EPSRC Centre for Doctoral Training in Industrially Focused Mathematical Modelling

Homogenisation approaches for non-static microscale structures

Clint Wong

Contents

1. Introduction

The understanding of fluid interactions with vegetative deformable structures is an important challenge: it has a wide range of applications including flood control, environmental conservation, and energy production.

We focus on fluid flow over a uniform and submerged vegetated region. In particular, we consider the situation where the vegetated region consists of individual reeds that are deformable. An example of this situation is shown in Figure 1.

Figure 1: Water flow through a submerged vegetated region. *Source: Blue Carbon Lab*

Modelling fluid flow through vegetation is an ongoing challenge. This is due to the geometry and the deformable nature of the vegetation. The resultant flow is also turbulent.

The modelling of fluid flow through vegetations has been, and still is, an ongoing challenge, due to the fact that the hydrodynamics are controlled by both the macro-scale and micro-scale properties. The key challenges include how to capture: (i) the elastic deformation of the vegetation, (ii) the geometry of the vegetation, and (iii) the turbulent behaviour of the flow.

The velocity profile of the fluid is coupled with deformation of the vegetation. As a result, the (location of the) interface between the two is unknown and has to be found as part of the solution. The vegetative layer, more commonly referred to as the canopy, is also challenging to model, due to its heterogeneity and the complex geometry of individual plants within it. For example, a tree has components with length scales that differ in multiple orders of magnitude, from leaves to branches to the trunk. The plants can also grow in a certain way and experience permanent deformation due to the prevailing direction of the flow. Finally, flow through vegetation is typically turbulent in real-life scenarios. As a result, the irregularity of the flow makes it challenging to describe. Turbulence is generated predominantly due to the strong velocity differences between the flow above and through the canopy.

Modelling fluid-flow through vegetation is a complex multi-scale problem that has typically demanded sophisticated experimental or numerical studies. The Coastal and Hydraulics Laboratory of the U. S. Army Corps of Engineers is interested in whether simpler models are able to produce both quantitative and qualitative agreement with more complex models; we seek to answer this question. We highlight the previous research by Ghisalberti and Nepf [1] and Mattis et al. [2], which forms the basis for this study.

Based on their experimental work on flow past a submerged canopy, Ghisalberti and Nepf [1] develop a uni-directional flow model (a one-dimensional flow from upstream to downstream, with the fluid velocity depending on the distance from the channel bed) incorporating rigid reeds. Their predictions were found to be in good agreement with the experimental data, but it is unclear whether the number of fitting parameters used in their complex model can be reduced. In contrast, Mattis et al. [2] consider and solve a 3D model for flow through a canopy using the immersed boundary method, a numerical technique which significantly reduce the computational cost of simulating individual reeds in the canopy. Although their results on the velocity profile are qualitatively promising, they are not able to achieve quantitative agreement with experimental data.

We use the framework by Mattis et al. [2] to construct a simple and compact model that is analogous to the model by Ghisalberti and Nepf [1]. Our derivation utilises the technique of homogenisation, where we derive the bulk properties of the canopy instead of modelling individual plants. Our aim is to explain the physical phenomena observed in Ghisalberti and Nepf [1].

2. Flow through a submerged canopy

We consider a two-dimensional horizontal channel containing water of depth H. The channel bed is flat and is entirely covered by a vegetative canopy that consists of individual reeds of length h. The reeds are all assumed to be fully submerged under water. A schematic of the setup is shown in Figure 2.

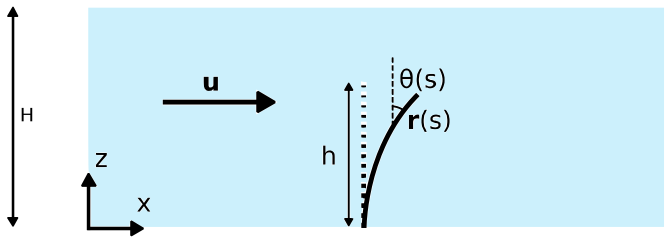

Figure 2: Schematic diagram for the two-dimensional channel flow in the x-z plane. The solid curve denotes the centre-line of a single flexible reed in the canopy that is rooted to the channel bed. The dotted line with length h denotes the undeformed configuration of the reed.

The motion of the fluid and the deformation of the reeds is coupled. The governing equations for both the fluid and the reeds must be solved simultaneously.

