



THE DEVELOPMENT OF **AIRVIZ**

Jonathan Hollist, MacroAir R & D Engineer



TABLE OF CONTENTS

Introduction.....	2
Timely Flow Simulation.....	3
Testing of the Results.....	3
Simulation Accuracy.....	11
Conclusions.....	13

INTRODUCTION

Since their invention in 1998 the efficacy and efficiency of HVLS fans has been well known to all who experience them. But for someone who has never stood under one of these massive fans and felt the cool air cascade down on them, or stood away from the fan and felt the cooling horizontal jet from head to toe, it is difficult to understand how well the fans work.

Wanting to provide a way for customers, contractors, engineers and architects alike to see and understand the performance of our HVLS fans, we created AirViz™. The flow simulation tool, using computational fluid dynamics to determine the average air speed in the occupied space, provides a visualization of the air flow from our fans within a designated space. Computational fluid dynamics or CFD is a technique used to simulate flow of a fluid.

Contrary to popular usage, a fluid is not just a liquid, it is anything that flows; this includes gases, like air or liquids. CFD takes the space where the flow is to be simulated and divides it into a grid. It then solves the complicated equations that govern fluid flow, known as the Navier-Stokes equations, for each cell in the grid. There are often thousands or millions of cells in a simulation. Depending on the number of cells along with the complicated nature of the equations, a CFD simulation can take hours, or days for a powerful desktop computer to solve.

There are many CFD software packages on the market; the one we use for AirViz is an add-in of SolidWorks called SolidWorks Flow Simulation. With this software package we can model the building, place the fans in appropriate locations and add specific parameters. After the simulation is complete we can post-process the data by getting various images and videos of the air flow, and average air speed in the occupied space.

The various software packages that use CFD are very powerful and fairly accurate but they do have their limits. First and foremost is computational power. Only within the last 10 years have desktop computers been able to handle complicated flow simulations. Even today flows with geometries that move or rotate, like a fan, take days or even weeks to solve. Perhaps this is why no other HVLS manufacturer, to the best of our knowledge, has done this sort of thing before. Also with a CFD solution one cannot expect perfect accuracy. Since approximations are used for things like turbulent flow, simulated velocities at a particular point can only be expected to be within about +/-30 percent of their real value. However, for a larger space, the averaged accuracies reduce down to a much more respectable +/- two percent.

This paper explores what we did to overcome these limitations. We will outline how we came up with a technique that provides for a timely flow simulation. We will show how we tested that technique and show the accuracy of our results.

TIMELY FLOW SIMULATION

When setting up a CFD model in SolidWorks Flow Simulation, as with any other software program, the boundary conditions are vital. The boundary conditions tell the simulation how to react when it gets to the edge of the space that it is simulating. In most cases this is simply a wall that has been modeled in SolidWorks. Sometimes it is an open vent into an external environment that we are not simulating, or a source of air like an air-conditioning duct. It could also be a moving object like a fan blade.

Ideally, a flow simulation for a HVLS fan would include the spinning blades as a boundary condition in a large space or room. This would allow for the simulation to predict, with the best accuracy, the flow in that room. Unfortunately, due to the rotating geometry and the large spaces involved, the simulation could not be solved on our system in a timely manner. Taking this long to solve is not feasible on a large-scale basis, so we decided to define a stationary boundary condition that would mimic the input and output of a rotating HVLS fan.

To do this we simulated the moving blades in a small space and adapted that information to the newly defined boundary conditions. This was a painstaking process of going back and forth between the small moving model and the larger stationary model. A process we called “tuning the fan” or “tuning the simulation” to meet the specifications for each fan that we offer. The details of the boundary conditions that we defined will not be discussed here as they are proprietary to the manufacturer and are only applicable to MacroAir fan models. Running a flow simulation with the new boundary conditions takes only a few hours.

