

Numerical Modeling of Conjugate Heat Transfer in Fluid Network

Alok Majumdar
NASA/Marshall Space Flight Center
Huntsville, Alabama

ABSTRACT

This paper describes a finite volume procedure of conjugate heat transfer for network flow analysis. The analysis domain is discretized into a system of fluid and solid nodes, flow branches and conductors of heat between solid to solid, solid to fluid and solid to ambient. The system of equations that are solved simultaneously to predict flow and thermal properties include fluid mass and energy conservation equations at the fluid node, momentum conservation equation in fluid branch and energy conservation equation at solid node. The governing equations for solid and fluid node are coupled through solid to fluid heat transfer which is expressed as a function of flow properties and temperature of solid nodes. The numerical algorithm has been incorporated into the Generalized Fluid System Simulation Program (GFSSP), a finite volume based network flow analysis code. The paper also presents verification of the numerical model by comparing the GFSSP solution of conduction-convection problem with analytical solution and demonstrates the application of the code in predicting the chilldown of a cryogenic transfer line.

INTRODUCTION

Fluid network modeling with conjugate heat transfer has many applications in aerospace engineering. In modeling unsteady flow with heat transfer, it is important to know the variation of wall temperature in time and space to calculate heat transfer between solid to fluid. Since wall temperature is a function of flow, a coupled analysis of solid and fluid temperature is necessary. In cryogenic applications, modeling of conjugate heat transfer is of great importance to correctly predict chill down of transfer lines and boil-off rate of propellant in cryogenic storage tank due to heat leakage. Development of accurate, robust and economic numerical model is a critical need for design and operation of such systems. This paper describes the progress we have made at Marshall Space Flight Center in recent years to develop this capability using a general-purpose flow network code, Generalized Fluid System Simulation Program (GFSSP) [1]. This capability will be demonstrated by modeling the chill down of a cryogenic transfer line.

The operation of a cryogenic propulsion system, such as those found in spacecraft and missiles, requires transfer line chilldown before establishing a steady flow of cryogenic fluid between various system components. It is necessary to know how long it takes to chilldown a given transfer line for satisfactory operation. When liquid cryogen, for example, hydrogen, at saturation temperature begins flowing through a tube initially at

ambient temperature, the liquid instantly vaporizes near the tube wall. Thus a cross section of the flow will have an outer vapor ring with a saturated liquid core. As the flow moves downstream, the liquid core evaporates, and the vapor becomes superheated. As the tube wall cools, the liquid core penetrates farther and farther downstream. Eventually, the tube becomes filled with liquid. Due to change in fluid density, the average velocities are significantly higher in the vapor region of the tube. Prediction of chill down time requires modeling of these transient phenomena and understanding of how they affect heat transfer from the tube wall to the flowing cryogen.

Cross et al [2] and Majumdar and Steadman [3] used earlier version of GFSSP to model chilldown of cryogenic transfer line. The earlier version of GFSSP did not have the capability of conjugate heat transfer. Therefore, solid nodes were modeled through user subroutines where conservation equations for solid node were solved at the beginning of each time step and the solid temperatures were used to calculate heat transfer to the fluid node. Although Cross et al successfully compared numerical solution with known analytical solution for a short tube; Majumdar & Steadman was not able to match the experimental data too well for long tube. It was recognized that the modeling of conjugate heat transfer with user subroutines lacks flexibility and efficiency and are more prone to coding error. Therefore, it was decided to extend code's capability to include solid nodes and develop connectivity between fluid and solid node such that conservation equations for both solid and fluid nodes are simultaneously solved.

MATHEMATICAL FORMULATION

The finite volume formulation requires governing equations to be expressed in conservative form. The rate of change of a conserved property in a given control volume is expressed as the vector sum of transported property from neighboring control volumes together with source or sink terms. The unknown variables in the flow circuit of figure 1 are pressure, temperatures of fluid and solid and flowrate. These variables are solved from the equations listed in Table 1.

Table 1. Mathematical Closure

<u>Unknown Variables</u>	<u>Equations to Solve</u>
1. Pressure	1. Mass Conservation Equation
2. Flowrate	2. Momentum Conservation Equation
3. Fluid Temperature	3. Energy Conservation Equation of fluid
4. Solid Temperature	4. Energy Conservation Equation of solid
5. Mass	5. Thermodynamic Equation of State

Figure 1 shows a typical network consisting of fluid, solid and ambient nodes. The next task is to develop a generalized form of each governing equation that would be representative of all possible thermo-fluid systems.

