Computational Fluid Dynamics Study of an Air Cyclone in Reduced Pressure and Gravity

Brad Frye

Department of Physics, Carthage College, Kenosha, WI

7 May 2010

Abstract

We investigate the performance of a small air cyclone under varying gravity and pressure conditions, using a computational fluid dynamics (CFD) program. We simulate dust loading in the cyclone and calculate filtration efficiency as a function of particle diameter, under different conditions of ambient pressure and external gravitational field, and we study the effects of varying ambient pressure on the pressure drop across the cyclone. We also utilize a theoretical model for calculating pressure drop for our cyclone, and we compare it against simulation data to show that it accurately predicts pressure drop.

Contents

1 Introduction

1.1 Background

Cyclone filters are devices which remove particulate matter from air that is passed through the filter. The process of filtration is depicted in Fig. 1. Dusty air is introduced to the cyclone through the tangential inlet, and the air flows in a downward spiral due to the cyclone body's cylindrical shape. The air reverses direction in the conical section, and exits the cyclone through the vortex finder outlet. Dust particles that move outward to the inside wall are trapped in the lower velocity boundary flow, and eventually move downward and exit a dust outlet for collection. Particles can be captured due to the centrifugal and inertial forces which act on them; the inertial filtration is thus more effective for particles of greater mass. Cyclone filters are generally characterized by their geometry and the particle size that can be filtered 50% of the time, known as the d_{50}. Cyclone filters are commonly used for the abatement of particulate air pollution in agricultural and industrial facilities.

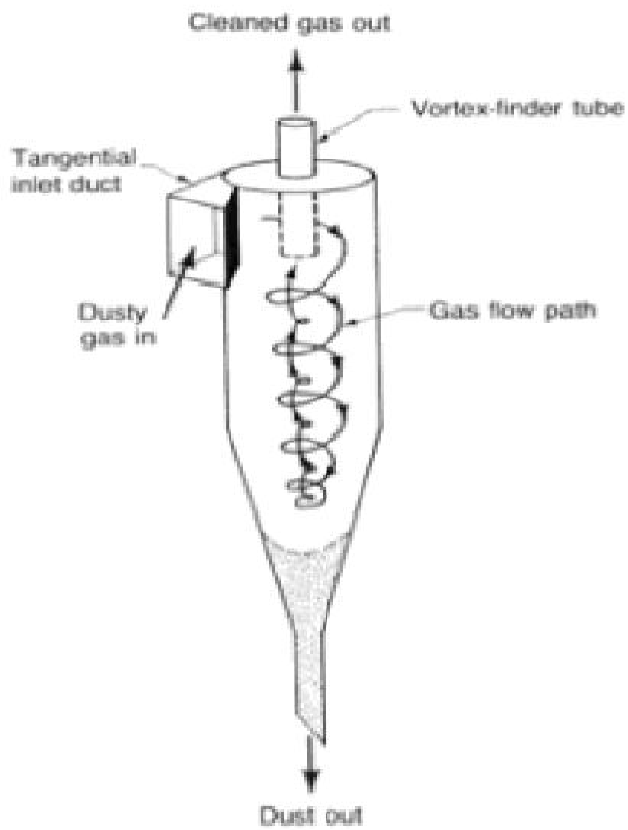

Figure 1: Basic function of a cyclone filter

Cyclonic filtration has been proposed as an effective method for mitigation of lunar dust in future lunar habitats. The lunar surface is covered with a very fine dust, known as lunar regolith, which is formed by high velocity micrometeorite impacts. The dust is very fine, ranging from 50 microns to submicron diameters; the distribution of lunar dust particle sizes is shown in Fig. 2. The particles have jagged edges from the formation process, known as comminution. In addition, the dust carries electrostatic charge, due to the ionizing UV and X-ray radiation imparted on the dust from the sun during the lunar day [Carrier, 2005].

Astronauts that landed on the moon in the Apollo program experienced a few problems due to the dust. Due to the abrasiveness and fineness of the regolith, it is damaging to machinery and spacesuit fiber. Its electric charge makes it readily cling to anything, covering the astronauts to introduce dust into the Lunar Lander on their spacesuits. After the moonwalk of the Apollo XVII mission in 1972, astronaut Harrison Schmitt experienced minor hay fever-like respiratory symptoms and eye irritation due to airborne dust in the Lunar Lander cabin.

While short term exposure did not cause serious health problems, breathing the dust over a longer period could be more hazardous. Since lunar dust contains very fine silicates, it is possible that long term inhalation could cause silicosis, a lung disease common to miners on earth who breathe similar substances. Another concern is the possibility of reactivity of the dust in the pressurized habitats. Lunar dust is subject to high energy conditions such as solar wind, cosmic radiation, and

