

**Similitude And Approximations In Engineering,**  
**Vijay Gupta**  
**Applied Mechanics**  
**Indian Institute of Technology Delhi**  
**Week - 07**  
**Lecture - 25**

Welcome back. We continue with our discussion of the relaxation of the modeling rules. In the last lecture, we had introduced the concept of weak laws and strong laws. In today's lecture, we will consider the application of the same principle when the relaxation cannot be applied to the whole domain but to segments of this. This is called segmented modeling. In many cases we cannot ignore a governing law throughout the entire range of investigation.

## Segmented Modelling

In many cases, we cannot ignore a governing law throughout the entire range of investigation, but we can break up the phenomenon either spatially or temporally, and use fewer laws in each of the regions

- Boundary layers in fluid flows: regionally segmented.
- Phenomenon changes behaviour as time passes: surface tension kicking in in spread of oil spills after many days
- Phenomenon displays different behavior in different directions or at different speeds.

But we can break up the phenomena either spatially or temporally and use fewer laws in each of the regions. Boundary layers in a fluid flow, as we have discussed this before, is regionally segmented. One region near the wall where different laws are applied: viscous forces is important. And the region away from the wall, which is called the outer flow where the viscous forces can be neglected.

The change in phenomena as the time passes. We have done one example of the spreading of an oil slick where the surface tension kicks in after many days. We have also done examples where the phenomenon displays different behavior in different directions or at different speeds. We will do a systematic study of this now. Sequential modeling, sequential time wise.

## Sequential modelling

### One phase following another

In modeling automobile crashes the crash phase can be clearly distinguished from the post-crash phase.

- During the crash phase, which usually lasts only for a fraction of a second, tyre friction forces are negligibly small compared to the large inertial forces of the impacting cars. Hence, the crash phase is governed by inertial forces and by energy dissipation in partially elastic and imperfectly smooth bodies.
- Tyre friction forces do not come into play until the beginning of the post-crash phase, which may last for several seconds. The energy dissipation due to impact need no longer be modelled.

One phase following another. In modeling automobile crashes, the crash phase can be clearly distinguished from the post crash phase. During the crash phase which usually lasts only for a fraction of a second, tire friction forces are negligibly small compared to the large inertial forces of the impacting cars. Hence, the crash phase is governed by inertial forces and by energy dissipation in partially elastic and imperfectly smooth bodies of the cars. The tire friction forces do not come into play until the beginning of the post crash phase which may last for several seconds. The energy dissipation due to impact needs no longer be modeled in this phase. So, two phases are modeled using different sets of laws.

## Sequential modelling

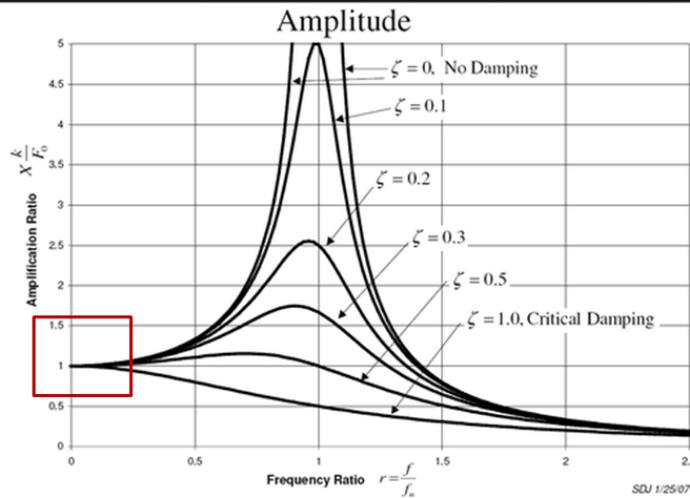
### Another example of sequential modelling:

Settling of microscopic particles at very low Reynolds number:

- Initially for a few milliseconds, both the inertia of the fluid as well as that of solid is required
- Then the fluid flow may be considered quasi-steady, and its inertia ignored, but the solid is still accelerating
- Ultimately, the solid is travelling at terminal speed and inertia of both the solid as well as fluid can be ignored thereafter.

