Visualization Tool for the Dynamics of Gravity Driven Flows

Introduction

A graphical user interface (GUI) has been developed in Matlab that simulates the draining of a tank under various conditions. The flow dynamics for a variety of different tank and exit piping configurations includes the solution of an initial value problem (IVP) that involves a nonlinear relationship between the elevation head that is driving the flow and the instantaneous flow rate out of the tank. Another thing that complicates the nonlinear relationship in the model is the treatment of the head loss in the exit pipe due to friction effects. Thus, the mathematical model for the general case requires the use of an ODE solver to numerically integrate the IVP, coupled with a nonlinear equation solver to determine the instantaneous flow rate at each point in time.

The Gravity Driven Flows (GDF) application was designed to be particularly useful for gaining insight into the fluid mechanics of the tank-draining problem. In addition, however, it also ties together many concepts from various courses in the Chemical Engineering curriculum (such as Differential Equations, Numerical Methods, and the Unit Operations laboratory courses, as well as the Fluid Mechanics course). This application was developed as part of a multi-year NSF-funded project at UMass-Lowell entitled "Multi-Semester Interwoven Problem for Teaching Basic Core STEM Material Critical to Solving Dynamic Systems Problems." The overall project focus is to improve student understanding of basic STEM (Science, Technology, Engineering, Mathematics) material. This project has been initiated to better integrate the material from a variety of math and engineering science courses across several semesters so that the students can better understand the relationship of basic STEM material to an ongoing problem. The GDF program, which addresses the dynamics of gravity driven flows, is just one of several tools that was developed as part of this overall project.

General Model Development for the GDF Application

Consider a flow system in which an incompressible liquid is added to a storage tank at a known rate, $Q_{in}(t)$, and is discharged from the tank through a single exit pipe at a rate $Q_{out}(t)$ that is related to the height of fluid in the tank, $h(t)$. The discharge is through a piping system of length L_p that is open to the environment. In general, the exit pipe can be placed vertically or horizontally, or have both vertical and horizontal components, with lengths L_v and L_h, respectively. The inlet to the tank can be shut off, with the top "empty" portion of the tank filled with air. This portion may be pressurized or open to the environment, as desired, for a particular case of interest.

A mass balance for the single-phase isothermal liquid within the tank is given as

rate of change of mass = mass flow rate in - mass flow rate out

In mathematical terms, this simple mass balance equation can be written as

$$\frac{d}{dt}M = \rho\frac{d}{dt}V = \rho\frac{dV}{dh}\frac{dh}{dt} = \rho(Q_{in} - Q_{out}) \tag{1}$$

where Q has units of fluid volume per unit time and the fluid volume within the tank, V, is given by $V = \int A(h)dh$, where A(h) is the free surface area at fluid height h. Since the liquid is isothermal and incompressible, the density is constant -- as implied in the manipulations performed in eqn. (1). Thus, with constant ρ and the definition of the liquid volume, eqn. (1) reduces to

DYNamic SYStems   Multi-Semester Interwoven Project for Teaching Basic Core
STEM Material Critical to Solving Dynamic Systems Problems   Rev 033106

Page 1   UMASS LOWELL

$$A(h)\frac{dh}{dt} = Q_{in} - Q_{out} \tag{2}$$

A relationship between $Q_{out}(t)$ and $h(t)$ at any time t can be derived by considering the general energy equation for "quasi" steady-state incompressible flow with no energy sources or sinks (see any standard fluid mechanics text):

$$\frac{P_1}{\gamma} + \alpha_1 \frac{v_1^2}{2g} + z_1 - h_L = \frac{P_2}{\gamma} + \alpha_2 \frac{v_2^2}{2g} + z_2 \tag{3}$$

where we have used standard notation ($\gamma = \rho g$ is the specific weight, α is the kinetic energy correction factor, etc.), with h_L representing the head loss due to friction in the system. In this equation, point 1 represents the free surface of the fluid in the tank and point 2 is at the free jet at the end of the exit pipe that is open to the atmosphere. Each term in this expression has units of energy per unit weight of fluid. Since energy divided by force gives distance (e.g., ft-lbf/lbf = ft), each term is referred to as head -- that is, the sum of the pressure head, velocity head, and elevation head at point 1, minus any head losses in the system due to viscous friction, must equal the total head at point 2. Thus, eqn. (3) is the general mechanical energy equation from fluid mechanics assuming that there is no mechanical device in the exit stream to add or remove energy from the system.

