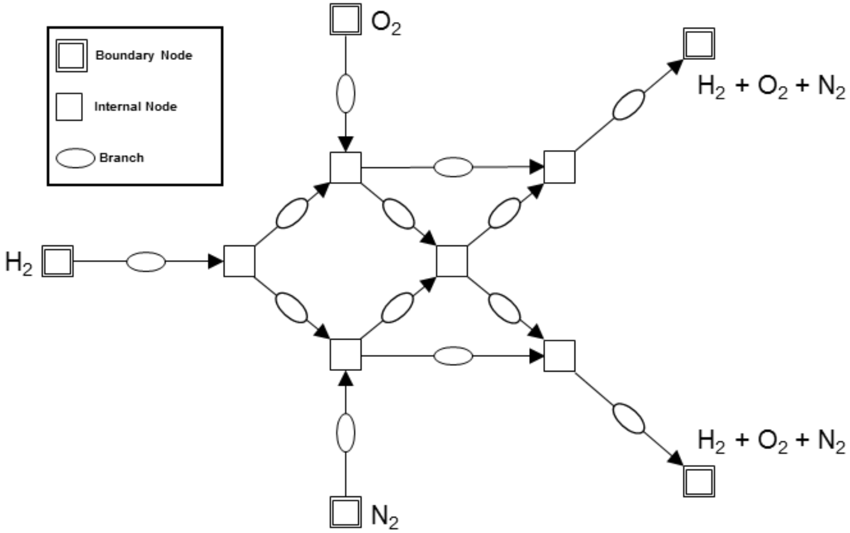
Generalized Fluid System Simulation Program
(Version 702)



Alok Majumdar & Andre LeClair

NASA/Marshall Space Flight Center

GFSSP Training Class

Kennedy Space Center

April 2-4, 2024

Course Overview

- **GFSSP** is a general-purpose computer program
  - Developed at Marshall Space Flight Center (MSFC)
  - Used to analyze Steady-State and Time-dependent Complex Flow Networks
    - Flow rates
    - Pressures
    - Temperatures
    - Concentrations

- **GFSSP Version 702** Training
  - Provides basic introduction and advanced capabilities in **GFSSP**
  - Course designed to quickly teach new users how to use **GFSSP**
  - Lectures and Tutorials cover engineering flow network problems
    - Eight Core Lectures (CL)
    - Nine Lectures on Applications (LA)
    - Six step-by-step Tutorial Problems (TP)
    - Five Challenge Problems

For more information about **GFSSP**:
https://www.nasa.gov/gfssp

Background (1/2)

- **G**eneralized **F**luid **S**ystem **S**imulation **P**rogram (**GFSSP**) Objective
  - Provide a generalized and easy-to-use flow analysis tool

- Started development in 1994
  - General purpose computer program to compute flow network parameters
    - Pressure
    - Temperature
    - Flow distribution in flow network
    - With solid to fluid (conjugate) heat transfer
  - Initially developed to analyze
    - Turbopump Internal Flow
    - Propulsion Systems Transient Flow

Background (2/2)

- History & Ongoing Development
  - Version 1.4 (Steady State); Released in 1996
  - Version 2.01 (Thermodynamic Transient); Released in 1998
  - Version 3.0 (User Subroutine); Released in 1999
  - Graphical User Interface, VTASC; Developed in 2000
  - Selected for NASA Software of the Year Award in 2001
  - Version 4.0 (Fluid Transient and post-processing capability); Released in 2003
  - Version 5.0 (Conjugate Heat Transfer capability); Released in 2006
  - Educational Version; Released in 2011
  - Version 6.0 (Multi-Dimensional Capability); Released in 2014
  - Version 701 (Psychrometric Properties and MLI); Released in December 2015
  - New GUI, MIG, developed in 2017-2018
  - **Version 702**; Test release in Jan. 2020, Updated in Aug. 2020, Feb. 2024
    - Additional heat transfer correlations
    - Common block replaced with modules

Course Outline (1/3)

- 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-3)
  6. Pre & Post Processor – Part 2 (CL-4)
  7. Fluid Transient (LA-2)
  8. Tutorial on Water-hammer (TP-2)

Course Outline (2/3)

- Day 2 Morning

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

- Day 2 Afternoon

4. Rotating Flow, Turbopump, Heat Exchanger (LA-4)
5. Pressure & Flow Regulator (LA-5)
6. Tutorial on Pressure Regulator (TP-4)
7. Multi-D Modeling and Psychrometric Properties (LA-6)
8. Conjugate Heat Transfer (LA-7)
9. Tutorial on Transfer Line Chilldown (TP-5)

Course Outline (3/3)

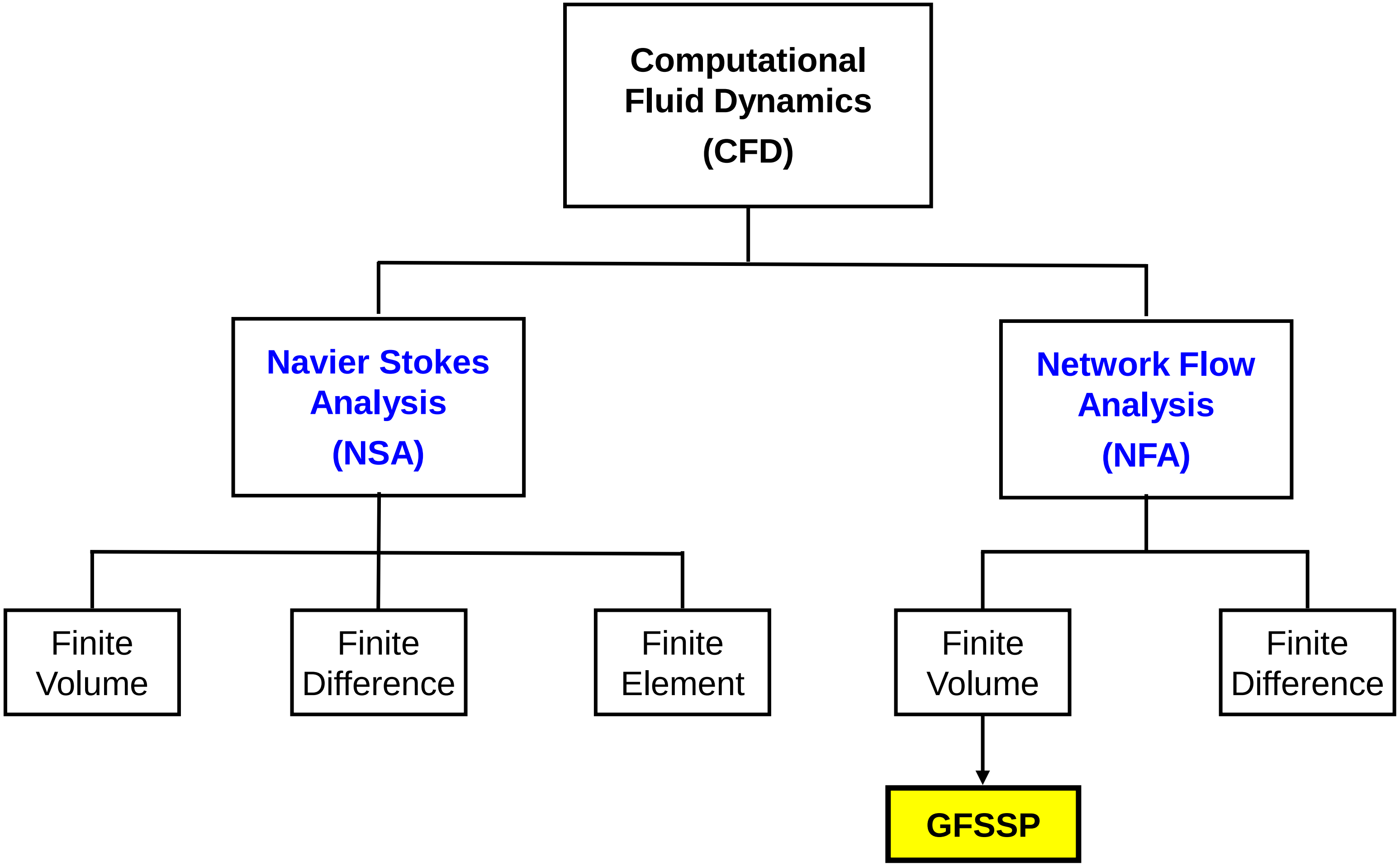
- Day 3 Morning

1. Data Structure (CL-6)
2. User Subroutine (CL-7)
3. Fluid Mixture & Two-phase Flow (LA-8)
4. Tutorial on Propellant Recirculation (TP-6)

- Day 3 Afternoon

5. Model Integration & Future Developments (CL-8)
6. Open Session

Navier Stokes or Network Flow Analysis (1/2)



Computational Fluid Dynamics (CFD)

Navier Stokes Analysis (NSA)

Network Flow Analysis (NFA)

Finite Volume

Finite Difference

Finite Element

Finite Volume

Finite Difference

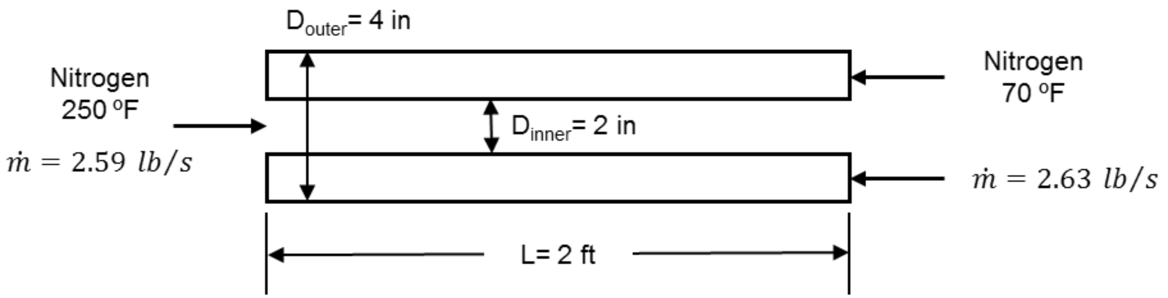
GFSSP

Navier Stokes or Network Flow Analysis (2/2)

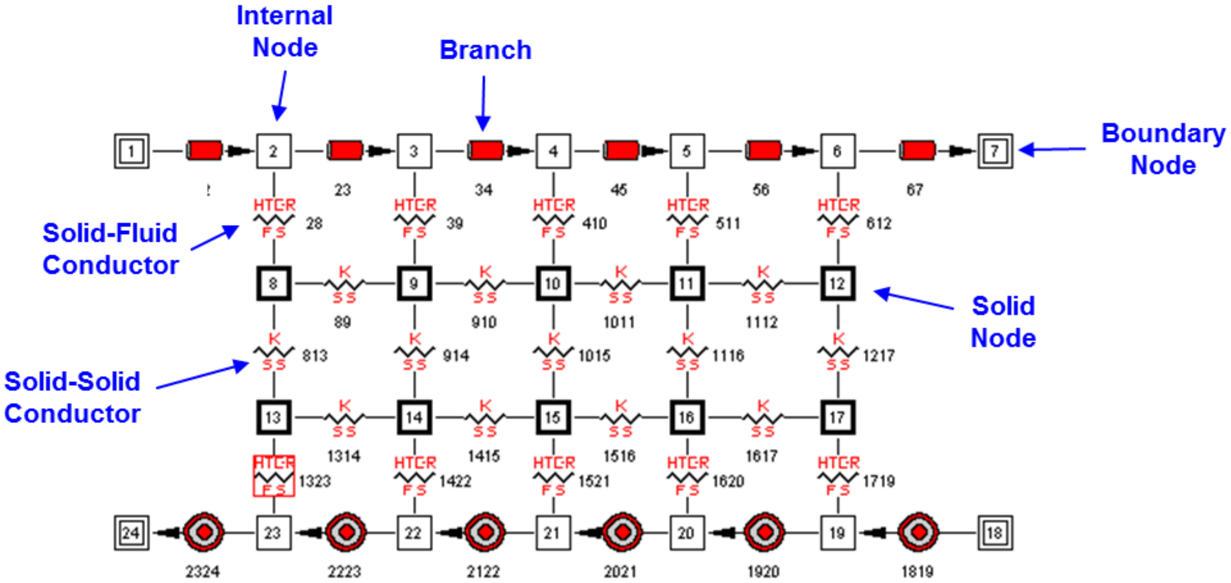
Navier Stokes Analysis	**Network Flow Analysis**
− Suitable for detailed flow analysis within a component	− Suitable for flow analysis of a system consisting of several components
− Requires fine grid resolution to accurately model transport processes	− Uses empirical laws of transport process
− Used after preliminary design	− Used during preliminary design

Network Definition (1/2)

- Flow Problem



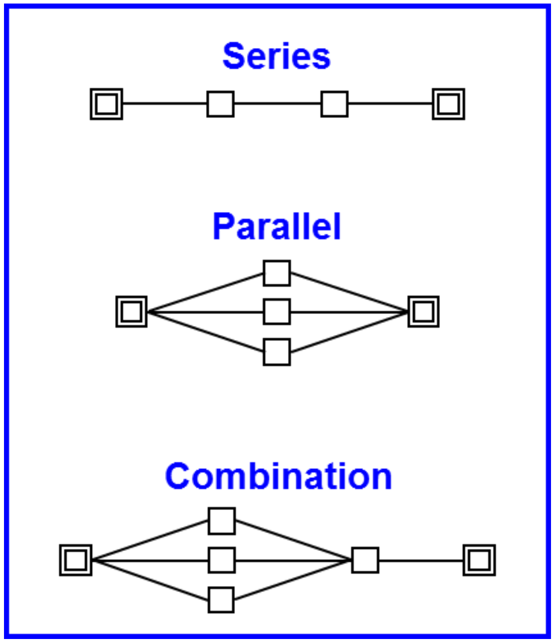
- **GFSSP** Model



Network Definition (2/2)

- Network Symbols
  - ⧈ Boundary node
  - □ Internal node
  - ─▬─ Branch
- Boundary Nodes
  - All dependent variables must be specified
- Internal Nodes
  - All dependent variables
    - Must be guessed for steady flow
    - Must be initially specified for transient flow

Network Types



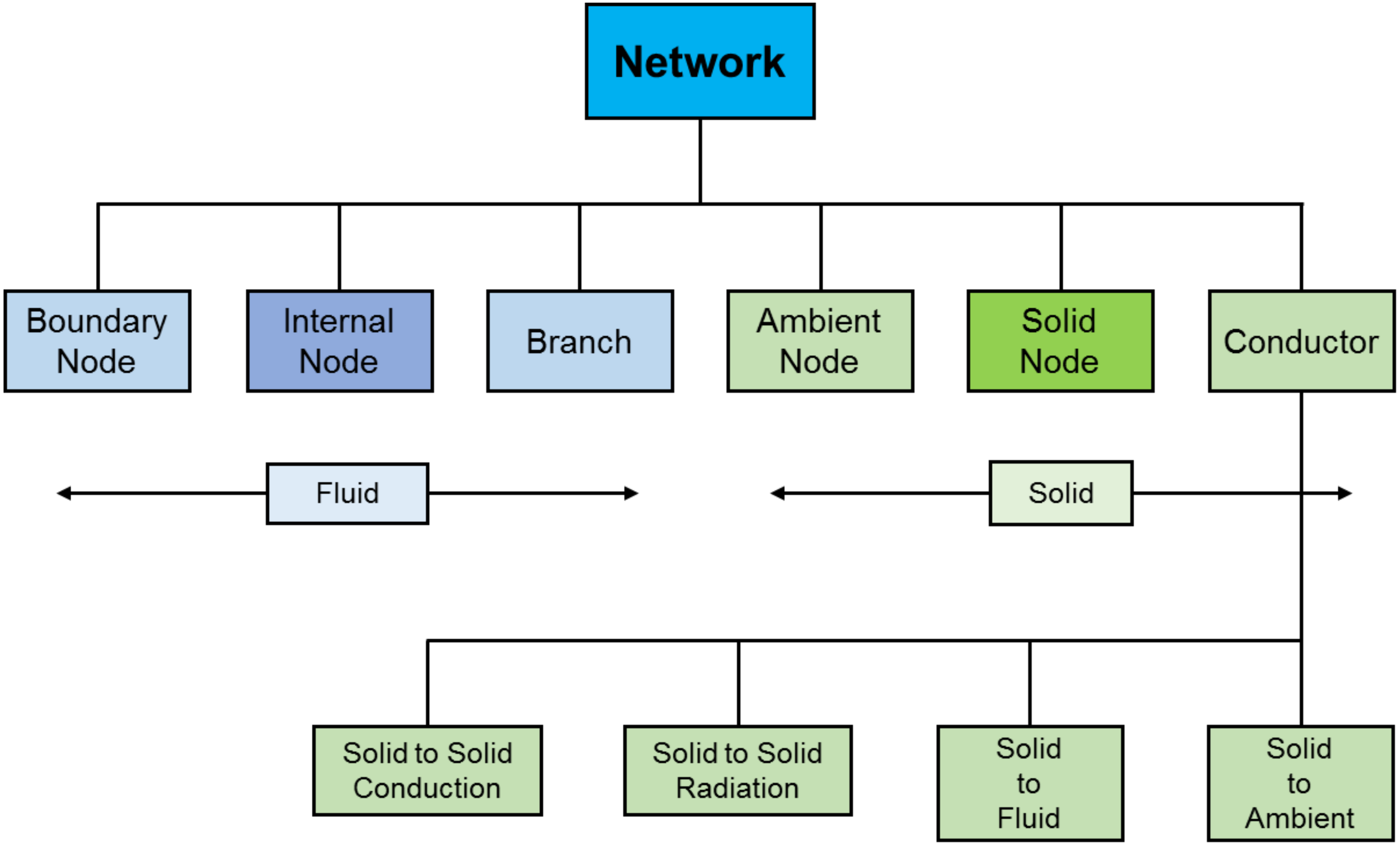
Units and Sign Conventions

- Units

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

- Sign Conventions
  - Mass Input to Node: positive (+)
  - Mass Output from Node: negative (-)
  - Heat Input to Node: positive (+)
  - Heat Output from Node: negative (-)

Data Structure



Network

Boundary Node

Internal Node

Branch

Ambient Node

Solid Node

Conductor

Fluid

Solid

Solid to Solid Conduction

Solid to Solid Radiation

Solid to Fluid

Solid to Ambient

Mathematical Formulation (1/3)

- 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 Concentration	5. Conservation Equations for Mass Fraction of Species
6. Mass	6. Thermodynamic Equation of State

Mathematical Formulation (2/3)

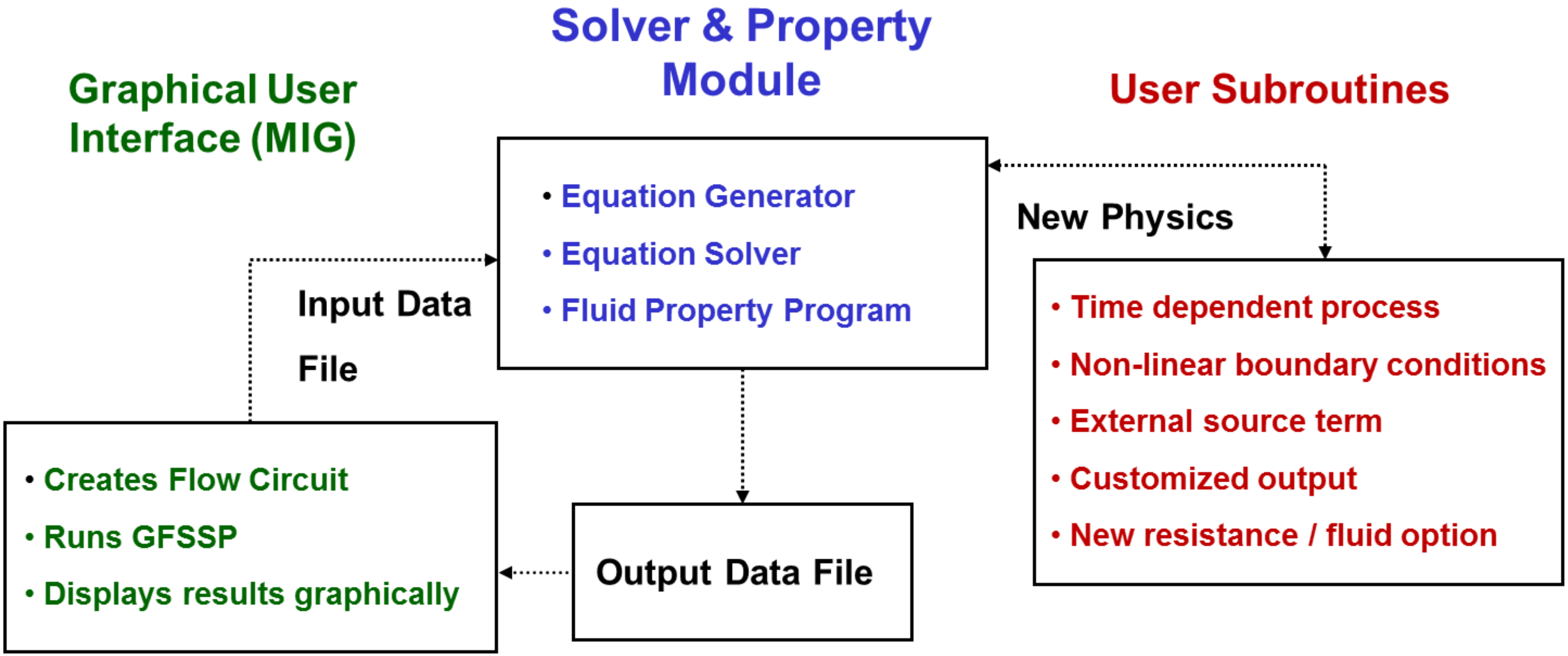
- Auxiliary Variables
  - Thermodynamic Properties
  - Flow Resistance Factor
  - Heat Transfer Coefficient

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 (3/3)

- Governing equations
  - Can generate an infinite number of solutions
- Unique solution obtained with a given set of boundary conditions
- User provides the boundary conditions

Program Structure



Graphical User Interface (MIG)

Solver & Property Module

User Subroutines

- Equation Generator
- Equation Solver
- Fluid Property Program

New Physics

- Time dependent process
- Non-linear boundary conditions
- External source term
- Customized output
- New resistance / fluid option

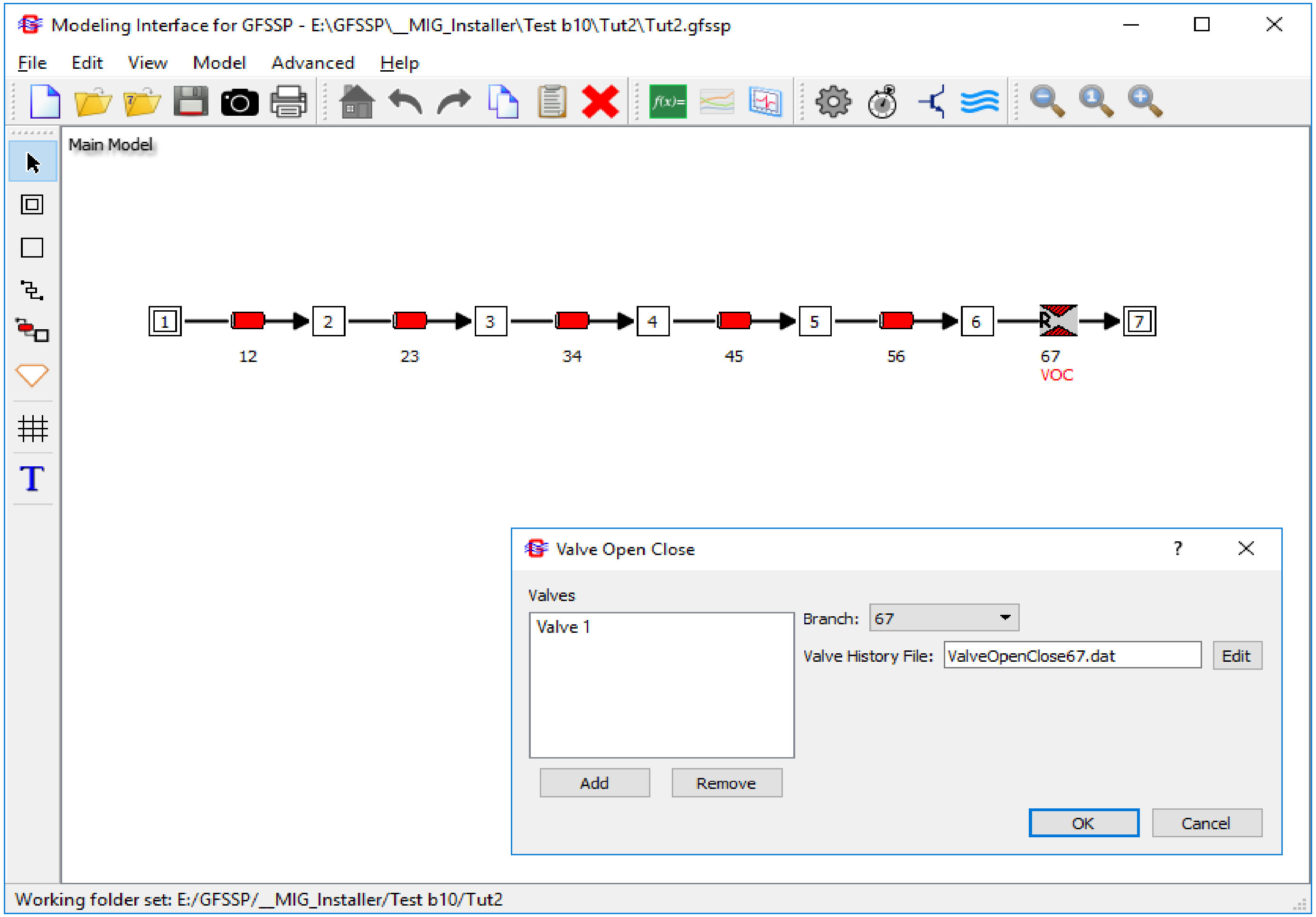
Input Data File

- Creates Flow Circuit
- Runs GFSSP
- Displays results graphically

Output Data File

Graphical User Interface (1/2)

- **MIG**: Model Building



Valves: Valve 1

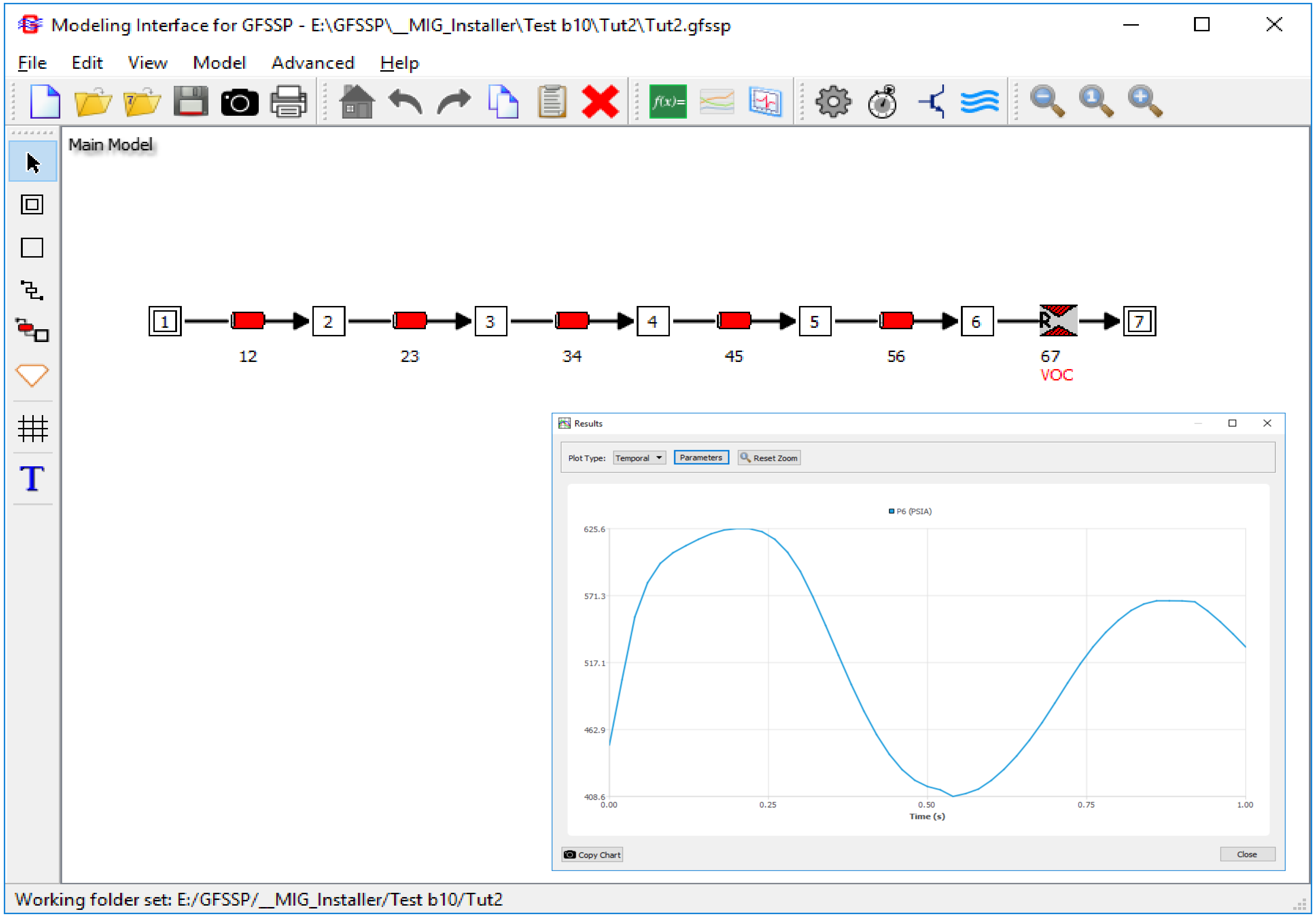
Branch: 67

Valve History File: ValveOpenClose67.dat

Working folder set: E:/GFSSP/__MIG_Installer/Test b10/Tut2

Graphical User Interface (2/2)

- **MIG**: Model Results

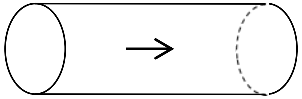


Modeling Interface for GFSSP - E:\GFSSP__MIG_Installer\Test b10\Tut2\Tut2.gfssp

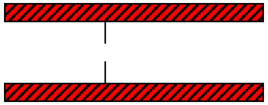
Plot Type: Temporal

Working folder set: E:/GFSSP/__MIG_Installer/Test b10/Tut2

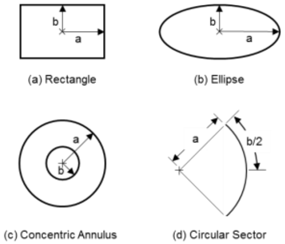
Resistance Options



1. Pipe Flow



2. Flow Through a Restriction



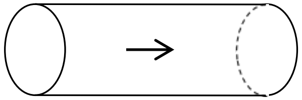
(a) Rectangle

(b) Ellipse

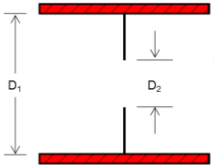
(c) Concentric Annulus

(d) Circular Sector

3. Non-Circular Duct

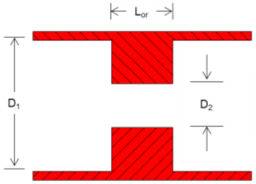


4. Pipe Flow with Entrance & Exit Losses



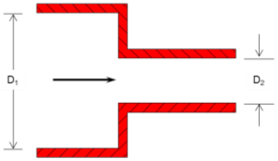
Where:
D_1 = Pipe Diameter
D_2 = Orifice Throat Diameter

5. Thin, Sharp Orifice



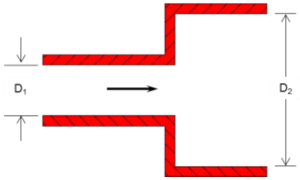
Where:
D_1 = Pipe Diameter
D_2 = Orifice Throat Diameter
L_{or} = Orifice Length

6. Thick Orifice



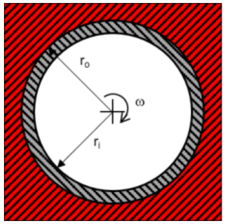
Where:
D_1 = Upstream Pipe Diameter
D_2 = Downstream Pipe Diameter

