A Finite Volume Procedure for Thermo-Fluid System Analysis in a Flow Network

Alok Majumdar
NASA/Marshall Space Flight Center
Huntsville, Alabama

Abstract

This paper describes a finite volume procedure for network flow analysis in a thermo-fluid system. A flow network is defined as a group of inter-connected control volumes called ñnodesò that are connected by ñbranch equations are solved at the nodes and momentum conservation equations are solved at the branches. The flow network also includes solid nodes to account for fluid to solid heat transfer. The heat conduction equation is solved at the solid nodes in conjunction with the flow equations. The properties of a real fluid are calculated using a thermodynamic property program and used in the conservation equations. The system of equations describing the fluid-solid network is solved by a hybrid numerical method that is a combination of the Newton-Raphson and successive substitution method. This procedure has been incorporated into a general-purpose computer program, the Generalized Fluid System Simulation Program, GFSSP. This paper also presents the application and verification of the method by comparison with test data for several applications that include (a) internal flow in a rocket engine turbo-pump, (b) pressurization and loading of a cryogenic propellant tank, (c) fluid transient during sudden opening of valve for priming of an evacuated feed line, and (d) chilldown of a cryogenic transfer line with phase change and two phase flows.

1. Introduction

The need for a generalized numerical method for thermo-fluid analysis in a flow network has been felt for a long time in the aerospace industry. Designers of thermo-fluid systems often need to know pressures, temperatures, flowrates, concentrations, and heat transfer rates at different parts of a flow circuit for steady state or transient conditions. Such applications occur in propulsion systems for tank pressurization, internal flow analysis of rocket engine turbo-pumps, chilldown of cryogenic tanks and transfer lines and many other applications of gas-liquid systems involving fluid transients and conjugate heat and mass transfer. Computer resource requirements to perform time-dependent three-dimensional Navier-Stokes Computational Fluid Dynamic (CFD) analysis of such systems are prohibitive and therefore are not practical. A possible recourse is to construct a fluid network consisting of a group of flow branches such as pipes and ducts that are joined together at a number of nodes. They can range from simple systems consisting of a few nodes and branches to very complex networks containing many flow branches simulating valves, orifices, bends, pumps and turbines. In the analysis of existing or proposed networks, node pressures, temperatures and concentrations at the system boundaries are usually known. The problem is to determine all internal nodal pressures, temperatures, and

concentrations, as well as branch flow rates. Such schemes are known as Network Flow Analysis methods, and they use largely empirical information to model fluid friction and heat transfer. The oldest method for systematically solving a problem consisting of steady flow in a pipe network is the Hardy Cross method [1]. The original method was developed for hand calculations, but it has also been widely employed for use in computer-generated solutions. But as computers allowed much larger networks to be analyzed, it became apparent that the convergence of the Hardy Cross method was very slow or even failed to provide a solution in some cases. The other limitation of this method is its inability to extend to unsteady, compressible flow and heat transfer.

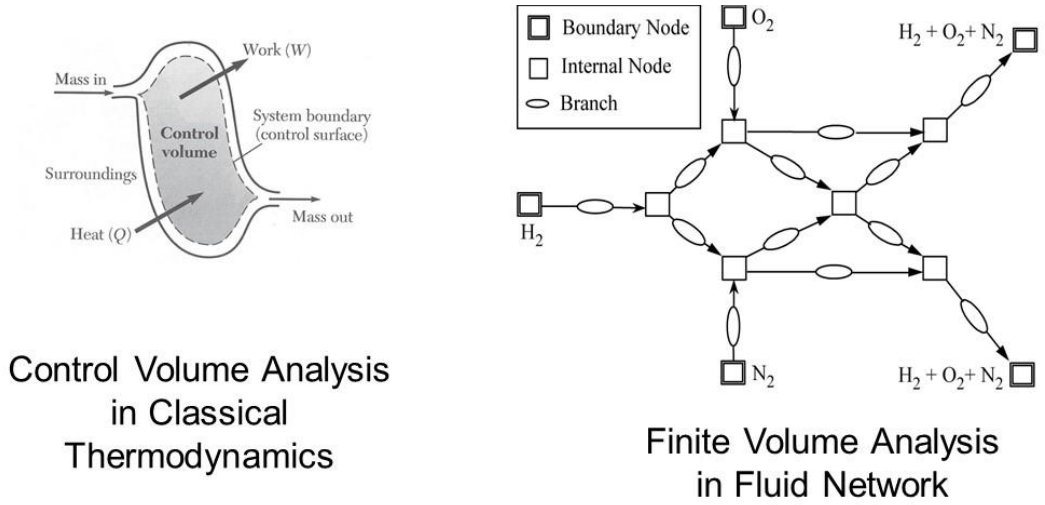


Figure 1. Extension of Control Volume Analysis to Finite Volume Analysis in Fluid Network

Finite volume procedures are an extension of the control volume analysis performed in classical thermodynamics for mass and energy conservation (Figure 1). Therefore, a finite volume procedure is a logical choice for solving network flow which is a collection of inter-connected control volumes. Finite volume procedure was first developed by Professor Spalding and his students at Imperial College [2] to solve the Navier-Stokes equations in two dimensions. The Navier-Stokes equations were expressed in terms of stream function and vorticity using an upwind scheme [3] to ensure numerical stability for high Reynolds number flows. The governing equations are derived using the principle of conservation of variables. The system of equations was solved by a successive substitution method. This method was successfully applied to solve many recirculating flows which were never solved before. The Navier-Stokes equation in three dimensions was solved in its primitive form by Patankar and Spalding [4]. They used a staggered grid where pressures were located at the center of the control volume whereas velocities were located at the boundaries of the control volume. This finite volume procedure was known as the SIMPLE (Semi-Implicit Pressure Linked Equation) algorithm. It uses the mass conservation equation to develop pressure corrections using a simplified momentum equation. The pressures and velocities are corrected iteratively until the solution is converged. Turbulence was modeled by defining an effective viscosity which is a function of turbulence properties such as turbulence energy and its dissipation rate and known as

Launder and Spalding [5] developed the turbulence model equations are solved in conjunction with the mass and momentum conservation equations. The SIMPLE algorithm and two-equation model of turbulence have been implemented in many Computational Fluid Dynamics (CFD) codes in later years.