For viscous internal flow problems, the friction loss in a piping system of constant area is given by

$$h_L = f\frac{L_p}{D_h}\frac{v^2}{2g} + \left(\sum_i K_i\right)\frac{v^2}{2g} \tag{4}$$

where D_h is the hydraulic diameter (pipe inside diameter, D_p, for a circular pipe), L_p is the length of the pipe, f is the dimensionless Darcy friction factor, and K_i is the loss coefficient associated with the ith "minor loss" component within the piping system (accounts for valves, elbows, etc.).

For laminar flow situations, the friction factor, f, is given by

$$f = \frac{64}{Re} \qquad \text{(laminar flow)} \tag{5}$$

and, for turbulent flows, it can be approximated with reasonable accuracy with the explicit Swamee-Jain correlation,

$$f = \frac{0.25}{\left[\log\left(\frac{\varepsilon/D_h}{3.7} + \frac{5.74}{Re^{0.9}}\right)\right]^2} \qquad \text{(turbulent flow)} \tag{6}$$

where ε/D_h is the relative roughness, the dimensionless Reynolds number, Re, is given by

$$Re = \frac{\rho D_h v}{\mu} \tag{7}$$

and all the other parameters have their common meaning (i.e. ρ is density, μ is the viscosity, g is the acceleration due to gravity, etc.).

Finally, to close the loop on all the various equation dependencies, we note that the average flow velocity, v, in the piping system can be written in terms of the volumetric flow rate, Q, or

DYNamic SYStems   Multi-Semester Interwoven Project for Teaching Basic Core
STEM Material Critical to Solving Dynamic Systems Problems

Rev 033106

Page 2

$$v = \frac{\text{flow rate}}{\text{flow area}} = \frac{Q}{A_f} \qquad (8)$$

Also, in general, the change in elevation head, $z_1 - z_2$, that appears in eqn. (3) is related to the sum of h(t) and to the vertical distance covered by the exit pipe, L_v, which is described by the particular geometry of interest for any given problem ($L_p = L_v + L_h$). Note also that the usual approximation that v_1 is much less than v_2 is not made here, since we want to account for nearly any user-defined tank geometry and size. Instead, we can relate the free surface fluid velocity, v_1, and the velocity in the piping system, $v_2 = v$, via the continuity equation, $Q_1 = Q_2 = Q$ or $A v_1 = A_f v_2 = A_f v$, which gives

$$v_1 = \frac{A_f}{A} v_2 = \frac{A_f}{A} v \qquad (9)$$

and this can be used to eliminate the dependence on v_1 in eqn. (3). Also, since the free surface fluid velocity distribution is nearly uniform, the kinetic energy correction factor at this point is set to unity -- that is, $\alpha_1 = 1.0$ in eqn. (3).

Finally, we also note that, in general, the relationships given in eqns. (3) - (9) can not be manipulated algebraically to give an explicit expression for Q(t) in terms of h(t) -- this can only be done for a few very simple situations. However, in all cases, these relationships can be written in implicit form as:

$$F(Q, h) = 0 \qquad (10)$$

which can be solved numerically using a number of standard root finding algorithms. In particular, given a specific value of h, one could use Matlab's *fzero* routine, for example, to "find the value of Q such that F(Q) = 0," thus producing the desired relationship between h and Q.

