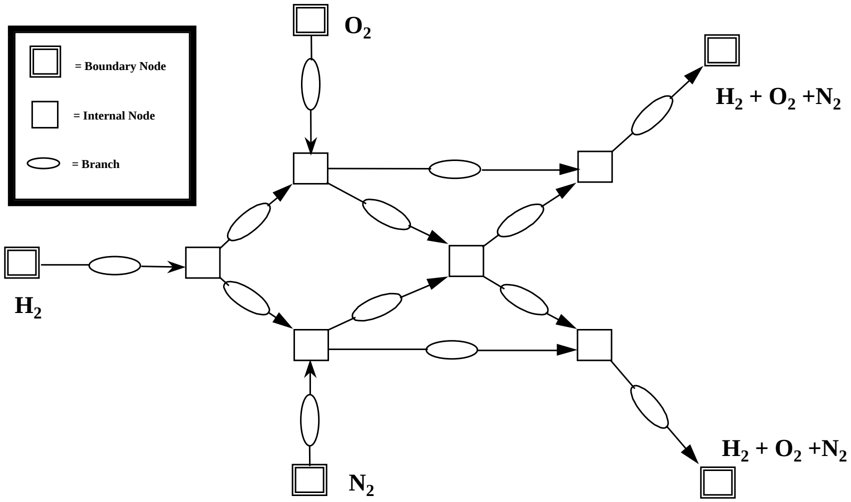
Generalized Fluid System Simulation Program (Version 6)



Alok Majumdar & Andre Leclair

NASA/Marshall Space Flight Center

GFSSP Training Class for Launch Service Program

November 27-29, 2012

Kennedy Space Center

Course Abstract

This three-day hands-on course provides basic introduction as well as advanced capabilities of GFSSP. GFSSP is a general-purpose computer program developed at Marshall Space Flight Center for analyzing steady state and time-dependent flow rates, pressures, temperatures, and concentrations in a complex flow network. For more information about GFSSP, please visit http://gfssp.msfc.nasa.gov/. The version 604 of the code will be released through this course. The course quickly teaches new users to use GFSSP to solve engineering flow network problems through lectures and tutorial problems. There are seven core lectures, nine lectures on application and six step-by-step tutorials and several challenge tutorials.

BACKGROUND

- GFSSP stands for Generalized Fluid System Simulation Program
- It is a general-purpose computer program to compute pressure, temperature and flow distribution in flow network with solid to fluid (conjugate) heat transfer
- It was primarily developed to analyze
  - Internal Flow Analysis of Turbopump
  - Transient Flow Analysis of Propulsion System
- GFSSP development started in 1994 with an objective to provide a generalized and easy to use flow analysis tool

BACKGROUND

DEVELOPMENT HISTORY & ONGOING DEVELOPMENTS

- Version 1.4 (Steady State) was released in 1996
- Version 2.01 (Thermodynamic Transient) was released in 1998
- Version 3.0 (User Subroutine) was released in 1999
- Graphical User Interface, VTASC was developed in 2000
- Selected for NASA Software of the Year Award in 2001
- Version 4.0 (Fluid Transient and post-processing capability) is released in 2003
- Version 5.0 (Conjugate Heat Transfer capability) is released in 2006.
- Educational Version was released in 2011
- Version 6.0 (Multi-Dimensional Capability) will be released in 2012
- Psychrometry & Chemical Reaction are under development

COURSE OUTLINE

Day 1 Morning

1. Introduction & Overview (CL-1)
2. Pre & Post Processor – Part I (CL-2)
3. Compressible Flow (LA-1)
4. Tutorial on Converging-Diverging Nozzle (TP-1)

Day 1 Afternoon

5. Resistance & Fluid Options (CL-4)
6. Fluid Transient (LA-2)
7. Tutorial on Water hammer (TP-2)

Day 2 Morning

1. Mathematical Formulation (CL-5)
2. Tank Pressurization,Control & Relief Valve (LA-3)
3. Tutorial on Tank Pressurization & Control Valve (TP-3)

Day 2 Afternoon

4. Pressure & Flow Regulator (LA-4)
5. Tutorial on Pressure Regulator (TP-4)
6. Cryogenic Propellant Loading (LA-5)
7. Tutorial on Transfer Line Chilldown (TP-5)

COURSE OUTLINE

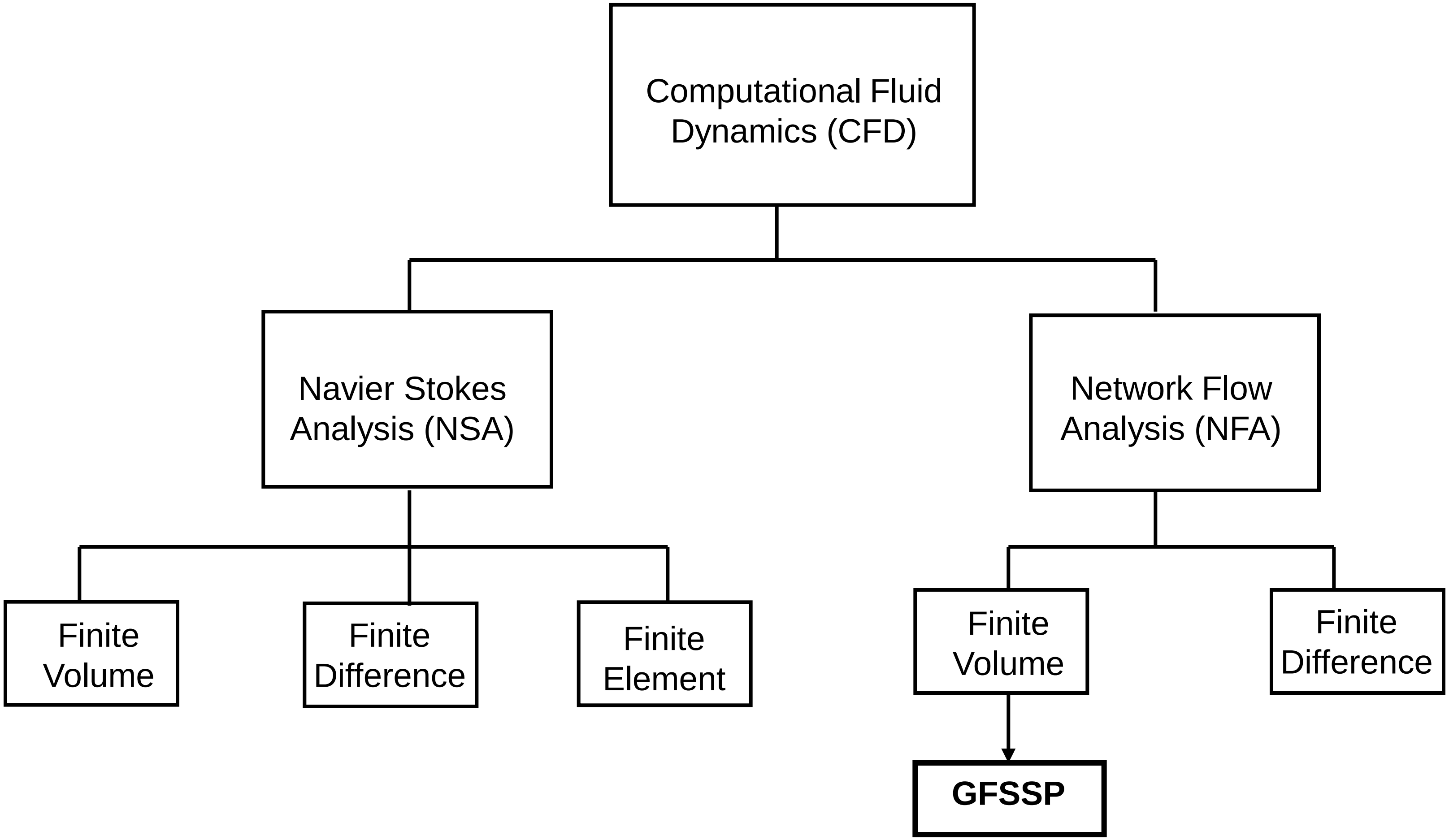
Day 3 Morning

1. Data Structure (CL-6)
2. User Subroutine (CL-7)
3. Fluid Mixture & Two-phase Flow (LA-6)
4. Tutorial on Propellant Recirculation (TP-6)

Day 3 Afternoon

5. Rotating Flow ,Turbopump & Heat Exchanger (LA-7)
6. Multi-Dimensional Flow Modeling (LA-8)
7. Open Session

NETWORK FLOW OR NAVIER STOKES ANALYSIS



NETWORK FLOW OR NAVIER STOKES ANALYSIS

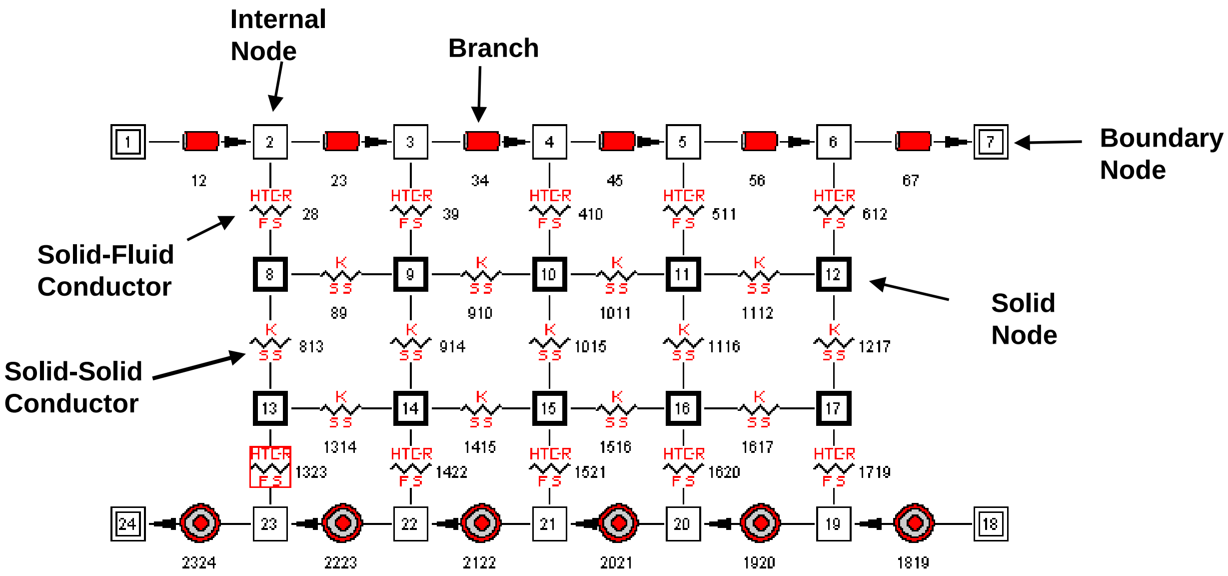
Navier Stokes Analysis

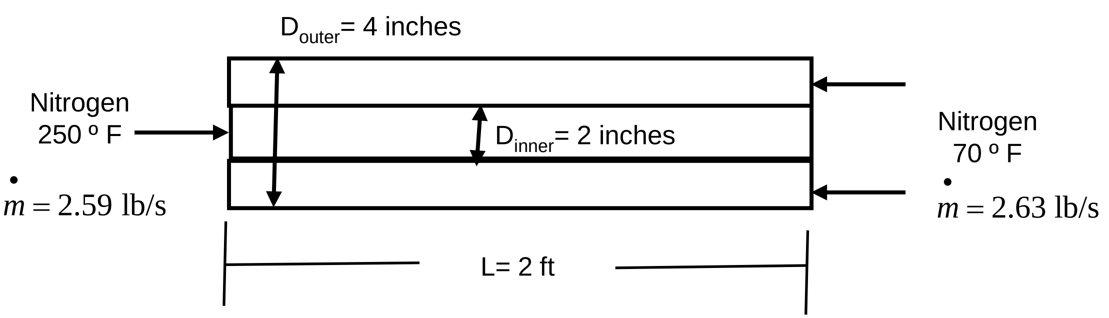
- Suitable for detailed flow analysis within a component
- Requires fine grid resolution to accurately model transport processes
- Used after preliminary design

Network Flow Analysis

- Suitable for flow analysis of a system consisting of several components
- Uses empirical laws of transport process
- Used during preliminary design

NETWORK DEFINITION





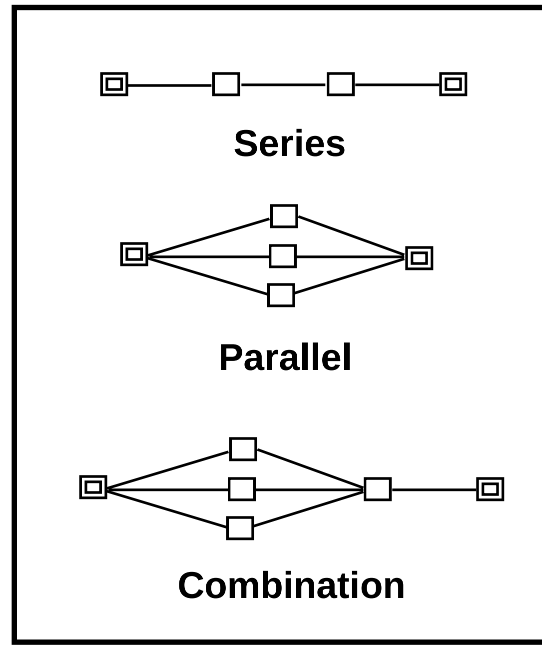
$\dot{m} = 2.59$ lb/s

$\dot{m} = 2.63$ lb/s

NETWORK DEFINITIONS

- **Network:**

  - **Boundary node**
  - **Internal node**
  - **Branch**



- **At boundary nodes, all dependent variables must be specified**

- **At internal nodes, all dependent variables must be guessed for steady flow and specified for transient flow.**

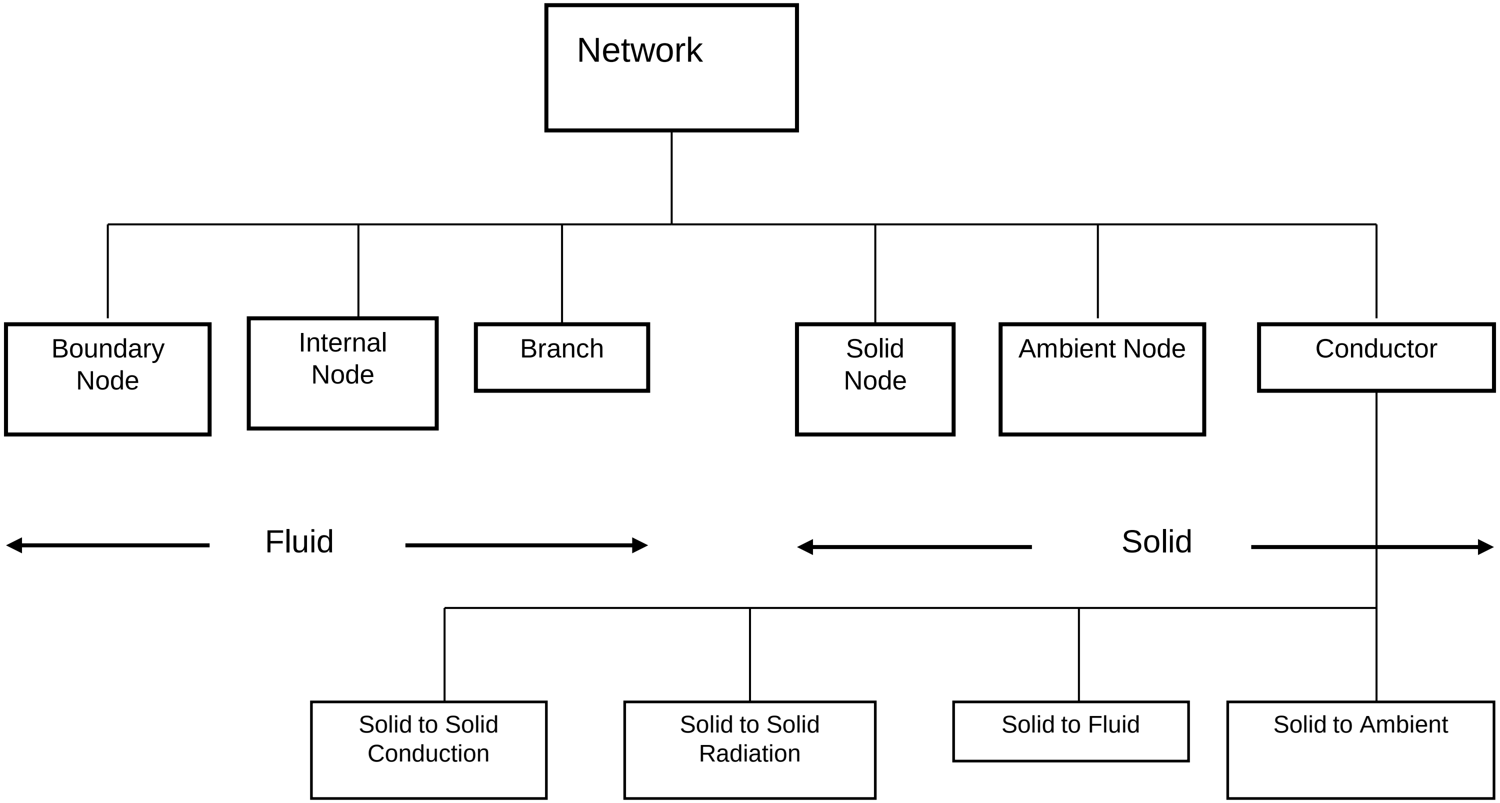
UNITS AND SIGN CONVENTIONS

- Units

	External (input/output)	**Internal** (inside GFSSP)
– Length	- inches	- feet
– Area	- $inches^2$	- $feet^2$
– Pressure	- psia	- psf
– Temperature	- °F	- °R
– Mass injection	- lbm/sec	- lbm/sec
– Heat Source	- Btu/s OR Btu/lbm	- Btu/s OR Btu/lbm

- Sign Convention
  - Mass input to node = positive
  - Mass output from node = negative
  - Heat input to node = positive
  - Heat output from node = negative

DATA STRUCTURE



MATHEMATICAL FORMULATION

Principal Variables:

Unknown Variables	**Available Equations to Solve**
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. Specie Concentrations	5. Conservation Equations for Mass Fraction of Species
6. Mass	6. Thermodynamic Equation of State

MATHEMATICAL FORMULATION

Auxiliary Variables:
Thermodynamic Properties, Flow Resistance Factor & Heat Transfer Coefficient

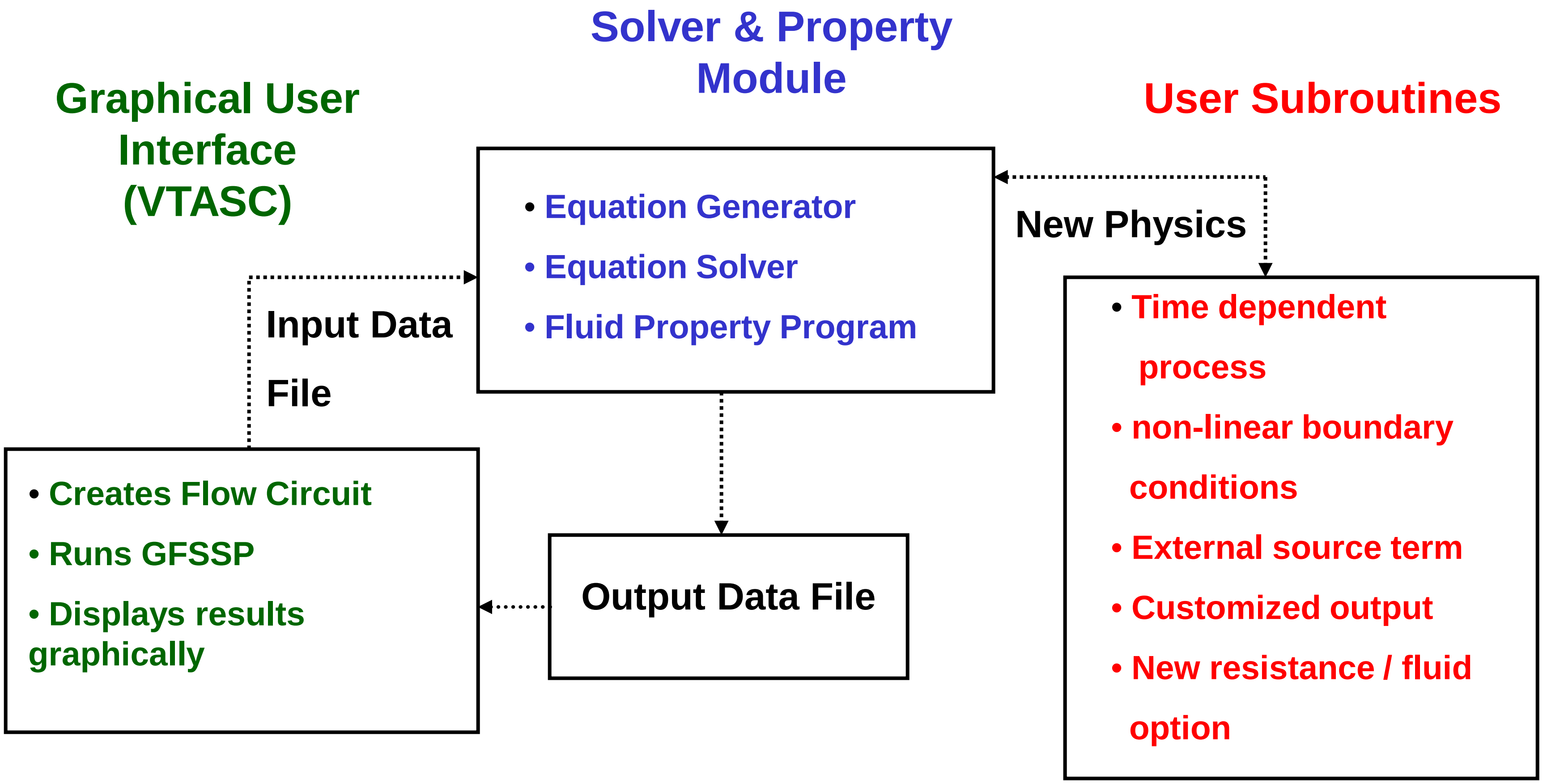
Unknown Variables	**Available Equations to Solve**
Density Specific Heats Viscosity Thermal Conductivity	Equilibrium Thermodynamic Relations [GASP, WASP & GASPAK Property Programs]
Flow Resistance Factor Heat Transfer Coefficient	Empirical Relations

MATHEMATICAL FORMULATION

BOUNDARY CONDITIONS

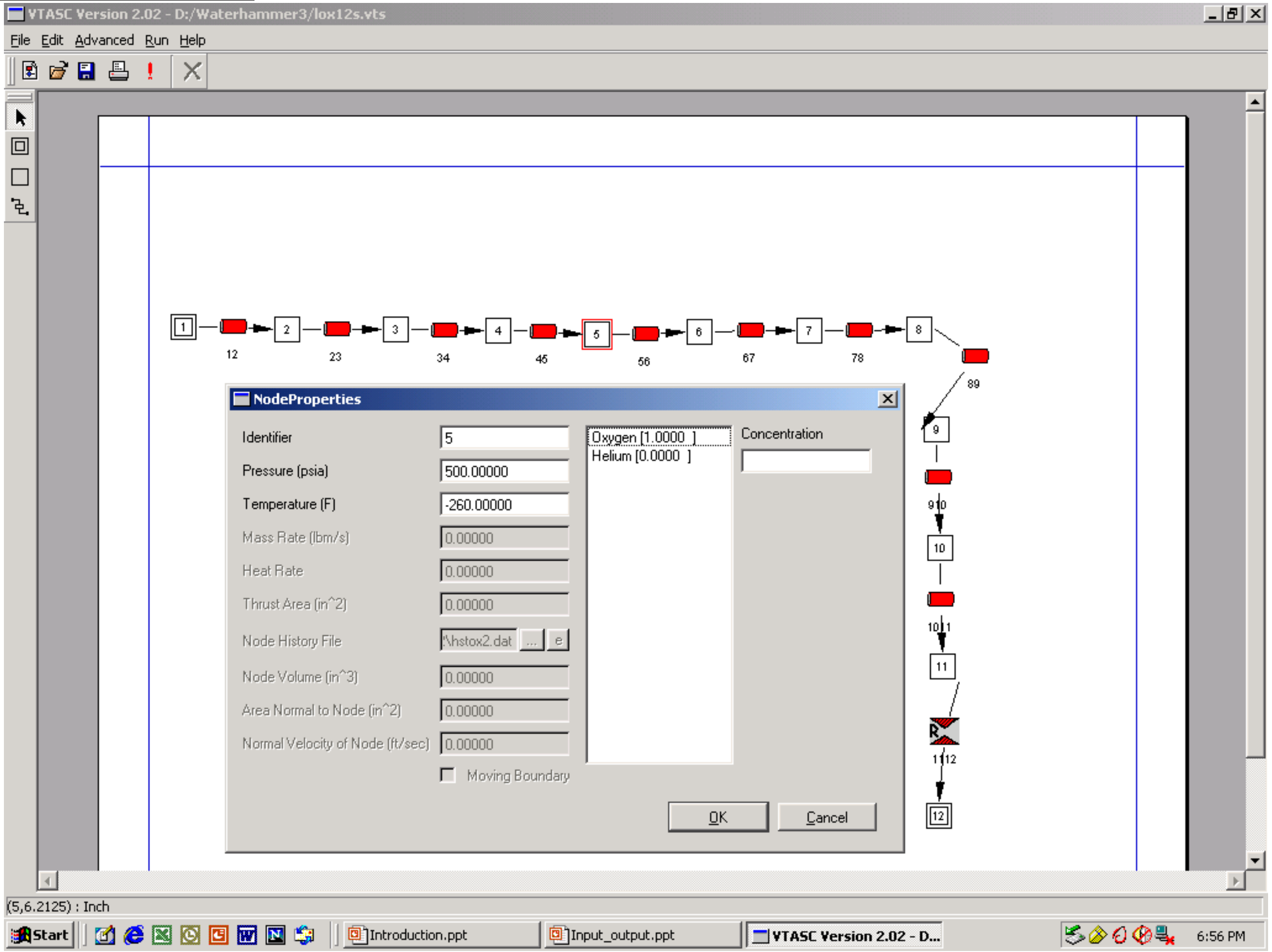
- Governing equations can generate an infinite number of solutions
- A unique solution is obtained with a given set of boundary conditions
- User provides the boundary conditions

PROGRAM STRUCTURE



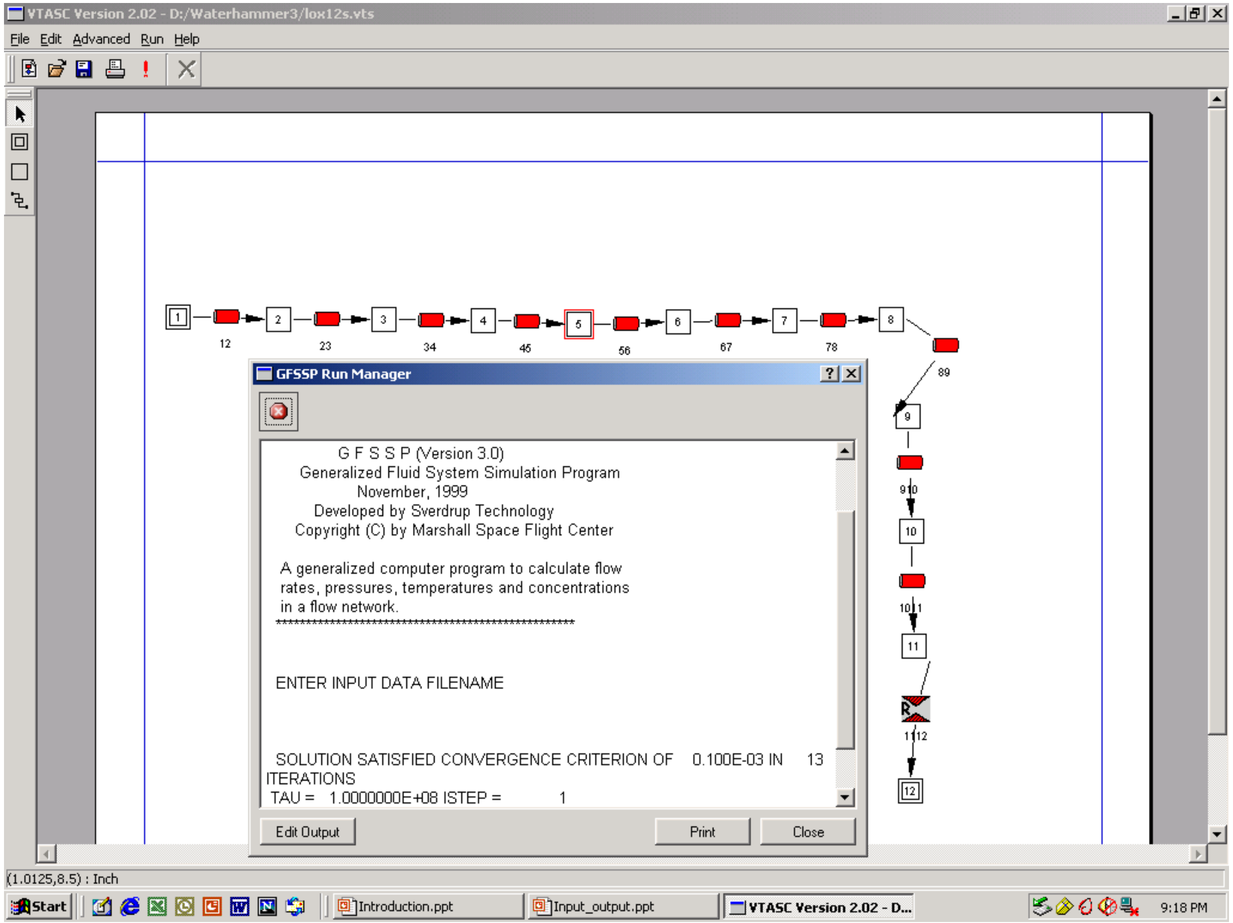
GRAPHICAL USER INTERFACE - 1

MODEL BUILDING



GRAPHICAL USER INTERFACE

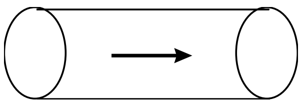
MODEL RUNNING



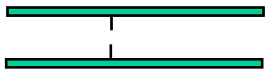
GRAPHICAL USER INTERFACE

MODEL RESULTS

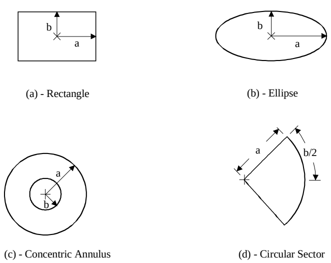
RESISTANCE OPTIONS



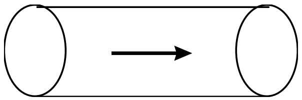
1. Pipe Flow



2. Flow Through a Restriction

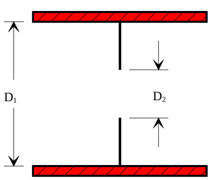


3. Non-Circular Duct



4. Pipe Flow with Entrance & Exit Losses

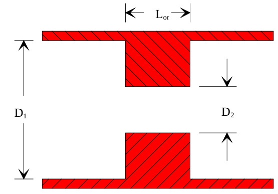
Thin Sharp Orifice



Where:
D_1 = Pipe Diameter
D_2 = Orifice Throat Diameter

5. Thin, Sharp Orifice

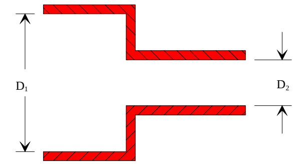
Thick Orifice



Where:
D_1 = Pipe Diameter
D_2 = Orifice Throat Diameter
L_{or} = Orifice Length

6. Thick Orifice

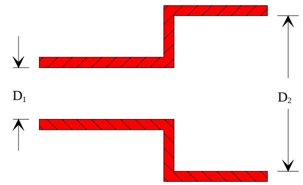
Square Reduction



Where:
D_1 = Upstream Pipe Diameter
D_2 = Downstream Pipe Diameter

7. Square Reduction

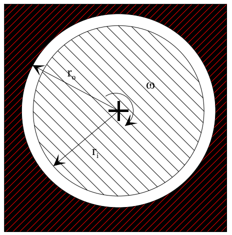
Square Expansion



Where:
D_1 = Upstream Pipe Diameter
D_2 = Downstream Pipe Diameter

8. Square Expansion

Rotating Annular Duct

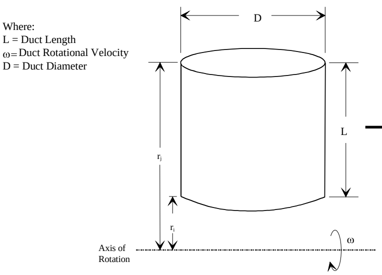


Where:
L = Duct Length (Perpendicular to Page)
b = Duct Wall Thickness ($b = r_o - r_i$)
ω = Duct Rotational Velocity
r_i = Duct Inner Radius
r_o = Duct Outer Radius

9. Rotating Annular Duct

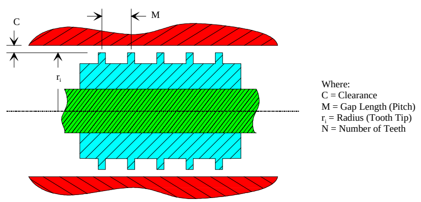
RESISTANCE OPTIONS

Rotating Radial Duct

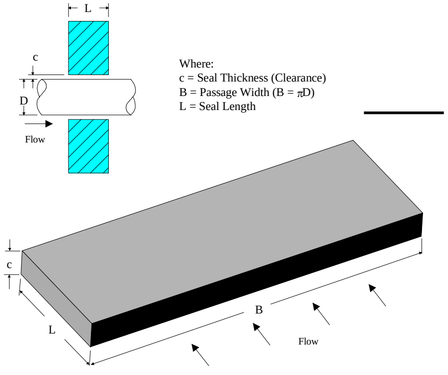


10. Rotating Radial Duct

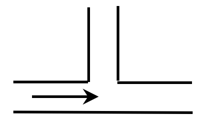
Labyrinth Seal



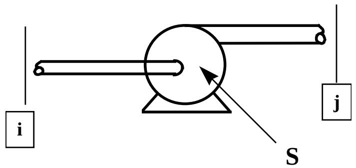
11. Labyrinth Seal



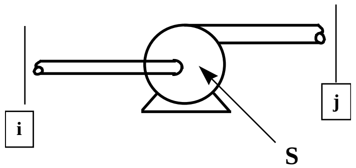
12. Face Seal



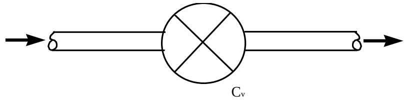
13. Common Fittings & Valves



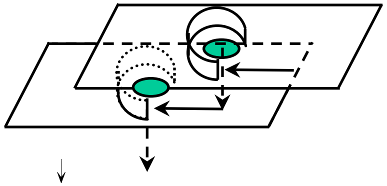
14. Pump Characteristics



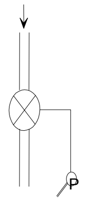
15. Pump Power



16. Valve with Given Cv

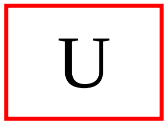


17. Viscojet



18. Control Valve

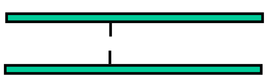
RESISTANCE OPTIONS



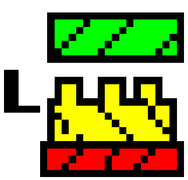
19. User Defined

20. Heat Exchanger Core

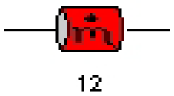
21. Parallel Tube



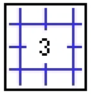
22. Compressible Orifice



23. Labyrinth Seal (Egli Correlation



24. Fixed Flowrate



25. Cartesian Grid

FLUID OPTIONS

GASP & WASP

Index	Fluid	Index	Fluid
1	HELIUM	7	ARGON
2	METHANE	8	CARBON DIOXIDE
3	NEON	9	FLUORINE
4	NITROGEN	10	HYDROGEN
5	CARBON MONOXIDE	11	WATER
6	OXYGEN	12	RP-1

FLUID OPTIONS

GASPAK

Index	Fluid	Index	Fluid
1	HELIUM	18	HYDROGEN SULFIDE
2	METHANE	19	KRYPTON
3	NEON	20	PROPANE
4	NITROGEN	21	XENON
5	CO	22	R-11
6	OXYGEN	23	R12
7	ARGON	24	R22
8	CO_2	25	R32
9	PARAHYDROGEN	26	R123
10	HYDROGEN	27	R124
11	WATER	28	R125
12	RP-1	29	R134A
13	ISOBUTANE	30	R152A
14	BUTANE	31	NITROGEN TRIFLUORIDE
15	DEUTERIUM	32	AMMONIA
16	ETHANE	33	IDEAL GAS
17	ETHYLENE		

FLUID OPTIONS

- Fluids not available in GASP/WASP or in GASPAK can be modeled by providing property tables
- GFSSP needs Molecular Weight and tables for following thermodynamic and transport properties:
    - Enthalpy
    - Entropy
    - Density
    - Viscosity
    - Thermal Conductivity
    - Specific Heat
    - Specific Heat Ratio

ADDITIONAL OPTIONS

- Variable Geometry
- Variable Rotation
- Variable Heat Addition
- Turbopump
- Heat Exchanger
- Tank Pressurization
- Control Valve
- Valve Open/Close
- Conjugate Heat Transfer
- Pressure Regulator
- Flow Regulator
- Relief Valve
- Multi-dimensional flow

Example Problems

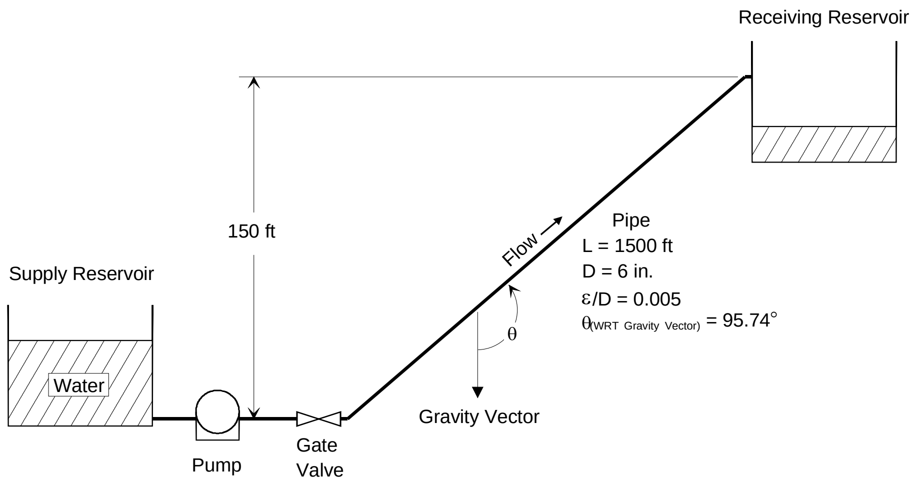
GFSSP User's Manual includes twenty-five example problems

The Purpose is to:

1. Demonstrate the major features of the code
2. Validate the solution by comparing with Textbook solution, Experimental data, if available

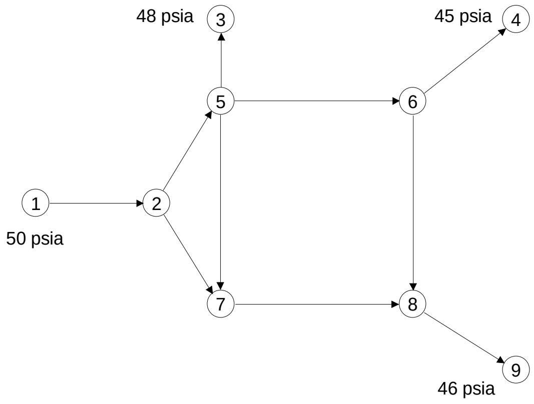
	EXAMPLE																									
FEATURE	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**	**15**	**16**	**17**	**18**	**19**	**20**	**21**	**22**	**23**	**24**	**25**	**FEATURE**
Conjugate Heat Transfer													13	14									23			Conjugate Heat Transfer
Constant Property		2					7																		25	Constant Property
Cyclic Boundary																				20						Cyclic Boundary
Fixed Flow Regulator																						22				Fixed Flow Regulator
Flow Regulator																	17									Flow Regulator
Gravity	1																						23			Gravity
Heat Exchanger					5						11									20						Heat Exchanger
Ideal Gas								8								16	17									Ideal Gas
Long Inertia			3			6						12						18	19							Long Inertia
Fluid Mixture				4						10		12											23			Fluid Mixture
Model Import																							23			Model Integration
Moving Boundary							7		9																	Moving Boundary
Multi-dimensional Flow																									25	Multi-dimensional Flow
Non-Circular Duct							7																			Non-Circular Duct
Phase Change														14												Phase Change
Pressurization (Tank)										10		12														Pressurization (Tank)
Pressure Regulator																16										Pressure Regulator
Pressure Relief Valve																								24		Pressure Relief Valve
Pump	1											12														Pump
Turbo Pump											11															Turbo Pump
Turbo Pump-Internal Flow																					21					Axial Thrust
Unsteady								8	9	10		12		14	15	16	17					22	23	24		Unsteady
User Fluid																				20						User Fluid
User Subroutine										10		12						18	19	20						User Subroutine
Valve O/C															15											Valve O/C
Variable Geometry									9																	Variable Geometry
Water Hammer															15											Water Hammer

Example 1 - Simulation of a Flow System Consisting of a Pump, Valve and Pipe Line



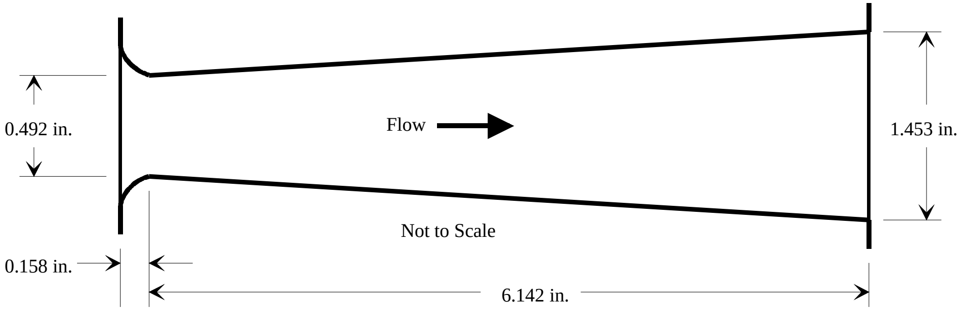
Feature: Gravity, Pump Characteristics, Pipe & Valve

Example 2 - Simulation of a Water Distribution Network



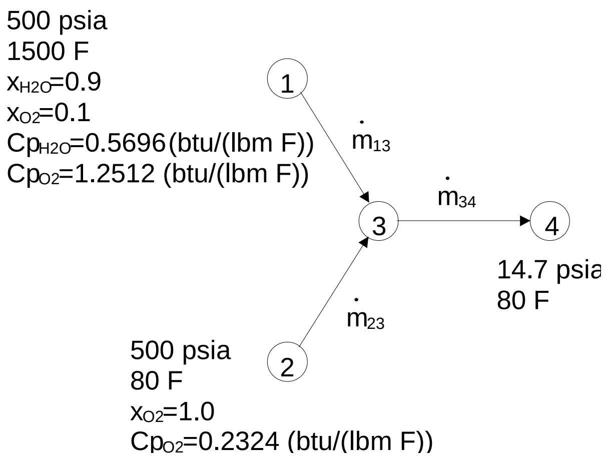
Feature: Constant Property Fluid Option & Comparison with Hardy-Cross Method

Example 3 - Simulation of Compressible Flow in a Converging-Diverging Nozzle



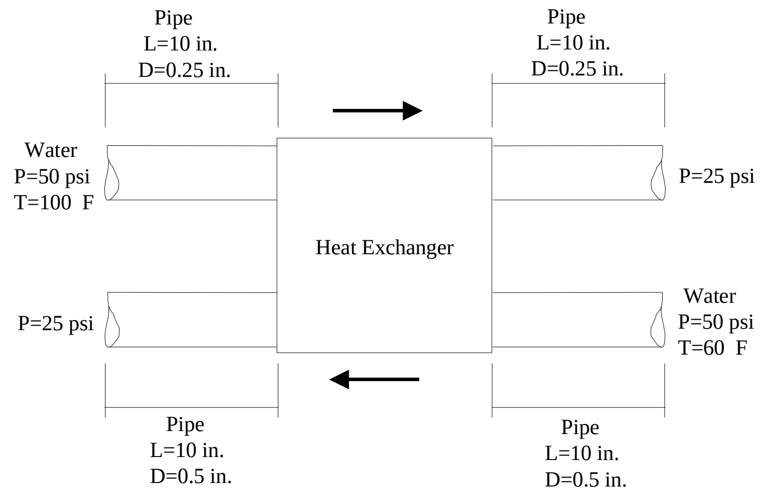
Feature: Inertia in Momentum Conservation, Comparison with Isentropic Solution

Example 4 - Simulation of the Mixing of Combustion Gases and a Cold Gas Stream



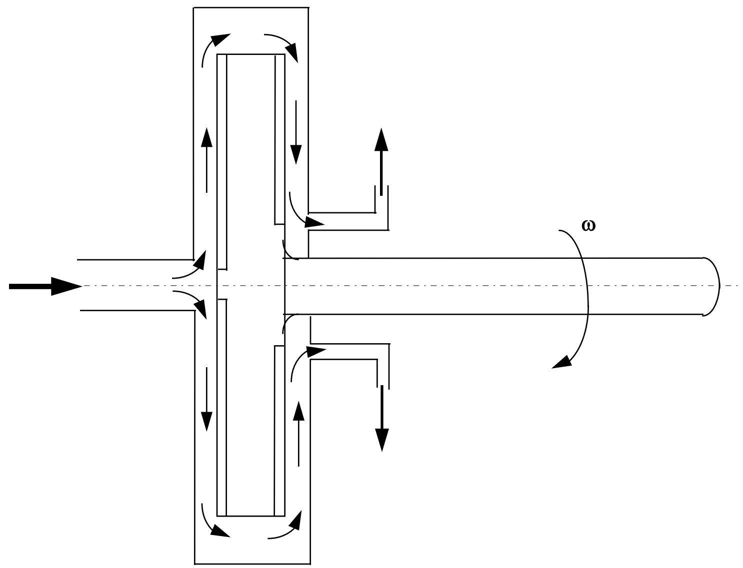
Feature: Fluid Mixture, Comparison with Textbook Solution

Example 5 - Simulation of a Flow System Involving a Heat Exchanger



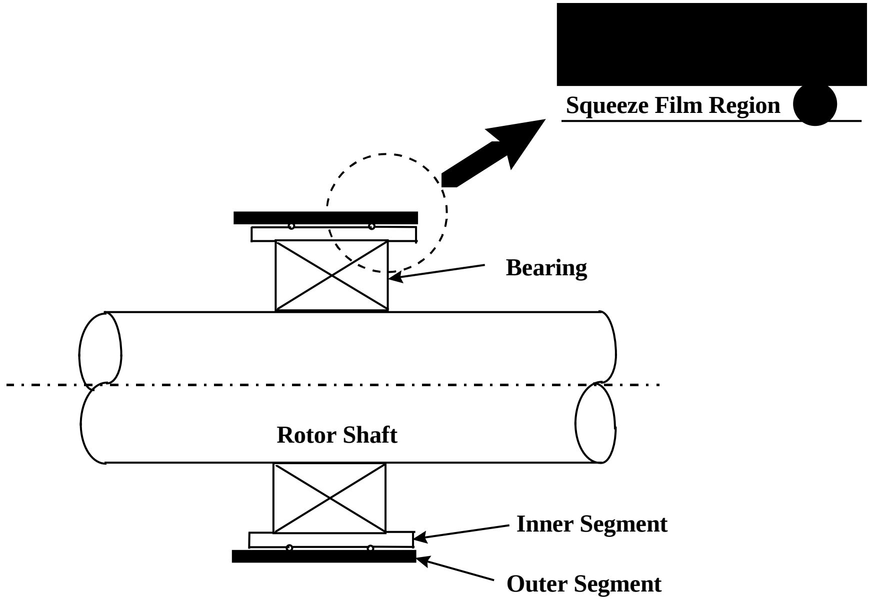
Feature: Heat Exchanger Option, Comparison with Textbook Solution

Example 6 - Radial Flow on a Rotating Radial Disk



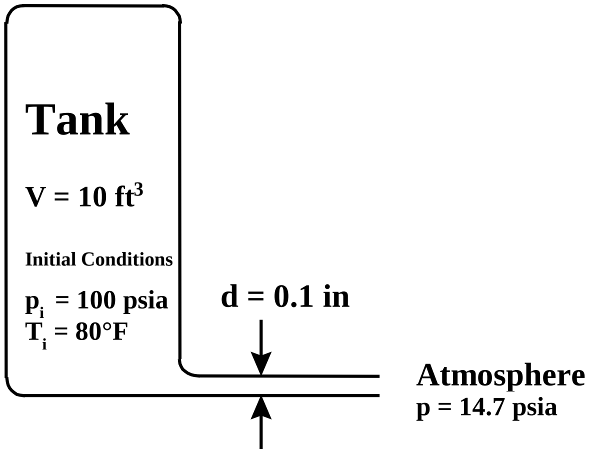
Feature: Rotating Flows, Comparison with Textbook Solution

Example 7 - Flow in a Long Bearing Squeeze Film Damper



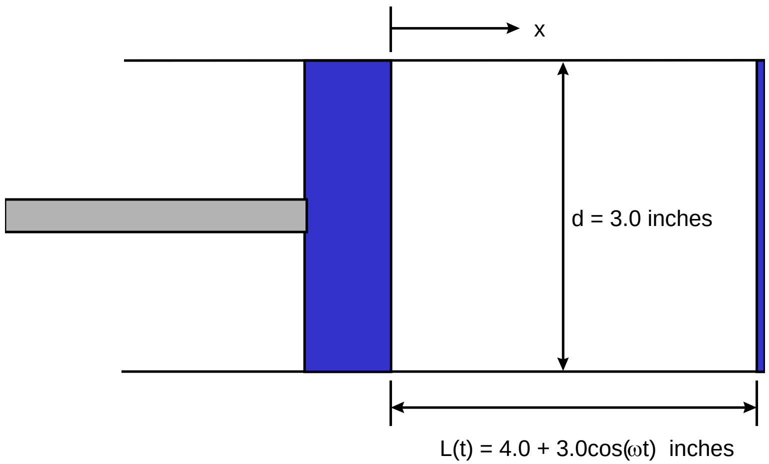
Feature: Moving Boundary, Comparison with Test Data

Example 8 - Simulation of the Blow Down of a Pressurized Tank



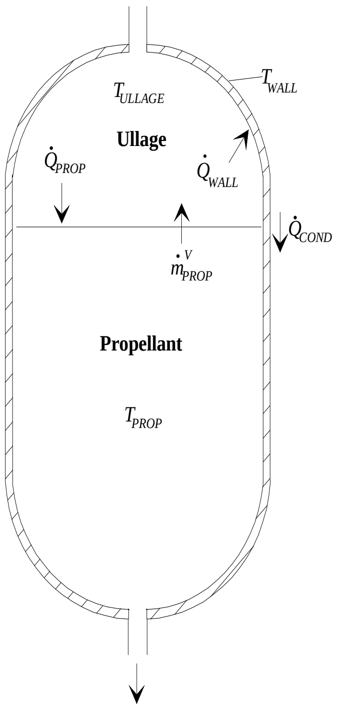
Feature: Unsteady Flow, Comparison with Analytical Solution

Example 9 - A Reciprocating Piston-Cylinder



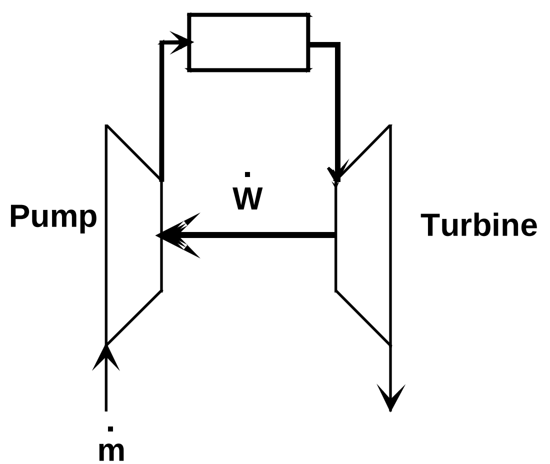
Feature: Variable Geometry, Moving Boundary, and Comparison with Analytical Solution

Example 10 - Pressurization of a Propellant Tank



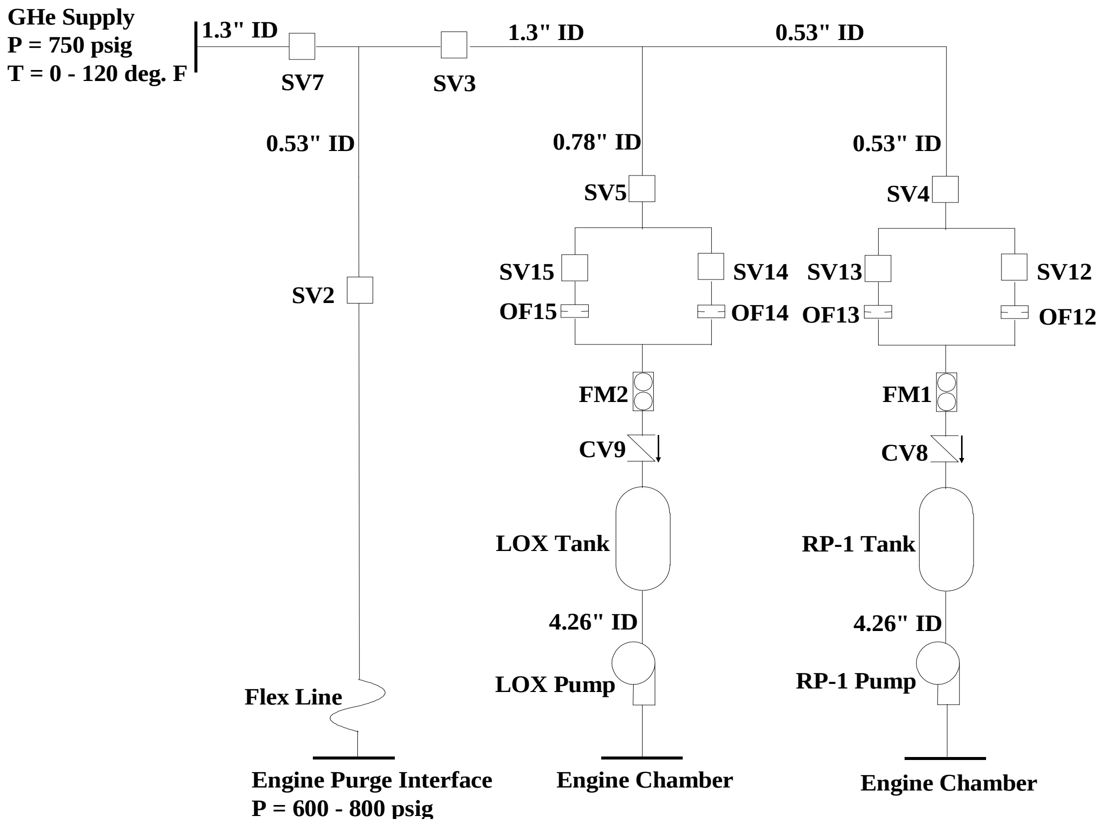
Feature: Tank Pressurization, User Subroutine for Mass Transfer, Comparison with published data

Example 11 - Power Balancing of a Turbopump Assembly



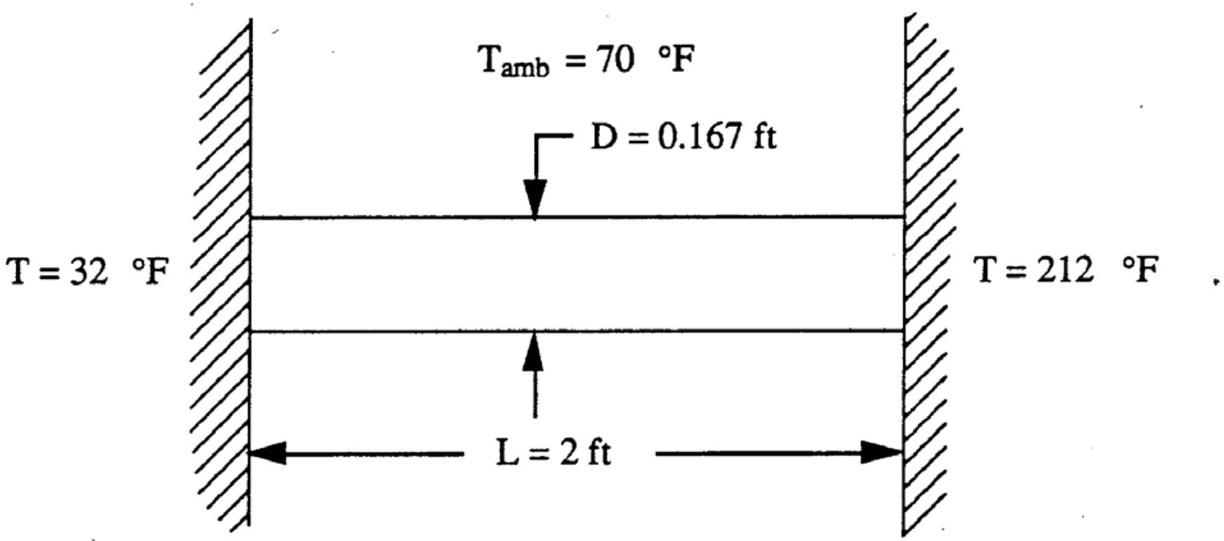
Feature: Turbopump & Heat Exchanger Option, External Heat Input

Example 12 - Helium Pressurization of LOX and RP-1 Propellant Tanks



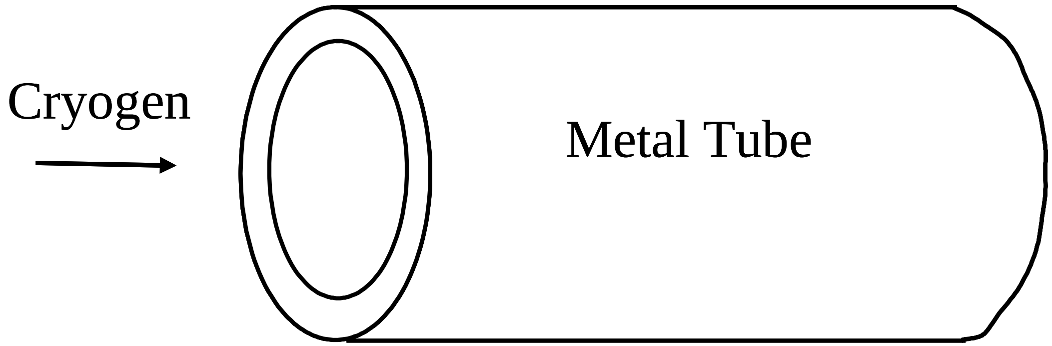
Feature: Multiple Tank Pressurization with control valve & Comparison with test data

Example 13 - Steady State & Transient Conduction Through a Circular Rod, With Convection



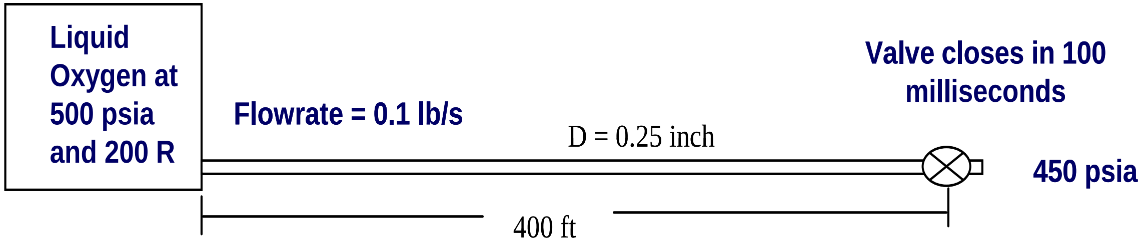
Feature: Conjugate Heat Transfer and comparison with analytical solution

Example 14 - Chilldown of a Short Cryogenic Pipeline



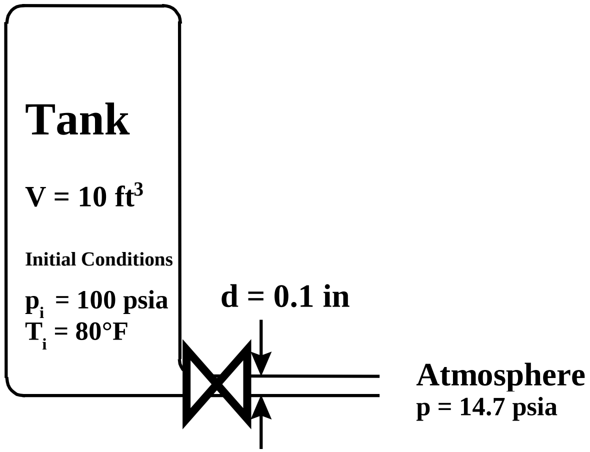
Feature: Conjugate Heat Transfer with phase change and comparison with analytical solution

Example 15 – Simulation of Fluid Transient Following Sudden Valve Closure



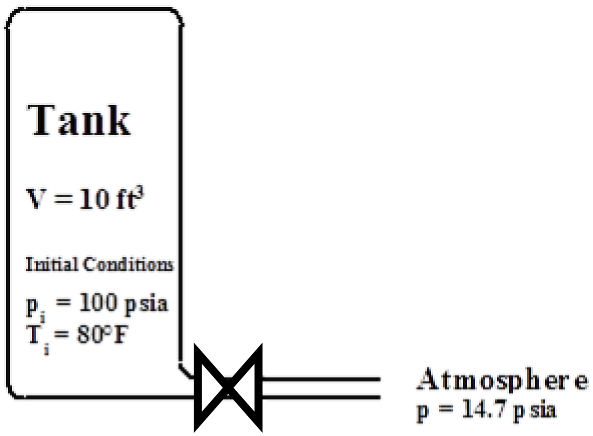
Feature: Valve Open/Close, Restart and Comparison with Method of Characteristic Solution

Example 16 - Simulation of a Pressure Regulator Downstream of a Pressurized Tank



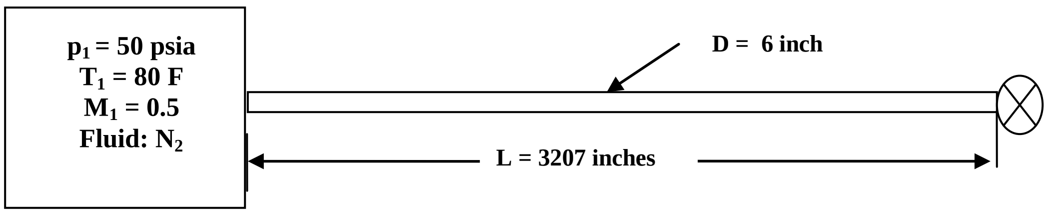
Feature: Pressure Regulator Option (Iterative & Marching Algorithm)

Example 17 - Simulation of a Flow Regulator Downstream of a Pressurized Tank



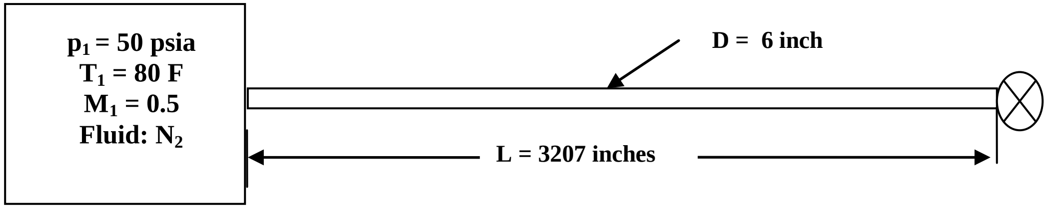
Feature: Flow Regulator Option (Iterative & Marching Algorithm)

Example 18 - Simulation of a Subsonic Fanno Flow



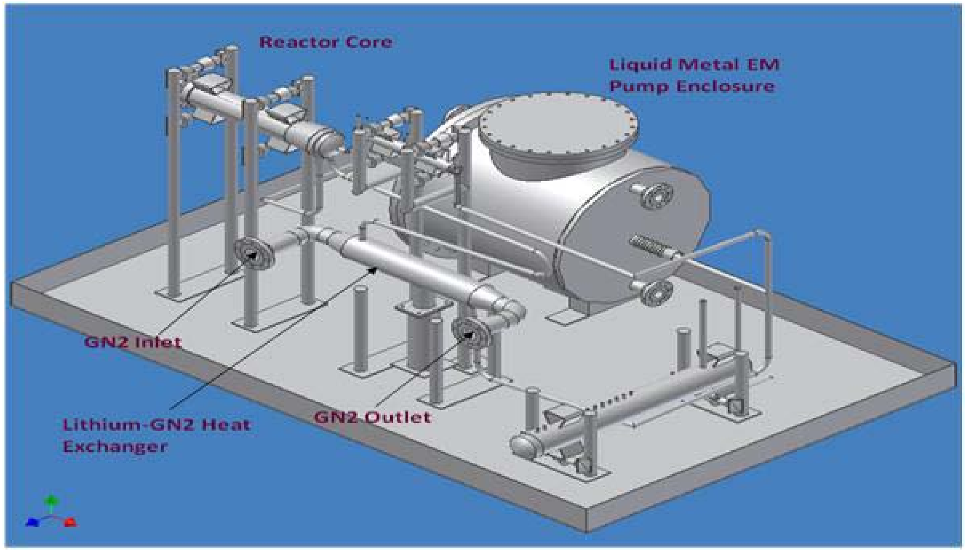
Feature: Compressible Flow with Friction, User Subroutine to set constant friction factor, and comparison with Text Book solution

Example 19 - Simulation of a Rayleigh Flow



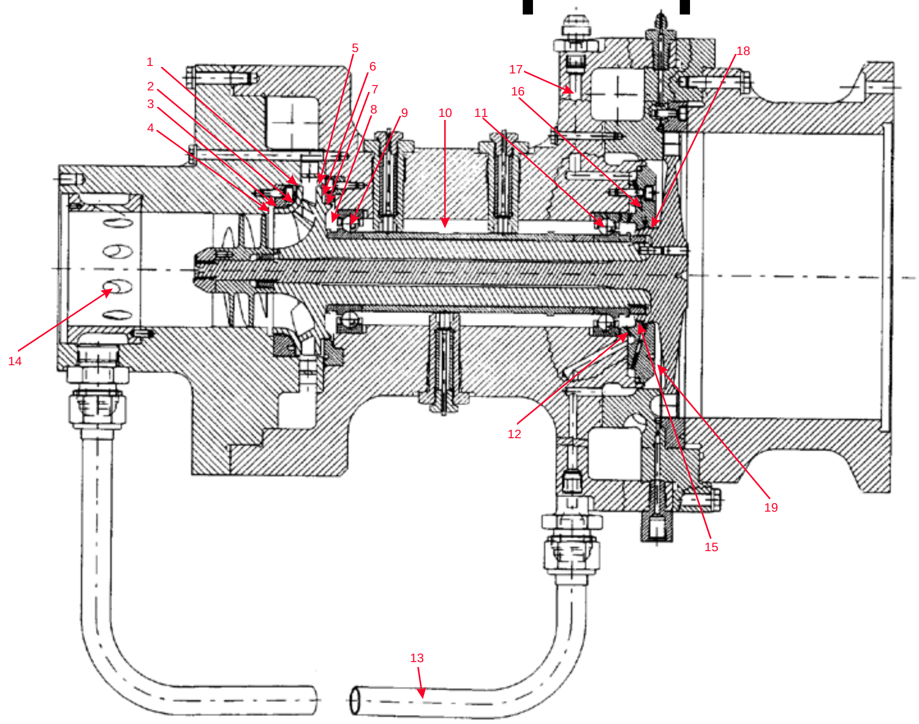
Feature: Compressible flow with heat transfer, User Subroutine to set zero friction factor, and comparison with analytical solution

Example 20 - Simulation of a Lithium Loop Model



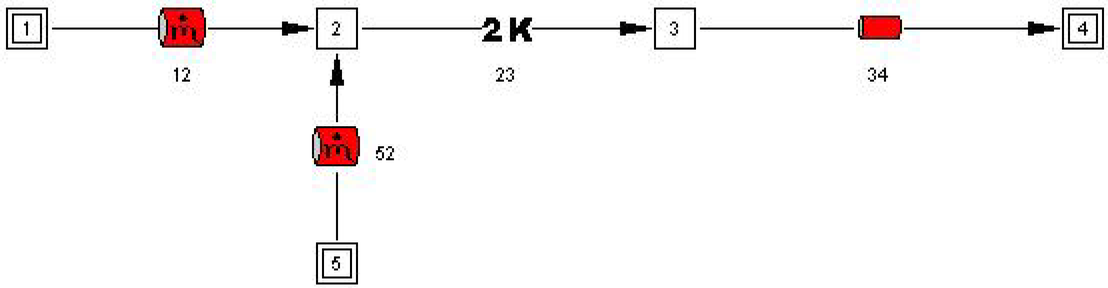
Feature: Closed Loop with cyclic boundary, Use of user-specified property, Heat Exchanger & User Subroutine to model Electro-Magnetic Pump

Example 21 – Axial Thrust Calculation in a Turbopump



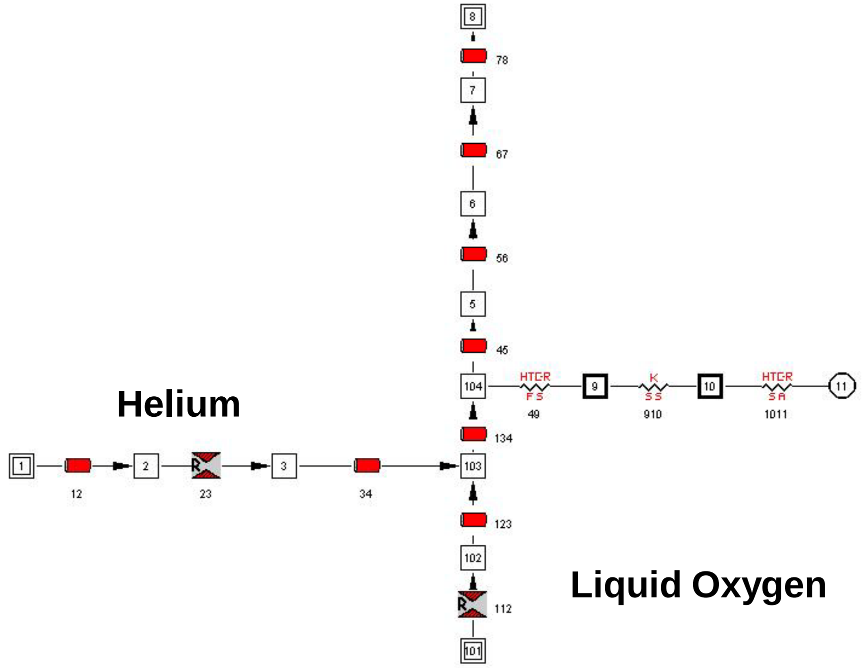
Feature: Axial Thrust, Rotating Flow, Mixture, Parallel Tube and comparison with test data

Example 22 – Simulation of a Fluid Network with Fixed Flow Rate Option



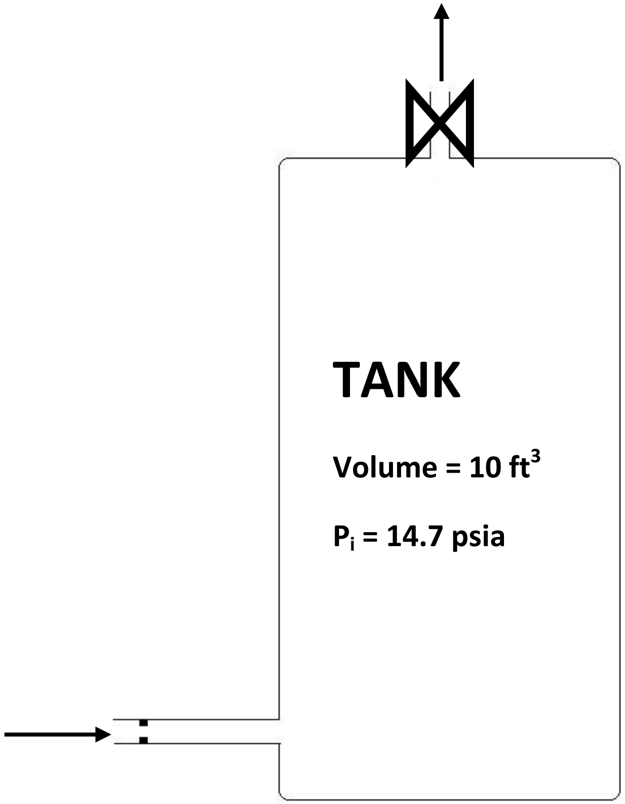
Feature: Fixed Flow Rate Option for Unsteady Flow

Example 23 – Helium-Assisted Buoyancy-Driven Flow in a Vertical Pipe Carrying Liquid Oxygen with Ambient Heat Leak



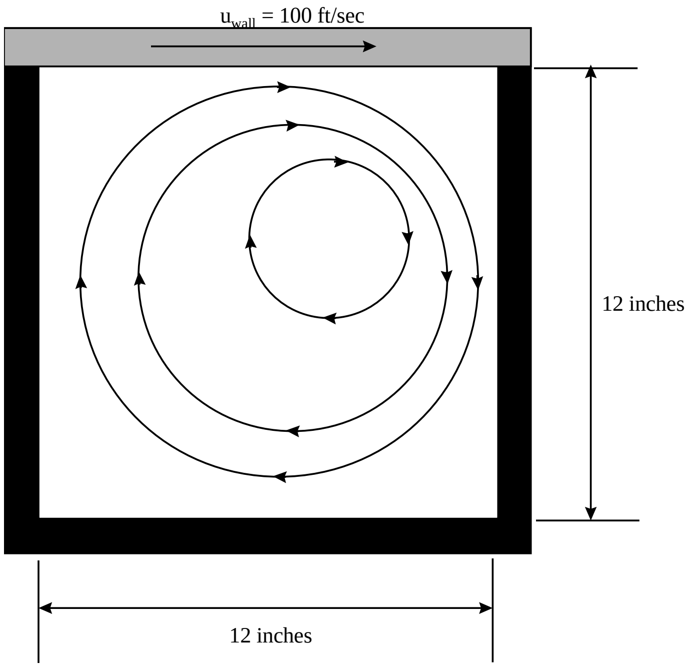
Feature: Phase Change in Fluid Mixture, Buoyancy Driven Flow, Model Import and Conjugate Heat Transfer

Example 24 – Simulation of Relief Valve in a Pressurized Tank



Feature: Relief Valve Option

Example 25 – Two-dimensional Recirculating Flow in a Driven Cavity



Feature: Multi-dimensional option, Grid Generation and Post Processing with Tecplot

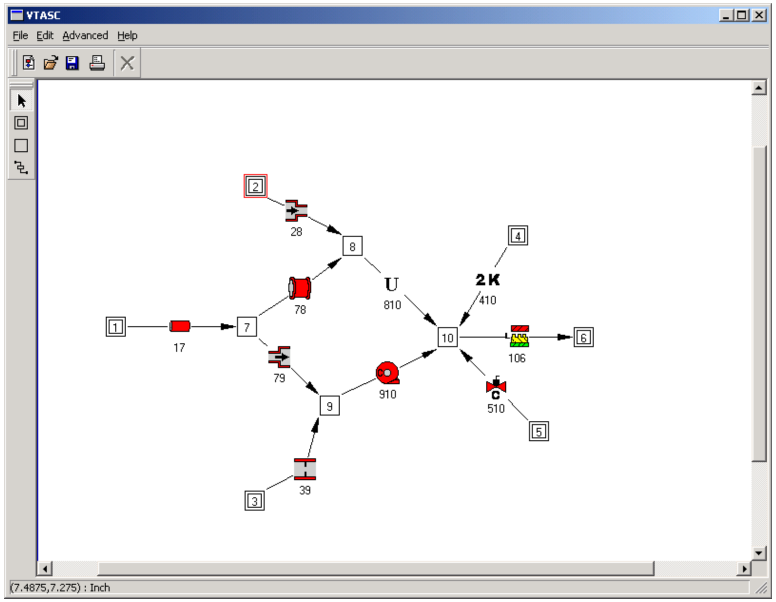
SUMMARY

- GFSSP is a finite volume based Network Flow Analyzer
- Flow circuit is resolved into a network consisting of nodes and branches
- Mass, energy, and species conservation are solved at internal nodes. Momentum conservation is solved at branch
- Generalized data structure allows generation of all types of flow network
- Modular code structure allows user to add new capabilities with ease

SUMMARY

- Unique mathematical formulation allows effective coupling of thermodynamics and fluid mechanics
- Numerical scheme is robust; adjustment of numerical control parameters is seldom necessary
- Intuitive Graphical User Interface makes it easy to build, run, and evaluate numerical models
- GFSSP has been successfully applied in various applications that included
  - Incompressible & Compressible flows
  - Phase change (Boiling & Condensation)
  - Fluid Mixture
  - Thermodynamic transient (Pressurization & Blowdown)
  - Fluid Transient (Waterhammer)
  - Conjugate Heat Transfer
- Twenty five Example Problems illustrate use of various code options

INPUT/OUTPUT THROUGH A GRAPHICAL USER INTERFACE - VTASC



GFSSP Version 6 Training Course

Content

- Overview
- VTASC Description
- VTASC Steady State Demonstration
- Input/Output Files

Overview

Visual Thermo-fluid dynamic Analyzer for Systems and Components (VTASC) is a program designed to efficiently build flow network models for use in the GFSSP program.

- Visually Interactive
  - "Drag and Drop" Paradigm
  - Model Building, Running, and Post-Processing in one environment
- Self-Documenting
  - Hard copy of flow network
  - Bitmap image of flow network for inclusion into papers and presentations

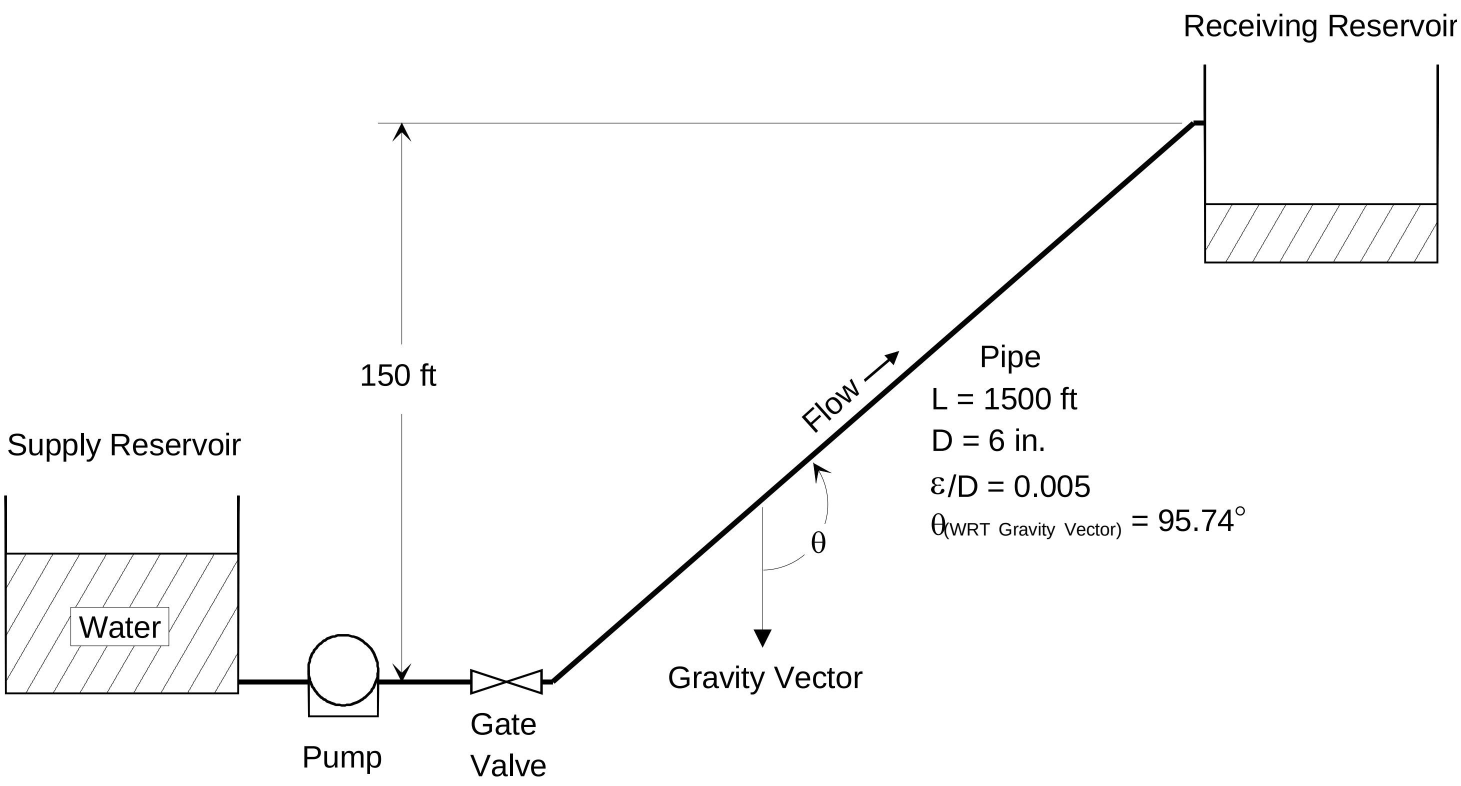
Overview

- Eliminates errors during model building process
  - Automatic node and branch numbering
  - Save and restore models at any point in the model building process
  - Robust
- Pushbutton generation of GFSSP input file
  - Steady and Transient cases
  - Advanced features such as Turbopump, Tank Pressurization, and Heat Exchangers
- Run GFSSP directly from VTASC window
  - GFSSP Run Manager acts as VTASC/GFSSP interface

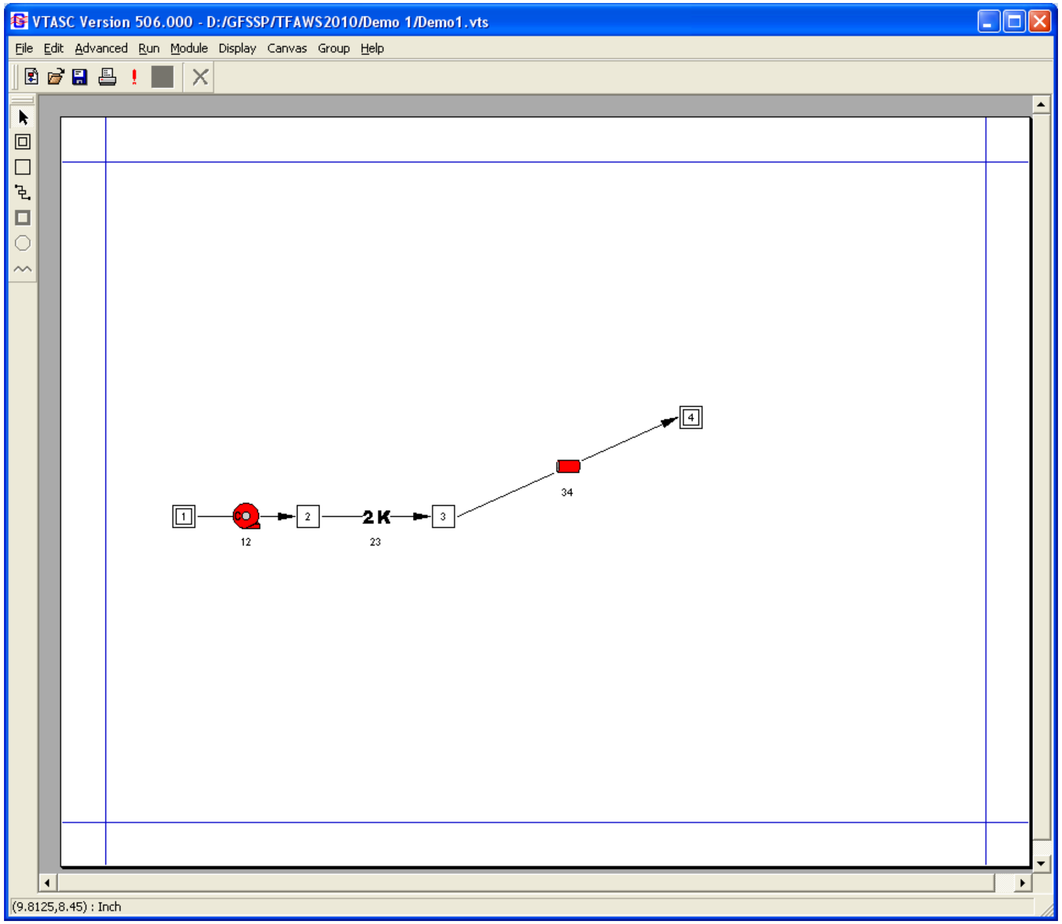
Overview

- Post-processing capability allows quick study of results
  - Pushbutton access to GFSSP output file
  - Point and click access to output at each node and branch
  - Built-in plotting capability for transient cases
  - Capable of plotting through Winplot
- User Subroutines can be developed and integrated
  - Editing, compiling, and linking to create a new executable can be done in VTASC Window
  - New executable replaces the default executable

VTASC DEMONSTRATION PROBLEMS

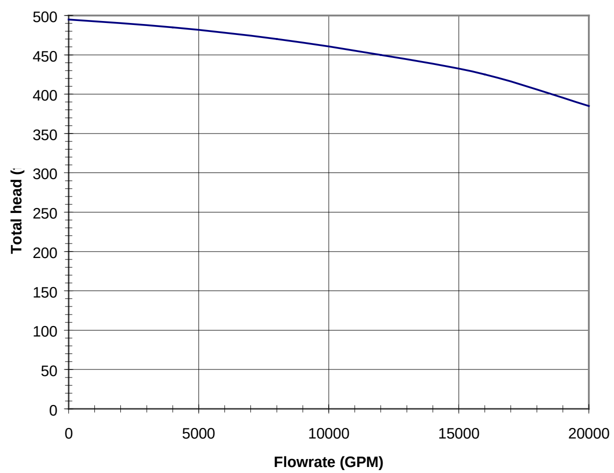


Build Model on VTASC Canvas



$\dot{m}$

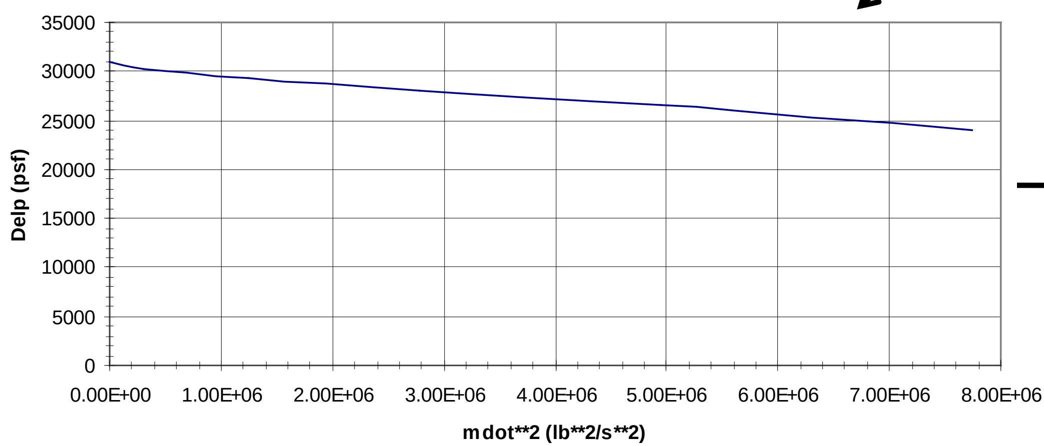
Determination of Pump Characteristics



1.) Manufacturer's Pump Curve (Head vs. Flowrate)

Q (GPM)	mdot (lb/s)	Head (ft)	Δp (psf)	$mdot^2$ $(lb/s)^2$
0	0	495	30,888	0
4000	556.13	485	30,264	3.093e05
8000	1112.3	470	29,328	1.2372e06
12000	1668.4	450	28,080	2.784e06
16000	2224.5	425	26,520	4.9484e06
20000	2781	385	24,024	7.734e06

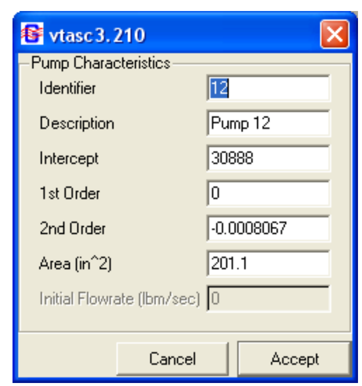
2.) Convert to lb/s and psf



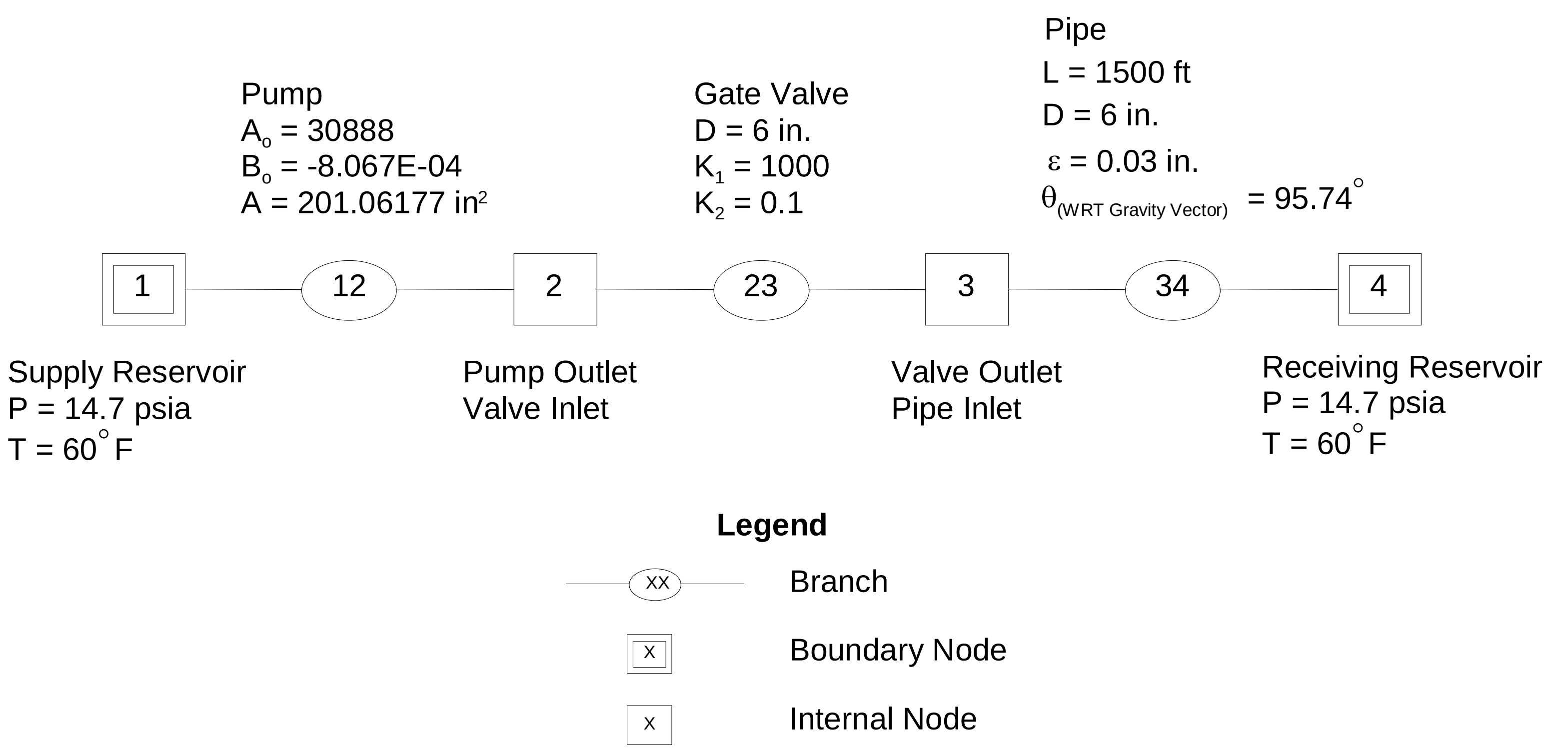
3.) Plot delP vs. $mdot^2$

4.) Curve fit:

$$\Delta P = 30{,}888 - 8.067\times10^{-4}\,\dot{m}^2$$



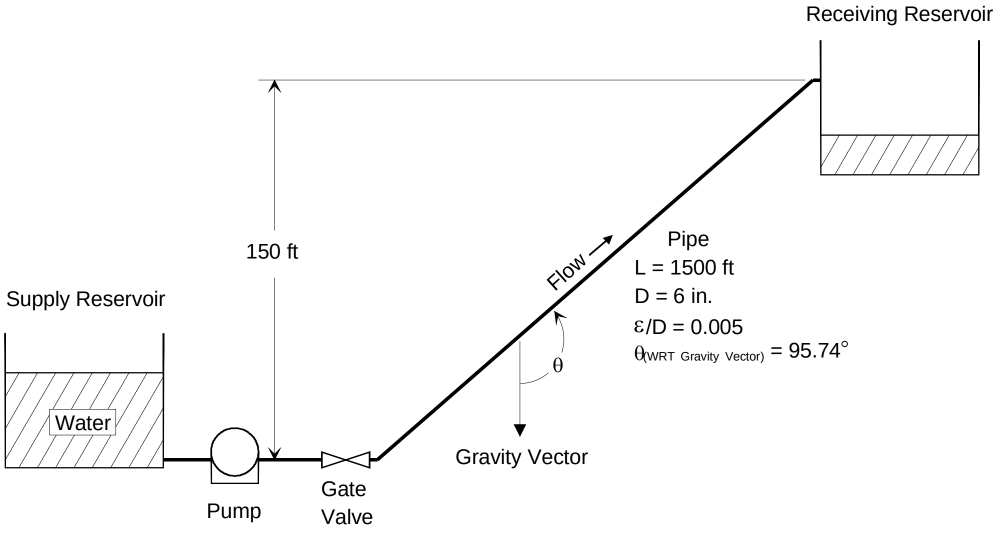
VTASC DEMONSTRATION PROBLEMS



INPUT DATA FILE

Example 1 Flow System Schematic

- GFSSP data files are created by VTASC
- Structure of the data file are classified into 20 sections
- Example 1 will be used to explain the content of input data file



INPUT DATA FILE

```
GFSSP VERSION
   300
ANALYST
ALOK MAJUMDAR
INPUT DATA FILE NAME
ex1.dat
OUTPUT FILE NAME
ex1gp.out
TITLE
PUMP-SYSTEM CHARACTERISTICS
```

← **Title Information**

```
USETUP
     F
DENCON   GRAVITY   ENERGY   MIXTURE   THRUST   STEADY   TRANSV   SAVER
     F        T        T        F        F        T        F        F
HEX   HCOEF   REACTING   INERTIA   CONDX   ADDPROP   PRINTI   ROTATION
     F        F        F        F        F        F        T        F
BUOYANCY   HRATE   INVAL   MSORCE   MOVBND   TPA   VARGEO   TVM
     F        F        F        F        F        F        F        F
SHEAR   PRNTIN   PRNTADD   LAMINAR   TRANSQ
     F        T        T        T        F
PRESS   INSUC   VARROT
     F        F        F
NORMAL   SIMUL   SECONDL
     T        T        F
```

← **Logical Variables**

```
NNODES    NINT   NBR   NF
     4        2      3      1
RELAXK   RELAXD   RELAXH CC NITER
  1.000    0.500    1.000   0.100E-03   500
NFLUID(I), I= 1,NF
    11
```

← **Node, Branch & Fluid Information** (NNODES … NF)

← **Solution Control Variable** (RELAXK … NITER)

← **Fluid Designation** (NFLUID)

INPUT DATA FILE

```
  NODE INDEX
     1     2
     2     1
     3     1
     4     2
 NODE  PRES (PSI)  TEMP(DEGF)  MASS SOURC  HEAT SOURC  THRST AREA
     1  0.1470E+02  0.6000E+02  0.0000E+00  0.0000E+00  0.0000E+00
     2  0.1000E+02  0.6000E+02  0.0000E+00  0.0000E+00  0.0000E+00
     3  0.4803E+02  0.6000E+02  0.0000E+00  0.0000E+00  0.0000E+00
     4  0.1470E+02  0.6000E+02  0.0000E+00  0.0000E+00  0.0000E+00
 INODE   NUMBR     BRANCH 1   BRANCH 2   BRANCH 3   BRANCH 4   BRANCH 5
     2       2          12         23
     3       2          23         34
BRANCH   UPNODE    DNNODE    OPTION
    12         1        2       14
    23         2        3       13
    34         3        4        1
BRANCH OPTION -14:  PUMP CONST1, PUMP CONST2, AREA
    12 30888.00000    -0.00081   201.06177
BRANCH OPTION -13:  DIA, K1, K2, AREA
    23      6.00000  1000.00000     0.10000     28.27431
BRANCH  OPTION -1:  LENGTH, DIA, EPSD, ANGLE, AREA
    34 18000.00000     6.00000     0.00500     95.74000     28.27431
BRANCH   NOUBR    NMUBR
    12       0
    23       1        12
    34       1        23
```

Node Numbering & Designation

Node Variables

Node Branch Connection

Branch Flow Designation and Resistance Option

Resistance Option Information

Inertia Information

INPUT DATA FILE

```
BRANCH    NODBR     NMDBR
    12         1        23
    23         1        34
    34         0
BRANCH
          12
  UPSTRM BR.     ANGLE
  DNSTRM BR.     ANGLE
           23      0.00
BRANCH
          23
  UPSTRM BR.     ANGLE
           12      0.00
  DNSTRM BR.     ANGLE
           34      0.00
BRANCH
          34
  UPSTRM BR.     ANGLE
           23      0.00
  DNSTRM BR.     ANGLE
 NODE DATA FILE
FNODE.DAT
 BRANCH DATA FILE
FBRANCH.DAT
```

Inertia Information (arrow pointing to the BRANCH inertia data)

Restart File (arrow pointing to the node/branch data file lines)

OUTPUT DATA FILE

Output Data File can be classified into following categories:

- Title and Data Files
- Logical Variables
- Node & Branch Information
- Fluid Information
- Boundary Conditions
- Solution Results
- Convergence Message

OUTPUT DATA FILE

1. Titles & Data Files

```
TITLE      :PUMP-SYSTEM CHARACTERISTICS
ANALYST    :ALOK MAJUMDAR
FILEIN     :ex1.dat
FILEOUT    :EX1.OUT
```

2. Logical Variables

```
LOGICAL VARIABLES
   DENCON   =  F
   GRAVITY  =  T
   ENERGY   =  T
   MIXTURE  =  F
   THRUST   =  F
   STEADY   =  T
   TRANSV   =  F
   SAVER    =  F
   HEX      =  F
```

OUTPUT DATA FILE

3. Node & Branch Information

```
NNODES   =   4
NINT     =   2
NBR      =   3
NF       =   1
NVAR     =   5
NHREF    =   2
```

4. Fluid Information

```
FLUIDS:  H2O
```

OUTPUT DATA FILE

5. Boundary Conditions

BOUNDARY NODES

NODE	P (PSI)	T (F)	RHO (LBM/FT^3)	AREA (IN^2)
1	0.1470E+02	0.6000E+02	0.6237E+02	0.0000E+00
4	0.1470E+02	0.6000E+02	0.6237E+02	0.0000E+00

6. Solution Results (Nodes)

SOLUTION
INTERNAL NODES

NODE	P(PSI)	TF(F)	Z	RHO	EM(LBM) (LBM/FT^3)	QUALITY
2	0.2290E+03	0.6003E+02	0.1186E-01	0.6241E+02	0.0000E+00	0.0000E+00
3	0.2288E+03	0.6003E+02	0.1185E-01	0.6241E+02	0.0000E+00	0.0000E+00

OUTPUT DATA FILE

7. Solution Results (Branches)

BRANCHES

BRANCH	KFACTOR (LBF-S^2/(LBM-FT)^2)	DELP (PSI)	FLOW RATE (LBM/SEC)	VELOCITY (FT/SEC)	REYN. NO.	MACH NO.	ENTROPY GEN. BTU/(R-SEC)	LOST WORK LBF-FT/SEC
12	0.000E+00	-0.214E+03	0.191E+03	0.219E+01	0.241E+06	0.183E-02	0.000E+00	0.000E+00
23	0.764E-03	0.193E+00	0.191E+03	0.156E+02	0.644E+06	0.130E-01	0.210E-03	0.848E+02
34	0.591E+00	0.214E+03	0.191E+03	0.156E+02	0.644E+06	0.130E-01	0.162E+00	0.657E+05

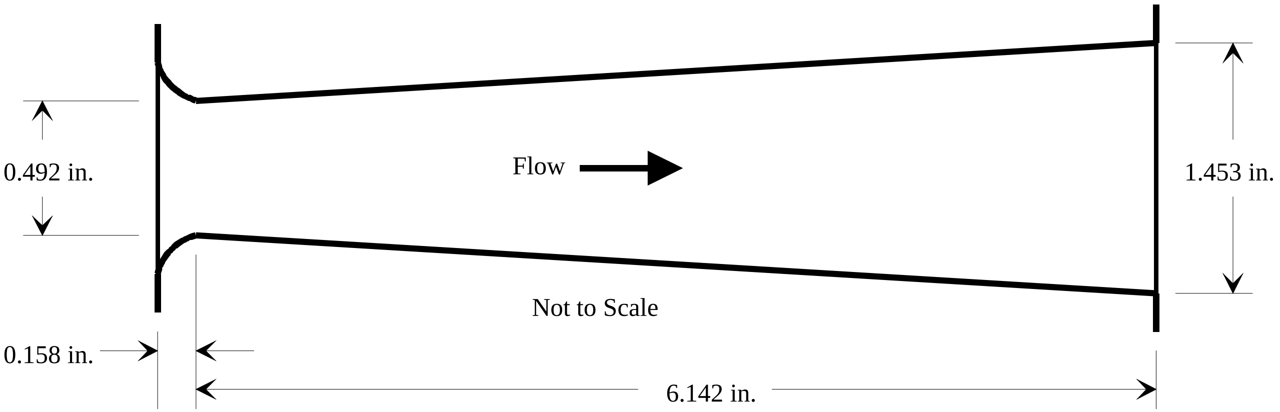
8. Convergence Information

```
***** TOTAL ENTROPY GENERATION =    0.163E+00 BTU/(R-SEC) ******

**** TOTAL WORK LOST =    0.120E+03 HP *****


SOLUTION SATISFIED CONVERGENCE CRITERION OF    0.100E-03 IN      6 ITERATIONS
TAU =    0.100000E+09ISTEP =            1
```

SUMMARY

- VTASC is a flow network model builder for use with GFSSP
- Flow networks can be designed and modified interactively using a “Point and Click” paradigm
- Generates GFSSP Version 6.04 compatible input files
- Winplot can be activated from VTASC for post processing
- User Subroutines can also be developed, compiled, and linked from VTASC

Compressible Flow

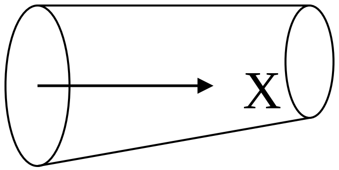


CONTENT

- One-dimensional Compressible Flow
- Compressible Flow Modeling in GFSSP
- Converging- Diverging Nozzle (Example 3 & Tutorial 1)
- Fanno Flow (Example 18)
- Rayleigh Flow (Example 19)

Assumptions

- The properties are function of x only



$$A = A(x); p = p(x); \rho = \rho(x); u = u(x); T = T(x)$$

Governing Equations

Mass Conservation:

$$\frac{d\rho}{\rho} + \frac{dA}{A} + \frac{dV}{V} = 0$$

Momentum Conservation:

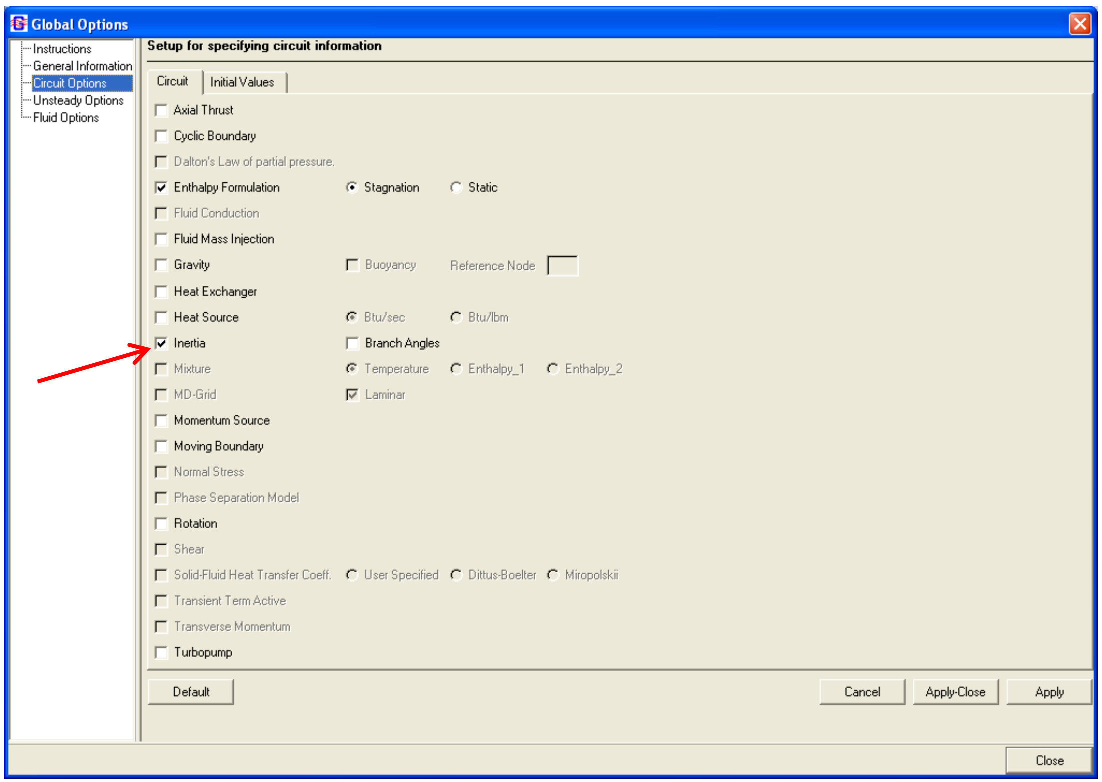
$$\frac{dp}{p} + \frac{\gamma M^2}{2}\frac{fdx}{D} + \gamma M^2 \frac{dV}{V} = 0$$

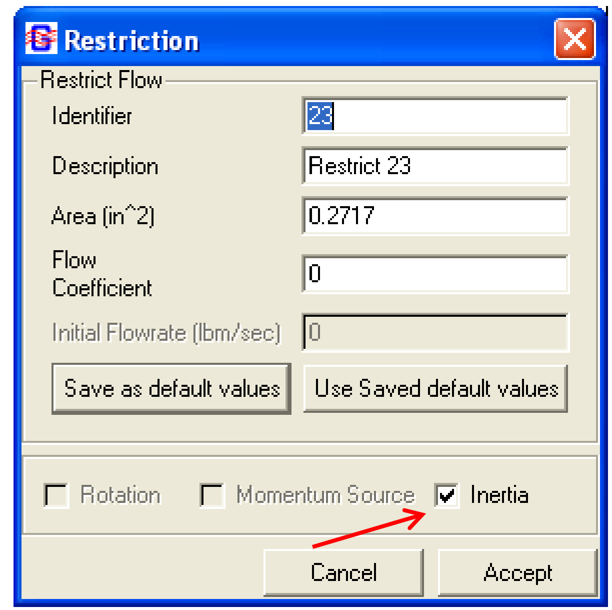
Where, M = Mach no. = $\frac{V}{C} = \frac{V}{\sqrt{\gamma \frac{p}{\rho}}}$

Analytical Solution

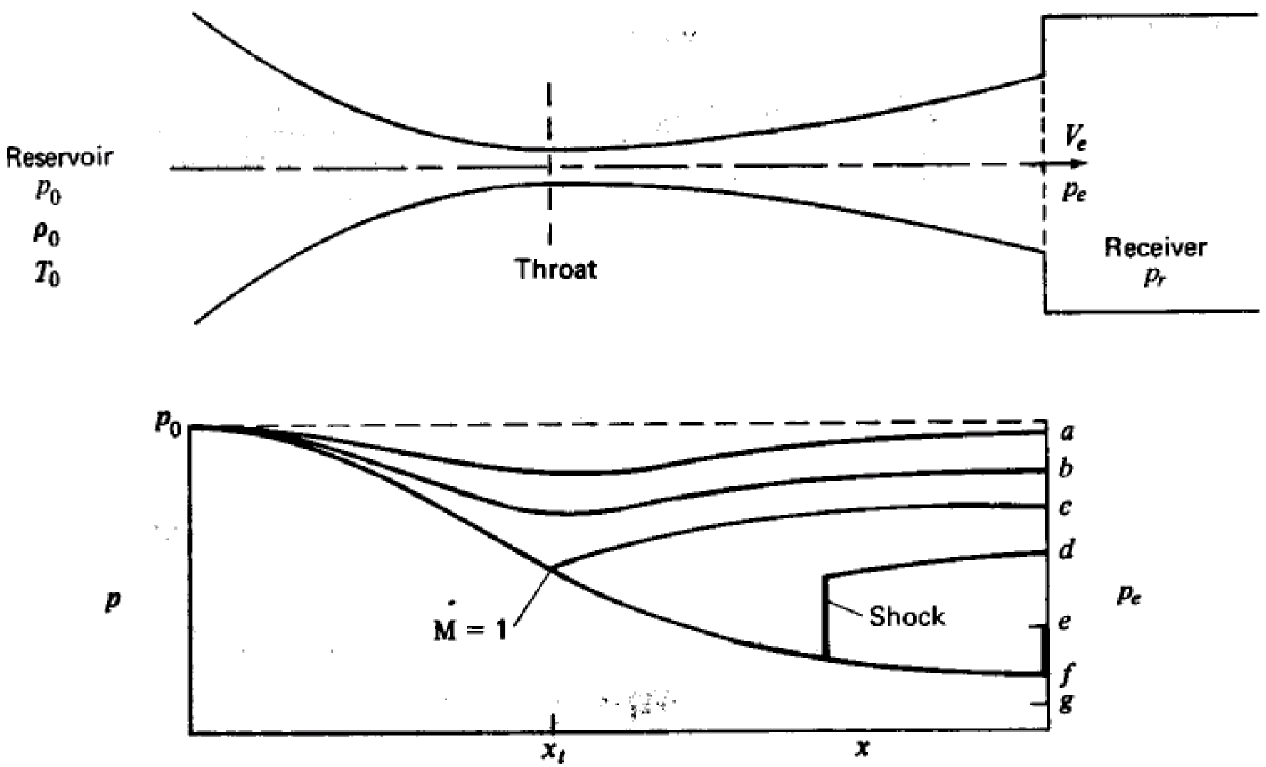
$$\frac{dM}{dx} = \frac{M\left(1 + \frac{\gamma - 1}{2}M^2\right)}{\left(1 - M^2\right)}\left[\gamma M^2 \frac{f}{D} + \frac{\left(1 + \gamma M^2\right)}{2T_0}\frac{dT_0}{dx} - \gamma M^2 \frac{1}{A}\frac{dA}{dx}\right]$$

- GFSSP considers all fluids to be compressible at all speeds, however, inertia term in Momentum Conservation Equation needs to be activated for high speed flows





Effect of varying back pressure



a & b : Subsonic Flow

c: Sonic flow at throat; rest subsonic flow

d: Shock wave in diverging section

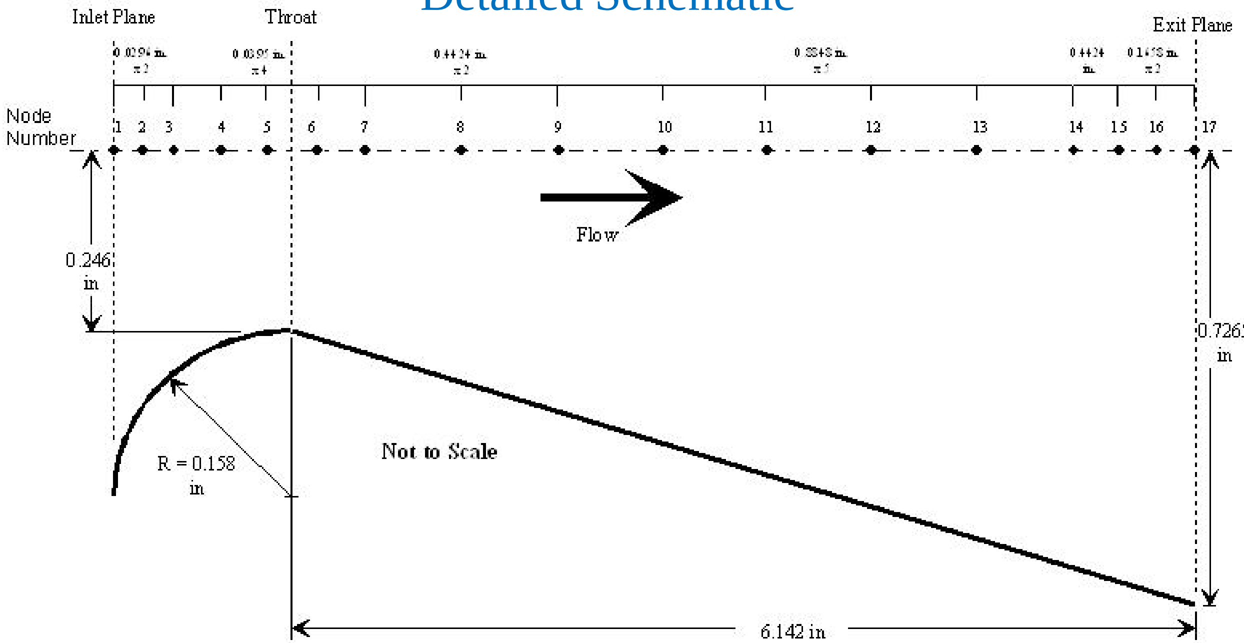
e: Shock wave at exit plane

f: Supersonic flow in diverging section

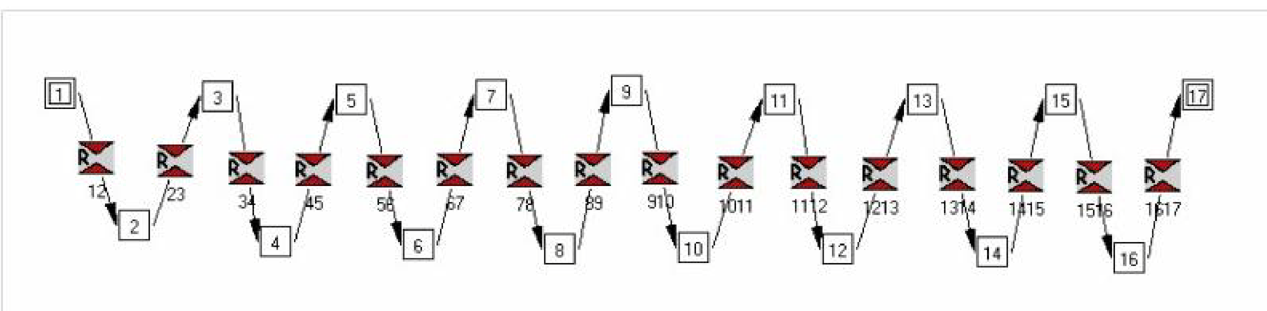
g: Same as f, further expansion occurs at outside nozzle

Example 3 - Simulation of Compressible Flow in a Converging-Diverging Nozzle

Detailed Schematic



VTASC Model



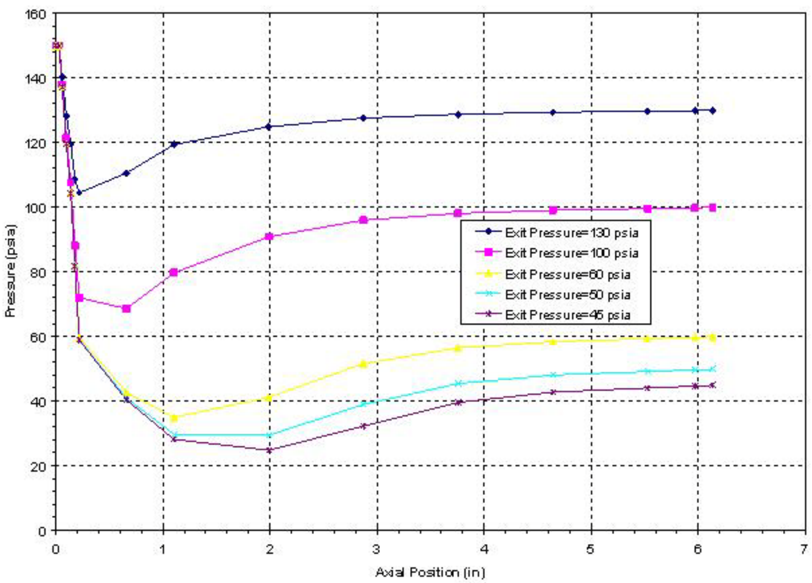
Example 3 – Converging-Diverging Nozzle

Boundary Conditions

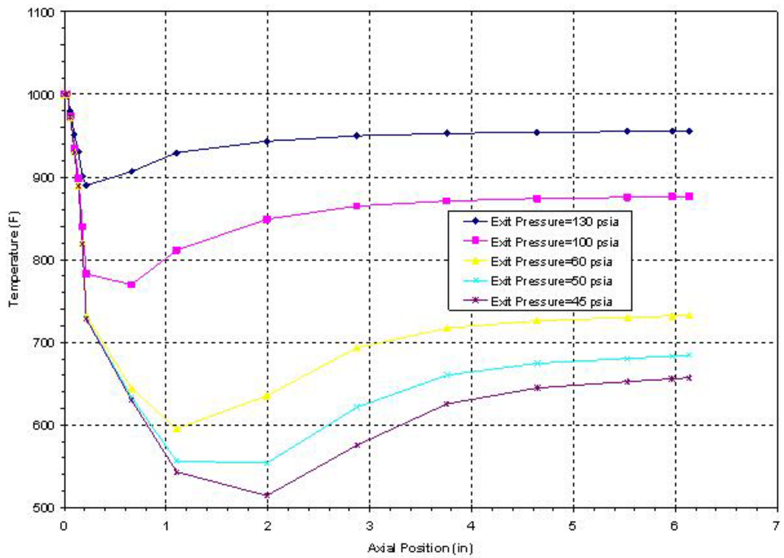
P_1 (psia)	T_1 (°F)	P_{17} (psia)	T_{17} (°F)
150	1000	134	1000
150	1000	100	1000
150	1000	60	1000
150	1000	50	1000
150	1000	45	1000

Predicted Mass Flow Rate with Varying Exit Pressure

P_{exit} (psia)	$\dot{m}$ (lbm/s)
134	0.279
100	0.329
60	0.336
50	0.337
45	0.337



Predicted Pressures for the Isentropic Steam Nozzle



Predicted Temperatures for the Isentropic Steam Nozzle

Example 3 – Converging-Diverging Nozzle

Comparison with Isentropic Solution

$$\dot{m} = A_{throat} P_{inlet} \sqrt{\frac{g_c \gamma}{R\, T_{inlet}} \left(\frac{2}{\gamma + 1} \right)^{\left(\frac{\gamma + 1}{\gamma - 1}\right)}}$$

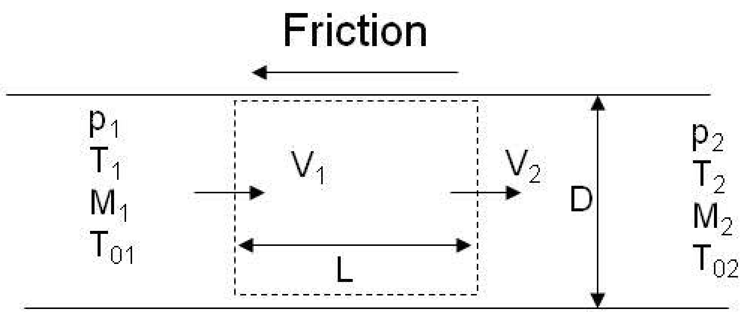
$$P_{Inlet} = P_{Static} \left(1 + \left(\frac{\gamma - 1}{2} \right) M^2 \right)^{\frac{\gamma}{\gamma - 1}}$$

$$P_{Inlet} = (150 \text{ psia}) \left(1 + \left(\frac{1.2809 - 1}{2} \right) (0.342)^2 \right)^{\frac{1.2809}{1.2809 - 1}} = 161.6 \text{ psia}$$

$$\dot{m} = \left(0.19012 \text{ in}^2\right) \left(161.6 \ \frac{\text{lbf}}{\text{in}^2}\right) \sqrt{\frac{32.174 \frac{\text{lbm} - \text{ft}}{\text{lbf} - \text{s}^2} \ (1.281)}{85.83 \frac{\text{lbf} - \text{ft}}{\text{lbm} - {}^0 \text{R}} 1460^0 \text{R}} \left(\frac{2}{1.281 + 1} \right)^{\frac{2.281}{0.281}}} = 0.327 \ \frac{\text{lbm}}{\text{s}}$$

GFSSP Predicted Mass Flowrate is 0.337 lbm/sec (within 3 %)

Flow with Friction (Fanno Flow)



APPENDIX 26.C
Fanno Flow Factors
($k = 1.4$)

M	p/p^*	$a/a^* = \rho^*/\rho$	T/T^*	p_0/p_0^*	$4fL/D$
0.00	∞	0.	1.200	∞	∞
0.05	21.903	0.0547	1.199	11.592	280.02
0.10	10.944	0.1094	1.197	5.822	66.922
0.12	9.116	0.131	1.1965	4.864	45.408
0.14	7.809	0.153	1.195	4.182	32.511
0.16	6.829	0.175	1.194	3.673	24.198
0.18	6.066	0.196	1.192	3.278	18.543
0.20	5.455	0.218	1.1905	2.963	14.533
0.25	4.355	0.272	1.185	2.403	8.483
0.30	3.619	0.3257	1.178	2.035	5.299
0.35	3.092	0.379	1.171	1.778	3.453
0.40	2.696	0.431	1.163	1.590	2.308

Fanno Curve

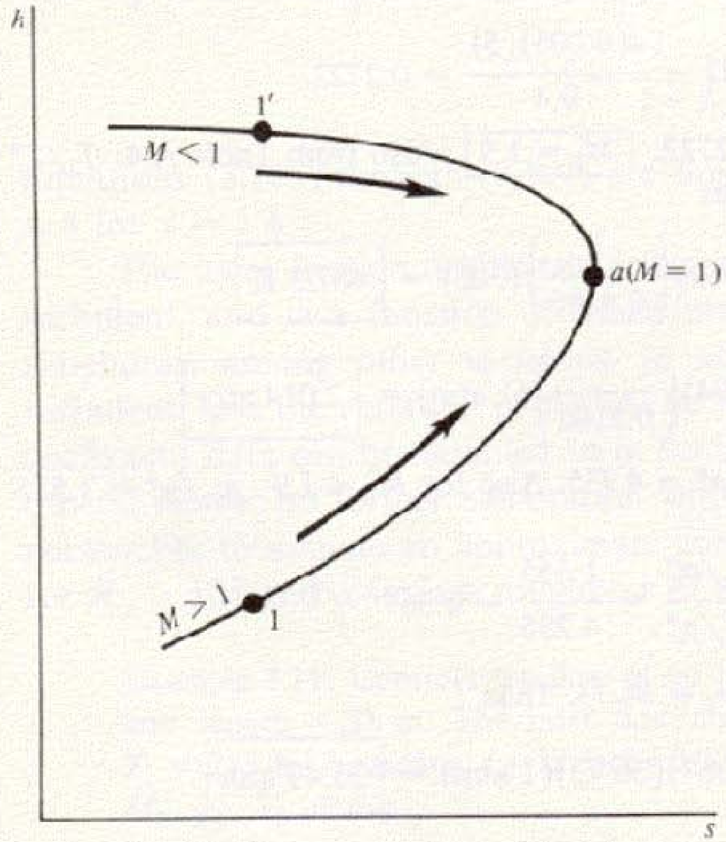
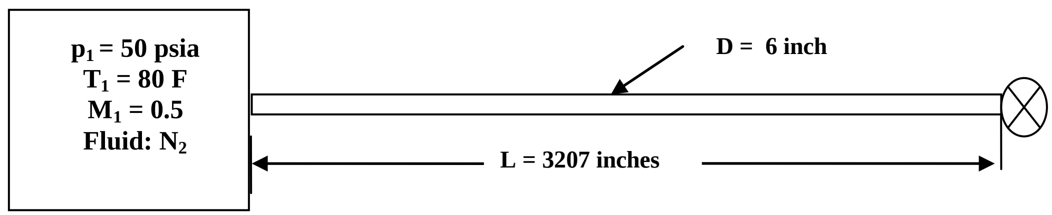
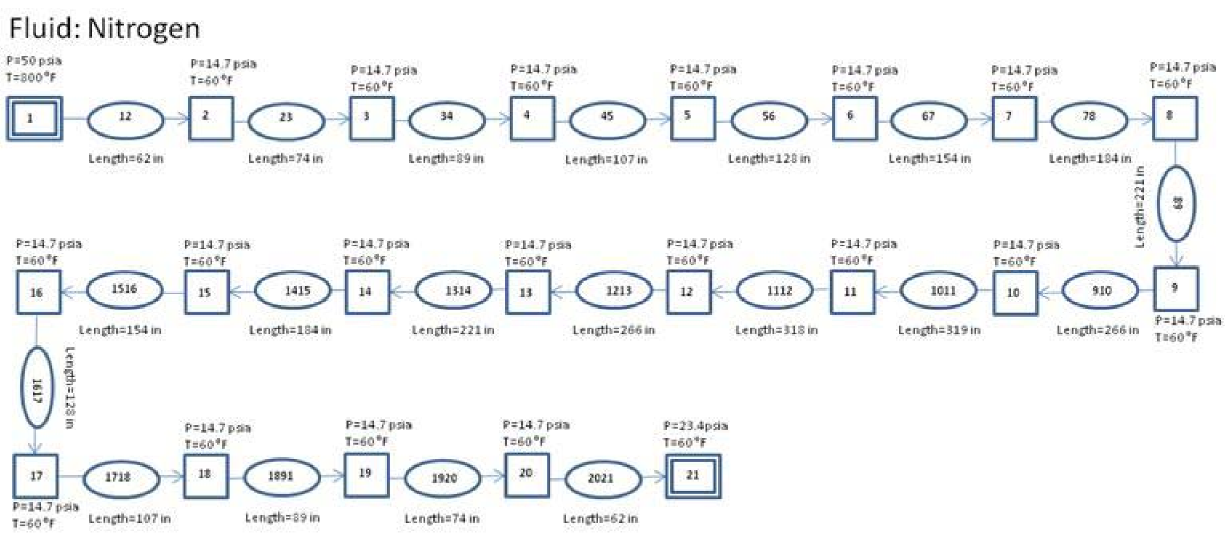


FIGURE 3.14
The Fanno curve.