Navier-Stokes based CFD codes, however, are not suitable for thermo-fluid system analysis. It is not practical to solve for three dimensional Navier-Stokes equations in conjunction with turbulence model equations to model a thermo-fluid system consisting of many fluid components such as pumps, pipes, valves, orifices and bends. On the other hand, it is possible to solve a one-dimensional momentum equation with empirical correlations to model frictional effect to determine flow and pressure distribution in a flow network consisting of many such fluid components within reasonable computer time. A modified form of the SIMPLE algorithm was used to compute flow distribution in manifolds [6,7], where one-dimensional mass and momentum equations were solved using the Colebrook equation [8] for friction factor to account for viscous effect. Numerical predictions compared well with experimental data. However, this approach cannot be extended for any given flow network. A generalized flow network cannot be constructed using a structured co-ordinate system. In order to develop a numerical method to analyze any given flow network, the conservation equations for mass, momentum and energy must be written using an unstructured co-ordinate system. This paper presents a finite volume procedure for calculating flow, pressure and temperature distribution in a generalized fluid network for steady-state, transient, compressible, two-phase and with or without heat transfer. The thermo-fluid system network is discretized into fluid nodes and branches, solid nodes and conductors. The fluid nodes are connected with branches, and scalar properties such as pressure, enthalpy and concentrations are stored in the fluid nodes, and vector properties such as flowrates and velocities are stored in the branches. Solid nodes and fluid nodes are connected by solid to fluid conductors. The conservation equations for mass and energy are solved at fluid nodes and momentum conservation equations are solved at fluid branches in conjunction with the thermodynamic equation of state for real fluids. The energy conservation equation for a solid is solved at the solid nodes. The system of equations is solved by a hybrid numerical method which consists of both the Newton-Raphson and Successive Substitution methods. This procedure has been incorporated into a general purpose computer program, GFSSP (Generalized Fluid System Simulation Program) [9-11]. This paper describes several applications of GFSSP that include (a) internal flow in a rocket engine turbo-pump, (b) compressible flows in ducts and nozzles, (c) pressurization and loading of a cryogenic propellant tank, (d) fluid transient during sudden opening of a valve for priming of a partially evacuated propellant feed line, and (e) chilldown of cryogenic transfer line with phase change and two phase flows.

2. Mathematical Formulation

The mathematical formulation to solve numerically the flow in a network offers a different kind of challenge than solving the Navier-Stokes equations in three dimensions. The Navier-Stokes equations are usually written for the co-ordinate systems which are topologically Cartesian. In a topologically Cartesian system of co-ordinates, a control

volume can have a maximum of six neighboring control volumes: east, west, north, south, high and low. The data structure for a three dimensional co-ordinate system can be adapted for deriving the conservation equations for mass, momentum and energy. On the other hand, a fluid network cannot be fully represented in a three dimensional Cartesian co-ordinate system which has a limitation on the maximum number of neighbors. A fluid network is n-dimensional where n can assume any number. Therefore, its data structure is unique. In the following section the network definition and data structure of a flow network will be described. This will be followed by the description of governing equations which will include the conservation equations of mass, momentum, energy, and mixture species, as well as auxiliary equations such as the thermodynamic equation of state and empirical equations for friction and heat transfer.

2.1 Network Definitions

A flow network is first discretized into nodes and branches prior to the development of the governing equations. The defining parameters of a network are explained with the help of the example of a counter-flow heat exchanger shown in Figure 2. In this example hot fluid in the central tube is cooled by cold fluid in the annulus. The two fluid streams are exchanging energy by heat conduction and convection. This physical system is represented by a network of fluid and solid nodes. The fluid paths in the central tube and annulus are represented by a set of internal and boundary fluid nodes connected by fluid branches. The branch represents a fluid component such as a pipe, orifice, valve or pump. In this particular case the pipe and annulus are chosen as branch options. The mass and energy conservation equations are solved at the internal fluid nodes and the momentum equations are solved at the branches. It may be noted that this concept is similar to the staggered grid concept of the SIMPLE algorithm [4]. The walls, through which heat is transferred from hot fluid to cold fluid, are discretized both axially and radially. Solid to fluid conductors connect solid and fluid nodes and calculate the convective heat transfer rate, and solid to solid conductors connect solid nodes and calculate conduction heat transfer. The energy conservation of a solid is solved at the solid nodes accounting for heat transfer with neighboring solid and fluid nodes.

