T.C.

MARMARA UNIVERSITY

MECHANICAL ENGINEERING DEPARTMENT

# INVESTIGATION OF BLADELESS FAN FLOW STRUCTURES

**SERCAN ALTIN**

**SUPERVISOR**

**Asst. Prof. Mustafa YILMAZ**

**İSTANBUL 2014**

## CONTENTS

## 1. ABSTRACT

My study is based on Dyson Air Multiplier (hereafter fan). I have identified that the main component which allows fan to work is the unique design of its airfoil. At first I decided to determine the experimental design and results of a near-surface analysis of a two-dimensional cross section of the airfoil. In order to mold the airfoil within a foam exoskeleton, I used fiberglass. A null displacement manometer was used to measure the air flow velocities at the boundary layer and a high-speed camera was used to observe the flow.

A wind tunnel is used in aerodynamic research in order to study the effects of air moving past solid objects. Air is blown through a duct equipped with a viewing port and instrumentation where models or geometrical shapes are mounted for study. By using a series of fans, air is moved through the tunnel. For larger wind tunnels several meters in diameter, a single large fan is not practical, and so instead multiple fans are used in parallel to provide sufficient air flow, such as in the 30-by 60-Foot Tunnel at NASA's Langley Research Center.

FIG. 1.— Dyson Air-Multiplier AM01(6)

Due to the fan blade’s motion, the air flow created by the fans entering the tunnel is itself highly turbulent. The air moving through the tunnel needs to be relatively turbulence-free and laminar [1].Closely spaced vertical and horizontal air vanes are used to solve this problem via smoothing out the turbulent air flow before reaching the subject of the testing thus increasing the overall length of the tunnel. This design is less than ideal for a wind tunnel but it is still the prevalent design.

Dyson Ltd claims that the Dyson Air Multiplier (Figure 1), creates laminar airflow with no buffeting or uneven airflow, a characteristic that a wind tunnels settling chamber wants to reduce. Designers of wind tunnel are quite interested in this elimination of buffeting and the creation of steady laminar flow (Figure 2). Dyson Ltd’s results from laser Doppler anemometry indicate that there are two regions close to the source of flow that offer steady laminar flow [2]. There are two reagions of opportunity to test a target: one is right in the middle of the source and the other region lies 20 inches downstream. There is no publicly available record of this study. Because of the time restrictions with the project I was only be able to look at the near surface effects of the airfoil.

I have found a method to test the feasibility of this design for a wind tunnel by analyzing the laminar area around the source. The process of building and constructing a full-scale version of a Dyson Air Multiplier inspired wind tunnel without understanding the under-lying processes is not cost effective. I believe that the unique airfoil design is the secret to the Dyson Air Multiplier. In order to evaluate the other characteristics of the Dyson Air Multiplier to produce an ideal wind tunnel, it might be needed to perform the further experiments. In here I find out a method for the construction and testing of my airfoil.

I used a two-dimensional cross section of their circular design for my implementation. To construct an airfoil with a constant chord length made out of fiberglass, the cross section was extruded. The air was supplied from a compressed air source. Null displacement manometer is used in testing for steady state flow. The manometer was used to measure air flow properties around the boundary layer, after a steady state flow was detected. The manometer probe has a small diameter thus while it may be able to measure velocities along the boundary layer, but it does not have resolution to detect the boundary layer close to the slit.

## 1.1.WHAT IS A MECHANICAL FAN?

The mechanical fan is a machine that consists of a blade assembly powered by an electrical motor, however, this can also be a handheld device that is simply moved back and forth to move the air. Manufactured in a variety of sizes and styles, the mechanical fan can be run on household current or battery power to move hot and cold air. From the propeller-like, axial-style fan found in box fans and ceiling fans to the centrifugal fan, the mechanical fan is manufactured in many styles. Each type provides a particular advantage when installed in a specific application.[12]

## 1.2.FAN TYPES

There are three main types of fans used for moving air: axial, centrifugal (also called radial) and cross flow (also called tangential).

The axial-flow fans have blades that force air to move parallel to the shaft about which the blades rotate. Axial fans blow air across the axis of the fan, linearly, hence their name. This is the most commonly used type of fan, and is used in a wide variety of applications, ranging from small cooling fans for electronics to the giant fans used in wind tunnels. This style of fan uses a blade or a system of blades to move air at right angles or parallel to the shaft that the blade assembly is mounted to. In a typical home-use box fan, a five-bladed fan is turned by a small, electrical motor to move air throughout a room. Commonly equipped with three speeds, this type of mechanical fan is often used in warm climates to maintain a comfortable climate in the home. This is also the type of fan that is used in many different types of electrical devices to provide cooling to the components or to discharge built-up heat from the device.[12]

The centrifugal fan has a moving component (called an impeller) that consists of a central shaft about which a set of blades form a spiral pattern. Centrifugal fans blow air at right angles to the intake of the fan, and spin (centrifugally) the air outwards to the outlet. The centrifugal fan is commonly called a squirrel cage. It moves air by spinning an impeller, a circular, drum-like device consisting of raised slats, through the use of an electric motor and a rubber belt. Used in furnaces, car heaters and many air-conditioning systems, this type of mechanical fan is capable of moving large volumes of air, often at slow speeds. Similar to the centrifugal fan is the cross-flow fan, this mechanical fan is used primarily in heating and cooling applications and also uses an impeller to move the air.[12]

An impeller rotates, causing air to enter the fan near the shaft and move perpendicularly from the shaft to the opening in the scroll-shaped fan casing. A centrifugal fan produces more pressure for a given air volume, and is used where this is desirable such as in leaf blowers, air mattress inflators, and various industrial purposes. They are typically noisier than comparable axial fans. [16]

The cross flow fan has a squirrel cage rotor (a rotor with a hollow center and axial fan blades along the periphery). Tangential fans take in air along the periphery of the rotor, and expel it through the outlet in a similar fashion to the centrifugal fan. Cross flow fans give off an even airflow along the entire width of the fan, and are very quiet in operation. They are comparatively bulky, and the air pressure is low. Cross flow fans are often used in air conditioners, automobile ventilation systems, and for cooling in medium-sized equipment such as photocopiers. The action of a fan or blower causes pressures slightly above atmospheric, which are called "plenums."[13]

Fans typically go along with electric motors. An electric motor's poor low speed torque and powerful high speed torque are a natural match for a fan's load. Fans are often attached directly to the motor's output, with no need for gears or belts. The electric motor is either hidden in the fan's center hub or extends behind it. For big industrial fans, 3-phase asynchronous motors are commonly used. Smaller fans are often powered by shaded pole AC motors, or brushed or brushless DC motors. AC-powered fans usually use mains voltage, while DC-powered fans use low voltage, typically 24 V, 12 V or 5 V. Cooling fans for computer equipment exclusively use brushless DC motors, which produce much less electromagnetic interference. [13]

## 2. INTRODUCTION

The Dyson Air Multiplier was announced on 18 October 2009 as an electric fan, intended to provide smoother airflow and, having no exposed rotating blades, operating in a safer manner than conventional bladed fans. The fan works by drawing 27 litres of air per second in through an inlet in the base pillar and forcing it through an outlet in the upper ring. The jet of air travels over the aerofoil shape of the ring, creating local low pressure, thereby pulling air from behind it as it decelerates in a process known as inducement, a property of Bernoulli's principle. Once the air exits the ring it entrains the air in front and alongside, producing an airflow of 405 l/second. Using this process, a small brushless impeller in the fan's base can power a much larger air outlet without exposing any blades. [17]

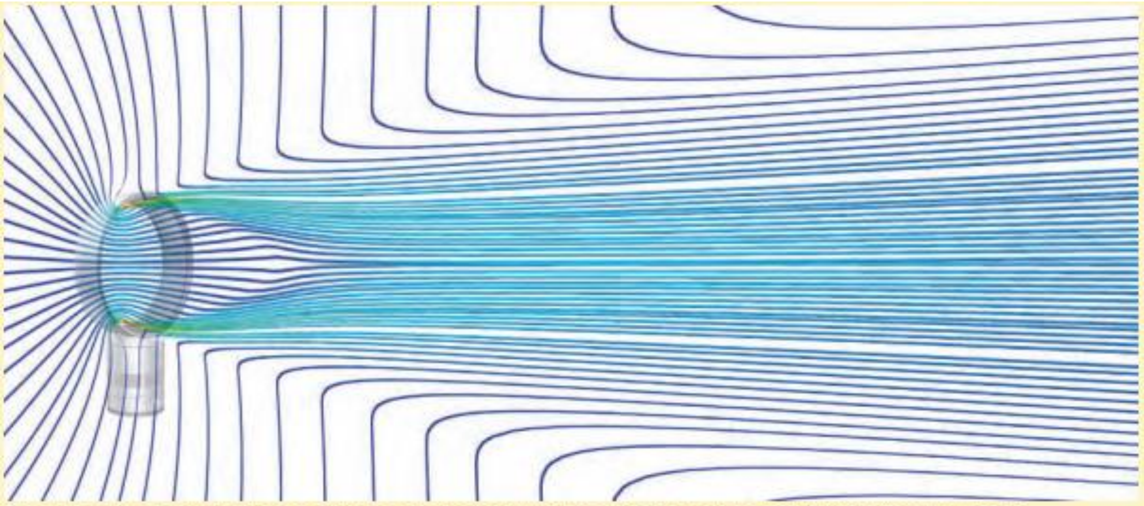
It’s immediately obvious that the Dyson Air Multiplier is no ordinary fan. There are no blades, and it looks more like modern art than an appliance that cools a room. The lack of blades is Dyson’s groundbreaking innovation. The Air Multiplier uses loop amplifiers to increase airflow, similar to the science behind turbochargers and jet engines. It’s engineered to draw in a small amount of air at the base and accelerate the air flow as it circulates around the aerodynamic collar (it can amplify airflow 15- to 18-fold, depending on the model). As a result, the Air Multiplier produces a smooth stream of air, like a gentle breeze, rather than roughly chopping up the air like traditional bladed fans do. A dimmer switch provides precise airflow control, and the pivoting base makes it easy to control the direction of the air. Plus, with no blades or a grille, Dyson’s fans are safe for people and pets and very easy to clean.

The blending of science and design in the Air Multiplier is seamless. It’s what we’ve come to expect from James Dyson, the maverick designer and engineer who devised “cyclone technology” to create vacuums that don’t need messy bags and won’t lose suction. Dyson’s vacuums took the appliance industry by storm, and his bladeless fans are set to do the same, one cool breeze at a time. [18]

## 2.1.ANSYS AND BLADELESS

### 2.1.1. Fluid Flow Simulation Advances Acclaimed New Fan Design.

Released to wide international recognition in 2009 including being named to Time magazine's list of best gadgets — the Dyson Air Multiplier™ fan is a both technical and stylistic re-imagining of the household fan. By accelerating air over a ramp, the design eliminates fan blades as well as the buffeting and turbulence associated with these household appliances. From the outset, however, Dyson engineers faced the challenge of developing and optimizing the design of an original new fan without the benefit of previous experience with this type of design. Historically, the company relied on physical prototyping for design development, but resulting cost and time constraints limit the ability to evaluate hundreds of design candidates needed to optimize a new product. To complement experimental testing and reduce development time for this new fan, Dyson's engineers used fluid dynamics software from ANSYS to evaluate up to 10 different designs per day. The idea for the Dyson Air Multiplier fan originated when the engineering team was testing the Dyson Airblade™ hand dryer. This drying device works by generating a thin sheet of air moving at 400 mph that pushes water off the user's hands. Observing the side effect that the sheet of air was dragging a considerable portion of the surrounding air with it, the team conceived a new idea: to produce a thin, high-speed sheet of air that drags surrounding air a process known as inducement through a fan. Airflow leaving the product then drags along more flow a process known as entrainment and the Dyson Air Multiplier fan was the result. This unique approach eliminated the need for the external blades of a conventional fan and provided a much smoother movement of air that feels like a natural breeze. For the new, bladeless fan, Dyson engineers developed a basic design concept in which air is drawn into the base of the unit by an impeller, accelerated through an annular aperture and then passed over an airfoil-shaped ramp that channels its direction. The initial design had an amplification ratio how much air is dragged along for each unit of primary flow of six to one, which needed to be improved substantially for the finished product. The conventional approach to evaluating designs was to use rapid physical prototyping for the annular ring. However, with each ring taking several days to build, combined with additional time for measurement and hand finishing and, finally, several days to assemble and test the ring, a total of two weeks would have been required to evaluate each prospective design. Dyson faced similar fluid flow design challenges in previous development projects, especially with its lines of vacuum cleaners and hand dryers. As before, the company's engineers overcame such problems by using ANSYS FLUENT


Air is initially drawn in through the base of the fan and is injected through the annular slot, inducing more flow from the surrounding air.

software to simulate fluid flow without the need for a physical prototype. Being able to visualize fluid flow throughout the solution domain helped engineers to gain an intuitive understanding of the design, leading to rapid improvements. The software's ability to divide the domain into subdomains substantially speeded up the process of making design changes. For example, the sub domains in and around the annular ring contained a very dense mesh to maximize accuracy in this critical area. After making a change, the team had to remesh only the subdomain that contained the change, and thus the time for remeshing was reduced from over an hour to about 10 minutes.