Another example of sequential modeling is the settling of microscopic particles in very low Reynolds number flow. This also, we had done earlier. Initially for a few milliseconds both the inertia of the fluid as well as that of the solid are required. The fluid is accelerating, the solid is accelerating. Then, if the Reynolds number is very low, the fluid may be considered quasi-static after the first phase, and its inertia ignored. But the solid is still accelerating. Ultimately, the solid body is travelling at the terminal speed and the inertia both of the solid as well as of the fluid can be ignored thereafter. We had analyzed this in quite some details.

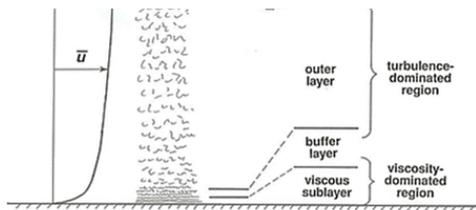
# Regional modelling



Regional modeling where the segmentation is done on space. In modeling of slow moving off-road vehicles, the forcing frequency is quite low. And all the sprung and unsprung masses are considered as rigid bodies. Consequently, those masses are subjected only to Newton's law of inertia and gravitation. This represents the amplification of a spring dashpot mass model that we discussed.

For low frequency ratios,  $f$  divided by  $f_n$ , the natural frequency, When this is low, this is the mass controlled motion in the red box. We can neglect all springiness in the system. And we can model this as if only the inertial forces are important and the gravity forces. On the other hand, all springs have negligible small mass, follow only Hooke's law of elasticity, And all shock absorbers also having negligible mass follow only the law of viscous friction. So, the masses of springs and of dashpots are ignored. Hence, the whole vehicle can be partitioned into separate regions, where different laws apply. So, that the masses are considered to be without any elasticity. The springs and dashpots are considered to be without any mass. Under these circumstances, modeling is considerably relaxed.

# Regional modelling



In turbulent flow of a fluid through a pipe or channel — inertial effects are confined to the main stream whereas viscous effects are confined to the vicinity of the walls. This difference permits a number of relaxations. Because the major effects of viscous friction are restricted to the vicinity of the walls (where a large velocity gradient exists), the law of viscous friction need not be expressed in truly general representative terms that would apply to any conceivable situation. Instead, it can be specified in terms of the boundary layer in which viscous friction takes place.

Another case of regional modeling is, of course, the turbulent flow of a fluid through a pipe or channel. The inertial effects are confined to the main stream whereas, the viscous effects are confined to the vicinity of the walls. This difference permits a number of relaxation. Because the major effects of viscous friction are restricted to the vicinity of the wall where a large velocity gradient exists, the laws of viscous friction need not be expressed in truly general repetitive terms that would apply to any conceivable situation. Instead, it can be specified in terms of the boundary layer in which viscous friction takes place.

We apply this principle to an unrelated topic, the introduction of hydraulic radius in flow through non-circular pipes or in open channels. In such flows, the inertial effects are confined to the main stream whereas, viscous effects are confined to the vicinity of the walls. Consider this elliptical tube. I drew elliptical tube because it was easy to draw on a power point, but it could be a tube of any section. The region within this is divided into two regions. The dark grey represents the boundary layer near the wall within which the viscous effects play an important role.

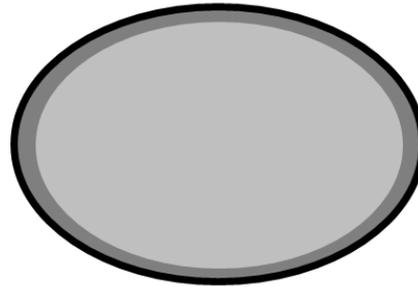
# Hydraulic radius

In the turbulent flow of a fluid through a pipe or channel, the inertial effects are confined to the mainstream whereas viscous effects are confined to the vicinity of the walls. This permits a number of relaxations.

$$F_v \sim \tau(P_w L) \sim (\mu V / \delta)(P_w L)$$

$$F_i \sim (\rho L A) V^2 / L \sim \rho A V^2$$

From these we get a Reynolds number-like Pi-number  $\rho V / \mu \cdot A / P_w \cdot \delta / L$ .