Our aim is to solve for the fluid velocity and the configuration of the reeds within the canopy. Since the fluid applies a load onto each reed, the fluid loses momentum as a result. We consider mass and momentum balances and solve for the fluid velocity and reed configurations, assuming that each reed as an inextensible elastic beam with a circular cross-section with diameter b. Each reed is rooted perpendicularly to the channel bed, and consistent to our two-dimensional framework, undergoes planar deformation (see Figure 2). Thus, for a given load, we deduce the beam configuration using the theory of elasticity. However, we emphasise that the motion of the fluid and the solids are coupled; the Navier-Stokes equations and the beam equations have to be solved simultaneously. To develop a simple and compact model, we consider the flow to be uni-directional (a one-dimensional flow from upstream to downstream that only depends on the distance from the channel bed) and we homogenise the influence of the individual reeds.

Homogenisation

We can approximate the momentum loss at each point in the canopy by carefully averaging momentum loss in a local area

In real-world scenarios, the canopy has a complex micro-structure. Even with the simplifications that we have made in the previous section, it is typically impractical to monitor the effects of individual reeds in the flow. Since there is a separation of length scales between the size of the reeds and the size of the river, in developing an efficient method to model the system, we can derive the bulk properties of the canopy by abstracting the physics from the reed scale. In particular, we are interested in how fluid momentum is lost by flowing through an array of reeds.

Rather than considering all the individual reeds in the momentum equation, we derive a single collective term that represents the influence on the fluid flow of all the reeds, by averaging the momentum loss on the reed scale. To illustrate this homogenising procedure, consider the disk of radius R shown in Figure 3. The total momentum loss in this area is the sum of the momentum loss from each reed within the disk; the corresponding average momentum loss at any point within the disk is the total divided by the area. Provided that the radius of the disk is sufficiently small compared with the size of the flow domain, the local flow in that area will be approximately uniform. As a result, the momentum loss from each reed within the disk is approximately equal. By taking the limit where R is infinitesimally small, we deduce a single collective term which approximates the local momentum loss at each point within the canopy.

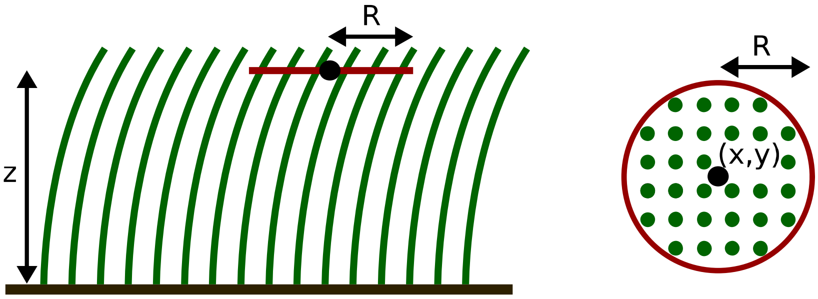

Figure 3: Schematic diagrams of a circle of radius R (red) encircling a collection of reeds (green) in the canopy. The circle is centred at the black dot, with coordinates (x, y, z). (left) Side-view of the circle. (right) Top-view of the circle.

The homogenised sink term, F has the form $F = \bar{N}f/cos\theta$, where f is the local momentum loss due to a single beam, θ is the local angle of deflection (see Figure 2), and $\bar{N}$ is the number of reeds per unit area. We highlight that the sink term is now proportional to the density of reeds in the canopy.

From here onwards, we will consider the particular case of a steady flow over a distribution of reeds of equal diameter and length. In real-life scenarios, the liquid flowing in the channel will have a free surface, and the water depth will be a function of both space and time. For simplicity, we do not model the free surface and so we treat the water depth, H, as a constant. We apply a constant pressure gradient along the channel, with a no shear conditionon the velocity at the top of the channel. The simplifications allow our problem to be solved as a system of simple equations with respect to z, the distance from the bottom of the channel. We present and discuss our findings in the next section.

3. Flow over rigid and flexible beams

In this section, we highlight our findings in this study. We first non-dimensionalise the governing equations and discuss the key parameters that govern the flow over a submerged canopy. We then present and analyse our numerical results, varying the different dimensionless parameters. Finally, we compare our results with the experimental data from Ghisalberti and Nepf [1] and discuss the effect of turbulence.

Dimensionless parameters

We non-dimensionalise the governing equations to abstract the key parameters that are intrinsic to the system. They also quantify the relative importance between different physical effects. With our model, the key dimensionless parameters are:

- $Re = UH/\nu$, the Reynolds number, which is the ratio between inertial and viscous forces. In this expression, U is the velocity scale and ν is the kinematic viscosity of water;
- $P = \Delta p/\rho U^2$, the pressure gradient along the channel. In this expression, Δp is the dimensional pressure gradient imposed and ρ is the density of water;
- $B = EI/\rho C_D b H^3 U^2$, the stiffness of the reed. In this expression, EI is the flexural rigidity of the reed and C_D is the geometric drag coefficient of the reed;
- $\tilde{\lambda} = C_D \bar{N} b H$, the canopy density;
- $\tilde{h} = h/H$, the submergence ratio of the canopy.

