2018 Building Performance Analysis Conference and
SimBuild co-organized by ASHRAE and IBPSA-USA
Chicago, IL
September 26-28, 2018

A PSEUDO-TRANSIENT METHOD FOR MODELING FLOW NETWORKS

Aaron Powers
Johnson Controls, Inc., Memphis, TN

ABSTRACT

A flow network is a modeling technique often used to model fluid flow in buildings and HVAC systems. This technique allows for fluid flows and pressures to be simulated without resorting to a full CFD analysis. A flow network is simulated by applying the steady-state conservation of mass principle at discrete points called nodes to create a system of nonlinear equations which can be solved by a variety of methods. Most solvers for building simulation flow networks are based on some derivative of Newton's method. These solvers can show fast convergence when good initial guesses are provided but can be slow or divergent otherwise, especially in large networks. This paper introduces a new fictitious transient term to the nodal conservation of mass equations which allows for the equations to be decoupled and solved using explicit methods. This effectively removes the need for a multivariable nonlinear solver when simulating flow networks. The effects of a single model input parameter on stability and accuracy was investigated. Test cases were used to compare the new modeling technique to a traditional steady-state model using a Levenberg Marquardt solver. The results show that the new method creates almost no loss in accuracy while greatly improving the simulation time, especially for shorter time steps and larger models.

INTRODUCTION

A flow network is a lumped element model of fluid flow in buildings. Lumped element models simplify the spatial complexities of a physical system into a finite state space. This allows for the simulation of fluid flow in buildings without resorting to full CFD methods. The state space of a building flow network is comprised of a number of pressure nodes which are connected through various flow elements. An example schematic of a flow network is shown in Figure 1.

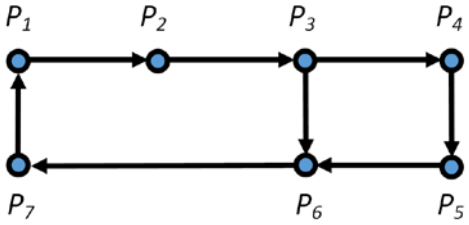

Figure 1 Example flow network schematic

Consider the flow through a single element connecting two nodes as shown in Figure 2.

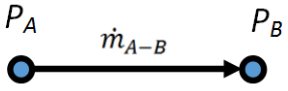

Figure 2 Flow between two pressure nodes

The fluid flow through a single element is generally a function of the pressure differential between the inlet and outlet nodes (Eq 1.).

$$\dot{m} = f(P_A - P_B) \quad (1)$$

Due to the incompressibility of flow in building system components, the flow functions are generally algebraic rather than differential. In other words, the fluid mass inside a component does not vary in time. Components such as pipes, ducts, and window openings will have a decreasing pressure in the direction of fluid flow while components such as pumps and fans will have an increasing pressure in the direction of fluid flow.

There are two types of pressure nodes in a flow network: variable nodes and boundary nodes. Variable nodes are those around which the conservation of mass applies. Boundary nodes have specified pressures independent of the system and can either consume or produce flow. Consider a single variable pressure node with a number of flows entering and leaving as shown in Figure 3.

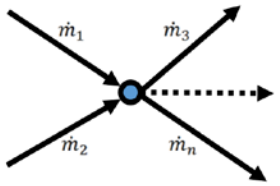

Figure 3 Nodal mass balance

Applying the conservation of mass to the node results in the following equation.

$$\sum_{i=1}^{n} \dot{m}_i = 0 \tag{2}$$

In a flow network with *n* total variable pressure nodes, this results in an *n* dimensional nonlinear system. Since the equations are coupled and algebraic, iterative methods must be used to solve for the unknown pressures at each time step.

Flow networks have been implemented in a number of programs, mostly in the application of passive airflow modeling (Dols and Polidoro, 2016; EnergyPlus, 2015; Walton, 1990; AIRNET, 1989). The application of the steady conservation of mass equation as a part of a larger nonlinear building system has also been implemented in other programs (Clark 1985; TRNSYS 2009). These programs all use an iterative method to solve for unknown pressures and flow at each time step. The solvers implemented in these programs include the method of successive substitution and various derivatives of Newton's method.

