Similitude And Approximations In Engineering,
Vijay Gupta
Applied Mechanics
Indian Institute of Technology Delhi
Week - 02
Lecture – 08

Welcome back.

Some rules of thumb for similarity

Flow situation	Should match (modelling rules)	Most Important modelling rule
No free surface, with cavitation, (bad pumps, siphon, torpedoes)	Eu, Re	Eu
With free surface, no cavitation, (ships, dams, harbours, off-shore platforms, etc.)	Re, Fr	Fr
No free surface, no cavitation, (submarines, airplanes, enclosed flows in pipes, etc.)	Re	Re
With free surface, with cavitation, (high speed ships)	Eu, Re, Fr	Fr, Eu

In this lecture, we will discuss some simple applications of similitude to fluid mechanics. In the last lecture, we have developed some rules of thumb for similarity. We discussed that in the flow situation where with no free surface, but with cavitation that happens in badly designed pumps, siphons and torpedoes, we should match Euler number and Reynolds number. The dependence on Reynolds number is rather weak. So only the Euler number is needed to be matched in most cases.

The dependence of Froude number drops out because there being no free surface. But when we have free surface, but no cavitation like in ships, dams, harbors, offshore platforms etcetera. The Euler number is unimportant because there is no cavitation. So that we can take the pressure difference which characterizes the problem as $\frac{1}{2}\rho V_o^{\ 2}$.

So that the Euler number drops out and only Reynolds number and Froude number are important. Again the dependence on Reynolds number is rather low and so we need to match Froude number. In cases when there is no free surface, no cavitation, like in submarines operating far below the surface in the deep sea, airplanes, and in closed flow in pipes, etcetera, we need to match only Reynolds number. In the situation where we have free surface as well as cavitation like in high speed ships, all the three parameters, Euler number, Reynolds number and Froude number need to match. Let us do an example of a blimp.

Blimp is a lighter than air aircraft which floats in the sky. To estimate the power requirement of a blimp, travelling at 10 meters per second in air, it is proposed to test a 120th scale model of it in water. What should be the velocity of model in the water and what will be the prediction rule for the power required? So that if we measure the power required for moving the model blimp in water, we can from this determine what the power required would be for the actual prototype blimp in air. Since we are working in atmosphere, we can work with gauge pressures, there is no cavitation involved. We can work with gauge pressures and therefore, we can take the characteristic pressure difference $(\Delta p)o$ as the dynamic pressure, the stagnation pressure minus the static pressure, that is one half rho v naught squared so that the Euler number is dropped from consideration.

There is no free surface, so that we can work with non gravitational pressure difference and so the Froude number is also not significant. The only pi number that needs to be matched is Reynolds number. And if we match the Reynolds number, that means the value of the Reynolds number for the model is set equal to the value of Reynolds number for the prototype. This is the expression and from this we get the velocity of the model should be 15.7 meters per second when we use 10 meters per second for $V_{o,p}$, that is, for the prototype velocity.

$\frac{L_{o,p}}{L_{o,m}}$ is 20, we are using one twentieth model. We use the density of the prototype that is density of air divide the density of water, viscosity of water and viscosity of air. To determine the prediction rule for power required. Note that the power required is to overcome the drag and the drag at such speeds for bodies like a blimp is dominated by pressure drag which is the flow wise component of pressure force integrated over the entire blimp surface. So, $D = \int_{entire\ surface} \left(p - p_o\right) i.\, dA = \int_{entire\ surface} p_g i.\, dA$. Normalization of the right hand side using the characteristic gauge pressure as $\frac{1}{2}\rho V_o^{\ 2}$, the dynamic pressure, and using A_o ,the characteristic area, we get $\frac{D}{\left(\frac{1}{2}\rho V_o^2 \bullet A_o\right)} = \int_{entire\ surface^*} p_g^* i.\, dA^*$.

Now, notice that this is in terms of the values which are normalized. When the flow over the model blimp is the same as flow over the prototype blimp, the values of the normalized variables are identical. That is the consequence of similarity. And so, this integral would be the same in prototype as well is in the model. That means this value, which incidentally is defined as drag coefficient, is the same in the prototype and in the model.