TESTING THE RESULTS

Our original simulated results corresponded nicely with general flow data that we have collected through our many years of experience with the manufacturer. However, this peaked our interest about how precisely our simulated results matched real life. To perform real life comparison tests we turned to the National Orange Show, which has a large indoor pavilion that we placed four of our fans inside. We then modeled the building in SolidWorks with everything that we felt pertinent to the air flow as seen in Figure 1.

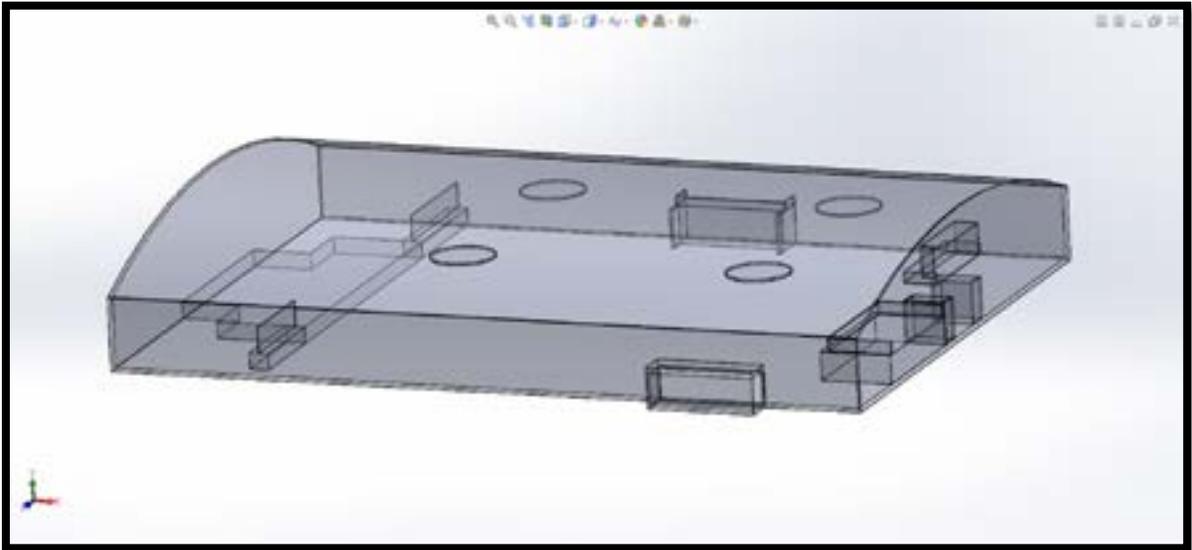


Figure 1: Orange Show Pavilion SolidWorks Model

For ease of measurement, we ran the flow simulation on the building with one single fan. We stored the results to compare with real life data.

To collect the physical data we chose locations along various radial lines, starting at the center of the fan and going as far as 100 feet as seen in Figure 2.

