FSI-02-TN60

Modeling Roughness Effects in Open Channel Flows

D.T. Souders and C.W. Hirt
Flow Science, Inc.

Overview

Flows along rivers, through pipes and irrigation channels encounter resistance that is proportional to the roughness of bounding walls. Roughness can vary considerably from smooth steel, to concrete or sand, pebbles, and even large boulders. In traditional hydraulics the influence of roughness has been cataloged in the form of a roughness coefficient based on data obtained from a wide range of field and laboratory observations.

Roughness coefficients are typically defined in one of two ways: (1) Chézy's resistance coefficient, or (2) Manning's n. To understand these terms we must first state the conditions under which they are defined. For this discussion, and much more information about flow resistance caused by roughness, the reader is referred to Open Channel Hydraulics by Ven Te Chow and published by McGraw-Hill, reissued 1988.

Flow losses are defined in terms of a uniform flow state defined as steady flow with a fixed discharge and flow depth. In practice this usually means the flow in a channel of uniform cross section and having a constant slope such that the gravitational acceleration down the slope is balanced by frictional resistance at the boundaries of the channel.

Chézy derived a simple formula to describe this uniform state,

$$V = C\sqrt{R_h S},$$

where V is the mean flow velocity in ft/s, R_h is the hydraulic radius in ft (i.e., flow cross sectional area divided by wetted perimeter), and S is the channel slope. C is called Chézy's coefficient of flow resistance, or simply Chézy's C. (A more general expression in which C would be dimensionless is obtained by inserting a factor of the acceleration of gravity under the square root.)

The Manning formula, which is meant to do the same thing as Chézy's but from a more empirical point of view, is

$$V = \frac{1.49}{n} R_h^{\,2/3} S^{1/2},$$

in which n is a parameter called Manning's n. The relationship between Chézy's C and Manning's n is easily shown to be

$$C = \frac{1.49}{n} R_h^{\,1/6}.$$

Chow gives an extensive table (see Table 5-6 and photographs of actual channels) for Manning's n that covers a wide range of artificial and natural channels.

If one tries to numerically compute flow in an open channel the question immediately arises of how to specify the flow loss at a rough channel surface? Computations of the flow profile in a channel are only affected by roughness through a viscous shear stress in the grid elements adjacent to the boundaries. The remainder of the flow must adjust itself to the wall shear stresses through viscous forces, which for water are very small unless the flow is turbulent. This means that a turbulence model must be used in the simulations.

In this Technical Note we describe a simple test problem that has been used to study wall roughness effects in open channel flows. There are two issues. First, what are the proper boundary conditions to impose on the turbulence transport variables at rough boundaries? Second, how does one insure a consistency between those boundary conditions and the boundary conditions for mean quantities such as velocity or momentum?

<u>Modeling the Effects of Roughness</u>

When a turbulence transport model (e.g., k-epsilon or RNG) is used in numerical simulations there must be suitable boundary conditions for the turbulence quantities (i.e., turbulent kinetic energy *k* and turbulent dissipation ε) that reflect the roughness of the boundary. Typically, these boundary conditions are set using so called "wall functions." That is, it is assumed that a logarithmic velocity profile exists near a wall, which can be used to compute an effective shear stress at the wall. One does this by fitting the log profile to the computed tangent velocity closest to the wall, u_{tan}, at the distance of that velocity component from the wall, y_0. For a smooth wall the equation is,

$$u_{\tan} = u_s \left[\frac{1}{k} \ln\left(\frac{r u_s y_0}{m} \right) + 5.0 \right] \qquad \textit{smooth wall}$$

In this equation κ is the von Karman constant and the equation is solved for the wall shear stress velocity, u_s. The u_s value is then used to define wall boundary conditions for the turbulent variables *k* and εpsilon (see, for example, <u>Turbulence Modeling for CFD</u>, by David Wilcox and published by DCW Industries, Inc., 1998). For the k-epsilon model the boundary conditions are:

$$k = \frac{u_s^2}{\sqrt{c_m}}, \qquad e = \frac{u_s^3}{k y_0}.$$

For rough walls the same procedure is used, but the logarithmic function has a slightly different form,

$$u_{\tan} = u_s \left[\frac{1}{k} \ln\left(\frac{y_0}{rough} \right) + 8.5 \right] \qquad \textit{rough wall}$$