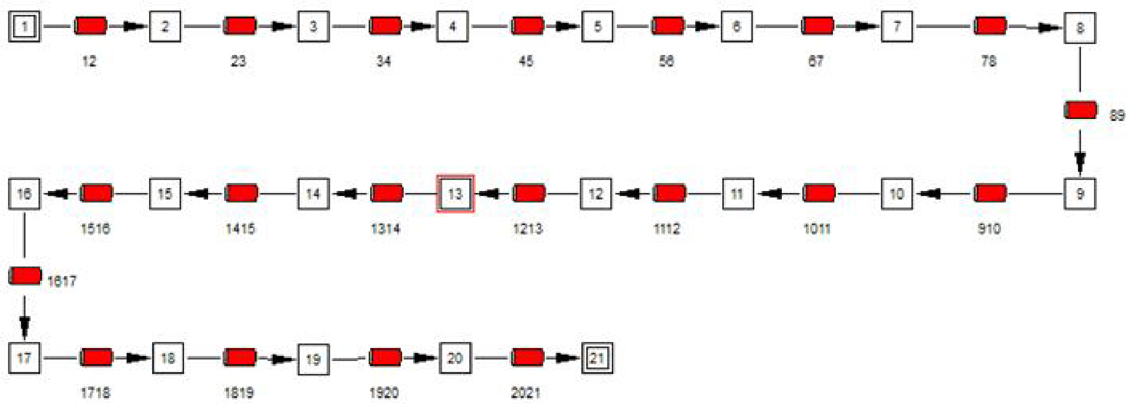
- Mach number increases for Subsonic Flow
- Flow can be choked in a long, thin pipe due to friction
- Mach number decreases for Supersonic Flow
- Entropy increases in both cases due to friction

Example 18 - Simulation of a Subsonic Fanno Flow





VTASC Model

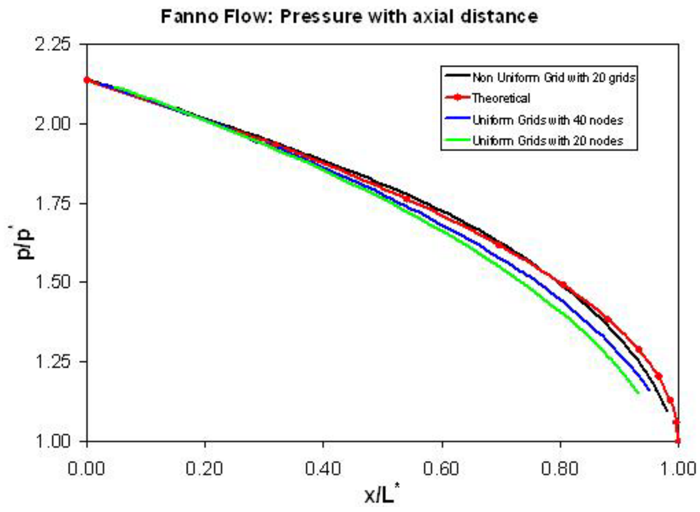


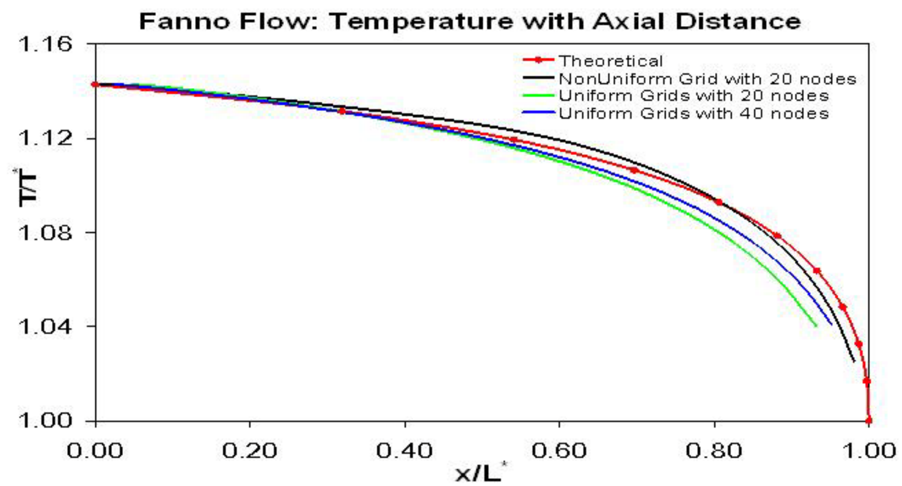
Boundary Conditions

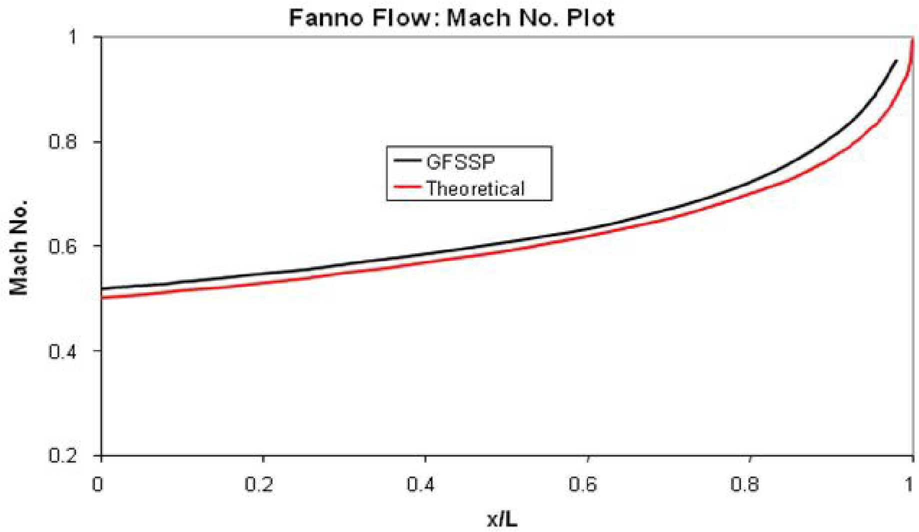
Boundary Node Number	Pressure (psia)	Temperature (ºF)
1	50	80
21	23.4	60

In the User Subroutine Friction Factor was set to 0.002, which was used for analytical solution

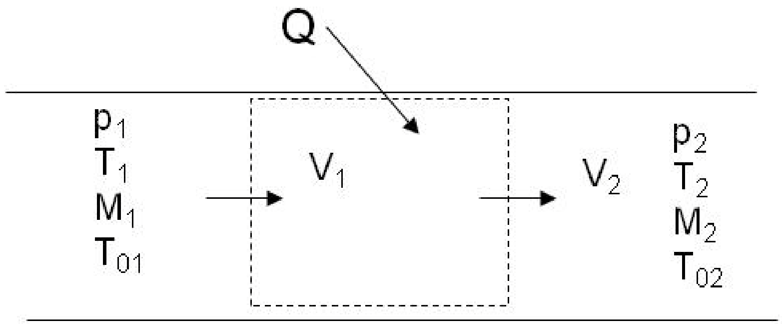
Comparison with Analytical Solution







Flow with Heat Transfer (Rayleigh Flow)



APPENDIX 26.D
Rayleigh Flow Factors
($k = 1.4$)

M	p/p^*	p_0/p_0^*	T/T^*	T_0/T_0^*	$a/a^* = \rho^*/\rho$
0.00	2.400	1.268	0.000	0.000	0.000
0.05	2.392	1.266	0.0143	0.0119	0.00598
0.10	2.367	1.259	0.056	0.0468	0.0237
0.12	2.353	1.255	0.079	0.0667	0.0339
0.14	2.336	1.251	0.107	0.089	0.0458
0.16	2.317	1.246	0.137	0.115	0.0593
0.18	2.296	1.241	0.1708	0.143	0.0744
0.20	2.273	1.235	0.2066	0.1735	0.091
0.25	2.207	1.218	0.304	0.257	0.138
0.30	2.131	1.198	0.409	0.3468	0.192
0.35	2.048	1.178	0.514	0.439	0.251

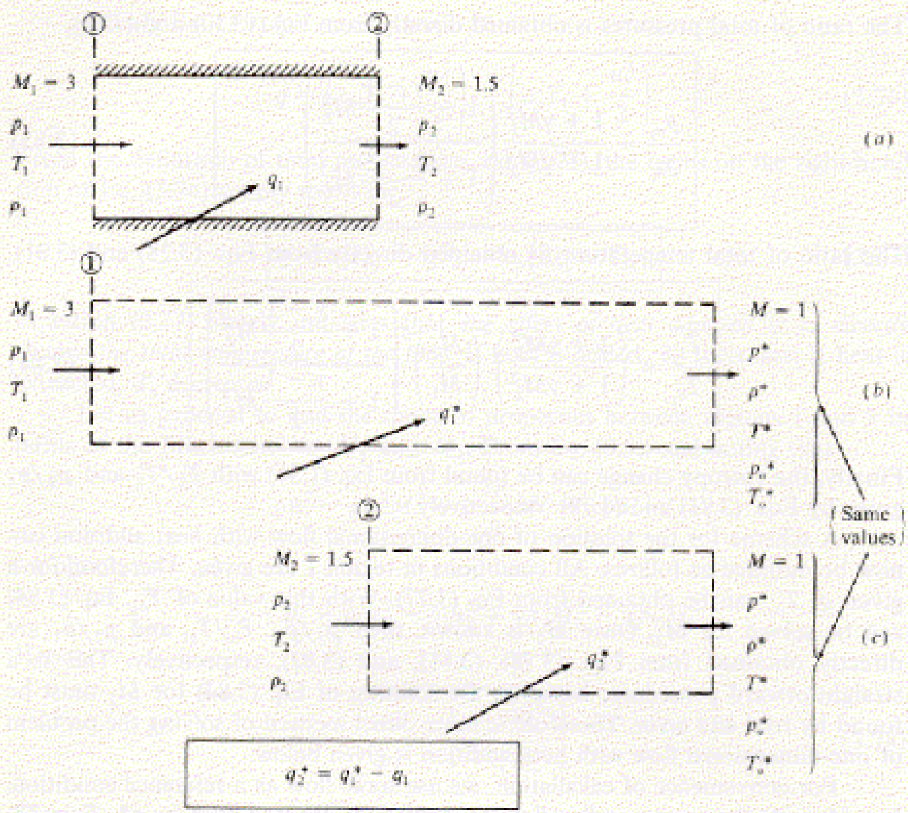


FIGURE 3.11
Illustration of the meaning of the starred quantities at Mach 1 for one-dimensional flow with heat addition.

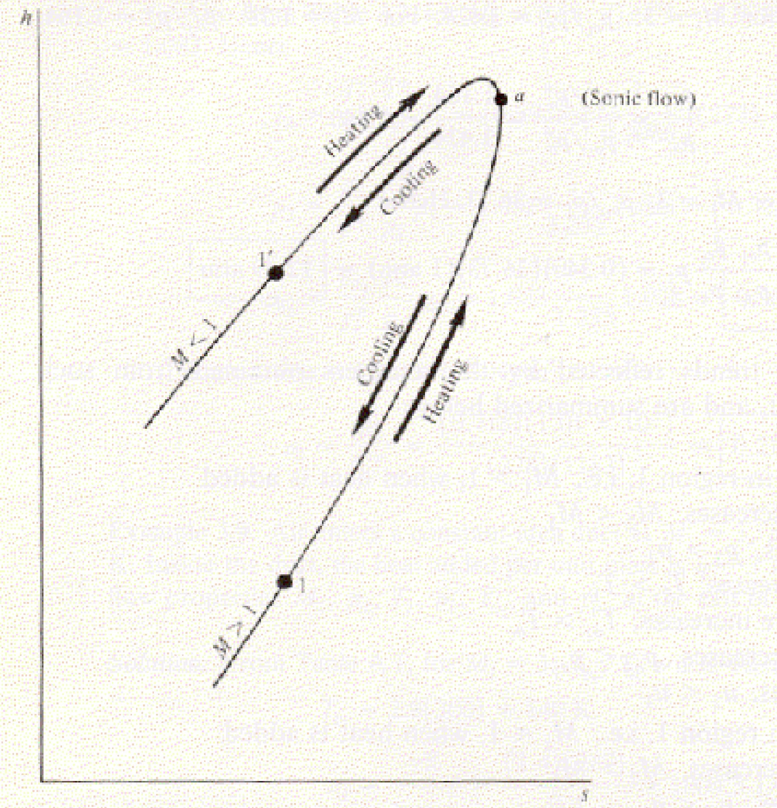
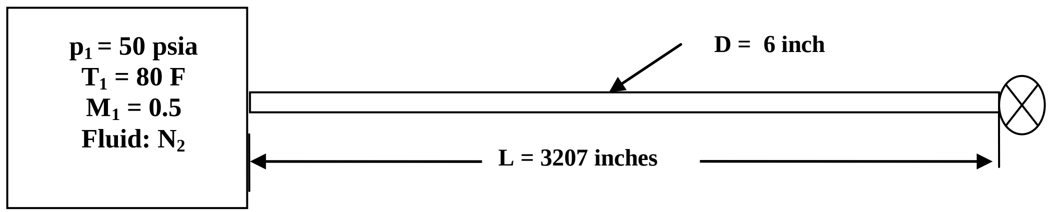
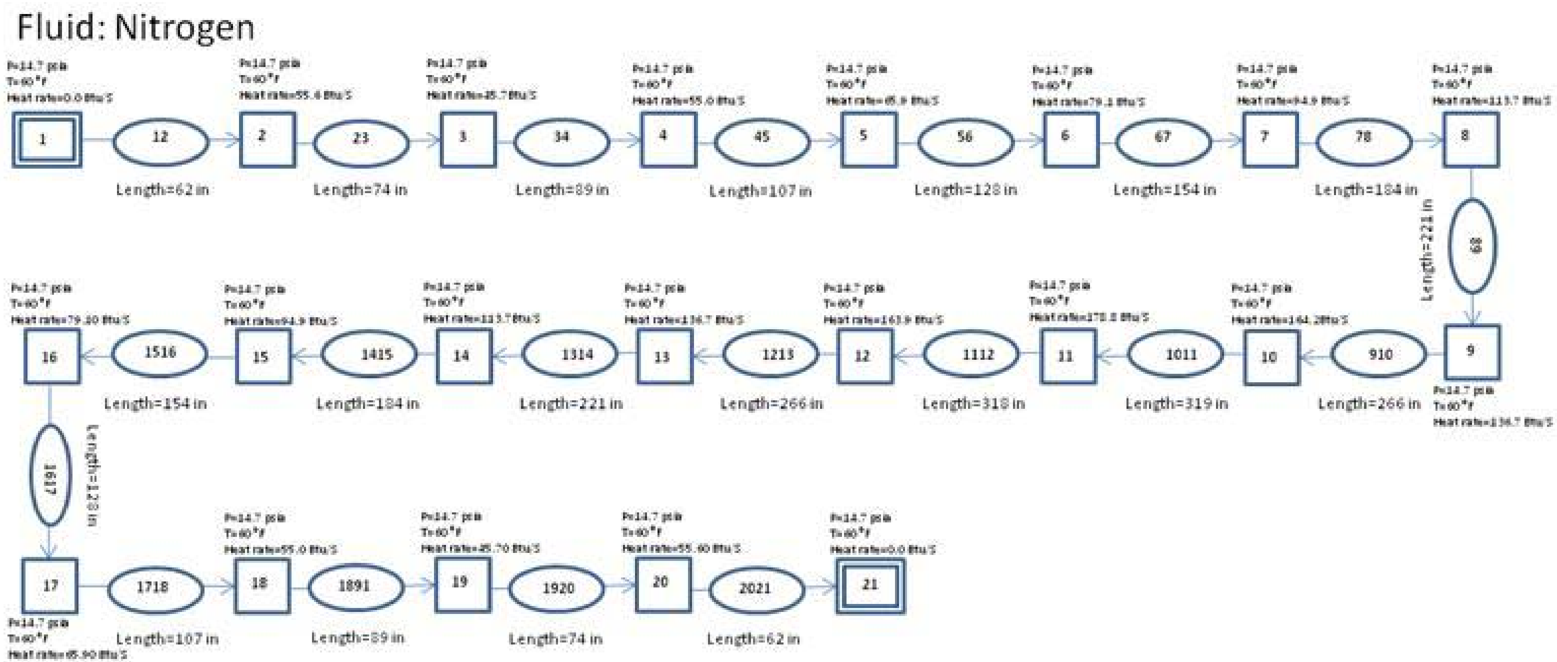


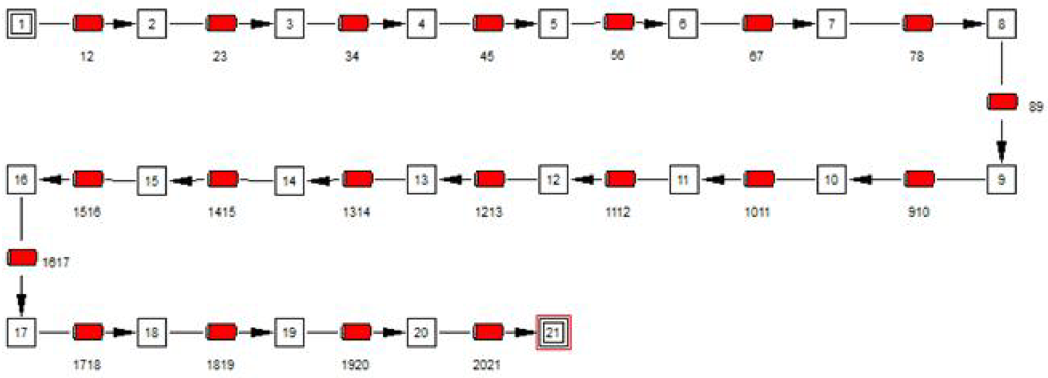
FIGURE 3.12
The Rayleigh curve.

Example 19 - Simulation of a Rayleigh Flow





VTASC Model

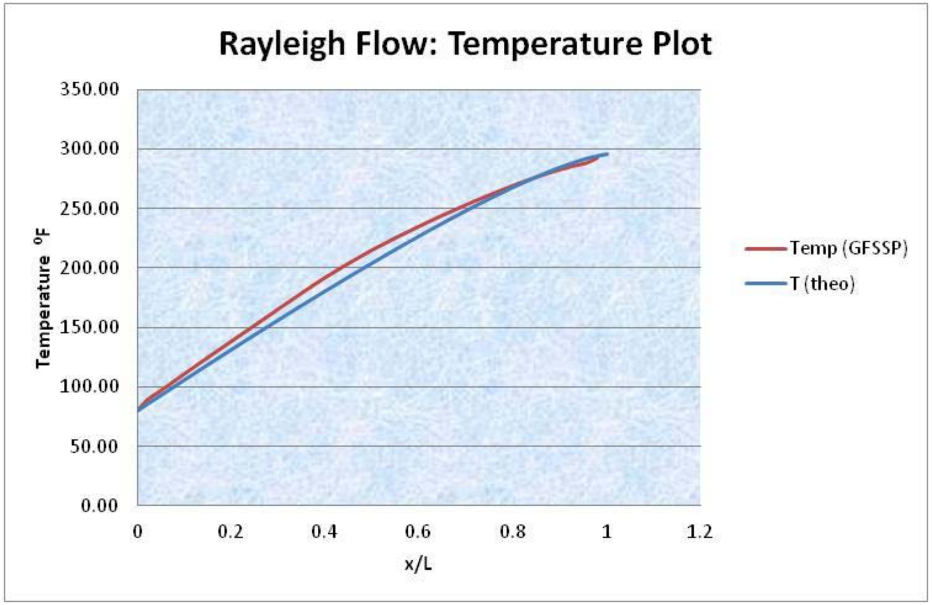


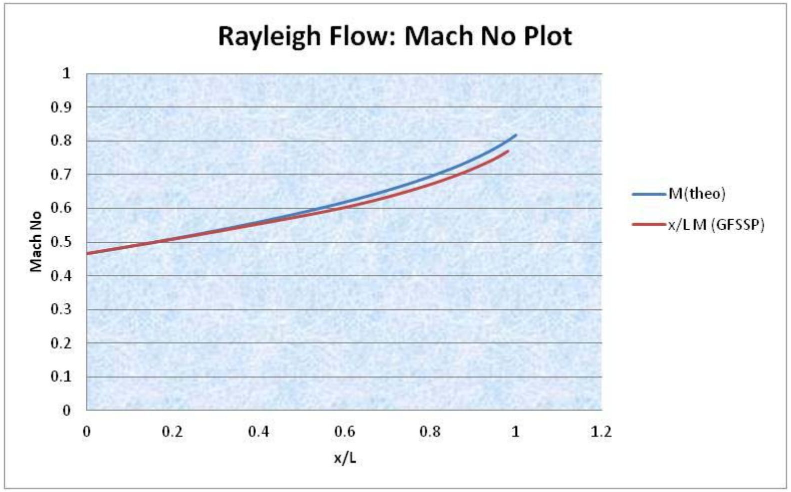
Boundary Conditions

Boundary Node Number	Pressure (psia)	Temperature (ºF)
1	50	80
21	35	40

In the User Subroutine Friction Factor was set to zero to eliminate frictional effect

Comparison with Analytical Solution



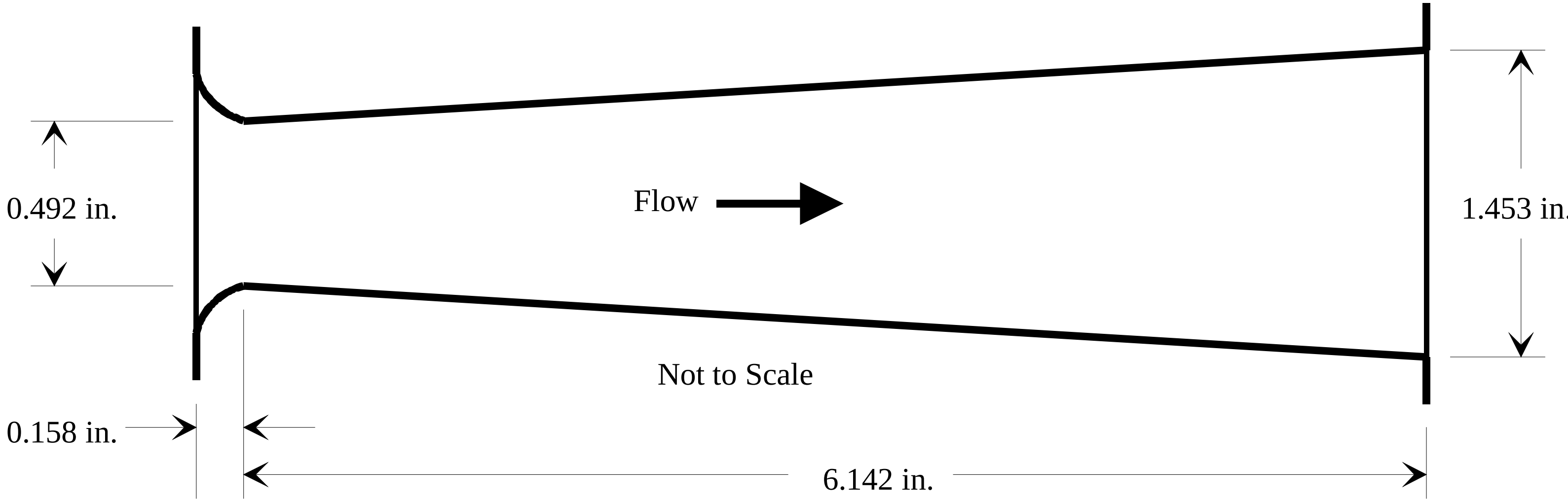


CONCLUSIONS

- GFSSP can model subsonic compressible flows for ideal and real gas
- For compressible flows, inertia term in the momentum conservation equation needs to be activated to account for fluid acceleration due to large density and area change
- GFSSP predictions for compressible flows have been validated by comparing with analytical solutions of three classical compressible flow problems:
  - Converging-Diverging Nnozzle
  - Subsonic Flow with Friction (Fanno Flow)
  - Subsonic Flow with Heat Transfer (Rayleigh Flow)

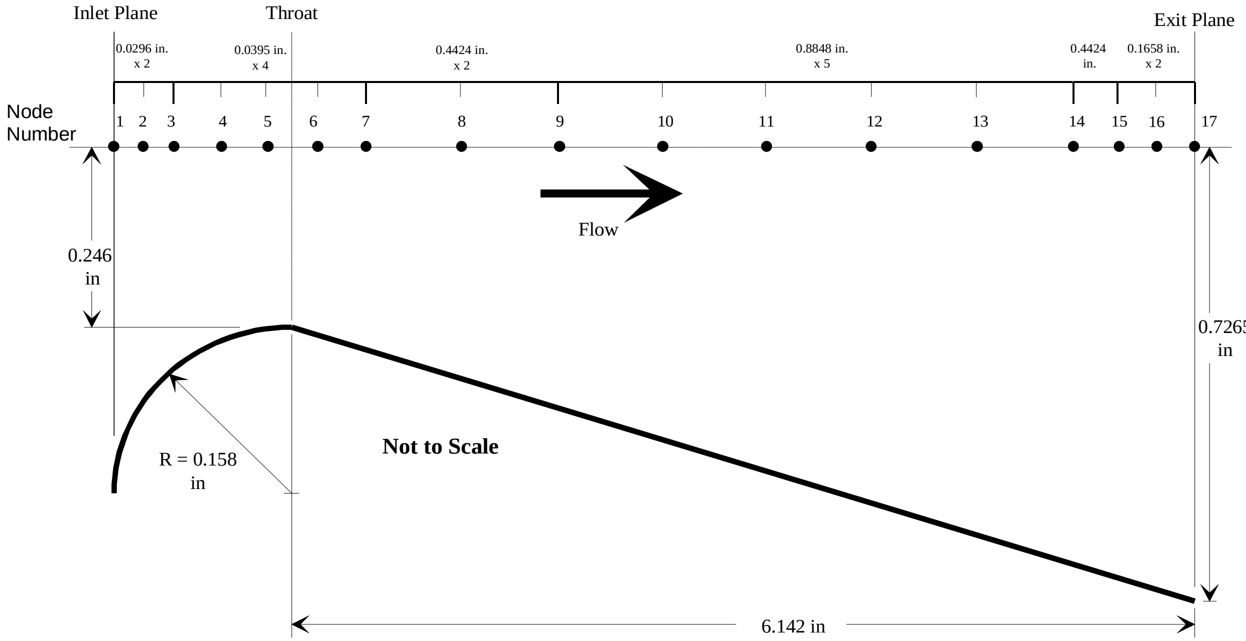
Tutorial – 1

Simulation of Compressible Flow in a Converging-Diverging Nozzle



Converging-Diverging Nozzle Geometry

Marshall Space Flight Center
GFSSP Training Course



Problem Considered:

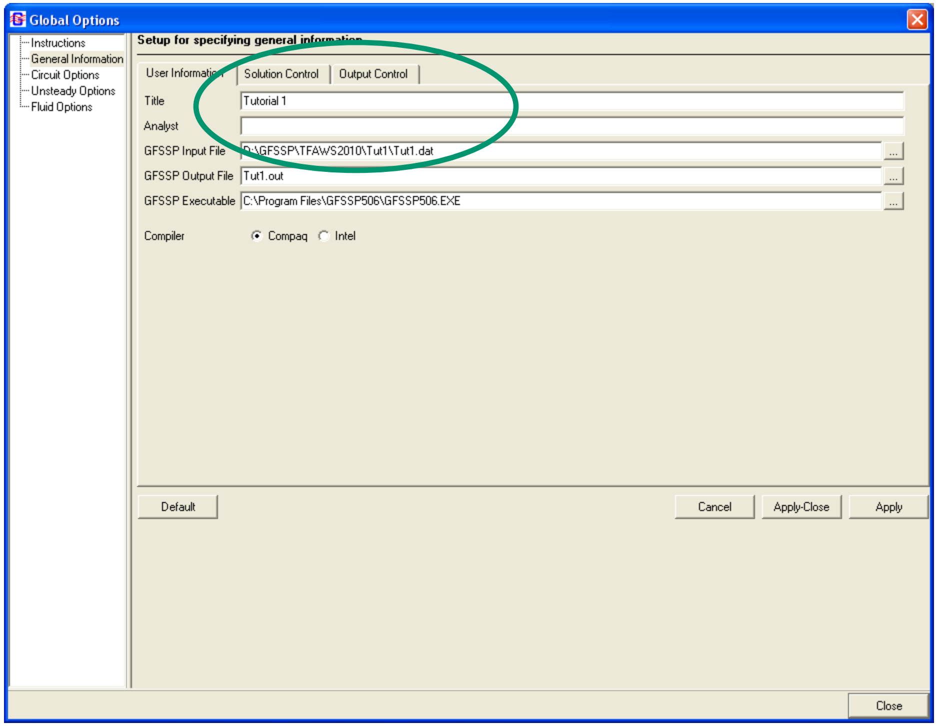
- One-dimensional pressure and temperature distribution
- Flow rates in subsonic and supersonic flow

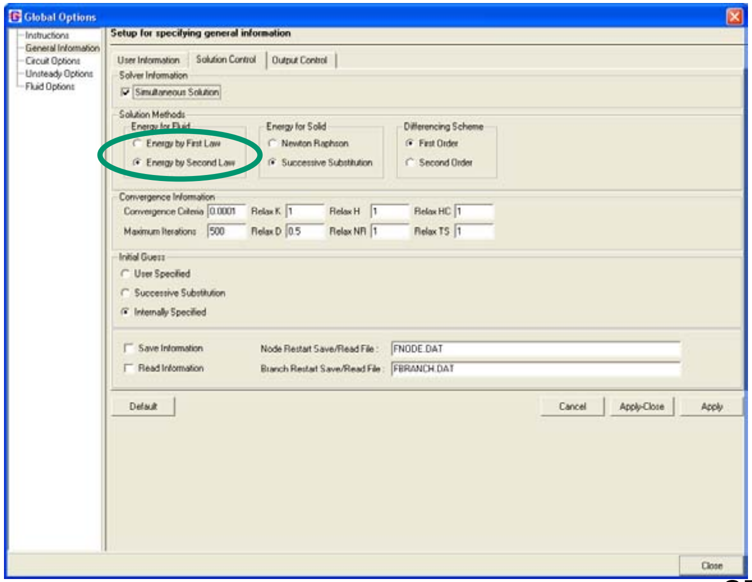
(This is a simplified version of Example 3 in the GFSSP User's Manual)

Program Options

Input data file : tut1.dat

Output data file : tut1.out



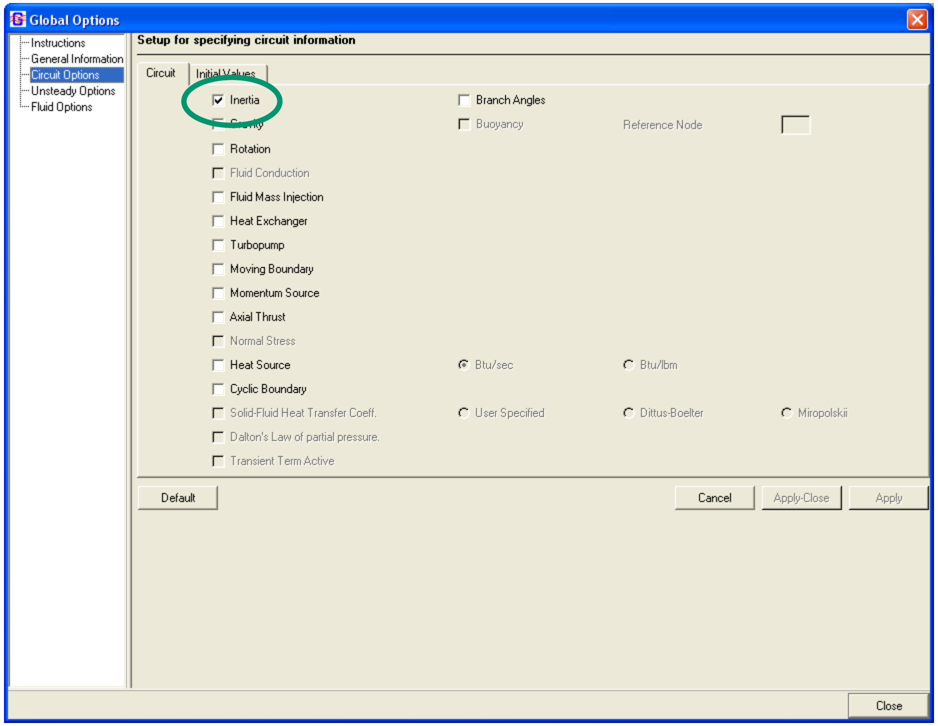


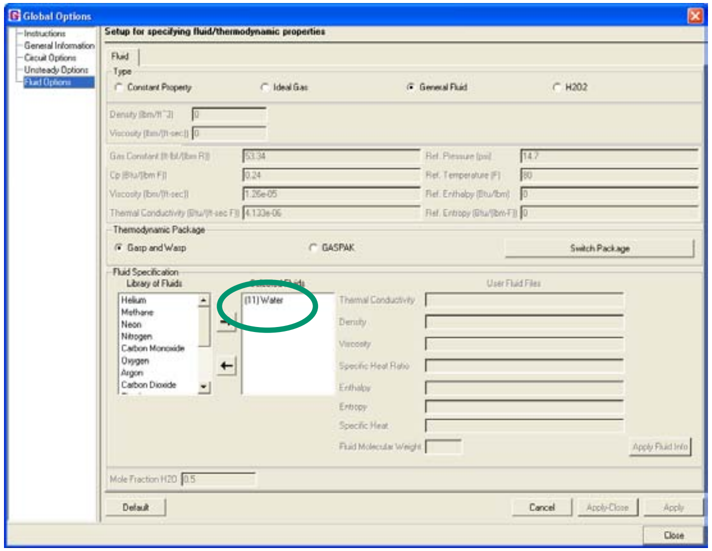
Second Law Option

Program Options (cont.)

Activate INERTIA option globally

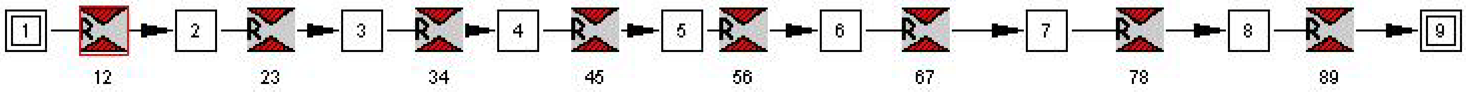
- This only means that the inertia term becomes <u>selectable</u> in the branches



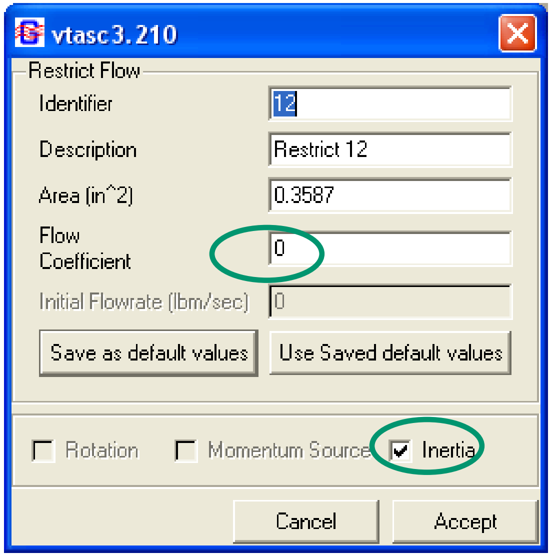


Fluid is steam (water)

Branch Geometry

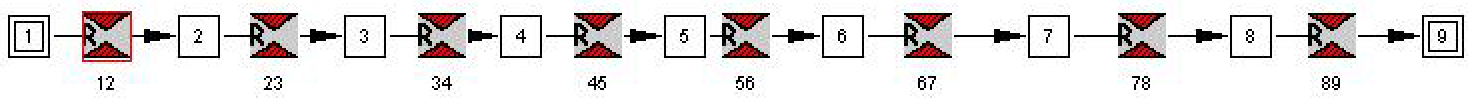


Branch	Area (in²)
12	0.3587
23	0.2243
34	0.1901
45	0.2255
56	0.3948
67	0.7633
78	1.2520
89	1.6286

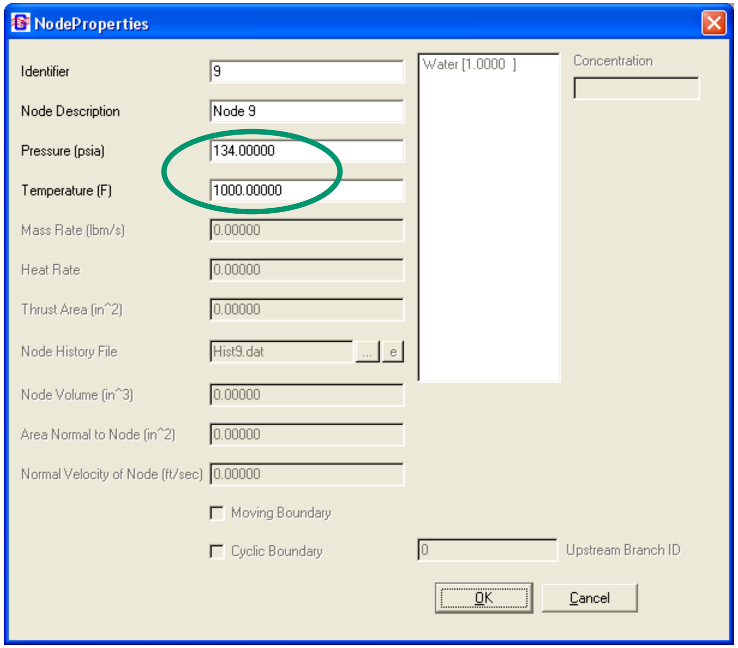


- Set restriction flow coefficient to 0.0 (isentropic)
- Activate inertia term in each branch

Boundary Conditions



- Run five cases
- Adjust downstream pressure for each case



Run	P1 (psia)	T1 (F)	P9 (psia)	T9 (F)
1	150	1000	134	1000
2	150	1000	100	1000
3	150	1000	60	1000
4	150	1000	50	1000
5	150	1000	45	1000

Results of Parametric Computations

Determine the choked flowrate

Run	P9 (psia)	F (lb_m/s)
1	134	
2	100	
3	60	
4	50	
5	45	

- How does the choked flowrate compare to the hand-calculated value of 0.327 lb_m/s?

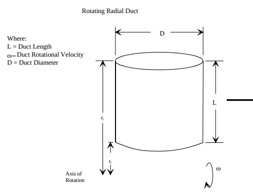
$$\dot{m} = A_{throat} P_{inlet} \sqrt{\frac{g_c \gamma}{R\, T_{inlet}} \left(\frac{2}{\gamma+1}\right)^{\left(\frac{\gamma+1}{\gamma-1}\right)}} = \left(0.19012\, in^2\right)\left(161.6\, \frac{lbf}{in^2}\right)\sqrt{\frac{32.174 \frac{lbm-ft}{lbf-s^2}(1.281)}{85.83 \frac{lbf-ft}{lbm-{}^0R} 1460^0 R}\left(\frac{2}{1.281+1}\right)^{\frac{2.281}{0.281}}} = 0.327\, \frac{lbm}{s}$$

- How does the throat temperature (T4) compare to the hand-calculated value of 799 °F?

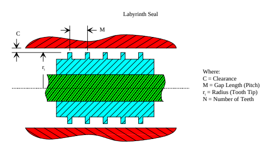
Study of the Results

- Study *tut1.out* to note the following facts:
  - Pressure is decreasing from inlet to throat and increases from throat to exit in subsonic flow (Exit Pressure = 134 psia)
  - Temperature follows a similar trend; temperature changes due to expansion and compression
  - Entropy remains constant due to isentropic assumption
  - With lower exit pressure, flow becomes supersonic in the diverging part and becomes subsonic with the formation of shock wave
  - Flowrate remains constant with exit pressure once choked flow rate is reached

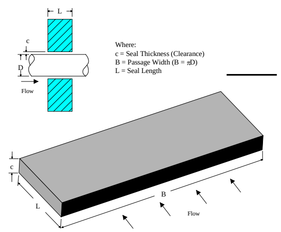
RESISTANCE & FLUID OPTIONS



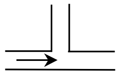
10. Rotating Radial Duct



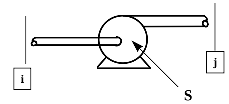
11. Labyrinth Seal



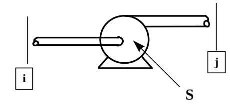
12. Face Seal



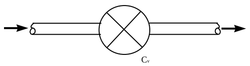
13. Common Fittings & Valves



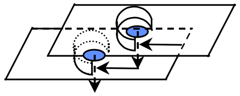
14. Pump Characteristics



15. Pump Power



16. Valve with Given Cv



17. Viscojet

RESISTANCE OPTIONS

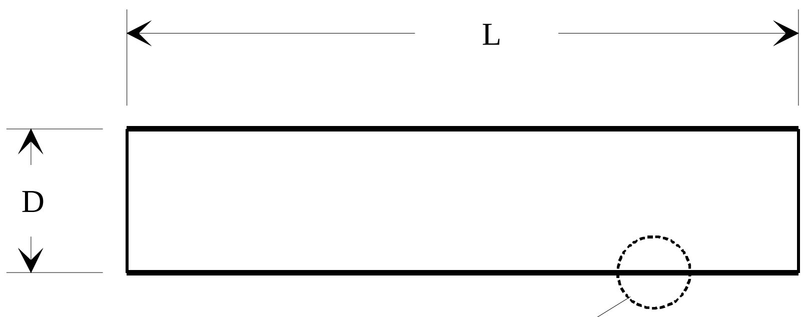
GFSSP can model flow in the following passages:

- Circular and non-circular pipes/ducts
- Flow through a restriction
- Thick and thin orifice
- Square expansion and reduction
- Rotating radial and annular ducts
- Labyrinth Seal
- Flow between closely spaced parallel plates (Face Seals)
- Common fittings and valves
- Pump characteristics
- Pump power
- Joule-Thompson device
- Control Valve
- Heat Exchanger Core
- Parallel Tube
- Compressible Orifice
- Fixed Flow
- Two-Dimensional Cartesian Grid

RESISTANCE OPTION 1

Pipe Resistance Option Parameters

PIPE FLOW



DETAIL A

DETAIL A

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

Flow Resistance Factor Calculated from:

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

For Re < 2300, Friction Factor is:

$$f = \frac{64}{\mathrm{Re}_D}$$

For Re > 2300, Friction Factor calculated from the Colebrook Equation:

$$\frac{1}{\sqrt{f}} = -2\log\left[\frac{\varepsilon}{3.7D} + \frac{2.51}{\mathrm{Re}\sqrt{f}}\right]$$

RESISTANCE OPTION 2

FLOW THROUGH A RESTRICTION

$$K_f = \frac{1}{2 g_c \rho_u C_L^2 A^2}$$

In Classical Fluid Mechanics, Head Loss, H, is Expressed as:

$$\Delta H = K \frac{u^2}{2g}$$

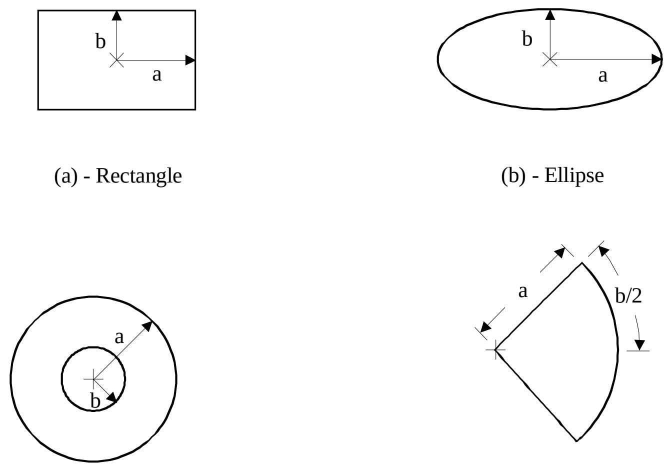
K and CL are Related by: $$C_L = \frac{1}{\sqrt{K}}$$

If the User sets C_L to 0, the code will set K_f to 0 (invisid flow through the branch)

RESISTANCE OPTION 3

NON-CIRCULAR DUCT

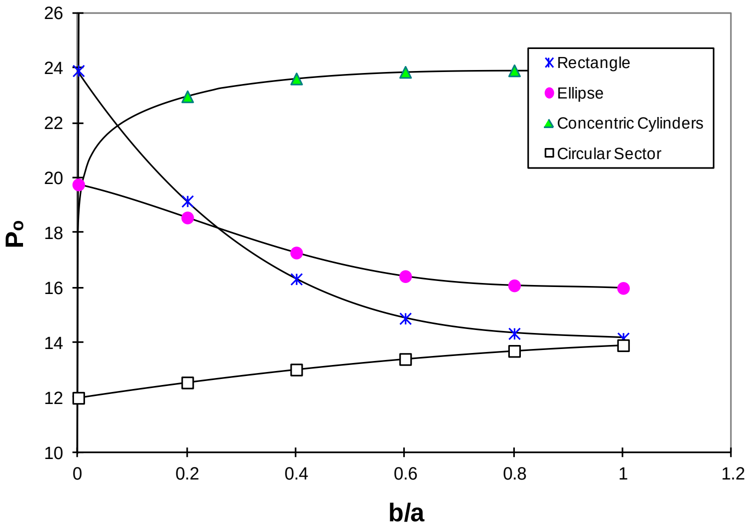
Four cross-sections currently considered:



(a) - Rectangle (b) - Ellipse

(c) - Concentric Annulus (d) - Circular Sector

Poiseuille Number Relationship For Laminar Flow



$$Po = C_f Re$$

RESISTANCE OPTION 3 - CONTINUED

NON-CIRCULAR DUCT - CONTINUED

Reynolds Number based upon Hydraulic Diameter D_h (= 4A/P)

Laminar Flow **(Re_{D_h} < 2300)**

$$f = \frac{4Po}{Re_{D_h}}$$

Turbulent Flow **(Re_{D_h} > 2300)**

1. Compute Effective Diameter

$$D_{eff} = \frac{16D_h}{Po}$$

2. Compute Effective Reynolds Number

$$Re_{eff} = \frac{\dot{m}}{\mu}\frac{D_{eff}}{A}$$

3. Use Effective Diameter & Reynolds Number in Colebrook Equation:

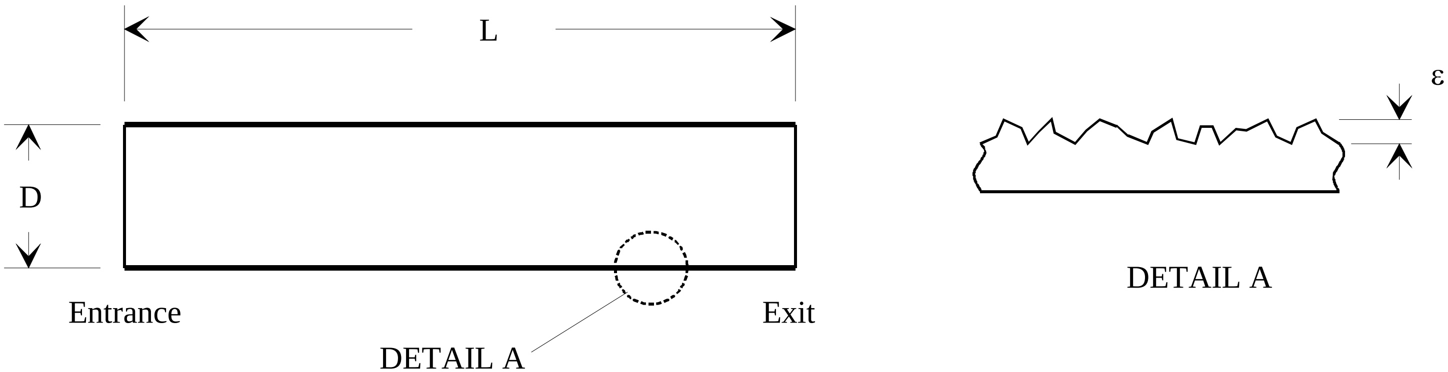
$$\frac{1}{\sqrt{f}} = -2\log\left[\frac{\varepsilon}{3.7D} + \frac{2.51}{Re\sqrt{f}}\right]$$

Flow Resistance Factor Calculated from: $K_f = \frac{8fL}{\rho_u \pi^2 D_h^5 g_c}$

RESISTANCE OPTION 4

PIPE FLOW WITH ENTRANCE AND EXIT LOSS

Pipe With Entrance and/or Exit Loss



Where:

D = Pipe Diameter
L = Pipe Length
ε = Absolute Roughness
K_i = Entrance Loss Coefficient
K_e = Exit Loss Coefficient

Flow Resistance Factor:

$$K_f = \frac{8K_i}{\rho_u \pi^2 D^4 g_c} + \frac{8 f L}{\rho_u \pi^2 D^5 g_c} + \frac{8K_e}{\rho_u \pi^2 D^4 g_c}$$

RESISTANCE OPTION 5

THIN SHARP ORIFICE

Flow Resistance Factor:

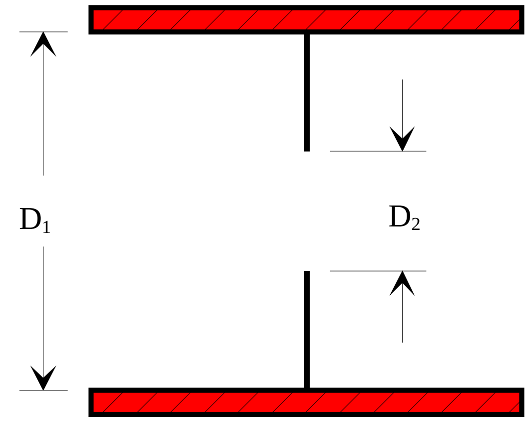
$$K_f = \frac{K_1}{2 g_c \rho_u A^2}$$

Where:

$$K_1 = \left[2.72 + \left(\frac{D_2}{D_1}\right)^2 \left(\frac{120}{Re_{D_1}} - 1\right)\right]\left[1 - \left(\frac{D_2}{D_1}\right)^2\right]\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right] \quad \text{for } Re_{D_1} \le 2{,}500$$

$$K_1 = \left[2.72 - \left(\frac{D_2}{D_1}\right)^2 \left(\frac{4000}{Re_{D_1}}\right)\right]\left[1 - \left(\frac{D_2}{D_1}\right)^2\right]\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right] \quad \text{for } Re_{D_1} > 2{,}500$$

Thin Sharp Orifice



Where:
D_1 = Pipe Diameter
D_2 = Orifice Throat Diameter

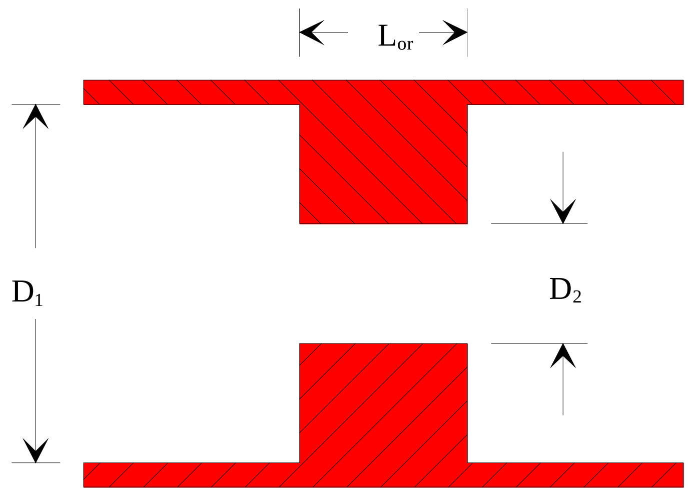
RESISTANCE OPTION 6

THICK ORIFICE

Flow Resistance Factor:

$$K_f = \frac{K_1}{2 g_c \rho_u A^2}$$

Where:



Where:
D_1 = Pipe Diameter
D_2 = Orifice Throat Diameter
L_{or} = Orifice Length

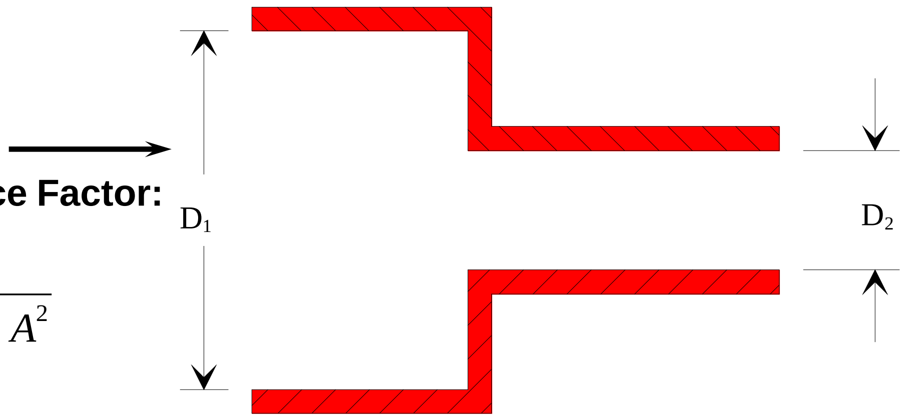
$$K_1 = \left[2.72 + \left(\frac{D_2}{D_1}\right)^2 \left(\frac{120}{Re_{D_1}} - 1\right)\right]\left[1 - \left(\frac{D_2}{D_1}\right)^2\right]\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right]\left[0.584 + \frac{0.0936}{\left(L_{or}/D_2\right)^{1.5} + 0.225}\right] \quad \text{for } Re_{D_1} \le 2{,}500$$

$$K_1 = \left[2.72 - \left(\frac{D_2}{D_1}\right)^2 \left(\frac{4000}{Re_{D_1}}\right)\right]\left[1 - \left(\frac{D_2}{D_1}\right)^2\right]\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right]\left[0.584 + \frac{0.0936}{\left(L_{or}/D_2\right)^{1.5} + 0.225}\right] \quad \text{for } Re_{D_1} > 2{,}500$$

RESISTANCE OPTION 7

SQUARE REDUCTION

Square Reduction



Where:
D_1 = Upstream Pipe Diameter
D_2 = Downstream Pipe Diameter

Flow Resistance Factor:

$$K_f = \frac{K_1}{2 g_c \rho_u A^2}$$

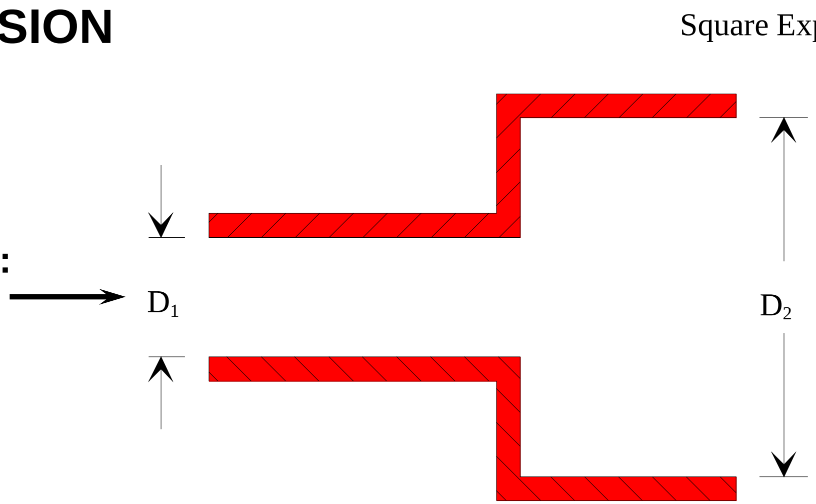
Where:

$$K_1 = \left[1.2 + \frac{160}{Re_{D_1}}\right]\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right] \quad \text{for } Re_{D_1} \le 2{,}500$$

$$K_1 = \left[0.6 + 0.48f\right]\left(\frac{D_1}{D_2}\right)^2\left[\left(\frac{D_1}{D_2}\right)^2 - 1\right]^2 \quad \text{for } Re_{D_1} > 2{,}500$$

RESISTANCE OPTION 8

SQUARE EXPANSION



Where:
D_1 = Upstream Pipe Diameter
D_2 = Downstream Pipe Diameter

Flow Resistance Factor:

$$K_f = \frac{K_1}{2 g_c \rho_u A^2}$$

Where:

$$K_1 = 2\left[1 - \left(\frac{D_1}{D_2}\right)^4\right] \quad \text{for } Re_{D_1} \le 4{,}000$$

$$K_1 = \left[1 + 0.8f\right]\left[1 - \left(\frac{D_1}{D_2}\right)^2\right]^2 \quad \text{for } Re_{D_1} > 4{,}000$$

Version 603 has the capability to switch between Option 7 and 8 depending upon the flow direction

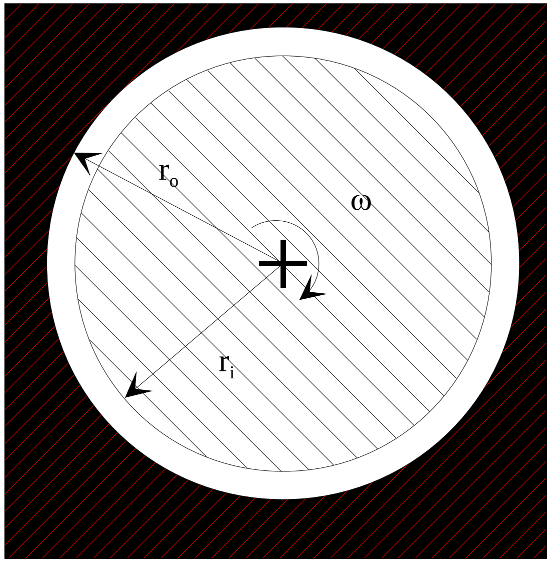
RESISTANCE OPTION 9

ROTATING ANNULAR DUCT

Rotating Annular Duct

Flow Resistance Factor:

$$K_f = \frac{f\ L}{\rho_u \pi^2 A^2 g_c (r_o - r_i)}$$



Where:
L = Duct Length (Perpendicular to Page)
b = Duct Wall Thickness ($b = r_o - r_i$)
ω = Duct Rotational Velocity
r_i = Duct Inner Radius
r_o = Duct Outer Radius

Where:

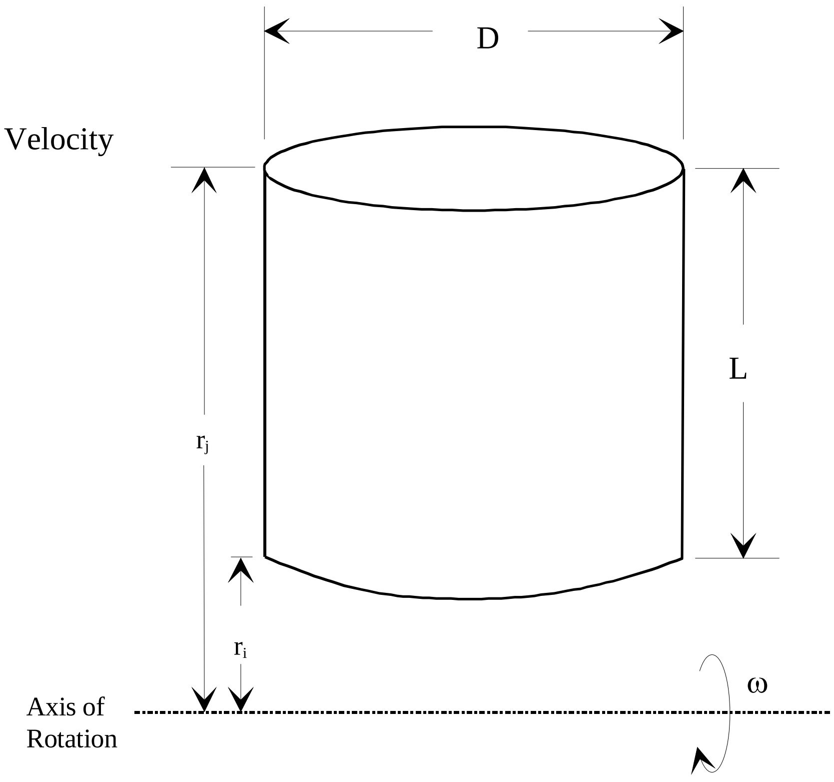
$$\frac{f}{f_{0T}} = \left[1 + 0.7656\left(\frac{\omega\, r_i}{2u}\right)^2\right]^{0.38}$$

$$f_{0T} = 0.077(Ru)^{-0.24}, \qquad Ru = \frac{\rho_u u 2(r_o - r_i)}{\mu}$$

RESISTANCE OPTION 10

ROTATING RADIAL DUCT

Where:
L = Duct Length
ω = Duct Rotational Velocity
D = Duct Diameter



Flow Resistance Factor:

$$K_f = \frac{8\, f\, L}{\rho_u \pi^2 D^5 g_c}$$

Where:

$$\frac{f}{f_{0T}} = 0.942 + 0.058\left[\left(\frac{\omega D}{u}\right)\left(\frac{\omega D^2}{\nu}\right)\right]^{0.282}$$

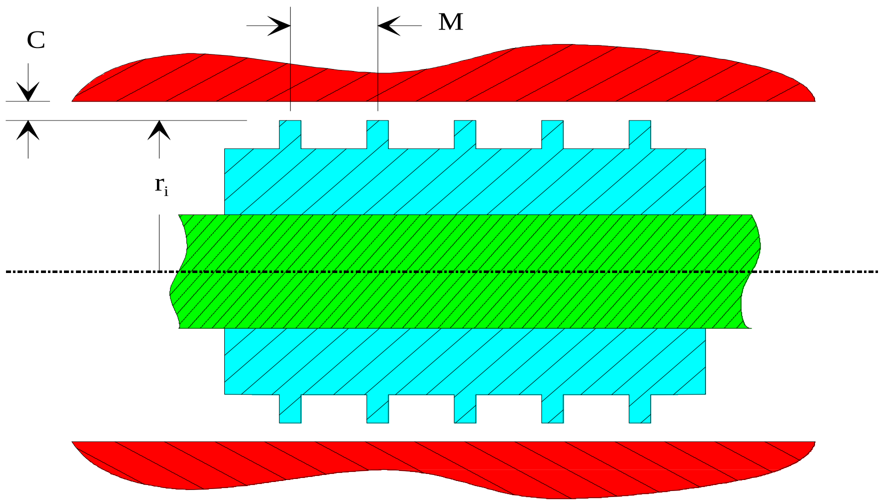
$$f_{0T} = 0.0791\,(Ru)^{-0.25}$$

$$Ru = \frac{\rho_u u\, 2(r_o - r_i)}{\mu}$$

RESISTANCE OPTION 11

LABYRINTH SEAL

Labyrinth Seal



Where:
C = Clearance
M = Gap Length (Pitch)
r_i = Radius (Tooth Tip)
N = Number of Teeth
α = Step Seal Factor (~0.9)

Flow Resistance Factor (Modified Dodge Eqn):

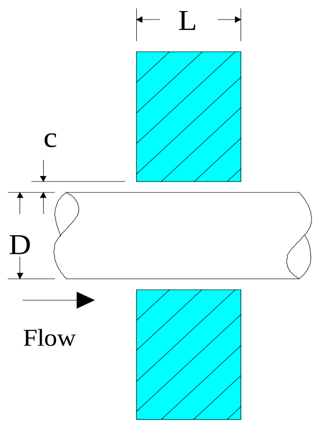
$$K_f = \frac{\left(\frac{1}{\varepsilon^2} + 0.5\right) N + 1.5}{2 g_c \rho_u \alpha^2 A^2}$$

Where:

$$\varepsilon = \sqrt{\frac{1}{\left\{1 - \left[\frac{C(N-1)/M}{N(\{C/M\} - 0.02)}\right]\right\}}}$$

RESISTANCE OPTION 12

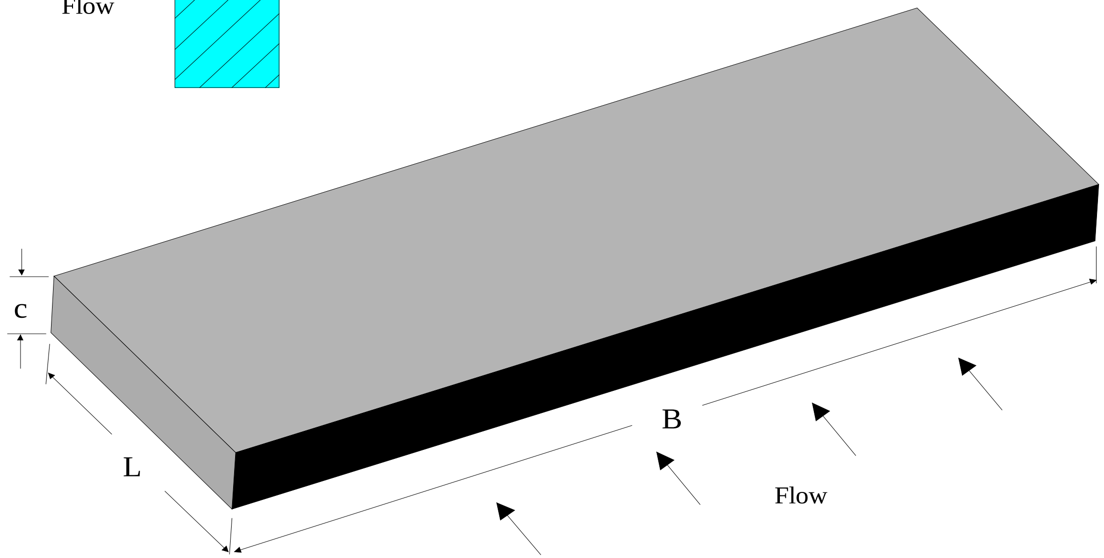
FACE SEAL



Where:
c = Seal Thickness (Clearance)
B = Passage Width (B = πD)
L = Seal Length

Flow Resistance Factor:

$$K_f = \frac{12 \mu L \rho}{\pi \, g_c \, D c^3 |\dot{m}|}$$



RESISTANCE OPTION 13

COMMON FITTINGS AND VALVES

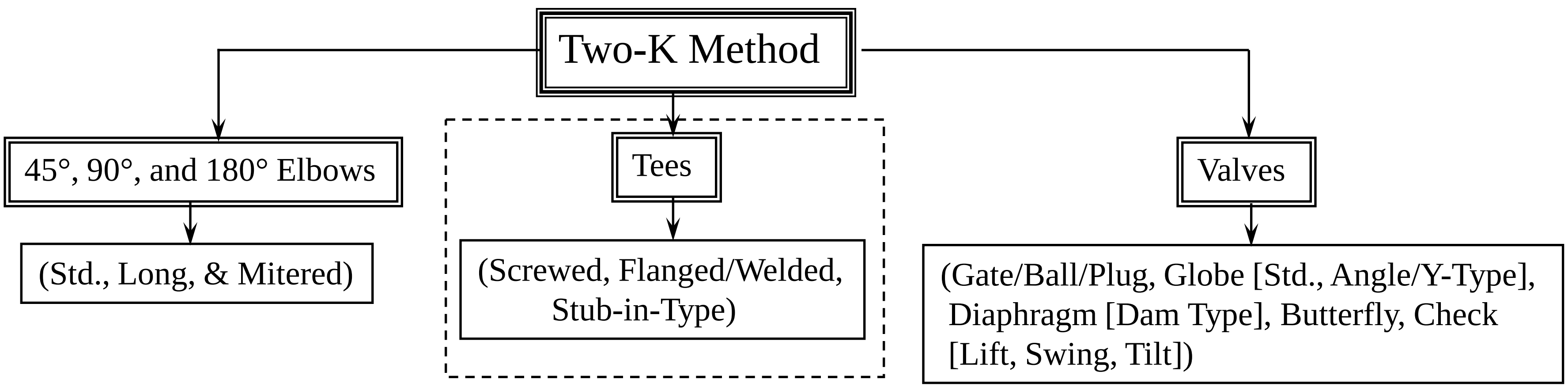
Flow Resistance Factor:

$$K_f = \frac{K_1/\mathrm{Re} + K_\infty(1 + 1/D)}{2 g_c \rho_u A^2}$$

Where:

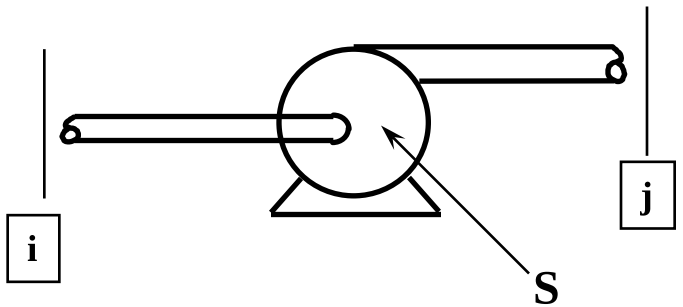
K_1 = K for the fitting at Re = 1;
K_∞ = K for the fitting at Re = ∞ ; (K_2 in GFSSP)
D = Internal diameter of attached pipe, in.

The Following Common Fittings and Valves Can Be Modeled using This Option:



RESISTANCE OPTION 14

PUMP CHARACTERISTICS

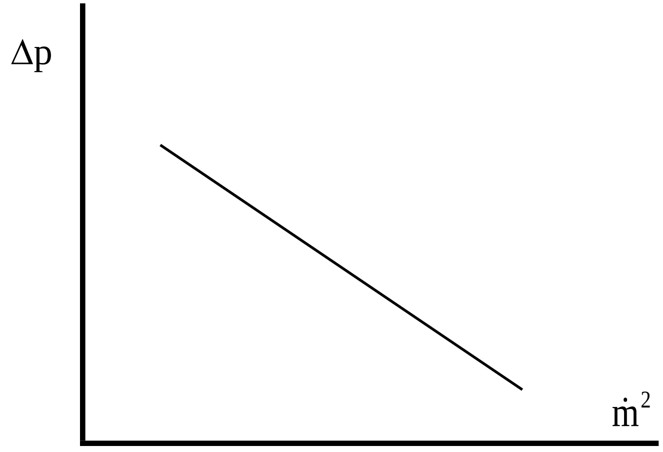


This Option Considers the Branch as a Pump with Given Characteristics. The Pump Characteristics are Expressed in the Pressure Rise:

$$\Delta p = A_o + B_o \dot{m} + C_o \dot{m}^2$$

Where: Δp = Pressure Rise in lbf/ft^2

$\dot{m}$ = Flow Rate in lbm/sec

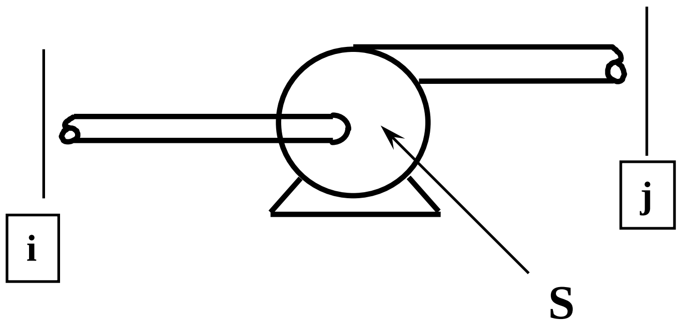


The Momentum Source Used to Induce the Desired Flow is:

$$S = \Delta p \; A$$

RESISTANCE OPTION 15

PUMP POWER

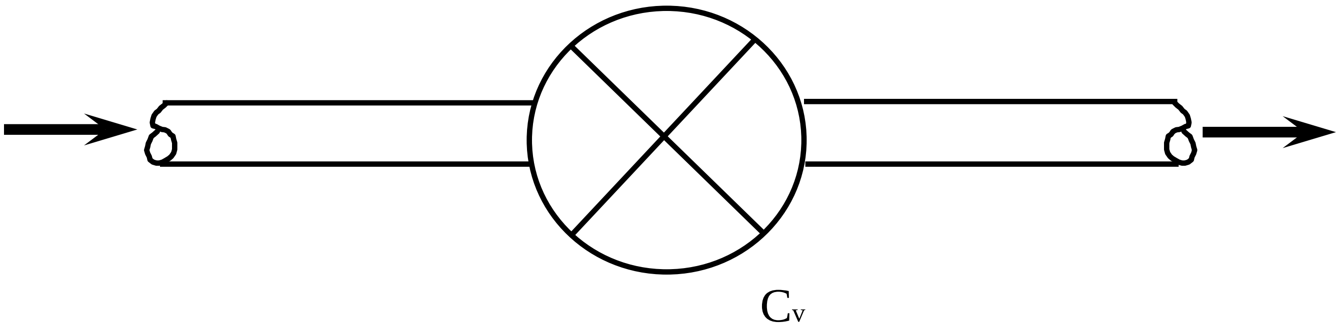


This Option Considers the Branch as a Pump with a Given Horsepower, P, and Efficiency, η. The Momentum Source, S, used to Induce the Desired Flow is Expressed as:

$$S = \frac{550\,\rho_u P \eta A}{\dot{m}}$$

RESISTANCE OPTION 16

VALVE WITH GIVEN CV



This Option Considers the Branch as a Valve with a Given C_v.
The Flow Resistance Factor for this Branch is Expressed as:

$$K_f = \frac{4.6799 \times 10^5}{\rho_u C_v^2}$$

RESISTANCE OPTION 17

VISCOJET (JOULE-THOMPSON DEVICE)

This option considers the branch as a Visco Jet which is a specific type of flow resistance with relatively large flow passages with very high pressure drops. The flow rate through the Visco Jet is given by:

$$w = 10000\, k_v \frac{V_f}{L_{ohm}} \sqrt{\Delta p\, S.G.}\,(1 - x)$$

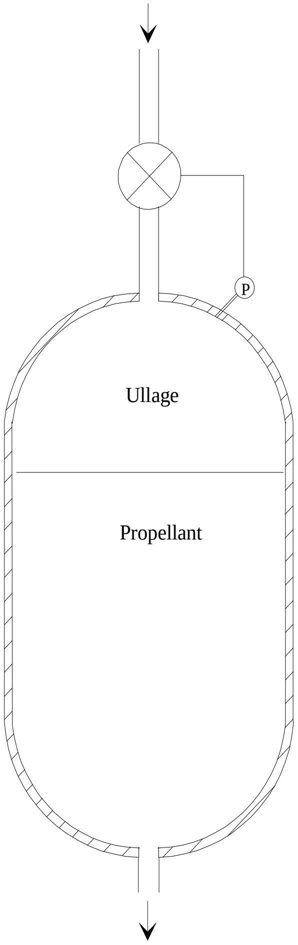
Where: w = the flow rate in lbm/hr,
L_{ohm} = the resistance of the fluid device $\left(\frac{\sqrt{lb_f / in^2}}{lb_m / hr}\right)$,
k_v = an empirical factor,
S.G. = Specific Gravity,
x = the downstream fluid quality, calculated by the code
and V_f = the viscosity correction factor.

For this option, K_f can be expressed as:

$$K_f = \frac{18.6624}{S.G.}\left(\frac{L_{ohm}}{V_f k_v (1 - x)}\right)^2$$

RESISTANCE OPTION 18

CONTROL VALVE



- Pressure monitored at arbitrary point downstream of valve
- Valve maintains pressure within user specified tolerance
  - Closes when pressure exceeds maximum value
  - Opens when pressure drops below minimum value
- Flow resistance factor calculated using same equations as Option 2

RESISTANCE OPTION 18 - CONTINUED

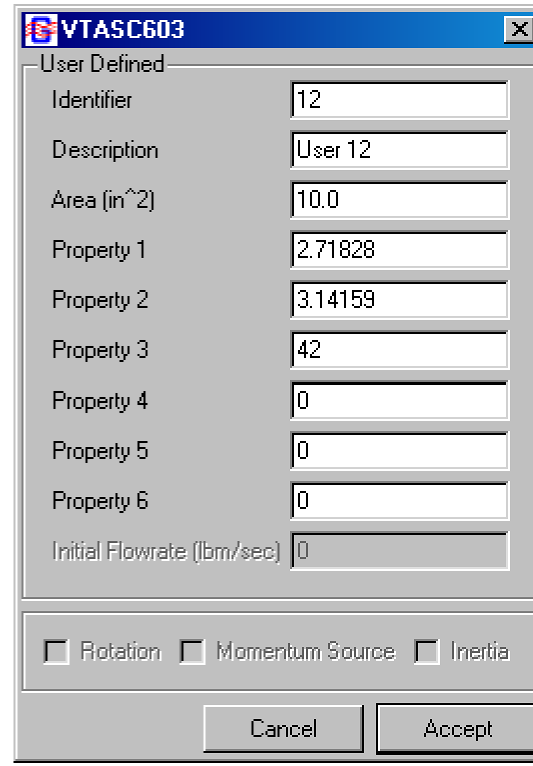
SUB-OPTIONS

- Instantaneous - Valve is either fully open or fully closed at any given time.
- Linear - Valve open/close transient is modeled as a linear operation.
- Non-linear - Valve open/close transient is modeled as some user specified non-linear operation.

RESISTANCE OPTION 19

User Defined U

- This option allows users to create a new resistance option that is not available in GFSSP library.
- Once this option is chosen, user is required to supply the coding for calculating K_f for this option in the User Subroutine
- In the preprocessor the user is required to supply the cross-sectional area of the branch.
- User has the option of supplying up to six branch parameters via the preprocessor



RESISTANCE OPTION 20

Heat Exchanger Core

A_s – Wetted Surface Area

A_c – Minimum Free Flow Area

σ – Ratio of Free Flow Area to Frontal Area

K_c- Contraction Loss Coefficient

K_e- Expansion Loss Coefficient

$$K_f = \frac{\left(K_c + 1 - \sigma^2\right) + 2\left(\frac{\rho_1}{\rho_2} - 1\right) + f\,\frac{A_s}{A_c}\,\frac{\rho_1}{\rho_{avg}} - \left(1 - \sigma^2 - K_e\right)\frac{\rho_1}{\rho_2}}{2\rho_1 g_c A_c^2}$$

RESISTANCE OPTION 21

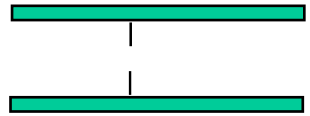
Parallel Tube

$$K_f = \frac{8 fL}{\rho_u \pi^2 D^5 g_c n^2}$$

• This option is an extended version of Resistance Option 1 where n is the numbers of parallel tubes

RESISTANCE OPTION 22

Compressible Orifice



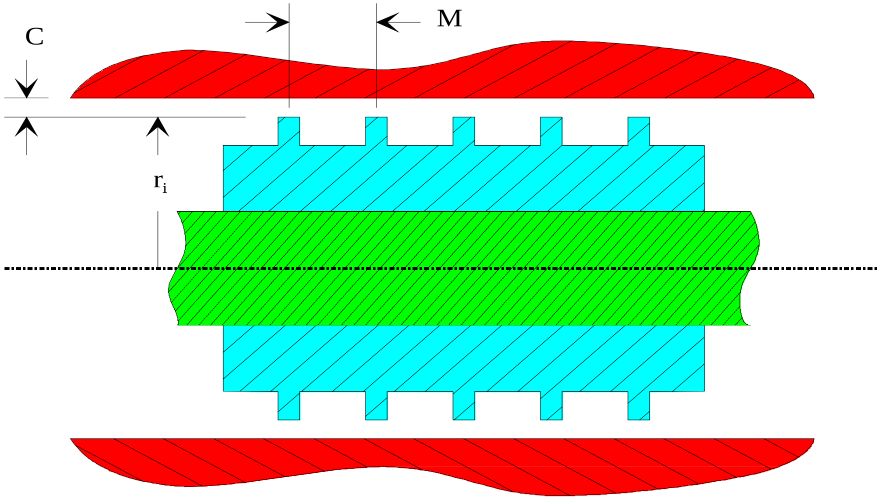
$$\dot{m}_{ij} = C_{L_{ij}} A \sqrt{p_i \rho_i g_c \frac{2\gamma}{\gamma - 1} \left(p_{cr}\right)^{2/\gamma} \left[1 - \left(p_{cr}\right)^{(\gamma - 1)/\gamma}\right]}$$

- This option considers branch as an orifice for compressible flow
- Flowrate is calculated from a simplified momentum equation
- There is no need to calculate K_f for this option
- The input to this option is identical to option 2 (Flow through Restriction)

RESISTANCE OPTION 23

LABYRINTH SEAL (EGLI Correlation)

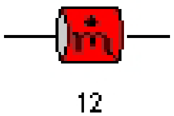
Labyrinth Seal



Where:
C = Clearance
M = Gap Length (Pitch)
r_i = Radius (Tooth Tip)
N = Number of Teeth
α = Step Seal Factor (~0.9)

RESISTANCE OPTION 24

Fixed Flowrate

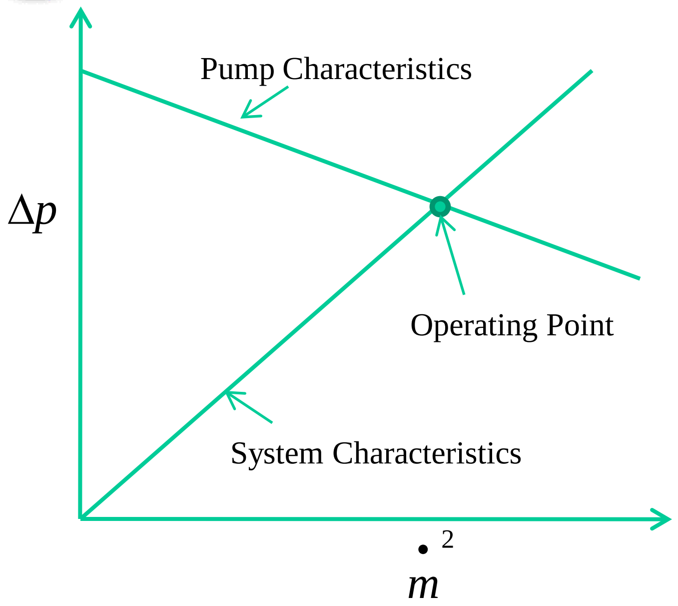


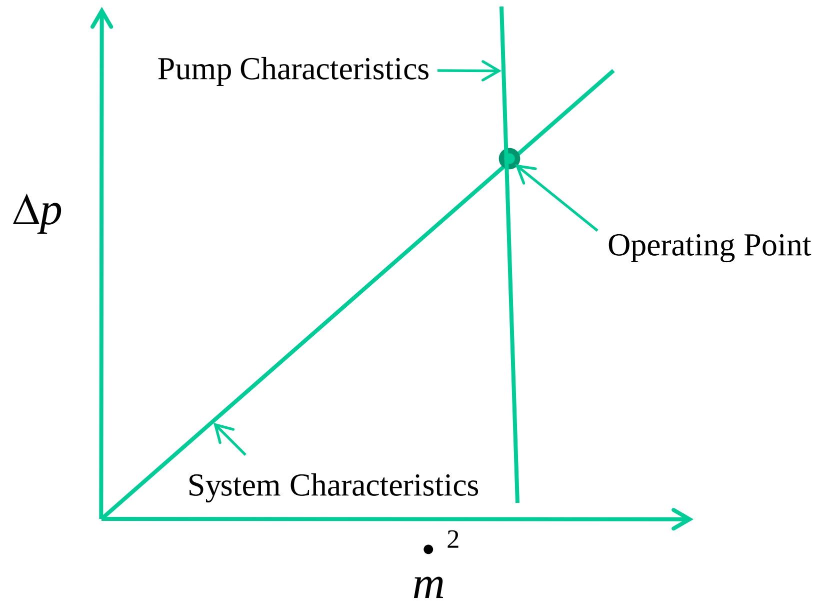
•A new branch option has been introduced to fix flowrate in a given branch

• The Fixed Flow branch can only be located adjacent to a Boundary Node

• For unsteady option, a history file will be needed to specify flow rate and area at all time steps

Algorithm for Fixed Flow Option(Schallhorn)

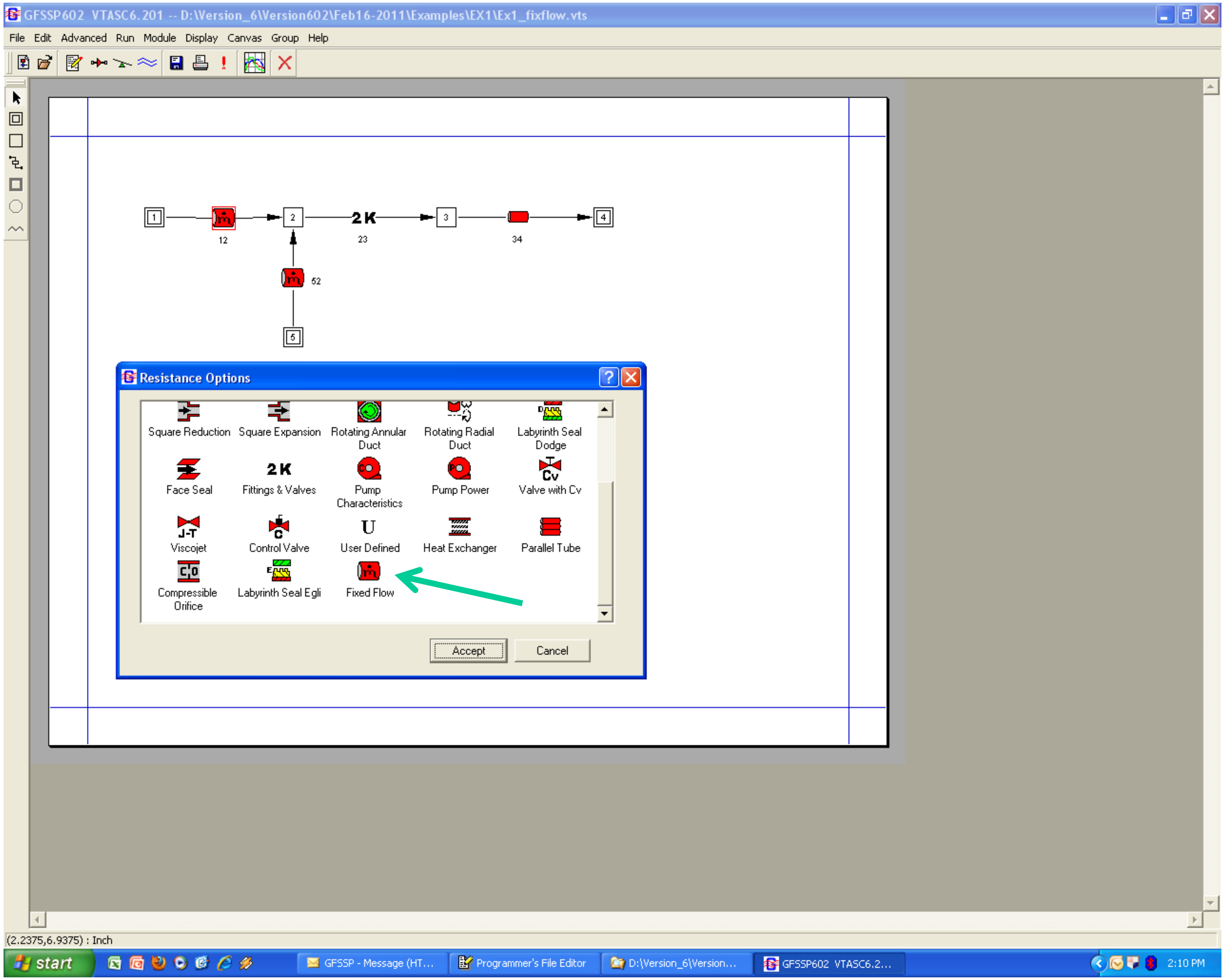




$$\Delta p = A + C\,\dot{m}\left|\dot{m}\right| \quad \text{where } A = \alpha\,\dot{m}\left|\dot{m}\right|\;;\; C = -\alpha;\, where\;\; \alpha = 1\times 10^{25}$$

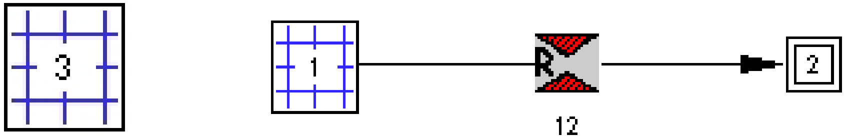
Substituting A and C , one gets: $\dot{m} = \dfrac{\dot{m}\left|\dot{m}\right|}{\left|\dot{m}\right|}$

New Resistance Option – Fixed Flow



RESISTANCE OPTION 25

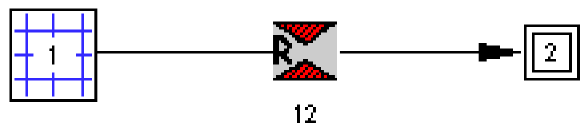
Linear Cartesian Grid

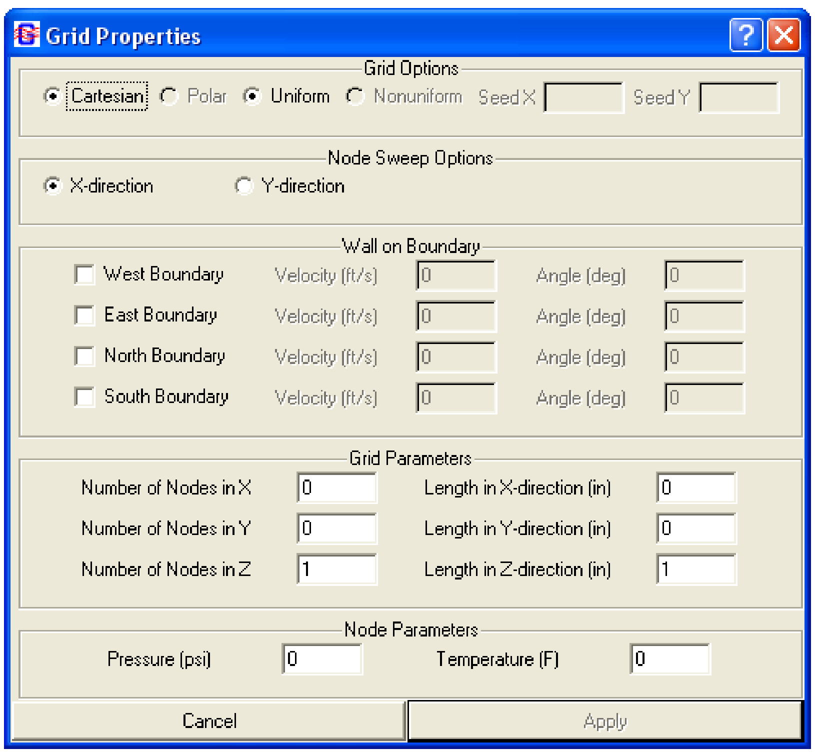


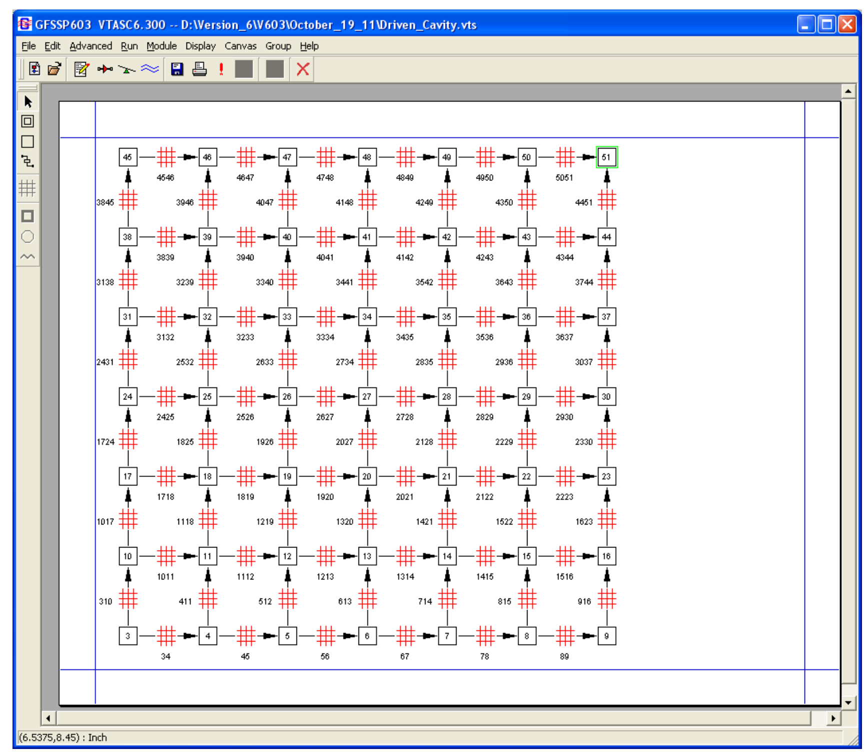
- Multi-dimensional flow calculation capability has been introduced
- User can “Enable Grid Generation” under “Advanced” pull down menu option
- This will make Linear Cartesian Grid available for drag and drop
- By right clicking the image user will have the option to generate the grid

Linear Cartesian Grid

- Generation and display of Two-dimensional







RESISTANCE OPTIONS SUMMARY

- Most fluid systems can be modeled using available options
- Option 2 can be used as a generic option where C_L must be computed from a known pressure drop vs. flowrate characteristics.
- User can add new resistance options by User Subroutines
- Multi-Dimensional Grid Generation will be expanded to include Cylindrical Polar Co-ordinate

FLUID OPTIONS

- GFSSP requires the following thermodynamic and thermophysical properties of fluids for the solution of the governing equations:
  - Density [$\rho(T, p)$]
  - Absolute Viscosity [$\mu(T, p)$]
  - Thermal Conductivity [$k(T, p)$]
  - Specific Heat at Constant Pressure [$C_p(T, p)$]
  - Ratio of Specific Heats [$\gamma(T, p)$]

- GFSSP requires these properties at every node at each iteration. These properties are supplied by thermodynamic property programs integrated into GFSSP.

AVAILABLE FLUIDS IN GASP/WASP

Working Fluid
Argon
Carbon Monoxide
Carbon Dioxide
Fluorine
Helium
Hydrogen
Methane
Neon
Nitrogen
Oxygen
Water
Kerosene (RP-1)
User Defined (Constant Property Fluid)

Properties Calculated Using: GASP & WASP

Properties Found in Lookup Table

User Supplies ρ and μ
(NOTE: The Energy Equation cannot be used with this Fluid Option)

AVAILABLE FLUIDS IN GASPAK

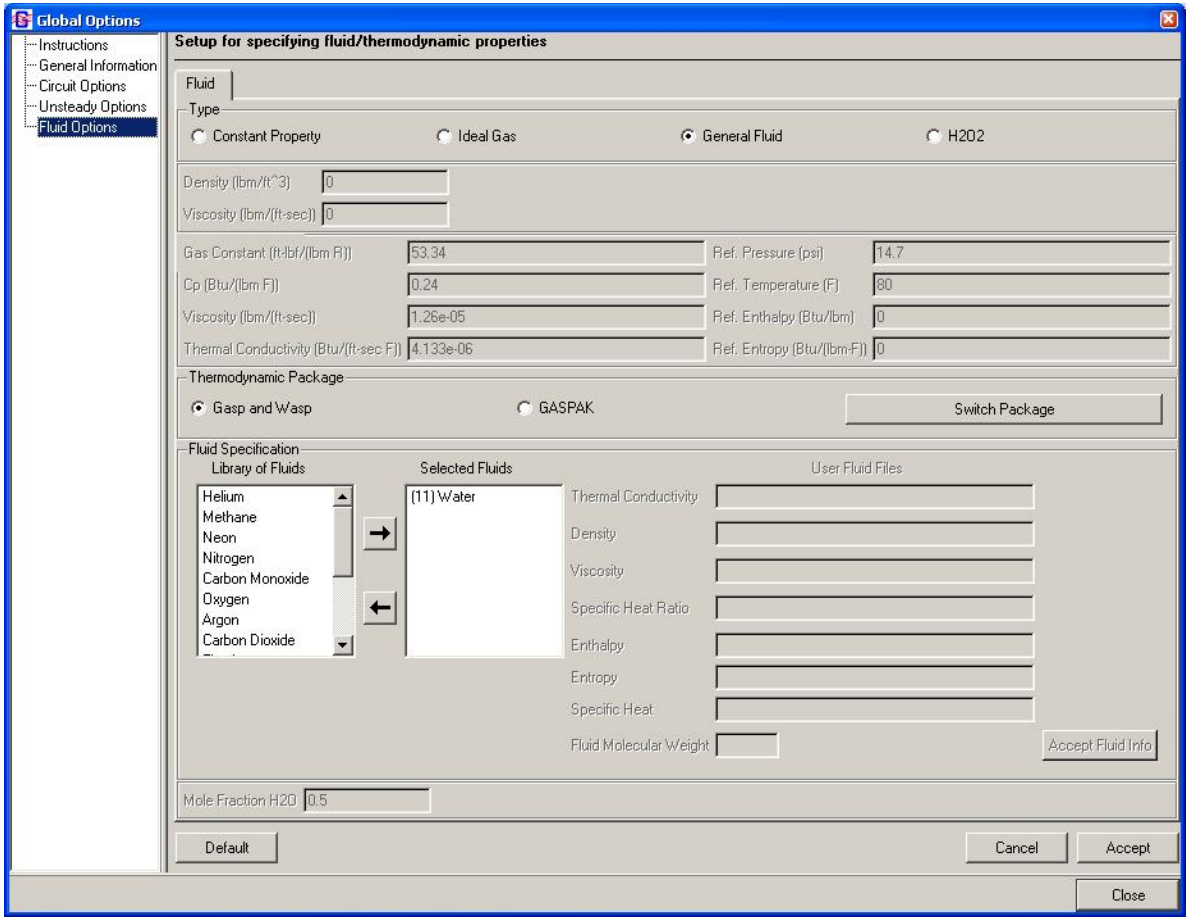
- **User can choose GASPAK by setting ADDPROP to TRUE**
- **GASPAK has a library of 32 fluids; in addition an option of ideal gas is also provided when GASPAK is set to .TRUE.**

Index	Fluid	Index	Fluid
1	HELIUM	19	KRYPTON
2	METHANE	20	PROPANE
3	NEON	21	XENON
4	NITROGEN	22	R-11
5	CO	23	R12
6	OXYGEN	24	R22
7	ARGON	25	R32
8	CO_2	26	R123
9	PARAHYDROGEN	27	R124
10	HYDROGEN	28	R125
11	WATER	29	R134A
12	RP-1	30	R152A
13	ISOBUTANE	31	NITROGEN TRIFLUORIDE
14	BUTANE	32	AMMONIA
15	DEUTERIUM	33	IDEAL GAS
16	ETHANE	34	HYDROGEN PEROXIDE
17	ETHYLENE	35	AIR
18	HYDROGEN SULFIDE		

Provision of Using Fluid Not Available in Fluid Library

- User can add fluids in the library by providing property table
- GFSSP requires following property table:
  - Thermal Conductivity
  - Density
  - Viscosity
  - Specific Heat at Constant Pressure
  - Specific Heat Ratio
  - Enthalpy
  - Entropy

Thermodynamic Property Table



No. of Pressure points

No. of Temperature points

```
15          30
    0.5100E+03    0.5600E+03    0.6100E+03    0.6600E+03    0.7100E+03
    0.7600E+03    0.8100E+03    0.8600E+03    0.9100E+03    0.9600E+03
    0.1010E+04    0.1060E+04    0.1110E+04    0.1160E+04    0.1210E+04
    0.1260E+04    0.1285E+04    0.1310E+04    0.1335E+04    0.1360E+04
    0.1385E+04    0.1410E+04    0.1435E+04    0.1460E+04    0.1510E+04
    0.1560E+04    0.1660E+04    0.1760E+04    0.1860E+04    0.1902E+04
    0.6000E+01    0.2300E+00    0.2280E+00    0.2250E+00    0.2230E+00
    0.2210E+00    0.2190E+00    0.2170E+00    0.2160E+00    0.2150E+00
    0.2130E+00    0.2120E+00    0.2110E+00    0.2105E+00    0.2100E+00
    0.2090E+00    0.2087E+00    0.2085E+00    0.2083E+00    0.2080E+00
    0.2083E+00    0.2087E+00    0.2090E+00    0.2093E+00    0.2097E+00
    0.2099E+00    0.2100E+00    0.2105E+00    0.2110E+00    0.2120E+00
    0.2130E+00
    0.7000E+01    0.2300E+00    0.2280E+00    0.2250E+00    0.2230E+00
    0.2210E+00    0.2190E+00    0.2170E+00    0.2160E+00    0.2150E+00
    0.2130E+00    0.2120E+00    0.2110E+00    0.2105E+00    0.2100E+00
    0.2090E+00    0.2087E+00    0.2085E+00    0.2083E+00    0.2080E+00
    0.2083E+00    0.2087E+00    0.2090E+00    0.2093E+00    0.2097E+00
    0.2099E+00    0.2100E+00    0.2105E+00    0.2110E+00    0.2120E+00
    0.2130E+00
```

First pressure point, p(1)

Second pressure point, p(2)

30 Temperature points written in free format

30 CP values corresponding to 30 temperature points at p(1)=6.0 psi

30 CP values corresponding to 30 temperature points at p(2)=7.0 psi

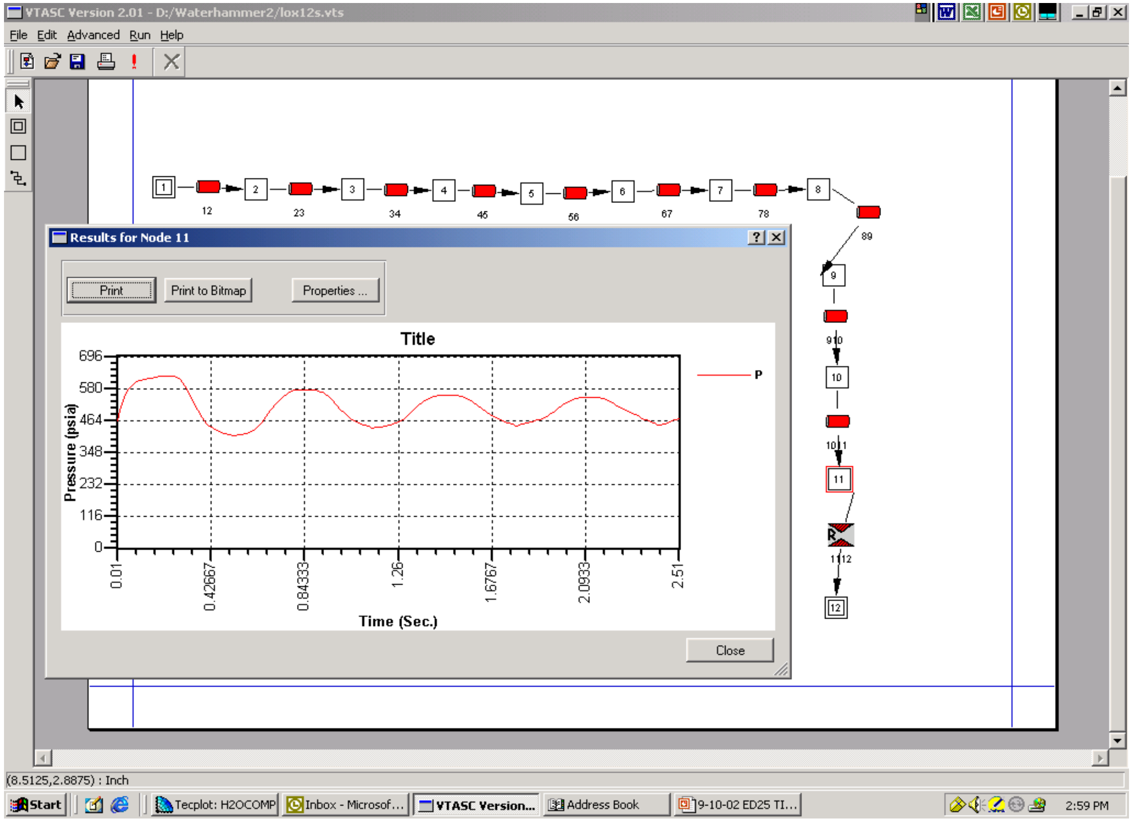
Read Statements :

```
READ(NRP1DAT,*) NP1,NT1
READ(NRP1DAT,*) (T1(J),J=1,NT1)
DO I = 1,NP1
     READ (NRP1DAT,*) P1(I),(PHI1(I,J,K),J=1,NT1)
ENDDO
```

FLUID OPTIONS SUMMARY

- GFSSP considers both gas and liquid as real fluid ; Liquid is also modeled as compressible fluid
- GASP/WASP/GASPAK provide higher order equation of state to calculate properties of liquid and vapor state over a wide range
- Table look-up provision can be used to add new fluid to the library
- Ideal Gas option can also be used

FLUID TRANSIENT



CONTENT

- Classification of Unsteady Flow
- Causes of Transient
- Methods of Analysis
- Valve Closing
  - Comparison with Method of Characteristics
  - Gas liquid mixture
  - Condensation
  - Fluid Transient in Branch Line
  - User Subroutine
- Valve Opening
- Conclusions

CLASSIFICATION OF UNSTEADY FLOW

- Quasi-steady flow is a type of unsteady flow when flow changes from one steady-state situation to another steady-state situation
  - Time dependant terms in conservation equation is not activated
  - Solution is time dependant because boundary condition is time dependant
- Unsteady flow formulation has time dependant terms in all conservation equations
  - Time dependant term is a function of density, volume and variables at previous time step
- GFSSP provides option for first order or second order differencing scheme

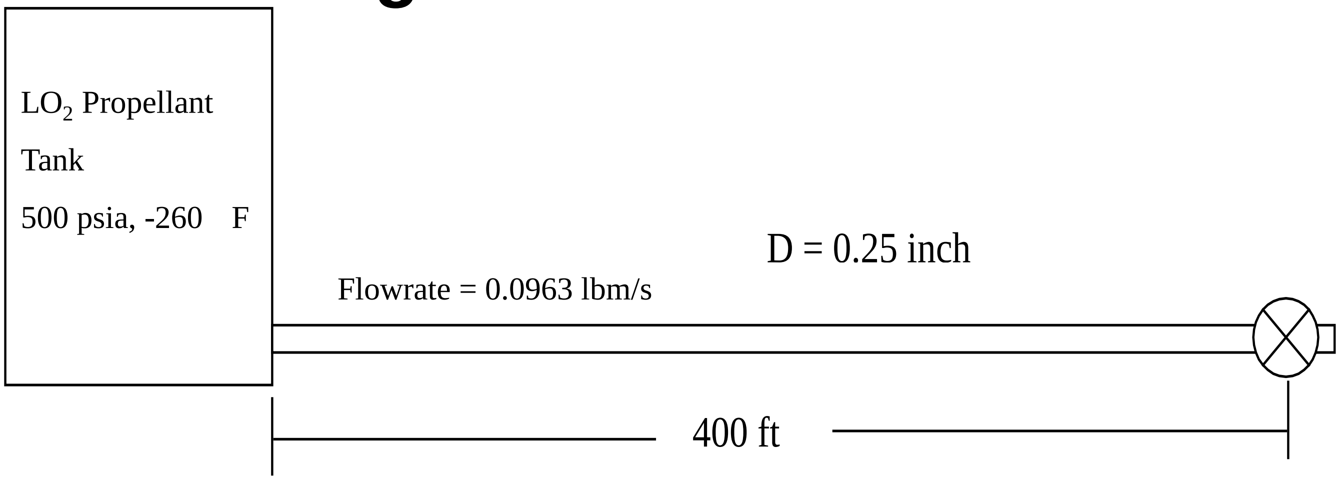
CAUSES OF TRANSIENT

- Changes in valve settings, accidental or planned
- Starting or stopping of pumps
- Changes in power demand of turbines
- Action of reciprocating pumps
- Changing elevation of reservoir
- Waves in reservoir
- Vibration of impellers or guide vanes in pumps or turbines
- Unstable pump characteristics
- Condensation

METHODS OF ANALYSIS

- Arithmetic Method
- Graphical Method
- Finite Difference Method
  - Method of Characteristics
  - Predictor-Corrector
- Impedance Method
- Finite Volume Method (GFSSP)

Example 15 – Simulation of Fluid Transient Following Sudden Valve Closure



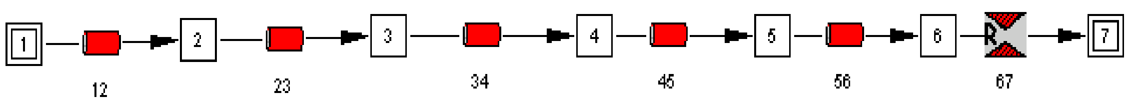
Valve Closure History

Time (sec)	Area (in^2)
0.00	0.0491
0.02	0.0164
0.04	0.0055
0.06	0.0018
0.08	0.0006
0.10	0.00

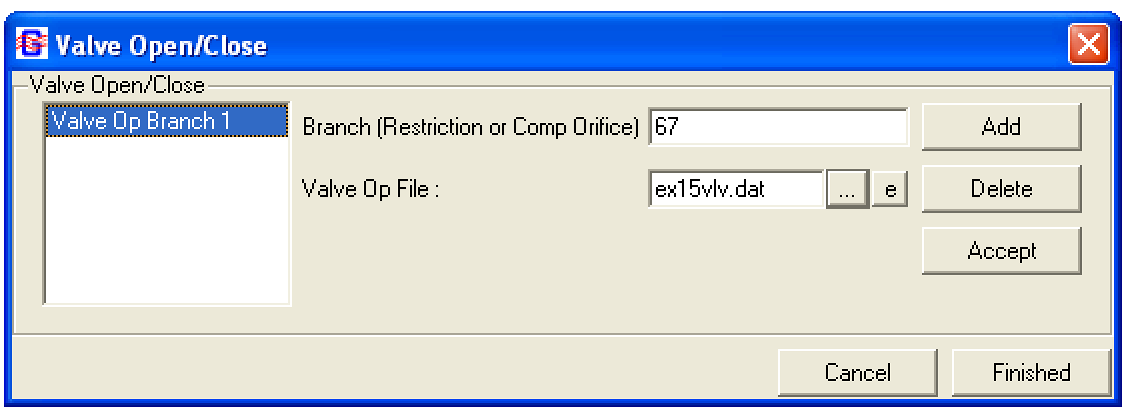
Objectives of Analysis:

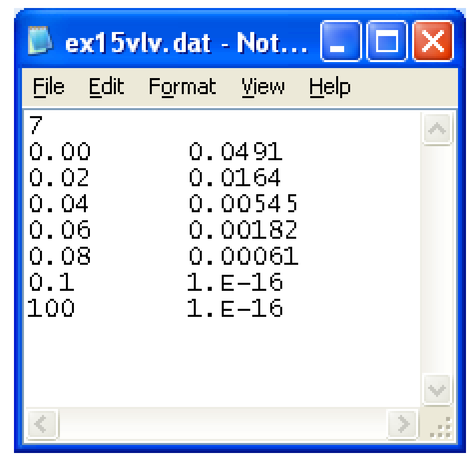
- Maximum Pressure
- Frequency of Oscillation

GFSSP Model



- Run a steady state model with 450 psia ambient condition
- Run unsteady model with steady state solution as initial value





Result

Estimate of Time Step from Courant Number:

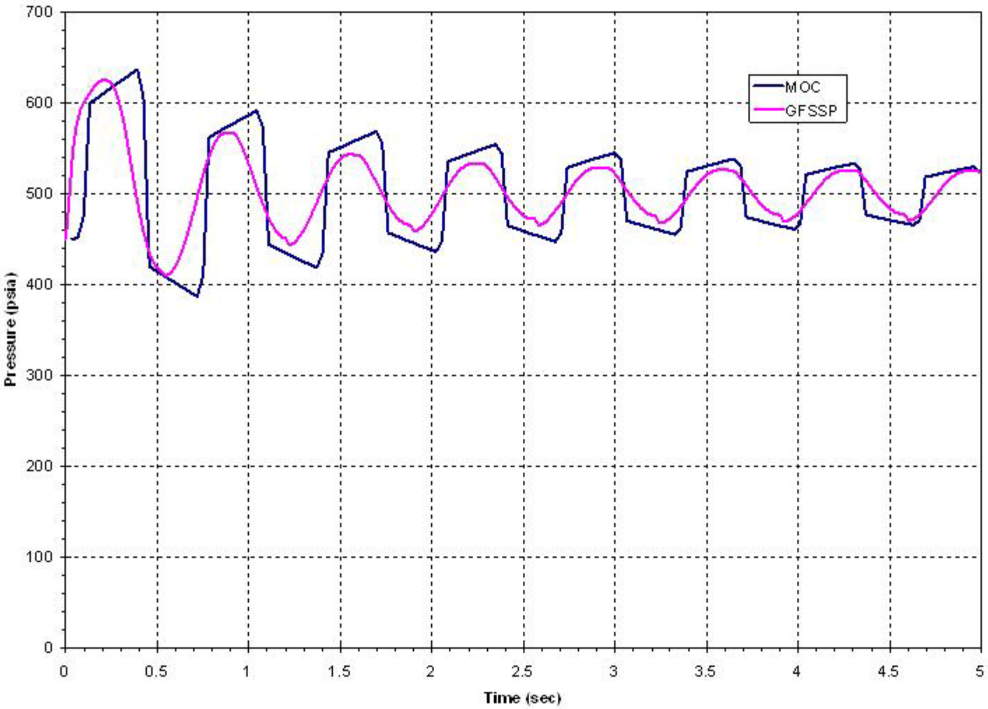
$$\text{Courant Number} = \frac{4L_{branch}}{a_{fluid}\Delta\tau} \geq 1$$

The speed of sound (a_{fluid}) for LOX is 2462 ft/sec

$$Courant\ \ Number = 6.5 \text{ for}$$

$$\Delta\tau = 0.02 \ \text{ sec}$$

$$for\ \ L_{branch} = 200\,ft$$



Description of Test cases

P_{tank} = 500 psia

T_{tank} = -260 ° F (Oxygen) = 70 ° F (Water) = -414 ° F (Hydrogen)

Case No.	Fluid	Number of Branches	Time Step (sec)	Sound Speed (ft/sec)	Flowrate (lb/sec)	p_{max} (psia)	Period of Oscillation (sec)
1	LO_2	10	0.01	2462	0.0963	626	0.65
2	LO_2	20	0.005	2462	0.0963	632	0.65
3	LO_2	5	0.02	2462	0.0966	620	0.65
4	H_2O	10	0.005	4874	0.071	704	0.33
5	LH_2	10	0.02	3577	0.0278	545	0.43
6	LO_2 & GHe (0.1%)	10	0.01	1290**	0.0963	580	1.24
7	LO_2 & GHe (0.5%)	10	0.01	769**	0.0963	520	2.08
8*	LO_2 (2 Phase) x_{exit} = 0.017	10	0.01	-----	0.0963	550	1.17
9*	LO_2 (2 Phase) x_{exit} = 0.032	10	0.01	------	0.0963	538	1.22
10	LO_2		0.01	2462	0.0963	611	0.65

* Pressure oscillations are due to condensation

** Estimated from period of oscillation [$a=4L/\lambda$]

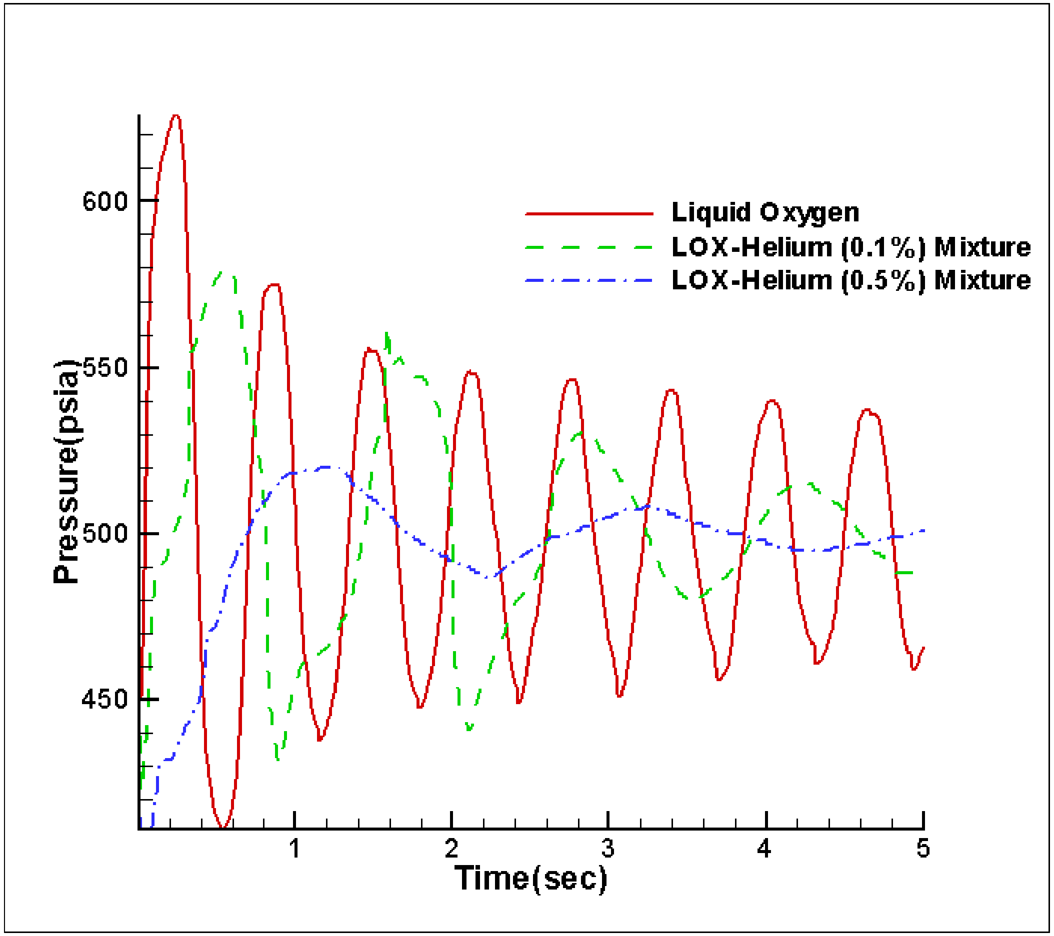
- Time step for each test case is so chosen that Courant number is larger than unity

COMPARISON BETWEEN GFSSP & MOC SOLUTION

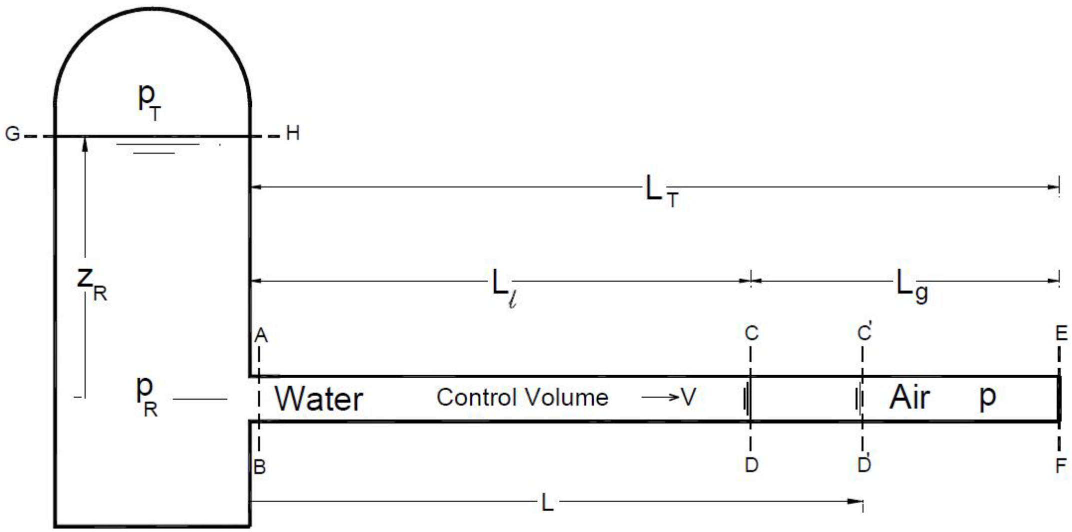
Fluid	Flowrate (lb/s)	Velocity (ft/s)	Friction Factor (Used in MOC solution)	Sound Speed (ft/s)	Max. Pressure rise above supply pressure (psi)		Period of Oscillation (sec)	
					MOC	GFSSP	MOC	GFSSP
Water	0.071	3.34	0.0347	4892	214	204	0.33	0.33
Oxygen	0.0963	4.35	0.0196	2455	136	126	0.65	0.65
Hydrogen	0.0278	19.01	0.0157	3725	61	45	0.43	0.43

Majumdar, A. K. and Flachbart, R. H., "Numerical Modeling of Fluid Transients by a Finite Volume Procedure for Rocket Propulsion Systems", Paper No. FEDSM2003-45275, Proceedings of ASME FEDSM'03, 4th ASME/JSME Joint Fluids Engineeing Conference, Honolulu, Hawaii, July 6-10, 2003.