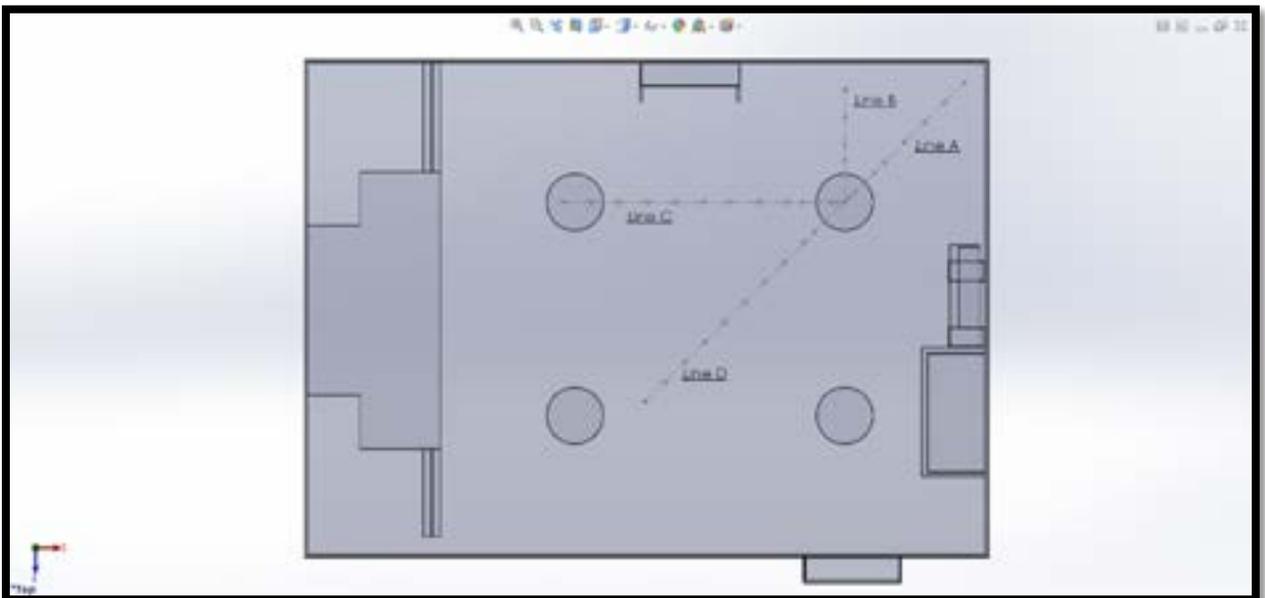


Figure 2: Measurement Positions

Chart 1 below shows the locations measured for each radial line.

Distance (ft)	Line A	Line B	Line C	Line D
0	X		X	
2.5	X		X	
5	X		X	
10	X	X	X	X
15	X	X	X	X
20	X	X	X	X
30	X	X	X	X
40	X	X	X	X
50	X		X	X
60	X		X	X
70			X	X
80			X	X
90			X	X
100			X	X

Chart 1: Locations of Measurements

Four anemometers, like the one shown in Figure 3 below, were mounted on a test stand.



Figure 3: Close-up of Anemometer

The anemometers measured the air speed at four different heights 1.5 feet, 3 feet, 4.5 feet, and 6 feet. With 40 different measurement locations and four different heights, we collected a total of 160 points of data.

Measurements were taken in three steps. First, the test stand was placed at a desired location (as seen in Figure 4 below).



Figure 4: Test Stand and Data Logging Computer

Second, we allowed about one minute for the flow to steady out after our movements. Finally, we recorded the data for all of heights at that location. Data was collected for six minutes with an air speed recorded every second. Figure 5 shows the results of the data points taken for a typical location.