The incorporation of rough-wall boundary conditions into ***FLOW-3D***® has been accomplished through the implementation of three procedures (in code Versions beginning with 8.1):

1. Smooth and rough wall expressions for the near-wall logarithmic velocity profiles are combined into a single expression by introducing a modified viscosity at a boundary that reflects the enhanced momentum exchange expected because of roughness.
2. The modified logarithmic velocity profile is used to compute a local shear stress in elements adjacent to boundaries and this value is then used to define the values of the turbulent transport variables (*k*,*e*) at the boundary. (Steps 1 and 2 are in Version 8.0.)
3. Wall shear stresses computed from the logarithmic velocity profile are included in the momentum equations for the mean flow velocity.

Step 1, the blending of the smooth and rough wall velocity profiles, is accomplished by assuming that roughness makes an additional contribution to the molecular viscosity μ, forming an effective viscosity defined as

$$m_{eff} = m + r \bullet a \bullet u_s \bullet rough.$$

Here u_s is the shear velocity tangent to the wall surface, ρ is the fluid density, *a* is a constant, and *rough* is the roughness value input for the solid surface. The idea behind this proposition is that roughness elements introduce perturbations in the flow characterized by u_s and *rough* that increases the transfer of momentum between the fluid and the boundary.

The convenience of the μ_{eff} expression can be appreciated by substituting it in place of μ into the smooth-wall logarithmic profile,

$$u_{\tan} = u_s \left[\frac{1}{k} \ln\left(\frac{r u_s y_0}{m + r a u_s rough} \right) + 5.0 \right] \qquad \textit{combined}$$

When roughness is zero and μ_{eff} is simply equal to the molecular viscosity the logarithmic velocity profile is the correct expression for a smooth wall.

However, when the roughness contribution to μ_{eff} is much larger than the molecular value the logarithmic profile reduces to something that looks like the rough wall expression,

$$u_{\tan} = u_s \left[\frac{1}{k} \ln\left(\frac{y_0}{a \bullet rough} \right) + 5.0 \right]$$

If we further define a=0.246, then this becomes the accepted expression for rough walls, with the constant 5.0 replaced by 8.5.

Thus, the use of an effective viscosity that is the sum of the molecular viscosity and a roughness contribution at a rough-wall boundary offers a convenient way to make a continuous transition between smooth and rough wall boundary conditions.

The *combined* velocity profile defined above is used by ***FLOW-3D***® to compute the shear stress at all no-slip solid boundaries in conjunction with turbulence transport models. However, if the Reynolds number based on the shear velocity $\rho u_s y_0/\mu$ is less than 5.0 this indicates that the flow at the boundary is in the laminar sublayer. In this case the shear velocity is set equal to the value corresponding to a laminar shear stress,

$$u_s = \sqrt{\frac{m u_{\tan}}{r y_0}} \qquad when \quad \frac{r u_s y_0}{m} \leq 5.0$$

For laminar flow in general, or with non-transport turbulence models such as the large eddy simulation model, we use μ_{eff} with *a*=0.246 to compute the wall shear stress,

$$t_{wall} = \frac{m_{eff} u_{\tan}}{y_0}$$

Although the law-of-the-wall velocity profile is normally confined to wall regions, in channel flow this profile extends approximately through the entire depth of flow. In this connection, it must be remembered that establishing this profile in the bulk of the flow away from boundaries is the job of the turbulence model. Unfortunately, there is no guarantee that the correct logarithmic profile will be created by a given turbulence model. In fact, we shall see in the next section that turbulence models have definite limitations.

Channel Flow Test Problem

We select a generic example of an open-channel flow as a test problem to test the rough-wall boundary condition treatment described in the previous section. The channel is assumed to be rectangular in cross section, 5 ft wide and containing water to a depth of 2.5 ft, which gives a hydraulic radius of 1.25 ft (i.e., 2.5*5/10). The slope of the channel is modeled by adjusting the components of gravity. In this case the slope was chosen to be 0.017 (1 degree) in our example.

Since the free surface is to be flat and parallel to the bottom of the channel in “uniform flow conditions,” the top boundary has been replaced by a free-slip rigid surface. Symmetry about a vertical mid-plane means that only one half of the channel must be modeled. To allow the flow to develop but remain uniform in the flow (axial) direction, periodic boundary conditions were used in that direction. This choice greatly reduces the computational effort since it removes the axial dependency and changes a three-dimensional problem into an equivalent two-dimensional one.

