrain crane, a design problem was identified with its HVAC fresh air intake system. It was noticed that whenever the air conditioning unit of the crane was switched on, the air being expelled from the ductwork was hotter than the ambient air temperature. The initial solution to the perceived problem was that there must have been some components that were assembled incorrectly or that the HVAC system had

not been properly charged with refrigerant. However, upon further investigation, it was surmised that there was an inherent flaw within the design of the HVAC system, as was supported by the condition stated earlier.

It was recognized that there was a need to develop an optimal location for fresh air intake into the crane HVAC system in order to stop superheated air from the inside of the engine compartment being introduced into the system which was rendering the system ineffective. The optimal location sought would have the following characteristics: 1) It would be located out of the superheated airflow, 2) be as hidden from plain sight as practical, 3) be shielded from road spray and debris, 4) be located in an easily accessible area for filter maintenance, and 5) would require the minimal amount of material and labor to relocate the intake.

First, initial testing was performed on the existing design and the critical temperature data was recorded under various operating conditions of the crane to establish a baseline and to better understand the problem with the existing design. Next, the 3D CAD model of the existing design was created for the CFD analysis. Then, the CFD analysis was performed on the CAD model. Finally, the new design was developed based on the results of the CFD analysis and ultimately was implemented into the production system.

Initial Testing and Data Collection

Initial testing was performed to establish a baseline fresh air intake temperature for on-highway travel operation. Background information was gathered about the problem of the airflow from the inside of the engine compartment for the fresh air intake system. It was observed that while the machine was in mobile on-road operation, there would be a significant amount of hot air coming through the HVAC system even though the air conditioner was activated.

Road tests were conducted and the critical temperature was measured using appropriately placed thermocouples. The critical temperature observed was the vent temperature inside the driver's cab which determined whether superheated air was being introduced into the HVAC system under various operating conditions. Figure 1 shows the location for the original fresh air intake. It was discovered during the initial long road run testing that this location was not an appropriate location to be drawing outside air into the HVAC system. This, however, was not obvious when the machine was stationary and the HVAC system appeared to function normally. Furthermore, by preventing air from entering the intake filter and vent, it was observed that the HVAC system functioned as intended.