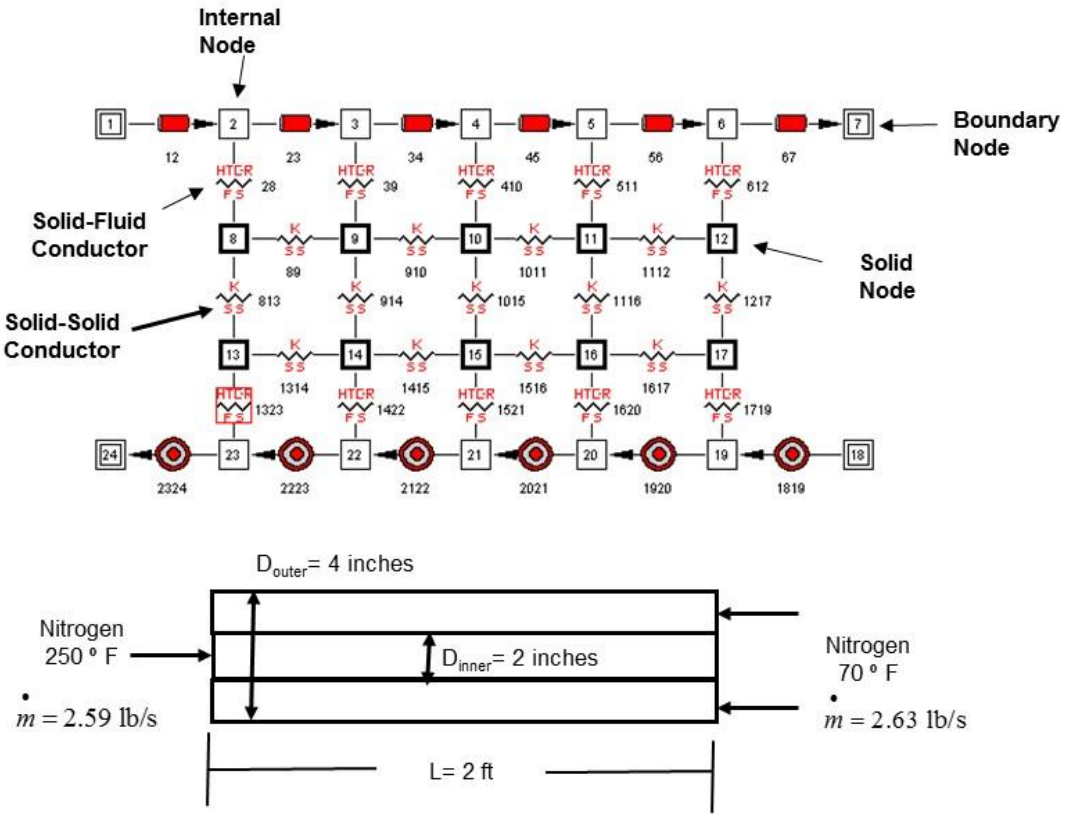


Figure 2. Flow network representing a counter-flow heat exchanger

2.2 Data Structure

In a flow network, the layout of the nodes cannot be represented by a structured co-ordinate system (Figure 1). There is no origin and no preferred co-ordinate direction to build the network of nodes and branches. In a structured co-ordinate system, the array of nodes can be constructed in the pre-specified co-ordinate direction. In 1-D flow network, each node has two neighbors; in 2-D flow network, each node has four neighbors and in 3-D flow network, each node has six neighbors. In a typical flow network, a node can have ñnò number of neighbors. Therefore, a u to define an unstructured flow network.

Any flow network can be constructed with three elements: 1) Boundary Node, 2) Internal Node and 3) Branch. Each element has properties. Internal nodes and branches, where the conservation equations are solved, have two kinds of properties: Geometric and Thermo-fluid. There are two types of geometric properties: relational and quantitative. The data structure of the flow network is shown in Figure 3. The relational geometric property allows nodes and branches to know their neighbors. Thermo-fluid properties include pressure, temperature, enthalpy, density, viscosity, etc.

Figure 3. Data Structure for Network Flow Analysis

Each node is designated by an arbitrary number and assigned a pointer to the array where node numbers are stored. The pointers are necessary to access the thermodynamic and thermo-physical properties of the node. The relational properties of the node include the number of branches connected to it and the names of those branches. Figure 4 shows an example of these two relational properties of a node in a given network.

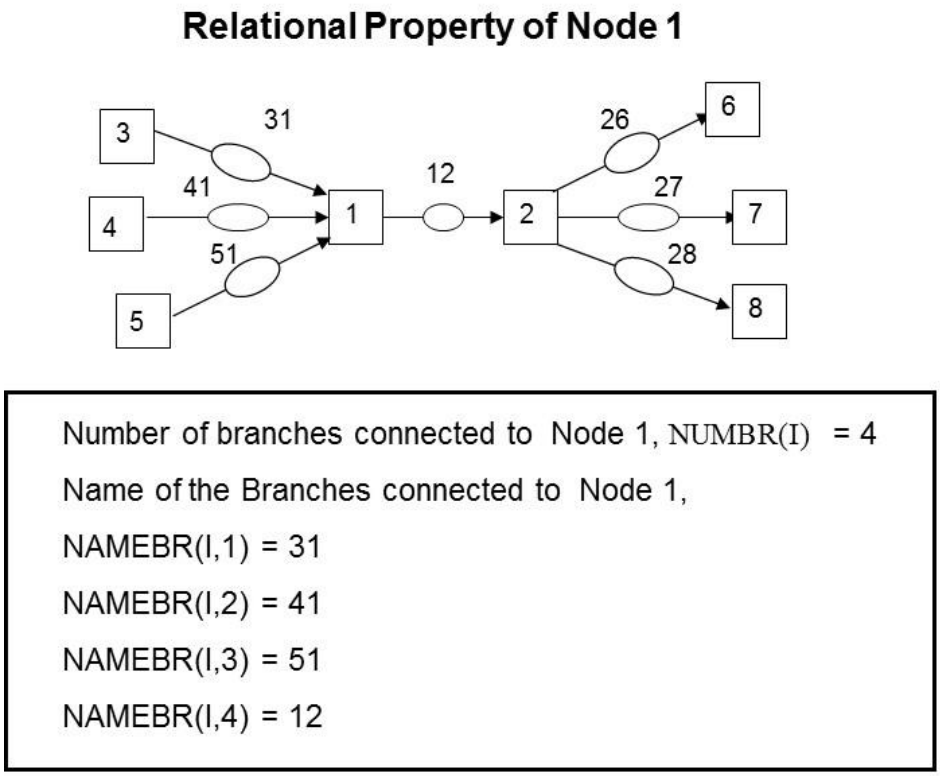


Figure 4. Example of Relational Property of a Node

Like the nodes, each branch is also designated by an arbitrary number and assigned a pointer to the array where branch numbers are stored. The relational properties of the branch include a) the names of the upstream and downstream nodes, b) the number of upstream and downstream branches, and c) the names of the upstream and downstream branches. Figure 5 shows an example of relational properties of a branch in a given network.

