International Journal of the Physical Sciences Vol. 7(2), pp. 327 - 337, 9 January, 2012
Available online at http://www.academicjournals.org/IJPS
DOI: 10.5897/IJPS11.873
ISSN 1992 - 1950 ©2012 Academic Journals

*Full Length Research Paper*

# Modeling flow regime transition in intermittent water supply networks using the interface tracking method

**Stephen Nyende-Byakika[1]*, Gaddi Ngirane-Katashaya[2] and Julius M. Ndambuki[1]**

[1]Department of Civil Engineering, Tshwane University of Technology, Pretoria, South Africa.
[2]Department of Civil Engineering, Makerere University, Kampala, Uganda.

Accepted 5 December, 2011

**For several ailing water distribution networks in the world, during conditions of excessive withdrawals or insufficient water production, pressures fall to very low or even negligible values and consequently, no water can be supplied. Usually, most pipes have water which either fills or nearly fills their cross sectional areas but the pressure to push it out is absent. With conditions changing from pressurized to no pressure (free surface flow), existing water supply models are unable to simulate either free surface flow or the transition between free surface to pressurized flow. In this study, transient "low pressure-open-channel flow" (LPOCF) conditions were analyzed. The interest of this research lies in the coexistence of free surface and pressurized flow regimes with the aim of understanding the pressurization process of pipes. This was represented by a flow regime transition from free surface to pressurized flow through a moving interface along the pipeline. Results revealed the merits of applying full dynamic wave equations in the solution of transient LPOCF conditions in water distribution networks.**

**Key words:** Free surface flow, full dynamic equations, low pressures, open channel flow, pressurization, water supply, flow regime transition, mixed flow.

## INTRODUCTION

Growing demand for water as a result of increasing urban populations, industrialization and rising water consuming lifestyles puts stress on existing water supply systems. In order to cater for additional demand, distribution networks are expanded often beyond their design capacities, which creates bottlenecks such as development of transient flow conditions ranging from excessive pressures and fluctuating pressures to open-channel flow situations (Nyende-Byakika et al., 2010). This culminates into low pressures with low flows and sometimes no flow at all, thereby compromising service levels and giving planners and engineers a complicated task of supplying the additional resource in sufficient and reliable quantities in the most feasible way possible. Such problems would best be solved through infrastructural upgrades; however, this is an expensive option not easily affordable in many developing countries that are often faced with this problem.

In order to meet regulatory requirements and customer expectations, water utilities are feeling a growing need to explain better the movement and transformations undergone by water introduced into their distribution systems (Rossman, 2000). If understanding of network behaviour under adverse conditions could be obtained and the impact of these conditions established, networks would be managed better, and more satisfactory customer service would be offered (Nyende-Byakika, 2011). However, modeling intermittent water supply systems of pipeline networks is a challenging task because these systems are not fully pressurized but networks with high

*Corresponding author. E-mail: stenbyak@gmail.com.

water demands, very low pressures and sometimes restricted by water supply hours per day. Many systems exhibit open channel flow behaviour due to excessive low transient pressure conditions when some sections operate as gravity flow systems under low reservoir conditions and get pressurized under high reservoir level conditions as force mains with low pressures. The alternate emptying and refilling followed by pressurization and depressurization of water pipelines make it problematic to apply standard hydraulic models because of low transient pressures and pipes flowing partially full (Ingeduld et al., 2006). This type of situation is difficult to analyze using conventional approaches and may require special treatment different from that of fully pressurized systems, with more sophisticated/complex algorithms and robust scenario management to model. Ingeduld et al. (2006) notes that hydraulic models of intermittent water supply need to simulate the "charging" process in pipes and this requires integration of continuity and motion equations to indicate the positions of the water front in the network at any time.

Due to the fact that most water supply models operate on the assumption that pressure is sufficient to deliver adequate flows, in situations of low transient pressures, the models do not give reliable results. Thus, in order to address this issue, a tool that treats transient pressures and flows which lie along the continuum between open and closed systems in both time and space has got to be developed. This research was aimed at augmenting existing knowledge on supply of water during transient low pressure open-channel flow (LPOCF) conditions. Knowledge obtained would aid the provision of water supply services even under extreme situations of low flows and low pressures. This will not only improve the understanding of piped water supply systems but also ensure sustainable supply of the basic need for survival.

In this paper, the authors studied co-existence of pressurized and free surface flow regimes in a network; a situation sometimes referred to as mixed flow. Simulation of flow regime changes between pressurized and free surface flow was done; a condition which current water distribution models do not tackle. This aids in understanding the development and propagation of pressure surges in pipelines so that during the transient state while supply is low, pressures can be determined simultaneously with discharges that can be availed to consumers.

## Transient low pressure – open channel flow conditions

It is worth noting that in water supply situations where intermittent flow is manifested, transient LPOCF conditions can best be described as unsteady flow since flow depth and velocity vary with time and as gradually varied flow since the rate of change of flow depth and velocity along the channel is very low. Gradually varied flow is a non-uniform flow whose spatial rate of flow is sufficiently low to imply translatory wave motion of long wavelength and low amplitude (Chadwick et al., 2004) such that the assumption of parallel streamlines and hydrostatic pressure distributions is reasonable. In this research therefore, LPOCF conditions were modeled as unsteady gradually varied flow.