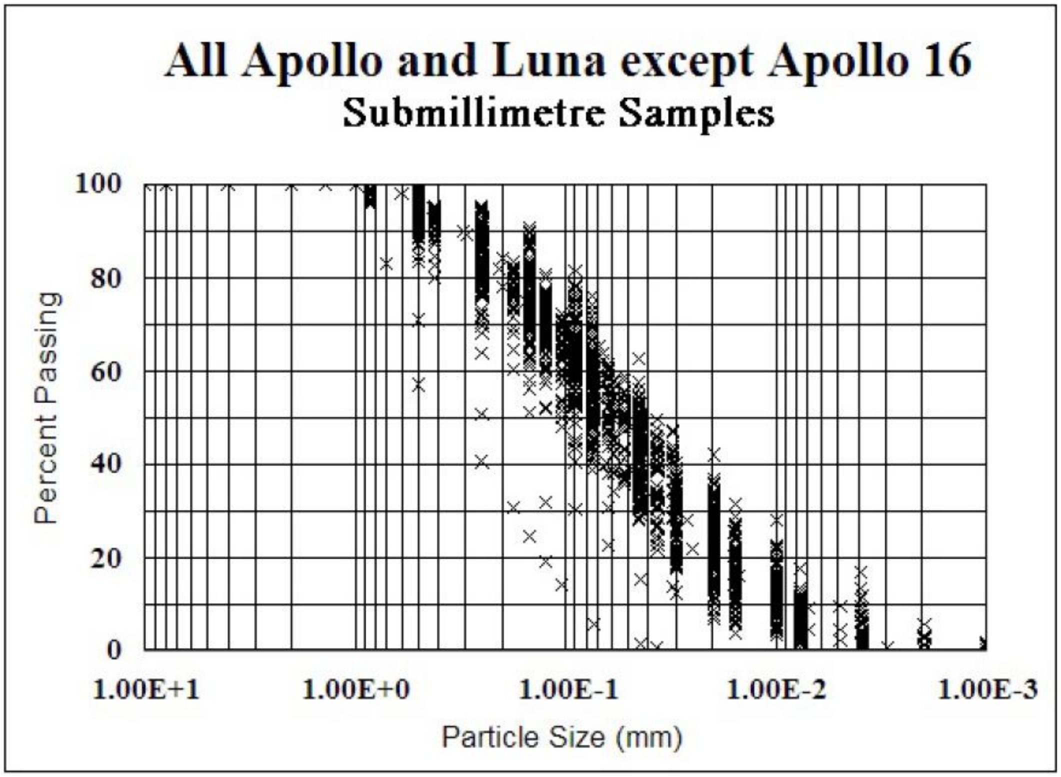

Figure 2: Particle size distribution of lunar dust samples from Apollo/Luna missions [Carrier, 2005]

micrometeoroids, producing excited energy states in the particles, which are retained in the presence of the moon's ultra high vacuum. For this reason the dust may become chemically reactive in the humid pressurized environments, which could be extremely hazardous to the crew members [NASA GRC, 2008].

A 2.5-day workshop hosted by NASA Glenn Research Center entitled "NASA Lunar Dust Filtration and Separation Workshop" was held to discuss the main technological challenges faced by NASA in overcoming the lunar dust problem. Improved lunar dust filtration capability was prioritized as mission critical to all future lunar missions. Participants of the workshop compared current state-of-the-art filtration technologies with predicted dust-loading scenarios, and devised possible filtration systems to be used on the Altair Lunar Lander, and possibly lunar outpost habitats and pressurized rovers.

The proposed solution is a multistage system: The two main stages are the pre-filter and high efficiency filter. The pre-filter should remove the bulk of the larger particles to reduce the load on the high efficiency filter, which will remove the ultra fine particles. The proposed bulk filtration system is a three stage system, which consists of a screen, inertial separator, and sintered metal filter (the screen and sintered metal filter are both porous filtration media.) A list of recommended features for the bulk filtration system was produced:

1. The ability to capture 100 percent of particles larger than 20 micron, and most of the particles down to 5 micron
2. No expandable filtration media
3. Extended (5-year) operational life
4. Less than 249 Pa (1-inch H_2O) pressure drop
5. Constraints for weight, power, and volume (to be determined)
6. Flow requirements (to be determined)
7. Minimal maintenance

The use of a cyclone filter easily satisfies some of these conditions: It is simply a piece of metal, with no moving parts or filtration media, so the second and third criteria are satisfied. Although the fifth criterion is not specific, cyclones can be made small and lightweight, while still maintaining modest filtration efficiency. Finally, cyclones are very low-maintenance, only requiring that the dust is removed from the dust cup periodically. Additionally, cyclone filters can handle a relatively heavy dust load, compared to other filtration methods [NASA GRC, 2008].

The requirements for minimum filtration efficiency and maximum pressure drop help to constrain the design of the cyclone. In general, increasing the efficiency of a cyclone comes at the cost of an increase in pressure drop [Faulkner *et al.*, 2006]. The optimal cyclone filter for this implementation should have high efficiency for particle sizes between 5 and 20 microns, with a pressure drop well below 1-inch H_2O.

1.2 Research Objectives

The focus of this research is a better understanding of the efficiency and pressure drop of a cyclone, under conditions that will be seen on the moon. This thesis is a continuation of research carried out in the summer of 2008, the purpose of which was to verify experimental data found in the Systems Engineering Educational Discovery (SEED) Program, which took place in the spring of 2008 [Pennington *et al.*, 2008]. This experiment attempted to answer the question posed by our NASA SEED advisor, Dr. Juan Agui: What effect does gravity have on the filtration efficiency of an inertial separator? The result of our ground (earth gravity) and microgravity tests showed that gravity did not significantly affect the efficiency of our cyclone.

In order to further understand these results, additional research was carried out over the summer, in a Summer Undergraduate Research Experience (SURE) at Carthage College [Crosby *et al.*, 2008].

We created a model of our previously tested cyclone in CFDesign [Blue Ridge Numerics, Inc.], and ran air flow simulations under different gravity conditions. Using massed particle traces characteristic of the mass distribution of our lunar simulant, we calculated fractional efficiencies of the cyclone. The results agreed well with our experimental findings, showing that different gravity conditions had negligible effects on efficiency. In addition, we developed a simple theoretical model of cyclonic filtration. Looking at forces incident on a particle entrained in airflow, we showed mathematically that the probability of the particle being captured is independent of the gravitational force.

This project was conducted in collaboration with Juan Agui and Jeffrey Mackey, researchers for the ISRU (In Situ Resource Utilization) program at NASA Glenn Research Center (GRC). We have previously worked with Dr. Agui, in two consecutive SEED projects, and our CFD model made for the SURE research project helped to support his experimental research. Agui's colleague Jeffrey Mackey is currently designing a reduced pressure inertial filtration experiment to be flown on a reduced gravity aircraft, and he has proposed that we carry out computational simulations of reduced pressure filtration to support his research.

Our previous work characterized our cyclone under normal ambient earth pressure, approximately 14.7 psi (740 Torr). Since the ultimate goal in characterizing cyclonic filtration at reduced gravity is to utilize the technology in lunar habitats, the conditions must match those of the planned lunar base. The proposed lunar habitats have ambient pressures of 8.3 psi (430 Torr), and airlock pressures of 4.0 psi (207 Torr). The goal of this research is to model air cyclone filtration in varying pressures as well as varying gravity fields, in order to answer the following questions: How does the pressure affect the filtration efficiency? What are the effects of a changing ambient pressure on the flow characteristics (such as pressure gradient, vorticity, etc.) of the cyclone? Does gravity have a greater effect on efficiency at lower pressures?