The light grey region in the middle away from the walls is a region where viscous effects can be neglected and only inertial effects need to be considered. So, how does it help us? The viscous forces on the wall are like tau, the viscous stress times the area of the wall which is written as the weighted parameters times the length of the perpendicular to the plane of the screen. So, the area here is represented by  $(P_w L)$ , the weighted parameters times the length.  $\tau$  of course, is the shear stress and there is  $\mu$  times the velocity gradient, and the velocity gradient is in a thin region of thickness  $\delta$ . So, tau is estimated by  $(\mu V / \delta)(P_w L)$ . So, this is the viscous force. The inertial force on the other hand is in the main region over the whole area. So, that  $F_i \sim (\rho L A) V^2 / L$ , where L is the characteristic dimension of the cross section of the pipe. And so, this gives us a force of inertia as  $\rho A V^2$ . From these we get a Reynolds number like pi number  $\rho V / \mu \cdot A / P_w \cdot \delta / L$ .

Now, there is a leap of faith. Consider two pipes of the same length carrying the same fluid at the same average speed. Two pipes same length carrying the same fluid at the same average speed. It stands to reason that the boundary layer thickness in the two pipes would be the same and flow would be similar if  $A / P_w$  of the two pipes are same. If one pipe is circular the value of its  $A$  divided by  $P_w$  is obtained as  $\pi D^2 / 4$ , the area, and the wetted parameter  $\pi D$  for a circular pipe.

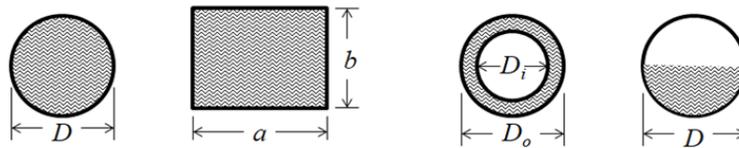
So, this gives you  $\frac{A}{P_w} = \frac{\frac{\pi}{4} D^2}{\pi D} = D / 4$  where  $D$  is the diameter of the pipe. Thus the diameter of a circular pipe the flow through which may be considered similar to the flow through a given non circular pipe should be 4 times  $A / P_w$ . This is termed as the hydraulic diameter  $D_h$  of the pipe. Thus we could use all the formulation of a circular pipe for a non circular pipe if the diameter  $d$  in that formulation is replaced by the hydraulic diameter  $D_h$  which is  $4A / P_w$ .

Again thus the pressure drop in a non circular pipe can be predicted quite well by calculating the pressure drop in a circular pipe of diameter equal to  $4A/P_w$ .

Four different pipes a circular pipe, a rectangular pipe, a annular pipe where the flow is between two diameters  $D_i$  and  $D_o$  and a half filled pipe. I calculate the hydraulic diameters of these four pipes. In the last column are the hydraulic diameters. Using these hydraulic diameters for diameter in the Moody chart we can find out the head losses.

## Hydraulic radius

Thus, the pressure drop in a non-circular pipe can be predicted quite well by calculating the pressure drop in a circular pipe of diameter equal to  $4A/P$ .

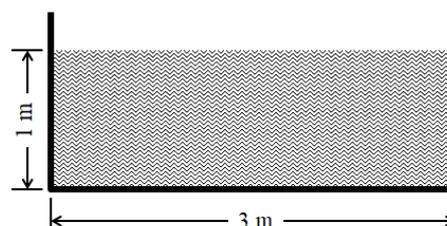


Cross-section	P	A	$D_h = 4A/P$
Circular, diameter D	$\pi D$	$\frac{\pi}{4} D^2$	D
Rectangular, a × b	$2(a + b)$	ab	$\frac{2ab}{a + b}$
Annular, $D_o$ and $D_i$	$\pi(D_o + D_i)$	$\frac{\pi}{4}(D_o^2 - D_i^2)$	$(D_o - D_i)$
Circular, half-full	$\frac{\pi}{2} D$	$\frac{\pi}{8} D^2$	D

The same principle can be applied to open channels. For open channels the pressure does not change and the change in non gravitational pressure  $P = p + \rho g z$  is related to the change in elevation.