In unsteady non-uniform flow, the discharge $Q$, varies as a function of time and length along the pipe and all the hydraulic parameters of a cross-section change as a function of time and length that is, water depth, cross-sectional area and water surface width. Thus, unsteady flow equations are key to the understanding of the unsteady flows in pipelines. The (full dynamic wave) equations that are used to solve unsteady gradually varied flow are the Saint Venant / shallow water equations which were derived in 1871 by A.J.C Barre de Saint Venant based upon the following assumptions (Chadwick et al., 2004):

(1) Flow is one-dimensional, that is, velocity is uniform over a cross section and the water level across the section is horizontal.
(2) The streamline curvature is small and vertical accelerations are negligible, hence the pressure is hydrostatic. Gradually varied unsteady flow implies translatory wave motion of long wave length and low amplitude in which case the assumption of parallel streamlines and hydrostatic pressure distributions is reasonable.
(3) Effects of boundary friction and turbulence can be accounted for through resistance laws analogous to those used for steady state flow.
(4) The average channel bed slope is small so that the cosine of the angle it makes with the horizontal may be replaced by unity.

During the transition from free-surface flow to pressurized flow, a moving water interface advances into the free-surface region. There is need to track the interface in order to explain the development of pressures in a pipeline. This enables us to make a contribution in understanding the co-existence of pressurized and free-surface flows in a water supply network. The study of the flow regime transition also greatly enables understanding of the pressurization process in pipelines. However, a major problem with mixed flow analysis is the difficulty involved in treating the moving interface similar to surges (Song et al., 1983) and requires a considerable amount of computational effort to detect its generation and trace its movement.

## DEVELOPMENT OF MATHEMATICAL FORMULATIONS AND MODELS USED

Analysis of unsteady flow in this study was carried out using conservation of mass and conservation of momentum principles which yielded two governing algebraic equations because the flow and depth of the water surface were both unknown. Each computational element in the governing equations was written in terms of elevations and flows at the ends of the element. A computational element with respect to time was also considered and due to this, the algebraic governing equations involved not only the unknown flow and depth at two points along the channel but also at two points in time (Chadwick et al., 2004; Franz and Melching, 1997).

### Conservation of mass

The continuity equation for one-dimensional unsteady open channel flow can be expressed as (Chadwick et al., 2004; Franz and Melching, 1997):

$$\frac{\partial A}{\partial t}+\frac{\partial Q}{\partial x}=0 \qquad (1)$$

where *A* is cross sectional area of flow, *Q* is flow rate; *t* is the time interval being considered and $x$ is the length of the reach being considered. All quantities in the equation are algebraic expressions and can be positive or negative therefore, a negative outflow is an inflow. The equation is a statement of the conservation of mass principle on a per-unit-length basis and can also be stated as:

$$\frac{\partial y}{\partial t}+V\frac{\partial y}{\partial x}+y\frac{\partial V}{\partial x}=0 \qquad (2)$$

where *V* and *y* are the velocity and depth of flow, respectively.

### Conservation of momentum

The momentum conservation equation (Equation of motion) can be written in the form (Chadwick et al., 2004; Franz and Melching, 1997):

$$\frac{\partial Q}{\partial t}+\frac{\alpha\partial\left(Q^2/A\right)}{\partial x}+gA\frac{\partial h}{\partial x}-gA\left(S_0-S_f\right)=0 \qquad (3)$$

where *Q* is discharge, *A* is the flow cross sectional area, *h* is depth of flow, *g* is gravitational acceleration, $S_o$ and $S_f$ are channel slope and friction slope, respectively and $\alpha$ is an energy coefficient normally equated to unity for SI units.

Equations 2 and 3 are called Saint Venant equations. These governing equations for gradually varied unsteady flow in open channel can also be expressed as:

$$\frac{\partial y}{\partial t}+\left(\frac{A}{T}\right)\frac{\partial V}{\partial x}+V\frac{\partial y}{\partial x}=0 \qquad (4)$$

$$\frac{\partial V}{\partial t}+V\frac{\partial V}{\partial x}+g\frac{\partial y}{\partial x}=g\left(S_0-S_f\right) \qquad (5)$$

where *T* is the top width of flow. Equations 4 and 5 represent the continuity and dynamic equation, in non-conservation form, respectively. The friction slope is given by $S_f=\frac{fV|V|}{2D}$ for pressurized flows and $S_f=\frac{n^2V|V|}{R_h^{4/3}}$ for free-surface flows, with $f$ the Darcy-Weisbach friction factor, $n$ the Manning's Coefficient, $D$ the pipe diameter and $R_h$ the hydraulic radius.

Different types of formulations can be given for the Saint Venant equations depending on the problem. Equations of continuity and motion for a one-dimensional unsteady flow in an open channel can be restated as (Song et al., 1983; Trajkovic et al., 1999; Leon, 2007; Gomez and Achiaga, 2008):

$$\frac{\partial y}{\partial t}+v\frac{\partial y}{\partial x}+\frac{c^2}{g}\frac{\partial v}{\partial x}=0 \qquad (6)$$

and

$$g\frac{\partial y}{\partial x}+\frac{\partial v}{\partial t}+v\frac{\partial v}{\partial x}+g\left(S_f-S_0\right)=0 \qquad (7)$$

in which *c* is the gravity wave celerity given by:

$$c=\sqrt{\frac{gA}{T}} \qquad (8)$$

Corresponding equations for pressurized or closed conduit flow can be written as (Song et al., 1983):

$$\frac{\partial y}{\partial t}+v\frac{\partial y}{\partial x}+\frac{a^2}{g}\frac{\partial v}{\partial x}=0 \qquad (9)$$

$$g\frac{\partial y}{\partial x}+\frac{\partial v}{\partial t}+v\frac{\partial v}{\partial x}+g\left(S_f-S_0\right)=0 \qquad (10)$$

in which *a*, the speed of the water hammer wave and *y* should be regarded as the piezometric head measured from the pipe invert rather than the flow depth.