So, $\frac{D_p}{D_m} = \frac{\left(\frac{1}{2}\rho V_o^2 \bullet A_o\right)_p}{\left(\frac{1}{2}\rho V_o^2 \bullet A_o\right)_m}$. When the work done would be drag times the velocity, and using this expression we get this or that the ratio of the power requirement for the prototype divided by power requirement for the model, $\frac{W_p}{W_m} = \frac{D_p \times V_{o,p}}{D_m \times V_{o,m}} = \frac{\left(\rho V_o^3 A_o\right)_p}{\left(\rho V_o^3 Ao\right)_m} = 0.125$. So, the prototype uses only one eighth power of the power used by model. The model would be using 8 times the power of the prototype.

Why is this so? This largely because the model is moving in water and the water is lot denser, almost a 1000 time denser, and therefore, you need to do a lot of work and spend lot of power in moving the blimp at a similar speed.

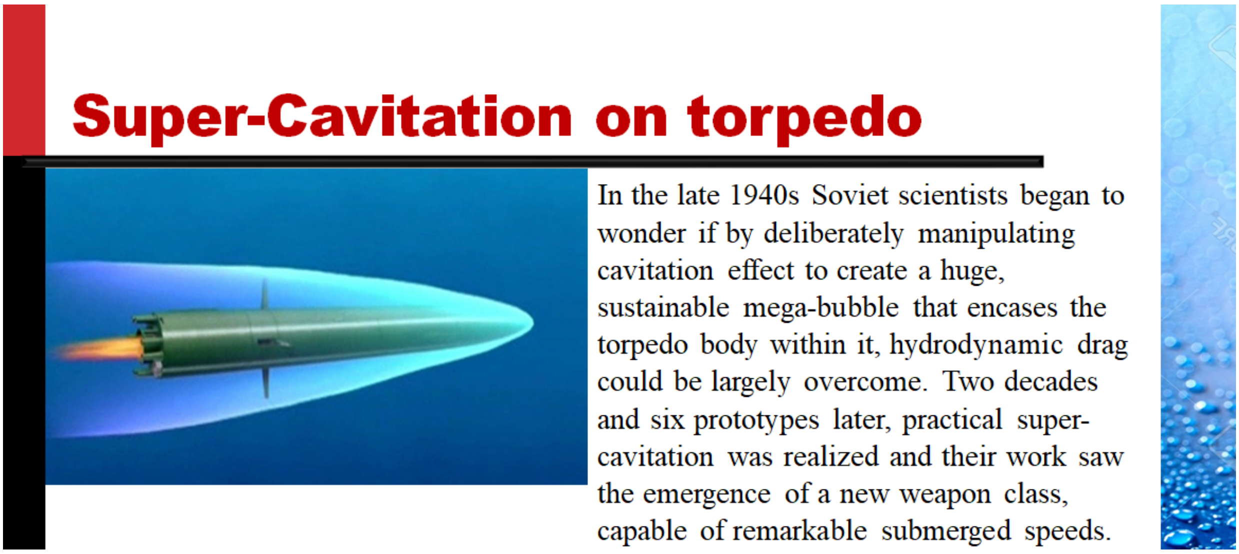
Let us do another example, an example where cavitation is involved. Cavitation on torpedo fins. Torpedoes when they move, they move at very high speeds and on the fins the velocity is very large and so cavitation occurs. This cavitation is quite harmful when the pressure increases on the vapor bubble that is when the liquid speed decreases, the bubble collapses, collapses in the manner shown, and there is a kind of water jet impinges on the nearby surfaces.

This causes surface erosion. This picture denotes the cavitation damage done on the impeller of a water pump. I could not find an image of a damage done to the torpedo fins but the damage is quite serious and this is an example of the damage done. The problem that we have is that the torpedo diameter 20 cm is seen to be on the verge of cavitation at a speed of 30 m per second when running 10 m below the ocean surface. The ocean is at temperature of 20 degree Celsius.

Why do we need to specify the temperature? Because cavitation is occurring, and so the vapor pressure is important, and the vapor pressure depends very strongly on the temperature. So we need to know the vapor pressure of water at 20 degree Celsius. The problem is to determine at what depth must the torpedo run if it is travelling at 50 m per second. It is travelling that fast so the pressure drop would be larger. Since the cavitation pressure is the same, the temperature is 20 degree Celsius then it means the ambient pressure must be larger and that would mean that the torpedo if it is to avoid cavitation must run much deeper than 10 m below the ocean surface.