## Hydraulic radius

For open channels the pressure does not change and the change in non-gravitational pressure  $\mathcal{P} = p + \rho g z$  is related to change in elevation



So, if this was a rectangular open channel then the wetted parameter of this is 1 meter plus 3 meters plus 1 meter 5 meters and the area of the flow is 3 into 1 3 meters squared. So, we can

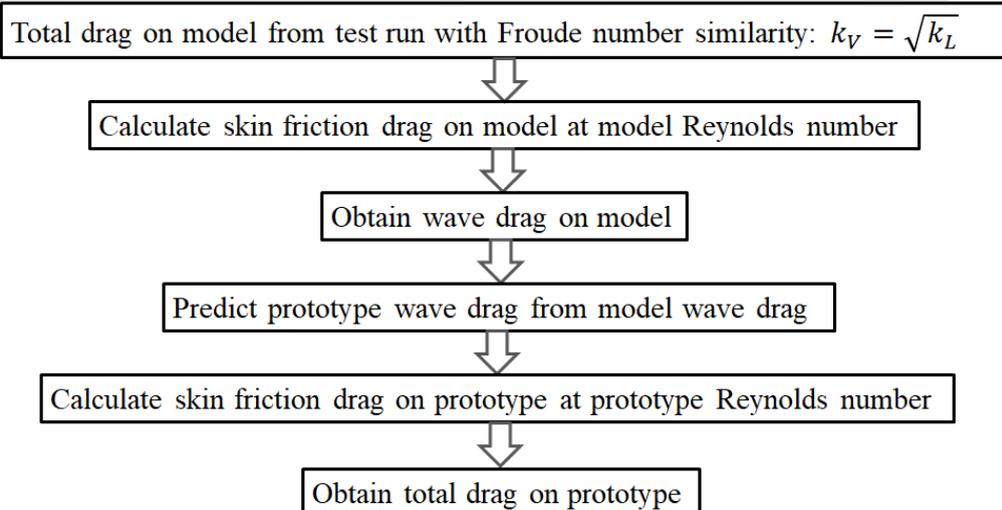
calculate the hydraulic diameter as  $4A/P_w$  and treat this as a closed pipe of that diameter. The results that we get are very close to result we get from much better analysis.

Let us do one more examples of this segmented modeling. Testing of ship models we are interested in finding out the drag on the ship. The drag on the ship as it moves in open ocean consists of two parts. One the viscous friction termed as a skin friction on the surface of the ship in contact with water. The other arises is related to the energy that is dissipated in the waves that move away from the ship. So, the drag of the ship has two components the skin friction drag and the wave drag. The wave resistance and the skin friction modeling would require the Reynolds number and Froude number match.

Reynolds number invariance gives  $k_v = k_v/k_L$ ,  $\nu$  for kinematic viscosity, and the Froude number invariance gives  $k_v = k_L^{1/2}$ . If the same fluid is used in the model as in the prototype then the modeling without drastic relaxation becomes impossible unless we use  $k_L$  is equal to 1, which is hardly any modeling. But if different fluids are used then  $k_v$  is like  $k_L^{3/2}$ , makes it possible to choose an appropriate  $k_L$ . If I can choose a fluid with the kinematic viscosity which is much lower than kinematic viscosity of water.

But even a small length scale of 1, 100 this  $k_L$  of 100 would require a  $k_v$  of 1000. That means the kinematic viscosity of the fluid used in model study should be 1000th of the kinematic viscosity of water. Fluids of that low viscosity are not available. So, this avenue is not open to us. Because of these conflicting scaling requirements, the hull friction, which is the skinned friction, and the wave resistance cannot be modeled simultaneously.

## Testing ship models



However, with friction effects confined predominantly to the hulls boundary layer and the wave raises predominantly to the hulls boundary layer and the gravity inertia effects restricted to the waves around the ship it is argued that the two raises can be determined separately. The total drag which depends upon Reynolds number and Froude number is broken up into two

parts. One the skin friction drag which depends upon Reynolds number and the wave drag which depends upon the Froude number. Now, skin friction drag is a well studied topic and can be estimated from the skin friction on a flat plate. The sides and the bottom of a ship can be modeled as flat plates and skin friction on them could be obtained analytically using the correlations for flat plates analytically.

So, how is how are the ship tested? We find the total drag on the model from a test run with Froude number similarity. So, we use a  $k_L$  which could be pretty large and run the test at a velocity given by the scale factor  $k_V = \sqrt{k_L}$  determine the total drag. Then calculate the skin friction drag on this model at the Reynolds number of the model determine at what Reynolds number was the test run and calculate using the flat plate formula the skin friction drag on the model. We subtract this skin friction drag from the total drag to obtain the wave drag on the model. The wave drag on the model must be scaled by using the Froude similarity.

