

Fluid Dynamics

Chapter 8 – Engineering models

last edited April 3, 2021
by Olivier Cleynen – <https://fluidmech.ninja/>

8.1	Motivation	161
8.2	Comparing influences: the weighted momentum balance	161
8.2.1	Principle	161
8.2.2	The non-dimensional Navier-Stokes equation	162
8.2.3	The flow parameters of Navier-Stokes	164
8.2.4	Flow parameters obtained as force ratios	166
8.2.5	The Reynolds number in practice	166
8.3	Making models	167
8.4	Comparing results: coefficients	169
8.4.1	Principle	169
8.4.2	Force coefficients	169
8.4.3	Power coefficient, and other coefficients	170
8.4.4	Non-dimensionalizing all the problems	171
8.5	Solved problems	172
8.6	Problems	173

These notes are based on textbooks by White [22], Çengel & al.[25], Munson & al.[29], and de Nevers [17].

8.1 Motivation

In this chapter, we develop tools to analyze scale effects in fluid mechanics. This study should allow us to answer two questions:

- How can we adequately reproduce a flow in an enlarged or reduced version?
- How do forces and powers change when the flow is scaled?

8.2 Comparing influences: the weighted momentum balance

8.2.1 Principle

In many practical engineering situations which involve fluid flow, investigating the flow in the real application is impossible or impractical. In those cases it is common practice to build scaled-up or scaled-down versions of the flow in a laboratory. This brings up the question: how should the model flow properties be adapted to represent the original flow? For example, if the model is half as small as the original, should the velocity be halved? Or perhaps doubled?

The answer to this problem is as follows: two flows are *dynamically similar* (i.e. representative of one another) when their *flow parameters* are the same. In order to understand what this means, we will need to look back at the Navier-Stokes equation (the momentum balance equation we derived in chapter 6), re-writing it in a non-dimensional form. Onwards!



Video: when their movement is filmed and then viewed sped-up, fog banks can appear to flow like water. What time-lapse rate is required to achieve physical similarity?

by Simon Christen (sv1)
<https://vimeo.com/69445362>

Advice from an expert

Making models is useful when you have a large machine, say, a jet airliner or a submarine, and you want to study different configurations without building a new full-scale machine each time. But there are other applications: imagine trying to study the flow of molten metal in a furnace, the flow of blood in a beating heart, or the flow of air around a mosquito's wings. Making models and understanding scale effects will allow you to investigate fluid flow in all kind of inaccessible locations. That makes the math worth the effort!



8.2.2 The non-dimensional Navier-Stokes equation

Here, we want to obtain an expression for the Navier-Stokes equation for incompressible flow which allows for an easy comparison of its constituents. We start with the original equation, which we derived in chapter 6 as equation 6/42 p. 125:

$$\rho \frac{\partial \vec{V}}{\partial t} + \rho(\vec{V} \cdot \vec{\nabla})\vec{V} = \rho \vec{g} - \vec{\nabla} p + \mu \vec{\nabla}^2 \vec{V} \quad (8/1)$$

What we would now like to do is *separate the geometry from the scalars* in this equation. The principle is to express each vector \vec{A} as the multiple of its length A and a *non-dimensional* vector \vec{A}^* , which has the same direction as \vec{A} but only unit length.

In order to achieve this, we introduce a series of non-dimensional physical terms, starting with non-dimensional time t^* , defined as time t multiplied by the frequency f (in Hertz) at which the flow repeats itself:

$$t^* \equiv f t \quad (8/2)$$

A flow with a very high frequency is highly unsteady, and the changes in time of the velocity field will be relatively important. On the other hand, the lower the frequency, and the closer the flow is to being steady. In all cases, as we observe the flow, non-dimensional time t^* progresses from 0 to 1, after which the solution is repeated.

We then introduce non-dimensional velocity \vec{V}^* , a unit vector field equal to the velocity vector field divided by its own (scalar field) length V :

$$\vec{V}^* \equiv \frac{\vec{V}}{V} \quad (8/3)$$

Pressure p is non-dimensionalized differently, since in fluid mechanics, it is the pressure *changes* across a field, not their absolute value, that influence the velocity field. For example, in eq. 8/1 $\vec{\nabla} p$ can be replaced by $\vec{\nabla}(p - p_\infty)$ (in which p_∞ can be any constant faraway pressure). Now, the pressure field $p - p_\infty$ is non-dimensionalized by dividing it by a reference pressure difference

$p_0 - p_\infty$, obtaining:

$$p^* \equiv \frac{p - p_\infty}{p_0 - p_\infty} \quad (8/4)$$

If p_0 is taken to be the maximum pressure in the studied field, then p^* is a scalar field whose values can only vary between 0 and 1.

Non-dimensional gravity \vec{g}^* is simply a unit vector equal to the gravity vector \vec{g} divided by its own length g :

$$\vec{g}^* \equiv \frac{\vec{g}}{g} \quad (8/5)$$

And finally, we define a new operator, the *non-dimensional del* $\vec{\nabla}^*$,

$$\vec{\nabla}^* \equiv L \vec{\nabla} \quad (8/6)$$

which ensures that vector fields obtained with a non-dimensional gradient, and the scalar fields obtained with a non-dimensional divergent, are “scaled” by a reference length L .