2 Previous Work

Our 2008 SEED experiment was the first to collect and publish data on inertial filtration in lunar gravity, and, if successful, Jeffrey Mackey's experiment will be the first to find data on cyclonic filtration in reduced ambient pressure and gravity. This thesis builds on these research projects, using their findings as points of comparison in order to further develop an understanding of cyclonic filtration, in the context of lunar environment conditions. In particular, the 2008 SURE project is essential, due to its similarity to this study in terms of research methods. The results found in these projects are summarized in Fig. 3.

Fortunately, a large amount of computational, experimental, and theoretical research has been conducted on the properties of small air cyclones. In designing a filtration system with a cyclone, it is important to have a useful way of determining the effectiveness and cost of using the cyclone fileter. For this reason, several empirical cyclone models have been developed which can predict the collection efficiency of a cyclone, which describes how effectively cyclones filter particulate matter. The collection efficiency is expressed as a fraction of collected particles divided by the

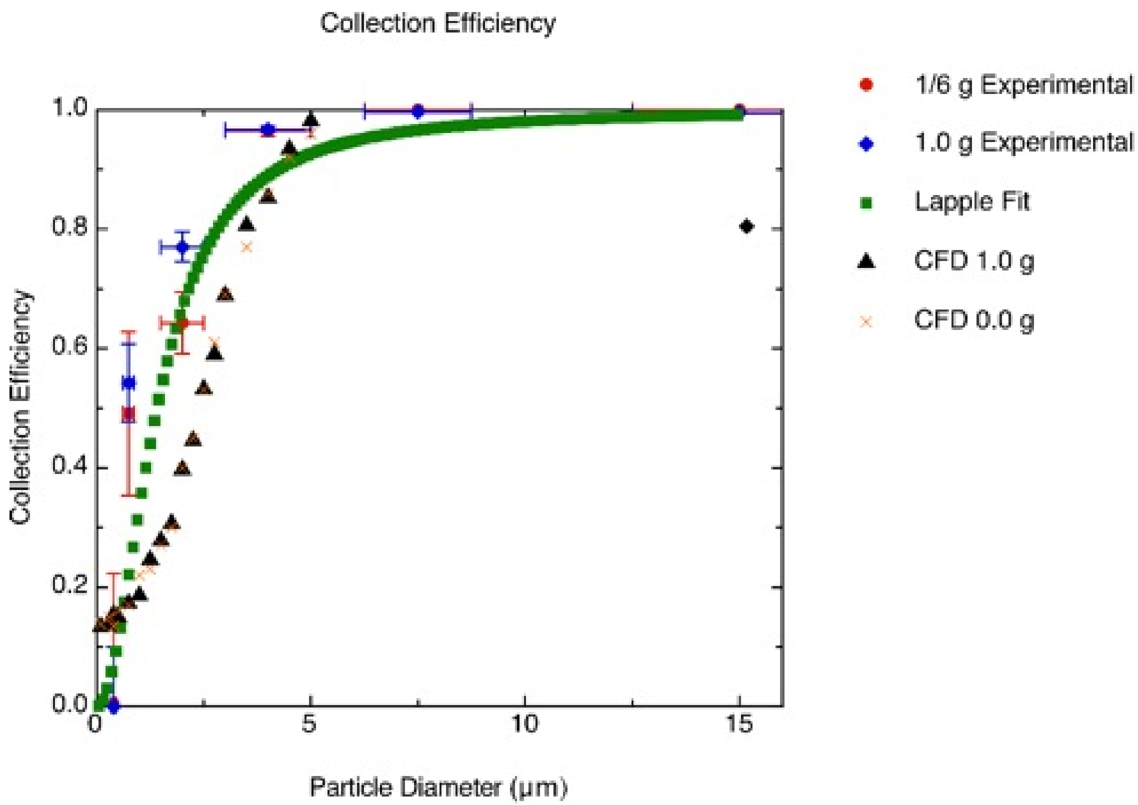

Figure 3: Results from 2008 SEED and SURE research. Shown are the experimental results (● and ⋄), Lapple curve (■), and computational results (▲ and ×). See Equation 1 for the definition of cyclone collection efficiency [Pennington *et al.*, 2008, Crosby *et al.*, 2008]

total number of particles introduced into the cyclone; the equation for efficiency is:

$$\epsilon = \frac{N_{collected}}{N_{total}} = \frac{N_{total} - N_{escaped}}{N_{total}}. \tag{1}$$

In measuring the efficiency of a cyclone, a certain number (N_{total}) of particles is initially placed into the system, and the researcher must count either the number of particles that are captured ($N_{collected}$) or the number that exit through the cyclone's upper outlet ($N_{escaped}$). The first and most common empirical model used to predict cyclone efficiency is the Lapple model, a model which was developed by Shepherd and Lapple based on experimental cyclone research. We used the Lapple model to plot an efficiency curve for our cyclone parameters in our SEED research, as shown in Fig. 3. Empirical models are useful because we can see how different parameters affect the efficiency of the cyclone. The Lapple model uses Eqs. 1-3 to calculate the efficiency of a cyclone; the equation to determine the cyclone's characteristic d_{50} is:

$$d_{50} = \sqrt{\frac{9\mu W}{2\pi N V_i(\rho_p - \rho_g)}}. \tag{2}$$

Here d_{50} is the diameter of a particle that will be collected 50% of the time, μ is the dynamic viscosity of air, W is the inlet width, N is the effective number of turns inside the cyclone, V_i is the inlet velocity, ρ_p is the density of the dust, and ρ_g is the density of air. The Lapple model defines the number of turns in the cyclone by:

$$N = \frac{1}{W}(L_{cylinder} + \frac{L_{cone}}{2}). \tag{3}$$

Here $L_{cylinder}$ is the length of the cylindrical region of the cyclone, and L_{cone} is the length of the conical section (for the geometry of the cyclone used in our research, refer to Fig. 4). Particles with diameter less than d_{50} will be collected less than 50% of the time, so a smaller d_{50} corresponds to a more efficient cyclone. Therefore, according to this model, increasing the inlet velocity, number of turns, or particle density will increase efficiency, as will decreasing the inlet width, air viscosity, or air density. The efficiency of the cyclone at any particle diameter can be extrapolated by:

$$\epsilon_i = \frac{1}{1 + (d_{50}/d_i)^2}. \tag{4}$$

Here ϵ_i is the fractional efficiency of a particle i, and d_i is the diameter of particle i. From this equation, an efficiency curve can be plotted for a given cyclone geometry.