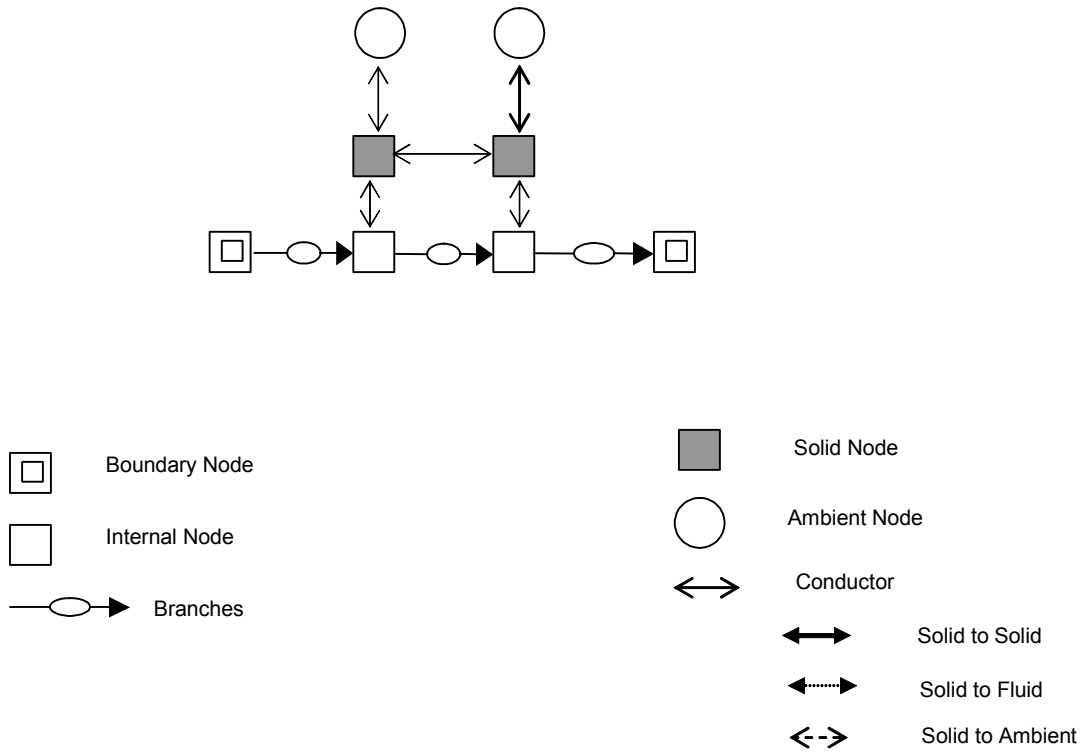


Figure 1. A network consisting of fluid node, solid node, flow branches and conductors

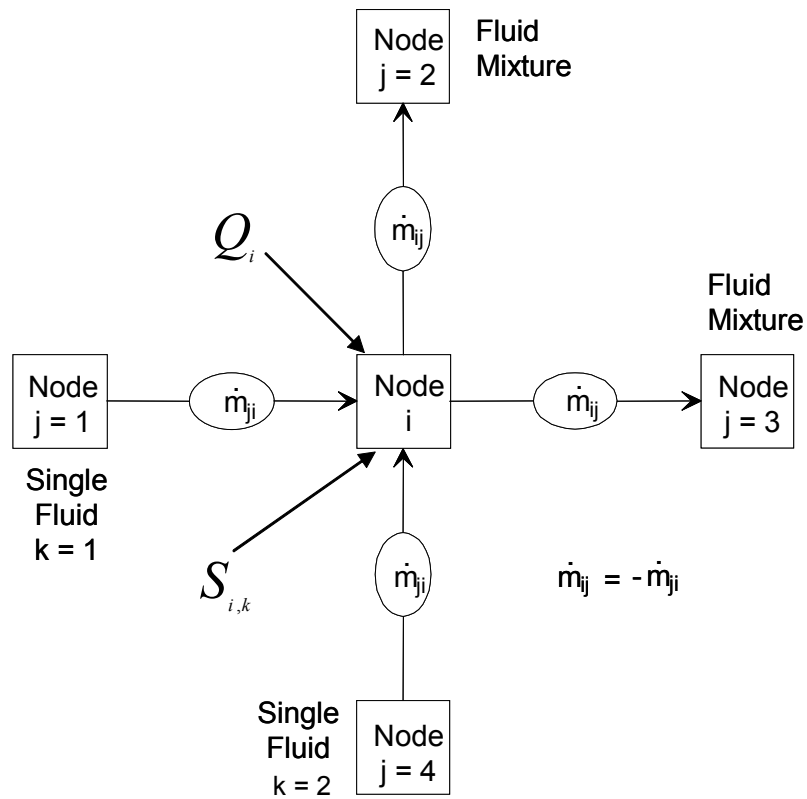


Figure 2. Schematic of connections between Nodes by Branches and the indexing practice

Mass Conservation

The mass conservation equation at the i^{th} node (Figure 2) can be written as

$$\frac{m_{\tau+\Delta\tau} - m_{\tau}}{\Delta\tau} = - \sum_{j=1}^{j=n} \dot{m}_j \quad (1)$$

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

Momentum Conservation

The momentum conservation equation at the ij branch can be written as

$$\begin{aligned} \frac{(mu)_{\tau+\Delta\tau} - (mu)_{\tau}}{g_c \Delta\tau} + MAX\left|\dot{m}_{ij}, 0\right| (u_{ij} - u_u) - MAX\left|-\dot{m}_{ij}, 0\right| (u_{ij} - u_u) = \\ (p_i - p_j)A_{ij} - K_f \dot{m}_{ij} \left|\dot{m}_{ij}\right| A_{ij} \end{aligned} \quad (2)$$

The left hand side of the momentum equation contains unsteady and inertia term. The pressure and friction force appear in the right hand side of the equation. The unsteady term represents rate of change of momentum with time. For steady state flow, time step is set to an arbitrary large value and this term is reduced to zero. The inertia term is important when there is a significant change in velocity in the longitudinal direction due to change in area and density. An upwind differencing scheme is used to compute the velocity differential. The pressure term represents the pressure gradient in the branch. The pressures are located at the upstream and downstream face of a branch. Friction was modeled as a product of K_f and the square of the flow rate and area. It may be noted that $\dot{m}_{ij} \left|\dot{m}_{ij}\right|$ has been used instead of \dot{m}_{ij}^2 . Recognizing the flowrate is a vector quantity; this technique is used to ensure that friction always opposes the flow. K_f is a function of the fluid density in the branch and the nature of the flow passage being modeled by the branch. For a pipe K_f can be expressed as

$$K_f = \frac{8fL}{\rho_u \pi^2 D^5 g_c} \quad (3)$$

For a valve, K_f can be expressed as

$$K_f = \frac{1}{2 g_c \rho_u C_L^2 A^2} \quad (4)$$

The friction factor, f , in equation (3) is calculated from Colebrook equation [4], which is expressed as

$$\frac{1}{\sqrt{f}} = -2 \log \left[\frac{\varepsilon}{3.7D} + \frac{2.51}{\text{Re} \sqrt{f}} \right] \quad (5)$$

Energy Conservation Equation of Fluid

The energy conservation equation for fluid node i, shown in Figure 2, can be expressed following the first law of thermodynamics and using enthalpy as the dependant variable. The energy conservation equation based on enthalpy can be written as

$$\frac{m \left(h - \frac{p}{\rho J} + \frac{1}{2} \frac{\rho u^2}{g_c J} \right)_{\tau+\Delta\tau} - m \left(h - \frac{p}{\rho J} + \frac{1}{2} \frac{\rho u^2}{g_c J} \right)_{\tau}}{\Delta\tau} \quad (6)$$

$$= \sum_{j=1}^{j=n} \left\{ \text{MAX} \left[-\dot{m}_{ij}, 0 \right] \left[h_j + \frac{1}{2} \frac{\rho_j u_j^2}{g_c J} \right] - \text{MAX} \left[\dot{m}_{ij}, 0 \right] \left[h_i + \frac{1}{2} \frac{\rho_i u_i^2}{g_c J} \right] \right\} + \dot{Q}_i ,$$

where

$$\dot{Q}_i = h_c A (T_{\text{solid}} - T_{\text{fluid}}) . \quad (6a)$$

The MAX operator represents the upwind formulation. \dot{Q}_i represents solid to fluid heat transfer that will be described in more detail in a following section.