These new terms allow us to replace the constituents of equation 8/1 each by a non-dimensional “unit” term multiplied by a scalar term representing its length or value:

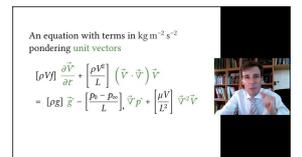
$$\begin{aligned} t &= \frac{t^*}{f} \\ \vec{V} &= V \vec{V}^* \\ p - p_\infty &= p^* (p_0 - p_\infty) \\ \vec{g} &= g \vec{g}^* \\ \vec{\nabla} &= \frac{1}{L} \vec{\nabla}^* \end{aligned}$$

Now, inserting these in equation 8/1, and re-arranging, we obtain:

$$\begin{aligned} \rho \frac{\partial}{\partial \frac{1}{f} t^*} (V \vec{V}^*) + \rho (V \vec{V}^* \cdot \frac{1}{L} \vec{\nabla}^*) (V \vec{V}^*) &= \rho g \vec{g}^* - \frac{1}{L} \vec{\nabla}^* [p^* (p_0 - p_\infty) + p_\infty] + \mu \frac{\vec{\nabla}^{*2}}{L^2} (V \vec{V}^*) \\ \rho V f \frac{\partial \vec{V}^*}{\partial t^*} + \rho V V \frac{1}{L} (\vec{V}^* \cdot \vec{\nabla}^*) \vec{V}^* &= \rho g \vec{g}^* - \frac{1}{L} \vec{\nabla}^* [p^* (p_0 - p_\infty)] + \mu V \frac{1}{L^2} \vec{\nabla}^{*2} \vec{V}^* \\ [\rho V f] \frac{\partial \vec{V}^*}{\partial t^*} + \left[\frac{\rho V^2}{L} \right] (\vec{V}^* \cdot \vec{\nabla}^*) \vec{V}^* &= [\rho g] \vec{g}^* - \left[\frac{p_0 - p_\infty}{L} \right] \vec{\nabla}^* p^* + \left[\frac{\mu V}{L^2} \right] \vec{\nabla}^{*2} \vec{V}^* \end{aligned} \quad (8/7)$$

In equation 8/7, the terms in brackets each appear in front of non-dimensional (unit) vectors. These bracketed terms all have the same dimension, i.e. $\text{kg m}^{-2} \text{s}^{-2}$. Multiplying each by $\frac{L}{\rho V^2}$ (of dimension $\text{m}^2 \text{s}^2 \text{kg}^{-1}$), we obtain a purely non-dimensional equation:

$$\left[\frac{fL}{V} \right] \frac{\partial \vec{V}^*}{\partial t^*} + [1] (\vec{V}^* \cdot \vec{\nabla}^*) \vec{V}^* = \left[\frac{gL}{V^2} \right] \vec{g}^* - \left[\frac{p_0 - p_\infty}{\rho V^2} \right] \vec{\nabla}^* p^* + \left[\frac{\mu}{\rho V L} \right] \vec{\nabla}^{*2} \vec{V}^* \quad (8/8)$$



Video: Figuring out the non-dimensional Navier-Stokes equation

by Olivier Cleynen (CC-BY)
<https://youtu.be/KD3M1qUgN1A>

Equation 8/8 does not bring any information on top of the original incompressible Navier-Stokes equation (eq. 8/1). Instead, it merely separates it into two distinct kinds of components. The first, in square brackets, are a scalar fields (purely numbers), which indicate the magnitude of the acceleration field. The others are unit vector fields (fields of oscillating vectors, all of length 1, and noted with stars), which represent the geometry (direction) of the acceleration field. In this form, we can more easily observe and quantify the weight of the different terms relative to one another. At this point, it is time to introduce the notion of *flow parameters*.

8.2.3 The flow parameters of Navier-Stokes

From here on, in equation 8/8, we convene to call the terms written in brackets *flow parameters*, and label them with the following definitions:

$$[\text{St}] \equiv \frac{f L}{V} \quad (8/9)$$

$$[\text{Eu}] \equiv \frac{p_0 - p_\infty}{\rho V^2} \quad (8/10)$$

$$[\text{Fr}] \equiv \frac{V}{\sqrt{g L}} \quad (8/11)$$

$$[\text{Re}] \equiv \frac{\rho V L}{\mu} \quad (8/12)$$

This allows us to re-write eq. 8/8 as:

$$[\text{St}] \frac{\partial \vec{V}^*}{\partial t^*} + [1] \vec{V}^* \cdot \vec{\nabla}^* \vec{V}^* = \left[\frac{1}{\text{Fr}^2} \right] \vec{g}^* - [\text{Eu}] \vec{\nabla}^* p^* + \left[\frac{1}{\text{Re}} \right] \vec{\nabla}^{*2} \vec{V}^*$$

And finally, we arrive at the simple, elegant and formidable *non-dimensional incompressible Navier-Stokes equation* expressed with flow parameters:

$$[\text{St}] \frac{\partial \vec{V}^*}{\partial t^*} + [1] \vec{V}^* \cdot \vec{\nabla}^* \vec{V}^* = \frac{1}{[\text{Fr}]^2} \vec{g}^* - [\text{Eu}] \vec{\nabla}^* p^* + \frac{1}{[\text{Re}]} \vec{\nabla}^{*2} \vec{V}^* \quad (8/13)$$

Equation 8/13 is an incredibly useful tool in the study of fluid mechanics, for two reasons:

1. It allows us to quantify the **relative weight** of the different terms in a given flow.

In this way, it serves as a compass, helping us determine which terms can safely be neglected in our attempts to find a particular flow solution, merely by quantifying the four parameters [St], [Eu], [Fr], and [Re].

2. It allows us to obtain **dynamic similarity** between two flows at different scales.

In order that a fluid flow be representative of another flow (for example in order to simulate airflow around an aircraft with a model in a wind tunnel), it must generate the same vector field \vec{V}^* . In order to do this, we must generate an incoming flow with the same four parameters [St], [Eu], [Fr], and [Re].



Let us therefore take the time to explore the signification of these four parameters.