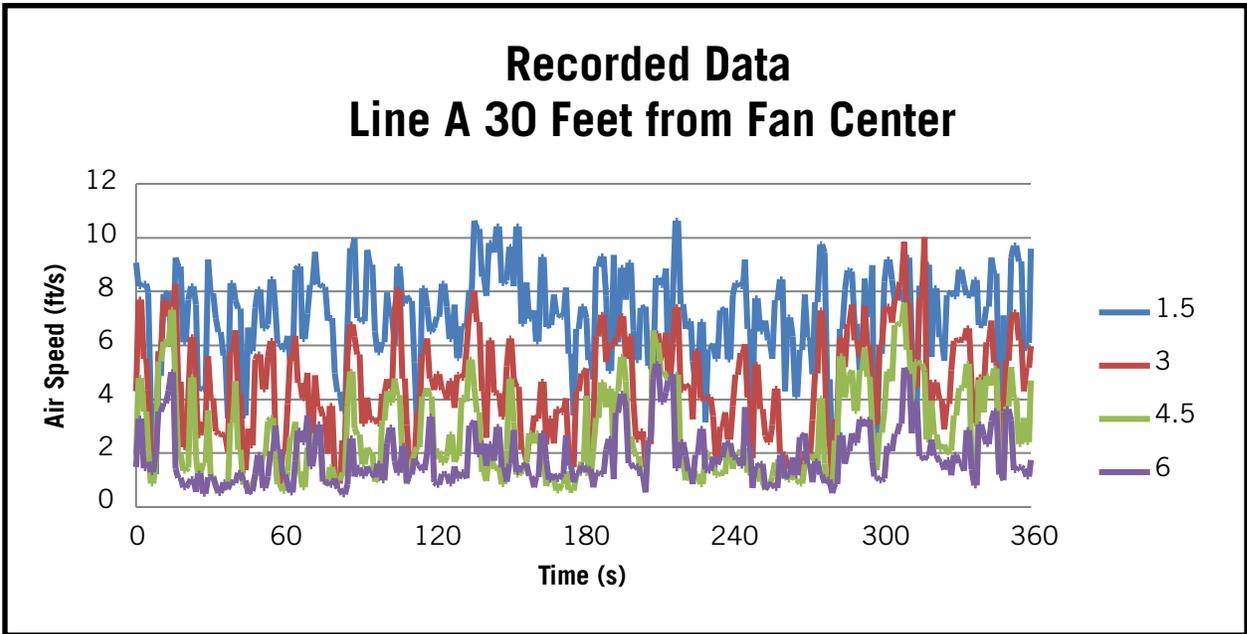


Figure 5: Recorded Data for a Typical Location

As can be seen from the chart there is a large amount of data that fluctuates a great deal. In order to make this data more manageable we chose the average of the six minutes of data along with the standard deviation of that data.

One good way to look at the data is to chart it at a given height for each of the radial lines. Figure 6 shows this chart for line A at 1.5 feet. The center dashed line is the average value at each point and the outside solid lines are the spread of data at plus and minus one standard deviation. Thus 68 percent of the recorded air speed data falls within this “highway.”