7. Square Reduction



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

8. Square Expansion



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

Fluid Options

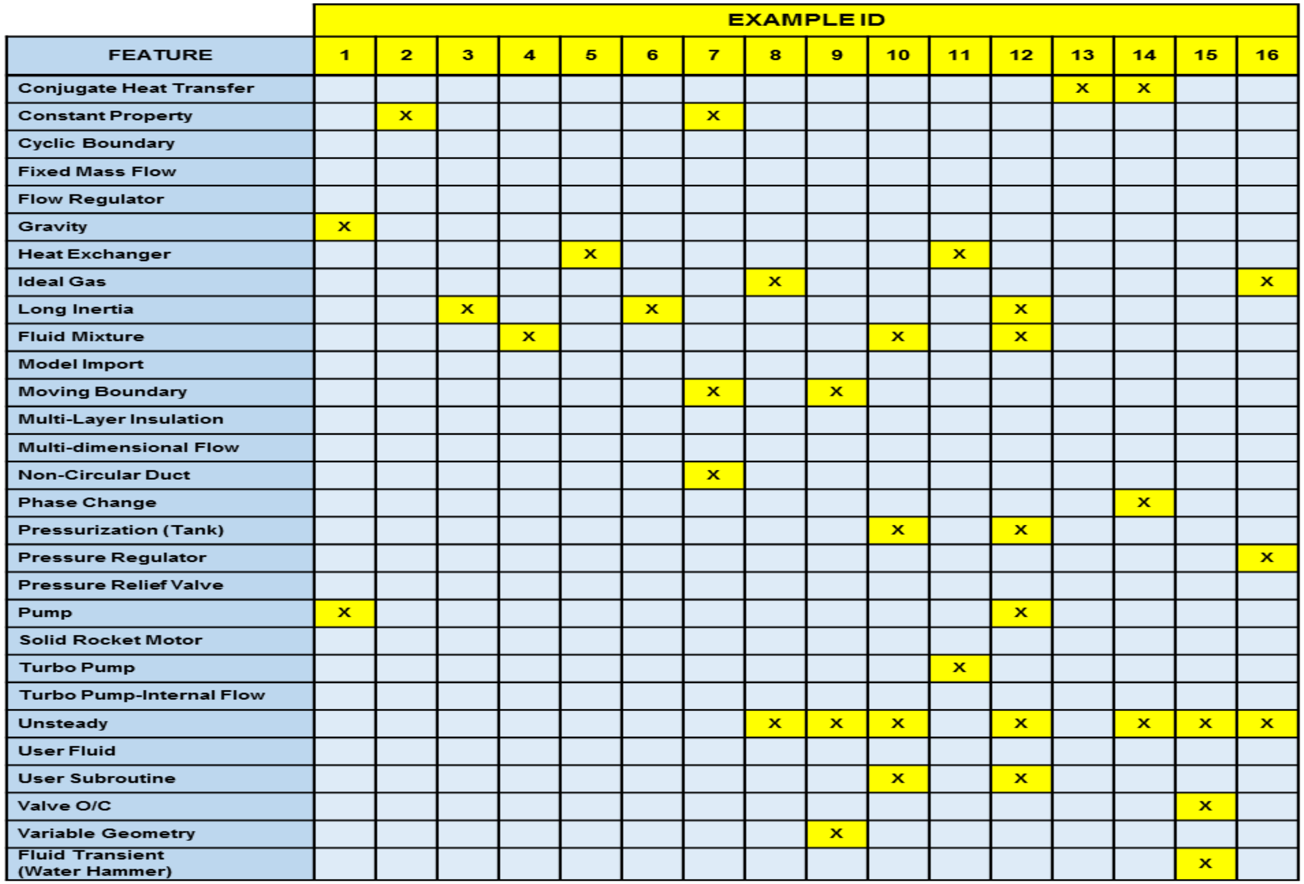
ID Number	SOURCE / FLUID	ID Number	SOURCE / FLUID	ID Number	SOURCE / FLUID
1	GASP He	51	GASPAK He	69	GASPAK Kr
2	GASP CH_4	52	GASPAK CH_4	70	GASPAK Propane
3	GASP Ne	53	GASPAK Ne	71	GASPAK Xe
4	GASP N_2	54	GASPAK N_2	72	GASPAK R-11
5	GASP CO	55	GASPAK CO	73	GASPAK R-12
6	GASP O_2	56	GASPAK O_2	74	GASPAK R-22
7	GASP Ar	57	GASPAK Ar	75	GASPAK R-32
8	GASP CO_2	58	GASPAK CO_2	76	GASPAK R-123
9	GASP F_2	59	GASPAK H_2 (para)	77	GASPAK R-124
10	GASP H_2 (para)	60	GASPAK H_2 (normal)	78	GASPAK R-125
11	WASP H_2O	61	GASPAK H_2O	79	GASPAK R-134A
12	RP-1 Tables	62	GASPAK RP-1 (liq)	80	GASPAK R-152A
		63	GASPAK Isobutane	81	GASPAK N_2F_3
33	Ideal Gas	64	GASPAK Butane	82	GASPAK NH_3
		65	GASPAK Deuterium	84	GASPAK H_2O_2
37	User Fluid 1	66	GASPAK Ethane	86	GASPAK Air
38	User Fluid 2	67	GASPAK Ethylene		
39	User Fluid 3	68	GASPAK H_2S		

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
- Fluid Mixture
- Psychrometric Calculation
- Multi-Layer Insulation

Example Problems (1/4)

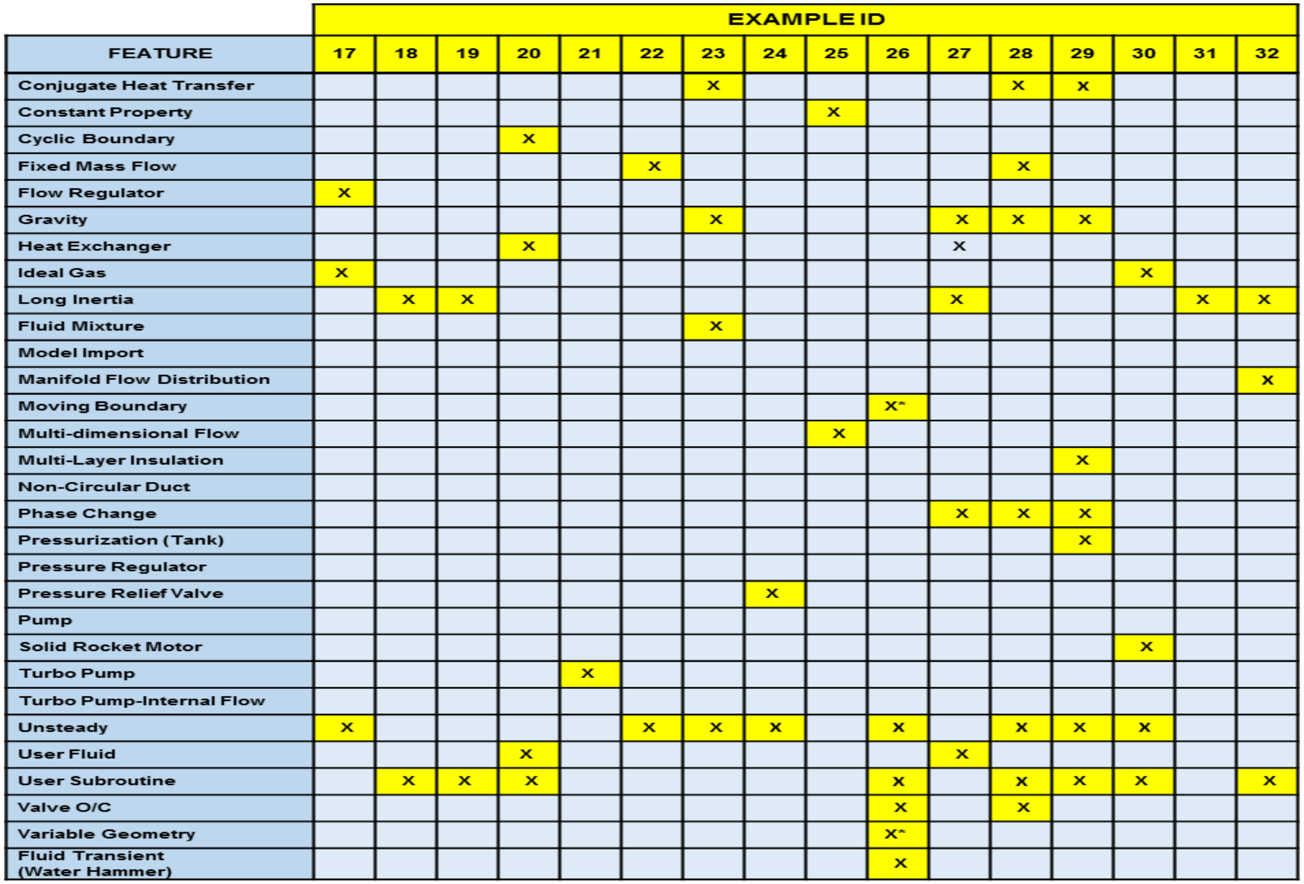
- **GFSSP** User’s Manual: Example Problems 1 - 16 (1/2)
  - Demonstrates major features of GFSSP
  - Provides validation by comparison with textbook solution and/or experimental data



	EXAMPLE ID															
FEATURE	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Conjugate Heat Transfer													X	X		
Constant Property		X					X									
Cyclic Boundary																
Fixed Mass Flow																
Flow Regulator																
Gravity	X															
Heat Exchanger					X						X					
Ideal Gas								X								X
Long Inertia			X			X						X				
Fluid Mixture				X						X		X				
Model Import																
Moving Boundary							X		X							
Multi-Layer Insulation																
Multi-dimensional Flow																
Non-Circular Duct							X									
Phase Change														X		
Pressurization (Tank)										X		X				
Pressure Regulator																X
Pressure Relief Valve																
Pump	X											X				
Solid Rocket Motor																
Turbo Pump											X					
Turbo Pump-Internal Flow																
Unsteady								X	X	X		X		X	X	X
User Fluid																
User Subroutine										X		X				
Valve O/C															X	
Variable Geometry									X							
Fluid Transient (Water Hammer)															X	

Example Problems (2/4)

- **GFSSP** User's Manual: Example Problems 17 - 32 (2/2)
  - Demonstrates major features of GFSSP
  - Provides validation by comparison with textbook solution and/or experimental data



	EXAMPLE ID															
FEATURE	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31	32
Conjugate Heat Transfer							X					X	X			
Constant Property									X							
Cyclic Boundary				X												
Fixed Mass Flow						X						X				
Flow Regulator	X															
Gravity							X				X	X	X			
Heat Exchanger				X							X					
Ideal Gas	X													X		
Long Inertia		X	X								X				X	X
Fluid Mixture							X									
Model Import																
Manifold Flow Distribution																X
Moving Boundary										X*						
Multi-dimensional Flow									X							
Multi-Layer Insulation													X			
Non-Circular Duct																
Phase Change											X	X	X			
Pressurization (Tank)													X			
Pressure Regulator																
Pressure Relief Valve								X								
Pump																
Solid Rocket Motor														X		
Turbo Pump					X											
Turbo Pump-Internal Flow																
Unsteady	X					X	X	X		X		X	X	X		
User Fluid				X							X					
User Subroutine		X	X	X						X		X	X	X		X
Valve O/C										X		X				
Variable Geometry										X*						
Fluid Transient (Water Hammer)										X						

*** Variable geometry & Moving Boundary handled by User Subroutine**

Example Problems (3/4)

- Example Models to be studied in closer detail (1/2)
  - Simple Flow Systems
    - Ex1: Steady-state Water Pumping System
    - Ex2: Water Distribution Network
    - Ex4: Mixing of Hot Combustion Gases with a Cold Gas Stream
    - Ex8: Blow Down of a Pressurized Tank
    - Ex16: Pressure Regulator Downstream of a Pressurized Tank
    - Ex17: Flow Regulator Downstream of a Pressurized Tank
    - Ex22: Fluid Network with the Fixed Flow Rate Option
    - Ex24: Relief Valve in a Pressurized Tank

  - Compressible Flow
    - Ex3: Converging-Diverging Nozzle
    - Ex18: Subsonic Flow with Friction (Fanno Flow)
    - Ex19: Subsonice Flow with Heat Transfer (Rayleigh Flow)

  - Fluid Transient
    - Ex15: Waterhammer after Sudden Valve Closure
    - Ex26: Fluid Transient after Sudden Valve Opening

Example Problems (4/4)

- Example Models to be studied in closer detail (2/2)
  - Tank Pressurization
    - Ex10: Simple Tank Pressurization
    - Ex12: Multiple Tank Pressurization with Control Valves

  - Conjugate Heat Transfer
    - Ex13: Steady-state Conduction through a Rod with Convection
    - Ex14: Chilldown of a Cryogenic Pipeline
    - Ex29: Self-Pressurization of a Cryogenic Propellant Tank Due to Boil-Off

  - Turbomachinery Applications
    - Ex6: Radial Flow on a Rotating Disk
    - Ex11: Power Balancing of a Turbopump Assembly
    - Ex21: Axial Thrust Calculation in the Simplex Turbopump

  - Miscellaneous
    - Ex5: Simple Heat Exchanger
    - Ex20: Lithium Loop Model
    - Ex23: Helium-Assisted, Buoyancy-Driven Flow in a LOx Recirculation Line
    - Ex25: Two-Dimensional Recirculating Flow in a Driven Cavity
    - Ex27: Boiling Water Reactor
    - Ex31: Psychrometrics of Air-Water Vapor Mixture

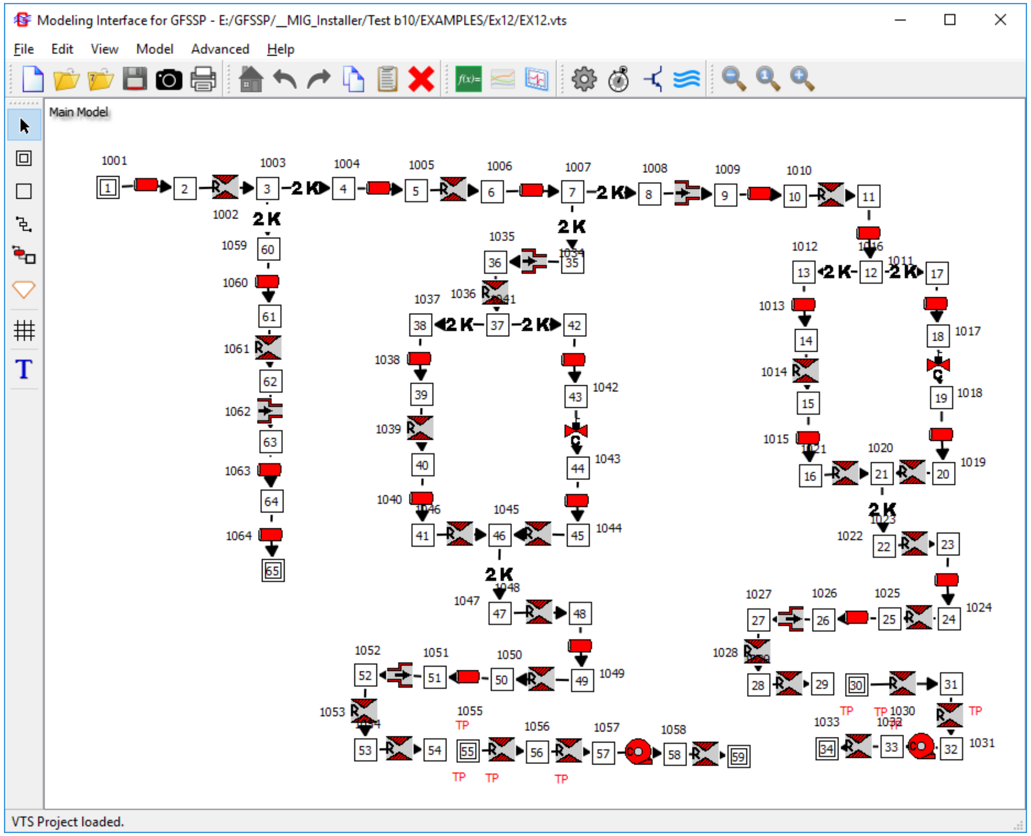
Summary (1/2)

- **GFSSP** is a finite volume based Network Flow Analyzer
- Flow circuit
  - Resolved into a network consisting of Nodes and Branches
- Mass, Energy, and Species conservation
  - Solved at Internal Nodes
- Momentum Conservation
  - Solved at Branches
- Generalized Data Structure
  - Allows generation of all types of flow network
- Modular Code Structure
  - Allows user to add new capabilities with ease
- Unique mathematical formulation
  - Allows effective coupling of thermodynamics and fluid mechanics

Summary (2/2)

- Robust Numerical Scheme
  - Numerical control parameters adjustment is seldom necessary
- Intuitive Graphical User Interface (**MIG**)
  - Makes it easy to build / run / evaluate numerical models
- **GFSSP** has been successfully applied in various applications
  - Incompressible & Compressible flows
  - Phase change (Boiling & Condensation)
  - Fluid Mixture
  - Thermodynamic transient (Pressurization & Blowdown)
  - Pressure and Flow Regulators
  - Fluid Transient (Waterhammer)
  - Conjugate Heat Transfer
  - Model Integration
- Example Problems (32)
  - Illustrate use of various code options

Input/Output Through a Graphical User Interface - MIG



Modeling Interface for GFSSP - E:/GFSSP/__MIG_Installer/Test b10/EXAMPLES/Ex12/EX12.vts

VTS Project loaded.

Content

- Overview
- **MIG** Description
- **MIG** Steady State Demonstration

MIG Overview (1/3)

- **Modeling Interface for GFSSP (MIG)**
  - Program designed to efficiently build flow network models for **GFSSP**

- Visually Interactive
  - "Drag and Drop" Paradigm
  - Model Building, Running, and Post-Processing in one environment

- Self-Documenting
  - Hard copy of flow network
  - JPG image of flow network for inclusion into papers and presentations

MIG Overview (2/3)

- Eliminates errors during model building process
  - Automatic node and branch numbering
  - Save and restore models at any point in the model building process
  - Built-in calculator
  - Input values can be defined as Symbols that can be easily changed for parametric studies

- Pushbutton generation of **GFSSP** input file
  - Steady and Transient cases
  - Advanced features such as Turbopump, Tank Pressurization, and Heat Exchangers

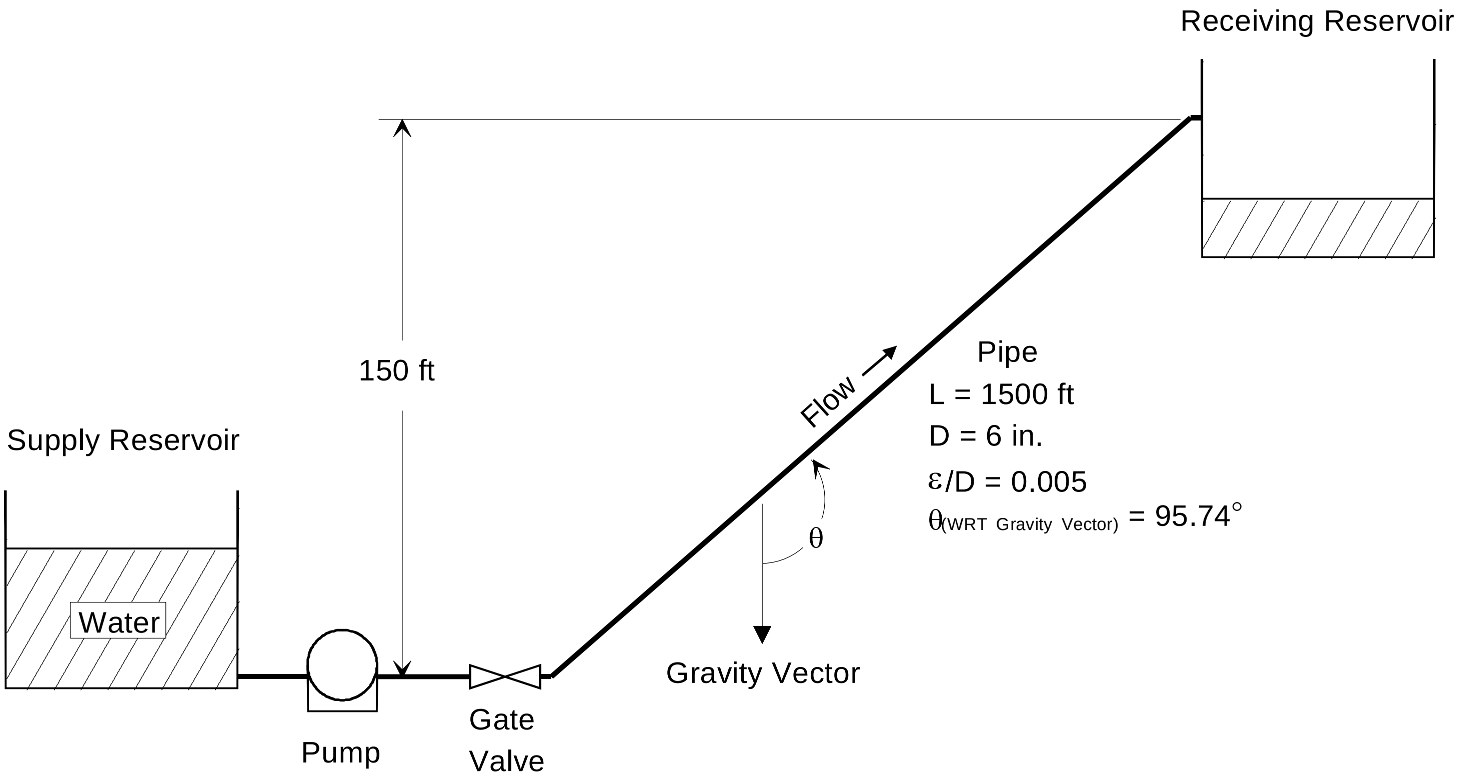
- Run **GFSSP** directly from **MIG** window
  - **GFSSP** Run Manager acts as **MIG/GFSSP** interface

MIG Overview (3/3)

- Post-processing capability allows quick study of results
  - Pushbutton access to **GFSSP** output file
  - Built-in plotting capability for transient cases
  - Capable of plotting through Winplot

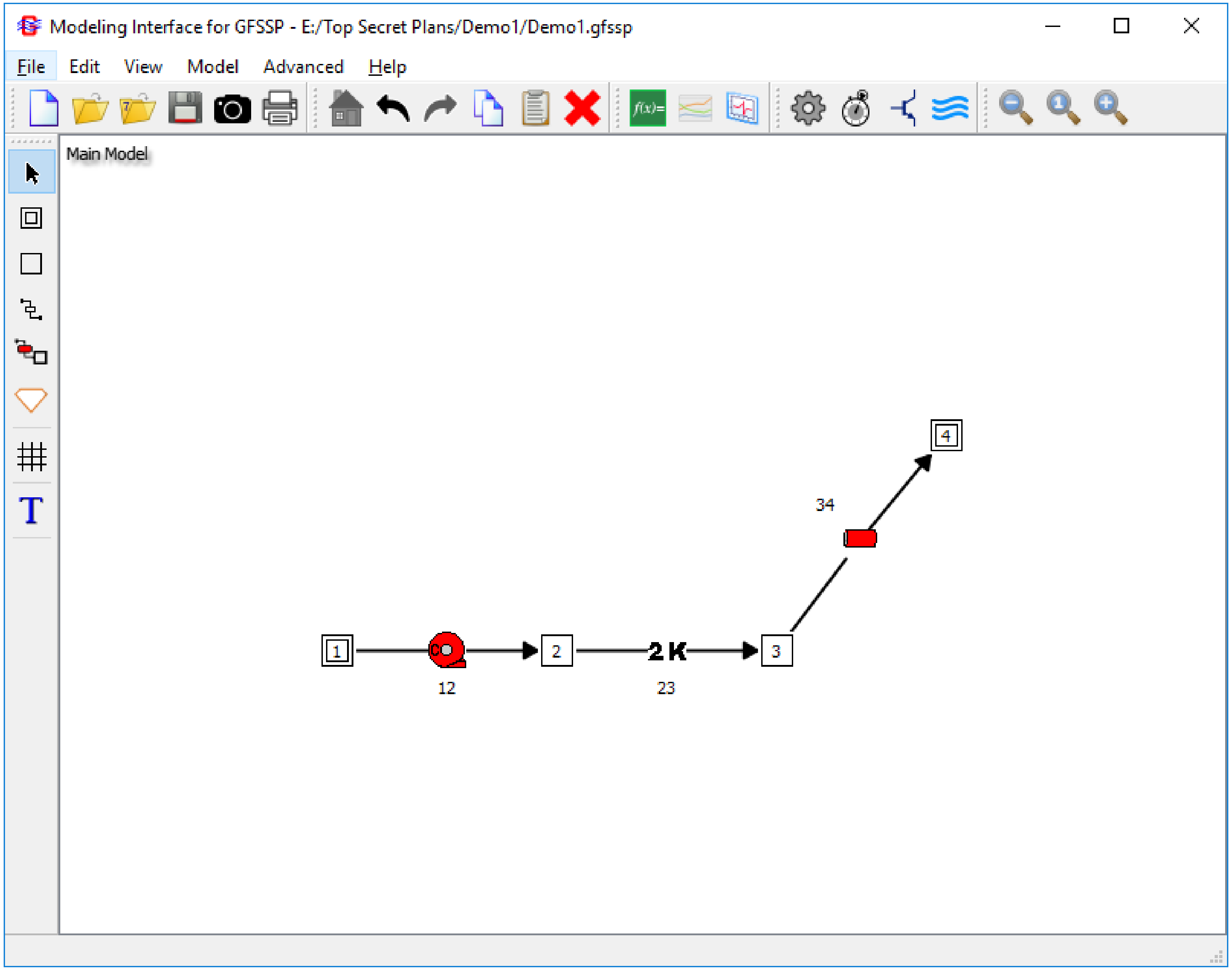
- Develop/Integrate User Subroutines using **MIG**
  - Edit and compile a dynamic link library (DLL) used by the main GFSSP executable

GFSSP Demonstration Problem 1



Pipe
L = 1500 ft
D = 6 in.
$\varepsilon/D = 0.005$
$\theta_{\text{(WRT Gravity Vector)}} = 95.74^\circ$

Build Model on MIG Canvas

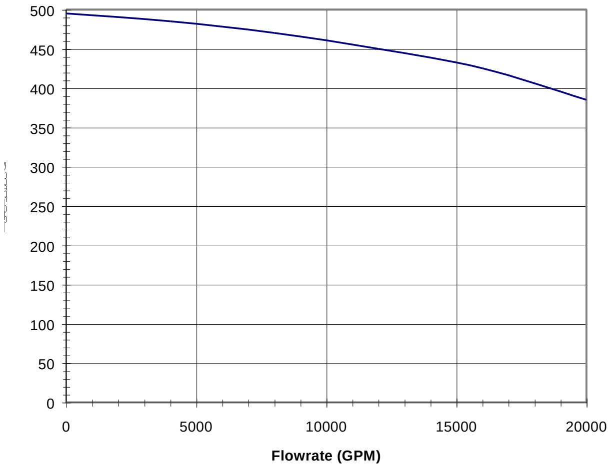


GFSSP 7.02 Preprocessor / Demo 1

Determination of Pump Characteristics

1) Manufacturer's Pump Curve

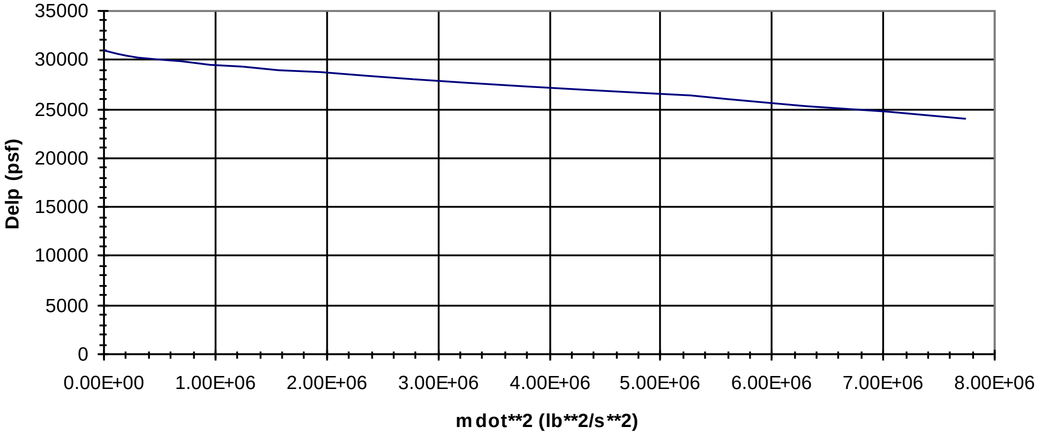
(Head vs. Flowrate)



2) Convert to lb/s and psf

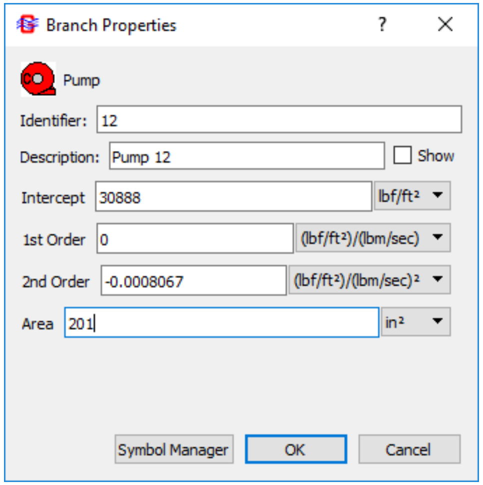
Q (GPM)	$\dot{m}$ (lb$_m$/s)	Head (ft)	ΔP (psf)	$\dot{m}^2$ (lb$_m$/s)^2
0	0	495	30888	0
4000	556.13	485	30264	$3.093\text{x}10^5$
8000	1112.3	470	29328	$1.2372\text{x}10^6$
12000	1668.4	450	28080	$2.784\text{x}10^6$
16000	2224.5	425	26520	$4.948\text{x}10^6$
20000	2781	385	24024	$7.734\text{x}10^6$

3) Plot ΔP vs. $\dot{m}^2$



4) Curve fit

$$\Delta\text{P} = 30888 - 8.067\text{x}10^{-4}\dot{m}^2$$



Pump

Identifier: 12

Description: Pump 12 ☐ Show

Intercept 30888 lbf/ft^2

1st Order 0 (lbf/ft^2)/(lbm/sec)

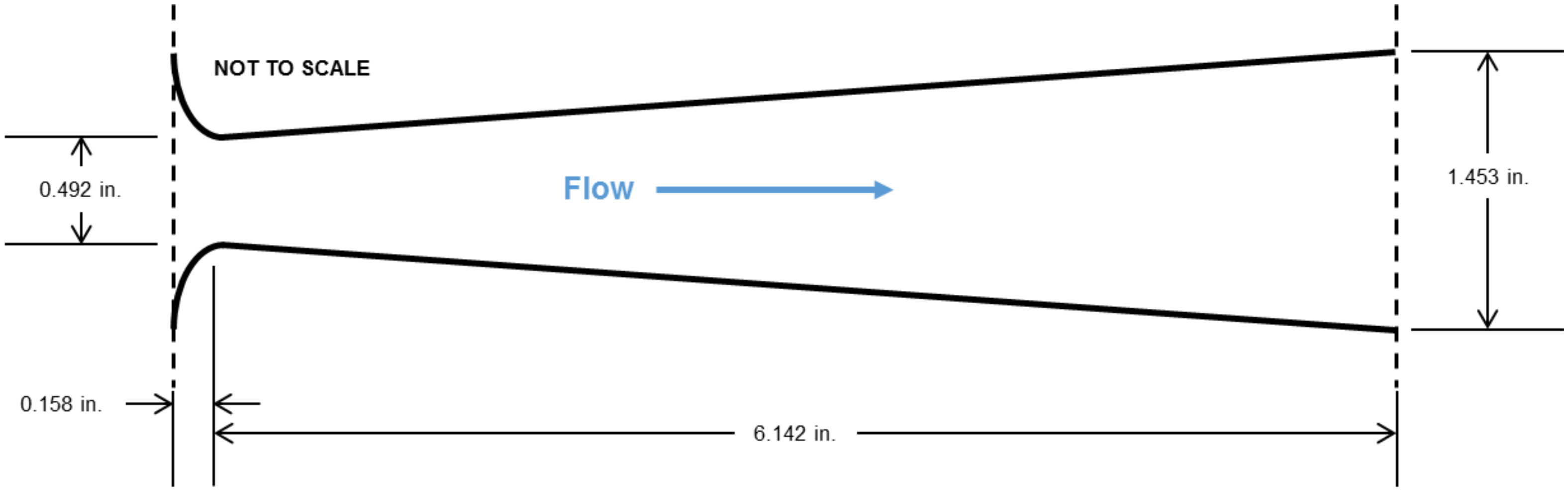
2nd Order -0.0008067 (lbf/ft^2)/(lbm/sec)^2