The Strouhal number $[St] \equiv f L/V$ (eq. 8/9) quantifies the influence of unsteadiness effects over the acceleration field. It does this by evaluating the importance of the representative frequency f at which the flow pattern repeats itself. Very high frequencies denote highly unsteady flows. When the frequency is very low, $[St]$ is very small; the flow is then quasi-steady and it can be solved at a given moment in time as if it was entirely steady.

The Euler number $[Eu] \equiv p_0 - p_\infty / \rho V^2$ (eq. 8/10) quantifies the influence of the pressure gradient over the acceleration field. It does this by comparing the largest relative pressure $p_0 - p_\infty$ to the flow of momentum in the flow field. The greater $[Eu]$ is, and the more the changes in the velocity field \vec{V} are likely to be caused by pressure gradients rather than viscosity, convection or unsteadiness.

The Froude number $[Fr] \equiv V / \sqrt{g L}$ (eq. 8/11) quantifies the relative importance of gravity effects. In practice, gravity effects only play an important role in free surface flows, such as waves on the surface of a water reservoir. In most other cases, gravity contributes only to a hydrostatic effect, which has little influence over the velocity field.

The Reynolds number $[Re] \equiv \rho V L / \mu$ (eq. 8/12, also eqs. 1/28 p. 23 & 7/24 p. 142) quantifies the influence of viscosity over the acceleration field. It does this by comparing the magnitude of inertial effects ($\rho V L$) to viscous effects (μ). When $[Re]$ is very large, viscosity plays a negligible role and the velocity field is mostly dictated by the inertia of the fluid. We return to the significance of the Reynolds number in §8.2.5 below.

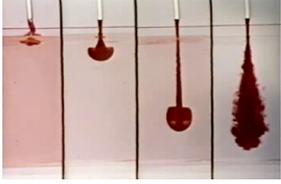
The Mach number $[Ma] \equiv V/c$ (eq. 1/10 p. 16) compares the flow speed V with that of the molecules within the fluid particles (the speed of sound c). $[Ma]$ does not appear in equation 8/13, because we already decided to restrict ourselves to non-compressible flows. If we hadn't, we would be re-expressing the pressure term in that equation as a function of $[Ma]$, quantifying the effect of changes in density.

These five flow parameters should be thought of *scalar fields* within the studied flow domain: there is one distinct Reynolds number, one Mach number etc. for each point in space and time. Nevertheless, when describing fluid flows, the custom is to choose for each parameter *a single representative value* for the whole flow. For example, when describing pipe flow, it is customary to quantify a representative Reynolds number $[Re]_D$ based on the average flow velocity $V_{av.}$ and the pipe diameter D (as we have seen with eq. 7/25 p. 142), while the representative Reynolds number for flow over an aircraft wing is often based on the free-stream velocity V_∞ and the wing chord length. Similarly, the flight Mach number $[Ma]_\infty$ displayed on an aircraft cockpit instrument is computed using the relative free-stream air speed V_∞ and the free-stream speed of sound c_∞ , rather than particular values measured closer to the aircraft.

8.2.4 Flow parameters obtained as force ratios

Instead of the mathematical approach covered above, the concept of flow parameter can be approached by *comparing forces* in fluid flows. This method is described for reference in Appendix A5 p. 256.

8.2.5 The Reynolds number in practice



Video: half-century-old, but timeless didactic exploration of how the dynamics of fluids change with the Reynolds number, with accompanying notes by G. I. Taylor[3]

by the National Committee for Fluid Mechanics Films (NCFMF, 1967[21]) (STYL)
<https://youtu.be/51-6QCJTajU>

Among the five non-dimensional parameters described above, the *Reynolds number* [Re] is by far the most relevant in the study of most fluid flows, and it deserves a few additional remarks. As we have seen, the Reynolds number is a measure of how little effect the viscosity has on the time-change of the velocity vector field:

- With low [Re], the viscosity μ plays an overwhelmingly large role, and the velocity of fluid particles is largely determined by that of their own neighbors;
- With high [Re], the momentum ρV of the fluid particles plays a more important role than the viscosity μ , and the inertia of fluid particles affects their trajectory much more than the velocity of their neighbors.

In turn, this gives the Reynolds number a new role in characterization of flows: it can be thought of as **the likeliness of the flow being turbulent over the length L** . Indeed, from a kinematic point of view, viscous effects are highly stabilizing: they tend to harmonize the velocity field and smooth out disturbances. On the contrary, when these effects are overwhelmed by inertial effects, velocity non-uniformities have much less opportunity to dissipate, and at the slightest disturbance the flow will become and remain turbulent (fig. 8.1). This is the reason why the quantification of a representative Reynolds number is often the first step engineers and scientists take when studying a fluid flow.

