

Fluid Transient Analysis during Priming of Evacuated Line

By

Alak Bandyopadhyay

Computer Science, Alabama A & M University, Normal, AL 35762

&

Alok K. Majumdar, Kimberley Holt

Propulsion System Department, NASA Marshall Space Center, Huntsville, AL 35812

Extended Abstract

Pressure surges are critical in the design of spacecraft propellant feed lines. The pressure transients that occur during priming of feed lines are very important in the design and analysis of liquid propulsion systems. During the start-up of the propulsion system of a spacecraft, the process of filling of an evacuated pipeline, is called priming. Priming can generate severe pressure peaks due to the slam (water hammer) of the propellant against a closed thruster valve. The downstream conditions strongly affect the pressure surge. In space systems, satellites, or interplanetary probes, the propellant lines are vacuum-pumped or filled with low pressure helium or nitrogen before the launch. Before operations, these lines are primed with a vaporizing liquid, sometimes in the presence of a non-condensable gas (NCG), which produces water hammer phenomena.

The objective of the current study is to use a finite volume based network flow solver (Generalized Fluid System Simulation Program, GFSSP [1]) for the numerical simulation of Priming in (a) a straight feedline (Figure 1a) and (b) a flow network (Figure 1b). The geometrical configurations and dimensions for the pipe and other components used for the current study are identical to experimental study of Prickett et al [2].

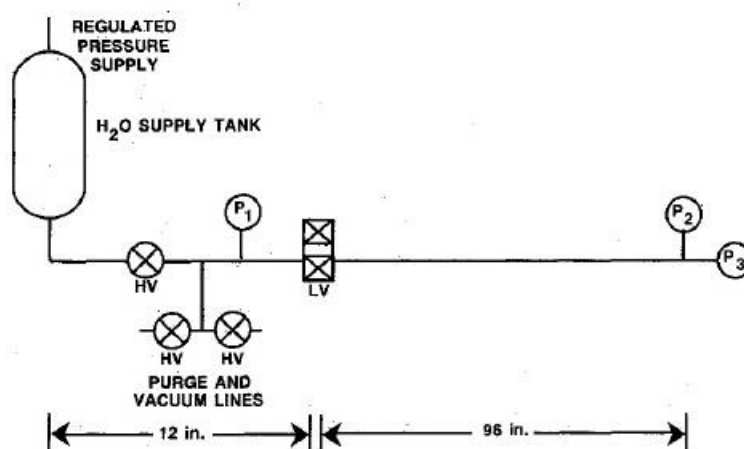


Figure 1a Schematic of Single Feed Line (courtesy: Prickett et al [2])

In the figure shown above are the supply reservoir on the left feeding water through the pipe line, hand valves (HV), the main supply valve (Latch Valve – LV), also shown the pressure transducers at three different points (P₁, P₂ and P₃). The latch valve opens almost instantaneously,

and water flows from the reservoir to the evacuated pipe. The pipe upstream of the latch valve is 12 inches long and 3/8 inches in diameter and the downstream of the latch valve, the pipe is 96 inches long and 1/4 inches in diameter. The supply pressure varies from 30 psia to 120 psia. The second problem considered in the current research is priming in a flow network as shown in figure 1(b) below.