Dyson engineers first modeled the initial prototype with the goal of validating the accuracy of the fluid flow model. Each case was simulated as a 2-D, steady-state, incompressible, turbulent air flow using the k-epsilon turbulence model. The attractiveness of the 2-D method
was the meshing simplicity and relatively short solution time, but the downside was a more simplistic flow field. However, the software from ANSYS was extremely consistent in predicting performance trends that were observed, which gave the engineering team confidence in the simulation results.The next step was to evaluate a series of design iterations with the primary goal of increasing the amplification ratio to move the maximum amount of air possible for a given size and power consumption. Dyson's engineers quickly homed in on three dimensions as having major impact on performance: the gap in the annular ring, the internal profile of the ring and the profile of the external ramp. The team was able to design and model up to 10 geometric variations of these dimensions in a single day and then to compute the results in a batch overnight.

Another major benefit of using fluid dynamics simulation was that the engineers were able to establish relationships between air velocity and delivered flow rate for various designs a key performance metric.Over the course of the design process, Dyson's engineers steadily improved the performance of the fan to the point that the final design has an amplification ratio of 15 to one, a 2.5-fold improvement over the six-to-one ratio of the original concept design. The team investigated 200 different design iterations using simulation, which was 10 times the number that would have been possible had physical prototyping been the primary design tool. Physical testing was used to validate the final design, and the results correlated well with the simulation analysis.With critical acclaim from many reviewers, the Dyson Air Multiplier fan has been a resounding success in the marketplace. By optimizing the design's performance and reducing the number of prototypes, simulation software from ANSYS made a notable contribution to that success.[14]

### 2.1.2. ANSYS Meshing [24]

Mesh generation is one of the most critical aspects of engineering simulation. Too many cells may result in long solver runs, and too few may lead to inaccurate results. ANSYS Meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. ANSYS Meshing technology has been built on the strengths of stand-alone, class-leading meshing tools. The strongest aspects of these separate tools have been brought together in a single environment to produce some of the most powerful meshing available.

The highly automated meshing environment makes it simple to generate the following mesh types:

- Tetrahedral
- Hexahedral
- Prismatic inflation layer
- Hexahedral inflation layer
- Hexahedral core
- Body fitted Cartesian
- Cut cell Cartesian

Consistent user controls make switching methods very straight forward and multiple methods can be used within the same model. Mesh connectivity is maintained automatically.

Different physics requires different meshing approaches. Fluid dynamicssimulations require very high-quality meshes in both element shape and smoothness of sizes changes. Structural mechanics simulations need to use the mesh efficiently as run times can be impaired with high element counts. ANSYS Meshing has a physics preference setting ensuring the right mesh for each simulation. [24]

In our case, the meshing size is applied as 1 000 000 triangular mesh.

### 2.1.3. ANSYS Meshing Features [23]

The meshing tools in the ANSYS Workbench platform were designed with the following guiding principles:

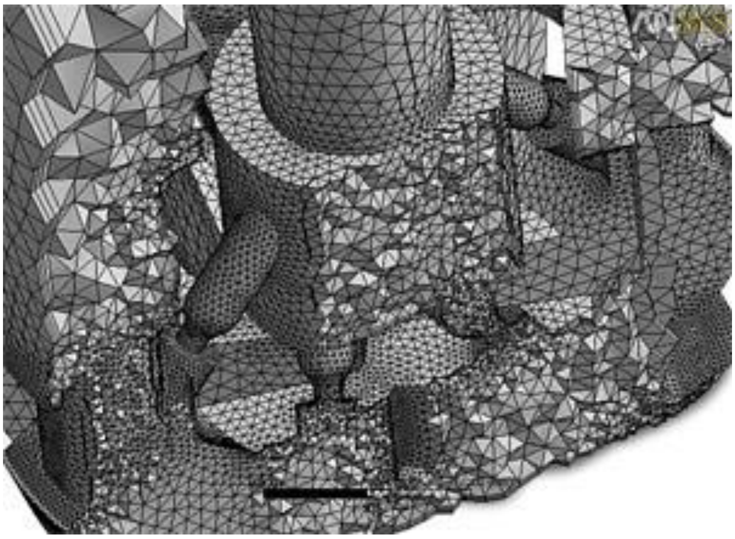
- Parametric: Parameters drive the system.
- Persistent: Model updates are passed through the system.
- Highly-automated: Baseline simulation can be performed with limited input.
- Flexible: Ability to add additional control without complicating the workflow
- Physics aware: Key off of physics to automate modeling and simulation throughout system
- Adaptive architecture: Open system that can be tied to a specific process: CAD neutral, meshing neutral, solver neutral, etc.

By integrating best-in-class meshing technology into a simulation-driven workflow, ANSYS Meshing provides a next-generation meshing solution.

### 2.1.4. Triangular (Tetrahedral) Meshing

The combination of robust and automated surface, inflation and tet meshing using default physics controls to ensure a high-quality mesh suitable for the defined simulation allows for push-button meshing. Local control for sizing, matching, mapping, virtual topology, pinch and other controls provides additional flexibility, if needed.

Automated CFD meshing includes inflation layers for complicated geometries, such as this drill bit model.

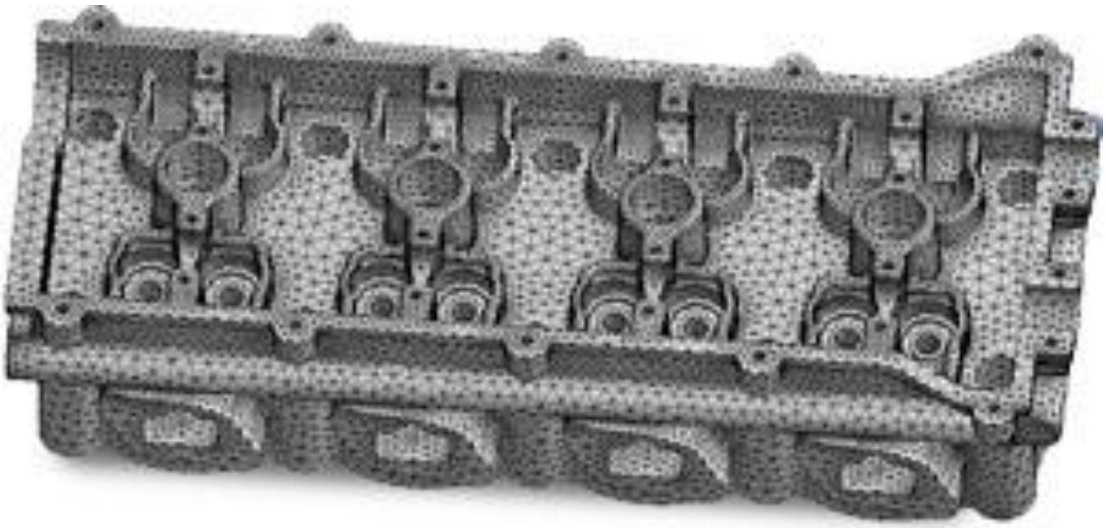


Automated structural meshing with well-shaped quadratic tet elements can be used for complicated geometries, such as this engine head.

### 2.1.5. Mesh Methods:

#### 2.1.5.1.Patch-Conforming Mesh Method

- This method uses a bottom-up approach (creates surface mesh then volume mesh).



- Multiple triangular surface meshing algorithms are employed behind the scenes to ensure a high quality surface mesh is generated the first time.
- From the surface mesh, inflation layers can be grown using several techniques.

- The remaining volume is meshed with a Delaunay advancing front approach that combines the speed of a elaunay approach with the smooth-transitioned mesh of an advancing front approach.
- Throughout this meshing process are advanced size functions that maintain control over the refinement, moothness and quality of the mesh.

#### 2.1.5.2.Patch-İndependent Mesh Method

- This method uses a top-down approach (creates volume mesh and extracts surface mesh from boundaries).
- Many common problems with meshing occur from bad geometry. If bad geometry is used as the basis to create the surface mesh, the mesh will often be undesireable (bad quality, connectivity, etc.).
- The patch-independent method uses the geometry only to associate the boundary faces of the mesh to the regions of interest thereby ignoring gaps, overlaps and other issues that present other meshing tools with countless problems.
- Inflation is done as a post step into the volume mesh. Since the volume mesh already exists, collisions and other common problems for inflation are known beforehand.

**Note:** For volume meshing, a tetrahedral mesh generally provides a more automatic solution with the ability to add mesh controls to improve the accuracy in critical regions. Conversely, a hexahedral mesh generally provides a more accurate solution but is more difficult to generate. [23]

## 2.2. ANSYS TURBULENCE MODELS

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number.

In principle, the Navier-Stokes equations describe both laminar and turbulent flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales, and would generally involve length scales much smaller than the smallest finite volume mesh, which can be practically used in a numerical analysis. The Direct Numerical Simulation (DNS) of these flows would require computing power which is many orders of magnitude higher than available in the foreseeable future.

To enable the effects of turbulence to be predicted, a large amount of CFD research has concentrated on methods which make use ofturbulence models. Turbulence models have been specifically developed to account for the effects of turbulence without recourse to a prohibitively fine mesh and direct numerical simulation.

### 2.2.1. Statistical Turbulence Models and The Closure Problem

When looking at time scales much larger than the time scales of turbulent fluctuations, turbulent flow could be said to exhibit average characteristics, with an additional time-varying, fluctuating component. For example, a velocity component may be divided into an average component, and a time varying component.

In general, turbulence models seek to modify the original unsteady Navier-Stokes equations by the introduction of averaged and fluctuating quantities to produce the Reynolds Averaged Navier-Stokes (RANS) equations. These equations represent the mean flow quantities only, while modeling turbulence effects without a need for the resolution of the turbulent fluctuations. All scales of the turbulence field are being modeled. Turbulence models based on the RANS equations are known as Statistical Turbulence Models due to the statistical averaging procedure employed to obtain the equations.

Simulation of the RANS equations greatly reduces the computational effort compared to a Direct Numerical Simulation and is generally adopted for practical engineering calculations. However, the averaging procedure introduces additional unknown terms containing products of the fluctuating quantities, which act like additional stresses in the fluid. These terms, called ‘turbulent’ or ‘Reynolds’ stresses, are difficult to determine directly and so become further unknowns.

The Reynolds (turbulent) stresses need to be modeled by additional equations of known quantities in order to achieve “closure.” Closure implies that there is a sufficient number of equations for all the unknowns, including the Reynolds-Stress tensor resulting from the averaging procedure. The equations used to close the system define the type of turbulence model.

### 2.2.2. Spalart-Allmaras Model

The Spalart-Allmaras model is a one-equation model that solves a modeled transportequation for the kinematic eddy (turbulent) viscosity. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity in turbomachinery applications.

In its original form, the Spalart-Allmaras model is effectively a low-Reynolds number model, requiring the viscosity-affected region of the boundary layer to be properly resolved. In ANSYS FLUENT, the Spalart-Allmaras model has been extended with a $y^+$ -insensitive wall treatment (Enhanced Wall Treatment), which allows the application of the model independent of the near wall$y^+$ resolution.

The formulation blends automatically from a viscous sublayer formulation to a logarithmic formulation based on $y^+$. On intermediate grids, 1< $y^+$ < 30 , the formulation maintains its integrity and provides consistent wall shear stress and heat transfer coefficients. While the $y^+$sensitivity is removed, it still should be ensured that the boundary layer is resolved with a minimum resolution of 10-15 cells.

The Spalart-Allmaras model was developed for aerodynamic flows. It is not calibrated for general industrial flows, and does produce relatively larger errors for some free shear flows, especially plane and round jet flows. In addition, it cannot be relied on to predict the decay of homogeneous, isotropic turbulence.

### 2.2.3. Standard k- ε Model

Two-equation turbulence models allow the determination of both, a turbulent length and time scaleby solving two separate transport equations. The standard k- ε model in ANSYS FLUENT falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [199] (p. 760). Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

The standard k- ε model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart.

In the derivation of the k- ε model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k- ε model is therefore valid only for fully turbulent flows.

As the strengths and weaknesses of the standard k- ε model have become known, modifications have been introduced to improve its performance. Two of these variants are available in ANSYS FLUENT: the RNG k- ε model and the realizable model.

### 2.2.4. Standard K-ω Model

The standard k-ω model in ANSYS FLUENT is based on the Wilcox k-ω model, which incorporates modifications for low-Reynolds number effects, compressibility, and shear flow spreading. One of the weak points of the Wilcox model is the sensitivity of the solutions to values for k and ω outside the shear layer (free stream sensitivity).While the new formulation implemented in ANSYS FLUENT has reduced this dependency, it can still have a significant effect on the solution, especially for free shear flows

The standard k-ω model is an empirical model based on model transport equations for the turbulence kinetic energy (k) and the specific dissipation rate (ω), which can also be thought of as the ratio of ε to k.

As the k-ω model has been modified over the years, production terms have been added to both the k and ω equations, which have improved the accuracy of the model for predicting free shear flows.

### 2.2.5. Transition SST Model

The transition SST model is based on the coupling of the SST k− ω transport equations with two other transport equations, one for the intermittency and one for the transition onset criteria, in terms of momentum-thickness Reynolds number. An ANSYS empirical correlation (Langtry and Menter) has been developed to cover standard bypass transition as well as flows in low free-stream turbulence environments.

In addition, a very powerful option has been included to allow you to enter your own user-defined empirical correlation, which can then be used to control the transition onset momentum thickness Reynolds number equation. To learn how to set up the transition SST model, see Setting Up the Transition SST Model.

## 3. GOVERNING EQUATIONS [22]

The set of equations solved by ANSYS CFX are the unsteady Navier-Stokes equations in their conservation form. Let's define the instantaneous equation of mass, momentum, and energy conservation. For turbulent flows, the instantaneous equations are averaged leading to additional terms.