A sample input file with data in units of lbm, ft, s is appended to this Note. Changes to the slope, dimensions of the channel, and computational resolution can easily be made to this file.

All computations were carried out until the average kinetic energy of the flow was no longer changing. This was typically after about 200 to 300s of problem time.

Validation of Model

Theoretical Results

The principal result for making comparisons between computations and theory (and/or experimental data) is flow rate, or equivalently, the mean axial velocity in the channel. The Chow reference (see Overview: p.204, Eqs.8.19-8.20) gives the following expressions for the mean velocity in smooth and rough channels,

$$V = u_s\left[A_{smooth} + 5.75\log\left(\frac{R_h u_s}{n}\right)\right] \qquad smooth$$

$$V = u_s\left[A_{rough} + 5.75\log\left(\frac{R_h}{k}\right)\right] \qquad rough$$

These expressions are arrived at by performing an approximate integration of the logarithmic velocity profile over the cross section of the channel. An exact integration cannot be done because it depends on the details of the three-dimensional shape of the channel. This leads to the introduction of the constants A_{smooth} and A_{rough}, which must be determined from empirical data.

Constant A_{smooth}=3.25 provides a fit to the pipe flow data of Nikuradse (see Chow reference for details). For rough channels, Chow cites a study by Keulegan using data by Bazin that indicates a range of values for A_{rough} extending from 3.23 to 16.92. Chow suggests A_{rough}=6.25 as a "mean" value. This variation can be partially explained by the fact that the shape and arrangement of roughness elements, not just their size, has an influence on the flow resistance generated by the elements.

Chow's tables of Manning's n require users to make choices for each type of channel and surface character (e.g., concrete, wood, sand, etc.). There are many factors that can affect flow rate. Some of these identified in Chow's book include variations in shape and size of the channel cross section, presence of obstructions or vegetation, and meandering.

Because of these variations, Chow's table of Manning's n lists minimum, normal, and maximum values for each type of channel. Typical ranges for concrete channels, for instance, show variations between the minimum and maximum values of 30% or more, while for rubble or riprap the variations are as much as 75%. Even in clean, straight, natural streams the range is 32%. Such variations mean that some latitude must be allowed when comparing computational results with data.

If we accept Chow's average flow velocity formula for a rough channel, and set it equal to the definition of Manning's n, then the relationship between n and roughness k in feet is given by,

$$\log\frac{R_h}{k} = \frac{0.046}{n} R_h^{1/6} - 1.088$$

In this expression the hydraulic radius R_h and k must both be expressed in units of feet. For use in ***FLOW-3D***® the roughness should be converted to the length units selected for the simulation.

Computational Results for Log Profile

Before doing channel flow simulations it is worthwhile to check how well the program is able to generate the logarithmic velocity profile above a solid surface. We do this with the channel-flow setup by replacing the sides of the channel with free-slip walls. The computed velocity profile depends only on the vertical coordinate z and should reproduce the theoretical log profile,

$$u = 5.75 u_s \log\left(\frac{9 y u_s}{n}\right) \qquad \textit{smooth boundary}$$

Figure 1 shows the computed velocity profile as a function of depth for both the k-epsilon and RNG turbulence models when applied to a smooth boundary. The k-epsilon-generated profile is a reasonable fit to the theoretical logarithmic dependence. This comparison confirms that the k-epsilon turbulence model is pretty much matching one of the test cases used to define its constant parameters.

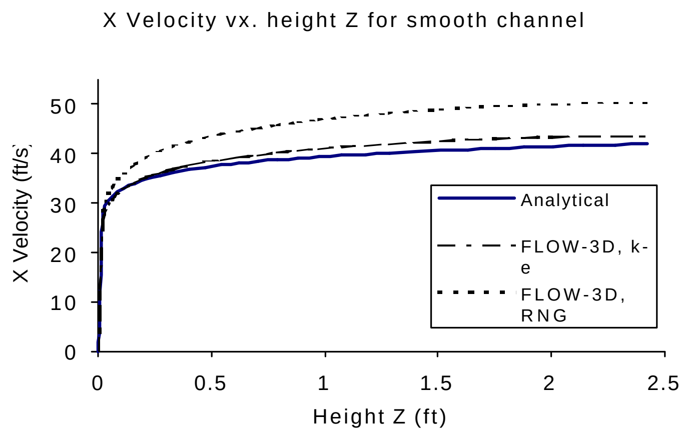

Figure 1. Comparisons of computed and theoretical log profiles for velocity above a smooth wall.