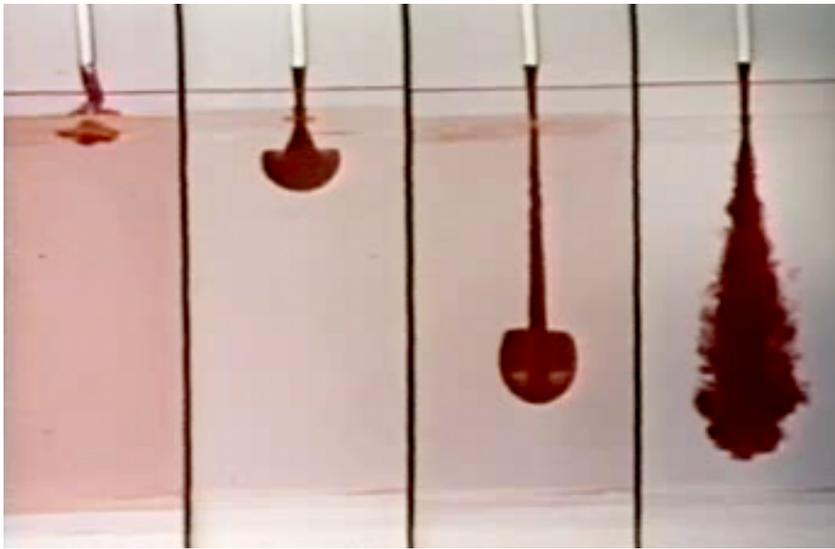


Figure 8.1: A viscous opaque fluid is dropped into a clearer receiving static fluid with identical viscosity. The image shows four different experiments photographed after the same amount of time has elapsed. The viscosity is decreased from left to right, yielding Reynolds numbers of 0,05, 10, 200 and 3 000 respectively. As described in eq. A/36 p. 256, a low Reynolds number indicates that viscous effects dominate the acceleration field. As the Reynolds number increases, the nature of the velocity field changes until it becomes clearly turbulent.

*By the National Committee for Fluid Mechanics Films (NCFMF, 1967[21]), with accompanying notes by Taylor[3]
Image ©1961-1969 Educational Development Center, Inc., reproduced under Fair Use doctrine
A screen capture from film Low Reynolds Number Flow at <http://web.mit.edu/hml/ncfmf.html>*

8.3 Making models

We now know that when we create a scaled-down or scaled-up version of one flow (as shown for example in fig. 8.2), we attempt to keep all flow parameters identical. In practice, this is extremely difficult to do, as we will see while going through this chapter's problem sheet, especially if ordinary fluids (air or water) are to be used. The practice in science and engineering is usually to focus on one or two key parameters, while ignoring others. If the budget allows for it, several models may be built, each focusing on one parameter (e.g. one model for compressibility effects, one model for viscous effects).



Figure 8.2: In order for the flow around a wind tunnel model to be representative of the flow around the real-size aircraft (here, a 48 m-wide Lockheed C-141 Starlifter), dynamic similarity must be obtained. The value of all flow parameters must be kept identical. This is not always feasible in practice.

*Wind tunnel photo by NASA (public domain)
Full-size aircraft photo CC-BY by Peter Long*

In the design and construction of models, practical constraints must be balanced against the need to reproduce the dynamics of fluids accurately. They include:

- Cost of production. The volume of a model typically increases with its length *cubed*, i.e. doubling its length multiplies its volume by a factor 8;
- Precision of manufacturing. Usually, the smaller the model, and the smaller the geometrical accuracy that can be reached;
- Ease of instrumenting. Carrying out measurements over extremely large or extremely small models may be challenging;
- Ease of optical access. Optical flow measurement devices are usually preferred because they do not obstruct the flow, but when the flow is internal to a model, they require transparent, flat walls to work correctly;
- Performance of large-scale laboratory equipment. Wind tunnels and water channels are expensive installations. Often, the characteristics of models is adapted to the equipment's availability and performance, rather than the other way around.

Finally, it must be noted that in this chapter, we focus on reproducing flow dynamics only. Other kinds of forces applying on models, and other physical phenomena, are not taken into account. For example, an airplane model in a tunnel may not be able to fly, and a boat model may not be able to float, because they are made of different materials than the original objects, and because weight and lift forces do not scale together (this is also the reason why large birds such as condors or swans do not look like, and cannot fly as slowly as mosquitoes and bugs! – these ideas are beautifully and smartly explored by Hendrik Tennekes [8, 23]). Similarly, heat transfer or chemical reaction rates, may be completely off in the model, unless the physical laws that govern them are also taken into account.

Advice from an expert

When your small wind tunnel already cost half a million euros to build, is run by fans with hundreds of kilowatts of power, and your new laser-Doppler velocity measurement system cost the lab 200 k€, it is clear that the available laboratory equipment will dictate the model size and the experimental flow conditions, not the other way around.

A well-designed experiment will always take advantage of the best performance of the available equipment. This is why fluid dynamicists like to measure the performance of their wind tunnel not with size or speed, but with a *tunnel Reynolds number* – the maximum Reynolds number you can subject a model to in the test section.



8.4 Comparing results: coefficients

8.4.1 Principle

The general rule that we follow when we compare measurements from experiments at different scale is that

in two physically similar experiments, the force and power coefficients are the same.

In other words, for scientists and engineers, the hard work is making a model where the flow coefficients are the same (and thus the original flow and the model are physically similar). Once this is done, the comparison of results is easy: the force and power coefficients are the same on the model and in the original flow.

8.4.2 Force coefficients

In order to compare forces, we want to have a sense of how flow-induced forces scale when fluid flows are scaled. A look back on chapter 2 (*Analysis of existing flows with one dimension*), and in particular eq. 2/7 p. 37, shows us that for a steady flow through a given control volume, the net force induced on the fluid is expressed by:

$$\vec{F}_{\text{net on fluid}} = \Sigma_{\text{net}} \left[\rho V_{\perp} A \vec{V} \right] \quad (8/14)$$

This equation tells us that the norm of the net force vector, F_{net} , is directly related to a term with dimensions of $\rho |V_{\perp}| AV$. In our selection of a scale by which to measure F_{net} , it is therefore sensible to include a term proportional to the the density ρ , a term proportional to the area A , and a term proportional to the square of velocity V . This “scale of fluid-induced force” is conventionally measured using the *force coefficient* C_F :

$$C_F \equiv \frac{F}{\frac{1}{2} \rho S V^2} \quad (8/15)$$

where F is the considered fluid-induced force (N);
 ρ is a reference fluid density (kg m^{-3});
 S is a reference surface area (m^2);
and V is a reference velocity (m s^{-1}).