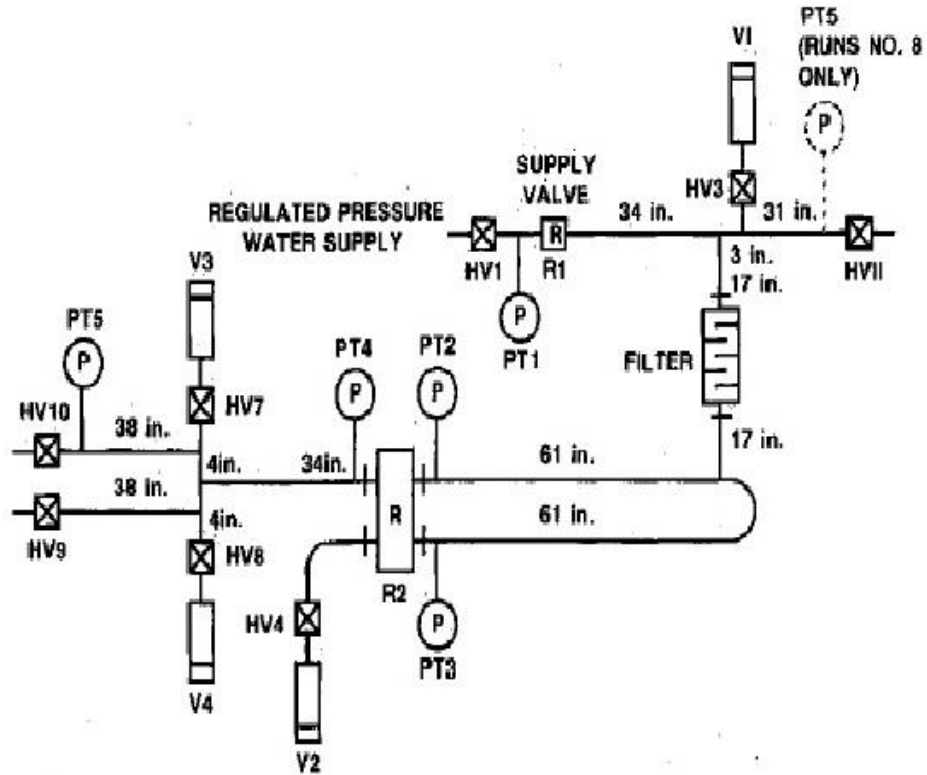


Figure 1b. Schematic of a flow network (courtesy: Prickett et al [2])

The physical problem as shown in Figure 1a is converted into the GFSSP's numerical model by dividing the entire domain into 8 pipe segments, and a restriction for the latch valve as shown in figure 2 below.

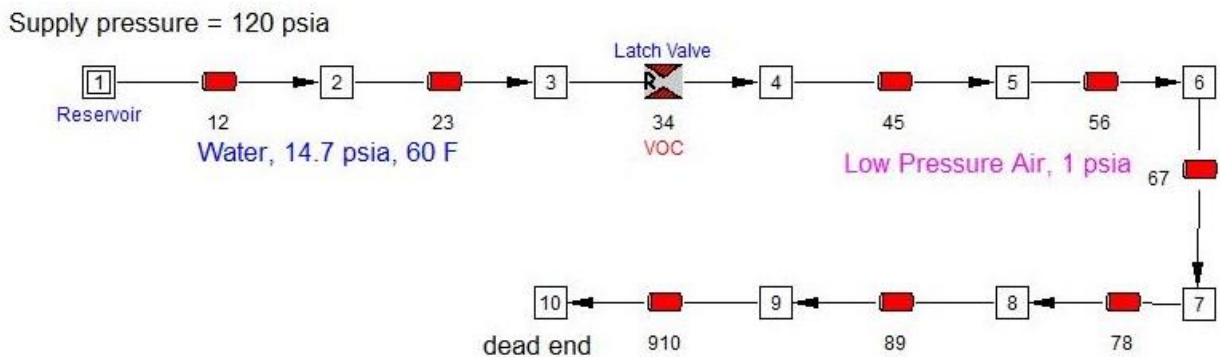


Figure 2. Computational Domain for the Single Feed Line

In order to model initially evacuated line downstream of the latch valve, air at very low pressure is assumed to be present in the pipe line, and the initial pressure of air (p_{air}) is a numerical adjusting parameter. In the current simulation this pressure is varied and gradually lowered to see when the numerical solution is virtually independent to this pressure. It has been found that this optimized air pressure is 1 psia, lowering air pressure below this does not change the result any further. Hence for numerical simulation, air pressure of 1 psia and a time step of 0.1 milliseconds are used for optimal values for the entire simulation. The peak pressure in the feed line is found to be 2279 psia as compared to the experimental results of Prickett et al. [2] of 2350 psia. Also the impact time, that is the time when maximum peak pressure occurs after the latch valve opens, is computed to be 0.173 second against experimental measurement of 0.17 seconds. Figure 3 shows the pressure at few of the nodal points including the pressure at dead end (node 10) as a function of time.