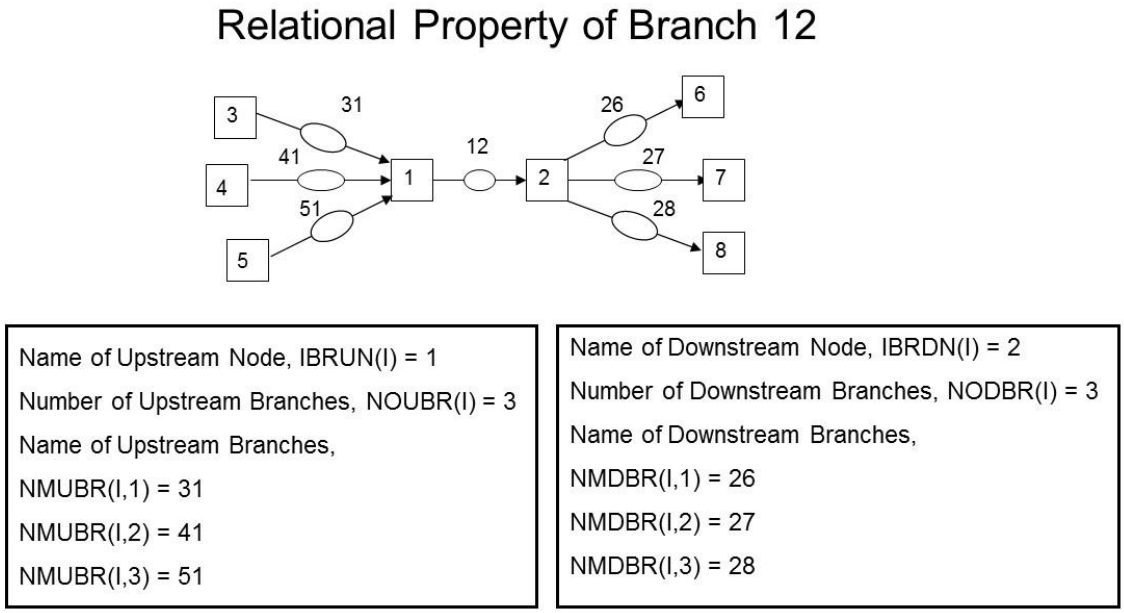


Figure 5. Example of Relational Property of a Branch

2.3 Governing Equations

The flow is assumed to be Newtonian, non-reacting and compressible. It can be steady or unsteady, laminar or turbulent, with or without heat transfer, phase change, mixing or rotation. Figure 6 displays a schematic showing adjacent nodes, their connecting branches, and the indexing system. In order to solve for the unknown variables, mass, energy and fluid species, conservation equations are written for each internal node and flow rate equations are written for each branch.

2.3.1 Mass Conservation Equation

The following is the mass conservation equation:

$$\frac{m_{t+\Delta t} - m_t}{\Delta t} = -\sum_{j=1}^{j=n} \dot{m}_{ij} \qquad (1)$$

Equation 1 requires that for the unsteady formulation, the net mass flow from a given node must equate to the rate of change of mass in the control volume. In the steady state formulation, the left side of the equation is zero. This implies that the total mass flow rate into a node is equal to the total mass flow rate out of the node.

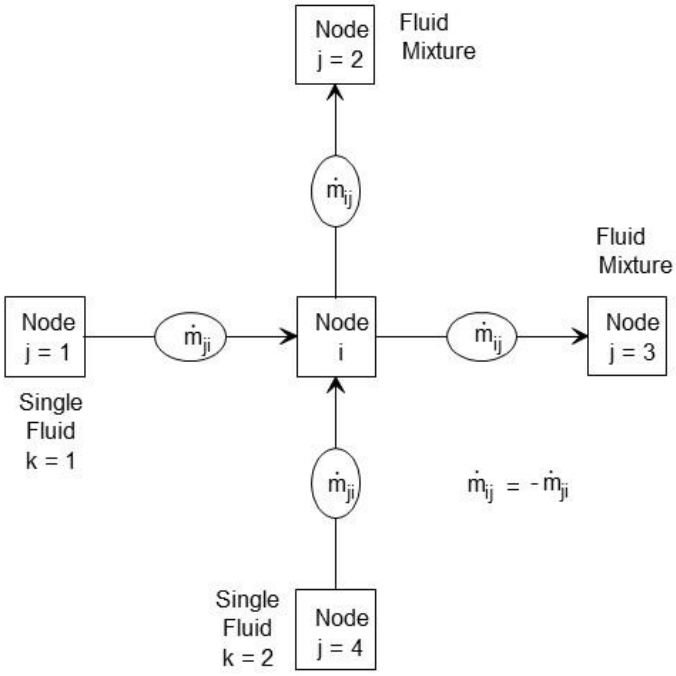


Figure 6. Schematic of Nodes, Branches and Indexing Practice

2.3.2 Momentum Conservation Equation

The flow rate in a branch is calculated from the momentum conservation equation (Equation 2) which represents the balance of fluid forces acting on a given branch. A typical branch configuration is shown in Figure 6. Inertia, pressure, gravity, friction and centrifugal forces are considered in the conservation equation. In addition to these five forces, a source term S has been provided in the equation to input pump characteristics or to input power to a pump in a given branch. If a pump is located in a given branch, all other forces except pressure are zero. The source term, S, is zero in all branches without a pump or other external momentum source.

$$\frac{(mu)_{t+\Delta t} - (mu)}{g_c \Delta t} + MAX\left|\dot{m}_{ij}, 0\right|(u_{ij} - u_u) - MAX\left|\dot{m}_{ij}, 0\right|(u_{ij} - u_u) =$$

-- -Unsteady --- ---------- Longitudinal Inertia ----------

$$(p_i - p_j)A_{ij} + \frac{\rho g V \cos\theta}{g_c} - K_f \dot{m}_{ij}\left|\dot{m}_{ij}\right| A_{ij} - \frac{K\rho r_{rot}^2 \omega^2 A}{g_c} - \rho A_{norm} u_{norm} u_{ij} / g_c + S \qquad (2)$$

--Pressure -- Gravity -- Friction -- Centrifugal -- Moving Boundary -- Source

Unsteady

This term represents the rate of change of momentum with time. For steady state flow, the time step is set to an arbitrary large value and this term reduces to zero.

Longitudinal Inertia

This term is important for compressible flows when there is a significant change in velocity in the longitudinal direction due to change in area and/or density. An upwind differencing scheme is used to compute the velocity differential.

Pressure

This term represents the pressure gradient in the branch. The pressures are located at the upstream and downstream face of a branch.

Gravity

This term represents the effect of gravity. The gravity vector makes an angle (θ) with the assumed flow direction vector. At $\theta = 180^{\circ}$ the fluid is flowing against gravity; at $\theta = 90^{\circ}$ the fluid is flowing horizontally, and gravity has no effect on the flow.

Friction

This term represents the frictional effect. Friction is modeled as a product of K_f, the square of the flow rate, and the area. K_f is a function of the fluid density in the branch and the nature of the flow passage being modeled by the branch. The calculation of K_f for different types of flow passages is described in a later section.