**The Continuity Equation**

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \boldsymbol{U}) = 0$$

**The Momentum Equations**

$$\frac{\partial (\rho \boldsymbol{U})}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} \otimes \boldsymbol{U}) = -\nabla p + \nabla \bullet \tau + \boldsymbol{S}_M$$

Where the stress tensor related to the strain rate by:

$$\tau = \mu\left(\nabla \boldsymbol{U} + (\nabla \boldsymbol{U})^T - \frac{2}{3}\delta \nabla \bullet \boldsymbol{U}\right)$$

**The Total Energy Equation**

$$\frac{\partial (\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} h_{tot}) = \nabla \bullet (\lambda \nabla T) + \nabla \bullet (\boldsymbol{U} \bullet \tau) + \boldsymbol{U} \bullet \boldsymbol{S}_M + \boldsymbol{S}_E$$

$$h_{tot} = h + \frac{1}{2}\boldsymbol{U}^2$$

### 3.1. The thermal energy equations

An alternative form of the energy equation, which is suitable for low-speed flows, is also available. To derive it, an equation is required for the mechanical energy, K.

$$K = \frac{1}{2}\boldsymbol{U}^2$$

The mechanical energy equation is derived by taking the dot product U of with the momentum equation:

$$\frac{\partial (\rho K)}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} K) = -\boldsymbol{U} \bullet \nabla p + \boldsymbol{U} \bullet (\nabla \bullet \tau) + \boldsymbol{U} \bullet \boldsymbol{S}_M$$

Subtracting this equation from the total energy equation yields the thermal energy equation:

$$\frac{\partial(\rho h)}{\partial t} - \frac{\partial p}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} h) = \nabla \bullet (\lambda \nabla T) + \boldsymbol{U} \bullet \nabla p + \tau : \nabla \boldsymbol{U} + \boldsymbol{S}_E$$

$$h = e + \frac{p}{\rho}$$

$$\frac{\partial(\rho e)}{\partial t} + \nabla \bullet (\rho \boldsymbol{U} e) = \nabla \bullet (\lambda T) + p \nabla \bullet \boldsymbol{U} + \tau : \nabla \boldsymbol{U} + \boldsymbol{S}_E$$

### 3.2. Ideal Gas Equation of state

For an Ideal Gas, the relationship is described by the Ideal Gas Law:

$$\rho = \frac{w(p + p_{\text{ref}})}{R_0 T}$$

where is the w molecular weight of the gas, and Ro is the universal gas constant.

### 3.3. Reynolds Averaged Navier-Stokes (RANS) Equations

As described above, turbulence models seek to solve a modified set of transport equations by introducing averaged and fluctuating components. For example, a velocity **U** may be divided into an average component $\overline{U}$, and a time varying component u.

$$U = \overline{U} + u$$

The averaged component is given by:

$$\overline{U} = \frac{1}{\Delta t} \int_{t}^{t+\Delta t} U dt$$

where is a time scale that is large relative to the turbulent fluctuations, but small relative to the time scale to which the equations are solved. For compressible flows, the veraging is actually weighted by density (Favre-averaging), but for simplicity, the following presentation assumes that density fluctuations are negligible.

For transient flows, the equations are ensemble-averaged. This allows the averaged equations to be solved for transient simulations as well. The resulting equations are sometimes called URANS (Unsteady Reynolds Averaged Navier-Stokes equations). Substituting the averaged quantities into the original transport equations results in the Reynolds-averaged equations given below

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \{\rho U \otimes U\} = \nabla \cdot \{\tau - \rho \overline{u \otimes u}\} + S_m$$

where is the molecular stress tensor.

The continuity equation has not been altered but the momentum and scalar transport equations contain turbulent flux terms additional to the molecular diffusive fluxes. These are the Reynolds stress and the Reynolds flux. These terms arise from the non-linear convective term in the un-averaged equations. They reflect the fact that convective transport due to turbulent velocity fluctuations will act to enhance mixing over and above that caused by thermal fluctuations at the molecular level. At high Reynolds numbers, turbulent velocity fluctuations occur over a length scale much larger than the mean free path of thermal fluctuations, so that the turbulent fluxes are much larger than the molecular fluxes.

The Reynolds-averaged energy equation is:

$$\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{U} h_{tot}) = \nabla \cdot \left(\lambda \nabla \tau - \rho \overline{\boldsymbol{u} h}\right) + \nabla \cdot (\boldsymbol{U} \cdot \tau) + S_E$$

This equation contains an additional turbulence flux term, compared with the instantaneous equation. The mean Total Enthalpy is given by:

$$h_{\text{tot}} = h + \frac{1}{2}\boldsymbol{U}^2 + k$$

Note that the Total Enthalpy contains a contribution from the turbulent kinetic energy, k, given by:

$$k = \frac{1}{2}\overline{\boldsymbol{u}^2}$$

Similarly, the additional variable equation becomes

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \boldsymbol{U} \phi) = \nabla \cdot (\Gamma \, \nabla \phi - \rho \overline{\boldsymbol{u} \phi}) + S_\phi$$

Turbulence models close the Reynolds-averaged equations by providing models for the computation of the Reynolds stresses and Reynolds fluxes. ANSYS CFX models can be broadly divided into two classes: eddy viscosity models and Reynolds stress models. [22]

## 4. THEORY

According to the Dyson's patients, the aim of usage of Air Multiplier to make use of Coanda surface is to exploit the Coanda effect [3]. Increase in lift for low-speed flying aircraft is an example of usage of this effect in several industrial and commercial applications [4].

The Air Multiplier uses friction in the air to push out its cool breeze. Around the rim of the circular fan is a little opening from which jets a very thin (1mm or so) stream of air at 55 mph. That thin stream of air pulls more air into the stream thanks to the aforementioned friction.

At the same time, the air that gets pushed away from the ring towards your beautiful face creates an area of low pressure (However, not quite a vacuum, but the same effect in the ring.) That low pressure pulls in more air from behind the machine (filling the gap) which is then in turn drawn into the air stream. That's called inducement.

The engineering challenge, however, is in keeping that stream of air flowing smoothly balancing how fast the air is travelling (its "inertia") with how thick it is (its "viscosity"). Physicists have an equation which represents this balance, and it results in something called the Reynolds number.

Some quick calculations (using 55mph air flow, a 1mm gap, 1.2 kg/m^3 air density and 18.27 * 10^-6 Pa s viscosity) show that the Air Multiplier has a Reynolds number of around 1615 - relatively low. That means that the air flow out is pretty smooth (or "laminar", as physicists call it) - something seen in this image of air speeds.(FIG 2).

If all those equations makes confused massively, think of a garden höse is a better way. If you turn on the tap half way, then the flow out of the hose is smooth - it stays together in a stream when it exits the end of the pipe until it hits your flowerbed. But if you squeeze the hose, making the exit smaller, or if you turn the tap on full, then the water sprays all over the place - much less easy to control. That would have a high Reynolds number.

In a fan, you want a smooth airflow which stays aimed at whatever you point it at for upto 5 or more metres away without going all over the place, wasting energy in turbulence. The Air Multiplier appears to have achieved this with its low Reynolds number

The "multiplier" part of the name refers to how much efficiency it saves. It sends approximately 15 times more air at you thanks to taking advantage of these physical processes than it actually has to drive through its own internal spinning turbine. Of course, regular fans gain some of these effects, too, though they're more likely to suffer inefficiencies due to turbulence caused by the speed of the blades and the "open" air around it. Dyson's turbine is enclosed in the body of the unit - the "handle" of the magnifying glass.[19]

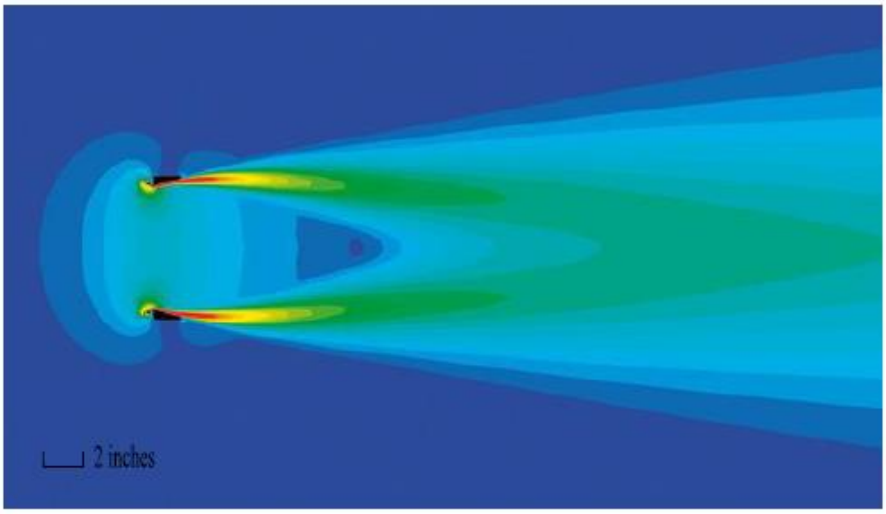



FIG. 2.— The smoothness of the resulting airflow was tested and proved using an optical technique called Laser Doppler Anemometry. Millions of tiny particles projected by the fan reflect thousands of readings a second, plotting air speed and direction. (2)

Henri Coanda states that this effect is achieved by a combination of sufficient fluid velocity flowing out of a suitable orifice and the result is entrainment of the surrounding fluid. In order to expand, Coanda observed that when the jet of fluid enters another large volume of

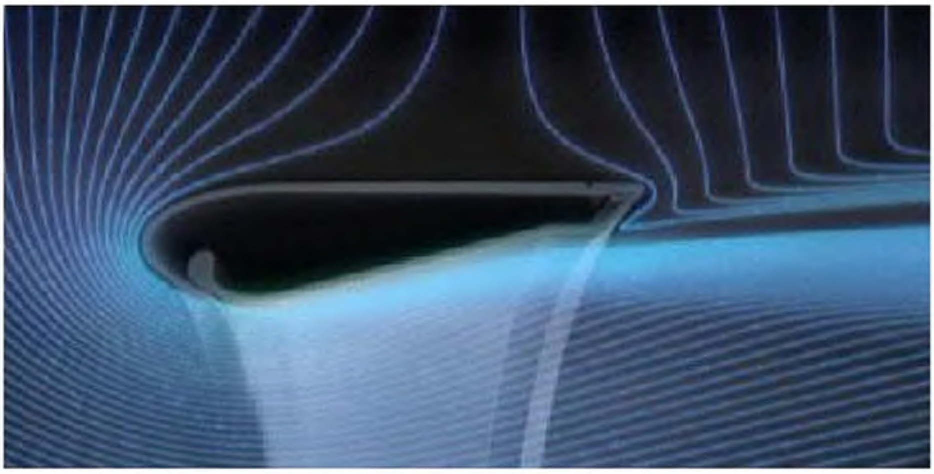


FIG. 3.— A cross sectional view of the Dyson Air-Multipliers airfoil with the theoretical streamlines superposed outside the airfoil. (7)

fluid, high velocity flowing out of a small area draws in additional flow from surrounding fluid. Besides, if a Coanda surface, one that curves away from said orifice, is present, the fluid flow will tend towards the surface, hugging it and bending away from the outlet [5]. There are several factors that varies the degree of this effect of entrainment and bending of flow such as the orifice size, fluid velocity and fluid properties like viscosity and density. An exact quantitative measure of entrainment and bending have not been documented outside of Dyson’s claims of 15-times volumetric-flow-multiplication. Although Dyson uses this explanation in their patent, there is more to the air multiplication. Coanda Effect, the Venturi Effect and Bernoulli's principle is used to explain the steps of airflow which Dyson's fan multiplies. At first, air which is supplied at the base of the fan is drawn in by an impeller [6],[3] and this results as high pressure in the hallow ring of the fan- the cross section of an airfoil- (Figure 3). Air flows across one side of the airfoil by propelling air out of a 1.3 mm slit located on the inside of the ring. This causes the entrainment of surrounding air, just downstream of the airfoil. The Venturi Effect is a factor which creates optimal air velocity that result in the Coanda Effect. once the air exists in the airfoil, it is funneled through a small slit, shown in Figure 4 [7]. Reduction in the area causes the fluid velocity to increase and this result in optimal entrainment of air.

Dyson uses Bernoulli's principle to explain that the air is drawn in from behind or induced. Once the air leaves the fan at a higher velocity than the air behind the fan, the area of

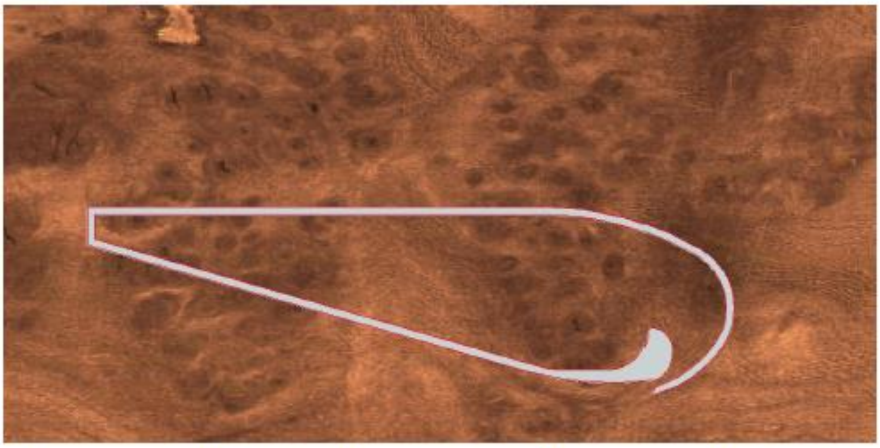


FIG. 4.— A side view of the Dyson airfoil to be constructed. The dimensions are 4.0” x 0.5” for the cross section. There is a small groove visible between the top and bottom of the air foil along with the notch present on the lower airfoil

low pressure is created. This difference between the pressures of the high velocity air and still air behind is what draws in the induced air. Air Multiplier's airflow 15 times its air intake is the result of the combination of entrainment from the Coanda Effect and inducement by Bernoulli's principle [3].