Once a routine or function is available to compute the flow rate, Q, for any given fluid height, h, one can simply call this file within another function file to evaluate dh/dt via eqn. (2). This effectively determines dh/dt knowing the current state of the system, h(t), at time t. This file becomes the ODE function file for use within a standard ODE solver (such as Matlab's *ode23* routine, for example). Thus, with a little programming logic and the help of some existing capability in Matlab (the *ode23* and *fzero* functions), one should be able to write an application to solve the above equations and actually simulate the dynamics of a particular tank and pipe system.

General Description of the Cases Studied

To see how all the above theory really works, we can propose a number of specific scenarios and actually study these systems in detail via numerical simulation to get a better understanding of the dynamics of gravity-driven flow in a single tank system. In particular, a Matlab graphical user interface (GUI) was developed that allows one to interactively vary a number of system parameters to observe how they affect the flow dynamics in a specific tank and exit piping configuration. The goal here was to maintain as much flexibility as possible and to provide a lot of detail for the system under study -- yet the GUI also had to have an intuitive look and feel for the user and be relatively easy to use for a variety of students with a wide range of backgrounds. Although flexibility and ease-of-use are often conflicting goals of any GUI, the Gravity Driven Flows (GDF) application interface, as shown in Fig. 1, provides a nearly optimum compromise between these conflicting requirements.
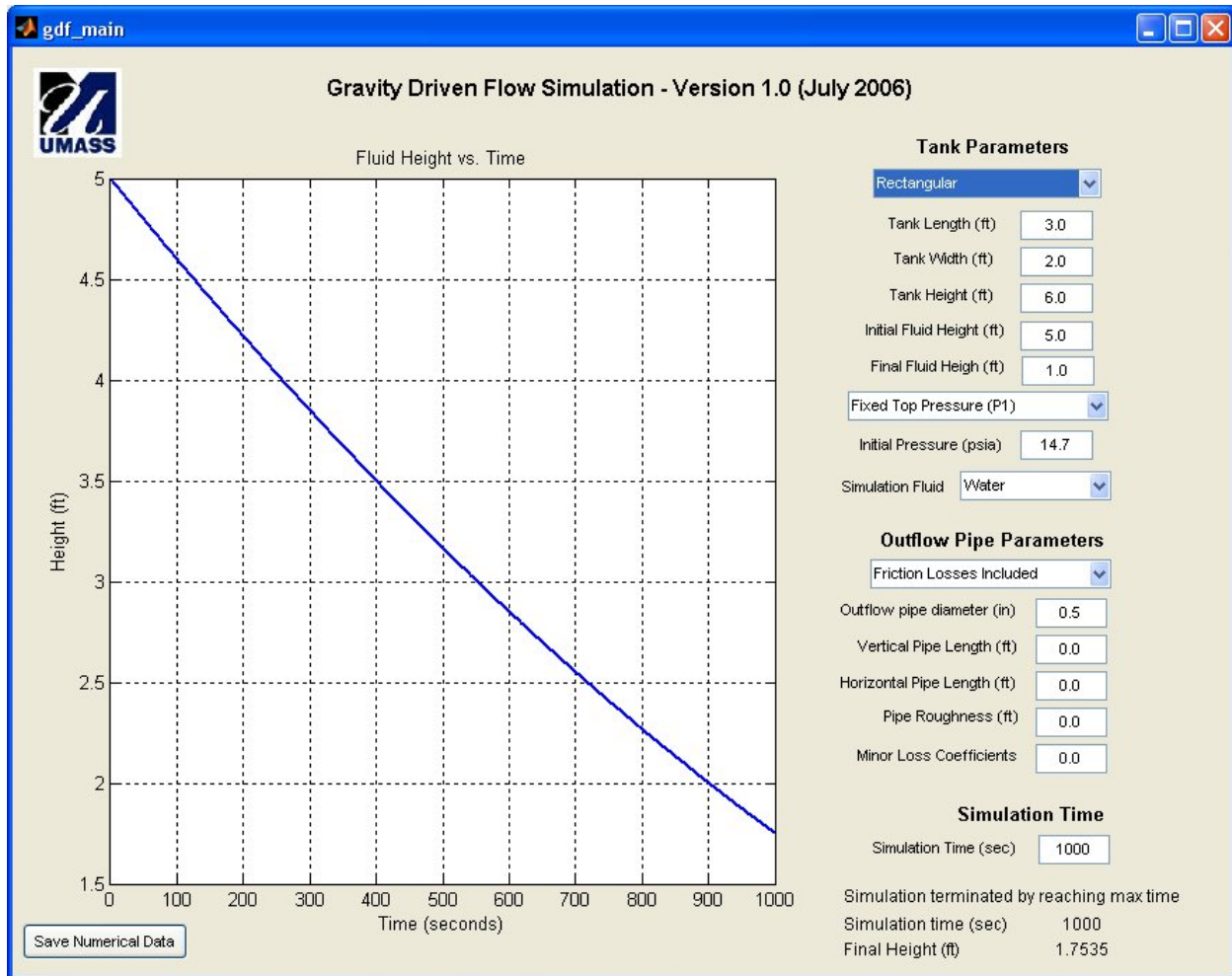
DYNamic SYStems   Multi-Semester Interwoven Project for Teaching Basic Core
STEM Material Critical to Solving Dynamic Systems Problems   Rev 033106