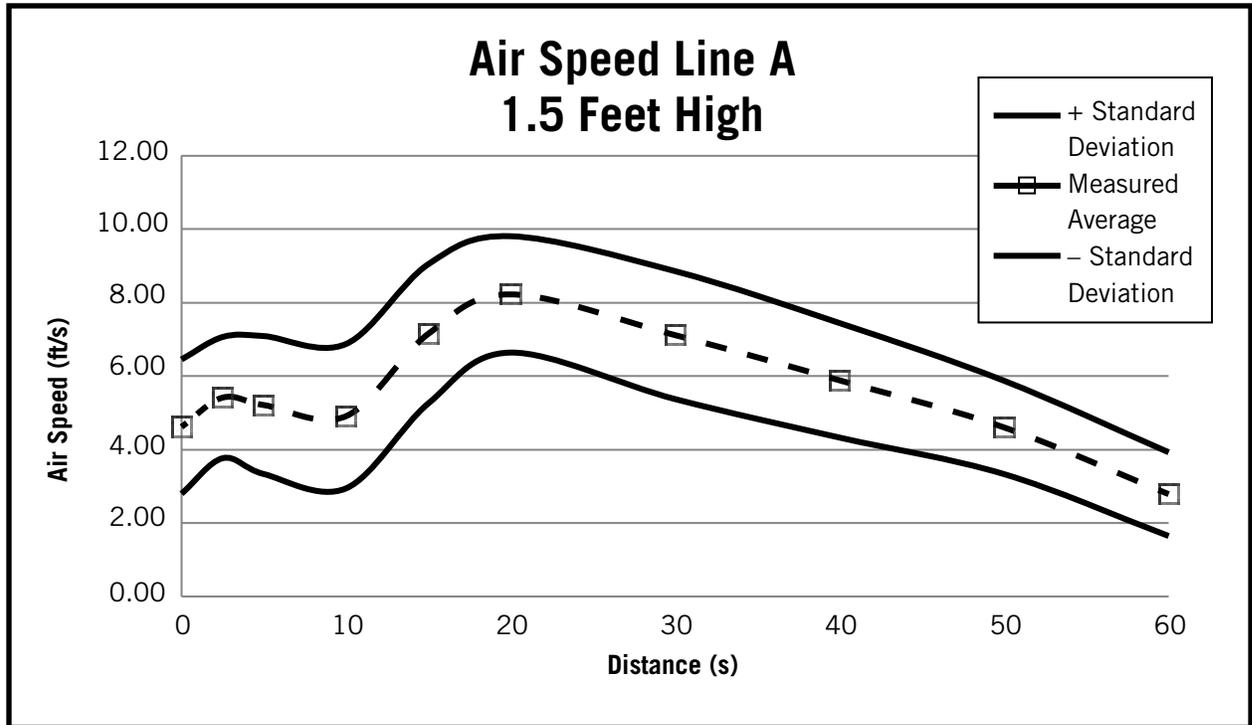


Figure 6: Recorded Air Speed Along a Typical Line and Height

We extracted the air speed from the simulation at each point that we measured to compare data. As seen in Figure 7 the results fall nicely within the range of measured data except for the points that fall directly under the fan.

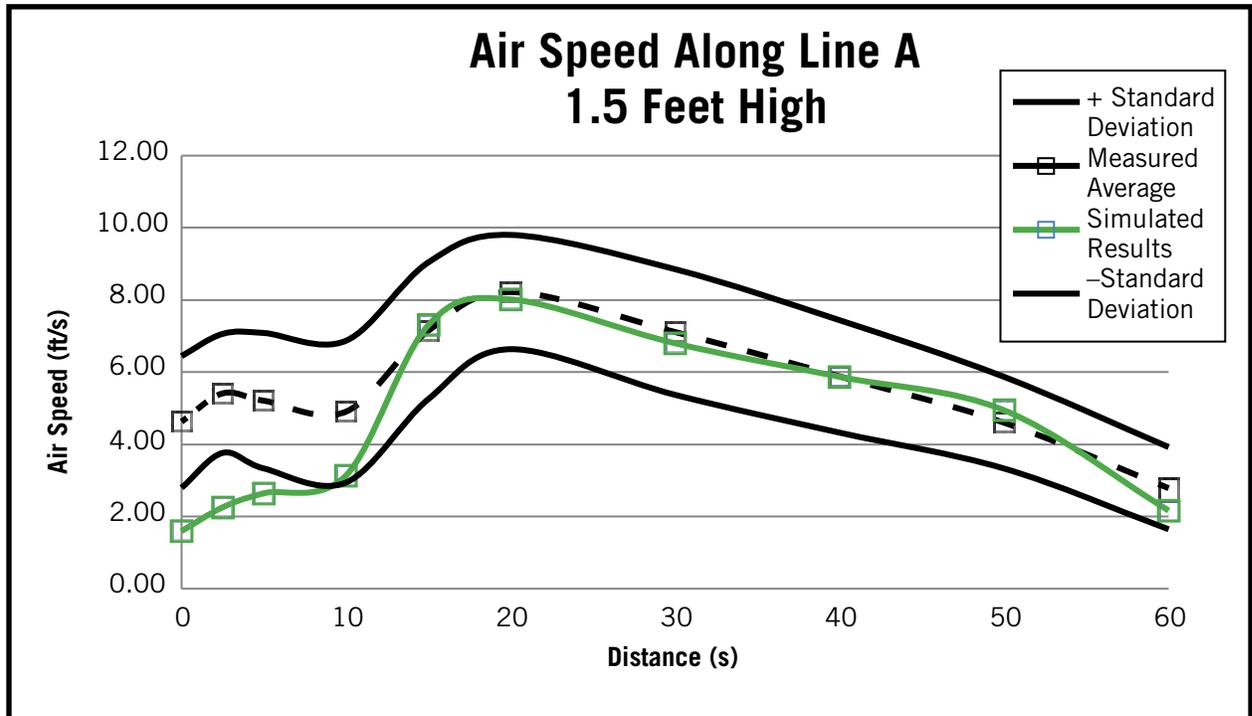


Figure 7: Recorded Air Speed with Simulated Air Speed Results

The results of each radial line and height are not presented here. The results in Figure 7 can be regarded as typical. Some of the points fit more closely in line with the measured average and some not so closely. A few points fall outside the highway of plus or minus one standard deviation. These points occur nearly or directly under the fan, where turbulent swirling flow is common. As pointed out in the introduction, due to the nature of the flow simulation, this was where we expected to lose some accuracy.