As mentioned earlier, Coanda notifies that the exact degree of bending can be adjusted for optimized entrainment according to the fluid's characteristics. Dyson's patients states that their inventors have found the optimal Coanda surface profile for entrainment of air, one with an airfoil cross section with a 16◦ angle between the top and bottom surfaces and whose cord length is a constant at approximately 1.5”.

## 5. THE MECHANIS OF THE AIR MULTIPLIER

TheAir Multiplier a fan with no blades is perhaps a touch misleading. There are blades in the fan (It just can't be seen, because they're hidden in the pedestal.). A motor rotates asymmetrically aligned blades to pull air into the device. According to Dyson, these blades can pull in up to 5.28 gallons (about 20 liters) of air per second.

The air flows through a channel in the pedestal up to the tube, which is hollow. The interior of the tube acts like a ramp. Air flows along the ramp, which curves around and ends in slits in the back of the fan. Then, the air flows along the surface of the inside of the tube and out toward the front of the fan. But how does the fan multiply the amount of air coming into the pedestal of the device?

Air is drawn into the base of the machine. The air is forced up into the loop amplifier and accelerated through the 1.3mm annular aperture, creating a jet of air that hugs the airfoil-shaped ramp. While exiting the loop amplifier, the jet pulls air from behind the fan into the airflow. At the same time, the surrounding air from the front and sides of the machine are forced into the air stream. The result is a constant uninterrupted flow of cooling air.

Air surrounding the edges of the fan will also begin to flow in the direction of the breeze. This process is called **entrainment**.

It boils down to physics. While it's true that the atmosphere is gaseous, gases obey the physical laws of fluid dynamics. As air flows through the slits in the tube and out through the front of the fan, air behind the fan is drawn through the tube as well. This is called **inducement**. The flowing air pushed by the motor induces the air behind the fan to follow.

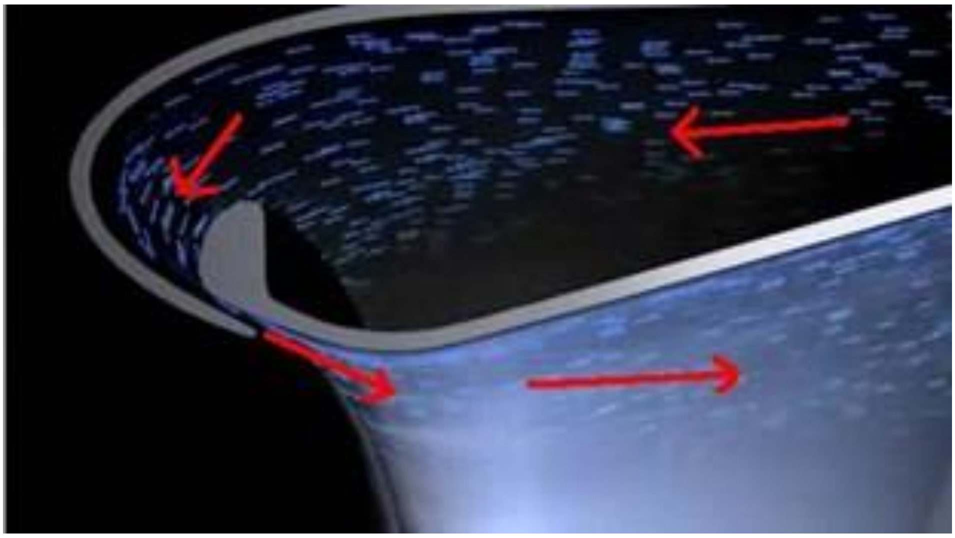
Through inducement and entrainment the Air Multiplier increases the output of airflow by 15 times the amount it takes in through the pedestal's motor. The result is a filled cylinder of air flowing smoothly without the choppiness of traditional fans.

The air flows through the channel in the pedestal, through a curved path, and comes out from small 16mm slits around the frame of the fan at a 16-degree angle slope. It can be thought that this just causes air to blow in the shape of the surface area of a cylinder, but because of the physical laws of inducement and entrainment, this allows for the surrounding air to also become drawn in from multiple areas around the fan. In other words, picture weather map, a small low-pressure region is created which actually draws the air in from behind it, like a forming tropical storm. This simultaneous push and pull of air creates a quiet, even, constant flow of cool breeze.

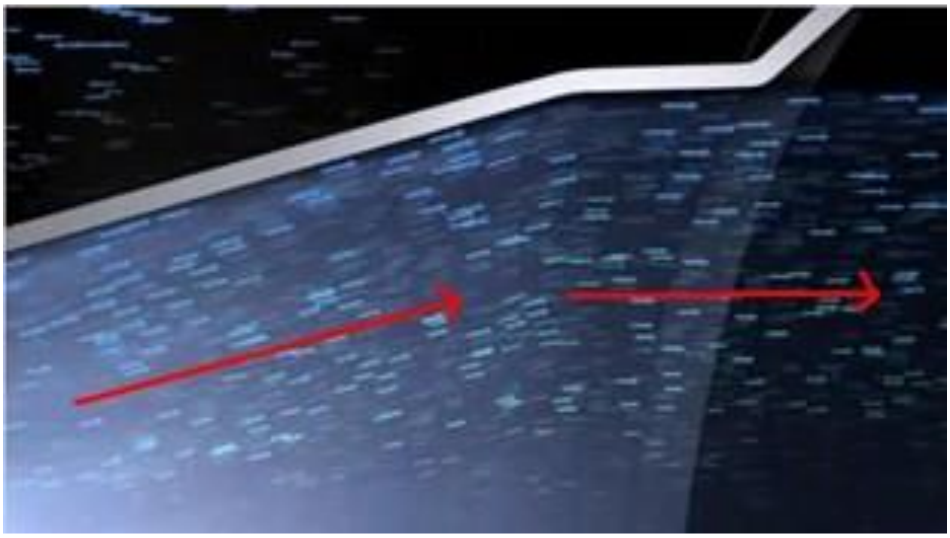
In addition to the blades in the pedestal stand, the air multiplier also makes use of a brushless electric motor, which rotates the nine asymmetrically-aligned blades. This provides for precise control of the speed of the fan, while staying relatively quiet compared to brushed motors. The pedestal motor adds extra push to the fast flowing air and shoots it up into the ring portion of the fan. [20]

The base contains a special energy efficient brushless motor, which draws in up to 33 litres of air per second from outside and directs it towards the hollow ring. The technology is somewhat similar to that used in jet engines.

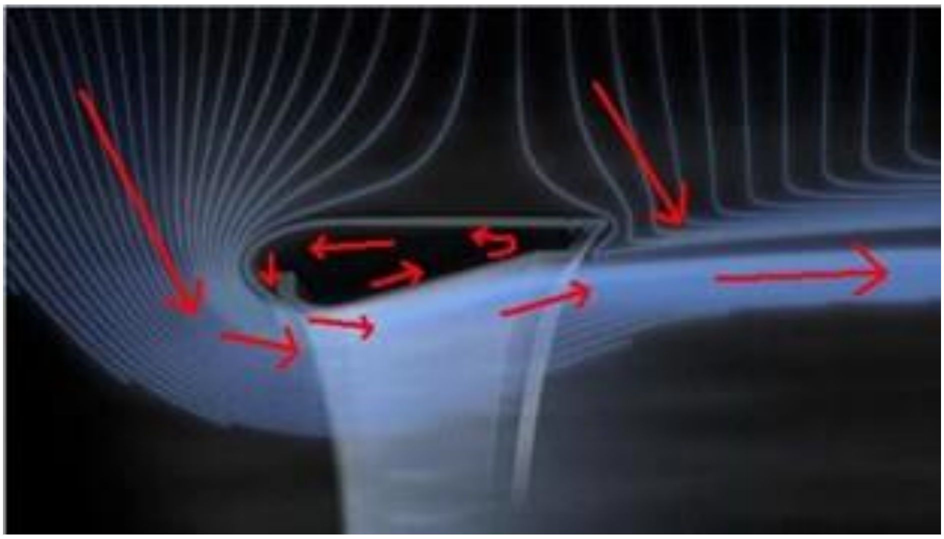
The ring is specially shaped so that the drawn air moves inside the ring in an annular fashion until it reaches the slit.



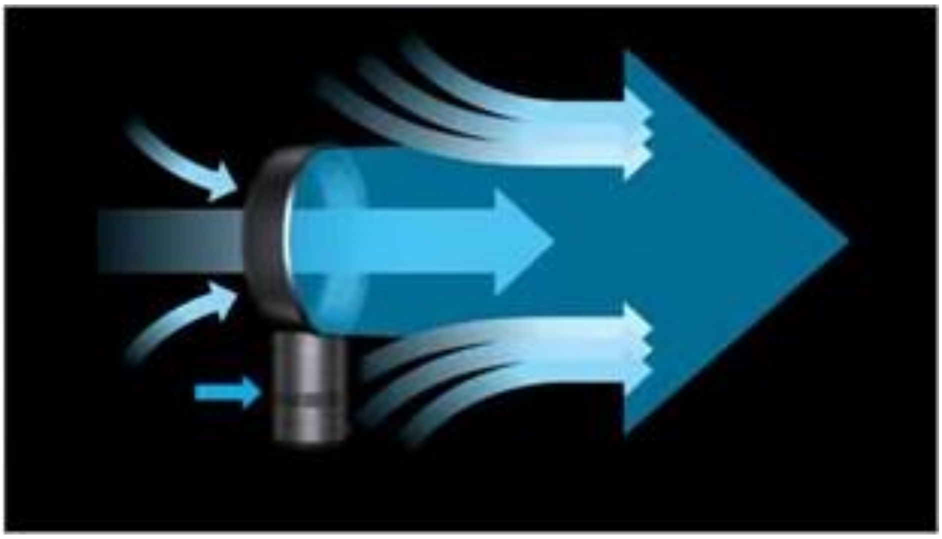
The air is accelerated through the slit as indicated above.



A special aerodynamic feature is this special air foil shaped ramp, which channelizes the air flow in a particular direction.



The air behind the machine is ‘induced’ into the machine generated air flow.



The process of ‘entrainment’ makes sure that air around the fan is effectively drawn into the air flow as well. This technology is based on the simple laws of fluid dynamics.

Dyson’s Air Multiplier is said to intensify the air flow up to 18 times.This basic technology is harnessed to come up with different designs.

It can also be clubbed with a heater, which makes it the perfect winter accessory. It is available at different ring diameters vis. 10 inch, 12 inch etc.[15]

## 6. FAN SELECTION CRITERIA

Many parameters are important in selecting a fan to meet the specified operating conditions such as:

- Fan diameter - to suit bay or cell size, Hudson blade type: “H”, “HW”, “B”, “C”, “D”, “K”,

- ACFM - Airflow required for design heat transfer duty,
- ASP - Actual static pressure from the sum of all resistances to airflow,
- Air temperature at plane of fan,
- Fan elevation above mean sea level or air density at fan, and
- Fan speed in either tip (FPM) or rotational speed (RPM).
- Also, a velocity recovery stack installed? Is there a noise limitation? How much allowable horsepower can the fan use? Will the fan avoid resonant frequency problems?

In the most simple case, design airflow, static pressure, and density are calculated for the fan “curve” conditions at 12,000 FPM tip speed and standard density with no noise limitations. The operating point is then plotted on a fan curve of the appropriate diameter.

The fan curve shows the correct pitch angle and horsepower required by the fan at the design point.

Selection is checked for the following:

(a) Confirm that the operating point is not close to a “stall” condition.

Typically, pitch angle safety margin is desired before reaching a stall point on the curve or overloading the motor. The American Petroleum Institute has specific limitations that must be met for ACHE applications.

(b) Confirm that the fan brake horsepower (BHP) is low enough that when environmental, drive and motor losses are added, the installed motor HP is not exceeded.

(c) Calculate the blade passing frequency, and compare it to the first mode resonance frequency for the selected blade (for Hudson fans).

(d) Repeat step (c) for the beam passing frequency. (Most structures have four main beams.)

(e) Repeat step (c) for 1x harmonic of fan RPM, and compare to first mode resonant frequency.

(f) Check resulting noise level against any noise limit specifications. Noise level is most often specified as sound pressure level (SPL) at a given distance from the tower or ACHE. Convert the SPL to a sound power level (PWL) to evaluate the selected fan. Some adjustments to tip speed and/or the number of blades may be required.

(g) Check air temperature at the fan. Does it exceed maximum fan operating or startup temperatures?.

## 7. FAN ENGINEERING NOMENCLATURE

**ACFM -** Actual cubic feet per minute of air moved by the fan.

**Actual Conditions** - Resistances related to actual inlet or outlet temperature and fan elevation above mean sea level compared to standard conditions.

**Air Density** - Air density at the plane of the fan based on standard or actual conditions.

**Beam Passing Frequency** - Number of times per revolution that one fan blade passes over a beam or strut-Blade Natural Frequency - Frequency at which a blade freely vibrates when it is struck in cycles/sec (Hz).