For the regimes that are relevant to this study, we note that Re is typically large and the flow is often turbulent. We will now analyse the numerical predictions of our model, using the parameters that we have presented here.

Numerical results

Although there is a vast amount of choice of parameters in the parameter space, varying Re and P gives results that one would anticipate. An increase in either parameter corresponds to an increase in the flow speed. Hence, we would also expect the beams to deflect more. This is consistent with physical intuition. However, it is not immediately obvious what role the canopy density plays, since $\tilde{\lambda}$ is a local property to the channel. Therefore, we focus our analysis on the interesting case where $\tilde{\lambda}$ is varied.

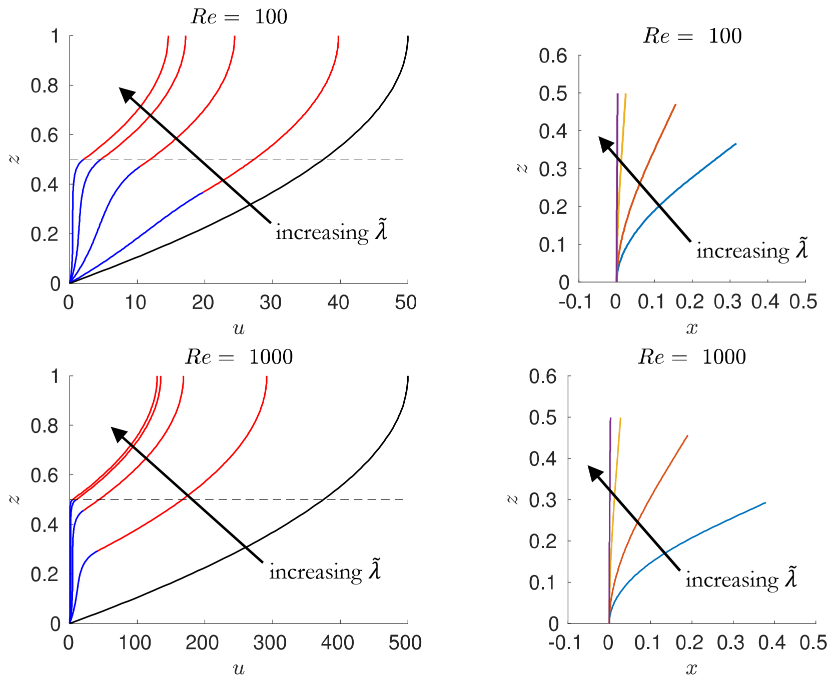

Figure 4: (left) Graphs showing how the velocity u changes across the channel, and (right) the corresponding shape of the reeds. The results are from numerical predictions of the model for canopy densities $\tilde{\lambda} = 10^{-2}, 10^{-1}, 10^{0}, 10^{1}$, and for two different values of Re. The red and blue curves indicate velocity profiles above and below the canopy respectively. The solid black curve is the analytical profile in the reed-free limit $\tilde{\lambda} = 0$ and the dashed line indicates the canopy height when there is no flow.

The velocity profile is approximately uniform in the canopy and parabolic above the canopy

We observe in Figure 4 that, for both Reynolds numbers, increasing the canopy density reduces the velocity for all values of z. Furthermore, as expected, the slowing of the flow in the canopy becomes more prominent as the Reynolds number is increased, since the homogenised sink term (in the fluid momentum equation) scales with the square of the velocity. When the canopy is sufficiently dense, we observe two distinct flow regimes within the channel. Above the canopy, the velocity profile remains parabolic. In contrast, the in-canopy flow is approximately uniform. A mathematical analysis of the governing equations suggests that the velocity within the canopy is given by $u \approx (2P/\tilde{\lambda})^{1/2}$. The phenomenon of having two distinct regimes is observed experimentally in flows through aquatic and terrestrial canopies.

Regarding the reed deformation, we notice that an increase in the canopy density reduces the mean reed deformation. This is a direct consequence of flows being slower when the canopy is dense, inducing smaller loads on individual reeds.

With the insights that we have gained from our numerical results, we would like to verify whether our simple model can quantitatively explain experimental results. In this work, we focus our verification on the case where the reeds are rigid and perpendicular to the channel bed.

Turbulence significantly influences the flow near the top of the canopy and above the canopy

A comparison with experimental results

We compare our predictions with the data in Ghisalberti and Nepf [1], and we find that our model overestimates the velocity both near the top of the canopy and above the canopy. This discrepancy is because we have neglected the effects of turbulence in our uni-directional model. Turbulence slows down the overall flow and experimental observations show that turbulence is prominent in regions that we have mentioned.