So, predict the wave drag of the prototype from the model wave drag. The drag coefficients the wave drag coefficient on the model must be equal to the wave drag coefficient on the prototype. Since you know the wave drag on the model we can calculate the wave drag coefficient on the model and this same drag coefficient would apply on the prototype and using the  $\frac{1}{2}\rho V^2$  times the area of the prototype, we can predict the prototype wave drag from the model drag. Now, we calculate the skin friction drag on the prototype at prototype Reynolds number. I know the prototype Reynolds number. So, I can use the flat plate formula to calculate this skin friction drag and then add this skin friction drag to the wave drag predicted in the earlier step to obtain the total drag on the prototype. This is the scheme of testing of ship model for drag that is employed almost exclusively.

Directional modeling. When governing laws work in distinct directions instead of distinct regions, directional modeling can relieve stringent scaling requirements. We have done this earlier in a different context.

For example, if the turbulent flow in rivers and estuaries were faithfully scaled in horizontal direction as well as in depth, river models would be so shallow that the turbulent flow would be suppressed, and in addition, unwanted surface tension would be exaggerated. With directional modeling the depth of rivers and estuary models are very often disproportionately magnified. We use a smaller scale factor for the depth than for the length and width. Fully turbulent flows in a slope channel is governed by Newton's law of inertia and the law of gravitation, but the inertial forces act predominantly in a horizontal direction while the gravitational forces act in a vertical direction. Since these two forces act in different dimensions we can use different scale factors in the two direction.

## Modelling large river basins



1943-1966

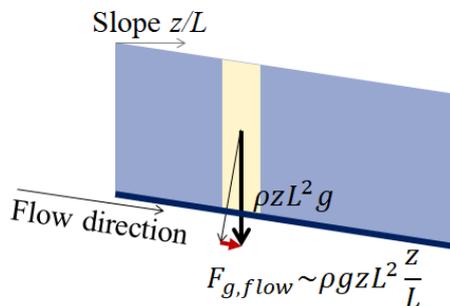
Scale: 1:2,000

Spread over 200 acres

This picture is from a section of a Mississippi River Basin model which is spread over 200 acres. This was constructed in 1943 and decommissioned in 1967 after giving wonderful results. After that the computational models came to help and it was no longer economical to maintain this very very expensive model. This model for length uses a scale of 1 to 2000.

So 2000 kilometers are driven by 1 kilometer. This is a huge model spread over 200 acres. In a river model inertial and gravitational forces are important, and this again, results in the Froude number similarity. Inertial force scale factor and the gravity force scale factors give you  $k_v = \sqrt{k_L}$  using the same fluid. So  $k_\rho$  is 1 and  $k_g$  is 1, and for a 1 is to 2000 scale model, this gives you  $k_v$  of 44.7. The surface tension forces scale factor for surface tension forces is like  $k_{F,s} = k_\sigma k_L$  which gives you the ratio of the surface tension force to the inertial force as  $\frac{F_s}{F_i} = \frac{\sigma}{\rho LV^2}$ . And for the prototype this gives you a value of  $2.6 \times 10^{-6}$ . Very low. So the surface tension force is selected in the prototype. For model flow, if we follow a same dimensions for the depth, 1 is to 2000, then this gives you the surface tension force or inertial force of the order of 10. Definitely, surface tension forces cannot be neglected in the model. So obviously a model run on the basis of Froude similarity would fail because though the surface tension force in the prototype is negligible it is non-negligible model flow. So the model flow is not really modeling the prototype flow correctly.

# Directional Modelling



$$F_{i,flow} \sim \rho z L^2 \frac{L}{t^2} \rightarrow \Pi_i = \frac{F_{i,flow} t^2}{\rho z L^3}$$

Their ratio leads to

$$\Pi = \frac{gz}{v^2} \text{ (Froude number)}$$

$$\text{And then, } k_Q = k_v k_L k_z = k_L k_z^{3/2}$$

The developed pi-number for channel flow allows for a large difference between the horizontal and vertical length scale factor. Vertical scale factor is reduced.

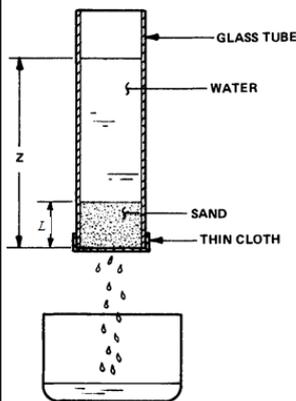
The Mississippi valley model used  $k_L = 100$  in the vertical direction.