### Solution of unsteady flow equations

An important family of equations that is often encountered in hydraulics is based on the following equation (Chadwick et al., 2004):

$$a\frac{\partial^2 f}{\partial x^2}+b\frac{\partial^2 f}{\partial y\partial x}+c\frac{\partial^2 f}{\partial y^2}=0 \qquad (11)$$

*f* , some variable/function such as velocity. If $b^2-4ac>0$ then a typical form is:

$$\frac{\partial^2 f}{\partial x^2} - c^2 \frac{\partial^2 f}{\partial y^2} = 0 \quad (12)$$

This is the hyperbolic equation which can be applied to unsteady flows. The Saint Venant/shallow wave equations are classified as partial differential equations of the hyperbolic type. All flow variables are functions of both time and distance along the channel. In other words, at a given location, the flow depth, discharge and the other flow variables vary with time. Likewise, at a fixed time, the flow variables change along the channel. For a given channel of known properties (cross sectional geometry, roughness factor and longitudinal slope), the unknowns are the discharge *Q* and flow depth *y*. The other flow variables such as the area *A* and the friction slope $S_f$ can be expressed in terms of *Q* and *y*. The independent variables are time *t* and distance along the channel.

**Method of characteristics**

The differential equations of Saint Venant cannot be solved analytically unless certain simplifications are carried out such as the neglect of certain terms and simplification of boundary conditions as is the case in the kinematic approximation and diffusion analogy and this can lead to serious errors (Tucciarelli, 2003). With the advent of the digital computer, numerical solutions can be obtained and thus, no simplifying assumptions to the basic equations need to be made. Thus, unsteady gravity flows have been traditionally modeled by numerically solving the one-dimensional equations of continuity and momentum. A number of different numerical methods are available to solve hyperbolic differential equations. The best known is the method of characteristics (MOC) (Chadwick et al., 2004; Chou, 2009). The method is widely used for transient flow in a closed conduit because it is simple and also provides good insight into behaviour of hyperbolic equations. In the method, the original set of partial differential equations (Equations 6 and 7) are transformed into two sets of simultaneous ordinary differential equations.

$$\frac{dx}{dt} = v \pm c \text{ and } \frac{dy}{dt} \pm \frac{c}{g}\frac{dv}{dt} \pm c\left(S_f - S_0\right) = 0 \quad (13)$$

Equation 13 is known as characteristic equations and is valid along two different characteristic lines. The first one is called a positive characteristic line and the second one is a negative characteristic line (Chadwick et al., 2004; Gomez and Achiaga, 2008). By discrediting the transformed equations we can obtain the velocity $v_p$ and depth of flow $y_p$ at point *P* as:

$$y_p = \frac{1}{C_R + C_S}\left[y_R C_S + y_S C_R\left(\frac{v_R - v_S}{g} - \Delta t.\left(S_{fR} - S_{fS}\right)\right)\right] \quad (14)$$

$$v_p = v_R - \frac{g}{C_R}\left(y_p - y_R\right) + g.\Delta t.\left(S_o - S_{fR}\right) \quad (15)$$

where $\left(\frac{g}{c}\right)_R = C_R$, $\left(\frac{g}{c}\right)_S = C_S$ and *R* and *S* are subscripts for locations defining positive and negative characteristic lines drawn from *P*, respectively. They are known quantities (velocity or depth) at the beginning of time step $\Delta t$ and can be obtained by linear interpolation. If we are at a boundary then one of the characteristic equations is outside of the problem boundary, so we need a boundary condition that will be the pressure head at this boundary or a relationship between pressure and head. When converted into finite difference equations, the characteristic equations lead to a set of simultaneous algebraic equations for the unknowns.

**Interface tracking method of analyzing mixed flow**

This class of flow regime transition models is exemplified by the works of Wiggert (1983), Hamam and McCorquodale (1982) and Li and McCorquodale (1999) by introducing a moving interface between the free surface and pressurised flow regimes. The method treats the two flow regimes separately but joined together by an interface which is regarded as a shock wave. Song et al. (1983) used the characteristic method for both open channel flow and closed conduit flow regimes. Identical equations and solution techniques were used throughout the system except for a special treatment at the interface. These models solve the ordinary differential equations (ODE) based on a momentum balance in a rigid column represented by the pressurised portion of the flow. In each time step, the ODE is solved and the velocity of the rigid column, speed, location and intensity of the shock wave is updated. The location of the pressurisation front is obtained using the continuity equation across the moving interface. The flow conditions near the interface are thus, calculated using a mass and momentum balance in a control volume. The free surface portion of the flow is solved by the method of characteristics. This model, also called a shock-fitting model is appropriate when the energy contained in the flow is sufficient to pressurize the flow through a hydraulic jump. The water depths and velocities near the interface are obtained using two shock-boundary conditions plus three characteristic equations (Politano et al., 2005).

If velocity changes are more gradual, acceleration of flow between two adjacent sections can be neglected and the flow near the interface can be simulated using momentum and mass balance in a moving control volume. This method facilitates accurate tracking of the interface conserving mass and is the approach that was used in this study.

**Modeling flow discontinuities in the transition region**

While the MOC is a valuable approach in the sense that it provides a deep understanding of the nature of shallow water fronts, the approach is limited by its inability to handle flow discontinuities (Figure 1). The hyperbolic nature of mass and momentum partial differential equations allows discontinuities in the solution in form of hydraulic bores (Vasconcelos, 2005). Figure 1 shows a smooth interface for illustration purposes only. Actually, the interface is modeled as a steep, near-vertical shock wave (Song et al., 1983) since it behaves as a moving internal hydraulic jump of extremely large magnitude representing abrupt flow change because of $c \to \infty$ as $T \to 0$ (Equation 8) for both pressurization and depressurization.