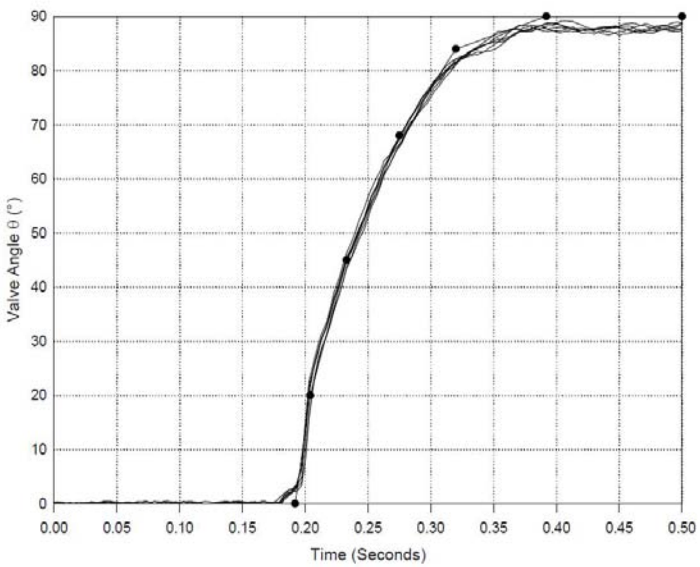
GAS LIQUID MIXTURE



Rapid Valve Opening



Top: Schematic of the Pipeline System with ball-valve location.

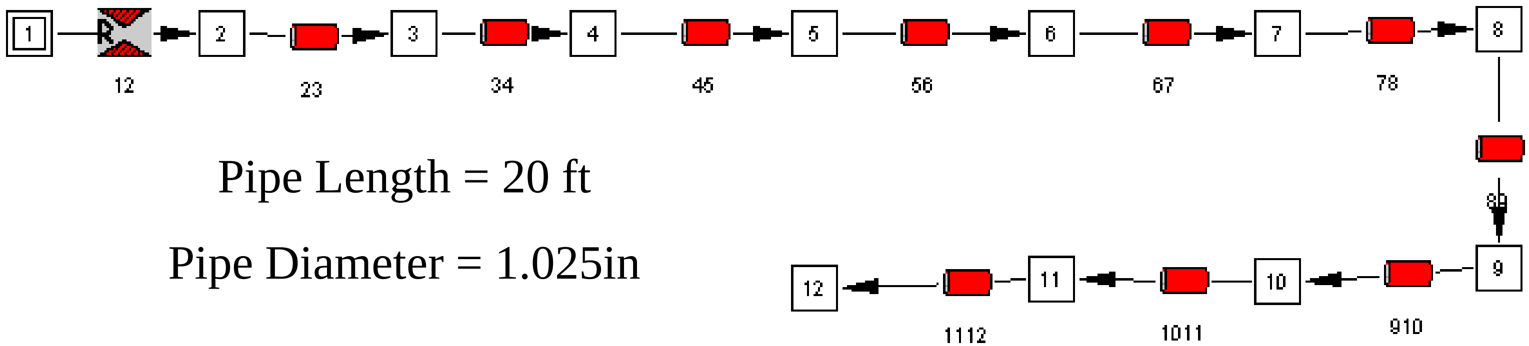


Side: Ball-Valve Opening History

Parameters: $\alpha = L_g/L_T$

$P_R = p_R/p_0$

GFSSP MODEL



Pipe Length = 20 ft

Pipe Diameter = 1.025in

- Node 1: Reservoir pressure = 29.4 to 102.9 psia
- Node 12: Volume change due to change in entrapped air volume.
- Air pressure at the end of node 12 is calculated using Adiabatic gas expansion relation for ideal gas.
- The valve opening area is based on ball-valve opening of the original problem.
- Momentum Source due to pressurization of water column due to air pressure.

Mathematical Formulation

- Volume Change with time for pseudo node 12 due to air pressure:

$$\Delta V_w = -\Delta V_{air} = m_{air} R_{air} T_{12} \Delta p / p_{12}^2$$

$$\Delta p = (p_{12} - p_{12}^*)$$

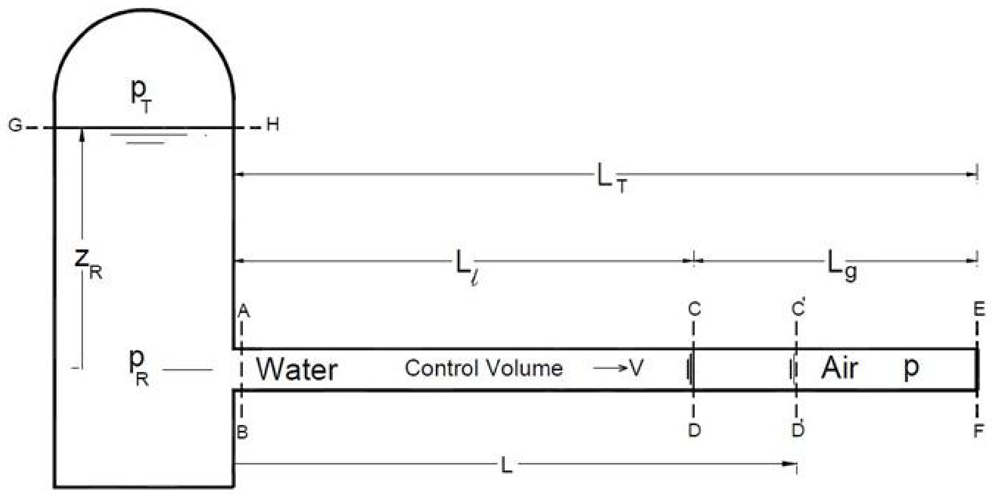
- Air Pressure Calculation: (Using Adiabatic Gas Relation):

$$p = \left(p^* V + p^* V^* \left[\left(\frac{V^*}{V} \right)^{\gamma - 1} - 1 \right] \right) / (V + \Delta V)$$

- p, V etc indicates pressure and Volume at the current time step and p^*, V^* indicates pressure and volume at the previous time step. The volume and pressure changes are with respect to time.

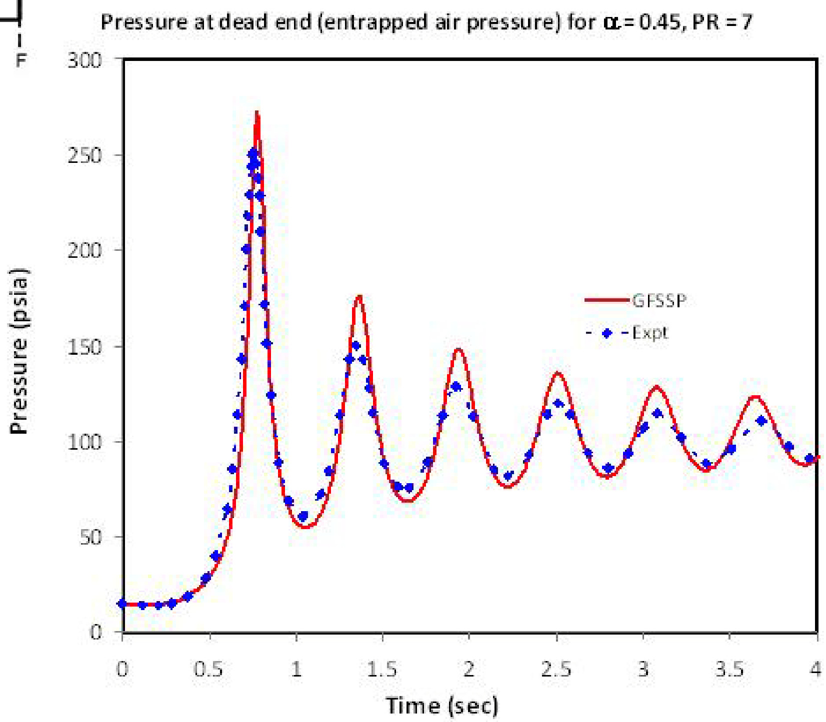
- Momentum Source in Node 12: $-(p_{air}-p_{12})A$

RESULTS



$$\alpha = L_g/L_T$$

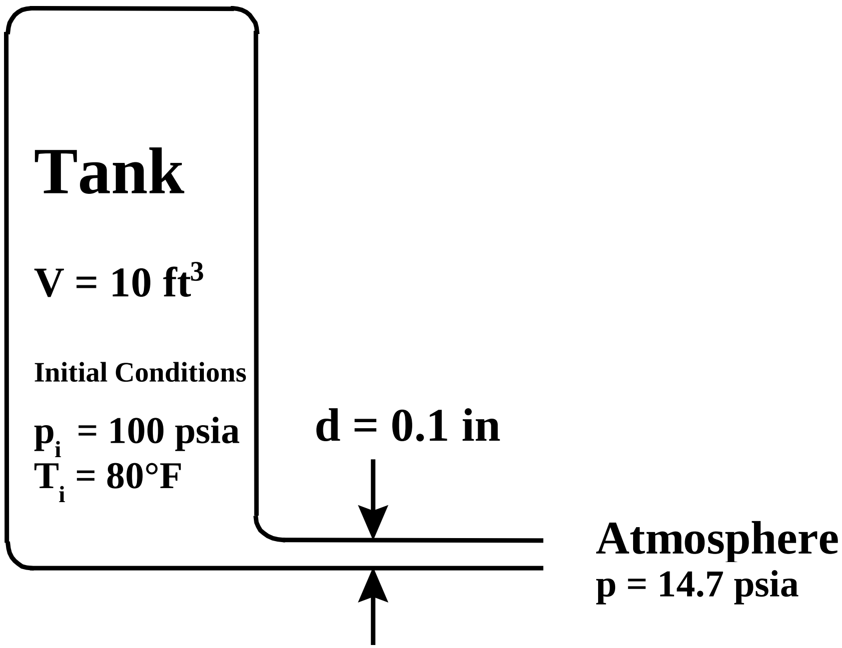
$$P_R = p_R/p_0$$

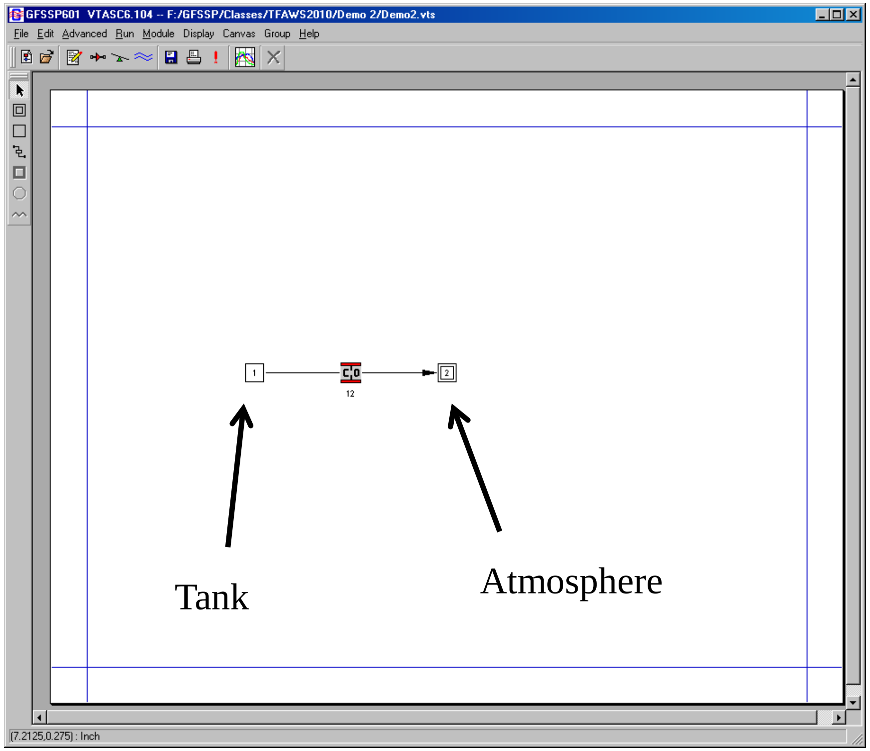


CONCLUSIONS

- GFSSP has been used to compute fluid transient following rapid valve closure and opening
- GFSSP predictions have been compared with MOC solution and experimental data:
  - Maximum pressure and frequency compares well
  - Discrepancies exist in damping rate primarily due to rigid pipe assumption
- Demonstrations have been made for
  - Two phase (Gas-Liquid) flow following valve closure
  - Condensation of liquid-vapor flow following valve closure
  - Sudden opening of cryogenic propellant in long pipeline
- Time step must satisfy Courant condition
- Predictions in all demonstration calculations show physical realism

VTASC DEMONSTRATION PROBLEMS -2



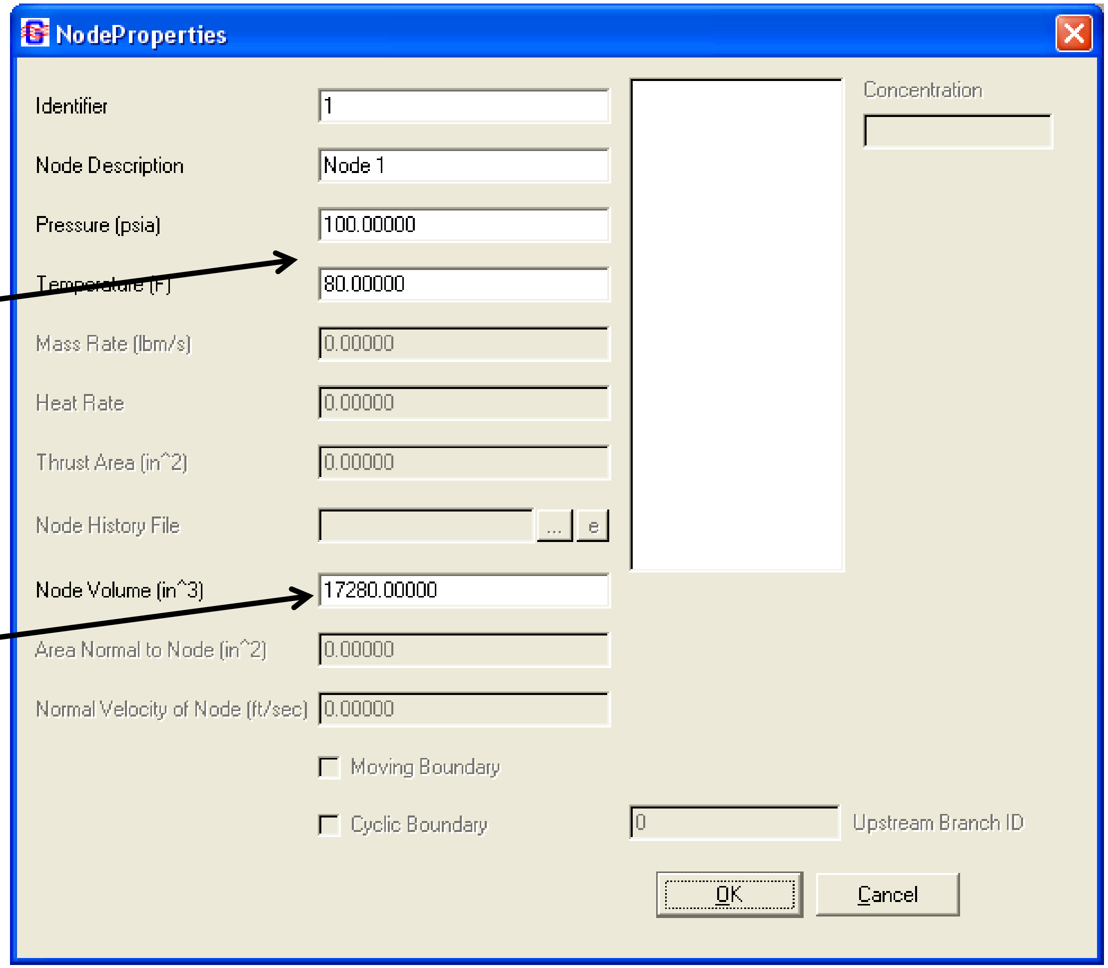


GFSSP 6.04 Preprocessor Demo 2

Demo 2:
Interior Node Initial Conditions

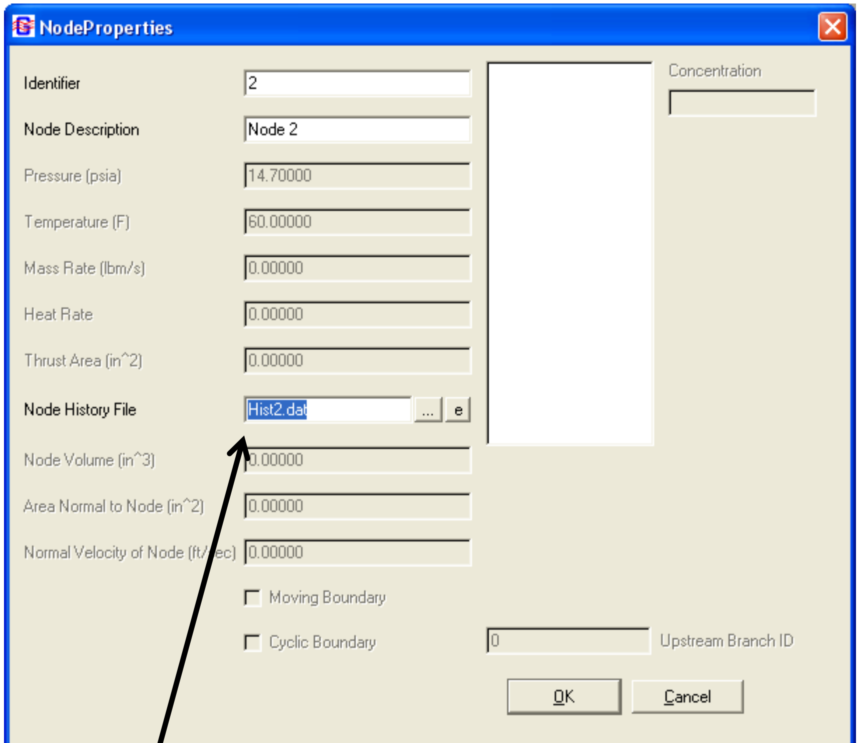
Initial P,T

Tank Volume



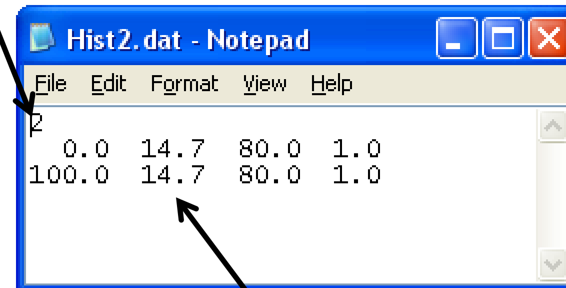
GFSSP 6.04 Preprocessor Demo 2

Demo 2: Transient Boundary Conditions



Specify History Filename

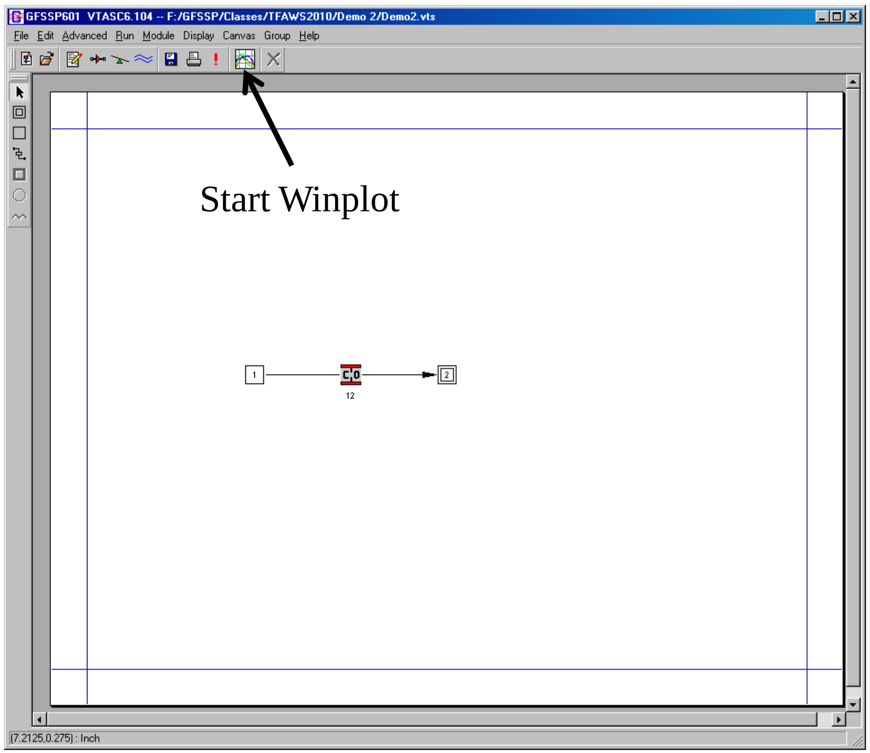
Number of Lines (min. 2)

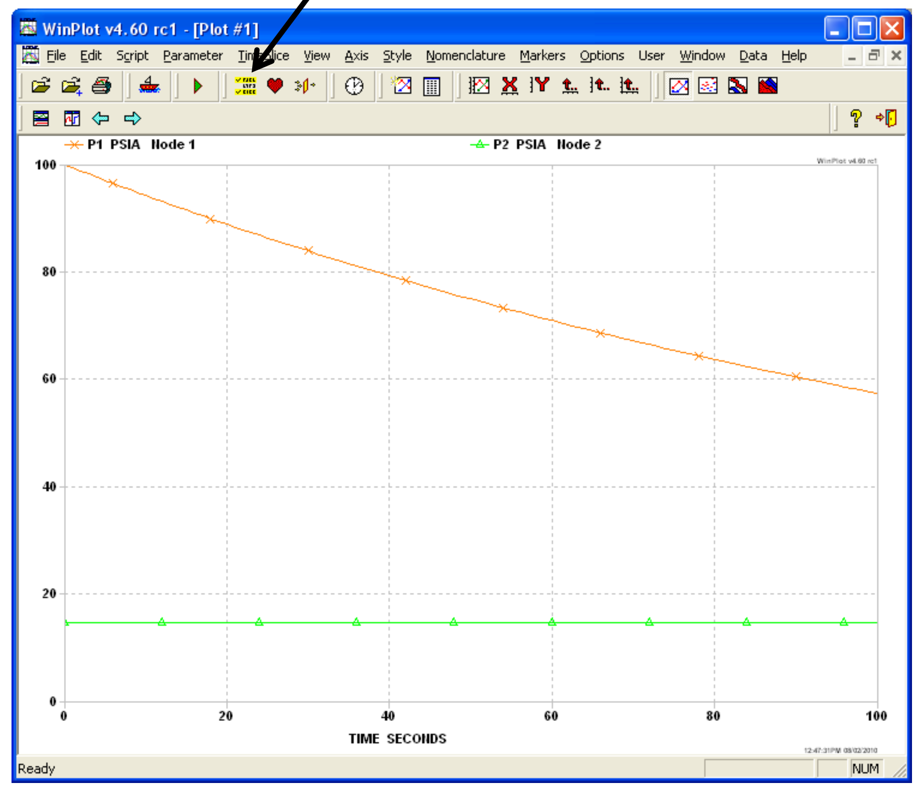


Time (s), P(psia), T(F), Mass Fraction

- GFSSP will interpolate transient boundary conditions from the history file
- Even if boundary conditions are constant, at least two lines must be given

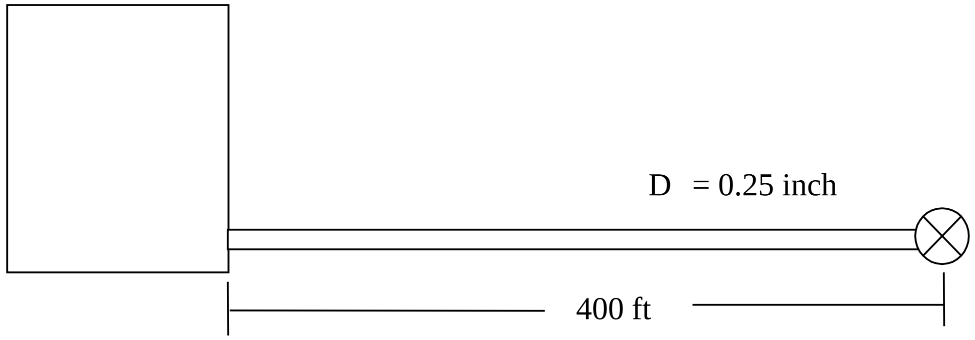
Demo 2:
Plotting Transient Results



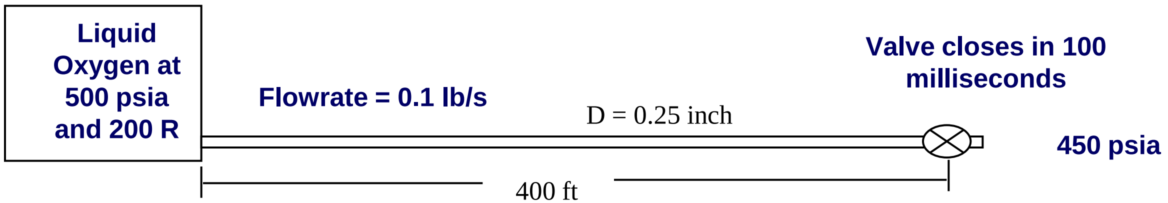


Tutorial – 2

SIMULATION OF FLUID TRANSIENT FOLLOWING SUDDEN VALVE CLOSURE



FLUID TRANSIENT SCHEMATIC

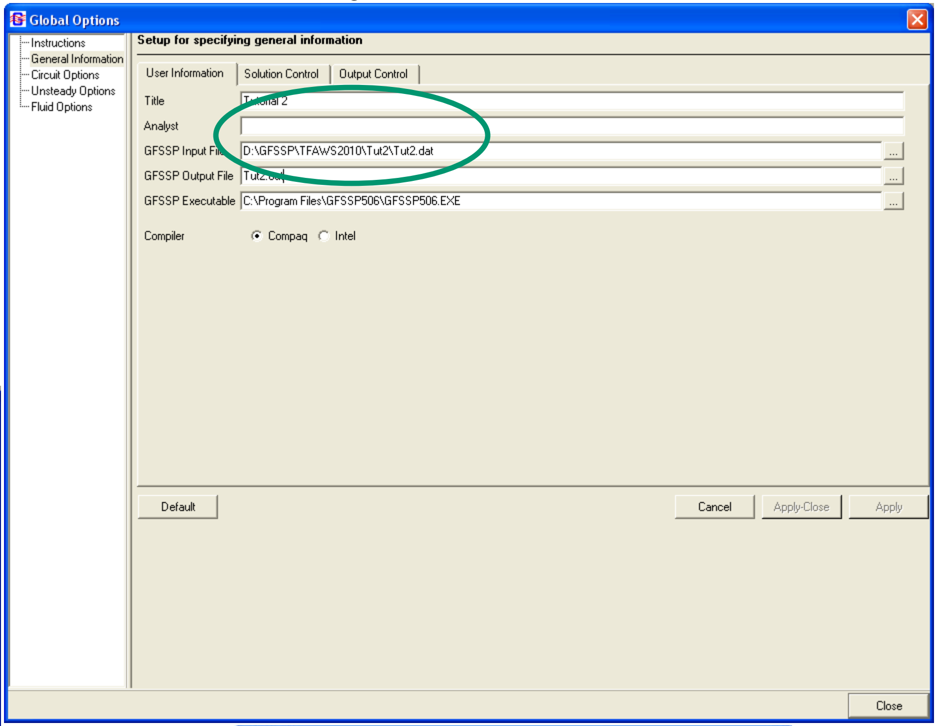


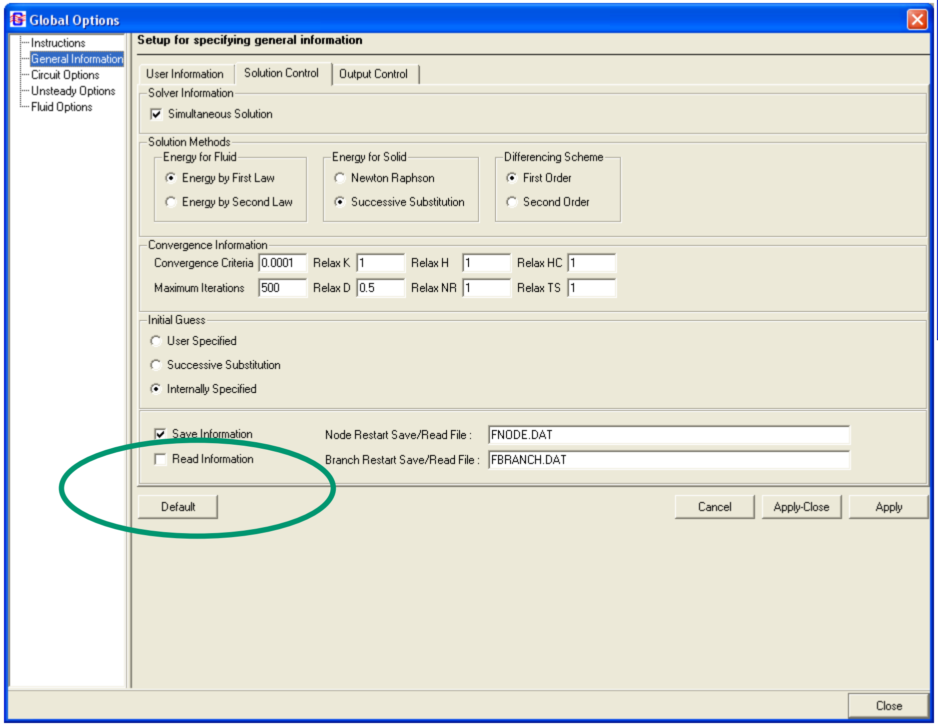
Problem Considered:

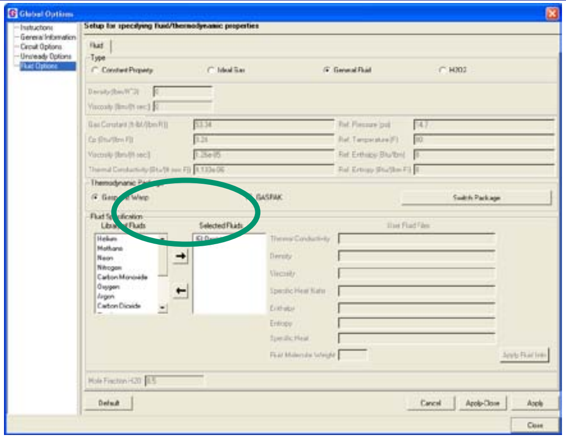
•Time dependent Pressure and Flow rate history during and after valve closure

Part 1: Build Steady State Model

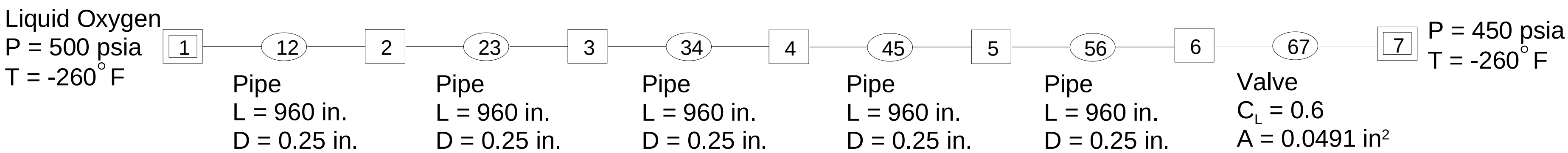
- Input File: Tut2.dat
- Output File: Tut2.out
- Check Save Information to save the steady-state solution in the restart files
- Fluid is LOx

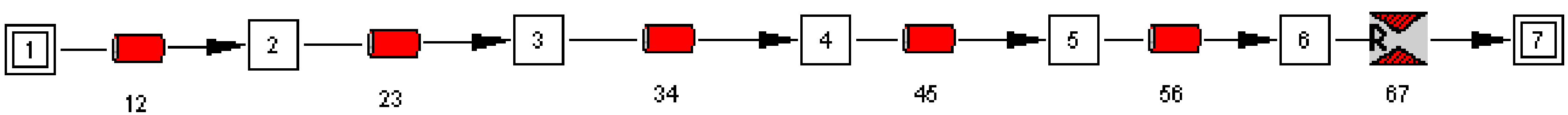




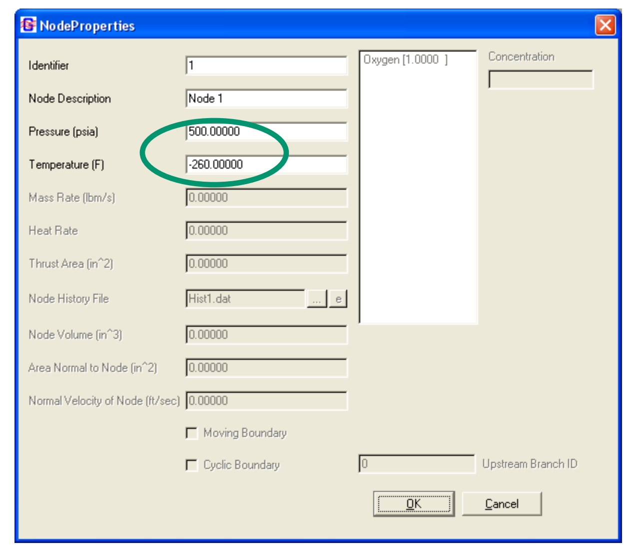


Part 1: Build Steady State Model (cont.)



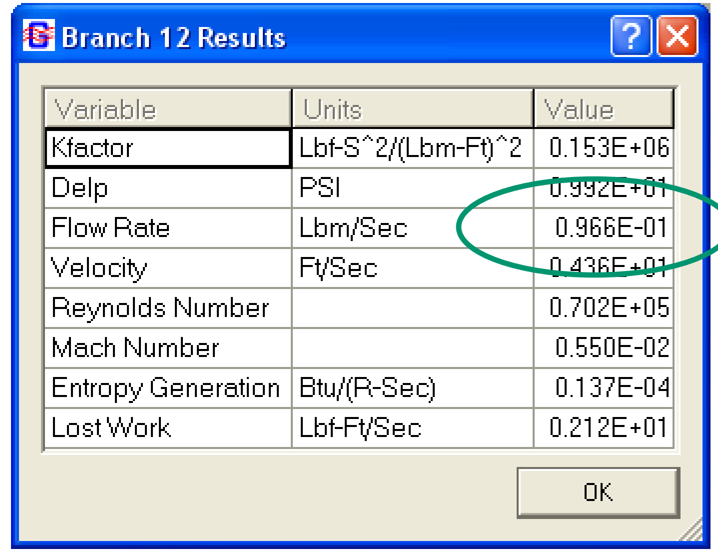


- Build the model on the canvas
  - Set boundary conditions
  - Set pipe and restriction parameters
  - Assume smooth pipe (ε=0)



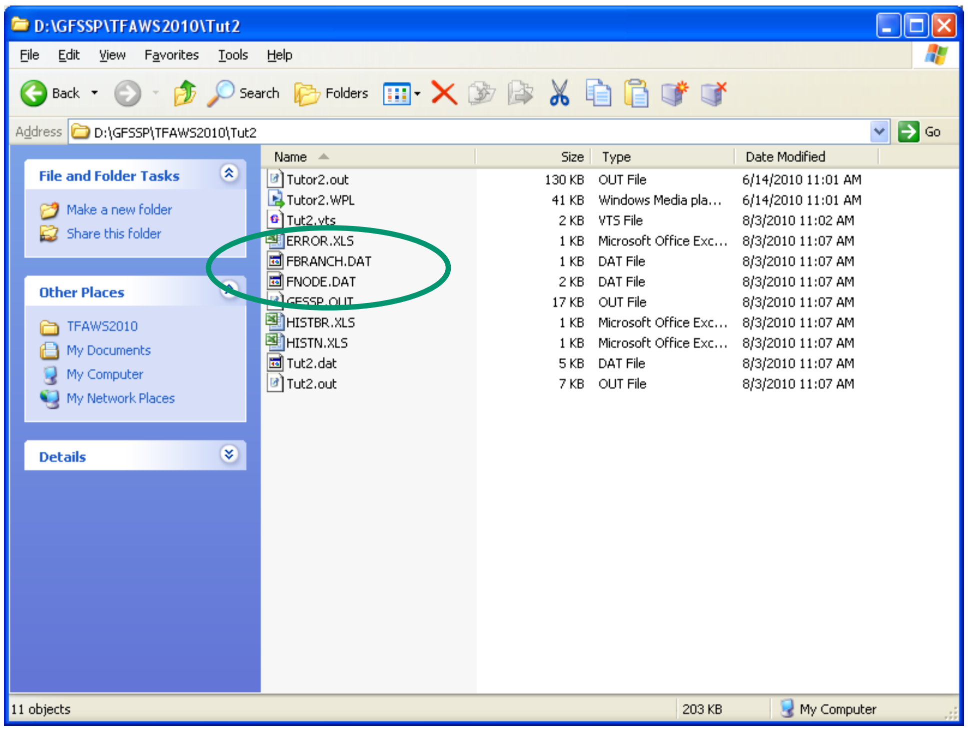
Part 1: Build Steady State Model (cont.)

- Run the steady state model
- Check that the flowrate is ≈0.1 lb_m/s
- Note that the results have been saved in the restart file



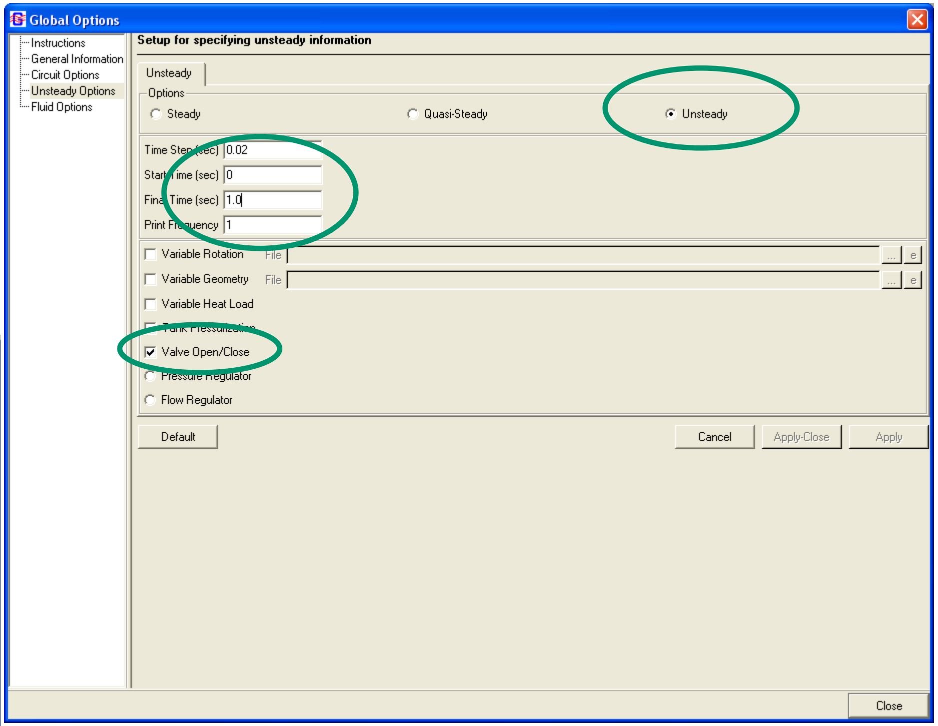
Branch 12 Results

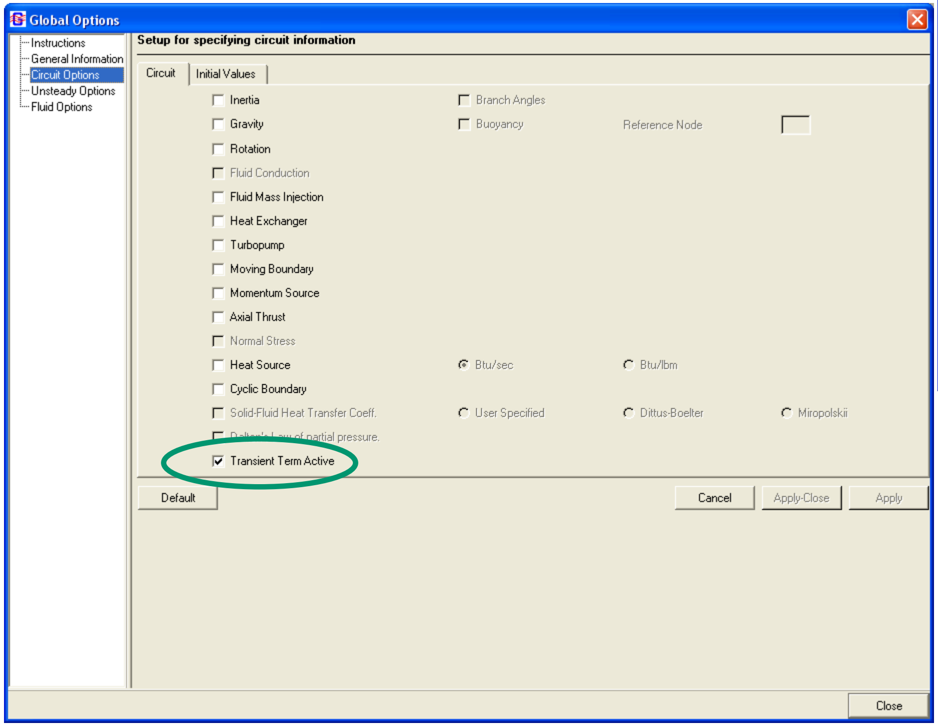
Variable	Units	Value
Kfactor	Lbf-S^2/(Lbm-Ft)^2	0.153E+06
Delp	PSI	0.992E+01
Flow Rate	Lbm/Sec	0.966E-01
Velocity	Ft/Sec	0.436E+01
Reynolds Number		0.702E+05
Mach Number		0.550E-02
Entropy Generation	Btu/(R-Sec)	0.137E-04
Lost Work	Lbf-Ft/Sec	0.212E+01



Part 2: Build Transient Model

- Convert the model to transient
  - Time step = 0.02 s
  - Run time = 1.0 s
  - Check Valve Open/Close Unsteady Option

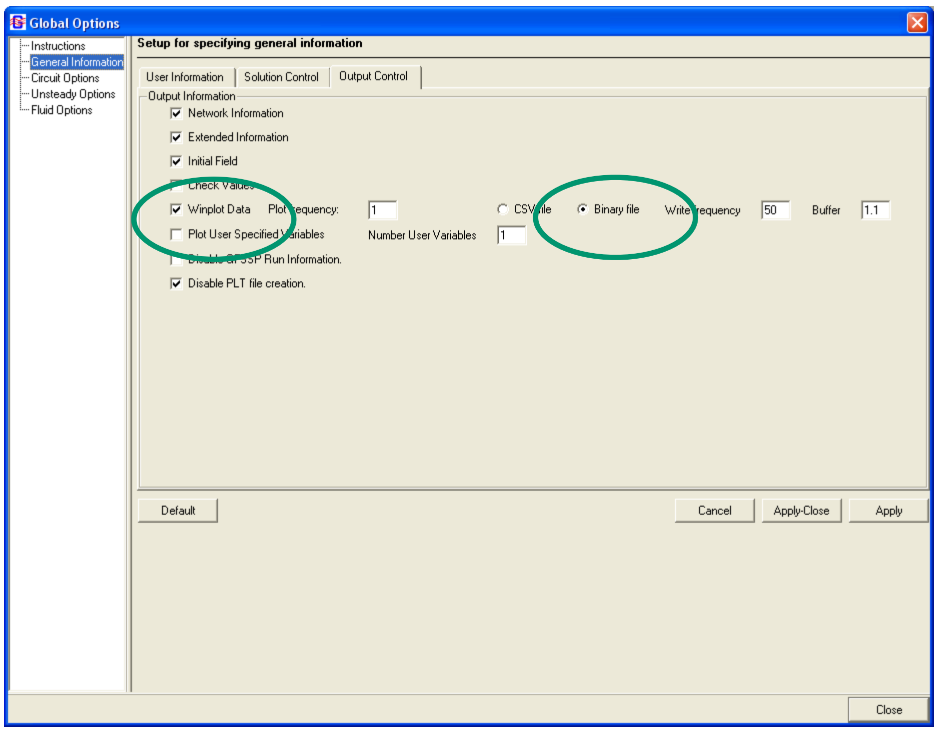


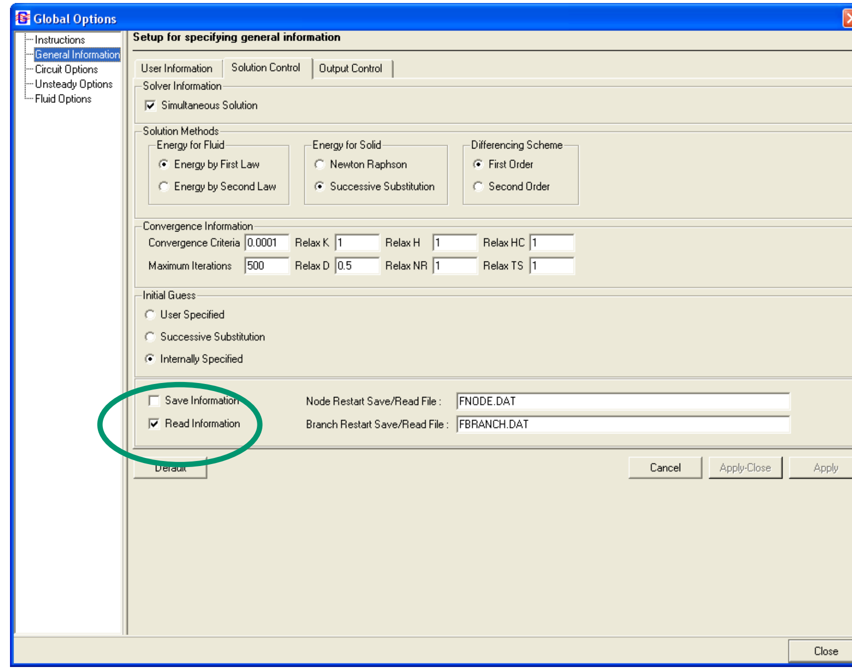


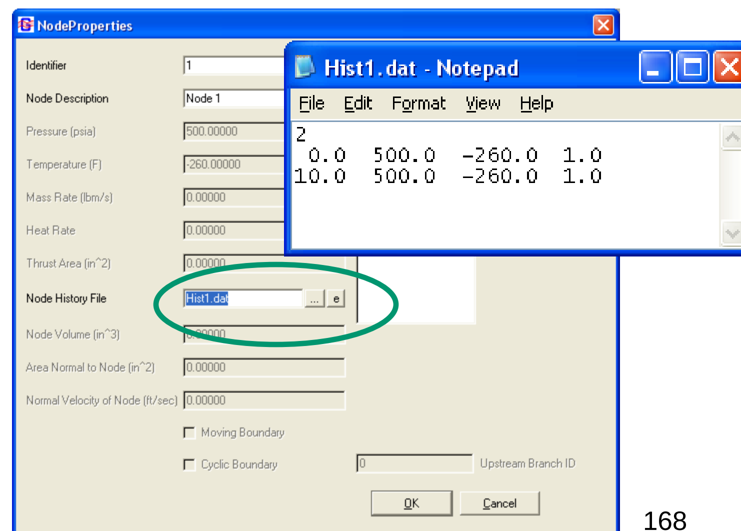
- Check Transient Term Active on Circuit Options page

Part 2: Build Transient Model (cont.)

- Uncheck SAVE restart file box
- Check READ restart file box
- Check Winplot output
- Create history files for boundary nodes





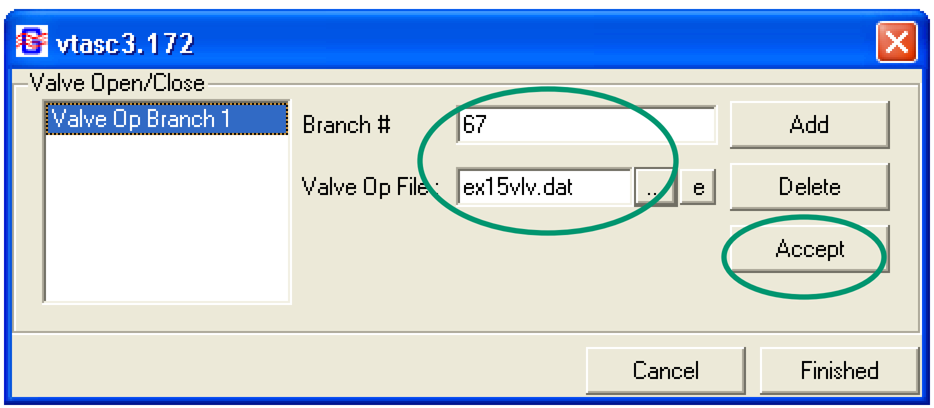


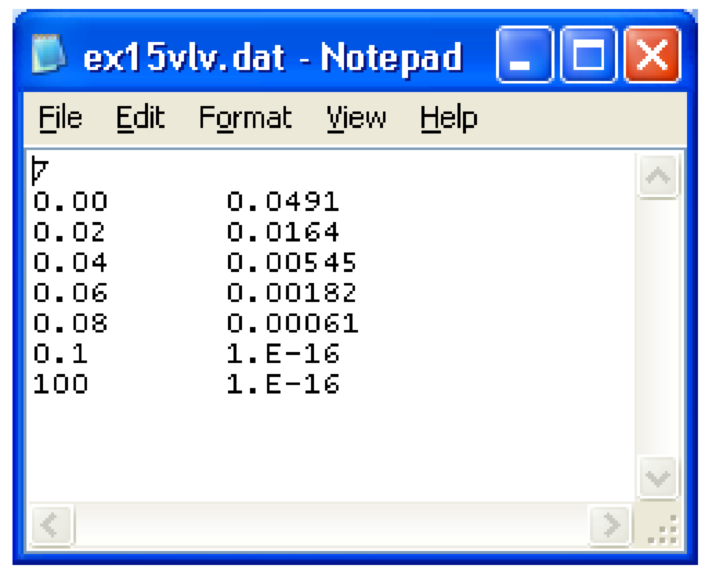
Part 2: Build Transient Model (cont.)

- Open the Valve Open/Close dialog box from the Advanced menu
- To represent the valve closing, the area of Branch 67 will vary as a function of time

Valve Closure History

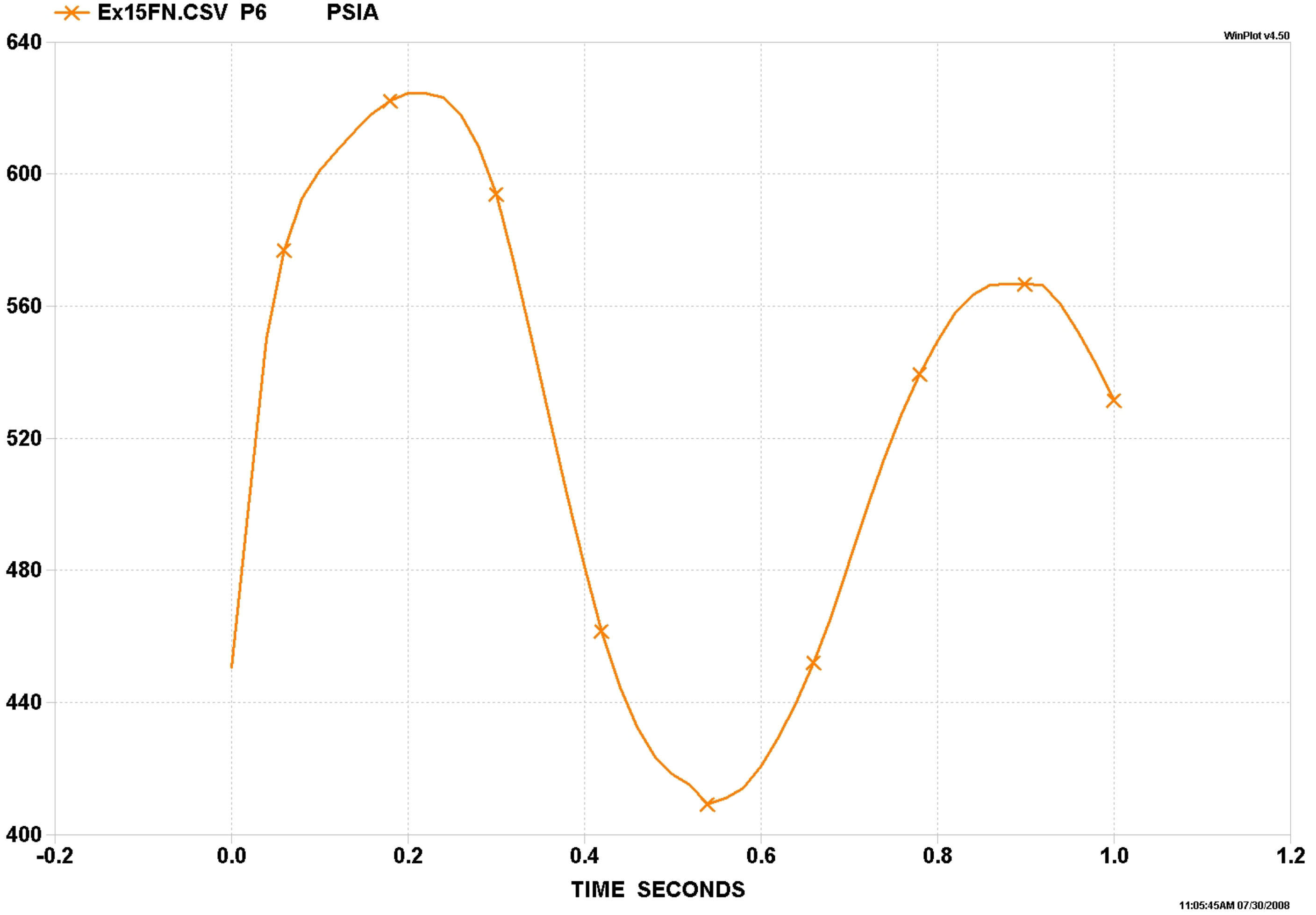
Time (Sec)	Area (in^2)
0.00	0.0491
0.02	0.0164
0.04	0.00545
0.06	0.00182
0.08	0.00061
0.10	0.00





Pressure History at Valve

Marshall Space Flight Center
GFSSP Training Course

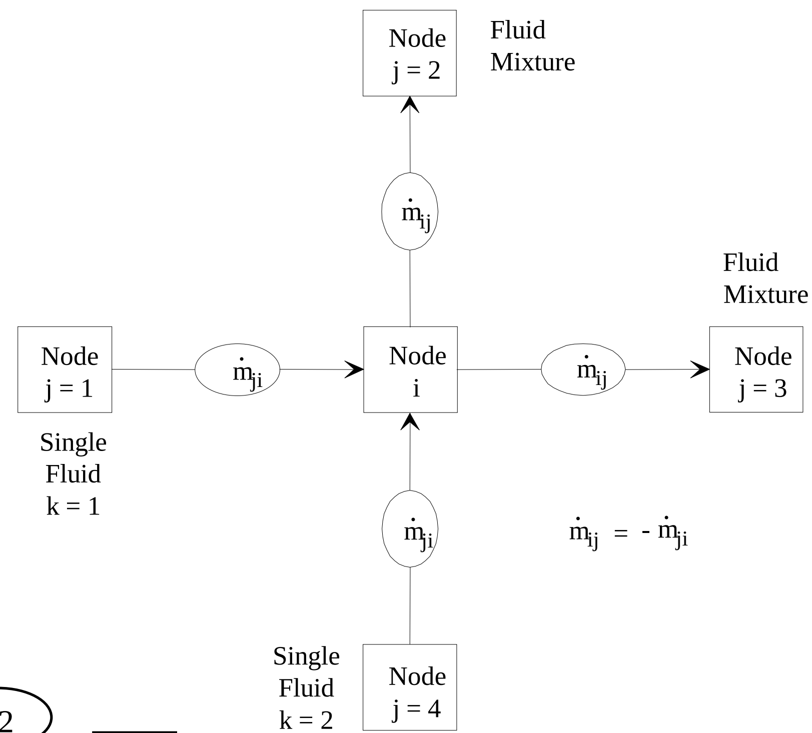


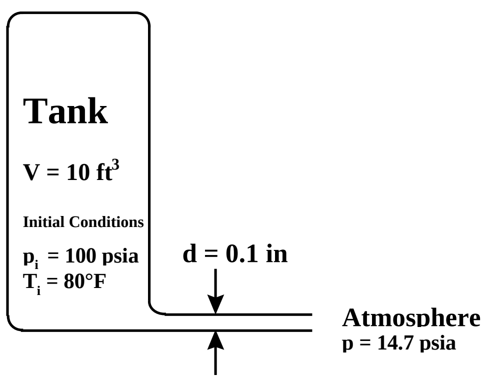
STUDY OF THE RESULTS

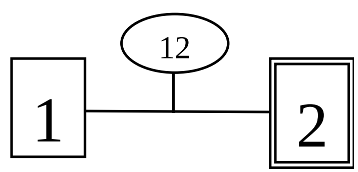
- Plot pressure and flowrate history
  - Peak pressure approximately 620 psia
- Estimate the predicted period of oscillation and compare with the following formula
  - Period of Oscillation = 4L/a
  - Where L = length of the pipe = 400 ft
  - And a = Speed of sound = 2462 ft/sec for LOX
- Plot compressibility (Z) history and note variation of compressibility with time

MATHEMATICAL FORMULATION

$$\frac{m_{\tau+\Delta\tau} - m_{\tau}}{\Delta\tau} = \sum_{j=1}^{j=n} \dot{m}_{ij}$$







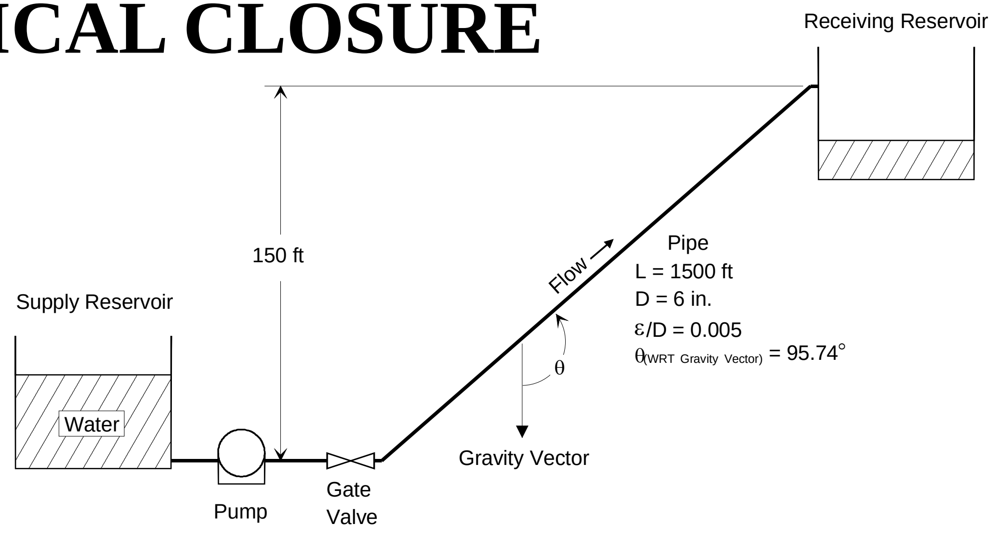
Content

- Mathematical Closure
- Governing Equations
- Solution Procedure

MATHEMATICAL CLOSURE

Problem of a Steady State Flow Network

- **Given** : Pressures and Temperatures at Boundary Nodes
- **Find** : Pressures and Temperatures at Internal Nodes and Flowrates in Branches

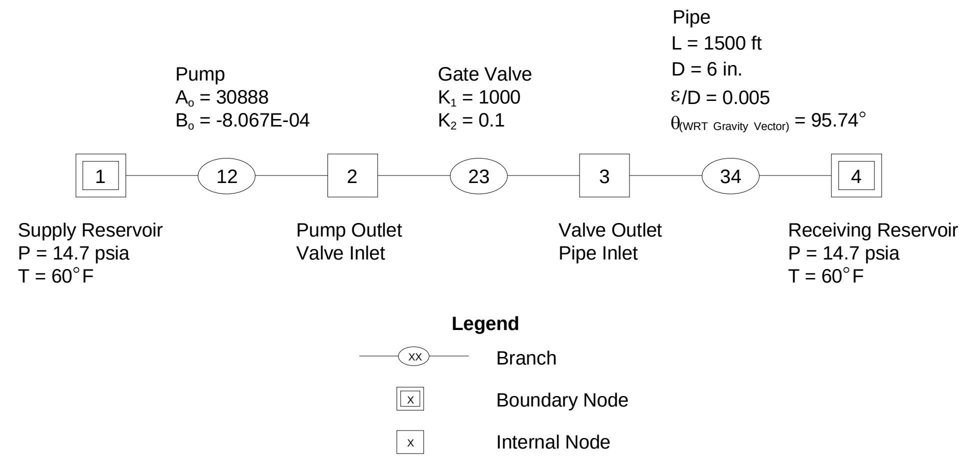


Primary Variables

$$p_2, p_3, T_2, T_3, \dot{m}_{12}, \dot{m}_{23}, \dot{m}_{34}$$

Secondary Variables

$$\rho_2, \rho_3, \mu_2, \mu_3$$



MATHEMATICAL CLOSURE

Problem of an Unsteady Flow Network

• **Given** : Pressures and Temperatures at Boundary Nodes and Initial Values at Internal Nodes

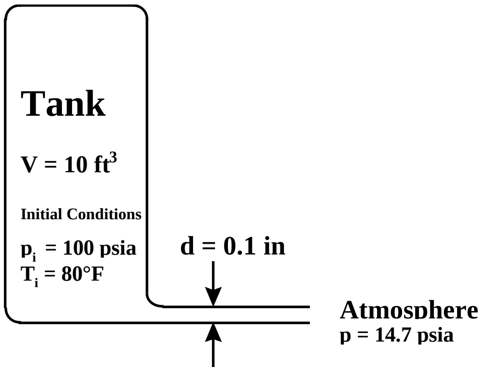
• **Find** : Pressures and Temperatures at Internal Nodes and Flowrates in Branches with Time.

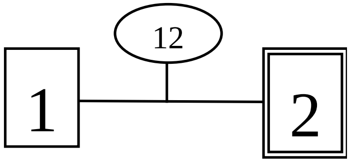
Primary Variables

$$p_1(\tau), T_1(\tau), m_1(\tau), \dot{m}(\tau)$$

Secondary Variables

$$\rho_1(\tau),\ \mu_1(\tau)$$





MATHEMATICAL CLOSURE

Problem of an Unsteady Flow with Conjugate Heat Transfer

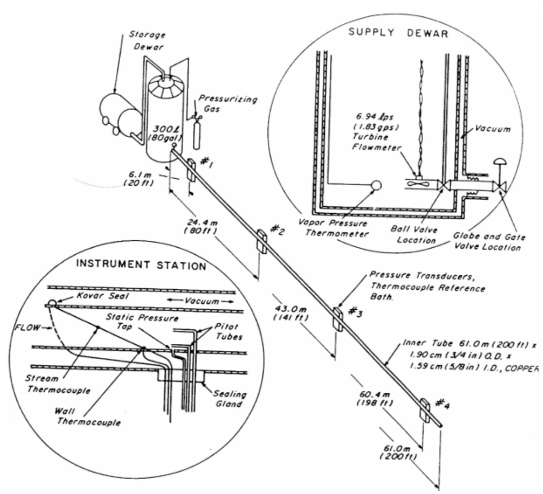
- **Given** : Pressures and Temperatures at Boundary Nodes and Initial Values at Internal Fluid Nodes and Solid Nodes
- **Find** : Pressures and Temperatures at Internal Nodes and Flowrates in Branches with Time.

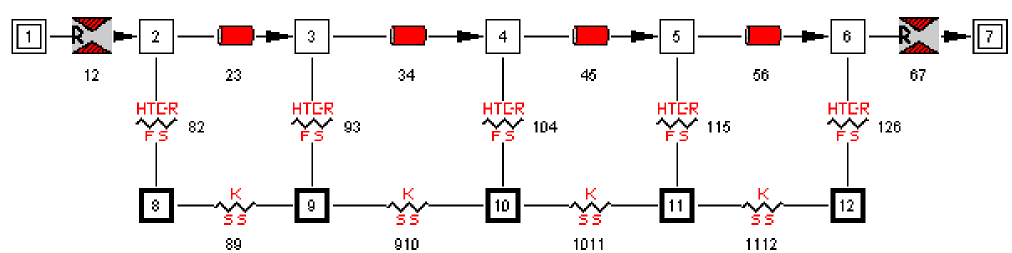
Primary Variables

$$p\ (\tau),\ T\ (\tau), m\ (\tau)\ , \dot{m}(\tau), T_s(\tau)$$

Secondary Variables

$$\rho_1(\tau), \mu_1(\tau), k(\tau), h_c(\tau)$$





Mathematical Closure

Unknown Variables	Available Equations to Solve
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. Species Concentration	5. Conservation Equations for Mass Fraction of Species
6. Mass	6. Thermodynamic Equation of State

MATHEMATICAL CLOSURE

Secondary Variables:

Thermodynamic & Thermophysical Properties

Unknown Variable	**Available Equations to Solve**
Density	
Specific Heats	Equilibrium Thermodynamic Relations
Viscosity	[GASP, WASP & GASPAK Property Programs]
Thermal Conductivities	
Friction Factor	Empirical Relations
Heat Transfer Coefficients	

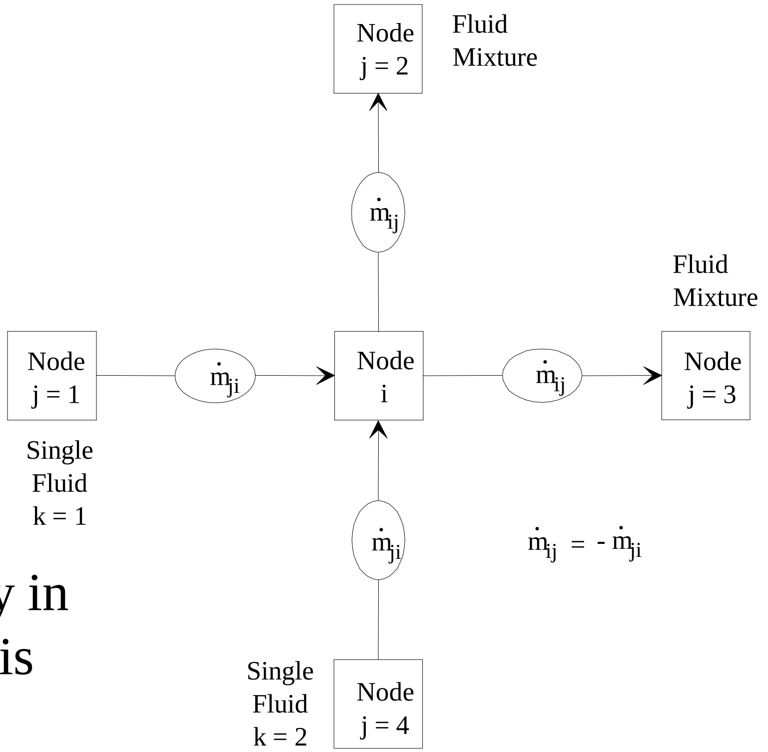
GOVERNING EQUATIONS

- Mass Conservation
- Momentum Conservation
- Energy Conservation of fluid
- Energy Conservation of solid
- Fluid Species Conservation
- Equation of State
- Mixture Property

GOVERNING EQUATIONS

MASS CONSERVATION EQUATION

$$\frac{m_{\tau+\Delta\tau} - m_{\tau}}{\Delta\tau} = \sum_{j=1}^{j=n} \dot{m}_{ij}$$



Note : Pressure does not appear explicitly in Mass Conservation Equation although it is earmarked for calculating pressures

GOVERNING EQUATIONS

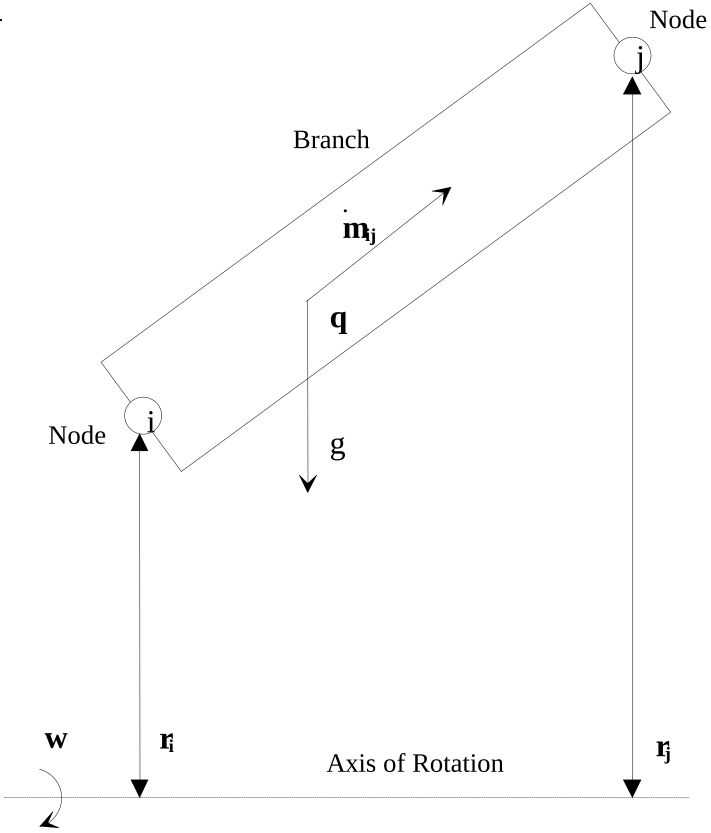
MOMENTUM CONSERVATION EQUATION

- Represents Newton's Second Law of Motion

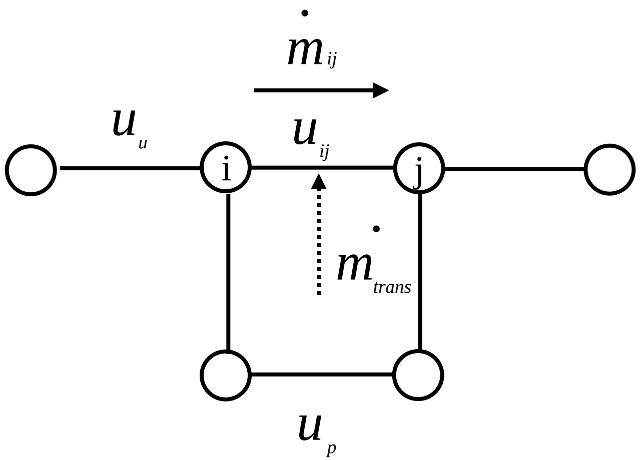
Mass x Acceleration = *Forces*

- Unsteady
- Longitudinal Inertia
- Transverse Inertia

- Pressure
- Gravity
- Friction
- Centrifugal
- Shear Stress
- Moving Boundary
- Normal Stress
- External Force



MOMENTUM CONSERVATION EQUATION



Mass x Acceleration Terms in GFSSP

Unsteady

$$\frac{(mu_{ij})_{\tau+\Delta\tau} - (mu_{ij})_{\tau}}{g_c \Delta\tau}$$

Longitudinal Inertia

$$MAX\left|\dot{m}_{ij},0\right|\left(u_{ij} - u_{u}\right) - MAX\left|-\dot{m}_{ij},0\right|\left(u_{ij} - u_{u}\right)$$

Transverse Inertia

$$+MAX\left|\dot{m}_{trans},0\right|\left(u_{ij} - u_{p}\right) - MAX\left|-\dot{m}_{trans},0\right|\left(u_{ij} - u_{p}\right)$$

MOMENTUM CONSERVATION EQUATION

Force Terms in GFSSP

Pressure

$$(p_i - p_j)A_{ij}$$

Gravity

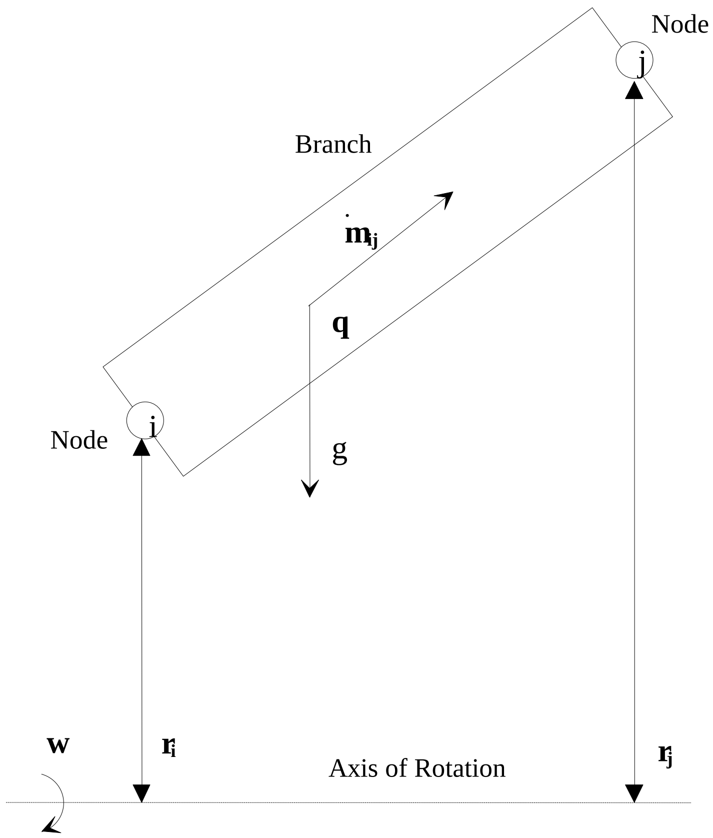
$$\frac{\rho g V Cos\theta}{g_c}$$

Friction

$$-K_f \dot{m}_{ij} \left|\dot{m}_{ij}\right| A_{ij}$$

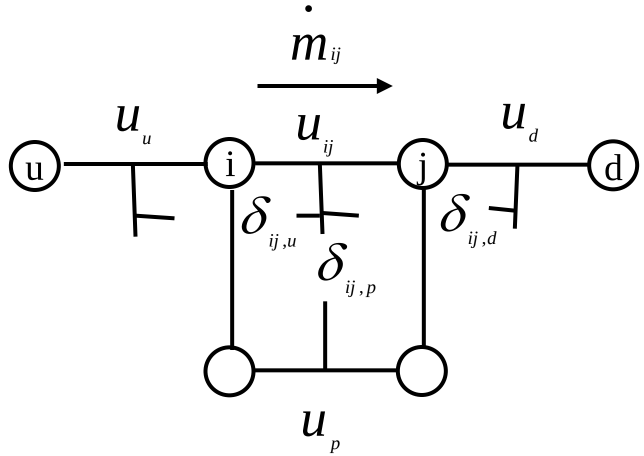
Centrifugal

$$\frac{\rho K_{rot}^2 \omega^2 A}{g_c}$$



MOMENTUM CONSERVATION EQUATION

Force Terms in GFSSP



Shear Stress

$$\mu \frac{u_p - u_{ij}}{g_c \delta_{ij,p}} A_s$$

Normal Stress

$$\left(\mu_d \frac{u_d - u_{ij}}{\delta_{ij,d}} - \mu_u \frac{u_{ij} - u_u}{\delta_{ij,u}} \right) \frac{A_{ij}}{g_c}$$

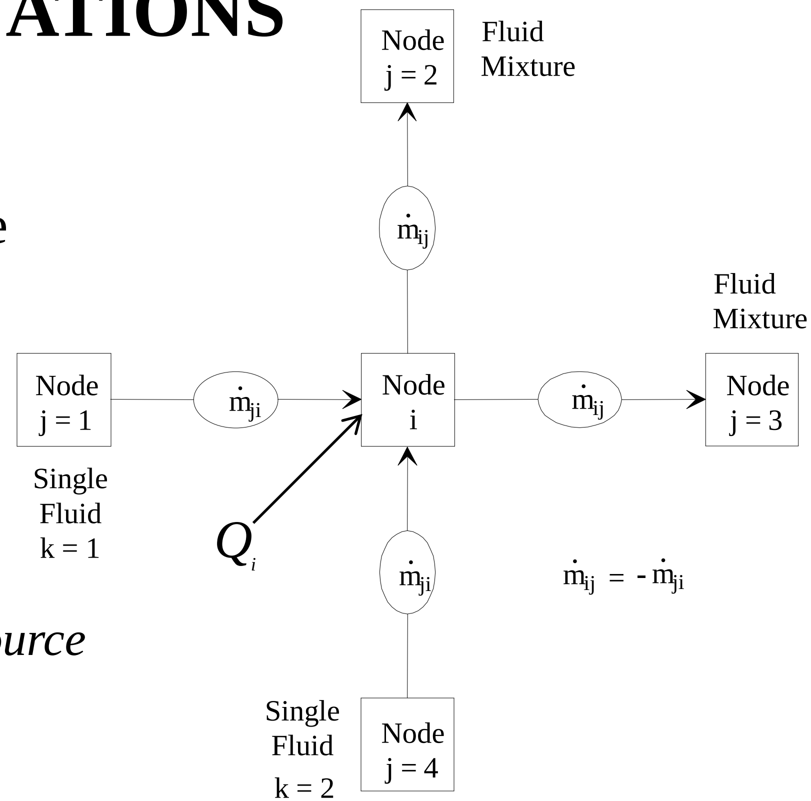
Moving Boundary

$$-\rho A_{norm} u_{norm} u_{ij} / g_c$$

GOVERNING EQUATIONS

ENERGY CONSERVATION EQUATION

- Energy Conservation Equation can be written in Enthalpy or Entropy
  - Based on Upwind Scheme



Enthalpy Equation

Rate of Increase of Internal Energy =

Enthalpy Inflow - Enthalpy Outflow + Heat Source

$$\frac{m\left(h-\dfrac{p}{\rho J}\right)_{\tau+\Delta\tau}-m\left(h-\dfrac{p}{\rho J}\right)_{\tau}}{\Delta\tau}=\sum_{j=1}^{j=n}\left\{MAX\left[-\dot{m}_{ij},0\right]h_j-MAX\left[\dot{m}_{ij},0\right]h_i\right\}+Q_i$$

GOVERNING EQUATIONS

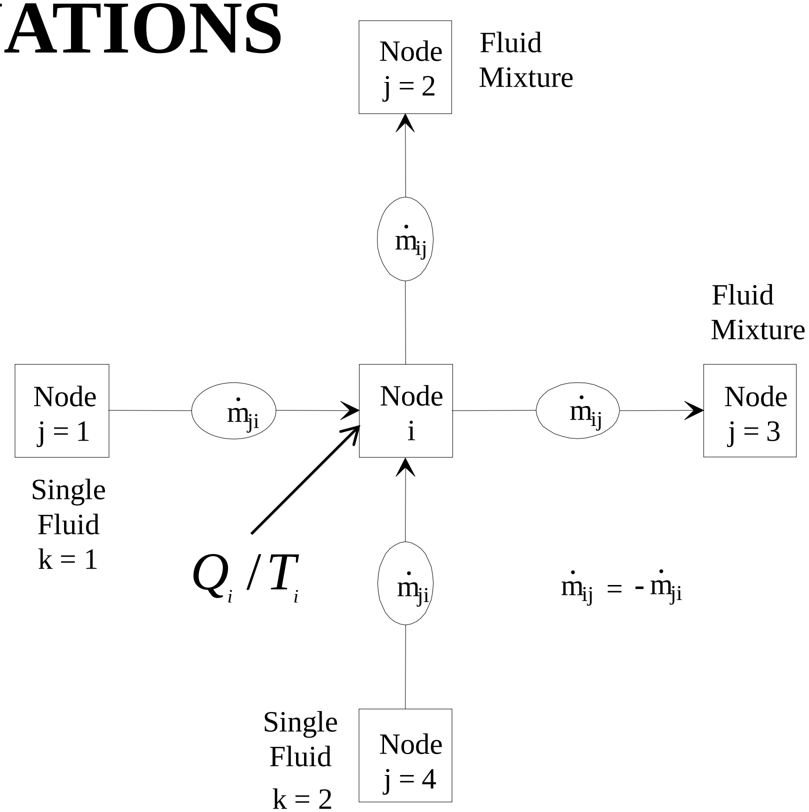
ENERGY CONSERVATION EQUATION

Entropy Equation

Rate of Increase of Entropy =

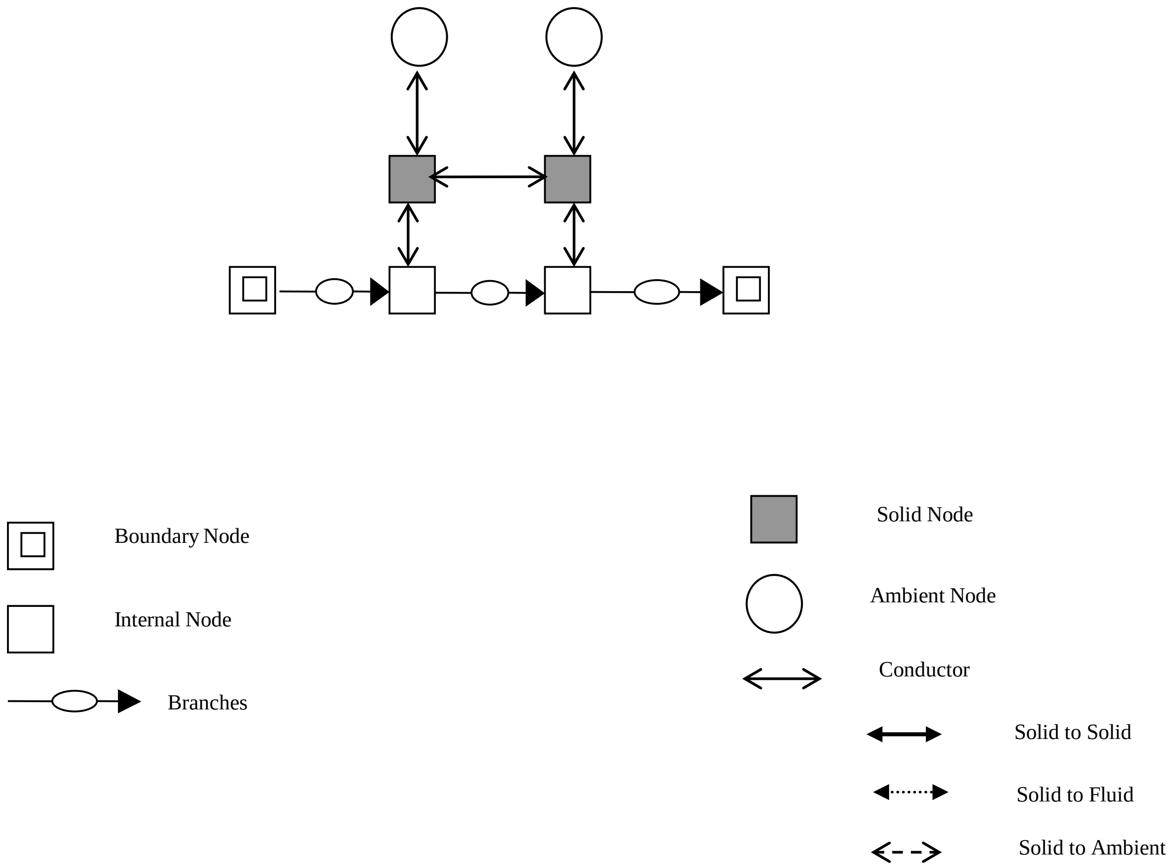
Entropy Inflow - Entropy Outflow +

Entropy Generation + Entropy Source

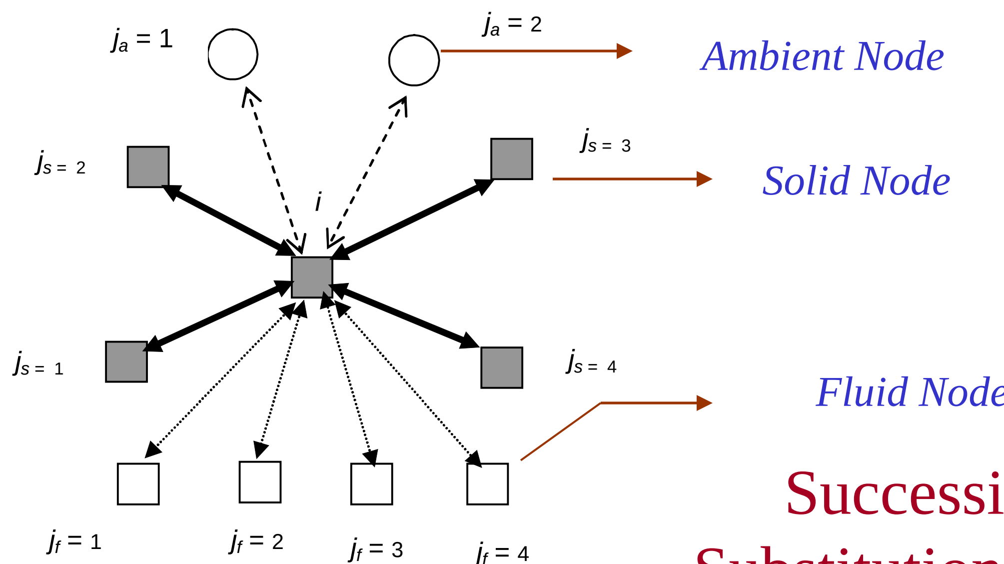


$$\frac{(ms)_{\tau+\Delta\tau}-(ms)_{\tau}}{\Delta\tau} = \sum_{j=1}^{j=n}\left\{MAX[-\dot m_{ij},0]s_j - MAX[\dot m_{ij},0]s_i\right\} + \sum_{j=1}^{j=n}\left\{\frac{MAX[-\dot m_{ij},0]}{|\dot m_{ij}|}\right\}\dot S_{ij,gen} + \frac{Q_i}{T_i}$$

Solid – Fluid Network for Conjugate Heat Transfer



Energy Conservation for Solid Node



Conservation Equation

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

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

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

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

Successive Substitution Form

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

Heat Transfer Coefficient

Dittus-Boelter Equation for Single Phase

$$Nu = \frac{h_c D}{k_v}$$

$$Nu = 0.023(\mathrm{Re}\ )^{0.8}(\mathrm{Pr}_v)^{0.33}$$

$$\mathrm{Re}\ = \frac{\rho u D}{\mu_v}$$

$$\mathrm{Pr}_v = \left(\frac{C_p \mu_v}{k_v}\right)$$

Modified Miropolski's Correlation for Two Phase Flow

$$Nu = 0.023(\mathrm{Re}_{mix})^{0.8}(\mathrm{Pr}_v)^{0.4}(Y)$$

$$\mathrm{Re}_{mix} = \left(\frac{\rho u D}{\mu_v}\right)\left[x + \left(\frac{\rho_v}{\rho_l}\right)(1-x)\right]$$

$$\mathrm{Pr}_v = \left(\frac{C_p \mu_v}{k_v}\right)$$

$$Y = 1 - 0.1\left(\frac{\rho_l}{\rho_v} - 1\right)^{0.4}(1-x)^{0.4}$$

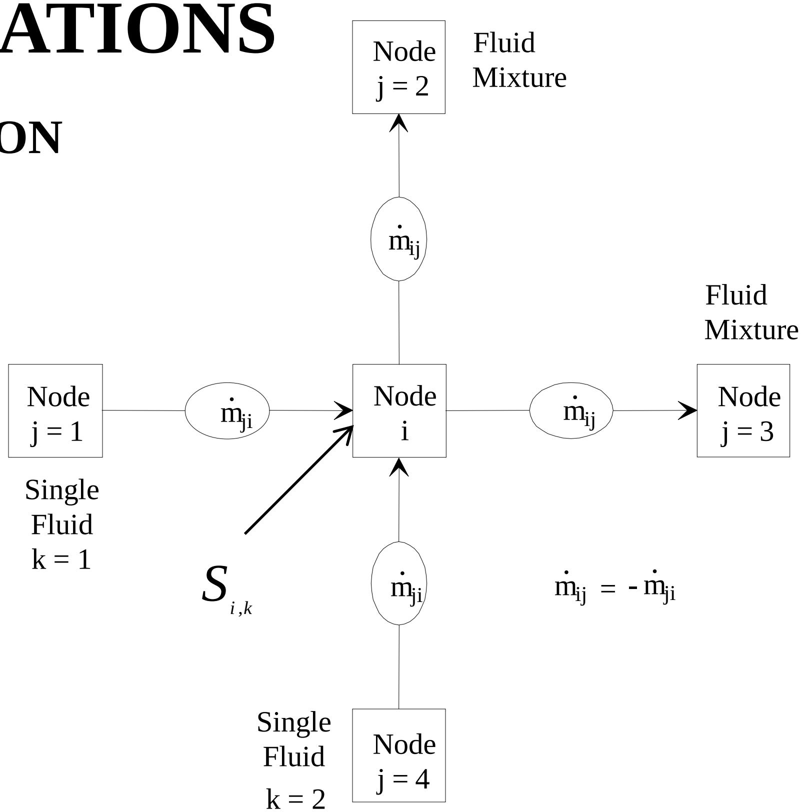
GOVERNING EQUATIONS

FLUID SPECIES CONSERVATION EQUATION

Rate of Increase of Fluid Specie =

Fluid Specie Inflow - Fluid Specie Outflow +

Fluid Specie Source



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

GOVERNING EQUATIONS

EQUATION OF STATE

For unsteady flow, resident mass in a control volume is calculated from the equation of state for a real fluid

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

Z is the compressibility factor determined from higher order equation of state

GOVERNING EQUATIONS

EQUATION OF STATE

- GFSSP uses two separate Thermodynamic Property Packages GASP/WASP and GASPAK
- GASP/WASP uses modified Benedict, Webb & Rubin (BWR) Equation of State
- GASPAK uses "standard reference" equation from
  - National Institute of Standards and Technology (NIST)
  - International Union of Pure & Applied Chemistry (IUPAC)
  - National Standard Reference Data Service of the USSR

GOVERNING EQUATIONS

Mixture Property Relation

Density

- Calculated from Equation of State of Mixture with Compressibility Factor

$$\rho_i = \frac{p_i}{z_i R_i T_i} \qquad R_i = \sum_{k=1}^{k=n} x_k R_k$$

- Compressibility Factor of Mixture is Mole average of Individual Components

$$z_i = \sum_{k=1}^{k=n} x_k z_k \qquad z_k = \frac{p_i}{\rho_k R_k T_k}$$

GOVERNING EQUATIONS

Mixture Property Relation

Thermophysical Properties

• Viscosity, Specific Heat and Specific Heat Ratios are calculated by taking Molar Average

$$\mu_i = \sum_{k=1}^{k=n} \bar{x}_k \mu_k$$

$$\gamma_i = \sum_{k=1}^{k=n} \bar{x}_k \gamma_k$$

$$C_{p,i} = \sum_{k=1}^{k=n} \frac{\sum_{k=1}^{k=n} C_{p,k} \bar{x}_k M_k}{\sum_{k=1}^{k=n} \bar{x}_k M_k}$$

GOVERNING EQUATIONS

Mixture Property Relation

Temperature

- Mixture Temperature is calculated from Energy Conservation Equation

$$(T_i)_{\tau+\Delta\tau} = \frac{\sum_{j=1}^{j=n}\sum_{k=1}^{k=nf} \mathrm{Cp}_k\, x_k T_j\, MAX\left[-\dot{m}_{ij}, 0\right] + \left(C_{p,i} m_i T_i\right)_\tau / \Delta\tau + Q_i}{\sum_{j=1}^{j=n}\sum_{k=1}^{k=nf} \mathrm{Cp}_k\, x_k\, MAX\left[\dot{m}_{ij}, 0\right] + \left(C_{p,i} m\right)_\tau / \Delta\tau}$$

Limitation

- Cannot handle phase change of mixture

GOVERNING EQUATIONS
Summary

- Familiarity with GFSSP's Governing Equations is not absolutely necessary to use the code
- However, working knowledge about Governing Equations is helpful to implement various options in a complex flow network
- A good understanding of Governing Equations is necessary to introduce new physics in the code

SOLUTION PROCEDURE

- Successive Substitution
- Newton-Raphson
- Simultaneous Adjustment with Successive Substitution (SASS)
- Convergence

SOLUTION PROCEDURE

- Non linear Algebraic Equations are solved by
  - Successive Substitution
  - Newton-Raphson
- GFSSP uses a Hybrid Method
  - SASS (Simultaneous Adjustment with Successive Substitution)
  - This method is a combination of Successive Substitution and Newton-Raphson

SOLUTION PROCEDURE

SUCCESSIVE SUBSTITUTION METHOD

STEPS:

1. Guess a solution for each variable in the system of equations
2. Express each equation such that each variable is expressed in terms of other variables: e. g. X = f (Y,Z) and Y = f (X,Z) etc
3. Solve for each variable
4. Under-relax the variable, if necessary
5. Repeat steps 1 through 4 until convergence

ADVANTAGES:

Simple to program; takes less computer memory

DISADVANTAGES:

It is difficult to make a decision in which order the equations must be solved to ensure convergence

SOLUTION PROCEDURE

NEWTON-RAPHSON METHOD

STEPS:

1. **Guess a solution for each variable in the system of equations**
2. **Calculate the residuals of each equation**
3. **Develop a set of correction equations for all variables**
4. **Solve for the correction equations by Gaussian Elimination method**
5. **Apply correction to each variable**
6. **Iterate until the corrections become very small**

ADVANTAGES:

No decision making process is involved to determine the order in which equations must be solved

DISADVANTAGES:

Requires more computer memory; difficult to program.

SOLUTION PROCEDURE

SASS (Simultaneous Adjustment with Successive Substitution) Scheme

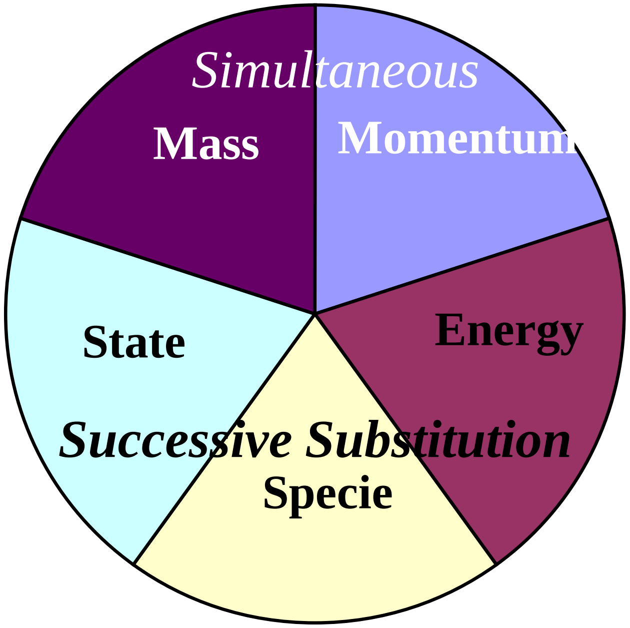
- SASS is a combination of successive substitution and Newton-Raphson method
- Mass conservation and flowrate equations are solved by Newton-Raphson method
- Energy Conservation and concentration equations are solved by successive substitution method
- Underlying principle for making such division:
  - Equations which have strong influences to other equations are solved by the Newton-Raphson method
  - Equations which have less influence to other are solved by the successive substitution method
- This practice reduces code overhead while maintains superior convergence characteristics

GFSSP Solution Scheme

SASS : **S**imultaneous **A**djustment with **S**uccessive **S**ubstitution

Approach : Solve simultaneously when equations are strongly coupled and non-linear

Advantage : Superior convergence characteristics with affordable computer memory



CONVERGENCE

- Numerical solution can only be trusted when fully converged
- GFSSP's convergence criterion is based on difference in variable values between successive iterations. Normalized Residual Error is also monitored
- GFSSP's solution scheme has two options to control the iteration process
  - Simultaneous (SIMUL = TRUE)
  - Non-Simultaneous (SIMUL = FALSE)

CONVERGENCE

Simultaneous Option

- Single Iteration Loop
  - First solve mass, momentum and equation of state by the Newton-Raphson (NR) scheme
  - Next solve energy and specie conservation equation by Successive Substitution (SS) scheme
  - Solution is converged when the normalized maximum correction, Δ_{max} is less than the convergence criterion

$$\Delta_{\max} = MAX\left|\sum_{i=1}^{N_E} \frac{\Phi_i'}{\Phi_i}\right|$$

N_E is the total number of equations solved by the Newton-Raphson scheme

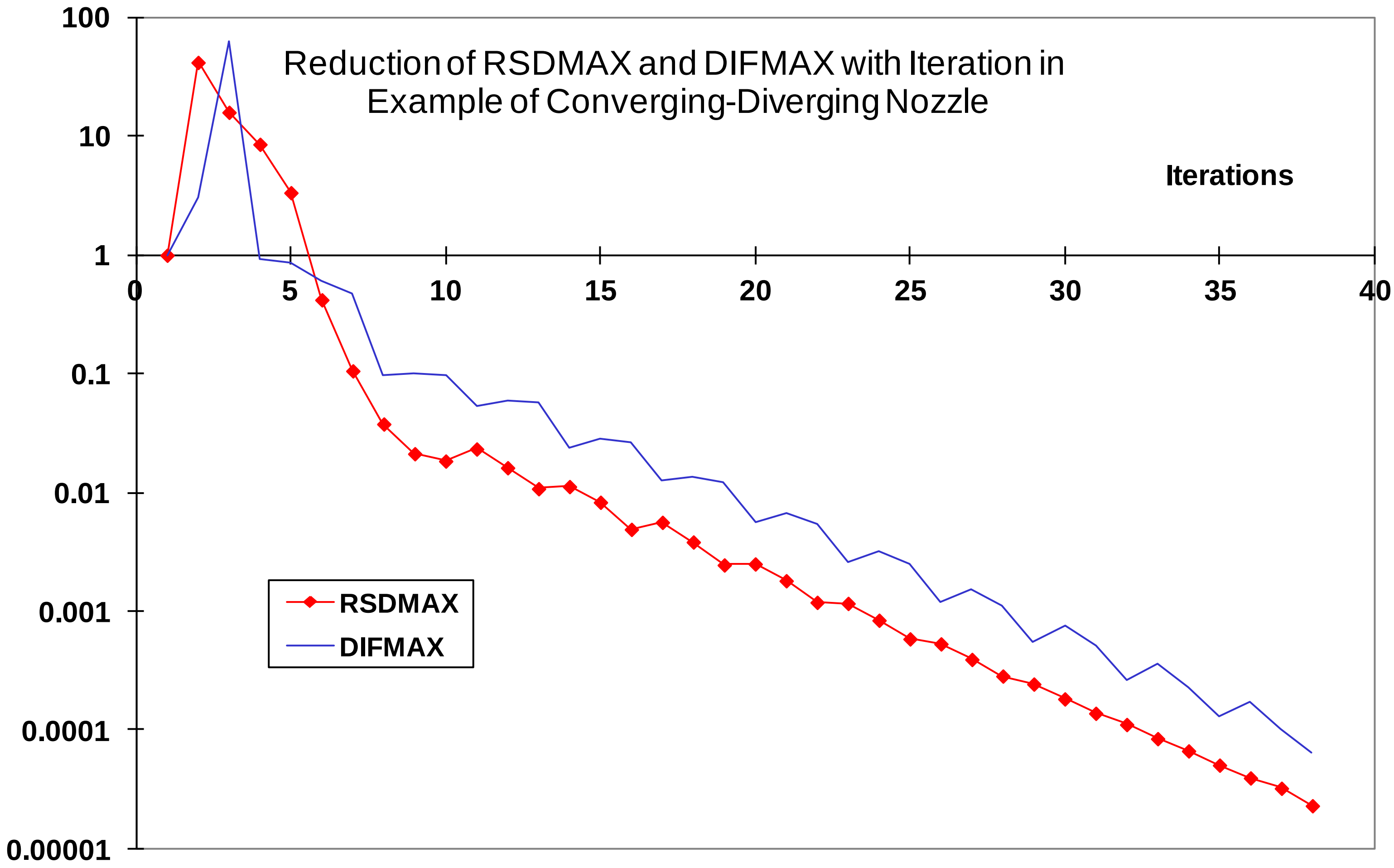
CONVERGENCE

Non-Simultaneous Option

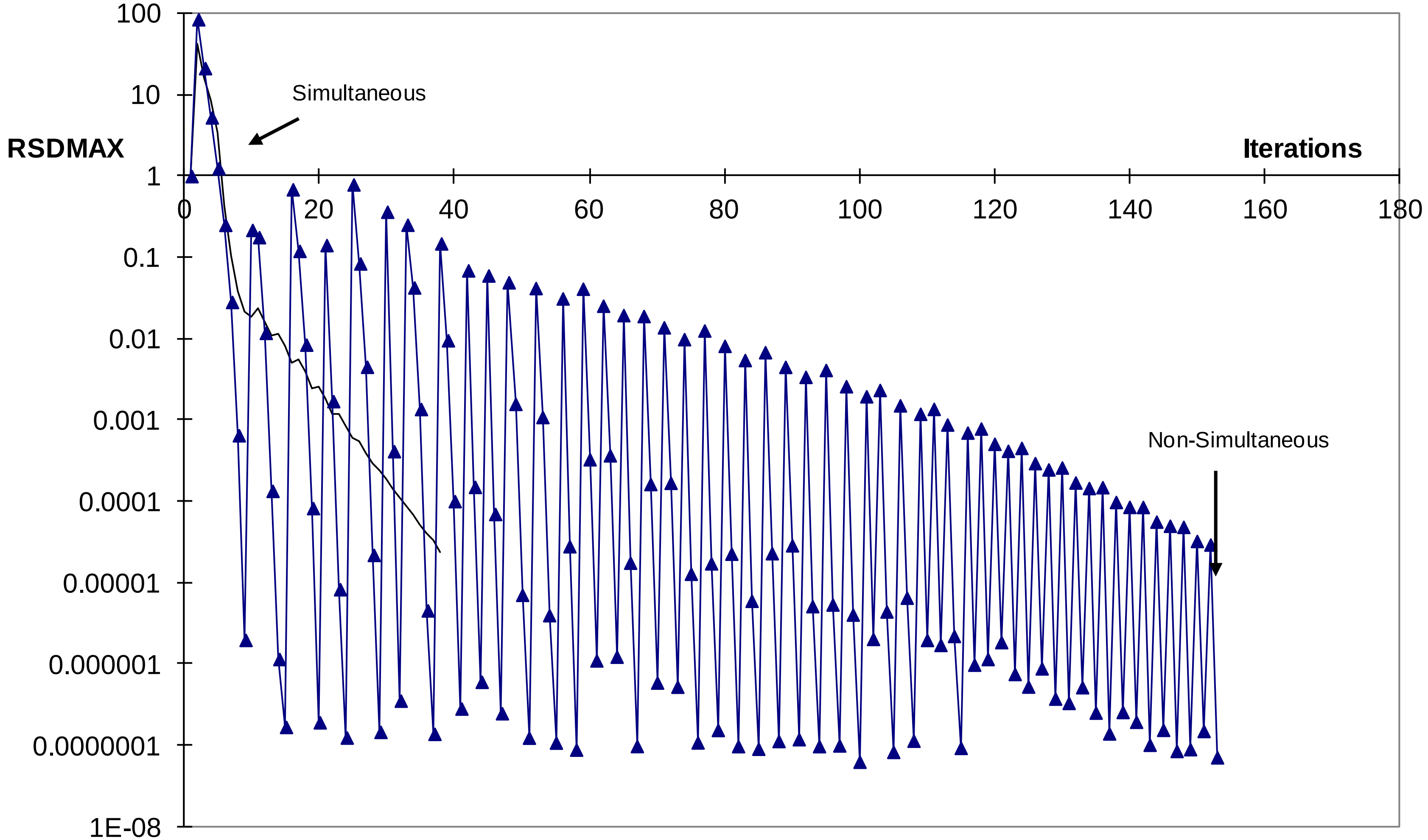
- Inner & Outer Iteration Loop
  - Mass, Momentum and Equation of state is solved in inner iteration loop by NR scheme
  - Energy and Specie conservation equations are solved in outer iteration loop by SS scheme
  - Convergence of NR scheme is determined by Δ_{max}
  - Convergence of SS scheme is determined by Δ^{o}_{max}

$$\Delta^{o}_{max} = MAX\left|\Delta_{K_f}, \Delta_{\rho}, \Delta_{h} \text{ or } \Delta_{s}\right| \qquad \Delta_{K_f} = MAX\left|\sum_{i=1}^{N_B}\frac{K'_f}{K_f}\right| \quad \text{etc.}$$

Convergence Characteristics For Simultaneous Option



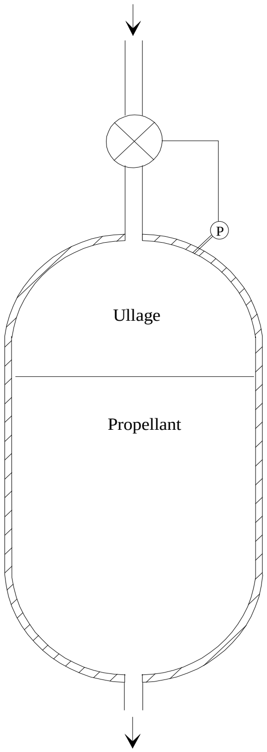
Comparison of Convergence Characteristics between Simultaneous and Non-Simultaneous Option in Converging-Diverging Nozzle



SOLUTION PROCEDURE
Summary

- Simultaneous option is more efficient than Non-Simultaneous option
- Non-Simultaneous option is recommended when Simultaneous option experiences numerical instability
- Under-relaxation and good initial guess also help to overcome convergence problem
- A lack of realism in problem specification can lead to convergence problem
- Lack of realism includes:
  - Unrealistic geometry and/or boundary conditions
  - Attempt to calculate properties beyond operating range

Tank Pressurization, Control Valves, and Relief Valves



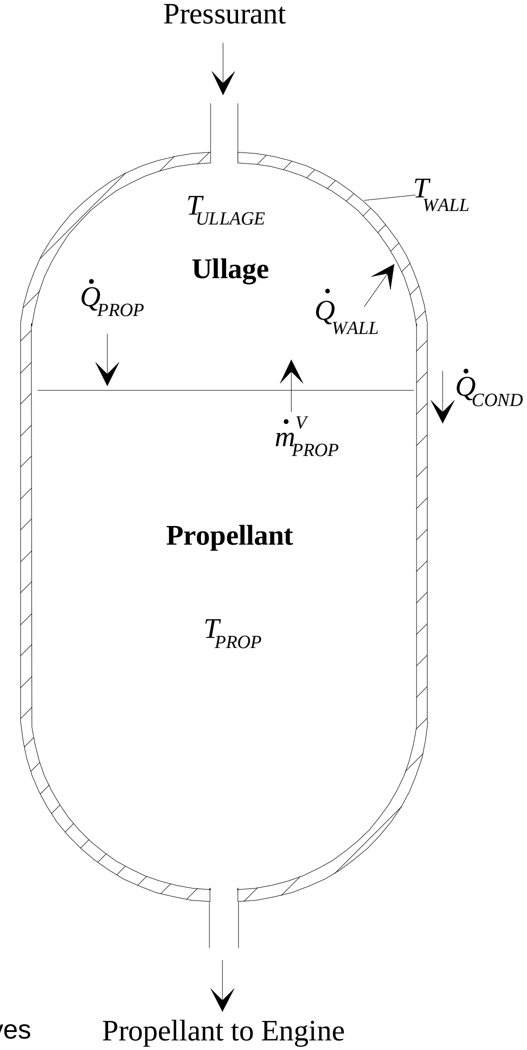
Content

- Tank Pressurization
- Control Valve
- Relief Valve

TANK PRESSURIZATION

- Predict the ullage conditions considering heat and mass transfer between the propellant and the tank wall

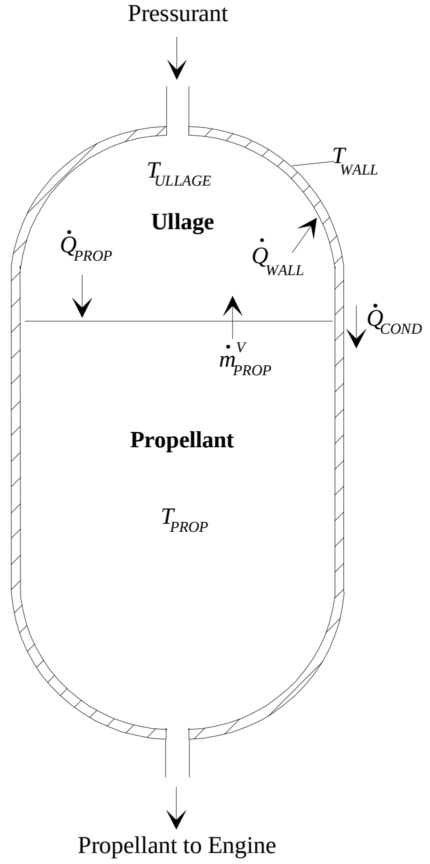
- Predict the propellant conditions leaving the tank



TANK PRESSURIZATION

ADDITIONAL PHYSICAL PROCESSES

- Change in ullage and propellant volume.
- Change in gravitational head in the tank.
- Heat transfer from pressurant to propellant.
- Heat transfer from pressurant to the tank wall.
- Heat conduction between the pressurant exposed tank surface and the propellant exposed tank surface.
- Mass transfer between the pressurant and propellant.