There are several challenges in implementing flow networks in building simulation. First is the issue of convergence of the nonlinear solver (Chen et. al. 2017). Second is the computational complexity and simulation time required by the use of nonlinear solvers. Finally, when coupled with controls, the use of implicit solvers can result in undesirable behavior. For example, on/off controllers can remain in a cycling manner for an indefinite amount of times (TRNSYS, 2009).

PSEUDO-TRANSIENT METHOD

Pseudo-transient methods are used to solve steady problems by creating an equivalent transient problem and simulating until steady-state is reached [Fletcher, 1998]. This method has been used mostly to solve CFD problems but has had little to no use in flow networks and building simulation. An equivalent transient flow network problem can be created by modifying the nodal mass conservation equation (Eq. 2) with a transient term and substituting the volumetric flow for the mass flow.

$$\sum_{i=1}^{n} \dot{V}_i = \frac{1}{a}\frac{dP_i}{dt} \tag{3}$$

Physically, this is analogous to simulating a compressible fluid with a constant proportionality between density and pressure. The a parameter is therefore a measure of the artificial compressibility of the fluid introduced into the problem; however, it is also simply a tunable constant of proportionality that is a mathematical necessity to the problem. This new system can now be solved explicitly using Euler's Method to discretize the time differential (APPENDIX II).

EFFECTS OF ALPHA AND TIME STEP

As with any explicit method, there is a tradeoff between accuracy and stability. The selection of the a parameter and dt can have a large impact on these metrics. Since the transience was artificially introduced to a steady problem, higher values of a and dt will generally lead to more error but may lead to faster convergence times. A simple test case, Test Case 1, was used to explore the effects of changing a (Figure 4). Note that the parameter a is not unitless. For the purposes of this study, the units of volumetric flow are GPM, the units for pressure are Ft. H_20, and the units for the time step are seconds. This then sets the units for a.

Figure 4 Test Case 1

This example includes a source node P_1, a sink node P_4, and two variable nodes P_2 and P_3. A pipe connects P_1 and P_2, and another connects P_3 and P_4. A linear two-way control valve connects P_2 and P_3. Element and node inputs are shown in the tables below. Descriptions of various element models can be found in Appendix I.

Table 1 Element inputs for Test Case 1

Element	Nominal Flow [GPM]	Nominal Pressure Drop [Ft. H_2O]	Pressure Exponent
Pipe 1	100	10	1.8
Valve	100	30	2.0
Pipe 2	100	10	1.8

Table 2 Node inputs for Example 1

Node	Initial Pressure [Ft. H_2O]	Boundary Node Pressure [Ft. H_2O]
P_1	N/A	80
P_2	0	N/A
P_3	0	N/A
P_4	N/A	0

The control valve is given a constant signal of 0.75. Under these conditions, the actual flow through the system should be approximately 82.1 GPM.

With dt set to 1 second, a was varied for several simulations of 300 seconds. The results are shown in Figure 5.

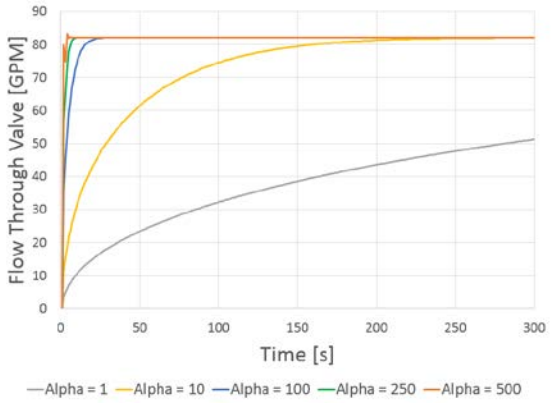

Figure 5 Effects of changing a

As the chart shows, higher values of a resulted in much faster convergence to the steady flow. In fact, for the $a = 500$ case, the flow through the valve is within 1% of the steady state flow after just five seconds. The $a = 500$ case shows a slight overshoot followed by small damped oscillations. As a continues to increase, the damped oscillations become larger (Figure 6). By $a = 750$, the flow becomes completely unstable (Figure 7).