**Solution of free surface side of the interface**

For flow downstream of the interface (free-surface flow), we need to

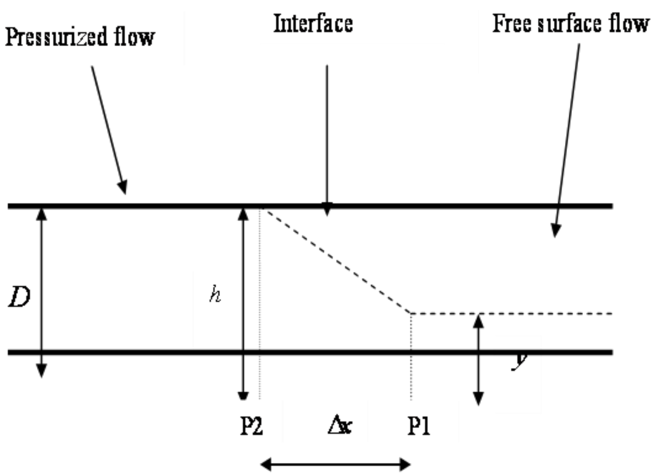


**Figure 1.** Control volume for the interface.

determine the flow depth and velocity. In the case of a surge that advances from upstream to downstream; the free-surface side of the interface can be solved separately to find the velocity $v_1$ and the depth $y_1$ at P1 (Figure 1), by using the characteristic flow equations in the free surface region.

### Solution of moving interface

Analysis of the moving interface or surge front requires determination of the interface location $x$ and velocity $w$. In this region, the equations that should be applied are the conservation of mass and momentum quantities given in Equations 17 and 20, respectively (Fuamba, 2003; Politano et al., 2005; Gomez and Achiaga, 2008).

### Continuity equation

The conservation of mass equation for the surge is:

$$\frac{d(A_1 - A_2)}{dt}\Delta x = A_1(v_1 - w) - A_2(v_2 - w) \quad (16)$$

where $A_1$, $v_1$ are respectively, the cross sectional area of flow and velocity at the upstream (pressurized) end of the interface; $A_2$, $v_2$ are respectively, the cross sectional area of flow and velocity at the downstream (free-surface) end of the interface; $\Delta x$, the length of the control volume that contains the interface; $w$, surge velocity; subscripts 1 and 2, locations at the free-surface and pressurized zones, respectively. If $\frac{d(A_1 - A_2)}{dt}$ is taken as zero as is practically expected during pressurization, then:

$$A_1(v_1 - w) = A_2(v_2 - w) \quad (17)$$

### Motion equation

In the derivation of the equation of motion, the momentum equation $F = ma$ was used where $F$ is the net force causing acceleration, $a$, while $m$ is the mass of the fluid. Also, $F = h\rho gA$ where $h$ is the depth of fluid; $\rho$ is the fluid density; $g$ is gravitational acceleration and $A$ is cross-sectional area of fluid. Therefore:

$$F = \frac{mv_2 - mv_1}{t} = \frac{m}{t}(v_2 - v_1) = \rho\frac{Vol}{t}(v_2 - v_1) = \rho Q(v_2 - v_1) = \rho A_1 v_1 (v_2 - v_1) \quad (18)$$

where $Q = A_1 v_1$ that is, mass is constant at $A_1 v_1$. The velocity at point 2, velocity at point 1 and the time between these velocities are represented as $v_2$, $v_1$ and $t$, respectively. $Vol$ is volume and $Q$ is discharge. This implies that:

$$(\bar{h}A_2 - \bar{y}A_1)\rho g = \rho A_1 v_1 (v_2 - v_1) \quad (19)$$

in which $\rho$ cancels out giving

$$\bar{h}A_2 - \bar{y}A_1 = (V_1 - w)\frac{A_1}{g}(v_1 - v_2) \quad (20)$$

after incorporating the interface velocity. $\bar{h}$ and $\bar{y}$ are depths from the water surface to the centre of gravity of the flow cross-sectional areas of the pressurized and free surface ends of the interface, respectively. The new interface position is found from the kinematic condition $\Delta x = w.\Delta t$.

### Solution of the pressurized side of the interface

For flow upstream of the interface (pressurized flow), we need to determine the velocity $v_2$ and pressure head $h_2$. The velocity of water at point P2 (Figure 1) is obtained from continuity Equation 17 as:

$$v_2 = w + \frac{A_1}{A_2}(v_1 - w) \quad (21)$$

The pressure at P2 is obtained from the characteristic equation for pressurized flow that does not cross the interface trajectory Equation 22 derived from Equation 15 for the reason that, in the transition region of two different flows (Figure 1), there are no valid characteristic equations that cross the interface trajectory

$$h_2 = h_{L2} + a\left(\frac{v_2 - v_{L2}}{g} - \Delta t(S_0 - S_{fL2})\right) \quad (22)$$

**Table 1.** Model input parameters.

| Input parameter | Value |
|---|---|
| Flow depth at the start of the pressurized characteristic equation (m) | 0.08 |
| Pressure at pressurized interface side (m) | 1.1 |
| Velocity at the start of the pressurized characteristic equation (m/s) | 2 |
| Friction slope | 0.0008 |
| Pipe length (m) | 1000 |
| Pipe diameter (m) | 0.1 |