**Blade Natural Frequency** - Frequency at which a blade freely vibrates when it is struck in cycles/sec (Hz).

**Blade Passing Frequency** - Number of times per revolution that a fan tip passes a point on the fan ring expressed in cycles/sec (Hz)- thought of as “how the fan interacts with the structure”.

**Brake Horsepower** - (BHP) - Net power required by the fan at actual conditions to perform the required design work.

**Chord** - Straight line distance between the leading and trailing airfoil edges.

**Fan Diameter** - Width/distance between opposite blade tips.

**Fan Laws** - Set of laws that predict performance changes if one or more parameters are changed from one fan or operating condition to another. These laws govern airflow, pressure capability and power required among many other parameters.

**Fan Ring Diameter** - Inside diameter of fan housing at the plane of fan.

**First Mode Resonant Frequency** - Frequency at which a blade freely vibrates when struck (“natural” frequency) in cycles /sec (Hertz).

**Forced Draft ACHE** - Fan is located below the heat transfer surface forcing ambient air through the bundle.

**Harmonic Frequency** - Integer multiples of fan RPM and expressed as 1x, 2x and 3x fan speed in cycles/second (Hertz). Harmonic frequency is checked against resonant frequencies to prevent vibration and fatigue.

**Induced Draft ACHE** - Fan is located above the heat transfer surface drawing ambient air through the bundle. The fan is exposed to the heated exhaust air.

**Leading Edge** - Thicker portion of the air foil that is the first part of the blade to meet the air.

**Net Free Area** - Net area at the plane of the fan through which all air must pass. Usually based on the nominal fan diameter minus seal disc area or hub diameter. Note that blade area is not considered.

**Pitch Angle** - Blade tip angle below the horizontal required to do the design work and move air upward. Hudson fans all rotate clockwise looking into the airflow.

**Resonant Frequency Safety Margin**- Percent difference between the closest resonant frequencies of 1st mode resonant frequency, blade and beam pass frequencies, and 1x Harmonics.

**SCFM** - Airflow rate moved at standard conditions, in standard cubic feet per minute.

**Solidity Ratio** - Measure of a fan’s pressure capability solidity sum of the tip widths divided by the fan circumference.

**Stall Point** - Fan operating condition where the boundary layer of air separates from the airfoil and causes turbulence. This can be compared to boat propeller or pump cavitation.

**Standard Conditions** – Resistances related to the standard density of air at 0.075 lbs/ft3 , at 70°F dry bulb temperature and sea level (29.92 in. Hg).

**Static Efficiency** - Fan efficiency based on static pressure and fan brake horsepower at the same density.

**Static Pressure** - Sum of all the system resistances against which the fan must work, expressed in inches of H2O. This is the useful work required from the fan. (Velocity pressure excluded).

**Tip Clearance** - Distance between the tip blade and the fan ring or housing, sometimes expressed as a percent of the fan diameter.

**Tip Speed** - Peripheral speed of the fan tip expressed in feet per minute.

**Total Efficiency -** Fan efficiency based on the total pressure and fan brake horsepower at the same density for standard or actual conditions.

**Total Pressure** - Sum of the static pressure and velocity pressures.

**Trailing Edge -** Thinner portion of the air foil.

**Velocity Pressure -** Parasitic loss caused by work done to collect all the air into the fan's inlet, expressed in in. of H2O. It is based on the fan's net free area at the plane of the fan.

**Velocity Recovery Stack** – Device frequently used in cooling towers but rarely in ACHEs. It captures the kinetic energy of the exit air velocity and converts it to useful work. It "reduces" air velocity and decreases the total pressure acting on the fan.

## 8. EXPERIMENTAL SETUP

According to experiments had previously read; they constructed airfoils that achieved airflow similar to that of the Dyson Air Multiplier. An airflow through the foil in a manner similar to the Dyson fan use to produce an air compressor. The air flowed into our airfoil on both sides. A null displacement manometer was used to measure flow velocity near the surface of our airfoil.

The airfoil has dimensions of 7.0" x 4.0"x 0.5" as seen in Figure 5. The 16◦ angle along the horizontal surface and the Coanda surface and the top section of the airfoil which was used by Dyson was recreated in our airfoil. The slit in the airfoil is 1.3 mm (0.0512");

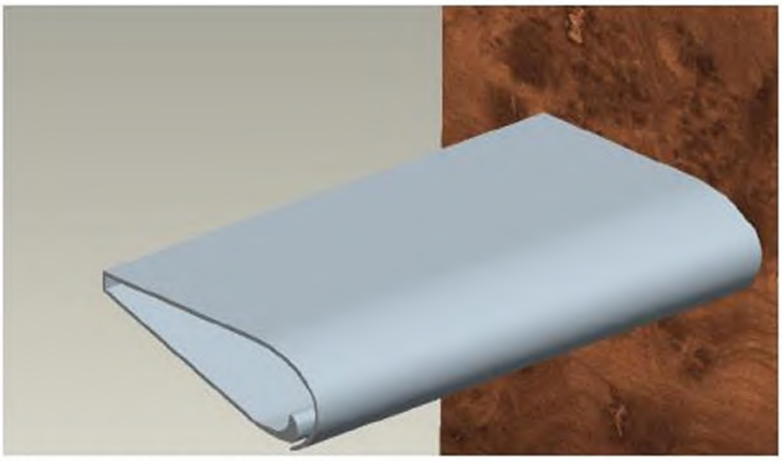


FIG. 5.— An angular view of the Dyson airfoil to be constructed. The dimensions are 7.0" x 4.0"x 0.5' for the cross section. The groove for air flow is faintly visible.

according to Dyson this can range from 1 mm to 5 mm (0.0394" to 0.197") [3].The construction of the airfoil was done by using fiberglass and a foam exoskeleton as can be seen in Figure6. The foam mold was acquired from another team using foam cores. A layer of 0.0089" thick 6 oz fiberglass was laid on the foam mold covered with wax paper and cling wrap.

The use of wax paper and cling wrap allowed for the foam exoskeleton to be easily separated from the fiberglass. A thin slow hardening epoxy was used to reduce the amount of

FIG. 6.— A lengthwise view of the foam mold used to construct the two halfs of the airfoil.

residue left on the surface of our fiberglass airfoil[8]. Another layer of epoxy was used to smooth out the surface of the parts, and then sanded to further smooth the bumps on the surface of the airfoil.

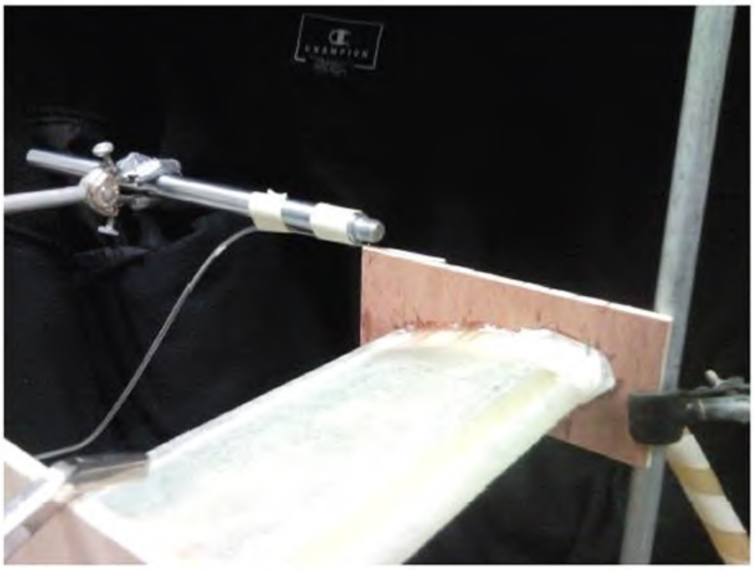


FIG. 7.— An angled view of the airfoil in setup during experimental tests in which it was used upside down. The manometer's capillary tube can be seen above the airfoil.

The bottom section, seen in Figure 4 is the Coanda surface. The top section connects with the bottom sec- tion on the left side of Figure 4. The two parts were built with the same foam exoskeleton, and both parts were cut using a jewelers saw whose small teeth make it ideal for this situation. The pieces were cut to meet the requirements for the top and bottom pieces as seen in Figure 4 and then attached to wood cross sections that kept the airfoil's shape. Attaching parts together was done with Plastic Fusion Epoxy Adhesive made by the Super Glue Corporation. A piece of wood 1.3 mm in thickness was used to space the gap between the airfoils during setting. The airfoil was then capped with two wooden boards and attached with epoxy to seal its sides. Holes were then drilled in both sides of the airfoil to allow for the air to enter through nozzles. The finished product can be seen in the experimental setup in Figure 7.

A Craftsman 919.165030 air compressor was used as the air source. It was capable of delivering 0.0118m³/s at steady state when they tested it with a flow meter. The Dyson Air Multiplier patent states it delivers 0.0297m³/s. Since our airfoil has an opening which is less than half of the opening surface of the Dyson fan, this volumetric flow rate was sufficient. The air source was supplied through both sides of the airfoil along the horizontal axis using nozzles connected to the air compressor with surgical tubing.

To test steady state airflow they used the null displacement manometer. The manometer has a near instant re- action rate, which allowed us to determine if we achieved a steady state flow. The capillary tube being used with the manometer has an outside diameter of 1.3 mm and an inside diameter of 0.97 mm. This is not small enough to measure the boundary layer at the slit, but it could do so farther downstream.Greater resolutions for measuring air flow velocities have been realized for low-pressure measurements by using a micrometer to read the vertical well displacement necessary to return the meniscus to its null

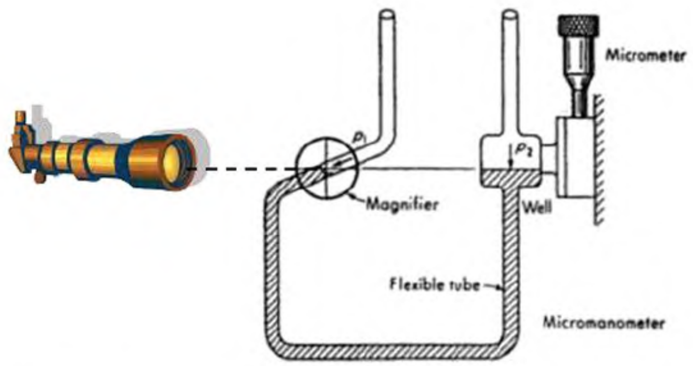



FIG. 8.— Schematic diagram of the null displacment manometer. A telescope is used to magnify the waterline in the tube. A micrometer is used to finely adjust the height of the funnel and thus the corresponding height of the water in the tube.

position as marked by a hair line [10]. This type of manometer is known as a null displacement manometer. This manometer is connected to the stainless steel capillary tube, that acts as a small probe with outer diameter 1.3 mm. Water was recommended and was used in this manometer. Our null displacement manometer was set up with a micrometer at the base of the funnel that could be adjusted to raise or lower the water level in the tube. A telescope was anchored to the same stand as the tube end of the manometer as shown in Figure 8. The end of the objective lens of the telescope was placed 15 cm from the marking on the null displacement manometer. The funnel containing water was embedded in the hemisphere of a baseball, and its height controlled by a micrometer. A baseball was drilled so that it had a hole through its center and was then cut in half. It was used to minimize inadvertent tilting that might arise from raising and lowering the funnel. The height of this funnel controlled the level in the associated tube and the height of this funnel from a zero gave us an air velocity. The micrometer had a resolution of 200 rad/inch, thus giving us a velocity resolution of 0.64 m/s from the manometer setup. Since this was being used to probe the boundary layers, where they expected to see 18 m/s, this resolution would suffice though it was less than ideal.

The manometer was first used at an angle of zero degrees with respect to the horizontal axis. Measurements were taken vertically, along the z-axis, every 1mm. This was done repeatedly at points every 1.27 cm (0.5”) in the x direction along the airfoil. The first point of measurement was taken at the lowest point of the curve along the airfoil. To simplify taking these measurements the airfoil was turned upside down. This set up can be seen in

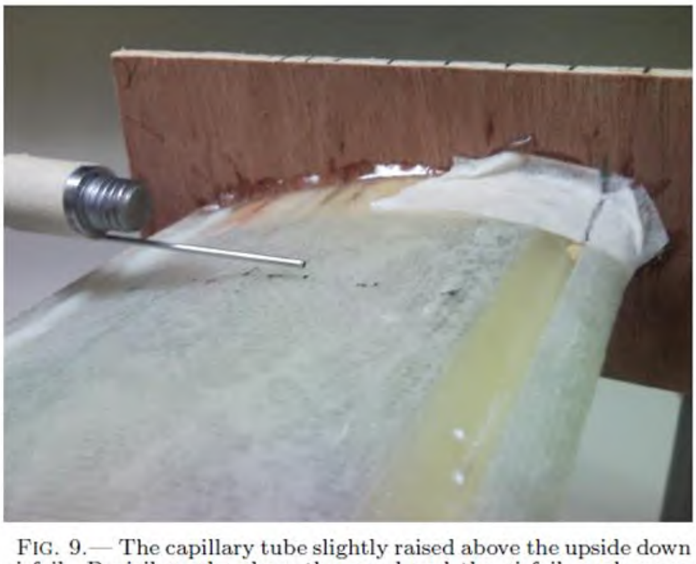


FIG. 9.— The capillary tube slightly raised above the upside down airfoil. Penicil marks along the wood and the airfoil mark every 1.27 cm (0.5”), the mark farthest to the right indicates the lowest point on the airfoil. The slit can be seen on the right side of the image.