Additionally, the anemometer in Figure 3 (page 6) has a difficult time handling flow directions greater than 60 degrees from center, which would occur under the fan, where the flow is generally vertically down, as opposed the horizontal orientation of the sensor.

SIMULATION ACCURACY

It is a temptation to look at accuracy of the simulation as a simple comparison at individual points; determining the percent difference between the simulation and the mean measured value. At some points the percent difference can be as high as 100 percent, or double the measured value. This may cause the observer to prematurely determine that the results of the simulation are inaccurate. This would be a mistake since the values measured by the anemometer alone can fluctuate up to 75 percent. Rather than get a distorted view at a single point we looked at the average air speed from a number of points. Chart 2 shows the measured average air speed, combined standard deviation, and percent differences for the combined points at each height. It also contains the combined values of all points.

Height (ft)	Averaged Measured Air Speed (ft/s)	Combined Standard Deviation (ft/s)	Averaged Simulated Air Speed (ft/s)	Difference of Standard Deviation (%)	Difference of Air Speed (%)
1.5	5.11	1.33	4.40	26.12%	13.92%
3	3.55	1.32	3.64	37.20%	2.43%
4.5	2.23	1.11	2.89	49.85%	29.49%
6	1.94	1.00	2.28	51.61%	17.55%
All Points	3.21	1.20	3.30	37.42%	2.90%

Chart 2: Combined Results

The combined results show a very good agreement between the measured data and the simulated data. The differences in air speed range from 2.34 percent to 29.49 percent. All simulated data falls within the range of the combined standard deviation of the measured values. This reinforces that the AirViz simulation will provide an accurate image of the general flow and give a good estimate for the average air speed in the occupied space.

Another way to look at the accuracy of the results is to determine if the differences between the measured and simulated results could be felt by someone standing under the fan. Figure 8 is a chart taken from ASHRAE Standard 55-2010.