TANK PRESSURIZATION

MATHEMATICAL MODELING OF PHYSICAL PROCESSES -1

Change in Ullage and Propellant Volume

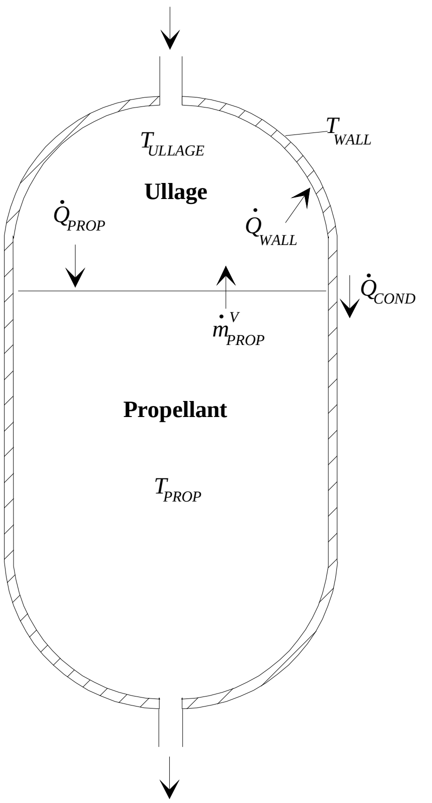
$$dV_{ullage} = \frac{\dot{m}_{prop} \Delta\tau}{\rho_{prop}} = -dV_{prop}$$

Conservation Equation of Volume

$$V_{ullage} + V_{prop} = V_{tank}$$

$$V_{prop}^{\tau+\delta\tau} = V_{prop}^{\tau} - dV_{prop}$$

$$V_{ullage}^{\tau+\delta\tau} = V_{ullage}^{\tau} + dV_{ullage}^{\tau+\delta\tau}$$



TANK PRESSURIZATION

MATHEMATICAL MODELING OF PHYSICAL PROCESSES -2

Change in Gravitational Head in the Tank

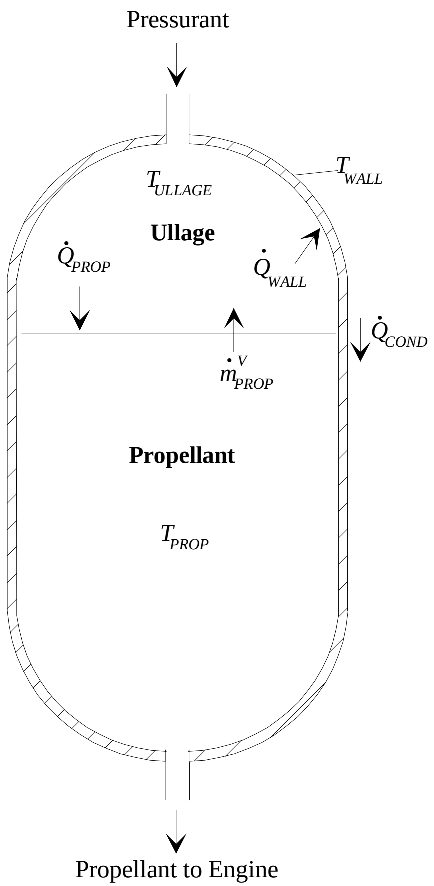
$$p_{tank\ bottom} = p_{ullage} + \frac{\rho_{prop} g H}{g_c}$$

Heat Transfer from Ullage to Propellant

$$\dot{Q}_{prop} = [h_c A]_{ullage-prop} \left(T_{ullage} - T_{prop}\right)$$

Heat Transfer Coefficient (Natural Convection)

$$h_c = K_H C \frac{k_f}{L_s} X^n$$



TANK PRESSURIZATION

MATHEMATICAL MODELING OF PHYSICAL PROCESSES -3

Heat Transfer from Ullage to Wall

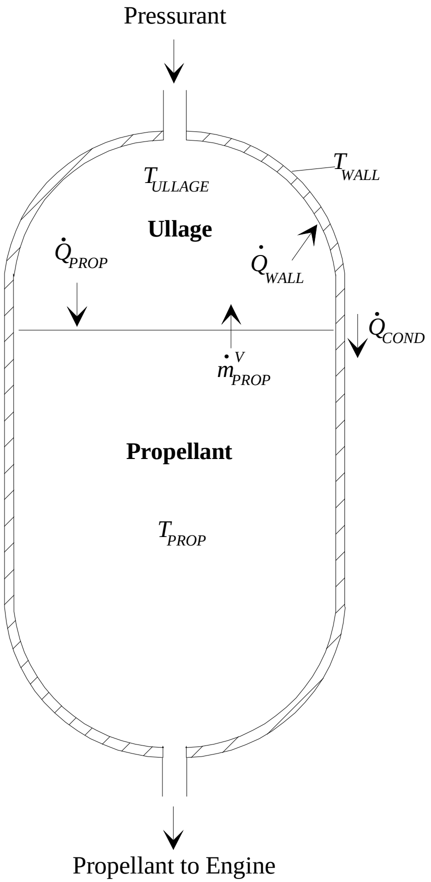
$$\dot{Q}_{wall} = [h_c A]_{ullage-wall}\left(T_{ullage} - T_{wall}\right)$$

Tank Wall Conduction

$$\dot{Q}_{cond} = k_{tank} A_{cond}\left(T_{wall} - T_{prop}\right)/(H/2)$$

Transient Heat Transfer in the Tank

$$m_{wall} c_{p_{wall}} \frac{\partial T_{wall}}{\partial \tau} = \dot{Q}_{wall} - \dot{Q}_{cond}$$



TANK PRESSURIZATION

MATHEMATICAL MODELING OF PHYSICAL PROCESSES -4

Mass Transfer from Propellant to Ullage

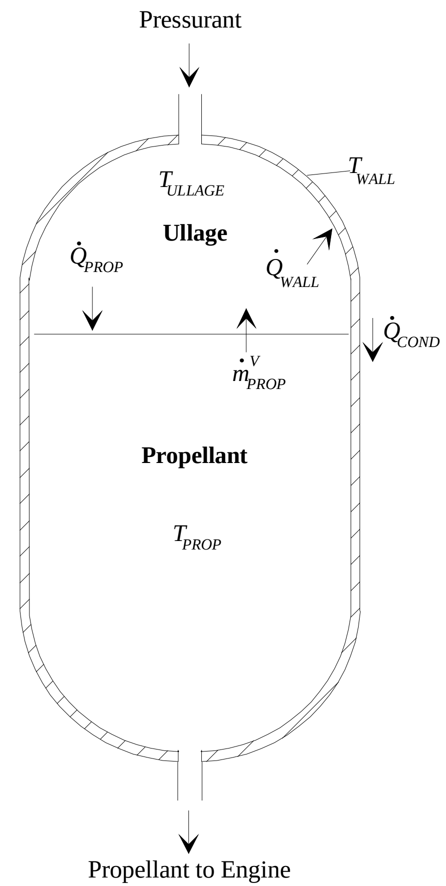
$$\dot{m}^{v}_{prop} = \frac{\dot{Q}_{prop}}{h_{fg} + c_{pf}\left(T_{sat} - T_{prop}\right)}$$

Vapor Pressure Relation

$$\ln p_{sat} = A + \frac{B}{T_{sat}} + C \ln T_{sat} + D T_{sat}$$

Enthalpy of Vaporization

$$h_{fg} = T_{sat}\left(\mathrm{v}_g - \mathrm{v}_f\right)\left.\frac{dP}{dT}\right|_{sat}$$

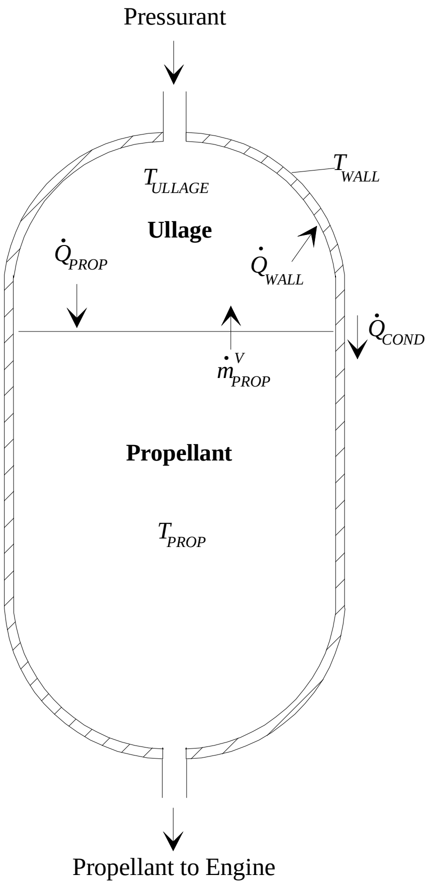


TANK PRESSURIZATION

CALCULATION STEPS

For each time step calculate

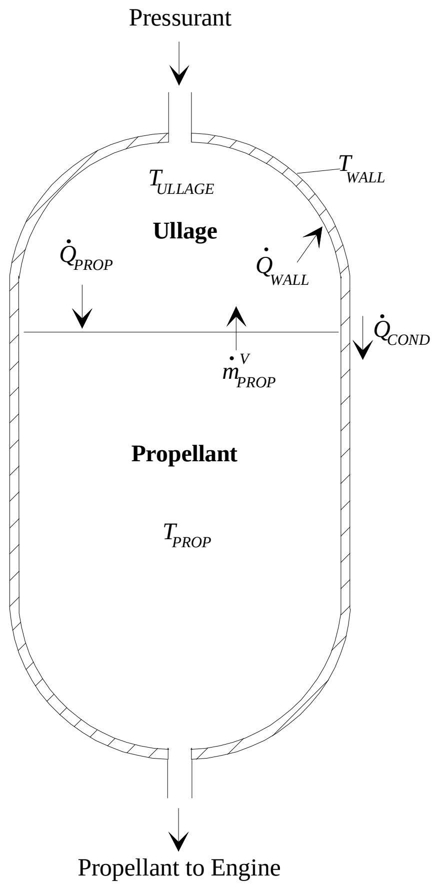
- Ullage and Propellant Volumes
- Tank Bottom Pressure
- Heat Transfer between pressurant and propellant and pressurant and wall
- Wall Temperature
- Mass Transfer from propellant to ullage (only with optional user subroutine)



TANK PRESSURIZATION

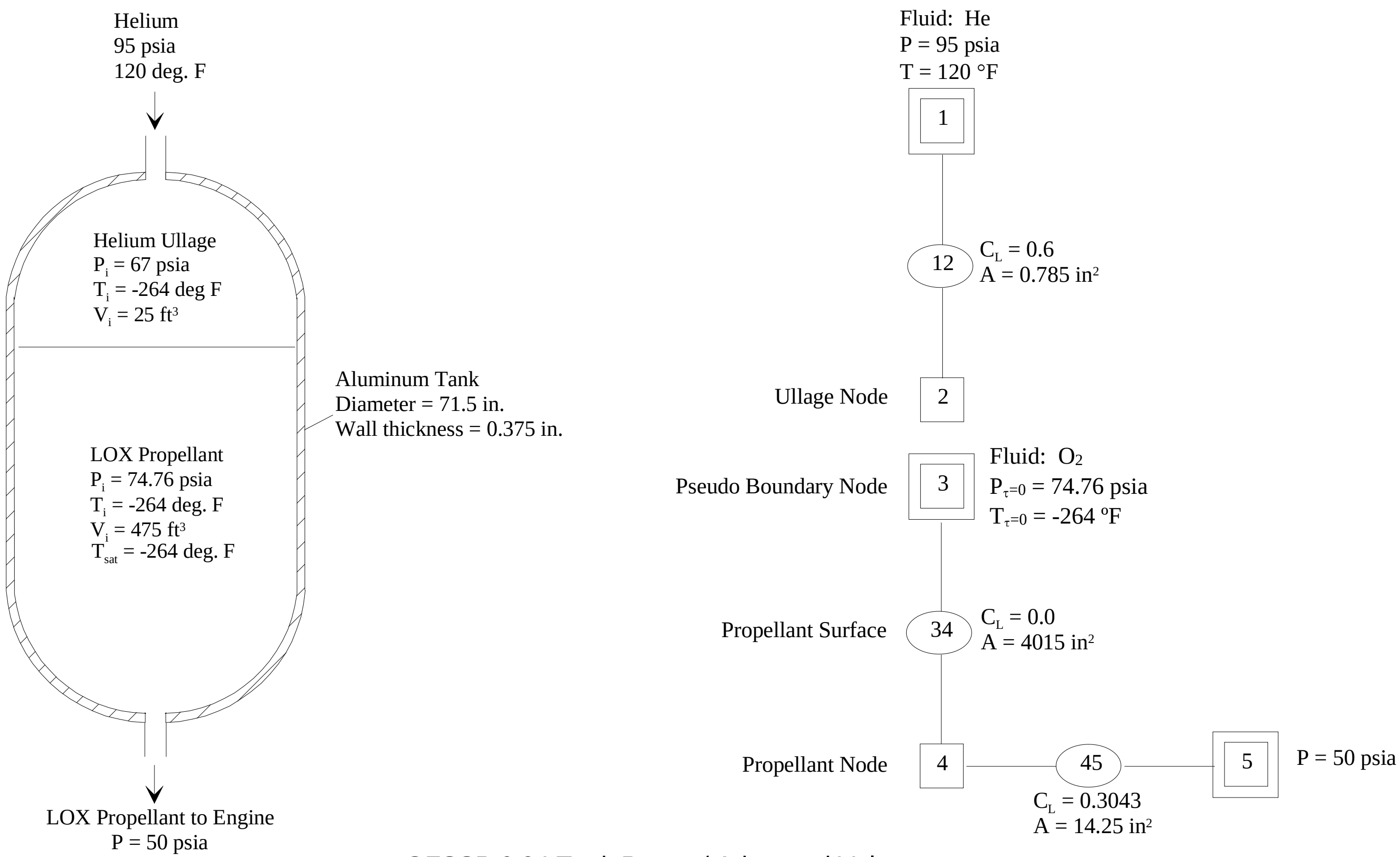
ADDITIONAL INPUT DATA FOR PRESSURIZATION

PRESS	Logical Variable to Activate the Option
NTANK	Number of Tanks in the Circuit
NODUL	Ullage Node
NODULB	Pseudo Boundary Node at interface
NODPRP	Propellant Node
IBRPRP	Branch number connecting NODULB & NODPRP
TNKAR	Tank Surface Area in Ullage at Start, in^2
TNKTH	Tank Thickness, in
TNKRHO	Tank Density, lbm/ft^3
TNKCP	Tank Specific Heat, Btu/lbm - R
TNKCON	Tank Thermal Conductivity, Btu/ft-sec-R
ARHC	Propellant Surface Area, in^2
FCTHC	Multiplying Factor in Heat Transfer Coefficient
TNKTM	Initial Tank Temperature, ^oF

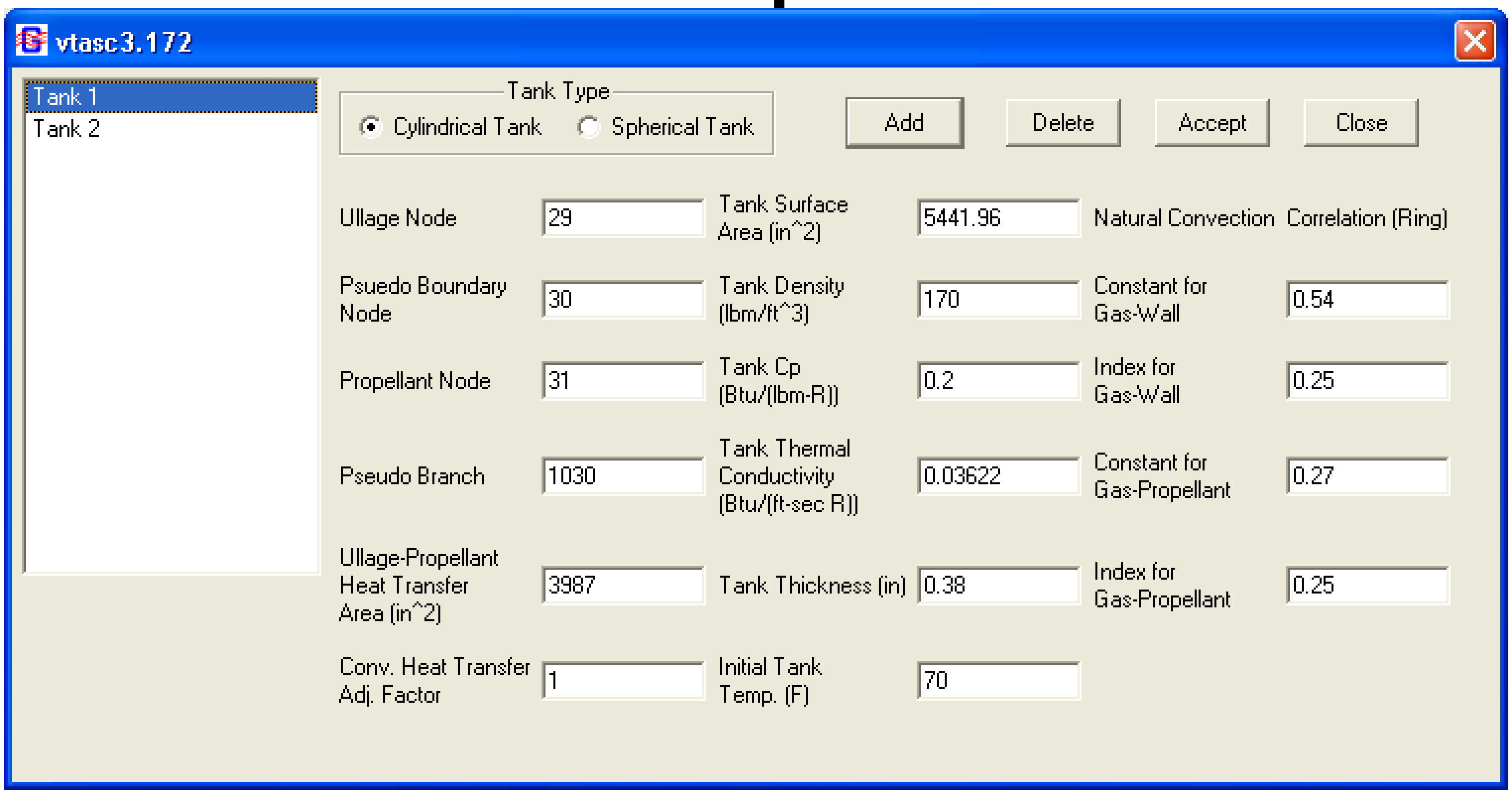


TANK PRESSURIZATION

EXAMPLE 10 TANK SCHEMATIC AND GFSSP MODEL



Additional Input for Pressurization Option



TANK PRESSURIZATION

EXAMPLE 10 PRESSURIZATION OUTPUT

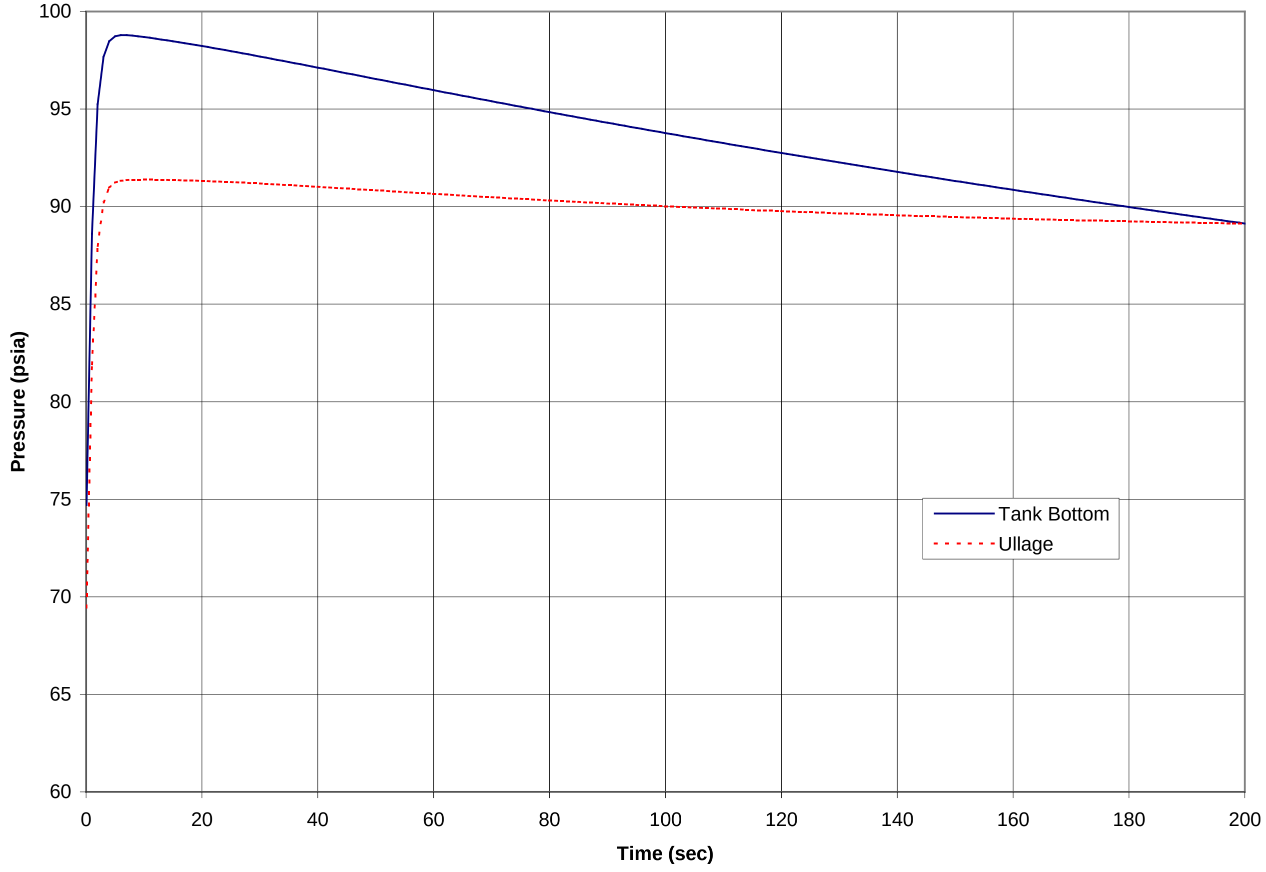
```
SOLUTION
INTERNAL NODES
    NODE      P(PSI)       TF(F)          Z           RHO         EM(LBM)     CONC
                                                 (LBM/FT^3)
                                                                                 HE          O2
     2     0.9138E+02 -0.1347E+03  0.1006E+01  0.1047E+00  0.5144E+01  0.9690E+00  0.0310
     4     0.9869E+02 -0.2640E+03  0.2310E-01  0.6514E+02  0.2937E+05  0.0000E+00  1.0000

BRANCHES
BRANCH   KFACTOR          DELP     FLOW RATE     VELOCITY     REYN. NO.    MACH NO.  ENTROPY GEN.  LOST WORK
 (LBF-S^2/(LBM-FT)^2)  (PSI)    (LBM/SEC)    (FT/SEC)                             BTU/(R-SEC)   LBF-FT/SEC
12     0.238E+05     0.362E+01   0.148E+00   0.445E+03   0.156E+06   0.129E+00   0.281E-02   0.127E+04
34     0.000E+00     0.000E+00   0.163E+03   0.899E-01   0.412E+06   0.114E-03   0.000E+00   0.000E+00
45     0.263E+00     0.487E+02   0.163E+03   0.253E+02   0.690E+07   0.323E-01   0.115E+00   0.176E+05

   NUMBER OF PRESSURIZATION SYSTEMS =   1
  NODUL   NODPRP   QULPRP    QULWAL      QCOND    TNKTM      VOLPROP     VOLULG
     2      4     1.9642     8.5069     0.0022   196.4447   450.8641    49.1359

  SOLUTION SATISFIED CONVERGENCE CRITERION OF    0.100E-02 IN   5 ITERATIONS
TAU =      10.0000     ISTEP =           100
```

Tank Output Units

QULPROP, Btu/s
QULWAL, Btu/s
QCOND, Btu/s
TNKTM, deg. R
VOLPROP, ft^3
VOLULG, ft^3

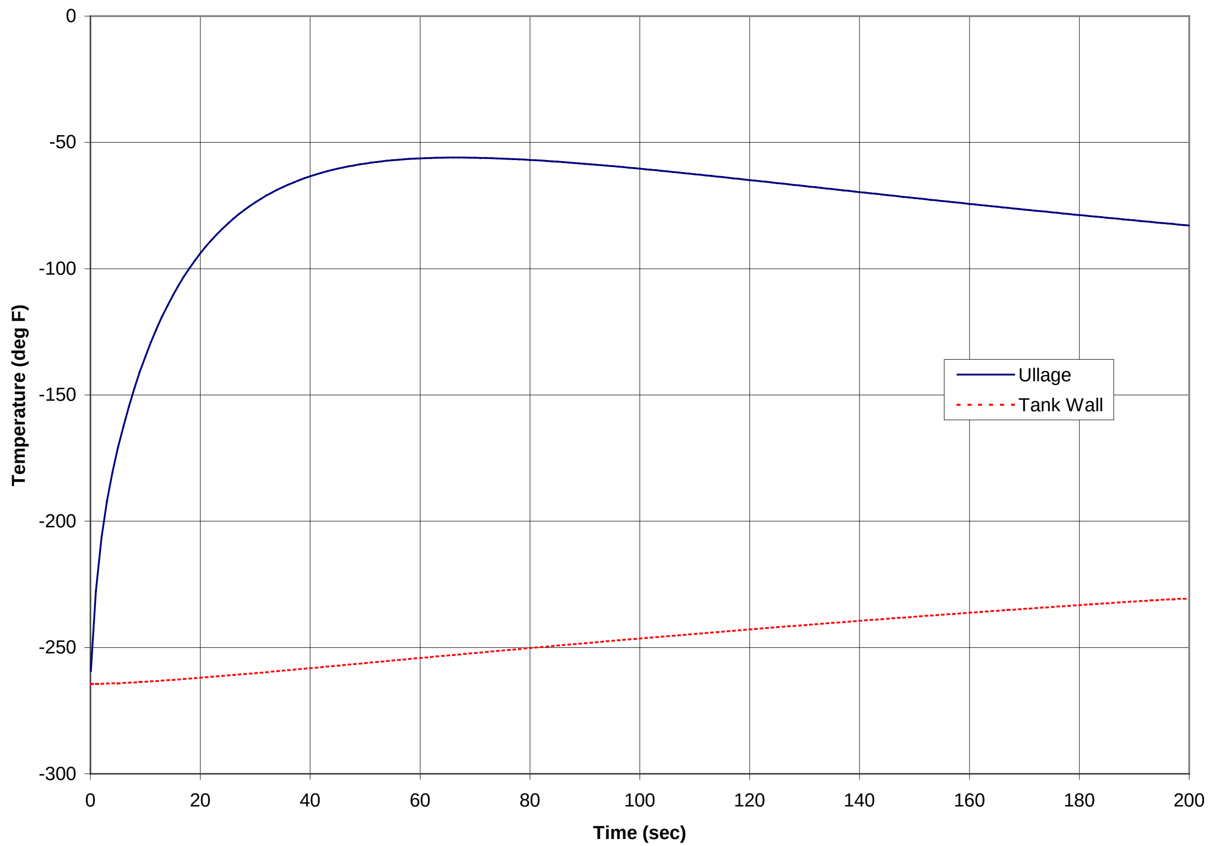
TANK PRESSURIZATION

EXAMPLE 10 ULLAGE AND TANK BOTTOM PRESSURE HISTORY



TANK PRESSURIZATION

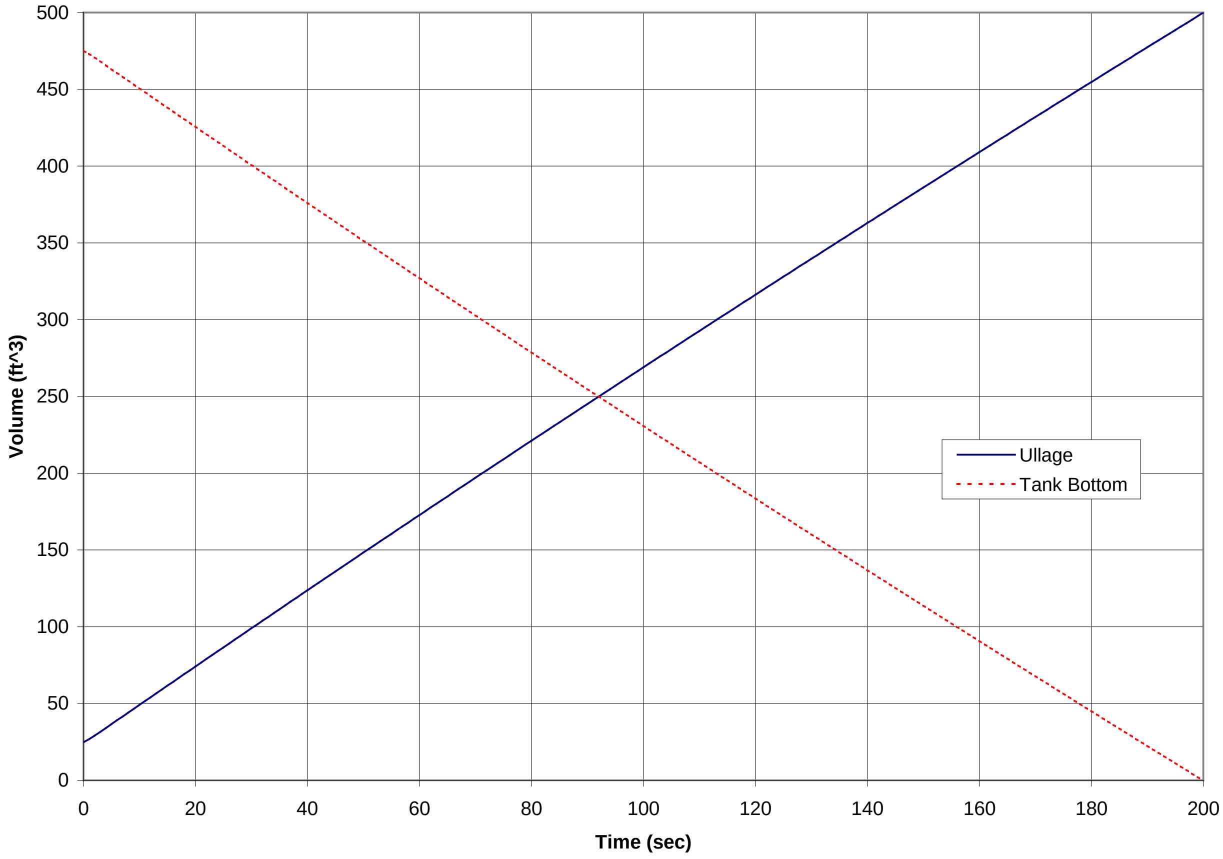
EXAMPLE 10 ULLAGE AND TANK WALL TEMPERATURE HISTORY



GFSSP 6.04 Tank Press. / Advanced Valves

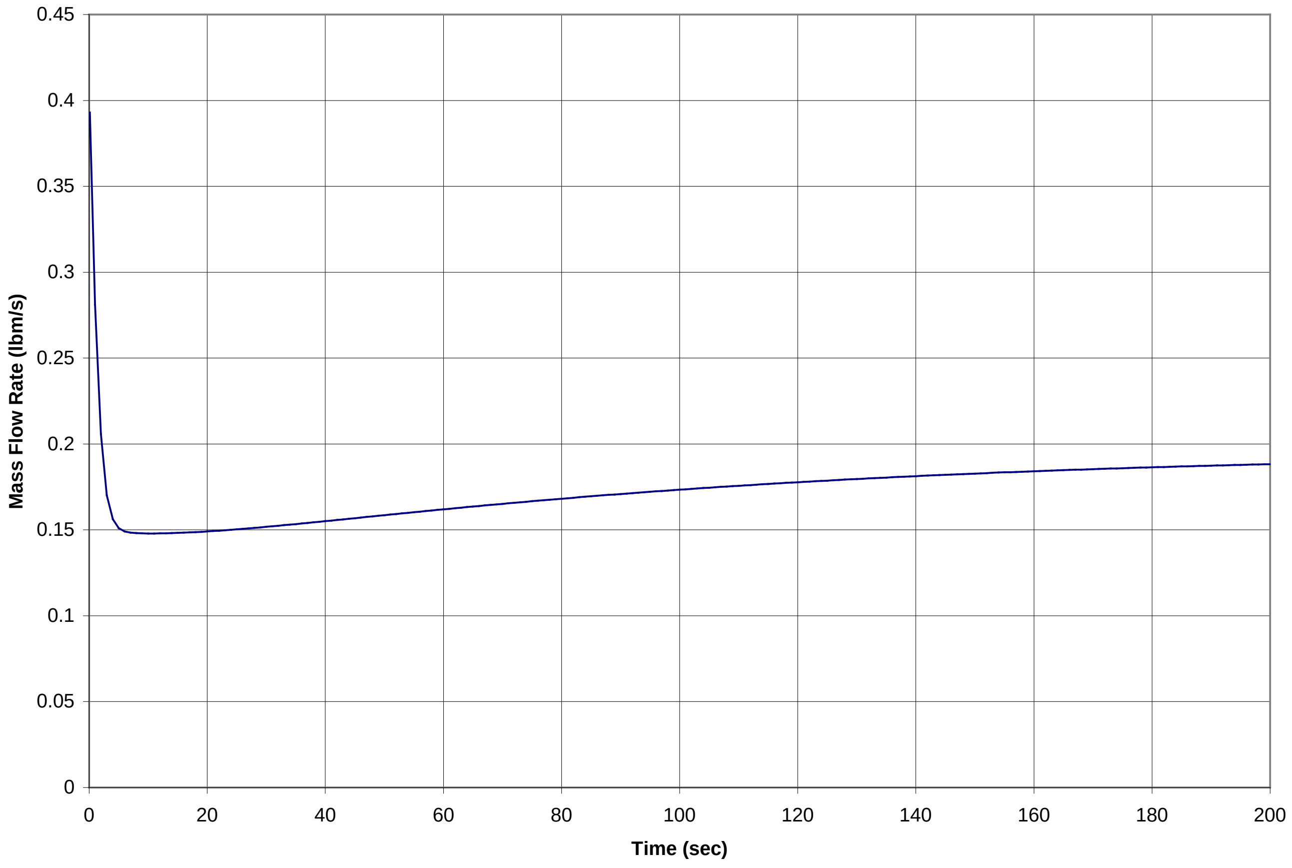
TANK PRESSURIZATION

EXAMPLE 10 PROPELLANT AND ULLAGE VOLUME HISTORY



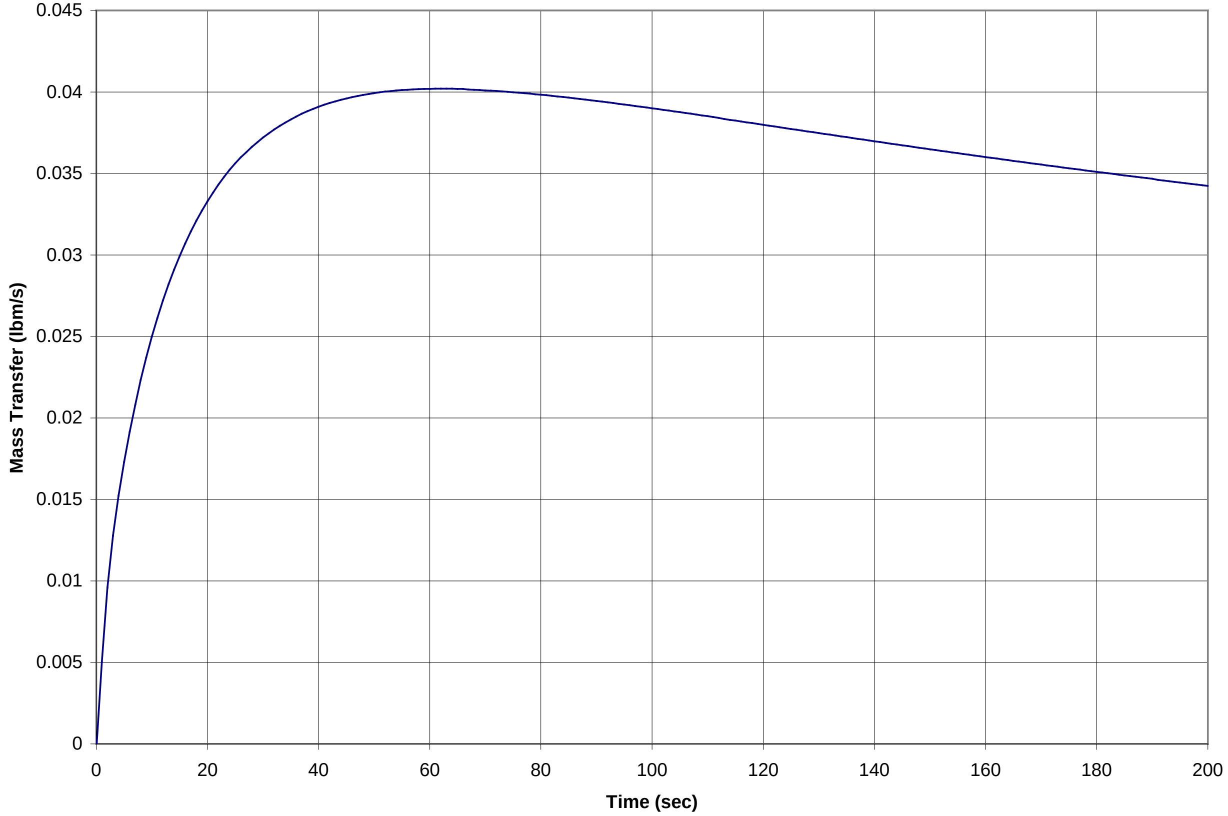
TANK PRESSURIZATION

EXAMPLE 10 HELIUM FLOW RATE HISTORY



TANK PRESSURIZATION

EXAMPLE 10 PROPELLANT TO ULLAGE MASS TRANSFER RATE HISTORY



TANK PRESSURIZATION

COLLAPSE FACTOR CORRELATION

The collapse factor is the ratio of actual pressurant consumption to an ideal pressurant consumption which assumes no heat or mass transfer. It is represented by the following formula.

$$\frac{w_p}{w_p^0} = \left\{ \left(\frac{T_0}{T_s} - 1 \right) \left[1 - \exp\left(-p_1 C^{p_2} \right) \right] \times \left[1 - \exp\left(-p_3 S^{p_4} \right) \right] + 1 \right\} \times \exp\left[-p_5 \left(\frac{1}{1+C} \right)^{p_6} \left(\frac{S}{1+S} \right)^{p_7} Q^{p_8} \right]$$

where

$$w_p^0 = \rho_G^0 \Delta V \qquad C = \frac{\left(\rho c_p^0 t \right)_w}{\left(\rho c_p \right)_G^0 D_{eq}} \frac{T_s}{T_0}$$

$$S = \frac{h_c \theta_T}{\left(\rho c_p \right)_G^0 D_{eq}} \frac{T_s}{T_0} \qquad Q = \frac{\dot{q} \theta_T}{\left(\rho c_p \right)_G^0 D_{eq} T_0}$$

TANK PRESSURIZATION
COLLAPSE FACTOR CORRELATION DEFINITIONS

C - ratio of wall to gas thermal capacitance

p_1-p_8 - fitted constants dependent on propellant

Q - ratio of ambient heat input to effective thermal capacitance of gas

S - modified Stanton number

T_0 - pressurant inlet temperature

T_s - propellant saturation temperature at initial tank pressure

w_p^0 - ideal pressurant mass consumption

TANK PRESSURIZATION

PRESSURIZATION MODEL VALIDATION

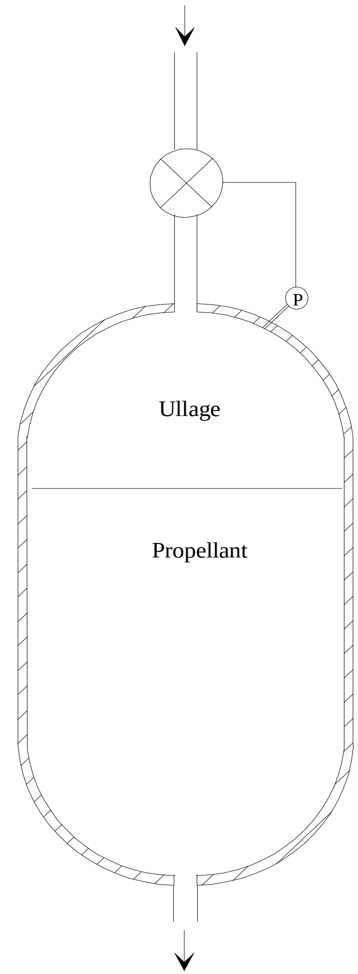
- GFSSP Collapse Factor Prediction: 1.44
- Epstein Correlation Collapse Factor Prediction: 1.58
- GFSSP Prediction Discrepancy: 8.86%

TANK PRESSURIZATION
SUMMARY

- GFSSP's transient capability further extended to model the pressurization of a propellant tank
- User activates this option and supplies additional tank information
- Code predicts the history of ullage and propellant conditions
- Example 10 illustrates the use of this option and also describes the verification of the numerical prediction

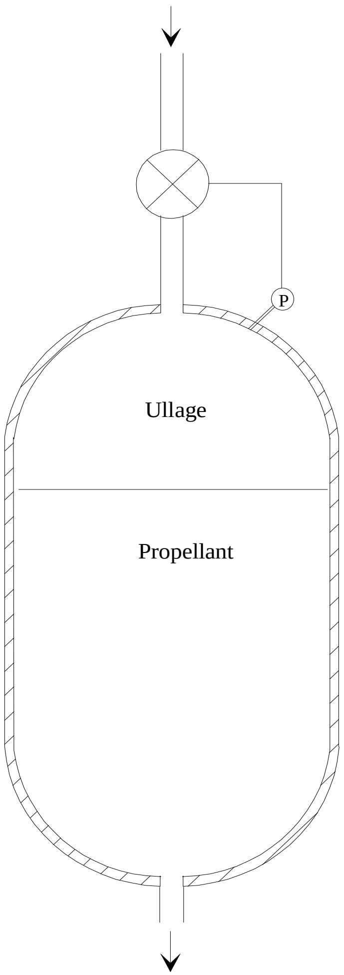
CONTROL VALVE

- Pressure monitored at arbitrary point downstream of valve
- Valve maintains pressure within user specified tolerance
  - Closes when pressure exceeds maximum value
  - Opens when pressure drops below minimum value
- Flow resistance factor calculated using same equations as Option 2



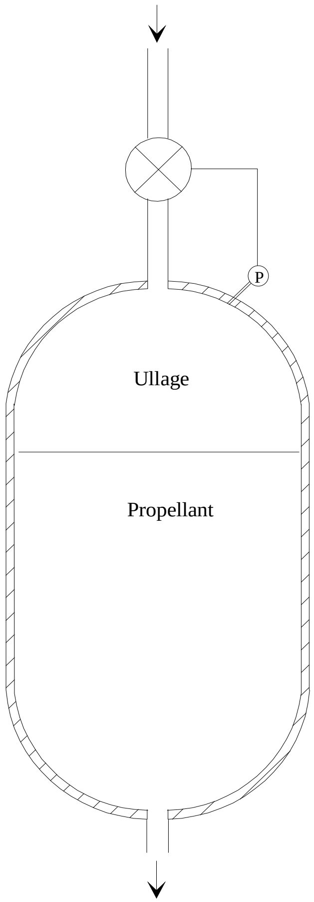
CONTROL VALVE
SUB-OPTIONS

- Instantaneous - Valve is either fully open or fully closed at any given time.
- Linear - Valve open/close transient is modeled as a linear operation.
- Non-linear - Valve open/close transient is modeled as some user specified non-linear operation.



CONTROL VALVE
BRANCH INPUTS -1

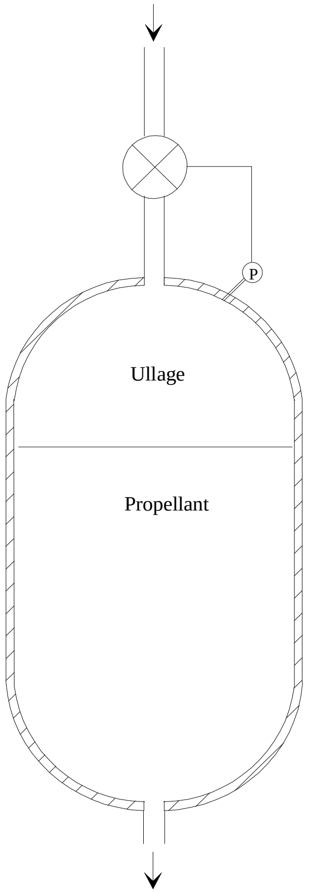
- Sub-option
- Flow Coefficient (C_L)
- Area (A)
- Control Node
- Valve Initial Position
- Pressure Tolerance File Name



CONTROL VALVE

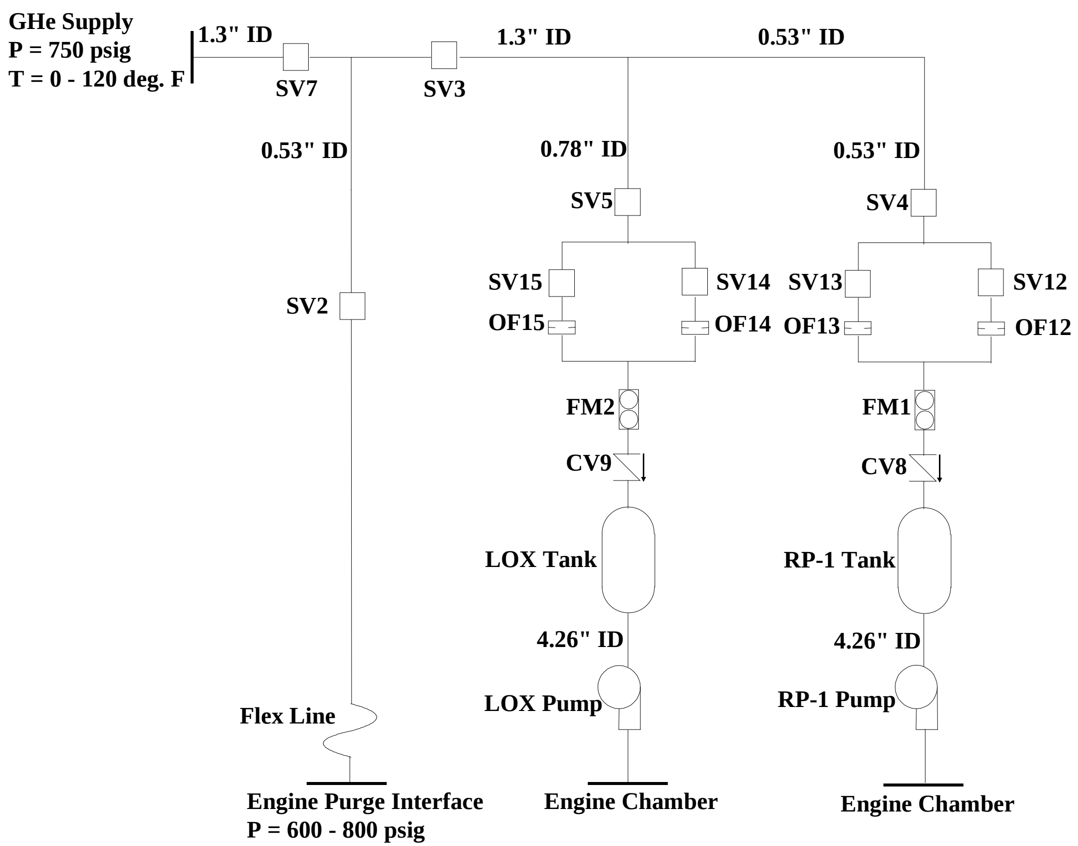
BRANCH INPUTS -2

- Linear Sub-option
  - Time to open/close
  - Number of steps to open/close
- Non-linear Sub-option
  - Open characteristics file name
  - Close characteristics file name



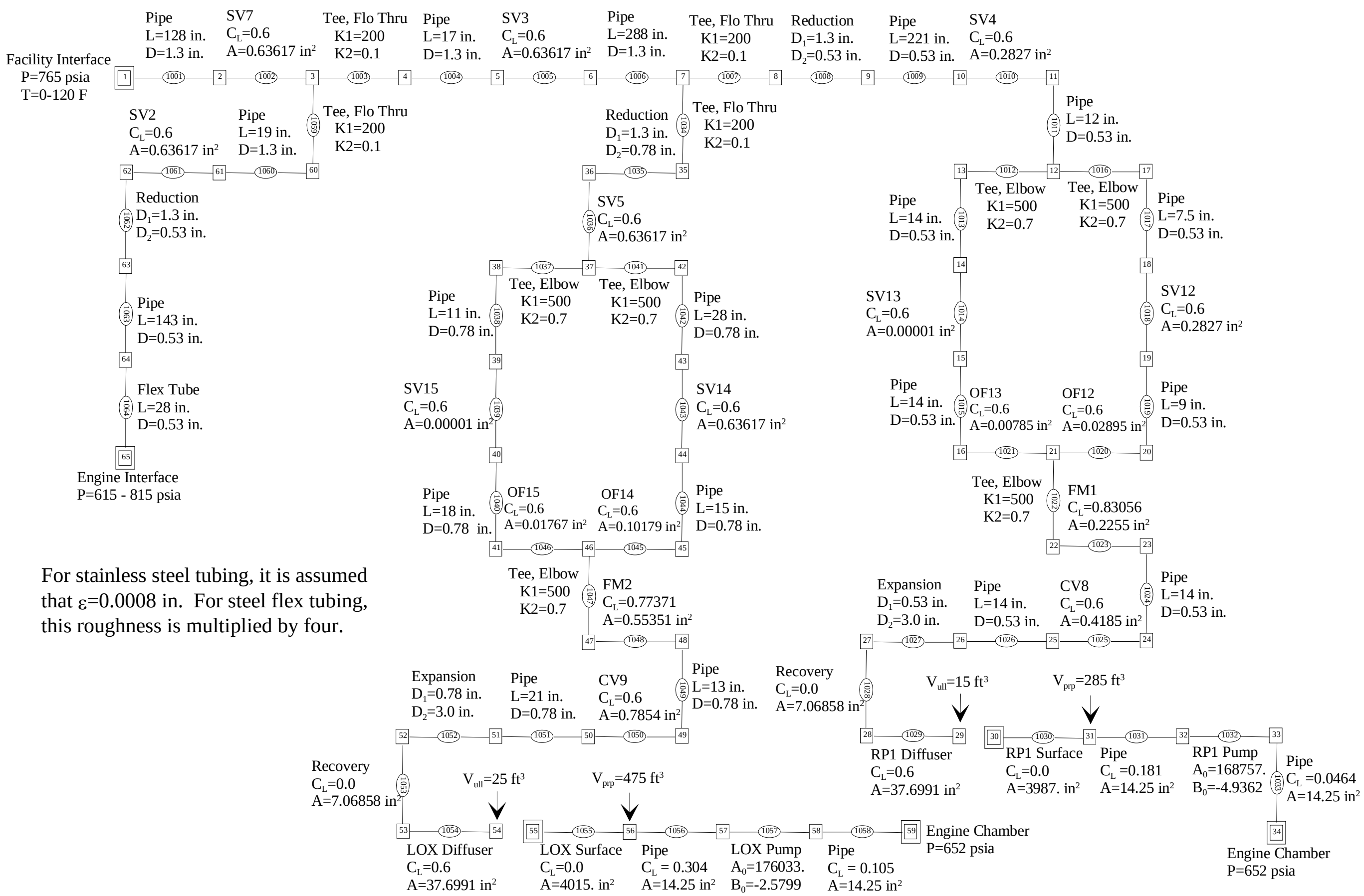
CONTROL VALVE

EXAMPLE 12 SCHEMATIC



CONTROL VALVE

EXAMPLE 12 GFSSP MODEL

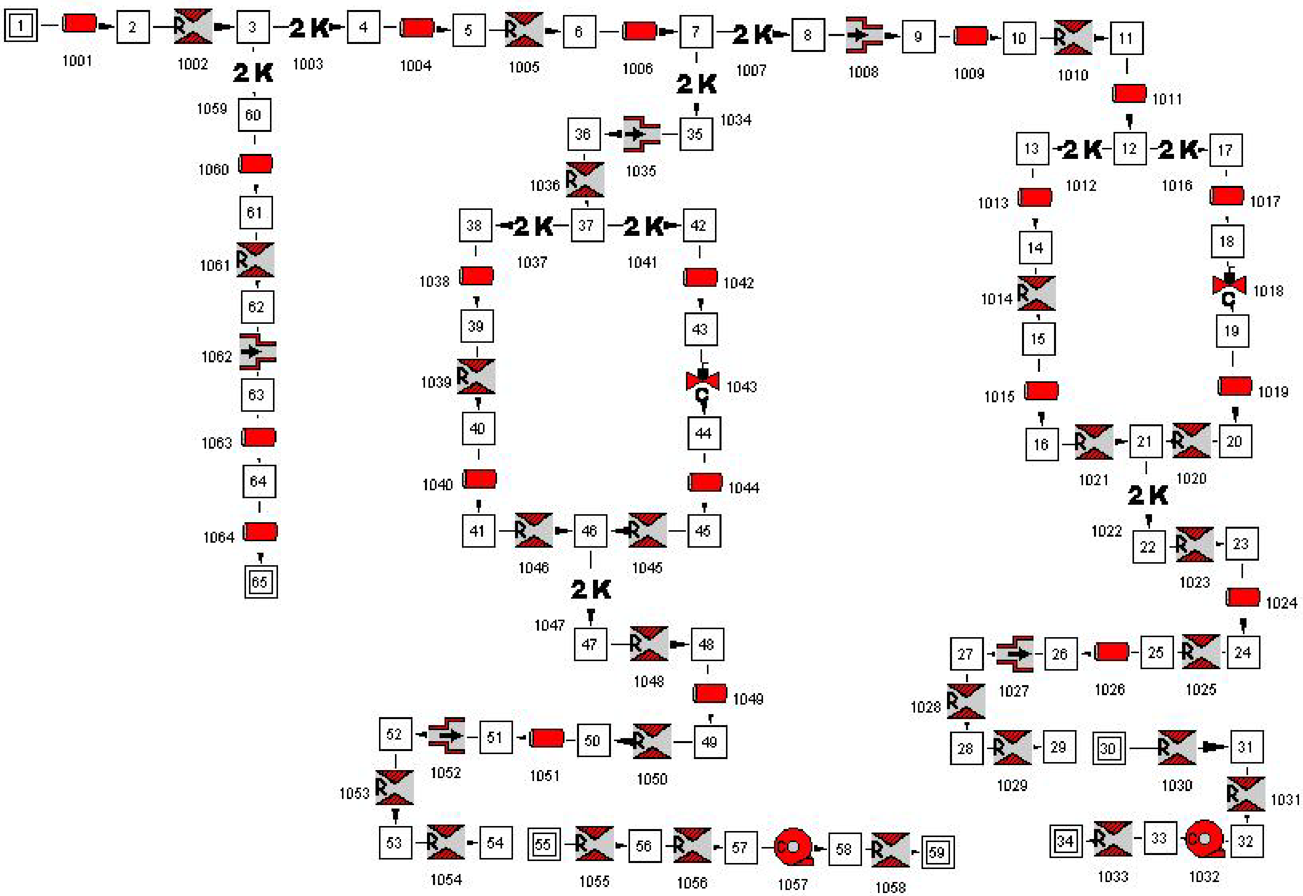


For stainless steel tubing, it is assumed that ε=0.0008 in. For steel flex tubing, this roughness is multiplied by four.

GFSSP 6.04 Tank Press. / Advanced Valves

CONTROL VALVE

EXAMPLE 12 VTASC CANVAS



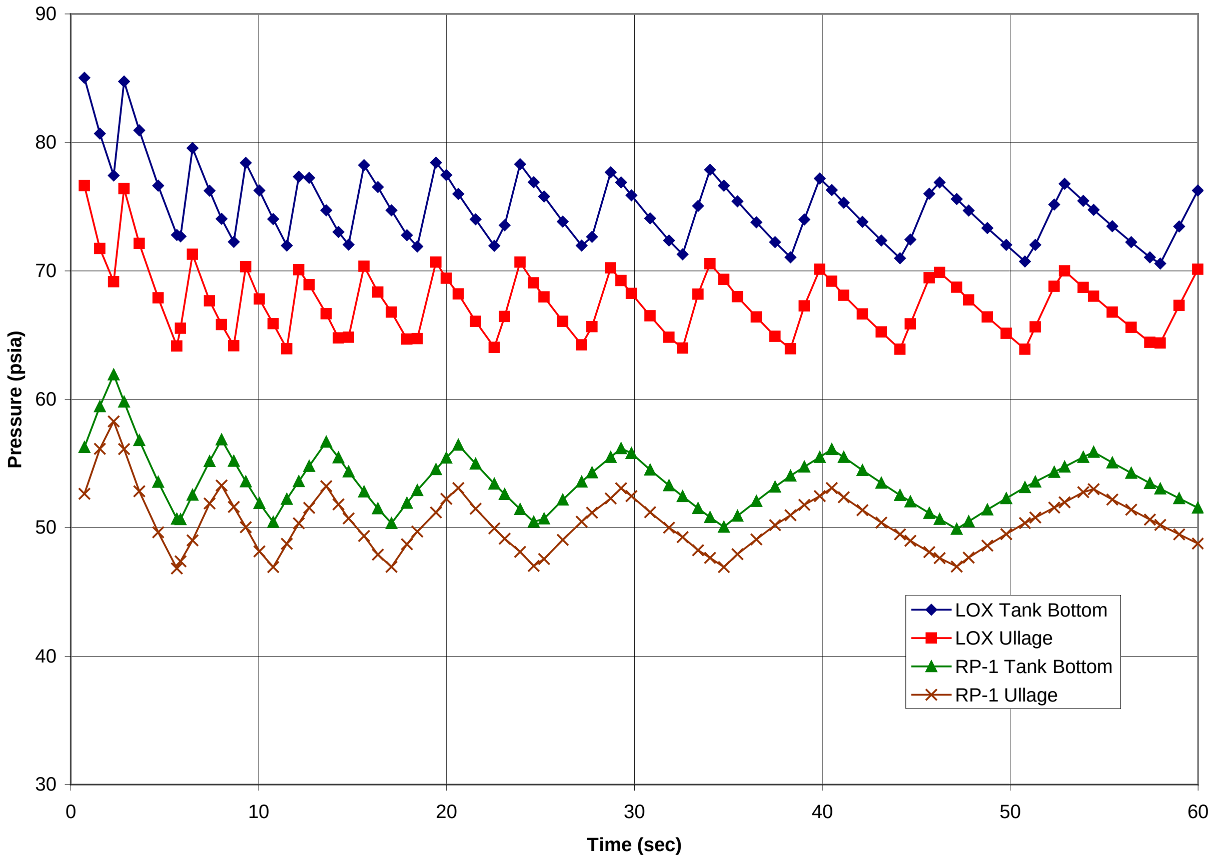
CONTROL VALVE

EXAMPLE 12 CONTROL VALVE INPUT

```
                             .
                             .
                             .
BR OPT-> 18-2, SUBOPT, FLOW COEF, AREA, CTRL NODE, INIT POS
  1018     2     0.60000     0.28270     29.0000   T
BR OPT-> 18-2(continued), CYCLE TIME, CYCLE STEPS, PR TOL FILE
           0.05000     5.00000
ex12rp1.dat
                             .
                             .
                             .
BR OPT-> 18-2, SUBOPT, FLOW COEF, AREA, CTRL NODE, INIT POS
  1043     2     0.60000     0.63617     54.0000   T
BR OPT-> 18-2(continued), CYCLE TIME, CYCLE STEPS, PR TOL FILE
           0.05000     5.00000
ex12lox.dat
                             .
                             .
                             .
```

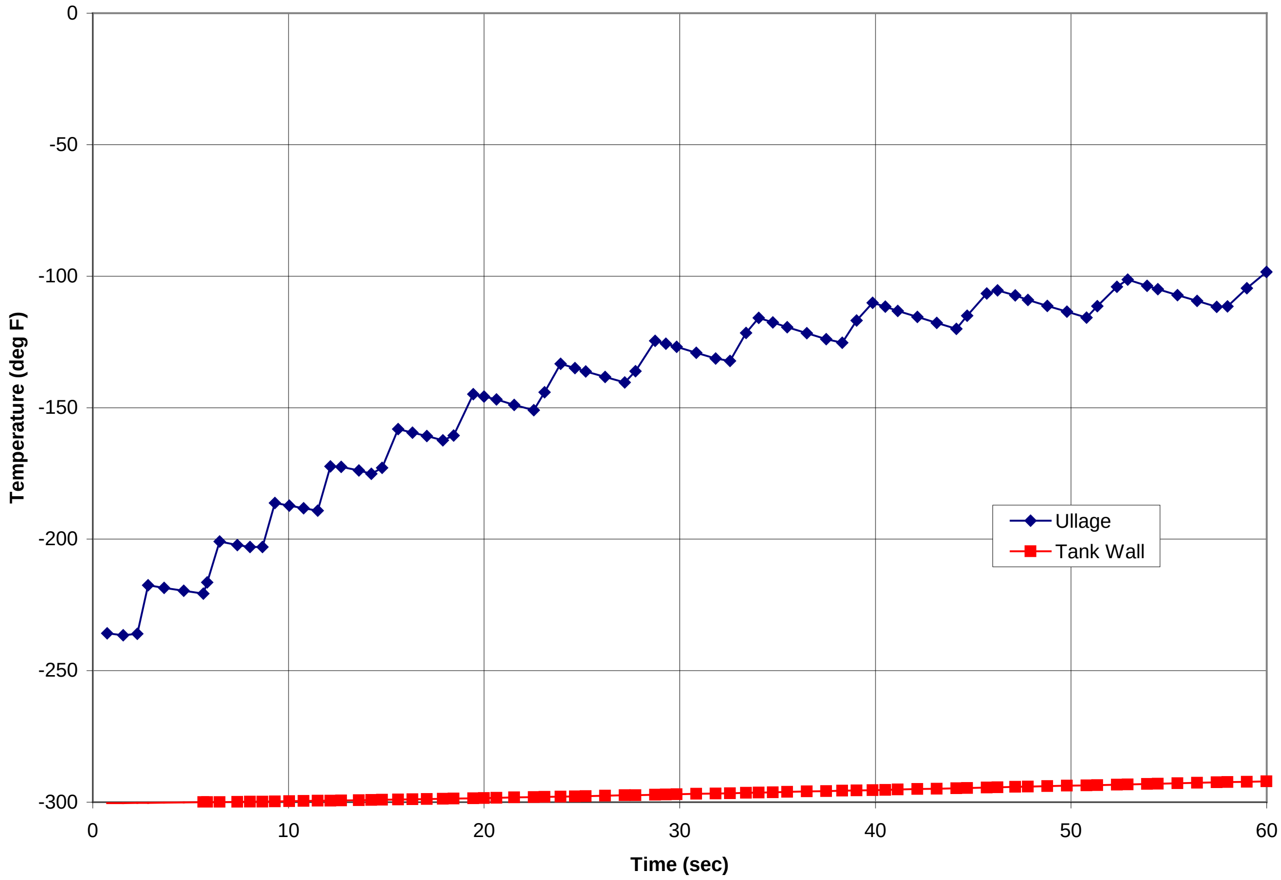
CONTROL VALVE

EXAMPLE 12 PROPELLANT TANK PRESSURE HISTORY



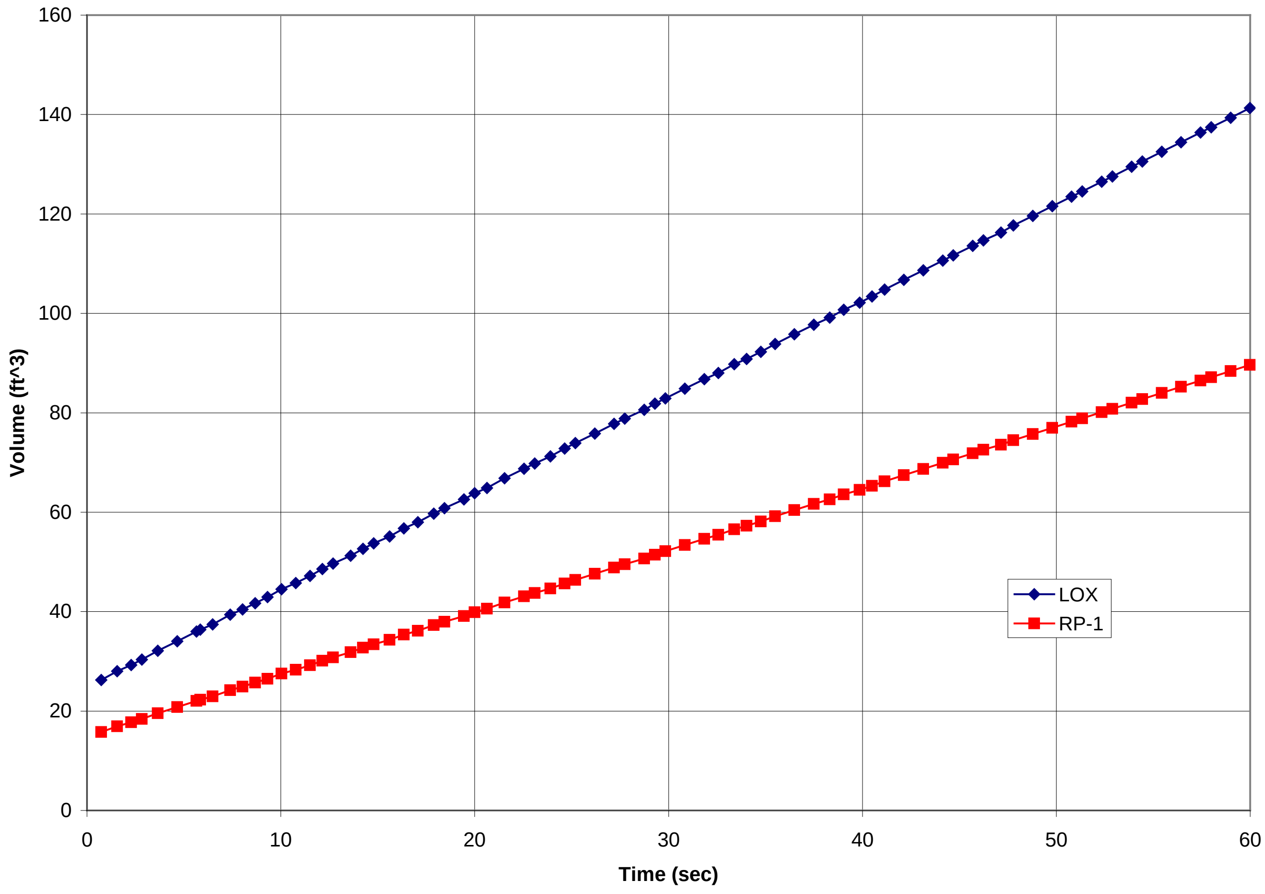
CONTROL VALVE

EXAMPLE 12 LOX TEMPERATURE HISTORY



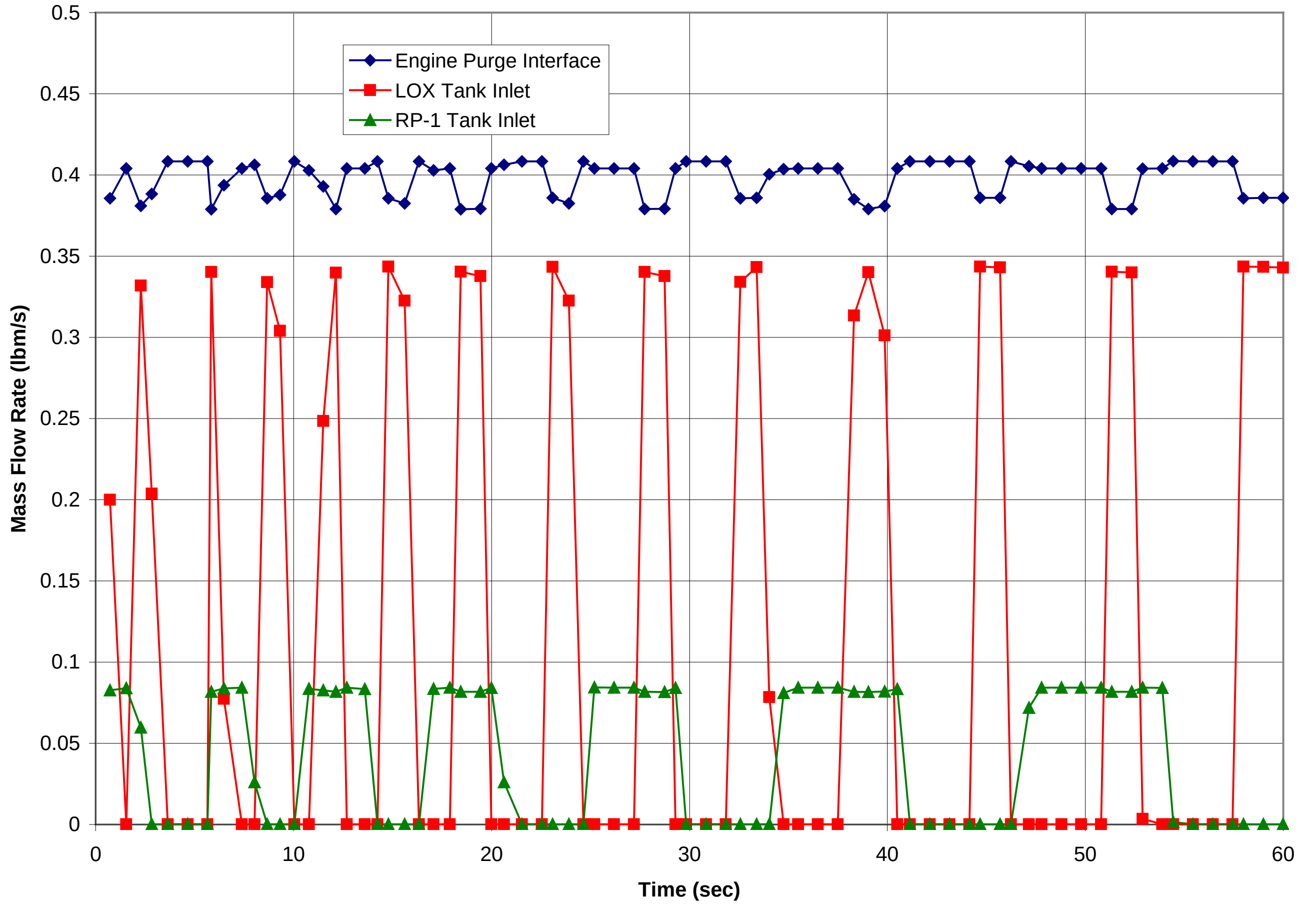
CONTROL VALVE

EXAMPLE 12 ULLAGE VOLUME HISTORY



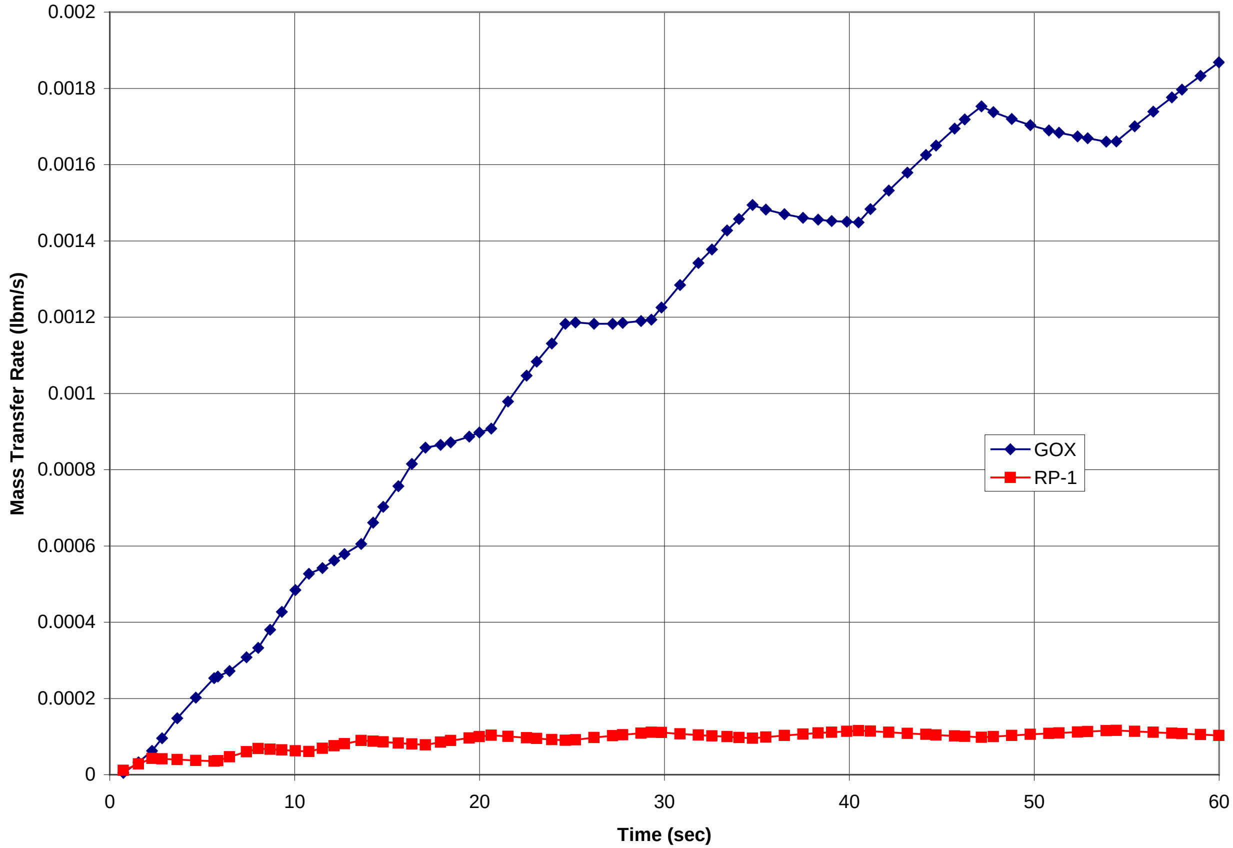
CONTROL VALVE

EXAMPLE 12 HELIUM FLOW RATE HISTORY



CONTROL VALVE

EXAMPLE 12 PROPELLANT TO ULLAGE MASS TRANSFER HISTORY



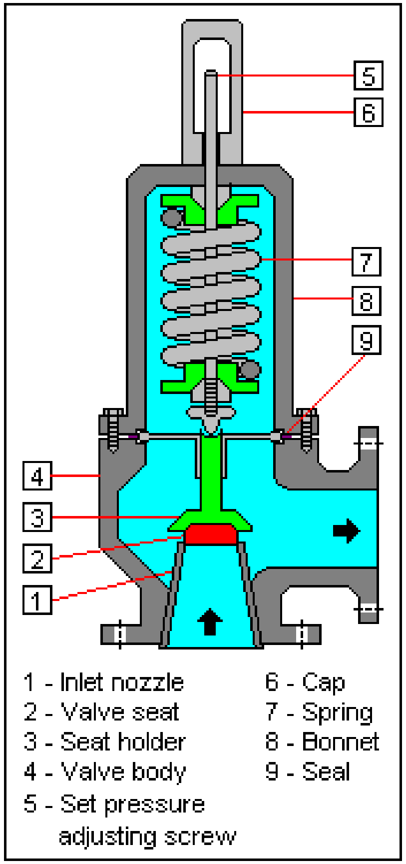
GFSSP 6.04 Tank Press. / Advanced Valves

CONTROL VALVE
SUMMARY

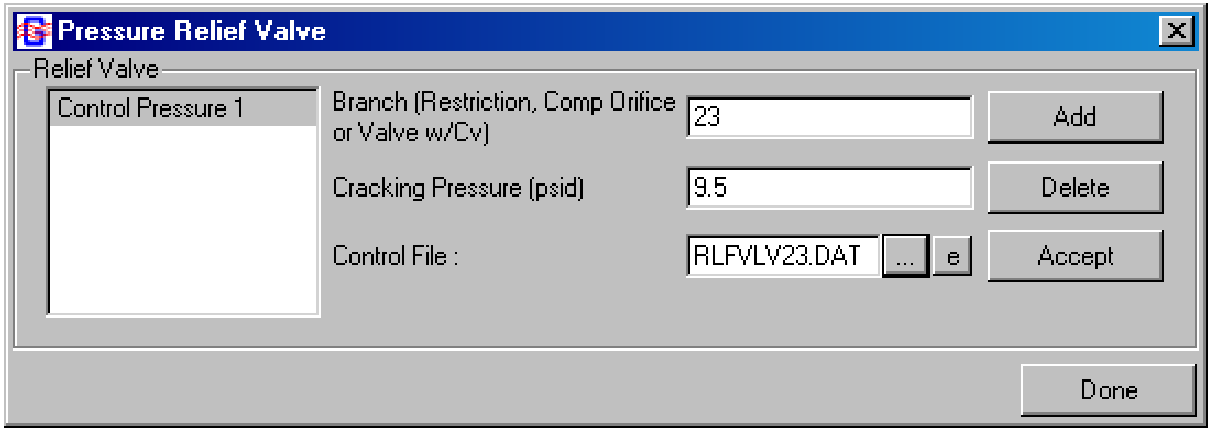
- New Branch Resistance Option added to GFSSP to model a Control Valve
- Valid only for transient models
- User provides the flow and operational characteristics of the valve
- Example 12 demonstrates the operation of the control valve option in GFSSP

RELIEF VALVE

- Distinct from Control Valve
- Monitors pressure differential across valve branch
- Valve opens when pressure exceeds cracking pressure
- Relief valve is an Advanced Option that may be linked to:
  - Restriction
  - Compressible orifice
  - Valve with Cv



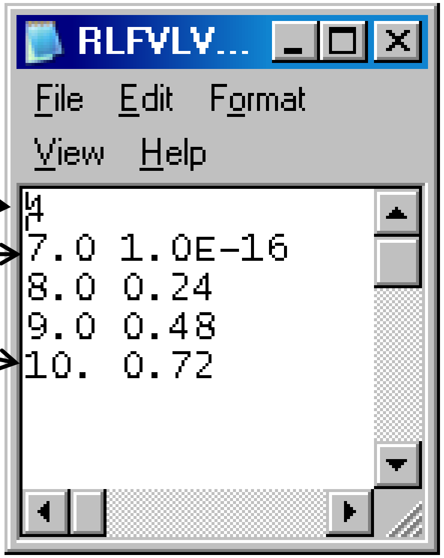
RELIEF VALVE INPUTS



- Branch ID number
- Valve cracking pressure differential (psid)
- Control file determines valve branch flow resistance as function of pressure differential

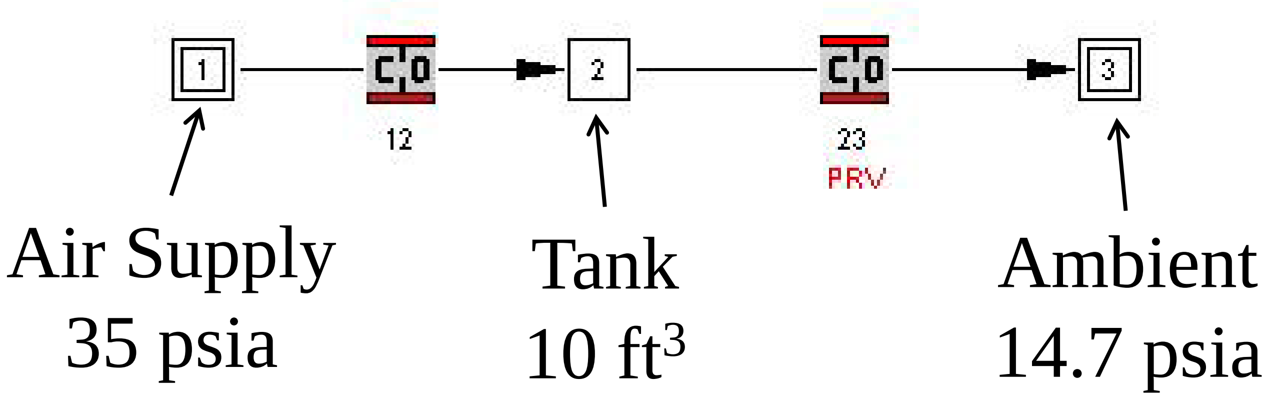
RELIEF VALVE CONTROL FILE

- Number of lines
- Reseating pressure (psid)
- Fully-open pressure (psid)
- Area (in^2) or Cv



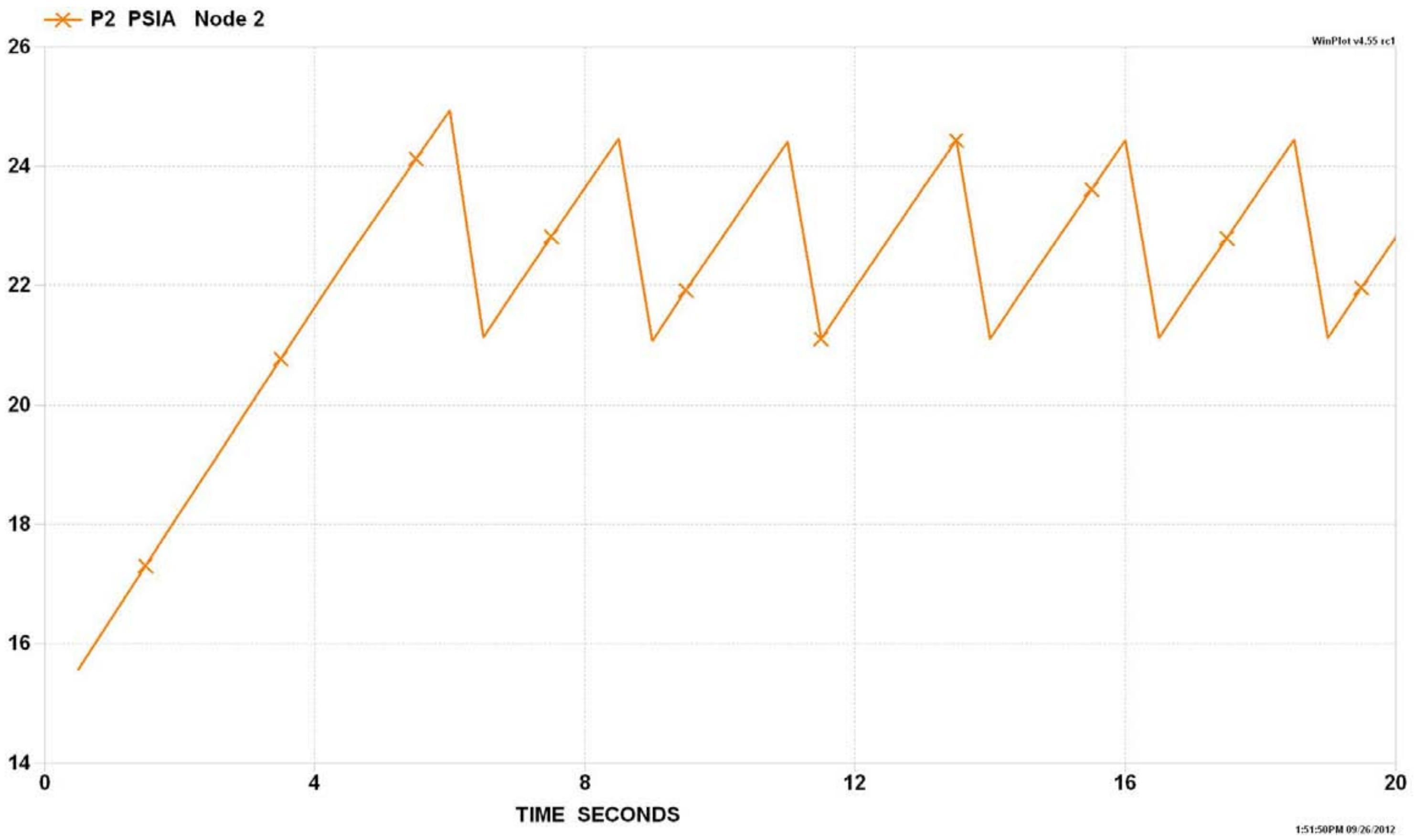
CONTROL VALVE

EXAMPLE 24 VTASC CANVAS



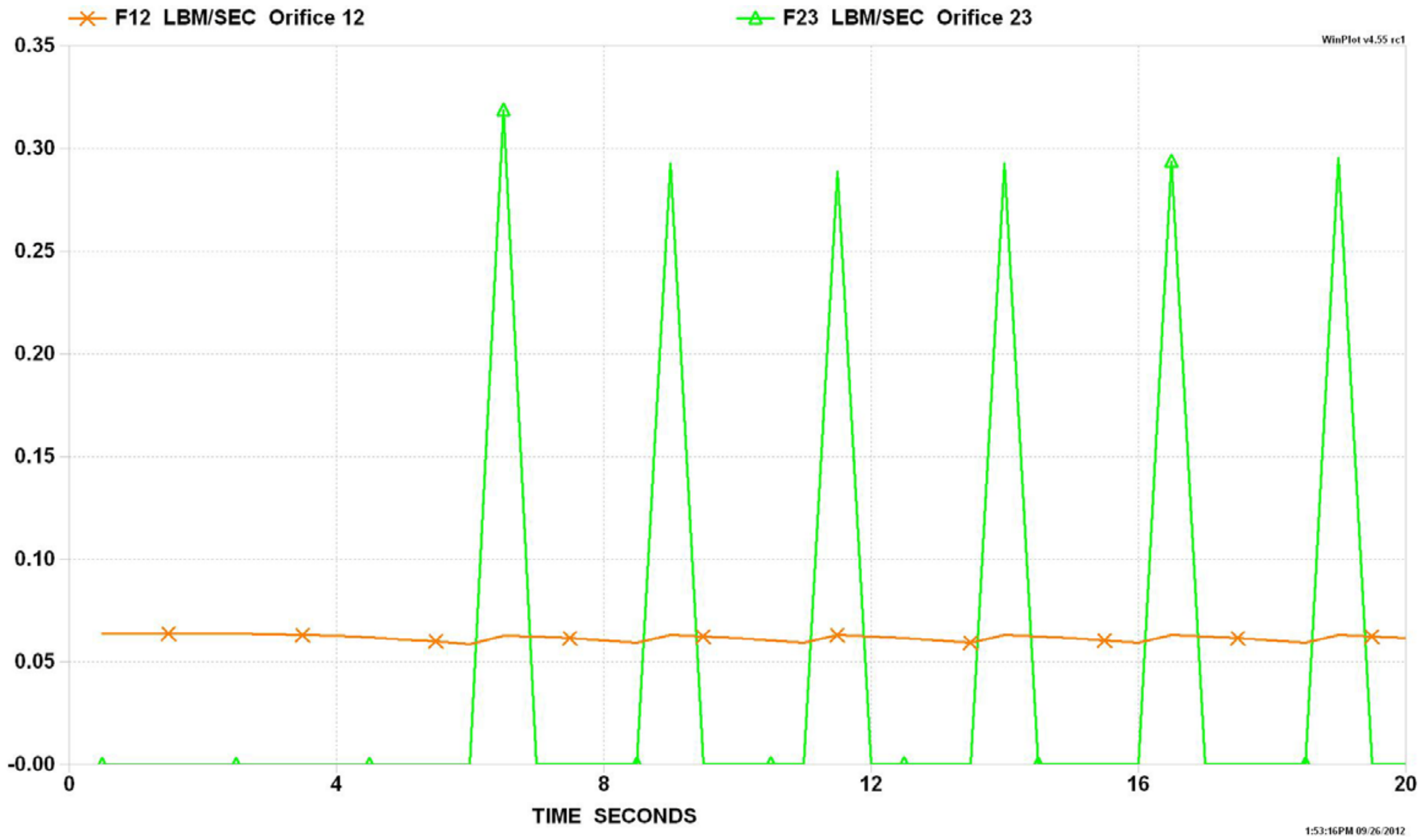
RELIEF VALVE

EXAMPLE 24 Tank Pressure History



RELIEF VALVE

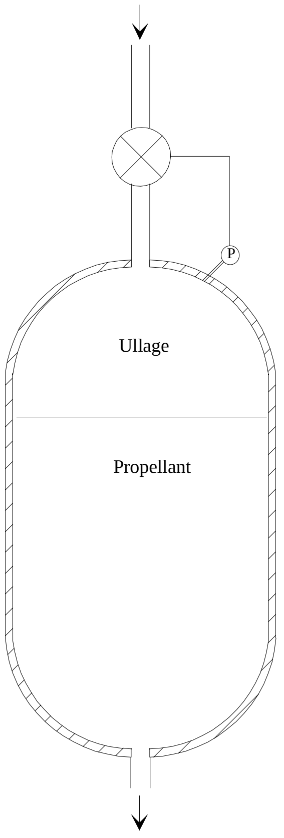
EXAMPLE 24 Flow Rate History

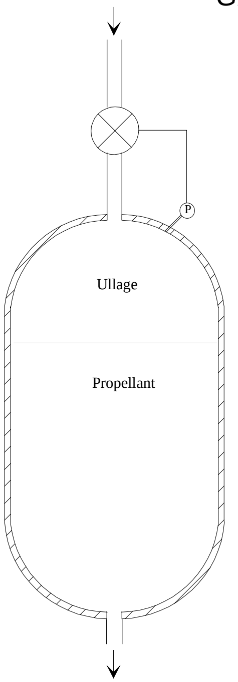


RELIEF VALVE
SUMMARY

- New Advanced Option added to GFSSP to model a Relief Valve
- Valid only for transient models
- User provides the cracking pressure and flow resistance characteristics of the valve
- Example 24 demonstrates the operation of the relief valve option in GFSSP

Tutorial – 3
Valve-Controlled Pressurization of a Propellant Tank





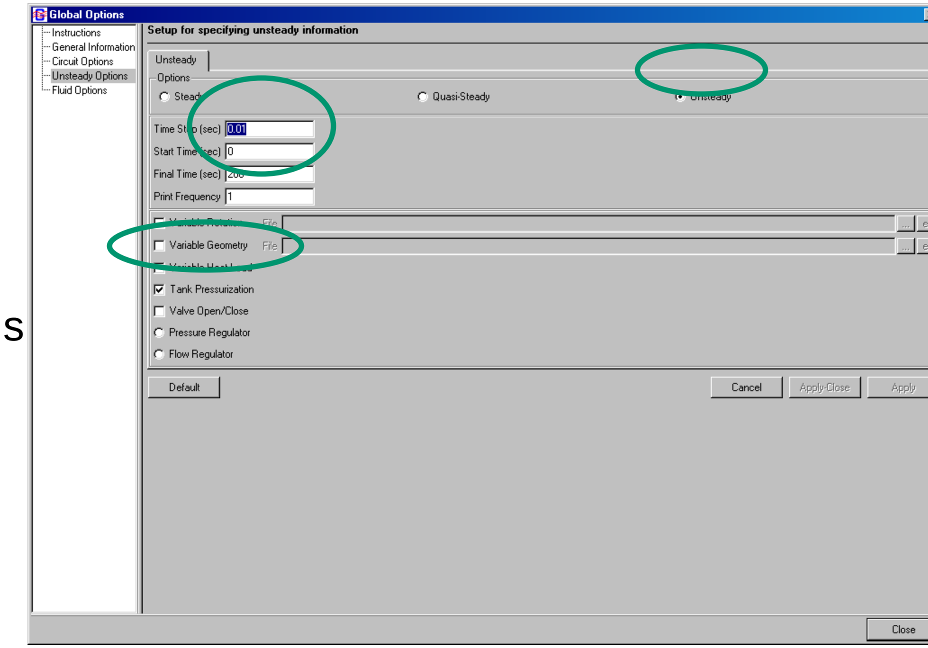
Problem Elements:

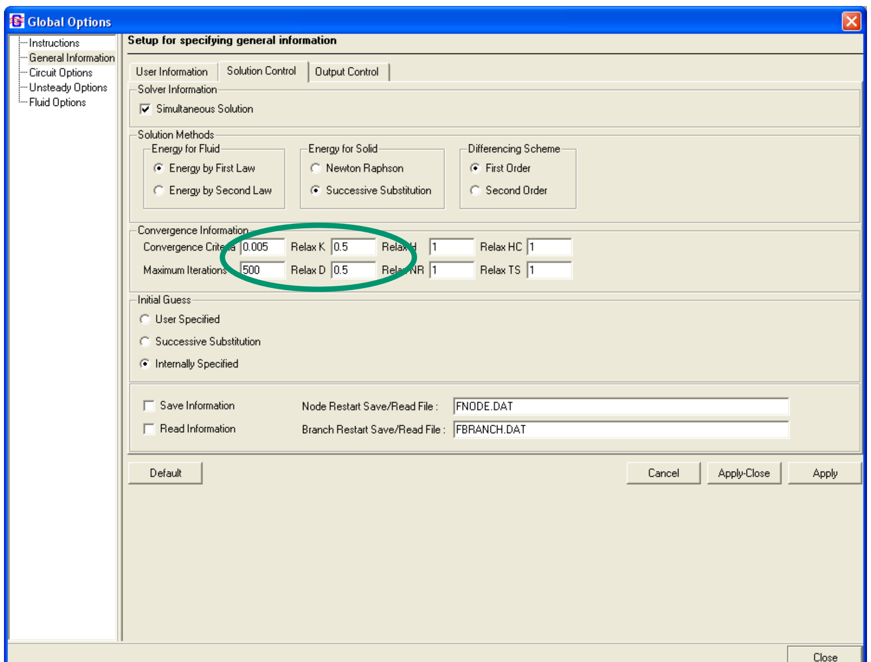
- Control tank pressure within a specified tolerance
  - Use control valve branch option
- Use tank pressurization advanced option
- Use 2 fluids (oxygen and helium)

Set Up Options

Marshall Space Flight Center
GFSSP Training Course

- User Information:
  - Input File: Tut3.dat
  - Output File: Tut3.out
- Unsteady Options
  - Time step = 0.01 s; Final time = 200 s
  - Check Tank Pressurization option

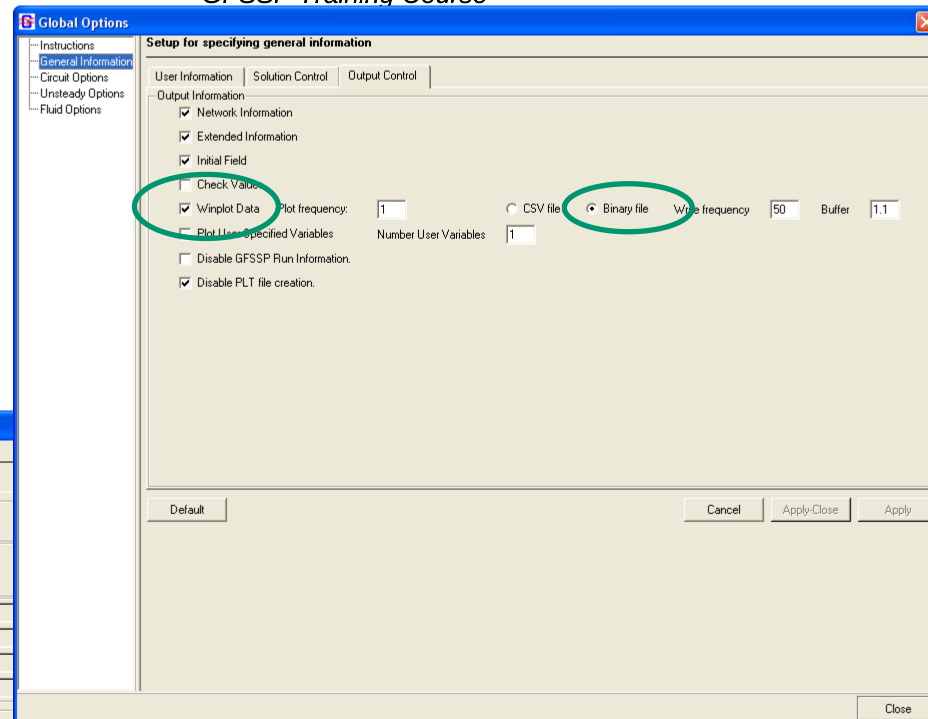


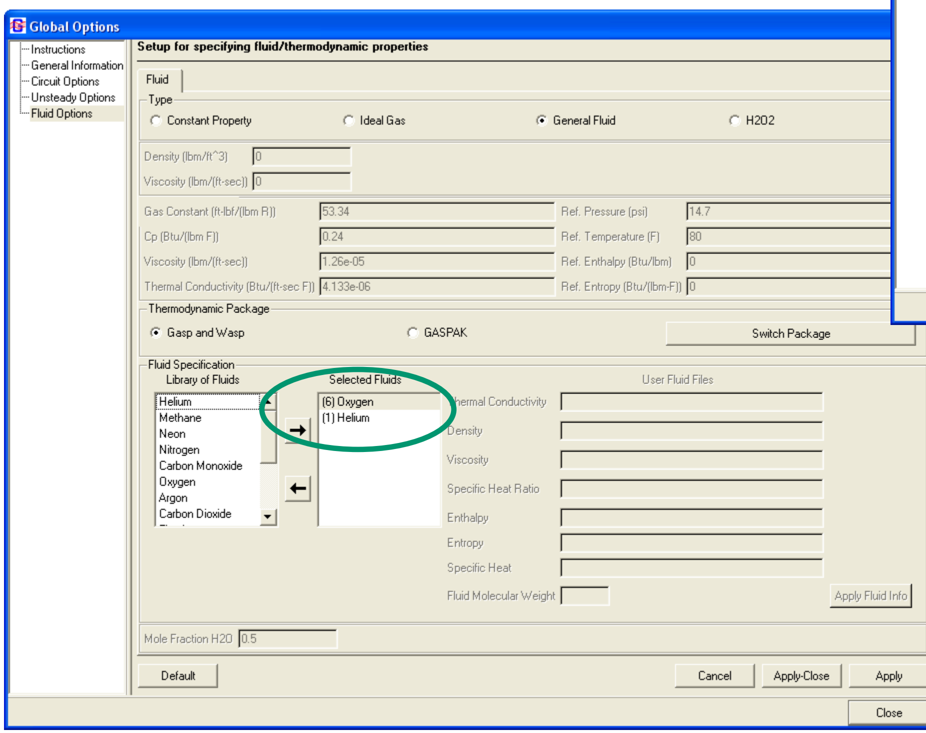


- Solution Control
  - Convergence Criteria = 0.005
  - RELAXK = 0.5

Set Up Options

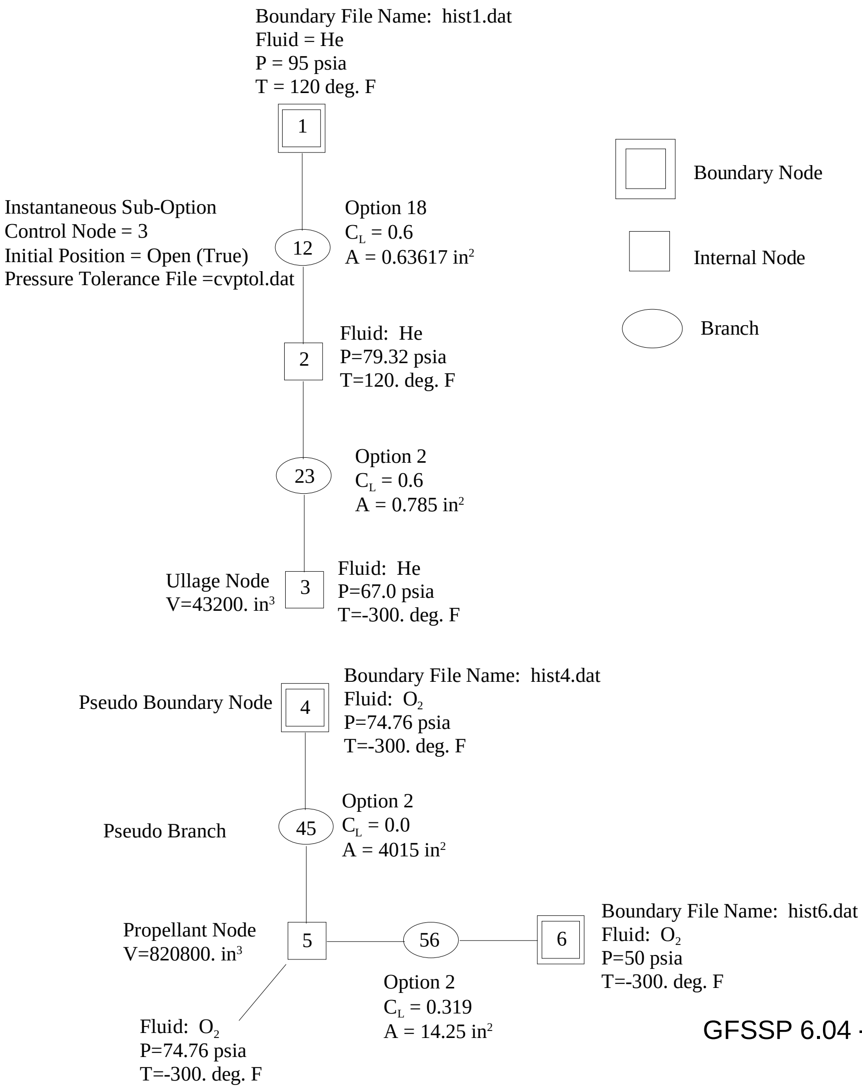
- Output Control
  - Select Winplot binary output

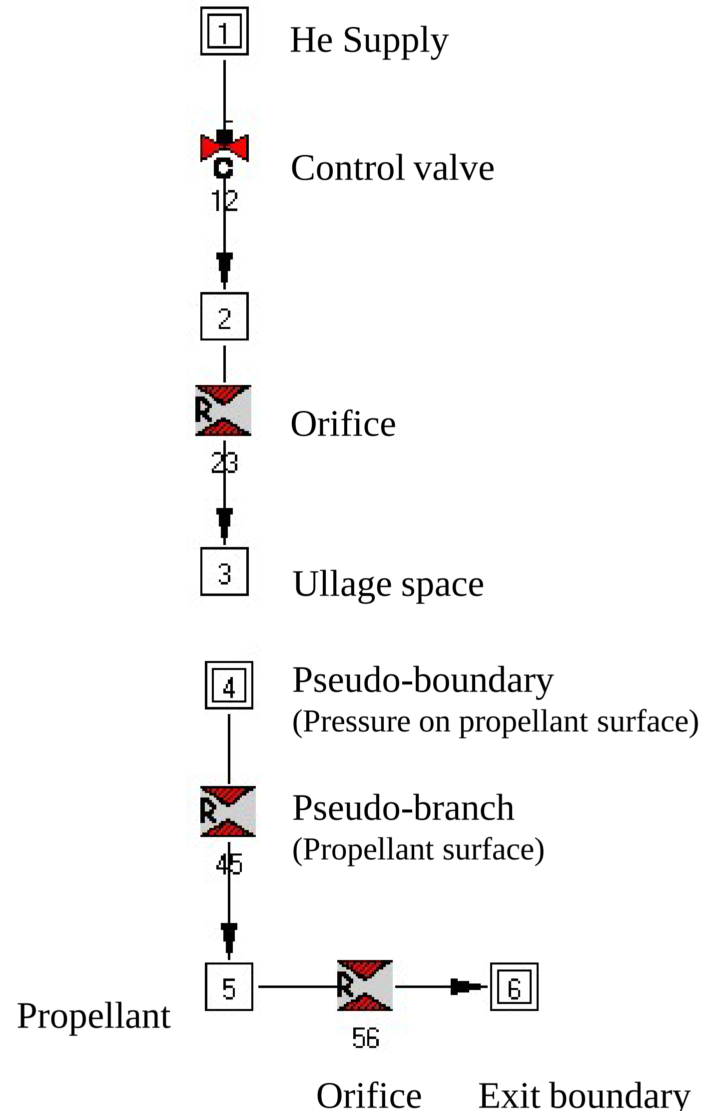




- Fluid Options
  - Select Oxygen first
  - Then select Helium

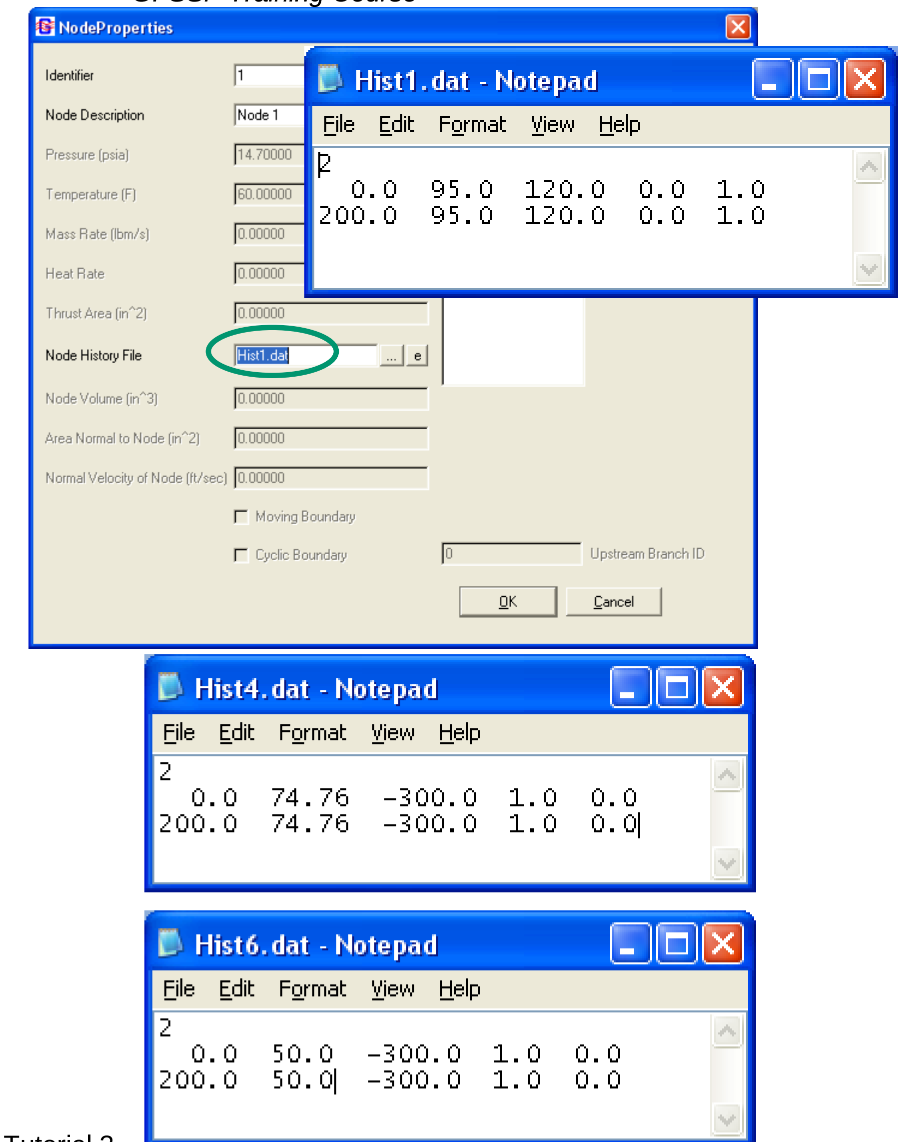
Build Model on Canvas





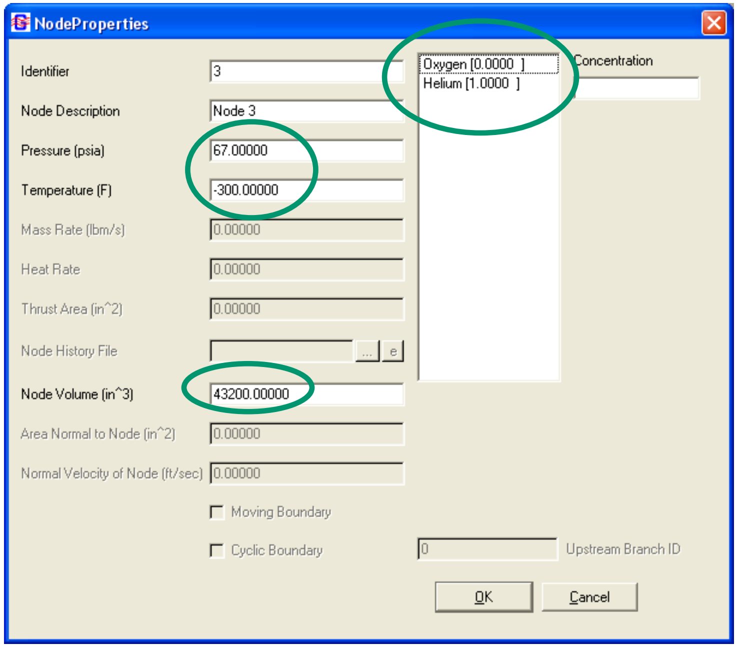
Set Up Boundary Nodes

- Node 1 is the helium supply
  - P = 95 psia, T = 120 °F
  - LO2 mass fraction = 0.0
  - He mass fraction = 1.0
- Node 4 is a pseudoboundary node
  - It separates the He from the LO2
  - History file is required, but pressure will be overwritten by Node 3 ullage pressure plus propellant head
  - P = 74.76 psia, T = -300 °F
  - LO2 mass fraction = 1.0
  - He mass fraction = 0.0
- Node 6 is the LO2 exit boundary
  - P = 50 psia, T = -300 °F
  - LO2 mass fraction = 1.0
  - He mass fraction = 0.0



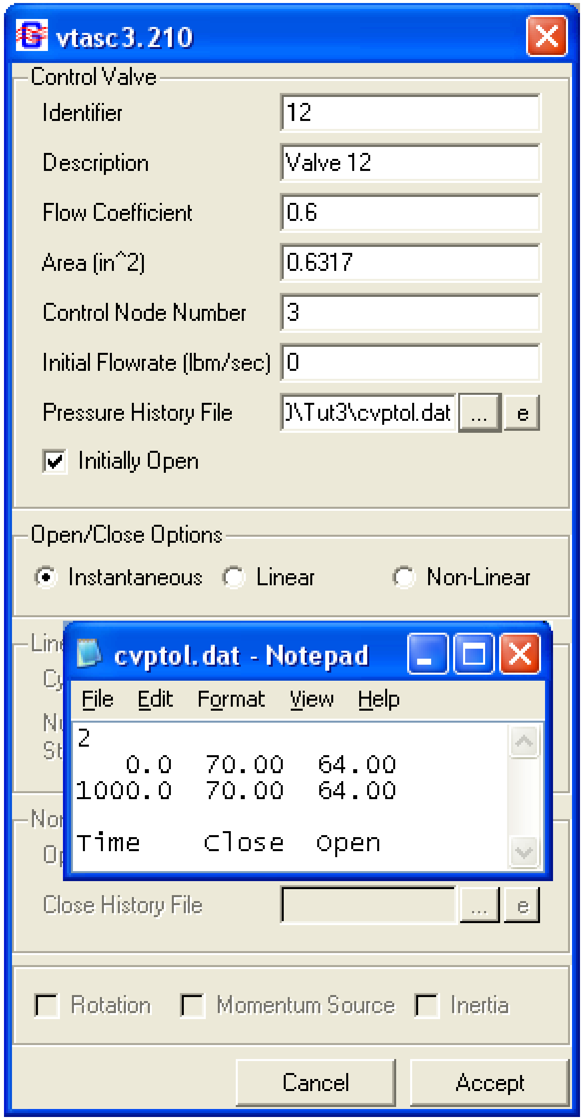
Set Up Interior Nodes

- Node 3 represents the ullage space
    - Initial P = 67 psia, T = -300.0 °F
    - Initial Volume = 43,200 in^3
    - He fraction = 1.0, LOx fraction = 0.0
- Node 5 represents the propellant space
    - Initial P = 74.76 psia, T = -300.0 °F
    - Initial Volume = 820,800 in^3
    - LOx fraction = 1.0, He fraction = 0.0
- Node 2 represents the small space between the control valve and the ullage inlet orifice
    - Initial P = 79.32 psia, T = 120 °F
    - Volume is negligible
    - He fraction = 1.0, LOx fraction = 0.0



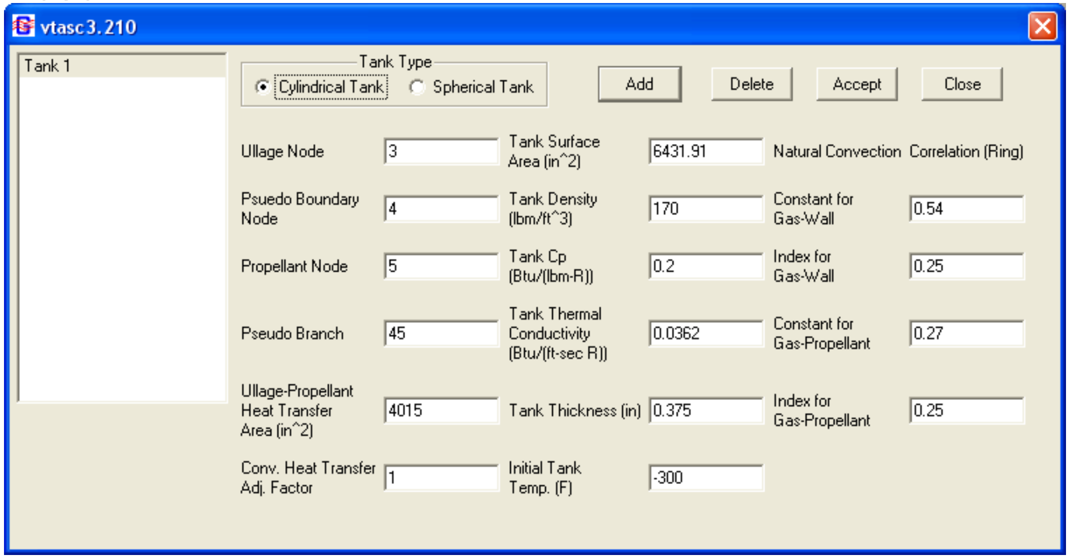
Set Up Branches

- Branch 12 is an instantaneous control valve
  - A = 0.6317 in^2, C_L = 0.6
  - It is controlled by pressure in Node 3
    - 70 psia – close
    - 64 psia – open
  - Valve is initially open
  - Requires a history file
- Branch 23 is the inlet orifice to the ullage
  - A = 0.785 in^2, C_L = 0.6
- Branch 45 represents the surface of the propellant
  - A = 4015 in^2, C_L = 0.0
- Branch 56 represents the orifice to the exit boundary
  - A = 14.25 in^2, C_L = 0.319



Tank Pressurization Option

- Open Pressurization Dialog from Advanced menu
- Click ADD



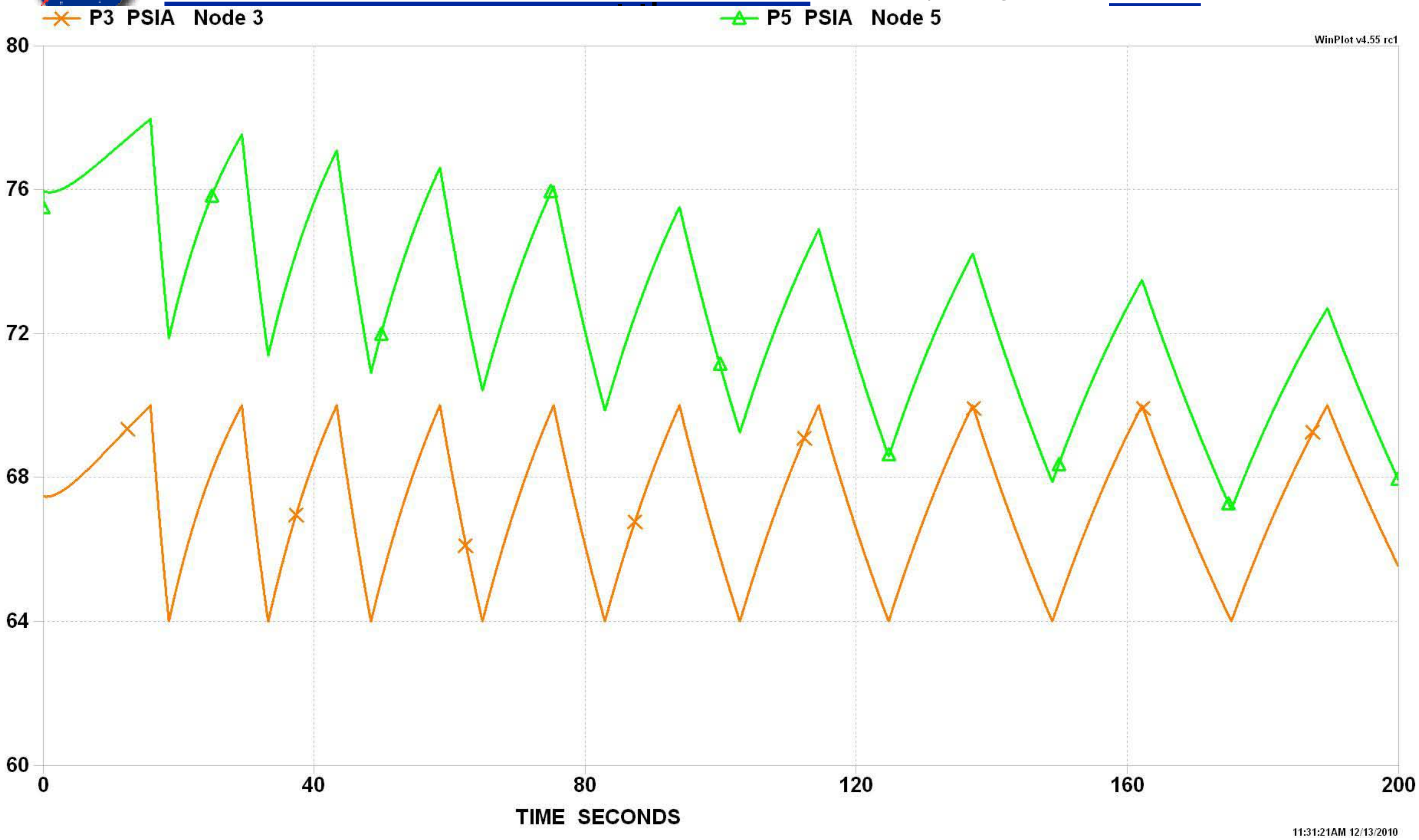
- Cylindrical aluminum tank
- Tank Surface Area: 6431.91 in^2
- Density: 170. lbm/ft^3
- Specific Heat: 0.2 Btu/lbm-R
- Thermal Conductivity: 0.0362 Btu/ft-s-R
- Ullage/Propellant Heat Transfer Area: 4015. in^2
- Wall Thickness: 0.375 in.
- Conv. Heat Transfer Adj. Factor: 1.0
- T_{tank}: -300 F

- Use default convection correlation coefficients
- Click ACCEPT, then CLOSE

Study of the Results

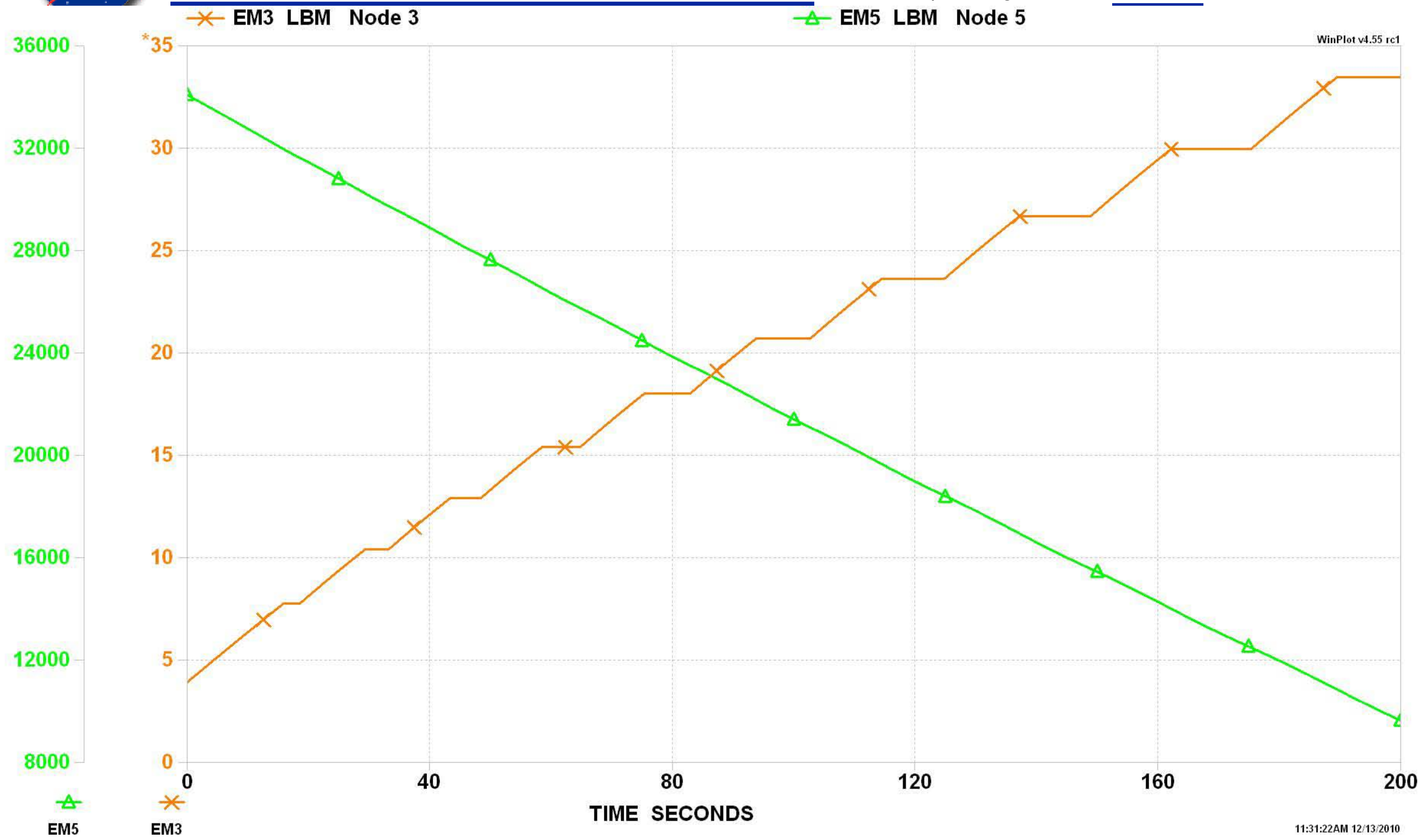
- Study *tut3.out* and *plot files* to note the following facts:
  - Ullage pressure is maintained between 64 and 70 psia by the control valve
  - Difference between ullage pressure and tank bottom pressure due to gravitational head
  - Tank bottom pressure decreases as propellant is expelled from the tank

Tank Pressure

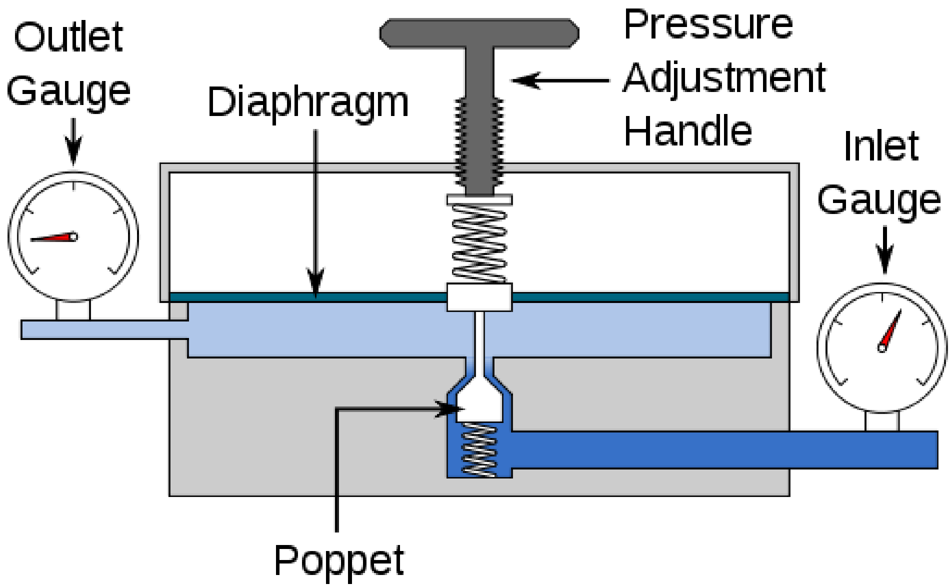


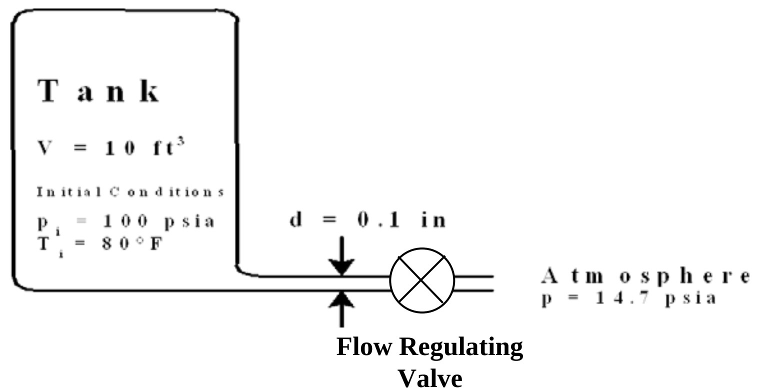
Tank Mass History

Marshall Space Flight Center



PRESSURE & FLOW REGULATOR



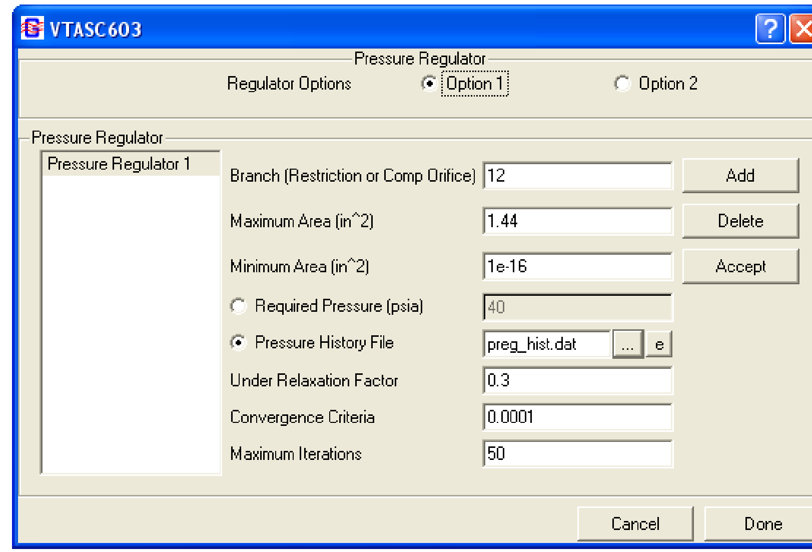


Modeling Pressure Regulator

- GFSSP has two built-in options (algorithms) to model a pressure regulator
  1. Iterative Algorithm
     - Applicable for single regulator and longer computation time
     - Serves as an example of how to adjust GFSSP solution to satisfy a given boundary condition
  2. Marching Algorithm (Schallhorn-Haas)
     - Capable of handling multiple regulators
     - Numerically stable and computationally efficient