Figure 1. Original Fresh Air Intake Location

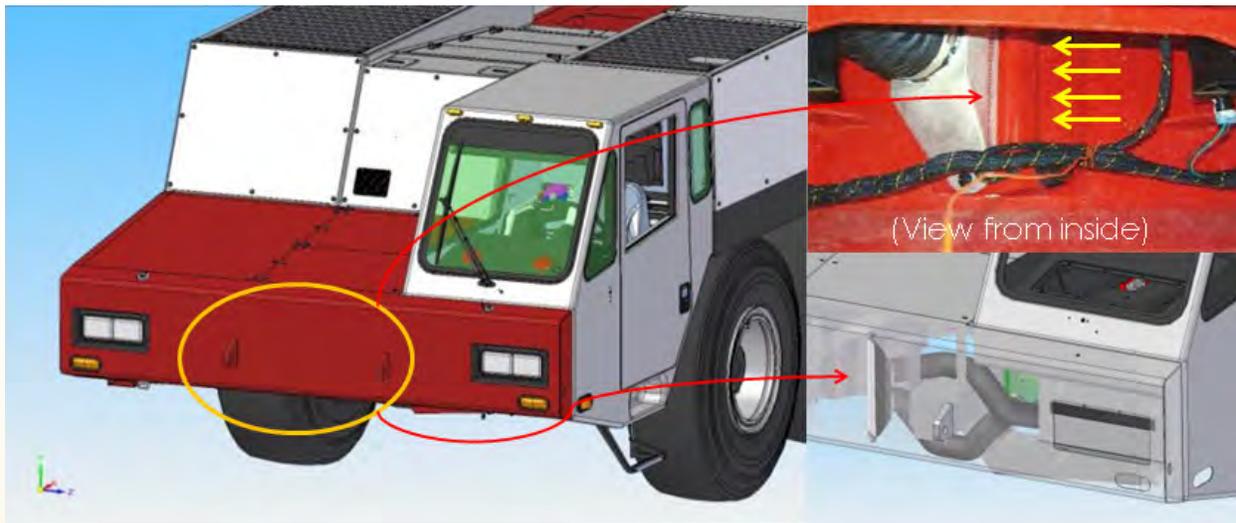


Table 1 summarizes the vent temperature data obtained from the thermocouples under four different operating conditions of the crane: 1) long road run test, 2) stationary test, 3) short road run test, and 4) short road run test (fresh air taped closed and prevented from entering the intake filter and vent). As can be seen from Table 1, the average vent temperature was approximately 135°F during the long road run test. When in stationary mode, the average vent temperature was approximately 47°F which was deemed acceptable. During the short road run test, this temperature rose to nearly 65°F with the air conditioner fully functioning. For the final trial, the intake for outside air was taped closed so that no outside air

entered the system. The road test was then repeated and the vent temperature inside the cab maintained 43°F. This proved: 1) that there is superheated ambient air that is under the hood and behind the front bumper, and 2) that the superheated ambient air was being introduced into the HVAC system causing inadequate cooling on warm/hot days.

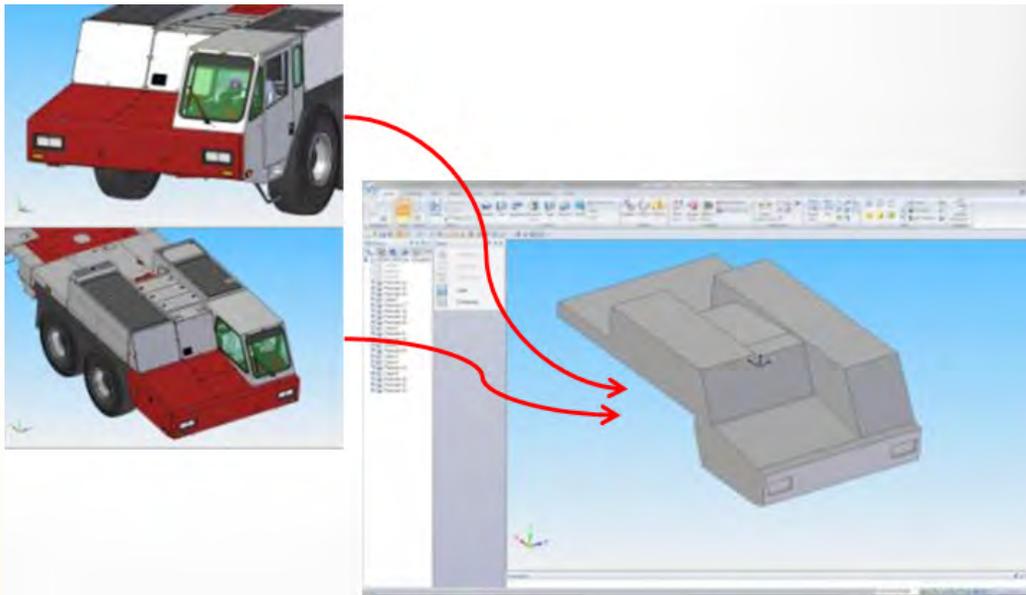
Table 1. Test Data Summary

Crane operating condition	Average Dash Vent Temperature (°F)
Long road run test	135°F
Stationary mode	47°F
Short road run test	65°F
Short road run test (fresh air taped closed and prevented from entering the intake filter and vent)	43°F

3D CAD Modeling

Once the initial testing was completed, a three dimensional (3D) CAD model of the front end of the crane was created in order to perform the CFD analysis. The first and most lengthy part of this task was to assemble a complete 3D model assembly of the complete front end of the carrier portion of the crane. Numerous complex and detailed sub-assemblies were combined to produce this model (See Figure 2).