Let us determine that. As 10 m is fairly large compared to the torpedo size of 20 cm, the Froude number does not need to be matched. We can ignore the free surface of the water. And we can replace the pressure by the non-gravitational pressures given by p script as $P = p + \rho g z - p_o$. But since the cavitation is involved, the Euler number is now to be matched and the Euler number must be defined with $P_o - P_v$ taken as the characteristic modified pressure difference.

That is these two values, $\left[\frac{\rho V_o^2}{P_o - P_v}\right]_p = \left[\frac{\rho V_o^2}{P_o - P_v}\right]_m$, one for the prototype and the other for the model must match. The value of $\left(P_o - P_v\right)$ is $p_o - p_v - \rho g z$ which could be at the level at the surface of the sea, $-\left(p_v + \rho g z - p_o\right)$ when the cavitation occurs. Taking the datum at the ocean surface where p is p0 and z is equal to 0, we determine the value of $\left(P_o - P_v\right) = p_o - p_v - \rho g z$. For the model, this value is determined to be 200 kilo Pascal when you plug in all the given values. The equivalence of the Euler number between the model and the prototype, then gives the value of $\left(P_o - P_v\right)_p$ for the prototype as 555.7 kilo Pascal, almost two and a half times that for the first case. And from this we can determine the depth must be 45.2 meters. So, if the torpedoes has to operate at a much larger speed, then it must move deeper. So from 10 meters to 45.2 meters to avoid cavitation. I hope this example clarifies the use of modeling rules in cases where cavitation is involved.



Cavitation is not always damaging. In the late 1940s, Soviet scientists began to wonder about deliberately manipulating cavitation effects to create a large, huge, sustainable mega bubble that encases the torpedo body, within it, hydrodynamic drag could be largely overcome. So that if we can create a cavitation bubble such that the whole body of the torpedo is enclosed within this as shown here, then this is all vapor. So the drag on the torpedo would be greatly reduced. After two decades of work and six prototypes later, practical super cavitation was realized and the work saw the emergence of new weapon class capable of remarkable submerged speeds. So the drag is much lower, the speeds can be much larger. Cavitation instead of going against us is now helping us.

Rayleigh-Stokes flow or Stokes second problem

Consider a fluid in the half-space above an infinite flat plate oscillating in the x-direction in its plane with a velocity $V_o \sin \omega t$ where $\omega/2\pi$ is the frequency of oscillation. Determine the pi-numbers that can be used for establishing similarity.

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

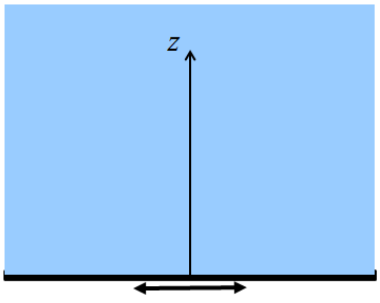
$$\rho\frac{\partial u}{\partial t}=\mu\frac{\partial^2 u}{\partial z^2}$$

The no-slip condition

$u = V_o \sin \omega t$ on the plate $z = 0$,

and the condition far away

$u(\infty, t) = 0$ for all times.



Now let us do a little more complicated problem. This is a problem which is known as Rayleigh-Stokes flow or the Stokes second problem in fluid mechanics. The problem situation is simple. We have a flat plate which is oscillating in a liquid which was initially coincident not moving at all and this plate is vibrating at a speed which is given by V_o sin $sin\ \omega t$.

That is within amplitude V_o and a circular frequency ω. Because of the no slip condition, the fluid adjacent to the plate starts moving and as it moves it causes shear stress on the less stress of the fluid and the effect of the movement of the plate penetrates upward. But soon the direction of motion changes, so everything reverses. And because of this at any given time the effect of the motion of the plate does not penetrate to more than a fixed distance. We have to determine what is the extent of the region of fluid near the plate that is affected by the motion of the plate.

We start with the governing equation. $\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial z^2}\right)$ is the equation of motion of the fluid when there is only one component of velocity u and there is no pressure gradient in x, y or z direction and so the equation of motion simplifies to $\rho\frac{\partial u}{\partial t}=\mu\frac{\partial^2 u}{\partial z^2}$. This unsteady flow with only one component of velocity which is a function only of z and of course, t. This partial differential equation is not very difficult to solve.