Centrifugal

This term in the momentum equation represents the effect of the centrifugal force. This term will be present only when the branch is rotating as shown in Figure 7. K_{rot} is the factor representing the fluid rotation. K_{rot} is unity when the fluid and the surrounding solid surface rotate with the same speed. This term also requires knowledge of the distances from the axis of rotation between the upstream and downstream faces of the branch.

Moving Boundary

This term represents the force exerted on the control volume by a moving boundary. This term is not active for multi-dimensional calculations.

Source

This term represents a generic source term. Any additional force acting on the control volume can be modeled through the source term. In a system level model, a pump can be modeled by this term. A detailed description of modeling a pump by this source term, S, appears in a later section.

In a system level thermo-fluid model, compressible flow through an orifice is often an option for a branch. Under that circumstance, instead of solving equation 2, a simplified form of momentum equation is solved to calculate flowrate through an orifice. If the ratio of downstream to upstream pressure is less than the critical pressure ratio:

$$\frac{p_j}{p_i} < p_{cr}, \qquad (3a)$$

where:

$$p_{cr} = \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}}, \qquad (3b)$$

then the choked flow rate in the branch is calculated from:

$$\dot{m}_{ij} = C_{L_{ij}} A \sqrt{p_i \rho_i g_c \frac{2\gamma}{\gamma - 1} \left(p_{cr}\right)^{2/\gamma} \left[1 - \left(p_{cr}\right)^{(\gamma-1)/\gamma}\right]} \qquad (3c)$$

If $\frac{p_j}{p_i} > p_{cr}$ the un-choked flow rate in the branch is calculated from:

$$\dot{m}_{ij} = C_{L_{ij}} A \sqrt{p_i \rho_i g_c \frac{2\gamma}{\gamma - 1} \left(\frac{p_j}{p_i}\right)^{2/\gamma} \left[1 - \left(\frac{p_j}{p_i}\right)^{(\gamma-1)/\gamma}\right]} \qquad (3d)$$

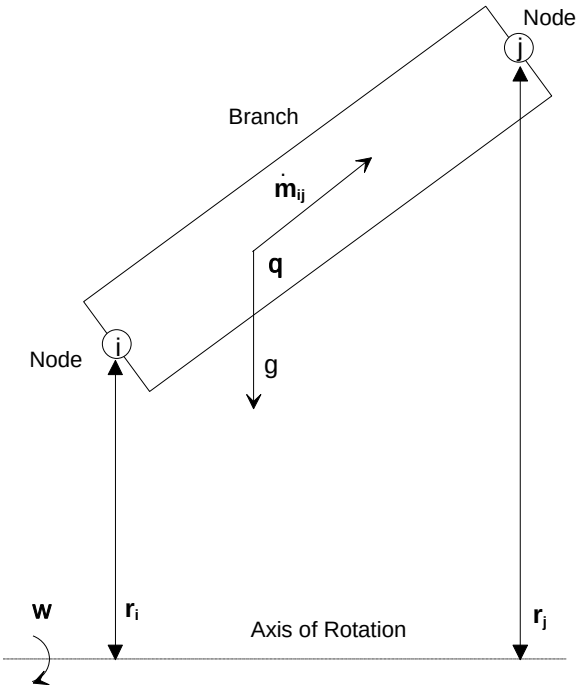


Figure 7 - Schematic of a Branch Showing Gravity and Rotation

2.3.3 Energy Conservation Equations for fluid and solid

Energy Conservation Equation of Fluid

The main purpose of the energy conservation equation in fluid flow calculations is to obtain fluid properties which are primarily functions of pressure and temperature. While pressures are calculated from the mass conservation equation, to obtain temperatures and other properties, the energy equation must be solved. The energy conservation equation can be expressed in terms of enthalpy or entropy. Once pressure and enthalpy or pressure and entropy are known, all thermodynamic and

thermo-physical properties can be evaluated by using the available computer programs [12-14] that calculate properties of common fluids.

The energy conservation equation in terms of enthalpy for node *i*, shown in Figure 6, can be expressed as:

$$\frac{m\left(h-\frac{p}{\rho}\right)_{t+\Delta t}-m\left(h-\frac{p}{\rho}\right)_{t}}{\Delta t}=$$

$$\sum_{j=1}^{j=n}\left[MAX\left[-\dot{m}_{ij},0\right]h_j-MAX\left[\dot{m}_{ij},0\right]h_i\right]+\frac{MAX\left[-\dot{m}_{ij},0\right]}{\left|\dot{m}_{ij}\right|}\left[\left(p_i-p_j\right)+K_{ij}\dot{m}_{ij}^{2}\left(\eta_j A\right)\right]+Q_i \tag{4}$$

Equation 4 shows that for transient flow, the rate of increase of internal energy in the control volume is equal to the rate of energy transport into the control volume minus the rate of energy transport from the control volume plus the rate of work done on the fluid by the pressure force plus the rate of work done on the fluid by the viscous force plus the rate of heat transfer into the control volume. The term $\left(p_i-p_j\right)\eta_j A_{ij}$ represents work input to the fluid due to rotation or having a pump in the upstream branch of the node *i*. The term $K_{ij}\dot{m}_{ij}^{2}\eta_j A_{ij}$ represents viscous work in the upstream branch of the node *i* where η_j and A_{ij} are velocity and area of the upstream branch.

The energy conservation equation based on entropy is shown in Equation 5.

$$\frac{(ms)_{t+\Delta t}-(ms)_t}{\Delta t}=\sum_{j=1}^{j=n}\left[MAX\left[-\dot{m}_{ij},0\right]s_j-MAX\left[\dot{m}_{ij},0\right]s_i\right]+\sum_{j=1}^{j=n}\frac{MAX\left[-\dot{m}_{ij},0\right]}{\left|\dot{m}_{ij}\right|}\dot{S}_{ij,gen}+\frac{Q_i}{T_i} \tag{5}$$

The entropy generation rate due to fluid friction in a branch is expressed as

$$\dot{S}_{ij,gen}=\frac{\dot{m}_{ij}\Delta p_{ij,viscous}}{\rho_u T_u J}=\frac{K_f\left(\dot{m}_{ij}\right)^3}{\rho_u T_u J} \tag{5a}$$

The first term in the right hand side of the equation represents the convective transport of entropy from neighboring nodes. The second term represents the rate of entropy generation in branches connected to the ith node. The third term represents entropy change due to heat transfer.