Figure 2. 3D CAD Model of the Front End of the Crane

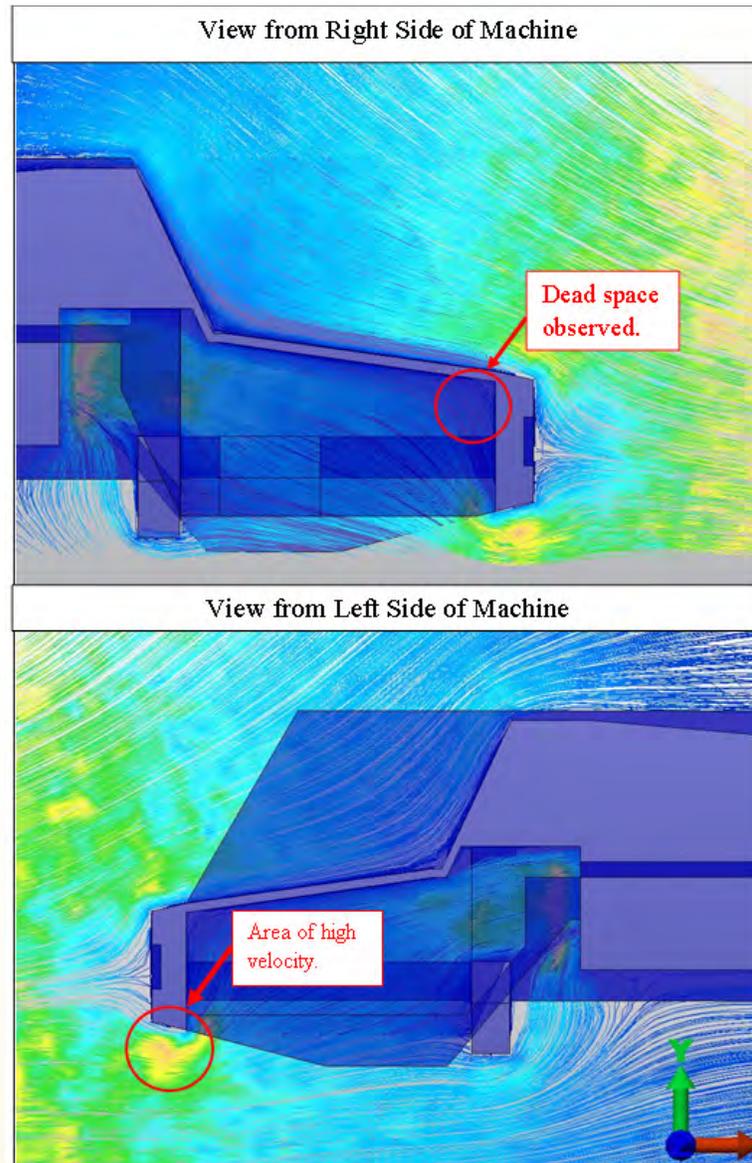


Upon completion of the highly detailed assembly model, a simplified 3D model was required, in order to perform the CFD analysis. Due to the complex geometries of the crane and its sub-assemblies, the full model could not be utilized during the CFD analysis. This would have required intense computational resources and time, which wasn't feasible. Thus, a simplified "box" representation was constructed to represent the complete front end of the crane in the 3D CAD environment. These types of models can often be referred to as Boolean solids or "dumb" solids and are the results of various solid shapes being joined (Madsen et al., 2002). As with any engineering design process, this was an iterative process. The initial Boolean solid was as simple as possible, and during subsequent revisions to the solid, the level of detail was increased to a feasible level, in order to produce more accurate CFD results. During the creation of the 3D model, trial CFD analyses were run to determine if the model was too complex to allow a result to be generated in a feasible amount of time. Thus, there were numerous iterations of this simplified 3D model creation. In the end, a sufficiently complex model was finalized for the final CFD analysis.

CFD Analysis

During the CFD analysis, the simplified 3D model was used to attempt to simulate how the air flows around the front end of the crane while it is travelling on the highway at 55 MPH. Due to extreme complexities in the 3D model creation phase of the project, there were multiple iterations of the CFD analysis. During this process of continually refining the 3D model, each subsequent CFD analysis provided additional insight as to how the air was moving around the front end of the crane. Once the model was deemed sufficient, the final analyses were composed and then studied in order to determine the most suitable placement for the relocated fresh air intake. The CFD analysis was performed using Autodesk Simulation Multiphysics 2012 software, and appropriate renderings were created to display the results of the study (see Figure 3).