Pressure Regulator Option – 1
Iterative Algorithm

- Purpose: To control pressure at a given node by adjusting the flow area of the upstream branch
- Implementation
  - Step 1
    - Edit
    - Options
    - Unsteady Options
      - Pressure Regulator
  - Step 2
    - Advanced
      - Pressure Regulator
- Application
  - Example 16 – Simulation of a Pressure Regulator downstream of a pressurized tank



History File

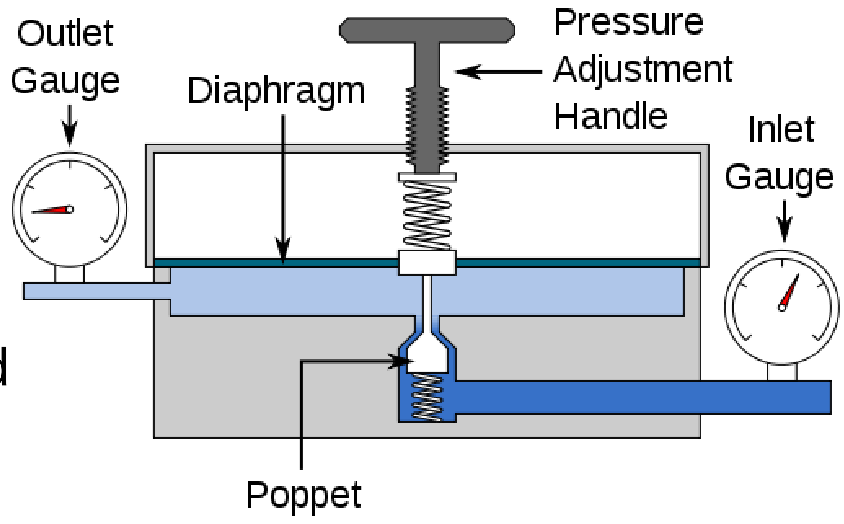
Number of lines in the file → 4

Time in Seconds | Pressure in psia

0	35.00
10	35.00
10.01	40.00
1000	40.00

Pressure Regulator – Option 2
Marching Algorithm

- In Marching Algorithm, area is guessed and adjusted only once in each time step
- Adjustment of area is calculated based on difference between calculated and desired pressure
- Area adjustment can be done by backward differencing algorithm (Schallhorn-Majumdar) or forward looking algorithm (Schallhorn-Hass)
- Schallhorn-Hass Algorithm has been implemented in GFSSP Version 603 as Option 2



Backward Differencing Algorithm

- In Backward Differencing Algorithm, area change is calculated from the previous time step:

$$A_{new} = A_{\tau} - \frac{\partial A}{\partial p}\left(p_{\tau} - p_{reg}\right)$$

η_{relax} = relaxation factor/reaction lag,

$$\frac{\partial A}{\partial p} \approx \left| \frac{A_{\tau} - A_{\tau-\Delta\tau}}{p_{\tau} - p_{\tau-\Delta\tau}} \right|,$$

Forward Looking Algorithm

- In Forward Looking Algorithm, previous time step result is not used, instead area is calculated from the following expression:

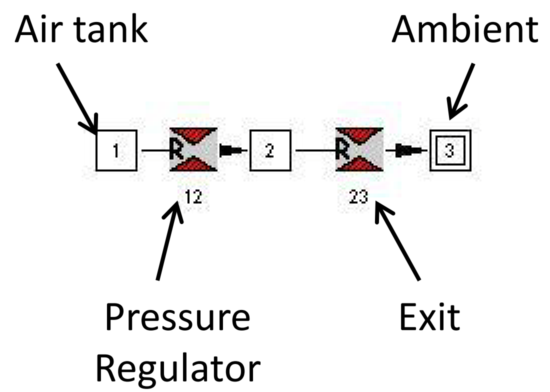
$$A^{*}_{\tau+\Delta\tau} = \begin{cases} \min\left(\left[A_{\tau} + \eta_{relax}\left(A_{new} - A_{\tau}\right)\right] A_{max}\right), \\ \max\left(\left[A_{\tau} + \eta_{relax}\left(A_{new} - A_{\tau}\right)\right] 0\right) \end{cases}$$

where,

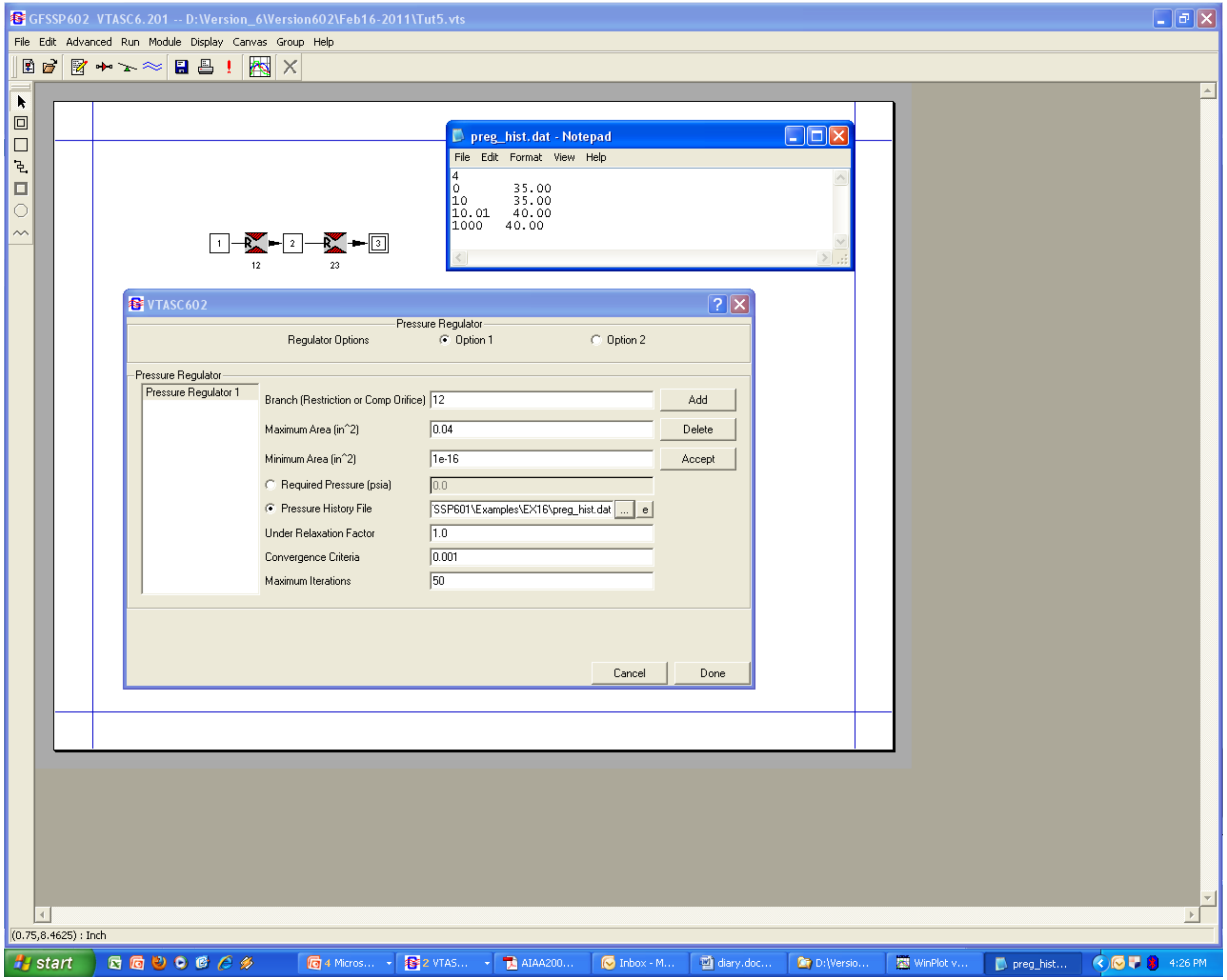
$$A_{new} = A_{\tau}\left(\frac{p_{reg}}{p_{\tau}}\right)^{3}\left(e^{\left(\frac{p_{reg}}{p_{\tau}}-1\right)}\right),$$

Application Results

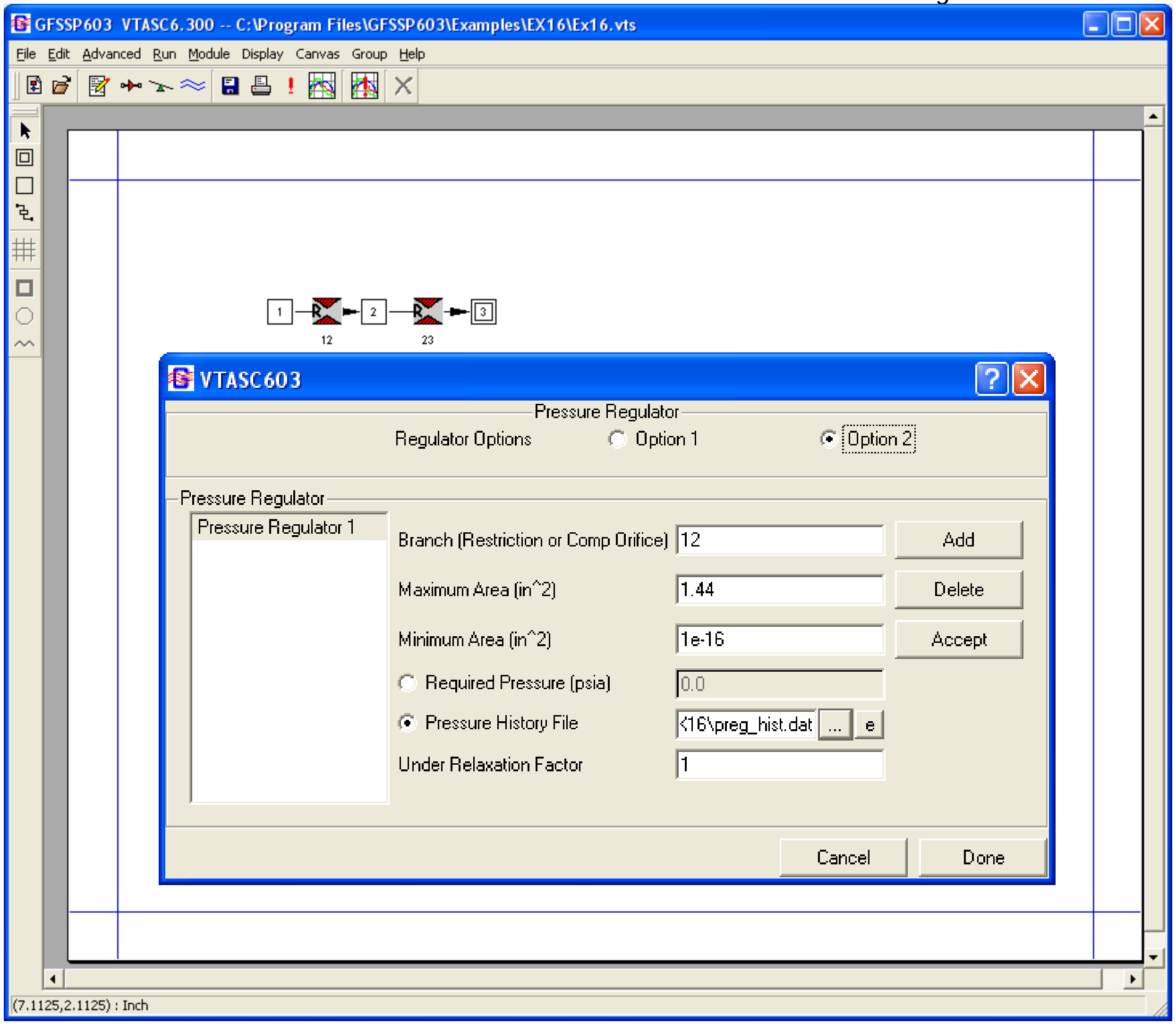
Reference: Forward Looking Pressure Regulator Algorithm for Improved Modeling Performance with the Generalized Fluid System Simulation Program by Paul Schallhorn & Neal Hass, AIAA Paper No. 2004-3667



Iterative Algorithm – Option 1

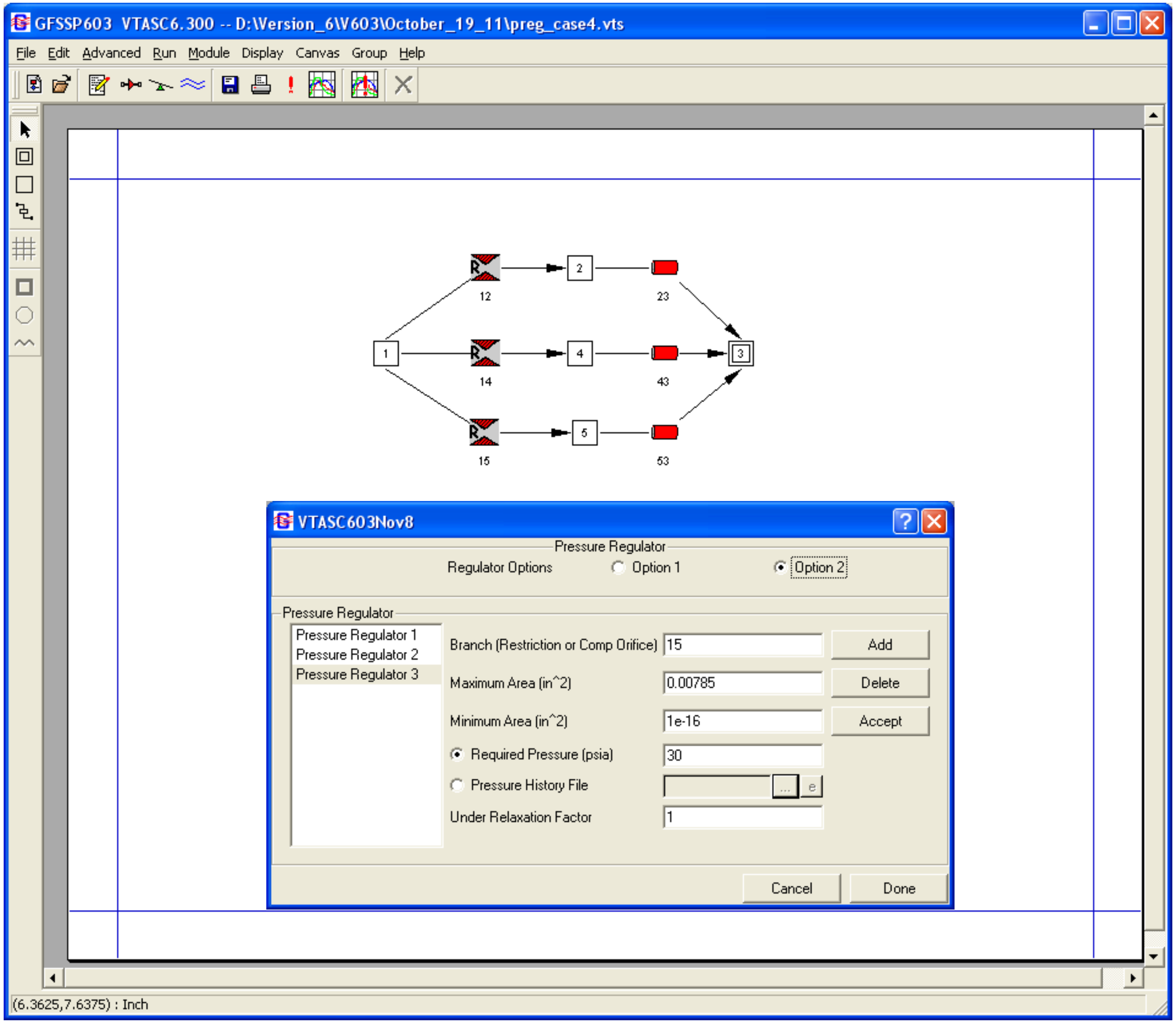


Marching Algorithm – Option 2



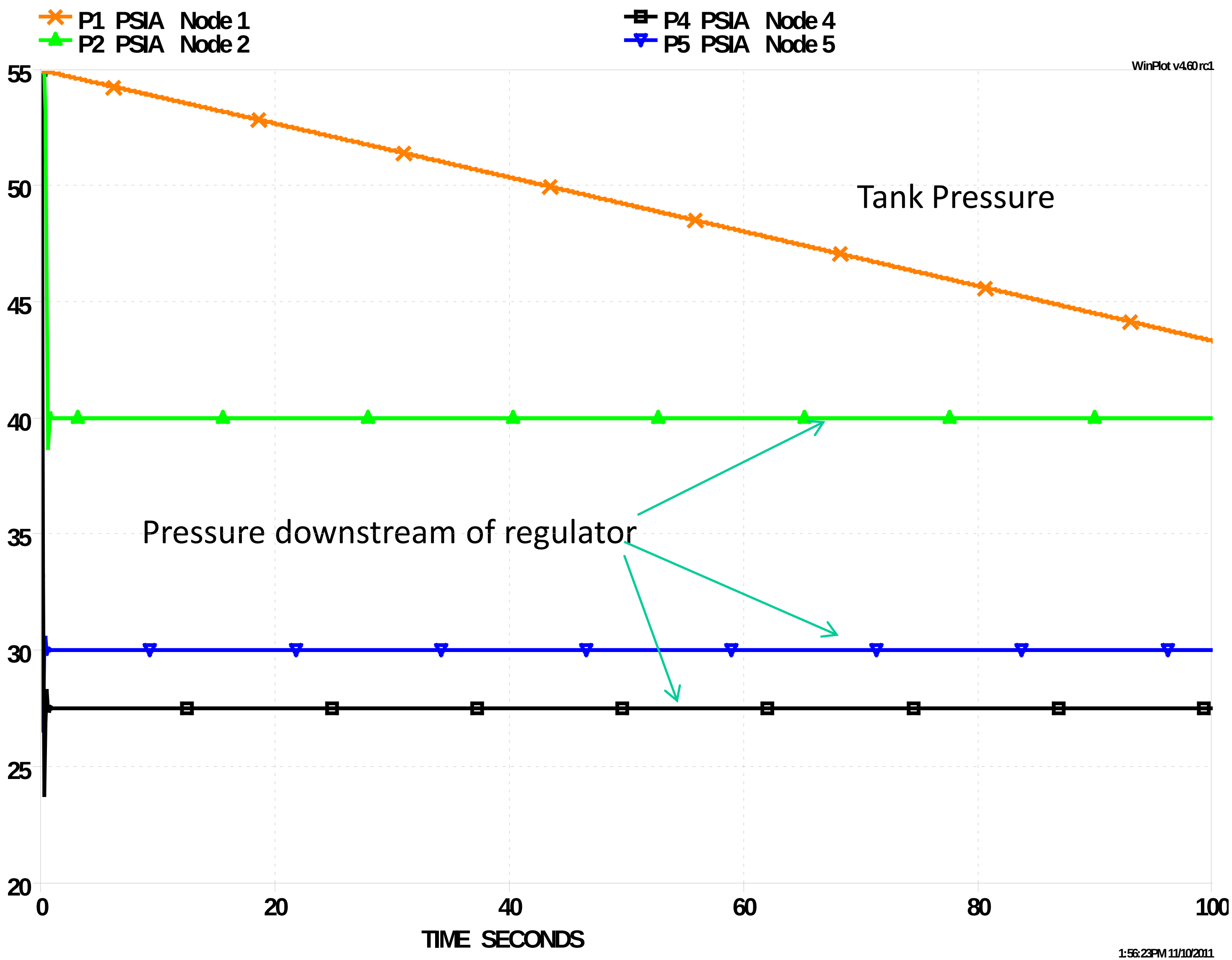

P2 PSIA
P2 PSIA
WinPlot v4.60 rc1
Option 1 (Iterative) – Orange
Option 2 (Marching) - Green
100
80
60
40
20
0
0
40
80
120
160
TIME SECONDS
2:07:10PM 11/28/2011

Forward Looking Pressure Regulator – Option 2



Pressure History

(Schallhorn & Haas Algorithm)

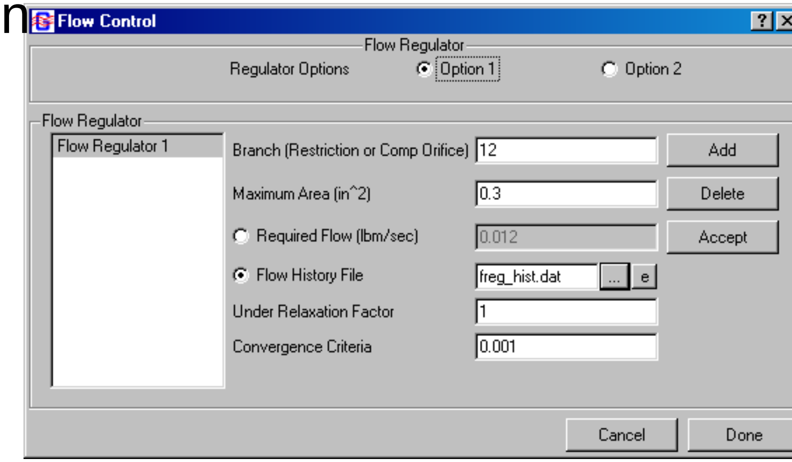


Modeling Flow Regulator

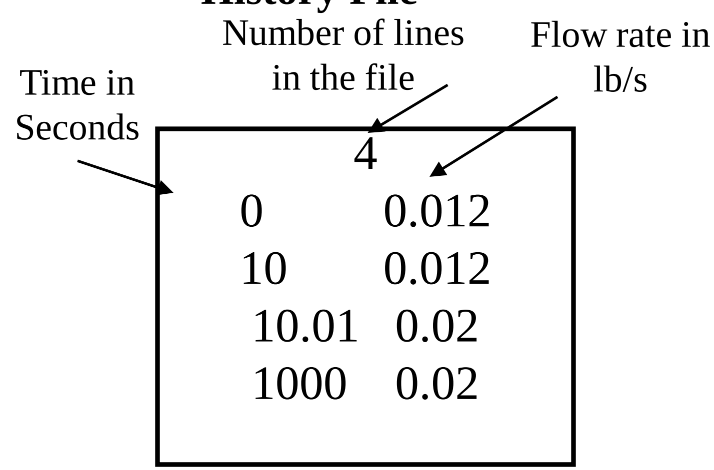
- GFSSP has two built-in options (algorithms) to model a flow regulator
  1. Iterative Algorithm
     - Applicable for single flow regulator; requires longer computation time
     - Serves as an example of how to adjust GFSSP solution to satisfy a given boundary condition
  2. Time-Marching Algorithm
     - Adjusts area once per time-step, based on backwards-differencing functional derivative dF/dA
     - Capable of handling multiple flow regulators

Flow Regulator Option – 1
Iterative Algorithm

- Purpose: To control flow rate in a given branch by adjusting the branch area
- Implementation
  - Step 1
    - Edit
    - Options
    - Unsteady Options
      - Flow Regulator
  - Step 2
    - Advanced
      - Flow Regulator
- Application
  - Example 17 – Simulation of a Flow Regulator downstream of a pressurized tank



History File



Flow Regulator Algorithm

The required flow area is determined by the Newton-Raphson scheme with the following steps:

1. Assume an Area, A^*
2. Compute the deviation, $f = \dot{m}_{req} - \dot{m}$
3. Estimate the gradient

$$f' = \frac{\partial f}{\partial A} = -\frac{2 g_c \rho_u C_L^2 A \, \Delta p}{\dot{m}}$$

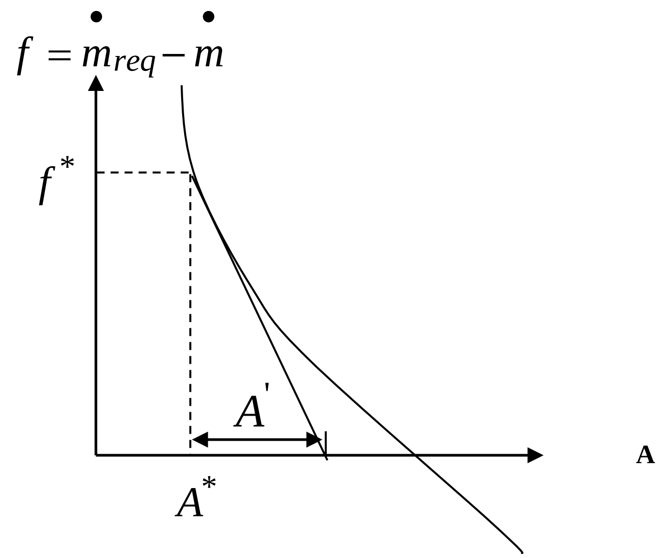
4. Estimate the correction in the Area

$$A' = -\frac{f^*}{f'}$$

5. Compute the new Area

$$A = A^* + \alpha A' \quad where \ 0 < \alpha < 1$$

6. Repeat steps 2 – 5 until $f^* \to 0$



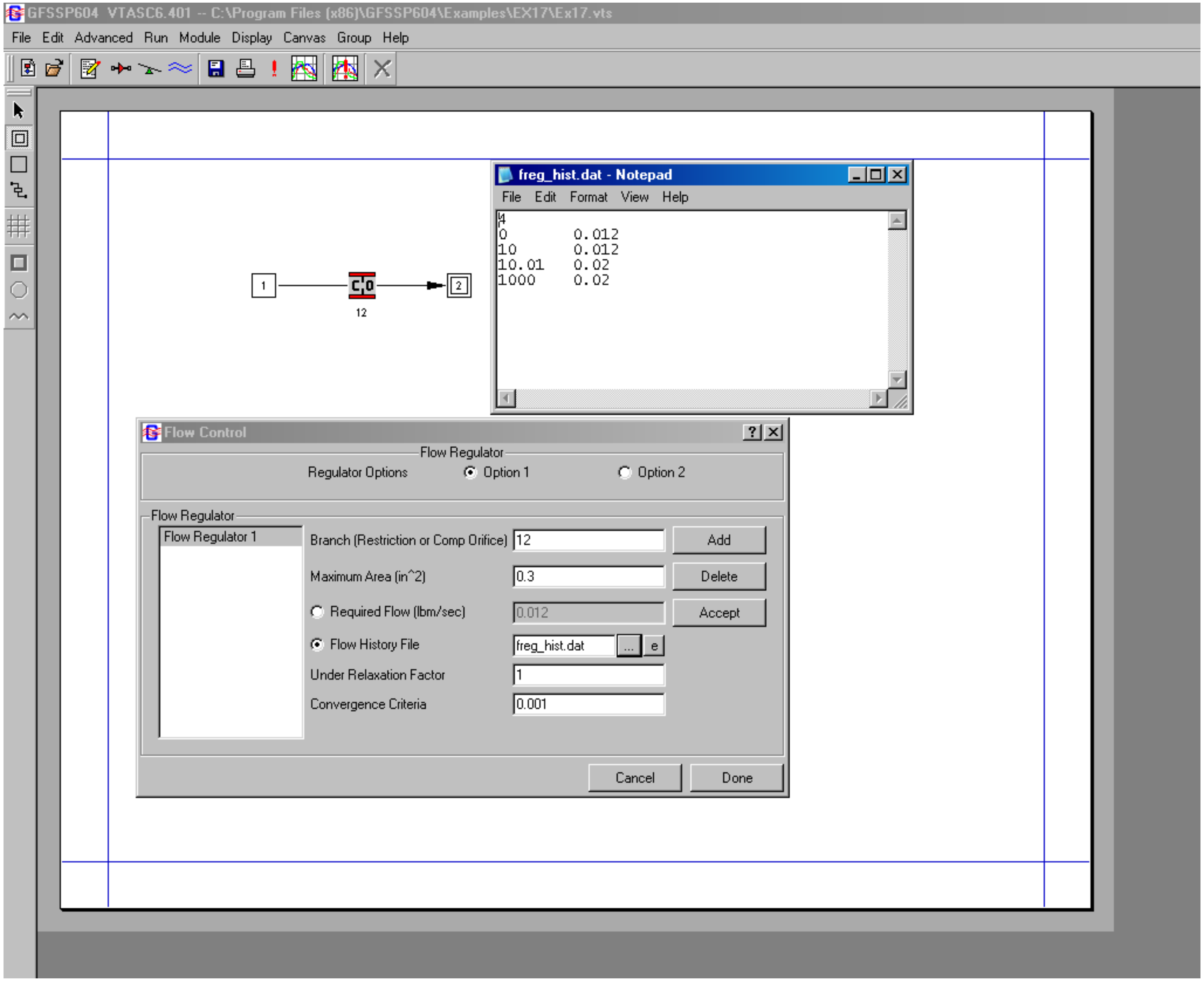
Flow Regulator – Option 2
Time-Marching Algorithm

- In the Time-Marching Algorithm, area is adjusted only once at the beginning of each time step
- Adjustment of area is calculated based on the functional derivative dF/dA and the difference between the calculated and desired flow rate
- CAUTION: If other elements of the model have significant effect on the flow rate, calculation of dF/dA by backwards differencing may lead to numerical instability. May require under-relaxation.

$$A_{\tau+\Delta\tau} = A_{\tau} - \eta_{relax} \frac{\left(F_{\tau} - F_{req}\right)}{\frac{dF}{dA}}$$

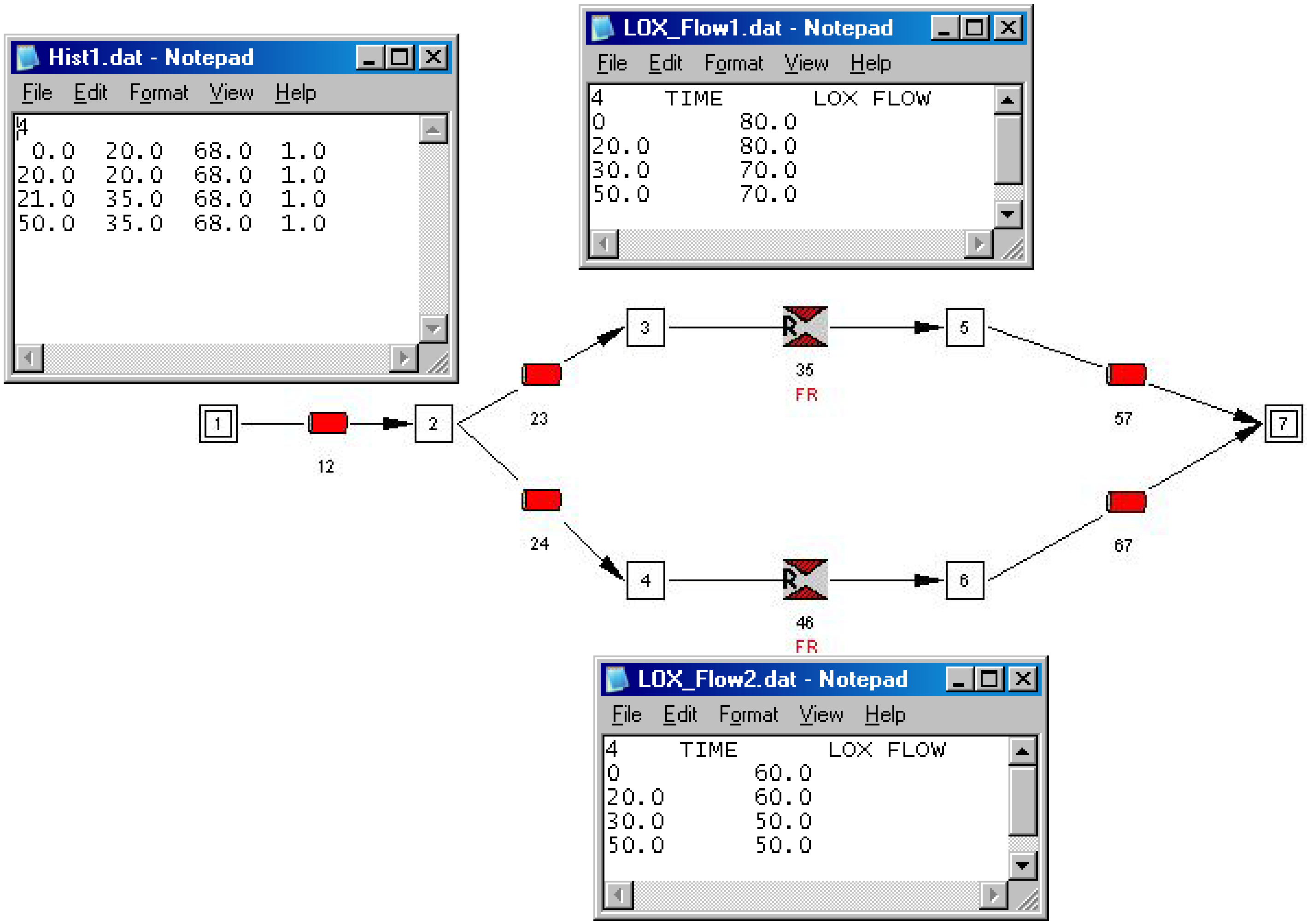
$$\frac{dF}{dA} = \frac{\left(F_{\tau} - F_{\tau-\Delta\tau}\right)}{\left(A_{\tau} - A_{\tau-\Delta\tau}\right)}$$

Flow Regulator – Example 17



Comparison between Iterative and Time-Marching Algorithms

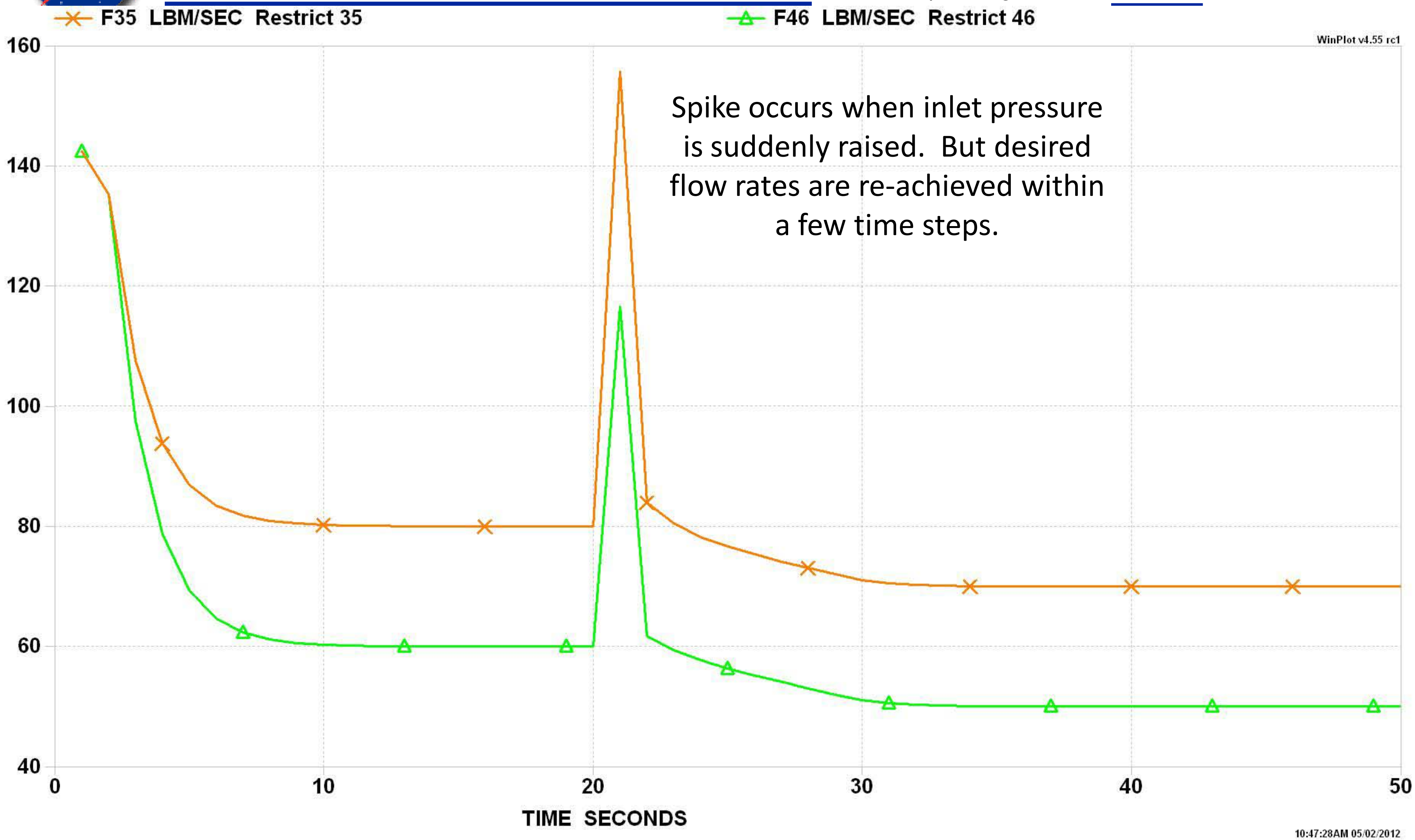
Example of Multiple Flow Regulators



GFSSP v604 -- Pressure and Flow Regulators

Flow Rate History

Marshall Space Flight Center



Summary

- Pressure & Flow Regulator Options have been made available to include in any unsteady flow simulation
- Pressure Regulator has two options: Iterative (Option -1) and Marching (Option -2)
- Option-2 has the flexibility of using multiple regulators and runs faster
- Flow Regulator has only two options: Iterative (Option-1) and Marching (Option-2)
- Option-2 has the flexibility of using multiple regulators and runs faster; however, it may require relaxation for numerical stability
- Fixed Flow Branch Option can also be used to regulate flow in multiple branches

Tutorial – 4

Modeling a Pressure Regulator

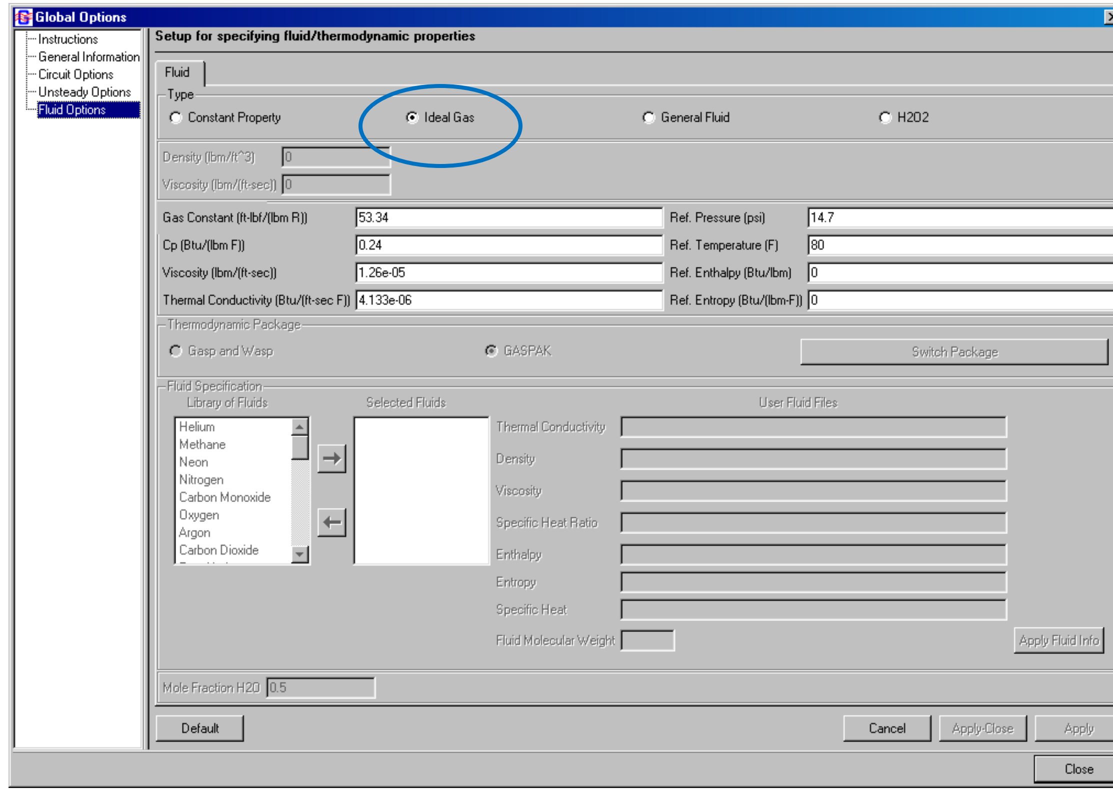
In this project, you will:

•Use GFSSP's built-in pressure regulator options to model the regulated blowdown of a tank of compressed air

•Learn the difference between the two pressure regulator options

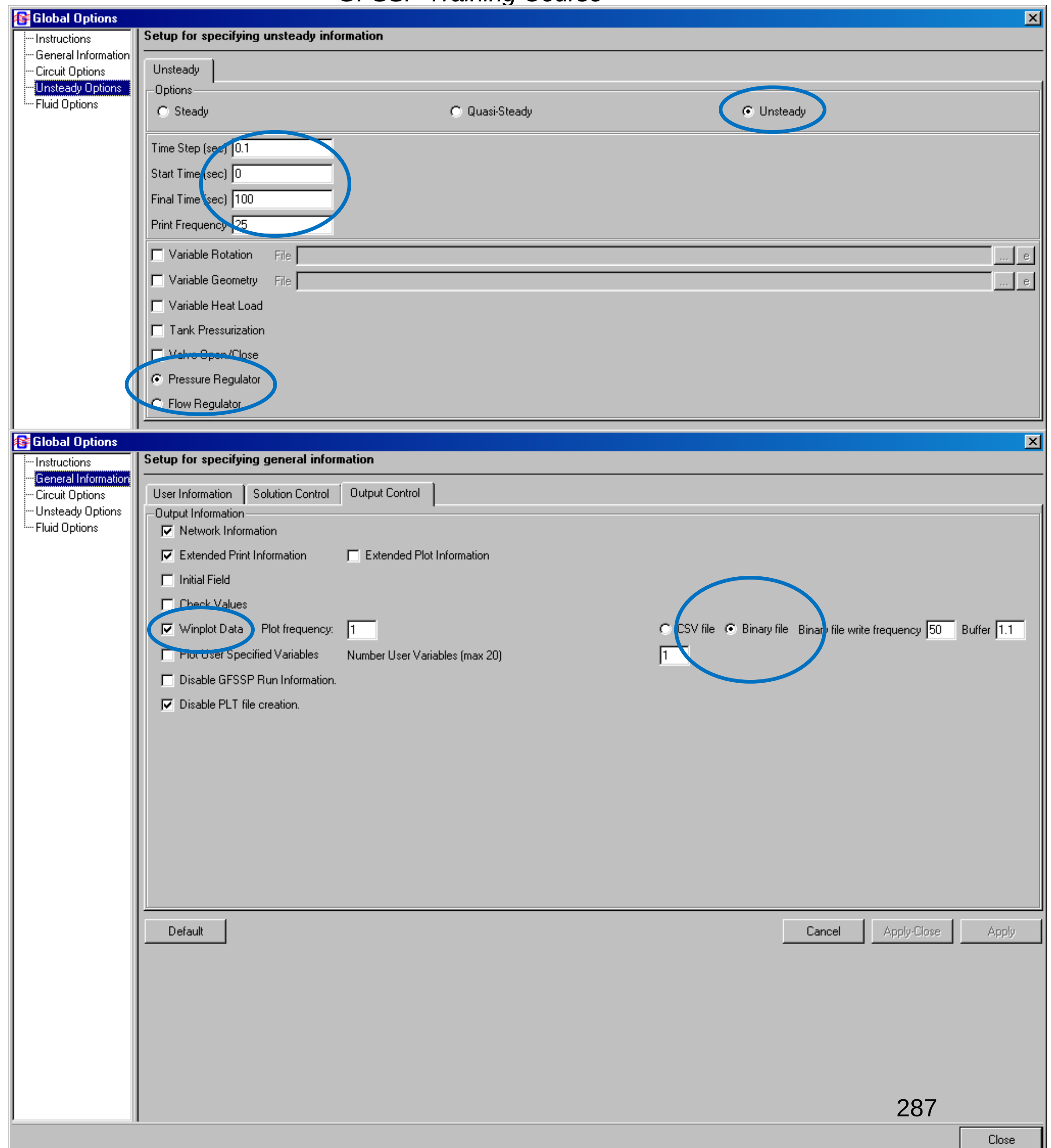
Set Up Options

- •VTASC File: Tut4.vts
- •User Information
  - •Input File: Tut4.dat
  - •Output File: Tut4.out
- •Fluid is Ideal Gas
  - • Defaults to air properties

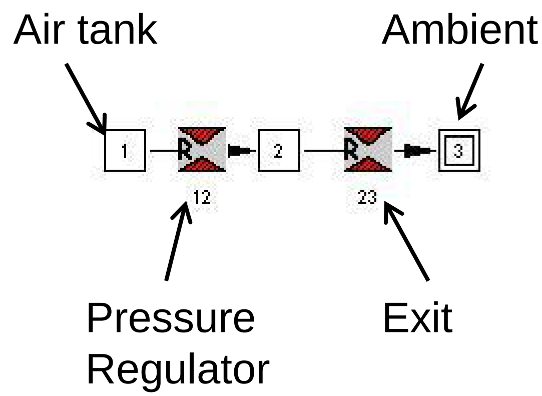


Set Up Options (cont.)

- Transient run
    - Time step: 0.1 s
    - Final time: 100 s
    - Print Freq: 25
    - Select Pressure Regulator option
- Output Control
    - Select Winplot binary output

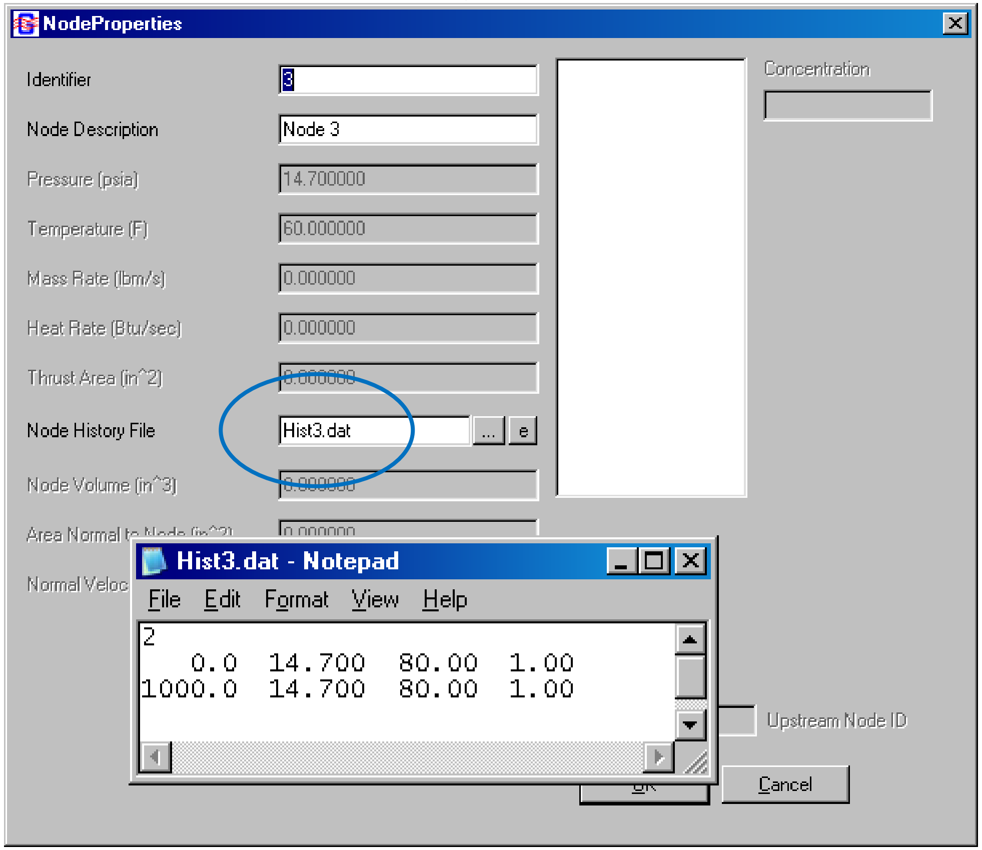


Build Model on Canvas



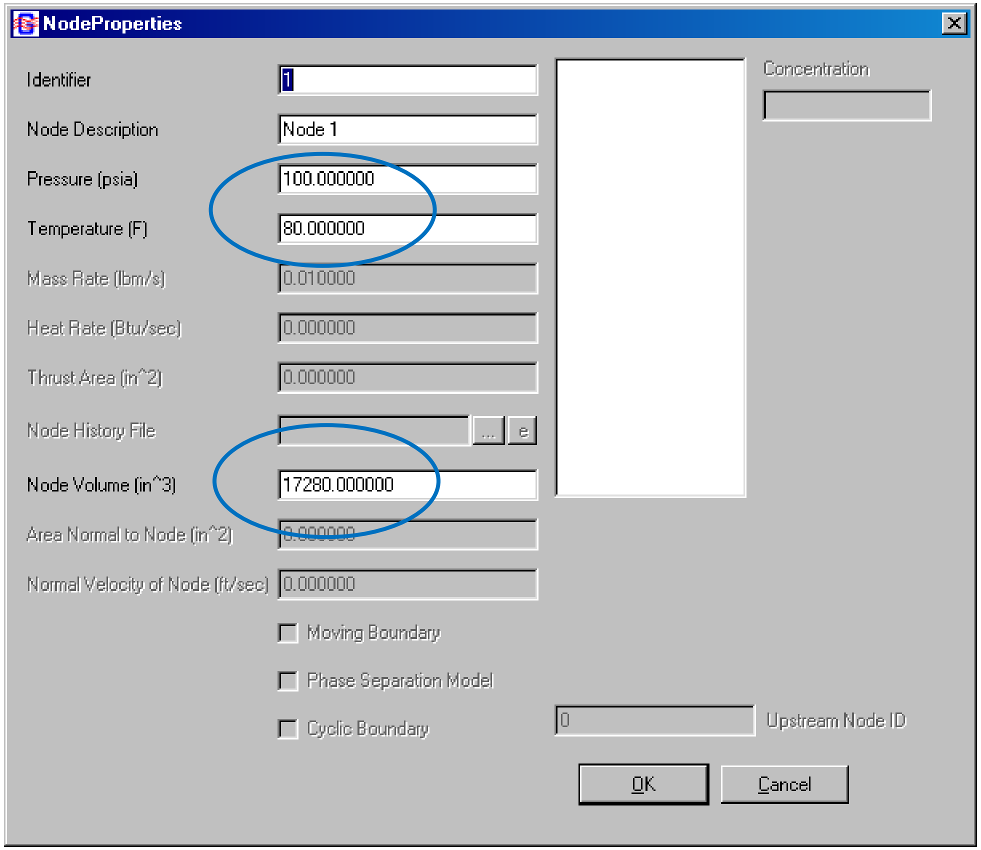
Set Up Transient Boundary Condition

- Node 3:
  - P = 14.7 psia
  - T = 80.0 °F



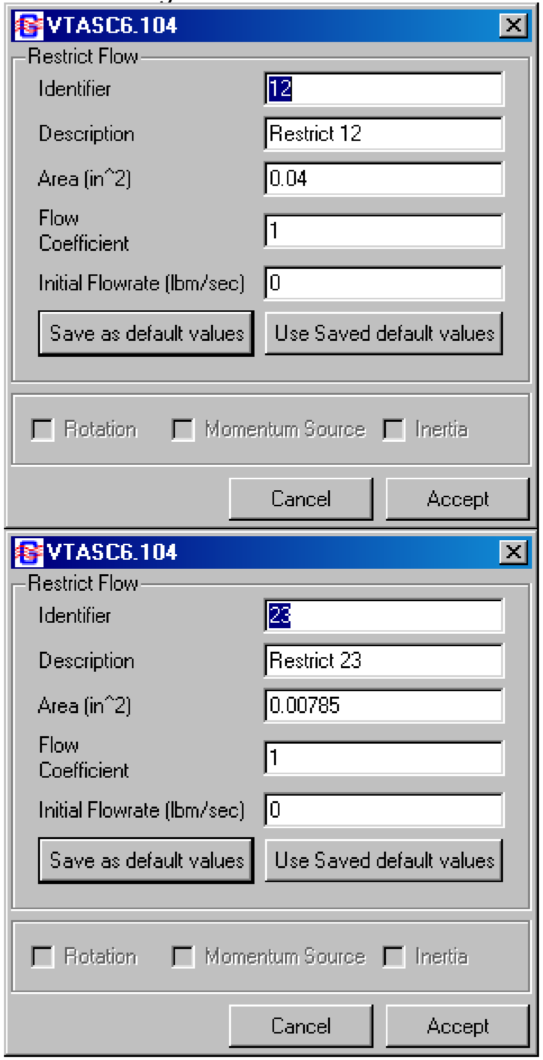
Set Up Internal Nodes

- Node 1:
  - Initial P = 100.0 psia
  - Initial T = 80.0 °F
  - Vol. = 10 ft^3 = 17280 in^3
- Node 2:
  - Initial P = 14.7 psia
  - Initial T = 80.0 °F
  - Vol. = 100 in^3



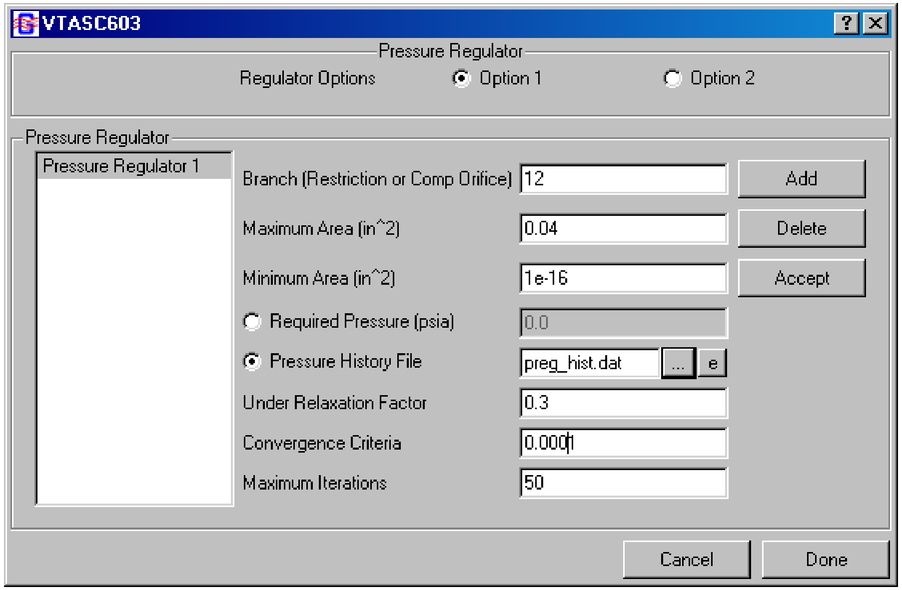
Set Up Fluid Branches

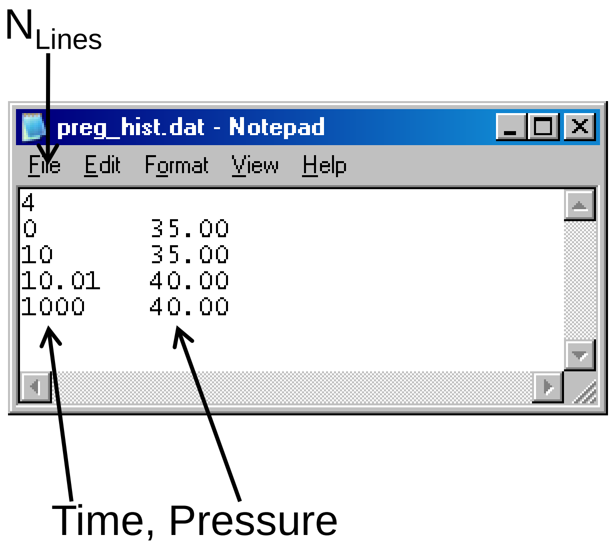
- Branch 12: Pressure Regulator
  - Initial A = 0.04 in^2
  - C_L = 1.0
- Branch 23: Exit
  - A = 0.00785 in^2
  - C_L = 1.0



Set Up Pressure Regulator Option 1

- Select Advanced/Pressure Regulator
- Make sure the “Option 1” radio button is selected
- Click “Add”
- Fill in the dialog boxes
- Create a pressure history data file: preg_hist.dat
- Click “Accept”, “Done”
- For each time step, GFSSP will adjust the area of Branch 12 to maintain the desired pressure in the downstream node.

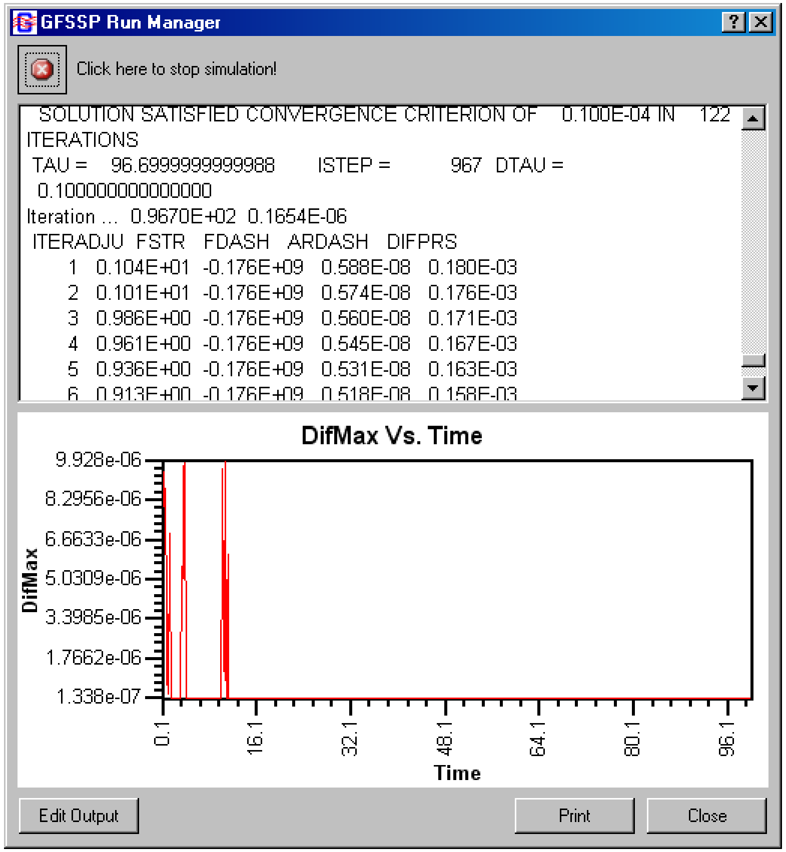




Result of Pressure Regulator Model

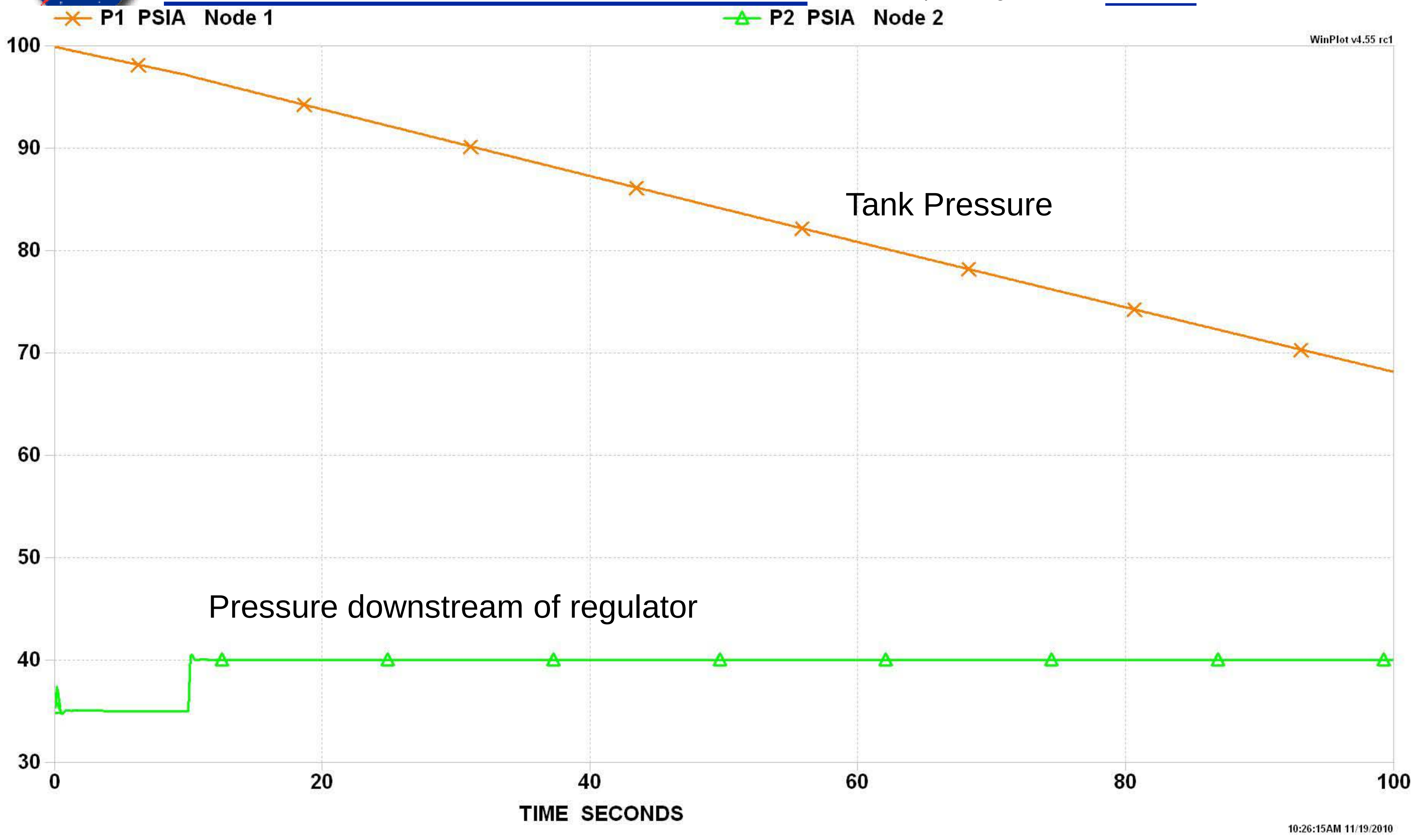
Marshall Space Flight Center
GFSSP Training Course

- Run the model
- Note that in each time step GFSSP is adjusting the area of Branch 12 to meet the desired pressure.
- What effect do you think this has on run time?

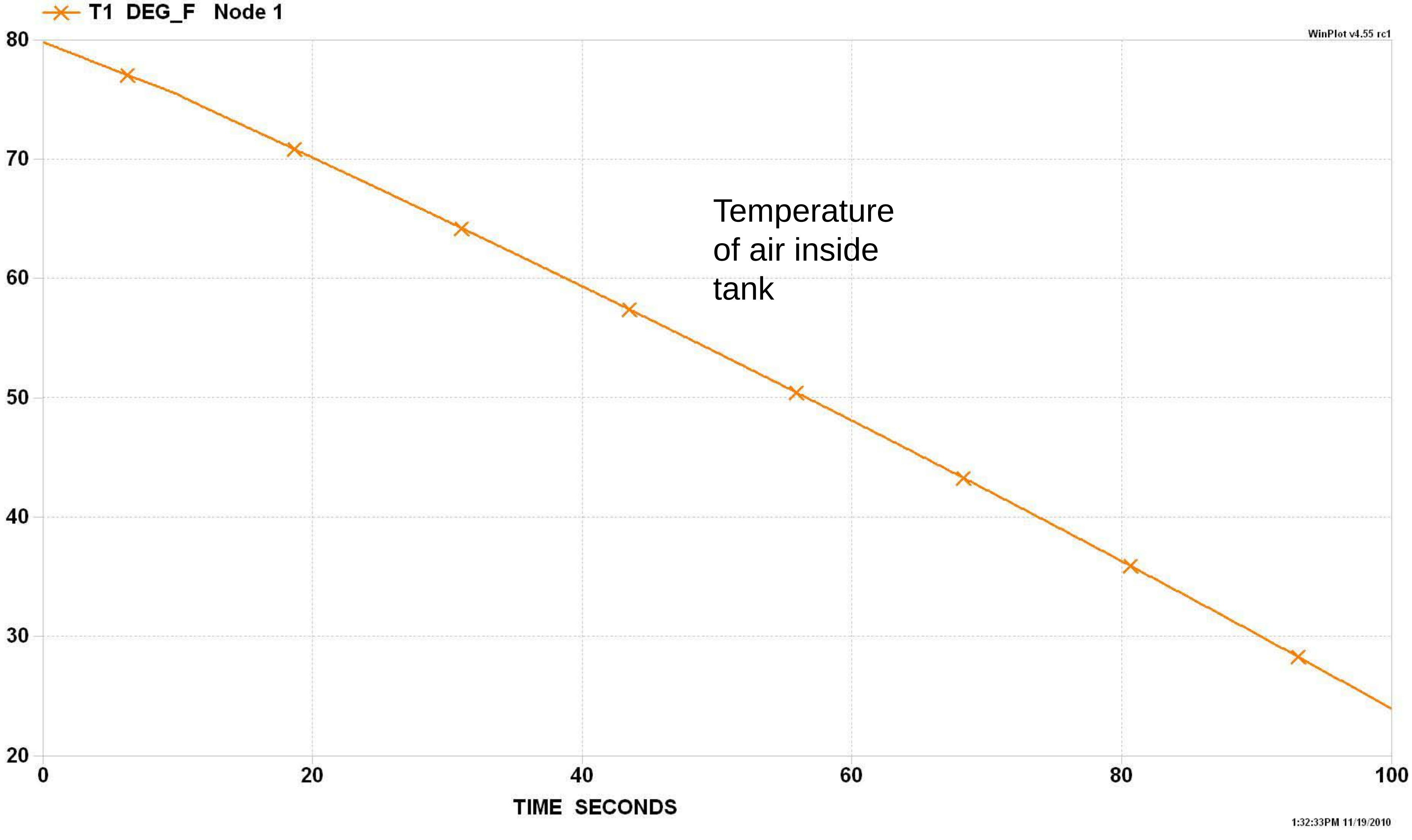


Pressure History

Marshall Space Flight Center

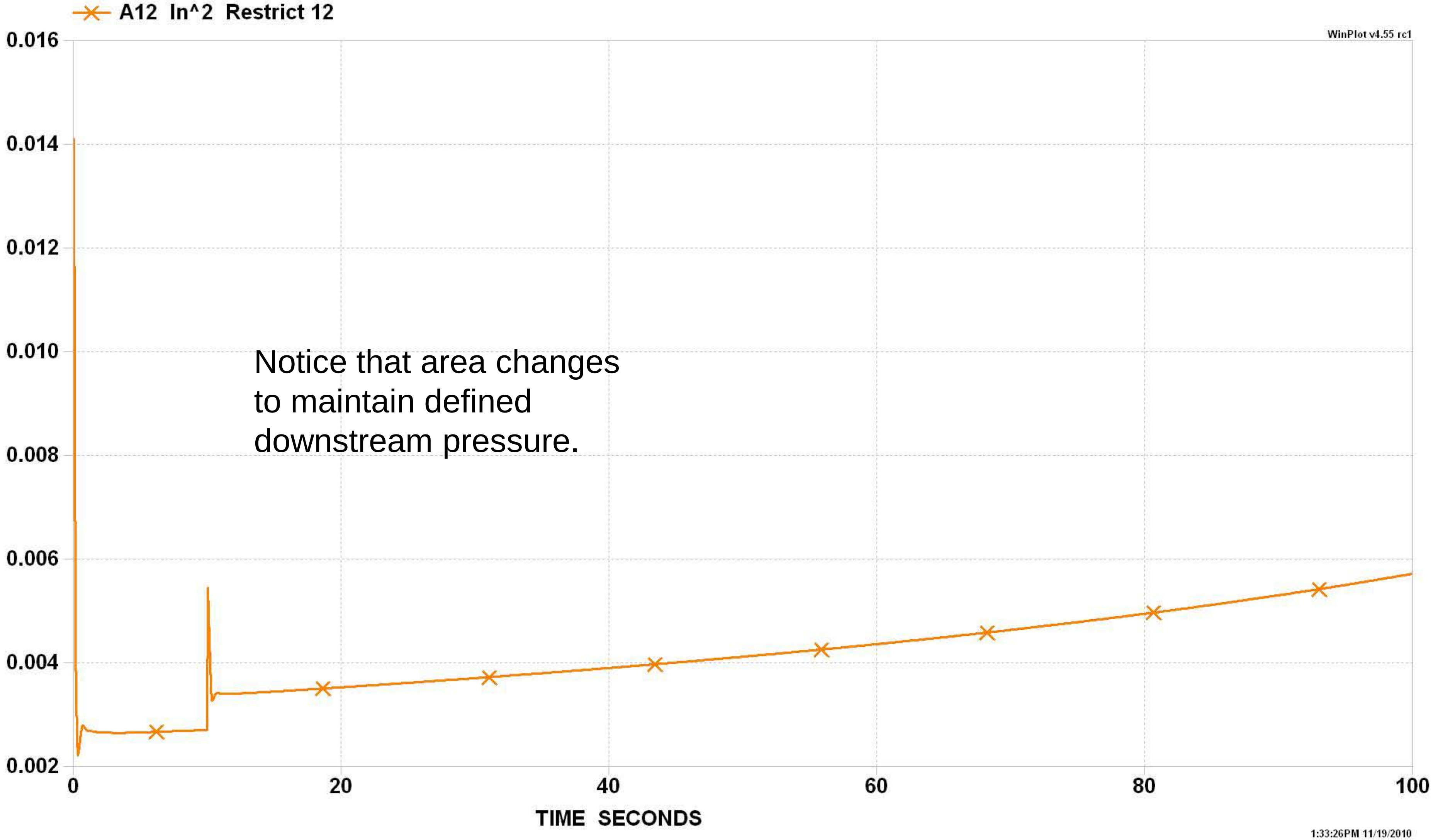


Temperature History



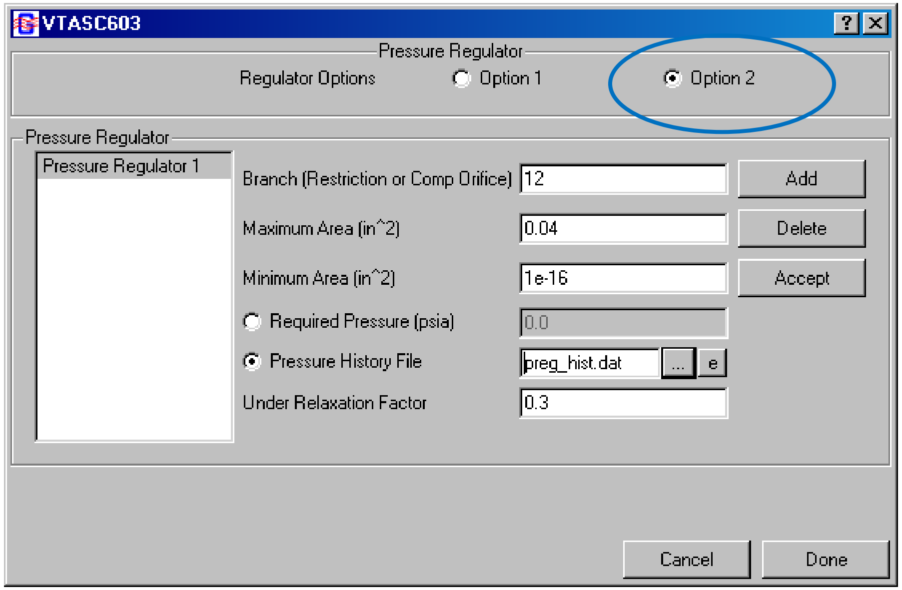
Pressure Regulator Area History

Marshall Space Flight Center



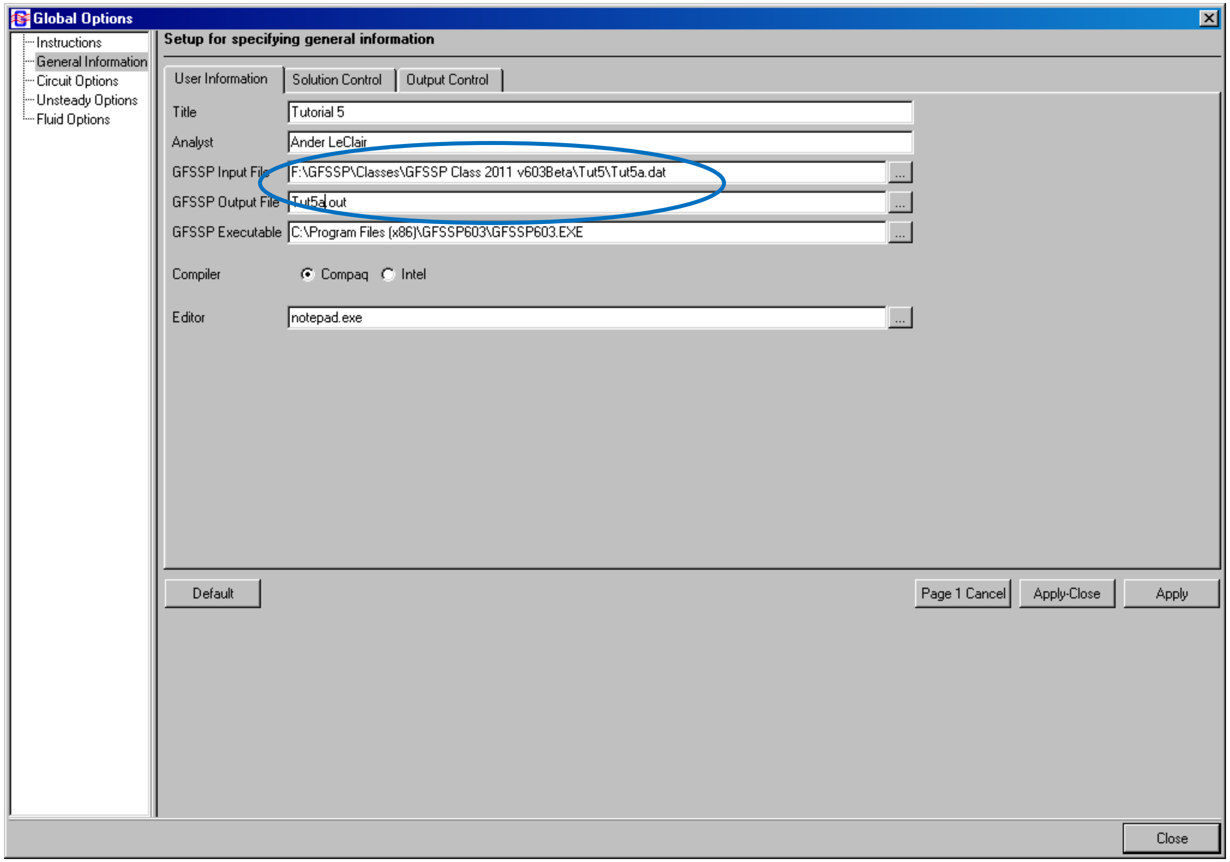
Part 2: Modeling the Forward-Looking Pressure Regulator

- Now we will repeat the exercise using a different pressure regulator algorithm
- Go to Advanced/Pressure Regulator
- Delete the current pressure regulator
- Select “Option 2” radio button
- Click “Add”
- Fill in the dialog boxes
- Click “Accept”, “Done”



Rename the GFSSP Files

- On the edit options page, rename the input and output files:
    - Tut4a.dat
    - Tut4a.out
- This will prevent overwriting the results of the first pressure regulator



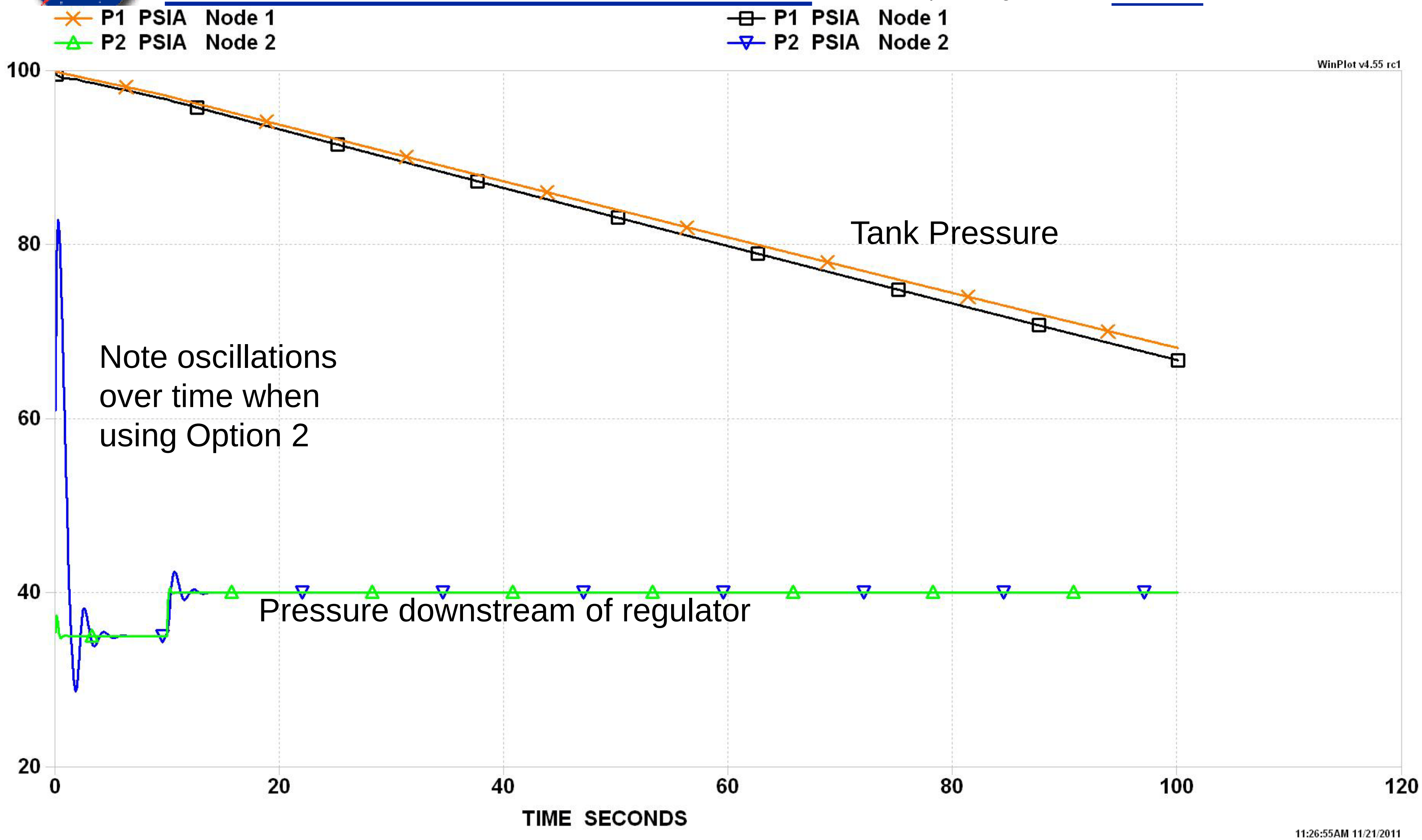
Result of Pressure Regulator Model

- Run the model
- Note that this model runs faster. Why?
  - GFSSP's Option 1 pressure regulator iterates the branch area at every timestep to meet the required pressure. Therefore each timestep is run 10-20 times. It's like a regulator that reacts instantaneously.
  - The Option 2 regulator adjusts the area just once at the beginning of each time step, based on a relation developed by Schallhorn and Haas. It reacts in a finite amount of time, as would a real pressure regulator.

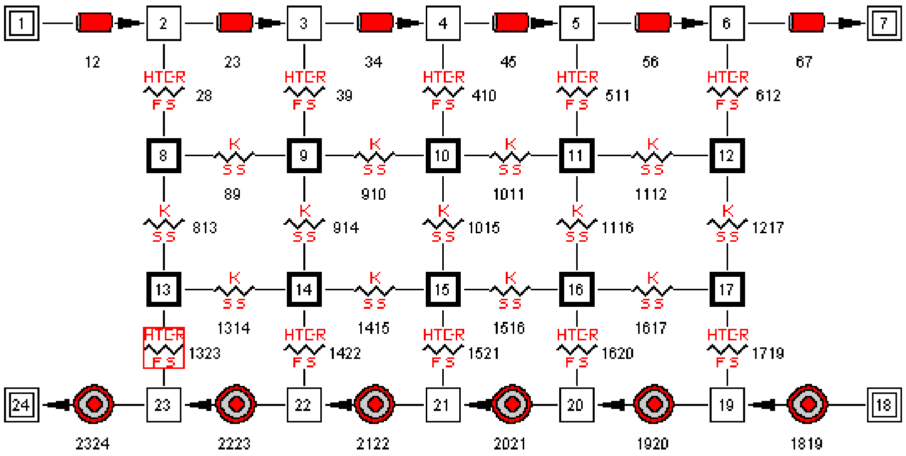
$$A_{New} = A_{\tau}\left(\frac{P_{req.}}{P_{\tau}}\right)^{3} e^{\left(\frac{P_{req.}}{P_{\tau}}-1\right)}$$

- Plot the new Option 2 results (Tut5a.WPL) over the Option 1 results (Tut5.WPL)
- Time permitting, try rerunning Option 2 with a different relaxation factor and note its effect on the pressure oscillations.

Pressure History



Conjugate Heat Transfer Applications

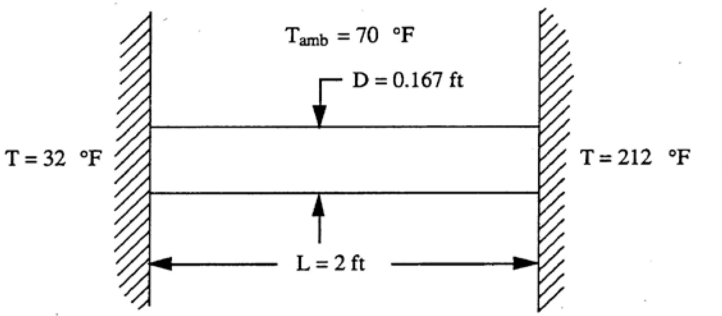


Conjugate Heat Transfer

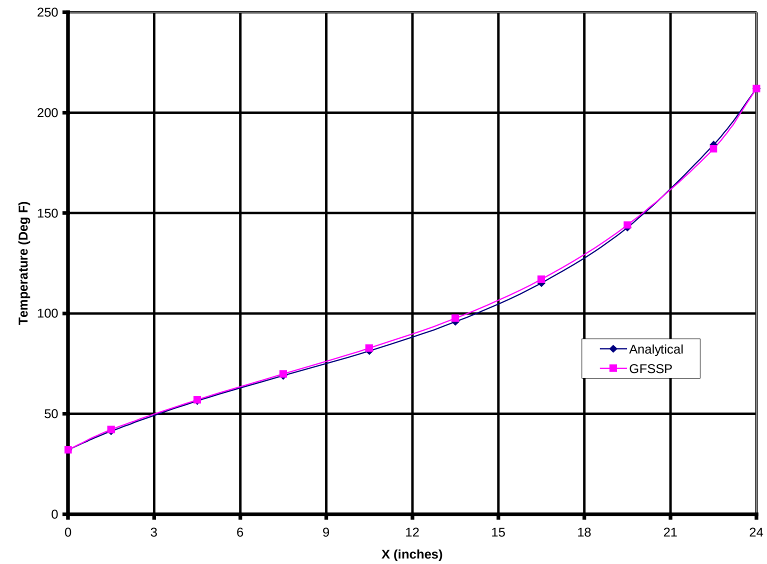
Why do we need it?

- In many applications fluid flow and heat transfer are strongly coupled
- Typical examples in Propulsion Systems are:
  - Pressurization of cryogenic propellant tank
  - Chilldown of cryogenic transfer line
  - Regenerative cooling of engine nozzle
- Integration of separate models of fluid flow and heat transfer is difficult to construct and converge to a correct solution
- A better approach is to build a conjugate model using one solver module to solve for fluid and solid properties

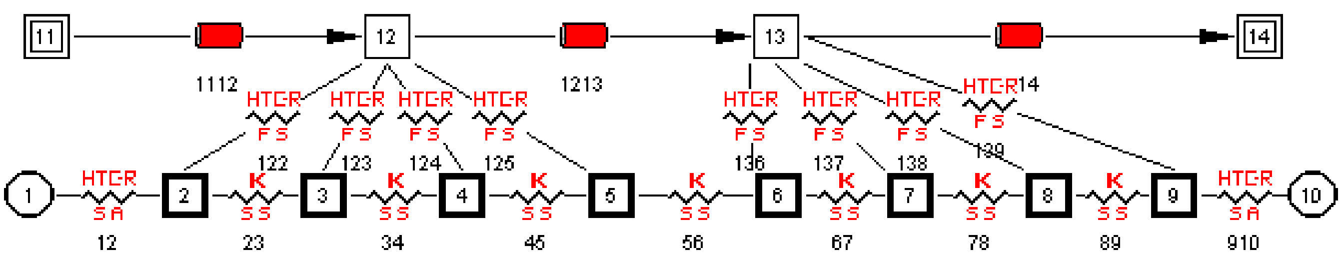
Verification of Conjugate Heat Transfer Results



Problem Considered

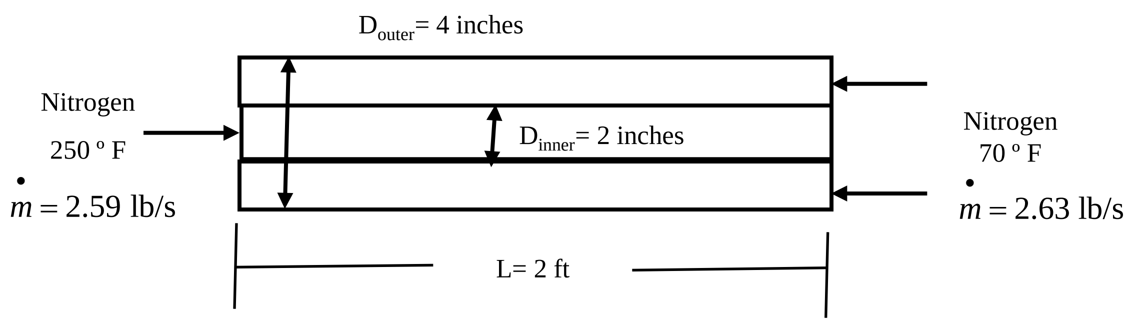


Comparison with Analytical Solution

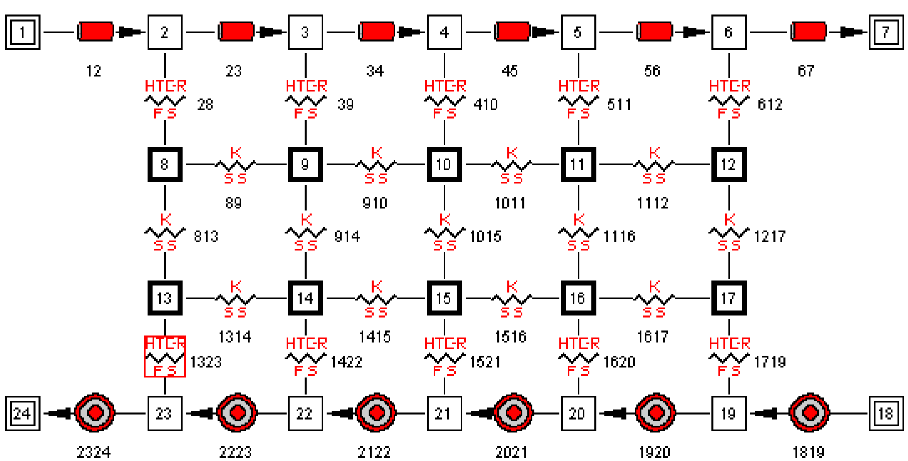


GFSSP Model

Verification of Counter-flow Heat Exchanger Results



T$_6$=246 ° F



T$_{23}$ = 73.91 ° F

Heat Balance

Heat lost by hot leg =

$$\dot{m}_{23}\, C_p\left(T_1 - T_6\right) = 2.5879\ Btu/s$$

Heat gained by cold leg =

$$\dot{m}_{1819}\, C_p\left(T_{23} - T_{18}\right) = 2.577\ Btu/s$$

Heat Exchanger Effectiveness

GFSSP

$$\varepsilon = \frac{q}{q_{\max}} = \frac{\dot{m}_{23}\, C_p\left(T_1 - T_6\right)}{\left(\dot{m} C_p\right)_{\min}\left(T_{h,in} - T_{c,in}\right)} = \frac{2.5879}{116.46} = .0222$$

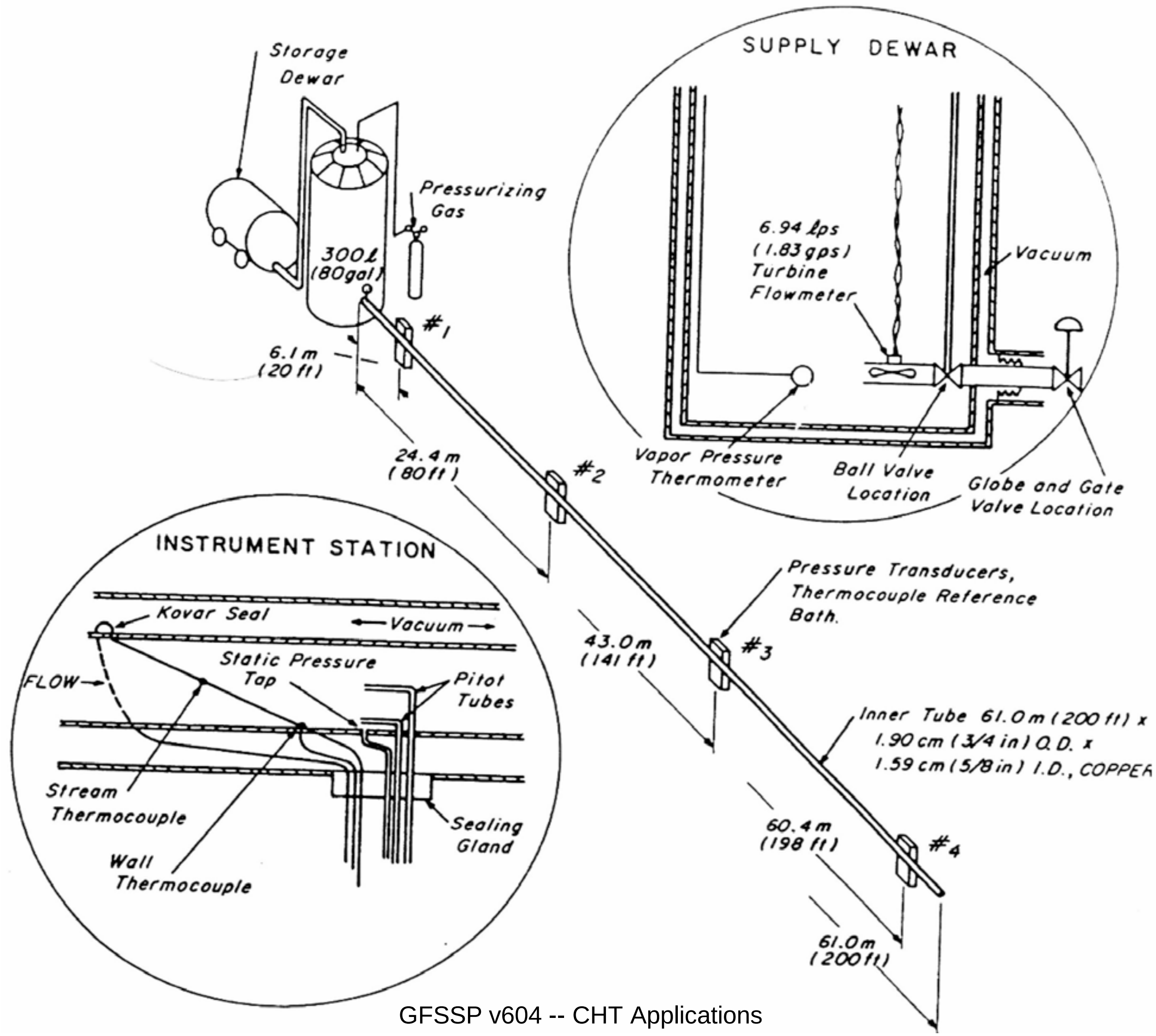
Effectiveness-NTU Relationship

$$\varepsilon = \frac{1 - e^{-NTU\left(1 - \frac{C_{\min}}{C_{\max}}\right)}}{1 - \frac{C_{\min}}{C_{\max}} e^{-NTU\left(1 - \frac{C_{\min}}{C_{\max}}\right)}} = 0.02194$$

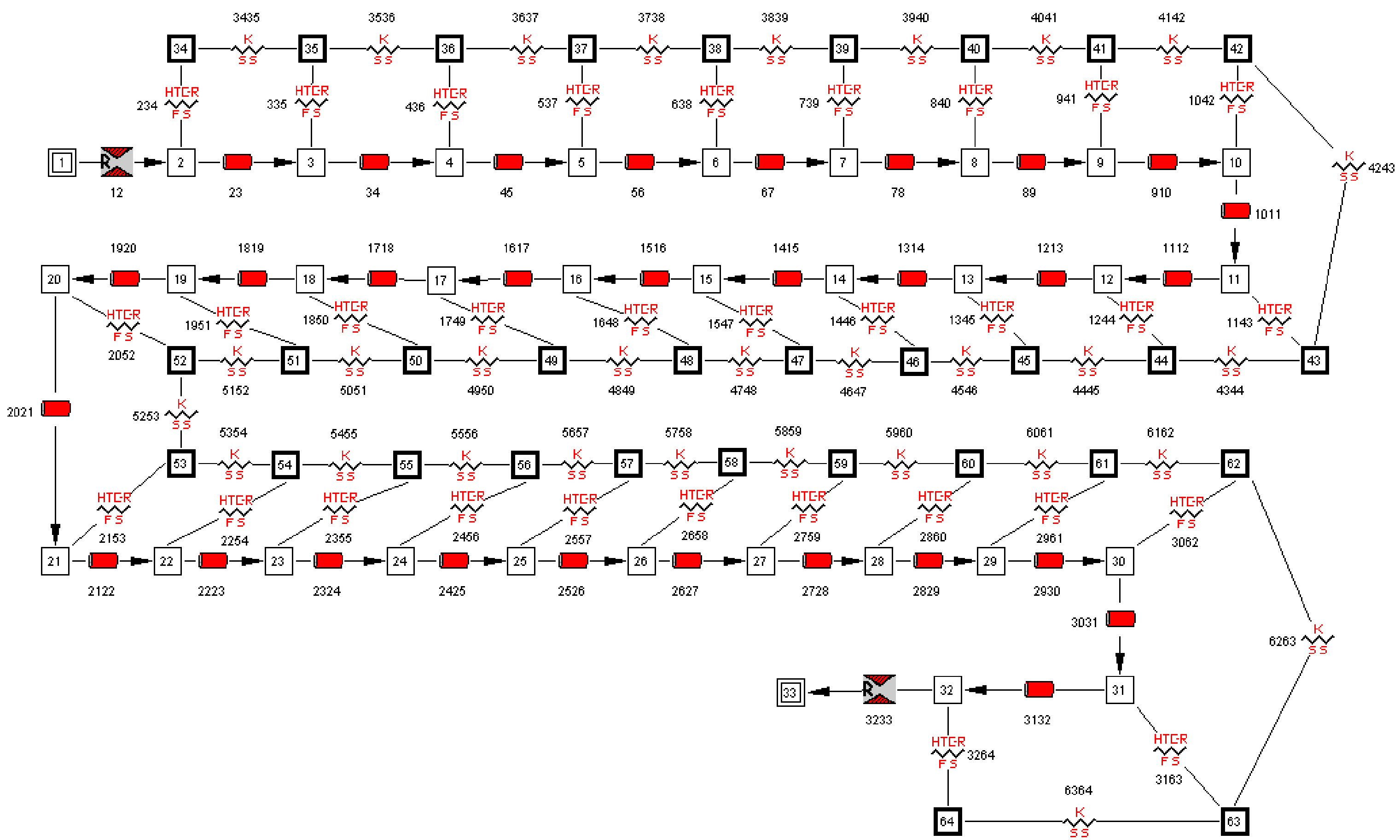
Advanced Applications

- Modeling Cryogenic Transfer Line
  - Validation of CHT Capability
- Modeling of Propellant Loading
  - Model Verification by comparing with Space Shuttle Data
- Pressurization of Space Shuttle's LH2 Tank
  - Model Verification by comparing with Space Shuttle Data

NBS Test Set-up of Cryogenic Transfer Line



GFSSP Model of Cryogenic Transfer Line



Comparison with Test Data

Saturated LH_2 chilldown time for various driving pressures

Driving Pressure (psia)	Saturation Temperature (°F)	Experimental Chilldown Time (s)	Predicted Chilldown Time (s)
74.97	−411.06	68	70
86.73	−409.08	62	69
111.72	−406.4	42	50
161.72	−402.13	30	33

Subcooled LH_2 chilldown time for various driving pressures. LH_2 is subcooled at –424.57 ºF

Driving Pressure (psia)	Experimental Chilldown Time (s)	Predicted Chilldown Time (s)
36.75	148	150
61.74	75	80
86.73	62	60
111.72	41	45
136.72	32	35
161.7	28	30

Saturated LN_2 chilldown time for various driving pressures

Driving Pressure (psia)	Saturation Temperature (°F)	Experimental Chilldown Time (s)	Predicted Chilldown Time (s)
61.74	−294.09	165	185
74.97	−289.71	150	160
86.73	−286.24	130	140

Subcooled LN_2 chilldown time for various driving pressures. LN_2 is subcooled at –322.87 ºF

Driving Pressure (psia)	Experimental Chilldown Time (s)	Predicted Chilldown Time (s)
36.75	222	250
49.97	170	175
61.74	129	140
74.97	100	100
86.73	85	90

Comparison of temperature histories for subcooled LH_2 for various driving pressures

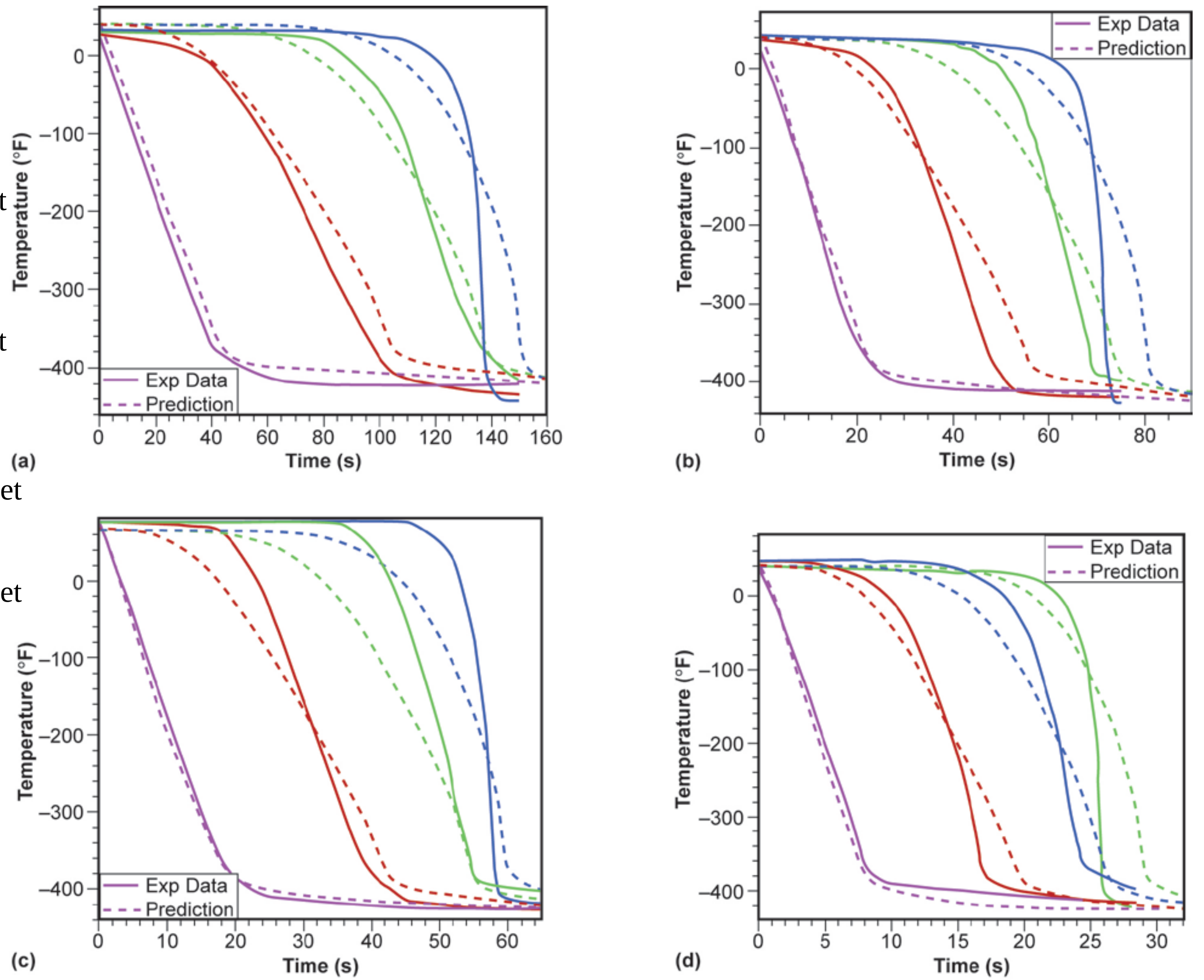
p=36.74 psia p=61.72 psia

Station #1 (violet)
—20 ft from tank inlet

Station #2 (red)
—80 ft from tank inlet

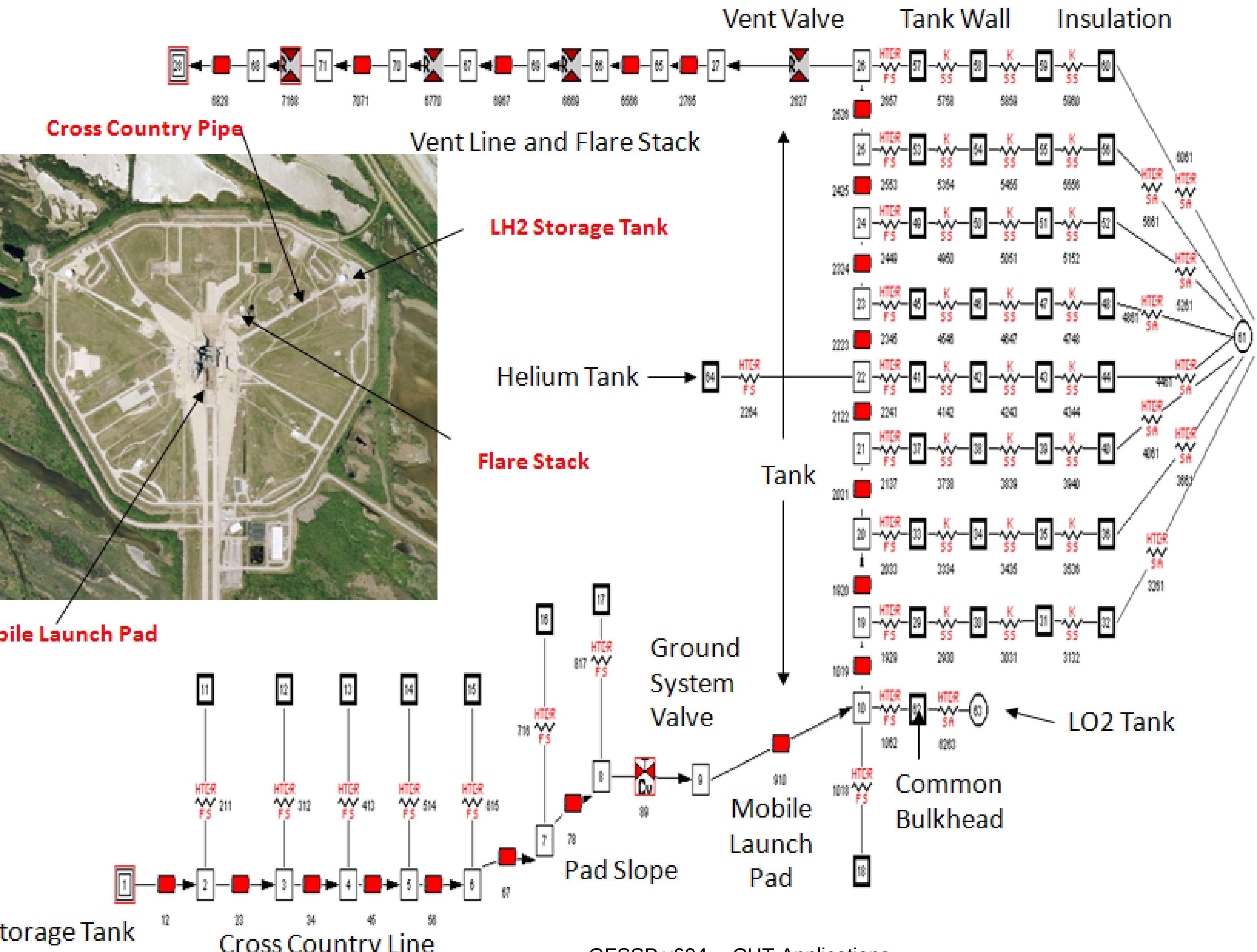
Station #3 (green)
—141 ft from tank inlet

Station #4 (blue)
—198 ft from tank inlet



p=86.7 psia p=161 psia

GFSSP Model of LH2 Tank Loading of Ares I Upper Stage



GFSSP v604 -- CHT Applications

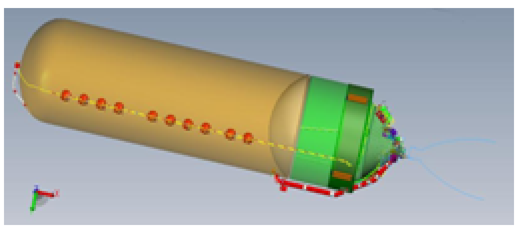
Requirements for Propellant Loading

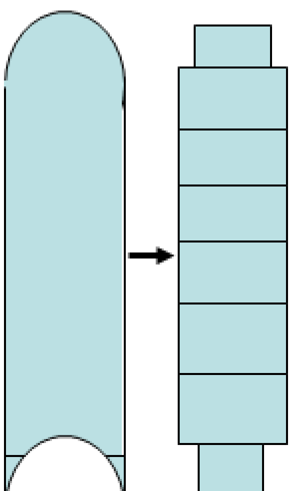
- **LH2 Loading**
  - Slow fill – 2 lb/sec until Tank is 5% full
  - Fast fill – 15 lb/sec until Tank is 95% full
  - Topping – 2 lb/sec until Tank is 100% full
  - Replenish – 1 lb/sec to allow replenishment due to boil-off

- **LO2 Loading**
  - Slow fill – 9 lb/sec until Tank is 5% full
  - Fast fill – 93 lb/sec until Tank is 95% full
  - Topping – 9 lb/sec until Tank is 100% full
  - Replenish – 1 lb/sec to allow replenishment due to boil-off

- **Pre-chill**
  - Chilling of both tanks should start simultaneously to maintain a favorable thermal gradient across Common Bulkhead
  - LH2 loading can only start after completion of LO2 loading followed by 15 minutes of pressure test
  - Tank pressure must not exceed 10 psig during loading

Input Data for Integrated Ground System, LH_2 Tank and Flare Stack Model of Propellant Loading





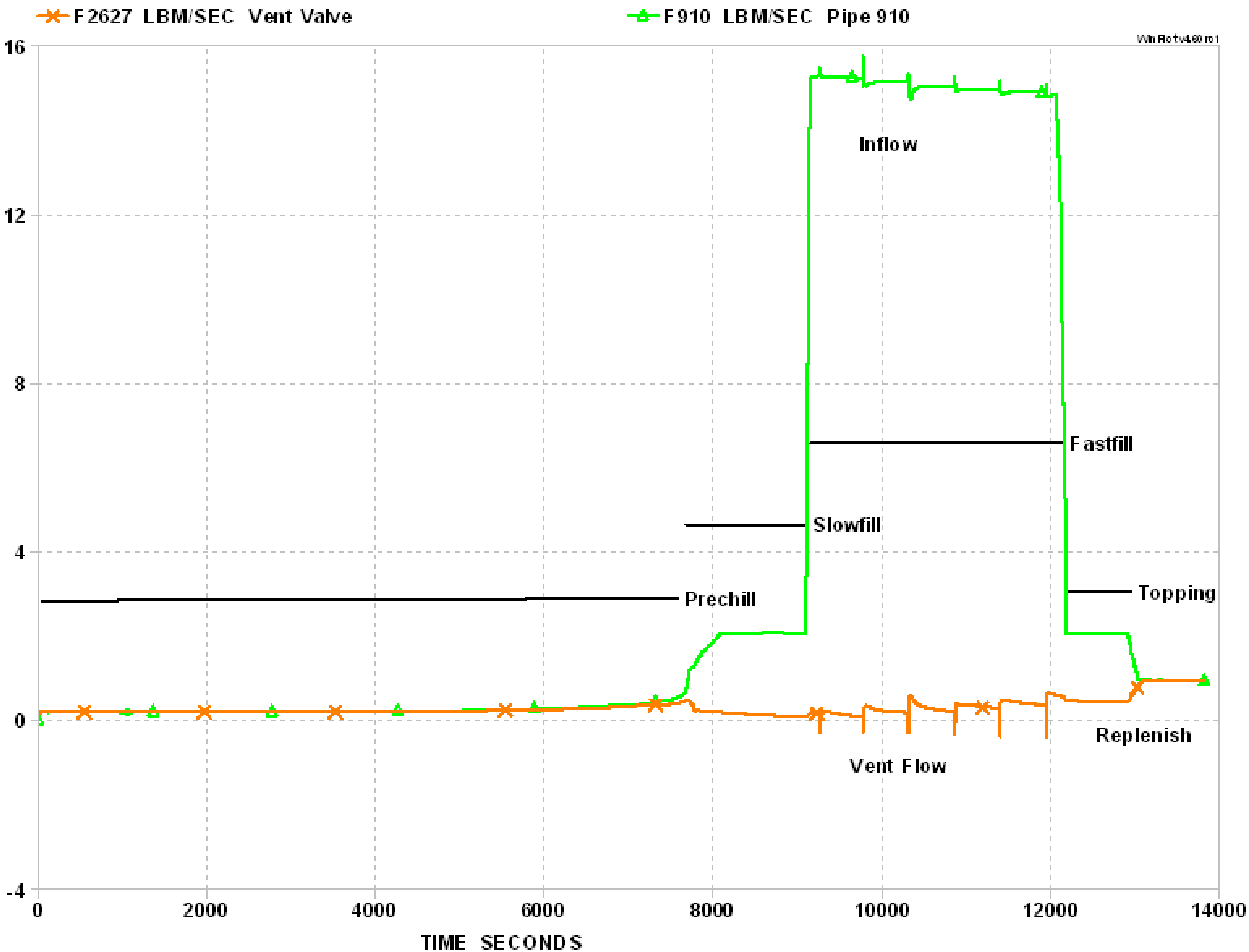
GFSSP v604 -- CHT Applications

LH2 Storage Tank Pressure	46.3 psia
Ambient Temperature	85 ° F
LH2 Propellant Load	48593 lb
Pre-Chill Valve C_v	16
Slow Fill & Topping Valve C_v	12
Fast Fill Valve C_v	140
Replenish Valve C_v	5.64
Vent Valve Area	20.94 in^2
Vent Valve C_d	0.552
Ground System Pipe Length and Volume	1910 ft / 879 ft^3
Flare Stack Pipe Length and Volume	1305 ft /1605 ft^3
Tank Volume	11,620 ft^3
Ground System Pipe Mass	29314 lb
Tank Mass	8742 lb
Foam Mass	673 lb
Metal (Al-Li) thickness	0.1934 in
Foam (BX-265) thickness (Tank Barrel)	1 in
Foam (BX-265) thickness (Dome)	0.5 in
Common Bulkhead Conductance	0.045 Btu/hr-ft^2-F

Summary Result for LH2 Loading

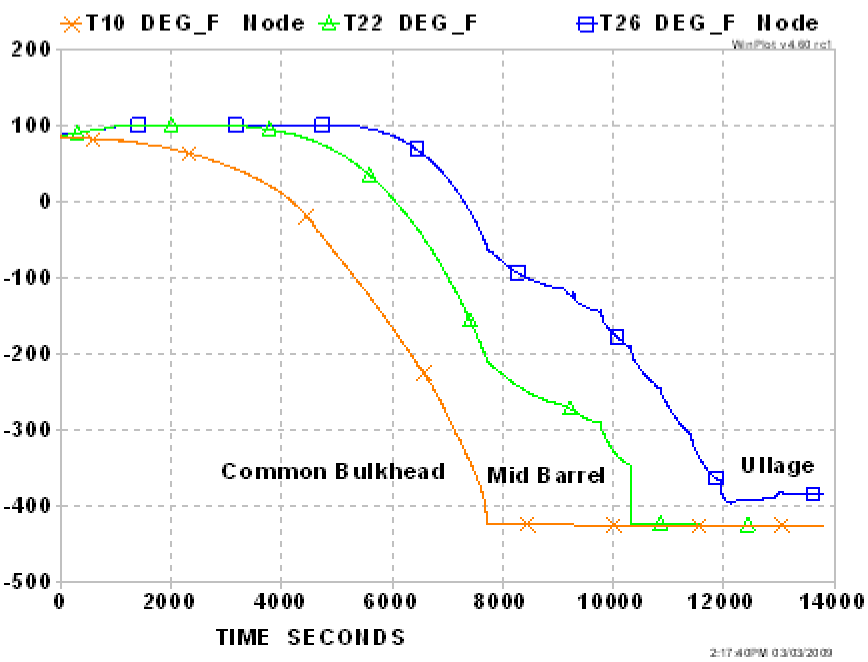
Pre-chill Time (after start)	129 Minutes
5% Tank Fill Time (after pre-chill)	23 Minutes
95% Tank Fill Time (after pre-chill)	73 Minutes
100% Tank Fill Time (after pre-chill)	87 Minutes
Tank Chill-down Time (after start)	194 Minutes
Maximum Tank Pressure (pre-chill)	15.94 psia
Maximum Ullage Pressure (Replenish)	14.89 psia
Maximum Vent Flowrate	0.94 lb/sec
Amount of GH2 Vented	3993lb
Minimum Foam Surface Temperature	6.2 F

Tank Inflow rate and Vent flow rate

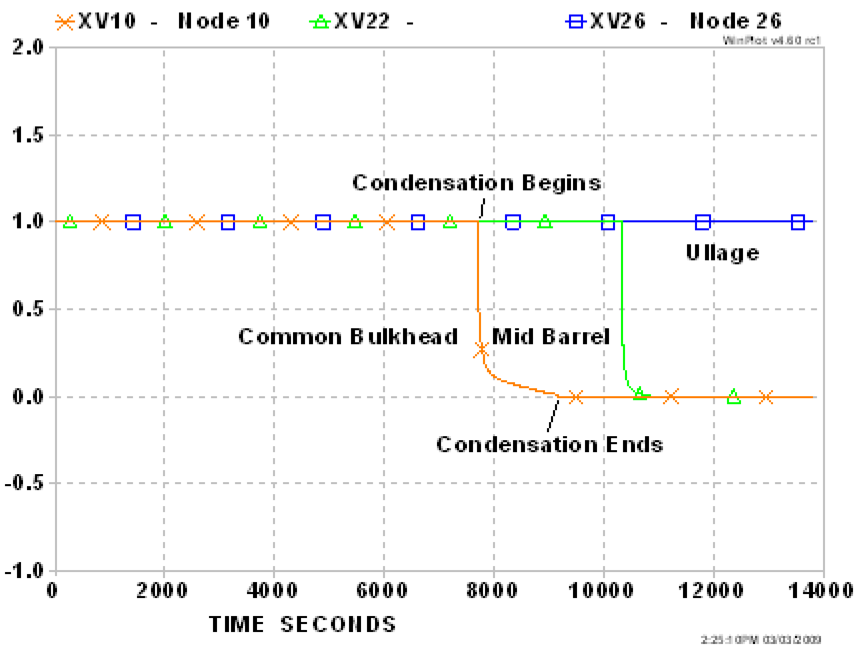


Propellant Temperature and Quality in LH2 Tank

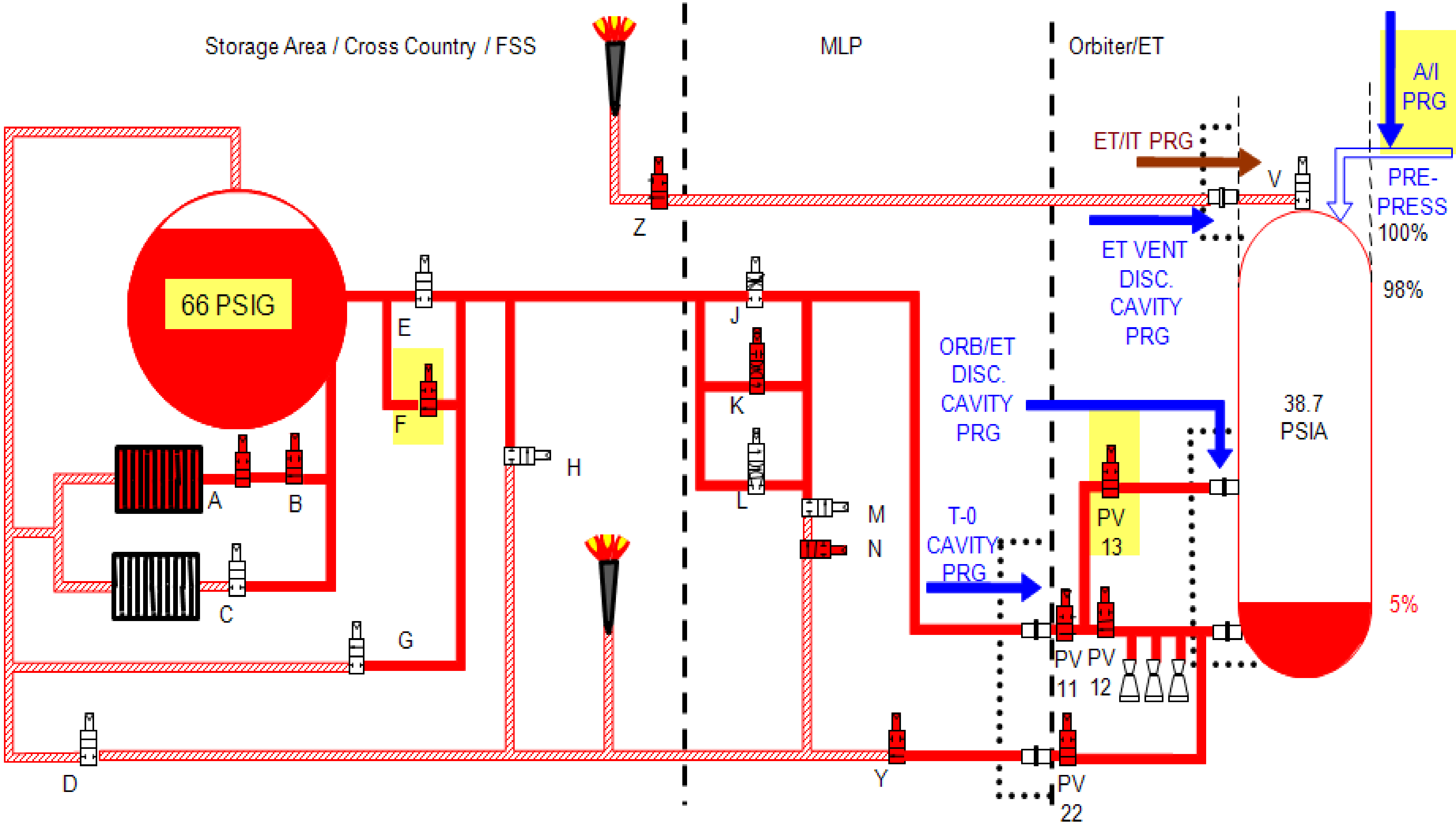
Hydrogen Temperature



Quality (Vapor Fraction)



Shuttle ET LH_2 Propellant Loading



Shuttle ET LH_2 Propellant Loading

Loading Phase	Start Time (Approx.)	Flowrate (lb_m/s)
Transfer Line Chill	T-7h55m	≈1
Pressurize Storage Tank and ET	T-7h51m	10
Slow Fill to 5%	T-7h42m	10
Fast Fill to 72%	T-7h5m	73
Fast Fill to 85%	T-6h39m	52
Reduced Fast Fill to 98%	T-6h18m	10
Topping and Replenish (not modeled)	T-5h54m	≈1

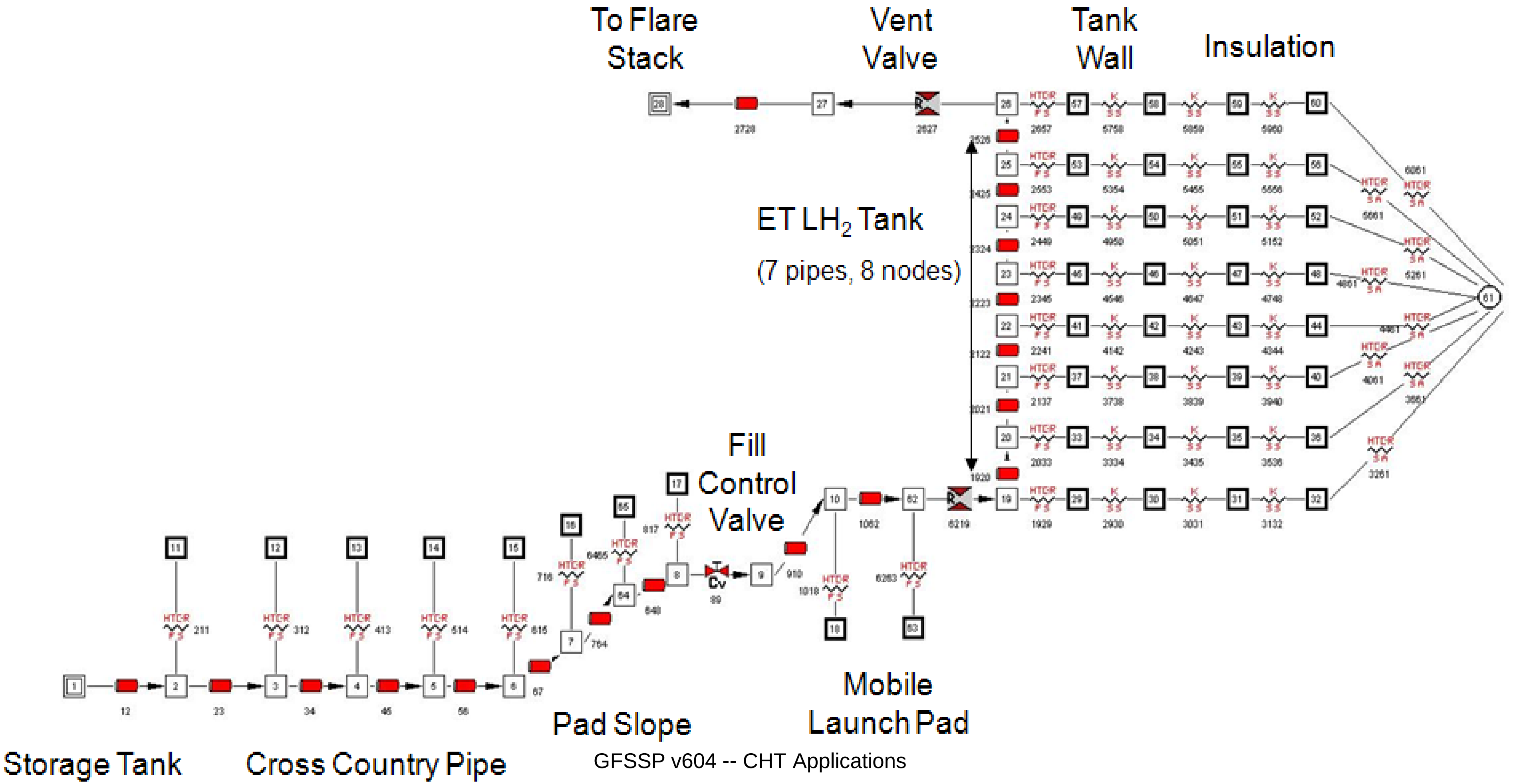
KSC LH_2 Facility Properties

- **Cross-country pipeline**
  - ¼ mile of 10” Invar pipe, vacuum-jacketed
  - 26,400 lb_m
  - Δz = 79’
- **Mobile launch platform**
  - 334’ of 8” and 10” stainless steel pipe, vacuum-jacketed
  - 6100 lb_m
  - Δz = 43’

ET LH_2 Tank Properties

- **Tank mass: 23,600 lb_m**
- **LH_2 mass: 227,600 lb_m**
- **Length: 97 ft**
- **Diam: 27.6 ft**
- **Insulation: 2078 lb_m**
  - ≈ 1.0 in NCFI on barrel and aft dome
  - ≈ 0.75 in BX-265 on forward dome
- **Surface area: 8550 ft^2**
- **Vent: $C_dA = f(\Delta P) \approx 18\ in^2$**
  - Open during facility line chilldown
  - Cycles open and closed during slow/fast fill to maintain 24-27 psig

ET LH_2 GFSSP Model

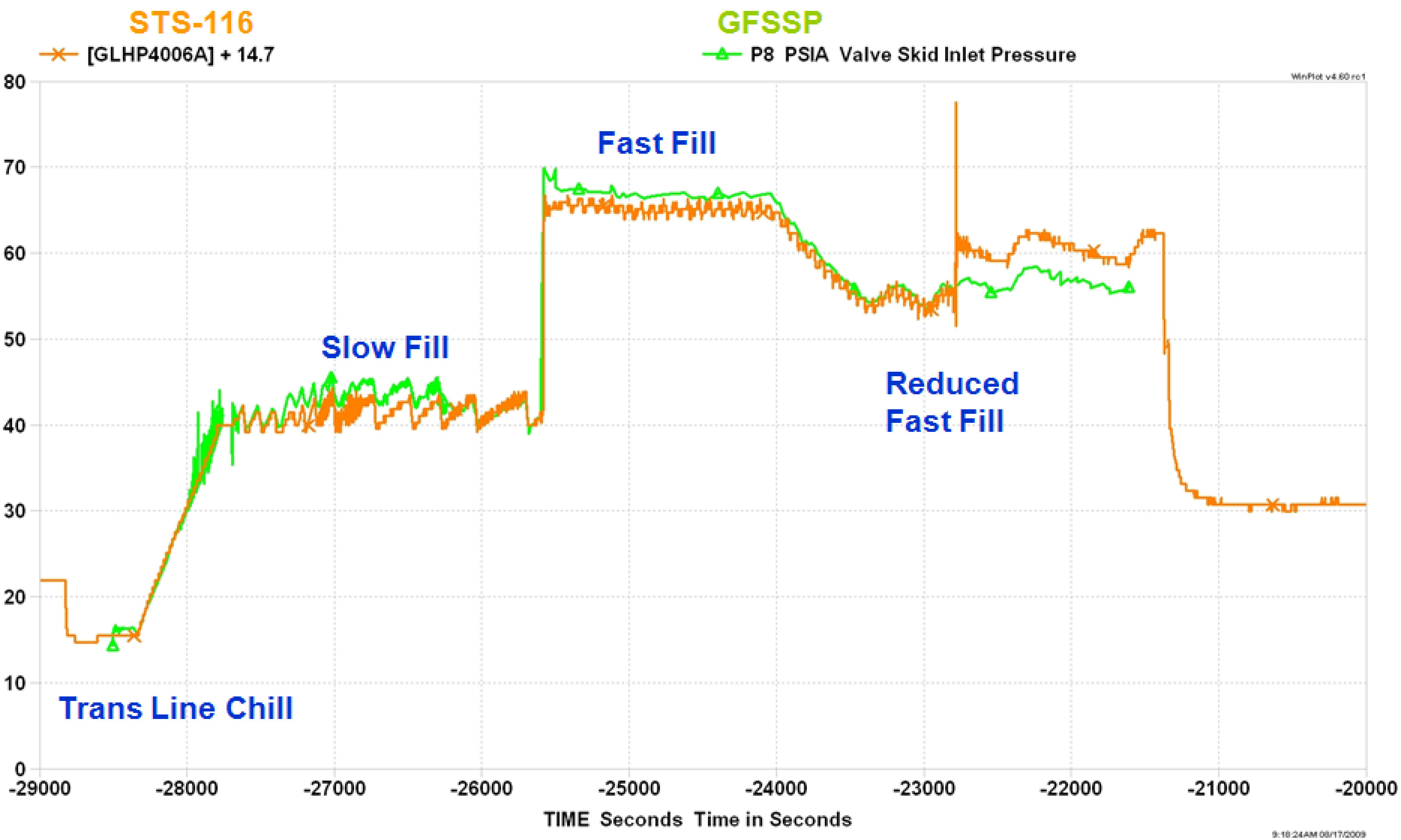


Comparison

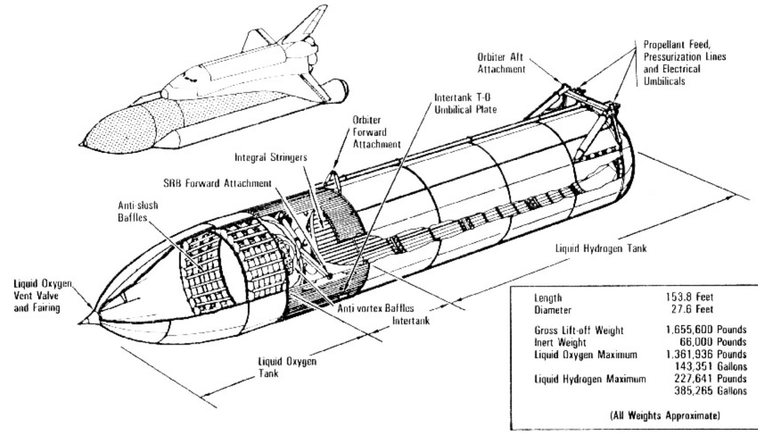
	STS-116	GFSSP
5% Full	48 min (T-7h7m)	50 min (T-7h5m)
98% Full	119 min (T-5h56m)	116 min (T-5h59m)
Tank chilled (to -420 F)	N/A	106 min (T-6h9m)
H_2 Vented During Loading	N/A	4931 lb_m
Heat Leak (through tank walls)	* 68 - 140 BTU/s	96 BTU/s

* Not measured. Estimate from ET System Definition Handbook.