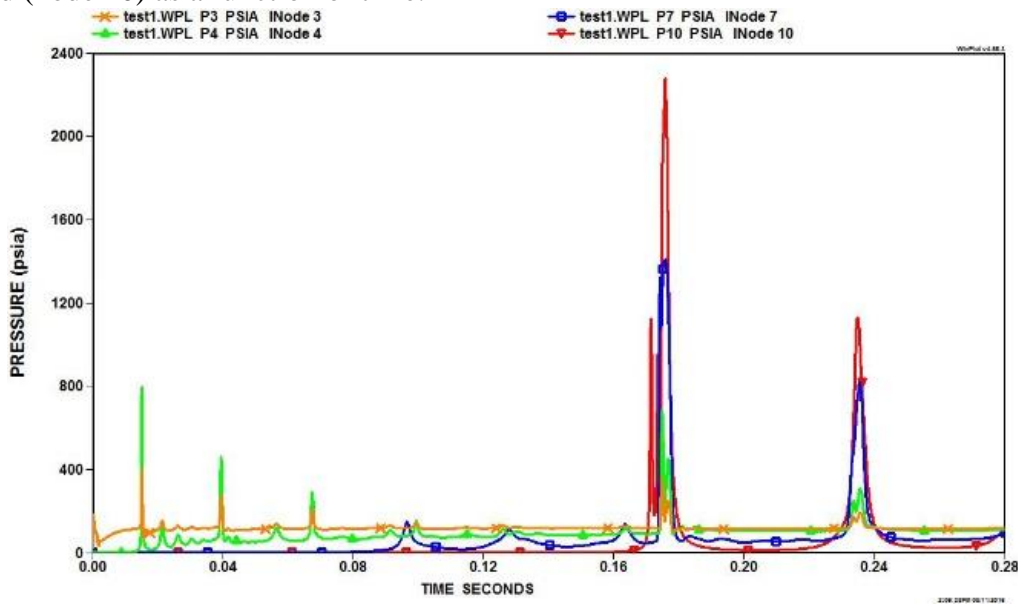


Figure 3. Pressure at various nodes in straight feedline

Priming in flow network as shown in Figure 1b is quite complex and consists of several hand valves (HV), latch valve (supply valve), volume reservoirs (indicated by V1 through V4), and pipe lines. The fluid (water) flows from a reservoir through the hand valve (HV1) and controlled by the latch valve (indicated by R1) and it branches out into three directions: a) going straight to hand valve 2 (HV11), which is considered a dead end to compare with one of the test run of Prickett et al, b) branching out through hand valve HV3, and ending into a volume reservoir, and c) branching out through a filter and rest of the network. The volume of the reservoirs are 3.55, 14.55, 5.68, 4.68 cubic inches respectively for volume V1, V2, V3 and V4. The hand valves are used for control of system purging and evacuation. In the current numerical model they are assumed to be fully open. The liquid filter is modeled suitably using the given pressure drop as a function of discharge rate as given by the relation below.

$$\Delta p, \text{ psia} = 15.92(\text{GPM})^{1.18} \quad (\text{GPM} = \text{discharge rate in gallons per min})$$

The GFSSP model for the flow network is as shown in Figure 4 below:

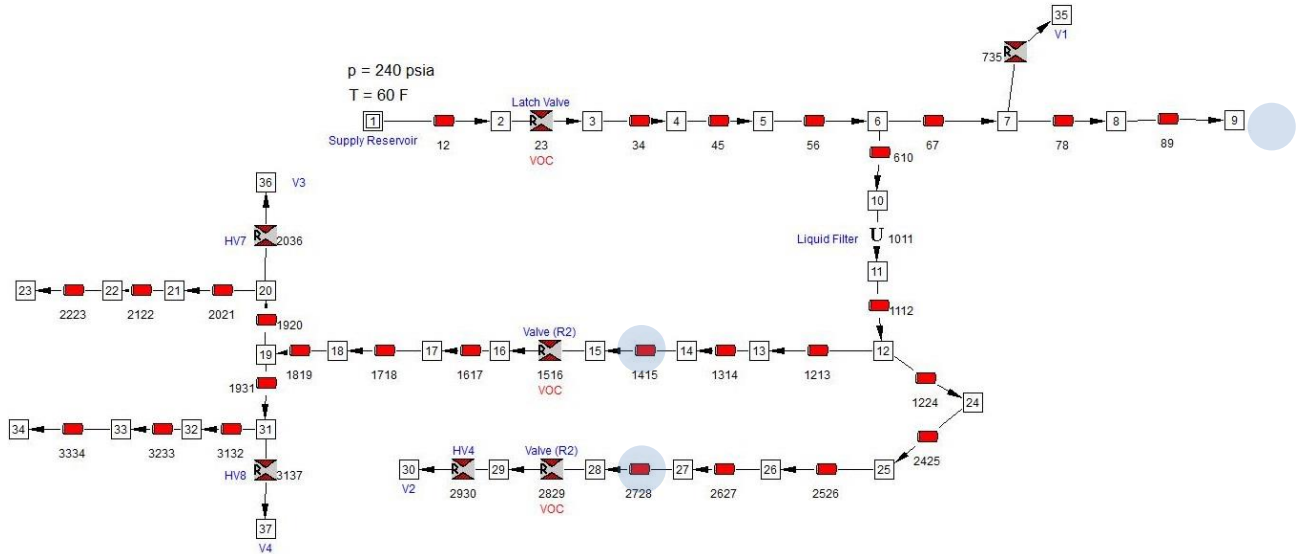


Figure 4. Computational Domain for the Flow Network

As observed from the numerical simulation, the peak pressure occurs at the dead end of the straight branch (node 9), and numerically it is found to be little above 3500 psia. The pressure on the bottom branch, the peak occurs at node 28 and is found to be 1837 psia as compared to test data as reported by Prickett et al as 1800 psia. The numerical results for the pressure at the dead end of the straight region (as shown by node 9) could not be compared with the experimental data of Prickett et al. [2] as it has not been reported by them. The current research also studied the effect of unsteady friction factor model [3,4] on the priming pressure distribution as compared to the conventional steady state friction factor model and peak pressure and will be later reported in the full paper.

Overall, it has been concluded that the two-fluid model used in the current research were able to produce very good agreement of the pressure results as compared with experimental data available in the literature (Prickett et al. [2]). In the case of priming in straight pipe, the numerical prediction match very well with the experimental data with less than 1 % error. The unsteady friction factor helped reducing the peak pressure in the network model. There needs to be more testing done in this regard using the unsteady friction model. The work can be extended for water hammer analysis with real propellants which can be of more practical and real space craft use.

References

- ¹Majumdar, A.K., LeClair, A.C., Moore, R., Schallhorn, P.A., Generalized Fluid System Simulation Program, Version 6.0, NASA/TM—2013–217492, NASA Marshall Space Flight Center, Huntsville, AL, October 2013, <<https://gfssp.msfc.nasa.gov>>.
- ²Prickett, R.P., Mayer, E., and Hermel, J., “Water Hammer in Spacecraft Propellant Feed Systems”, AIAA-88-2920, *Journal of Propulsion and Power*, vol. 8, no. 3, May-June 1992, pp.592–597.

³Brunone, B., Kearney, W., Mecarelli, M., Ferrante, M., “Velocity Profiles and Unsteady Pipe Friction in Transient Flow,” *J. Water Resource Planning and Management*, Vol. 126, No. 4, pp. 236–244, 2000.

⁴Dehkordi, M.M., Firoozabadi, B.D., “Effects of Unsteady Friction Factor on Column Separation,” *Proc. IMECE 2007*, ASME International—Mechanical Engineering Congress and Exposition, Seattle, WA, pp. 1–7, Nov. 2007