Equation of State

The resident mass in the i^{th} control volume can be expressed from the equation of state for a real fluid as

$$m = \frac{pV}{RTz} \quad (7)$$

For a given pressure and enthalpy the temperature and compressibility factor in equation 6 is determined from the thermodynamic property program developed by Hendricks et al [5,6].

Energy Conservation Equation of Solid

Typically a solid node can be connected with other solid nodes, fluid nodes and ambient nodes. Figure 3 shows a typical arrangement where a solid node is connected with other solid nodes, fluid nodes and ambient node. The energy conservation equation for solid node i can be expressed as:

$$\frac{\partial}{\partial\tau} (m C_p T_s^i) = \sum_{j_s=1}^{n_{ss}} \dot{q}_{ss} + \sum_{j_f=1}^{n_{sf}} \dot{q}_{sf} + \sum_{j_a=1}^{n_{sa}} \dot{q}_{sa} + \dot{S}_i \quad (8)$$

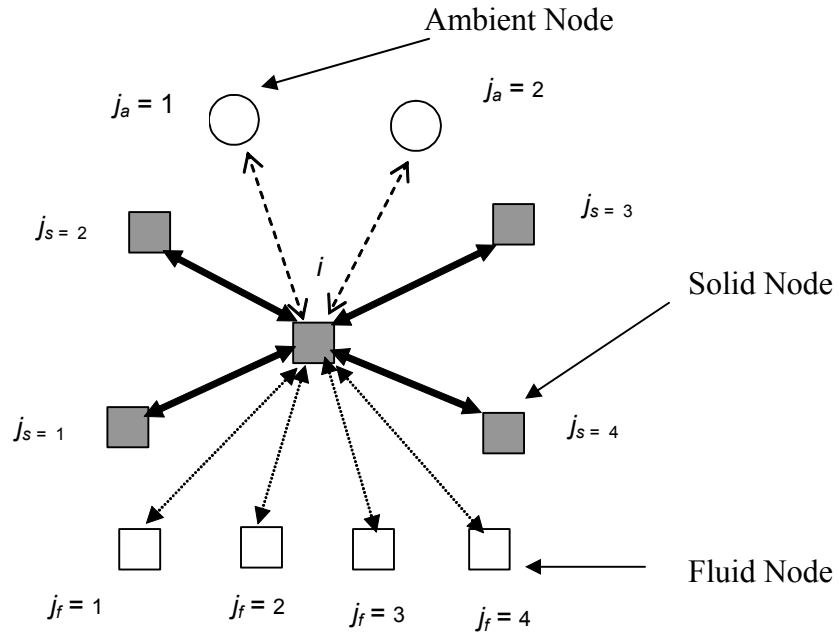


Figure 3. A schematic showing the connection of a solid node with neighboring solid, fluid and ambient nodes

The left hand side of the equation represents 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

$$\dot{q}_{ss} = k_{ij_s} A_{ij_s} / \delta_{ij_s} (T_s^{j_s} - T_s^i) \quad (9)$$

$$\dot{q}_{sf} = h_{ij_f} A_{ij_f} (T_f^{j_f} - T_s^i) \quad (10)$$

$$\dot{q}_{sa} = h_{ij_a} A_{ij_a} (T_a^{j_a} - T_s^i) \quad (11)$$

The heat transfer rate can be expressed as a product of conductance and temperature differential. The conductance for equations (9), (10) and (11) are:

$$C_{ij_s} = \frac{k_{ij_s} A_{ij_s}}{\delta_{ij_s}} ; C_{ij_f} = h_{ij_f} A_{ij_f} ; C_{ij_a} = h_{ij_a} A_{ij_a} \quad (12)$$

where effective heat transfer coefficients for solid to fluid and solid to ambient nodes are expressed as:

$$\begin{aligned} h_{ij_f} &= h_{c,ij_f} + h_{r,ij_f} \\ h_{ij_a} &= h_{c,ij_a} + h_{r,ij_a} \end{aligned} \quad (13)$$

$$\begin{aligned}
h_{r,ij_f} &= \frac{\sigma \left[(T_f^{j_f})^2 + (T_s^i)^2 \right] \left[T_f^{j_f} + T_s^i \right]}{1/\varepsilon_{ij_f} + 1/\varepsilon_{ij_s} - 1} \\
h_{r,ij_a} &= \frac{\sigma \left[(T_a^{j_a})^2 + (T_s^i)^2 \right] \left[T_a^{j_a} + T_s^i \right]}{1/\varepsilon_{ij_a} + 1/\varepsilon_{ij_s} - 1}
\end{aligned} \tag{14}$$

Equation 8 can be rearranged to determine T_s^i .

$$T_s^i = \frac{\sum_{j_s=1}^{n_{ss}} C_{ij_s} T_s^{j_s} + \sum_{j_f=1}^{n_{sf}} C_{ij_f} T_f^{j_f} + \sum_{j_a=1}^{n_{sa}} C_{ij_a} T_a^{j_a} + \frac{(mC_p)_m T_{s,m}^i}{\Delta\tau} + \dot{S}}{\frac{mC_p}{\Delta\tau} + \sum_{j_s=1}^{n_{ss}} C_{ij_s} + \sum_{j_f=1}^{n_{sf}} C_{ij_f} + \sum_{j_a=1}^{n_{sa}} C_{ij_a}} \tag{15}$$

SOLUTION PROCEDURE

The pressure, enthalpy, and resident mass in internal nodes and flowrate in branches are calculated by solving equations (1), (6), (7), and (2) respectively. The temperature of the solid node was calculated from equation (15). A combination of the Newton-Raphson method and the successive substitution method has been used to solve the set of equations. The mass conservation (2), momentum conservation (3) and resident mass (7) equations are solved by the Newton-Raphson method. The energy conservation equations for fluid and solid are solved by the successive substitution method. The temperature, density and viscosity are computed from pressure and enthalpy using a thermodynamic property program [5,6]. Figure 4 shows the flow diagram of the Simultaneous Adjustment with Successive Substitution (SASS) scheme. The iterative cycle is terminated when the normalized maximum correction Δ_{\max} is less than the convergence criterion C_c . Δ_{\max} is determined from

$$\Delta_{\max} = \max \left| \sum_{i=1}^{N_E} \frac{\phi'_i}{\phi_i} \right| \tag{16}$$

The convergence criterion is set to 0.001 for all models presented in this paper. The details of the numerical procedure are described in Reference 7.