The RNG model does not do as well. It is believed that this discrepancy is partly caused by the use of approximate wall boundary conditions for the turbulence variables. ***FLOW-3D***® uses the same boundary conditions for both the k-epsilon and RNG models, however, some corrections should be made to the value of turbulence viscosity in regions of low turbulence intensity. Near a rigid boundary, for example, the RNG model may need some refinement in the boundary value of turbulent viscosity. The needed correction will be included in Version 8.1 of ***FLOW-3D***®.

A similar comparison between computations and theory can be made for a **rough** boundary. For example using a roughness of k=0.024, the computed velocity is greater than the theoretical prediction over most of the flow region. Furthermore, the computed results are sensitive to the grid used for the numerical solutions. In particular, the velocity profiles are most sensitive to the size of the grid element used at the boundary, Fig.2, and least sensitive to the total number of grid elements, Fig.3. The best results are obtained with the largest grid element (0.2ft) at the boundary and the total number of elements is not particularly large (say 12).

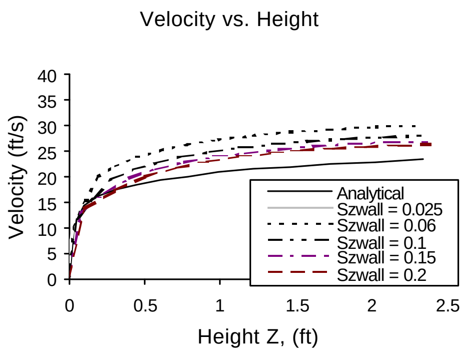

Figure 2. Comparison of computed profiles with different sizes for element at the boundary.

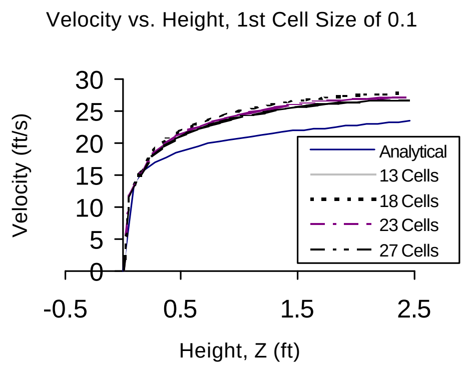

Figure 3. Comparison of computed profiles using different resolutions, but a fixed element size of 0.1ft at the boundary.

These findings are well known but not always acknowledged. Wilcox (*loc.*) describes the problem well and shows that accurate numerical results can only be generated by putting enough grid elements near a boundary and then imposing the analytical solution ("wall function") at the first 4 (or more) grid points. The basic problem is the singular behavior of the turbulence variables near a wall boundary. Unfortunately, Wilcox's remedy is difficult to implement and undermines the usefulness of a simple "wall function" boundary condition.

Wilcox suggests that an alternative is to use a relatively large grid element size adjacent to a boundary ("large" means covering the region of rapid variation in turbulence variables and relying on the wall function to set the proper average values in this region). For the channel flow test case, we use a boundary cell size of 0.2 with 12 cells to cover the depth of flow (2.5ft) to satisfy these conditions.

<u>Computational Results for Channel</u>

A series of computations for the complete channel were then carried out for a range of roughness values. A typical result, Fig.4, shows a weak secondary flow in the cross section of the channel that is generated by non-uniform shear stresses in the region of the channel corners.

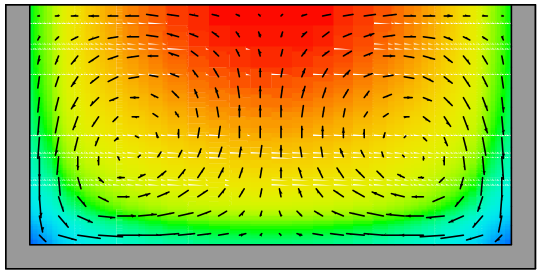

Figure 4. Cross sectional flow generated in a smooth channel.
Secondary velocities are less than 0.3% of the mean axial flow speed.

