Understanding Rocket Aerodynamics

Learning Objective	In this lesson the student will be introduced to aerodynamics. The drag formula and laminar and turbulent flow will be discussed. Rocket design will also be covered.
Grade Level	10 – 12

– Introduction –

The advantages of studying aerodynamics are numerous. From the design of new vehicles to the understanding of how a ball, puck or shuttlecock moves in sports.

Aerodynamics and The Basics of Drag

Figure 1 – High aerodynamic drag as demonstrated by a hand turned sideways.

We will start the discussion on drag and how it can be calculated. Drag is the resistance that is caused when an object moves through a fluid such as air or water known as aerodynamic or hydrodynamic drag respectively.

An easy way to think about aerodynamic drag is to picture yourself driving down a highway with the windows open. If you were to stick your hand out the window such that your palm is fully exposed and your hand is perpendicular to the road (see figure 1), you would notice a significant force against your hand in the form of the air hitting it. You will notice that this force increases as you go faster. As we will discuss below aerodynamic drag increases by a power of two when velocity increases.

If you were to then turn your hand so that it is parallel to the ground below you will notice a much smaller force on your hand (see figure 2). If air did not have a density you would not feel any force against your hand.

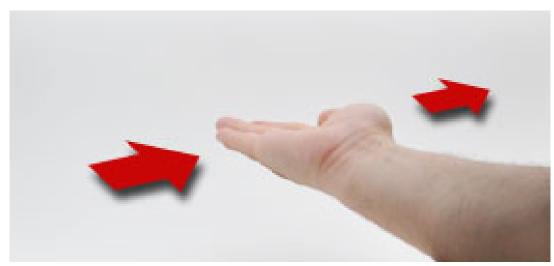

Figure 2 – Low aerodynamic drag as demonstrated by a hand turned facing up.

In fact the air around us has a density of [1.22521 kg/m³](). As a comparison the density of water is [1000 kg/m³]() or almost 1000 times that of air. That is why it is easier to move through air than it is through water.

The Formula for Drag

Years of research has given us the formula for drag which can be applied to air, water and other fluids.

The formula is shown below:

$$F_D = \frac{1}{2} \rho u^2 C_D A$$

Written out the formula would be:

Drag = ½ Mass Density of the Fluid X Velocity² X Drag Coefficient X Reference Area

Thus if we were to think of the example of sticking our hand outside the window of the car we can see that the drag will increase by a power of two with an increase in velocity. As well when we turn our hand to the side we can see that the drag decreases as the area decreases. We can also see that if we increase the Mass Density of the Fluid (ρ) let's say from air to water, the drag increases although by not as great a rate as when the velocity increases.

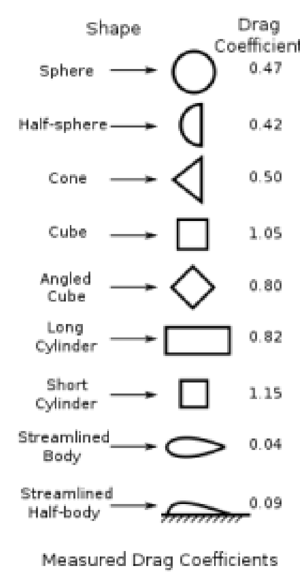

Figure 3 – Drag coefficients for various shapes

The Drag Coefficient

The drag coefficient is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment such as air or water. It is represented by the symbol C_D in the equation above. The drag coefficient for various shapes is seen in figure 3.

As we can see from the diagram, the Streamlined Body shape has a much lower $C_D$than the Cube. This would make sense if you think back to our example of placing our hand outside the window of a moving car. A flat hand against the wind feels a lot more force than a hand turned up parallel to the road.

The C_D of a Model Rocket

A wind tunnel is an excellent tool for measuring drag on a model. Wind tunnels are used in the automotive industry to test new car designs as well as in the aerospace industry to test drag for aircraft.