While the Lapple efficiency is considered the standard cyclone model and is simple to use, several studies [Faulkner *et al.*, 2006, L. Wang et al., 2006] have found that it give inaccurate results. Faulkner and Shaw (2006) determined experimentally that efficiency did not vary significantly over a range of inlet velocities; they concluded that the inlet velocity should be set as low as possible to achieve the desired efficiency, since an increase in velocity will give only a small increase in efficiency for a large increase in pressure drop. Since the Lapple model efficiency is simply calculated, it will still be compared against the efficiencies found by simulation, even though it is approximate.

Another important property of a cyclone is its pressure drop, which describes the energy cost of using a cyclone filter. Several empirical and theoretical models have been developed to predict the pressure drop of a cyclone, given its flow conditions; Lapple and Shepherd also developed a simple empirical model for pressure drop, but more recent models have shown closer agreement with experimental findings.

L. Wang *et al.* (2006) stated that the Lapple pressure drop model and other empirical models were unsatisfactory due to inaccuracy or complexity, and they developed a new theoretical model to predict pressure drop. Their model, which calculates the pressure loss as the sum of five individual sources of pressure loss, was shown by experiment to be more accurate than several other models. Their model is still much more complex than the Lapple model, though, requiring the knowledge of both inlet and outlet velocities, as well as inner cyclone tangential and axial velocities, and the length traveled by the air through the cyclone. The analysis focuses on a few standard cyclone geometries, deriving dimensionless variables which depend on geometry; the equations for these quantities are not provided, so we are unable to use this model.

Karagoz *et al.* (2005) also developed a theoretical model for calculating a cyclone's pressure drop. It also showed close agreement with experimental findings, and is relatively simpler than the Wang model. This model considers a tube of flow which moves through the cyclone, and the pressure drop is dependent on the interactions and geometrical deformations of this tube. The full calculation of pressure drop uses at least ten equations and more than twenty terms; for brevity, the main equations are shown only, and the end result is given.

The main result of this study is shown in the following equation:

$$K = \frac{R_0}{\alpha - R_0}((1-\alpha)^{-2(1-\frac{R_0}{\alpha})} - 1). \tag{5}$$

Here α is a dimensionless variable which is dependent on the cyclone geometry, R_0 is a dimensionless quantity that depends on frictional conditions of the cyclone's flow, as well as the geometry, and K is a dimensionless value known as the pressure loss coefficient. The value K describes the pressure drop of a cyclone compared against the air density and inlet velocity, as such:

$$K = \frac{2\Delta p}{0.5\rho_{air}V_0^2}. \tag{6}$$

Here Δp is the pressure difference across the cyclone, and V_0 is the speed of the air at the inlet. Using the set of equations given in the study, we calculated the constants α and R_0, and then found the pressure loss coefficient K. Using our cyclone geometry and flow condition values, we found K to be 2.303. Then, solving for Δp, we calculated the pressure drop to be 1.043-in. H_2O. We use this value as a comparison against the value found by simulation to check that the simulation is producing accurate flow. Interestingly, this model shows that a change in ambient pressure will cause some variation in pressure drop. For a given value of K, the pressure drop will decrease with ambient pressure, due to the dependence on air density, which will decrease in decreased pressure. Additionally, K will actually show a slight increases with reduced pressure, due to the relationship of the R_0 value with kinematic viscosity, which is inversely proportional to pressure. This is a very weak mathematical dependence; we believe that the pressure drop's dependence on density will dominate the dependence on viscosity, resulting in a positive correlation with ambient pressure, or a reduction in pressure drop at reduced pressure conditions [Karagoz et al., 2005, Landau *et al.*, 1987, Benenson *et al.*, 2006].

3 Method

3.1 Fluid Mechanics Governing Equations

Problems in fluid mechanics involve solving for the velocity of the flow, or flow field. As in other fields of physics, the fundamental equations of fluid mechanics are either conservation equations or special forms of Newton's laws. One of these equations is the continuity equation:

$$\frac{\delta \rho}{\delta t} + \rho(\nabla \cdot \mathbf{v}) = 0 \tag{7}$$

Here ρ is the density of the fluid, and $\mathbf{v}$ is the velocity vector field. This equation represents the conservation of mass: an increase/decrease of mass at a point in space must equal the mass moving spatially in or out of that point. The flow field can be solved for using Euler's equation:

$$\rho \left[\frac{\delta \mathbf{v}}{\delta t} + (\mathbf{v} \cdot \nabla)\mathbf{v} \right] = \mathbf{F} - \nabla p \tag{8}$$

This equation is the equivalent of Newton's second law for fluids. The expression in brackets on the left side of the equation is the substantive derivative of velocity, meaning that the velocity derivative of a small volume of fluid is taken as it moves through space. The values on the right side are the forces on the fluid: $\mathbf{F}$ represents any body forces on the fluid, such as gravity, and the gradient of pressure ∇p is the pressure force per unit volume.

The continuity and Euler equations can be used to solve for the fluid's velocity field, given a set of boundary conditions. The boundary conditions in fluid mechanics problems are flow conditions, such as pressure, or the flow rate of the fluid, and the geometry of any solid bodies in the flow. Interesting problems usually involve fluids flowing through or around solid objects.