It has to be solved with the no slip condition at the plate which would say that the velocity u would be equal to the velocity of the plate at any given instant V_o sin $sin\ \omega t$. And far away the velocity would be 0 for all times. We define the normalized velocity $u^* = \frac{u}{V_o}$, the amplitude of the velocity of the plate, and we define $t^* = \omega t$. $1/\omega$ is the characteristic time. So ωt becomes a non-dimensionalized normalized time.

And we define a normalized z variable $z^* = \frac{z}{L_o}$, where L_o is some characteristic length in the z direction. What is that characteristic length? We do not know. In fact this is where half the story is. So L0 is as yet unknown. If we do the transformation, the normalized equation looks like this. With the boundary condition that $u^* = \sin sin\, t^*$ on $z^* = 0$, and $u^*(\infty, t^*) = 0$. They are far away boundary conditions. And so u^* now is a function of two independent normalized variables, z^* and t^*, and one parameter $\frac{\nu}{\omega L_o^2}$.

Now comes an interesting part. This is an unsteady problem and there is a viscous flow. Viscosity is central to the problem. We cannot neglect viscosity. So both terms in this equation must be retained. None of the terms must drop out. Since this derivative of normalized variable and this derivative of normalized variable by definition are of order one, normalized. And so this coefficient must be of order one and this gives you a definition of L_0. Because if this is of order one, then L_0 must be equal to $\frac{\nu}{2\omega}$. If you use this, L_0 should be of order $\frac{\nu}{2\omega}$. We can set it equal to this. So L_0 is equal to $\frac{\nu}{2\omega}$. This graph shows the variations of the velocity profile with time for three different cases of omega.

This is for the fastest omega. This curve which goes only up to about this height, the three curves. One of the curves which is swinging around is goes up to the height about 5, the second curve goes up to height about 2.5 and the third one goes to the height about 1.2. The curve that moves the lowest has the largest value of ω, that is, the fastest variation.

So the penetration is small. The effect goes only up to about this height. So L_0 can be interpreted as the penetration depth of the effect of the motion of the base plate into the fluid. When L_0 is large, that is for a given fluid, ω is small, the plate is moving slowly or the velocity is changing slowly, the frequency is low, then the depth of penetration is largest. Let us do one more problem.

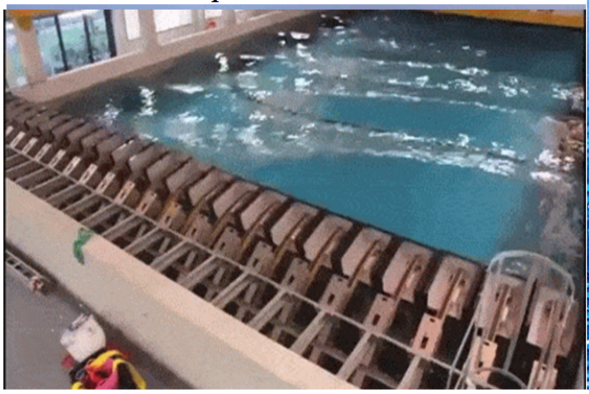
Abo8ut ocean waves and currents. An offshore oil drilling platform is expected to encounter waves of 4 meter height at a frequency of 0.1 Hertz and a steady current of 1 m/s. Determine the parameters for the model wave channel where a one-sixteenth model of the platform can be tested.

These are the kind of channels used for testing this. big plungers produce waves within the ocean of the required frequency and required amplitude and the required height.

Ocean waves and currents

An off-shore oil-drilling platform is expected to encounter waves of 4-m height at 0.1 Hz frequency, and a steady current of 1 m/s. Determine the parameters for the model wave-channel where a one-sixteenth model of the platform can be tested.

The governing equation for this flow about an off-shore drilling platform would contain the unsteady term and the convective acceleration term and the term representing the pressure forces and the gravity forces which play important roles in this situation. The viscous forces are rather unimportant and can be ignored.



There is a flow velocity that is also superimposed on this and the model rig is placed in this channel for doing the study. The governing equation for this flow about an offshore drilling platform would contain the unsteady term and the convective acceleration term because of the steady current and the term representing pressure forces and gravity forces which play important roles in the situation: $\rho\left[\frac{\partial V}{\partial t} + (V.\nabla)V\right] = -\rho g k - \nabla p$. The viscous forces are rather unimportant and can be ignored. So the equations would contain these terms, unsteady term, convective acceleration term, the gravity term and the pressure term. And when we non-dimensionalize this by defining $x^* = x/L_o$, a characteristic length, $V^* = V/V_o$, a characteristic velocity, $t^* = \frac{t}{\tau}$, a characteristic time. Or if the frequency is f_o, then we can define $t^* = f_o t$), and $p^* = p/p_o$.