Area 201 in^2

Summary

- **MIG** is a flow network model builder for use with **GFSSP**
- Interactive “Point and Click” paradigm to design/modify flow networks
- Generates **GFSSP** compatible input files
- Develop/Compile/Link User Subroutines linked from **MIG**
- Winplot can be activated from **MIG** for post-processing

Compressible Flow



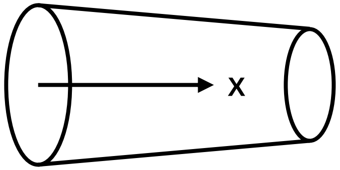
Content

- One-dimensional Compressible Flow
- Compressible Flow Modeling in **GFSSP**
- Converging - Diverging Nozzle (Example 3 & Tutorial 1)
- Example 18: Subsonic Flow with Friction (Fanno Flow)
- Example 19: Subsonic Flow with Heat Transfer (Rayleigh Flow)

One-Dimensional Compressible Flow

- Assumptions
  - 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:

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

$$where\ \mathrm{M} = \mathrm{Mach\ no.} = \frac{V}{c} = \frac{V}{\sqrt{\gamma \frac{p}{\rho}}}$$

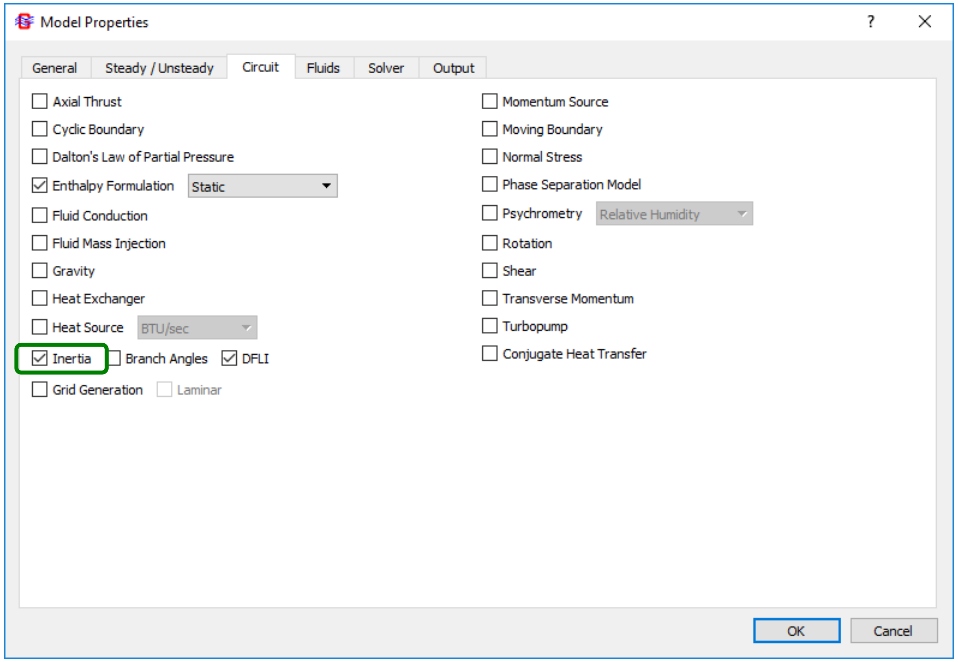
- Analytical Solution

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

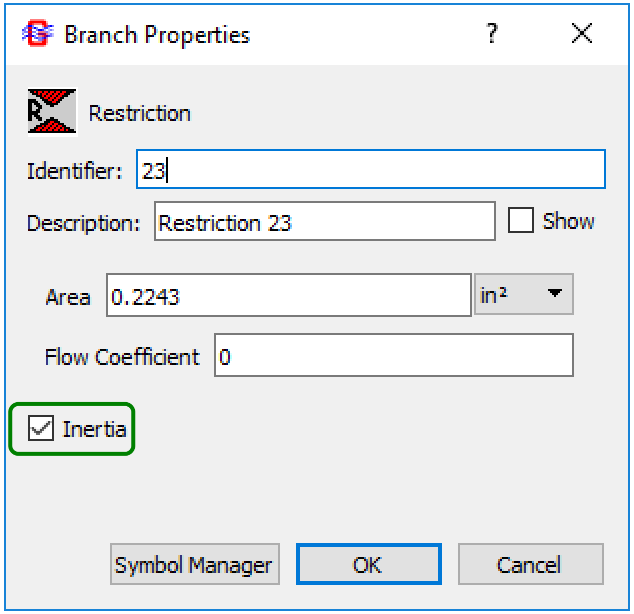
Compressible Flow Modeling in GFSSP

- **GFSSP** considers all fluids to be compressible at all speeds
  - Must activate Inertia term in Momentum Conservation Equation for high speed flows

- Once Inertia term is activated in a branch
  - Upstream pressure becomes static pressure
    - Pressure in the upstream boundary node is always a stagnation pressure



☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☑ Enthalpy Formulation: Static
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☐ Heat Exchanger
☐ Heat Source: BTU/sec
☑ Inertia ☐ Branch Angles ☑ DFLI
☐ Grid Generation ☐ Laminar
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry: Relative Humidity
☐ Rotation
☐ Shear
☐ Transverse Momentum
☐ Turbopump
☐ Conjugate Heat Transfer

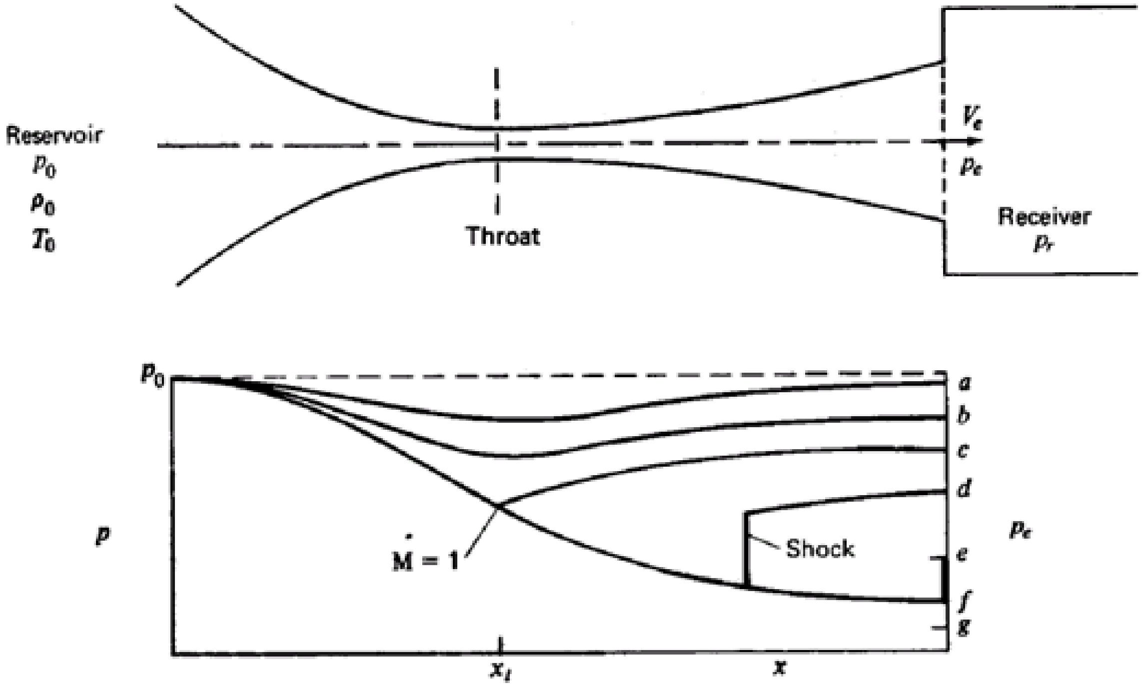


Restriction
Identifier: 23
Description: Restriction 23 ☐ Show
Area 0.2243 in²
Flow Coefficient 0
☑ Inertia

Converging-Diverging Nozzle

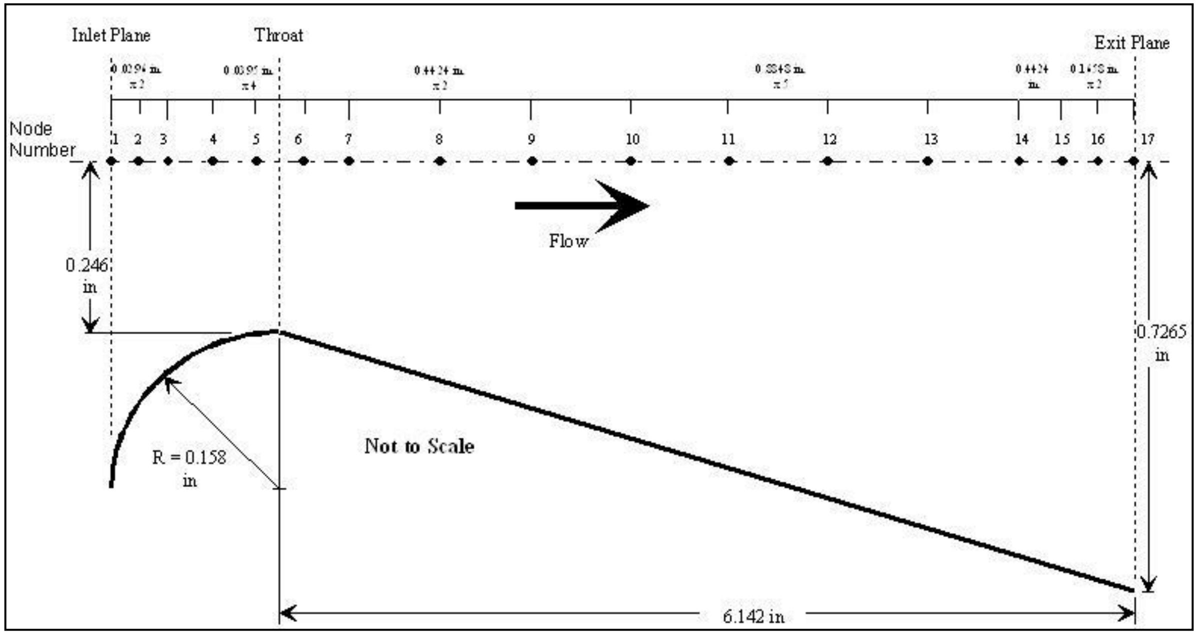
- Effect of Varying Back Pressure

– *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 outside nozzle

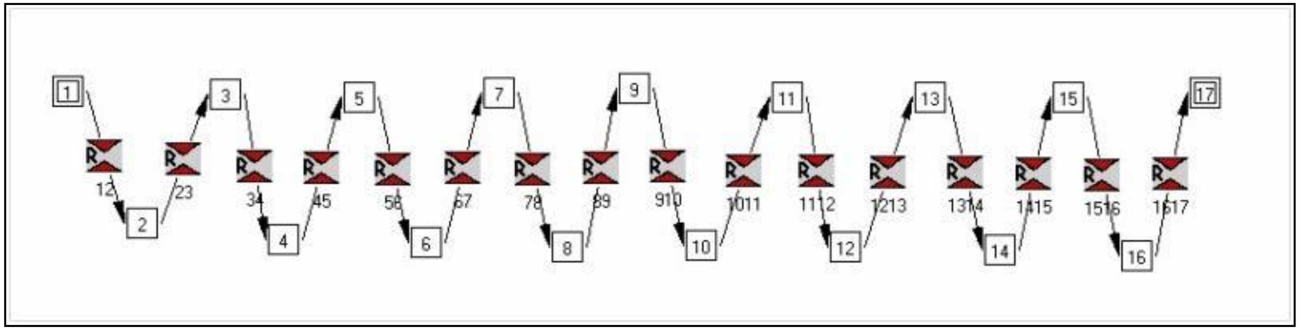


Ex3: Converging-Diverging Nozzle (1/3)

- Detailed Schematic



- MIG Model



Ex3: Converging-Diverging Nozzle (2/3)

- Inputs

Boundary Conditions

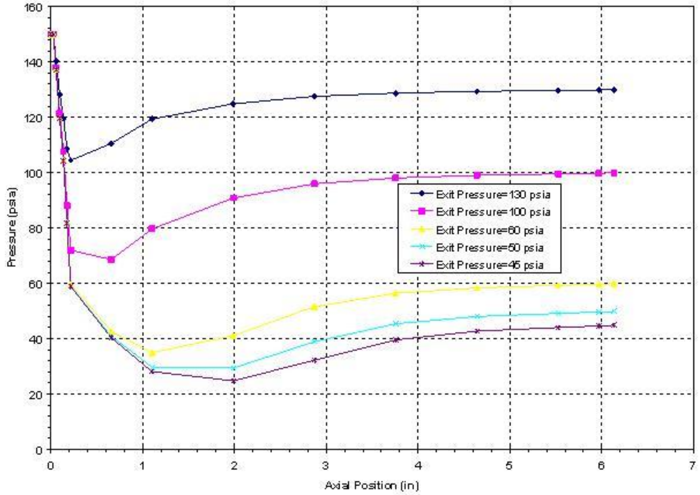
P_1 (psia)	T_1 (°F)	P_{17} (psia)	T_{17} (°F)
150	1000	134	1000
150	1000	100	1000
150	1000	60	1000
150	1000	50	1000
150	1000	45	1000

- **GFSSP** Predictions

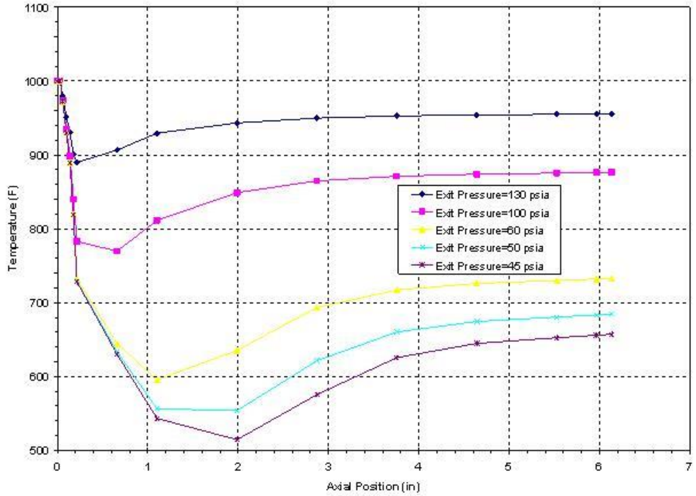
Predicted Mass Flow Rate with Varying Exit Pressure

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

Predicted Pressures for Isentropic Steam Nozzle



Predicted Temperatures for Isentropic Steam Nozzle



Ex3: Converging-Diverging Nozzle (3/3)

- **Isentropic Solution**

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

$$\mathrm{P_{inlet}} = \mathrm{P_{static}}\left(1+\left(\frac{\gamma-1}{2}\right)M^2\right)^{\frac{\gamma}{\gamma-1}}$$

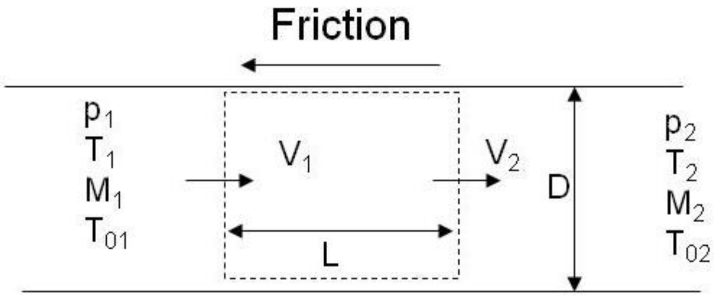
$$= (150\ \mathrm{psia})\left(1+\left(\frac{1.2809\ -1}{2}\right)0.342^2\right)^{\frac{1.2809}{1.2809-1}} = 161.6\ \mathrm{psia}$$

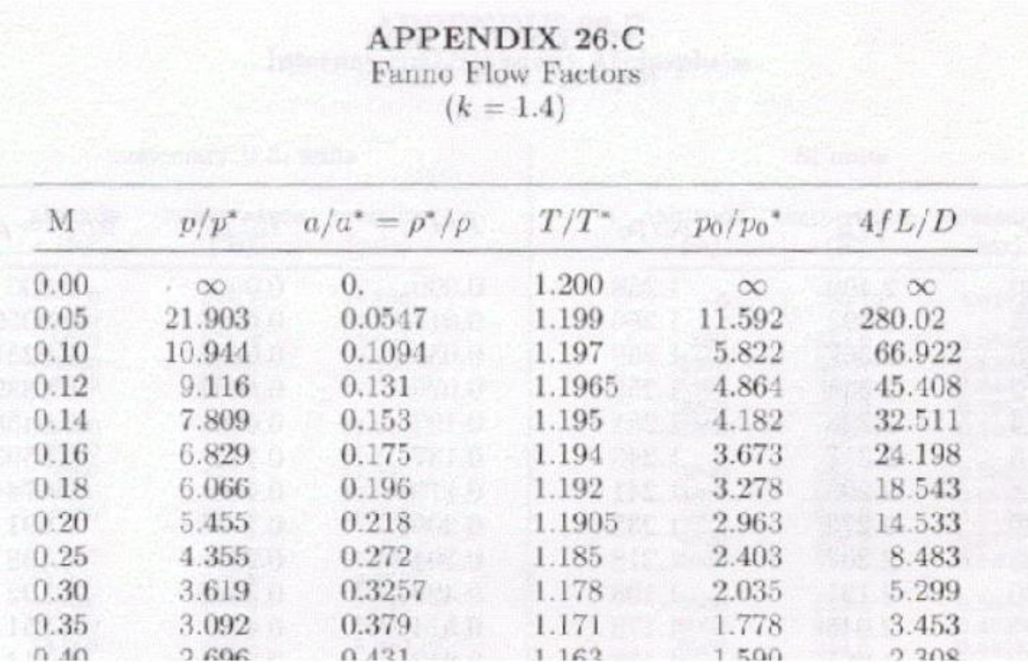
$$\dot{m} = (0.19012\ \mathrm{in}^2)(161.6\ \frac{\mathrm{lb_f}}{\mathrm{in}^2})\sqrt{\frac{32.174\frac{\mathrm{lb_m-ft}}{\mathrm{lb_f-s^2}}(1.281)}{85.83\frac{\mathrm{lb_f-ft}}{\mathrm{lb_m-°R}}(1460°\mathrm{R})}\left(\frac{2}{1.281+1}\right)^{\left(\frac{2.281}{0.281}\right)}} = \mathbf{0.327\ \frac{lb_m}{s}}$$

- **GFSSP-predicted $\dot{m} = 0.337\ \mathrm{lb_m/s}$** (within 3%)

Subsonic Flow with Friction (1/2)

- Fanno Flow





APPENDIX 26.C

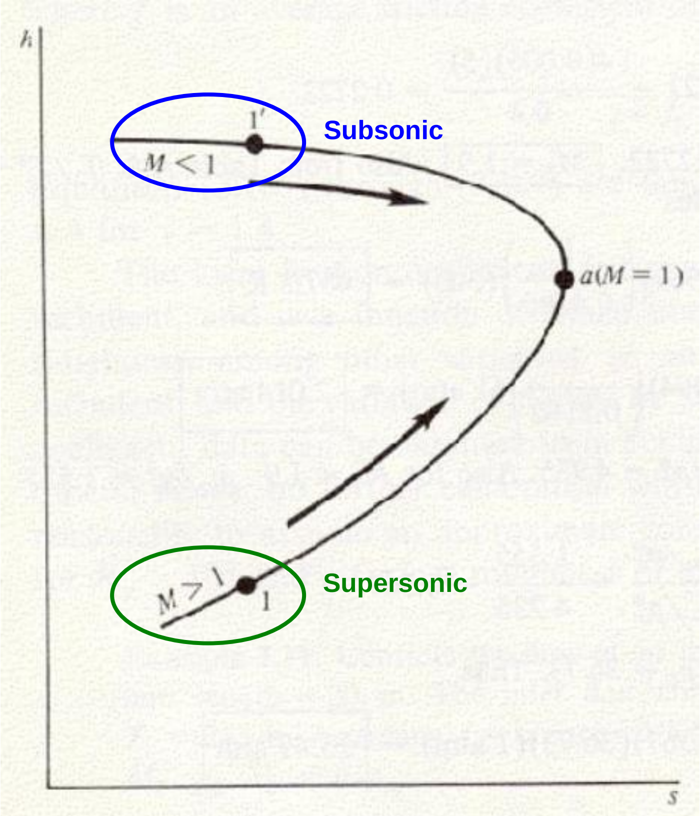
Fanno Flow Factors

$(k = 1.4)$

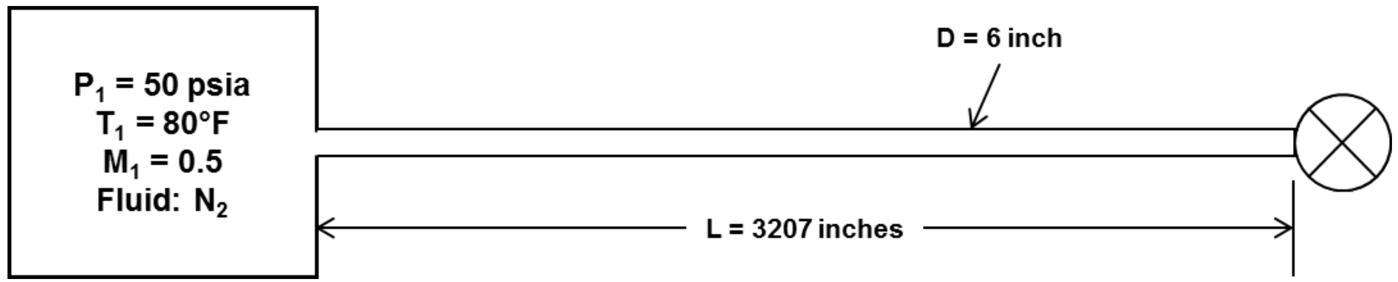
M	p/p^*	$a/a^* = \rho^*/\rho$	T/T^*	$p_0/p_0{}^*$	$4fL/D$
0.00	∞	0.	1.200	∞	∞
0.05	21.903	0.0547	1.199	11.592	280.02
0.10	10.944	0.1094	1.197	5.822	66.922
0.12	9.116	0.131	1.1965	4.864	45.408
0.14	7.809	0.153	1.195	4.182	32.511
0.16	6.829	0.175	1.194	3.673	24.198
0.18	6.066	0.196	1.192	3.278	18.543
0.20	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

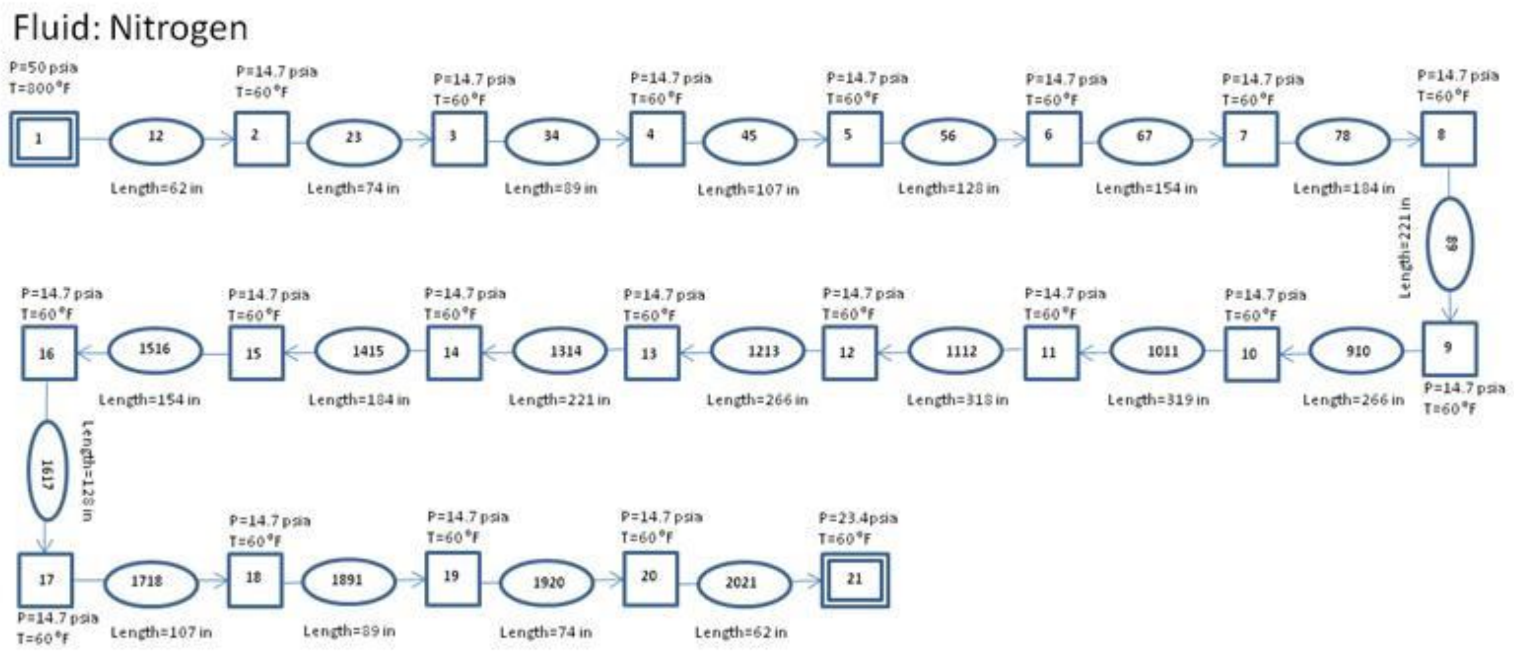
Subsonic Flow with Friction (2/2)

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



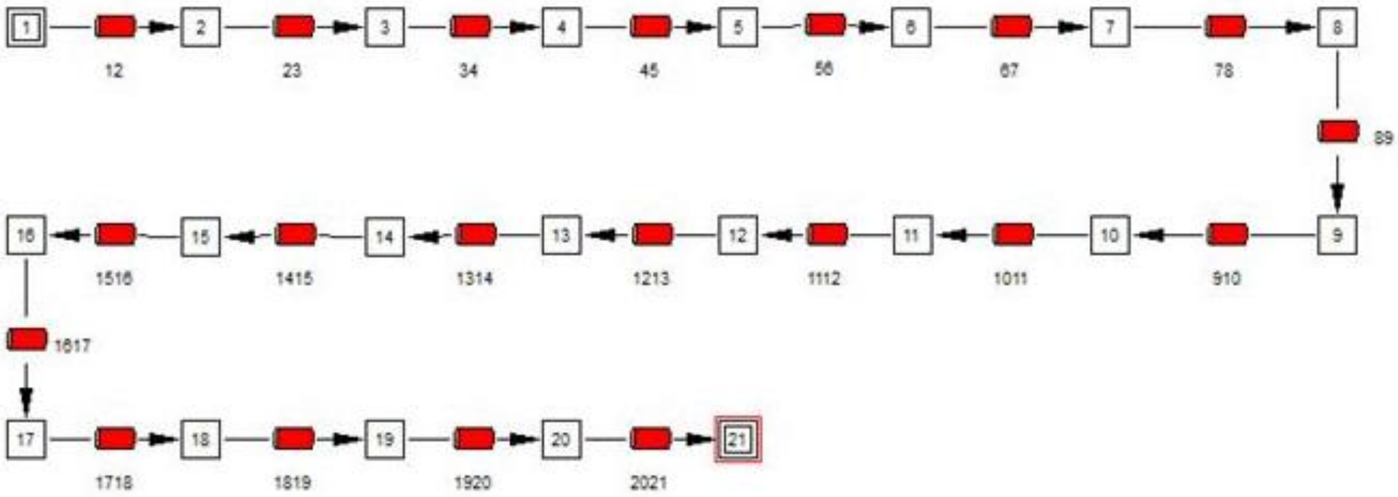
Ex18: Subsonic Flow with Friction (1/3)





Ex18: Subsonic Flow with Friction (2/3)

MIG Model



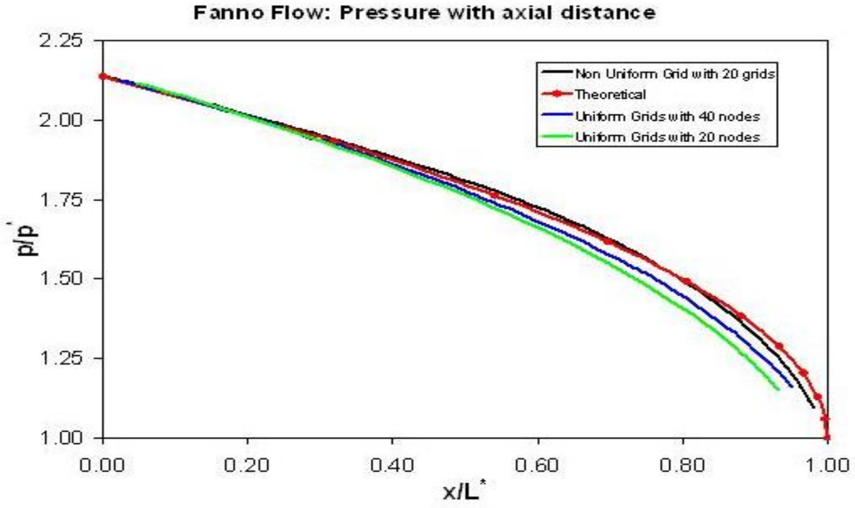
Boundary Conditions

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

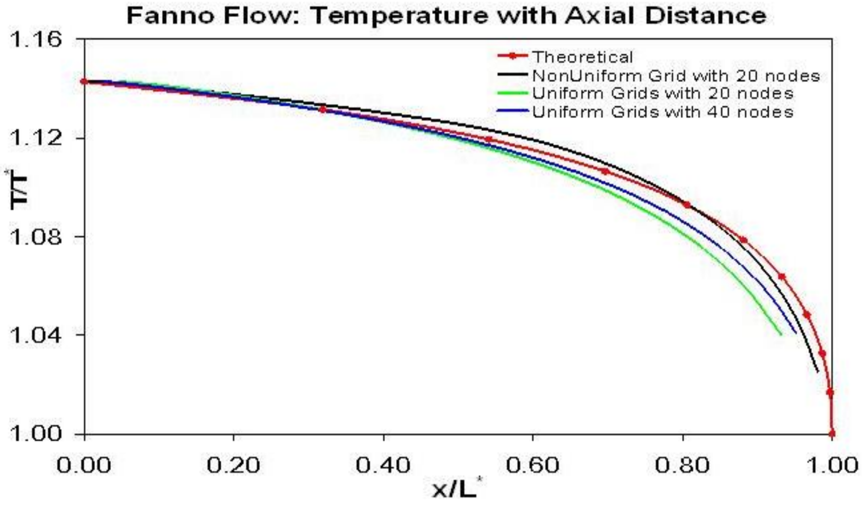
In the User Subroutine:
Friction Factor was set to 0.002
(also used for analytical solution)

Ex18: Subsonic Flow with Friction **(3/3)**

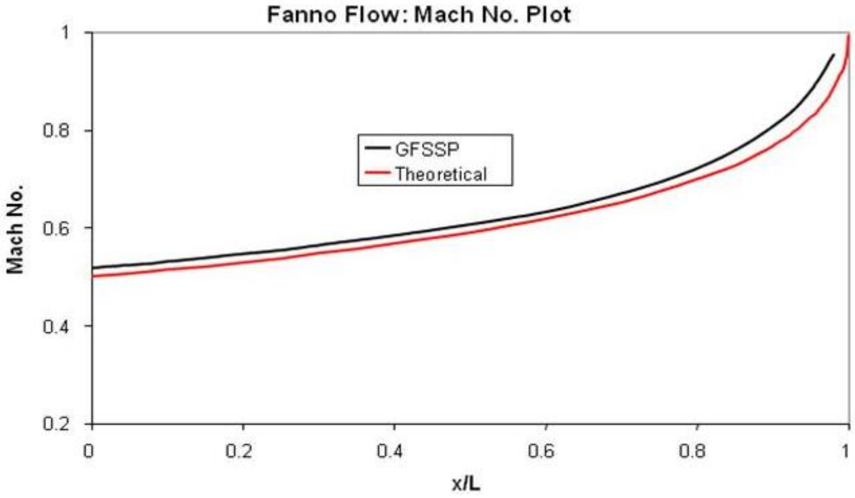
- Comparison with Analytical Solution



Fanno Flow: Pressure with axial distance



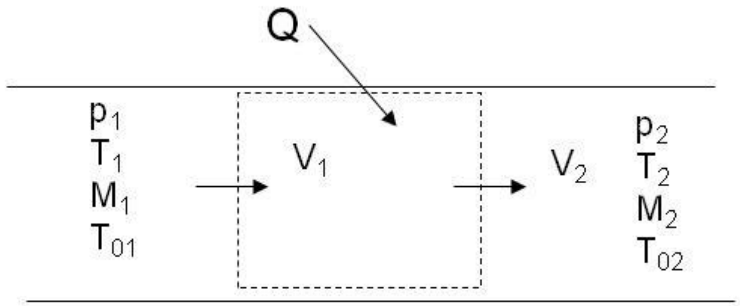
Fanno Flow: Temperature with Axial Distance