Page 3

Fig. 1 Typical screen showing the GDF interface.

There are only a few parameter limitations for the GDF application, as follows:

1. There is no inflow to the tank - thus, $Q_{in} = 0$ in all cases since the focus in on a draining tank situation.

2. The exit piping configuration has the same diameter throughout the system -- thus, v represents the average flow velocity and it is constant throughout the piping system.

3. The exit pipe opens to the atmosphere -- thus, $P_2 = P_{atm} = 14.7$ psia = 0 psig.

4. The working fluid is an incompressible liquid at room temperature - thus, the fluid properties used are appropriate for $T \approx 70\ ^oF$.

5. To avoid conceptual problems as the fluid height, h, approaches zero, the tank drain time is defined as the time required for the fluid height to decrease from the initial height, h_i, to the final height, h_f, where these are user specified, with the added constraint that h_f must be greater than two times the exit pipe diameter, D_p, -- thus, $h_f \geq 2D_p$.

6. Finally, there are also some obvious geometry constraints that are checked for consistency, such as: the initial fluid height must be less than or equal to the maximum tank height, the desired final fluid

height must be less than or equal to the initial height, and all the tank geometry parameters must be greater than zero.

These restrictions are relatively minor and, in turn, they allow considerable user flexibility in specifying the particular tank-draining scenario of interest. Within the GUI, the user is allowed to pick the liquid of interest from a pre-defined list, select the shape of the tank from a list of standard shapes, specify the dimensions of the selected tank, decide if the system is open or closed, and select several features for specifying the flow characteristics of the exit piping system. In short, the GDF GUI should allow one to simulate a variety of practical configurations and situations that, with some study, should allow the user to gain considerable insight into the physics of this fluid flow problem.

The Model Details

To fully understand each simulation, one needs detailed documentation of the equations and procedures implemented within the Matlab application, and this section is devoted to providing this information. Of course, the continuity of mass and the basic viscous flow equations from Fluid Mechanics have already been described in some detail -- and eqns. (1) – (10) are the basic equations solved here. In addition to the fundamental theory, however, each option selected within the GUI specifies some piece of the puzzle by explicitly defining one or more of the parameters and variables given in the defining equations. Each of these options is defined in the following subsections.