And this is the equation that results: $\frac{f_o L_o}{V_o} \bullet \frac{\partial V^*}{\partial t^*} + V^* \bullet \nabla^* V^* = -\left(\frac{p_0}{\rho V_o^2}\right)\nabla^* p^* - \left(\frac{g L_o}{V_o^2}\right)k$. This number is named after a scientist, Vincenz Strouhal and is abbreviated as St, a very important number that plays an important role in unsteady flows. So in addition to Euler number and the Froude number, we have Strouhal number also that plays a part. Here the prototype length, characteristic length in the prototype, can be taken as 4 meters the velocity in the prototype is 1 m/s, the steady current velocity and the frequency in the prototype is 0.1 Hertz. This is the given data.

We can introduce a non-gravitational pressure difference $P = (p + \rho g z) - p_o$. So that terms on the right hand side combine to give only one term, this term $-\frac{(\Delta p)_o}{\rho V_o^2}\nabla^* P^*$. As before, we argue that since there is only one pressure, the atmospheric, there is no other pressure, there is no cavitation, the characteristic pressure difference can be taken to make $\frac{(\Delta p)_o}{\rho V_o^2}$ of order 1. So there is only one pi number that occurs in the equation that is $\frac{f_o L_o}{V_o}$. This is Strouhal number. But we need to consider the boundary conditions as well, as we have discussed before.

Ocean waves and currents

When the non-gravitational gauge pressure $\mathcal{P} = (p + \rho g z) - p_o$ is introduced in the3 boundary condition, this modifies to $\mathcal{P} = \rho g z_f$ at $z = z_f$, which, on non-dimensionalization ($\mathcal{P}^* = \mathcal{P}/(\Delta p)_o$ and $z^* = z/L_o$), gives $\mathcal{P}^* = \frac{\rho g L_o}{(\Delta p)_o} z_f^*$ at $z^* = z_f^*$.

With $(\Delta p)_o = \frac{1}{2}\rho V_o^2$ this can be recast as $\mathcal{P}^* = \frac{2}{\mathrm{Fr}^2} z_f^*$ at $z^* = z_f^*$

Thus, the modelling rules require Strouhal number $\frac{f_o L_o}{V_o}$ and the Froude number $\frac{V_o}{\sqrt{g L_o}}$ to be identical in the prototype and the model for similarity