The Euler equation describes ideal flow, meaning that it is incompressible and frictionless. Real flow must account for internal friction of the fluid, which requires the incompressible form of the Navier-Stokes equation:

$$\rho \left[\frac{\delta \mathbf{v}}{\delta t} + (\mathbf{v} \cdot \nabla)\mathbf{v} \right] = \mathbf{F} - \nabla p + \eta \nabla^2 \mathbf{v} \tag{9}$$

Here η is the dynamic viscosity (g/cm s), and the new term is the frictional force. Viscosity is the measure of a fluid's resistance to flow; the dynamic viscosity is a value which is independent of pressure (as opposed to the kinematic viscosity μ, which is inversely proportional to pressure.) Along with the continuity equation, the Navier-Stokes equation accurately describes turbulent, incompressible flow of a Newtonian fluid [Landau *et al.*, 1987, Benenson *et al.*, 2006]. However, being a nonlinear coupled differential equation, it is very difficult even to find a numerical solution for sufficiently complex flows; this is why we resort to computational methods to model the flow behavior of a cyclone. Computational fluid dynamics programs generally find an approximate solution to the Navier-Stokes equation by taking its time average, which is then called the Reynolds Averaged Navier-Stokes equation.

In taking the time average, an additional apparent stress term called the Reynolds stress emerges. This makes the equation unsolvable: There aren't enough equations for the new unknowns, so the mathematical model is not closed. While it is possible to produce more equations to determine

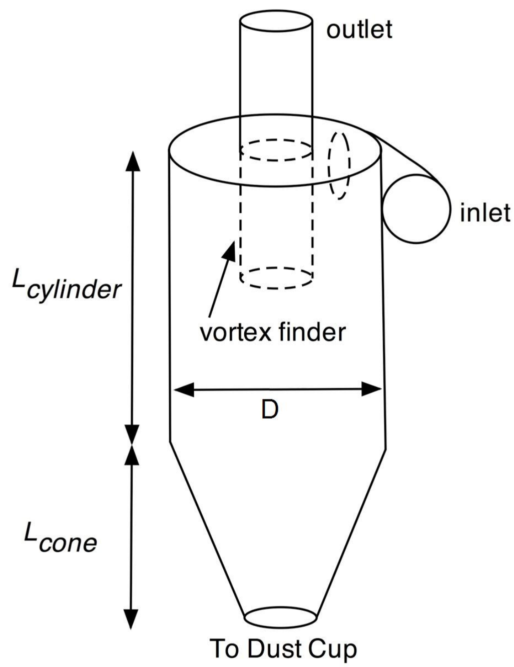

Figure 4: The geometry of the cyclone filter used in this study, the CMPI cyclone. The inlet, outlet, and vortex finder diameter is $1.0in$. The diameter of the cylinder is $2.0in$, the cylinder length $L_{cylinder} = 8in$, and the cone length $L_{cone} = 2in$

these unknowns, more unknowns are created in that process, which can continue ad infinitum; this is known mathematically as the closure problem. The solution to the problem is to utilize one of many turbulence models, which provide definitions of the unknown terms, often by applying ad hoc constants to the equations. The different turbulence models should be applied strategically to appropriate flow scenarios; this usually involves some trial and error by the user to find the right fit [Blue Ridge Numerics, Inc., Boas, 2006].

3.2 CFD Workflow

Several studies [Kaya *et al.*, 2008, B. Wang *et al.*, 2003, Gimbun et al., 2005] have found that computational fluid dynamics models can accurately predict the flow of cyclone filters. However, each of these studies referred to the commercial CFD program Fluent 3D, which is considered one of the more robust CFD's available. We chose to use CFDesign, which has the advantages of having a relatively low cost and a user interface that is easy to learn, plus the fact that we have experience using it from past research. Unfortunately, CFDesign is not as advanced as Fluent, so the flow solution may not be as accurate as those found in the studies.

CFDesign solves the flow of a system using the finite element method. To analyze a 3-dimensional object, the program requires the user to import a 3D Computer Assisted Drawing (CAD) from one of several 3D CAD programs; we used the CAD software SolidWorks [SolidWorks, Inc.]. The

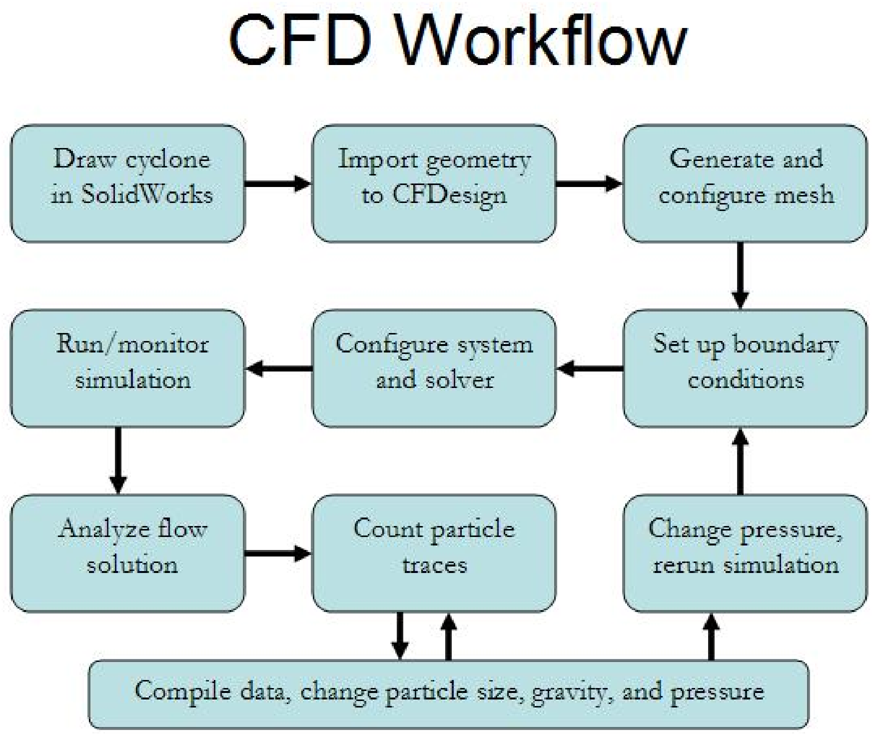

Figure 5: Diagram of CFD workflow

geometry of the cyclone filter analyzed is shown in Fig. 4. Our geometry was provided by our collaborator Jeffrey Mackey, who is using the same geometry cyclone for his experiments; the geometry is called the CMPI cyclone, named after the company that produced it. A diagram of the CFD Workflow is shown in Fig. 5.