As well, a wind tunnel can be used to test model rocket designs. An excellent report on this is "Model Rocket Drag Analysis using a Computerized Wind Tunnel" by John S. DeMar. In his report John tests various model rocket designs in a wind tunnel by measuring the drag and then working backwards to determine the C_D of each design.

Parasite Drag and Aerodynamics

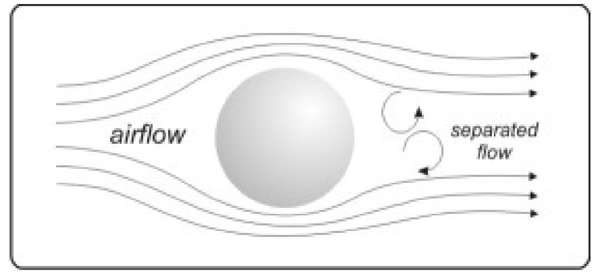

Figure 4 – Aerodynamic drag around a sphere.

In a section above, we discussed the aerodynamic drag felt when one puts their hand out the window of a moving car such that it is perpendicular to the ground. Most of the aerodynamic drag experienced in this case is pressure drag, the difference in air pressure from the front of your hand to the back of your hand. When the hand is turned such that it is parallel to the ground, pressure drag is reduced although still present, and skin friction (surface) drag is created along the surface of the hand. The sum of the pressure drag and skin friction drag is known as parasite drag.

Figure 4 shows airflow around a sphere traveling through the air. As you can see the airflow is broken up once it hits the surface of the sphere and a wake of "separated flow" is created on the other side. This separated flow is caused by the surface drag and the result is pressure drag on the back of the sphere.

If surface drag did not exist, the air flow at the back of the sphere would mirror the front of the sphere as it moved through the air and the separated flow would not exist.

Viscosity

According to Dictionary.com, viscosity is the property of a fluid that resists the force tending to cause the fluid to flow. A fluid with a high viscosity, such as honey, will have a high resistance to movement. Water, for example, has a much lower viscosity and flows relatively easy. Air has viscosity as well and for most applications the viscosity of air can be neglected. For objects moving at high rates of speed such as model rockets, the viscosity of air causes friction and thus affects the movement of the object. It is the viscosity of the air which causes surface drag.

Boundary Layer

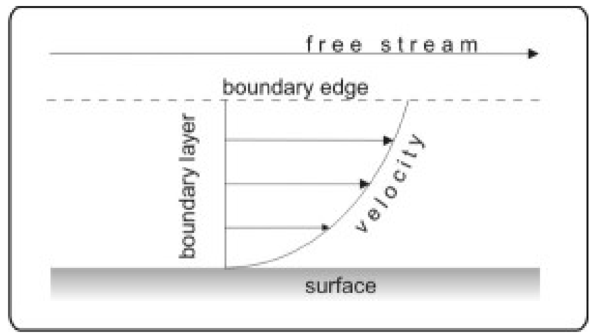

Figure 5 – Boundary Layer

Picture if you will the surface of a moving object and the flow of air over it. Right at the surface the velocity of the air is zero. The velocity starts to climb as the distance from the surface increases to the point where it is equal to the maximum air flow over the object (see figure 2). The distance between these two points is called the “boundary layer”.

Laminar vs. Turbulent Flow

When the air flow is smooth and the velocity rises evenly through the boundary layer, the air flow is known as “laminar flow.” Uneven flow through the boundary layer is called “turbulent flow.” Turbulent flow creates a larger boundary area and thus more drag than laminar flow. The boundary layer will tend to have laminar flow initially as the air moves across or down the object.

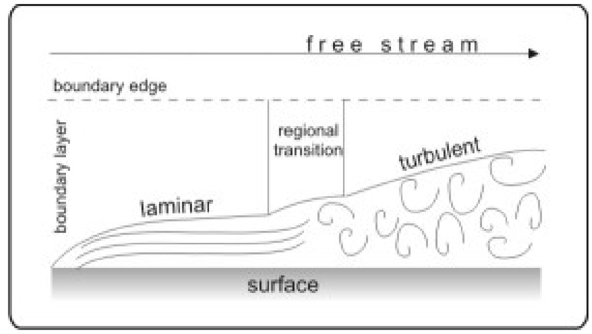

Figure 6 – Laminar vs Turbulent Flow with Reynolds Number

Imperfections such as rough or jagged surfaces increase the chances of air flow turning turbulent through the boundary layer. Distance from the leading edge also contributes to the transition to turbulent flow. This is something we can take into account in our rocket designs. Figure 3 shows air flow in the boundary layer turning from laminar to turbulent.

