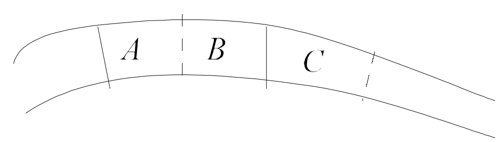
Performance

5. More Aerodynamic Considerations

There is an alternative way of looking at aerodynamic flow problems that is useful for understanding certain phenomena. Rather than tracking a particle of fluid, we can isolate a region of the fluid (called a control volume) and observe the effects of forces (of all kinds) on the fluid that is in that control volume. It should be noted that the fluid in the control volume is changing with time, as some enters and some leaves. (Sometime such problems are called variable mass problems because the mass in the control volume is changing). Again, for the purposes of this course it is convenient to restrict our attention to fluid flow through a stream tube or a duct of some sort, i.e. one-dimensional flow.

To start, we will consider flow in a duct and divide it into two overlapping chunks of fluid. The initial chunk of fluid is the fluid that includes regions A and B and is observed at time t_1. The second chunk of fluid contains the same fluid particles that have moved to a new position indicated by regions B and C. Hence the region B is common to both. We can now apply Newton's second law to the ***same group*** of fluid particles. At time t_1 the particles occupy region AB, and at time t_2 the particles occupy region BC. We can write Newton's laws regarding the change in momentum of the particles from time 1 to time 2:



$$\vec{F} = \frac{d}{dt}\left(mom\vec{e}ntum\right) = \lim_{t \Rightarrow 0} \frac{\Delta\left(mom\vec{e}ntum\right)}{\Delta t} \qquad (1)$$

If we designate $(mom\vec{e}ntum)$ my $\vec{m}$, Eq. (1) becomes:

$$\vec{F} = \lim_{\Delta t \Rightarrow 0} \frac{\Delta \vec{M}}{\Delta t} = \lim_{(t_2 - t_1) \Rightarrow 0} \frac{(\vec{M}_B(t_2) + \vec{M}_C(t_2)) - (\vec{M}_A(t_1) + \vec{M}_B(t_1))}{t_2 - t_1} \qquad (2)$$

or

$$\vec{F} = \lim_{t_2 - t_1 \Rightarrow 0} \left\{ \frac{\vec{M}_B(t_2) - \vec{M}_B(t_1)}{t_2 - t_1} + \frac{\vec{M}_C(t_2) - \vec{M}_A(t_1)}{t_2 - t_1} \right\} \qquad (3)$$

We now focus on the region B. What is originally in region A is flowing into region B, and what is in region C has flowed out of region B. So the first term on the right hand side of Eq. (3) is how the momentum changes in region B, and the second term is time rate of change of momentum flowing out of region B minus the time rate of change of momentum flowing into region B, or the net momentum change leaving B (called the momentum flux). It is important to note that these are vector equations and must be written for each component.

Special Case - steady, 1-D flow

For the special case of steady flow, the first term on the right hand side of Eq. (3) is equal to zero (for steady flow d( )/dt = 0 at a point). For flow through a stream tube, the momentum flux out through a boundary of the control volume is given by (it will be negative if it is flowing into the control volume):

$$\dot{\vec{M}} = (\rho A V)\,\vec{V} \tag{4}$$

where here $\dot{\vec{M}}$ = momentum flux
V = magnitude of velocity
$\vec{V}$ = velocity vector

Note that the quantity $\rho A V = \dot{m}$, the mass flow rate through the stream tube, and that Eq. (4) is evaluated on the boundary of the control volume. Hence we can write the final equation for steady, 1-D flow:

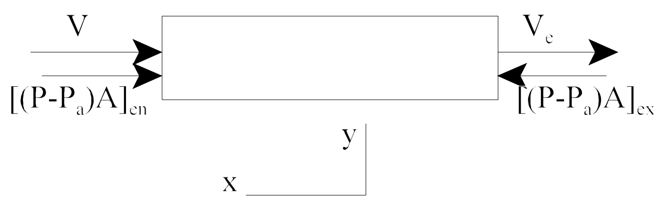
$$\vec{F} = \sum\nolimits_{boundary} \dot{m}\,\vec{V} \tag{5}$$

The use of this equation is best illustrated by an example.

Example

Consider a jet engine that draws air in the front at the flight speed V, and after igniting the fuel, accelerates the air to an exit speed of V_e relative to the engine. Find the thrust of the engine.

We can consider an envelope around the engine to be the control volume. All we need to look at is the boundary of the control volume.



Here the pressure is indicated as $P - P_a$, where P_a is the atmospheric pressure, because the atmospheric pressure acts on the entire control volume, and only the difference from that pressure contributes to the unbalance force.

We can apply Eq. (5) to the control volume representing the flow through a jet engine. We will consider the x component of the equation. The sign convention is as follows: the mass flow rate into the control volume is considered negative, and the mass flow rate out of the control volume is considered positive. The velocity components in the vector are either positive or negative according how they are directed along the axis of interest. Then with these sign conventions we have:

$$F_{x_{\text{engine on fluid}}} - [(P - P_a)A]_{en} + [(P - P_a)A]_{ex} = -\dot{m}\,(-V) + \dot{m}\,(-V_{ex})$$

However, the thrust of the engine is the force of the fluid on the engine. So we can replace F_x with -T and rearrange the equation to obtain: (Note that atmospheric pressure acts everywhere on the control volume so that only the difference of it and the actual pressure will cause an unbalanced force).

$$T = \dot{m}\,(V_{ex} - V_{en}) + [(P - P_a)A]_{ex} - [(P - P_a)A]_{en} \qquad \text{(a)}$$