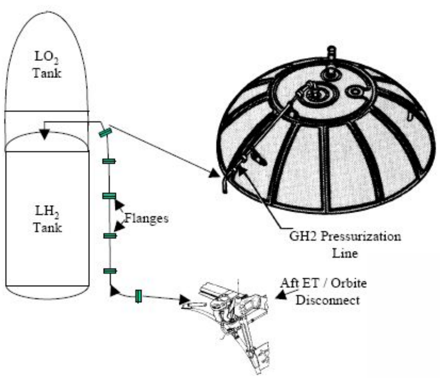
Pressure at Valve Skid

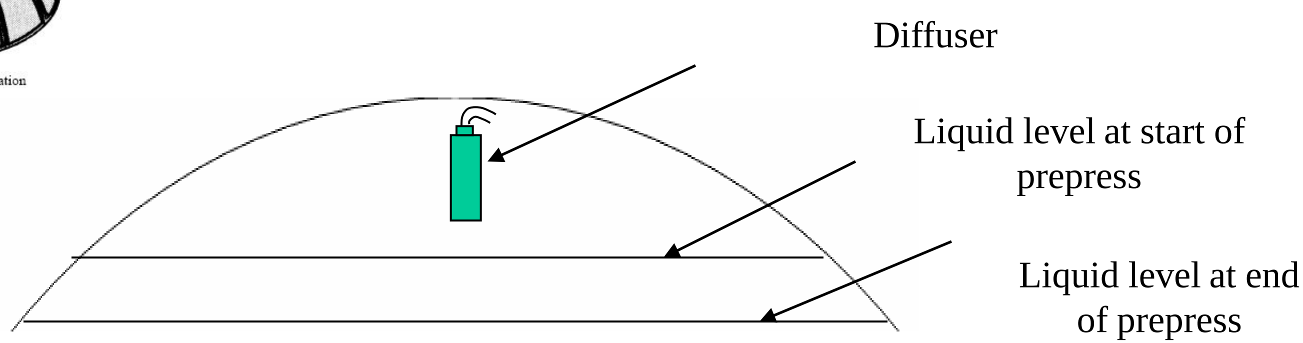


Pressurization of Space Shuttle's LH2 Tank

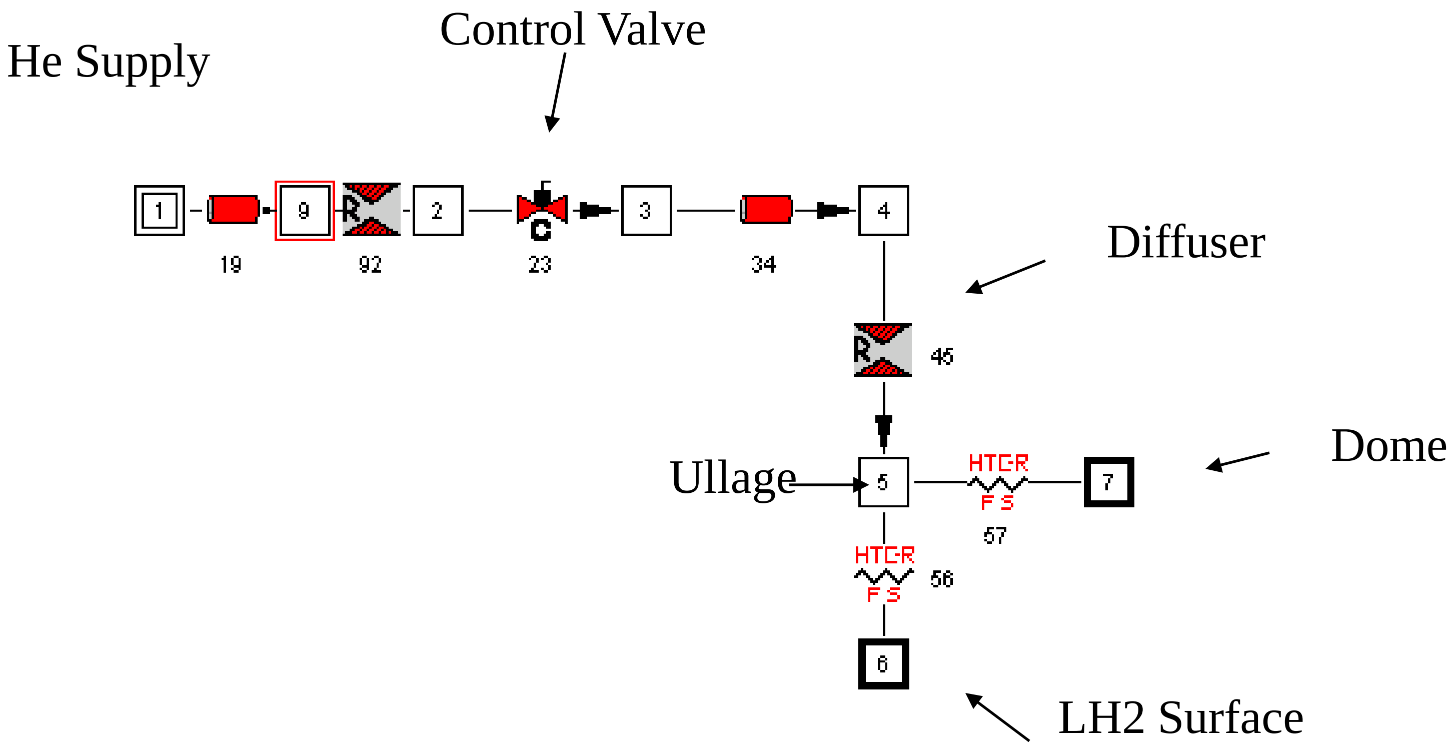


Lightweight External Tank

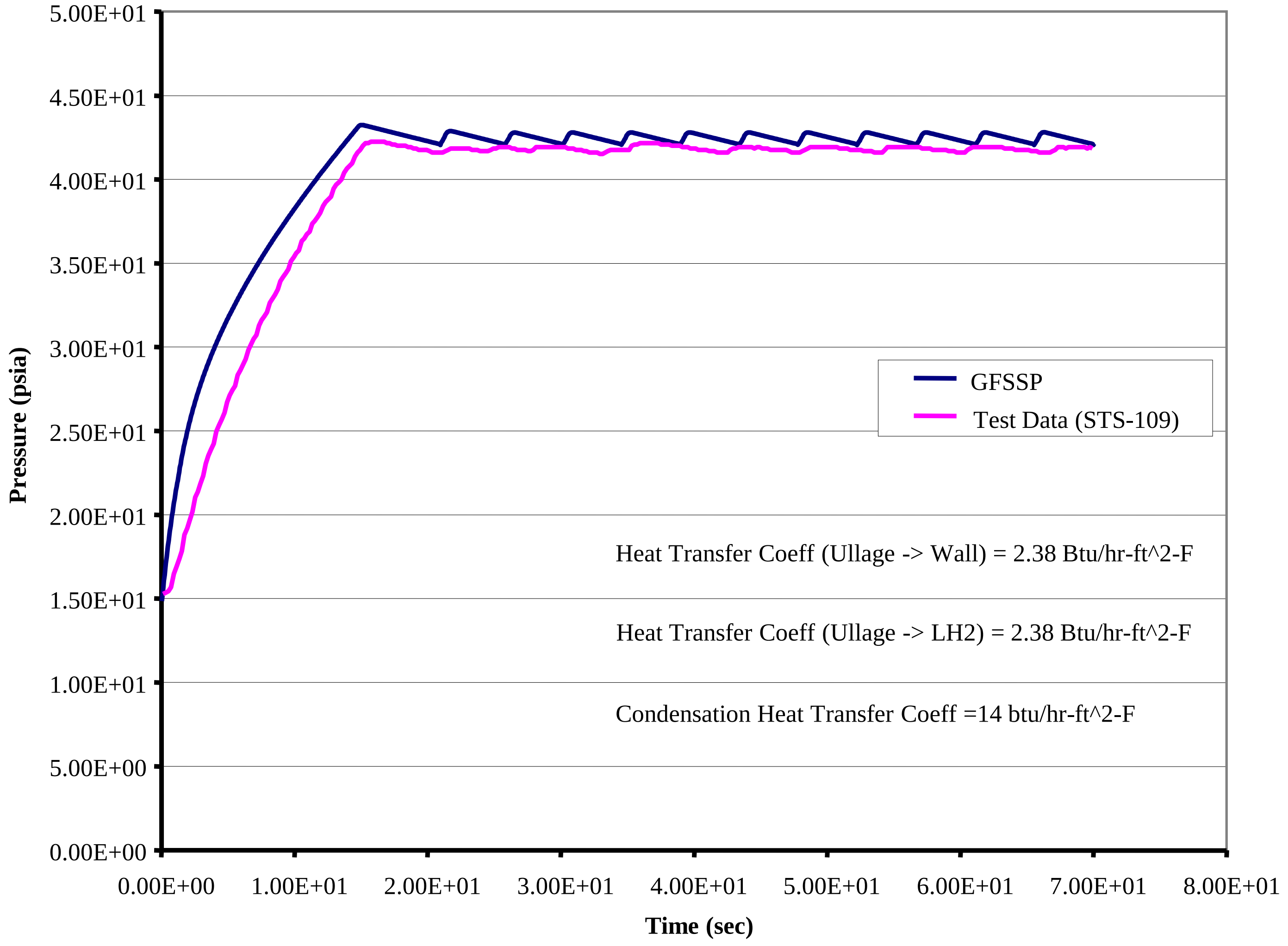




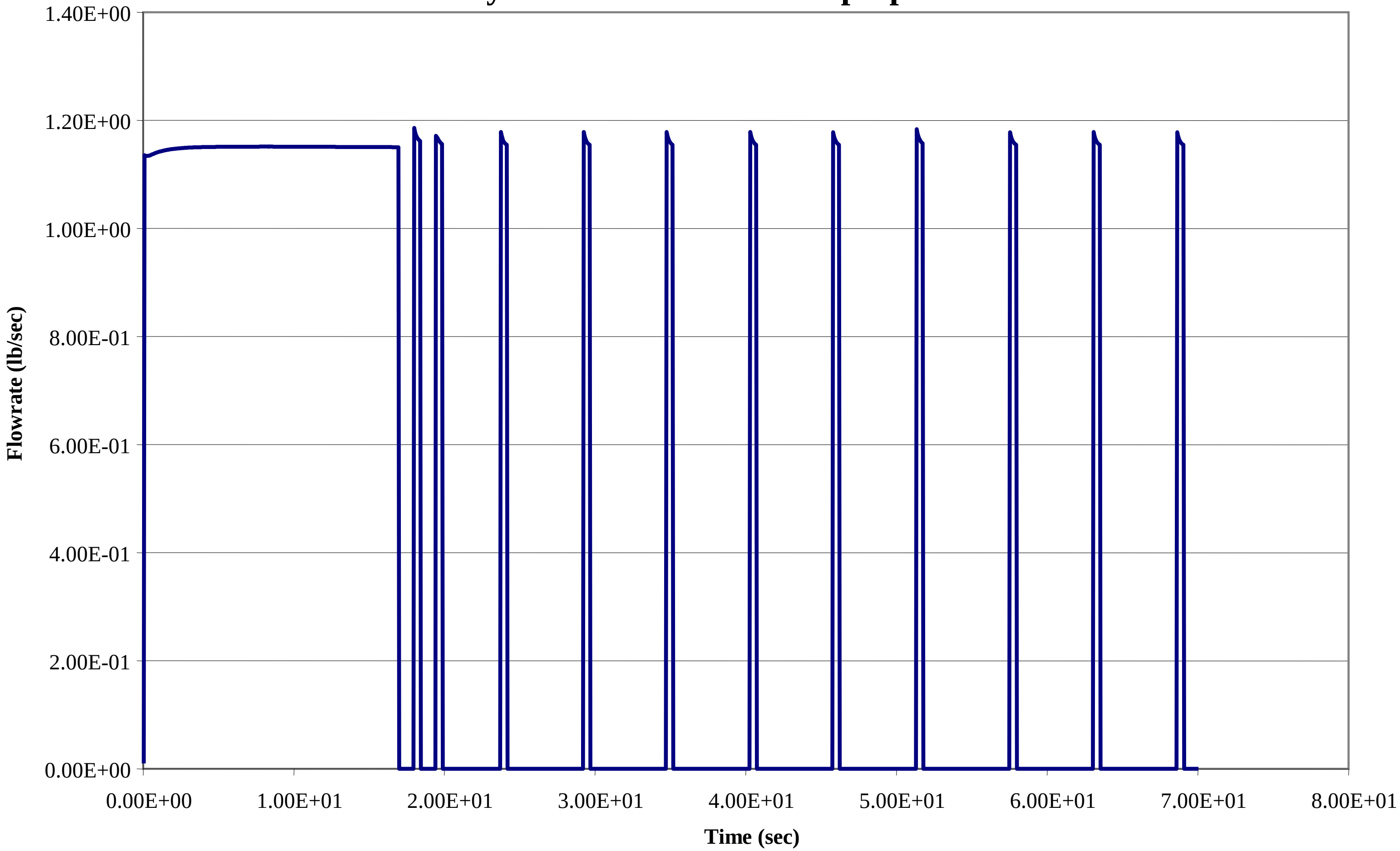
GFSSP Model of ET Hydrogen Tank Pressurization



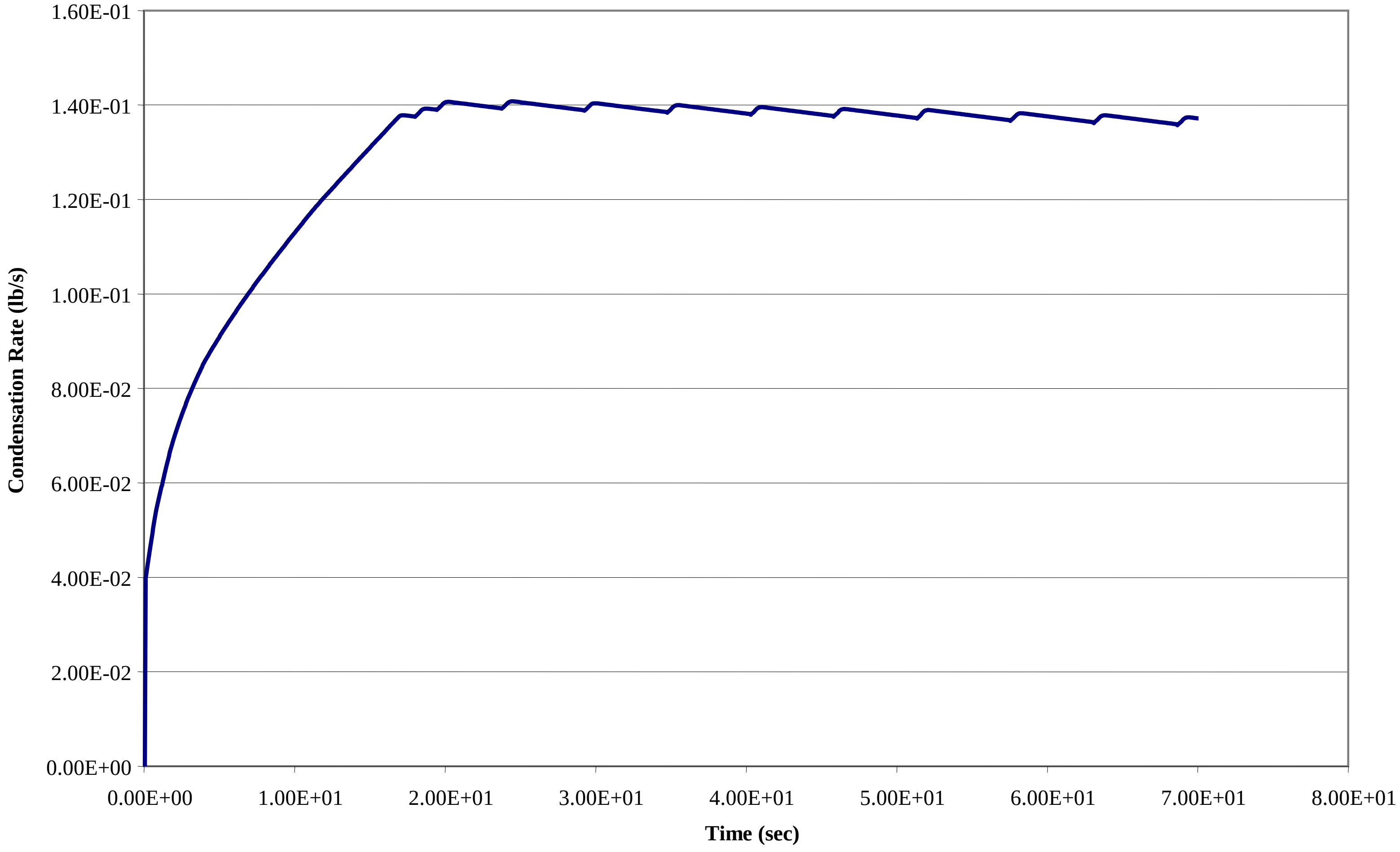
Ullage Pressure History in LH2 Tank (STS-109)



Helium flowrate history in to ullage for base case
9 cycles in 45 seconds after prepress



Condensation Rate of Hydrogen during Pressurization

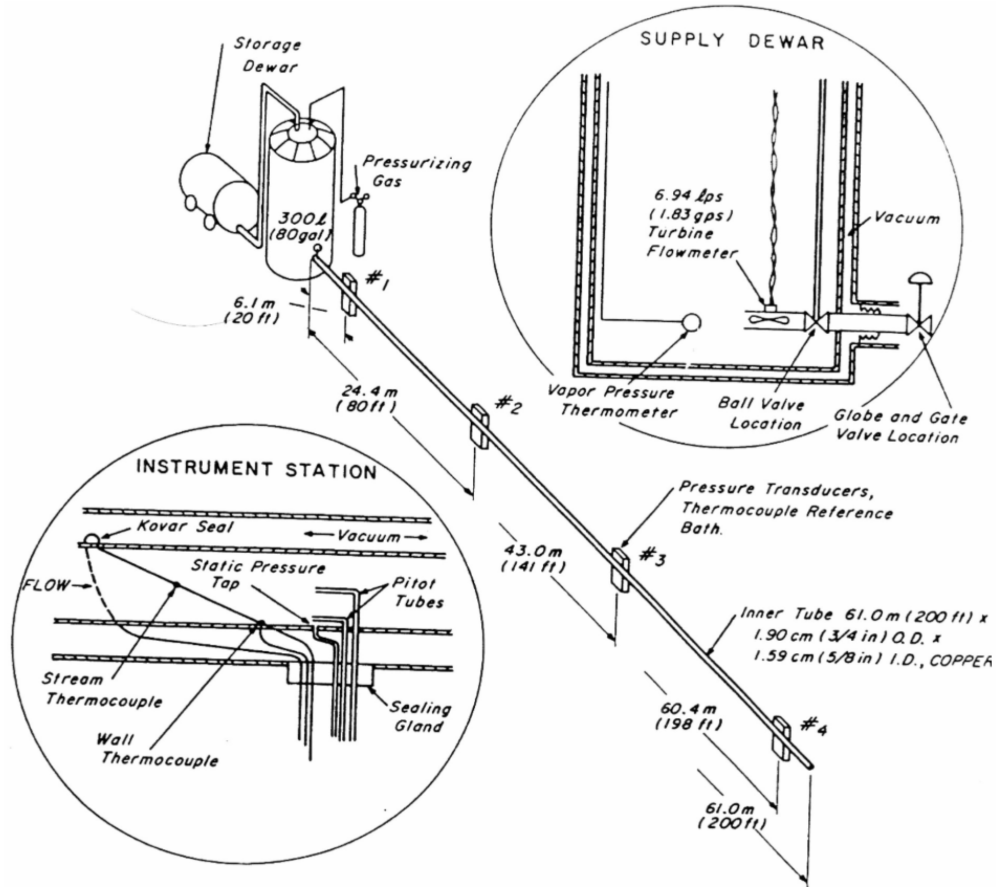


Summary

- GFSSP Version 6 allows Users to model Conjugate Heat Transfer (CHT)
- Solid to Solid and Solid to Fluid Heat Transfer capability was added in the GFSSP framework
- GFSSP's Graphical User Interface VTASC allows user to construct, run, and view results for network consisting of fluid and solid nodes
- For Heat Transfer Coefficients Dittus-Boelter and Miropolosky Correlations (For Two Phase Flow) have been included
- Other correlations can be implemented through User Subroutine
- GFSSP's CHT capability has been validated by comparing with test data
- Examples 13 and 14 illustrate the use of Conjugate Heat Transfer applications

Tutorial – 5

CHILLDOWN OF CRYOGENIC TRANSFER LINE

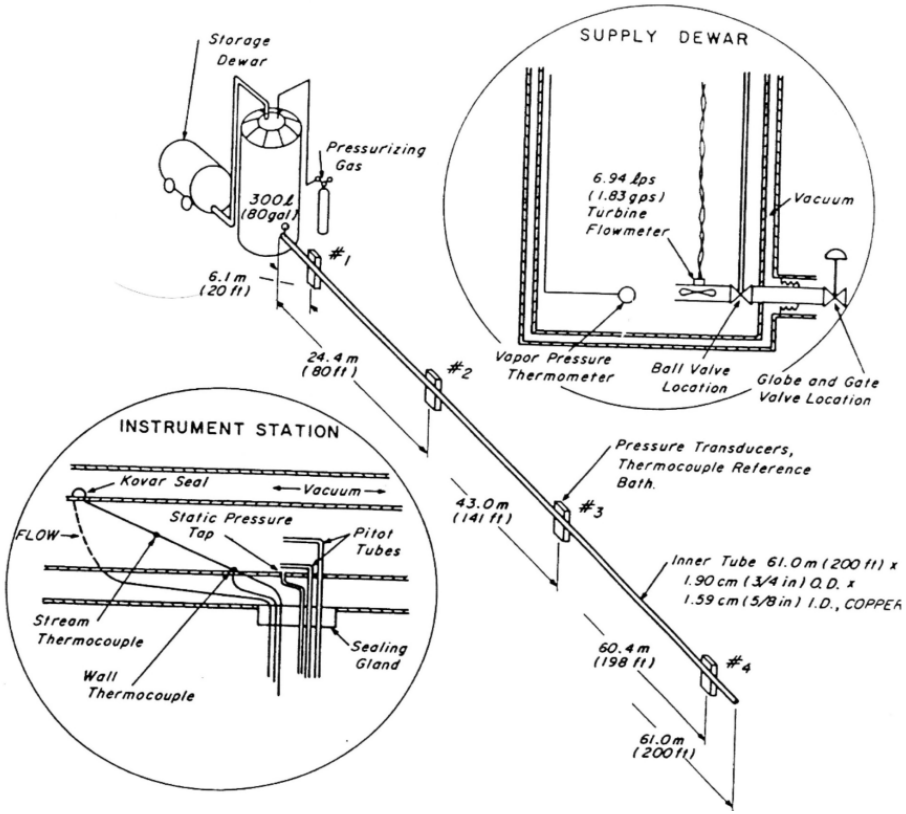


GFSSP 6.04 -- Tutorial 5

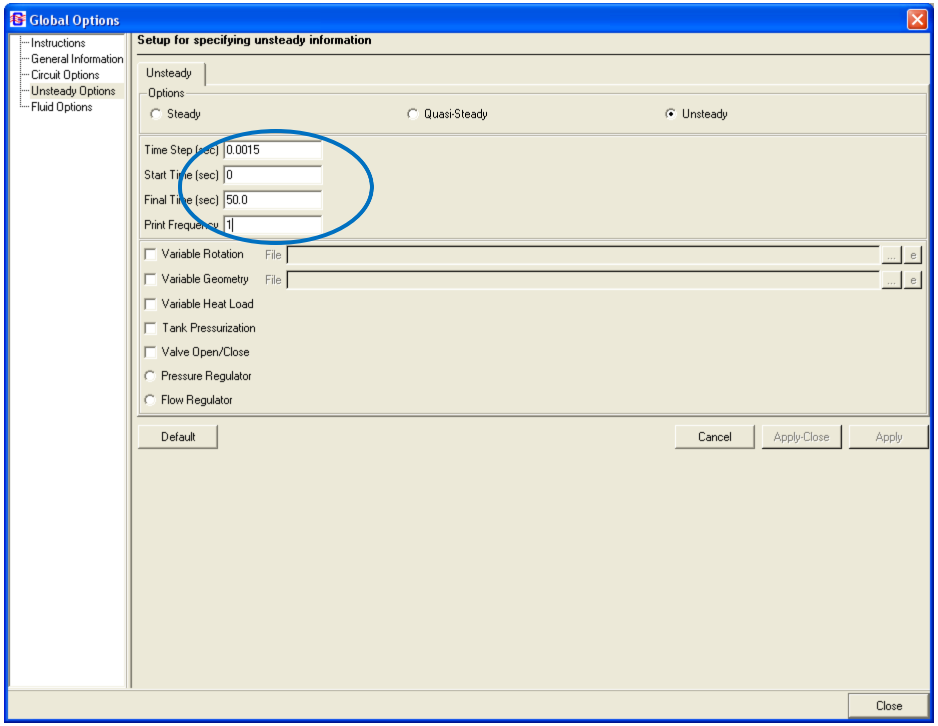
CHILLDOWN OF TRANSFER LINE SCHEMATIC

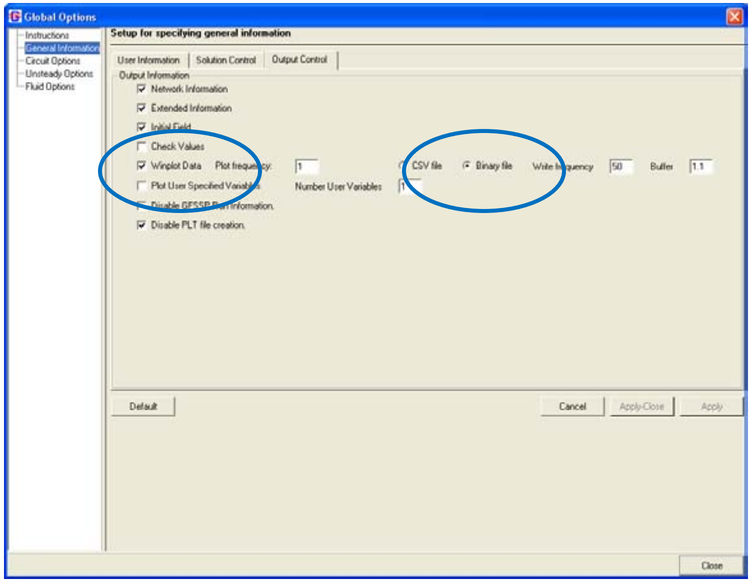
Problem Considered:

•Time dependent Pressure, Temperature and Flow Rate history during chilldown



- Select Advanced/Enable Conjugate Heat Transfer
- User Information
    - Input File: Tut5.dat
    - Output File: Tut5.out
- Unsteady Options
    - Time Step: 0.0015 s
    - Final Time: 50.0 s

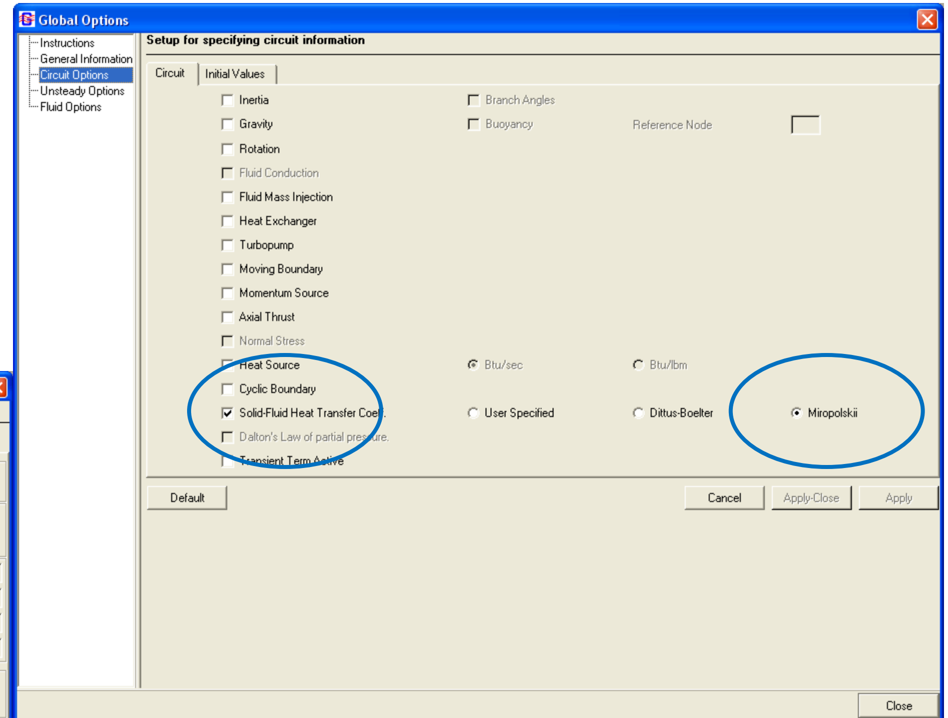


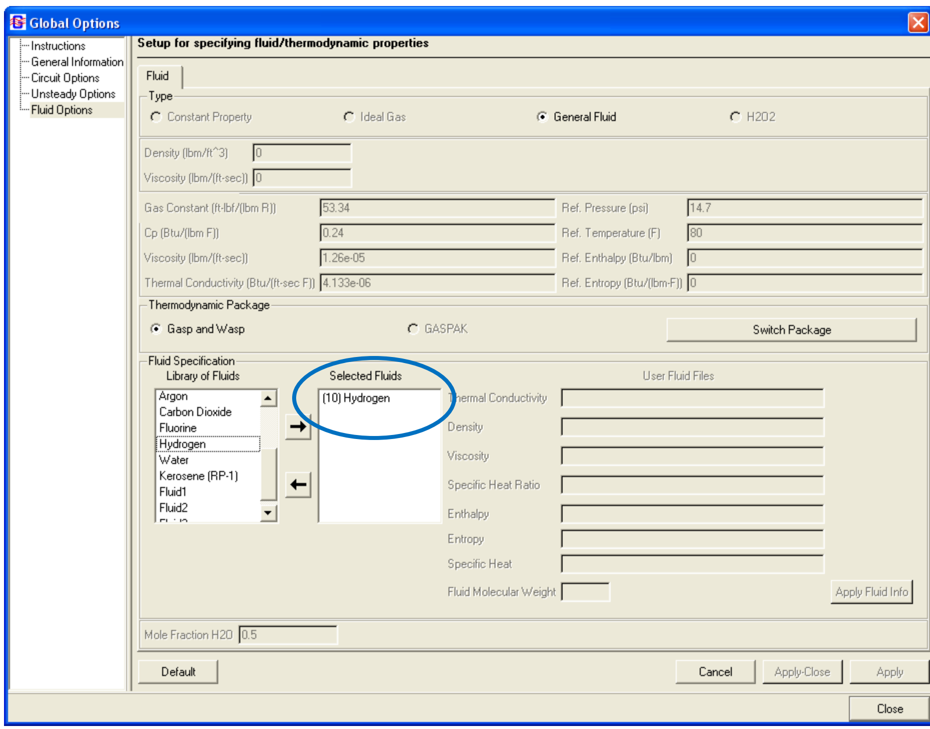


- Output Control
    - Winplot binary output

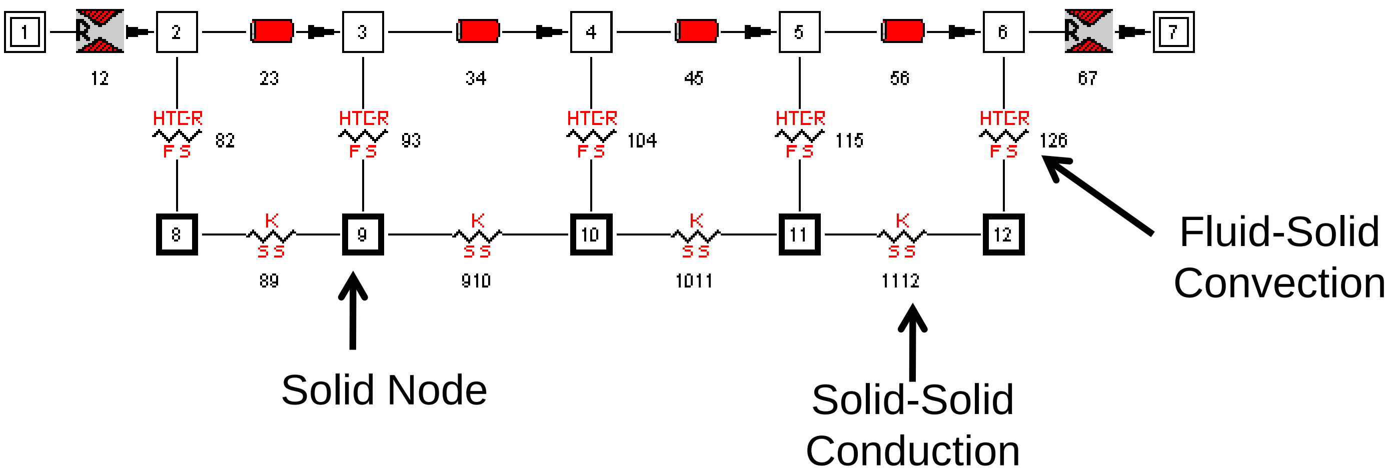
Set Up Options (cont.)

- Circuit Options
  - Solid-Fluid Heat Transfer Coeff.
  - Miropolski Correlation
- Fluid Options
  - Hydrogen





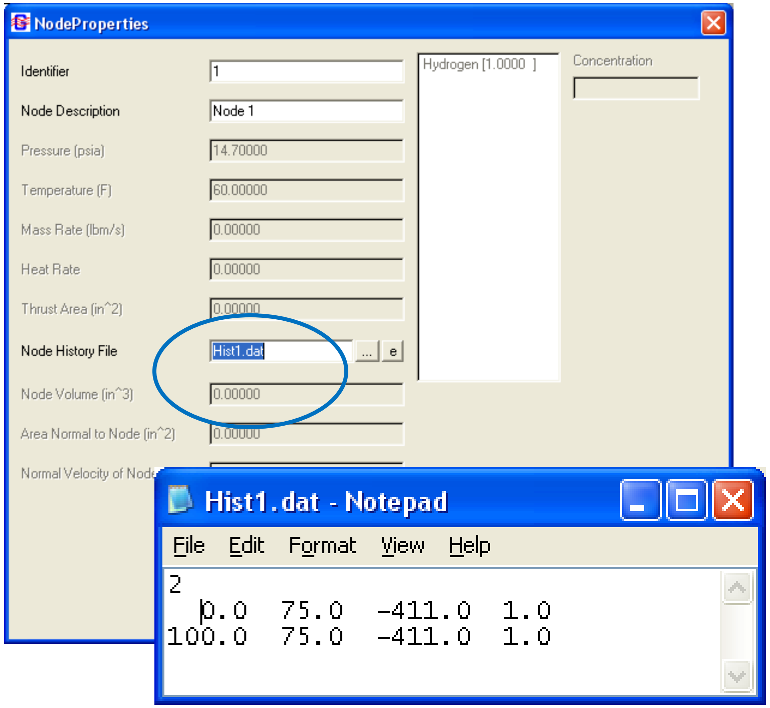
Build Model on Canvas

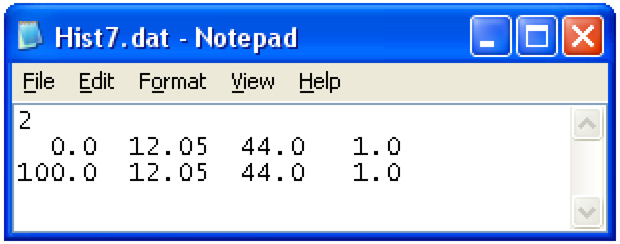


Now is a good time to save your Tut5.vts file.

Set Up Transient Boundary Conditions

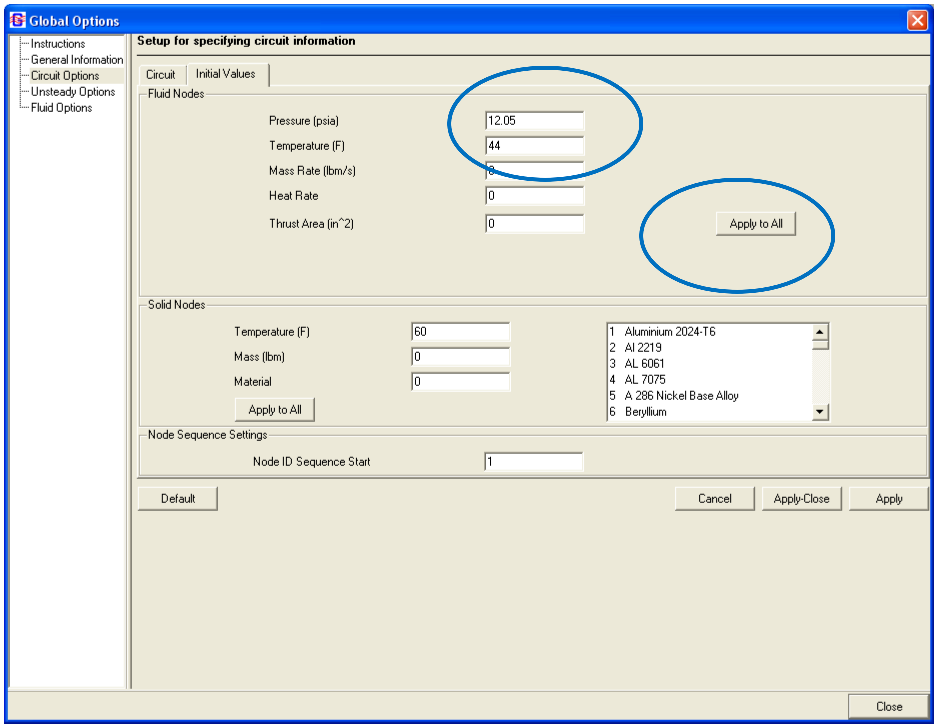
- Node 1: Inlet from Dewar
  - P = 75 psia
  - T = -411 °F
- Node 7: Outlet to Ambient (Boulder, CO)
  - P = 12.05 psia
  - T = 44 °F

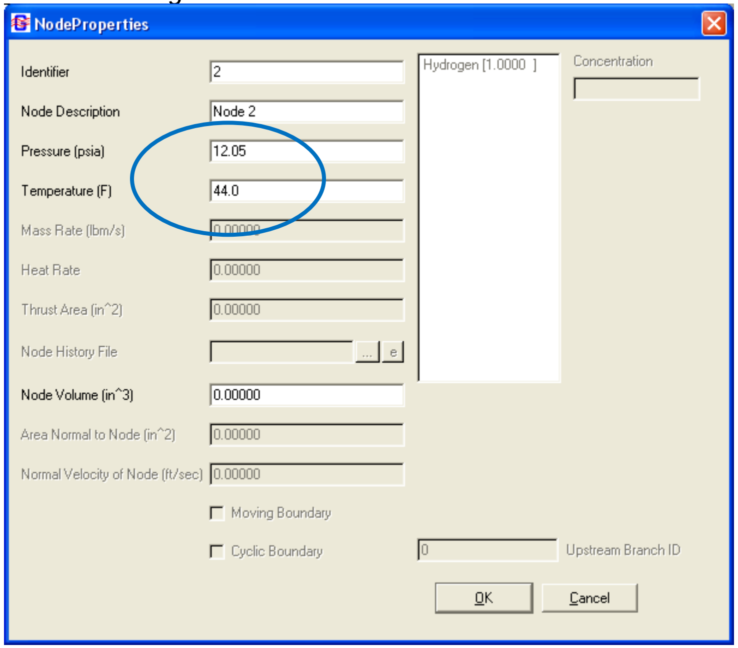




Set Up Interior Node Initial Conditions

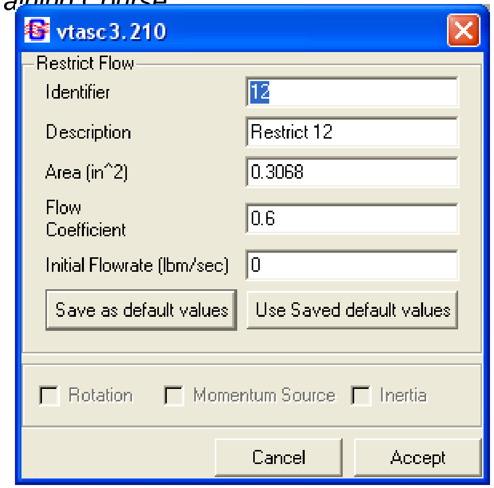
- Nodes 2-6
  - P = 12.05 psia
  - T = 44 °F
  - Volume not required – GFSSP will calculate from pipe dimensions
- Shortcut method to set all initial conditions:

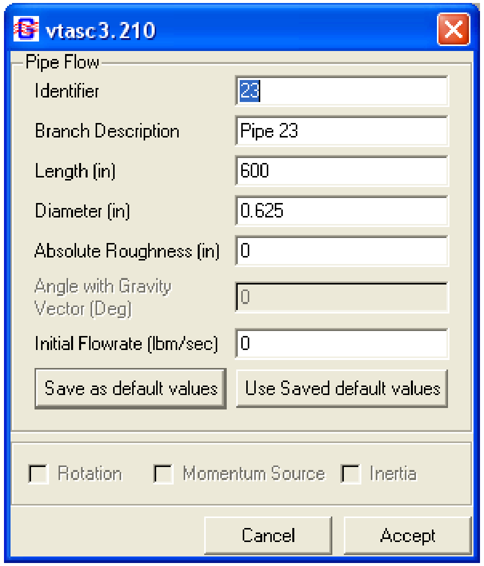




Set Up Fluid Branches

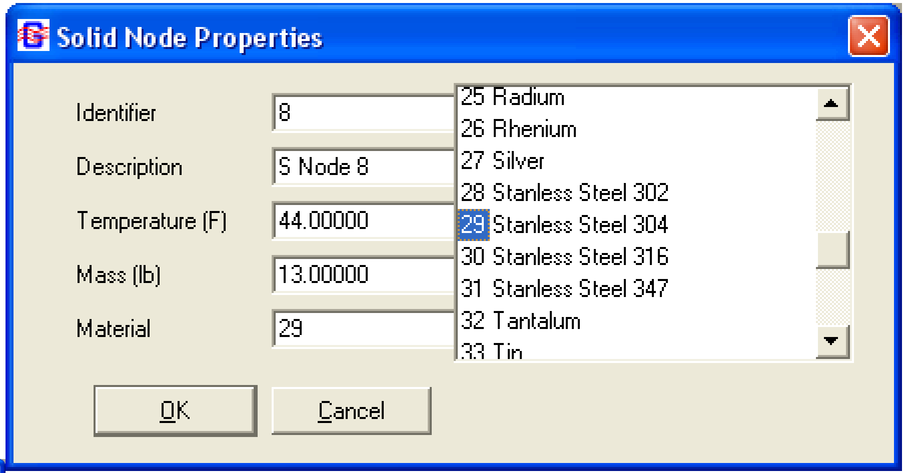
- Branch 12: Inlet valve
  - $A = 0.3068$ in^2, $C_L = 0.6$
- Branch 67: Exit
  - $A = 0.3068$ in^2, $C_L = 1.0$
- Branch 23, 34, 45, 56: Pipes
  - L = 200 ft / 4 = 50 ft = 600 in
  - D = 0.625 in
  - Smooth pipe: $\varepsilon = 0$

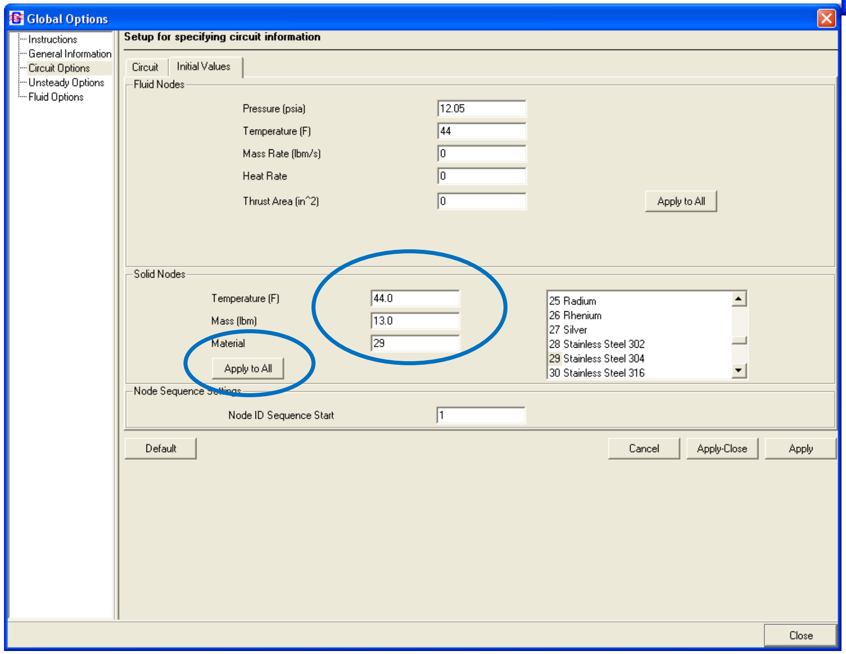




Set Up Solid Nodes

- Pipe is 65 lb_m of SS304
- Nodes 8 – 12
    - Initial T = 44 °F
    - Mass = 65 lb_m / 5 = 13 lb_m
    - Material 29
- Shortcut method for setting all solid node initial conditions:





Set Up Conductors

- Fluid-Solid Convection
  - Total Wetted Area:

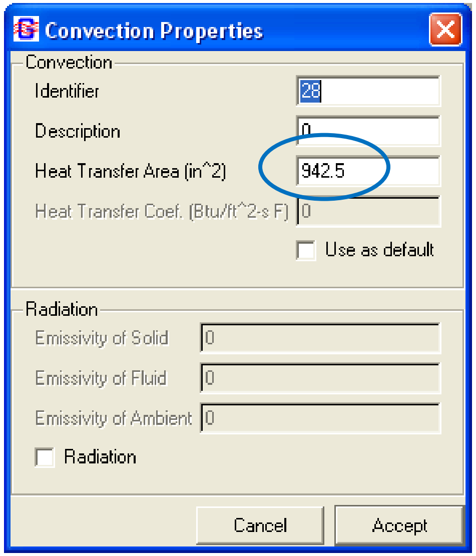
$$A = \pi DL = \pi(0.625\ in)(2400\ in) = 4712\ in^2$$

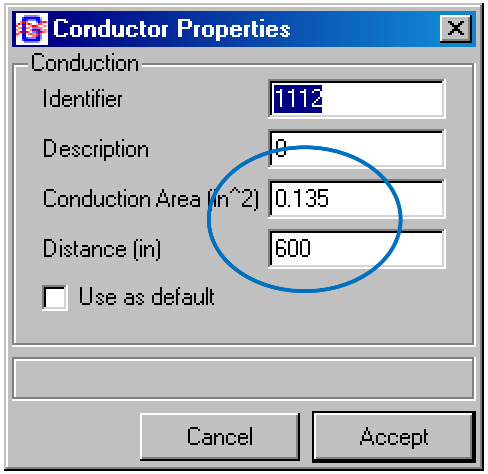
  - Per convector: 942.5 in^2
- Solid-Solid Conduction
  - Cross-Sectional Area:

$$A = \frac{\pi}{4}\left(OD^2 - ID^2\right)$$

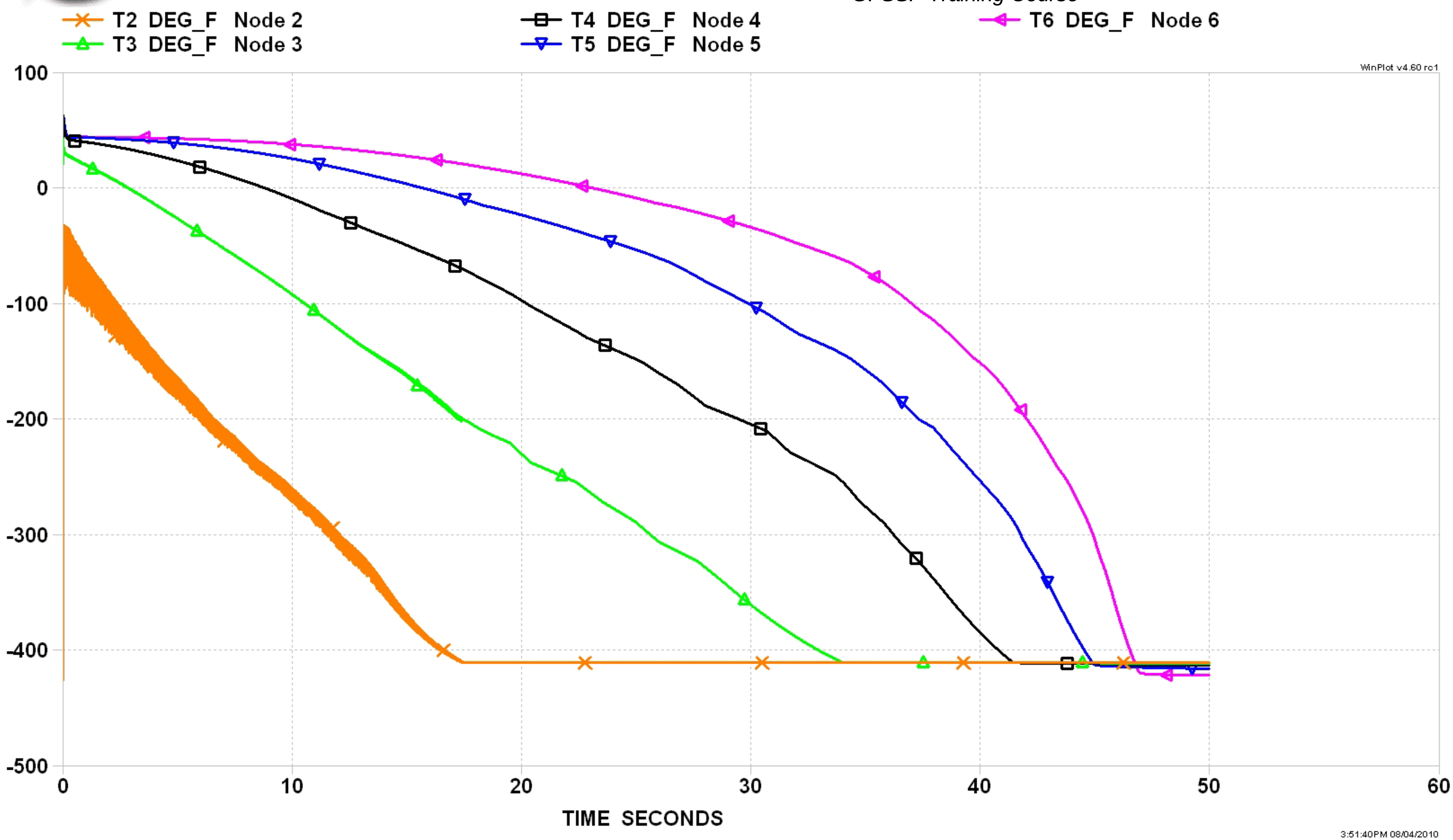
$$= \frac{\pi}{4}\left[(0.75\ in)^2 - (0.625\ in)^2\right] = 0.135\ in^2$$

  - Length per conductor: 50 ft = 600 in



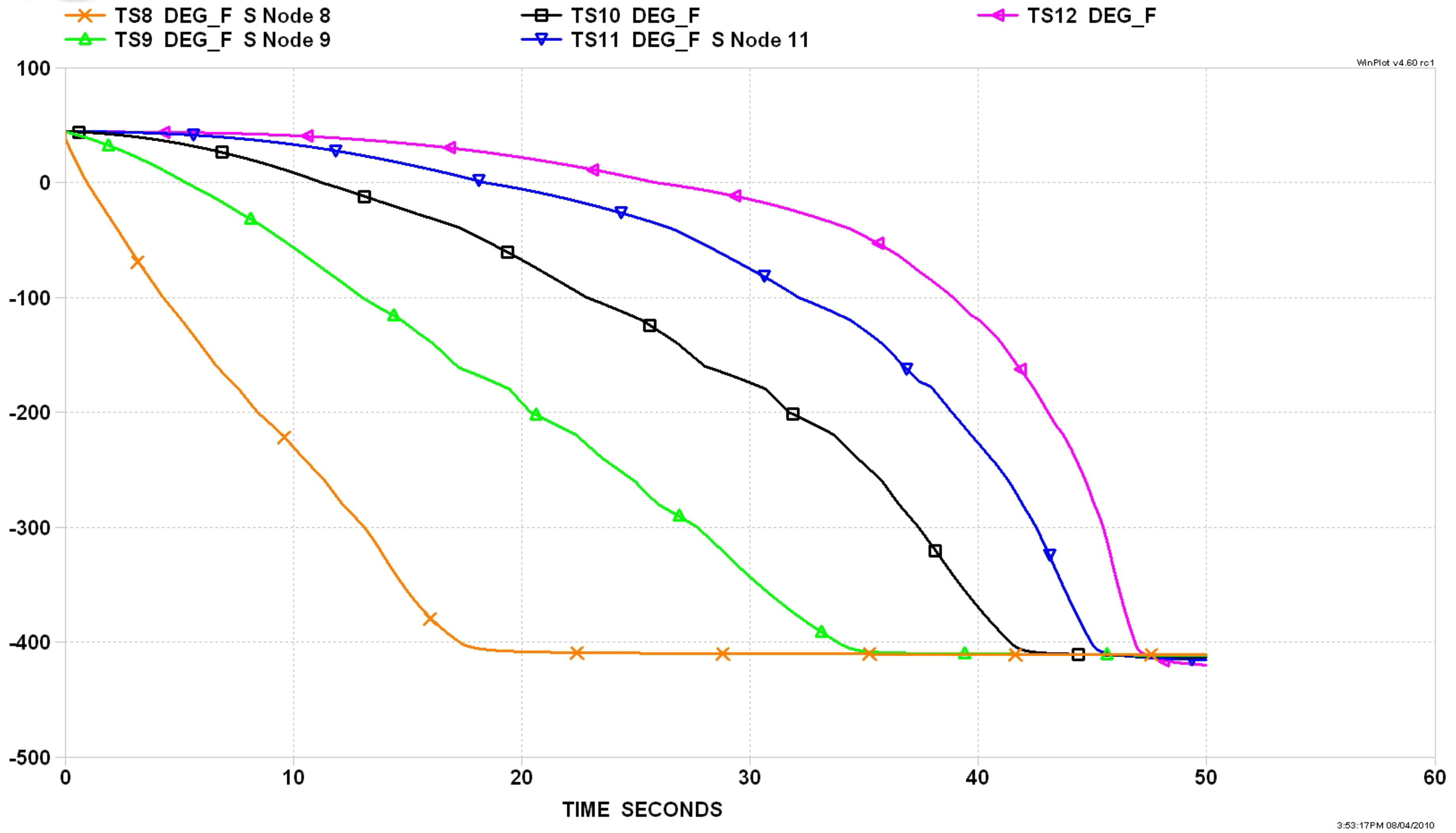


Fluid Temperature



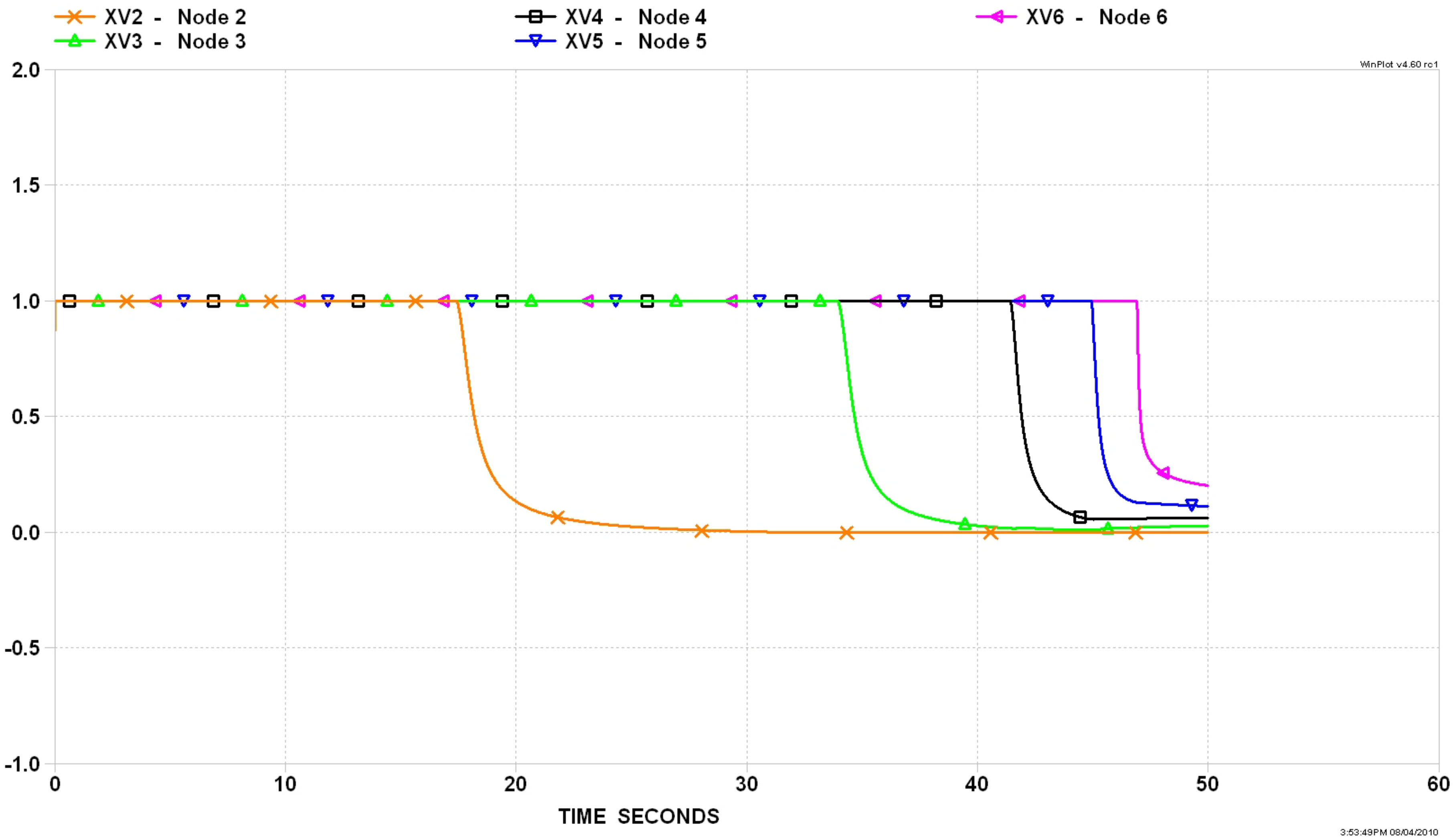
Solid Temperature

Marshall Space Flight Center
GFSSP Training Course



Quality (Vapor Mass Fraction)

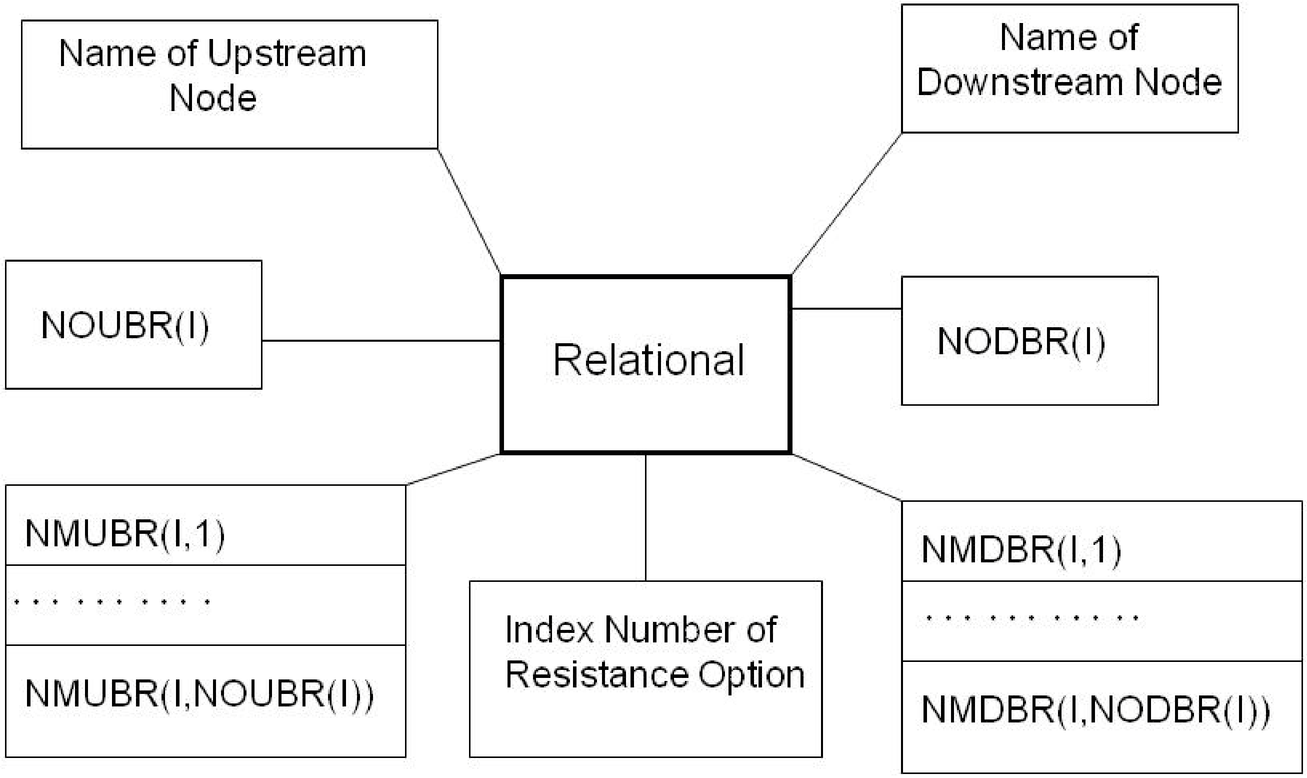
Marshall Space Flight Center
GFSSP Training Course



STUDY OF THE RESULTS

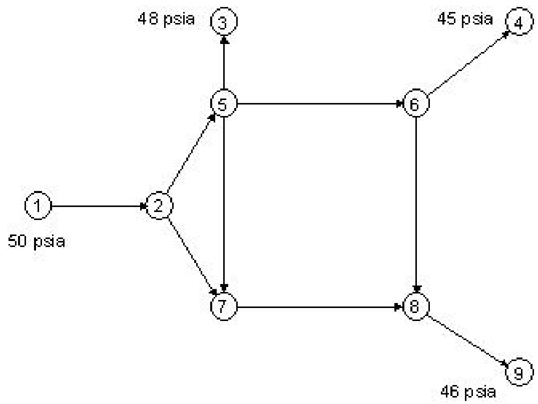
- Plot pressure, flowrate and fluid and solid temperature history
- Estimate the predicted chilldown time
- Observe the phase change behavior

DATA STRUCTURE

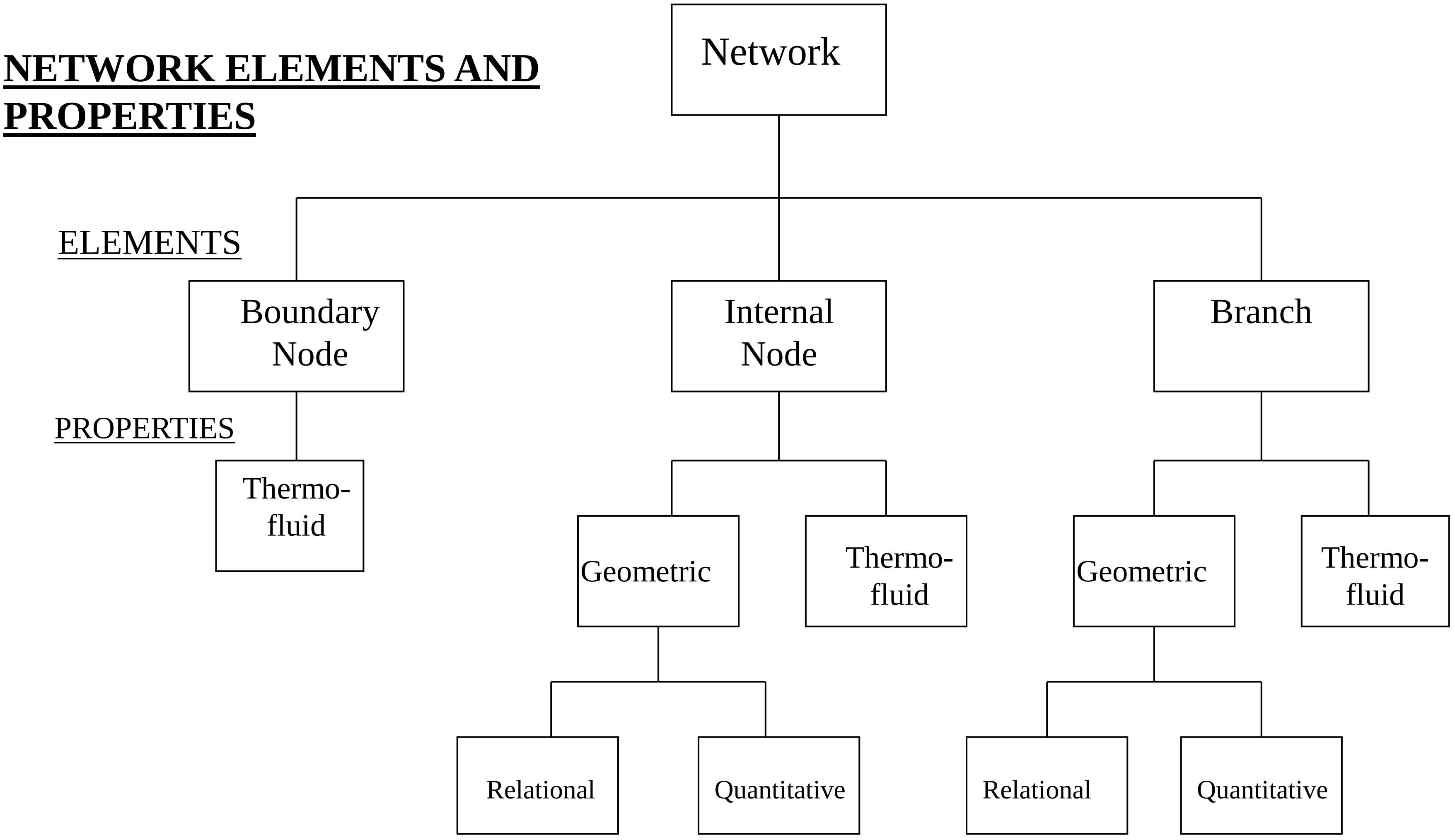


Importance of Data Structure in Network Flow Analysis

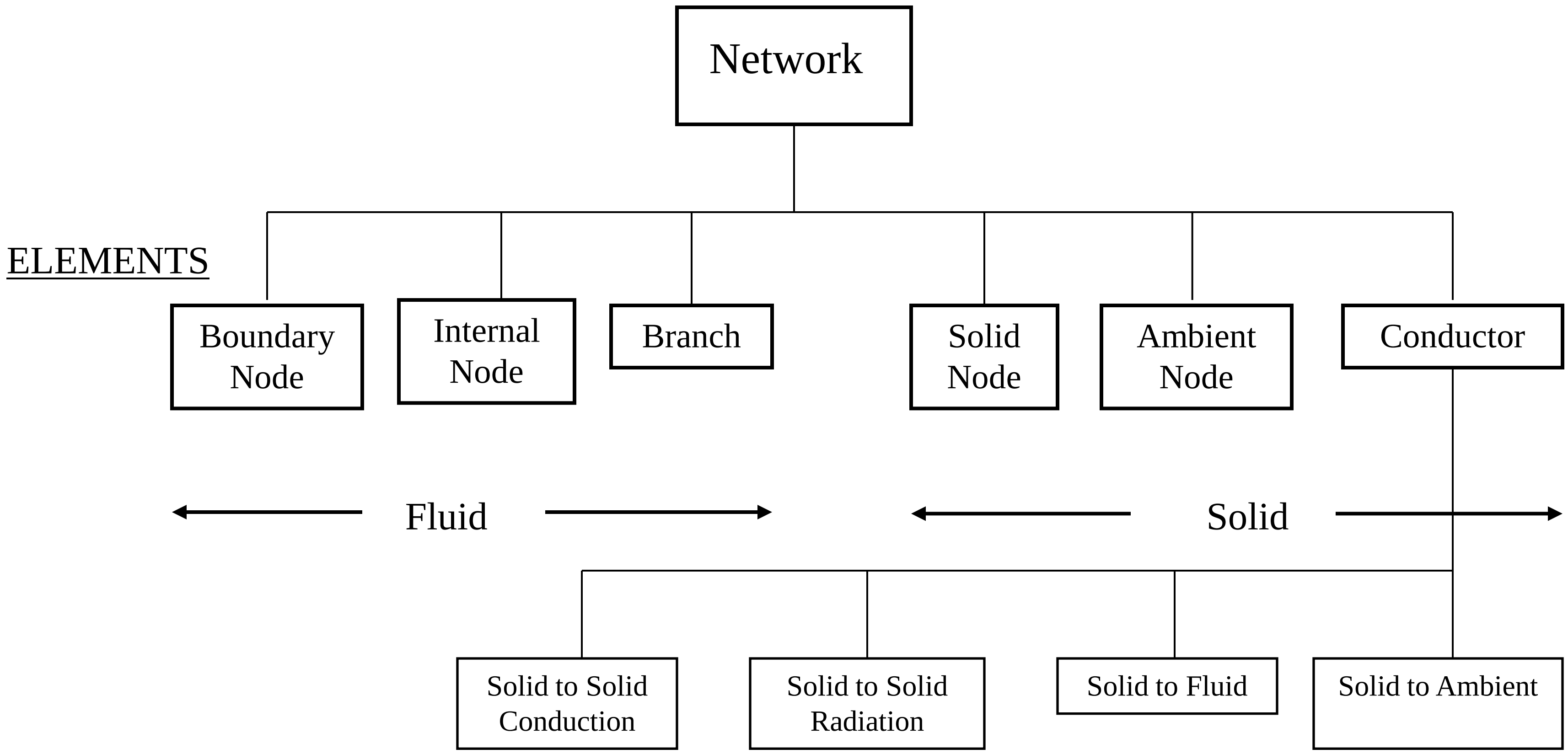
- In a flow network layout of nodes are not structured
  - There is no origin and co-ordinate direction to build the array of nodes
- In a structured system, array of nodes can be constructed in different co-ordinate direction
  - In one-dimensional flow network, each node has two neighbors
  - In two-dimensional flow network, each node has four neighbors
  - In three-dimensional flow network, each node has six neighbors
- In a typical flow network a node can have “n” number of neighbors
- Therefore, a unique data structure is needed to define a flow network



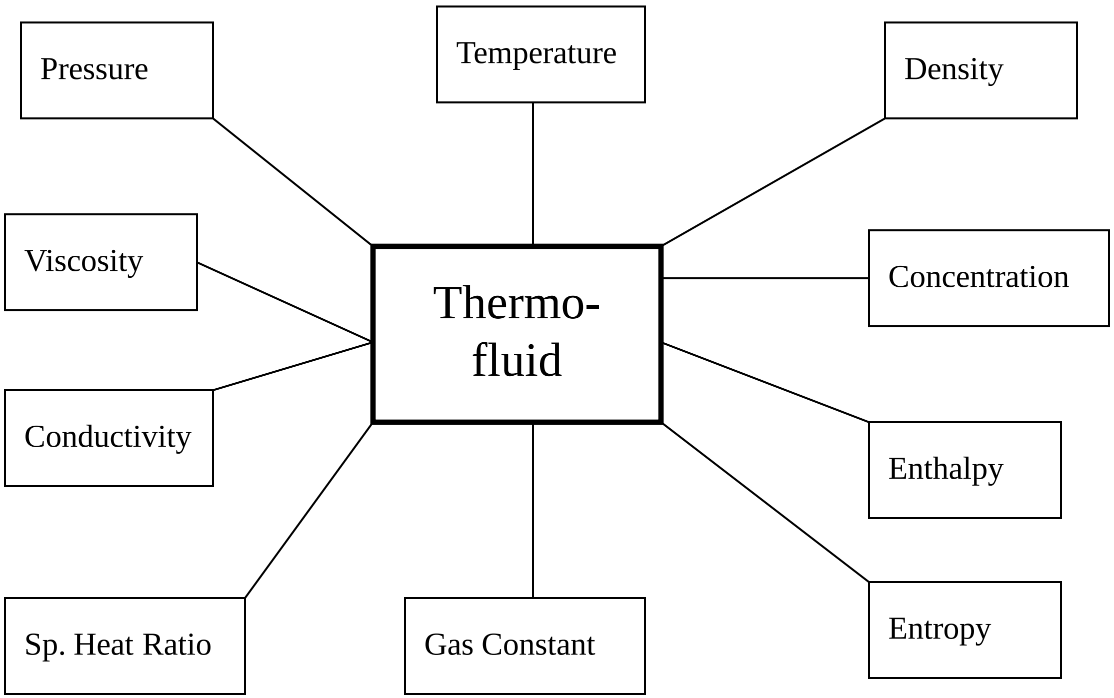
DATA STRUCTURE FOR FLOW ANALYSIS



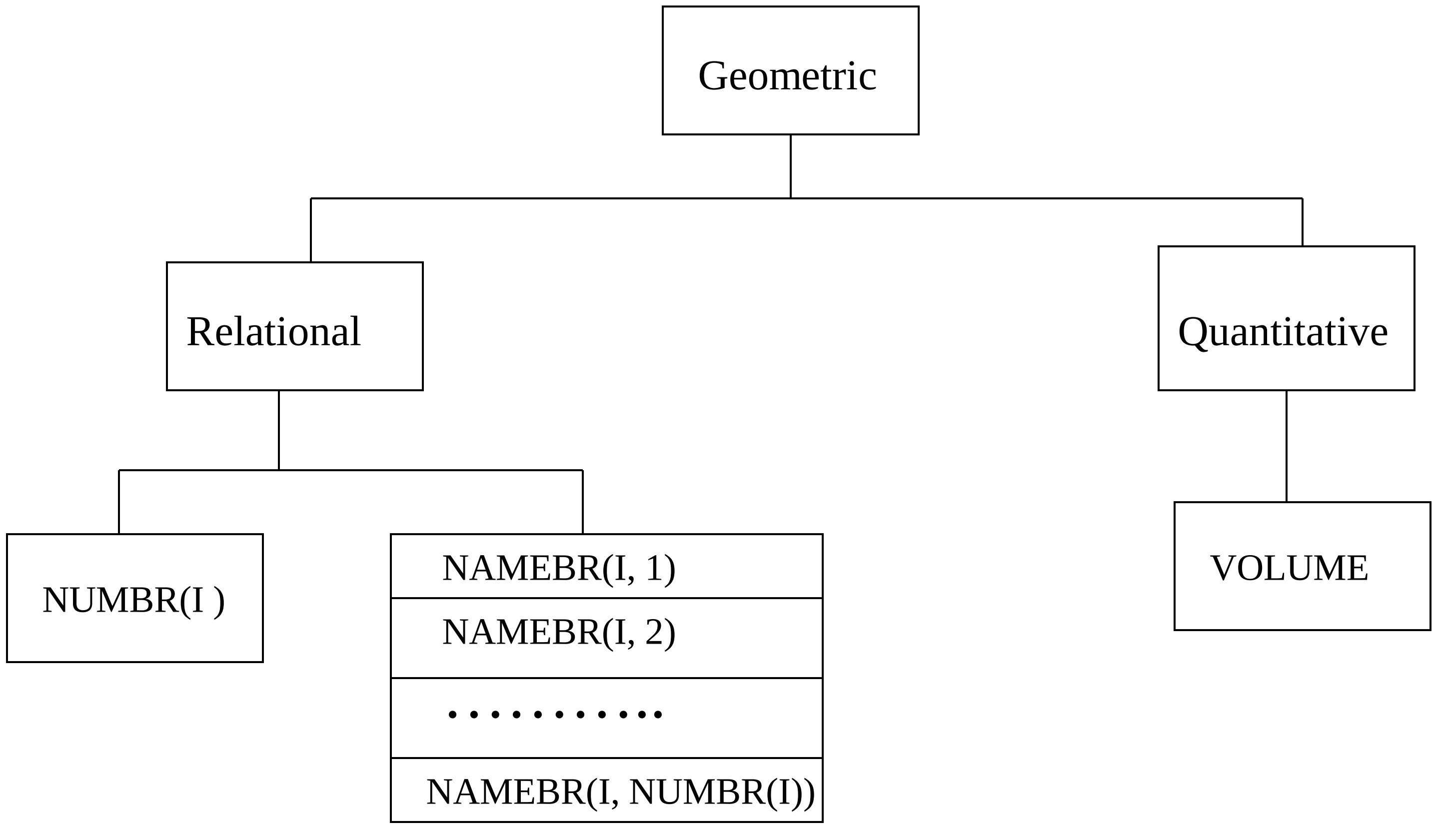
Extended Data Structure for Conjugate Heat Transfer



INTERNAL & BOUNDARY NODE THERMOFLUID PROPERTIES



INTERNAL NODE GEOMETRIC PROPERTIES

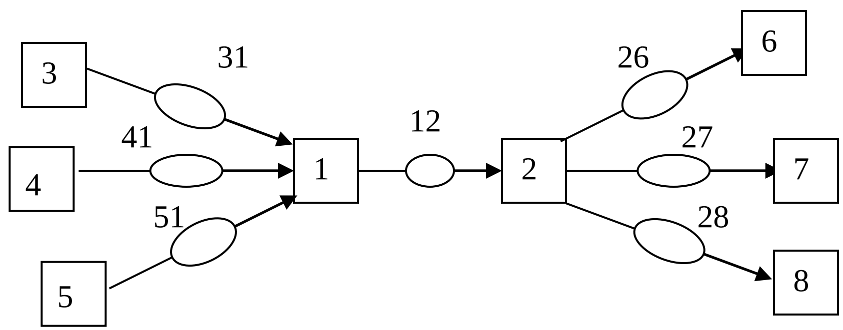


NUMBR – Number of branches connected to the node

NAMEBR – Name of the branches connected to the node

EXAMPLE OF NODE RELATIONAL PROPERTY

Relational Property of Node 1



Number of branches connected to Node I, NUMBR(I) = 4

Name of the Branches connected to Node I,

NAMEBR(I,1) = 31

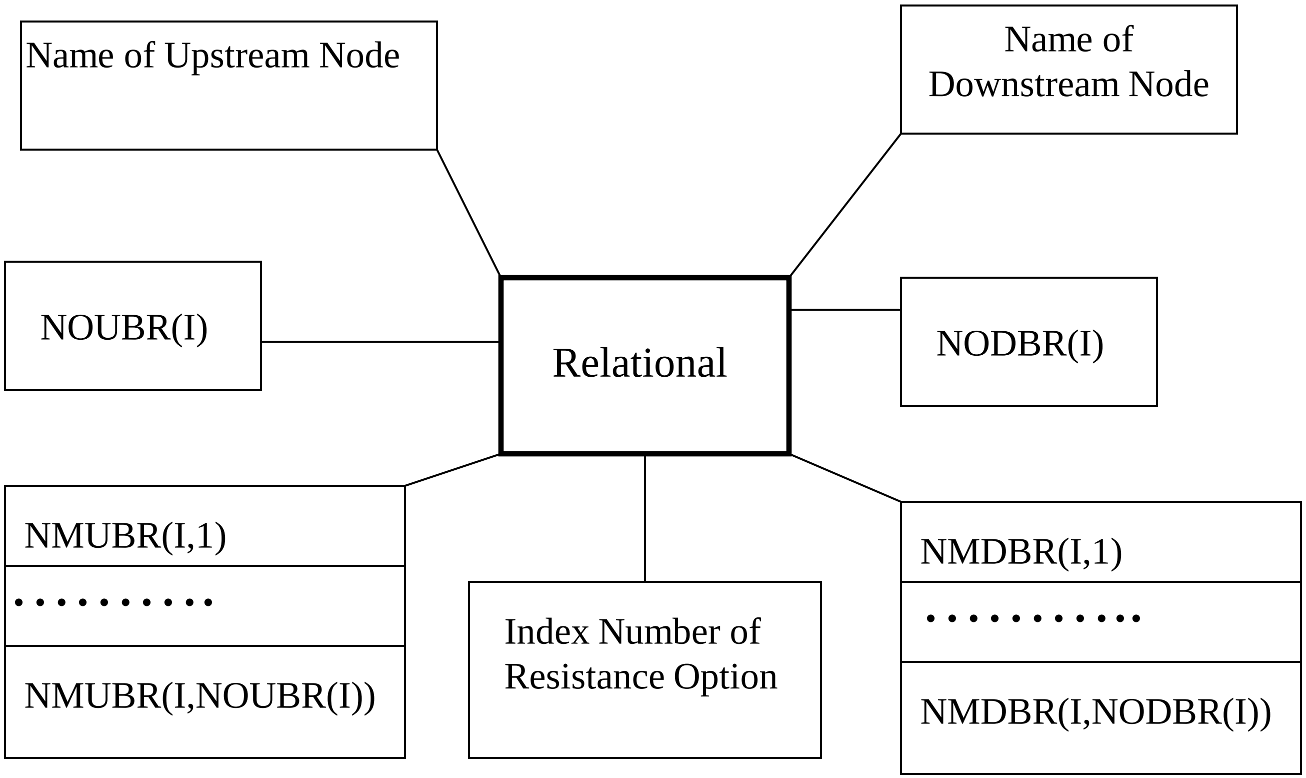
NAMEBR(I,2) = 41

NAMEBR(I,3) = 51

NAMEBR(I,4) = 12

BRANCH PROPERTIES

GEOMETRIC -RELATIONAL



NOUBR – Number of Upstream Branches

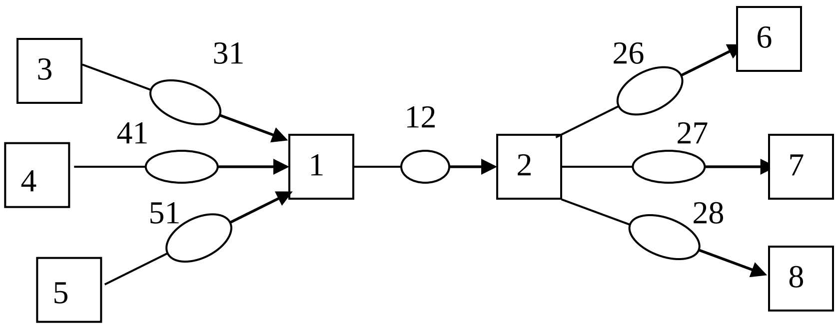
NMUBR – Name of Upstream Branches

NODBR – Number of Downstream Branches

NMDBR – Name of Downstream Branches

EXAMPLE OF BRANCH RELATIONAL PROPERTY

Relational Property of Branch 12



Name of Upstream Node, IBRUN(I) = 1

Number of Upstream Branches, NOUBR(I) = 3

Name of Upstream Branches,

NMUBR(I,1) = 31

NMUBR(I,2) = 41

NMUBR(I,3) = 51

Name of Downstream Node, IBRDN(I) = 2

Number of Downstream Branches, NODBR(I) = 3

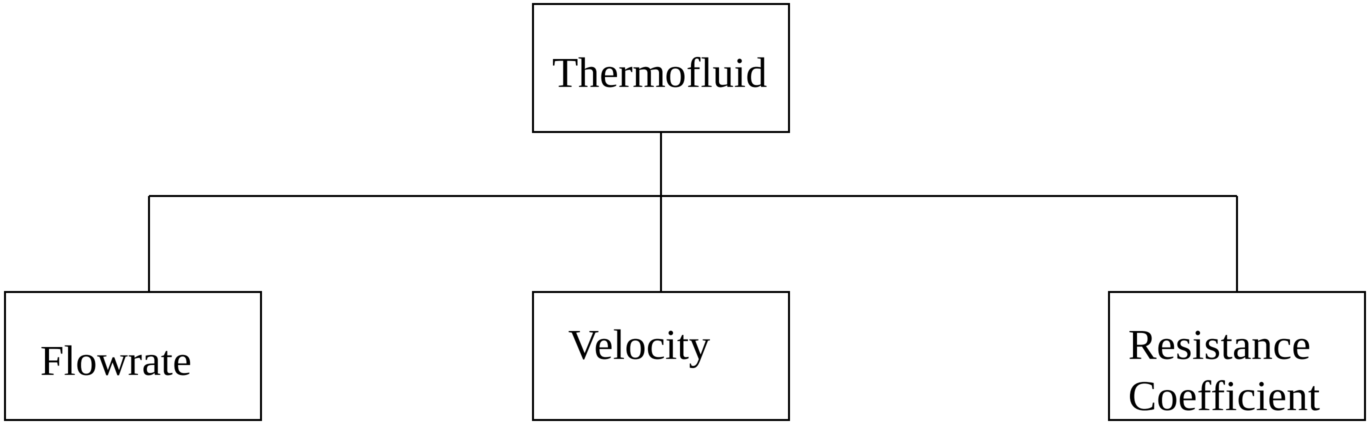
Name of Downstream Branches,

NMDBR(I,1) = 26

NMDBR(I,2) = 27

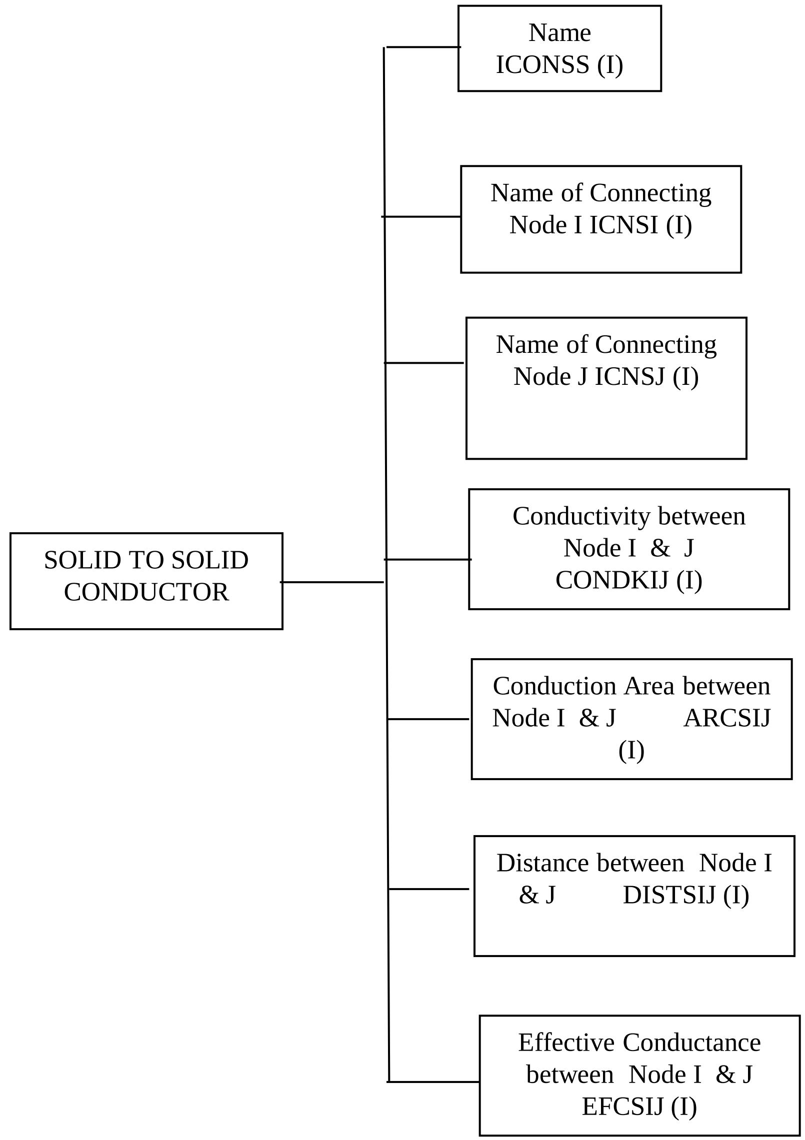
NMDBR(I,3) = 28

BRANCH PROPERTIES THERMOFLUID

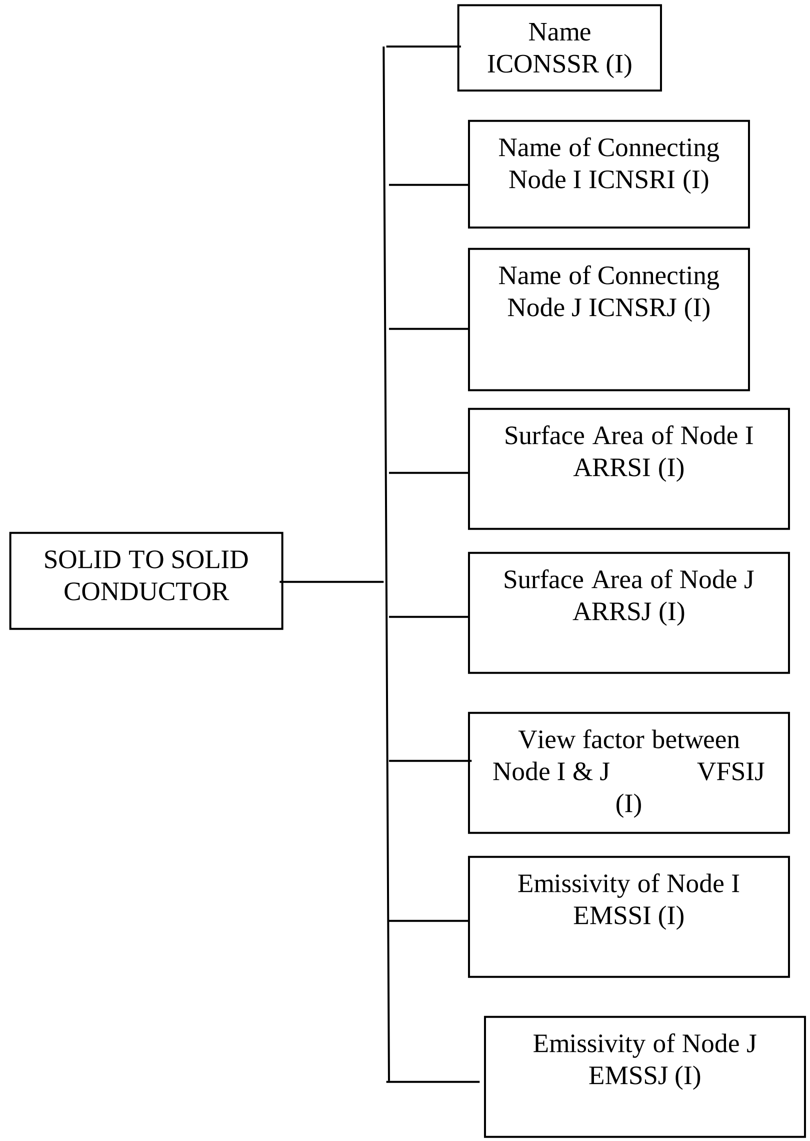


SOLID NODE PROPERTIES

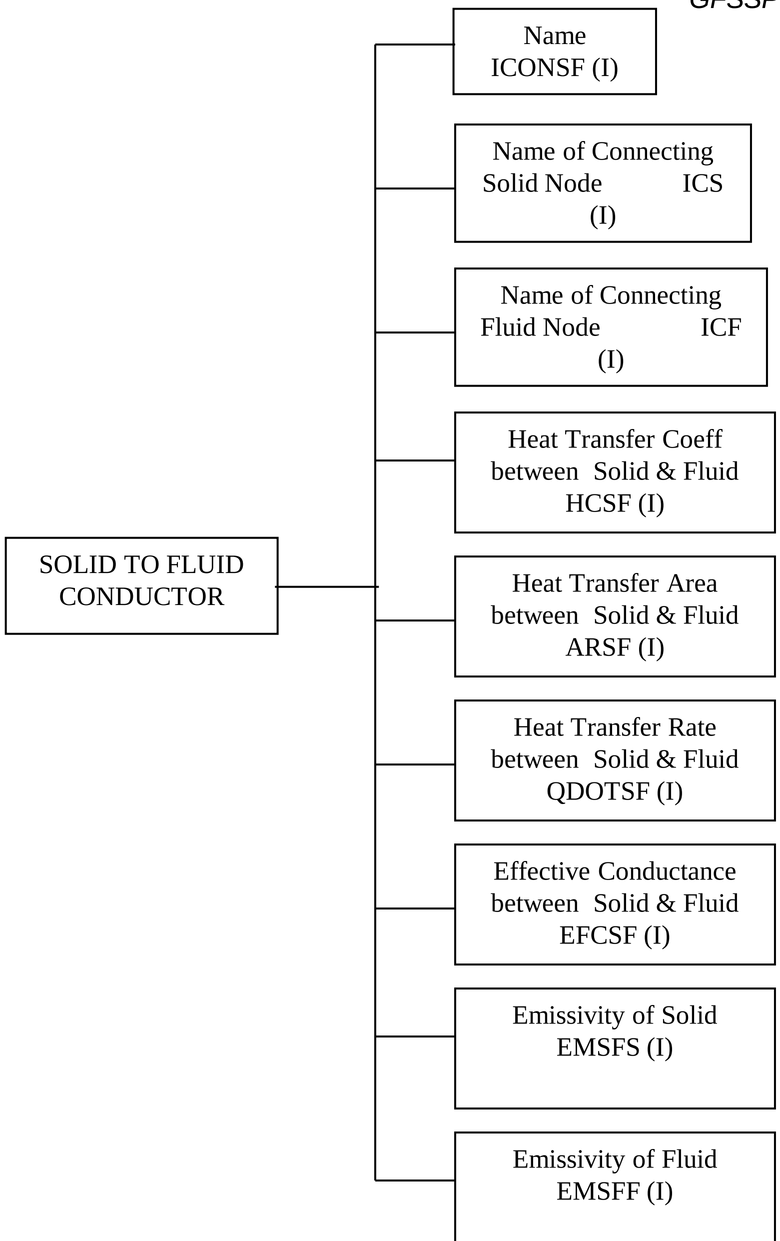
SOLID TO SOLID CONDUCTOR



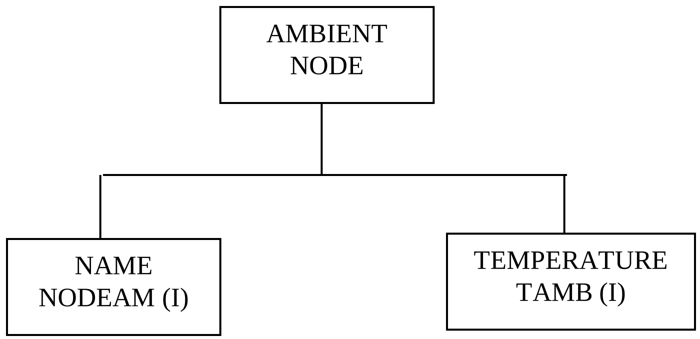
SOLID TO SOLID RADIATION CONDUCTOR



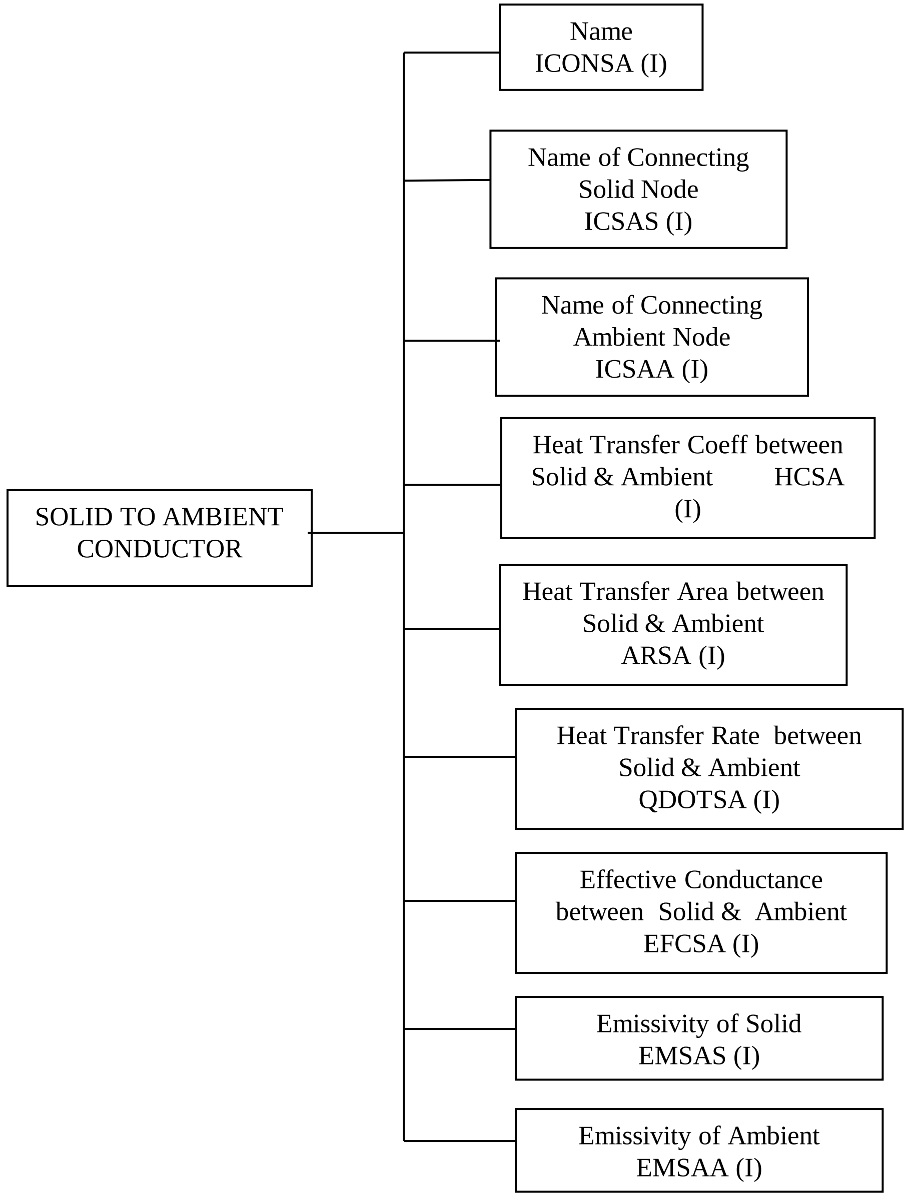
SOLID TO FLUID CONDUCTOR



AMBIENT NODE



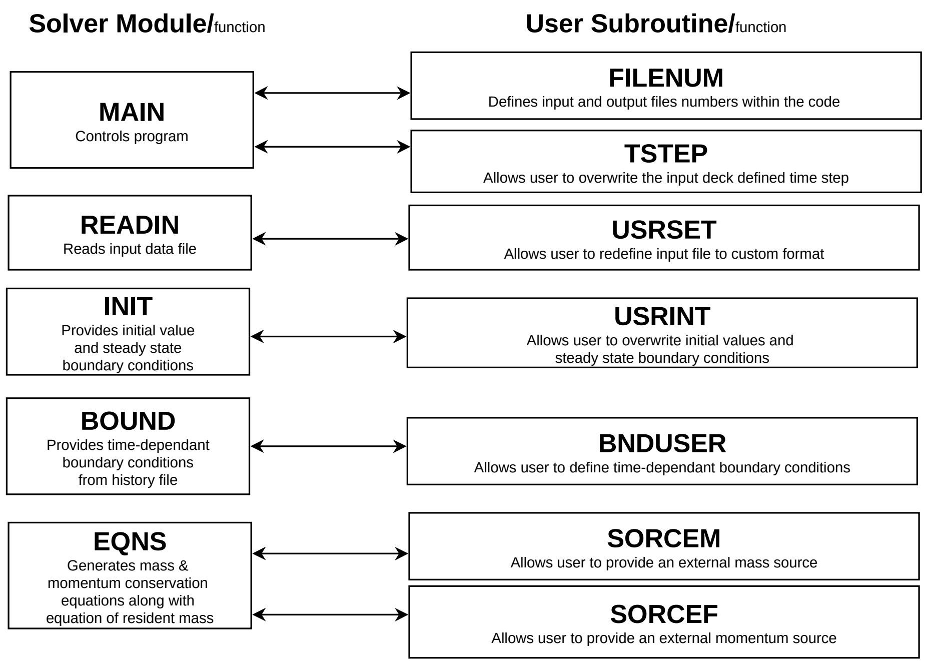
SOLID TO AMBIENT CONDUCTOR



SUMMARY

- GFSSP's data structure allows building any network system. The only limit is the dimension of the array which can be increased easily if needed
- The present version of the code provides allocation for
  - Fluid Node : 300 ; Solid Node : 100 ; Ambient Node : 100
  - Branch: 500 ; Solid to Solid, Solid to Fluid, Solid to Amb. Cond.: 100
  - Number of Branches to a node: 50
  - Number of species in a Mixture: 10
  - Number of Tanks for Pressurization System: 5
  - Number of Control & Relief Valves: 10
  - Number of Pressure & Flow Regulator: 10
- A knowledge of GFSSP's Data Structure will be required for development of User Subroutine

USER SUBROUTINE



Contents

- Motivation & Benefit
- Program Structure
- Solution Algorithm
- Solver-User Subroutine Interaction
- Data Structure
- Indexing Subroutines
- Examples

MOTIVATION AND BENEFIT

- Motivation: To allow users to access GFSSP solver module to develop additional modeling capability
- Benefit: GFSSP users can work independently without Developer's active involvement

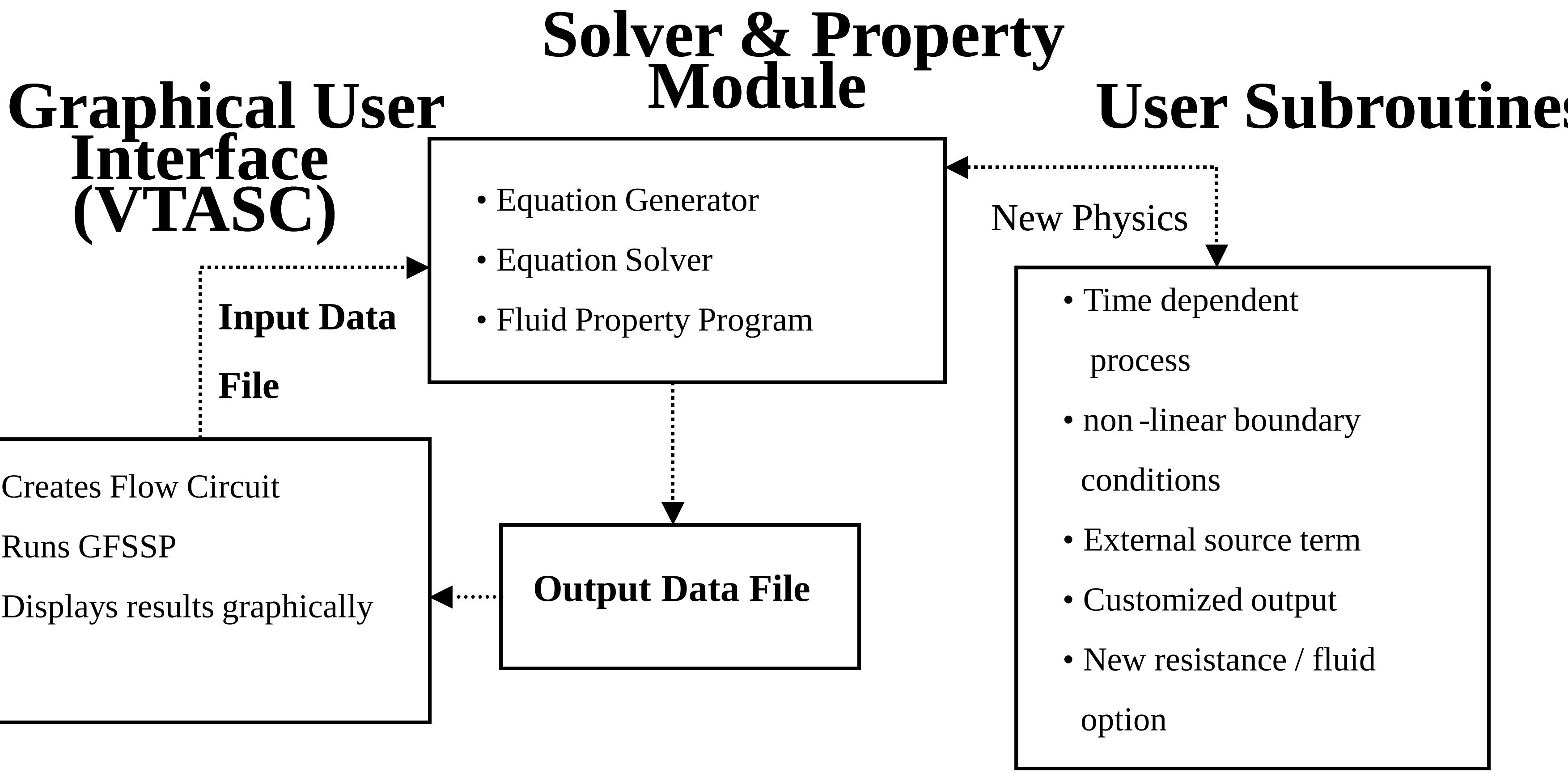
How do they work?

- A series of subroutines are called from various locations of solver module
- The subroutines do not have any code but includes the common block
- The users can write FORTRAN code to develop any new physical model in any particular node or branch

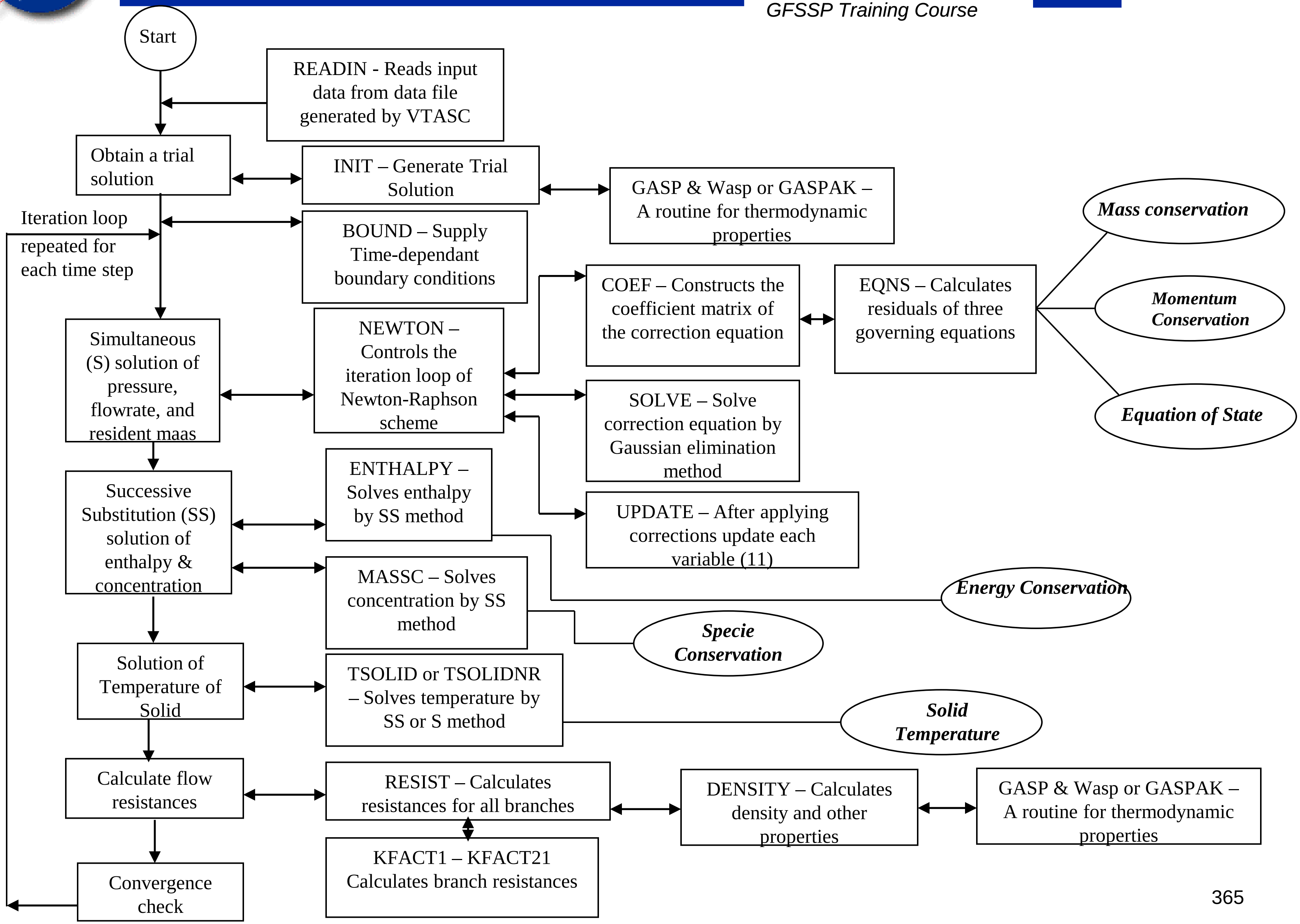
What users need to do?