Using the 3D geometry CFDesign imports from the CAD program, the CFD automatically generates a mesh to best approximate the shape of the object. The purpose of the mesh is to create a system of nodes three dimensionally about the object; the program will solve fluid dynamics equations at each node to find some flow variable at that point. The solver uses an "upwind advection scheme" to apply dependent relations to adjacent elements, and create a solution at each time step in the form of a field of dependent variables. The program allows the user to adjust the quality of the mesh by increasing or decreasing the number of nodes at any point. Having a greater amount of nodes will better approximate the actual flow in that region, but it will also increase the amount of processing the solving requires, which will quickly make the process very computationally intensive. The user should strategically alter the mesh quality of the object to get the most accurate flow without making the simulation take too long to solve; for example, a region with complex, turbulent flow should be given a finer mesh to best approximate the small scale fluctuations in velocity that are present, while a region with steady, laminar flow should have a coarser mesh applied to avoid wasting computational resources. The user also decides whether the flow is compressible or incompressible. Compressible flow occurs at very high flow velocities; if the velocity is less than one third of 1 mach, or about 113 m/s, then it can safely be assumed that the flow is incompressible [Benenson *et al.*, 2006]. Our flow is about 10 m/s, so it is incompressible, which greatly simplifies the calculations.

The next step in the process is establishing the boundary conditions. This mostly means specifying the flow conditions of the object's inlets and outlets. At each opening, the user can indicate a specific flow velocity, gauge pressure, mass flow rate, or volume flow rate. Giving the program the correct boundary conditions is a crucial step in the simulation, and it is also the most difficult. Making the choice of flow velocity, mass flow, or volume flow, can be confusing as can or specifying whether a certain outlet has a flow condition, pressure condition, both, or no condition. For an inexperienced user, it requires a great deal of accurately characterizing the flow. This step is also where the user specifies the materials of each part of the object, for example if a device is made of steel or aluminum, or selecting from a wide variety of gases or liquids for the flowing medium. There are many other ways the user can customize the system in this step, such as specifying heat sources, temperature gradients, porous materials, fan blades, and so on; but we do not require any of these options, at least for now.

After setting up the boundary conditions, the user must input their specifications to the solver before running the simulation. Setting the number of iteration steps is important: not iterating the adequate number of times will not fully solve the flow, but too many will take an excessively long time, and possibly lead to round-off error. The user will then specify if the solution should be steady state or transient. A transient solution is selected if there is some time-variant aspect of the system, such as a rotating fan blade, or an inlet condition that is chosen to vary with time. These situations usually take longer to solve, and often require some sort of special solving technique to get it to converge. Our system will have a steady state solution, meaning that none of the flow parameters are changing through further iterations, because we do not require time variance in any way. Another important option is which turbulence model to apply to the flow. The optimal turbulence model for our highly swirling flow is the Reynolds Stress Model (RSM) [Kaya *et al.*, 2008]. Unfortunately, RSM is not available in CFDesign; we must use the RNG k-e model, which does not produce accurate flow since it assumes isotropic turbulence [Kaya *et al.*, 2008, B. Wang *et al.*, 2003]. The RNG k-e model uses renormalization group theory to calculate the turbulent kinetic energy and turbulent dissipation rate [Blue Ridge Numerics, Inc.].

Once all of these steps have been completed, CFDesign is ready to run the simulation. The time it takes the solution to converge is dependent on many factors, such as mesh quality, turbulence model, boundary conditions, and number of time steps; a quick run would be about twenty minutes, but it may take an hour or two for a very fine mesh, or the solution may diverge, meaning that it could not find a steady state solution, and the user must reassess the system. After training ourselves thoroughly in the program, we consistently produced converging solutions. When the simulation is running, the user may monitor the solver. A large array of flow characteristics is plotted versus the number of time steps completed on a graph, such as velocity vector components, kinetic energy, pressure, and turbulence. If the solution is converging, the curves will eventually flatten out, meaning that they are keeping steady over successive time steps. A divergent solution will have some of the characteristics accelerating instead of becoming steady, which will result in an inaccurate flow. Monitoring the solution can save the user time, by canceling a diverging

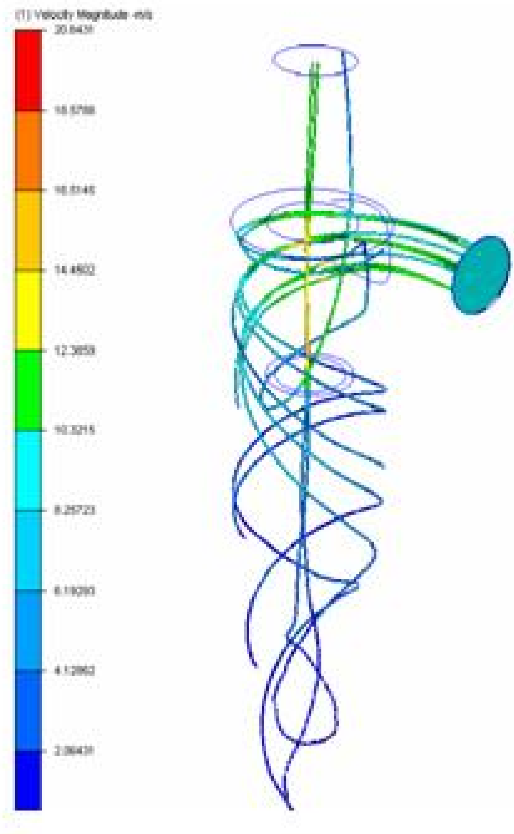

Figure 6: The flow behavior of ten dust particles is simulated. The traces are color-coded by velocity magnitude. These traces were given the mass of 0.5 micron regolith particles, under 1g acceleration. Three of the particles were not captured. [Blue Ridge Numerics, Inc.]

solution, or accepting a solution early if the number of time steps was overestimated.