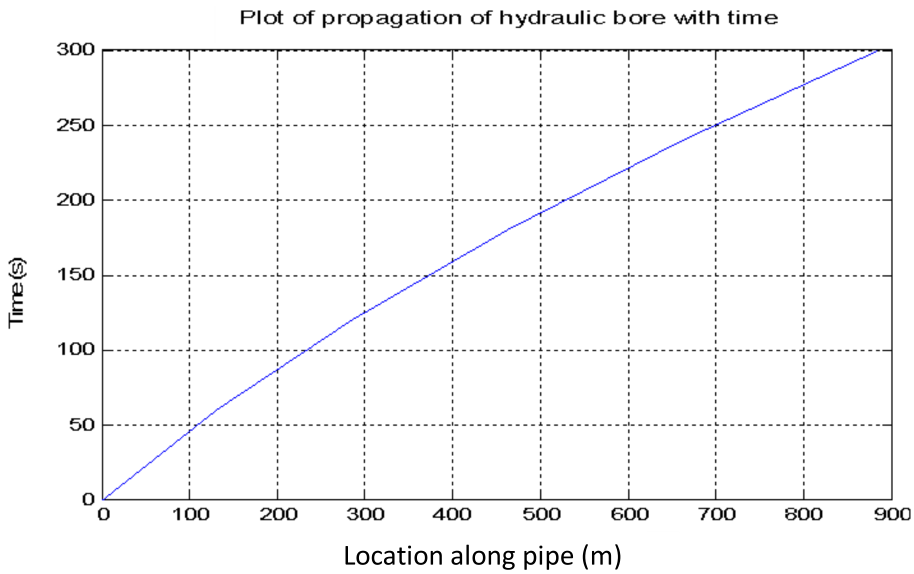


**Figure 2.** Behaviour of pressure surge.

where *a* is the celerity of the pressure wave; $h_{L2}$, $v_{L2}$ and $S_{fL2}$ are the head, velocity and friction slope at the beginning of the characteristic curve, respectively.

There is need to ensure that the size of the time step conforms to Courant's stability condition that ensures convergence of the finite difference equations (Chadwick et al., 2004). As we want information along the pipe to travel along the characteristic lines, we select the time interval $\Delta t$ such that $\Delta t \leq \frac{\Delta x}{|v \pm c|}$ for free surface flow and the one for pressurized flow has $c$ replaced with $a$. Thus, the size of $\Delta x$ and the wave celerity $c$ determine the size of the time interval. The parameter $a$ is much greater than $c$ so $\Delta t < \frac{\Delta x}{v + a}$ is the stability condition that is applied to the whole grid.

**Programming**

The aforementioned system of equations developed was programmed in MATLAB to simulate the pressurization of a pipeline. The model equations were discretised using a fixed-grid method with a first-order finite difference approximation. The resulting nonlinear equations were solved using the Newton-Raphson method.

## RESULTS

This section presents the results of the simulation of the pipe pressurization process. The simulation involved the following inputs: pipe length, pipe diameter, flow depth, pressure, velocity and friction slope. Flow depth and pressure values were obtained from initial boundary conditions (typical field values) and were measured from the pipe invert. The velocity was obtained from the initial conditions that were assumed. For the programme inputs in Table 1, the outputs are shown in Figures 2 and 3. Table 2 summarizes key model outputs that are analysed subsequently.

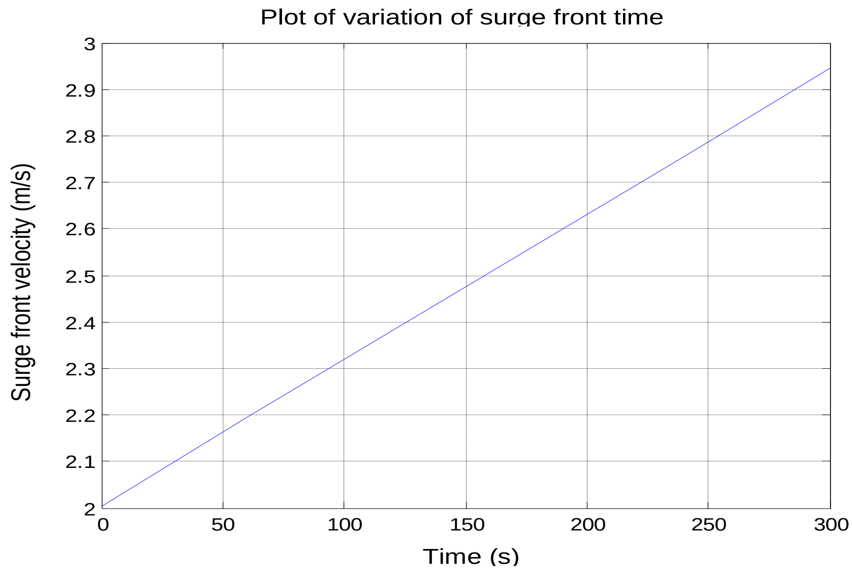


**Figure 3.** Behaviour of pressure surge velocity.

**Table 2.** Sample model outputs.

| Pressure (m) | Water velocity (m/s) | Interface velocity (m/s) | Time (s) | Location (m) |
|---|---|---|---|---|
| 0.4367 | 2.0035 | 2.0056 | 0 | 0 |
| 0.4358 | 2.1212 | 2.1935 | 60 | 131.6089 |
| 0.4332 | 2.2389 | 2.3814 | 120 | 285.7622 |
| 0.429 | 2.3566 | 2.5692 | 180 | 462.4551 |
| 0.423 | 2.4742 | 2.757 | 240 | 661.6828 |
| 0.4154 | 2.5919 | 2.9448 | 300 | 883.4408 |

## Surge front characteristics

The plot in Figure 2 reveals the rate of movement of the hydraulic bore (pressure surge or interface) along the pipeline with time. After approximately 45 s, the surge front had travelled 100 m giving a velocity of 2.2 m/s and after 250 s it had travelled 700 m giving a velocity of 2.8 m/s (Figure 3). This is a good indication of the rate of development of pressures along the pipeline and shows that the interface accelerates during pressurization as a result of the net force from the water that overcomes its gravitational and frictional resistance.