- Users need to compile a new file containing all user routines and link that with GFSSP to create a new executable

GFSSP – Program Structure



Flow chart of Solution Algorithm



Description of User Subroutines

Twenty two User Subroutines were provided:

- SORCEM: External Mass Source
- SORCEF: External Force
- SORCEQ: External Heat source
- SORCEC: External Concentration source
- KFUSER: New resistance option
- PRPUSER: New fluid property
- TSTEP: Variable time step during a transient run
- BNDUSER: Variable boundary condition during transient run

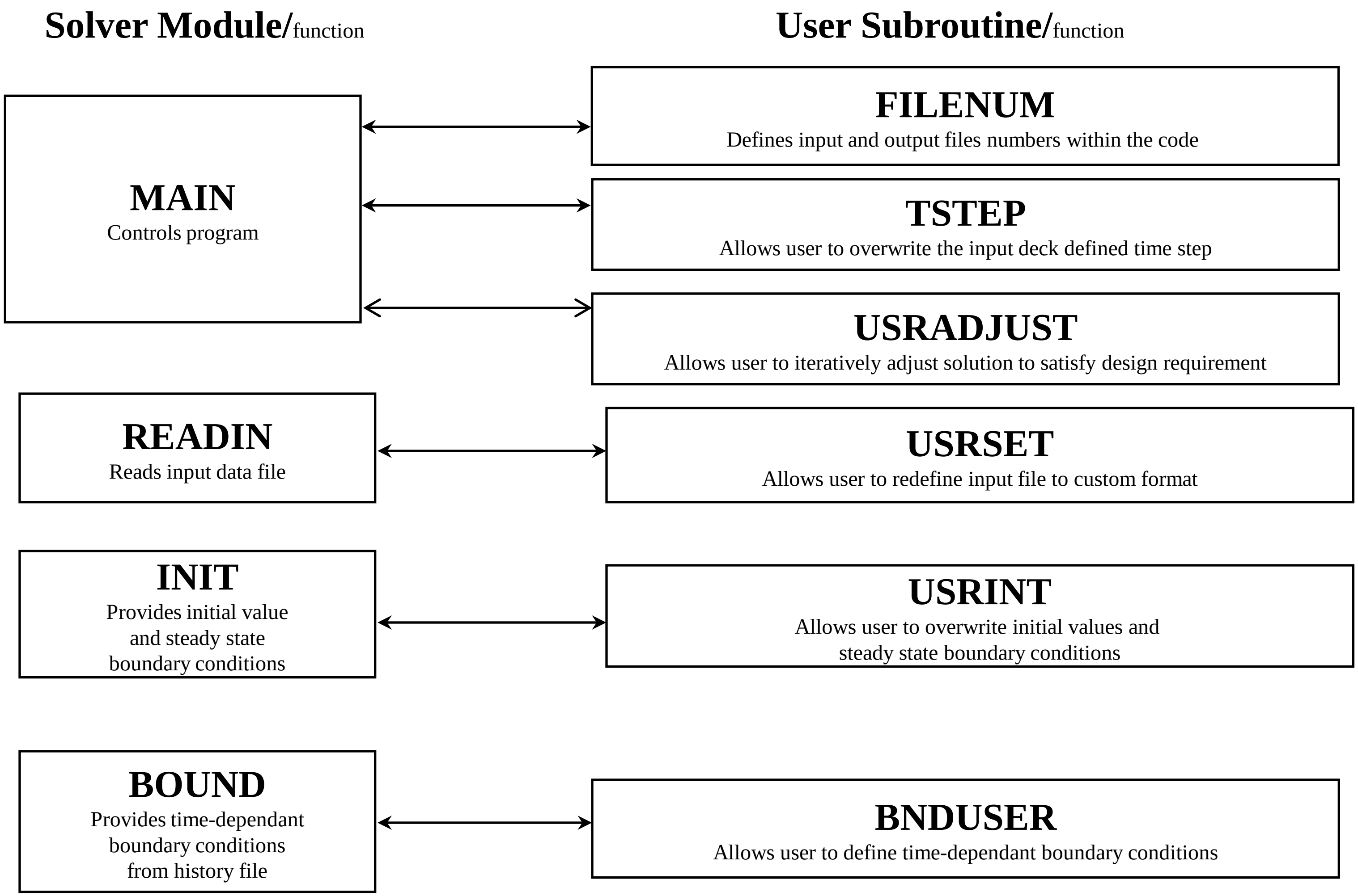
Description of User Subroutines

- USRINT: Provide initial values and steady state boundary conditions
- PRNUSER: Additional print out or creation of additional file for post processing
- FILNUM: Assign file numbers; users can define new file numbers
- USRSET: User can supply all the necessary information by writing their own code
- SORCETS: External Heat Source in Solid Node
- USRHCF: New Heat Transfer Correlation
- USRADJUST: Solution adjustment to satisfy design requirement

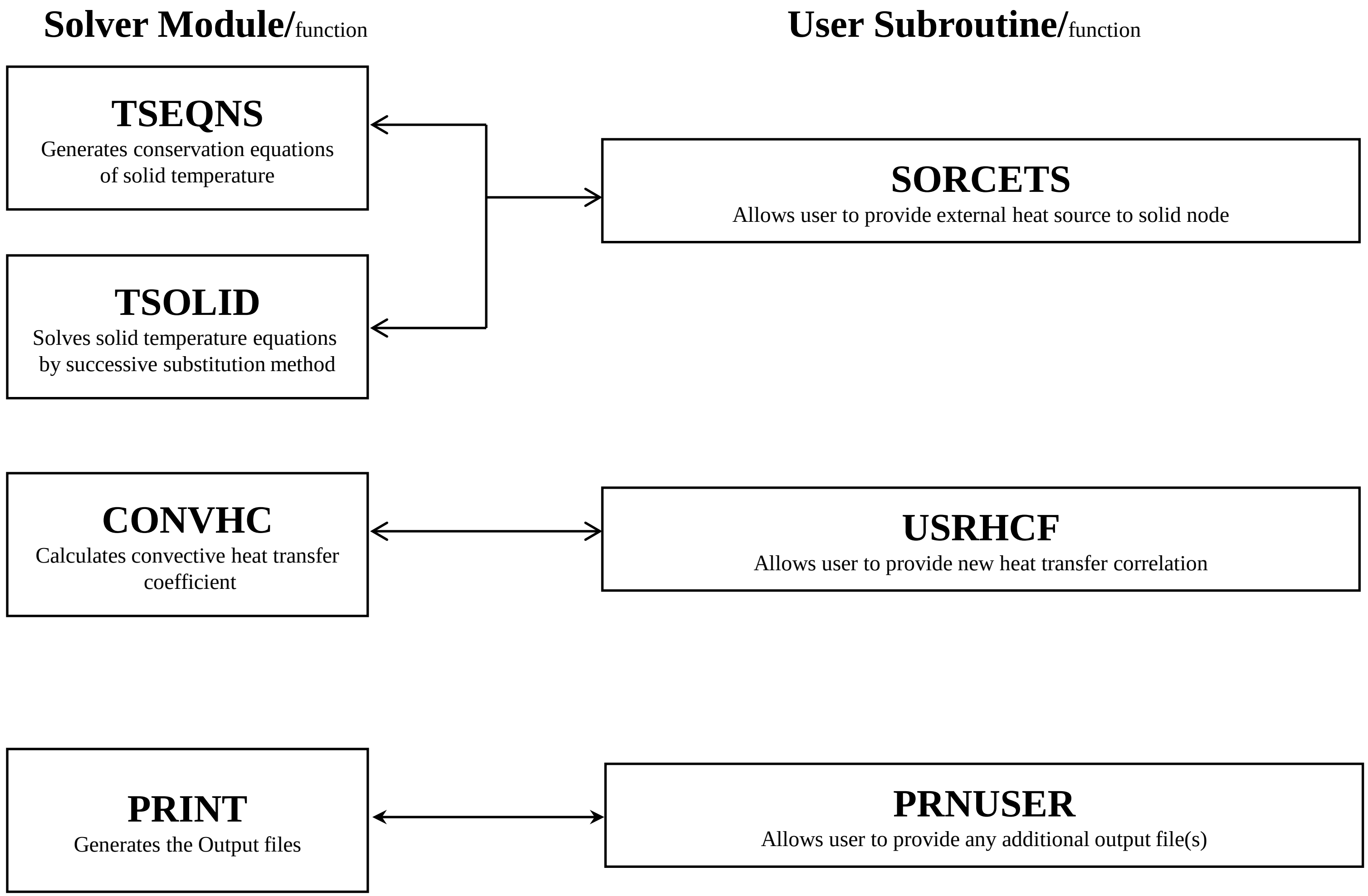
Description of User Subroutines

- PRPADJUST: Adjust Thermodynamic or Thermophysical Property
- TADJUST: Adjust Temperature, if necessary
- PADJUST: Adjust Pressure, if necessary
- FLADJUST: Adjust Flowrate, if necessary
- HADJUST: Adjust Enthalpy, if necessary
- SORCEHXQ Add heat sources to component Enthalpy Equation in Mixture (Enthalpy Option -2)
- USRMDG: Adjust Input Parameters for Multi-D Flow

Solver-User Subroutine Interaction (Partial List)



Solver Module/function		**User Subroutine**/function
EQNS Generates mass & momentum conservation equations along with equation of resident mass	↔	**SORCEM** Allows user to provide an external mass source
	↔	**SORCEF** Allows user to provide an external momentum source
ENTHALPY/ ENTROPY Generates and solves the energy conservation equation	↔	**SORCEQ** Allows user to provide an external heat source
MASSC Generates and solves the specie conservation equation	↔	**SORCEC** Allows user to provide an external specie source
RESIST Calculates the flow resistance coefficient	↔	**KFUSER** Allows user to define new resistance options
DENSITY Calculates the fluid properties	↔	**PRPUSER** Allows user to define new fluid properties


Solver Module/function
User Subroutine/function
TSEQNS
Generates conservation equations of solid temperature
TSOLID
Solves solid temperature equations by successive substitution method
SORCETS
Allows user to provide external heat source to solid node
CONVHC
Calculates convective heat transfer coefficient
USRHCF
Allows user to provide new heat transfer correlation
PRINT
Generates the Output files
PRNUSER
Allows user to provide any additional output file(s)

Indexing Subroutine

SUBROUTINE INDEXI (NUMBER, NODE, NNODES, IPN)
or
SUBROUTINE INDEXI (NUMBER, IBRANCH, NBR, IB)

This subroutine determines the pointer of node or branch

Input Variables:
NUMBER: *Node* or *Branch* Number
NODE/IBRANCH: Array for storing *Node* or *Branch* Number
NNODES/NBR: Number of *Nodes* or *Branches*

Output Variable:
IPN/IB: Location of *Node* or *Branch* in Array (Pointer)

USE OF SUBROUTINE INDEXI

Location	1	2	3	4	5
NODE	100	200	300	400	500
INDEX	2	1	1	1	2
P	5125.50	4785.23	3876.45	2557.85	1668.25
TF	560.00	555.25	525.34	500.25	480.00

NNODE = 5

NINT = 3

Address location of a node (say node number 400).

```
NUMBER = 400
CALL INDEXI (NUMBER,NODE,NNODES,IPN)
```

In this Example IPN = 4

P(IP) = 2557.85

TF(IP) = 500.25

Indexing Subroutines

SUBROUTINE INDEXA (NUMBER, NODEAM, NAMB, IPAN)
This subroutine determines the pointer of Ambient Node.

Input Variables:
NUMBER: Ambient Node Number
NODEAM: Array for storing Ambient Node Number
NAMB: Number of Ambient Nodes

Output Variable:
IPAN: Location of Ambient Node in Array (Pointer)

SUBROUTINE INDEXS (NUMBER, NODESL, NSOLIDX, IPSN)
This subroutine determines the pointer of Solid Node.

Input Variables:
NUMBER: Solid Node Number
NODESL: Array for storing Solid Node Number
NSOLIDX: Number of Solid Nodes

Output Variable:
IPSN: Location of Solid Node in Array (Pointer)

Indexing Subroutines

SUBROUTINE INDEXSSC (NUMBER, ICONSS, NSSC, ICSS)
This subroutine determines the pointer of Solid to Solid Conductor.

Input Variables:
NUMBER: Solid to Solid Conductor Number
ICONSS: Array for storing Solid to Solid Conductor Number
NSSC: Number of Solid to Solid Conductors

Output Variable:
ICSS: Location of Solid to Solid Conductor in Array (Pointer)

SUBROUTINE INDEXSFC (NUMBER, ICONSF, NSFC, ICSF)
This subroutine determines the pointer of Solid to Fluid Conductor.

Input Variables:
NUMBER: Solid to Fluid Conductor Number
ICONSF: Array for storing Solid to Fluid Conductor Number
NSFC: Number of Solid to Fluid Conductors

Output Variable:
ICSF: Location of Solid to Fluid Conductor in Array (Pointer)

Indexing Subroutines

SUBROUTINE INDEXSAC (NUMBER, ICONSA, NSAC, ICSA)
This subroutine determines the pointer of Solid to Ambient Conductor.
Input Variables:
NUMBER: Solid to Ambient Conductor Number
ICONSA: Array for storing Solid to Ambient Conductor Number
NSAC: Number of Solid to Ambient Conductors

Output Variable:
ICSA: Location of Solid to Ambient Conductor in Array (Pointer)

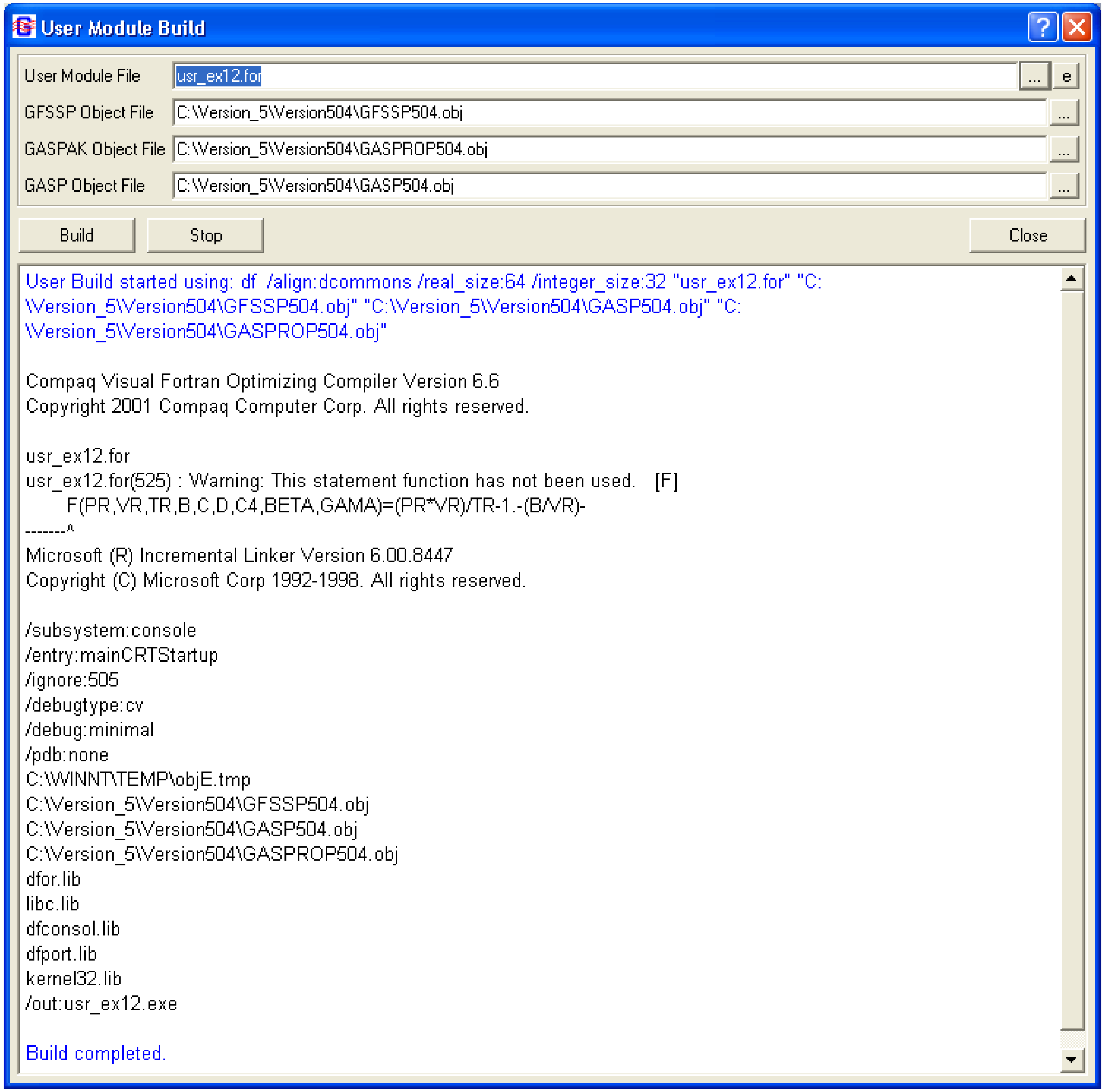
SUBROUTINE INDEXSSRC (NUMBER, ICONSSR, NSSR, ICSSR)
This subroutine determines the pointer of Solid to Solid Radiation Conductor.
Input Variables:
NUMBER: Solid to Solid Radiation Conductor Number
ICONSSR: Array for storing Solid to Solid Radiation Conductor Number
NSSR: Number of Solid to Solid Radiation Conductors

Output Variable:
ICSSR: Location of Solid to Solid Radiation Conductor in Array (Pointer)

Example of a Typical User Subroutine

```
C*********************************************************
      SUBROUTINE USRHCF(NUMBER,HCF)
C     PURPOSE: PROVIDE HEAT TRANSFER COEFFICIENT
C*********************************************************
      INCLUDE 'COMBLK.FOR'
C*********************************************************
      EQUIVALENCE (USRVAR1(1),HL)
      DATA FACTHC,CONHC,FNHC/1.0,0.15,0.33/
C     ESTIMATE HEAT TRANSFER COEFFICIENT IN ULLAGE NODES FROM FREE
C     CONVECTION CORRELATION
      NUMF=ICF(NUMBER)
      CALL INDEXI(NUMF,NODE,NNODES,IPN)
      NUMS=ICS(NUMBER)
      CALL INDEXS(NUMS,NODESL,NSOLIDX,IPSN)
      BETA=1./TF(IPN)
      DELTAT=ABS(TF(IPN)-TS(IPSN))
      GR=HL**3*RHO(IPN)**2*G*BETA*DELTAT/(EMU(IPN)**2)
      PRNDTL=CPNODE(IPN)*EMU(IPN)/CONDF(IPN)
      HCF=FACTHC*CONHC*CONDF(IPN)*(GR*PRNDTL)**FNHC/HL
      RETURN
      END
C*********************************************************
```

Compiling & Linking User Subroutine



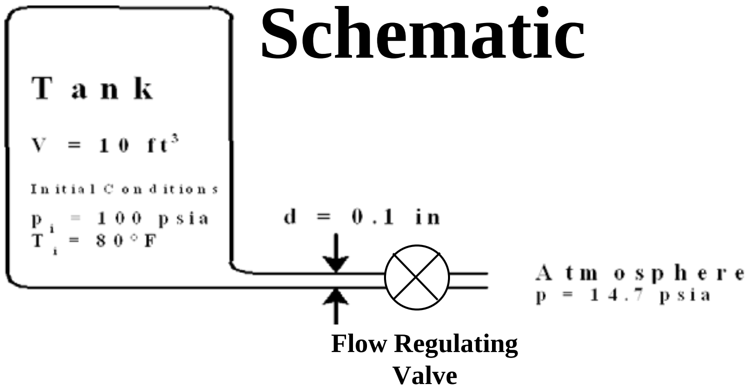
User Subroutine Applications

- Flow Regulator
- User Prescribed Heat Transfer Coefficient
- Example 18 - Simulation of a Subsonic Fanno Flow
- Example 20 - Simulation of a Lithium Loop Model

Modeling Flow Regulator in GFSSP

- A User Subroutine has been developed to model Flow Regulating Valve
- An iterative algorithm has been developed that adjusts flow area until predicted flow matches required flow
- The model has been tested for a blowdown problem (Example 8)

Flow Schematic



GFSSP Model

Flow Regulator Algorithm

The required flow area is determined by the Newton-Raphson scheme with the following steps:

1. Assume an Area, A^*
2. Compute the deviation, $f = \dot{m}_{req} - \dot{m}$
3. Estimate the gradient

$$f' = \frac{\partial f}{\partial A} = -\frac{2 g_c \rho_u C_L^2 A \, \Delta p}{\dot{m}}$$

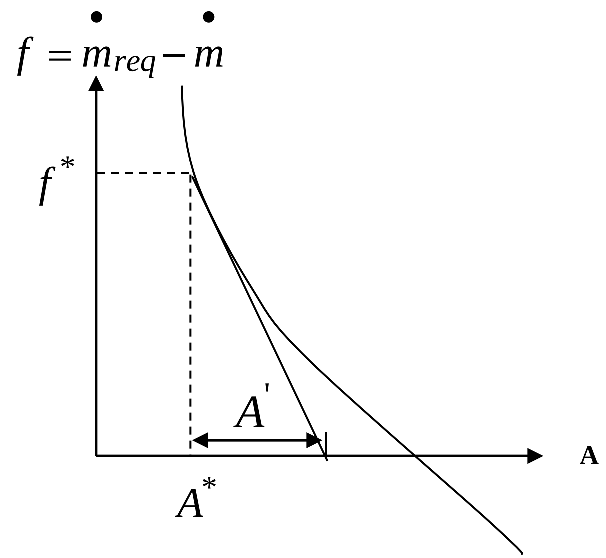
4. Estimate the correction in the Area

$$A' = -\frac{f^*}{f'}$$

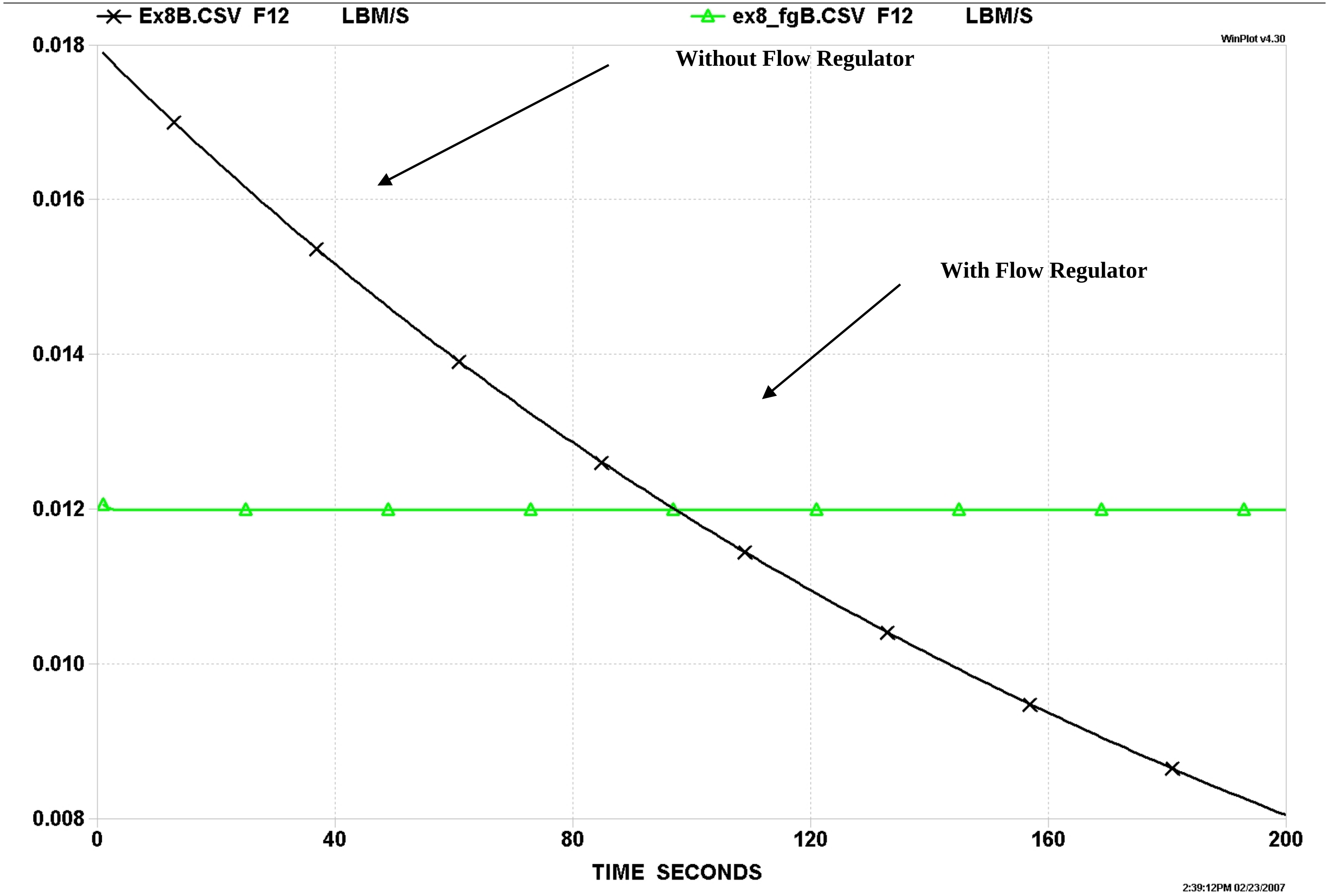
5. Compute the new Area

$$A = A^* + \alpha A' \quad where \; 0 < \alpha < 1$$

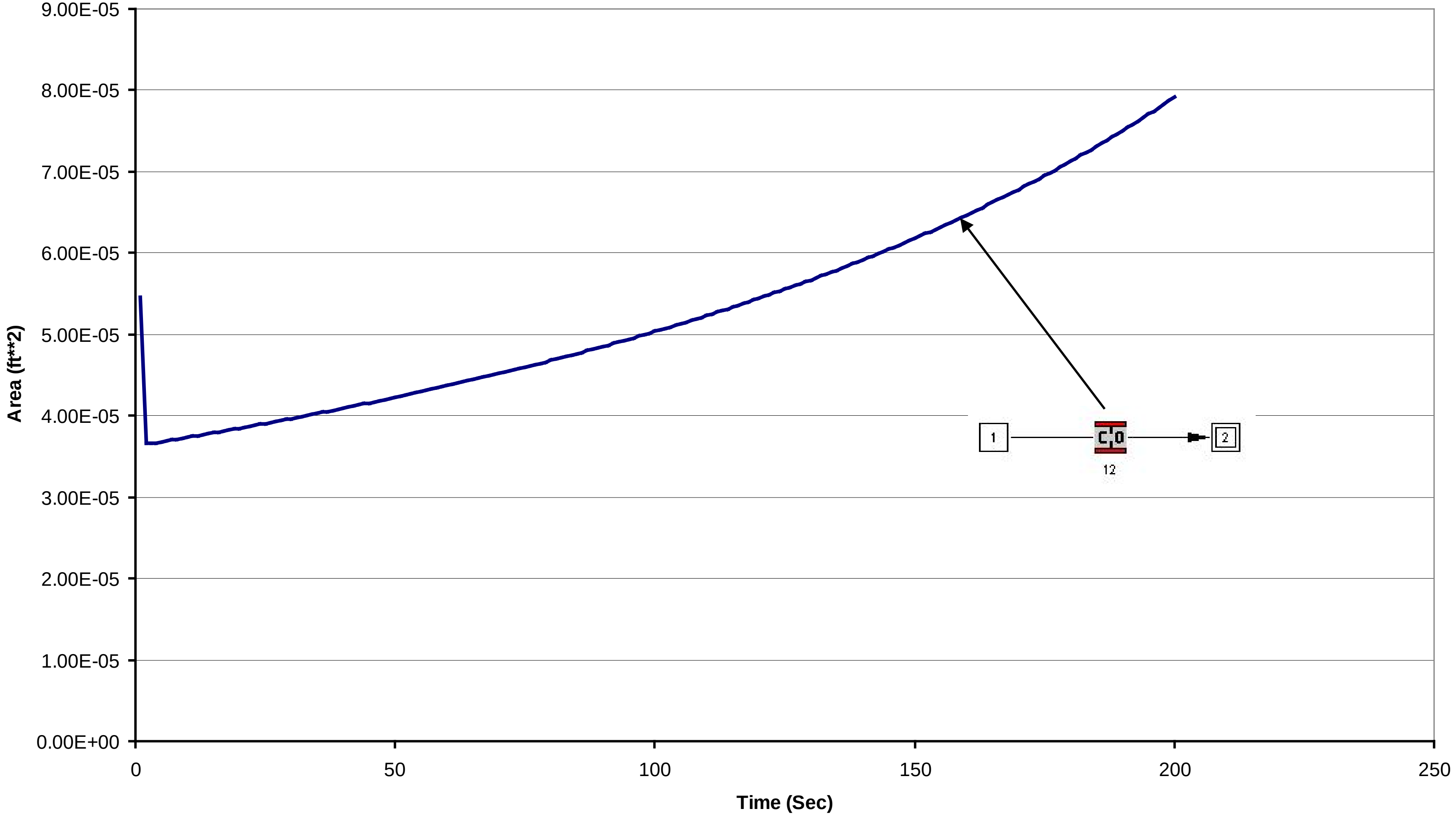
6. Repeat steps 2 – 5 until $f^* \rightarrow 0$



Results


Ex8B.CSV F12 LBM/S
ex8_fgB.CSV F12 LBM/S
WinPlot v4.30
Without Flow Regulator
With Flow Regulator
0.018
0.016
0.014
0.012
0.010
0.008
0
40
80
120
160
200
TIME SECONDS
2:39:12PM 02/23/2007

History of Area Change of Flow Regulating Valve

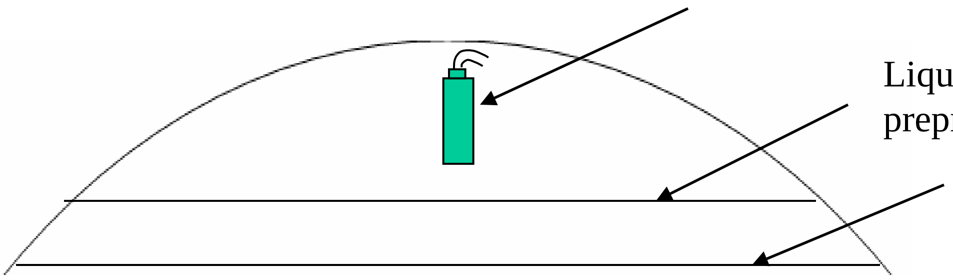


User Subroutine

```
C*********************************************************************
     SUBROUTINE USRADJUST
C    PURPOSE: ADJUST BOUNDARY CONDITION OR GEOMETRY FOR STEADY-STATE MODEL
C*********************************************************************
     INCLUDE 'COMBLK.FOR'
C*********************************************************************
C    ADD CODE HERE
     DATA RELAXAR,AREAMAX/1.0,0.002/
     DATA EMREQ/0.012/
     CALL INDEXI(12,IBRANCH,NBR,IB)
     CALL INDEXI(1,NODE,NNODES,IPUP)
     CALL INDEXI(2,NODE,NNODES,IPDN)
     FSTR=EMREQ-FLOWR(IB)
     ABSDELP=ABS(P(IPUP)-P(IPDN))
     FDASH=-2.*GC*RHO(IPUP)*BRPR1(IB)**2*ABSDELP*AREA(IB)
    &       /ABS(FLOWR(IB))
     ARDASH=-FSTR/FDASH
C    CHECK CONVERGENCE
     DIFMDOT=ABS(FSTR)/EMREQ
     IF (DIFMDOT .GT. 0.001.AND.ITERADJU.LT.15) THEN
      REPEAT =.TRUE.
      AREAOLD=AREA(IB)
      AREA(IB)=AREA(IB)+MIN(RELAXAR*ARDASH,0.05*AREA(IB))
      IF(AREA(IB).LT.0.0) AREA(IB)=AREAOLD
      AREA(IB)=MIN(AREA(IB),AREAMAX)
     ELSE
      REPEAT = .FALSE.
      ITERADJU=1
     ENDIF
     IF (REPEAT)
    &  WRITE (*,*) 'ITERADJU = ',ITERADJU, AREA(IB),ARDASH,DIFMDOT
     RETURN
     END
```

Annotations:
- ← Define Requirements (DATA statements)
- ← Obtain Indices (CALL INDEXI statements)
- ← Calculate gradient and correction (FSTR … ARDASH)
- ← Apply correction, check convergence and bound the solution (IF … ENDIF)
- ← Print convergence parameters (IF (REPEAT) WRITE)

User Prescribed Heat Transfer Coefficient



$$Nu = 0.15\left(Gr\ \Pr\right)^{0.33}$$

$$Nu = \frac{hL}{k}$$

$$Gr = \frac{L^3 \rho^2 g \beta \Delta T}{\mu^2}$$

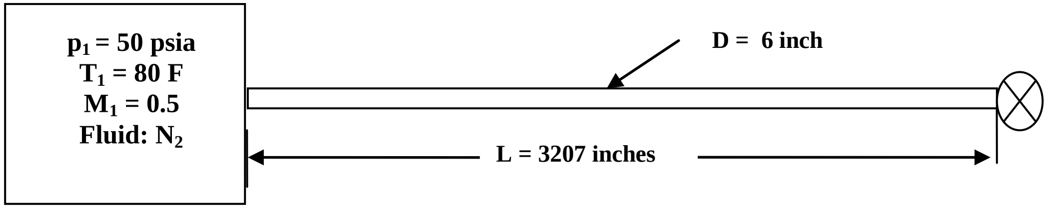
$$\Pr = \frac{C_p \mu}{k}$$

- In User Subroutine local property values are available in the common block
- Heat Transfer Coefficients are then calculated in SUBROUTINE USRHCF and returned to the SOLVER MODULE

User Subroutine to Calculate Heat Transfer Coefficient

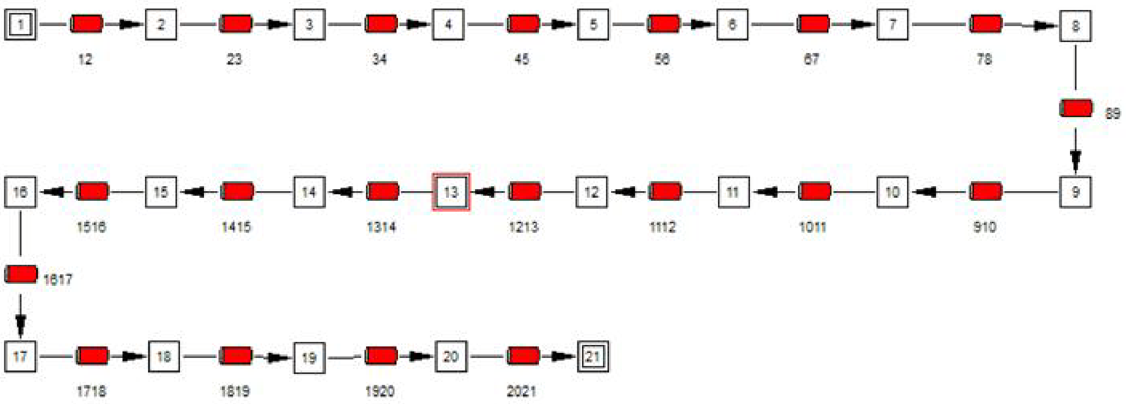
```
C*****************************************************************
      SUBROUTINE USRHCF(NUMBER,HCF)
C     PURPOSE: PROVIDE HEAT TRANSFER COEFFICIENT
C*****************************************************************
      INCLUDE 'COMBLK.FOR'
C*****************************************************************
      EQUIVALENCE (USRVAR1(1),HL)
      DATA FACTHC,CONHC,FNHC/1.0,0.15,0.33/
C     ESTIMATE HEAT TRANSFER COEFFICIENT IN ULLAGE NODES FROM FREE
C     CONVECTION CORRELATION
      NUMF=ICF(NUMBER)
      CALL INDEXI(NUMF,NODE,NNODES,IPN)
      NUMS=ICS(NUMBER)
      CALL INDEXS(NUMS,NODESL,NSOLIDX,IPSN)
      BETA=1./TF(IPN)
      DELTAT=ABS(TF(IPN)-TS(IPSN))
      GR=HL**3*RHO(IPN)**2*G*BETA*DELTAT/(EMU(IPN)**2)
      PRNDTL=CPNODE(IPN)*EMU(IPN)/CONDF(IPN)
      HCF=FACTHC*CONHC*CONDF(IPN)*(GR*PRNDTL)**FNHC/HL
      RETURN
      END
C*****************************************************************
```

Example 18 - Simulation of a Subsonic Fanno Flow



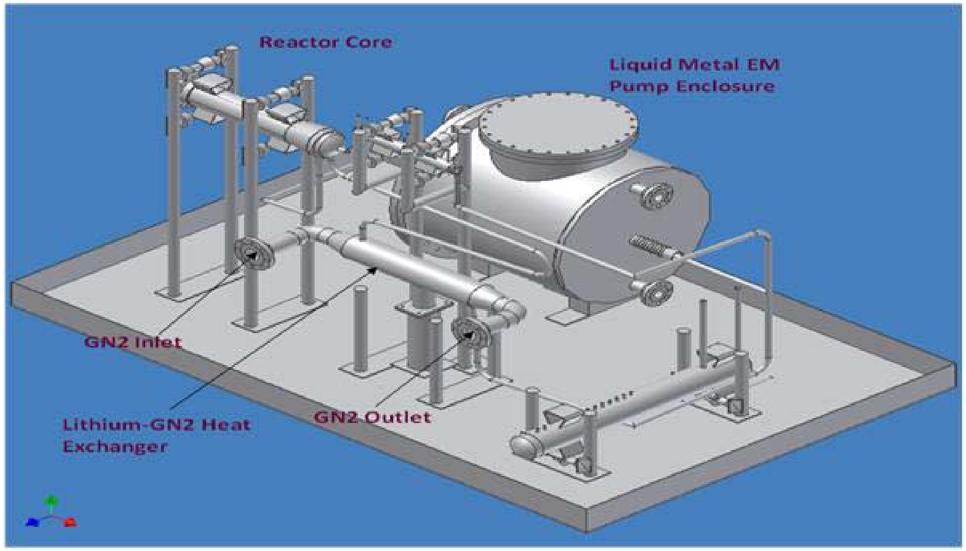
Feature: Compressible Flow with Friction, User Subroutine to set constant friction factor, and comparison with Text Book solution

VTASC Model & User Subroutine



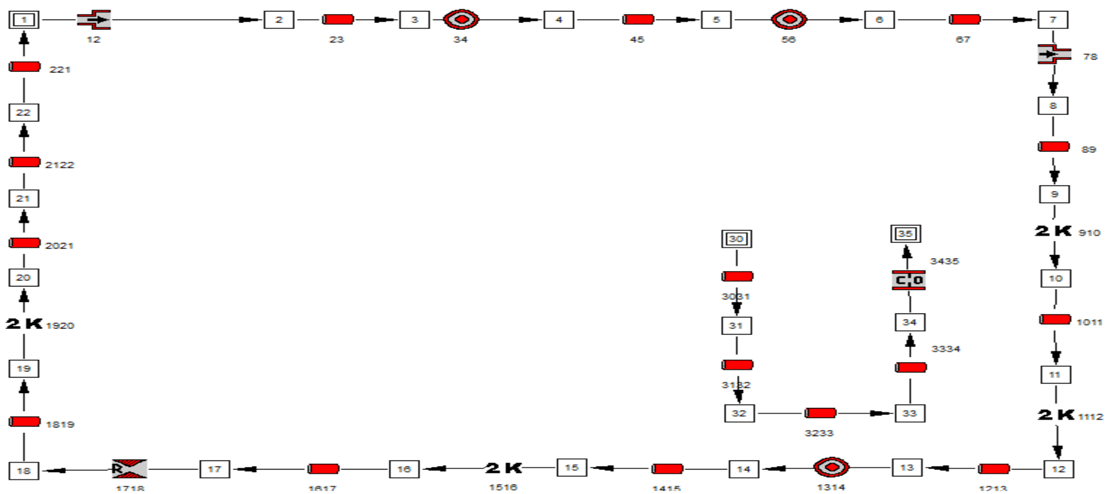

```
C*********************************************************************
      SUBROUTINE KFADJUST(I,RHOU,EMUU,RHOUL,EMUUL,RHOUV,EMUUV,ISATU,
     &                   AKNEW)
C     PURPOSE: ADJUST RESISTANCE IN A BRANCH
C*********************************************************************
      INCLUDE 'COMBLK.FOR'
C*********************************************************************
C     ADD CODE HERE
      IF(IOPT(I).EQ.1) THEN
       PIPEL=BRPR1(I)
       PIPED=BRPR2(I)
       F=0.002
       AKNEW=8.*F*PIPEL/(RHOU*GC*PI*PI*PIPED**5)
      ENDIF
      RETURN
      END
```

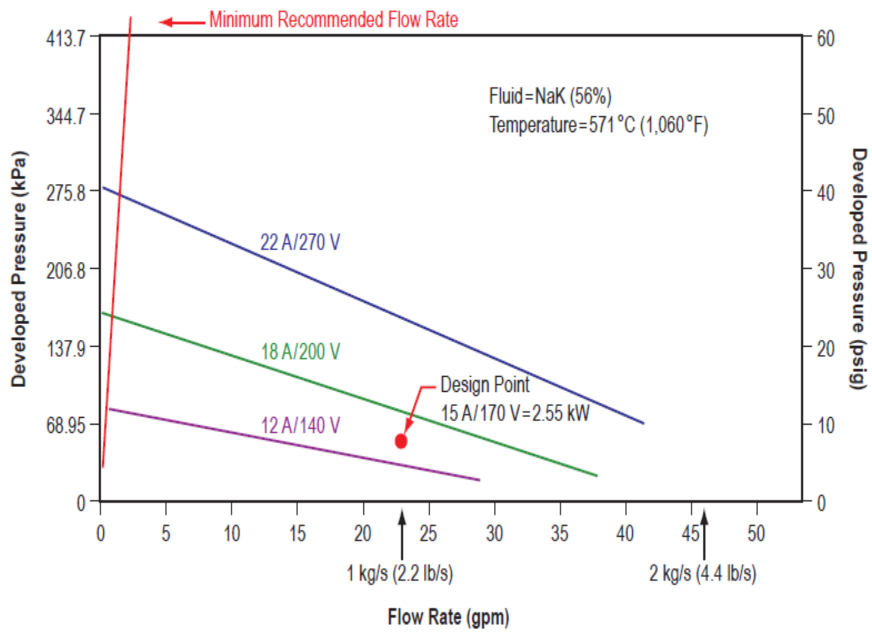
Example 20 - Simulation of a Lithium Loop Model



Feature: Closed Loop with cyclic boundary, Use of user-specified property, Heat Exchanger & User Subroutine to model Electro-Magnetic Pump

VTASC Model & Electromagnetic Pump Characteristics





```
      SUBROUTINE SORCEF(I,TERM0,TERM1,TERM2,TERM3,TERM4,TERM5,TERM6,
     &                  TERM7,TERM8,TERM9,TERM10,TERM100)
C     MODELING OF THERMO-ELECTRIC PUMP
      DIMENSION VOLT(50),FLWTE(50),DPTE(50,50)
            LOGICAL UNREAD
      DATA VOLTIN/170/
C     READ PUMP CHARACTERISTIC DATA FROM FILE
      IF (ITER.EQ.1.AND. (.NOT. UNREAD)) THEN
       OPEN (NUSR1,FILE='Ex20_pump.dat',STATUS='UNKNOWN')
       READ(NUSR1,*)  NFLW,NVOLT
       READ(NUSR1,*) (VOLT(JJ),JJ=1,NVOLT)
       DO II = 1,NFLW
        READ(NUSR1,*) FLWTE(II),(DPTE(II,JJ),JJ=1,NVOLT)
       ENDDO
            UNREAD = .TRUE.
      ENDIF ! IF (ITER.EQ.0)..
```

'Ex20_pump.dat'

```
5  3                          NFLW, NVOLT
140   200  270                VOLT(JJ), JJ=1,NVOLT
0.510  10.5  22    37         FLWTE(II), (DPTE(II,JJ),JJ=1,NVOLT)
1.021   9    19.5  33.5
1.531   7.5   17   30
2.042   6    14.5  26.5
2.598   4.5  12.0  23.0
```

Double Interpolation to find DELTAP for a given flowrate & Voltage

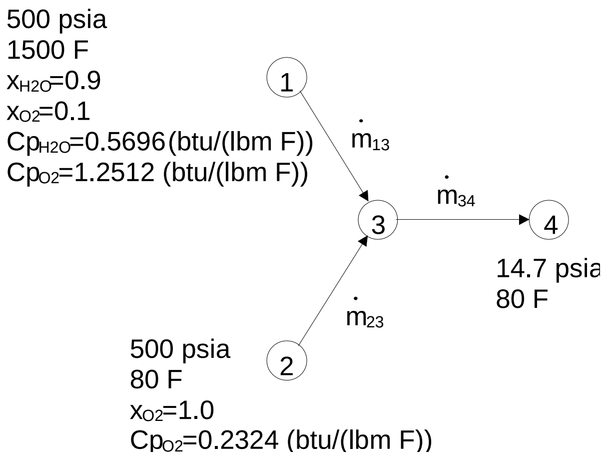

```
 IF (IBRANCH(I) .EQ. 1718) THEN
C    BRACKET THE FLOWRATE
      IR=0
      DO II =2,NFLW
        IF (FLOWR(I).GE.FLWTE(II-
1).AND.FLOWR(I).LE.FLWTE(II)) THEN
         IR=II
         GO TO 100
        ENDIF
      ENDDO
100   IF (IR.EQ.0) THEN
       IF (FLOWR(I).GT.FLWTE(NFLW)) IR=NFLW
       IF (FLOWR(I).LT.FLWTE(1)) IR=2
      ENDIF
C    BRACKET THE VOLT
      JR=0
      DO JJ = 2,NVOLT
        IF (VOLTIN.GE.VOLT(JJ-
1).AND.VOLTIN.LE.VOLT(JJ)) THEN
         JR=JJ
         GO TO 200
        ENDIF
      ENDDO
200   IF (JR.EQ.0) THEN
        IF(VOLTIN.GT.VOLT(NVOLT)) JR=NVOLT
        IF(VOLTIN.LT.VOLT(1)) JR=2
      ENDIF
```

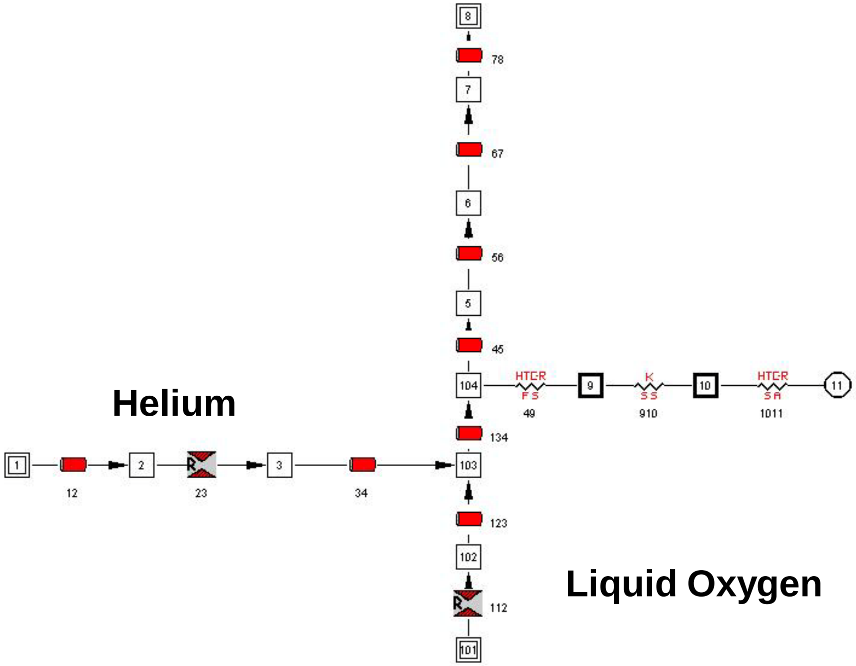
```
C    CALCULATE DELPTE
      FACTFLW=(FLOWR(I)-FLWTE(IR-1))/(FLWTE(IR)-FLWTE(IR-1))
      FACTV=(VOLTIN-VOLT(JR-1))/(VOLT(JR)-VOLT(JR-1))
      DELPTE=(1.-FACTFLW)*(1.-FACTV)*DPTE(IR-1,JR-1)
     &       +FACTFLW*(1.-FACTV)*DPTE(IR,JR-1)
     &       +FACTFLW*FACTV*DPTE(IR,JR)
     &       +(1.-FACTFLW)*FACTV*DPTE(IR-1,JR)
      TERM100=144*DELPTE*AREA(I)


      ENDIF ! IF (IBRANCH(I).EQ...)
```

SUMMARY

- User Subroutines can be used to add new capabilities that are not available to Users through Logical Options
- New capabilities may include:
  - Incorporating Design Specification; this may require iterative adjustment
  - User Specified Heat Transfer Coefficient
  - Incorporating a new physical model such as mass transfer
  - Customized output, Variable timestep etc.
- Checklist for User Subroutines
  - Identify subroutines that require modifications
  - Select GFSSP variables that require to be modified
  - Make use of GFSSP provided User variables in your coding

Fluid Mixture & Two-Phase Flows





CONTENT

- Enthalpy Formulation of Mixture Modeling
- Mathematical Formulation
- Applications:
  - Example 23 – Helium-Assisted Buoyancy-Driven Flow in a Vertical Pipe Carrying Liquid Oxygen with Ambient Heat Leak
  - Charging of Pogo Accumulator
- Summary

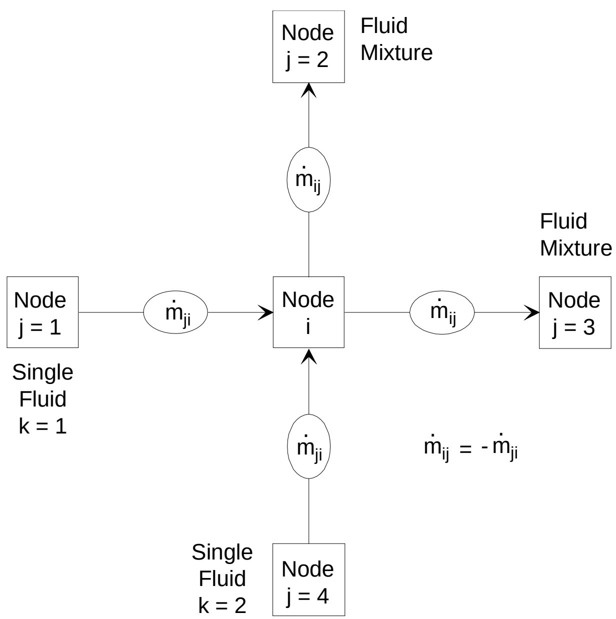
Enhanced Mixture Option

- The mixture capability in earlier versions of GFSSP does not allow phase change in any constituent of the mixture
- In liquid propulsion applications, there are situations where one of the constituent is saturated, i.e. mixture of liquid and vapor in equilibrium
  - For example during purging of liquid oxygen by ambient helium, a mixture of helium, LO2 and GO2 exist
- Why there is such limitation?
  - Because the energy conservation equation of the mixture is solved in terms of temperature
  - For calculating phase change energy equation for each species must be solved in terms of enthalpy

Mathematical Formulation

- Mass Conservation
  - Mixture Mass
  - Concentration of Species
- Momentum Conservation
  - Mixture Momentum
- Energy Conservation
  - Temperature option
    - Energy Conservation is formulated in terms of temperature
    - Applicable for gas mixture
  - Enthalpy option
    - Separate Energy Equations are solved for Individual Species
    - Applicable for liquid-gas mixture with phase change

Separate Energy Equation for Individual Species (SEEIS)



$$\frac{\left(m_i h_{ik} - \frac{p}{\rho_k J}\right)_{\tau+\Delta\tau} - \left(m_i h_{ik} - \frac{p}{\rho_k J}\right)_{\tau}}{\Delta\tau}$$

Transient Term

$$= \sum_{j=1}^{j=n} \left\{ MAX\left[-\dot{m}_{ij}, 0\right] h_{jk} - MAX\left[\dot{m}_{ij}, 0\right] h_{ik} \right\} + \dot{Q}_{ik} + \left\{ \pm \dot{Q}_{1\to 2}^{HES} \right\}$$

Advection Term *Source Term*

Thermodynamic Properties

- Temperature and other properties of individual species is calculated from node pressure and enthalpy of the species:

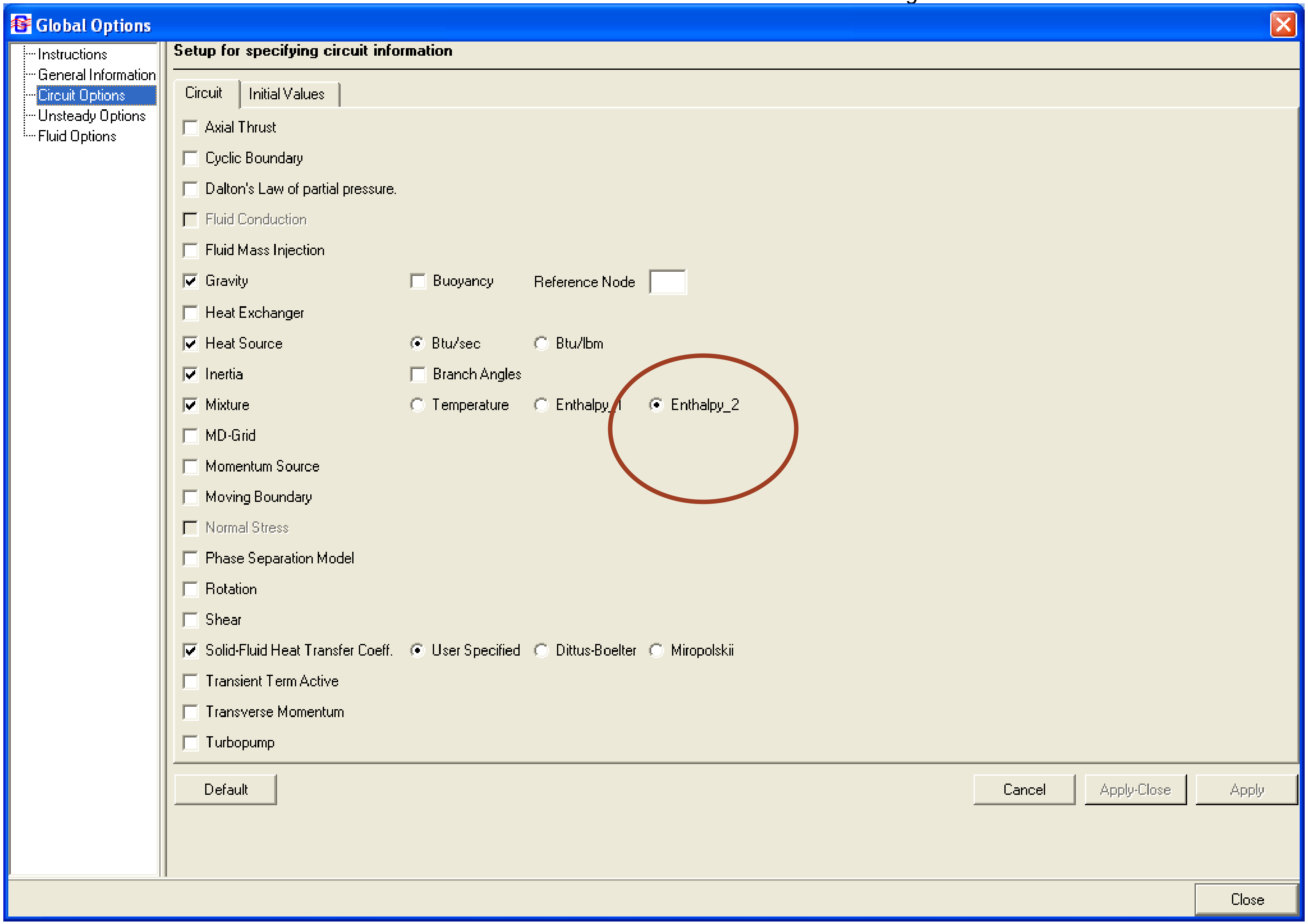
$$T_{ik} = f\left(p_i, h_{ik}\right)$$
$$\rho_{ik} = f\left(p_i, h_{ik}\right)$$
$$\mu_{ik} = f\left(p_i, h_{ik}\right)$$
$$K_{ik} = f\left(p_i, h_{ik}\right)$$
$$C_{p_{ik}} = f\left(p_i, h_{ik}\right)$$

- The nodal properties are calculated by averaging the properties of species:

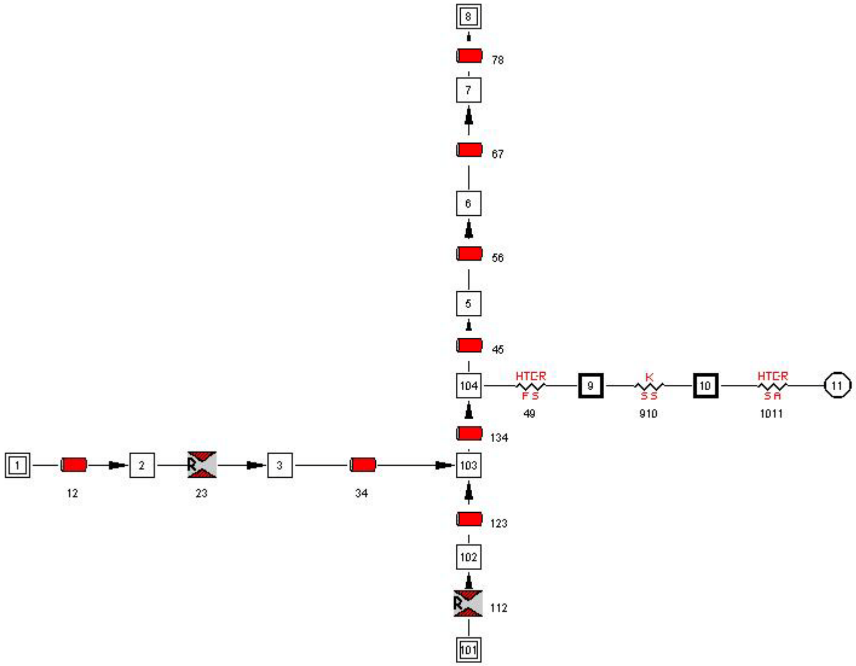
$$\rho_i = \sum_{k=1}^{n_f} \bar{c}_{ik}\ \rho_{ik}$$

$$\mu_i = \sum_{k=1}^{n_f} \bar{c}_{ik}\ \mu_{ik}$$

Select Mixture in Circuit Option

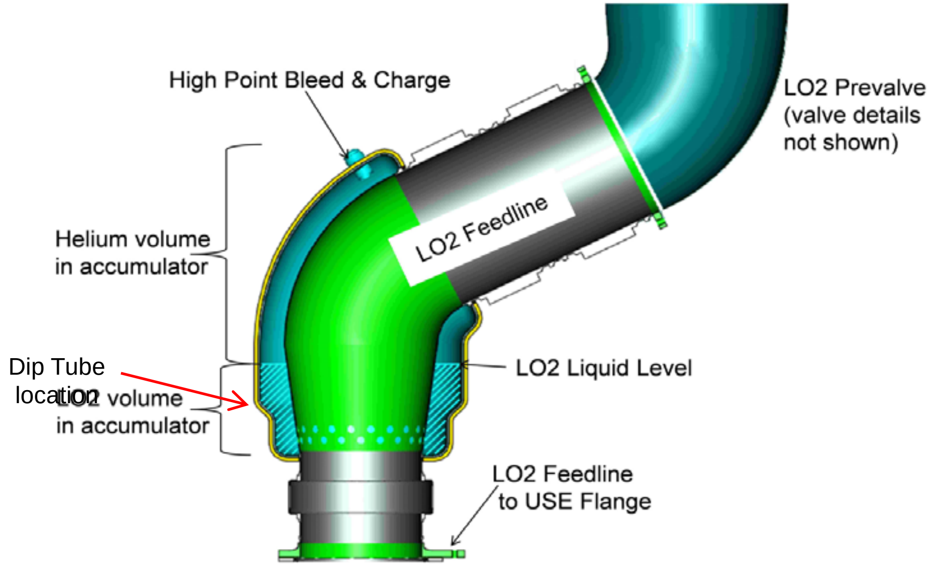


Example 23 – Helium-Assisted Buoyancy-Driven Flow in a Vertical Pipe Carrying Liquid Oxygen with Ambient Heat Leak



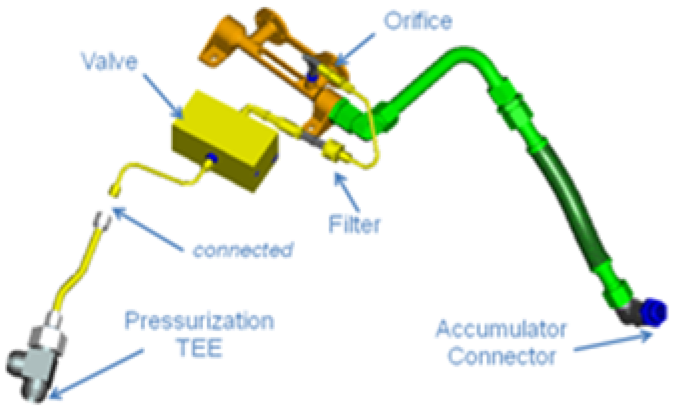
Feature: Phase Change in Fluid Mixture, Buoyancy Driven Flow, Model Import and Conjugate Heat Transfer

Charging of Pogo Accumulator

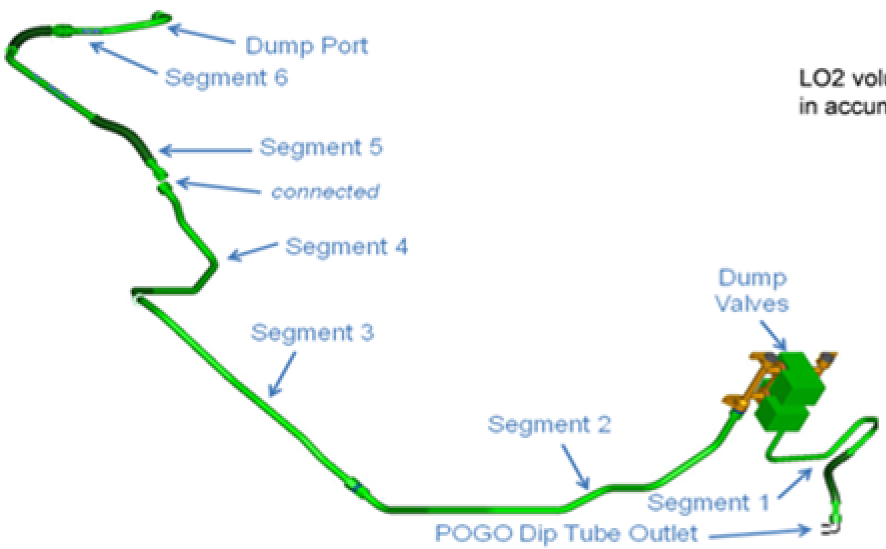


Problem Considered: An annular shaped pogo accumulator has been wrapped around a liquid oxygen feed line to a rocket engine turbopump. The lower portion of the accumulator communicates to feed line through a series of holes. The accumulator has a dip tube located a few inches above the communication port and connected to the dump line. The accumulator also has a high point bleed and charge port. Initially the accumulator is filled with liquid oxygen. Helium enters into the accumulator through high point bleed and charge port and displaces liquid oxygen to feed line and dump line. Once the liquid oxygen level drops to the location of dip tube, a stable helium bubble is retained in the accumulator that provides desired compliance. The objective of the model is to predict the history of charging process as well as the steady operation of the accumulator during engine run.

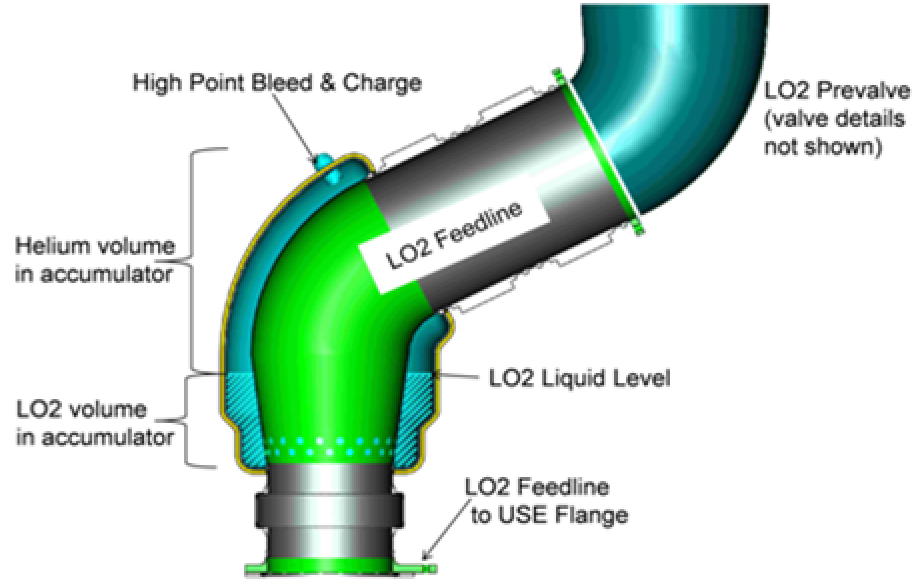
Pogo Accumulator with Charge Line and Dump line



Charge System

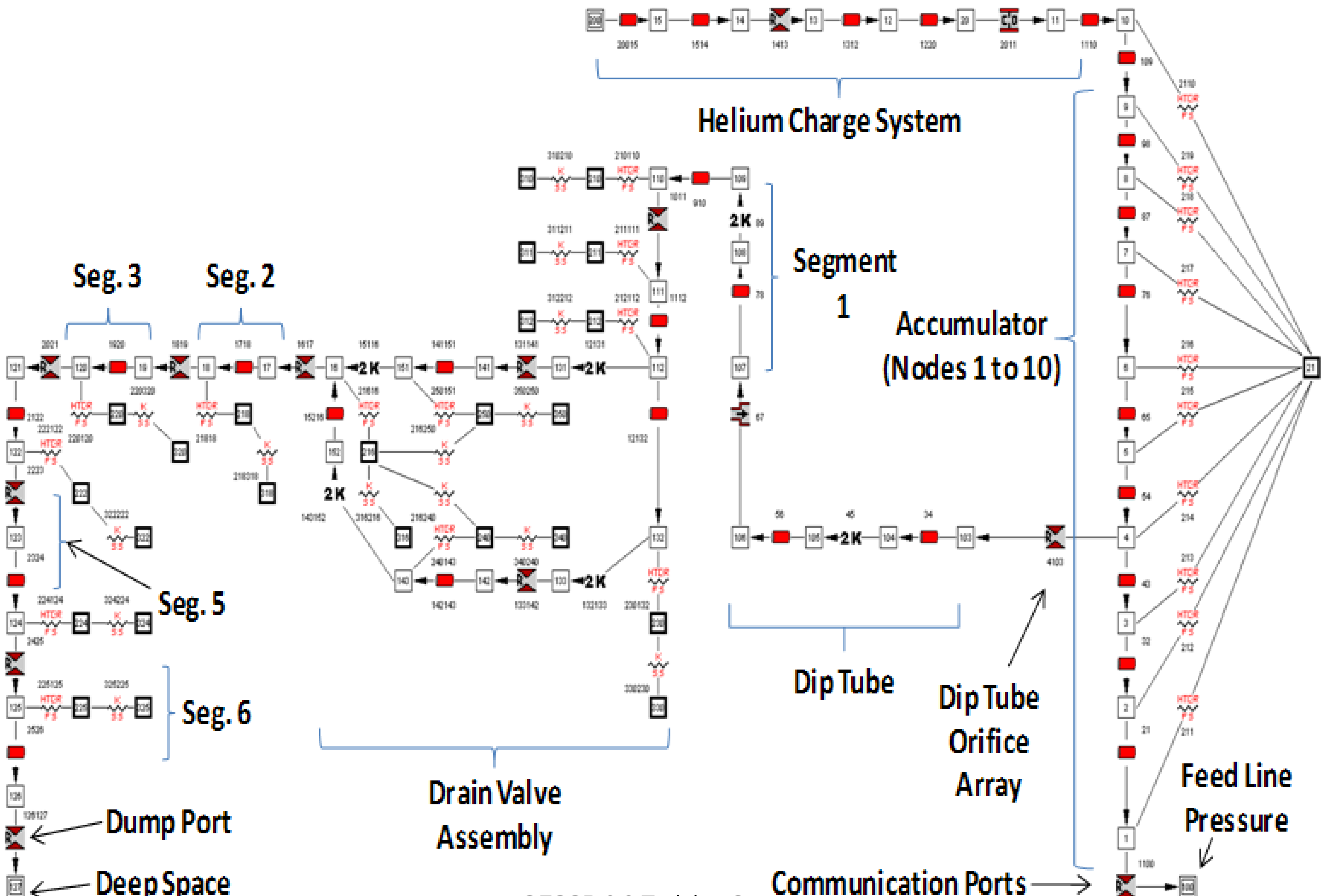


Dump System

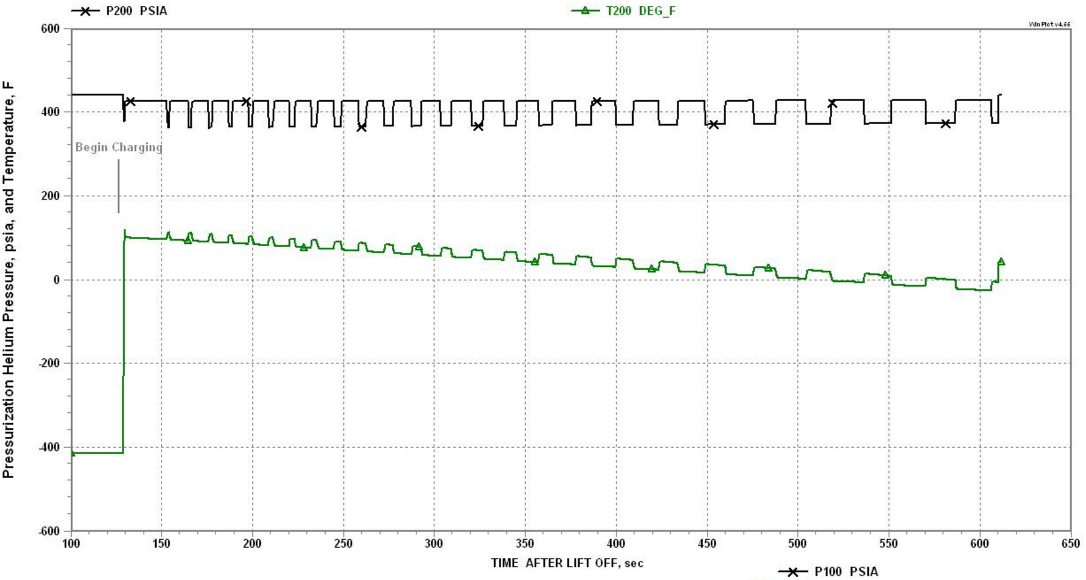


POGO Accumulator wrapped around LO2 Feed Line

GFSSP Model of Pogo Accumulator & Drain Line

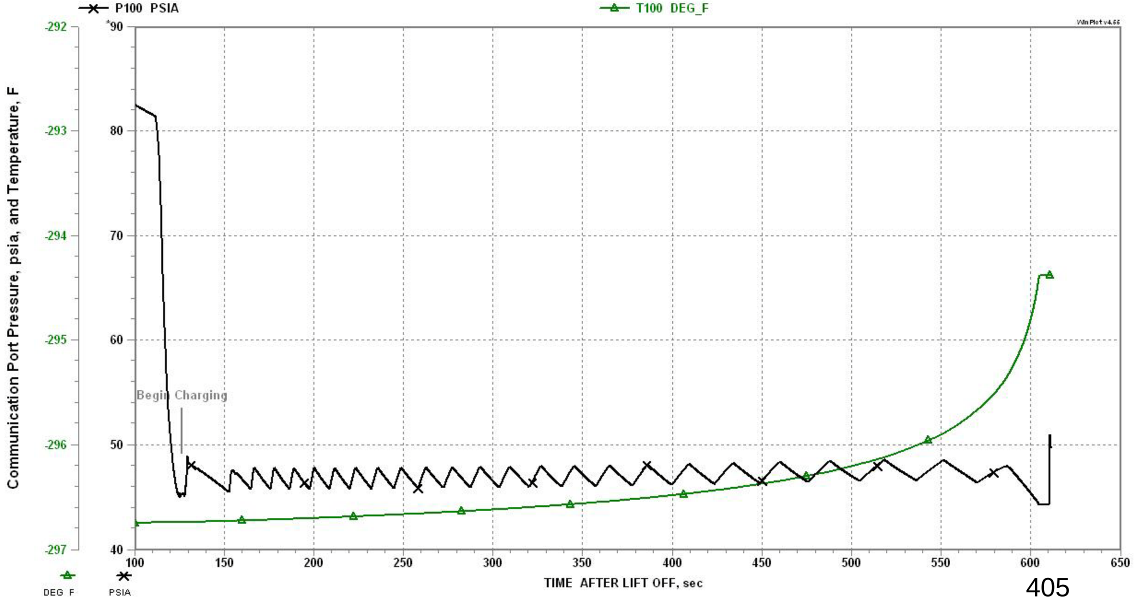


Boundary Conditions

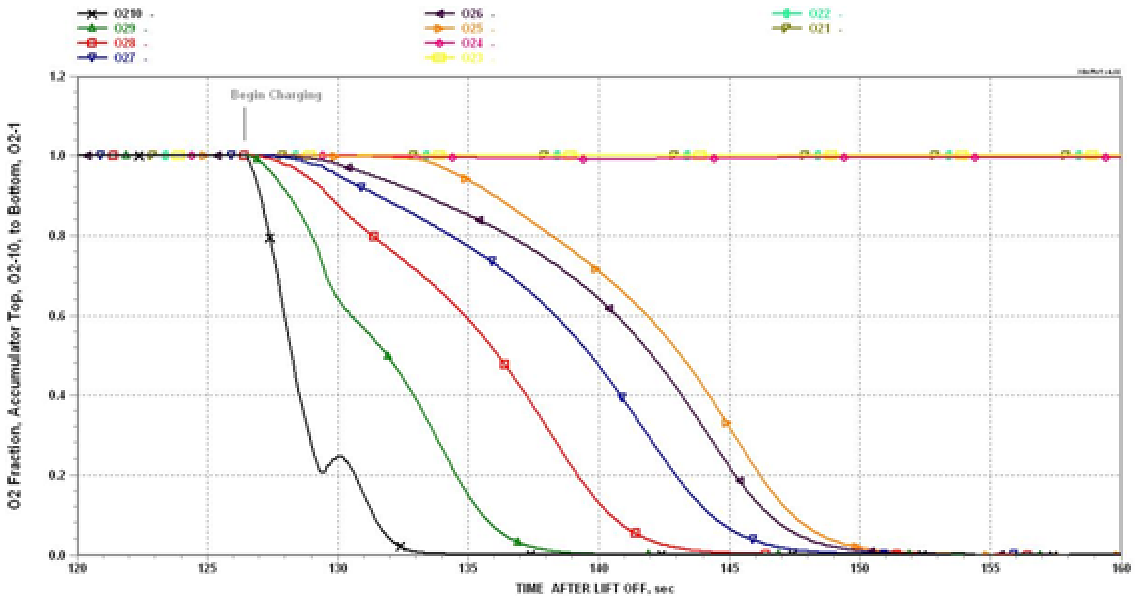


Pressure and Temperature history at the helium supply line

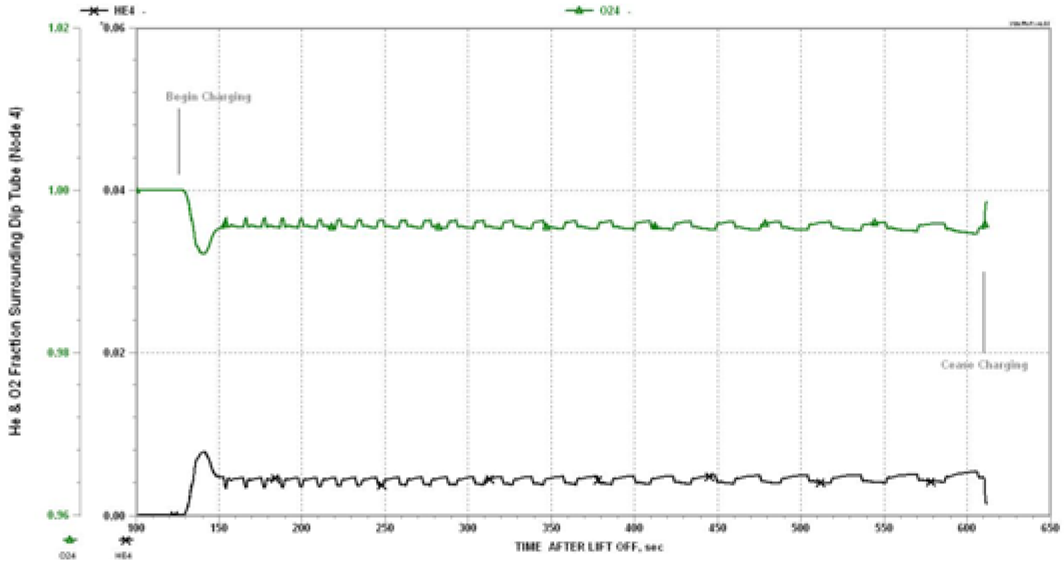
Pressure and Temperature history at the Communication Ports



Charging of helium & draining of He-LO2 mixture

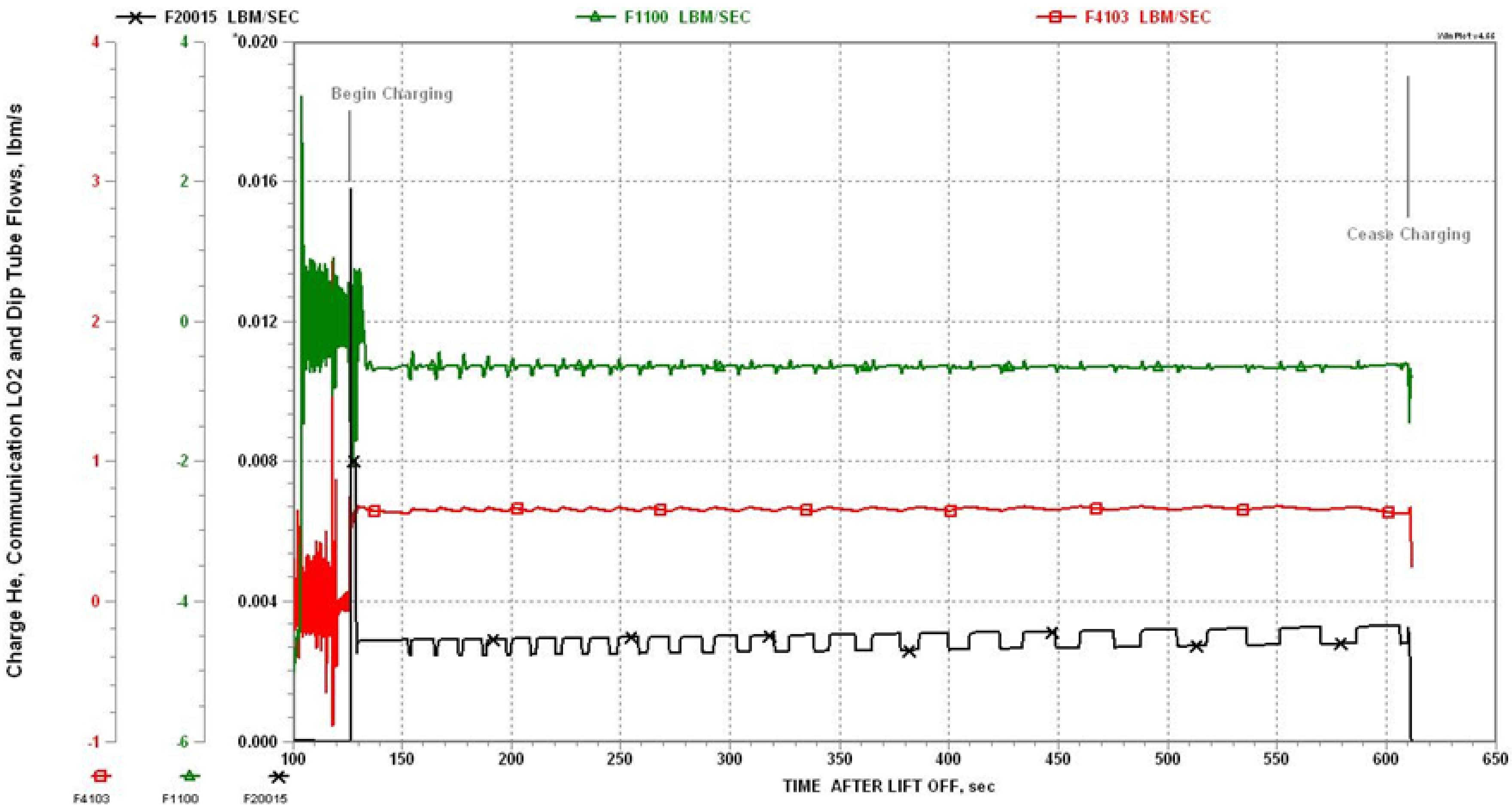


Displacement of Oxygen from the Accumulator during charging



Concentrations of Helium and Oxygen in the Dip Tube

Flowrates



Predicted Flowrates of (a) Helium into the Accumulator, (b) LO2 into the Accumulator, and (c) LO2 and Helium Mixture through the Dip Tube.

Summary

- An enthalpy based mixture formulation has been introduced in the code
- Energy equation for individual species has been solved to calculate enthalpy of each species
- Properties of individual species are computed from pressure and enthalpy
- Properties of the mixture are evaluated by averaging properties of the species
- New formulation allows calculation of phase change of species

Tutorial – 6

Modeling an Oxygen Recirculation Line

In this multi-step project, you will:

•Model LOx sitting stagnant in a vertical recirculation line

•Evaluate the effect of heat transfer on the flowrate

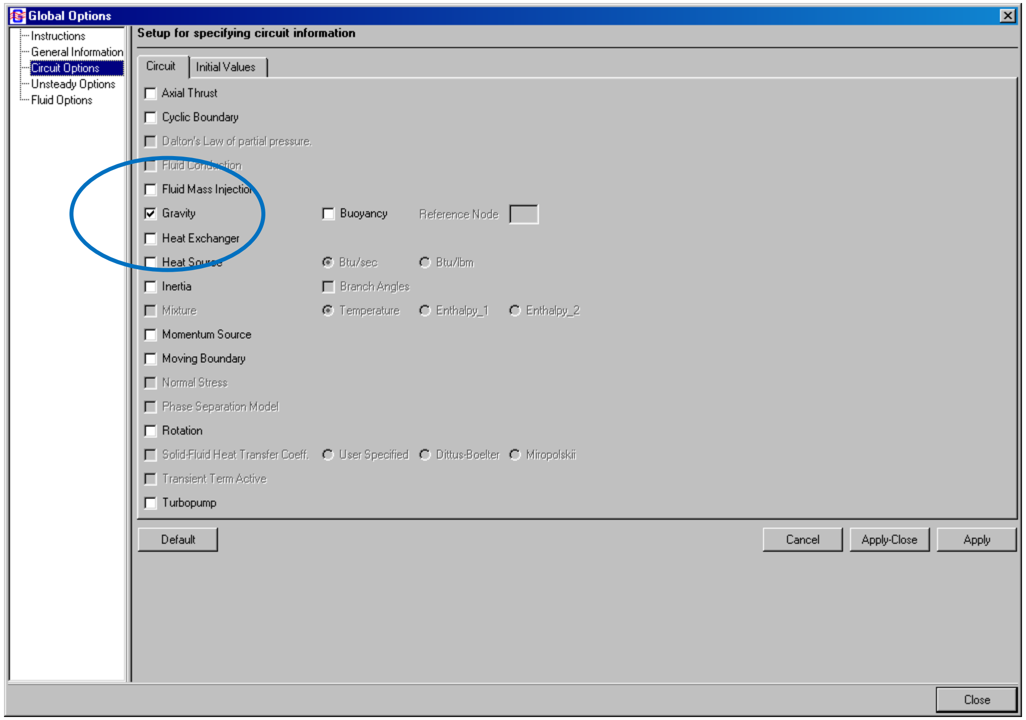
•Model a helium injector

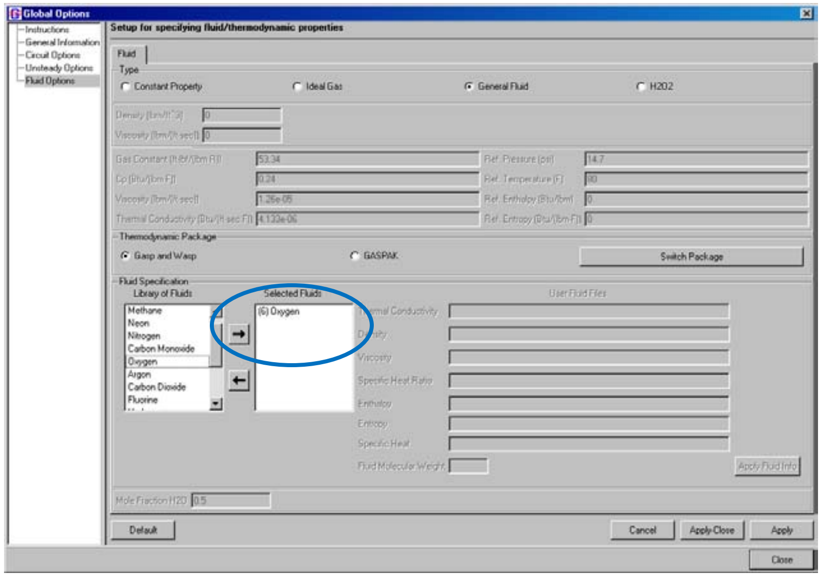
•Use VTASC's import option to combine the LOx recirculation and helium injector models

•Evaluate the LOx flowrate when both heat transfer and helium bubbling are applied

Part 1: LOx Sitting Stagnant

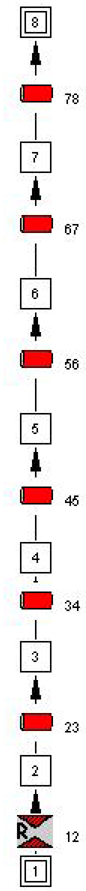
- VTASC File: Tut6.vts
- User Information
    - Input File: Tut6.dat
    - Output File: Tut6.out
- Circuit Options
    - Select Gravity
- Steady-State
- Fluid is Liquid Oxygen





Build Model on Canvas

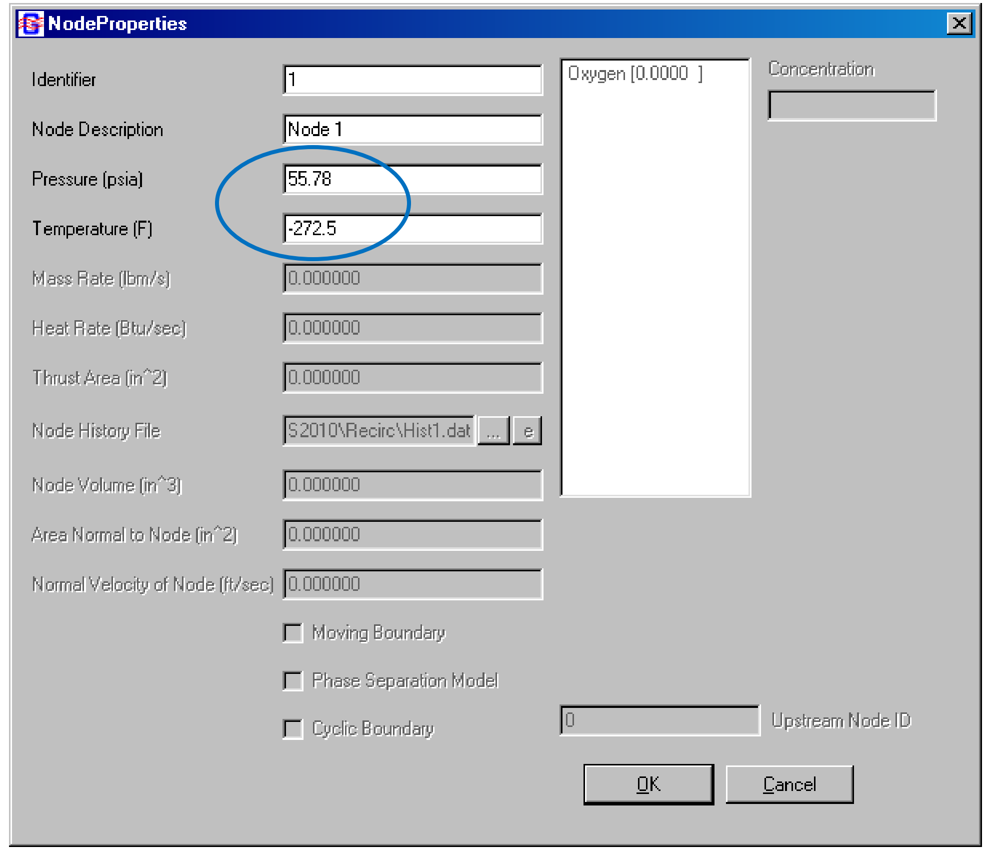
The recirculation line is simply a 6’ vertical smooth pipe of 1.87” diameter.



Now would be a good time to save your Tut6.vts file.

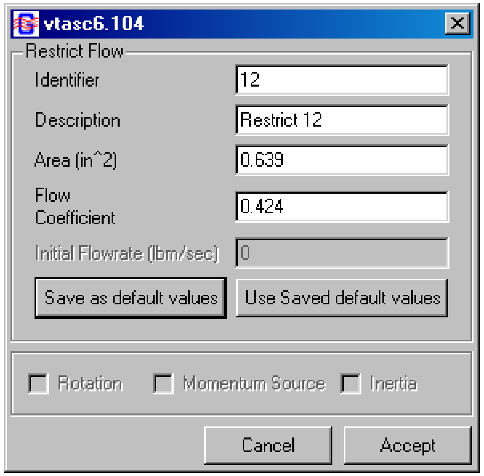
Set Up Steady-State Boundary Conditions

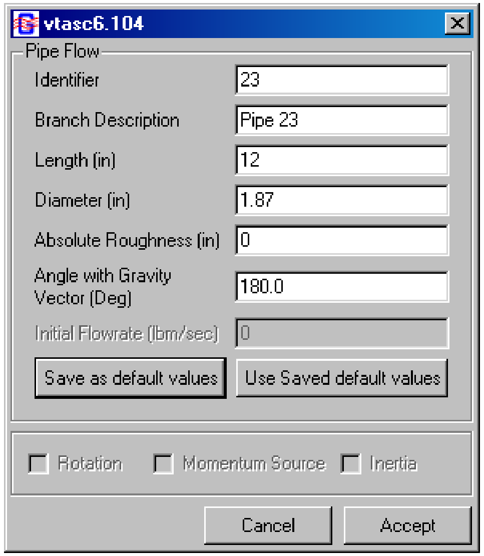
- Node 1:
  - P = 55.78 psia
  - T = -272.5 °F
- Node 8:
  - P = 53.0 psia
  - T = -272.5 °F
- The 2.78 psia ΔP corresponds to the hydrostatic head of 6 ft of Lox
- At this pressure, the LOx is approximately 1 °F subcooled.



Set Up Fluid Branches

- Branch 12: Inlet
  - $A = 0.639\ in^2$, $C_L = 0.424$
- Branch 23, 34, 45, 56, 67, 78: Pipes
  - L = 6 ft / 6 = 1 ft = 12 in
  - D = 1.87 in
  - Smooth pipe: $\varepsilon = 0$
  - Angle = 180° (vertical)





Result of Stagnant LOx Model

- Run the model
- Flow rate should be close to zero (< 0.01 lb/s)

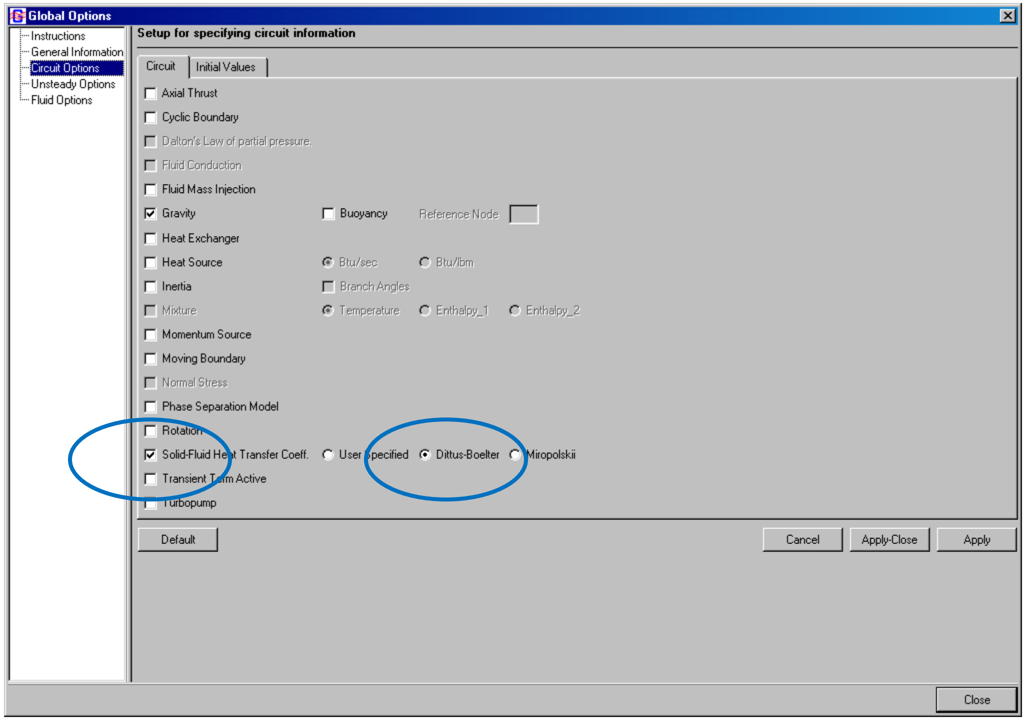
Tut6.out - Notepad

File Edit Format View Help

```
 SOLUTION
 INTERNAL  NODES
  NODE      P(PSI)       TF(F)         Z            RHO          EM(LBM)       QUALITY
                                                  (LBM/FT^3)
     2     0.5578E+02 -0.2725E+03  0.1332E-01  0.6674E+02  0.0000E+00  0.0000E+00
     3     0.5532E+02 -0.2725E+03  0.1321E-01  0.6673E+02  0.0000E+00  0.0000E+00
     4     0.5485E+02 -0.2725E+03  0.1310E-01  0.6673E+02  0.0000E+00  0.0000E+00
     5     0.5439E+02 -0.2725E+03  0.1299E-01  0.6673E+02  0.0000E+00  0.0000E+00
     6     0.5393E+02 -0.2725E+03  0.1288E-01  0.6673E+02  0.0000E+00  0.0000E+00
     7     0.5346E+02 -0.2725E+03  0.1277E-01  0.6673E+02  0.0000E+00  0.0000E+00

    NODE        H           ENTROPY      EMU        COND         CP           GAMA
              BTU/LB       BTU/LB-R   LBM/FT-SEC  BTU/FT-S-R  BTU/LB-R

     2     0.7133E+02  0.1525E+01  0.9488E-04  0.1882E-04  0.4194E+00  0.1944E+01
     3     0.7133E+02  0.1525E+01  0.9487E-04  0.1882E-04  0.4194E+00  0.1944E+01
     4     0.7133E+02  0.1525E+01  0.9487E-04  0.1882E-04  0.4194E+00  0.1944E+01
     5     0.7133E+02  0.1525E+01  0.9486E-04  0.1882E-04  0.4194E+00  0.1944E+01
     6     0.7133E+02  0.1525E+01  0.9486E-04  0.1881E-04  0.4195E+00  0.1944E+01
     7     0.7133E+02  0.1525E+01  0.9485E-04  0.1881E-04  0.4195E+00  0.1944E+01

 BRANCHES
BRANCH        KFACTOR         DELP           FLOW RATE    VELOCITY     REYN. NO.    MACH NO.     ENTROPY GEN.    LOST WORK
      (LBF-S^2/(LBM-FT)^2) (PSI)           (LBM/SEC)      (FT/SEC)                                BTU/(R-SEC)     LBF-FT/SEC
     12         0.658E+02     -0.653E-04   -0.951E-02     0.694E-01    0.170E+04    0.924E-04    0.582E-11    0.848E-06
     23         0.321E+00      0.463E+00   -0.951E-02     0.162E-01    0.819E+03    0.215E-04    0.284E-13    0.414E-08
     34         0.321E+00      0.463E+00   -0.951E-02     0.162E-01    0.819E+03    0.215E-04    0.284E-13    0.414E-08
     45         0.321E+00      0.463E+00   -0.951E-02     0.162E-01    0.819E+03    0.215E-04    0.284E-13    0.414E-08
     56         0.321E+00      0.463E+00   -0.951E-02     0.162E-01    0.819E+03    0.215E-04    0.284E-13    0.414E-08
     67         0.321E+00      0.463E+00   -0.951E-02     0.162E-01    0.819E+03    0.215E-04    0.284E-13    0.414E-08
     78         0.321E+00      0.463E+00   -0.951E-02     0.162E-01    0.819E+03    0.215E-04    0.284E-13    0.414E-08

 **************************************************

 TIME OF ANALYSIS WAS   1.562500000000000E-002  SECS

 **************************************************
```

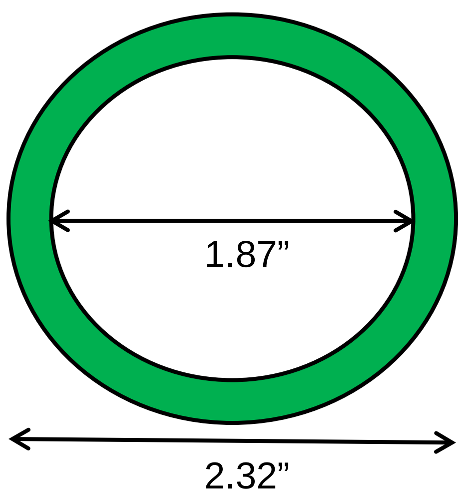
Part 2: Add Heat Transfer to Model

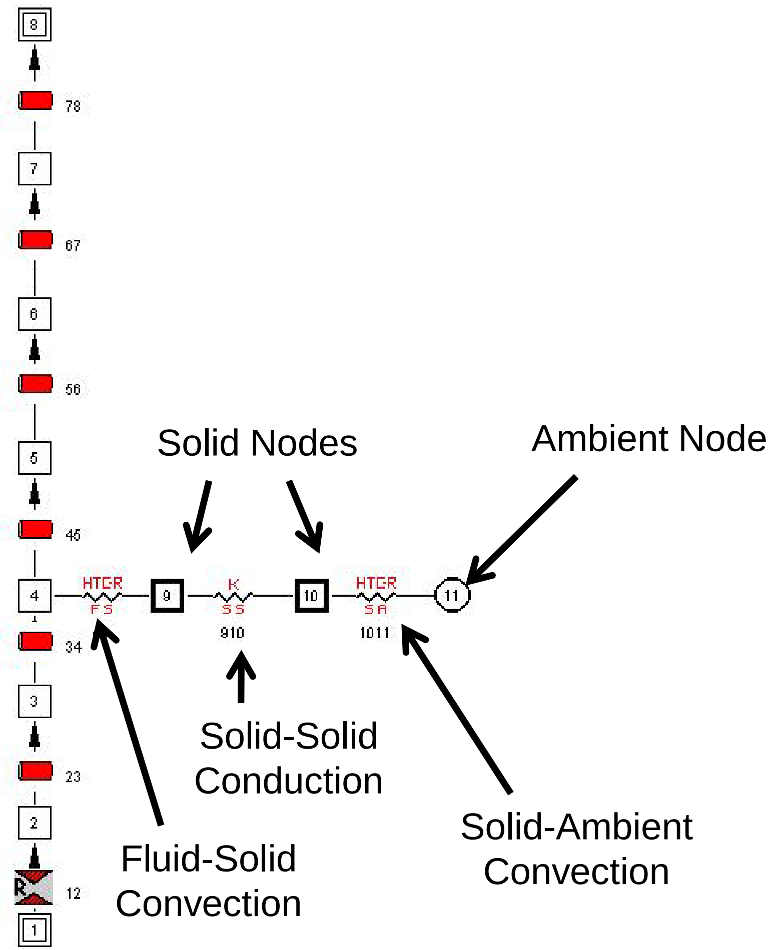
- Select Advanced/Enable Conjugate Heat Transfer
- Circuit Options
    - Add Dittus-Boelter heat transfer correlation



Update Model on Canvas

Allow 1' of the pipe to be exposed to ambient. The rest remains insulated.





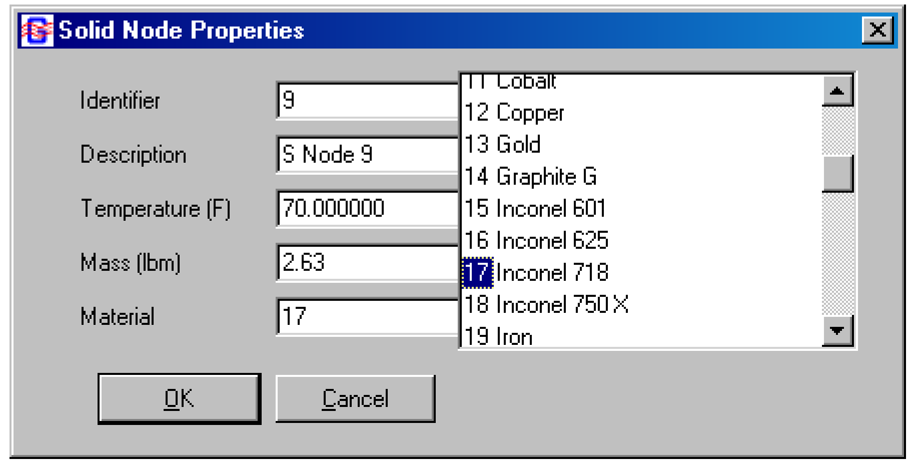
Set Up Solid Nodes

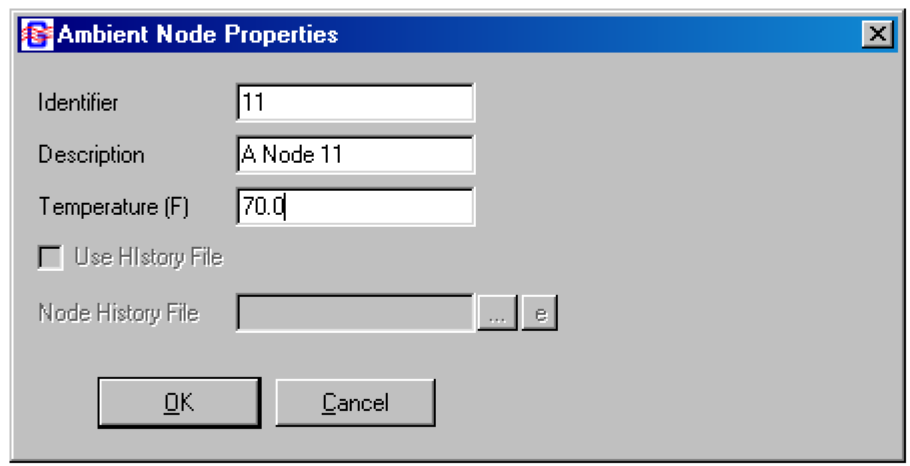
- Exposed pipe section is 5.26 lb_m of Inconel 718

$$m = \rho\left[\frac{\pi}{4}\left(OD^2 - ID^2\right)L\right]$$

$$= \left(0.296\frac{lb}{in^3}\right)\left[\frac{\pi}{4}\left(\left(2.32in\right)^2 - \left(1.87in\right)^2\right)\left(12in\right)\right] = 5.26lb$$

- Nodes 9 & 10
    - Guess T = 70 °F
    - Mass = 5.26 lb_m / 2 = 2.63 lb_m
    - Material 17
- Ambient Node 11
    - T_{amb} = 70 °F





Set Up Conductors

- Fluid-Solid Convection
  - Wetted Area:

$$A = \pi DL = \pi(1.87\ in)(12\ in) = 70.5\ in^2$$

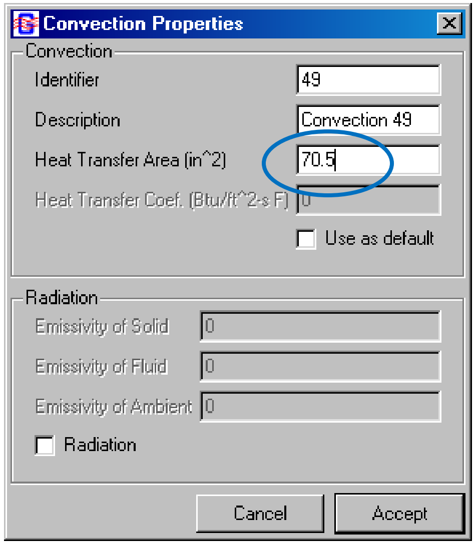
  - GFSSP will calculate h by Dittus-Boelter correlation
- Solid-Solid Conduction
  - “Average” Area:

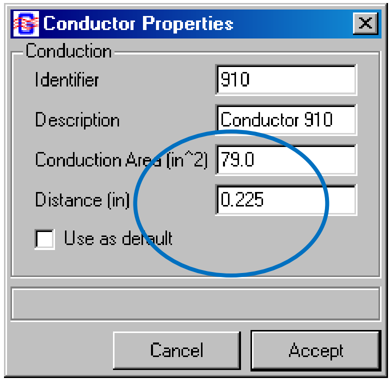
$$A = \pi D_{avg} L = \pi(2.095in)(12in) = 79.0in^2$$

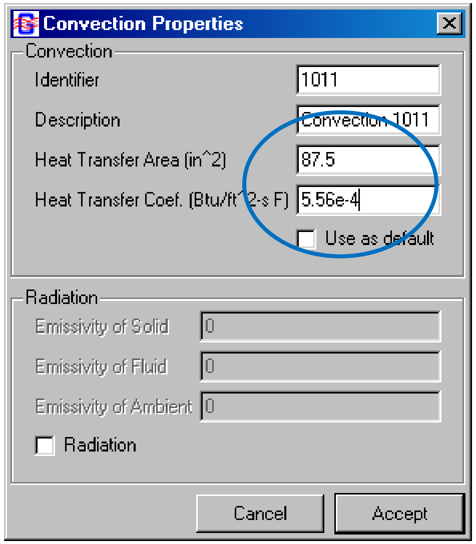
  - Distance (pipe thickness): 0.225 in
- Solid-Ambient Convection
  - Exposed Area:

$$A = \pi DL = \pi(2.32\ in)(12\ in) = 87.5\ in^2$$

  - Natural convection:
    - h = 2 BTU/hr-ft^2-°F
      = 5.56×10^{-4} BTU/s-ft^2-°F







Result of LOx Model with Heat Input

Marshall Space Flight Center
GFSSP Training Course

- Rerun the model
- Addition of 0.106 BTU/s of heat has increased flow rate to 0.172 lb/s

Tut6.out - Notepad

File Edit Format View Help

```
 SOLUTION
 INTERNAL  NODES
  NODE      P(PSI)      TF(F)          Z            RHO          EM(LBM)       QUALITY
                                                 (LBM/FT^3)
     2    0.5577E+02 -0.2725E+03  0.1332E-01  0.6667E+02  0.0000E+00  0.0000E+00
     3    0.5530E+02 -0.2725E+03  0.1321E-01  0.6667E+02  0.0000E+00  0.0000E+00
     4    0.5484E+02 -0.2710E+03  0.1305E-01  0.6639E+02  0.0000E+00  0.0000E+00
     5    0.5438E+02 -0.2710E+03  0.1294E-01  0.6639E+02  0.0000E+00  0.0000E+00
     6    0.5392E+02 -0.2711E+03  0.1320E-01  0.6546E+02  0.0000E+00  0.4100E-03
     7    0.5346E+02 -0.2713E+03  0.1402E-01  0.6331E+02  0.0000E+00  0.1425E-02

    NODE        H          ENTROPY       EMU         COND         CP          GAMA
              BTU/LB       BTU/LB-R   LBM/FT-SEC  BTU/FT-S-R   BTU/LB-R

     2    0.7134E+02  0.1525E+01  0.9487E-04  0.1882E-04  0.4194E+00  0.1944E+01
     3    0.7134E+02  0.1525E+01  0.9486E-04  0.1882E-04  0.4194E+00  0.1944E+01
     4    0.7196E+02  0.1525E+01  0.9294E-04  0.1871E-04  0.4199E+00  0.1959E+01
     5    0.7196E+02  0.1525E+01  0.9293E-04  0.1871E-04  0.4199E+00  0.1959E+01
     6    0.7196E+02  0.1525E+01  0.9246E-04  0.1871E-04  0.4198E+00  0.1959E+01
     7    0.7196E+02  0.1525E+01  0.9131E-04  0.1870E-04  0.4196E+00  0.1956E+01

 BRANCHES
BRANCH         KFACTOR        DELP        FLOW RATE    VELOCITY    REYN. NO.    MACH NO.    ENTROPY GEN.    LOST WORK
      (LBF-S^2/(LBM-FT)^2)   (PSI)        (LBM/SEC)    (FT/SEC)                             BTU/(R-SEC)     LBF-FT/SEC
     12         0.658E+02     0.185E-01    0.172E+00    0.727E+00    0.308E+05    0.966E-03    0.346E-07    0.504E-02
     23         0.115E+00     0.462E+00    0.172E+00    0.169E+00    0.148E+05    0.225E-03    0.604E-10    0.880E-05
     34         0.115E+00     0.461E+00    0.172E+00    0.169E+00    0.148E+05    0.225E-03    0.604E-10    0.880E-05
     45         0.115E+00     0.459E+00    0.172E+00    0.170E+00    0.151E+05    0.225E-03    0.602E-10    0.883E-05
     56         0.115E+00     0.459E+00    0.172E+00    0.170E+00    0.151E+05    0.225E-03    0.602E-10    0.883E-05
     67         0.116E+00     0.459E+00    0.172E+00    0.170E+00    0.152E+05    0.225E-03    0.618E-10    0.907E-05
     78         0.120E+00     0.461E+00    0.172E+00    0.171E+00    0.154E+05    0.225E-03    0.660E-10    0.966E-05

 SOLID NODES
  NODESL      CPSLD        TS
             BTU/LB F       F
     9    0.000E+00   -0.248E+03
    10    0.000E+00   -0.245E+03

 SOLID TO SOLID CONDUCTOR
  ICONSS      CONDKIJ       QDOTSS
            BTU/S FT F      BTU/S
   910    0.135E-02   -0.106E+00

 SOLID TO FLUID CONDUCTOR
 ICONSF     QDOTSF        HCSF            HCSFR
             BTU/S       BTU/S FT**2 F
    49    0.106E+00    0.925E-02      0.000E+00
```

Efficiency of Heat Leak as Pump

- Use the values from the output file to determine the following:
- Pump Power:

Flowrate → $\left(?\frac{lb}{s}\right)$; **P1 – P8** → $\left(?\frac{lb_f}{in^2}\right)$

$$\dot{W} = \frac{\dot{m}\Delta P}{\rho} = \frac{\left(?\frac{lb}{s}\right)\left(?\frac{lb_f}{in^2}\right)\left(144\frac{in^2}{ft^2}\right)}{\left(66.4\frac{lb}{ft^3}\right)\left(778\frac{lb_f - ft}{BTU}\right)} = ?\frac{BTU}{s}$$

- Heat Input: $\dot{Q} = ?\frac{BTU}{s}$

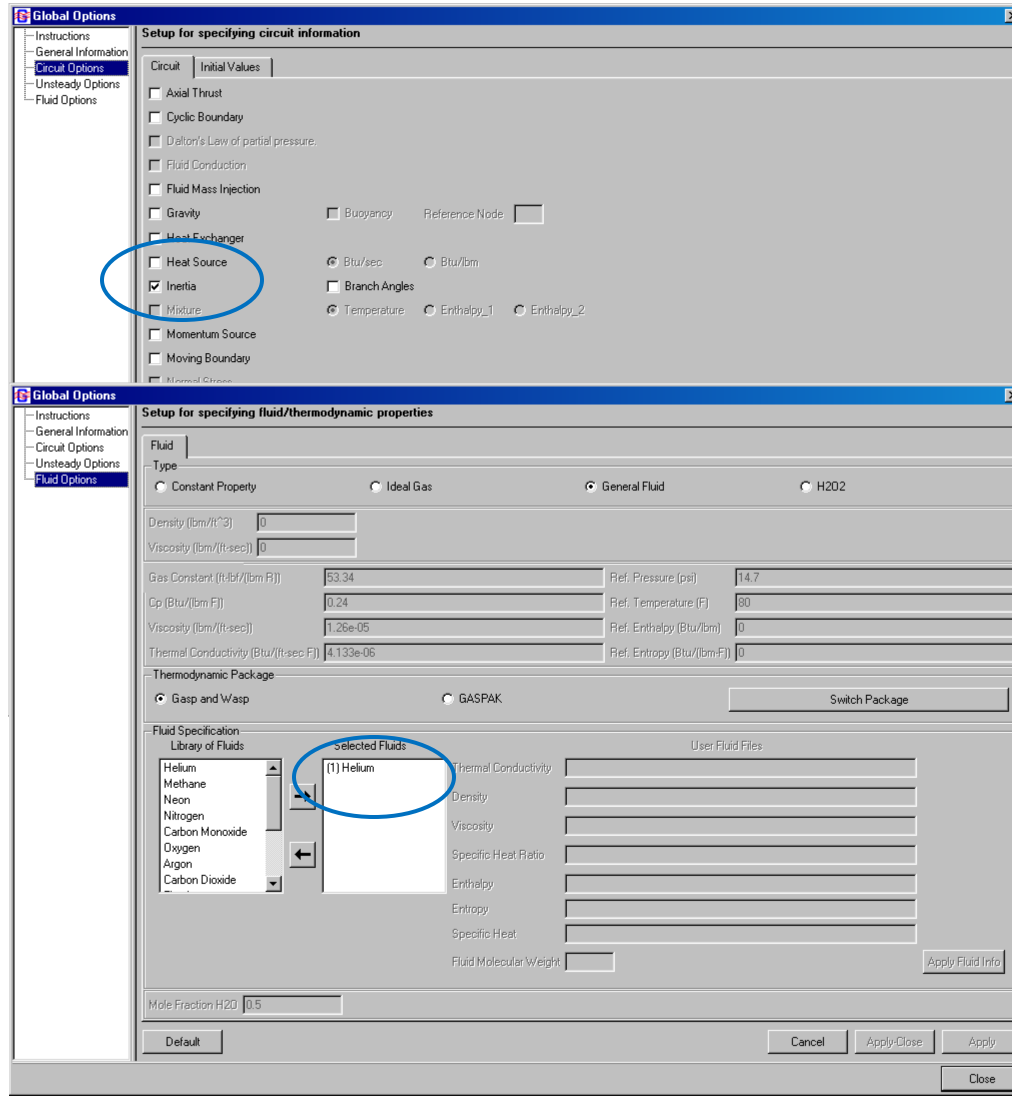
- Efficiency: $\eta = \frac{\dot{W}}{\dot{Q}} =$

- Carnot Efficiency:

$$\eta = 1 - \frac{T_C\left(°R\right)}{T_H\left(°R\right)} =$$

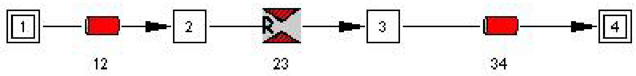
Part 3: Model a Helium Injector

- Save current tut6.vts file
- Start a new file tut6a.vts
- User information
  - Input file: Tut6a.dat
  - Output file: Tut6a.out
- Circuit options
  - Inertia
- Steady-state
- Fluid is helium



GFSSP 6.04 -- Tutorial 6

Build Model on Canvas



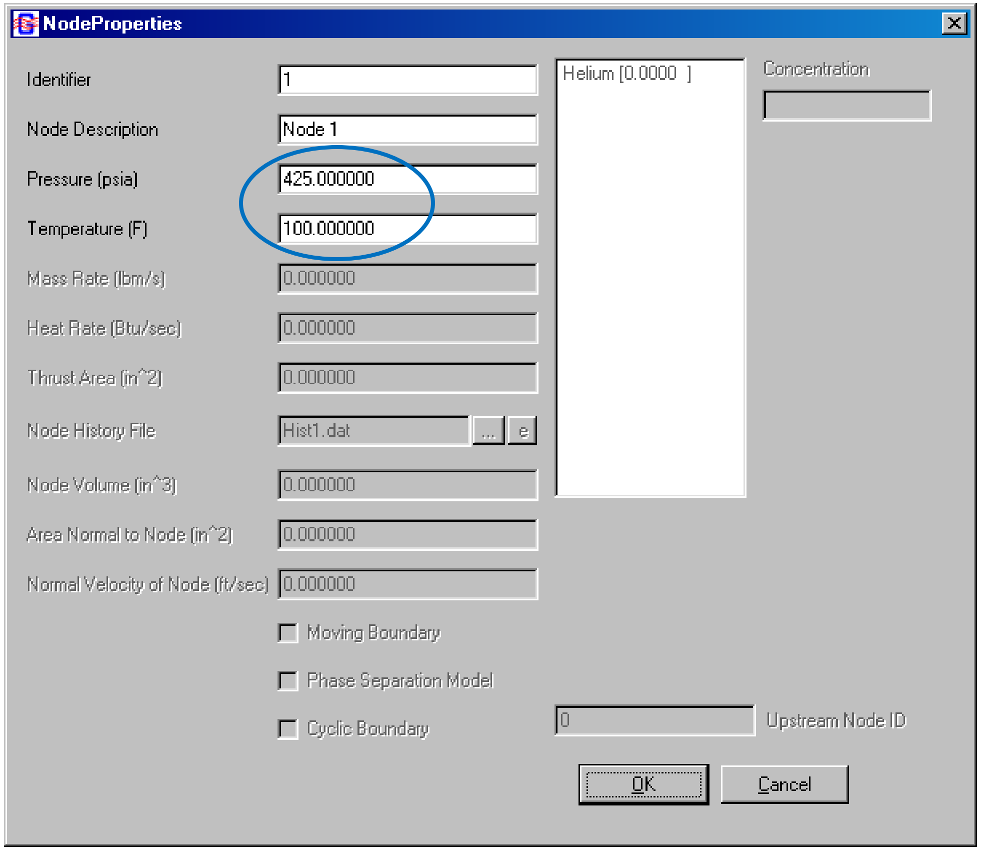
High Pressure
Helium Supply

Ambient

Now would be a good time to save your Tut6a.vts file.