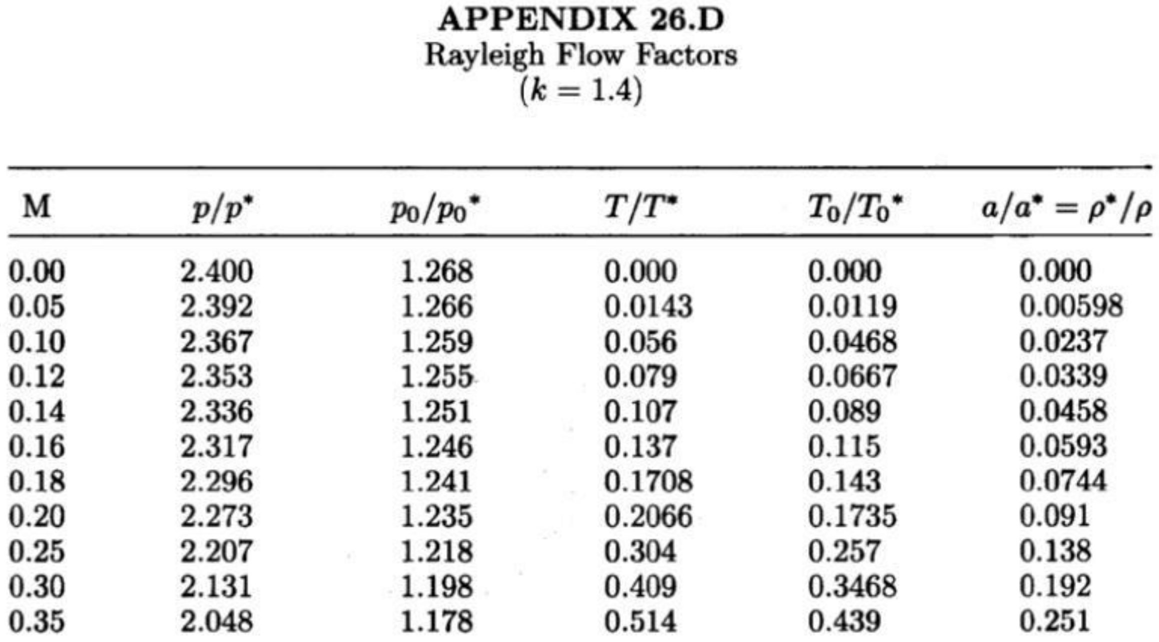


Fanno Flow: Mach No. Plot

Flow with Heat Transfer (1/2)

- 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

Flow with Heat Transfer (2/2)

- Concept of * (star) Quantities and Rayleigh Curve

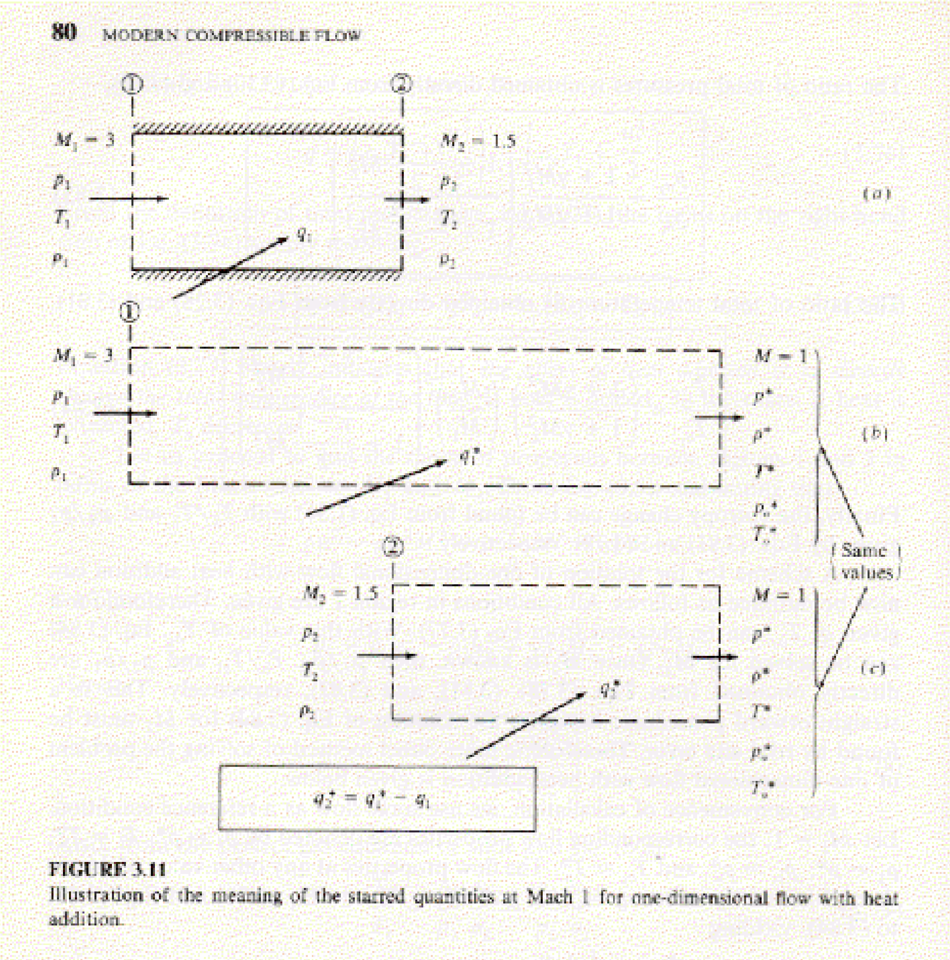


FIGURE 3.11
Illustration of the meaning of the starred quantities at Mach 1 for one-dimensional flow with heat addition.

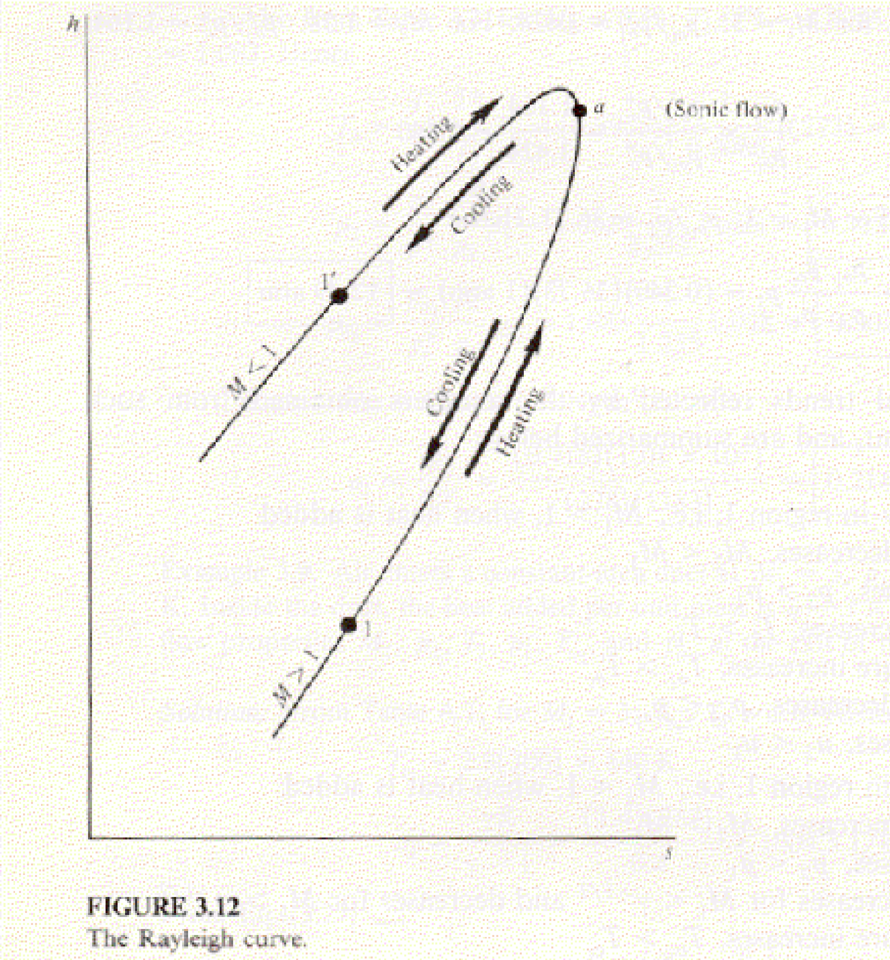
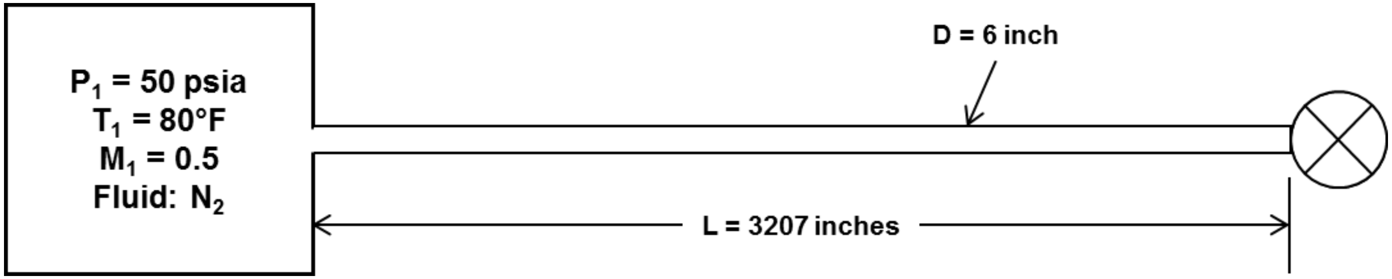
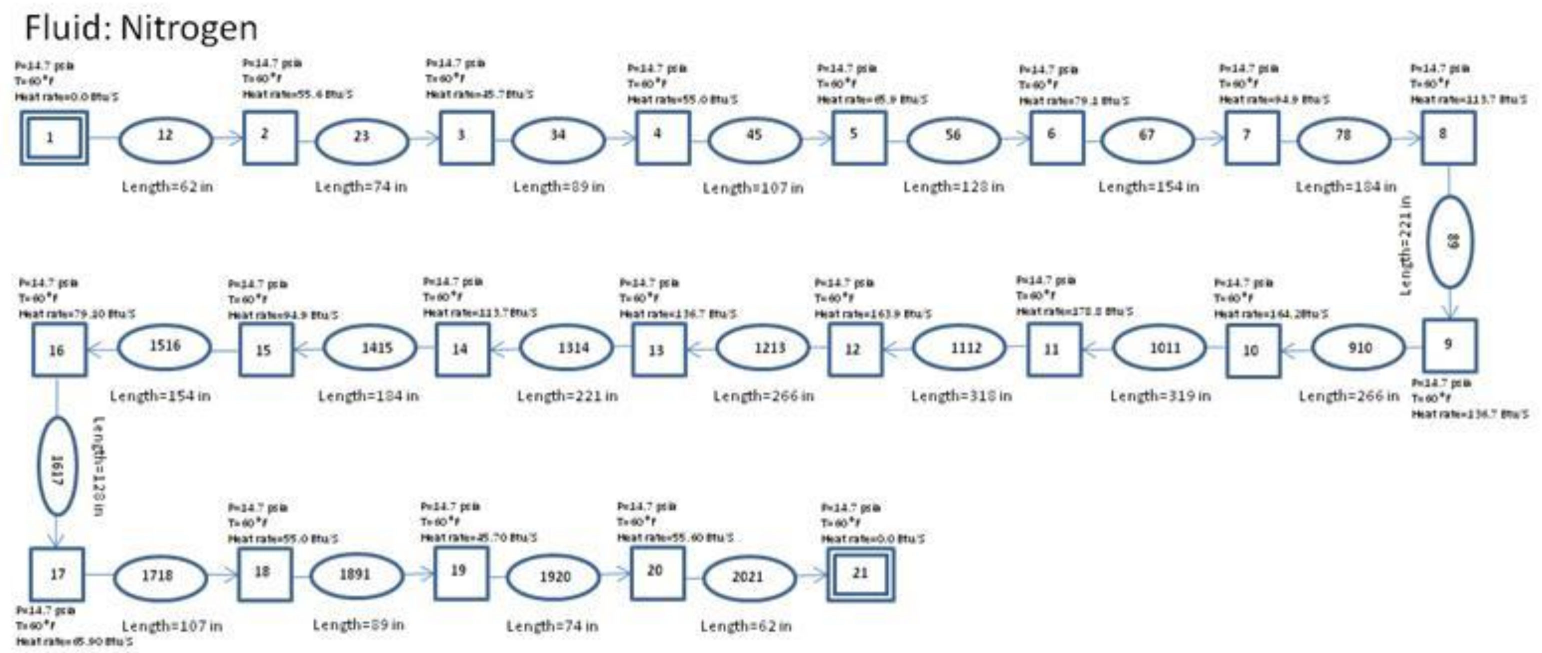


FIGURE 3.12
The Rayleigh curve.

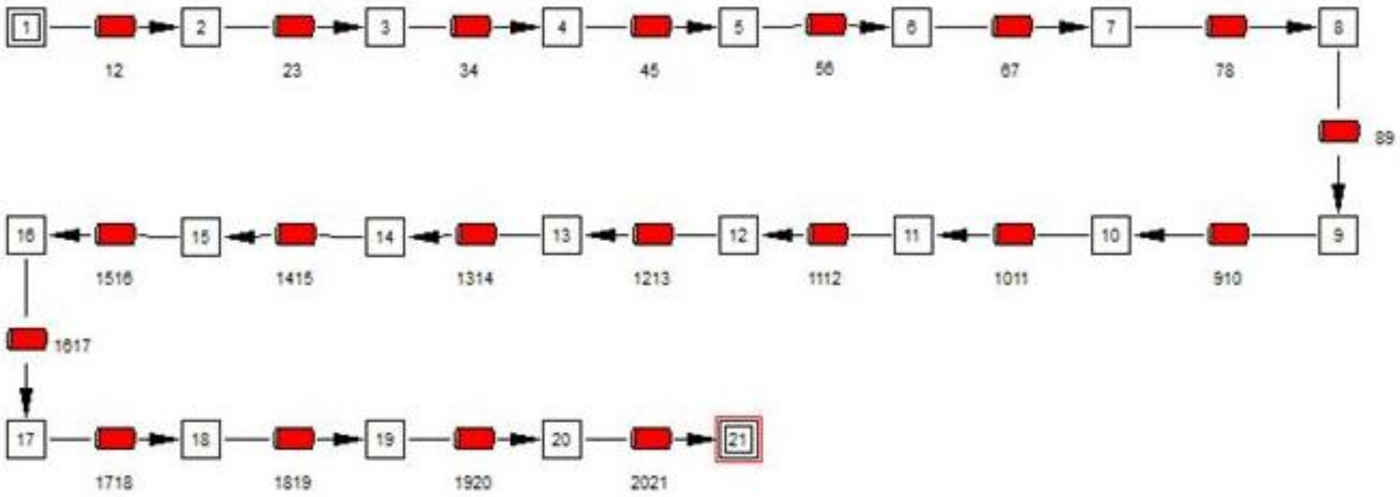
Ex19: Subsonic Flow with Heat Transfer (1/3)





Ex19: Subsonic Flow with Heat Transfer (2/3)

MIG Model



Boundary Conditions

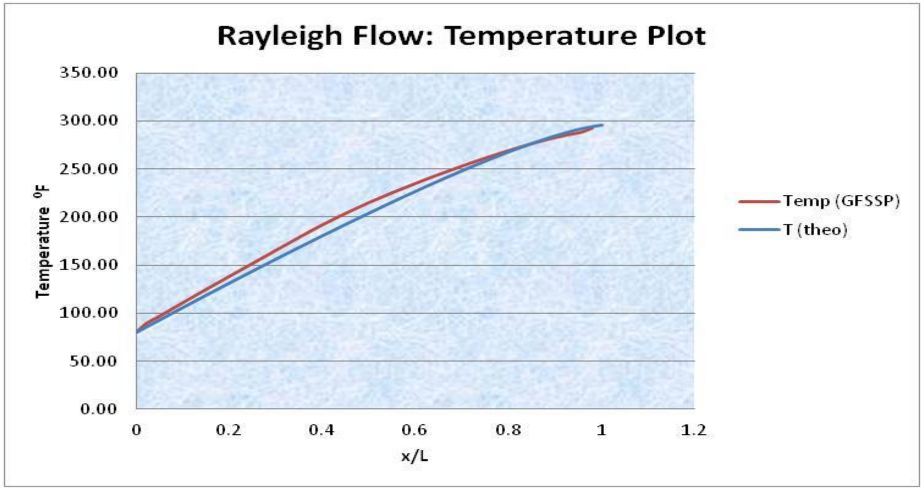
Boundary Node Number	Pressure (psia)	Temperature (ºF)
1	50	80
21	35	40

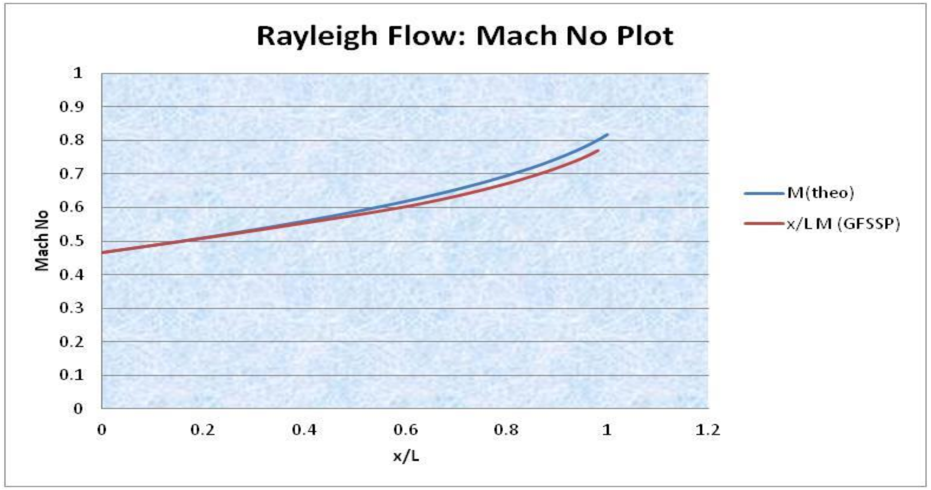
In the User Subroutine: Friction Factor was set to zero

(to eliminate frictional effect)

Ex19: Subsonic Flow with Heat Transfer (3/3)

- Comparison with Analytical Solution





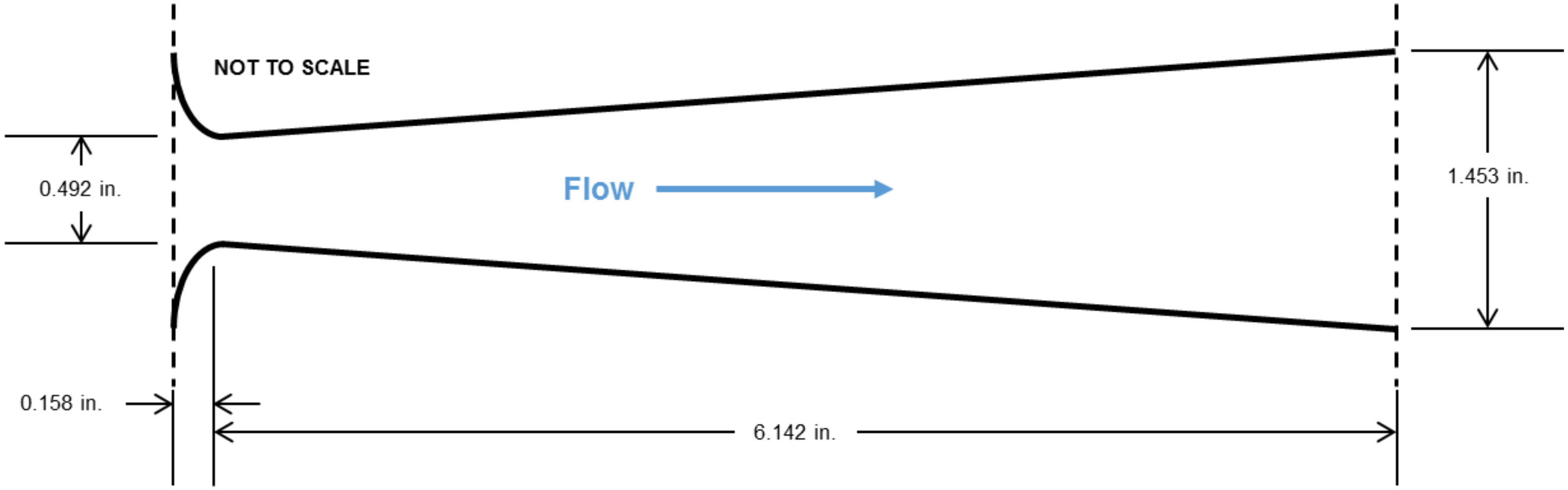
Summary

- Compressible Flow
  - **GFSSP** can model Ideal and Real gases
  - Inertia term in the momentum conservation equation needs to be activated
    - Accounts for fluid acceleration due to large density and area change

- **GFSSP** Predictions
  - Validated by comparing with analytical solutions for three classical compressible flow problems
    - Converging-Diverging Nozzle
    - Subsonic Flow with Friction (Fanno Flow)
    - Subsonic Flow with Heat Transfer (Rayleigh Flow)

Tutorial – 1

Simulation of Compressible Flow in a Converging-Diverging Nozzle

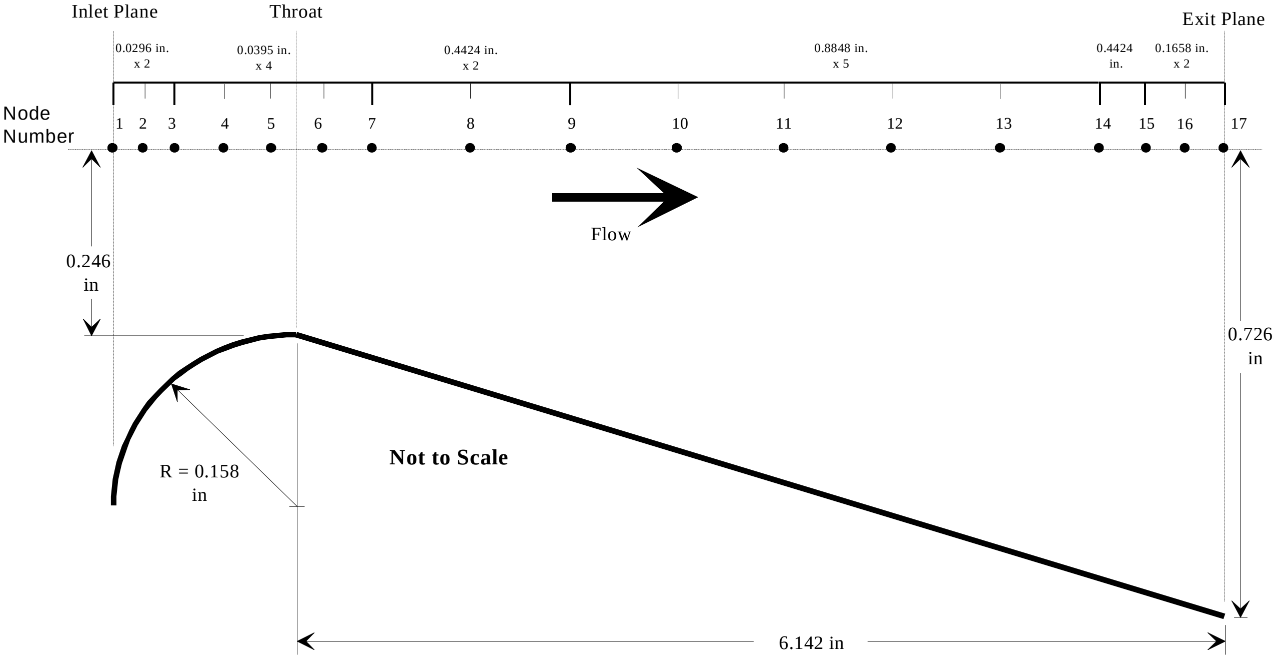


Converging-Diverging Nozzle Geometry

Problem Considered:

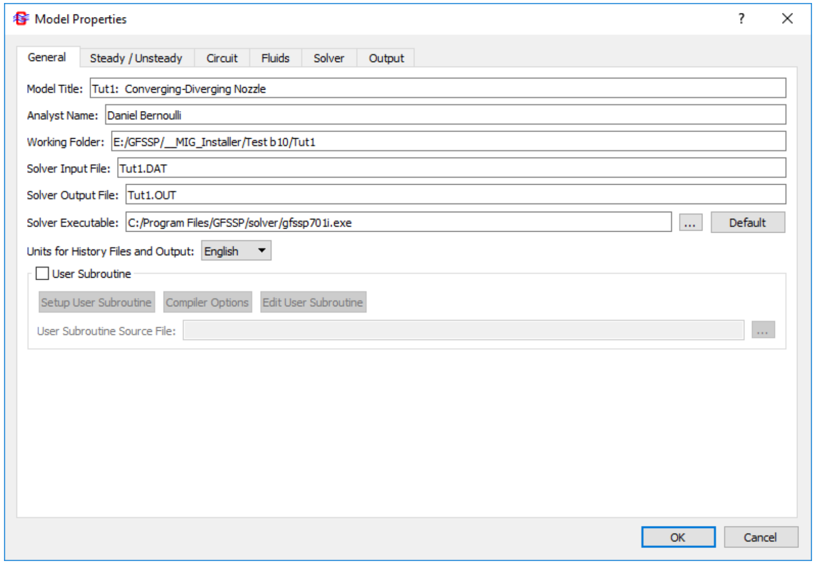
- One-dimensional pressure and temperature distribution
- Flow rates in subsonic and choked flow

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



Model Properties (1/2)

- Model file: Tut1.gfssp
- Input file: Tut1.DAT
- Output file: Tut1.OUT



Model Title: Tut1: Converging-Diverging Nozzle

Analyst Name: Daniel Bernoulli

Working Folder: E:/GFSSP/__MIG_Installer/Test b10/Tut1

Solver Input File: Tut1.DAT

Solver Output File: Tut1.OUT

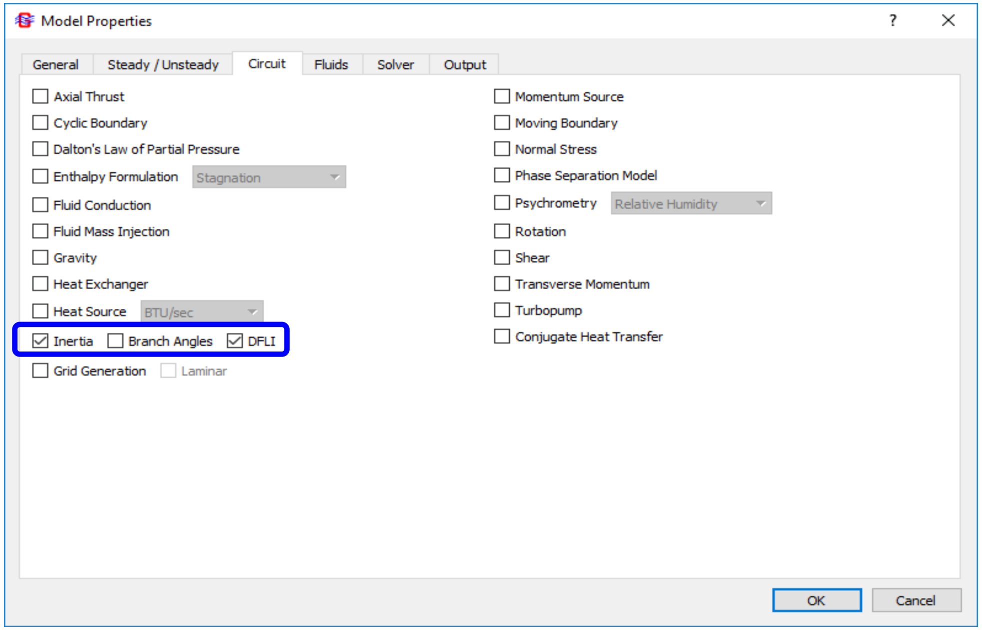
Solver Executable: C:/Program Files/GFSSP/solver/gfssp701i.exe

Units for History Files and Output: English

☐ User Subroutine

User Subroutine Source File:

- Activate INERTIA option *globally*
  - Means that the inertia term becomes <u>selectable</u> in the branches
  - Check **DFLI** box



☐ Axial Thrust

☐ Cyclic Boundary

☐ Dalton's Law of Partial Pressure

☐ Enthalpy Formulation Stagnation

☐ Fluid Conduction

☐ Fluid Mass Injection

☐ Gravity

☐ Heat Exchanger

☐ Heat Source BTU/sec

☑ Inertia ☐ Branch Angles ☑ DFLI

☐ Grid Generation ☐ Laminar

☐ Momentum Source

☐ Moving Boundary

☐ Normal Stress

☐ Phase Separation Model

☐ Psychrometry Relative Humidity

☐ Rotation

☐ Shear

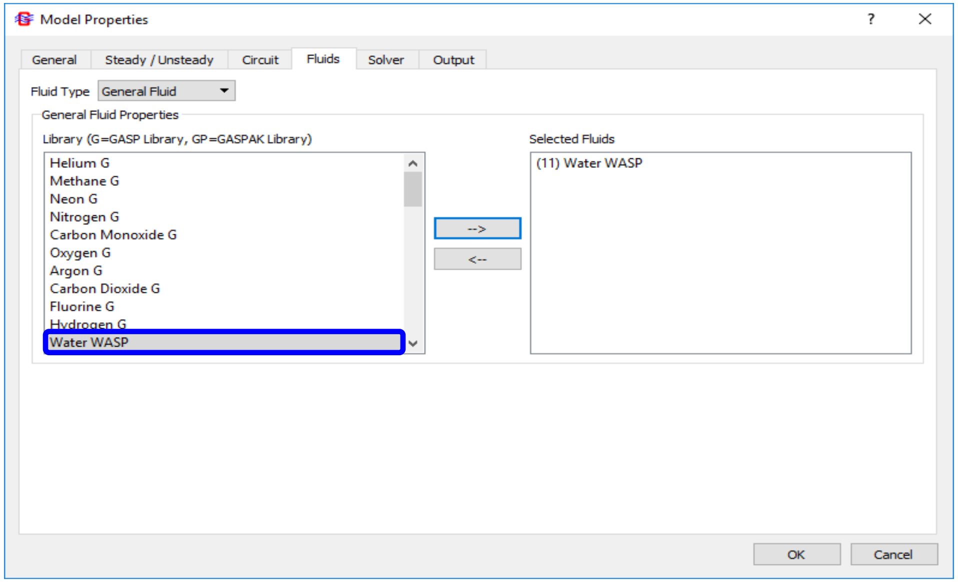
☐ Transverse Momentum

☐ Turbopump

☐ Conjugate Heat Transfer

Model Properties (2/2)

Fluid is steam
(water)



Fluid Type: General Fluid

General Fluid Properties

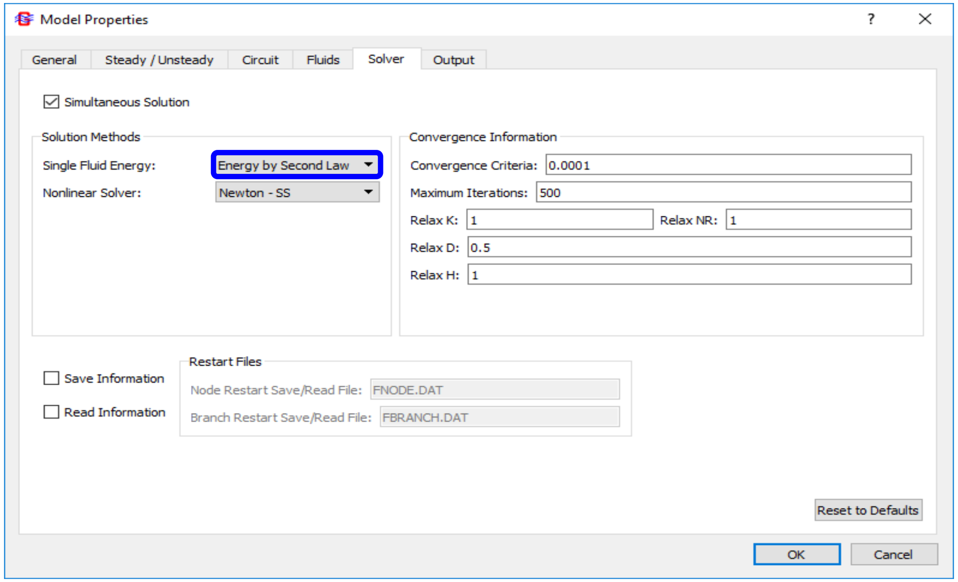
Library (G=GASP Library, GP=GASPAK Library)