Choice of Working Fluid

The pressure head term, P/γ, within the energy equation and the Reynolds number given by eqn. (7) include material properties for the working fluid of interest -- the specific weight, $\gamma = \rho g$, the density, ρ, and the dynamic viscosity, μ. These parameters can vary significantly among different fluids, and these differences can certainly affect the flow dynamics in a significant way. To illustrate this effect, one can select one of the following fluids within the GDF application (properties given at about 70 $^{\circ}$F):

Fluid	Specific Weight (lbf/ft^3)	Density (lbm/ft^3)	Dynamic Viscosity (lbm/ft-s)
water	62.3	62.3	6.56×10^{-4}
gasoline	42.5	42.5	1.96×10^{-4}
engine oil	55.4	55.4	5.8×10^{-1}

Note that, although the specific weight and density have the same numerical value in the English system, they are definitely different quantities, since they have different units. Also note that for numerical computation within the Matlab application, $g = 32.2$ ft/s^2 and the conversion factor, g_c, is given by $g_c = 32.2$ lbm-ft/lbf-s^2 (that is, the weight, $W = mg/g_c$, of 1 lbm at the earth's surface is 1 lbf). In general, one can never be too careful with units!

Open or Closed Tank

P_1 in eqn. (3) is the pressure at the free surface of the liquid in the tank. If the top of the tank is open to the atmosphere, then $P_1 = P_{atm} = 14.7$ psia. If the tank is closed and pressurized to some constant pressure, then P_1 is a user-specified constant.

Another option is to assume a simple closed tank with air as the gas above the fluid. The pressure in this space then obeys the perfect gas law relationship, $P_1 V_a = mRT$, where V_a is the air volume and P_1 is the absolute pressure. Since the air volume expands isothermally as the fluid volume decreases upon opening the exit drain pipe, $P_1 V_a$ is constant. Thus, for a fixed tank volume, V_t, and an initial fluid volume, $V_i = V(h_i)$, the desired pressure versus fluid height relationship is given by:

DYNamic SYStems   Multi-Semester Interwoven Project for Teaching Basic Core
STEM Material Critical to Solving Dynamic Systems Problems   Rev 033106

Page 5

UMASS LOWELL

$$P_1(h)V_a(h) = P_{1i}V_{ai} \qquad \text{or} \qquad P_1(h)\big[V_t - V(h)\big] = P_{1i}\big[V_t - V(h_i)\big]$$

$$\text{or} \qquad P_1(h) = \frac{V_t - V(h_i)}{V_t - V(h)}P_{1i} \qquad\qquad\qquad (11)$$

where V(h) is the fluid volume versus fluid height, which is determined from the tank geometry, and P_{1i} is the initial pressure at the top of the tank as specified via user input.
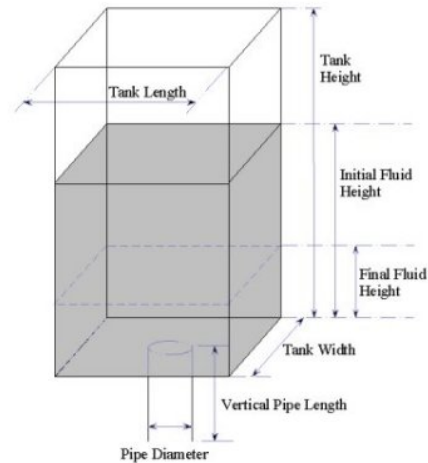
Tank Geometry Selection

A variety of standard tank geometries are available within the GDF application for detailed study. The geometry defines the free surface area, A(h), that appears in the fluid mass balance equation and the fluid volume, V(h), that appears in eqn. (11) for the pressure above the fluid in one of the closed tank options. Five tank geometry options are currently available, and the surface area and volume relationships, as well as a rough sketch of the basic geometry, are summarized below. For the rectangular geometry, the tank size is defined by a width, W, length, L, and height, H. For the vertical and horizontal cylinders, only two parameters are needed: a length, H or L, and a diameter, D, as defined via the sketches. For the spherical tank, only a single diameter is needed and, finally, for the truncated cone, two diameters and a height are needed to fully define the geometry. For this last case, D_L defines the lower area and D_U defines the upper area, where it is possible to have $D_L > D_U$. The flow dynamics are a strong function of the tank geometry and size, and the selections given here allow a lot of flexibility to address and study this behavior.