In effect, the force coefficient relates the magnitude of the force exerted by the fluid on an object (F) to the rate of flow of momentum towards the object ($\rho S V^2 = \dot{m} V$).

It is worth making a few remarks about this equation. First, it is important to realize that eq. 8/15 is a *definition*: while the choice of terms is guided by physical principles, it is not a physical law in itself, and there are no reasons to expect C_F (which has no dimension, and thus no unit) to reach any particular value in any given case. The choice of terms is also worth commenting:



Video: Comparing fluid-induced forces: using force coefficients
by Olivier Cleynen (CC-BY)
https://youtu.be/uGr_dloHWnI

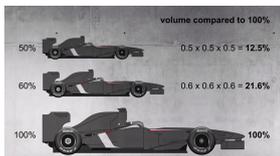
- F can be any fluid-induced force; generally we are interested in quantifying either the *drag* F_D (the force component parallel to the free-stream velocity) or the *lift* F_L (force component perpendicular to the free-stream velocity);
- the reference area S is chosen arbitrarily. It matters only that this area grow and shrink in proportion to the studied flow case. In the automobile industry, it is customary to choose the vehicle frontal area as a reference, while in the aeronautical industry the top-view wing area is customarily used. The square of a convenient reference length L can also be chosen instead;
- the choice of reference velocity V and density ρ is also arbitrary, since both these properties may vary in time and space within the studied fluid flow case. It matters merely that the chosen reference values are representative of the case; typically the free-stream (faraway) conditions V_∞ and ρ_∞ are used;
- the term $1/2$ in the denominator is a purely arbitrary and conventional value.

Force coefficients are meaningful criteria to compare and relate what is going on in the wind tunnel and on the full-size object: in each case, we scale the measured force according to the relevant local flow conditions.

Advice from an expert

Just like when selecting the length L in the Reynolds number, there is no absolute physical law that would indicate which reference S or V should be taken when calculating a force coefficient. The only important thing is that the definition should remain the same across all scaled flows.

However, a large body of professional customs and cultures usually exists in each sub-area of fluid dynamics. Make sure you consult the available literature and field conventions, so your results fit among them. Calculating the drag coefficient of a car using the car side-view area is not wrong, but it would go a long way to annoy colleagues and competitors!



Video: practical application of scale effects: translating measurements made on a 60% size formula one car wind tunnel model into their "real size" race car values, as in problem 8.5

by the Sauber F1 team (STYL)
<https://youtu.be/FQxmOQmnaGw>

For example, a flow case around a car A may be studied using a model B. *If dynamic similarity is maintained* (and that is by no means an easy task!), then the flow dynamics will be identical. The drag force $F_{D B}$ measured on the model can then be compared to the force $F_{D A}$ on the real car, using coefficients: since $C_{F_{D B}} = C_{F_{D A}}$ we have:

$$F_{D A} = C_{F_{D A}} \frac{1}{2} \rho_A S_A V_A^2 = C_{F_{D B}} \frac{1}{2} \rho_A S_A V_A^2 = \frac{F_{D B}}{\frac{1}{2} \rho_B S_B V_B^2} \frac{1}{2} \rho_A S_A V_A^2$$

$$F_{D A} = \left(\frac{\rho_A S_A V_A^2}{\rho_B S_B V_B^2} \right) F_{D B}$$

8.4.3 Power coefficient, and other coefficients

During our investigation of the flow using a small-scale model, we may be interested in measuring not only forces, but also other quantities such as power – this would be an important parameter, for example, when studying a pump, an aircraft engine, or a wind turbine.

We again go back to chapter 2 (*Analysis of existing flows with one dimension*), and in particular eq. 2/18 p. 40. This helps us recall that power gained or lost by a fluid flowing steadily through a control volume could be expressed as:

$$\dot{Q}_{\text{net}} + \dot{W}_{\text{shaft, net}} = \Sigma_{\text{net}} \left[\dot{m} \left(i + \frac{p}{\rho} + \frac{1}{2} V^2 + gz \right) \right] \quad (8/16)$$

Thus, the power gained or lost by the fluid is directly related to the magnitude of the scalar $\Sigma_{\text{net}} \rho |V_{\perp}| AV^2$. A meaningful "scale" for the power of a machine can therefore be the amount of energy in the fluid that is made available to it every second, a quantity that grows proportionally to ρAV^3 .

This "scale of fluid flow-related power" is conventionally measured using the *power coefficient* C_P :

$$C_P \equiv \frac{\dot{W}}{\frac{1}{2} \rho S V^3} \quad (8/17)$$

where \dot{W} is the power added to or subtracted from the fluid (W).

8.4.4 Non-dimensionalizing all the problems

In this chapter, we have focused on scaling forces and powers from one flow to the other. There are of course other parameters of interest, and for each, it is common practice to define a coefficient. We have already used the *pressure loss coefficient* $K_L \equiv |\Delta p| / \frac{1}{2} \rho V_{\text{av}}^2$ in chapter 7 (eq. 7/6 p. 137), and we shall soon use the *shear coefficient* c_f in the forthcoming chapter (eq. 10/6 p. 203).

There are many other examples; in fact, in fluid mechanics, the non-dimensionalization of problems is a very important work methodology. This helps us extrapolate from one or two experiments up to entire families of flows.

Advice from an expert

Now you should begin to see that there is more to dimensional analysis than just making models. Being good at comparing the influences of various factors on a fluid flow phenomenon means that you can generalize results. Pressure loss in a pipe, for example, is quantified with a single, suitably-designed parameter f , irrespective of say the flow speed or fluid density. Take a look back on the Moody diagram page 152: it has non-dimensional terms on all three sides! This is the reason why we only need one diagram, instead of a catalog of diagrams (e.g. one for each flow speed and diameter).