Energy Conservation Equation of Solid

Typically a solid node can be connected with other solid nodes, fluid nodes, and ambient nodes. Figure 8 shows a typical arrangement where a solid node is connected with other solid nodes, fluid nodes, and ambient nodes. The energy conservation equation for a solid node i can be expressed as:

$$\frac{\partial}{\partial t}\left(mC_p T_s^i\right) = \sum_{j_s=1}^{n_{ss}} \dot{q}_{ss} + \sum_{j_f=1}^{n_{sf}} \dot{q}_{sf} + \sum_{j_a=1}^{n_{sa}} \dot{q}_{sa} + \dot{S}_i \qquad (6)$$

The left hand side of the equation represents the rate of change of temperature of the solid node, i. The right hand side of the equation represents the heat transfer from the neighboring node and heat source or sink. The heat transfer from neighboring solid, fluid and ambient nodes can be expressed as

$$\dot{q}_{ss} = k_{ij_s} A_{ij_s} / d_{j_s} \left(T_s^{j_s} - T_s^i\right) \qquad (6a)$$

$$\dot{q}_{sf} = h_{ij_f} A_{ij_f} \left(T_f^{j_f} - T_s^i\right) \qquad (6b)$$

$$\dot{q}_{sa} = h_{ij_a} A_{ij_a} \left(T_a^{j_a} - T_s^i\right) \qquad (6c)$$

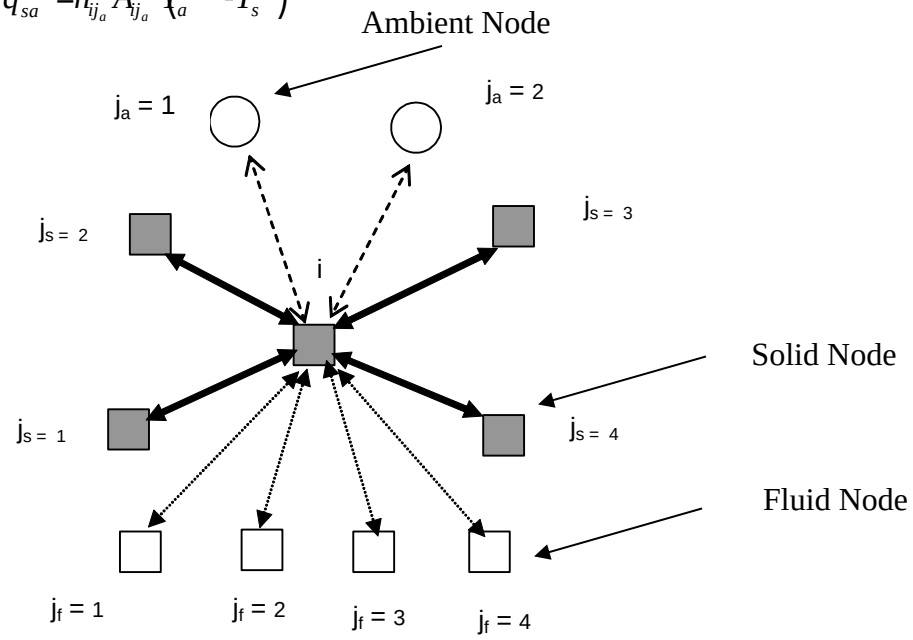


Figure 8. A schematic showing the connection of a solid node with neighboring solid, fluid and ambient nodes

The effective heat transfer coefficients for solid to fluid and solid to ambient nodes are expressed as the sum of the convection and radiation:

$$
\begin{aligned}
h_{ij_f} &= h_{c,ij_f} + h_{r,ij_f} \\
h_{ij_a} &= h_{c,ij_a} + h_{r,ij_a} \\
h_{r,ij_f} &= \frac{s\left(T_f^{j_f}\right) + \left(T_s^{i}\right)\left[T_f^{j_f} + T_s^{i}\right]}{1/e_{ij,f} + 1/e_{ij,s} - 1} \\
h_{r,ij_a} &= \frac{s\left[\left(T_a^{j_a}\right) + \left(T_s^{i}\right)\right]\left[T_a^{j_a} + T_s^{i}\right]}{1/e_{ij,a} + 1/e_{ij,s} - 1}
\end{aligned}
\qquad \text{(6d)}
$$

2.3.4 Equation of State and Thermodynamic Properties

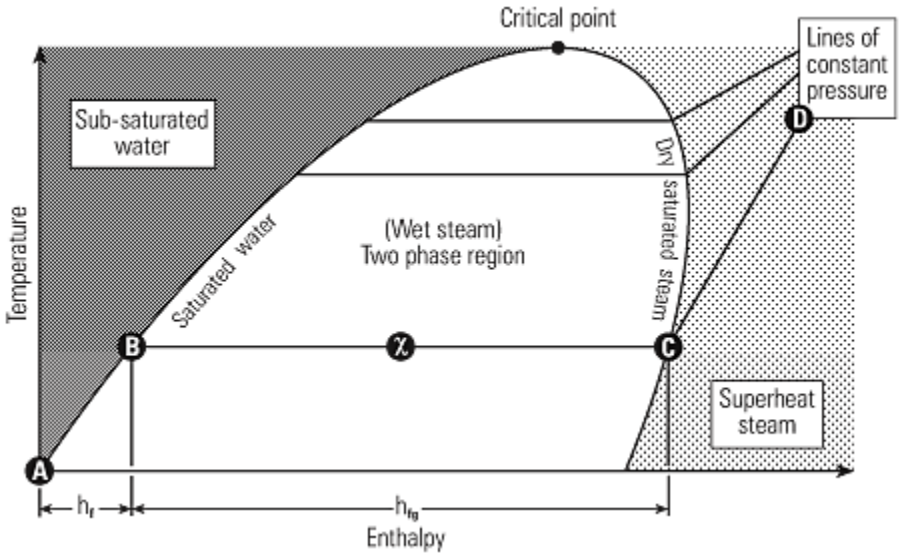


Figure 9. Thermodynamic state of a real fluid

The conservation equations for mass, momentum and energy contain thermodynamic and thermo-physical properties of a real fluid. A real fluid can exist in different states as shown in Figure 9: subcooled liquid (A), saturated liquid (B), a mixture of liquid and vapor (x), saturated vapor (C), and superheated vapor (D). The state of the real fluid in a given node is calculated from its pressure and enthalpy using a thermodynamic property program such as GASP [12] or GASPAK [14]. All these programs use accurate equations of state for thermodynamic properties and correlations for thermo-physical properties for common fluids.