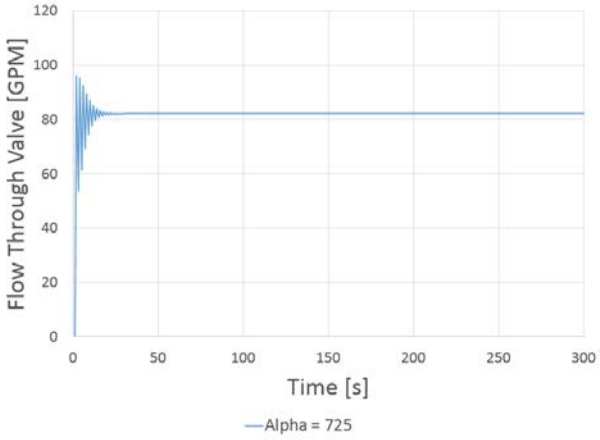

Figure 6 Damped flow oscillations for a *= 725 case*

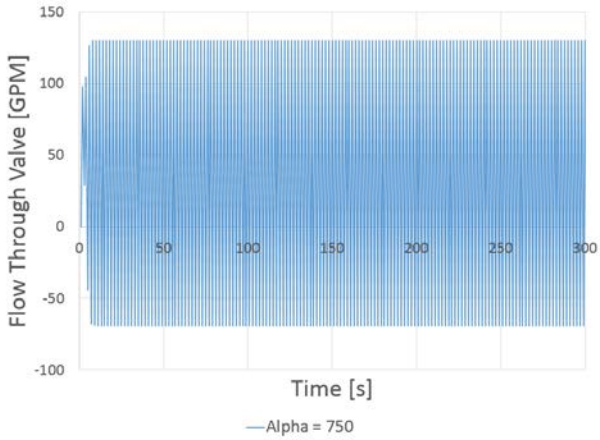

Figure 7 Unstable flow for a *= 750 case*

The structure of Eq 3 shows that the selection of dt has a similar effect on accuracy and stability. Specifically, the use of a smaller time steps increases the stability but increases the number of computations required to reach steady state.

TEST CASES

Two test cases were used to compare the pseudo-transient method to a traditional steady state model using a Levenberg Marquardt solver. The first test was Test

Case 1 as described in the previous section. However, for this example, the control signal for the valve was modified to

$$u = 0.5\left(1 + \sin^2\left(\frac{0.5rrt}{28800}\right)\right) \quad (4)$$

where t is the time in seconds. This function was selected to simulate the time scale at which loads typically change in a building.

The model was simulated for 10 hours, and the resulting valve flows are shown in Figure 8. As the plot shows, the flows simulated by the two methods are almost indistinguishable. This shows that there was almost no loss in accuracy by using the pseudo-transient method; however, the Levenberg-Marquardt simulation took approximately 3.9 times longer to execute. This was largely due to the overhead required to calculate the Jacobian matrix at each time step.

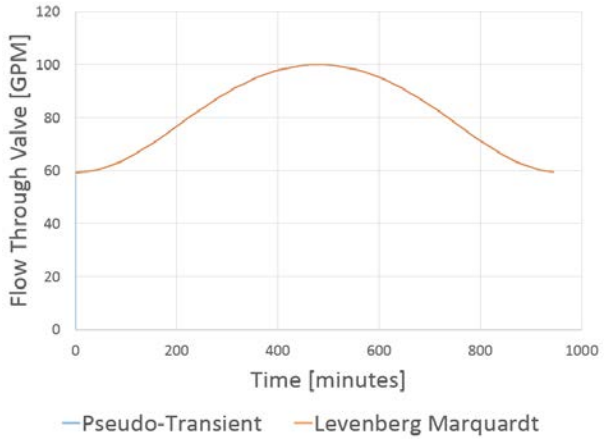

Figure 8 Results for Test Case 1

The second test case, Test Case 2, is shown schematically in Figure 9. This test consists of a pump serving three two-way valves which are plumbed in parallell off of a main header. The three valves were given control signals defined by