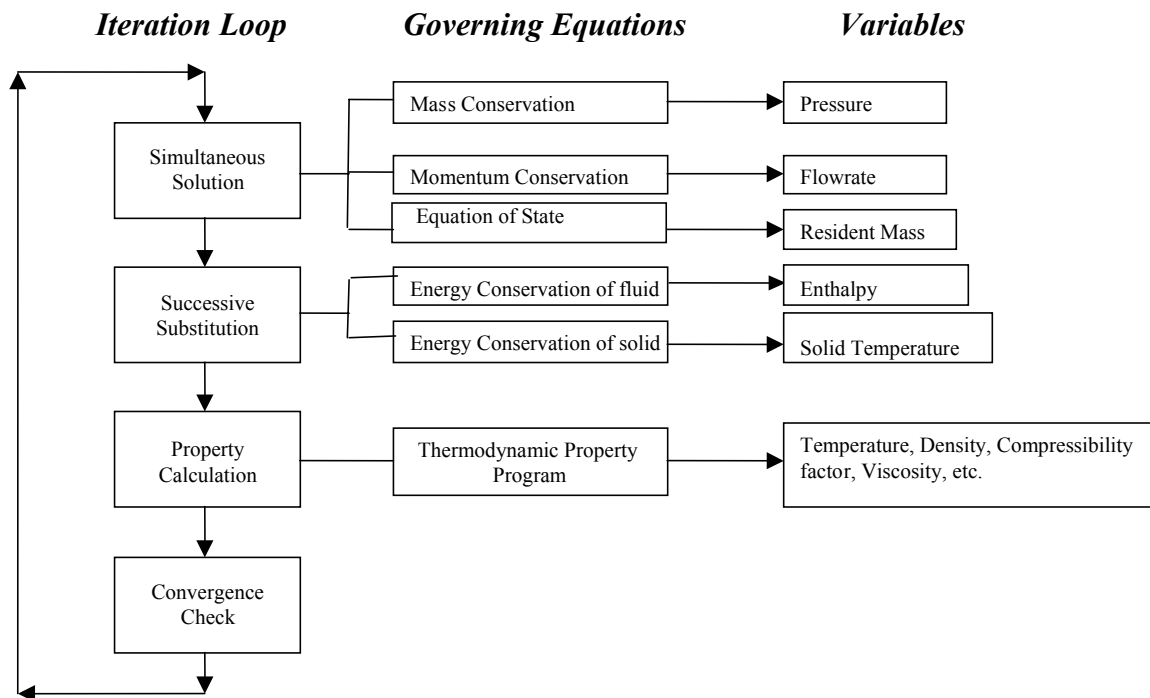


Figure 4. SASS (Simultaneous Adjustment with Successive Substitution) Scheme for solving Governing Equations

COMPUTER PROGRAM

GFSSP (Generalized Fluid System Simulation Program) embodies the mathematical formulation and solution procedure described in the previous sections. The program structure is shown in Figure 5. The program consists of three modules: Graphical User Interface, Solver and User Subroutines. VTASC (Visual Thermofluid dynamics Analyzer for Systems & Components) is the Graphical User Interface (GUI). VTASC allows user to create a flow circuit using a point and click paradigm. It creates an ASCII data file that is read by the solver module and reads the output data file for post processing the results. The solver module reads the data file generated by VTASC. It generates all governing equations from network data. The equations are solved by the iterative algorithm (SASS). It calls thermodynamic property programs to obtain the necessary properties during the iterative cycle.

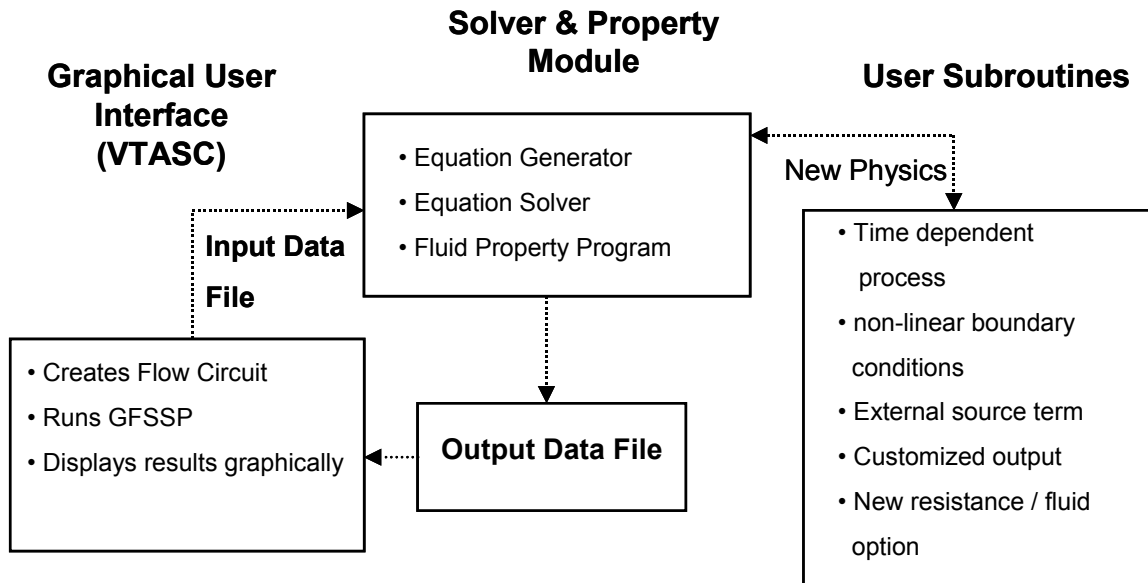


Figure 5. GFSSP Program Structure

RESULTS & DISCUSSION

The verification and validation of conjugate heat transfer capability in GFSSP was performed by comparing with known solution of a simple conduction-convection problem. The heat transfer in a homogenous circular rod between two walls was considered (Figure 6). The two walls are held at temperatures of 32 ° F and 212 ° F, respectively. The 0.167 ft diameter rod is 2 ft in length and is initially at a temperature of 70 ° F. The heat transfer coefficient between the rod and the ambient air is 1.14 Btu/ft²-hr-R and the thermal conductivity of the rod is 9.4 Btu/ft-hr-R. GFSSP's solution has been compared with closed form analytical solution in Figure 7. Two solutions matched well to confirm the accuracy of steady state formulation of GFSSP.