Helium G
Methane G
Neon G
Nitrogen G
Carbon Monoxide G
Oxygen G
Argon G
Carbon Dioxide G
Fluorine G
Hydrogen G
Water WASP

Selected Fluids

(11) Water WASP

Second Law Formulation
of
Energy Equation



☑ Simultaneous Solution

Solution Methods

Single Fluid Energy: Energy by Second Law
Nonlinear Solver: Newton - SS

Convergence Information

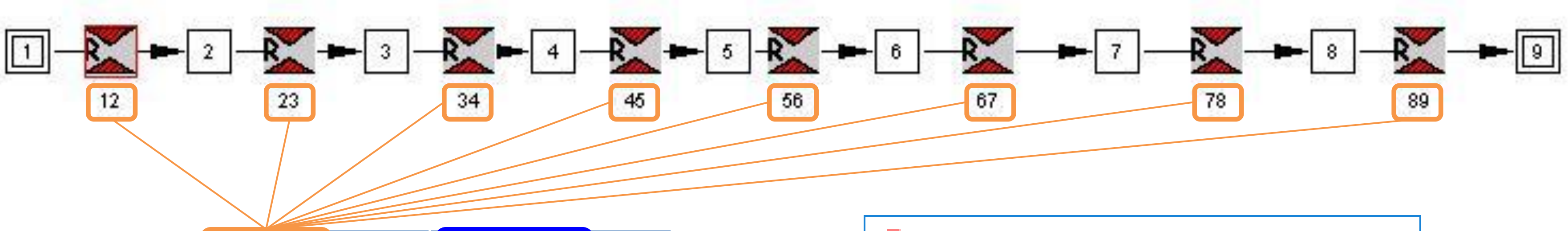
Convergence Criteria: 0.0001
Maximum Iterations: 500
Relax K: 1
Relax NR: 1
Relax D: 0.5
Relax H: 1

☐ Save Information
☐ Read Information

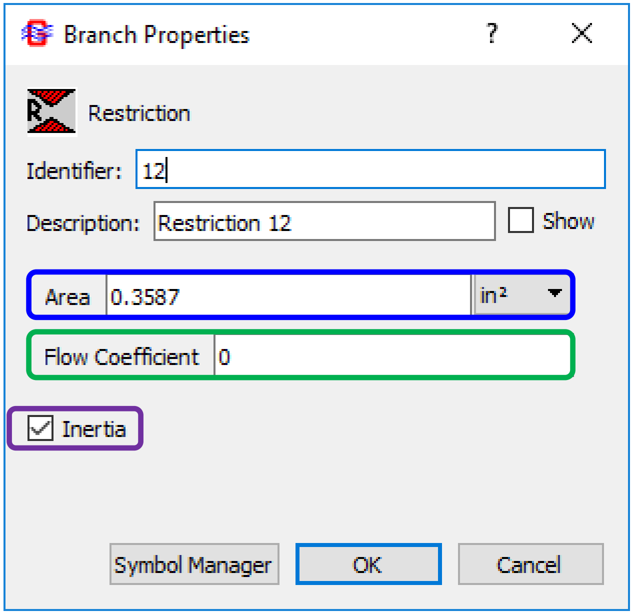
Restart Files

Node Restart Save/Read File: FNODE.DAT
Branch Restart Save/Read File: FBRANCH.DAT

Branch Geometry



Branch	Area (in^2)
12	0.3587
23	0.2243
34	0.1901
45	0.2255
56	0.3948
67	0.7633
78	1.2520
89	1.6286



Identifier: 12

Description: Restriction 12 ☐ Show

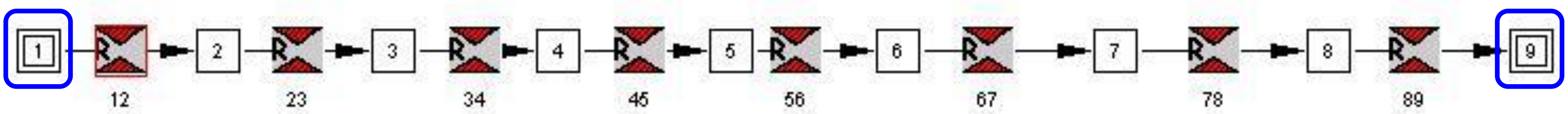
Area 0.3587 in^2

Flow Coefficient 0

☑ Inertia

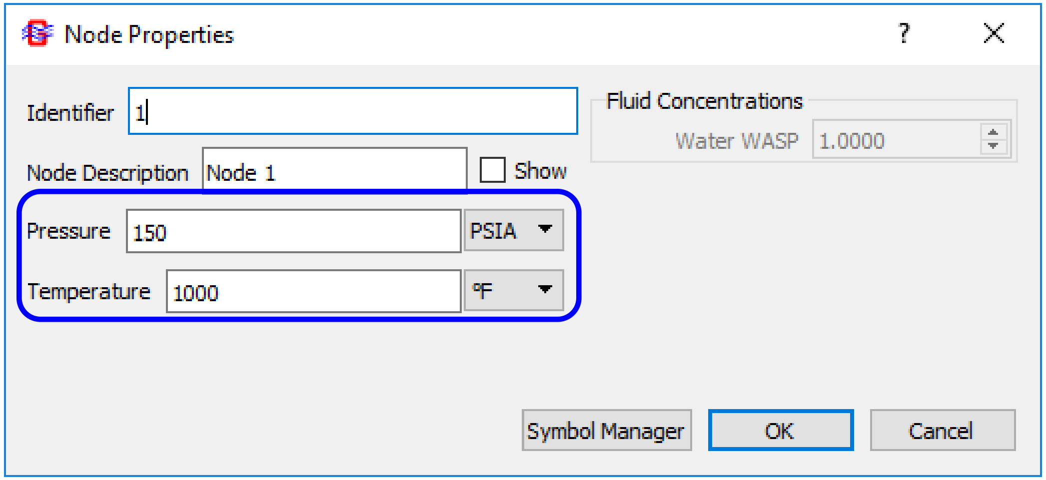
- Set restriction **Flow Coefficient** to 0.0 (isentropic – no friction)
- Activate **Inertia** term **in each branch**

Boundary Conditions



- **Node 1**
  - P = 150 psia
  - T = 1000 °F

- **Node 9**
  - P = 134 psia
  - T = 1000 °F*


Node Properties

Identifier: 1

Node Description: Node 1 ☐ Show

Pressure: 150 PSIA

Temperature: 1000 °F

Fluid Concentrations

Water WASP 1.0000

*Note: We don’t know exit temperature *a priori*, but because GFSSP uses an upwind scheme for the energy equation, we only need a reasonable guess.

Parametric Computational Results Comparison

- Run five cases, gradually decreasing the exit pressure (node 9)

Run	P_9 (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?

$$\dot{m} = \mathrm{A_{throat}P_{inlet}}\sqrt{\frac{g_c\gamma}{\mathrm{RT_{inlet}}}\left(\frac{2}{\gamma-1}\right)^{\left(\frac{\gamma+1}{\gamma-1}\right)}} = (0.19012\ in^2)(161.6\ \frac{lb_f}{in^2})\sqrt{\frac{32.174\frac{lb_m-ft}{lb_f-s^2}(1.281)}{85.83\frac{lb_f-ft}{lb_m-°R}(1460°R)}\left(\frac{2}{1.281+1}\right)^{\left(\frac{2.281}{0.281}\right)}} = \mathbf{0.327\ \frac{lb_m}{s}}$$

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

Study of the Results

Study ***tut1.out*** and note the following:

- **Pressure**
  - Decreases from inlet to throat
  - Increases from throat to exit in subsonic flow (Exit Pressure = 134 psia)
  - With lower Exit Pressure
    - Flow becomes supersonic in the diverging part of nozzle
    - Flow becomes subsonic with the formation of shock wave

- **Temperature**
  - Follows a similar trend
  - Changes due to expansion and compression

- **Entropy**
  - Remains constant due to isentropic assumption

- **Flowrate**
  - Remains constant with exit pressure once choked flow rate is reached

If Time Permits…

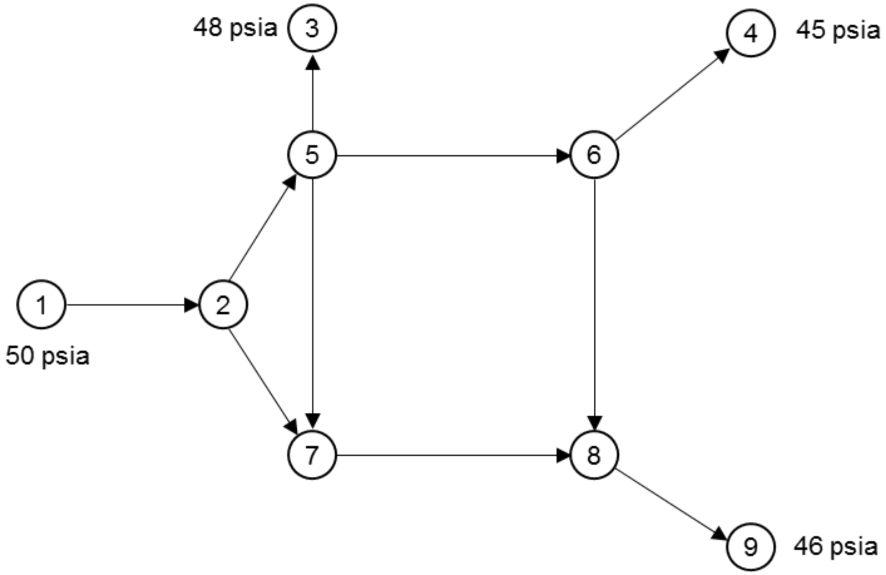
- Try re-running case 5 with “Energy by First Law” on the Solver tab
  - Flow rate is slightly different.
  - Note that enthalpy (H) is constant, and temperatures remain nearly constant (994 – 1000 °F). This is because **GFSSP** assumes stagnation enthalpy by default.

- Now change “Enthalpy Formulation” to “Static” on the Circuit tab.
  - Flow rate is slightly different.
  - Now temperatures are changing, because the energy equation includes a velocity term.
  - In **GFSSP**, temperatures are associated with nodes, but velocities are associated with branches, introducing some inaccuracy into the calculation of static enthalpies.
  - For an isentropic high-speed flow model such as this, the Second Law option is convenient, as it avoids the difficulties of static vs. stagnation enthalpy.

Challenge Problem 1 (1/2)

Simulation of a Water Distribution Network

Given: Water at room temperature enters the flow network shown below at 50 psia and exits at the given boundary pressures. Each branch of the network is a commercial steel pipe with the dimensions given in the table. The *relative* roughness (e/D) of the pipes is 0.0018.



Branch	**Length (inches)**	**Diameter (inches)**
12	120	6
25	2400	6
27	2400	5
57	1440	4
53	120	5
56	2400	4
64	120	4
68	1440	4
78	2400	4
89	120	5

Determine: the mass flow rate of each of the branches

Challenge Problem 1 (2/2)

Simulation of a Water Distribution Network

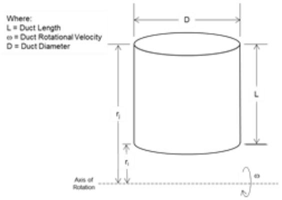
- How do your results compare to those determined by calculations using the Hardy Cross method of analyzing pipe networks?

- Hardy Cross method assumes a constant friction factor for the network

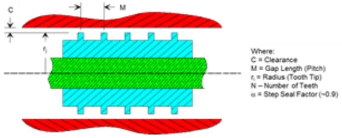
Branch	Flow Rate (lb_m/s)	
	Hardy-Cross	**GFSSP**
12	100.16	
25	63.59	
27	36.58	
53	44.43	
56	29.11	
57	-9.93	
64	47.07	
68	-17.99	
78	26.64	
89	8.66	

- **GFSSP** calculates a friction factor for each branch

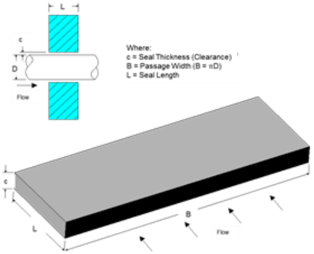
Resistance & Fluid Options



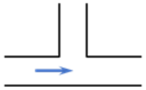
10. Rotating Radial Duct



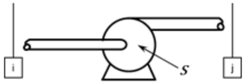
11. Labyrinth Seal



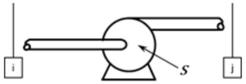
12. Face Seal



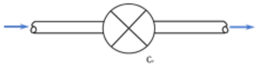
13. Common Fittings & Valves



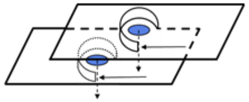
14. Pump Characteristics



15. Pump Power



16. Valve with Given C_v



17. Visco Jet

Friction Term in GFSSP's Momentum Equation

- In classical fluid mechanics, pressure drop can be related to dynamic pressure by a dimensionless constant K

$$\Delta \mathrm{P} = K\left(\frac{1}{2}\rho u^2\right)$$

- **GFSSP**'s momentum equation expresses friction losses in terms of flow rate

$$\Delta \mathrm{P} = \ K_f \dot{m}^2$$

- The relationship between K and K_f

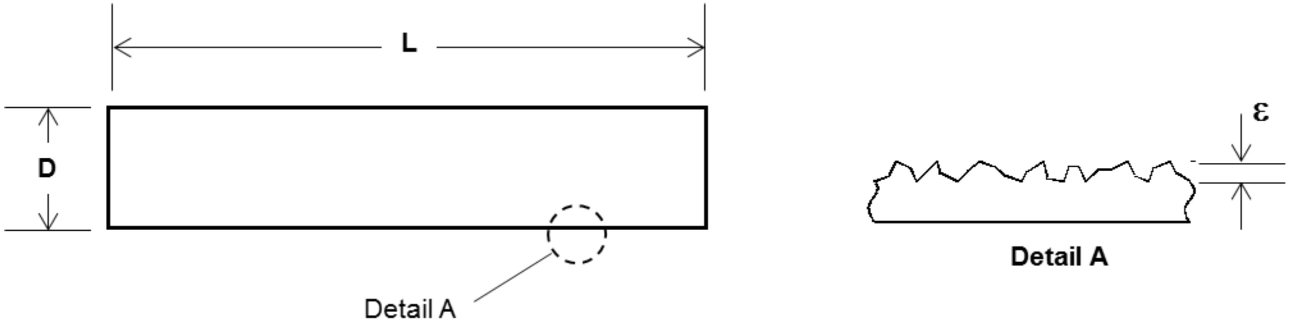
$$K_f \ = K\left(\frac{\frac{1}{2}\rho u^2}{(\rho A u)^2}\right) = K\left(\frac{1}{2\rho \mathrm{A}^2}\right)$$

- Note that K_f is not dimensionless
  - Units: $\left(lb_f/ft^2\right)/(lb_m/s)^2$

Resistance Option 1

- Pipe Flow

Pipe Resistance Option Parameters



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

For Re < 2300, Friction Factor (f)

$$f = \frac{64}{\mathrm{Re_D}}$$

For Re > 2300, Friction Factor (using Colebrook Equation)

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

Flow Resistance Factor

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

Resistance Option 1 (cont.)

- How was the equation for K_f derived?

$$\Delta\mathrm{P} = \frac{fL}{D}\left(\frac{1}{2}\rho u^2\right) = K_f \dot{m}^2$$

$$\dot{m} = \rho A u \qquad\qquad A = \frac{\pi}{4}D^2$$

$$\frac{fL}{D}\left(\frac{1}{2}\rho u^2\right) = K_f\left[\rho u\left(\frac{\pi}{4}D^2\right)\right]^2$$

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

Resistance Option 2 (1/2)

- Flow Through a Restriction (1/2)

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

- Loss Coefficient: C_L
  - Sometimes called “Flow Coefficient” or “Discharge Coefficient”
  - Smaller values of C_L indicate greater resistance
  - BUT, if User sets $C_L = 0$
    - GFSSP will set K_f to 0 (flag for inviscid flow through the branch)

Resistance Option 2 (2/2)

- Flow Through a Restriction (2/2)
- In classical fluid mechanics, head loss (ΔH) is expressed as:

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

- K and C_L are related by:

$$C_L = \frac{1}{\sqrt{K}}$$

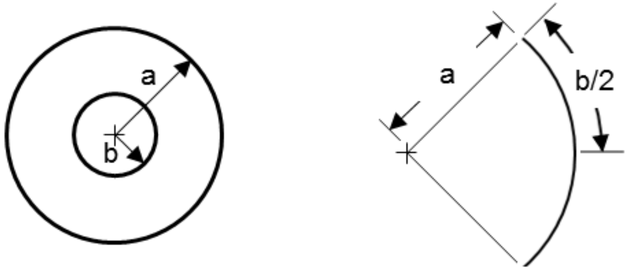
- Larger values of K indicate greater resistance
- In **GFSSP,** it is common to use the Restriction Option 2 as a generic branch
  - K-values from either the manufacturer, or from literature
  - K-values converted to Loss Coefficients, C_L

Resistance Option 3 (1/2)

- Non-Circular Duct (1/2)

Four cross-sections:

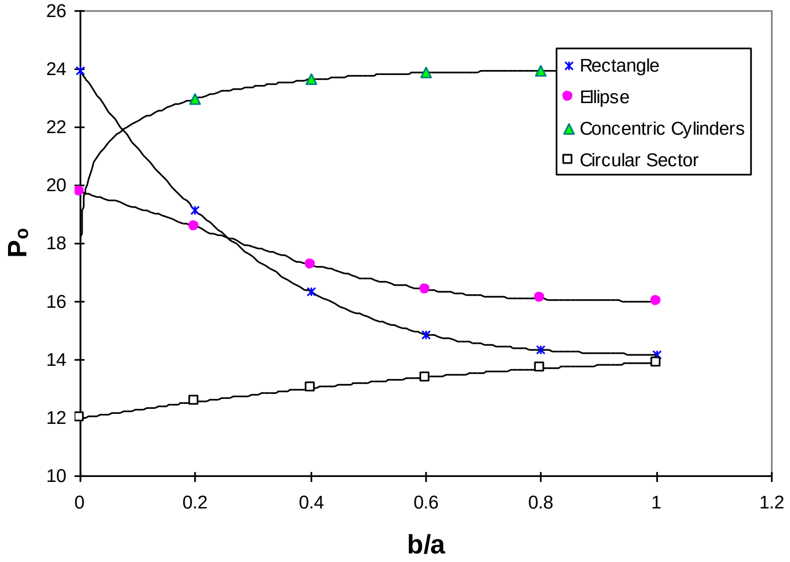
(a) Rectangle (b) Ellipse



(c) Concentric Annulus (d) Circular Sector

Poiseuille Number Relationship for Laminar Flow

$Po = C_f Re$



Resistance Option 3 (2/2)

- Non-Circular Duct (2/2)

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

1. Compute Hydraulic Diameter (D_h)

$$D_h = \frac{4A}{P}$$

2. Compute Effective Reynolds Number (Re_{D_h})

$$Re_{D_h} = \frac{\dot{m}}{\mu}\frac{D_h}{A}$$

3. Compute friction factor (f)

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

Turbulent Flow ($Re_{D_{eff}} > 2300$)

1. Compute Effective Diameter (D_{eff})

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

2. Compute Effective Reynolds number (Re_{eff})

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

3. Use D_{eff} & Re in Colebrook Equation

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

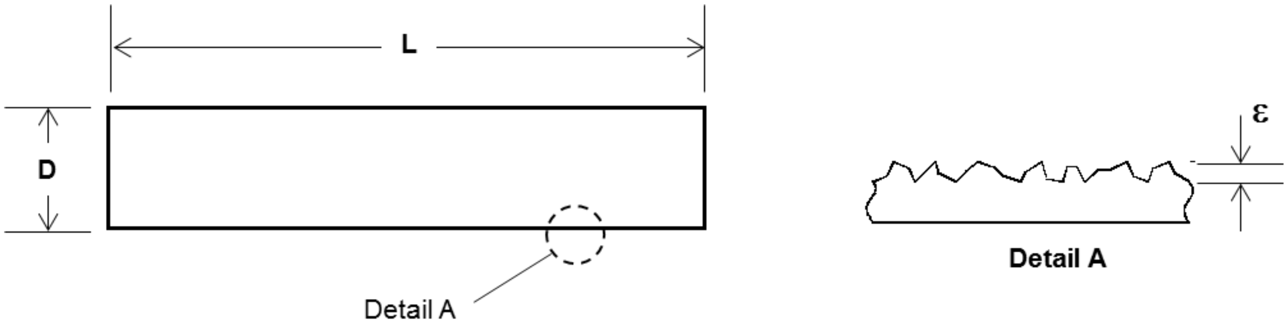
Flow Resistance Factor

$$K_f = \frac{fPL}{8g_c\rho_u A^3}$$

Resistance Option 4

- Pipe Flow with Entrance and Exit Loss

Pipe with Entrance and/or Exit Loss



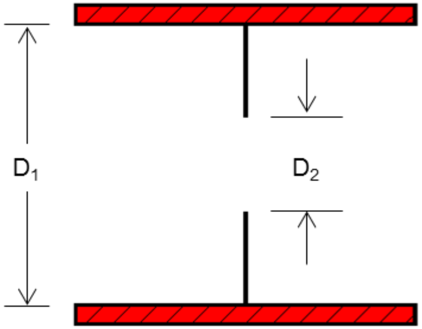
Where:
D = Pipe Diameter
L = Pipe Length
ε = Absolute Roughness
K_i = Entrance Loss Coefficient
K_e = Exit Loss Coefficient

Flow Resistance Factor

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

Resistance Option 5

- Thin Sharp Orifice



Where:
D$_1$ = Pipe Diameter
D$_2$ = Orifice Throat Diameter

Flow Resistance Factor

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

where:

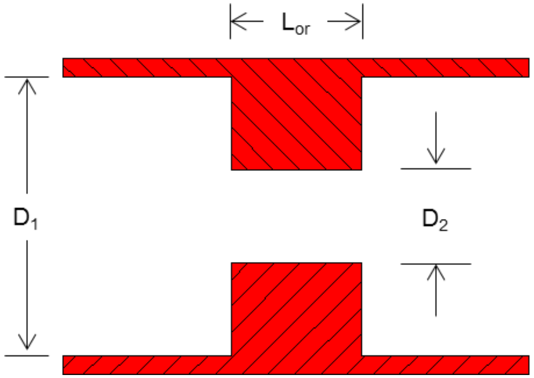
$$K_1 = \left[2.72 + \left(\frac{\mathrm{D}_2}{\mathrm{D}_1}\right)^2 \left(\frac{120}{\mathrm{Re}_{\mathrm{D}_\mathrm{h}}} - 1\right)\right]\left[1 - \left(\frac{\mathrm{D}_2}{\mathrm{D}_1}\right)^2\right]\left[\left(\frac{\mathrm{D}_1}{\mathrm{D}_2}\right)^4 - 1\right] \text{for } \mathrm{Re}_{\mathrm{D}_1} \leq 2500$$

$$K_1 = \left[2.72 + \left(\frac{\mathrm{D}_2}{\mathrm{D}_1}\right)^2 \left(\frac{4000}{\mathrm{Re}_{\mathrm{D}_\mathrm{h}}}\right)\right]\left[1 - \left(\frac{\mathrm{D}_2}{\mathrm{D}_1}\right)^2\right]\left[\left(\frac{\mathrm{D}_1}{\mathrm{D}_2}\right)^4 - 1\right] \text{for } \mathrm{Re}_{\mathrm{D}_1} > 2500$$

Note: This branch is only for incompressible flow.

Resistance Option 6

- Thick Orifice



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

Flow Resistance Factor

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

where:

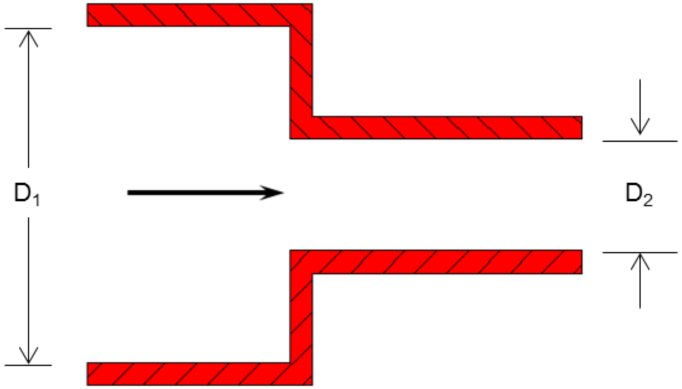
$$K_1 = \left[2.72 + \left(\frac{D_2}{D_1}\right)^2 \left(\frac{120}{Re_{D_h}} - 1\right)\right]\left[1 - \left(\frac{D_2}{D_1}\right)^2\right]\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right]\left[0.584 + \frac{0.0936}{(L_{or}/D_2)^{1.5} + 0.225}\right] \text{ for } Re_{D_1} \leq 2500$$

$$K_1 = \left[2.72 + \left(\frac{D_2}{D_1}\right)^2 \left(\frac{4000}{Re_{D_1}}\right)\right]\left[1 - \left(\frac{D_2}{D_1}\right)^2\right]\left[\left(\frac{D_1}{D_2}\right)^4 - 1\right]\left[0.584 + \frac{0.0936}{(L_{or}/D_2)^{1.5} + 0.225}\right] \text{ for } Re_{D_1} > 2500$$

Note: This branch is only for incompressible flow.

Resistance Option 7

- Square Reduction



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

Flow Resistance Factor

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

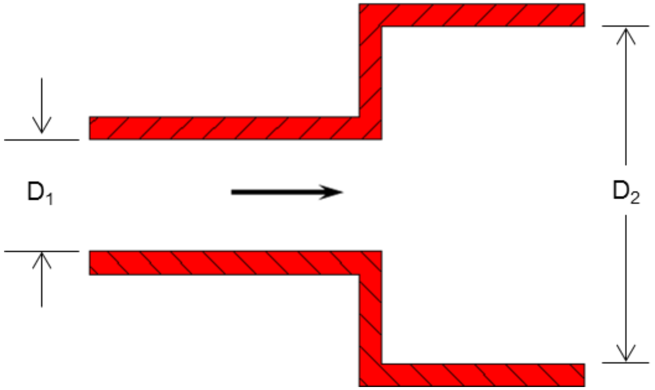
where:

$$K_1 = \left[1.2 + \frac{160}{\mathrm{Re}_{\mathrm{D}_1}}\right]\left[\left(\frac{\mathrm{D}_1}{\mathrm{D}_2}\right)^4 - 1\right] \text{ for } \mathrm{Re}_{\mathrm{D}_1} \leq 2500$$

$$K_1 = [0.6 + 0.48f]\left(\frac{\mathrm{D}_1}{\mathrm{D}_2}\right)^2\left[\left(\frac{\mathrm{D}_1}{\mathrm{D}_2}\right)^2 - 1\right]^2 \text{ for } \mathrm{Re}_{\mathrm{D}_1} > 2500$$

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:

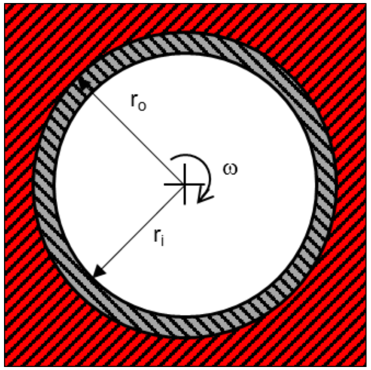
$$K_1 = 2\left[1 - \left(\frac{D_1}{D_2}\right)^4\right] \text{ for } \mathrm{Re}_{D_1} \leq 4000$$

$$K_1 = [1 + 0.8f]\left[1 - \left(\frac{D_1}{D_2}\right)^2\right]^2 \text{ for } \mathrm{Re}_{D_1} > 4000$$

***GFSSP** will automatically switch between Options 7 and 8 depending upon the flow direction*

Resistance Option 9

- 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

Flow Resistance Factor

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

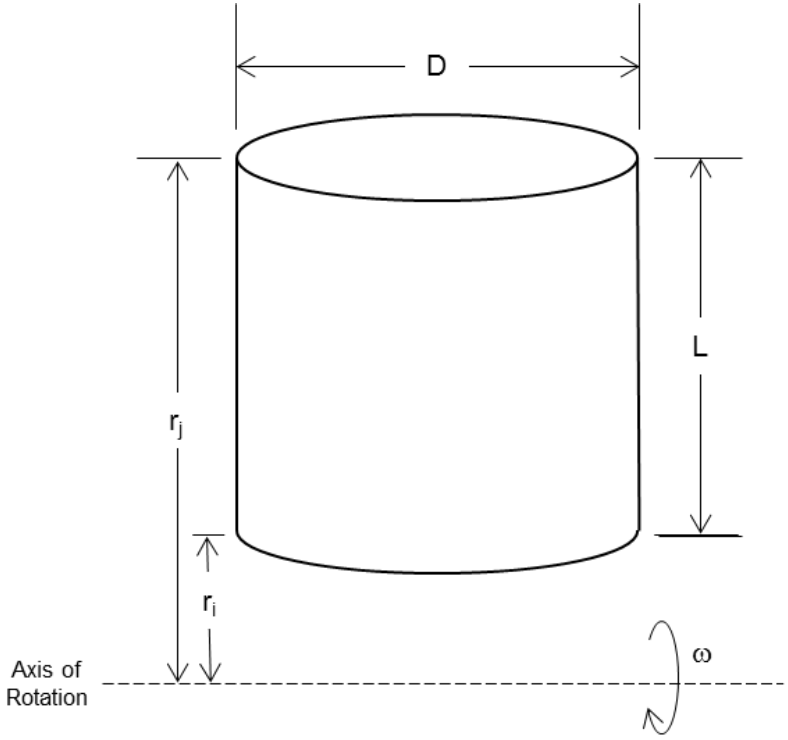
where:

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

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

Resistance Option 10

- Rotating Radial Duct



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

Flow Resistance Factor

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

where:

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

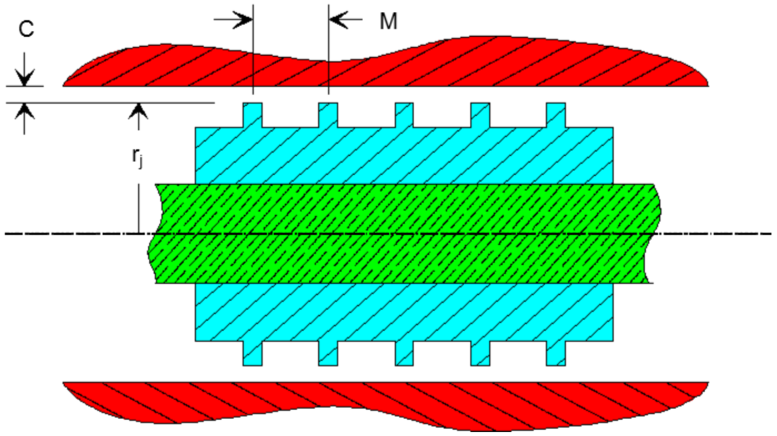
$$f_{0T} = 0.0791(Ru)^{-0.25}$$

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

Note: This branch only models the friction losses in the rotating duct. User must activate centrifugal term in momentum equation separately.