1. Rectangular Tank ($h_{max} = H$)
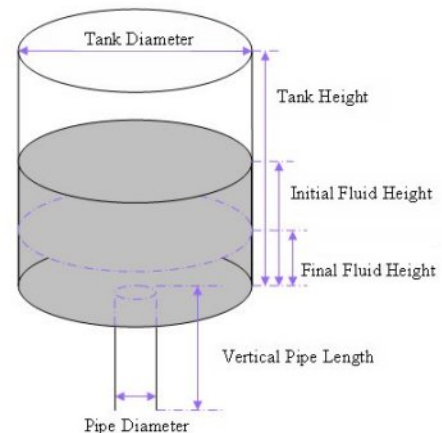
$$A(h) = WL$$

$$V(h) = WLh$$

$$V_t = WLH$$

2. Vertical Cylindrical Tank   ($h_{max} = H$)

$$A(h) = \frac{\pi}{4}D^2$$

$$V(h) = \frac{\pi}{4}D^2h$$

$$V_t = \frac{\pi}{4}D^2H$$

DYNamic SYStems   Multi-Semester Interwoven Project for Teaching Basic Core
STEM Material Critical to Solving Dynamic Systems Problems

Rev 053106

Page 6

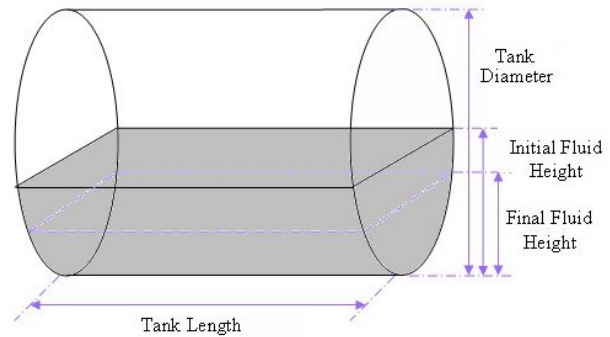UMASS LOWELL

3. Horizontal Cylindrical Tank   ($h_{max} = D$)

$$T(h) = 2\sqrt{h(D-h)}$$

$$A(h) = TL$$

$$\theta = 2\arcsin\left(\frac{2}{D}\sqrt{h(D-h)}\right)$$

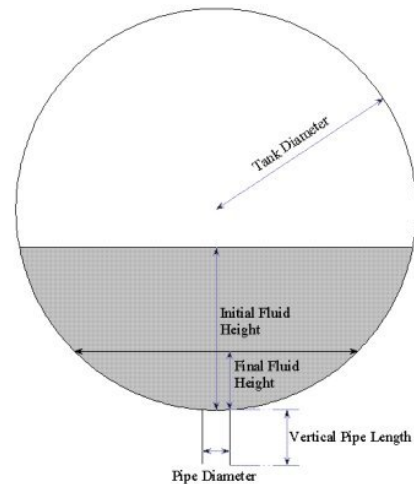$$V(h) = \frac{LD^2}{8}(2\pi - \theta + \sin\theta)$$

$$V_t = \frac{\pi}{4}D^2L$$

4. Spherical Tank   ($h_{max} = D$)

$$A(h) = \frac{\pi}{4}\left[D^2 - (2h-D)^2\right]$$

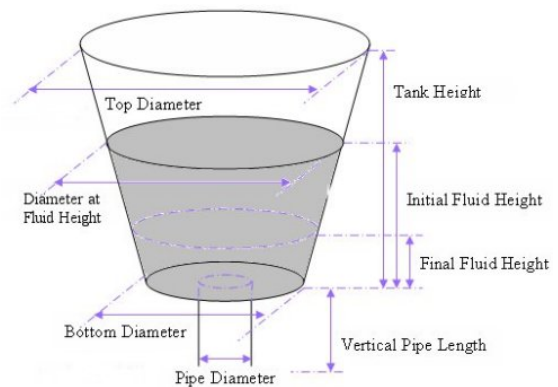$$V(h) = \frac{\pi}{24}\left[D^2(6h-D) - (2h-D)^3\right]$$