The Reynolds Number

Osborne Reynolds for which the Reynolds Number was named for, contributed much to the study of fluid dynamics. The formula for determining the Reynolds Number is shown below:

Reynolds Number = (Density X Velocity X Length) / Viscosity

The Reynolds Number may be used to determine whether or not air flow over an object is laminar or turbulent. Reynolds Numbers less than 100,000 indicate laminar flow, more than 1,000,000 indicate turbulent flow and in between these values means that the flow will be in the transition region (could be either laminar or turbulent). The numbers shown in figure 3 illustrate this.

An excellent example of calculating Reynolds Number for a model rocket is shown in the Estes document TR-11 by Dr. Gerald M. Gregorek. Dr. Gregorek uses a 12 in (30 cm) rocket traveling at

100 ft/s (30.5 m/s) in his example. Air density is 0.00238 slugs/ft^3 and air viscosity is 0.00000039 lb sec / ft^2. After substitution the RN number is calculated at 610,000 which puts it in the transition region.

Aerodynamic drag over a model rocket

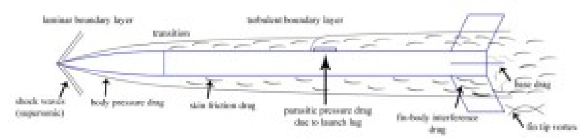

Figure 7 – Airflow over a rocket displaying the drag.

A model rocket is composed of 3 major parts, the nose cone, the body tube and the fins. Each one of these components contributes to the aerodynamic drag of the rocket. Breaking a model rocket down to its basic components and addressing the drag on each component is a good way to start our design. The overall aerodynamic drag on the rocket may be expressed as such:

$$\text{Total Drag} = D_{NC} + D_{BT} + D_F$$

Where D_{NC} is the drag from the nose cone, D_{BT} is the drag from the body tube and D_F is the drag from the fins. We will ignore the drag created from the launch lug for the purposes of this article.

In figure 7 we see aerodynamic drag over a typical model rocket. Observe that the flow over the rocket turns from laminar to turbulent further along the body of the rocket.

Of particular concern with this airflow is the parasitic drag that is produced from the launch lug. One of the goals of the rocket designer is to limit the amount of turbulent flow over the body of the rocket. This may be achieved by ensuring the surface area is smooth and free of unnecessary protrusions. Some designs actually eliminate the launch lug completely.

A new type of drag that we haven’t mentioned is the base drag at the end of the rocket. This can also be called pressure drag. Boat tails added to the end of a rocket will assist in reducing the base drag.

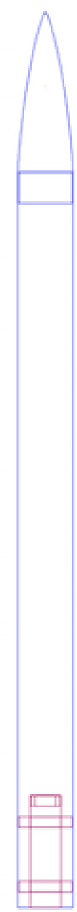

Figure 8 – Educator Rocket
without fins

Designing a Rocket

We will now design, analyze and build the Educator Rocket. We will use the program OpenRocket to analyze the drag of the each of the components. We will also modify these components and observe the drag before we build them.

OpenRocket is a free, open source rocket design and rocket flight simulation software. OpenRocket may be found by going to the url: http://openrocket.sourceforge.net.

Figure 8 shows the Sigma Rockets Educator Rocket (without fins as we will add them in this lesson) OpenRocket or .ork file. To obtain this file download it from here. The design consists of a 45.7cm (18 inch) long, 3.4cm diameter body tube and a Haack series 10cm long nose cone. The motor mount and centering rings are included in the design at the bottom (right side) of the rocket.