We have selected the chilldown of a short cryogenic transfer line as the validation of transient conjugate heat transfer problem. An aluminum tube of 26 inch length and 3/16 inch diameter was chilled by liquid hydrogen at -425 °F and 14.7 psia pressure. The tube was initially at 80 °F. The pressure at the outlet was set at 13.3 psia. The tube was discretized into 30 nodes and 29 branches as shown in Figure 8. Node 1 and 30 are boundary nodes where inlet and outlet conditions were specified. Flow temperatures and pressures were calculated at internal nodes 2 through 29. Each internal node was connected to a solid node (Node 31 through 58) by a solid to fluid conductor.

Figures 9 to 12 show comparison of predicted temperatures (solid and fluid), pressures and flowrates between Version 4 and Version 5 of GFSSP. Version 4 models conjugate heat transfer using user subroutines as described in Reference 2 & 3. Version 5 has the

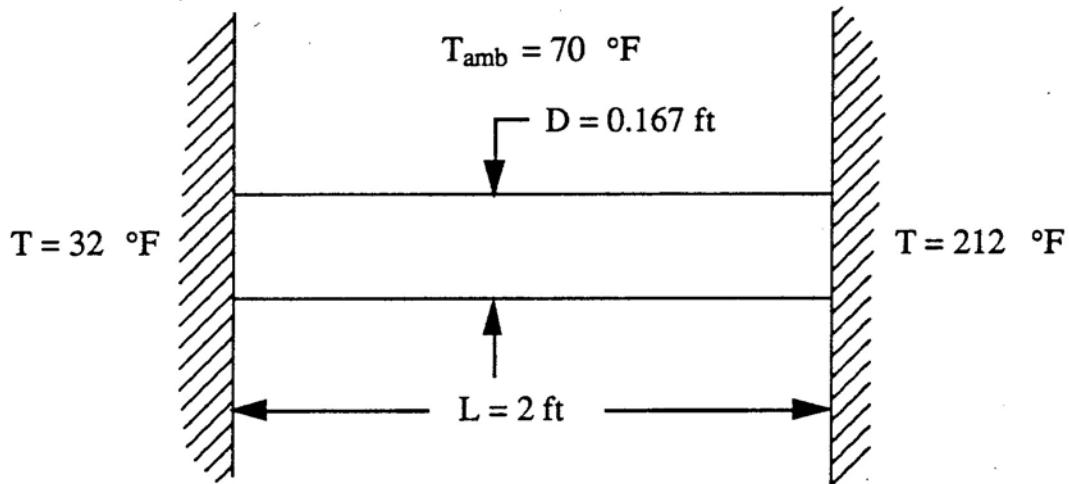


Figure 6. Schematic of circular rod connected to walls at different temperatures

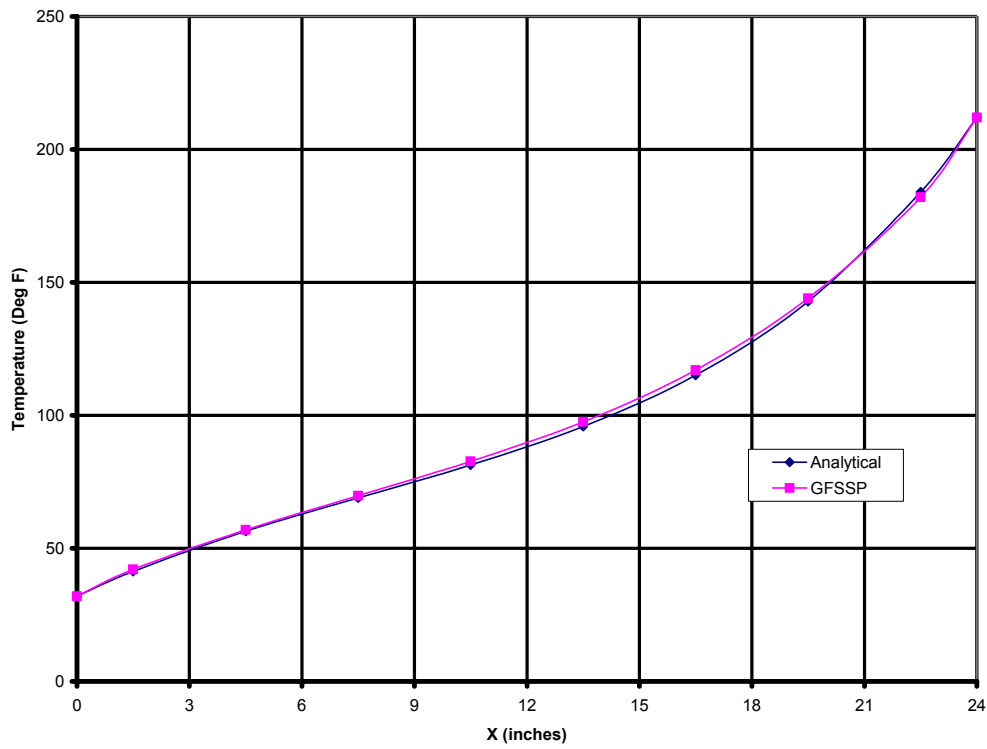


Figure 7. Comparison of GFSSP solution with analytical solution

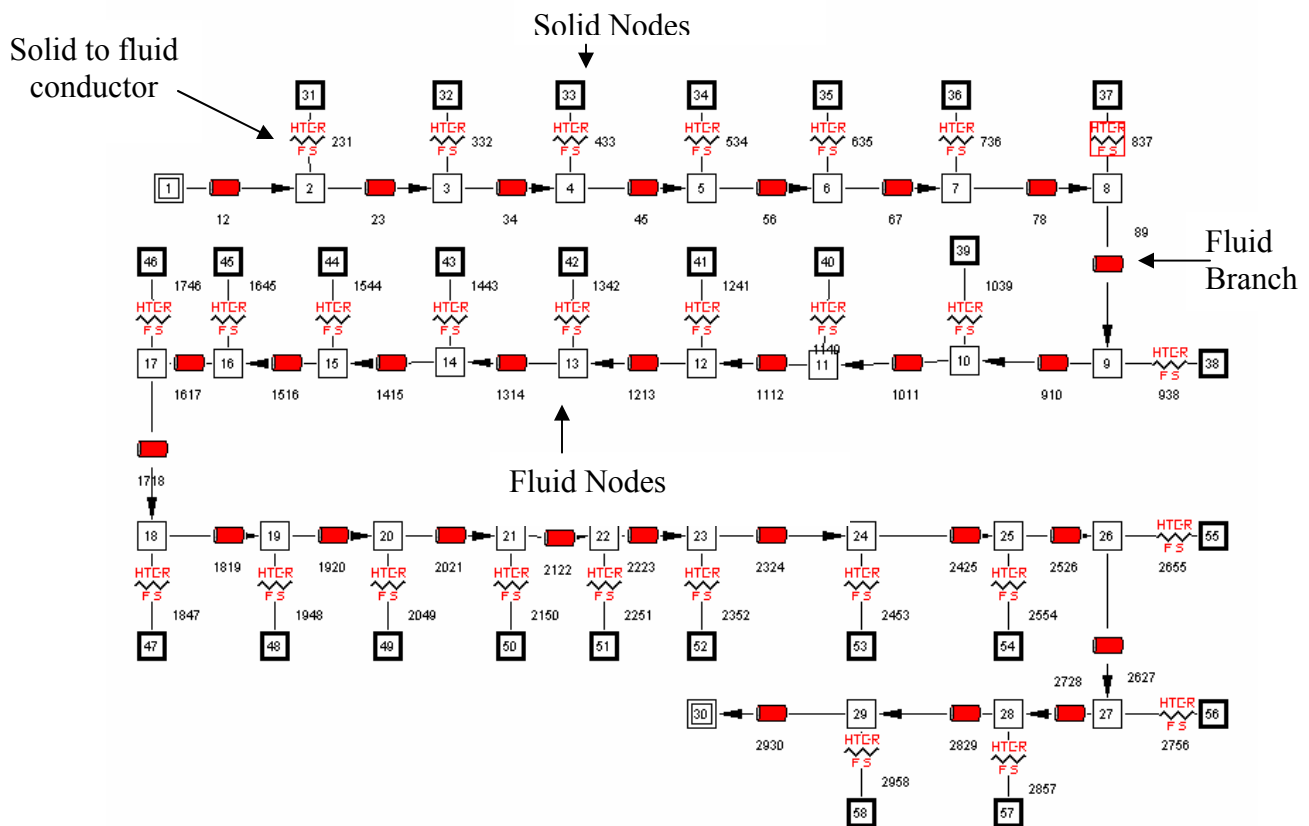
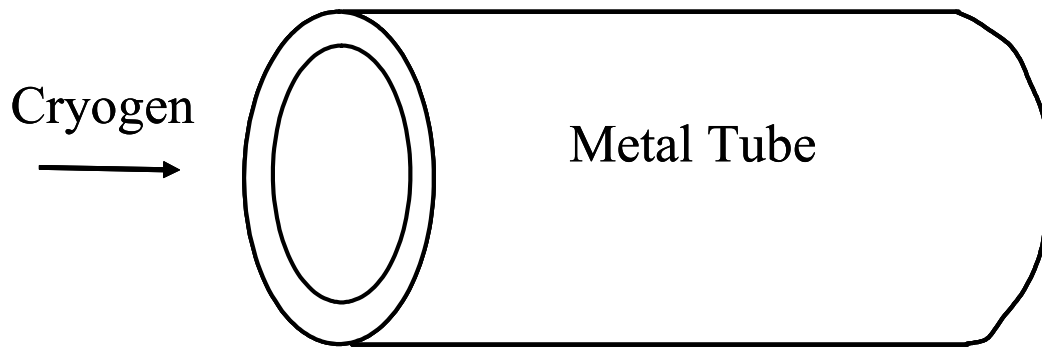


Figure 8. Schematic and GFSSP model of the chilldown problem

Tube Wall Temperature Comparison of GFSSP Versions 4 and 5

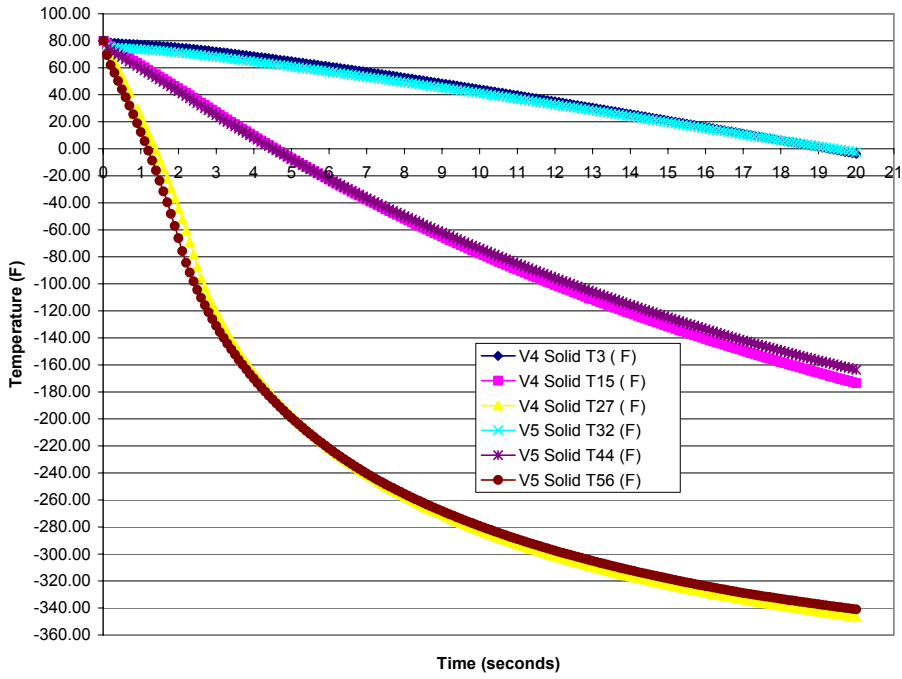


Figure 9. The predicted temperature history of tube wall at three axial locations