## Pressure characteristics

The plot in Figure 4 reveals development of pressure along a length of pipe at an instant. It can be observed that pressure builds up gradually along the entire pipeline in accordance with the propagation of the pressure surge or interface. Pressurization occurs in the direction in which the interface moves. As the interface advances forwards along the pipeline, whichever point it touches starts to get pressurized and the pressure at this point continues to increase ahead of the pressure at the subsequent points along the pipeline. It can be interpreted that pressure will increase with time at this point on the pipeline at the same rate as that of the propagation of the interface along the pipeline. It should be noted that the pressure values in Figure 4 and Table 2 are measured from the pipe bottom.

## Velocity characteristics

The plot in Figure 5 reveals the variation of water velocity (not interface velocity) as the interface moves along the pipeline during initial pressurization. This shows that

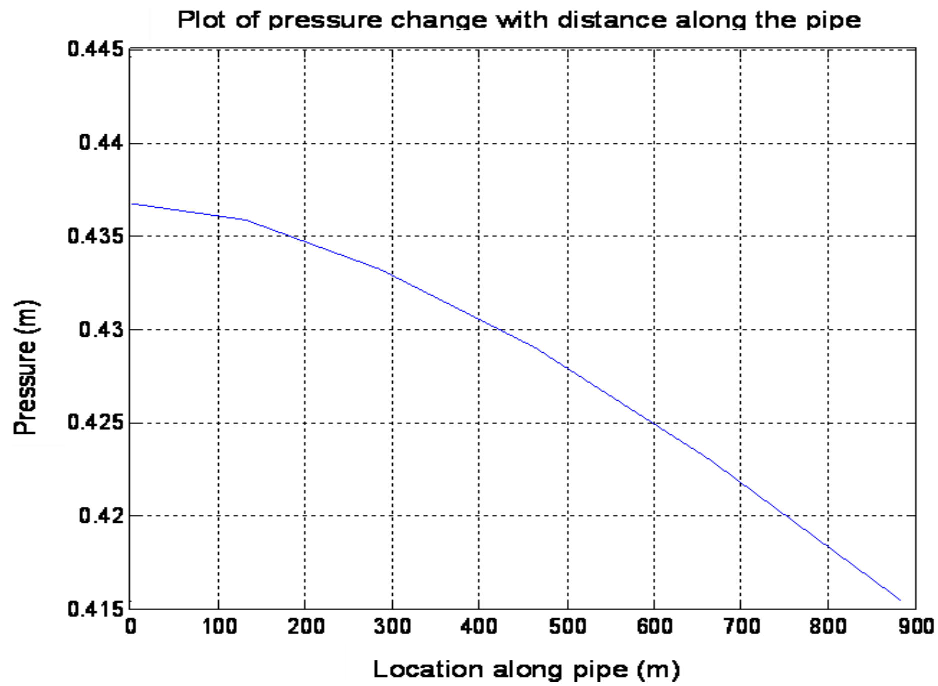


**Figure 4.** Pressure variation along pipeline.

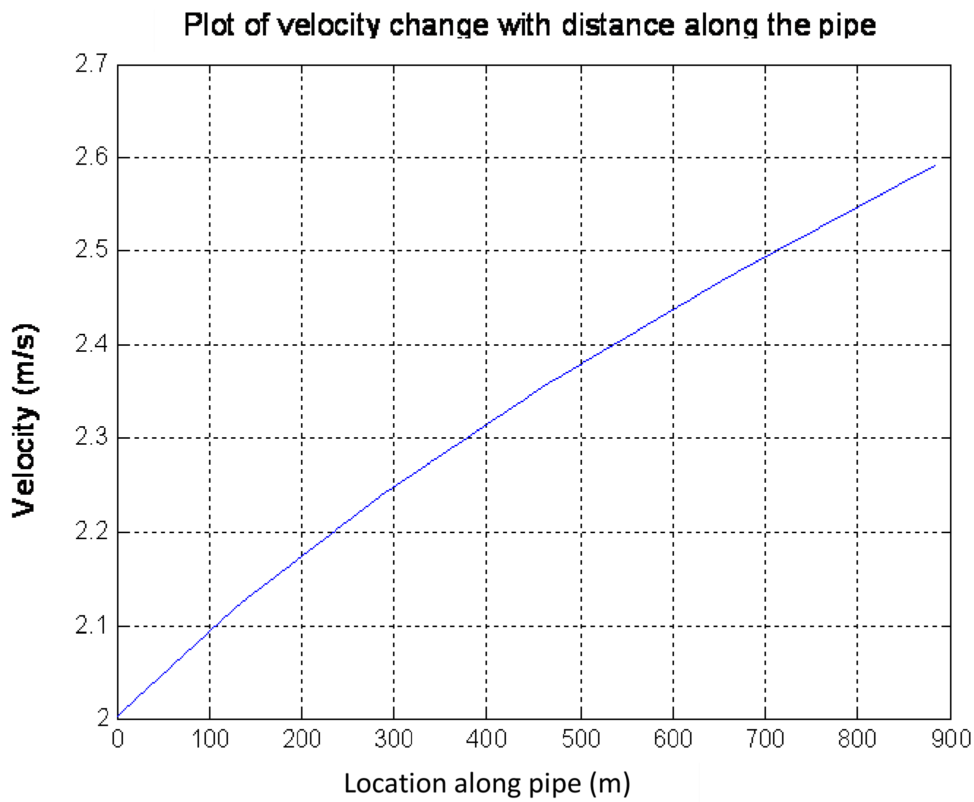


**Figure 5.** Velocity variation along pipeline.

**Table 3.** Comparison of surge front and water velocities.

| Location (m) | Velocity (m/s) | |
|---|---|---|
| | Surge front | Water |
| 100 | 2.2 | 2.09 |
| 700 | 2.8 | 2.5 |