Figure 9. To find all of the velocity vectors the manometer was turned to an angle of 45° and the velocities at that angle were then measured. Having these two measurements of velocities at different angles allowed us to find the actual velocity vector. For the math on how to derive the velocity vector from the two measured values. With the data they were able to construct a directional vector field to help analyze the boundary layer around the airfoil.To further our understanding of the characteristics of the flow they recorded video by using a high speed camera and fine dust. The airfoil was set up right side up and dust was dropped down from behind the airfoil. The camera was run for 2 seconds capturing at 500 fps. The camera was used so that it would show the entrainment and inducement of the surrounding air into the flow created by the airfoil.

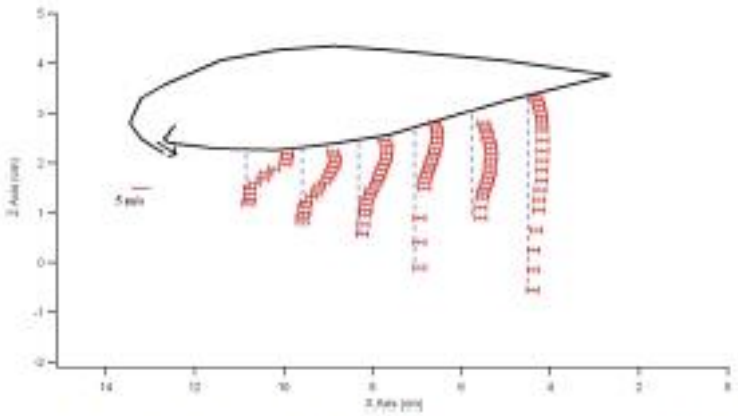


FIG. 10.— Velocity profiles along with error bars taken along the x-axis of the airfoil every 1.27 cm (0.5"). The manometer was used at an angle flat to the x-axis.

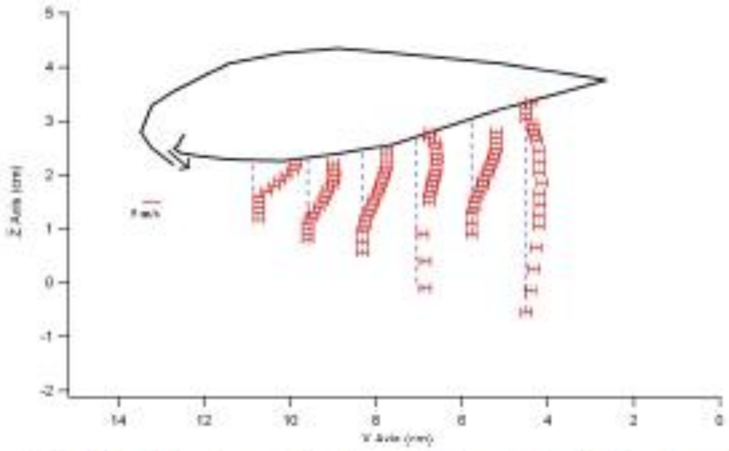


FIG. 11.— Velocity profiles along with error bars taken along the x-axis of the airfoil every 1.27 cm (0.5"). The manometer was used at an angle of 45° to the x-axis.

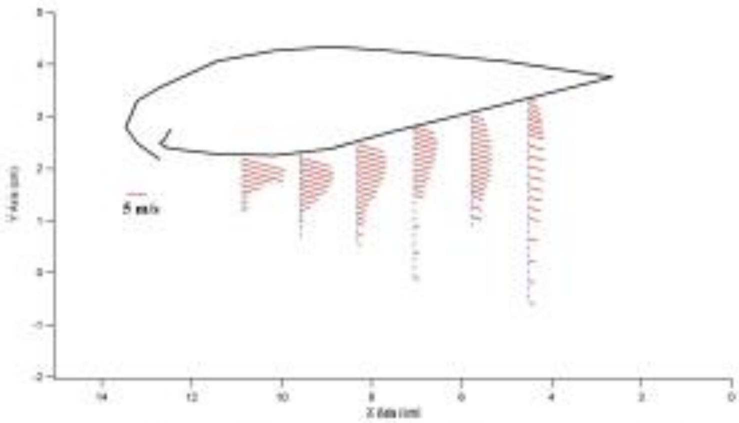


FIG. 12.— Velocity vectors for each profile on the airfoil that can be seen in Figure 10 and Figure 11.

## Reducing Noise

In spite of its luxurious looks and cutting-edge concept, the Dyson fan did have one notable flaw. It wasn't really very quiet. Dyson took note, and decided to revamp the second generation of its Multiplier.

The noise problem originated from turbulence. The Multiplier sucked air into its base, where it bounced around willy-nilly, creating chaos and noise. To pinpoint this noise, researchers placed the fan in a semi-anechoic (soundproof) chamber with 10 microphones listening for every whir and buzz.

Then they built translucent prototypes and passed ultraviolet paint and smoke through the device. High-speed cameras provided frame-by-frame playback, offering visual clues as to areas where air was bunching up and basically causing a ruckus.

The investigations addressed the turbulence problems by integrating Helmholtz cavities into the fan's base.

## Helmholtz Cavities and the Art of Noise

Helmhortz cavities make noise, of course. Figure out exactly how these cavities work, and then you can control that noise. By adding Helmhortz cavities of sorts into the base of the Multiplier, engineers increased air pressure, and ultimately these cavities began to work as silencers.

Car manufacturers are very familiar with the principles of Helmhortz cavities. They manipulate them to their advantage when quieting exhaust systems. In the case of the Multiplier, engineers basically tuned the cavities to specifically mute sounds in the range of 1,000 Hertz, which humans tend to find especially aggravating.

Their efforts (and those heaping mountains of research cash) paid off. According to Dyson, the second-generation fan is 75 percent quieter than its ancestor. And because air moves more smoothly and efficiently through the entire Multiplier, Dyson was able to scale back on the motor. They say the new motor requires 40 percent less power.

**For its quietness, the Noise Abatement Society awarded the Multiplier with a Quiet Mark award. The award goes to products that clamp down on unnecessary noise pollution.**

Air Multiplier fan is powered by an energy efficient brushless motor and air speed can be precisely djusted with a dimmer switch. Conventional fans are wired to run at just two or three settings.

No blades mean no need for a grille and it's safe and simple to clean. And because its motor is at its base, the Dyson Air Multiplier(TM) fan can be tilted with a touch, unlike a conventional top-heavy fan, which needs to be positioned with two hands and can topple easily. [20]

## 9. RESULTS

Data was taken at an angle of 0◦ and 45◦ to the x- axis. The first data set yielded Figure 10, while the second data set yielded Figure 11. These figures show the velocity profiles, along with their error bars, that were derived from many different manometer readings. The error comes from the resolution of the telescope in working with the manometer and oscillations of the water around the zero on the manometer.

Looking at the horizontal magnitudes of the flow vec- tors in Figure 11 we can see how the maximum velocities follow the airfoil surface. In the progression of the veloc- ity profiles the maximum velocity follows the surface of the airfoil or Coanda surface because of the Coanda ef- fect. This effect causes the fluid, in this case air, to bend upwards and follow the surface. The presence of this ef- fect helps to verify Dyson's patent. As the flow progesses downstream the velocity profile expands downwards and reaches a more even flow. Potentially there could be a laminar flow below or past the last data points. The boundary layer is outlined here by the decrease in velocity against the airfoil from the point of the maximum velocity. In the first two velocity profiles the boundary layer was unobserved, because it was too small to be observed with the size of our capillary tube. The velocity profile that is 2.54 cm (1") from the apex, at approximately 8 cm on the x-axis in Figure 10, the first appearance of the boundary layer is evident. The no-slip condition, where the velocity is zero at the surface, is not apparent here because of the diameter of the capillary tube. Although, the boundary layer can be seen tending towards zero at the airfoil surface.

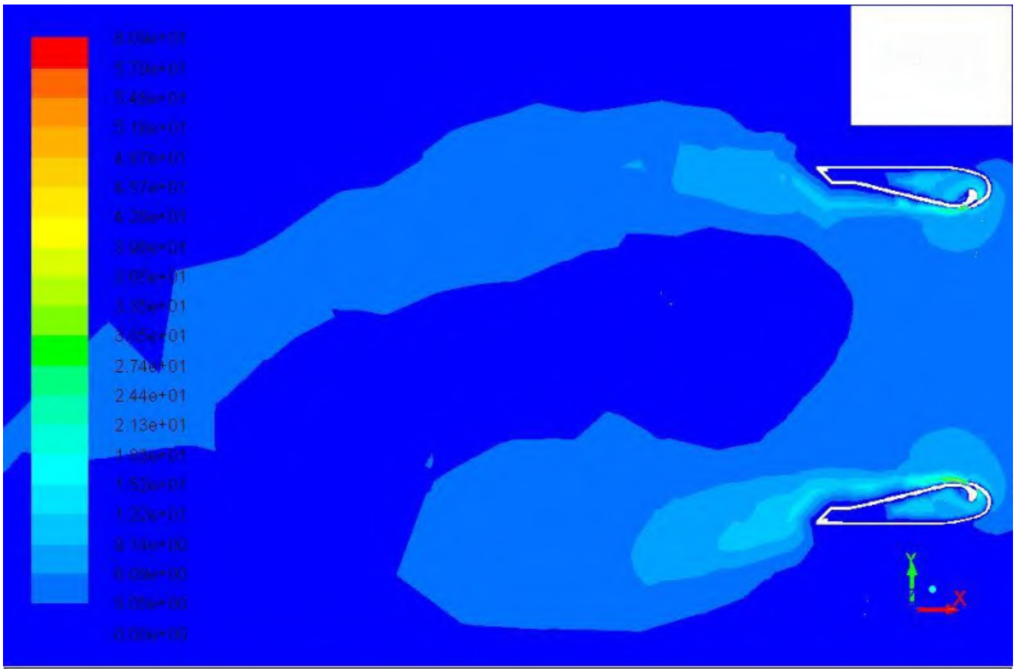
In Figure 12 we present the velocity vectors along the lower surface of the airfoil. From the data collected (par- allel to the horizontal axis and 45◦ below horizontal) we were able to calculate the angle of the actual velocity vector using the equations in Appendix A. These angles were then used to calculate the absolute velocity in the horizontal and vertical directions. The sum of these vec- tors is presented in Figure 12. The dashed vertical lines represent the locations along which velocity data was col- lected with the manometer probe. The solid lines ema- nating from these vertical lines are the velocity vectors calculated at those locations. No errors are presented for this data since this would obscure details and dominate features.

We observe a radical departure in our velocity vectors from the streamlines presented in Figure 3. The stream- lines in Figure 3 converge towards the lower surface of the airfoil but the velocity vectors, which represent stream- lines, in Figure 12 diverge from the same surface. There are a number of reasons for this: In ourfabrication of the airfoil we did not follow the designs laid out by Dyson Corp. for their fan but rather, due to time constraints, made one of similar dimensions. Due to the compar- atively large size of our Pitot tube we were unable to measure the boundary layer. Lastly, our airfoil surface was not completely smooth, introducing local variations from what was expected. This last point however should not introduce large errors and departures from theory but might explain some of the smaller irregularities.

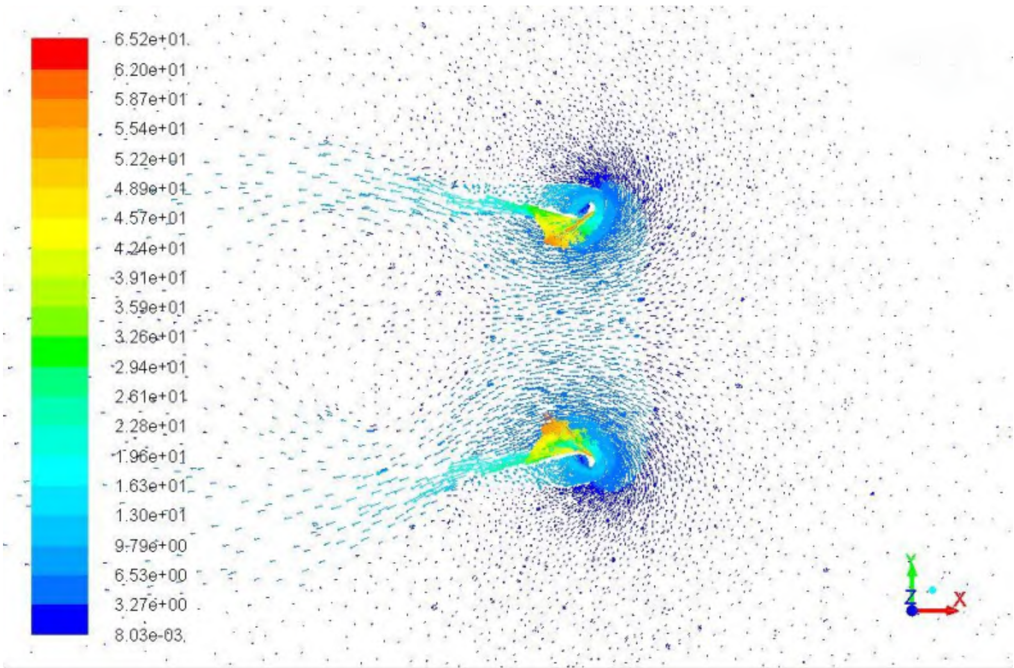
Images of the high-speed camera can be seen in Figure 13, these images were produced by dropping fine dust from a funnel behind the airfoil so that we could examine the flow patterns. In Figure 13 dust is drawn in from behind the airfoil, or induced. Large particles and clumps of dust fall down past but small particles get pulled into the flow indicating that there is air drawn in from behind the airfoil. Only the finer particle clouds were induced by the airfoil.