Resistance Option 11

- Labyrinth Seal



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

Flow Resistance Factor
(Modified Dodge Eqn)

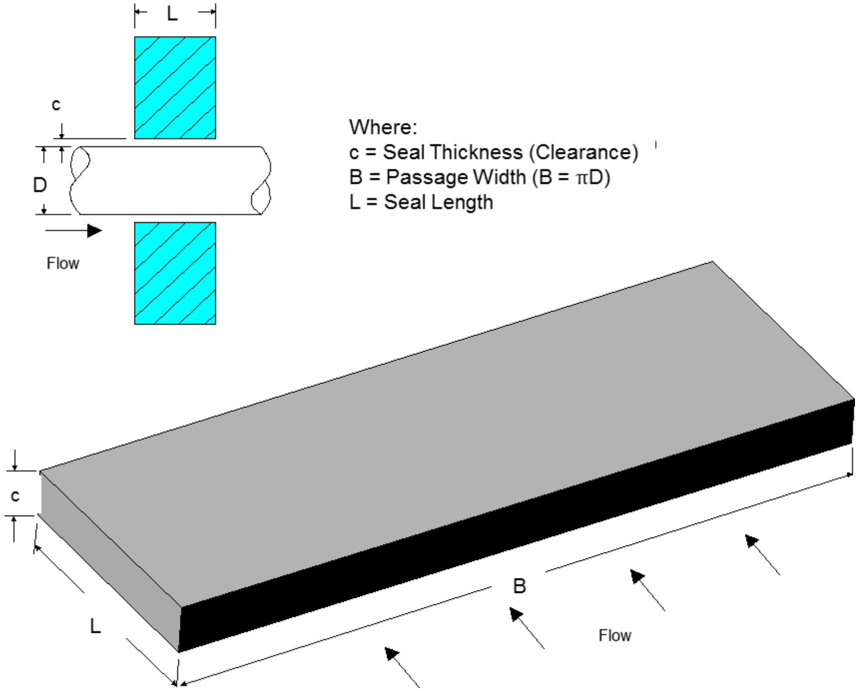
$$K_f = \frac{\left(\frac{1}{\varepsilon^2} + 0.5\right)\mathrm{N} + 1.5}{2 g_c \rho_u \alpha^2 \mathrm{A}^2}$$

where:

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

Resistance Option 12

- Face Seal



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

Flow Resistance Factor

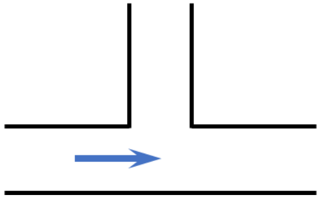
$$K_f = \frac{12\mu L\rho}{\pi g_c D c^3 |\dot{m}|}$$

Resistance Option 13

- Common Fittings and Valves

Flow Resistance Factor

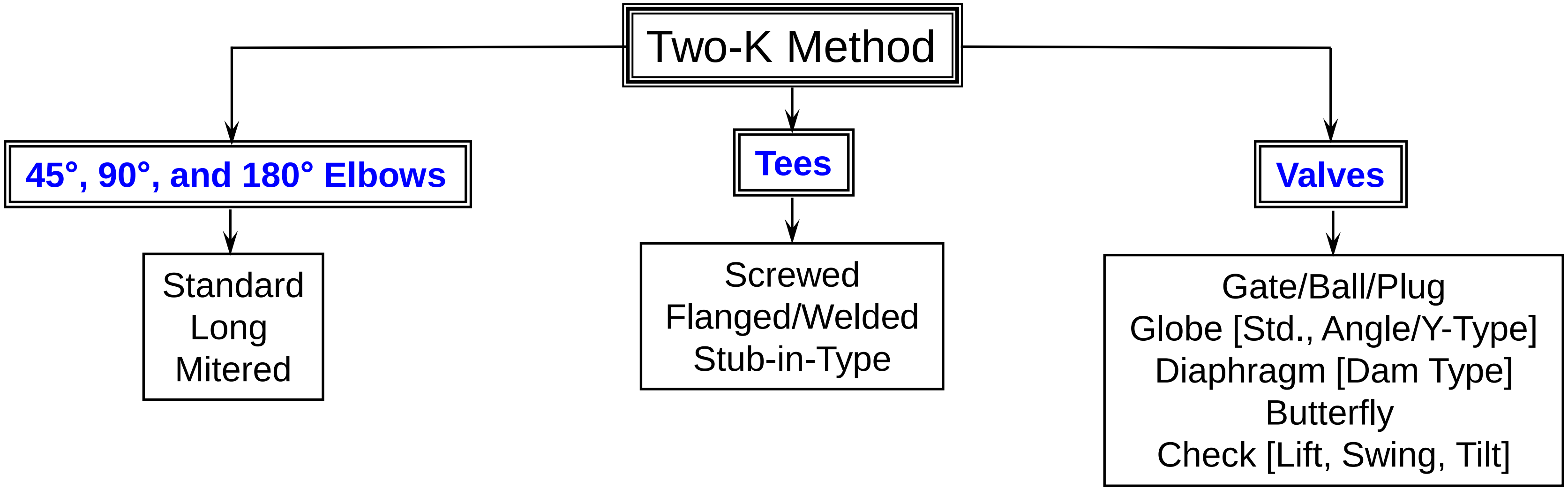
$$K_f = \frac{\frac{K_1}{\mathrm{Re}} + K_\infty\left(1 + \frac{1}{\mathrm{D}}\right)}{2 g_c \rho_u \mathrm{A}^2}$$



where:

K_1 = K for the fitting at $\mathrm{Re} = 1$
K_∞ = K for the fitting at $\mathrm{Re} = \infty$ (K_2 in **GFSSP**)
D = Internal diameter of attached pipe (in)

- Types of Fittings and Valves



Two-K Method

45°, 90°, and 180° Elbows

Standard
Long
Mitered

Tees

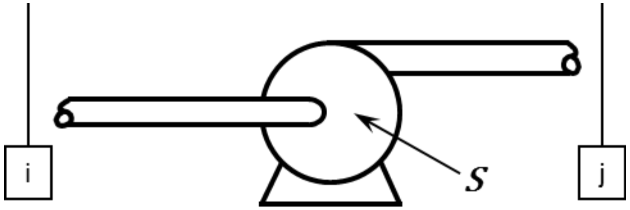
Screwed
Flanged/Welded
Stub-in-Type

Valves

Gate/Ball/Plug
Globe [Std., Angle/Y-Type]
Diaphragm [Dam Type]
Butterfly
Check [Lift, Swing, Tilt]

Resistance Option 14

- Pump Characteristics
  - Option 14 considers the branch as a pump with given characteristics

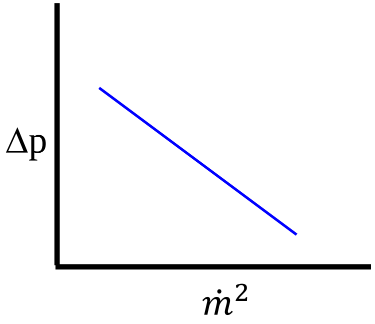


  - Pump Characteristics are expressed in the pressure rise

$$\Delta \mathrm{p} = A_0 + B_0\dot{m} + C_0\dot{m}^2$$

where:

$\Delta \mathrm{p} =$ Pressure Rise ($\mathrm{lb_f/ft^2}$)
$\dot{m} =$ Flow Rate ($\mathrm{lb_m/sec}$)

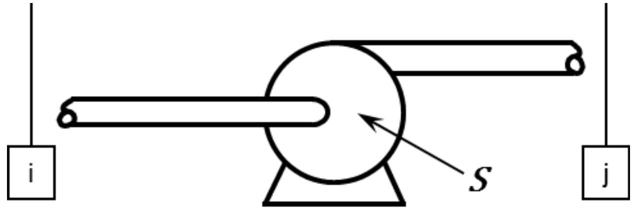


  - Momentum Source (S) used to induce the desired flow

$$S = \Delta \mathrm{p}\mathrm{A}$$

Resistance Option 15

- Pump Power
  - Considers the branch as a pump with a given horsepower (P) and efficiency (η)



  - Momentum Source (S) used to induce the desired flow

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

Pumps and the Energy Equation

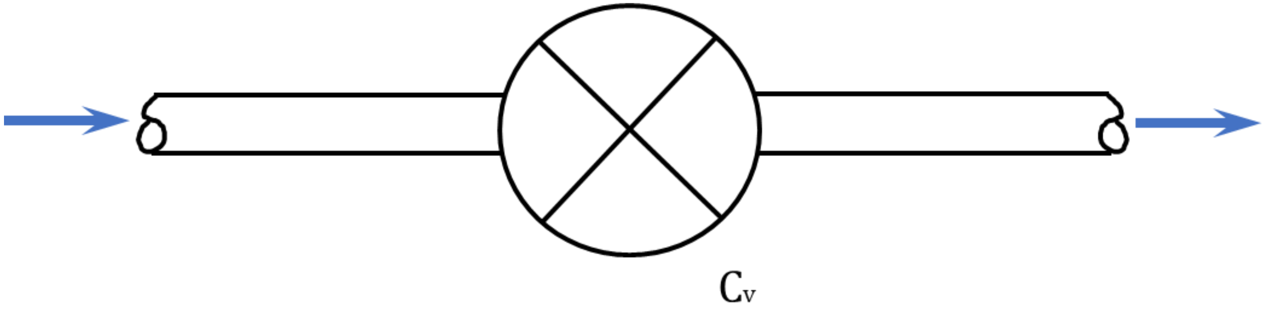
- NOTE: Pump options automatically add an energy source term to the 1st Law Energy Formulation.

$$S = \Delta P A u = \Delta P A \frac{\dot{m}}{\rho A} = \frac{\Delta P \dot{m}}{\rho}$$

- CAUTION: For a compressible fluid, density/velocity are not constant. This equation will be based on the inlet density, a lower value than the exit density, thus tending to overvalue the energy source.
- Therefore, if the pump option is used to model a compressor, exit temperatures may be overestimated.
- Options when modeling a compressor:
  - If possible, switch to Energy by 2nd Law (Entropy)
  - If Energy by 1st Law is required, break pump branch up into separate stages.

Resistance Option 16

- Valve with Given C_v
  - Branch considered as a valve with a given C_v



Flow Resistance Factor

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

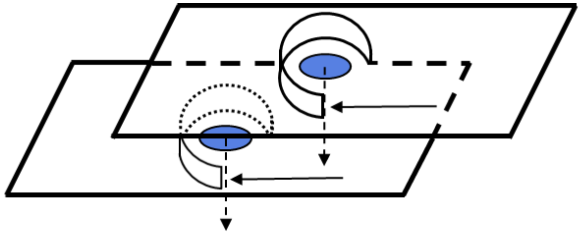
Resistance Option 17

- Visco Jet (Joule - Thomson Device)
  - Option 17 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.
  - Visco Jet flow rate is given by:

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

where:

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

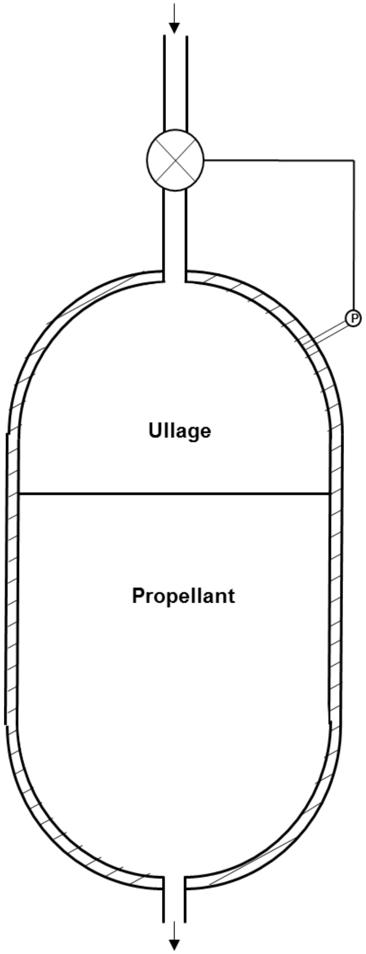


  - For Option 17, K_f is expressed as:

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

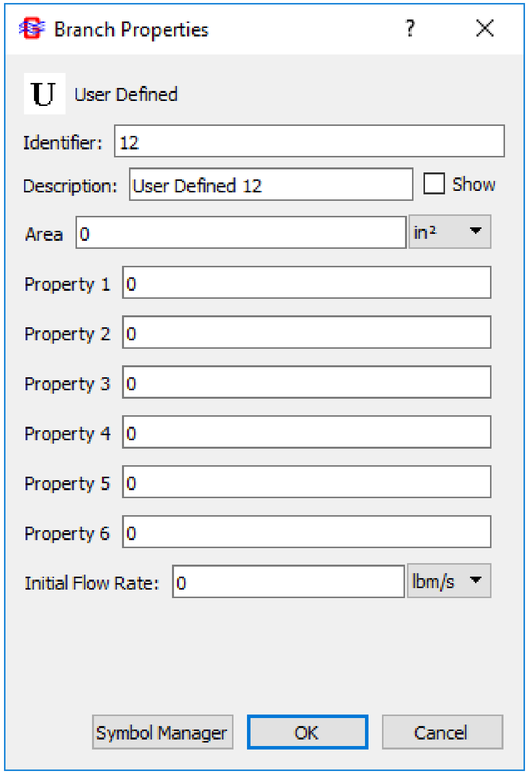
Resistance Option 18

- 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 (Restriction)



Resistance Option 18

- User Defined **U**
  - Allows User to create a new resistance not available in **GFSSP** library
  - User is required to supply Fortran coding for calculating K_f
  - User is required to supply the branch cross-sectional area via the preprocessor
  - User has the option of supplying up to six branch parameters via the preprocessor



User Defined

Identifier: 12

Description: User Defined 12 ☐ Show

Area 0 in²

Property 1 0

Property 2 0

Property 3 0

Property 4 0

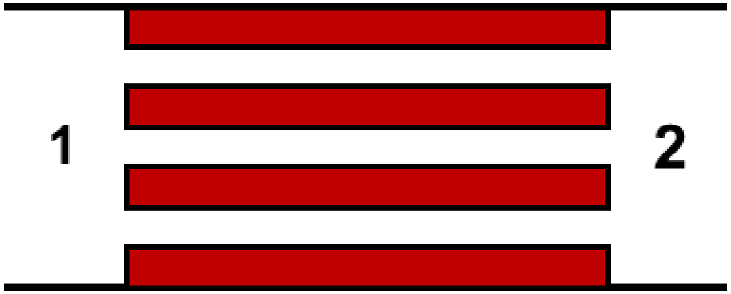
Property 5 0

Property 6 0

Initial Flow Rate: 0 lbm/s

Resistance Option 20

- Heat Exchanger Core



Flow Resistance Factor

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

where:

$\mathrm{A_s}$ = Wetted Surface Area
$\mathrm{A_c}$ = Minimum Free Flow Area
σ = Ratio of Free Flow Area to Frontal Area
K_c = Contraction Loss Coefficient
K_e =Expansion Loss Coefficient

Note: This branch only models the friction loss in the heat exchanger.
Heat transfer can be modeled separately with a heat source, or the heat exchanger advanced option.

Resistance Option 21

- Parallel Tube
  - Option 21 is an extended version of Option 1
    - n is the number of parallel tubes
  - Assumes uniform flow distribution

Flow Resistance Factor

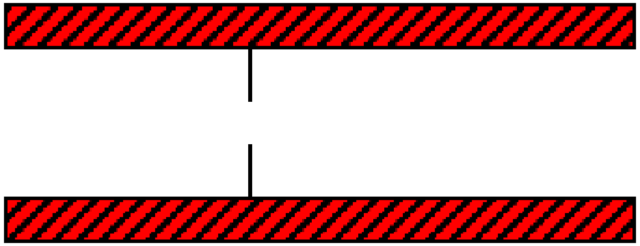
$$K_f = \frac{8 f \mathrm{L}}{\rho_u \pi^2 \mathrm{D}^5 g_c n^2}$$

Resistance Option 22 (1/2)

- Compressible Orifice (1/2)
  - Option 22 considers branch as an orifice for compressible flow
  - Flowrate is calculated from a simplified momentum equation
  - Input is identical to Option 2 (Restriction)
  - Flow will choke at the critical pressure ratio (P_{down}/P_{up})

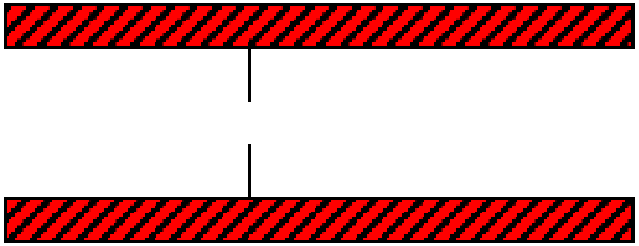
$$\mathrm{P_{cr}} = \left(\frac{2}{\gamma + 1}\right)^{\frac{\gamma}{\gamma-1}}$$

Gas	γ	P_{cr}
O2, N2, H2	1.4	0.53
He	1.66	0.49
CO2, CH4	1.3	0.55



Resistance Option 22 (2/2)

- Compressible Orifice (2/2)



  - If $\frac{P_{down}}{P_{up}} \leq P_{cr}$ (choked flow)

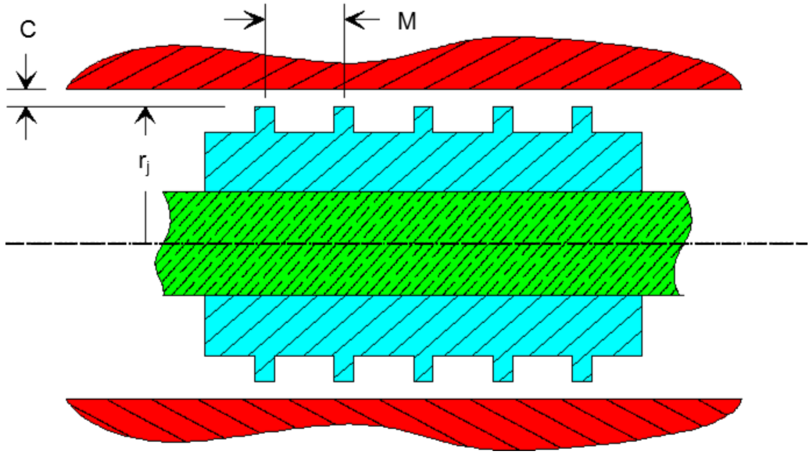
$$\dot{m} = C_L A \sqrt{P_{up}\rho_{up}g_c \frac{2\gamma}{\gamma-1} (P_{cr})^{\frac{2}{\gamma}} \left[1 - (P_{cr})^{\frac{\gamma-1}{\gamma}}\right]}$$

  - If $\frac{P_{down}}{P_{up}} > P_{cr}$

$$\dot{m} = C_L A \sqrt{P_{up}\rho_{up}g_c \frac{2\gamma}{\gamma-1} \left(\frac{P_{down}}{P_{up}}\right)^{\frac{2}{\gamma}} \left[1 - \left(\frac{P_{down}}{P_{up}}\right)^{\frac{\gamma-1}{\gamma}}\right]}$$

Resistance Option 23

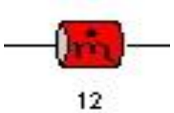
- Labyrinth Seal (EGLI Correlation)



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

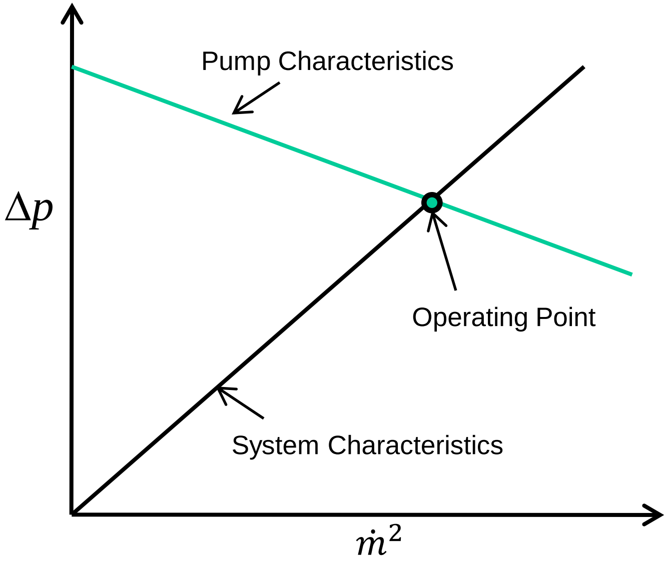
Resistance Option 24

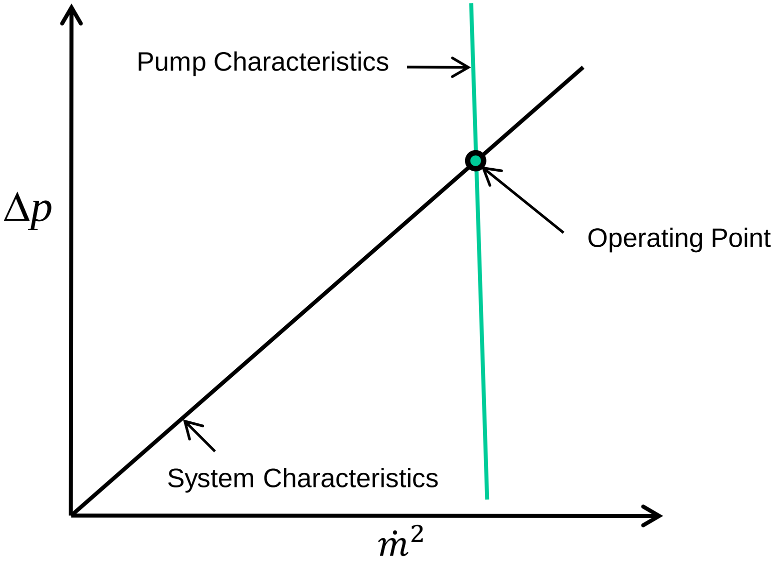
- Fixed Flowrate



- Fixed Flowrate branch uses a nearly vertical pump curve
  - Forces **GFSSP** to solve for a desired flow rate
- Fixed Flowrate branch can only be located adjacent to a Boundary Node
  - Replaces pressure boundary condition with a required flow
- User should always check that calculated flowrate is as expected
  - Tighter convergence criteria may be required
- Although the Fixed Flowrate branch works on the principal of a pump, it does NOT add an extra term to the 1st Law Energy Equation.

Algorithm for Fixed Flow Option (Schallhorn)





$$\Delta p = A + C\dot{m}|\dot{m}|$$

where:

$$A = \alpha\dot{m}|\dot{m}|$$

$$C = -\alpha$$

where:

$$\alpha = 1x10^{25}$$

Substituting A and C:

$$\dot{m} = \frac{\dot{m}|\dot{m}|}{|\dot{m}|}$$

Resistance Options Summary

- Most fluid systems can be modeled using available options
- Resistance Option 2 can be used as a generic option
  - C_L must be computed from a known pressure drop vs. flowrate characteristics
- User can add new resistance options through User Subroutines

Fluid Options

- **GFSSP** uses the following thermodynamic and thermo-physical properties of fluids for the solution of the governing equations
    - Density [ρ (T, p)]
    - Absolute Viscosity [μ (T, p)]
    - Thermal Conductivity [k (T, p)]
    - Specific Heat at Constant Pressure [C_p (T, p)]
    - Specific Heat Ratio [γ (T, p)]
    - Enthalpy [H (T, p)]
    - Entropy [S (T, p)]

- **GFSSP** requires these properties at every node, at each iteration

- Properties are supplied by thermodynamic property programs integrated into **GFSSP**

Integrated Fluid Property Programs

- **GASP/WASP**
  - Developed at NASA Glenn Research Center in 1970s
  - Uses modified Benedict, Webb, & Rubin (BWR) Equation of State
  - Fast and forgiving of out-of-range input
- **GASPAK**
  - Developed by Cryodata Inc. as an evolution of MIPROPS/NIST-12
  - Uses variable term Helmholtz equation
  - Based on:
    - National Institute of Standards and Technology (NIST)
    - International Union of Pure & Applied Chemistry (IUPAC)
    - National Standard Reference Data Service of the USSR
  - Fairly fast, but unforgiving of out-of-range input

Available Fluid Library

ID Number	SOURCE / FLUID	ID Number	SOURCE / FLUID	ID Number	SOURCE / FLUID
1	GASP He	51	GASPAK He	69	GASPAK Kr
2	GASP CH_4	52	GASPAK CH_4	70	GASPAK Propane
3	GASP Ne	53	GASPAK Ne	71	GASPAK Xe
4	GASP N_2	54	GASPAK N_2	72	GASPAK R-11
5	GASP CO	55	GASPAK CO	73	GASPAK R-12
6	GASP O_2	56	GASPAK O_2	74	GASPAK R-22
7	GASP Ar	57	GASPAK Ar	75	GASPAK R-32
8	GASP CO_2	58	GASPAK CO_2	76	GASPAK R-123
9	GASP F_2	59	GASPAK H_2 (para)	77	GASPAK R-124
10	GASP H_2 (para)	60	GASPAK H_2 (normal)	78	GASPAK R-125
11	WASP H_2O	61	GASPAK H_2O	79	GASPAK R-134A
12	RP-1 Tables	62	GASPAK RP-1 (liq)	80	GASPAK R-152A
		63	GASPAK Isobutane	81	GASPAK N_2F_3
33	Ideal Gas	64	GASPAK Butane	82	GASPAK NH_3
		65	GASPAK Deuterium	84	GASPAK H_2O_2
37	User Fluid 1	66	GASPAK Ethane	86	GASPAK Air
38	User Fluid 2	67	GASPAK Ethylene		
39	User Fluid 3	68	GASPAK H_2S		

Provision of Using Fluids Not Available in Fluid Library (1/2)

- User can add fluids in the library by providing property tables

- Tables can be used with 1st Law (enthalpy) energy formulation only

- **GFSSP** requires the following property tables
  - Thermal Conductivity (k)
  - Density (ρ)
  - Dynamic Viscosity (μ)
  - Specific Heat at constant pressure (C_p)
  - Specific Heat Ratio (γ)
  - Specific Enthalpy (h)
  - Specific Entropy (s)

Provision of Using Fluids Not Available in Fluid Library (2/2)

- If fluid properties knowledge is limited, some tables can be filled with dummy values
  - Entropy (s) is print-out value only; dummy values can be used

- If model does not use Conjugate Heat Transfer option
  - Thermal conductivity (k) is not required; dummy values can be used

- If model uses Mixture Temperature option
  - Specific heat (C_p) and specific heat ratio (γ) are required
  - Enthalpy (h) tables can be dummy values

- Enthalpy (h) tables can be constructed by integrating C_p over temperature

User-defined Fluid Table Inputs

Number of Pressure points (NP)

Number of Temperature points (NT)

NOTE: NP and NT must be the same in all seven user fluid files.

```
          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
```

Units of User-defined Fluid Tables

- To use SI units in fluid tables, SI units must also be enabled in **MIG**

- **GFSSP** installation directory contains a folder with utility programs for converting table units and converting a REFPROP output file to **GFSSP** format

Property Name	**English Units**	**SI Units**
Pressure (P)	psia	kPa
Temperature (T)	°R	K
Thermal Conductivity (k)	BTU/ft-s-°R	W/m-K
Density (ρ)	lb_m/ft^3	kg/m^3
Absolute Viscosity (μ)	lb_m/ft-s	$N\text{-}s/m^2$
Specific Heat Ratio (γ)	Dimensionless	Dimensionless
Specific Enthalpy (h)	BTU/lb_m	kJ/kg
Specific Entropy (s)	BTU/lb_m-°R	kJ/kg-K
Specific Heat (C_p)	BTU/lb_m-°R	kJ/kg-K

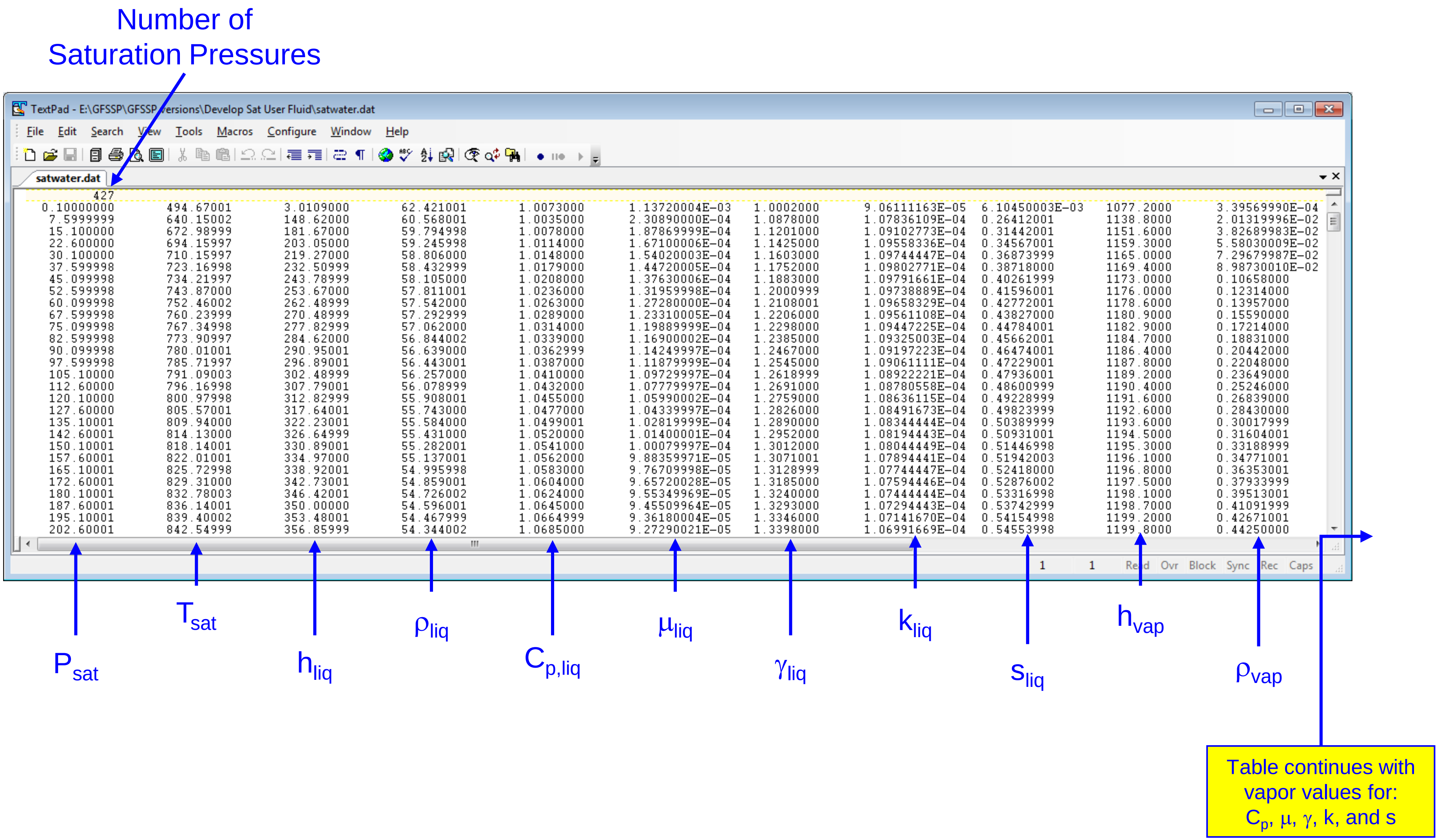
Saturated Properties of User-defined Fluids

- Seven User-defined fluid property tables
  - Not sufficient for modeling phase change

- User has the option of adding an eighth file
  - Saturated liquid and vapor properties as a function of saturation pressure

- **GFSSP** installation directory contains a folder with utility programs for converting REFPROP saturation properties to **GFSSP** format