One of the main objectives of using an accurate equation of state is to compute the compressibility factor z, which is used in the equation of state to compute the resident mass of the node:

$$m = \frac{pV}{RTz} \tag{7}$$

2.3.5 Species Conservation Equation

For a fluid mixture, thermodynamic and thermo-physical properties are also a function of the mass fraction of the fluid species. In order to calculate the properties of the mixture, the concentration of the individual fluid species within the branch must be determined. The concentration for the k[th] species can be written as

$$\frac{(m_i c_{i,k})_{t+\Delta t} - (m_i c_{i,k})_t}{\Delta t} = \sum_{j=1}^{j=n} \left[MAX(-\dot{m}_{ij}, 0) c_{j,k} - MAX(\dot{m}_{ij}, 0) c_{i,k} \right] + S_{i,k} \tag{8}$$

For transient flow, Equation 8 states that the rate of increase of the concentration of the k[th] species in the control volume equals the rate of transport of the k[th] species into the control volume minus the rate of transport of the k[th] species out of the control volume plus the generation rate of the k[th] species in the control volume.

2.3.6 Mixture Properties

A homogeneous mixture of multiple species in a given network can also be modeled provided the properties of the mixture are computed from the properties of the component species.

Temperature

In the absence of phase change, the temperature of the node can be calculated from a modified energy equation which is expressed in terms of specific heat and temperature.

$$(T_i)_{t+\Delta t} = \frac{\sum_{j=1}^{j=n} \sum_{k=f}^{k=n_f} Cp_{k,j} x_{k,j} T_j MAX(-\dot{m}_{ij}, 0) + (C_{p,i} m_i T_i)_t / \Delta t + Q_i}{\sum_{j=1}^{j=n} \sum_{k=f}^{k=n_f} Cp_{k,j} x_{k,j} MAX(\dot{m}_{ij}, 0) + (C_{p,i} m)_{t+\Delta t} / \Delta t} \tag{9}$$

Density

For Amagatôs model of partial volume, mixtur

$$\frac{1}{\rho_{mix}} = \sum_k \frac{x_k}{\rho_k} \tag{10}$$

ρ_k is evaluated at node pressure, p_i.

For Dalton of partial pressures, mixture density is expressed as:

$$\rho_{mix} = \sum_k \rho_k \qquad (11)$$

ρ_k is evaluated at partial pressure, p_k. which is product of molar concentration and node pressure, p_i.

Compressibility Factor

The compressibility factor of the mixture, z_i is expressed as

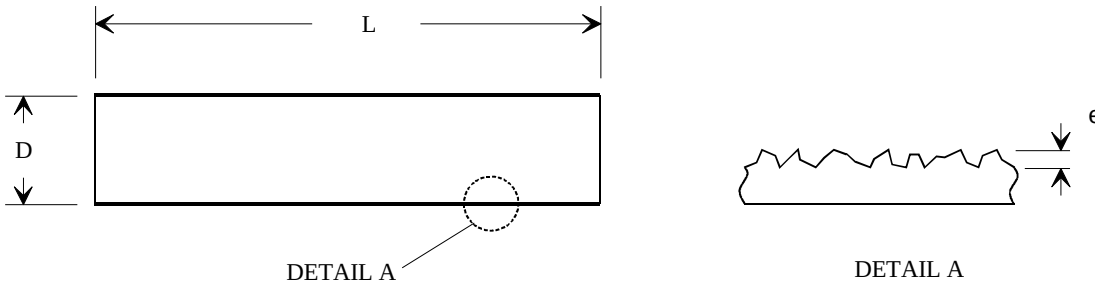
$$z_i = \sum_{k=1}^{k=n} x_k z_k \qquad (12)$$

where

$$z_k = \frac{p_i}{\rho_k R_k T_i} \qquad (12a)$$

2.3.7 Friction Calculation

It was mentioned earlier that the friction term in the momentum equation is expressed as a product of K_f, the square of the flow rate, and the flow area. Empirical information is necessary to estimate K_f . For pipe flow (Figure 10), length, *L*, diameter, *D*, and surface roughness, *e* are needed to compute friction.



Where:
D = Pipe Diameter
L = Pipe Length
e = Absolute Roughness

Figure 10 - Pipe parameters to compute friction

K_f can be expressed as:

$$K_f = \frac{8fL}{\rho_u \pi^2 D^5 g_c} \qquad (13)$$

The Darcy friction factor, f, is determined from the Colebrook Equation [8] which is expressed as:

tank was filled either with LH2 or LN2. At time zero, the valve at the left end of the pipe was opened, allowing liquid from the tank to flow into the ambient pipeline driven by tank pressure.

Fig. 16 Network flow model of the fluid system consisting of a tank, pipeline, and valve constructed with boundary nodes, internal nodes, and branches.