For the case of a rocket, where there is no inlet velocity and only an exit velocity,

$$T = \dot{m}\,V_{ex} + (P_{ex} - P_a)\,A_{ex} \qquad \text{(b)}$$

In the above equations, we have:

T	=	thrust
$(\ )_{en}$	=	entrance conditions
$(\ )_{ex}$	=	exit conditions
$\dot{m}$	=	mass flow rate out of engine
P	=	static pressure at location specified
P_a	=	local atmospheric pressure

Equation (a) applies to jet engines, and Eq. (b) applies to rockets and is known as the “rocket equation”

Dimensional Analysis

Dimensional analysis allows us to determine the important parameters in a problem so that we can 1) design our experiments to maximize the information obtained with the least number of experiments, 2) determine what parameters are important, and what form and relationship with regard to physical parameters govern the problem, and probably most important, 3) determine how to obtain valid experimental results for scaled models. The principle idea here is that the dimensions on one side of an equation, must equal the dimension on the other side of the equation. Lets start with a simple example that will exhibit all the ideas with out the complexity of a larger problem.

Suppose we assume that the aerodynamic forces are dependent on the variables, density, ρ, the airspeed, V, some geometric length, l, and the fluid viscosity, μ. We would like to know how these variables are related to the force, and how to best design an experiment to determine this relationship. W could just go into the wind tunnel and change each variable one at a time to see what happened, but this is cost and time consuming. Using dimensional analysis we can reduce this effort and determine important relationships. The force is assumed to be a function of the four variables:

$$F = f(\rho, V, l, \mu)$$

but we don't know what that function is. We assume that it is some algebraic relation that has the form:

$$F = f(\rho^{\alpha}, V^{\beta}, l^{\gamma}, \mu^{\delta})$$

Further, we can insist that the dimensions on both side of the equation must match,

$$\left[M L T^{-2}\right] = \left[M L^{-3}\right]^{\alpha}\left[L T^{-1}\right]^{\beta}\left[L\right]^{\gamma}\left[M L^{-1} T^{-1}\right]^{\delta}$$

Dimensional equivalence requires:

$$M^1 L^1 T^{-2} = M^{\alpha+\delta} L^{-3\alpha+\beta+\gamma-\delta} T^{-\beta-\delta}$$

or

$$\begin{aligned} 1 &= \alpha + \delta & & & \alpha &= 1 - \delta \\ 1 &= -3\alpha + \beta + \gamma - \delta & &\Rightarrow & \beta &= 2 - \delta \\ -2 &= -\beta - \delta & & & \gamma &= 2 - \delta \end{aligned}$$

Hence we can write:

$$F = K \rho^1 V^2 l^2 \left(\frac{\mu}{\rho V l} \right)^{\delta}$$

Divide through by the leading terms to get the ***force coefficient*** (introduce a ½ so that the result looks like the dynamic pressure)

Definition:

$$C_F = \frac{F}{\frac{1}{2} \rho V^2 l^2} \equiv \text{force coefficient}$$

Definition:

$R_e \equiv \dfrac{\rho V l}{\mu}$	**Reynolds Number**

We can now say that if we assume that the force depends on the variable indicated above, that the non-dimensional force coefficient is some function (unknown) of the Reynolds number, that is

$$C_F = f(R_e)$$

Hence in our experiment, we only need to vary the Reynolds number and measure the force coefficient, or do a one-parameter experiment to determine how the force coefficient depends on the Reynolds number. Further, we can note that if we tested a different size model, (l would change), all we need to do to match conditions is to make the Reynolds number the same. For example, if we used a model ½ the size of the full scale vehicle, we could run the wind tunnel twice as fast to keep the Reynolds number the same (assuming the same density and viscosity of the wind tunnel air).

In general, more parameters are needed to fully model the forces. To be quite general, we can assume that the force depends on the following variables:

$$F = f(\rho, V, l, \mu, a, g, p, \alpha)$$

where the added variables are

a	=	speed of sound
g	=	acceleration due to gravity
p	=	local pressure
α	=	angle of attack

If we complete the dimensional analysis in the same manner that we did before, solving for α, β, and γ in terms of the remaining exponents, we would find that:

$$C_F = f\left[\left(\frac{\rho V l}{\mu}\right), \left(\frac{V}{a}\right), \left(\frac{gl}{V^2}\right), \left(\frac{p}{\rho V^2}\right), \alpha\right]$$

$$= f\left[Re, M, Fr, Eu, \alpha\right]$$

where Reynolds number $= \dfrac{\rho V l}{\mu}$ all vehicles

Mach number $= \dfrac{V}{a}$ high speed aerodynamics

Froude number	=	$\frac{V}{\sqrt{g\,l}}$	surface ships, waves,
Euler number	=	$\frac{p}{\rho\, V^2}$	air cushion vehicles
attitude angle	=	α	all vehicles
force coefficient	=	$\frac{F}{1/2\, \rho\, V^2\, l^2}$	all vehicles

In the force coefficient equation, the term l^2 can represent an area. It is important to define the reference area used in a particular application. Typically the reference areas used for various vehicles are:

Aircraft	Planform (projected) area of the wing
Missiles	Largest cross sectional area
Submarines	Length of submarine squared
Surface ships	Wetted area
Air cushion vehicles	Area surrounded by skirt
Hydrofoils	Planform area of lifting surface
Helicopters	Area swept out by rotor blade
Propellers	"Disc" area swept out by blades
General shapes (automobiles)	Frontal area
Lifting surfaces	Planform area

Important forces in considering vehicle performance are lift and drag. Hence we must be able to estimate these accurately in order to get a good estimate of performance. In order to make these estimates, we must study basic aerodynamic shapes such as airfoils, which in turn are used to create wings, the primary lifting surface of the vehicle.