Format of the Saturated Property Table



Number of Saturation Pressures

```
        427
 0.10000000       494.67001      3.0109000      62.421001      1.0073000      1.13720004E-03  1.0002000      9.06111163E-05  6.10450003E-03  1077.2000      3.39569990E-04
 7.5999999        640.15002      148.62000      60.568001      1.0035000      2.30890000E-04  1.0878000      1.07836109E-04  0.26412001      1138.8000      2.01319996E-02
 15.100000        672.98999      181.67000      59.794998      1.0078000      1.87869999E-04  1.1201000      1.09102773E-04  0.31442001      1151.6000      3.82689983E-02
 22.600000        694.15997      203.05000      59.245998      1.0114000      1.67100006E-04  1.1425000      1.09558336E-04  0.34567001      1159.3000      5.58030009E-02
 30.100000        710.15997      219.27000      58.806000      1.0148000      1.54020003E-04  1.1603000      1.09744447E-04  0.36873999      1165.0000      7.29679987E-02
 37.599998        723.16998      232.50999      58.432999      1.0179000      1.44720005E-04  1.1752000      1.09802771E-04  0.38718000      1169.4000      8.98730010E-02
 45.099998        734.21997      243.78999      58.105000      1.0208000      1.37630006E-04  1.1883000      1.09791661E-04  0.40261999      1173.0000      0.10658000
 52.599998        743.87000      253.67000      57.811001      1.0236000      1.31959998E-04  1.2000999      1.09738889E-04  0.41596001      1176.0000      0.12314000
 60.099998        752.46002      262.48999      57.542000      1.0263000      1.27280000E-04  1.2108001      1.09658329E-04  0.42772001      1178.6000      0.13957000
 67.599998        760.23999      270.48999      57.292999      1.0289000      1.23310005E-04  1.2206000      1.09561108E-04  0.43827000      1180.9000      0.15590000
 75.099998        767.34998      277.82999      57.062000      1.0314000      1.19889999E-04  1.2298000      1.09447225E-04  0.44784001      1182.9000      0.17214000
 82.599998        773.90997      284.62000      56.844002      1.0339000      1.16900002E-04  1.2385000      1.09325003E-04  0.45662001      1184.7000      0.18831000
 90.099998        780.01001      290.95001      56.639000      1.0362999      1.14249997E-04  1.2467000      1.09197223E-04  0.46474001      1186.4000      0.20442000
 97.599998        785.71997      296.89001      56.443001      1.0387000      1.11879999E-04  1.2545000      1.09061111E-04  0.47229001      1187.8000      0.22048000
 105.10000        791.09003      302.48999      56.257000      1.0410000      1.09729997E-04  1.2618999      1.08922221E-04  0.47936001      1189.2000      0.23649000
 112.60000        796.16998      307.79001      56.078999      1.0432000      1.07779997E-04  1.2691000      1.08780558E-04  0.48600999      1190.4000      0.25246000
 120.10000        800.97998      312.82999      55.908001      1.0455000      1.05990002E-04  1.2759000      1.08636115E-04  0.49228999      1191.6000      0.26839000
 127.60000        805.57001      317.64001      55.743000      1.0477000      1.04339997E-04  1.2826000      1.08491673E-04  0.49823999      1192.6000      0.28430000
 135.10001        809.94000      322.23001      55.584000      1.0499001      1.02819999E-04  1.2890000      1.08344444E-04  0.50389999      1193.6000      0.30017999
 142.60001        814.13000      326.64999      55.431000      1.0520000      1.01400001E-04  1.2952000      1.08194443E-04  0.50931001      1194.5000      0.31604001
 150.10001        818.14001      330.89001      55.282001      1.0541000      1.00079997E-04  1.3012000      1.08044449E-04  0.51446998      1195.3000      0.33188999
 157.60001        822.01001      334.97000      55.137001      1.0562000      9.88359971E-05  1.3071001      1.07894441E-04  0.51942003      1196.1000      0.34771001
 165.10001        825.72998      338.92001      54.995998      1.0583000      9.76709998E-05  1.3128999      1.07744447E-04  0.52418000      1196.8000      0.36353001
 172.60001        829.31000      342.73001      54.859001      1.0604000      9.65720028E-05  1.3185000      1.07594446E-04  0.52876002      1197.5000      0.37933999
 180.10001        832.78003      346.42001      54.726002      1.0624000      9.55349969E-05  1.3240000      1.07444444E-04  0.53316998      1198.1000      0.39513001
 187.60001        836.14001      350.00000      54.596001      1.0645000      9.45509964E-05  1.3293000      1.07294443E-04  0.53742999      1198.7000      0.41091999
 195.10001        839.40002      353.48001      54.467999      1.0664999      9.36180004E-05  1.3346000      1.07141670E-04  0.54154998      1199.2000      0.42671001
 202.60001        842.54999      356.85999      54.344002      1.0685000      9.27290021E-05  1.3398000      1.06991669E-04  0.54553998      1199.8000      0.44250000
```

P_{sat}

T_{sat}

h_{liq}

ρ_{liq}

$C_{p,liq}$

μ_{liq}

γ_{liq}

k_{liq}

s_{liq}

h_{vap}

ρ_{vap}

Table continues with vapor values for: C_p, μ, γ, k, and s

Other Fluid Options

- **Constant Property Option**
  - Allows the user to model a fluid with constant density (ρ) and viscosity (μ)
  - Energy equation is not solved
  - Available only for steady-state models

- **Ideal Gas Option**
  - Allows the user to model an ideal gas
  - Uses constant viscosity (μ) and specific heat (C_p)

- **GFSSP Ideal Gas (default)**
  - AIR at room temperature values for viscosity (μ) and specific heat (C_p)

User-Coded Fluid (1/2)

- User subroutine **PRPUSER**

- Allows the user to overwrite any or all fluid properties

```
SUBROUTINE PRPUSER(I_GIVEN, I_NFLUID, Z_P, Z_T, Z_RHO, Z_H, Z_CP,
     +         Z_CV, Z_S, Z_GAMMA, Z_MU, Z_K, I_KR, Z_XV,
     +         Z_RHOL, Z_HL, Z_CPL, Z_CVL, Z_SL, Z_GAMMAL, Z_MUL, Z_KL,
     +         Z_RHOV, Z_HV, Z_CPV, Z_CVV, Z_SV, Z_GAMMAV, Z_MUV, Z_KV)
C      PURPOSE: ADD NEW FLUID PROPERTY
C      I_GIVEN: Inputs are:  (1) P/T  (2) P/H  (3) P/S  (4) Psat/X
```

User-Coded Fluid (2/2)

- User can choose which properties to overwrite
  - For example, overwrite only viscosity to match textbook solution
  - Define a new fluid using their own Fortran-coded property package

- **GFSSP** installation directory includes instructions for calling REFPROP from a User Subroutine
  - User must have
    - Installed REFPROP v9
    - Intel Fortran compiler

- Test cases have shown that REFPROP agrees well with GASP/WASP
  - Model run time is much slower

Fluid Options Summary

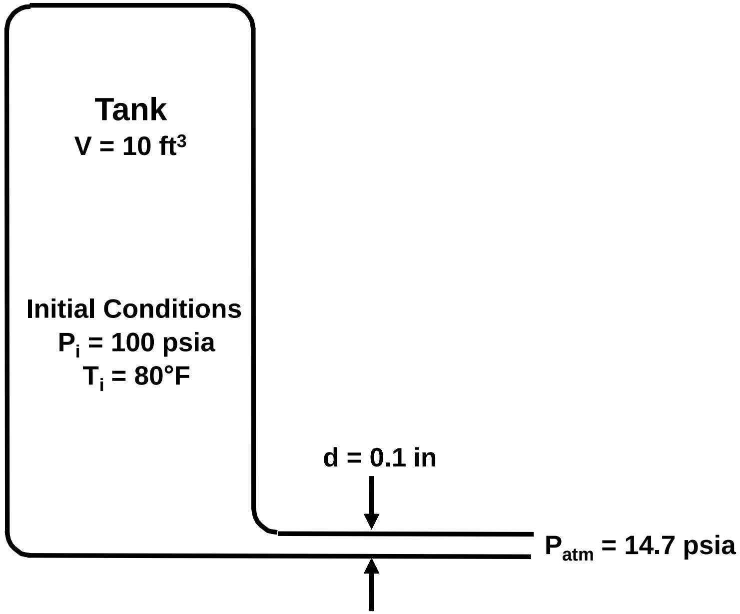
- **GFSSP** considers both gas and liquid as real fluid
  - Liquid is also modeled as compressible fluid

- GASP/WASP and GASPAK
  - Provide higher order equation of state to calculate properties of liquid and vapor state over a wide range

- Options to add new fluid to library
  - Table look-up provision
  - User-supplied Fortran code

- Constant Property and Ideal Gas options can also be used

GFFSP Demonstration Problem 2

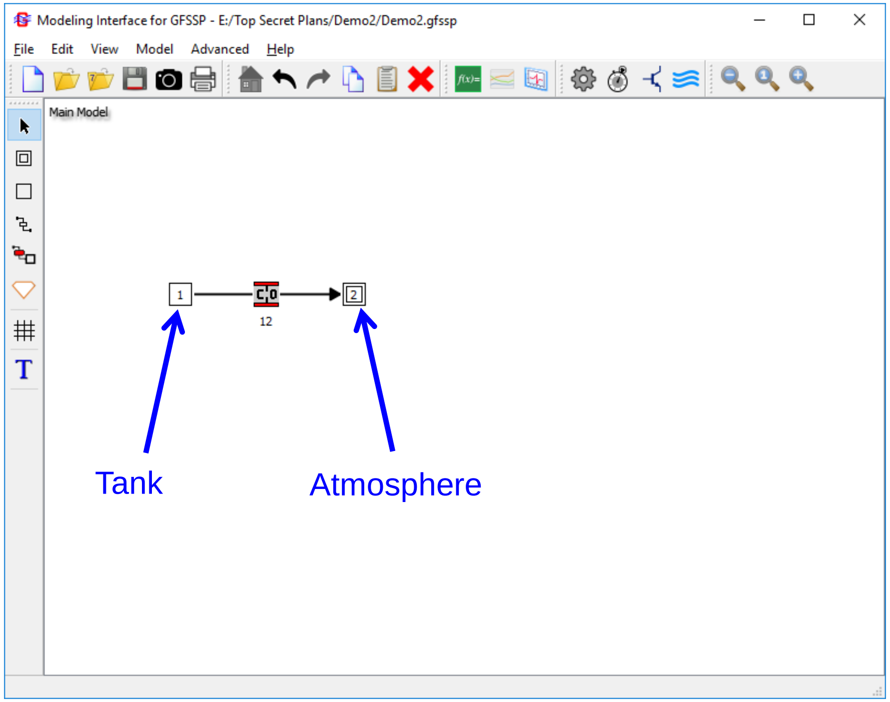


Tank
V = 10 ft^3

Initial Conditions
P_i = 100 psia
T_i = 80°F

d = 0.1 in

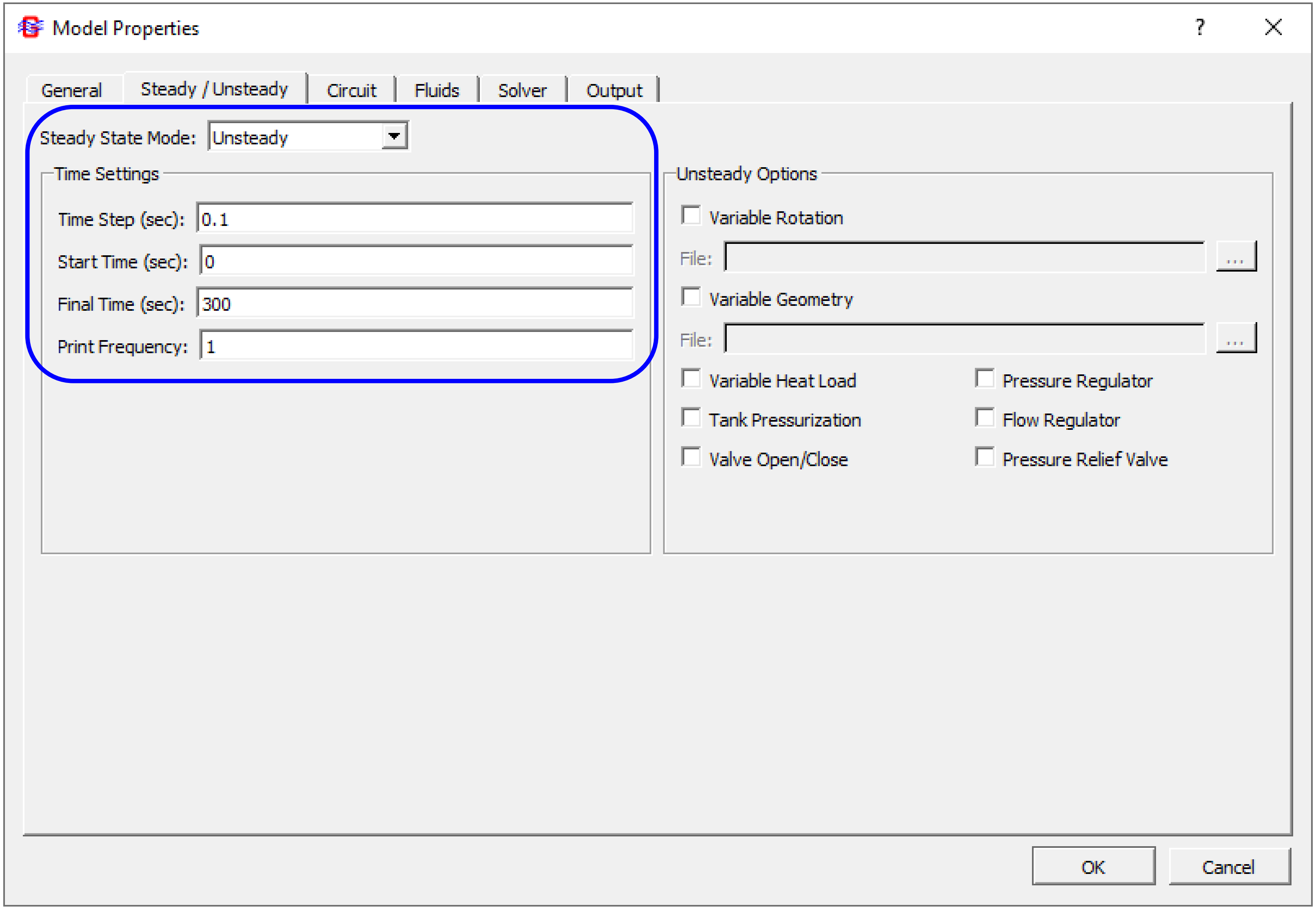
P_{atm} = 14.7 psia



Tank

Atmosphere

Creating an Unsteady Model



Model Properties

Steady State Mode: Unsteady

Time Settings

Time Step (sec): 0.1

Start Time (sec): 0

Final Time (sec): 300

Print Frequency: 1

Unsteady Options

☐ Variable Rotation

File:

☐ Variable Geometry

File:

☐ Variable Heat Load

☐ Tank Pressurization

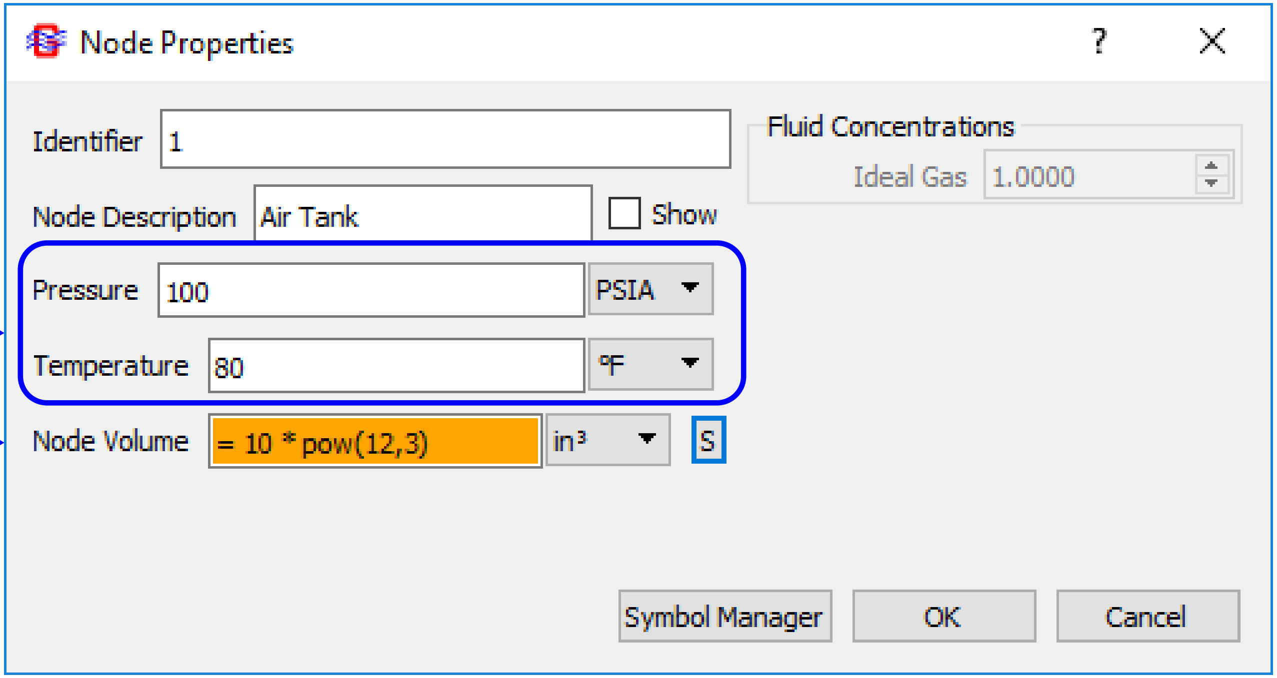
☐ Valve Open/Close

☐ Pressure Regulator

☐ Flow Regulator

☐ Pressure Relief Valve

Internal Node Initial Conditions



Node Properties

Identifier: 1

Node Description: Air Tank

☐ Show

Pressure: 100 PSIA

Temperature: 80 °F

Node Volume: = 10 * pow(12,3) in³

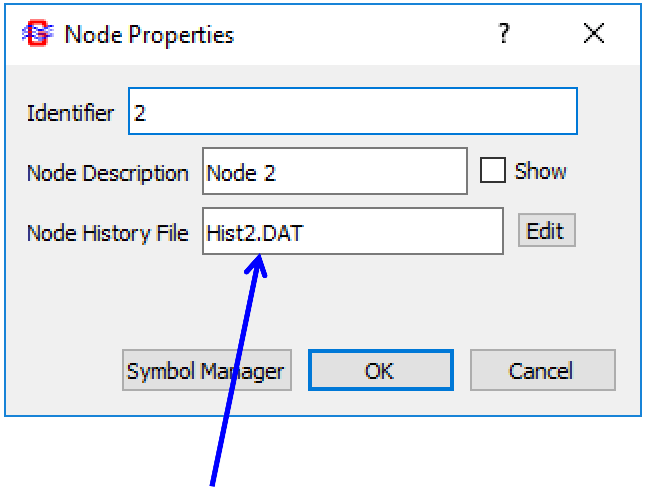
Fluid Concentrations

Ideal Gas: 1.0000

Initial P, T

Tank Volume

Transient Boundary Conditions

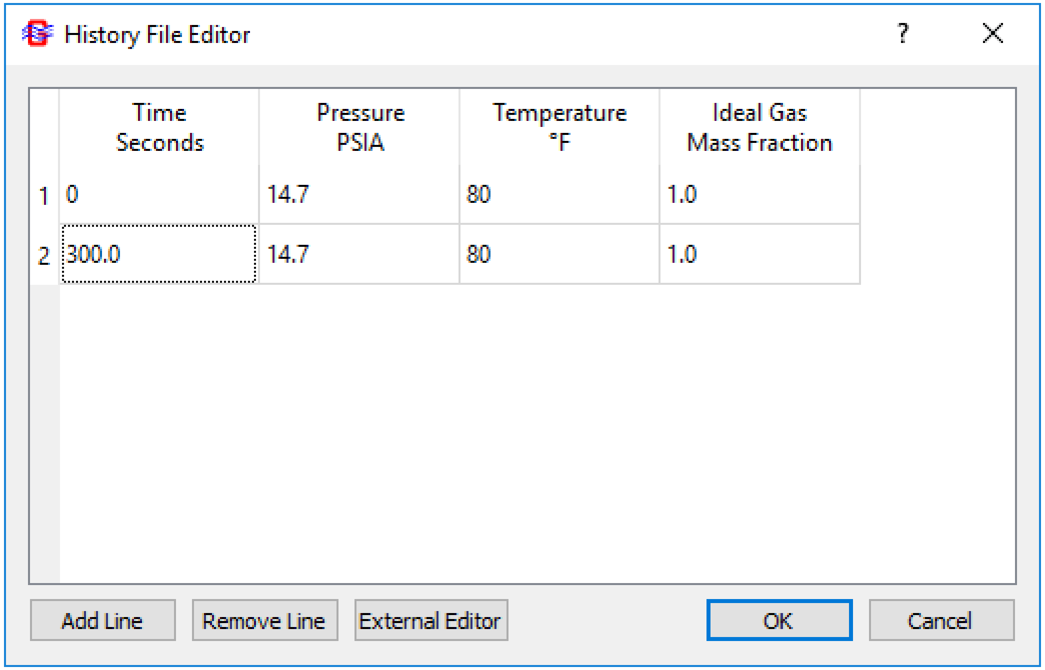


Identifier: 2

Node Description: Node 2 ☐ Show

Node History File: Hist2.DAT

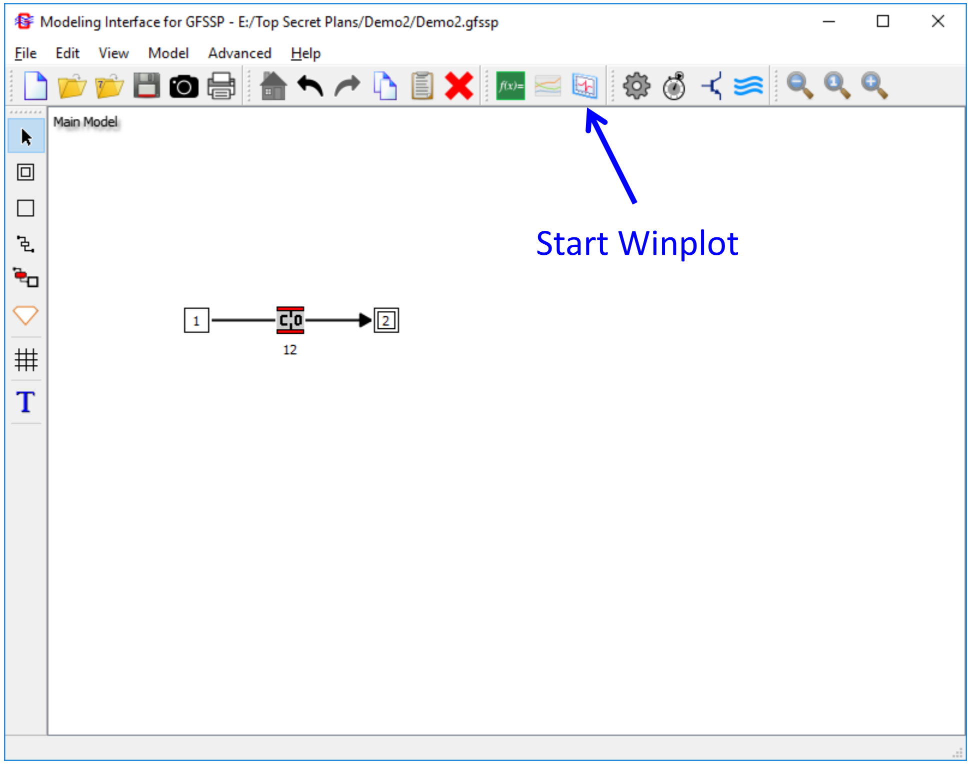
Specify History Filename



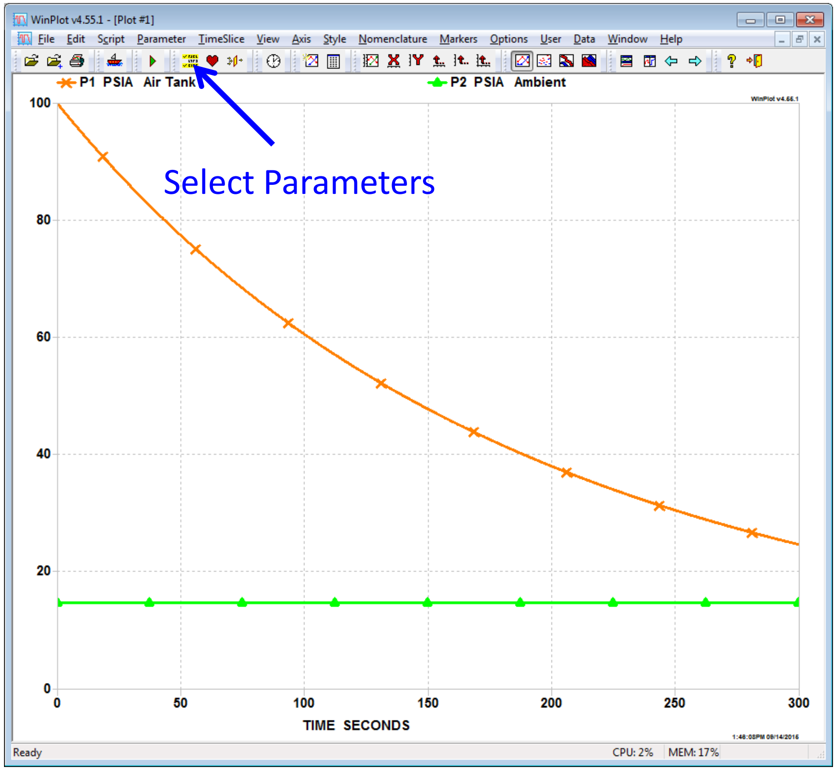
	Time Seconds	Pressure PSIA	Temperature °F	Ideal Gas Mass Fraction
1	0	14.7	80	1.0
2	300.0	14.7	80	1.0

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

Plotting Transient Results



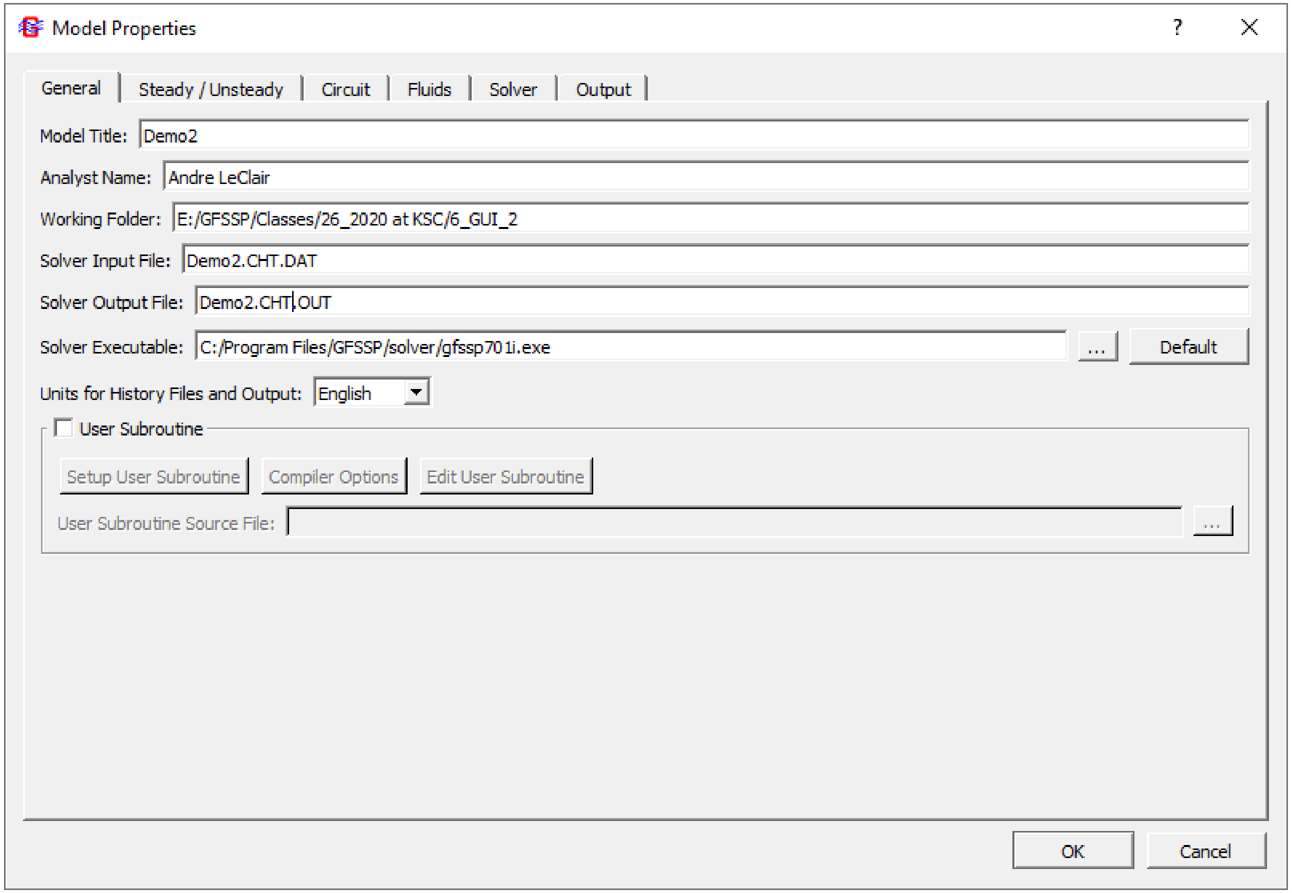
Start Winplot



Select Parameters

Add Conjugate Heat Transfer

- Previous solution was adiabatic (no heat transfer from wall)
- Now repeat problem with Conjugate Heat Transfer to model natural convection between the air and the warm tank wall.