Figure 16 shows a schematic of the network flow model that was constructed to simulate the cooling of the transfer line. The tube was discretized into 33 fluid nodes (two boundary nodes and 31 internal nodes), 31 solid nodes, and 32 branch nodes. The upstream boundary node represents the cryogenic tank, while the downstream boundary node represents the ambient where the fluid is discharged. The first branch represents the valve; the next 30 branches represent the transfer line. Each internal node was connected to a solid node (nodes 34 through 64) by a solid to fluid conductor. The heat transfer in the wall is modeled using the lumped parameter method, assuming the wall radial temperature gradient is small. The heat transfer coefficient of the energy equation for the

Table 2. Chilldown time for various driving pressures and temperatures for LH_2 and LN_2

Fluid	Driving Pressure (MPa)	Inlet State	Inlet Temperature (K)	Experimental Chilldown Time (Sec)	Predicted Chilldown Time (Sec)
LH_2	0.52	Saturated	27	68	70
LH_2	0.60	Saturated	28.11	62	69
LH_2	0.77	Saturated	29.6	42	50
LH_2	1.12	Saturated	31.97	30	33
LH_2	0.25	Saturated	19.5	148	150
LH_2	0.43	Subcooled	19.5	75	80
LH_2	0.60	Subcooled	19.5	62	60
LH_2	0.77	Subcooled	19.5	41	45
LH_2	0.94	Subcooled	19.5	32	35
LH_2	1.12	Subcooled	19.5	28	30
LN_2	0.43	Saturated	91.98	165	185
LN_2	0.52	Saturated	94.42	150	160
LN_2	0.60	Saturated	96.35	130	140
LN_2	0.25	Subcooled	76.00	222	250
LN_2	0.34	Subcooled	76.00	170	175
LN_2	0.43	Subcooled	76.00	129	140
LN_2	0.52	Subcooled	76.00	100	100
LN_2	0.60	Subcooled	76.00	85	90

Fig. 17 Downstream wall temperature (K) vs. time (s) for vertical upward LN2 chilldown runs

5. Launder,B.E., and Spalding, D. B..òThe Numerica
Flowsò,Computer Methods in, Vol. 3, 1974, pp
269-289
6. Majumdar, A. K., "Mathematical Modeling of Flows in Dividing and Combining Flow Manifold," Applied Mathematical Modeling, Volume 4, December 1980, pp. 424-431.
7. Datta, A. B. and Majumdar, A. K., "Flow Distribution in Parallel and Reverse Flow Manifolds," Int. J. Heat and Fluid Flow, Volume 2, No. 4, 1980.
8. Colebrook, C. F., òTurbulent Flow in Pi
Transition Between the Smooth and Rough P
London, vol. 11, pp. 133-156, 1938-1939.
9. Generalized Fluid System Simulation Program - Majumdar; Alok Kumar, Bailey; John W. ; Schallhorn; Paul Alan ; Steadman; Todd E. , United States Patent No. 6,748,349 June 8, 2004
10. Majumdar AK, LeClair AC, Moore R, Schallhorn PA. Generalized Fluid System Simulation Program, Version 6.0. NASA/TM ð 2013 ï217492; October 2013.
11. Majumdar, A. K., ñA Second Law Based Unst
Generalized Flow Simu-0834, 37th AIAA Aerospace
Sciences Meeting Conference and Exhibit, January 11-14, 1999, Reno, NV.
12. Hendricks, R. C., Baron, A. K., and Pell
Calculating the Thermodynamic and Transport Properties for Ten Fluids: Parahydrogen, Helium, Neon, Methane, Nitrogen, Carbon Monoxide, Oxygen, Fluorine, Argon, and C-7808, February, 1975.
13. Hendricks, R. C., Peller, A. K., òA Computer Code for Calculating W-7391, November, 1973.
14. Cryodata Inc., ñUser's Guide to GASPAK, V
15. Van Hooser, Katherine, Majumdar, Alok, Bailey, John, ñNumerical Pre
Transient Axial Thrust and Internal Flow
No. AIAA 99-2189, 35th AIAA/ASME/SAE/ASEE, Joint Propulsion Conference and Exhibit, June 21, 1999, Los Angeles, CA.
16. Schallhorn, Paul, Majumdar, Alok, Van Hooser, Katherine and Marsh, Matthew, ñFlow Simulation in Secondary Flow Passa
Paper No. AIAA 98-3684, 34th AIAA/ASME/SAE/ASEE, Joint Propulsion Conference and Exhibit, July 13-15, 1998, Cleveland, OH.
17. Bandyopadhyay, Alak and Majumdar, Alok, ò
with Friction and Heat Transfer using the Generalized Fluid System Simulation Program (GFSSP) ò Paper presented in Thermal Flui
Glenn Research Center, September 10-14, 2007
18. LeClair, Andre and Majumdar, Alok, ò Computatio
Propellant Loading of the Space Shuttle External Tank ò, Presented in AIAA Joint Propulsion Conference, Nashville, Tennessee, July, 2010
19. Majumdar, Alok, òTank Fill and Transfer Line Chilldown Analysis by GFSSPò, Paper presented in Thermal Fluid A
Space Research Center, July 29 ïAugust 2, 2013

Symbol	Description
A	Area (in^2)
C_L	Flow Coefficient
$c_{i,k}$	Mass Concentration of k^{th} Specie at i^{th} Node
c_p	Specific Heat (Btu/lb ^oF)
D	Diameter (in)
f	Darcy Friction Factor
g	Gravitational Acceleration (ft/ sec^2)
g_c	Conversion Constant (= 32.174 lb-ft/lb_f-sec^2)
h	Enthalpy (Btu/lb)
h_{ij}	Heat Transfer Coefficient (Btu/ft^2-sec- ^oR)
J	Mechanical Equivalent of Heat (778 ft-lb_f/Btu)
K_f	Flow Resistance Coefficient (lb_f-sec^2/(lb-ft)^2)
K	Non-dimensional Head Loss Factor
k	Thermal Conductivity (Btu/ft-sec- ^oR)
L	Length (in)
m	Resident Mass (lb)
	Mass Flow Rate (lb/sec)
p	Pressure (lb_f/ in^2)
Pr	Prandtl Number
Q,	Heat Source (Btu/sec)
Re	Reynolds Number (Re = UuD/ P)
R	Gas Constant (lb_f-ft/lb-R)
r	Radius (in)
S	Momentum Source (lb_f)
s	Entropy (Btu/lb-R)
T	Fluid Temperature (^{o} F)
T_s	Solid Temperature (^{o} F)
u	Velocity (ft/sec)
V	Volume (in^3)
x	Quality and Mass Fraction
z	Compressibility Factor

Greek

U	Density (lb/ft^3)
T	Angle Between Branch Flow Velocity Vector and Gravity Vector (deg),
Z	Angular Velocity (rad/sec)
H	Absolute Roughness (in)
H_{ij}	Emissivity
H/D	Relative Roughness
'h	Head Loss (ft)
P	Viscosity (lb/ft-sec)
	Kinematic Viscosity (ft^2/sec)