The computed average flow velocities along the channel are compared to the theoretical (empirical) results given by Chow in Fig.5. There is a constant offset in the computed mean velocity of about 2ft/s for the k-epsilon model and about 4ft/s for the RNG model. With the exception of the offset, both turbulence models predict a behavior with roughness that follows the theoretical predictions very well.

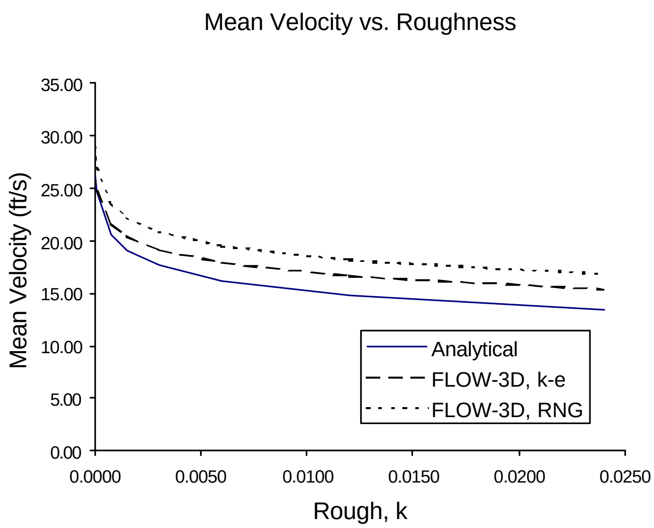

Figure 5. Comparison of computed and theoretical average velocity in channel as a function of roughness.

The offset in mean velocity seen in Fig.5 is associated with the constant A_{rough} in Chow's formula. As previously noted, A_{rough}=6.25, which was used in Fig.5 is a mean value taken from data where A_{rough} varied between 3.23 an 16.92. For the present test case, this variation in A_{rough}

translates to a variation in mean velocity from 2.65 ft/s to 14.00 ft/s. Based on this, we see that the computed results are well within the scatter of the experimental data.

Comments

The computed results are within the scatter of empirical data and, as Chow points out, there are many physical factors in a real channel that affect its flow rate.

Users are encouraged to use the new model, which simply means defining a roughness value for the boundaries of the channel. If the value of roughness is unknown it can be computed from Manning's n using the formula given at the end of the "Validation of Model" section. Be aware, however, that computed results for flow rates can be no more accurate than the data on which the above formulas are based – i.e., accept the values as a decent approximation but not the absolute truth.

APPENDIX: SAMPLE INPUT FILE FOR CHANNEL FLOW STUDY

```
  Roughness Study: Rough=0.0,  k-e
  S=0.017, Rad=1.25, K-Epsilson or RNG
 $xput
    remark='units are lbm,ft,s',
    twfin=300.0,      delt=0.01,
    itb=0,
    gx=0.562,         gz=-32.195,
    ifvis=3,
 $end
 $limits
 $end
 $props
    rhof=1.94,        mu1=0.0000234,
 $end
 $scalar
 $end
 $bcdata
    wl=4,             wr=4,
    wf=1,             wbk=1,
    wb=2,             wt=1,
 $end
 $mesh
    nxcelt=2,       nycelt=13,        nzcelt=13,
      px(1)=0.0,      py(1)=0.0,        pz(1)=-0.25,
      px(2)=2.5,      py(2)=2.5,        pz(2)=0.0,
                      py(3)=2.75,       pz(3)=2.5,
                      sizey(2)=0.20,    sizez(2)=0.20,
 $end
 $obs
    avrck=-2.1,
    nobs=1,         rough(1)=0.0,
    iob(1)=1,       zh(1)=0.0,
    iob(2)=1,       yl(2)=2.5,        zl(2)=0.0,
 $end
 $fl
    flht=100.0,     ui=20.0,
 $end
 $bf
 $end
 $temp
 $end
 $motn
 $end
 $grafic
 $end
 $parts
 $end
Documentation: A free-slip top is used to insure uniform flow conditions.
Periodic boundary conditions in x direction permits a steady flow in y-z to
be established with minimum amount of CPU time.  Symmetry is used in y
direction.
```