So what do we do? Look at a sloping river bed. If we consider a slice of the fluid on this the weight forces  $\rho z L^2 g$ .  $L$  is the horizontal dimension,  $z$  is a vertical dimension. The slope is  $z/L$ . And the component of the gravity force in the flow direction then is  $F_{i,flow} \sim \rho z L^2 \frac{L}{t^2}$ ,  $z L^2$  is the volume now,  $\frac{L}{t^2}$  is the acceleration.

So the ratio of the gravity forces this gravity force and this inertia force now leads to a Froude number which is  $\Pi = \frac{gz}{v^2}$  the same as before except  $L$  has been replaced by  $z$ . So this permits us to use a different scale in the  $z$  direction then the scale in the length direction. Then the scale factor for the volume flow rate is  $k_Q = k_v k_L k_z$  and since  $k_v$  is like  $\sqrt{k_z}$  so  $k_Q = k_L k_z^{3/2}$ . The developed pi number for channel flow allows for a large difference between the horizontal and vertical scale factors. The vertical scale factor is reduced the Mississippi valley model uses  $k_L$  is equal to 100 in the vertical direction. So that the Weber number becomes of order of 0.5, less than 1 and so the surface tension effects even in the model flow almost negligible.

This example illustrates a very important point in relaxation. Before disregarding the law we must satisfy ourselves of its insignificance not only in the prototype, but to the model as well. Too often a law of little importance to the prototype assumes great importance when applied to the model. Therefore, in very small models like harbors and estuaries surface tension is no longer insignificant unless we increase the depths.

## Flow through porous media



In the tests, sand of uniform grain size was held in the bottom of a glass tube by a thin cloth with negligible flow resistance; when the tube was filled with water, gravity made the water flow downward through the sand. Tests were performed with sands of three different grain sizes, with the depth of the sand layer made proportional to the grain size to maintain geometrical similarity.

Because the effects of gravitation depends on the water column's vertical height,  $z$ , while the inertial and viscous effects are confined to the layer of sand of the representative dimension,  $L$ , we can describe seepage by two representative lengths,  $z$  and  $L$ . Note that  $L$  represents the dimensions of the sand layer as well as grain and void sizes.

Let us do one more detailed example. Flow through porous media is governed by viscous friction between fluids and the solid grains of the porous media, the inertial force of the liquid and the weight of the fluid. Hence the correct modeling requires matching of the Reynolds and Froude number. This leads to the result that  $k_L$  must be 1 if the same fluid is used. Because from Reynolds number we get  $k_L = k_v/k_v$  and from Froude number we get  $k_L = k_v^2/k_g$ . If  $k_v$  and  $k_g$  are same these two equations can match only if  $k_L$  and  $k_v$  both are 1.

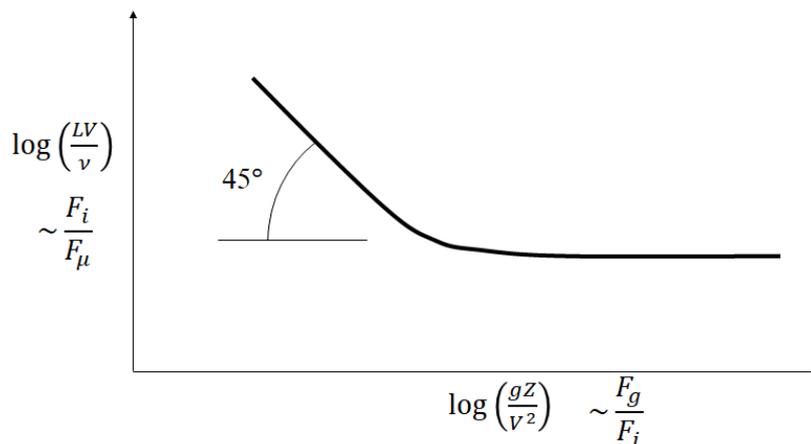
If we use a different fluid we need a fluid with scale factor kinematic viscosity as  $k_v = k_L^{3/2}$ . A requirement impossible to meet for large scale factors. Hence the problem is solved by employing two different length scale factors. Realizing that the two lengths a length  $L$  which characterizes the porous media the sand bed and a length  $z$  which characterizes the gravity force which is moving the water through the porous bed. In the test sand of uniform grain size was held in the bottom of a glass tube by thin cloth with negligible flow resistance.