Fluid Temperature Comparison for GFSSP Versions 4 and 5

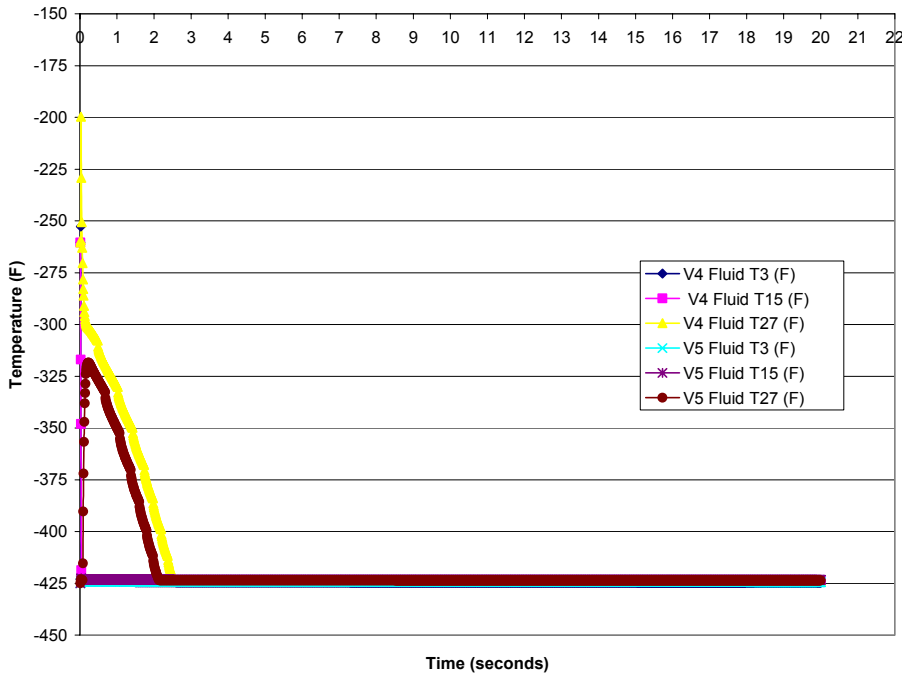


Figure 10. The predicted temperature history of fluid at three axial locations.