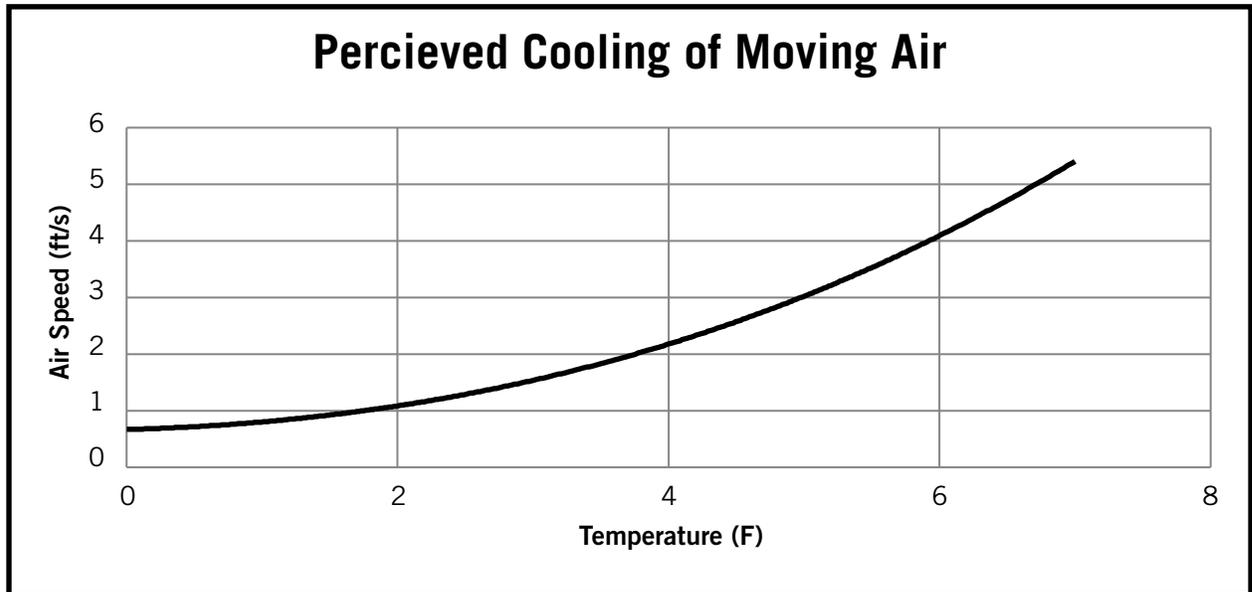


Figure 8: Percieved Cooling and Air Speed from ASHRAE 55-2010

Figure 8 shows the percieved cooling that comes from air movement. Matching each air speed in Chart 2 (page 4) with the cooling temperatures provides us with a temperature comparison for each height and all combined points, as shown below in Chart 3.

Height (ft)	Temp. From Measured Values (F)	Temp. From Simulated Values (F)	Temp. Difference (F)
1.5	6.76	6.28	0.48
3	5.60	5.68	0.08
4.5	4.12	4.94	0.82
6	3.67	4.18	0.51
All Points	5.28	5.37	0.09

Chart 3: Cooling Temperature Comparison

The temperature difference is 0.82°F at the 4.5 feet level. A difference of less than one degree can hardly be perceived by a person much less 0.09 degrees, which is found in the combination of all points. Thus the differences that exist between the measured values and the simulated values are very good when considering how they will be used.

CONCLUSIONS

Calculations using CFD are effective, but have limits. It requires a large amount of computer computation power and can be inaccurate in turbulent flows and at particular points. In this paper we have shown how these obstacles were overcome by establishing a stationary boundary condition that mimics the movement of our MacroAir fans. We also performed physical tests in a large enclosed area and compared them to simulated results.

While single point comparisons may not be ideal, overall averages of many points show great accuracy, all falling within the range of the measured values. The simulated average of all points gives the best results being only 2.9 percent different from the measured value. The perceived cooling from both the measured and simulated value would be the same. This gives us assurance that when we are providing a customer with an AirViz 3D video, image, or average air speed, we are providing an accurate service. This allows our customers to confidently see the airflow before they feel it.

AirViz gives customers an opportunity to try our MacroAir fans before they are installed. Through visualizing the potential performance and cost-saving prospects of our HVLS fans, companies can spec and test fan locations for optimum airflow, ultimately giving them confidence that our fans are the best air movement solution for their application.

About EnergyLogic's HVLS Fans by MacroAir

Since developing the first HVLS prototype in 1998, MacroAir continues to serve as the 'Engineers of Air™' through a commitment to innovation and design of the most durable and cost-effective commercial ceiling fans on the market. Our HVLS fans can be found in commercial, agricultural and retail establishments around the world. The AirVolution fan models are "UL Listed," a certification by Underwriter Laboratories, Inc., an independent product safety testing organization, verifying the quality of MacroAir products.

EnergyLogic's HVLS fans by MacroAir are trusted by companies such as Coca-Cola, CSX Transportation and NAPA Auto Parts, and the company is the approved HVLS fan supplier for automotive brands such as BMW, Chrysler, Ford, General Motors, Lexus, Mercedes-Benz, Nissan and Toyota under their Dealer Equipment Programs.

Mailing Address

5901 Crossings Blvd.
Antioch, TN 37013

Connect with Us

www.energylogic.com
Twitter: @energylogicllc
Facebook: facebook.com/EnergyLogic/
LinkedIn: linkedin.com/company/energylogic