Once the simulation is complete, the user may analyze the flow with a wide variety of tools. The analysis method we used to calculate our efficiency data was massed particle traces, as shown in Fig. 6. The particles were given density equal to that of lunar regolith, with diameters ranging from 0.1 micron to 10 micron. We chose this range of particle diameters because the Lapple model predicts these sizes to be filtered with nearly 0% and 100% efficiency, respectively; our simulations showed similar results. To calculate the efficiency for a particular set of traces, we counted the number of particles that escaped through the outlet, subtracted from the number of traces input to find the number that were collected, and divided by the number of traces input. The effective gravitational force on the particles is selected before each trace; we compiled the efficiency data for the aforementioned particle size distribution in both 0 g and 1 g gravitational accelerations. Data is taken in ten trials for each particle size for a given external gravity, in a given pressure condition; after all of the particle counts are taken for the pressure condition, a new simulation is run at the next pressure, and particle count data are taken again. Efficiency data is taken at pressures of 740 Torr, 430 Torr, and 207 Torr.

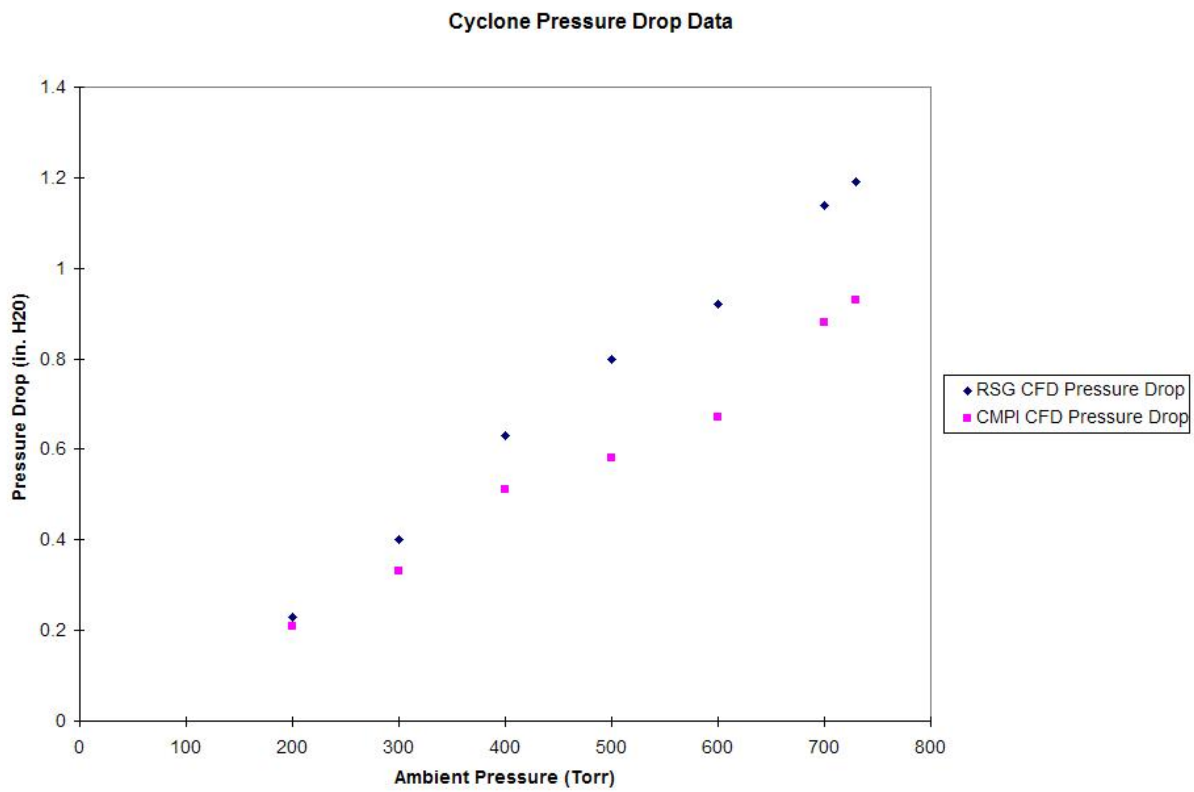

Figure 7: Results of the cyclone fluid dynamics calculations for pressure drop plotted against ambient pressure. The CMPI cyclone data are shown in pink, and the RSG data are shown in blue.

4 Discussion

4.1 Results

The efficiency of our cyclone over a range of particle sizes and varying pressures and gravities was calculated using CFDesign; unfortunately, our efficiency data were completely anamolous, and are unusable. CFDesign was unable to accurately simulate massed particle flow traces; the particles did not lose energy upon contact with the cyclone walls, when they were supposed to lose energy as determined by the coefficient of restitution parameter. This is caused by a known bug which did not exist in previous versions of CFDesign. This was the main cause of our aberrant data, although another possible contributing cause is the incapability of CFDesign to accurately produce the highly turbulent, swirling flow in our cyclone. The study conducted by Kaya *et al.* (2008) showed that the RNG k-epsilon turbulence model failed to produce accurate flow in a cyclone, which was the most theoretically sound model provided by CFDesign. Additionally, B. Wang *et al.* (2003) stated that the number of particle traces needed to produce statistically significant data is up to 300,000 traces. We seeded 100 traces for each diameter size at each gravity and pressure condition, and repeated for ten trials; thus we had 1000 traces per data point. The counting process in CFDesign is time consuming and labor intensive: The time spent taking efficiency data for this research was approximately 15 hours, without considering time taken to set up the simulations. We were unaware of the particle trace bug at the time, although counting 300,000 particle traces manually would not have been feasible.

We also calculated the pressure drop across the cyclone at each pressure condition; the data is summarized in Fig. 7. In addition, we tested pressure drop across the cyclone geometry from our 2008 SURE research, which is termed the RSG cyclone; this cyclone is identical to the CMPI cyclone, except with a shorter cylindrical section. The calculated pressure drop for our CMPI cyclone is 0.93-in. H_2O, and decreases linearally as external pressure is reduced. This value is close to the pressure drop as predicted by the Karagoz theoretical model, which was 1.043 in. H_2O; the reduction in pressure drop with reduction of ambient pressure confirms our predictions. The pressure drop accross the RSG cyclone has a slightly higher value at each pressure condition, and shows similar inclination.