Figure 3. Airflow Result Diagrams Showing Existence of Dead Space and Area of High Velocity



After the airflow result diagrams were created, they were analyzed to determine proper design and placement of the new fresh air intake. Design analysis is basically viewed as development and evaluation of a proposed design by utilizing objective thinking (Earle, 2000). While analyzing the results, there were several factors to be kept in mind: serviceability, accessibility, robustness and system simplification. As can be seen in Figure 3 the airflow increases significantly around the periphery of the front bumper. When the air enters under the front hood of the vehicle, it can be seen that there exists a dead space (dark blue area) with little or no airflow. It was this dead space that allowed the stagnant portion of the underhood air to be superheated and ultimately introduced into the HVAC system of the vehicle. The airspace directly under the

front bumper was observed to have a relatively high airflow (yellow colored area) and would ultimately allow ambient air to be introduced into the system as intended.

Component Relocation and Implementation

Following the CFD analysis, four different alternative locations were explored for the fresh air intake. They are listed below:

Location A – In front of cabin and on left side of the bumper.

Location B – On the top surface of the front sheet metal and hood area, centrally located.

Location C – On the front, center surface of the bumper. This location provides ram-air intake which uses the dynamic air pressure created by the vehicle motion.

Location D – On the bottom, sloped face of the bumper, centered with the HVAC components. This location provides some degree of ram air intake.

Multiple markups of existing CAD assemblies for the HVAC system were created, in addition to hand generated sketches. These markups and sketches attempt to convey the different possibilities that are available for the relocation of the air intake. Sketches are often used in place of complete mechanical drawings, especially when quick changes are desired (Spencer & Dygdon, 1968). In addition to sketches and markups, there were 3D CAD layouts generated showing the possible placements of the air intake. A concept evaluation was performed on each of the four potential locations in order to compare them with each other. Table 2 shows the results of the component relocation concept evaluation. There were 10 criteria used and each criterion was evaluated on a scale from -5 to +5, with -5 being the least desirable, and +5 being the most desirable. A score of zero indicated a neutral score and thus did not have a large impact on the scores. The most feasible and low cost solution (suitable location for the air intake) was chosen to implement. As can be seen from Table 2, *Location D* received the highest total score and thus, it was the logical choice for final implementation.

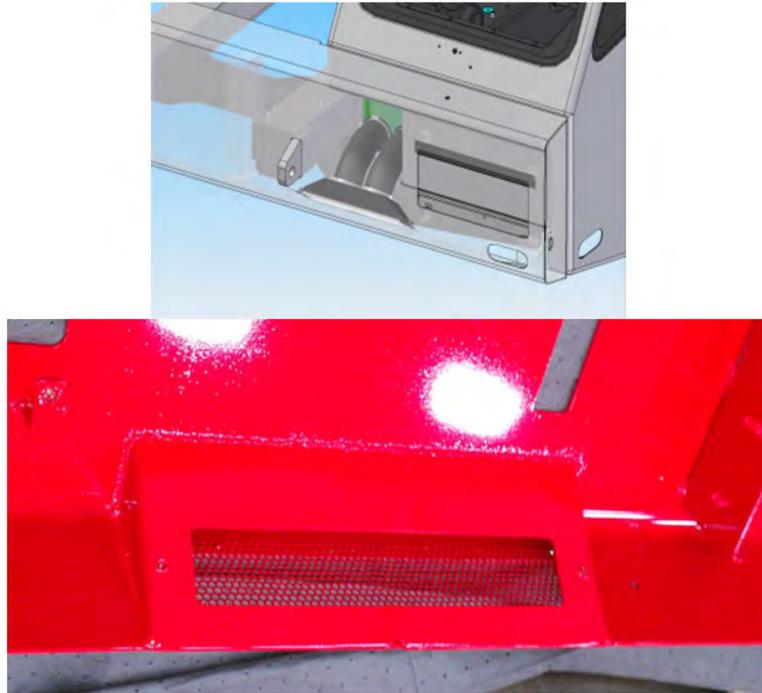
Table 2. Component Relocation Concept Evaluation

Scoring: -5 - Least Desirable, 0 - Neutral, +5 - Most Desirable

Criteria	Location A	Location B	Location C	Location D
Airflow received	4	3	5	5
Components requiring modification	1	-1	4	4
New components required	0	-1	0	0
Intake filter service access	-3	4	4	4
Component interference	-5	0	0	0
Water accumulation/drainage	5	-5	3	4
Safety issues	0	-5	0	0
Machine logo/signage interference	0	0	-5	0
Cost reduction	0	-5	0	0
Weight reduction	0	-5	-4	0
Total Score	2	-15	7	17