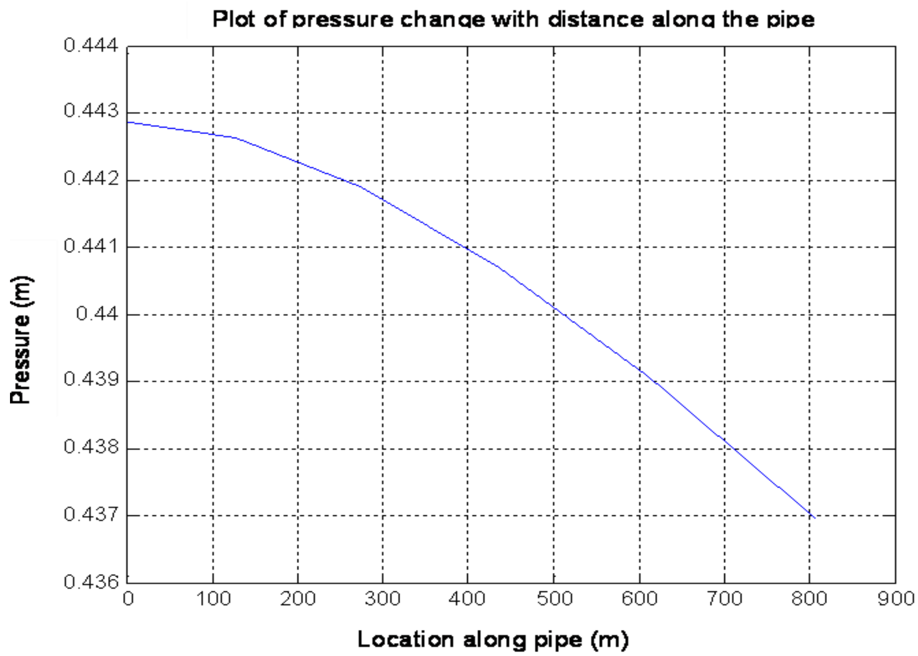


**Figure 6.** Pressure variation along pipeline.

water accelerates. It can be reasonably explained that the velocity is highest at the points that the interface just touches because it encounters no flow resistance at those points of free surface flow apart from friction. From Figures 3 and 5, it can be observed as would be expected that the surge velocity is higher than the water velocity. A comparison of surge velocity with water velocity is shown in Table 3 for values picked at two locations; 100 and 700 m along the pipeline.

The results obtained highlight the advantages of developing a fully dynamic and transient model. Not only is the surge front location and propagation accurately predicted during the transient flow phase between free surface and pressurized flows but flow conditions as well.

## Pipe size and pressurisation

It is shown that the pressure values obtained during the pressurization of the pipeline increase with pipe size. For example, the pressures at the various points are higher with a 200 mm pipe (Figure 6) than with the 100 mm pipe used in Figure 4. Table 4 illustrates a comparison of pressures along the pipeline in Figures 4 and 6. Figure 7 shows a comparison of both pressure results in one graph. The bigger the pipe size, the lower the frictional head losses arising from interaction of water with pipe walls. This consequently leads to higher pressures and further justifies the use of bigger diameter pipes in water supply.

## DISCUSSION

Results show that if we can track the movement of the hydraulic bore in the pipelines, and by obtaining the pressures along the pipeline in the network, we are able to predict what pressures are available at particular

**Table 4.** Comparison of pressures along pipelines for different pipe diameters.

| Pipe diameter (mm) | Location along pipeline (m) | | | | |
|---|---|---|---|---|---|
| | 0 | 200 | 400 | 600 | 800 |
| 100 | 437.0 | 434.5 | 430.5 | 425.0 | 418.5 |
| 200 | 442.8 | 442.3 | 441.0 | 439.2 | 437.0 |

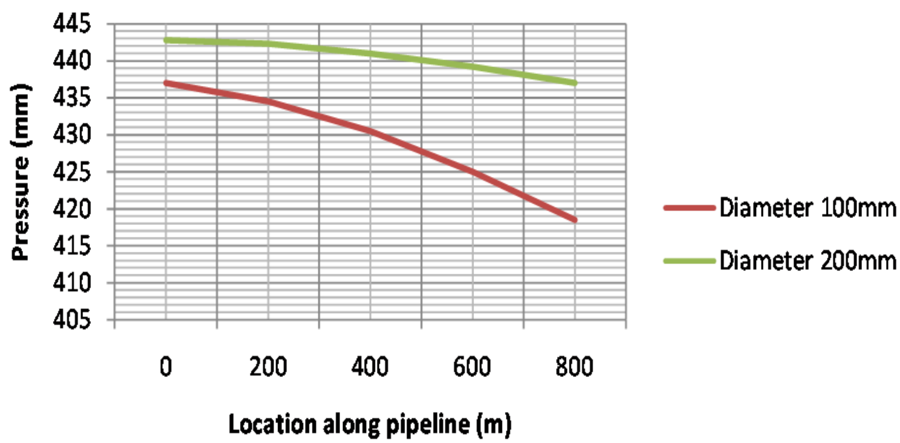


**Figure 7.** Pressure comparison for different pipe sizes.

locations in the network and the subsequent flows that can be enabled by these pressures. Pragmatic management decisions can then be made to ensure that water is available at different sections of the network. Such decisions can include closure or opening of valves, installation of adequate pipe and reservoir sizes in the network or a logical rationing program for water supply (Nyende-Byakika et al., 2010). In addition, if downstream conditions are adjusted accordingly, for example, by varying the valve aperture, then the model can predict the various pressures and flows that are enabled by the implemented actions while the network undergoes pressurization and depressurization.