And so, you should not be surprised when, after you show them the results of your first experimental fluid flow measurement campaign, your colleagues in the fluid dynamics laboratory ask you: "can you non-dimensionalize that diagram?"



Video: Understanding scale effects will save you a lot of money and embarrassment: the cautionary tale of Howard Hughes

by Olivier Cleynen (CC-BY)
<https://youtu.be/fF07XkAUELE>



XKCD #687: abusing dimensional analysis

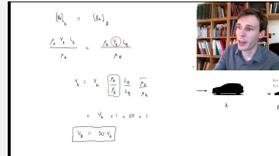
by Randall Munroe (CC-BY-NC)
<https://xkcd.com/687>

8.5 Solved problems

Scale model of a car



If we want to reproduce turbulent effects in the airflow around a car using a 1/20 model, how fast should the flow around the model be?

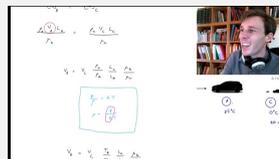


See this solution worked out step by step on YouTube
<https://youtu.be/4p99bCkZdho> (CC-BY Olivier Cleynen)

Wind tunnel speed

We reproduce the airflow around a full-size car moving in air at 25 °C. We use a 1/20 model in a wind tunnel. This tunnel is limited to 20 m s⁻¹, but the air is cooled ($T = 0$ °C).

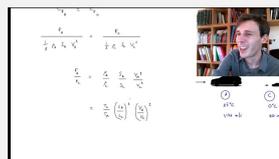
What is the real-car speed that is being simulated in the wind tunnel?



See this solution worked out step by step on YouTube
<https://youtu.be/gHf8ouKfAAU> (CC-BY Olivier Cleynen)

Wind tunnel measurement

In the setup from the previous example, how should the force measurements on the car be translated to values corresponding to the real car?



See this solution worked out step by step on YouTube
<https://youtu.be/Fn-zxYT9-KY> (CC-BY Olivier Cleynen)

Problem sheet 8: Engineering models

last edited September 19, 2020
by Olivier Cleynen – <https://fluidmech.ninja/>

Except otherwise indicated, assume that:

The atmosphere has $p_{\text{atm.}} = 1 \text{ bar}$; $\rho_{\text{atm.}} = 1,225 \text{ kg m}^{-3}$; $T_{\text{atm.}} = 11,3 \text{ }^\circ\text{C}$; $\mu_{\text{atm.}} = 1,5 \cdot 10^{-5} \text{ Pa s}$

Air behaves as a perfect gas: $R_{\text{air}} = 287 \text{ J kg}^{-1} \text{ K}^{-1}$; $\gamma_{\text{air}} = 1,4$; $c_{p \text{ air}} = 1005 \text{ J kg}^{-1} \text{ K}^{-1}$; $c_{v \text{ air}} = 718 \text{ J kg}^{-1} \text{ K}^{-1}$

Liquid water is incompressible: $\rho_{\text{water}} = 1000 \text{ kg m}^{-3}$, $c_{p \text{ water}} = 4180 \text{ J kg}^{-1} \text{ K}^{-1}$

The non-dimensional incompressible Navier-Stokes equation:

$$[\text{St}] \frac{\partial \vec{V}^*}{\partial t^*} + [1] \vec{V}^* \cdot \vec{\nabla}^* \vec{V}^* = \frac{1}{[\text{Fr}]^2} \vec{g}^* - [\text{Eu}] \vec{\nabla}^* p^* + \frac{1}{[\text{Re}]} \vec{\nabla}^{*2} \vec{V}^* \quad (8/13)$$

in which $[\text{St}] \equiv \frac{f L}{V}$, $[\text{Eu}] \equiv \frac{p_0 - p_\infty}{\rho V^2}$, $[\text{Fr}] \equiv \frac{V}{\sqrt{g L}}$, and $[\text{Re}] \equiv \frac{\rho V L}{\mu}$.

The force coefficient C_F and power coefficient C_P are defined as:

$$C_F \equiv \frac{F}{\frac{1}{2} \rho S V^2} \quad C_P \equiv \frac{\dot{W}}{\frac{1}{2} \rho S V^3} \quad (8/17)$$

The speed of sound c in air is modeled as:

$$c = \sqrt{\gamma R T} \quad (1/11)$$

Figure 8.3 quantifies the viscosity of various fluids as a function of temperature.