Recording the entrainment did not work out well, as the dust was too fine to show up on the high-speed camera. Despite the fact that the dust dispersed greatly after entering the region of flow, some of the dust was still pro- pelled nearly two meters downstream from the airfoil.

## 9.1. SPALART MODEL



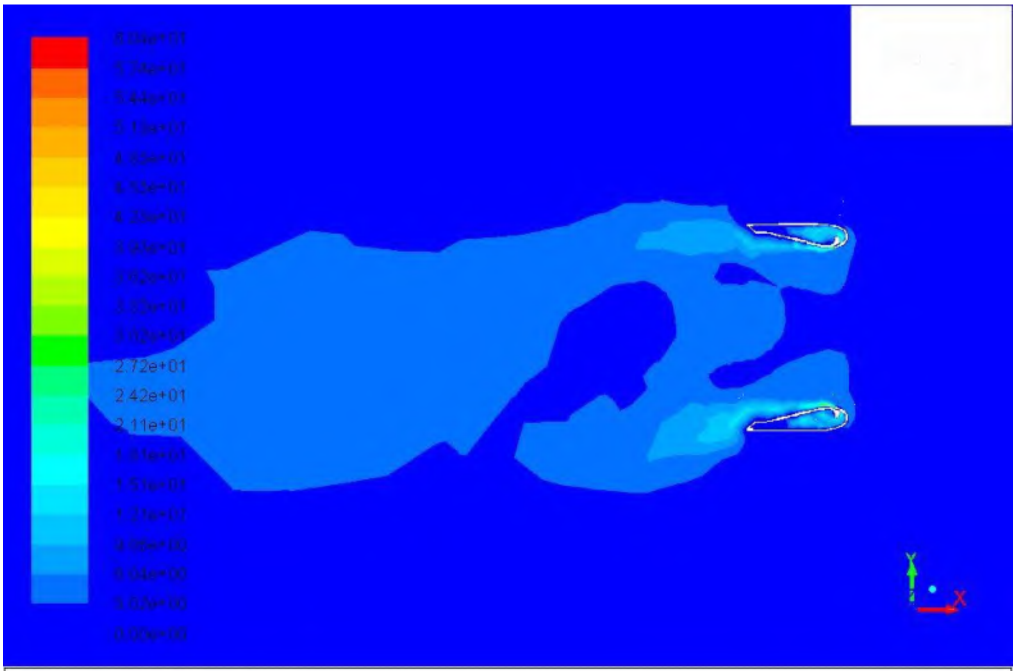

Contours of Velocity Magnitude (m/s)
Jun 06, 2014
ANSYS Fluent 15.0 (3d, dp, pbns, S-A)



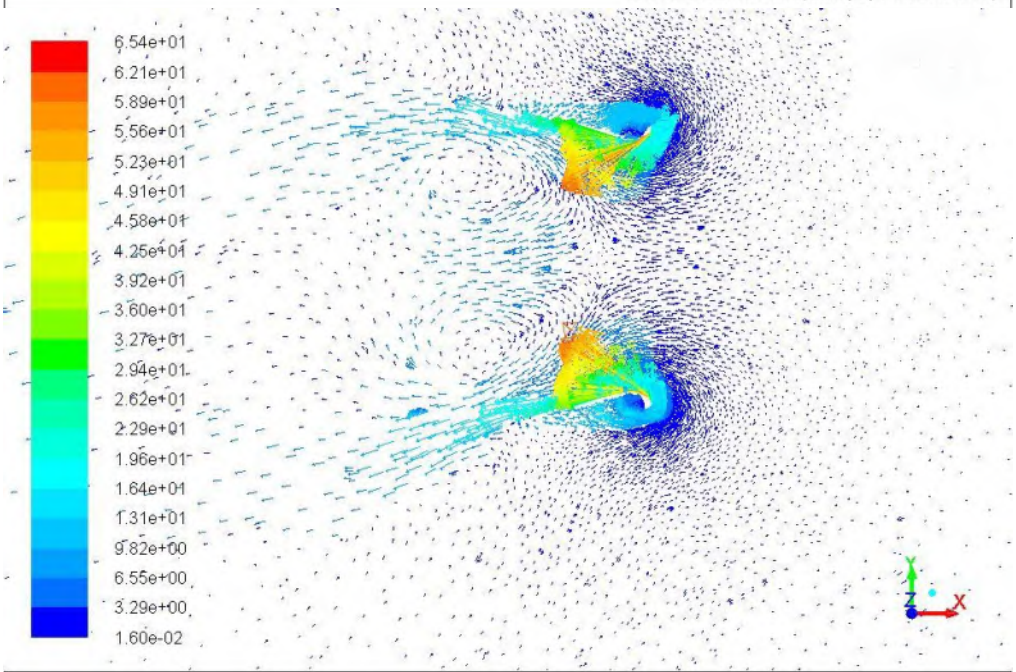

Velocity Vectors Colored By Velocity Magnitude (m/s)
Jun 06, 2014
ANSYS Fluent 15.0 (3d, dp, pbns, S-A)

When the inlet velocity is applied as 20 m/s, we obtained exit velocity as 60,9 m/s from spalart turbulance model analysis

## 9.2. THE TRANSITION SST MODEL



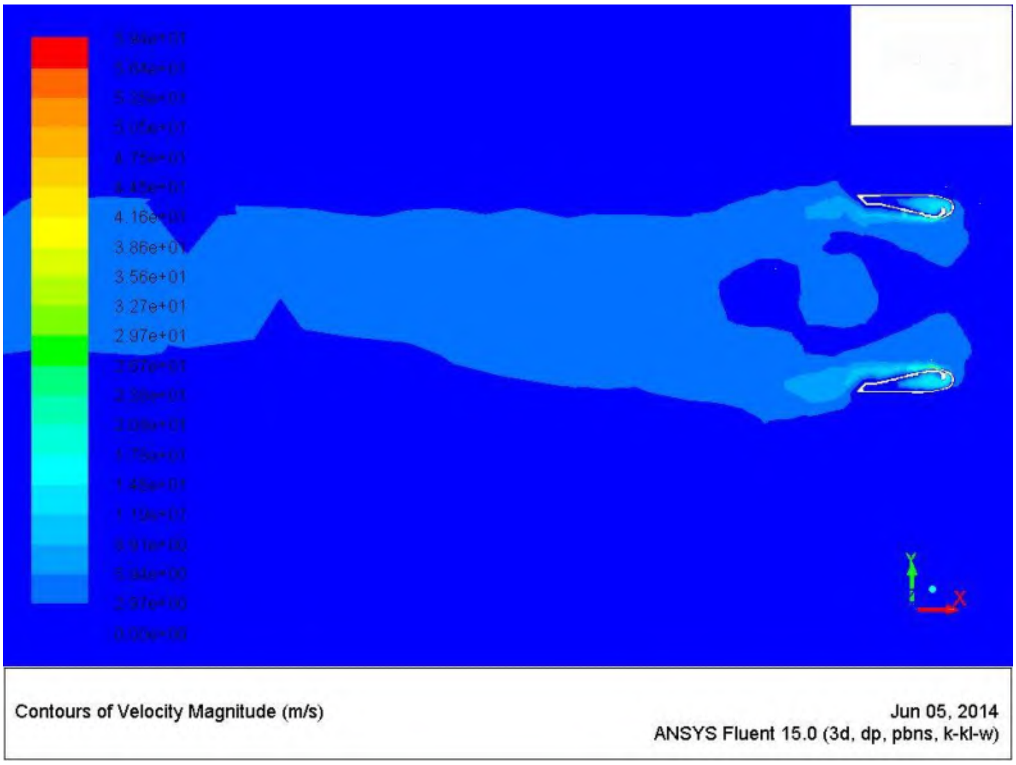

Contours of Velocity Magnitude (m/s) Jun 05, 2014
ANSYS Fluent 15.0 (3d, dp, pbns, trans-sst)

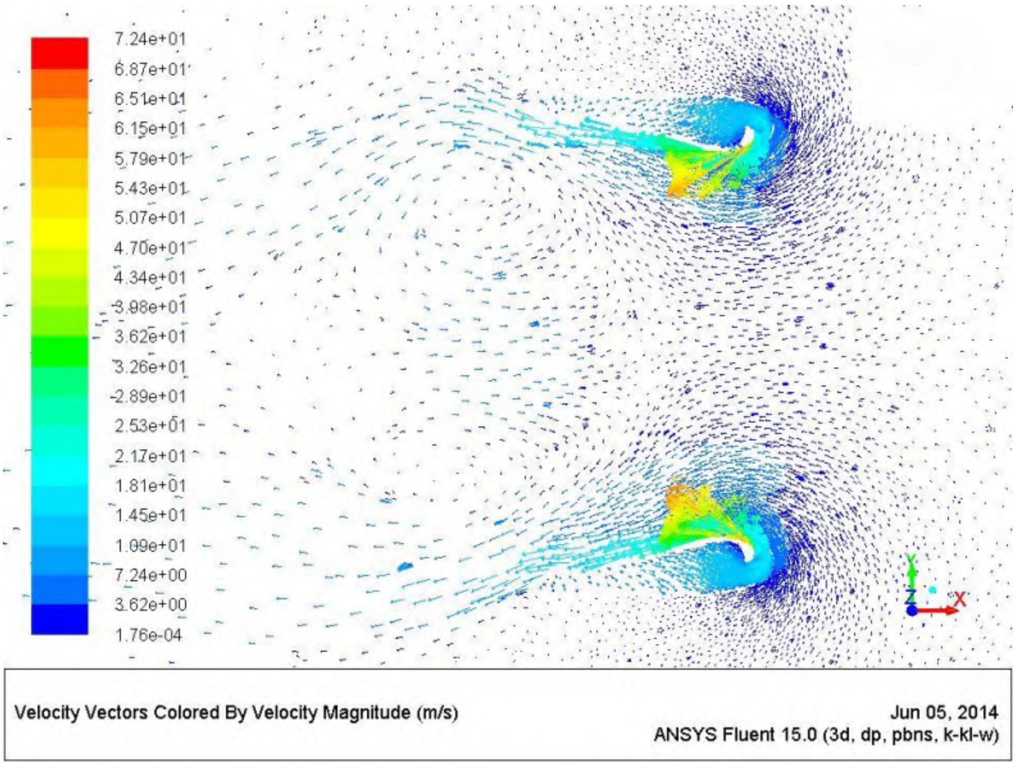



Velocity Vectors Colored By Velocity Magnitude (m/s) Jun 05, 2014
ANSYS Fluent 15.0 (3d, dp, pbns, trans-sst)

When the inlet velocity is applied as 20 m/s, we obtained exit velocity as 60,4 m/s from the transition SST model analysis

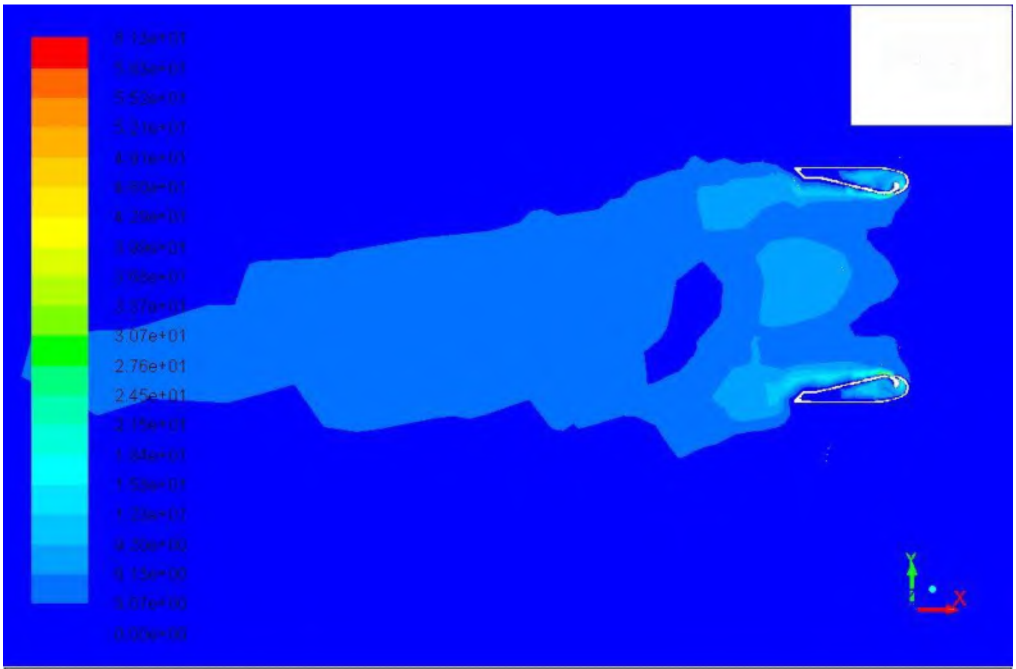
## 9.3. KL MODEL



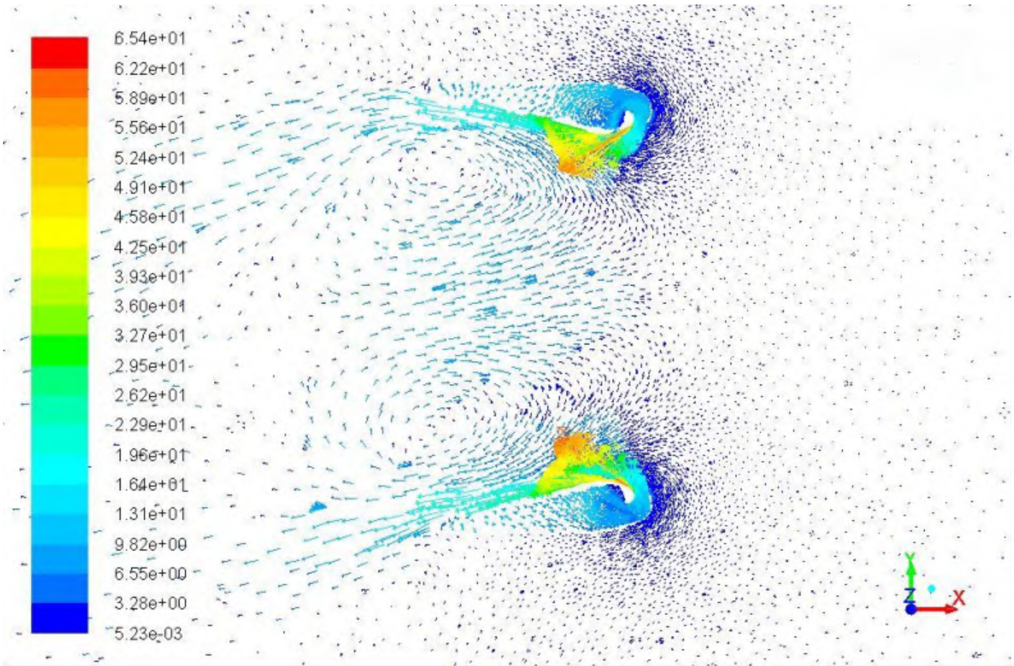




When the inlet velocity is applied as 20 m/s, we obtained exit velocity as 59,4 m/s from KL model analysis