Pressure Comparison for GFSSP Versions 4 and 5

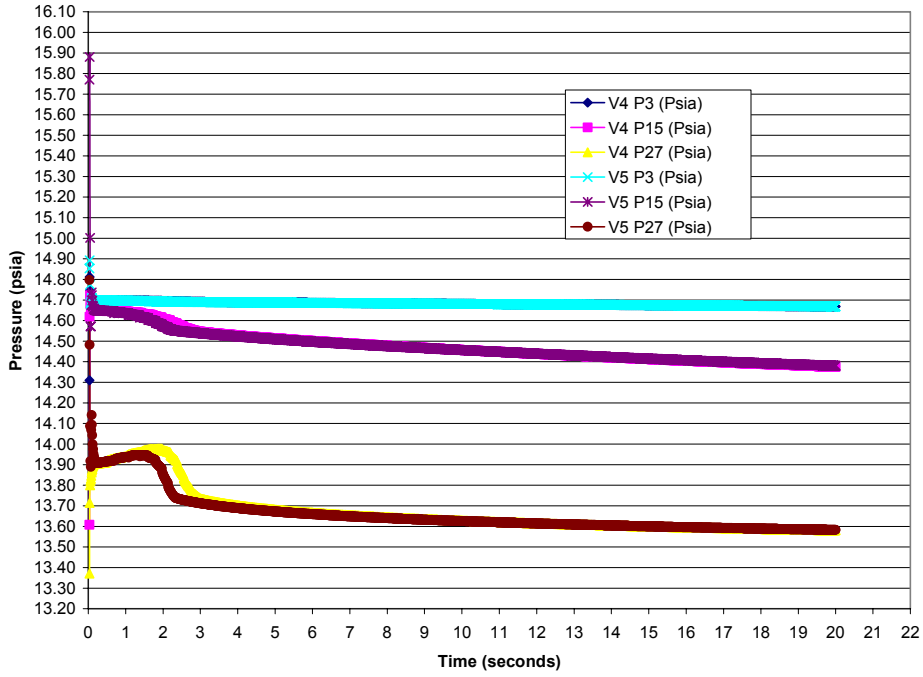


Figure 11. The predicted pressure history at three axial locations

Flow Rate Comparison for GFSSP Versions 4 and 5

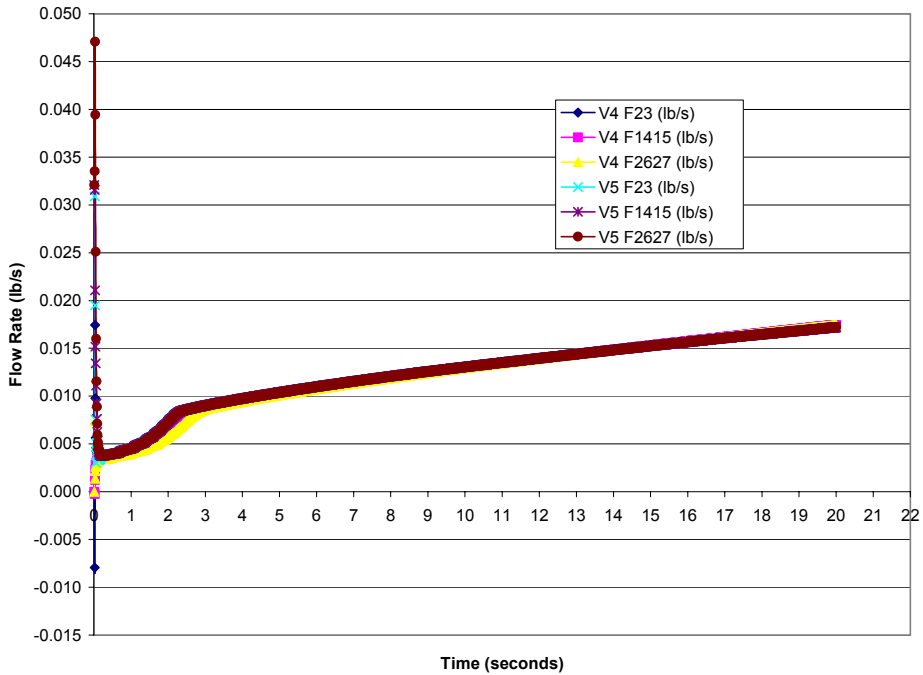


Figure 12. Predicted flowrate history at three axial locations

capability of modeling solid and fluid node as shown in Figure 8. Both versions use two phase heat transfer correlation of Miropolski [8]. All four plots exhibit good comparison between the predictions of two versions. The predictions of version 4 were verified with analytical data in reference 2. Therefore, the good comparison between version 4 and version 5 results validate the results of version 5.

CONCLUSIONS

GFSSP's capability has been extended to model conjugate heat transfer involving solid to fluid heat transfer. The new version offers more flexibility and ease of use than previous version which requires extensive programming and understanding of code's data structure to model conjugate heat transfer. The model results have been validated by comparing with analytical solution.

ACKNOWLEDGMENTS

The work has been performed at NASA/Marshall Space Flight Center under a Center Director's Discretionary Fund (CDDF) project. The author would like to acknowledge Mr. Ric Moore of UNITES contract for extending GFSSP's graphical user interface, VTASC and Mr. Johnny Maroney of Sverdrup Technology for engineering support.

REFERENCES

1. Majumdar, Alok, "Numerical Modeling of Unsteady Thermofluid Dynamics in Cryogenic System" 14th Annual Thermal & Fluids Analysis Workshop, August 18 – 22, 2003, Hampton, Virginia.
2. Cross, Matthew, Majumdar, Alok, Bennett, John & Malla, Ramesh, "Modeling of Chill Down in Cryogenic Transfer Lines", Journal of Spacecraft and Rockets, Vol. 39, No. 2, pp 284-289, 2002
3. Majumdar, Alok, and Steadman, Todd, "Numerical Modeling of Thermofluid Transients During Chillydown of Cryogenic Transfer Lines" 33rd International Conference on Environmental Systems (ICES), Paper No. 2003-01-2662, Vancouver, Canada, July 6-10, 2003.
4. Colebrook, C. F., "Turbulent Flow in Pipes, with Particular Reference to the Transition Between the Smooth and Rough Pipe Laws", J. Inst. Civil Engineering, London, vol. 11, pp. 133-156, 1938-1939.
5. Hendricks, R. C., Baron, A. K., and Peller, I. C., "GASP - A Computer Code for Calculating the Thermodynamic and Transport Properties for Ten Fluids: Parahydrogen, Helium, Neon, Methane, Nitrogen, Carbon Monoxide, Oxygen, Fluorine, Argon, and Carbon Dioxide", NASA TN D-7808, February, 1975.
6. Hendricks, R. C., Peller, I. C., and Baron, A. K., "WASP - A Flexible Fortran IV Computer Code for Calculating Water and Steam Properties", NASA TN D-7391, November, 1973.
7. Majumdar, Alok, "Generalized Fluid System Simulation Program (GFSSP) Version 3.0", Report No.: MG-99-290, Sverdrup Technology, Huntsville, Alabama, November, 1999.

8. Miropolskii, Z.L.: "Heat Transfer in Film Boiling of a Steam-Water Mixture in Steam Generating Tubes," *Teploenergetika*, Vol. 10, 1963, pp. 49–52; transl. AEC-tr-6252, 1964.

NOMENCLATURE

A	Area (in ²)
C	Conductance (Btu/s-°R)
C _L	Flow Coefficient
C _p	Specific Heat of Solid (Btu/lb-°F)
D	Diameter (in)
f	Friction Factor
g _c	Conversion Constant (= 32.174 lb-ft/lb _f -sec ²)
h	Enthalpy (Btu/lb); Heat Transfer Coefficient (Btu/ft ² -s-°R)
J	Mechanical Equivalent of Heat (= 778 ft-lb _f /Btu)
K _f	Flow Resistance Coefficient (lb _f -sec ² /(lb-ft) ²)
k	Conductivity of Solid (Btu/ft-s-°R)
L	Length (in)
M	Molecular Weight
m	Resident Mass (lb)
\dot{m}	Mass Flow Rate (lb/sec)
N _E	Number of Iterations
n _{ss}	Number of neighboring solid nodes for i th solid node
n _{sf}	Number of neighboring fluid nodes for i th solid node
n _{sa}	Number of neighboring ambient nodes for i th solid node
p	Pressure (lb _f /in ²)
\dot{Q}	Heat transfer rate (Btu/s)
•	Heat Transfer Rate from Solid to Solid Node, Btu/s
q_{ss}	Heat Transfer Rate from Solid to Fluid Node, Btu/s
•	Heat Transfer Rate from Solid to Ambient Node, Btu/s
q_{sa}	Heat Transfer Rate from Solid to Ambient Node, Btu/s
R	Gas Constant (lb _f -ft/lb-R)
Re	Reynolds Number
•	Heat Generation Rate at i th Solid Node
S_i	
T	Temperature (° F)
u	Velocity (ft/sec)
V	Volume (in ³)
z	Compressibility Factor

Greek

ρ	Density (lb/ft ³)
μ	Viscosity (lb/ft-sec)
$\Delta\tau$	Time Step (sec)
τ	Time (sec)
ε	Surface Roughness of pipe (in), emissivity
σ	Stefan-Boltzman Constant (=0.1714 x 10 ⁻⁸ Btu/h-ft ² -°R ⁴)

Subscript

a	Ambient
c	Convection
f	Fluid
i	Node
ij	Branch, Conductor
r	Radiation
s	Solid

Superscript

a	Ambient
f	Fluid
s	Solid