One method of incorporating turbulence in our model is to consider an "effective" Reynolds number, which quantifies the local turbulence that is present, depending on the distance from the channel bed. We consider four independent experiments that were performed by Ghisalberti and Nepf [1] for flow past a canopy with rigid beams. We use our model with an effective Reynolds number to predict the velocity, and we plot the output of our model, along with the experimental results, in Figure 5.

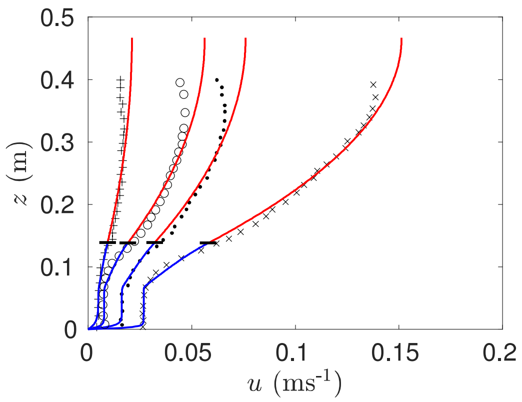

Figure 5: Predicting the velocity profile for flows within a flume with rigid beams. Symbols indicate experimental results that are extracted from Figure 3 and 9 in Ghisalberti and Nepf [1] for runs B (+), C (·), H (×), and J (◦). Solid lines indicate the corresponding predictions using the modified uni-directional model. The short horizontal bars indicate the height of the rigid canopy for each flow.

We see that, apart from the velocity near the free surface, our simple model is able to provide a reasonable fit for each set of experimental data. We are able to capture the crucial features of the flow, including the uniform profile near the channel bed and its transition to the parabolic profile near the top of the canopy.

4. Discussion, Conclusions & Recommendations

We have presented a simplified mathematical model for fluid flow through a vegetated region focusing, in particular, on the case where the flow is steady and the canopy is submerged under water. Our uni-directional model can be numerically solved much more simply than the more complete system used by Mattis et al. [2]. Our model relies on approximating the momentum loss within the canopy due to the reeds using a single homogenised sink term. Crucially, the simplifications then enable us to model the system as a system of simple equations governed by key dimensionless parameters identified in Section 3; these include B (the stiffness of the beams) and $\tilde{\lambda}$ (the canopy density).

Our key findings are:

- for flows with the same Reynolds number, the reeds in the canopy will deflect less as the canopy density increases;

- the flow is primarily uniform within the canopy, in the limit of large Reynolds numbers; this is in good agreement with the experimental observations by Ghisalberti and Nepf [1];
- turbulence significantly influences the velocity profile above the canopy. In order to account for these effects, we incorporated turbulence through an effective Reynolds number in our model, and this allowed us to better fit the numerical profiles to the experimental data in Ghisalberti and Nepf [1].

Expanding the model

Although we have developed a simple model for flow through and above submerged vegetation that can explain some real-world phenomena, our findings have also naturally provoked questions to be addressed in the future. In particular, as noted in the introduction, we are interested in a more thorough understanding of canopy turbulence, so that we may apply this knowledge to flows in other settings, including waves over canopies and two-phase flows through emergent canopies. A method of refining our turbulent flow model on turbulence is to return to the fitting parameters in Figure 4 and attempt to establish relations between them. Another approach we will take is to compare our current model with 3D numerical simulations, in which we will vary the dimensionless parameters independently. Our ultimate goal is to establish a systematic theoretical understanding of the complex interaction between vegetation and fluid flow in natural habitats.

5. Potential Impact

Our work is part of the long-term development of Proteus, an open source computational methods and simulation toolkit, which is under development by the U. S. Army Corps of Engineers, HR Wallingford LTD, and many university partners. Through the mathematical modelling of fluid flow through vegetation, our aim is to improve upon and extend the initial vegetation modelling approach in Proteus, which is described in Mattis et al. [2].

Christopher Kees, Research Hydraulic Engineer at the U. S. Army Corps of Engineers' Coastal and Hydraulics Laboratory, commented

"*I was excited to see some of the recent experimental results were predicted quite well by careful asymptotic analysis based on reasonable and intuitive assumptions. This work could lead to a very practical yet rigorous model of vegetation that could help provide rational guidance on the role of wetlands and marshes in attenuating waves and storm surge.*"

References

1. M Ghisalberti and HM Nepf (2004) *The limited growth of vegetated shear layers.* Water Resour. Res. **40** (7), 1-12.

2. SA Mattis, CN Dawson, CE Kees and MW Farthing (2015) *An immersed structure approach for fluid-vegetation interaction.* Adv. Water Resour. **80**, 1-16.