When the tube was filled with water gravity made the water flow downwards through the sand. Tests were performed with sands of three different grain sizes with the depth of the sand layer made proportional to the grain size to maintain geometric similarity. Because the effects of gravitation depends on the water column vertical height  $z$ , while the inertial and viscous effects are confined to the layer of sand with the characteristic dimension  $L$ , we can describe seepage by two representative lengths  $z$  and  $L$ . Note that  $L$  represent the dimension of the sand layer as well as the grain and void sizes. So, when we are using a scaled length  $L$ , this sand size has to be reduced.

With these relaxation we can write the representative versions of the three governing laws as follows. Inertia of the fluid flow in the sand layer: characteristic length is  $L$  here. Mass times acceleration, Mass is  $\rho AL$ ,  $L$  being the characteristic length, and the acceleration is  $V^2/L$ . So, this gives you pi number for inertial forces  $\frac{F}{\rho AV^2}$ .

Viscous forces in the same layer: characteristic length  $L$ . So, force is  $F \sim \mu V / L \cdot A$ . So, that the pi number is  $\Pi_v = \frac{FL}{\mu AV}$ . And the gravitational forces: now, this is because of the column of water or height  $z$ . So, this is  $F \sim \rho z A g$ , and this gives you  $\frac{F}{\rho z A g}$  as the pi number. The two modeling rules for the same fluid, water, so that  $\rho$  and  $\mu$  and  $g$  are scaled by a factor 1. From the first equation we get  $LV$  is invariant. From the second equation we get  $z/V^2$  is invariant.

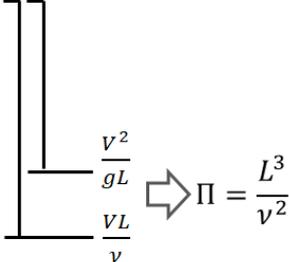
## Flow through porous media



This is the results from the test held on such a model as reported by Emori. This region  $\log \log\left(\frac{LV}{v}\right)$  is inversely proportional to  $\log \log\left(\frac{gZ}{v^2}\right)$ . If we look, if we compare if we look at this carefully we see that is inverse relation means that in this region the inertia force is unimportant, and only the viscous force need to be considered. But in the later region  $\frac{gZ}{v^2}$  is independent of  $\frac{LV}{v}$ . So, this leads to the conclusion that this region can be modeled as non-viscous. So, for larger value  $\frac{gZ}{v^2}$ . it is the inertia forces that count, not the viscous forces.

# Sediment waves

Inertia of solid particles	$\rho_s L^2 V^2$
Inertia of fluid	$\rho_f L^2 V^2$
Gravity of solid particles	$\rho_s g L^3$
Gravity of fluid	$\rho_f g L^3$
Viscous force	$\mu VL \sim \rho_f \nu VL$



One more example: one of the factors that unfavorably influence the transport rate of fluids in channel with sediment beds is the formation of ripples or miniature dunes on the bed. The small irregularities in the bed give rise to vortices whose increased velocity causes particles of the bed to move out of the range of the vortex and to settle down at some distance, thus serving to generate a new vortex, and so on. We need to determine the dominant mechanism here. What is that causes the sediment waves? You could think of five different forces. Inertia of the solid particles which would be like  $\rho_s L^2 V^2$ . Inertia of the fluid which will be like  $\rho_f L^2 V^2$ . The gravity of the solid particles, the gravity force, the weight of the solid particles, which will be  $\rho_s g L^3$ , and the viscous forces which would be like  $\mu VL$ ,  $\mu$  is the viscosity of the fluid. So,  $\mu$  is replaced here by  $\rho_f \nu$  the density of the fluid times kinematic viscosity  $\nu$ . Inertia of the fluid and viscous forces give you Reynolds number,  $\frac{VL}{\nu}$  should be invariant. And the inertia of the fluid and the gravity force will give you the Froude number  $\frac{V^2}{gL}$ . And from this we get  $\Pi = \frac{L^3}{\nu^2}$  as invariant. The pi number that control this fluid. If you use water as a model material, and 49.5 percent glycerol solution as prototype, we get  $k_\nu \sim 5.2$  and then  $k_L \sim 3$ , which is not large enough.