## 9.4. STANDARD K-ω MODEL




Contours of Velocity Magnitude (m/s)    Jun 06, 2014
ANSYS Fluent 15.0 (3d, dp, pbns, skw)




Velocity Vectors Colored By Velocity Magnitude (m/s)    Jun 06, 2014
ANSYS Fluent 15.0 (3d, dp, pbns, skw)

When the inlet velocity is applied as 20 m/s, we obtained exit velocity as 61,3 m/s from standard K-ω model analysis

## 10. CONLUSIONS

The airfoil design we constructed showed interesting airflow characteristics similar to those reported by Dyson Ltd’s. We were able to observe the inducement of the sur- rounding air using fine dust and the high-speed camera. The Coanda effect and the boundary layer were both ob- served near the surface of the airfoil.

Speed Inducement.png

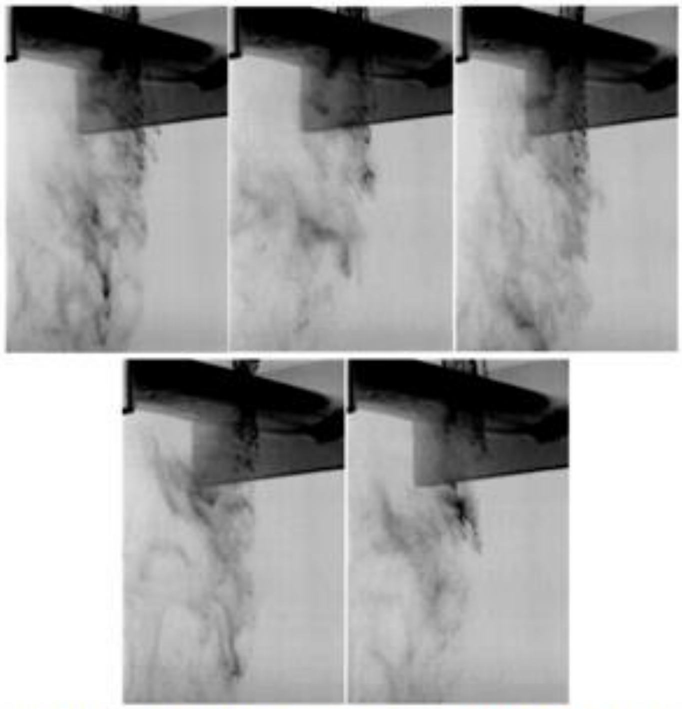


Fig. 13.— The figure shows a series of frames taken from the high-speed camera to illustrate inducement. The images have an unnatural contrast that is best for viewing the dust.

A large problem we faced with recording data came from our main instrument of measurement, the null dis- placement manometer. The manometer did not have a high resolution and as a result had a good deal of er- ror as compared to our therotical resolution. In the area behind the airfoil the manometer was unable to register any readings, since the velocities were below the resolu- tion of the manometer. Sometimes it would fail to keep a consistent zero in between trials. Possible error in the manometer was contributed to moisture in the thin tub- ing. A better instrument for measuring velocity would have improved our data.

The images acquired from the high-speed camera are unquantifiable since there were no measurements taken. The figures associated with the high-speed camera only illustrate the flow around the airfoil for the purpose of better understanding how the air moves through this area. Further studies could find a way to develop nu- merical data from this kind of observation or even other more complex types of recording data to quantify the in- ducement and entrainment of the airfoil design. In this way the 15-times volumetric-flow-multiplication reported by Dyson Ltd could be proven.

If we had more time we would have time to experiment with multiple airfoil geometeries. Initial experimentation included adjusting the angle and the slit spacing of the airfoil to maximize the volumetric flow. Further devel- opment would have been to put multiple airfoils together to replicate the Dyson Fan design.

While we did see that our airfoil created a region of nearly even flow on the outer edge of our airfoil, the velocity was still not high enough to be used as a practical wind tunnel at this scale. Furthermore we do not know if this area is laminar or not. If further work could prove that the flow in this region is laminar this design could be adapted and scaled to be used as a wind tunnel.

Neverthless, some disadvantages of this new technology can be sorted as:

- The wind velocity is insufficient when compared to regular fans of the same size.
- It is as noisy as a vacuum cleaner.
- It’s expensive. Read further for the rates.[15]

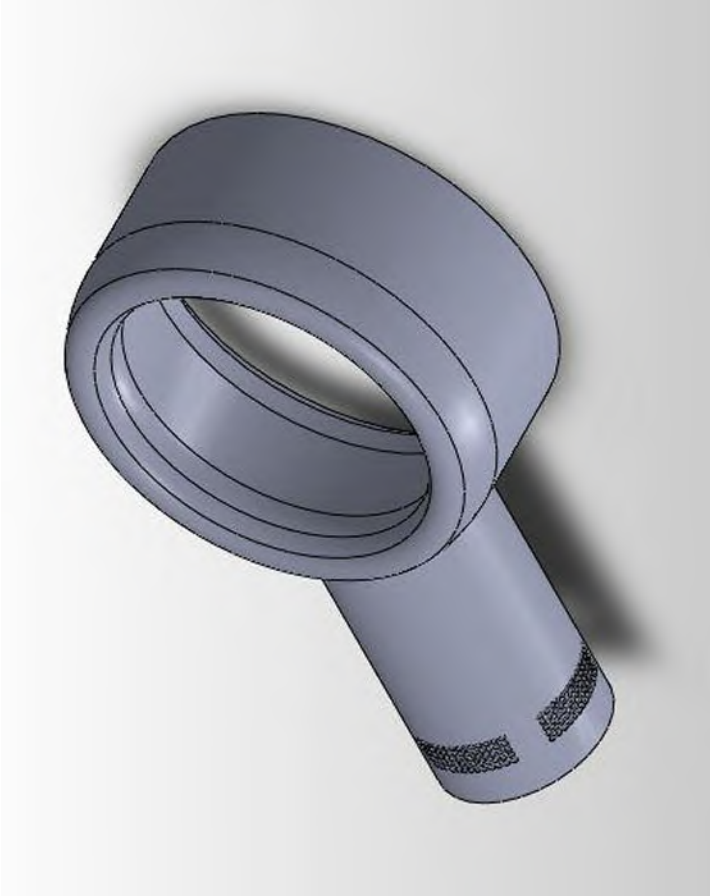
The Dyson Air Multiplier might be the next generation in fan technology. Gone are the fan blades you’ve seen on every other fan every. It’s replaced bye a hollow ring. Basically air is sucked into the base of the fan and then forced around the inside of the ring and shot out of one side of the ring. The ramp-like nature of the ring causes it to induce air to move from behind the ring through the hollow middle of the ring resulting in a fan-like experience. The technology, while cool, is quite loud- like small vacuum cleaner. [21].

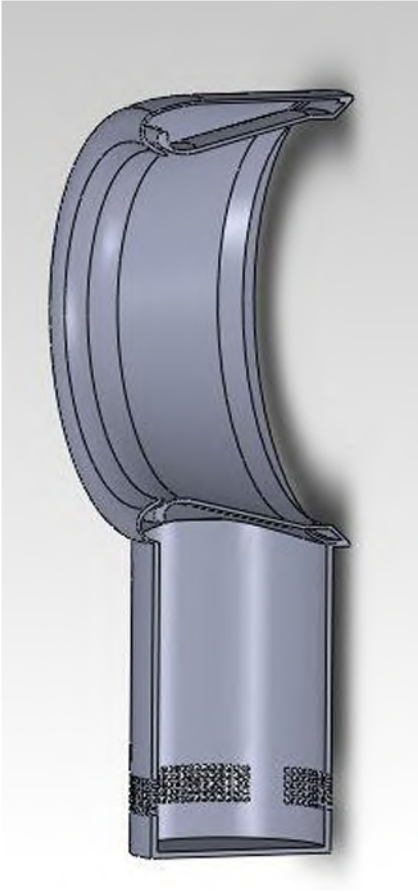
From the investigations we find that the exit velocity is three times more according to the inlet velocity.

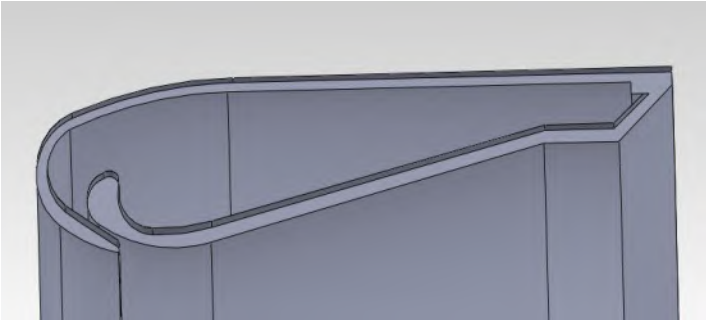
According to turbulent kinetic erengy results, better prediction were obtained by k ω model near aperture boulndaries.

In our analysis we try to obtain much more efficency by applying four different turbulence models and finally the best predicted result is obtained from k ω model.

## 11. SOLID DRAWING







## 12. REFERENCES

[1]Goldstein, E.,"Wind Tunnels, Don't Count Them Out,"
Aerospace America, Vol. 48 4, April 2010, pp. 38-43

[2]Dyson Bladeless Air Multiplies Laser Doppler Anemometry
Results, "http://www.dyson.com/insideDyson/", Date Accessed 19/02/2011.

[3]Gammack, P. D., Nicolas, F., Simmonds, K. J.. (2008). UK Patent Application. Application No. 0814866.0

[4]Tritton, D.J., Physical Fluid Dynamics, Van Nostrand Reinhold, 1977 (reprinted 1980), Section 22.7, The Coanda Effect.

[5]Coanda, Henri, (1936). Device for Deflecting a Stream of Elastic Fluid Projected into an Elastic Fluid. US Patent No. 2052869.

[6]Dyson Bladeless Air Multiplies, "http://www.dyson.com/fans/",
Date Accessed 19/02/2011.

[7]Dyson Bladeless Air Multiplies Airfoil Streamlines, "http://www.blokeish.com/blog/wp-content/uploads/2010/01/dyson-bladeless-air-muliplier-how- work.jpg", Date Accessed 19/02/2011

[8]Guide to Glass Reinforced Plastics, "http://www.ecfibreglasssupplies.co.uk/t-GlassReinforcedPlastics.aspx," Date Accessed: 18/02/2011

[9]Microprocessor-Based Portable Air Velocity
Kit, "http://www.omega.com/green/pdf/hh30a.pdf, "DateAccessed : 17/03/2011

[10]Richard W. Johnson, "The handbook of fluid dynamics," Springer-Verlag 1998

[11]Studying Near-Surface Effects of the Dyson Air-Multiplier Airfoil
"http://www.me.rochester.edu/" Date Accessed : 27/04/2011

[12] http://www.wisegeek.net/what-is-a-mechanical-fan.htm

[13] https://www.newworldencyclopedia.org/entry/Fan_(implement)

[14] http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/article/AA-V4-I2-Bladeless-is-More.pdf

[15] http://homeneeds.sulekha.com/all-you-need-to-know-about-bladeless-fans_599357_blog

[16] http://en.wikipedia.org/wiki/Mechanical_fan

[17] http://en.wikipedia.org/wiki/List_of_Dyson_products

[18] http://www.thegrommet.com/dyson-air-multiplier-1

[19] http://www.pocket-lint.com/news/98959-how-dysons-air-multiplier-works

[20] http://electronics.howstuffworks.com/gadgets/home/dyson-bladeless-fan1.htm

[21] http://craziestgadgets.com/2009/10/13/dysons-air-multiplier-bladeless-fan/

[22] http://product.caenet.cn/Uploadfiles/12872437250986625020081129090050986.pdf

[23]http://www.ansys.com/Products/Workflow+Technology/ANSYS+Workbench+Platform/ANSYS+Meshing/Features

[24]http://www.ansys.com/Products/Workflow+Technology/ANSYS+Workbench+Platform/ANSYS+Meshing