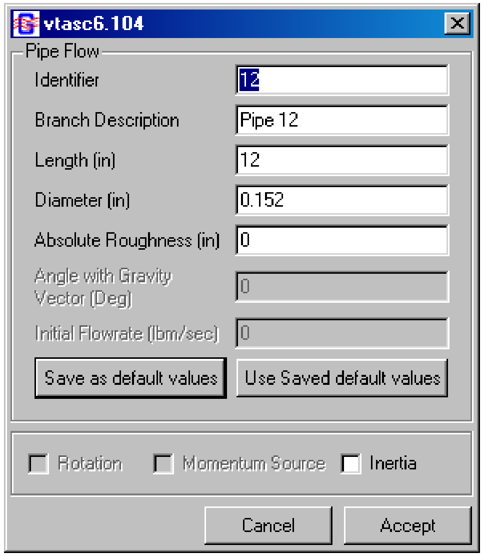
Set Up Steady-State Boundary Conditions

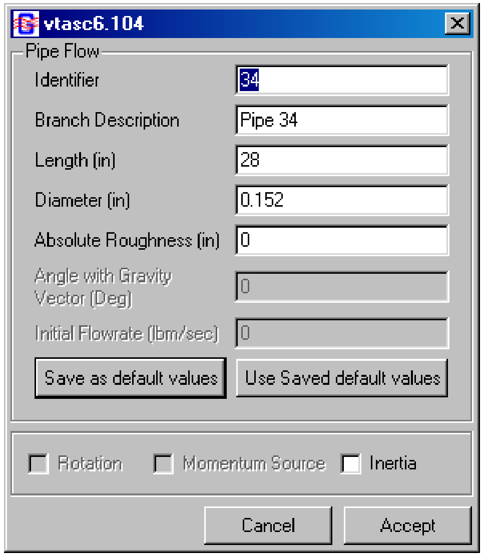
- Node 1:
  - P = 425 psia
  - T = 100 °F
- Node 4:
  - P = 14.7 psia
  - T = 60 °F

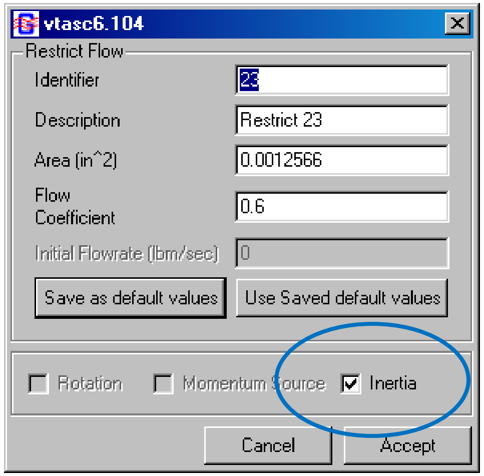


Set Up Fluid Branches

- Branch 12: Pipe
  - L = 12 in
  - D = 0.152 in
  - Smooth pipe: $\varepsilon = 0$
- Branch 23: Restrction
  - A = 0.0012566 in^2
  - $C_L = 0.6$
  - Inertia box checked
- Branch 34: Pipe
  - L = 28 in
  - D = 0.152 in
  - Smooth pipe: $\varepsilon = 0$







Result of Helium Injector Model

- Run the model
- The injector can provide 0.00163 lb/s of helium to ambient pressure

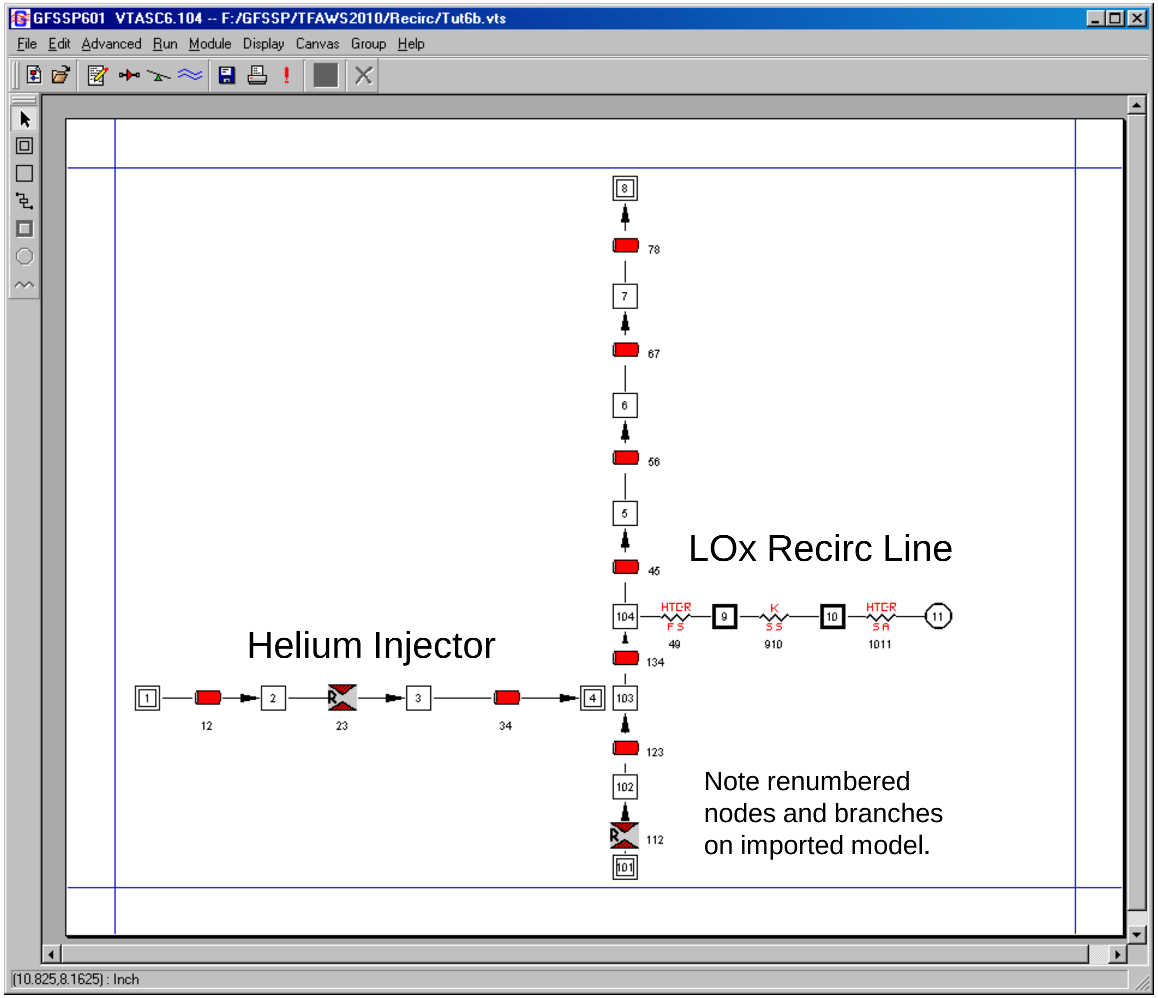
Tut6a.out - Notepad

File Edit Format View Help

```
 SOLUTION
 INTERNAL  NODES
  NODE      P(PSI)       TF(F)        Z            RHO          EM(LBM)      QUALITY
                                                (LBM/FT^3)
     2     0.4248E+03   0.1000E+03   0.1017E+01   0.2785E+00   0.0000E+00   0.1000E+01
     3     0.2164E+02   0.1033E+03   0.1001E+01   0.1432E-01   0.0000E+00   0.1000E+01

    NODE         H           ENTROPY      EMU          COND         CP           GAMA
              BTU/LB       BTU/LB-R   LBM/FT-SEC   BTU/FT-S-R   BTU/LB-R

     2     0.7052E+03   0.7460E+01   0.1415E-04   0.2598E-04   0.1243E+01   0.1669E+01
     3     0.7052E+03   0.7460E+01   0.1415E-04   0.2598E-04   0.1241E+01   0.1667E+01

 BRANCHES
BRANCH        KFACTOR         DELP         FLOW RATE    VELOCITY     REYN. NO.    MACH NO.     ENTROPY GEN.    LOST WORK
      (LBF-S^2/(LBM-FT)^2) (PSI)         (LBM/SEC)    (FT/SEC)                               BTU/(R-SEC)    LBF-FT/SEC
      12          0.824E+07     0.153E+00    0.163E-02    0.466E+02    0.116E+05    0.137E-01    0.296E-06    0.129E+00
      23          0.204E+10     0.403E+03    0.163E-02    0.673E+03    0.441E+05    0.197E+00    0.733E-04    0.319E+02
      34          0.374E+09     0.694E+01    0.163E-02    0.907E+03    0.116E+05    0.266E+00    0.260E-03    0.114E+03

 *************************************************

 TIME OF ANALYSIS WAS    0.000000000000000E+000   SECS

 *************************************************
```

Part 4: Combine He Injector with LOx Recirc Line

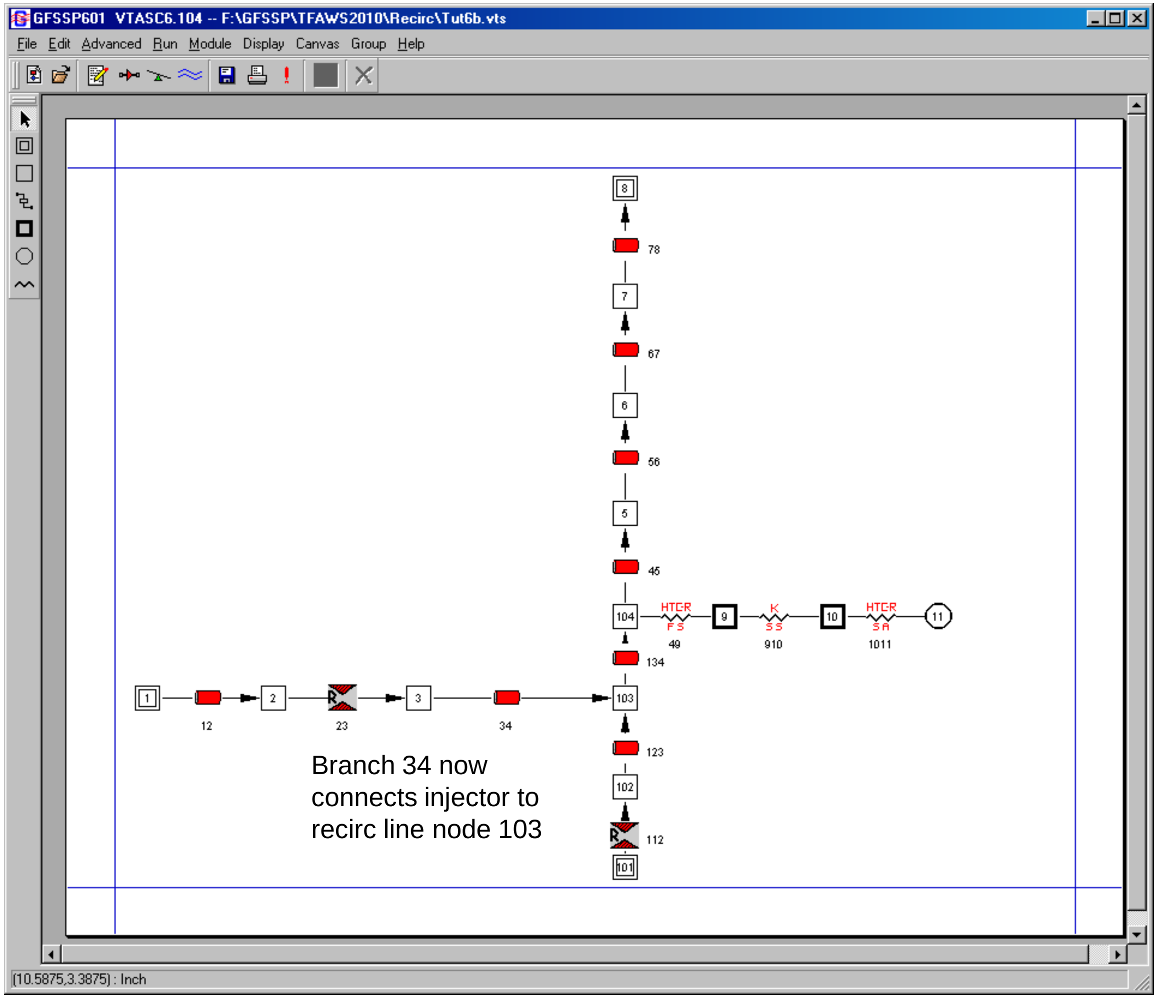
- <u>Save</u> current tut6a.vts file, then
- <u>Save As</u> tut6b.vts
- Select <u>File/Import Model</u>
  - Select LOx Recirculation Line model (tut6.vts)
  - To prevent conflict with existing nodes/branches, select an <u>offset of 100</u>
- Rearrange the canvas as necessary*



*HINT: To move a group of nodes/branches, select multiple items by holding down the CTRL key, then drag them across the canvas while holding down the SHIFT key.

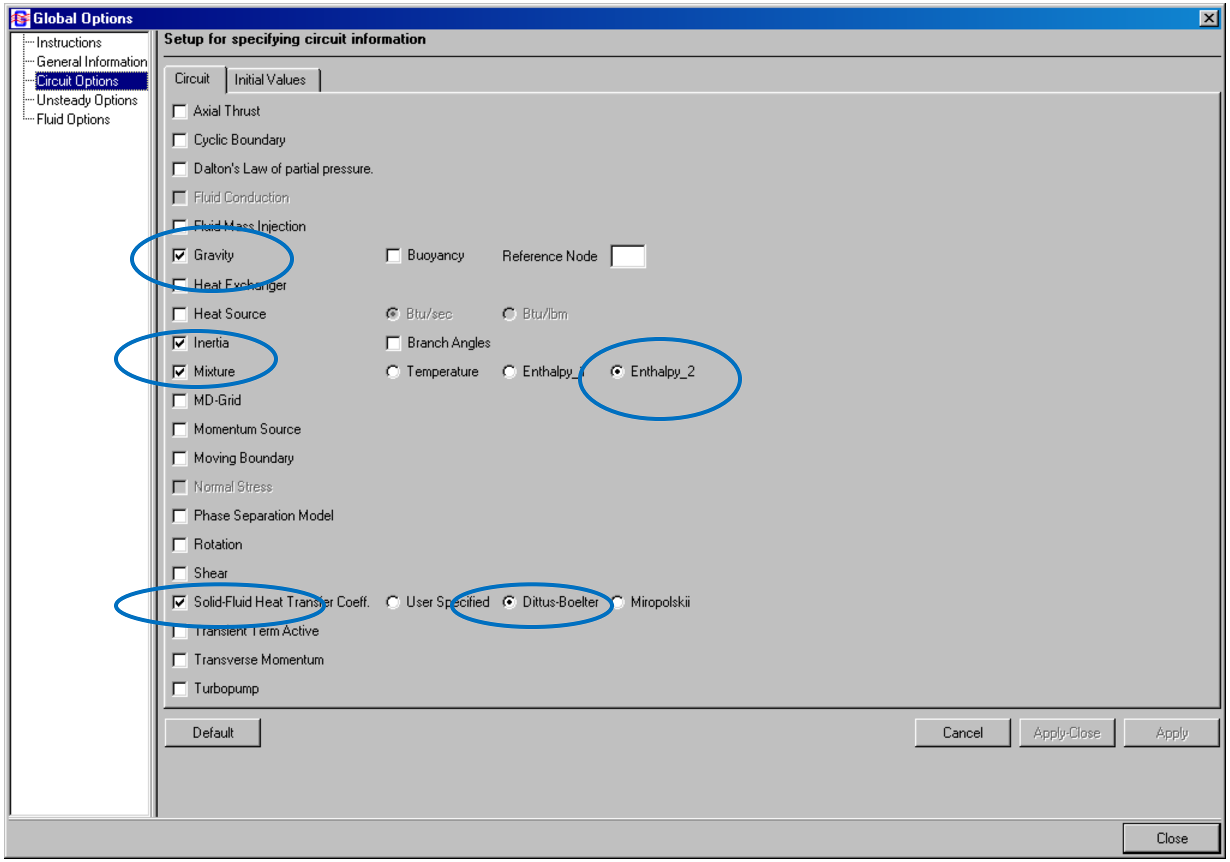
Connect Injector to Recirc Line

- Right-click Injector branch 34
- Select Change Branch Connection
- Change downstream node to Recirc Line node 103
- Delete Injector node 4



Set Up User Options

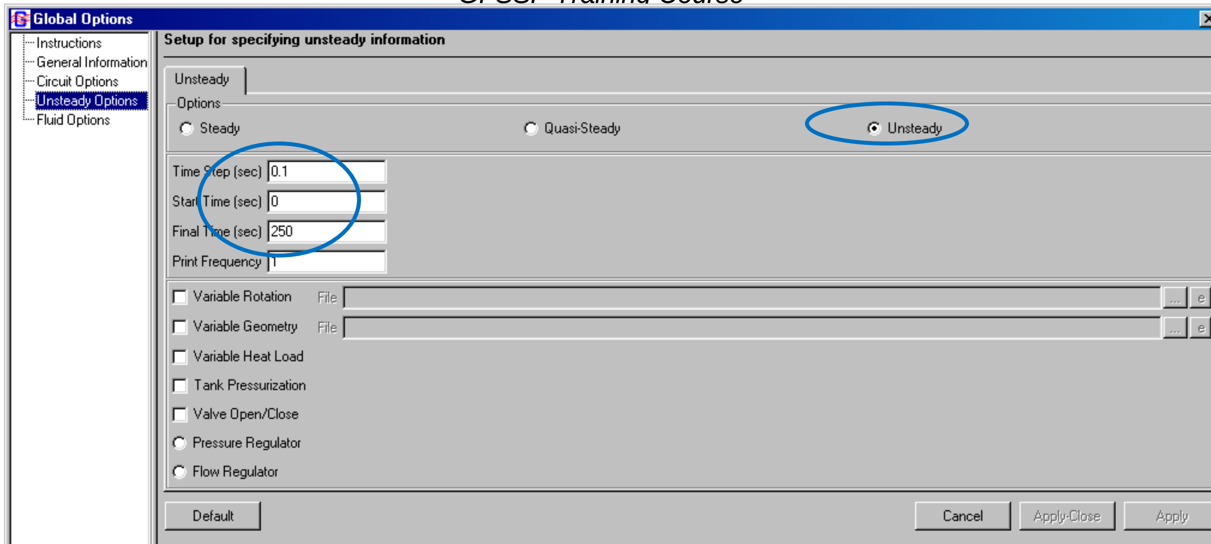
- User information
  - Input file: Tut6b.dat
  - Output file: Tut6b.out
- Circuit options*
  - Gravity
  - Inertia
  - Mixture: Enthalpy2
  - Dittus-Boelter

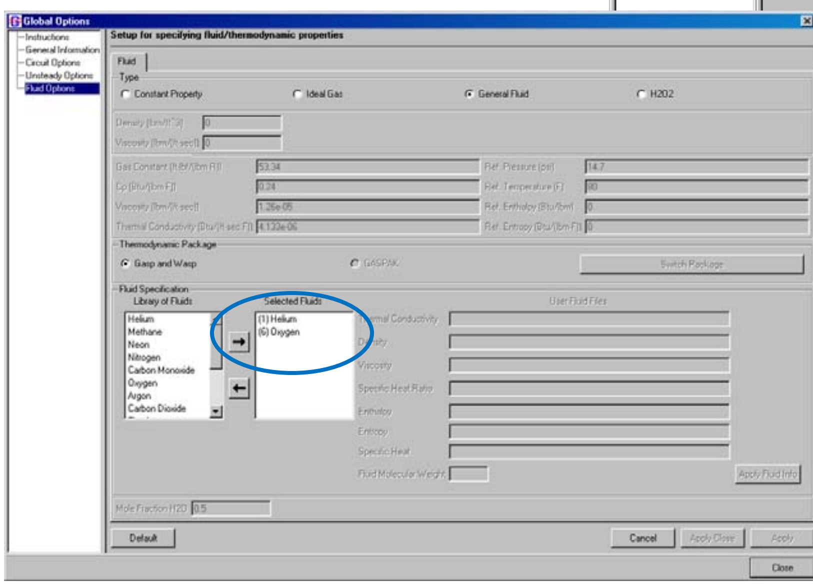


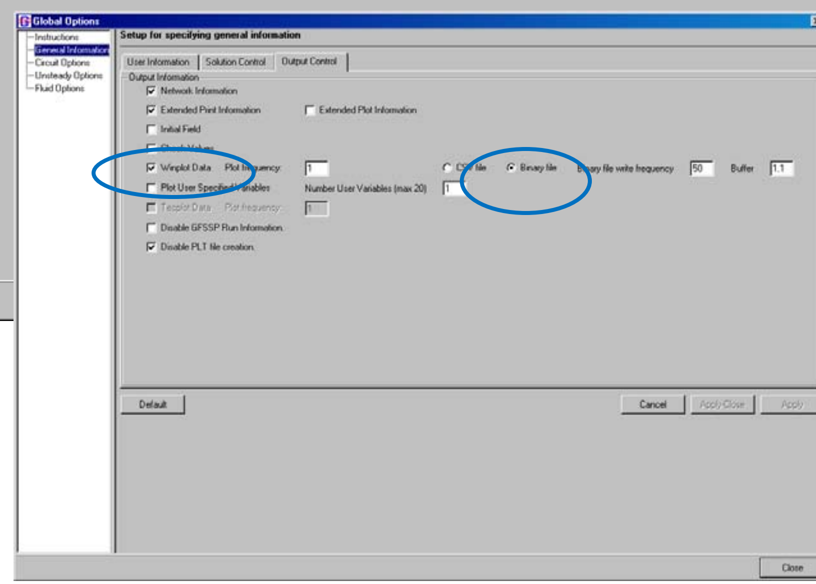
*NOTE: Many of these options will already be checked after they were imported.

Set Up User Options (cont.)

- Unsteady options
  - Unsteady
  - Time step: 0.1 sec
  - Final time: 250 sec
- Fluids are
  - Helium
  - Oxygen
- Winplot binary output

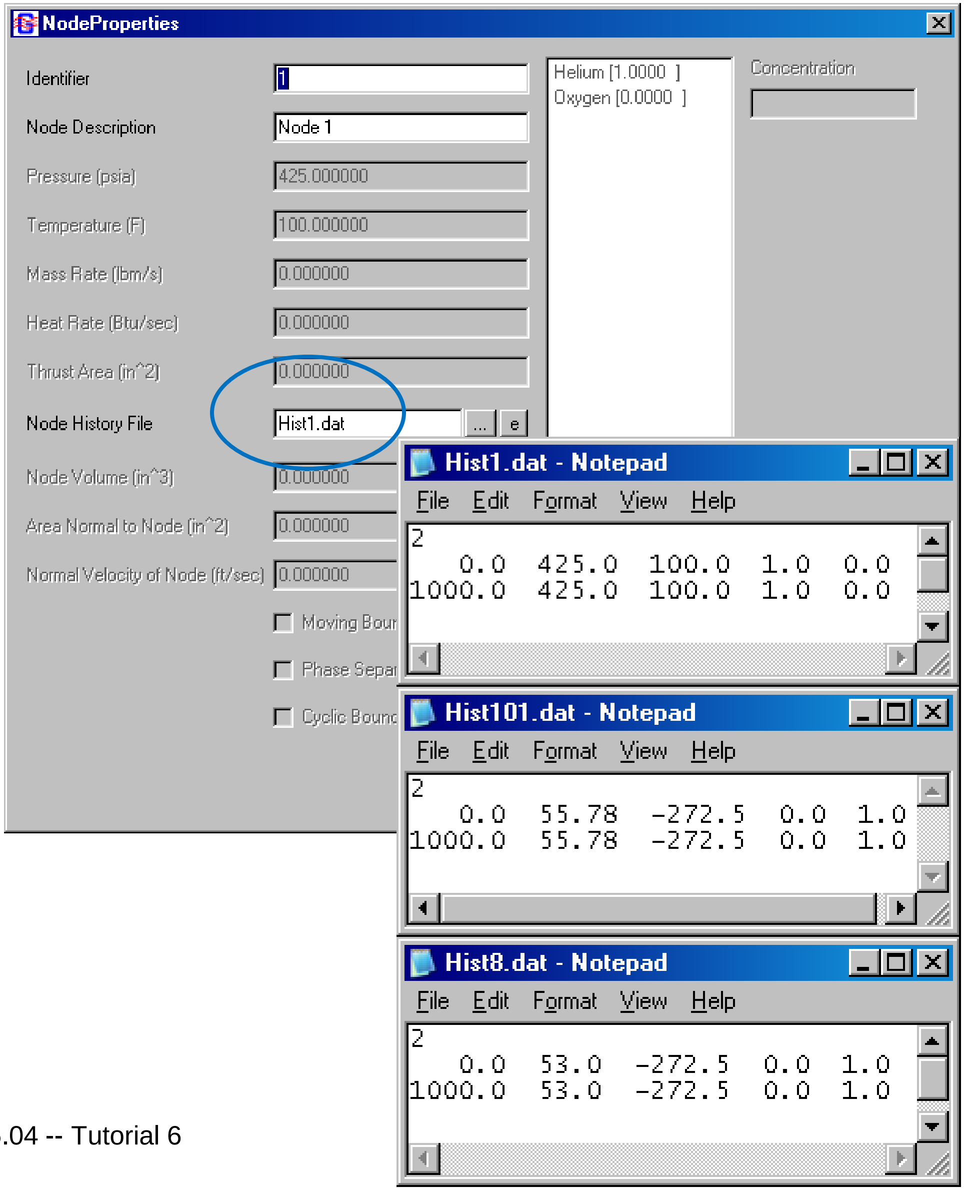






Complete Transient Boundary Conditions

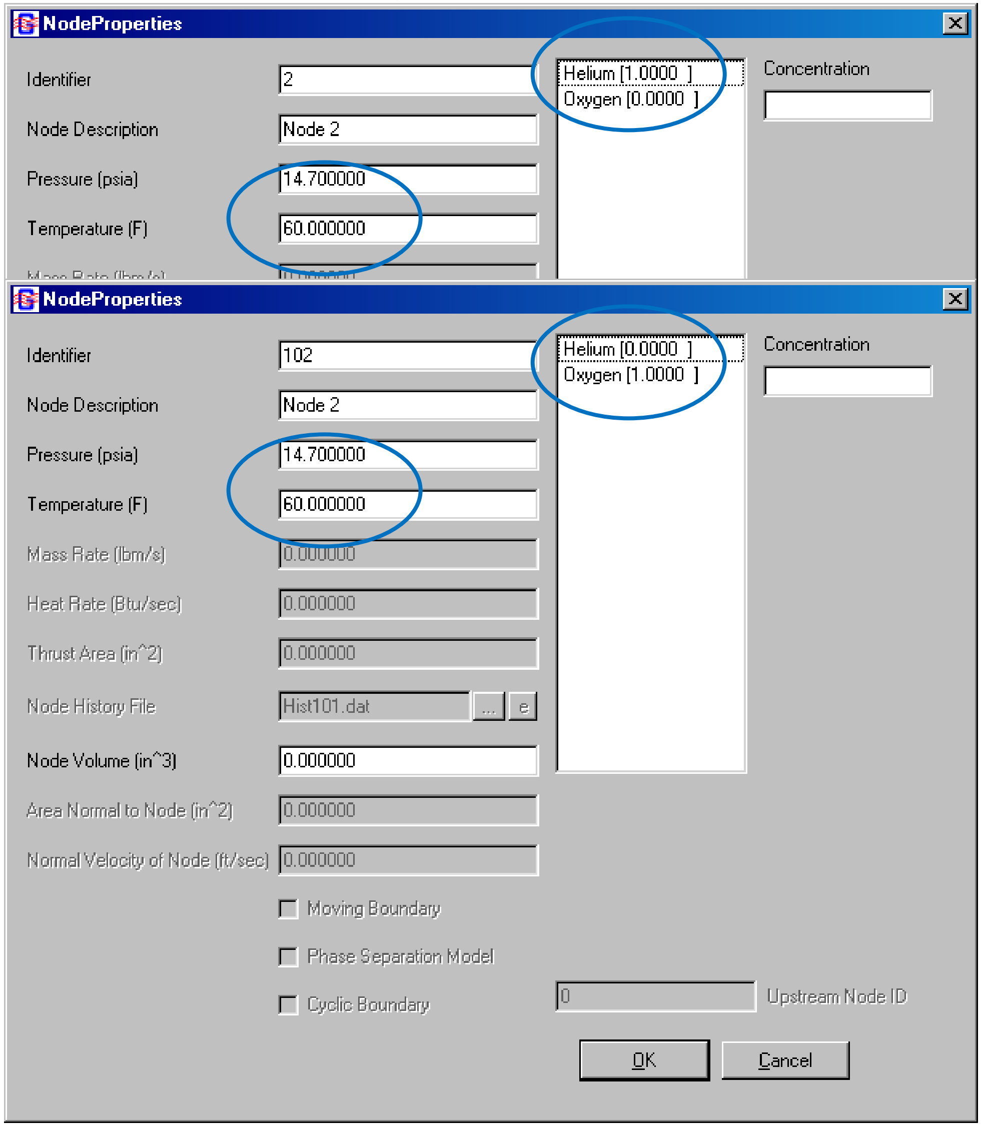
- Node 1:
  - P = 425 psia
  - T = 100 °F
  - He mass fraction: 1.0
  - LOx mass fraction: 0.0
- Node 101:
  - P = 55.78 psia
  - T = -272.5 °F
  - He mass fraction: 0.0
  - LOx mass fraction: 1.0
- Node 8:
  - P = 53.0 psia
  - T = -272.5 °F
  - He mass fraction: 0.0
  - LOx mass fraction: 1.0*



*Strictly speaking, node 8 will be some mixture of LOx and He. However, because it is downstream, setting the boundary mass fraction to pure LOx will not affect the calculations.

Complete Interior Node Initial Conditions

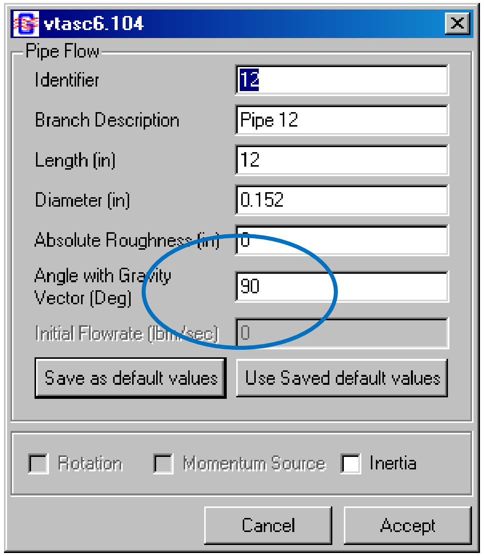
- Nodes 2 and 3:
  - P = 14.7 psia
  - T = 60 °F
  - He mass fraction: 1.0
  - LOx mass fraction: 0.0
- Nodes 102, 103, 104, 5, 6, and 7:
  - P = 14.7 psia
  - T = 60 °F
  - He mass fraction: 0.0
  - LOx mass fraction: 1.0

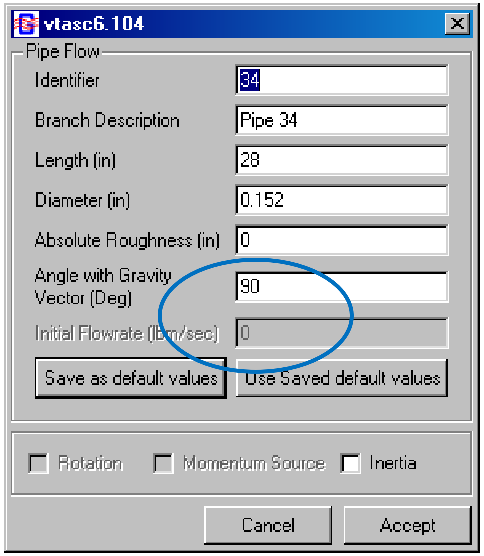


*NOTE: Node Volumes can be set to 0.0. GFSSP will calculate the volumes based on the pipe dimensions.

Complete Fluid Branches

- Branch 12: Pipe
  - L = 12 in
  - D = 0.152 in
  - Smooth pipe: ε = 0
  - Gravity angle: 90°
- Branch 34: Pipe
  - L = 28 in
  - D = 0.152 in
  - Smooth pipe: ε = 0
  - Gravity angle: 90°

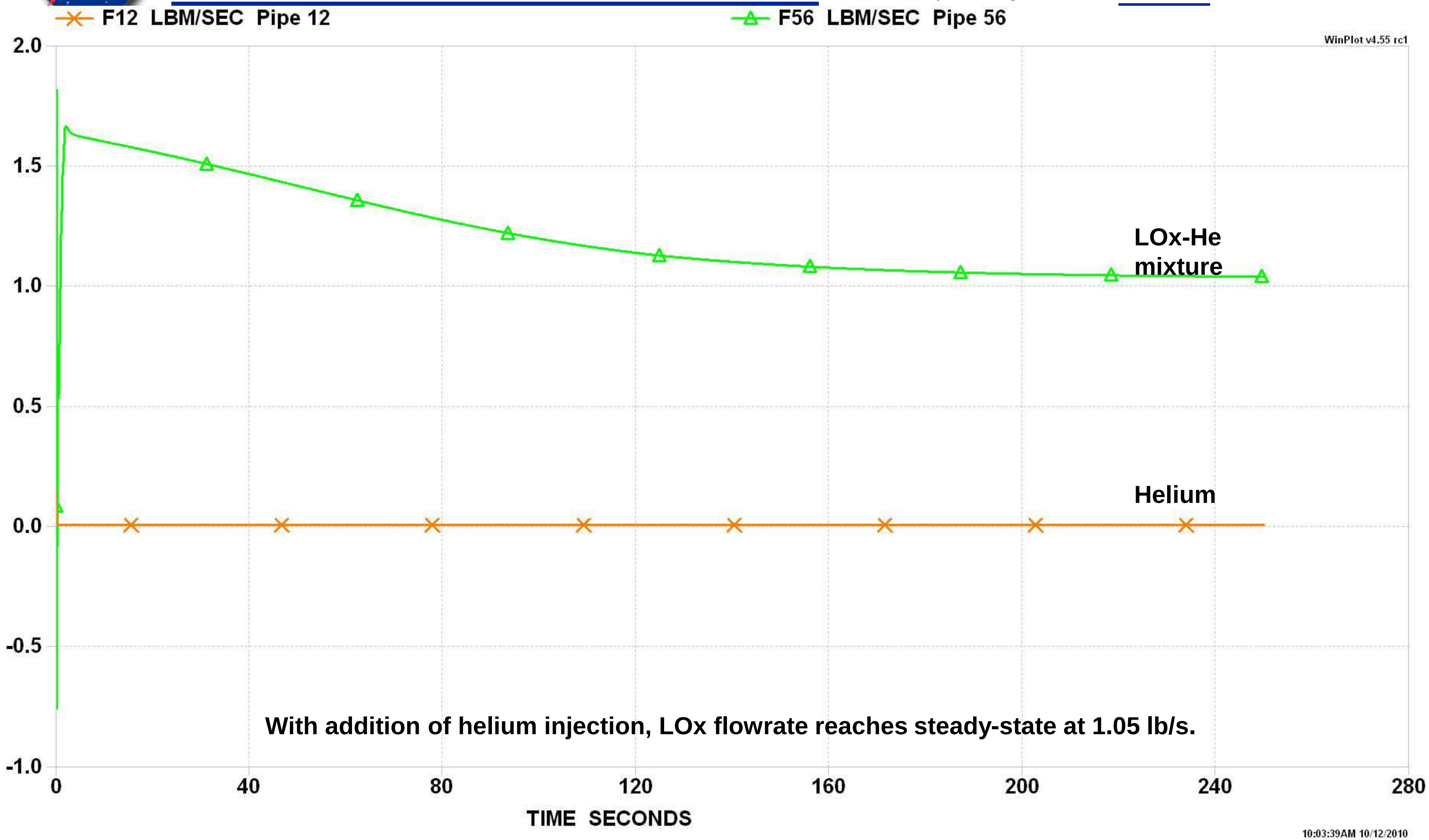




*NOTE: Most fluid branches will be already set after the model import. However, the gravity angle must be set in the helium injector pipes.

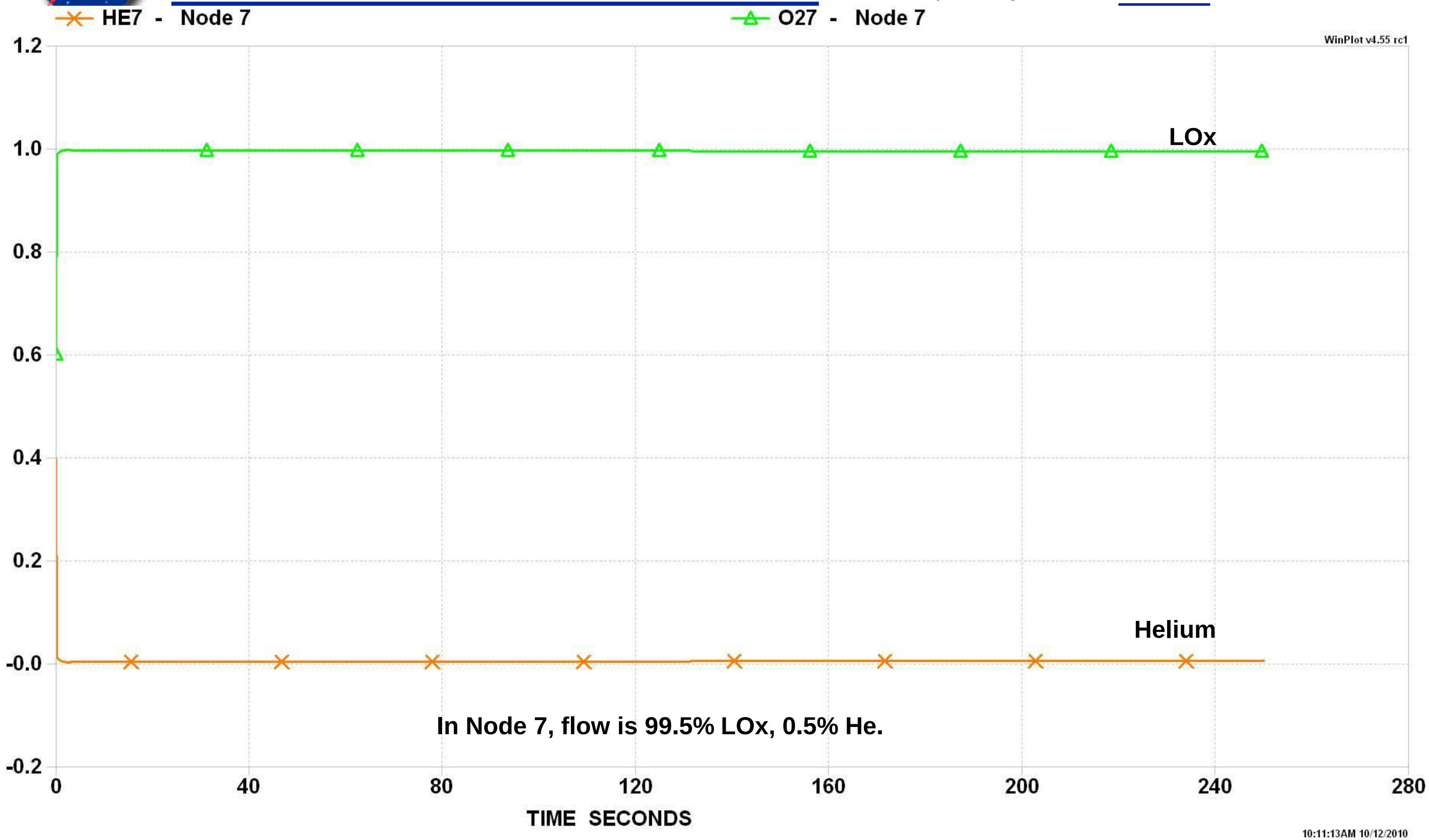
Flowrates

Marshall Space Flight Center

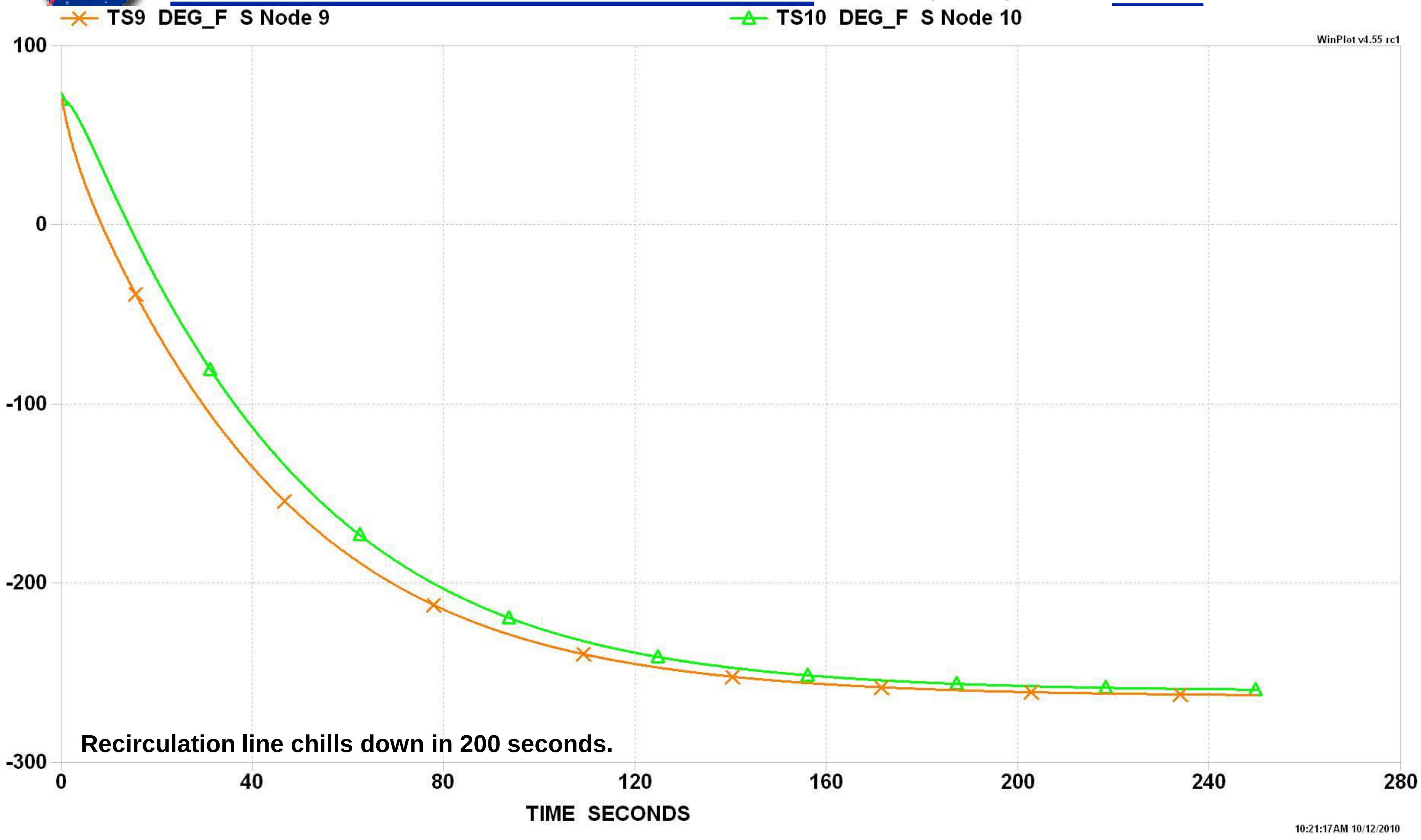


Mass Fractions

Marshall Space Flight Center



Solid Temperatures

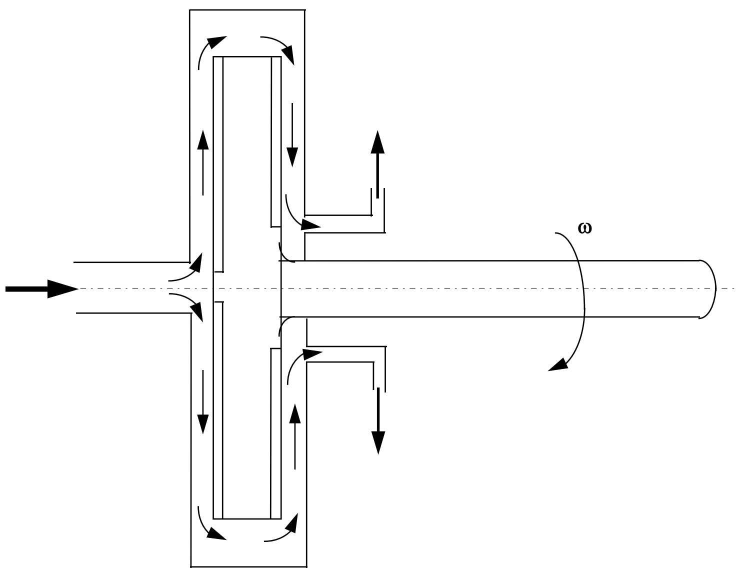


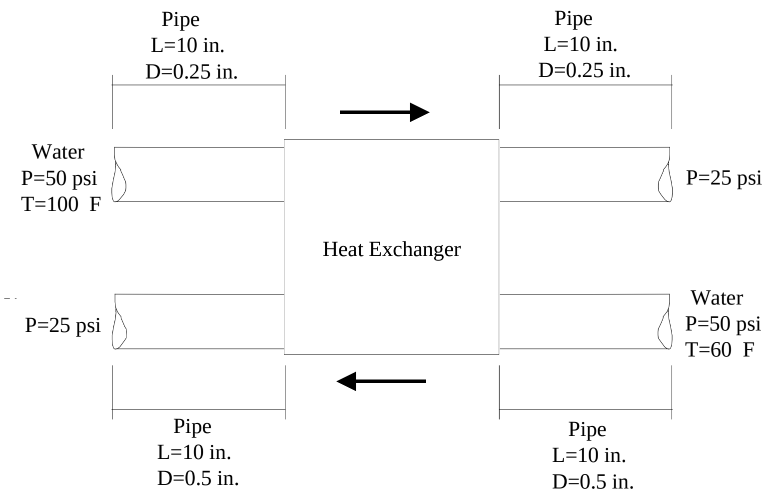
Recirculation line chills down in 200 seconds.

STUDY OF THE RESULTS

Model	Calculated Flowrate (lb_m/s)
Stagnant	<0.010
Heat Leak (0.106 BTU/s)	0.172
Heat Leak and He Injection	1.05

Rotating Flow, Turbopump & Heat Exchanger





CONTENT

- Centrifugal Force
- Axial Thrust
- Example 6 - Radial Flow on a Rotating Radial Disk
- Example 21 – Axial Thrust Calculation in a Turbopump
- Example 11 - Power Balancing of a Turbopump Assembly
- Heat Exchanger
- Example 5 - Simulation of a Flow System Involving a Heat Exchanger
- Summary

Centrifugal Force in Momentum Equation

Momentum Conservation Equation

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

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

$$+MAX\left|\dot{m}_{trans},0\right|\left(u_{ij}-u_p\right)-MAX\left|-\dot{m}_{trans},0\right|\left(u_{ij}-u_p\right)=$$

--------------- Transverse Inertia ---------------------

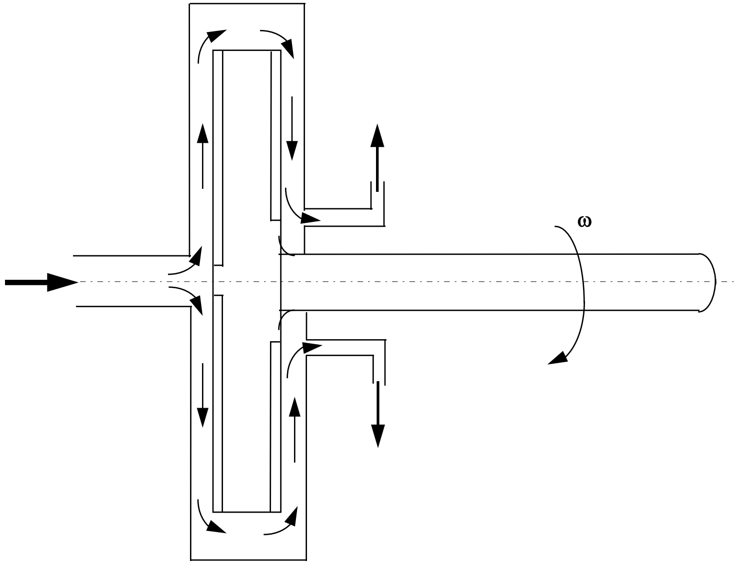
$$\left(p_i-p_j\right)A_{ij}+\frac{\rho g V Cos\theta}{g_c}-K_f\,\dot{m}_{ij}\left|\dot{m}_{ij}\right|A_{ij}+\frac{\rho K_{rot}^2\omega^2 A}{g_c}+\mu\frac{u_p-u_{ij}}{g_c\delta_{ij,p}}A_s$$

--Pressure-- -- Gravity -- -- Friction -- -- Centrifugal -- -- Shear --

$$-\rho A_{norm}u_{norm}u_{ij}/g_c+\left(\mu_d\frac{u_d-u_{ij}}{\delta_{ij,d}}-\mu_u\frac{u_{ij}-u_u}{\delta_{ij,u}}\right)\frac{A_{ij}}{g_c}+S \qquad (3.1.2)$$

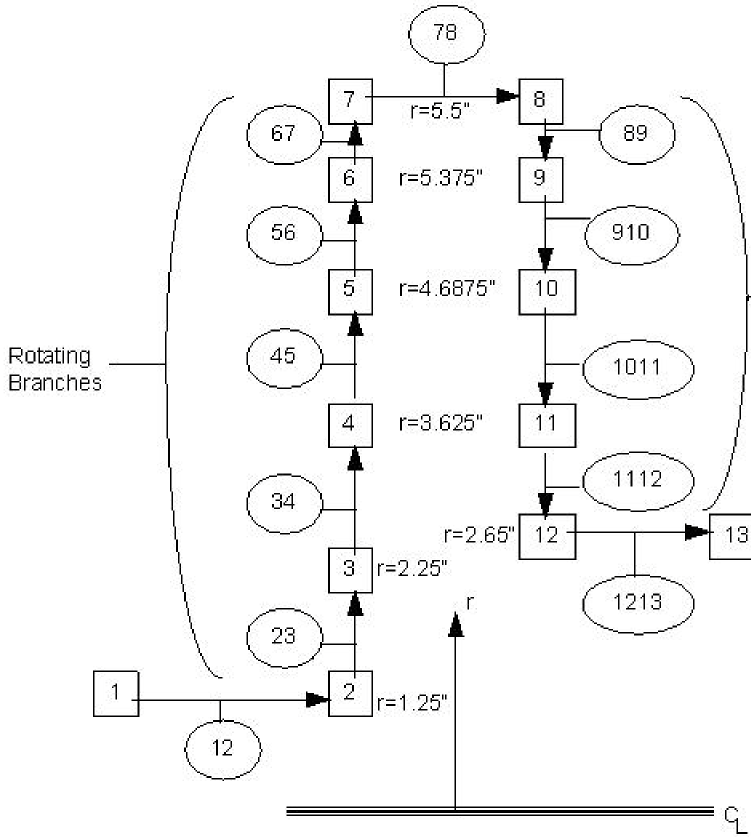
-- Moving Boundary-- -- Normal Stress --- -- Source --

Example 6 - Radial Flow on a Rotating Radial Disk

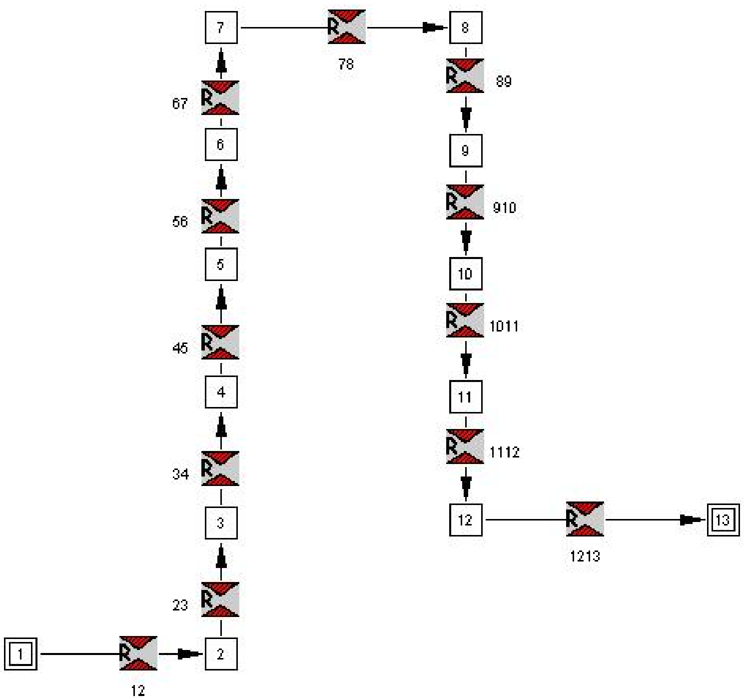


Feature: Rotating Flows, Comparison with Textbook Solution

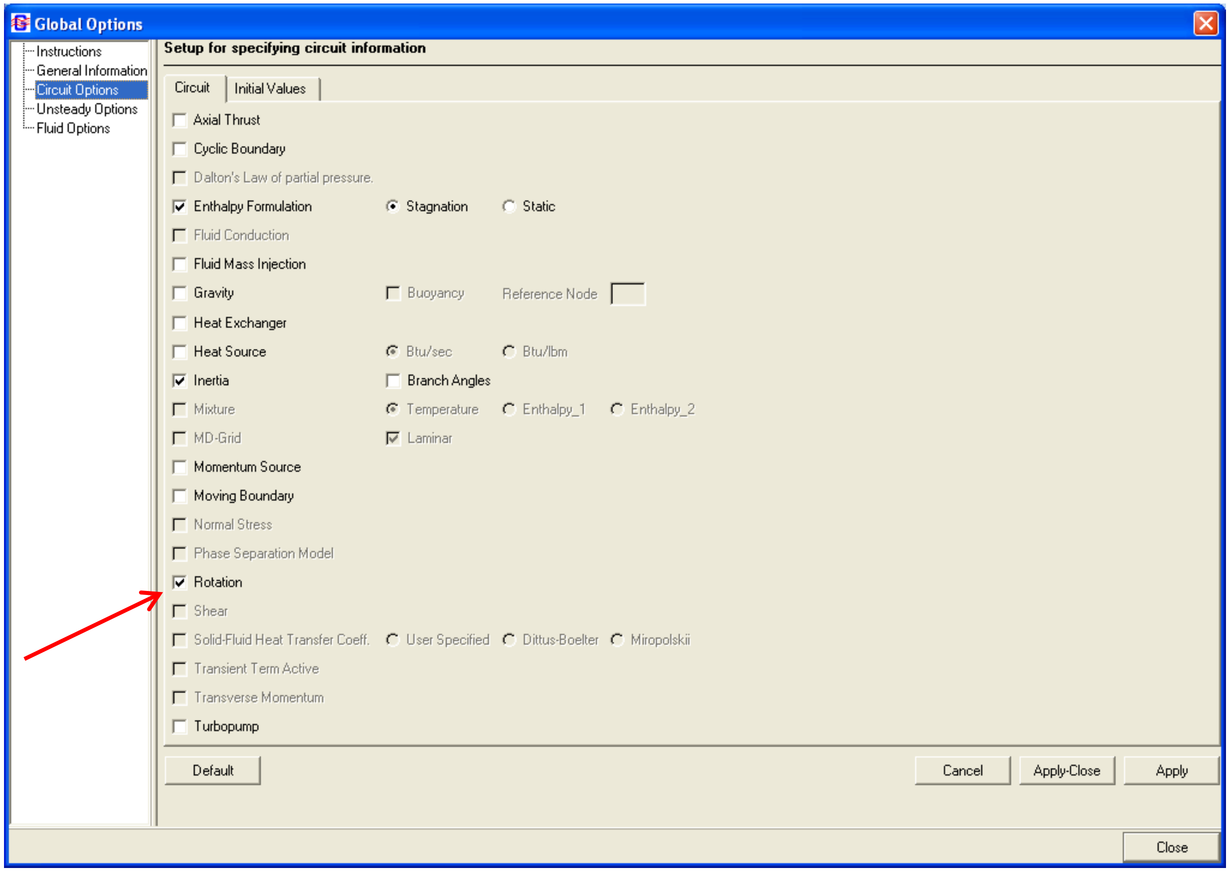
Detailed Schematic

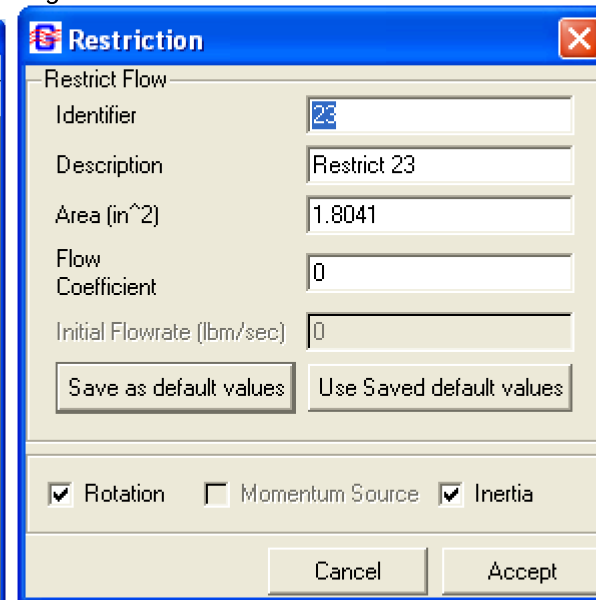


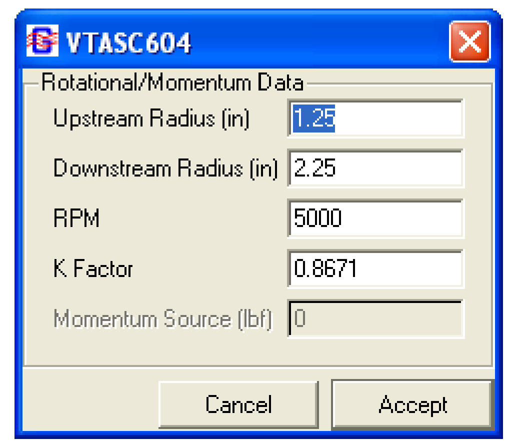
VTASC Model



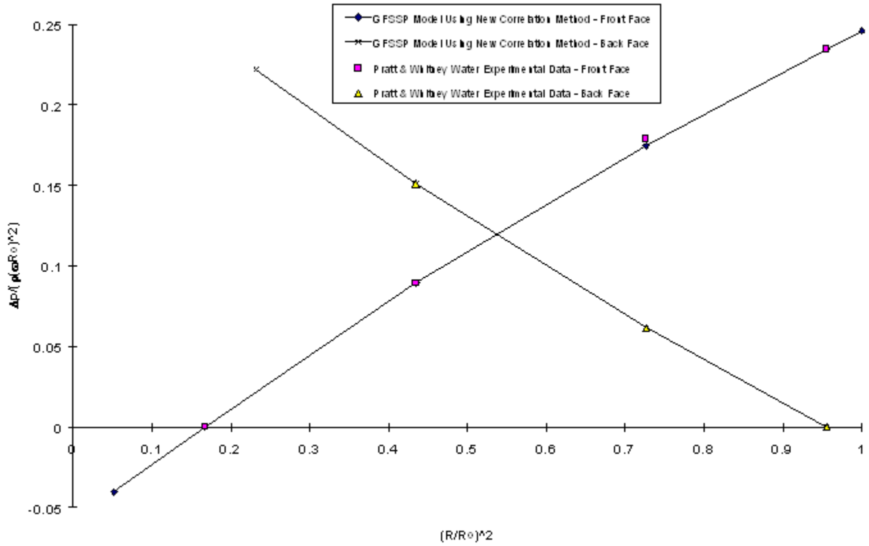
Activation of Rotational Term in VTASC





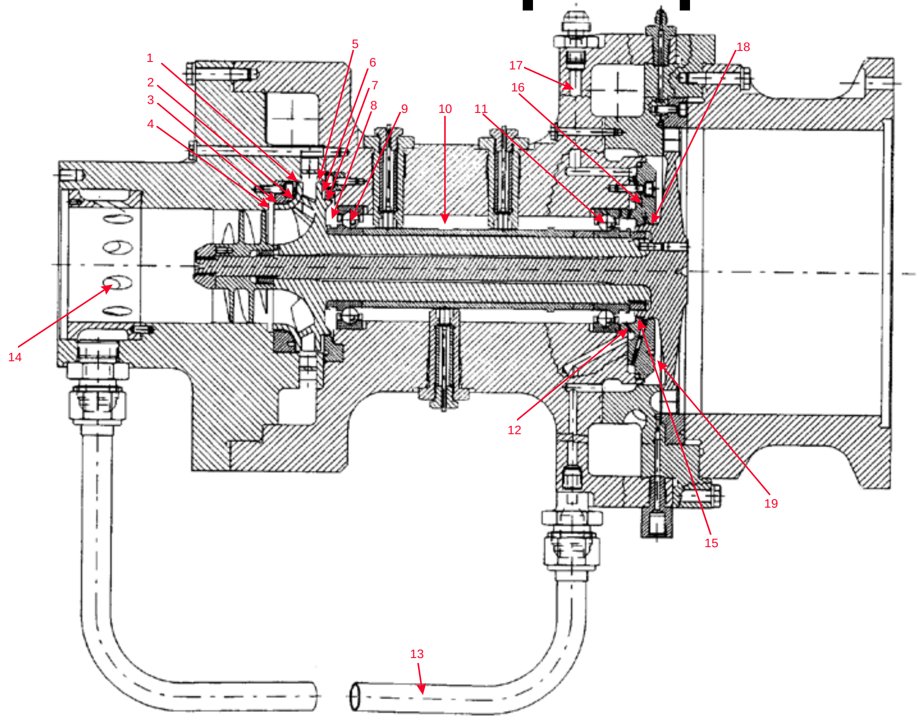


Comparison of GFSSP Model Results with Experimental Data



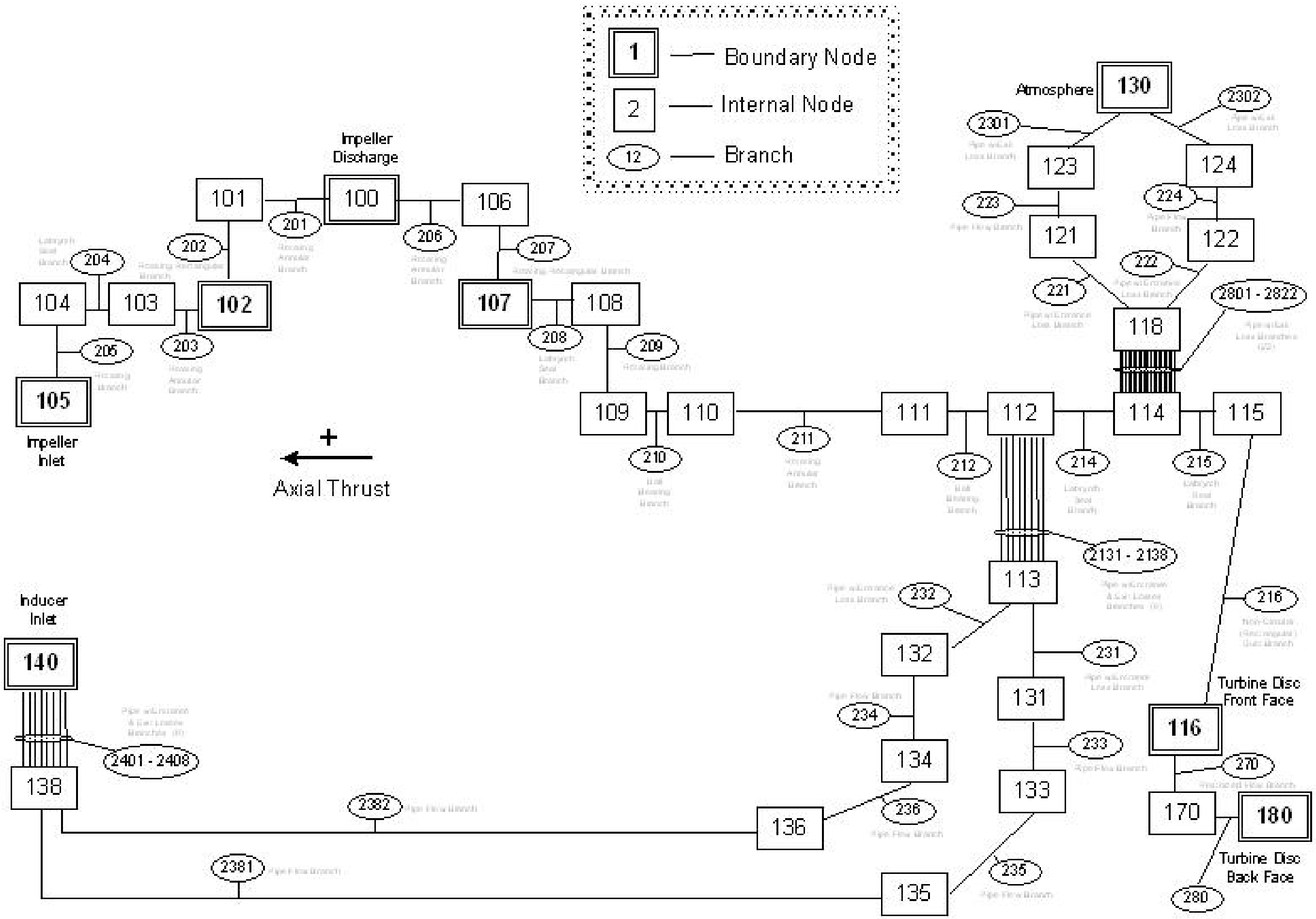
Schallhorn, P.A. and Majumdar, A. K.: "Numerical Prediction of Pressure Distribution Along the Front and Back Face of a Rotating Disc With and Without Blades," AIAA 97-3098, Presented at the 33rd Joint Propulsion Conference, Seattle, Washington, July 6-9, 1997

Example 21 – Axial Thrust Calculation in a Turbopump

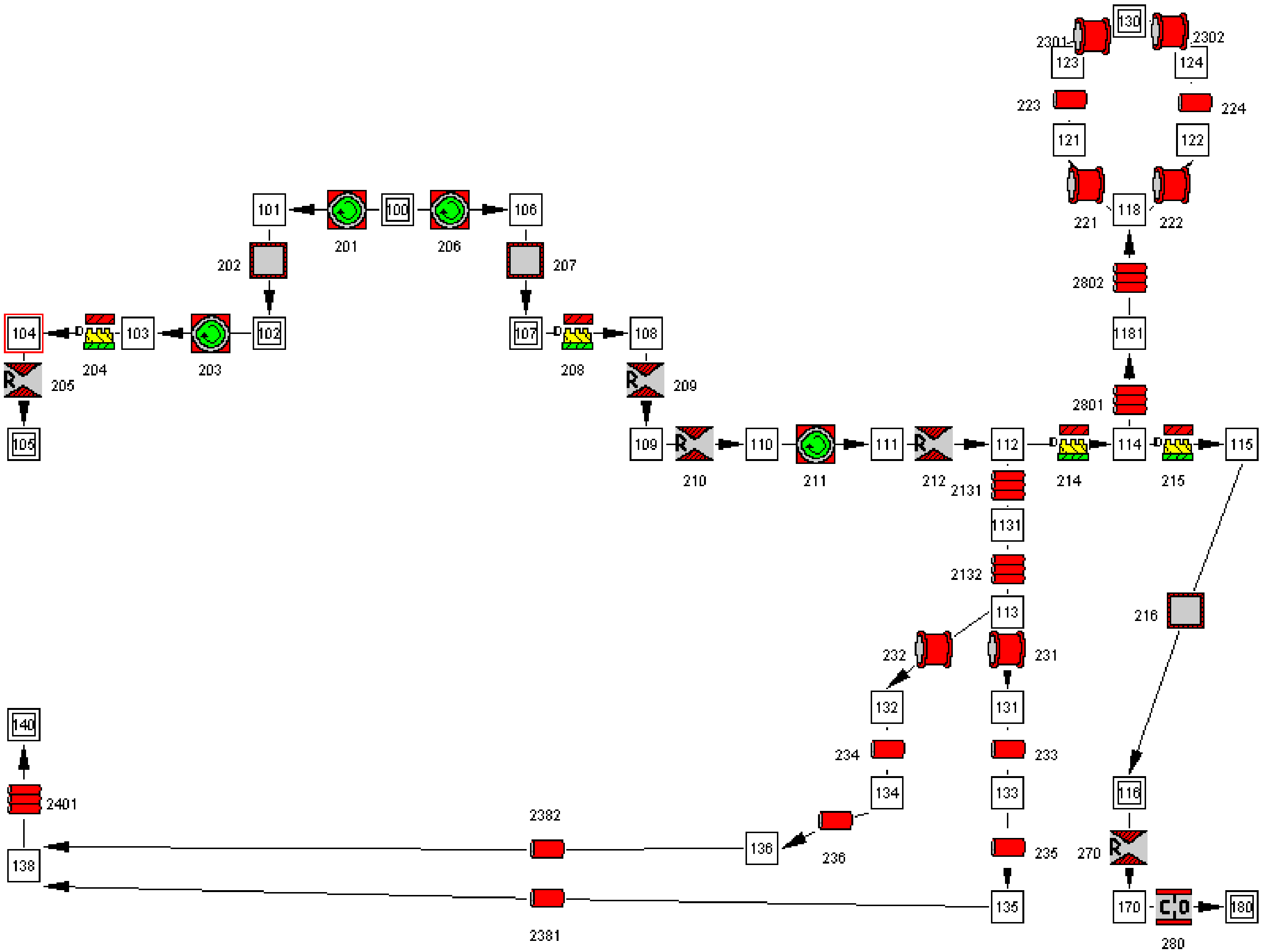


Feature: Axial Thrust, Rotating Flow, Mixture, Parallel Tube and comparison with test data

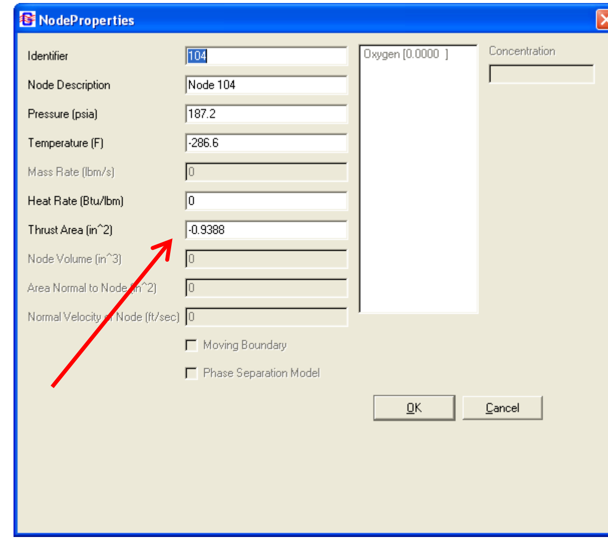
Simplex Turbopump Detailed Model

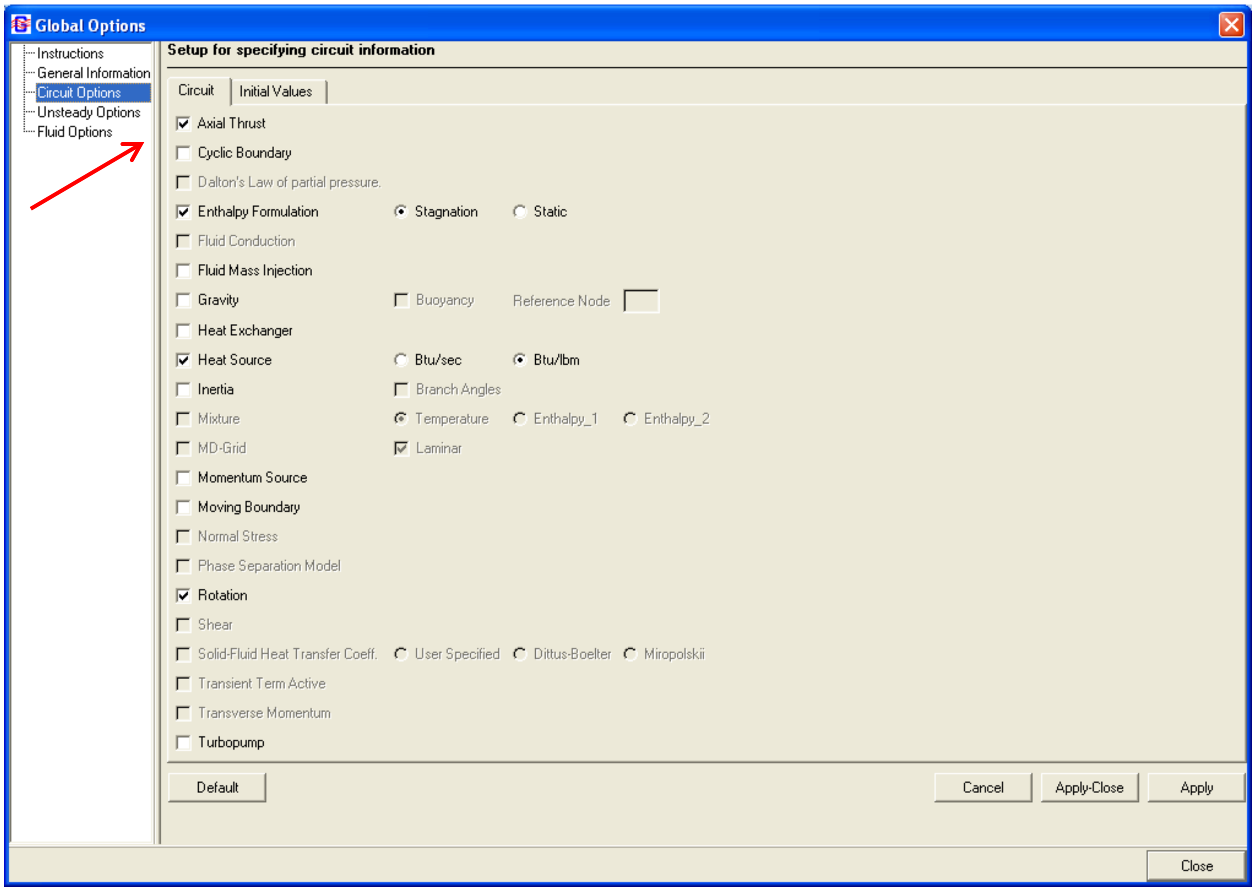


Simplex Turbopump VTASC Model



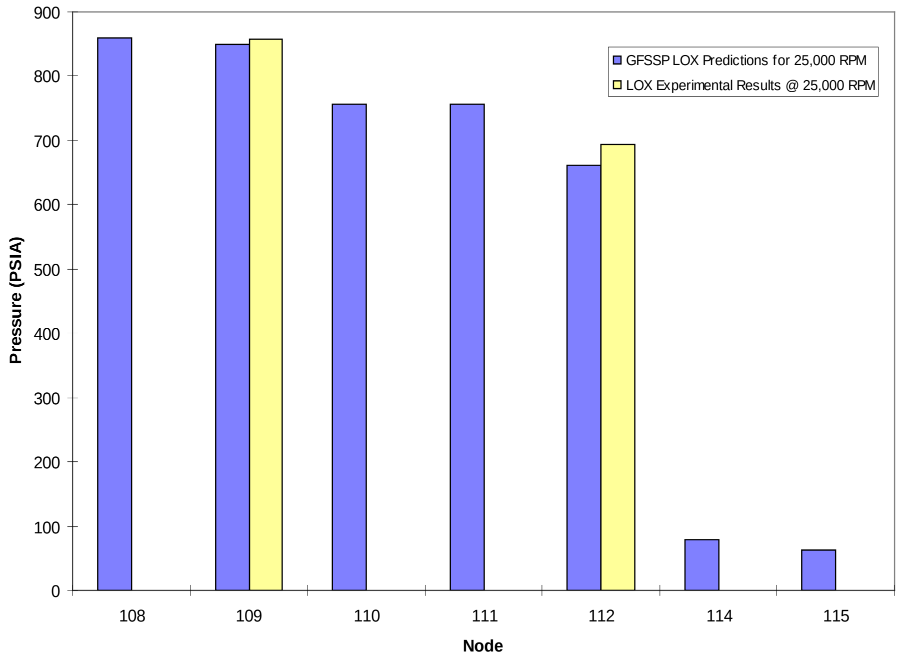
Activation of Axial Thrust in VTASC



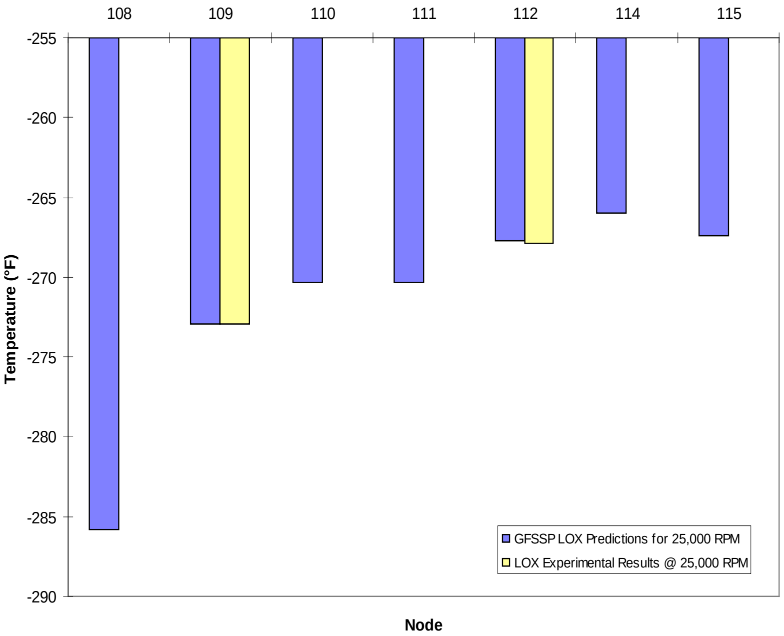


Comparison with Experimental Data

Pressure Predictions Compared to Experimental Data

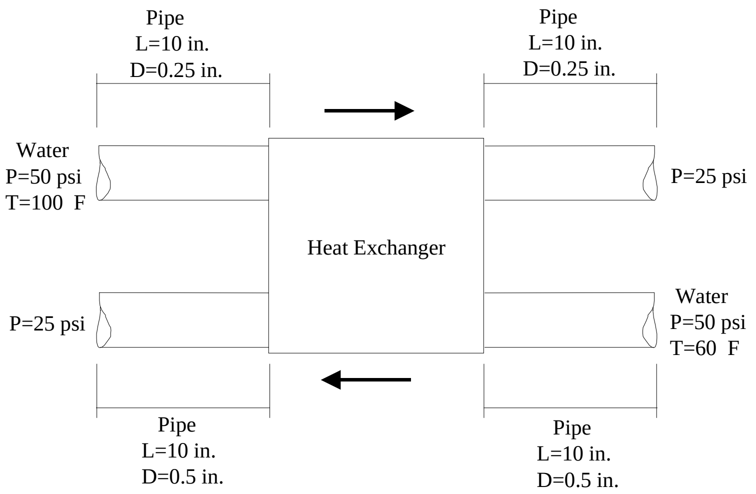


Temperature Predictions Compared to Experimental Data

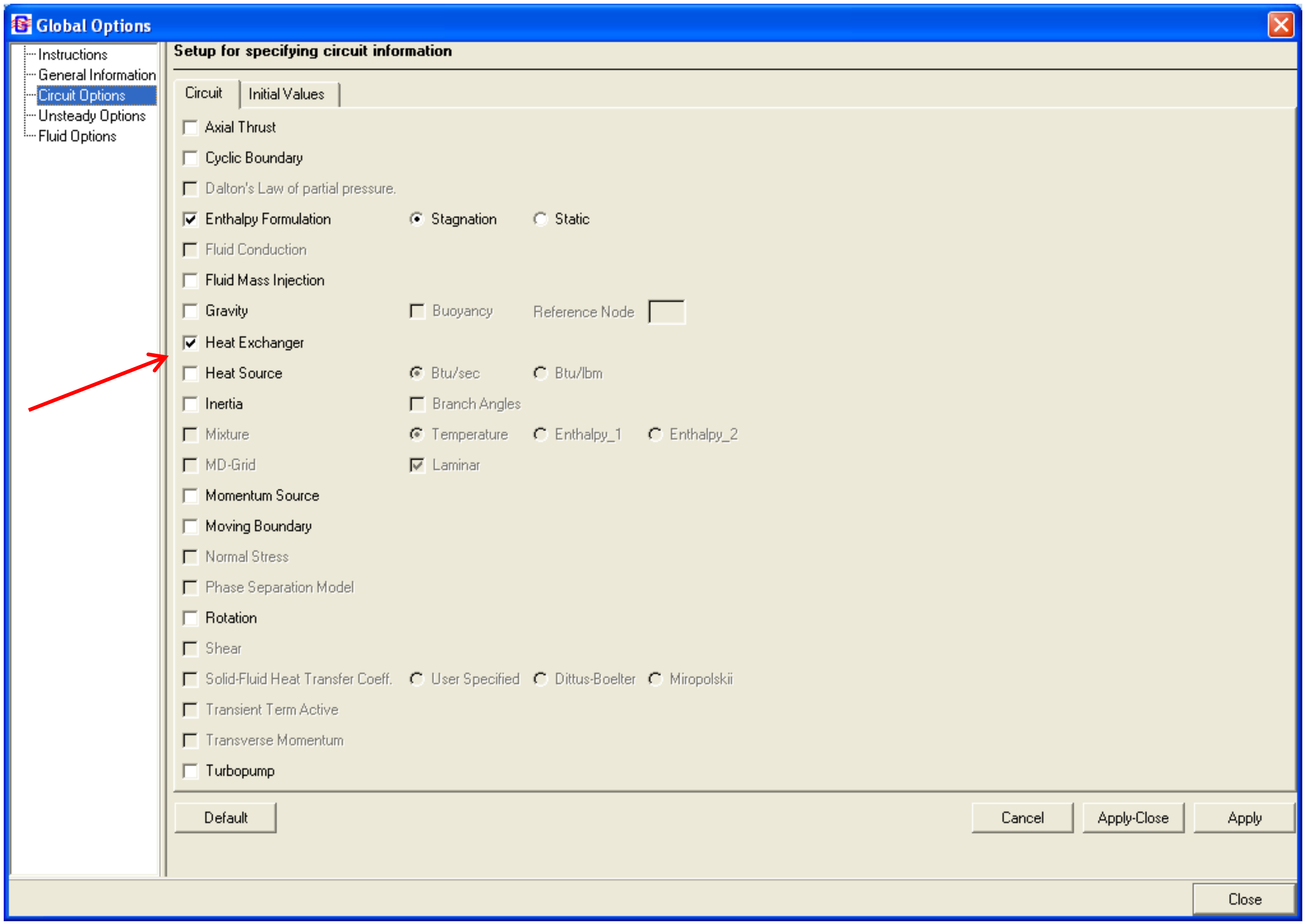


Schallhorn, Paul, Majumdar, Alok, Van Hooser, Katherine and Marsh, Matthew, “Flow Simulation in Secondary Flow Passages of a Rocket Engine Turbopump”, Paper No. AIAA 98-3684, 34th AIAA/ASME/SAE/ASEE, Joint Propulsion Conference and Exhibit, July 13-15, 1998, Cleveland, OH

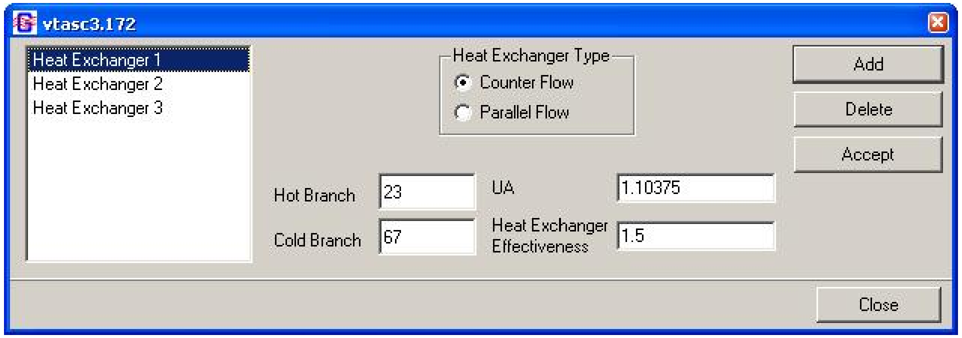
Example 5 - Simulation of a Flow System Involving a Heat Exchanger

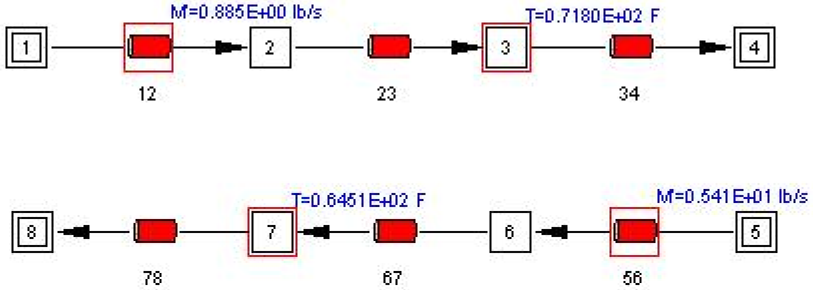


Feature: Heat Exchanger Option, Comparison with Textbook Solution

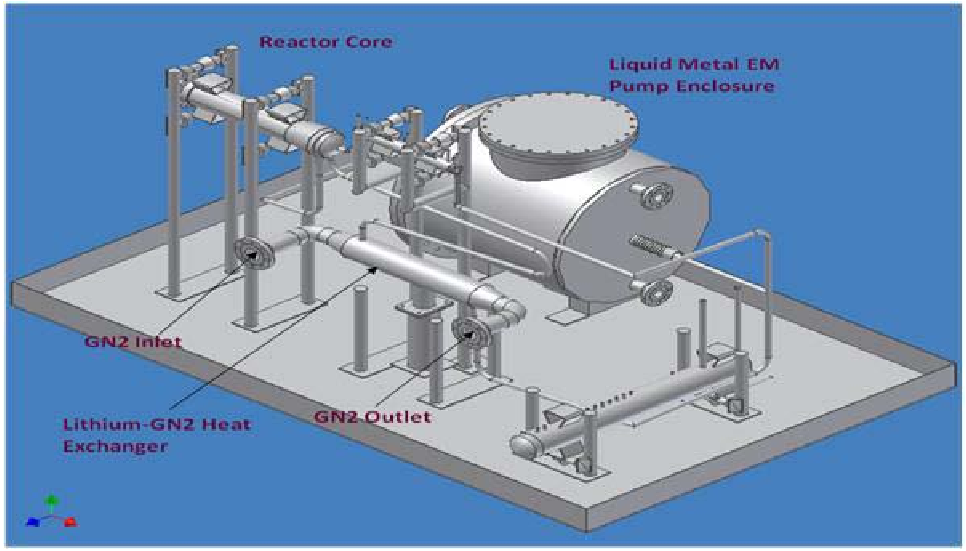

Global Options
Instructions
General Information
Circuit Options
Unsteady Options
Fluid Options
Setup for specifying circuit information
Circuit
Initial Values
Axial Thrust
Cyclic Boundary
Dalton's Law of partial pressure.
Enthalpy Formulation
Stagnation
Static
Fluid Conduction
Fluid Mass Injection
Gravity
Buoyancy
Reference Node
Heat Exchanger
Heat Source
Btu/sec
Btu/lbm
Inertia
Branch Angles
Mixture
Temperature
Enthalpy_1
Enthalpy_2
MD-Grid
Laminar
Momentum Source
Moving Boundary
Normal Stress
Phase Separation Model
Rotation
Shear
Solid-Fluid Heat Transfer Coeff.
User Specified
Dittus-Boelter
Miropolskii
Transient Term Active
Transverse Momentum
Turbopump
Default
Cancel
Apply-Close
Apply
Close

Heat Exchanger Option



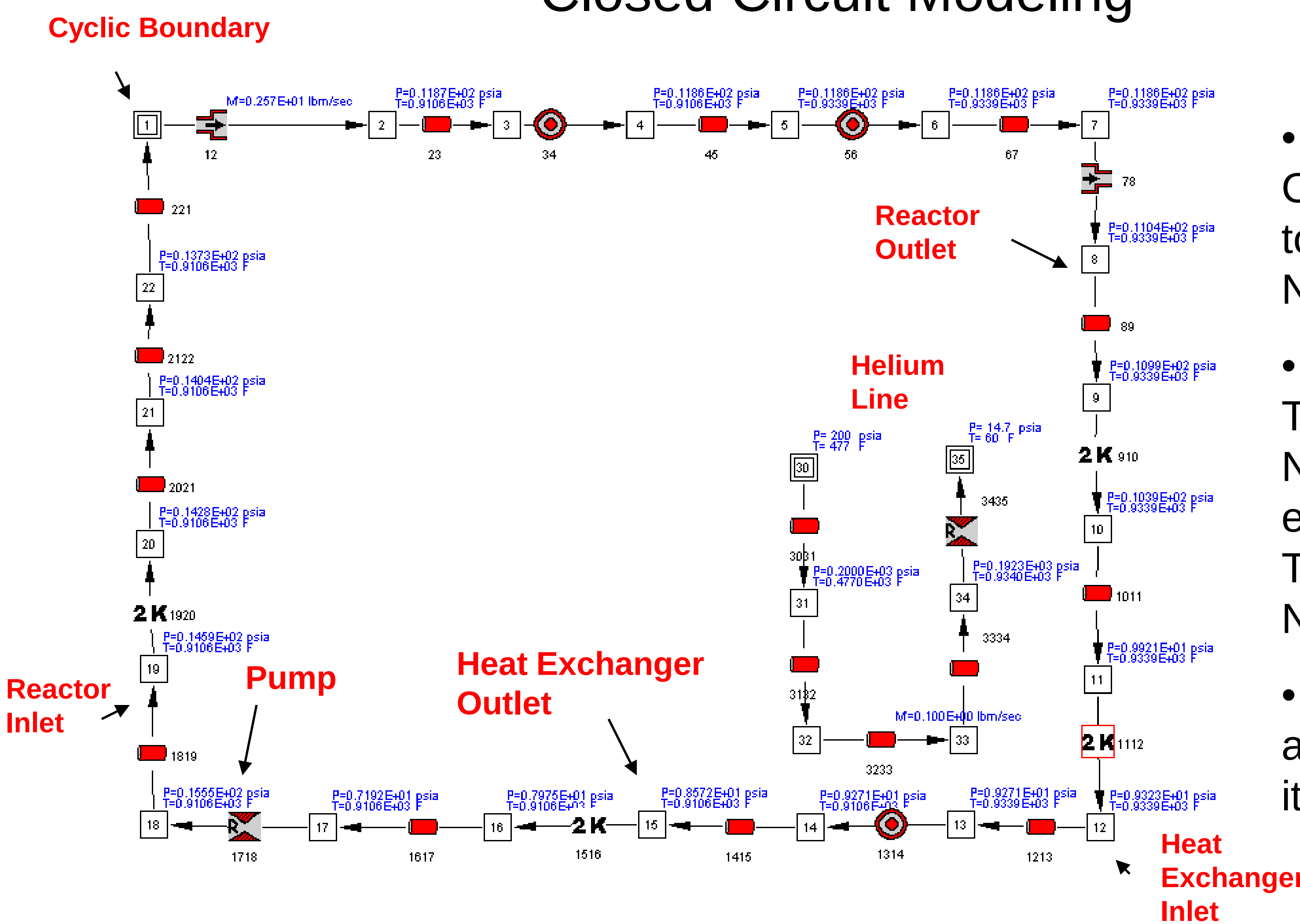


Example 20 - Simulation of a Lithium Loop Model



Feature: Closed Loop with cyclic boundary, Use of user-specified property, Heat Exchanger & User Subroutine to model Electro-Magnetic Pump

Closed Circuit Modeling

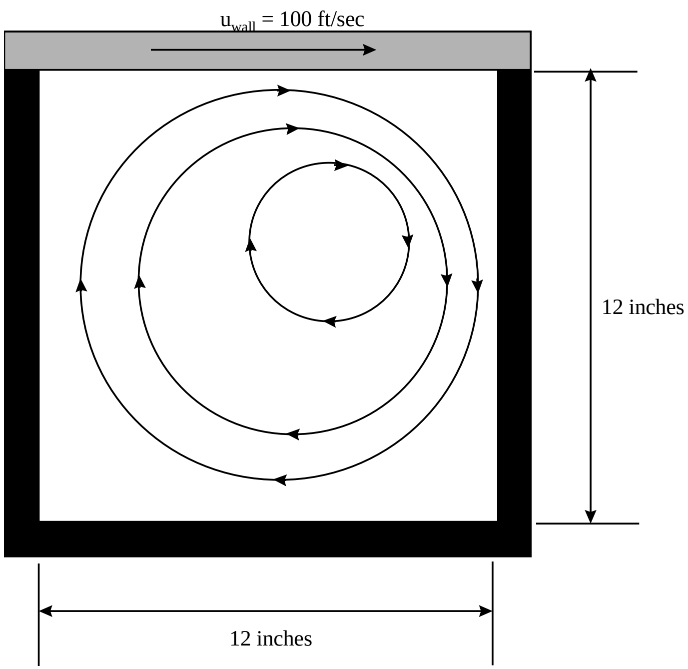


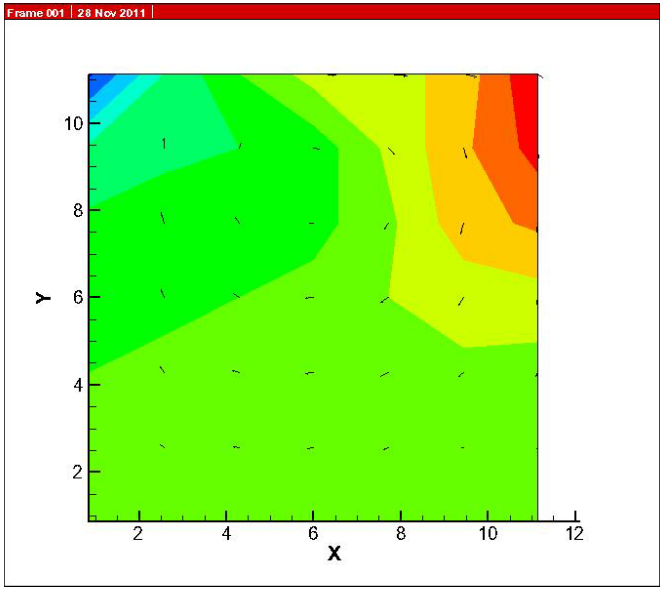
- Cyclic Boundary Condition needs to satisfied at Node 1
- This implies Temperature at Node 22 must be equal to Temperature at Node 1
- This must be achieved by iteration

Summary

- GFSSP's capability of modeling Rotational Flow and Internal Flow in Turbopump has been illustrated with Example 6 & 21
- In both examples, model predictions have been compared with Test Data and comparison was satisfactory
- This presentation illustrates the activation of Rotational Term and Axial Thrust Calculation in VTASC
- Reference 27 & 38 provide more details of these models
- GFSSP can model heat exchanger in a flow circuit and its application has been illustrated in Examples 5 & 20

Multi-Dimensional Flow Modeling





CONTENT

- Mathematical Formulation
- Validation of GFSSP Prediction
  - POISEULLE FLOW
  - COUETTE FLOW
  - DRIVEN CAVITY FLOW (Example – 25)
- Summary

Multi-D Terms in Momentum Equation

Momentum Conservation Equation

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

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

$$+MAX\left|\dot{m}_{trans},0\right|\left(u_{ij}-u_p\right)-MAX\left|-\dot{m}_{trans},0\right|\left(u_{ij}-u_p\right)=$$

--------------- Transverse Inertia ---------------------

$$\left(p_i-p_j\right)A_{ij}+\frac{\rho gVCos\theta}{g_c}-K_f\,\dot{m}_{ij}\left|\dot{m}_{ij}\right|A_{ij}+\frac{\rho K_{rot}^2\omega^2A}{g_c}+\mu\frac{u_p-u_{ij}}{g_c\delta_{ij,p}}A_s$$

--Pressure-- -- Gravity -- -- Friction -- -- Centrifugal -- -- Shear --

$$-\rho A_{norm}u_{norm}u_{ij}/g_c+\left(\mu_d\frac{u_d-u_{ij}}{\delta_{ij,d}}-\mu_u\frac{u_{ij}-u_u}{\delta_{ij,u}}\right)\frac{A_{ij}}{g_c}+S \qquad (3.1.2)$$

-- Moving Boundary-- -- Normal Stress --- -- Source --

Validation of GFSSP Prediction

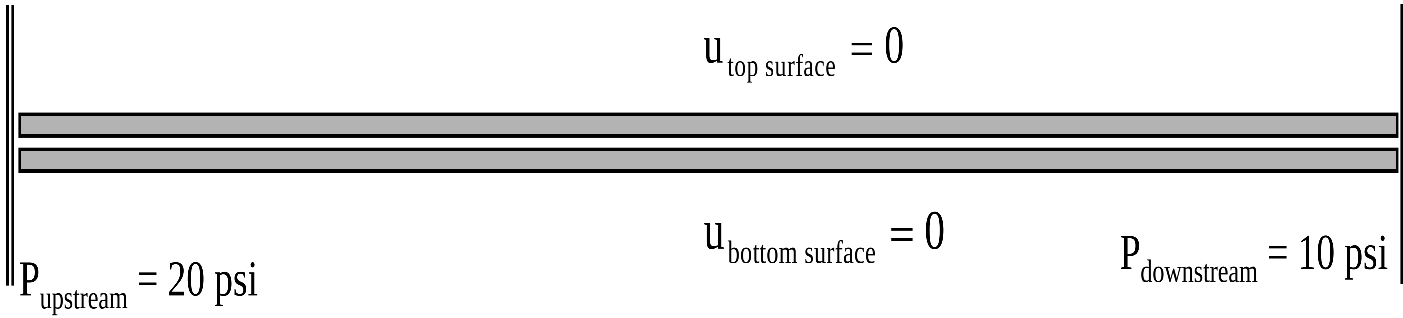
- Three Classical Fluid Dynamics Problems have been considered for Validation of GFSSP Prediction

  – POISEULLE FLOW
  – COUETTE FLOW
  – DRIVEN CAVITY FLOW

- Poiseulle Flow is shear dominated flow between two stationary flat plates
- Couette Flow is shear driven flow between one moving flat plate and one stationary flat plate
- Driven Cavity Flow is also shear driven recirculating flow in a rectangular cavity when top surface is moving with a constant velocity. Transverse momentum transfer is present in Driven Cavity Flow

Schallhorn, Paul and Majumdar, Alok, "Implementation of Finite Volume based Navier Stokes Algorithm within General Purpose Flow Network Code", 50th AIAA Aerospace Sciences Meeting held on 9-12 January, 2012 in Nashville, Tennessee.

POISEULLE FLOW



Length = 1000 inches

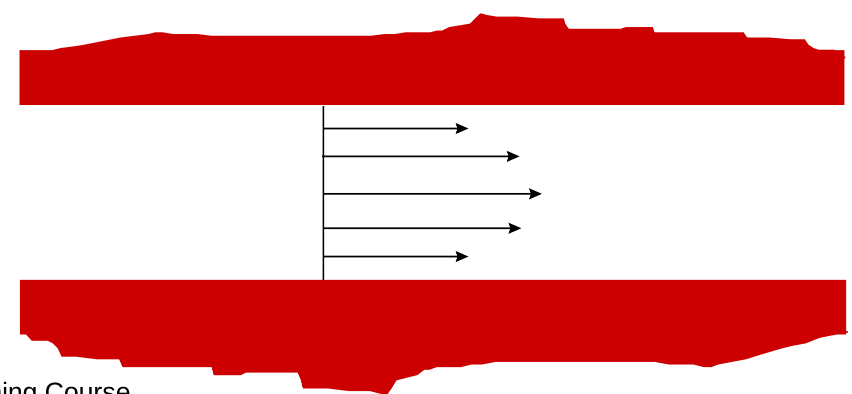
Distance between Plates = 1 inch

Fluid Density =12 lb/ft^3

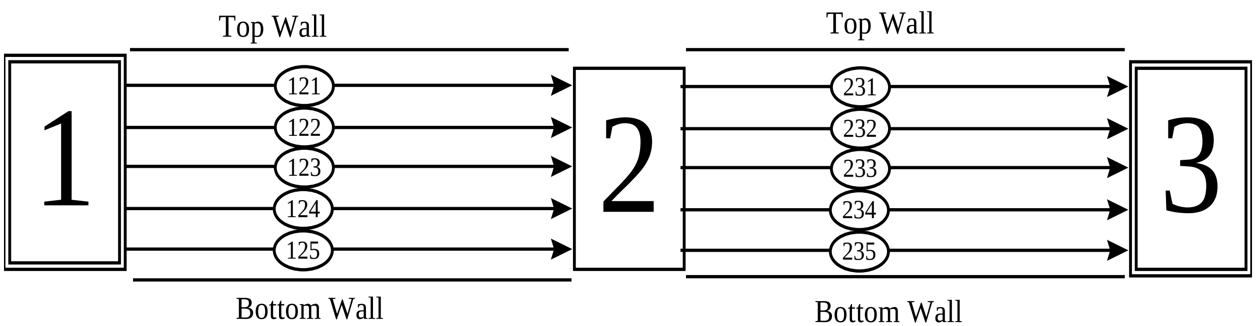
Fluid Viscosity = 1 lb/ft-sec

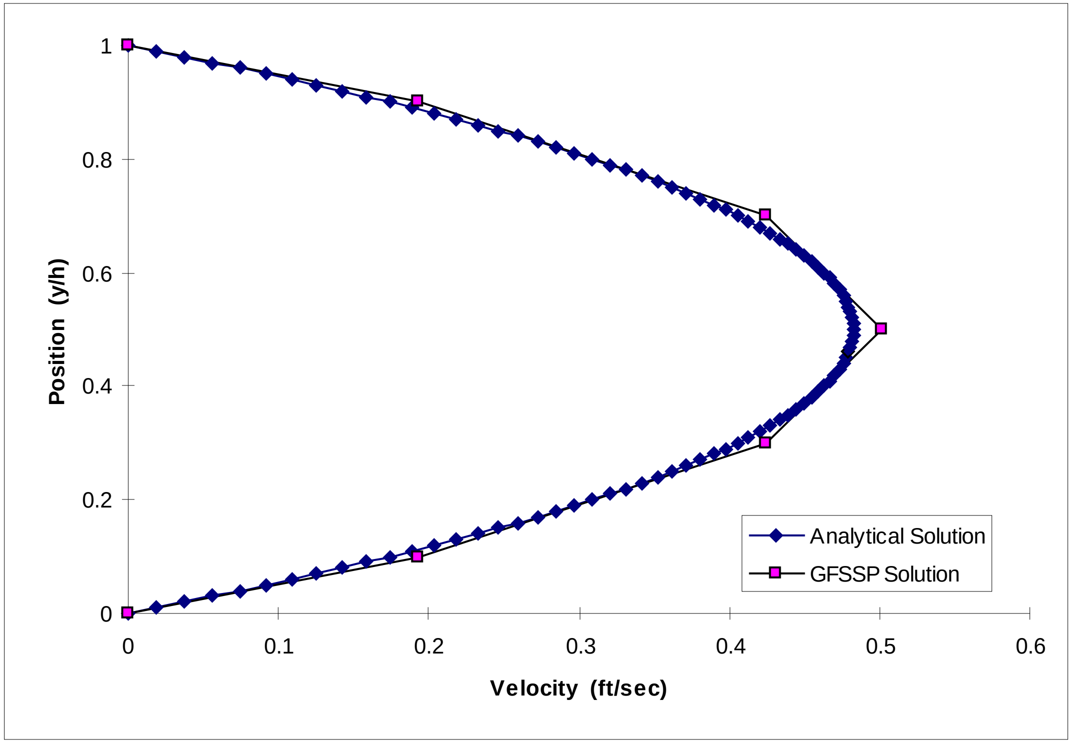
Analytical Solution:

$$u = 0.005\left(y - y^2\right)$$

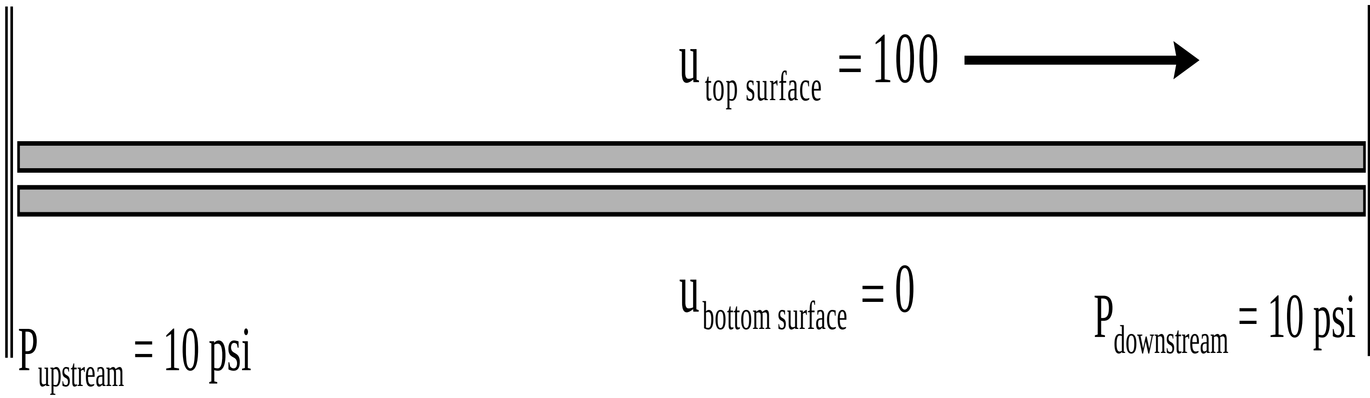


GFSSP model of Poiseulle Flow





COUETTE FLOW



Length = 1000 inches

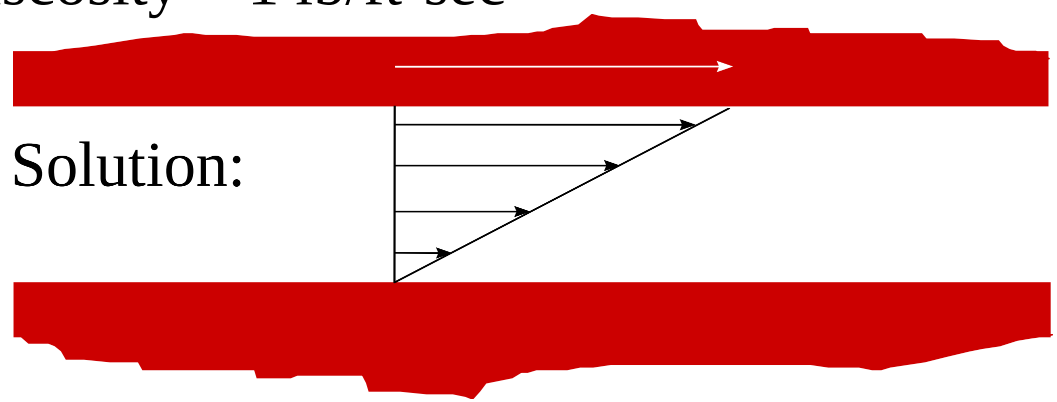
Distance between Plates = 1 inch

Fluid Density =12 lb/ft^3

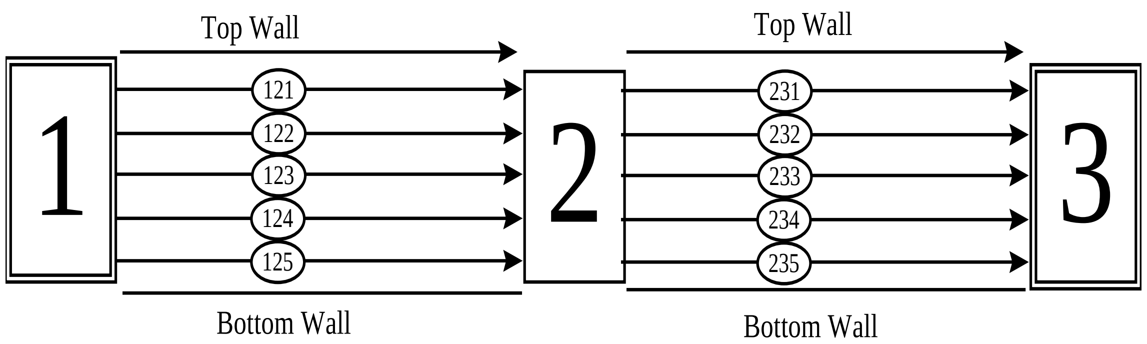
Fluid Viscosity = 1 lb/ft-sec

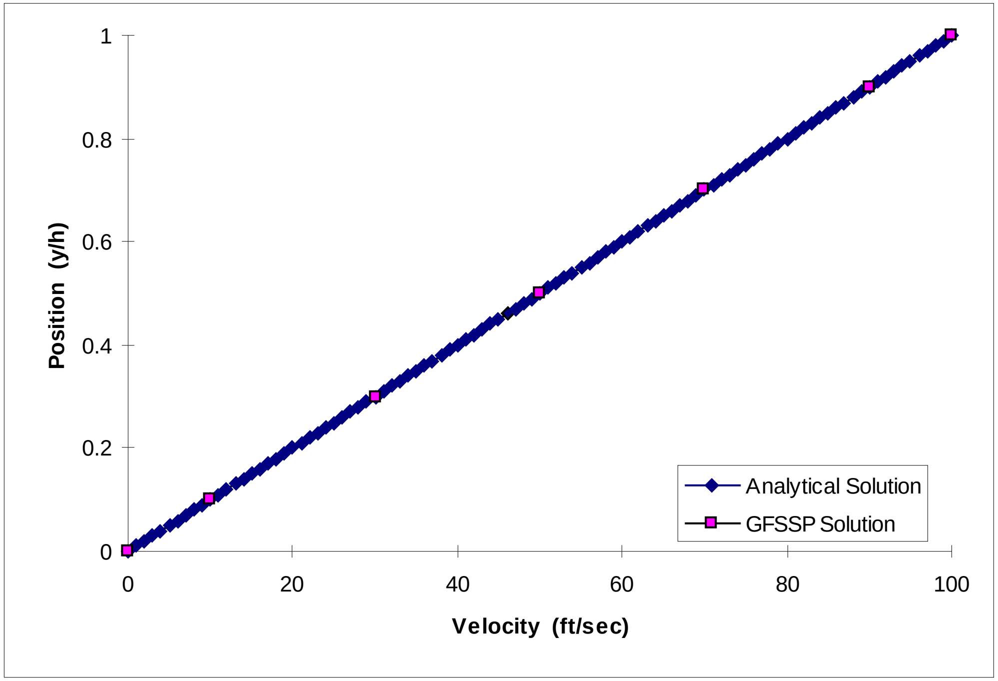
Analytical Solution:

$$u = 100y$$

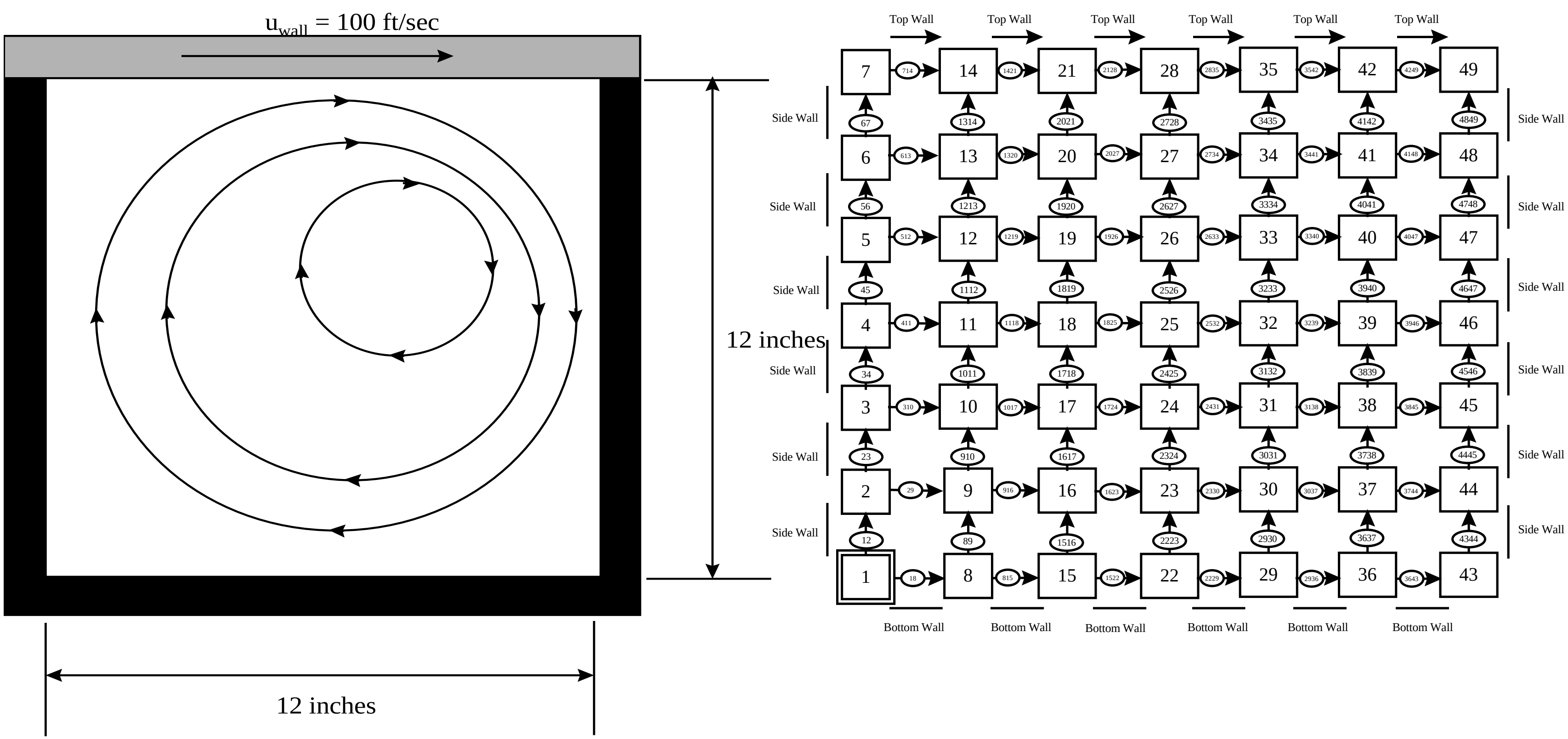


GFSSP model of Couette Flow





Example 25 – Two-dimensional Recirculating Flow in a Driven Cavity

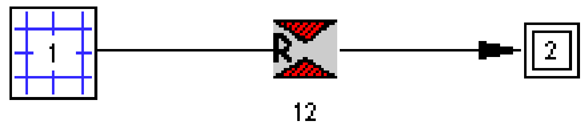


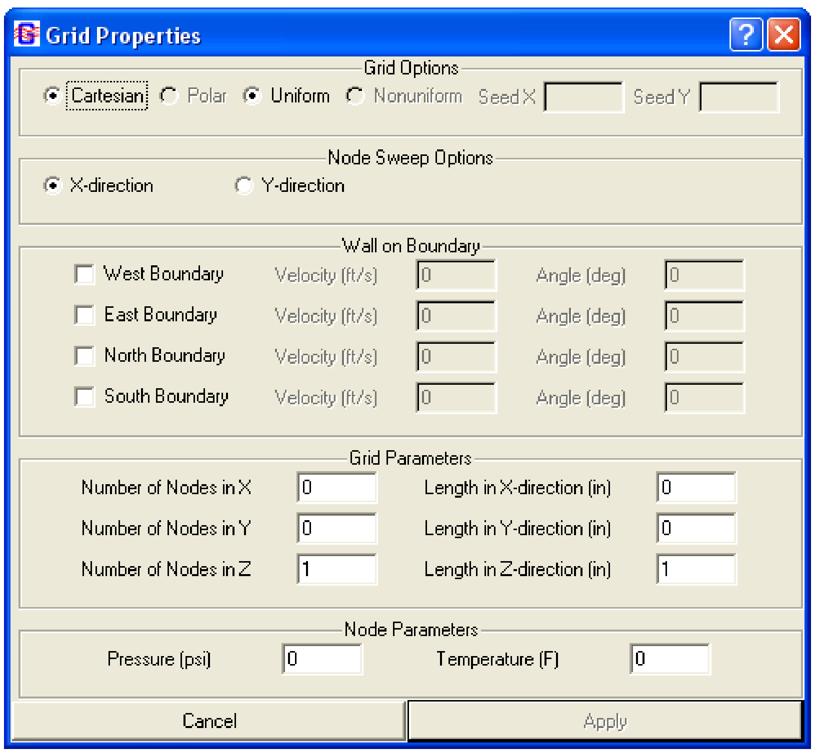
Density = 1 lb/ft^3 ; Viscosity = 1 lb/ft-sec

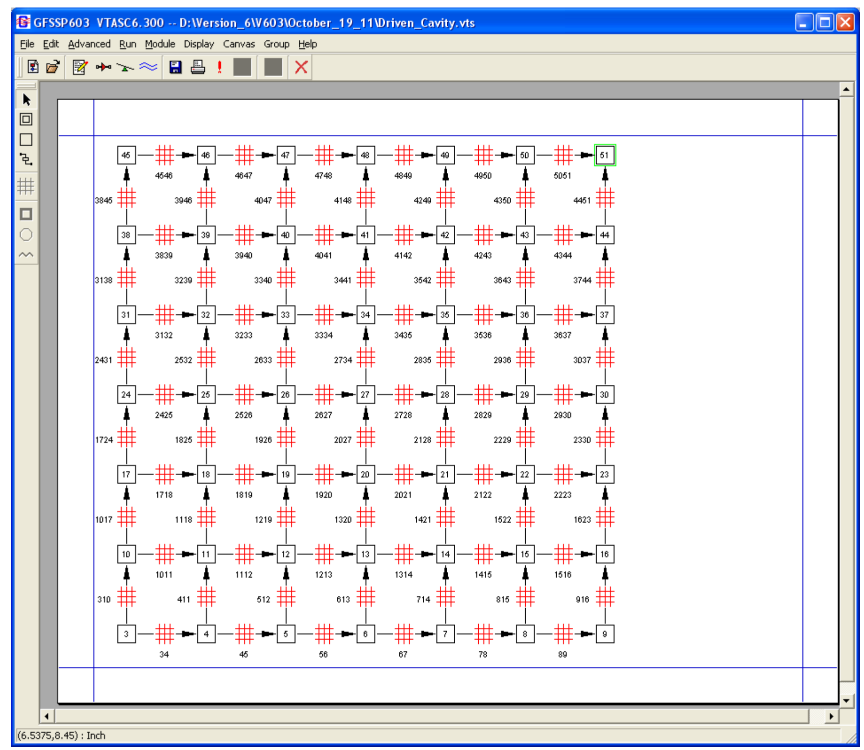
Reynolds Number = 100

Linear Cartesian Grid

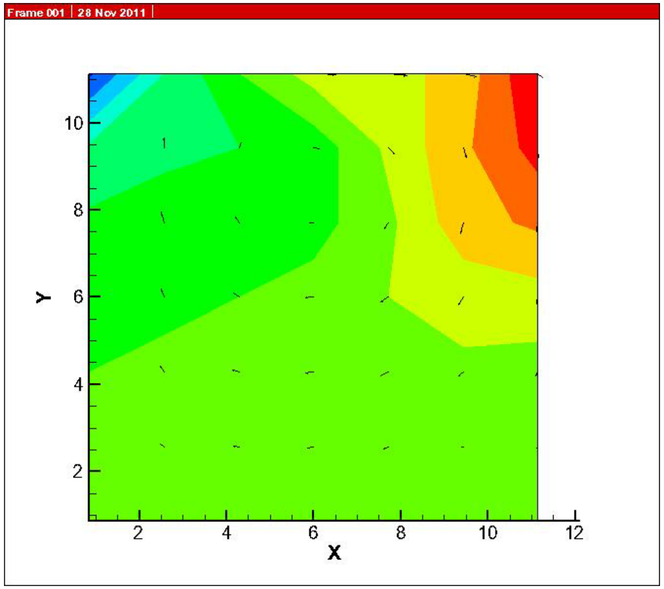
- Generation and display of Two-dimensional

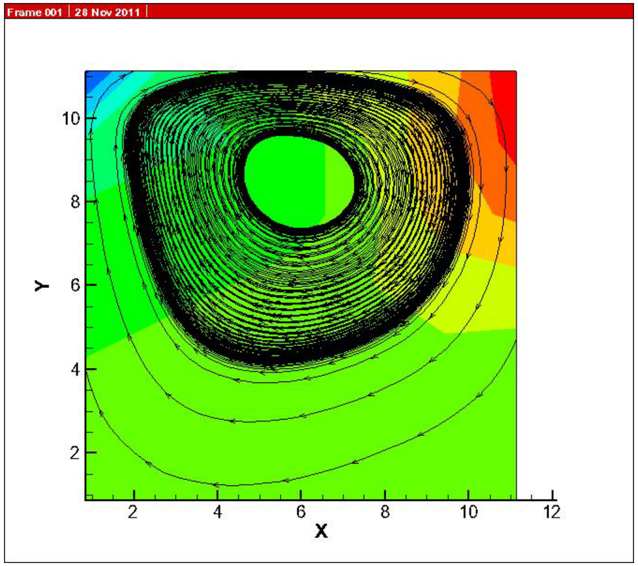


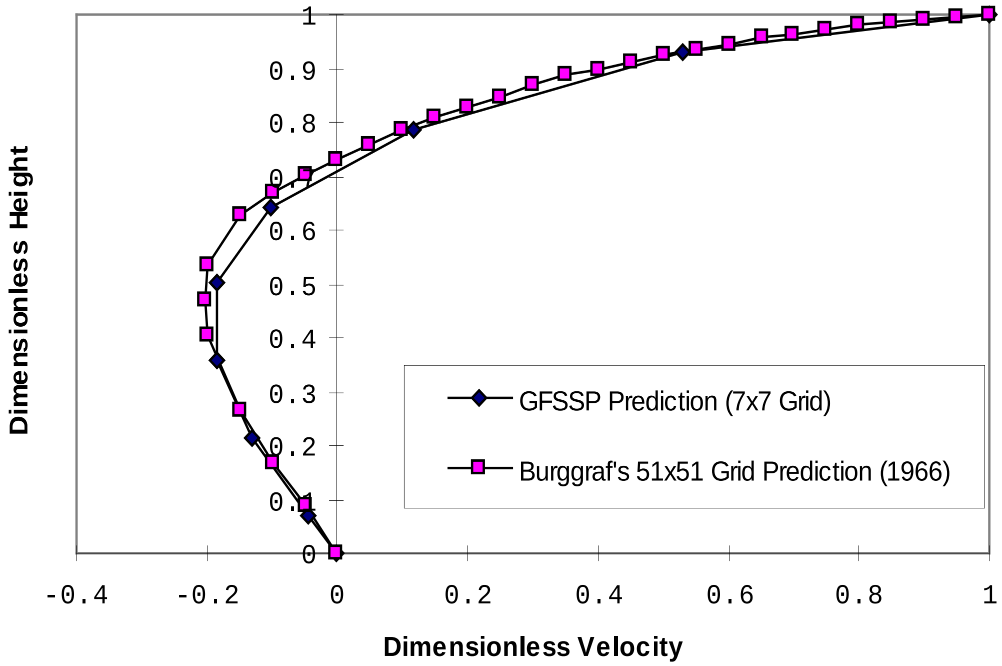




Comparison with Benchmark Solution







Summary

- Numerical Algorithm of GFSSP has been extended to calculate multi-dimensional flow
- GFSSP's unstructured nodal network accounts for transport of scalar variable in n-dimensional space
- One-dimensional momentum equation has been extended to include shear term and transport of longitudinal momentum due to transverse velocity
- The extended formulation has been validated by comparing the numerical prediction with three benchmark solutions:
  - Poiseulle Flow, Couette Flow and Flow in a Driven Cavity
- Future work will include Heat Transfer & Turbulent Flow