Load the program OpenRocket and then open the Educator .ork file.

We will be changing various components on the design rocket and checking their drag results in OpenRocket.

The Nose Cone

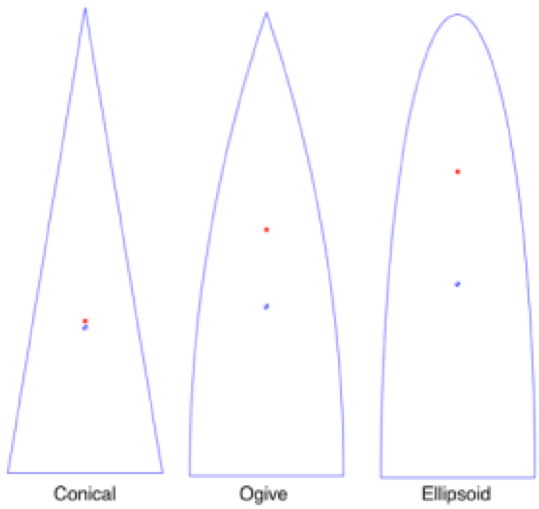

Figure 9 – Nose cones for design rocket.

The shape of the nose cone is a large contributing factor in its drag.

The nose cone is actually affected by both surface and pressure drag. We may determine the drag of different types of nose cones using a wind tunnel or we may use a software program such as OpenRocket to determine it.

To analyze the effect of nose cone designs in figure 9 we utilize the Drag Characteristics tab of Component Analysis function in OpenRocket. This function is found under the Analyze menu.

For our analysis we leave everything on our design rocket the same and change only the nose cone. Below is a table of the results for the analysis.

Nose Cone Type	**Pressure C_D**	**Friction C_D**	**% of Total Drag**	**Total Rocket C_D**

Nose Cone Type	Pressure C_D	Friction C_D	% of Total Drag	Total Rocket C_D
Conical	0.05	0.04	0.08	0.64
Ogive	0.00	0.05	0.05	0.60
Ellipsoid	0.03	0.05	0.07	0.63

As we can see the ogive nose cone design gives us the least amount of drag for the rocket and contributes the least percentage of total drag for our design. As well it gives our rocket a lower overall drag coefficient compared to the other two designs.

The Body Tube

We will continue our design using the ogive nose cone and move on to the body tube. For body tubes there are basically two types of aerodynamic drag acting against it: surface drag and base drag. Surface drag may be reduced by having a smooth surface and finish on the body tube. Base drag can be reduced significantly by applying a boat tail, or transition to the bottom of the rocket. For our analysis we will add and compare boat tails.

Boat Tail Length	Base C_D	Friction C_D	% of Total Drag	Total Rocket C_D
No Boat Tail	0.13	-	-	0.60
5 cm	0.02	0.02	0.11	0.58
7.5 cm	0.02	0.02	0.07	0.54

As we can see even though a 7.5 cm boat tail adds more length to the model rocket, it succeeds in reducing the drag. To allow for the boat tail we must change our design a little bit. First we have to change the length of our motor tube to account for the extra length from the boat tail. Then we need to position the centering rings closer to the bottom of the body tube to provide more support for the boat tail.

The Fins

When a cross wind hits a model rocket on its flight upwards it is the fins that provide the force to correct it. It is important to have the fins large enough for this correction and small enough so that they do not provide excessive weight to the rocket. We may also change the number of fins in our design as well as the angle on which they are attached to the body tube. To keep it simple we will stick to three fins. We will also keep the fin cant angle, the angle in which the fins intersect with the body, at zero (perpendicular to the body).

In OpenRocket the user may select one of two preset fin designs or may design their own fins. We will choose the preset Trapezoidal and alter one of its parameters for our design. We will modify the sweep length of the fins. The table below shows our analysis.

Sweep Length	Total Fin C_D	Total Rocket C_D
2.5cm	0.23	0.54
3.5cm	0.21	0.51
4.5cm	0.18	0.49

As we can see from our analysis the larger the sweep angle the lower the aerodynamic drag.