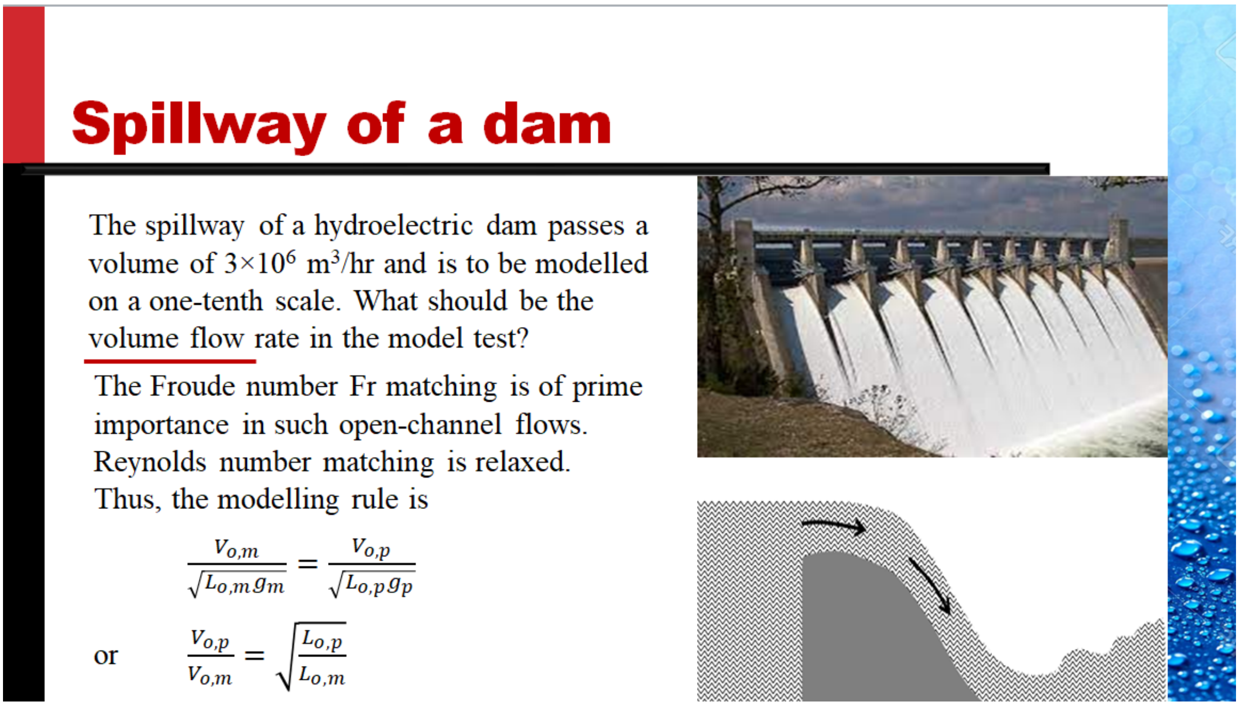
The presence of the free surface or liquid exposed to atmosphere introduces a boundary condition $p = p_o$ at $z = z_f(x, y)$. So there is the presence of a free surface and because of the free surface, Froude number now occurs in the boundary conditions, and that needs to be matched. This is a combination of Froude number and Euler number as discussed before. With $(\Delta p)_o = \frac{1}{2}\rho V_o^2$, the dynamic pressure, this can be recast as $P^* = \frac{2}{Fr^2} z_f^*$ at $z^* = z_f^*$. Notice that since there is only one pressure, the Euler number can be made to disappear.

And so we are left with only Froude number. Thus, the modeling rule requires Strouhal number and the Froude number to be identical in the prototype and the model for similarity. Matching of Froude number gives $\frac{V_{o,p}}{V_{o,m}} = \sqrt{\frac{L_{o,p}}{L_{o,m}}} = 16^{1/2} = 4$. So the velocities of the prototype would be 4 times the velocity of the model. The steady current velocity in the model should be only one fourth of the velocity in the prototype.

Strouhal number matching gives the frequency for the model to be 0.4 Hertz, 4 times that of the prototype. Prototype was oscillating with 0.1 Hertz. Since the flow geometries are to be similar in all aspects, the ratio of the height h of waves should be same as that of the characteristic lengths.

And so the height of the waves h_m is equal to height of the waves in the prototype times the length scale of the model divided by length scale of the prototype and this gives you 0.25 meters. So instead of 4 meter high waves we need only a quarter meter high waves in the model channel, something that is easy to achieve.

Let us do one more example. Modeling the spillway of the dams. Water stored in the dams and need to be drained, and when it is drained, since it is a great height it acquires a lot of velocity. And if the water falls freely then it will acquire such a velocity that will damage the surface the ground below, and so a spillway has to be carefully designed. Typically this spillway looks like this. The water coming down the spillway is, accelerates very much and so this portion has to be designed so that there is a hydraulic jump in which the fluid slows down and the level of water rises in the channel. The spillway of a hydroelectric dam passes a volume of 3×10^6 m^3/hr, and is to be modeled on a one tenth scale.



What should be the volume flow rate in the model test? Since this is a problem with the free surface, cavitation is not important. So Froude number matching is of prime importance in such open channel flows. Reynolds number matching could be relaxed because the dependence on Reynolds number is low as discussed in the beginning of this lecture. Thus, the modeling rule is simply the matching of Froude number. Froude number for the model should be same as Froude number for the prototype.

From this we get the velocity ratio $\frac{V_{o,p}}{V_{o,m}} = \sqrt{\frac{L_{o,p}}{L_{o,m}}}$. And the volume flow rate, volume flow rate should vary like the velocity times the area, and since it is geometrically similar, the area would be like characteristic length squared. So the volume flow rate in the protype would be $\left(V_o L_o^2\right)_p$ and for the model this should be m, and this gives you $10^{5/2}$. So given that the volume flow rate of the protype is 3 million meter cubed per hour, we obtain the volume flow rate in model should be about one hundredth, or little less than one hundredth of the flow rate in the model.

Thank you.