After the new location of the air intake was agreed upon, then the task of redesigning the components of the air intake system began. This phase of the project was one of the more tedious tasks due to the fact that the location chosen will impact multiple assembly groups, sub-assemblies, weldments and sub-weldments. During the creation of the new parts for the air intake, they constantly had to be checked and verified for form, fit and function as well as preventing interference with components that are to be located nearby. The entire process involved the following: 1) the location of the air intake, 2) the re-routing of the air intake lines that lead to the heater/evaporator core for the HVAC system, 3) revision to the sub-weldments affected by the relocation, 4) revisions to the higher level assemblies that call for the sub-weldments and piece parts, 5) proper Engineering Order (EO) documentation was generated to allow the changes to be implemented into the production system.

Figure 4 below shows the new design of the air intake system. The intake mounting and debris screen are located on the bottom sloped surface of the front bumper (*Location D*). As the CFD analysis shows, this was the optimal location to allow semi-ram air to be introduced into the system, without drawing superheated air from the inside of the engine compartment.

Figure 4. New Design of the Air Intake System.

Conclusions & Recommendations

The most important conclusion about the design studies conducted in this paper was simply that the fresh air intake could not remain in place where it originally resided, as it was non-functional. According to Earle (2000), the function is the most important characteristic of a design due to the fact that a product that functions improperly is viewed as a failure regardless of other features. Since the intake had to be moved, the only question was: Where does it go? From the CFD analysis, it was observed that if the intake were to draw its fresh air from anywhere around the periphery of the front bumper, that it would make a major improvement. It was seen that there was a slight dead or low flow area where the existing system was drawing the fresh air. By moving to an area of high flow, there was a significantly lesser chance of introducing superheated air into the HVAC system. It was recommended that the air intake vent be relocated as shown in Figure 4 and the new engineering drawings were produced. This location yields several improvements over the initial design, the greatest of which is the introduction of ambient air, and not superheated air from the inside of the engine compartment into the HVAC system. The new location also maintains easy serviceability and accessibility, while maintaining components that are robust enough to withstand the rigors of day to day operation of a mobile truck crane. Subsequent tests have verified that the new intake location allows the HVAC system to function as intended. During these tests it was noted that the system exceeds expectations (the driver had to adjust the air conditioning temperature control to a warmer temperature to remain comfortable).

References

- Anderson, J. J., Degroote, J., Degrez, G., Dick, E., Grundmann, R., and Vierendeels, J. (2009). *Computational Fluid Dynamics: An Introduction* (3rd ed.). (J. F. Wendt, Ed.) Berlin, Germany: Springer-Verlag.
- Earle, J. E. (2000). *Graphics for Engineers* (5th ed.). Upper Saddle River, NJ, Unites States: Prentice Hall.
- Farouki, R. T. (1999). Closing the gap between CAD model and downstream application (report on the SIAM Workshop on Integration of CAD and CFD, UC Davis, April 12-13, 1999), *SIAM NEWS*, 32 (5), 1-3.
- Kellar, W. P., Savill, A.M., and Dawes, W.N. (1999). Integrated CAD/CFD visualisation of a generic formula 1 car front wheel flowfield. *Lecture Notes in Computer Science*, 1593, 90-98.
- Li, H. F., Tu, J.Y., Subic, A., and Abu-Hijleh, B. (2007). The leakage analysis in a twin-screw supercharger by using an integrated CAD/CFD three-dimensional model. *Proceedings of International Conference on Computational Methods*, Singapore, 26 May 2007.
- Madsen, D. A., Folkestad, J., Schertz, K. A., Shumaker, T. M., Stark, C., and Turpin, J. L. (2002). *Engineering Drawing & Design* (3rd ed.). Albany, NY, United States: Delmar.
- Spencer, H. C., and Dygdon, J. T. (1968). *Basic Technical Drawing* (Second Revised ed.). New York, New York: The Macmillan Company.
- Watson, C. (2012). Integrating CFD into Mechanical Design. Retrieved July 15, 2012 from the World Wide Web: <http://www.ien.com/article/integrating-cfd-into/113847>