The main result of this research is that accurate collection efficiencies could not be found with CFDesign, due to insurmountable limitations of the program. The pressure drop data agrees well with the model we used, but that data may also have suffered from inaccuracy due to the inappropriate turbulent model. Our data does lead to some promising conclusions in the context of lunar habitat applications of cyclone filters. Our efficiency curves from our previous work show that 10 micron lunar dust particles are collected at 90% or greater efficiency, for all gravity conditions. Regarding the earlier mentioned proposed bulk filtration system and the corresponding list of desired traits, the efficiency data have shown that a cyclone filter could remove a sizable portion of the target particle sizes. Our measured pressure drops exceeded the recommended limit of 1.0-inch H_20 at earth atmospheric pressure, but dropped below that value at lower pressures; when the system is designed, the cyclone geometry and inlet flow rate should be configured to produce the desired pressure drop at the given ambient pressure.

4.2 Future Work

Overall, our data, while certainly aberrant, supports the utilization of cyclones as components of multi-stage filtration systems for lunar outpost habitats of future space missions. At the same time, the development of lunar dust mitigation techniques is critical, and so deserves a more thorough and legitimate analysis. For future work, the cyclone geometry should be analyzed using a higher end CFD program, such as Fluent 3D. Many contemporary cyclone studies using CFD have concluded that the RSM turbulence model is crucial to producing accurate flow. If the study is continued with CFDesign after the particle trace bug is resolved, more accurate efficiency data could be gathered. The data may be improved further if the number of particle traces tested is greatly increased. This can only be done if there were some way to automate the counting; as of this writing, CFDesign does not provide any such methods, and we were unable to design a particle counting program which satisfied our needs.

Once a reliable method of analysing flow in a cyclone is developed, further questions and conditions may be explored. Besides knowledge of efficiency and pressure drop dependence on external pressure and gravity, it would be useful to know how these values change with different cyclone geometries, flow rates, and air temperature. Building a full model which accurately predicts efficiency and pressure drop over all of these parameters would facilitate the optimization of a filtration system which implements a cyclone filter.

5 Acknowledgments

We are grateful to the Wisconsin Space Grant Consortium and Carthage College for financial support of the SEED and SURE research projects. We would like to acknowledge the contributions of Juan Agui at NASA Glenn Research Center for the conception of the original experiment, and Jeffrey Mackey for initiating the exploration of cyclone performance in reduced pressure. The author is also grateful to the natural science department and physics faculty for their support of this work.

References

[Benenson *et al.*, 2006] Benenson, Walter, Harris, John W., Stocker, Horst, and Lutz, Holger. Handbook of Physics. Spring Science, Business Media, Inc. (2006).

[Blue Ridge Numerics, Inc.] Blue Ridge Numerics, Inc., 650 Peter Jefferson Place, Suite 250, Charlottesville, VA 22911.

[Boas, 2006] Boas, Mary L. Mathematical Methods in the Physical Sciences. Third Edition. John Wiley & Sons Inc. (2006).

[B. Wang *et al.*, 2003] Wang, B., Xu, D. L., Xiao, G. X., Chu, K. W., Yu, A. B. Numerical Study of Gas-Solid Flow in a Cyclone Separator. Third International Conference on CFD in the Minerals and Process Industries. 10-12 December 2003.

[Carrier, 2005] Carrier, W. David. Geotechnical Properties of Lunar Soil. Lunar Sourcebook. September 2005. http://www.lpi.usra.edu/lunar/surface/carrier_lunar_soils.pdf.

[Crosby *et al.*, 2008] Crosby, Kevin M, Frye, Brad. Computational Fluid Dynamical Model of a Cyclone Separator in Microgravity. http://www.carthage.edu/dept/physics/cyclone-CFD.pdf (2008).

[Faulkner *et al.*, 2006] Faulkner, W. B., Shaw, B. W. Efficiency and Pressure Drop of Cyclones Across a Range of Inlet Velocities. Applied Engineering in Agriculture. 2006

[Gimbun et al., 2005] Gimbun, Jolius, Chuah, T. G., Choong, Thomas S. Y., Fakhru'l-Razi, A. A CFD Study on the Prediction of Cyclone Collection Efficiency. International Journal for Computational Methods in Engineering Science and Mechanics, Vol. 6, Issue 3, pgs. 161-168. July 2005.

[Karagoz et al., 2005] Karagoz, Irfan, Avci, Atakan. Modelling of the Pressure Drop in Tangential Inlet Cyclone Separators. Aerosol Science and Technology, Issue 39, pgs. 857-865. August 2005.

[Kaya *et al.*, 2008] Kaya, F, Karagoz, I. Performance Analysis of Numerical Schemes in Highly Swirling Turbulent Flows in Cyclones. Current Science, Vol. 94, No. 10, 25 May 2008.

[Landau *et al.*, 1987] Landau, L. D., Lifshitz, E.M. Fluid Mechanics. Pergamon Books Ltd. 1987.

[L. Wang et al., 2006] Wang, L., Parnell, C. B., Shaw, B. W., Lacey, R. E. A Theoretical Approach for Predicting Number of Turns and Cyclone Pressure Drop. Transactions of the ASABE. 2006.

[NASA GRC, 2008] NASA Glenn Research Center: Lunar Filtration and Separations Workshop. Ohio Aerospace Institute, Cleveland, Ohio, 2008.

[Pennington *et al.*, 2008] Pennington, C., Martin, E. Sorensen, E., Fritz, I., Frye, B., Crosby, K. M., Inertial Filtration in Lunar Gravity: SEED Project Report: http://www.carthage.edu/dept/physics/flight/Final_Report.pdf (2008).

[Sheperd *et al.*, 1939] Shepherd, C. B. and Lapple C. E. 1939. Flow Pattern and Pressure Drop in Cyclone Dust Collectors. Industrial and Engineering Chemistry. 31(8): 972-984

[SolidWorks, Inc.] SolidWorks Corporation, 300 Baker Avenue, Concord, MA 01742.