$$u_1 = 0.5\left(1 + \sin^2\left(\frac{0.5rrt}{28800}\right)\right) \quad (5)$$

$$u_2 = 0.5\left(1 + \sin^2\left(\frac{0.5rrt}{5000}\right)\right) \quad (6)$$

$$u_3 = 0.5\left(1 + \sin^2\left(\frac{0.5rrt}{35000}\right)\right) \quad (7)$$

The pump was controlled, through a VFD, with a proportional controller and a constant differential pressure setpoint across nodes P_6 and P_7.

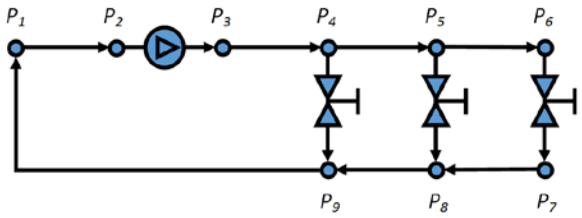

Figure 9 Schematic for Test Case 2

All initial pressures were set to 0 Ft. H_20. Element inputs are shown in the table below.

Table 3 Element Inputs for Test Case 2

Element	Nominal Flow [GPM]	Nominal Pressure Drop [Ft. H_2O]	Pressure Exponent
$Pipe_{1-2}$	225	10	1.8
Pump	225	-100	N/A
$Pipe_{3-4}$	225	10	1.8
$Pipe_{4-5}$	150	10	1.8
$Pipe_{5-6}$	50	10	1.8
Valve 1	75	50	2
Valve 2	100	30	2
Valve 3	50	10	2
$Pipe_{7-8}$	50	10	1.8
$Pipe_{8-9}$	150	10	1.8
$Pipe_{9-1}$	225	20	1.8

The pump is controlled to a constant differential pressure setpoint of 10 Ft. H_2O across nodes P_6 and P_7. The proportional gain for the pump controller was set to 0.25 $\frac{1}{Ft.H_2O}$. The resulting pump flow for both methods is shown in Figure 10.

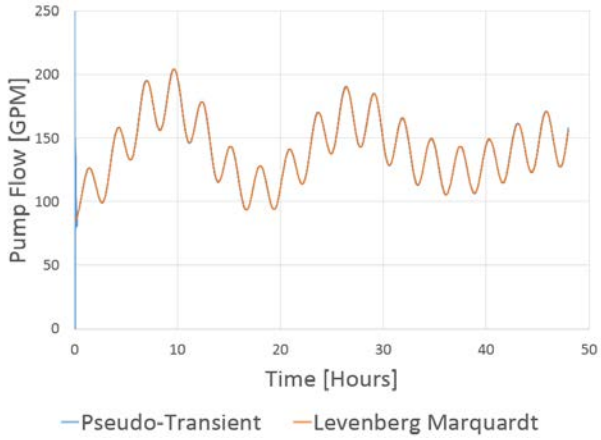

Figure 10 Results for Test Case 2

As with Test Case 1, the resulting simulated pump flows are nearly indistinquishable. These results show that there was almost no loss in accuracy by using the pseudo-transienit method despite executing approximately 27 times faster than the Levenberg Marquardt method.

Since each nodal pressure can be updated in constant time for each time step using the pseudo-transient method, the computational complexity is $O(n)$, where n is the number of variable nodes. The Levenberg Marquardt method (and other Newtonian methods requiring an inversion of the Jacobian) has a computational complexity of $O(n^3)$, which is set by the need to invert an $n \times n$ matrix at every time step. The difference in computational complexity shows why the difference in execution time grows with larger systems. This makes the pseudo-transient method especially well suited for large models.