And if we do that,  $k_\rho = 1.27$  for the two fluids must apply to solid particles as well. The density ratio  $k_\rho$  for the glycerol and water is 1.27. And so this must apply to the solid particle as well, same density scale factor should be used.

Coal particles with the density of  $1.26 \text{ kg/m}^3$  and the Araldite epoxy resin with the density of  $1.26 \text{ kg/m}^3$  could be used as the prototype and model sediments. So, it is possible to construct a model with glycerol and coal particles. But since length scale factors was not satisfactory large, it was decided to use water itself as a model fluid, and add relaxation. With the same fluid Reynolds and fluid numbers would result in conflicting model requirements. This could be avoided however if the inertial force of the sediments were much smaller than the rest of the forces and therefore could be neglected. Inertia of the fluid is  $\rho_f L^2 V^2$ . A further simplification results when without loss of generality will restrict the gravity to the wet weight of particle. So,

the weight force is now considered  $(\rho_s - \rho_f)gL^3$ . These two result in  $\frac{\rho_f V^2}{(\rho_s - \rho_f)gL}$ , a modified Froude number, which is known as a densimetric Froude number, as a pi number.

And the inertial and viscous forces give the Reynolds number as the other pi number.  $\frac{\rho_f VL}{\mu}$  for the same fluids gives  $k_V = 1/k_L$ . And the densimetric Froude number for the same fluid gives you  $k_V = \sqrt{k_{(\rho_s - \rho_f)} k_L}$ . And from this, by eliminating  $k_V$ , we get  $k_L = \sqrt[3]{k_{(\rho_s - \rho_f)}}$ .

The model tests were performed in a tilting frame 18 meter long and 65 cm wide. The model sediment was sand with a density of 2.65 kg/m<sup>3</sup>. The prototype sediment polystyrene with a density of 1.035 kg/m<sup>3</sup>. The fluid, water with a density of 1,000 kg/m<sup>3</sup>. With these data  $k_L = 3.6$ . The average grain diameter of the sand was 0.375. Therefore, the grain diameter of the polystyrene sediment was made 1.35 millimeter using the same scale factor of 3.6.

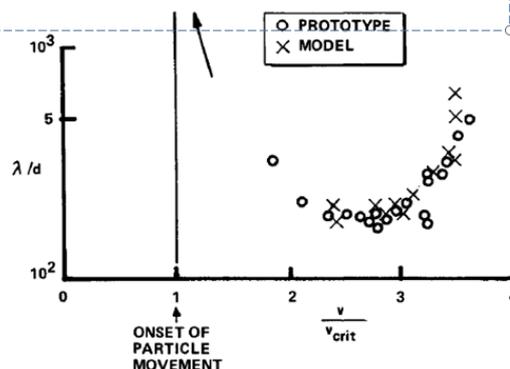
So, this was the testing arrangement. This flume was inclined at an angle alpha. Alpha adjusted such that it gave you the required flow rate. The same flume was used for model as well as the prototype test since it was assumed that the formation of sediment waves is a two dimensional process.  $k_V = \frac{1}{k_L} = 1/3.6$ . The necessary speed was achieved by tilting the flume by a very small angle. Model and prototype tilting angles were computed from the represented gravity components in the flow direction, that is,  $\rho_f g L^3 \alpha$ , obtained earlier, where alpha is the tilt angle.

Hence, the new pi number was introduced with inertial forces  $\frac{gL\alpha}{V^2}$ . For two dimensional flow Q is like  $Vbh$  where b is the flume width and h is the flow height. For constant flume width  $k_Q = k_V k_L = 1$ .

## Sediment waves

For two-dimensional flow,  $Q \sim Vbh$ , where  $b$  is the flume width, and  $h$  is the flow height. For constant flume width,  $k_Q = k_V k_L = 1$

Also,  $k_h = k_L = 3.6$



Also  $k_h = k_L = 3.6$ . And if you plot this, we get this. Onset of fluid particle motion is called  $V_{\text{critical}}$ . And this plots  $V / V_{\text{critical}}$  versus  $\lambda/d$ , the wavelength divided by the diameter of the grains. And we obtain this curve where all points collapse onto one curve suggesting that the formulation is correct.

Thank you very much. .