Pressurization in a pipe occurs due to a variety of reasons. If the supply head is sufficiently high, or if inflow rate is significantly larger than the outflow rate, the speed with which the pressurization wave moves can be very significant. It should be noted that pressurization does not always occur when the pipeline is full; it can also occur when the pipeline is not completely full as long as the initial head and discharge can allow sufficiently rapid filling. It can be realized that if the initial discharge and head are insufficient, lower water depths cannot produce the transition to pressure flow, but only an increase in water depth. On the other hand, greater water depths may not show a pressure front but almost an instantaneous transition to the pressure flow for the whole pipeline (Gomez and Achiaga, 2008), a case that is not envisaged under normal conditions because of the large pipeline lengths, diameters and slopes. In all cases, the transition occurs (and was modeled) through a moving interface that advances into the free surface portions of the system.

During rapid filling, a surge moving against the flow may develop a steep front but in cases of gradual filling, gentle slopes or depressurization, the surge may have a very smooth interface. Even if the interface is smooth, the transition between free-surface flow and pressurized flow cannot be continuous because the gravity wave speed (Equation 8) would be infinite at the point of transition where $T = 0$ i.e. $c \to \infty$ as $T \to 0$ and this represents an area of abrupt flow change creating a discontinuity which gives the Saint Venant equations their hyperbolic character (Fuamba, 2003). For this reason, it is always necessary to assume a discontinuity at the interface.

## Conclusion

In this paper, the pressurization process of pipes was studied and involved tracking the movement of the interface with the aim of determining where and when pressures would start to build up along the pipes. The process was analyzed and modeled in this work with a view of clearly understanding what happens during this phenomenon and consequently aiding engineers in ultimately designing systems and operations that take full advantage of this phenomenon. The motivation to study this flow phenomenon arose from the fact that it is the 'pressurization' stage that leads to the 'pressurized' state that is of profound interest to water supply managers and engineers. As a management tool, this would help inform when particular sections (nodes) of the pipes would build pressures thereby starting to release water and consequently, what actions should be taken for this to happen.

The results obtained highlight the advantages of developing a fully dynamic and transient model in the solution of transient LPOCF conditions in water distribution networks. Not only was the surge front location and propagation accurately predicted during the transient flow phase between free surface and pressurized flows but flow conditions as well.

Further studies should target setting up experiments and field tests to validate the results. They should also consider production of commercial models that tackle the dual character exhibited by several water supply systems, that is, co-existence of pressurized and free-surface flow conditions in the same network.

**REFERENCES**


Chadwick A, Morfett J, Borthwick M (2004). Hydraulics in Civil and Environmental Engineering. UK: Spon Press, p. 59.

Chou T (2009). The Method of Characteristics. University of California. Los Angeles.

Franz DD, Melching CS (1997). Full Equations model for the solution of the full, dynamic equations of motion for one-dimensional unsteady flow in open channels and through control structures. U.S. Geological Survey Water-Resources Investigations Report.

Fuamba M (2003). Contribution on Transient Flow Modelling in Storm Sewers. J. Hydraul. Res., 40(6): 685-693.

Gomez M, Achiaga V (2008). Mixed Flow Modelling Produced by Pressure Fronts from Upstream and Downstream Extremes. Proc. American Society of Civil Engineers.

Hamam MA, Mccorquodale JA (1982). Transient conditions in the transition from gravity to surcharged sewer flow. Canadian J. Civ. Eng., 9: 189-196.

Ingeduld P, Svitak Z, Pradhan A, Tarai A (2006). Modelling intermittent water supply systems with EPANET. 8th Annual WD Symposium EPA Cincinnati. August.

Leon AS (2007). Improved modelling of unsteady free surface, pressurised and mixed flows in storm sewer systems. PhD Thesis. University of Illinois at Urbana-Champaign.

Li J, Mccorquodale A (1999). Modelling mixed flow in storm sewers. J. Hydraul. Eng., 125(11): 1170-1180.

Nyende-Byakika S (2011). Modelling of Pressurised Water Supply Networks that may exhibit Transient Low Pressure-Open Channel Flow Conditions. PhD Thesis. Vaal University of Technology. Vanderbijlpark. South Africa.

Nyende-Byakika S, Ngirane-Katashaya G, Ndambuki JM (2010). Behaviour of stretched water supply networks. Nile Water. Sci. Eng. J., 3(1).

Politano M, Odgaard J, Klecan W (2005). Numerical Simulation of Hydraulic Transients in Drainage Systems. Mecanica Computacional, *p. 24*. Argentina.

Rossman AL (2000). EPANET Users' Manual. *National Risk Management Laboratory*. United States Environmental Protection Agency, Ohio.

Song CCS, Cardle JA, Leung KS (1983). Transient mixed flow models for storm sewers. J. Hydraul. Eng., 109(11): 1487-1504.

Trajkovic B, Ivetic M, Calomino F, Dippolito A (1999). Investigation of transition from free surface to pressurized flow in a circular pipe. Water Sci. Technol., 39(9): 105-112.

Tucciarelli T (2003). A new algorithm for a robust solution of the fully dynamic St. Venant equations. J. Hydraul. Res., 41(3): 239-246.

Vasconcelos JG (2005). Dynamic approach to the description of flow regime transition in storm water systems. PhD Thesis. University of Michigan.

Wiggert DC (1983). Transient flow in free-surface, pressurised systems. J. Hydraulics Division, ASCE, 98, No. HY1, 1972.