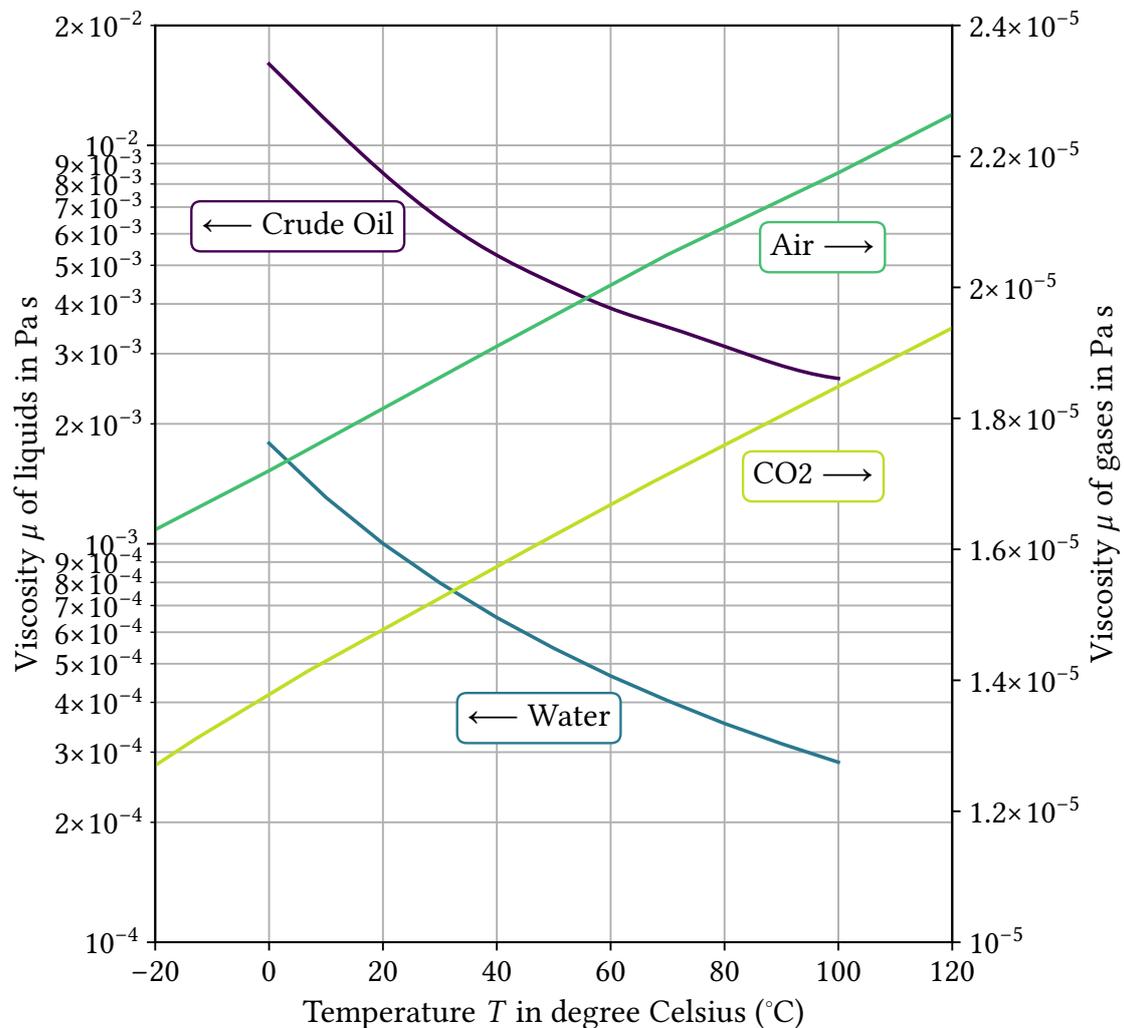


Figure 8.3: The viscosity of four fluids (crude oil, water, air, and CO₂) as a function of temperature. The scale for liquids is logarithmic and displayed on the left; the scale for gases is linear and displayed on the right.

Figure reproduced from figure 5.6 p. 99; CC-BY by Arjun Neyyathala & Olivier Cleynen

8.1 Reading quiz

Once you are done with reading the content of this chapter, you can go take the associated quiz at <https://elearning.ovgu.de/course/view.php?id=7199>

In the winter semester, quizzes are not graded.



8.2 Scaling a golf ball

CC-0 Olivier Cleynen

- 8.2.1. Write the definition of the Reynolds number and indicate the SI units for each term.
- 8.2.2. What is the consequence on the velocity field of having a low Reynolds number? (Briefly justify your answer, e.g. in 30 words or less)
- 8.2.3. Give one example of a high-Reynolds number flow, and one of a low-Reynolds number flow.

The standard golf ball has a diameter of 42,67 mm and a mass of 45,93 g (figure 8.4). A typical maximum velocity for such balls is 200 km h^{-1} .



Figure 8.4: A golf ball

Photo CC-BY by Bradley P. Johnson

A student wishes to investigate the flow over a golf ball using a model in a wind tunnel. Using a 3D printer, s/he prints an enlarged model with a diameter of 50 cm.

- 8.2.4. If the atmospheric conditions are identical, what flow velocity needs to be generated during the experiment in order to reproduce the flow patterns around the real ball?
- 8.2.5. Would the Mach number for the real ball then be reproduced?
- 8.2.6. If the temperature on the golf course is -5°C , and the temperature in the wind tunnel is 31°C (with identical pressure, 1 bar), what should the new wind tunnel velocity be?
- 8.2.7. If the enlarged model was made out of the same materials as the real ball, how heavy would it be?

8.3 Fluid mechanics of a giant airliner

An Airbus A380 airliner has an 80 m-wingspan and is designed to cruise at $[\text{Ma}] = 0,85$ where the air temperature and pressure are -40°C and 0,25 bar.

A group of students wishes to study the flow field around the aircraft, with a wind tunnel whose test section has a diameter of 1 m (in which, obviously, the model has to fit!). Together, they brainstorm various possibilities. They assume that the viscosity of the air is always $1,5 \cdot 10^{-5} \text{ Pa s}$

- 8.3.1. If the temperature inside the tunnel is maintained at $T_{\text{tunnel}} = 20^\circ\text{C}$, propose a combination of wind tunnel velocity and pressure which would enable the team to adequately model the effects of viscosity.
- 8.3.2. Which flow conditions would be required in a *water* tunnel with the same dimensions?

Realizing the conditions calculated in the questions above cannot be realized in practice, the group of students tries instead to study the effect of compressibility (while accepting that viscous effects may not be adequately modeled).

- 8.3.3. If the maximum velocity attainable in the wind tunnel is 80 m s^{-1} , which tunnel air temperature is required for compressibility effects to be modeled?