If simulation results are only of interest for large time scales (e.g. hourly), then the computational advantage of the pseudo-transient method goes away, and the execution time for the two methods become similar. This is due to the pseudo-transient method requiring relatively small time steps for stability and accuracy. As an example, the Levenberg Marquard method for Test Case 2 can be executed approximately 20% faster than the pseudo-transient method when the method uses hourly time steps. However, the use of smaller time steps is becoming more and more necessary, especially when simulating control systems.

CONCLUSION

This paper introduced a new pseudo-transient method for simulating flow networks for building simulation. The new method showed an accuracy similar to that of a more traditional Levenberg Marquardt solver. The new method also executed faster than the Levenberg Marquardt solver, especially for smaller time steps and larger models. Since it is explicit, the new method can be easier to code and debug compared to Newtonial solvers.

NOMENCLATURE

a: pump curve coefficient

C_v: valve flow coefficient

$\dot{m}$: mass flow rate

P: pressure

t: time

u: controller output (0-1)

$\dot{}$: volumetric flow rate

x: pressure exponent

F_{loss}: loss factor

h_a : specific enthalpy of moist air

h_D : mass transfer coefficient

h_{fg} : latent heat of vaporization for water

$h_{s,e}$: effective saturation specific enthalpy

h_Q : heat transfer coefficient for convection

Le_f: Lewis factor

Greek Symbols

Δ: difference

Subscripts

nom: nominal condition

j: discretized time index

REFERENCES

Chen, Z., Wen, J., Kearsley, A., and Pertzborn, A. (2017) Scaling Methods for Dynamic Building System Simulation in an HVACSIM+Environment. IBPSA Building Simulation 2017.

CLARK, D.R. (1985). HVACSIM+ Building Systems and Equipment Simulation Program Reference Manual. National Institute of Standards and Technology.

COMIS. (1990). Fundamentals of the Multizone Air Flow Model – COMIS. Oscar Faber Consulting Engineers.

Dols, W.S. and Polidoro, B.J. (2016). CONTAM User Guide and Program Documentation Version 3.2. National Institute of Standards and Technology

EnergyPlus Documentation. (2015). Engineering Reference.

Fletcher. (1998). Computational Techniques for Fluid Dynamics 1. Springer-Verlag Berlin Heidelberg.

TRNSYS Documentation (2009). Volume 4: Mathematical Reference. Solar Energy Laboratory, University of Wisconsin-Madison.

Walton, G.N. (1989). AIRNET – A Computer Program for Building Airflow Network Modeling. National Institute of Standards and Technology.

APPENDIX I: ELEMENT MODELS

Three element models used in this paper are described below. These models define how the flow through the element is related to the pressure difference across the element.

Pipe

$$\dot{m} = \dot{m}_{nom}\left(\frac{\Delta P}{\Delta P_{nom}}\right)^{1/x} \quad (8)$$

Two-way valve

$$\dot{m} = u C_{v,nom} \Delta P^{\frac{1}{x}} \quad (9)$$

where

$$C_{v,nom} = \frac{\dot{m}_{nom}}{\Delta P_{nom}^{\frac{1}{x}}} \quad (10)$$

Pump Controlled by VFD

$$\dot{m} = u\,\dot{m}_{nom}\,\frac{-a_2 - \sqrt{a_2^2 - 4a_3(a_1 - \Delta P^*)}}{2a_3} \quad (11)$$

where

$$\Delta P^* = \frac{\Delta P}{\Delta P_{nom} u^2} \quad (12)$$

$$a_1 = 1.35348296 \quad (13)$$

$$a_2 = 0.0159317 \quad (14)$$

$$a_3 = -0.36941442 \quad (15)$$

APPENDIX II: DISCRETIZAION OF EQ 3

The time derivative in Eq 3 can be discretized using Euler's method.

$$\sum_{i=1}^{n} a_{ij} = \frac{1}{\Delta t}\left(P_i^{j+1} - P_i^j\right) \quad (16)$$

The future time step pressure P_i^{j+1} can then be calculated explicitly.