$$V_t = \frac{\pi}{6}D^3$$

5. Truncated Cone-Shaped Tank   ($h_{max} = H$)

$$\text{let}\quad d(h) = D_L + \frac{(D_U - D_L)}{H}h$$

$$\text{then}\quad A(h) = \frac{\pi}{4}d^2$$

$$V(h) = \frac{\pi L}{12(D_U - D_L)}\left(d^3 - D_L^3\right)$$

$$V_t = \frac{\pi L}{12(D_U - D_L)}\left(D_U^3 - D_L^3\right)$$

<u>Exit Pipe Configuration</u>

As described above, the friction loss in the piping system connected to the tank certainly affects the flow rate and the drain time for a particular system. The diameter of the exit pipe, D_p, is probably the most important parameter, but the pipe vertical and horizontal lengths, L_v and L_h, respectively, the pipe roughness, ε, and the sum of the minor loss terms, $K = \sum K_i$, all play important roles in defining the dynamics of a particular system.

Within the Matlab GUI, the user can set each of the above parameters to specify a particular exit-piping configuration. In the default ***Friction Losses Included*** option, the user must specify the following parameters:

$$L_v = \text{vertical length of the exit pipe (then } z_1 - z_2 = h + L_v)$$

$$L_h = \text{horizontal length of exit pipe (then } L_p = L_v + L_h)$$

$$\varepsilon = \text{pipe roughness (for use in the friction factor correlation)}$$

K = sum of all individual minor loss coefficients in the piping system (including the initial contraction from the tank to the pipe, any elbows, valves, etc., that may be present).

In this option, α_2 is 1.0 for turbulent flow and $\alpha_2 = 2.0$ for laminar flow, which is determined based on the value of Re within the piping system. To keep things relatively simple and deterministic within the transition region, laminar flow is assumed if Re < 4000 and the turbulent flow values are used if Re $\geq$ 4000. With all these parameters, one can solve the nonlinear equations given in eqns. (3) - (11) using a numerical root-finding algorithm to find the instantaneous flow rate, Q_{out}, for the current value of fluid height, h, in the tank -- and, of course, knowing Q_{out} versus h allows one to solve the mass balance equations to give an estimate of the dynamic behavior of the system.

For the ***No Friction Losses*** option, only the pipe diameter, D_p, and vertical length, L_v, is set, with L_h, ε, and K set to zero internal to the code. With these parameters and the same treatment as above for the kinetic energy correction factor at the pipe exit, the energy equation with no head losses can be solved explicitly for Q for each value of fluid height (although, for ease in implementation within the GDF code, the same root-finding procedure is used regardless of the friction/no friction option).

<u>Simulation Results</u>

The results from each simulation, which focus on h(t) vs. t, for the particular system under study are plotted directly within the GUI window. In addition to the plotted results, summary numerical data for each case can also be saved in an ascii text file for use, as needed, in other applications. The data included here summarize all the key input parameters and options selected for the case, as well as vectors of h(t) and t for several specific time points. These data are sufficient to completely define all the important parameters for a given system.

Summary

A graphical user interface (GUI) has been developed in Matlab that simulates the draining of a tank under various conditions. The interface gives the user control over specifying the tank geometry and size parameters, the simulation fluid, the pressure and friction treatment options, as well as the specification of the exit pipe configuration. The user also specifies the initial and final fluid heights and the simulation time. The results from the simulation are summarized in a plot of fluid height vs. time. Overall, this GUI allows its users to easily visualize and to gain a better understanding of the dynamics of gravity driven flows.

The GDF application was designed to be particularly useful for gaining insight into the fluid mechanics of the tank-draining problem. In addition, however, it also ties together many

concepts from various courses in the Chemical Engineering curriculum (such as Differential Equations, Numerical Methods, and the Unit Operations laboratory courses, as well as the Fluid Mechanics course). The overall project focus is to improve student understanding of basic STEM (Science, Technology, Engineering, Mathematics) material. The GDF program, which addresses the dynamics of gravity driven flows, is just one of several tools that was developed as part of this overall project.