8.4 Scale effects on a dragonfly

A dragonfly (sketched in fig. 8.5) has a 10 cm wingspan, a mass of 80 mg, and cruises at 4 m s^{-1} , beating its four wings 20 times per second.

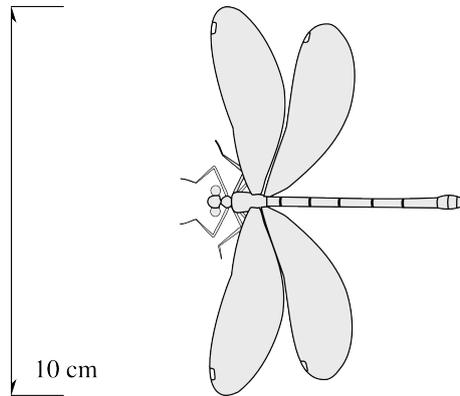


Figure 8.5: Plan view of a dragonfly

Figure by Olivier Cleynen, A. Plank, Drury, Dru, Westwood, J. O. (CC-0)

You are tasked with the investigation of the flight performance of the dragonfly, and have access to a wind tunnel with a test section of diameter 1 m.

- 8.4.1. Which model size and flow velocity would you use for this experiment?
- 8.4.2. How many wing beats per second would then be required on the model?
- 8.4.3. What would be the lift developed by the model during the experiment?
- 8.4.4. How much mechanical power would the model require, compared to the real dragonfly?

8.5 Formula One testing

You are leading the Aerodynamics team in a successful Formula One racing team. Your team races a car that is 5,1 m long, 1,8 m wide, has a 610 kW power plant, and a mass of 750 kg (figure 8.6). The car can reach a top speed of 310 km h^{-1} ; at that speed, the drag force is 7,1 kN.

In order to test different aerodynamic configurations for the car, your team invests in a 50-million-euro wind tunnel, in which you will run tests on a model. You currently have the choice between two model sizes: a 60 % model and a (smaller) 50 % model.

- 8.5.1. What would be the frontal area of each model, in proportion to the frontal area of the real car?
- 8.5.2. What would be the volume of each model, compared to the volume of the real car?
- 8.5.3. How much less weight would the 50 % model have than the 60 % model?



Figure 8.6: A 2017 Formula One racing car, driven by Pascal Wehrlein

Photo CC-BY by Jake Archibald

From here on, your team decides to use the 50 % car model.

- 8.5.4. If the ambient atmospheric conditions cannot be changed, which flow speed in the wind tunnel is required, so that the air flow around the real car is reproduced around the model?

You are preparing for a race where the air temperature will be 30 °C. The Formula One regulations forbid you from running your wind tunnel faster than 50 m s⁻¹.

- 8.5.5. Your team considers modifying the wind tunnel air temperature, in order to compensate for the limit in the air speed. If the temperature in the tunnel can be controlled between -10 °C and 40 °C, but the pressure remains atmospheric (1 bar), what is the maximum race-track speed that can be simulated in the wind tunnel?
- 8.5.6. In that case, by which factor should the model drag force measurements be multiplied in order to correspond to the real car?
- 8.5.7. The wind tunnel has a 2 m by 2 m square test cross-section. What is the power required to bring stationary atmospheric air outside ($T_{\text{atm.}} = 15 \text{ °C}$, $V_{\text{atm.}} = 0 \text{ m s}^{-1}$) to the desired speed and temperature in the test section?

NB: this exercise is inspired by an informative and entertaining video by the Sauber F1 team about their wind tunnel testing, which you are encouraged to watch at <https://youtu.be/KC0E0wU6inU>.

Answers

- 8.2**
- 1) See equation 8/12 and subsequent comments;
 - 2) See §8.2.5, in practice this can be summarized in two or three sentences;
 - 3) High-Re: air flow around an airliner in cruise; Low-Re: air flow around a dust particle falling to the ground.
 - 4) Match [Re]: $V_B = 4,74 \text{ m s}^{-1}$
 - 5) No, but it is very low anyway (no compressibility effects to be reproduced)
 - 6) Increase the velocity by 25 %
 - 7) $m_C = 73,9 \text{ kg}$
- 8.3**
- 1) With a half-aircraft model of half-width $L_2/2 = 80 \text{ cm}$, at identical speed and ambient temperature, we would need an air pressure $p_2 \approx 20 \text{ bar}$!
 - 2) In water, the density cannot be reasonably controlled, and we need a velocity $V_3 = 418 \text{ m s}^{-1}$!
 - 3) In air, at $V_4 = 80 \text{ m s}^{-1}$, Mach number can be reproduced at $T_4 = -251 \text{ °C}$ (although the Reynolds number is off). Wind tunnels used to investigate compressible flow around aircraft have very powerful coolers.
- 8.4**
- 1) With e.g. a model of span 60 cm, match the Reynolds number: $V_2 = 0,67 \text{ m s}^{-1}$;
 - 2) Match the Strouhal number: $f_2 = 0,56 \text{ Hz}$. Mach, Froude and Euler numbers will have no effect here;
 - 3) and 4) are left as a surprise!
- 8.5**
- 3) The third model would have 42 % less mass than the second;
 - 4) Maintaining [Re] requires $V_3 = 620 \text{ km h}^{-1}$ (fast!);
 - 5) Now the fastest track speed that can be studied is $31,7 \text{ m s}^{-1}$;
 - 6) Multiply force measurements by 1,39 to scale up to reality;
 - 7) The required work as power is 331 kW and the power to change temperature is $-6,654 \text{ MW}$ (better keep the cooled air in a closed circuit than feed the tunnel with outside air!)