Model Title: Demo2

Analyst Name: Andre LeClair

Working Folder: E:/GFSSP/Classes/26_2020 at KSC/6_GUI_2

Solver Input File: Demo2.CHT.DAT

Solver Output File: Demo2.CHT.OUT

Solver Executable: C:/Program Files/GFSSP/solver/gfssp701i.exe

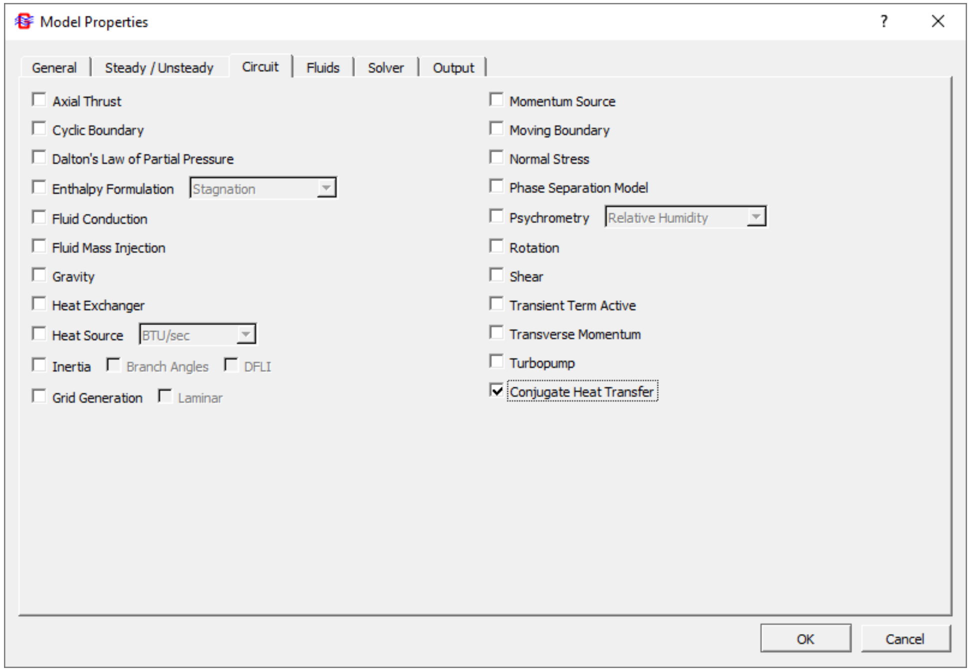
Units for History Files and Output: English

☐ User Subroutine

User Subroutine Source File:

Enable Conjugate Heat Transfer

- Check Conjugate Heat Transfer on Circuit tab



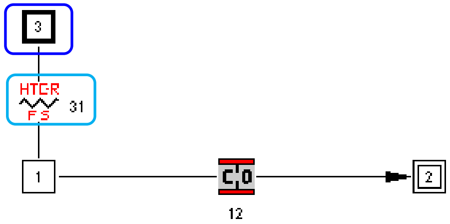
Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation — Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☐ Heat Exchanger
☐ Heat Source — BTU/sec
☐ Inertia ☐ Branch Angles ☐ DFLI
☐ Grid Generation ☐ Laminar

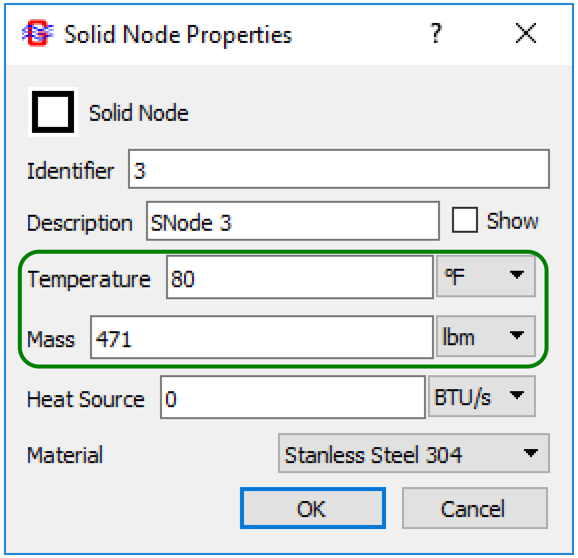
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry — Relative Humidity
☐ Rotation
☐ Shear
☐ Transient Term Active
☐ Transverse Momentum
☐ Turbopump
☑ Conjugate Heat Transfer

Add Solid Node 3, and Fluid-to-Solid Conductor 31



Enter Heat Transfer Information

- Input
  - Tank material, mass: SS304, 471 lb_m ; initially at 80 °F
  - Tank surface area: 3250 in^2
  - Heat transfer coefficient correlation is Vertical Plate Natural Convection
    - Characteristic length is tank diameter: 2.68 ft



Solid Node Properties

Solid Node

Identifier: 3

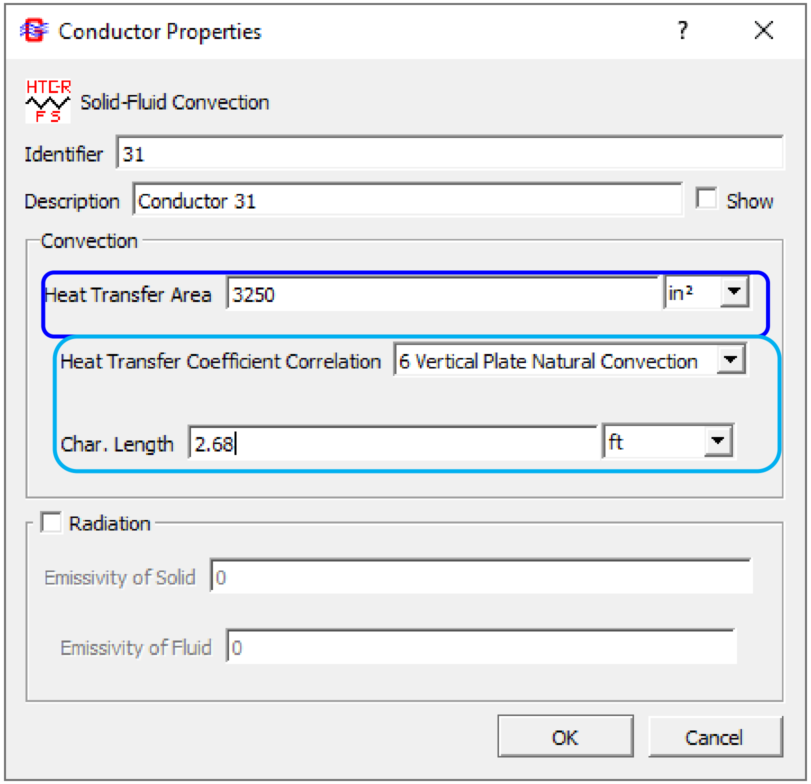
Description: SNode 3 ☐ Show

Temperature: 80 °F

Mass: 471 lbm

Heat Source: 0 BTU/s

Material: Stanless Steel 304



Conductor Properties

Solid-Fluid Convection

Identifier: 31

Description: Conductor 31 ☐ Show

Convection

Heat Transfer Area: 3250 in²

Heat Transfer Coefficient Correlation: 6 Vertical Plate Natural Convection

Char. Length: 2.68 ft

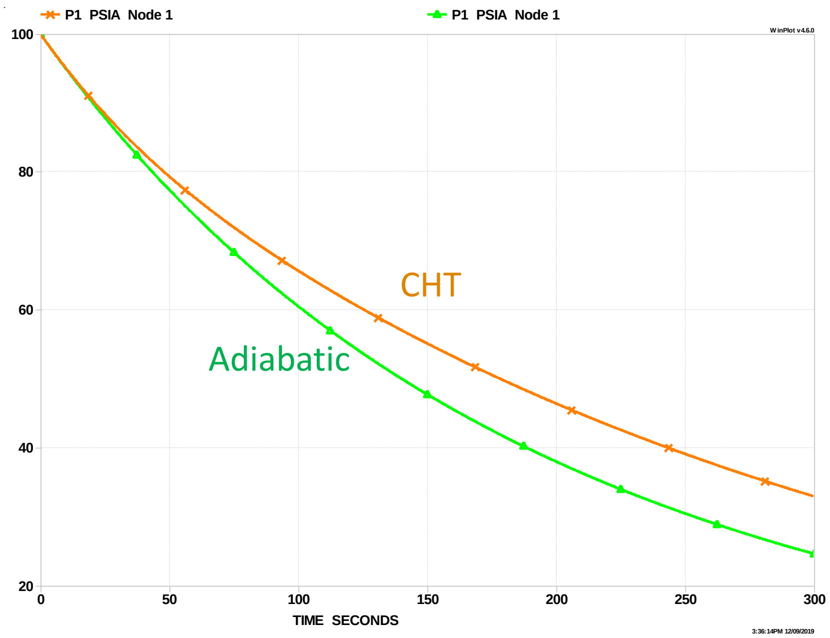
☐ Radiation

Emissivity of Solid: 0

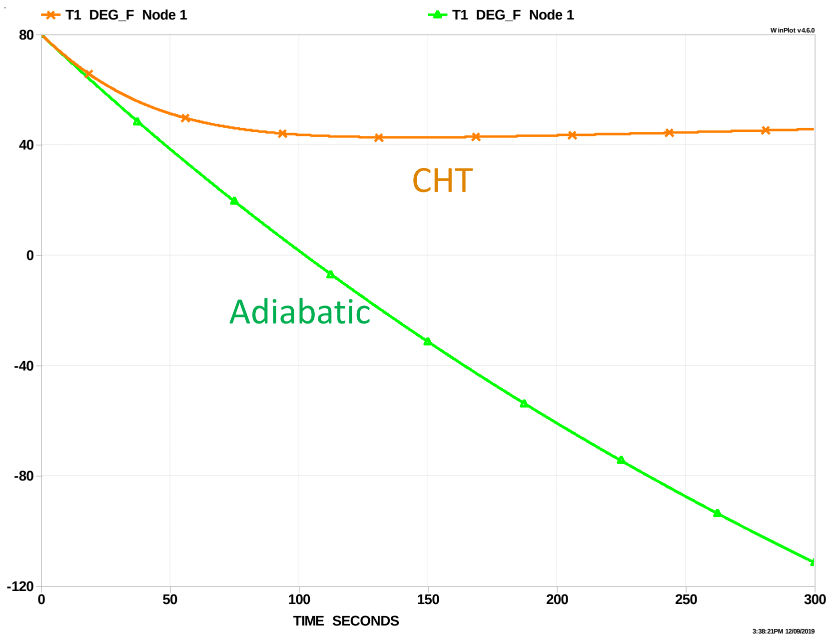
Emissivity of Fluid: 0

Compare Results (1/2)

- Compare Results (1/2)
  - Run the model and plot the results in Winplot
  - Plot Demo2.WPL file for comparison
    - ➢ Note: Pressure and temperature decrease more slowly when there is heat transfer from the tank wall



Pressure

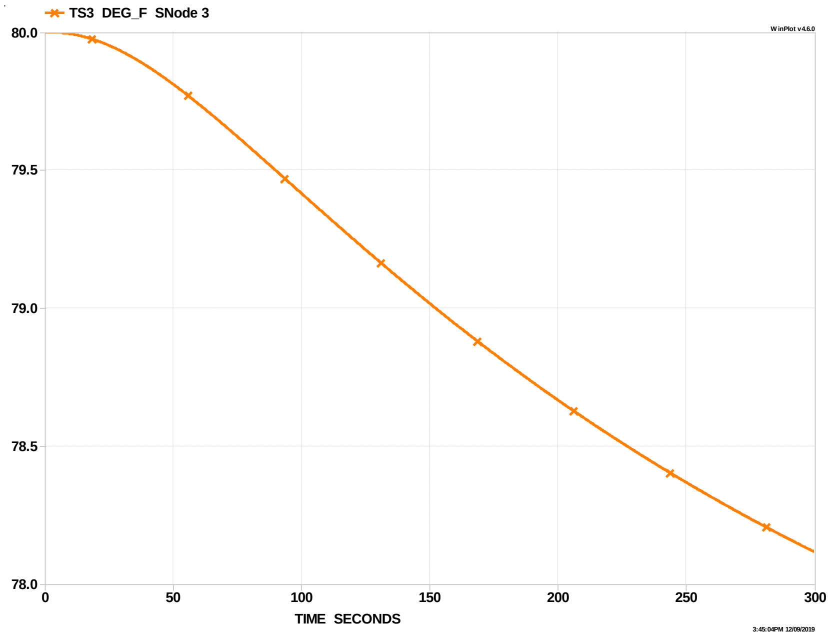


Temperature

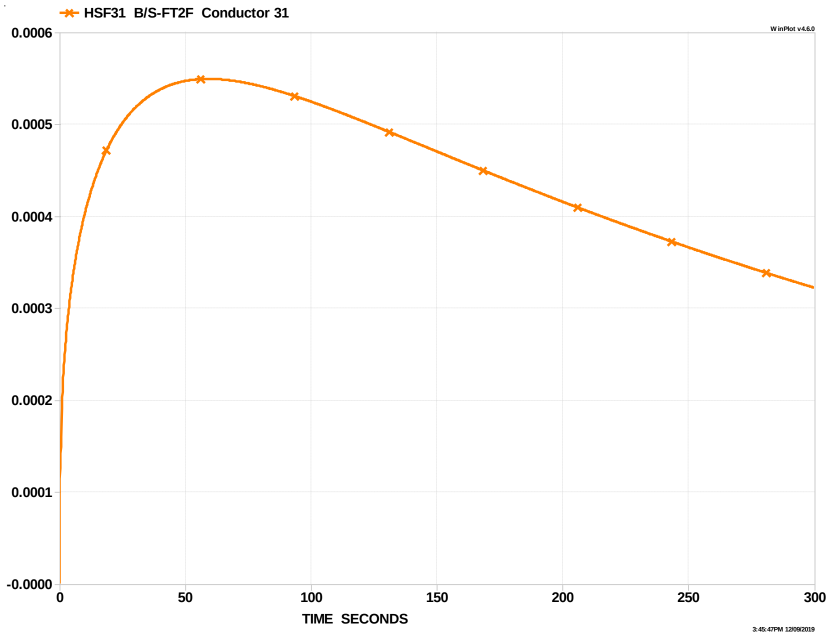
Compare Results (2/2)

- Compare Results (2/2)
  - Plot the tank wall temperature and the heat transfer coefficient
  - Note: Tank wall heat transfer coefficient is not constant over time

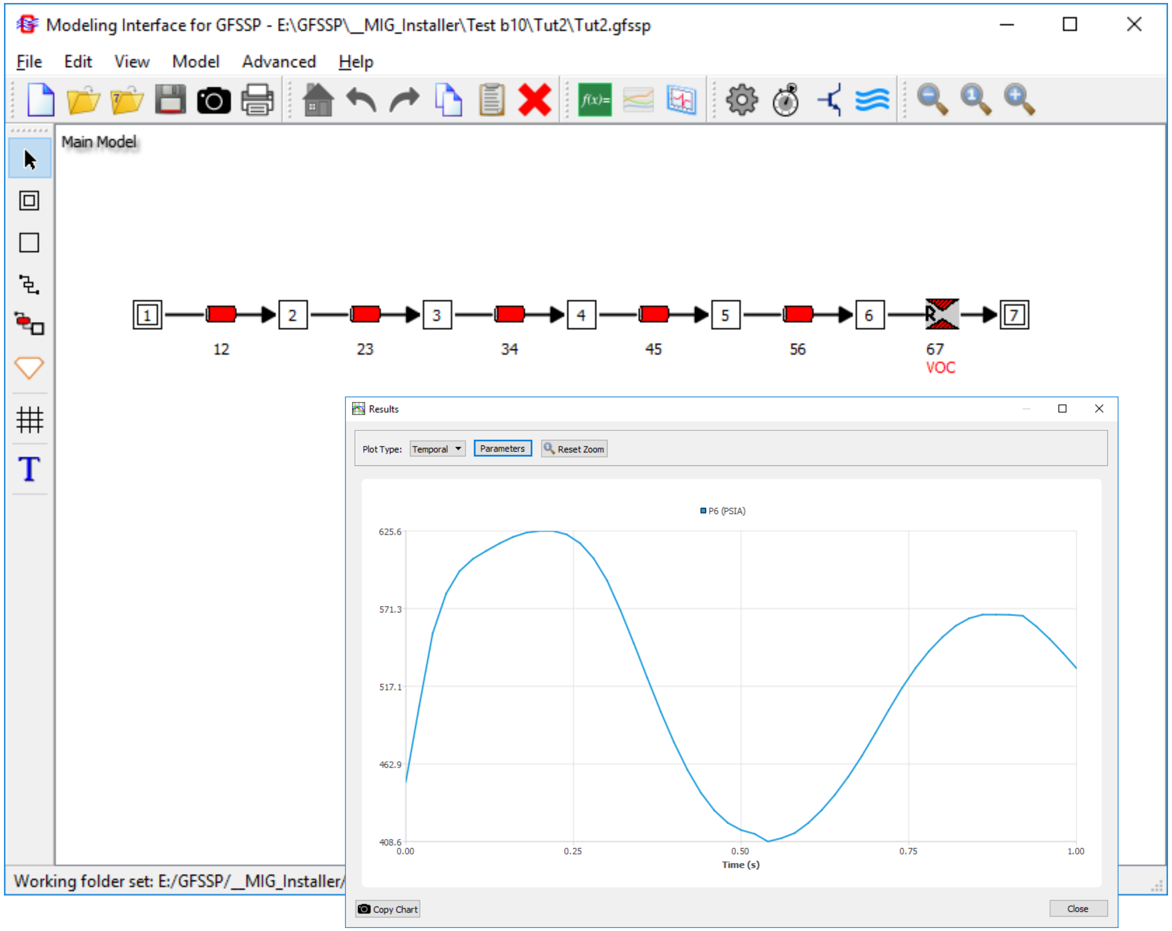
Tank Wall Temperature



Heat Transfer Coefficient



Fluid Transient



Modeling Interface for GFSSP - E:\GFSSP__MIG_Installer\Test b10\Tut2\Tut2.gfssp

Plot Type: Temporal

Working folder set: E:/GFSSP/__MIG_Installer/

Content

- Classification of Unsteady Flow
- Causes of Transient
- Methods of Analysis
- Valve Closing
- 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 term 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

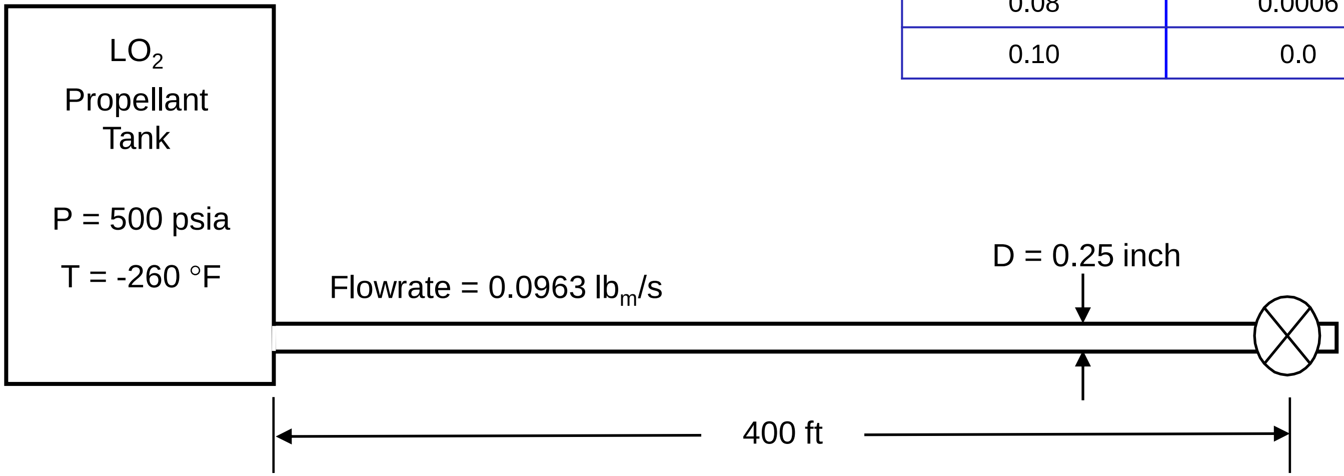
- Arithmetic Method
- Graphical Method
- Finite Difference Method
  - Method of Characteristics
  - Predictor-Corrector
- Impedance Method
- Finite Volume Method (**GFSSP**)

Ex15 – Simulation of Fluid Transient Following Sudden Valve Closure (waterhammer)

- Objectives of Analysis
  - Maximum Pressure
  - Frequency of Oscillation

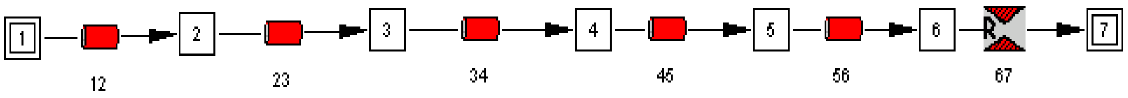
Valve Closure History

Time (sec)	Area (in^2)
0.0	0.0491
0.02	0.0164
0.04	0.0055
0.06	0.0018
0.08	0.0006
0.10	0.0



Ex15 - GFSSP Model (waterhammer)

- For this **GFSSP** model
  - Discretize total pipe length into 5 branches (80 ft. each)
  - Run a steady state model with 450 psia ambient condition
  - Run unsteady model with steady state solution as initial value

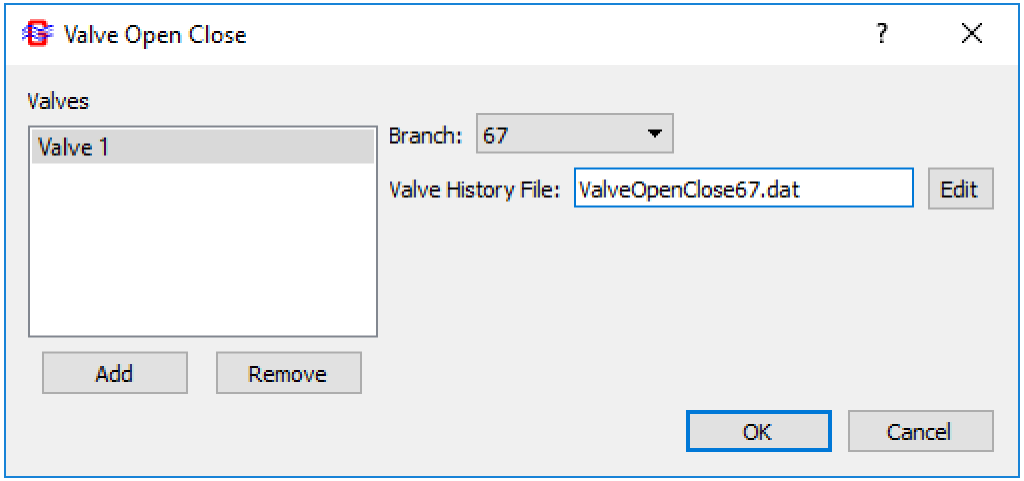


LO_2
Propellant
Tank

P = 500 psia
T = -260 °F

Valve

Ambient

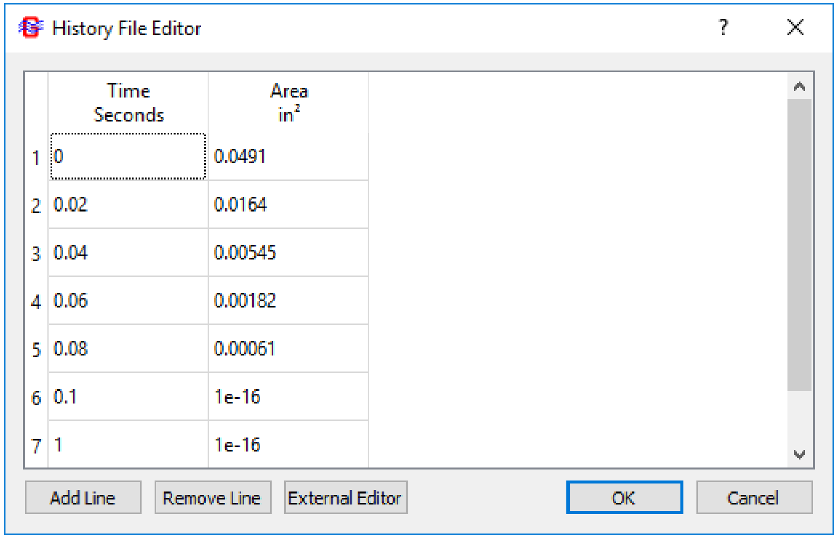


Valves

Valve 1

Branch: 67

Valve History File: ValveOpenClose67.dat



	Time Seconds	Area in^2
1	0	0.0491
2	0.02	0.0164
3	0.04	0.00545
4	0.06	0.00182
5	0.08	0.00061
6	0.1	1e-16
7	1	1e-16

Time Step Check

- Check assumed time step ($\Delta\tau$) with Courant Number

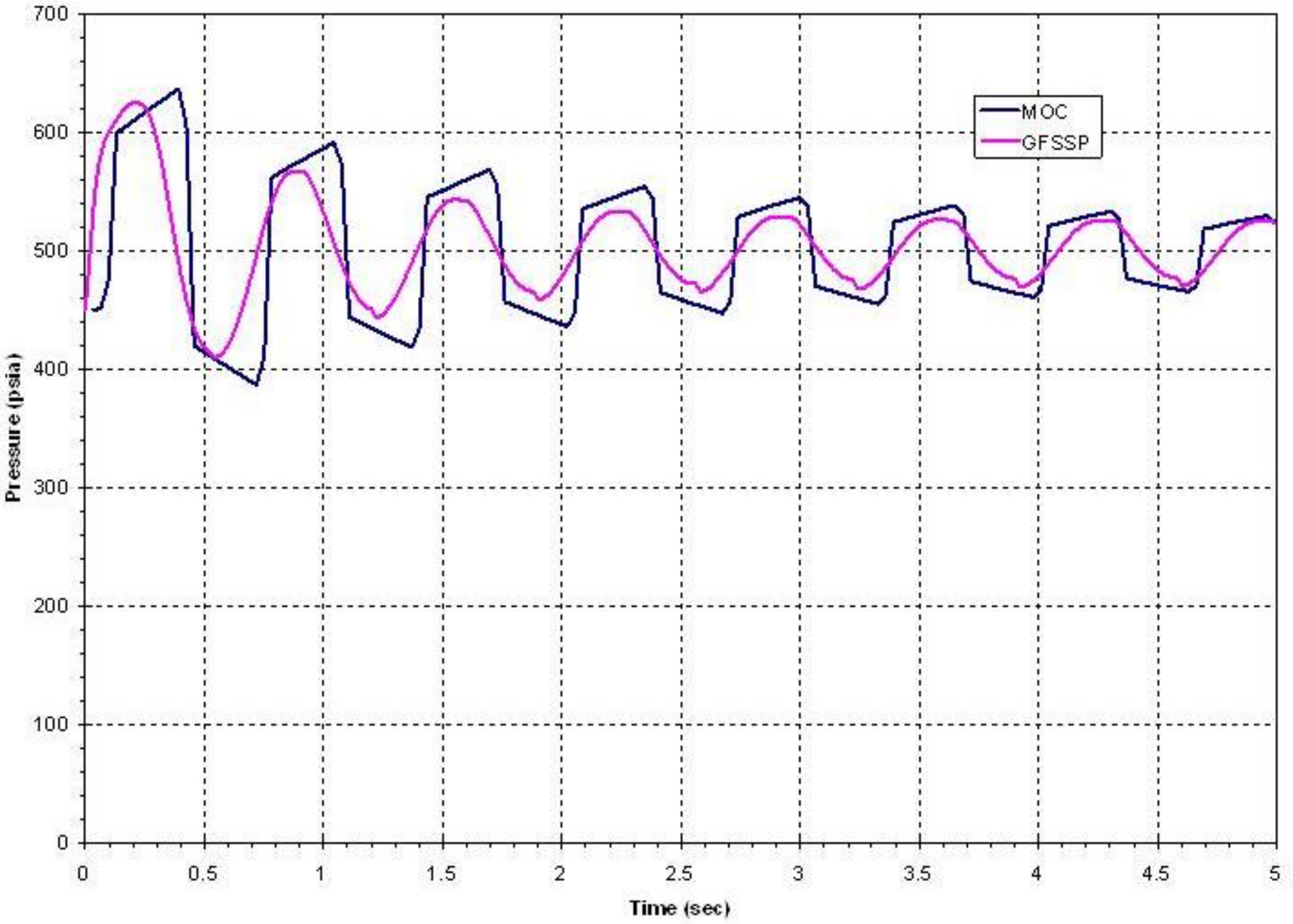
Courant Number $= \dfrac{4\mathrm{L}_{\mathrm{branch}}}{a_{fluid}\,\Delta\tau} \geq 1$ $\longrightarrow$ **Courant Number = 6.5**

where:
LOX speed of sound (a_{fluid}) is 2462 ft/sec

- Recheck Courant Number when any changes occur to $\mathrm{L}_{\mathrm{branch}}$ and/or $\Delta\tau$

Ex15 - Results

- Comparison between **GFSSP** and Method of Characteristics (MOC)



Description of Test Cases

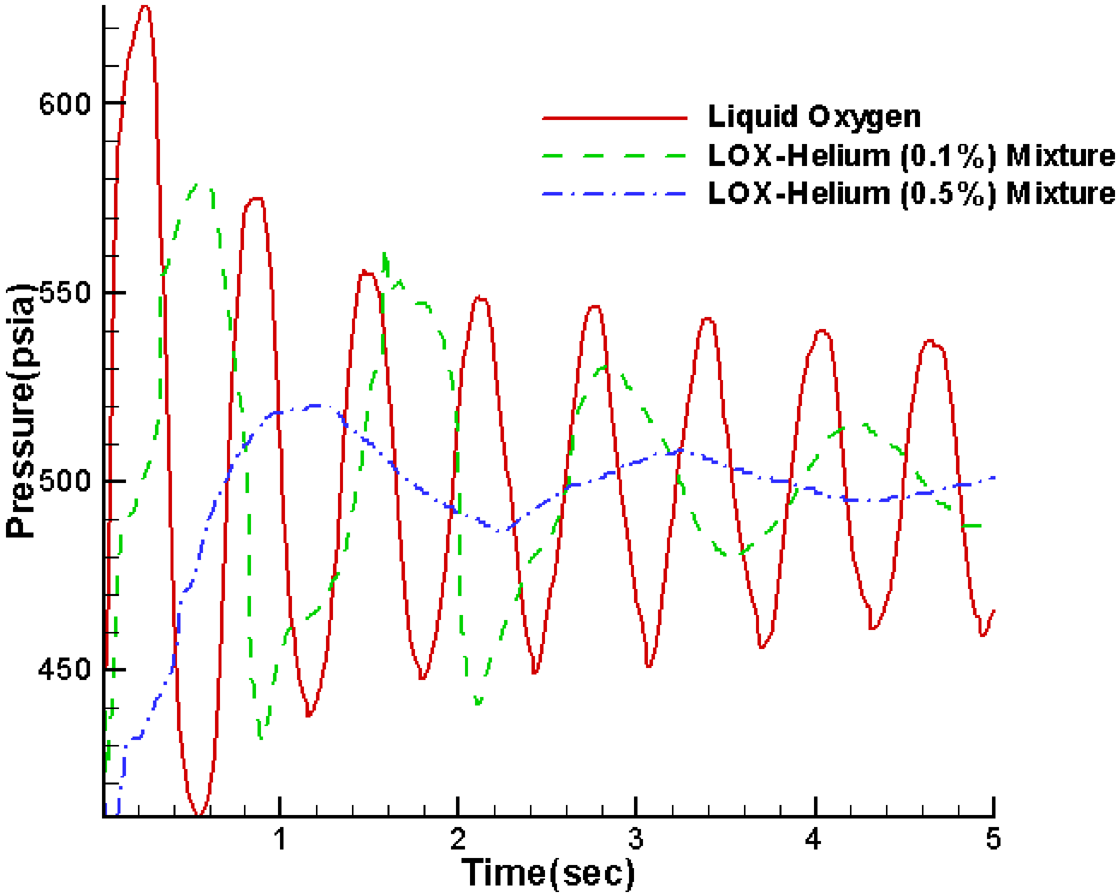
- Time step for each test case is so chosen that Courant Number ≥ 1

P_{tank} = 500 psia
LO_2: T_{tank} = -260 °F
H_2O: T_{tank} = 70 °F
LH_2: T_{tank} = -414 °F

Case No.	Fluid	Number of Branches	Time Step (sec)	Sound Speed (ft/sec)	Flowrate (lbm/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	10	0.01	2462	0.0963	611	0.65

* Pressure oscillations are due to condensation
** Estimated from period of oscillation [$a = 4L/\lambda$}

Gas Liquid Mixture



Comparison Between GFSSP & MOC Solution

"Numerical Modeling of Fluid Transients by a Finite Volume Procedure for Rocket Propulsion Systems", Majumdar, A. K. and Flachbart, R. H.

Paper No. FEDSM2003-45275, Proceedings of ASME FEDSM'03

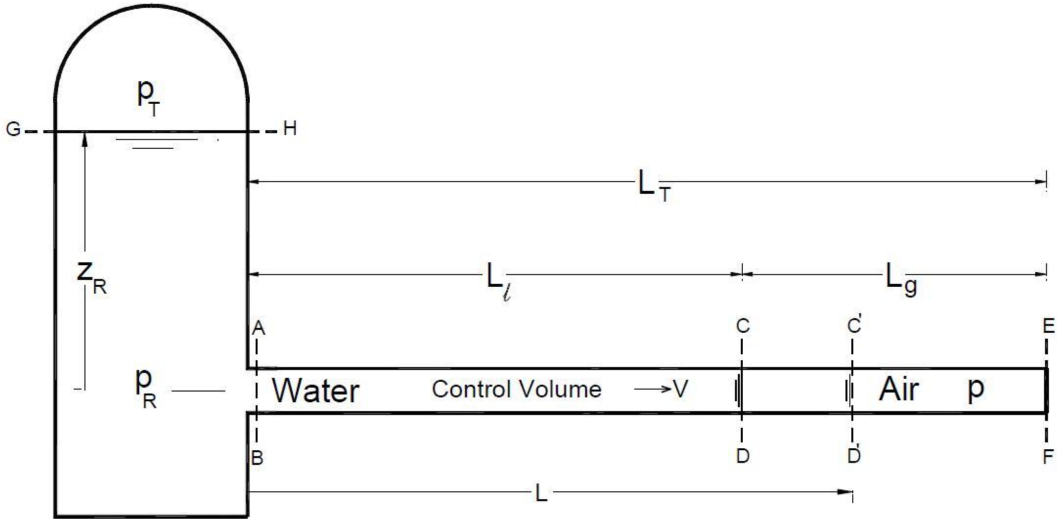
4th ASME/JSME Joint Fluids Engineering Conference, Honolulu, Hawaii, July 6-10, 2003

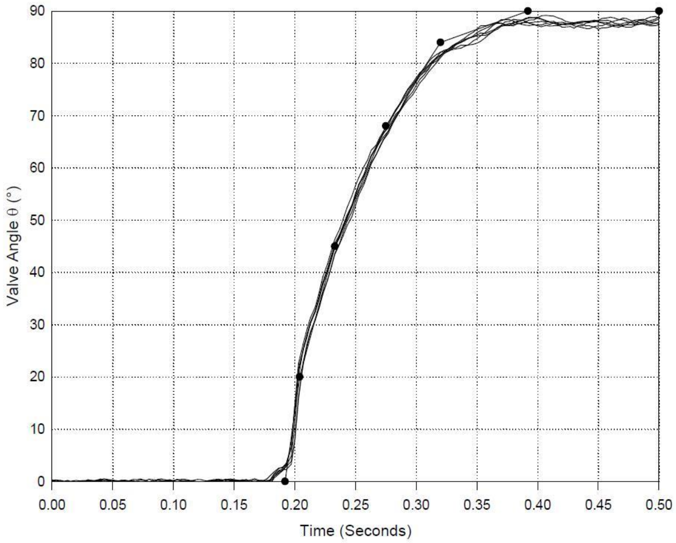
Fluid	Flowrate (lb_m/sec)	Velocity (ft/sec)	Friction Factor*	Speed of Sound (ft/sec)	Max. Pressure Rise Above Supply Pressure (psi)		Period of Oscillation (sec)	
					MOC	GFSSP	MOC	GFSSP
Water	0.071	3.34	0.0347	4892	214	204	0.33	0.33
Oxygen	0.0963	4.35	0.0196	2455	136	126	0.65	0.65
Hydrogen	0.0278	19.01	0.0157	3725	61	45	0.43	0.43

* Used in MOC solution

Rapid Valve Opening **(priming)**

Schematic of Pipeline System with Ball Valve location





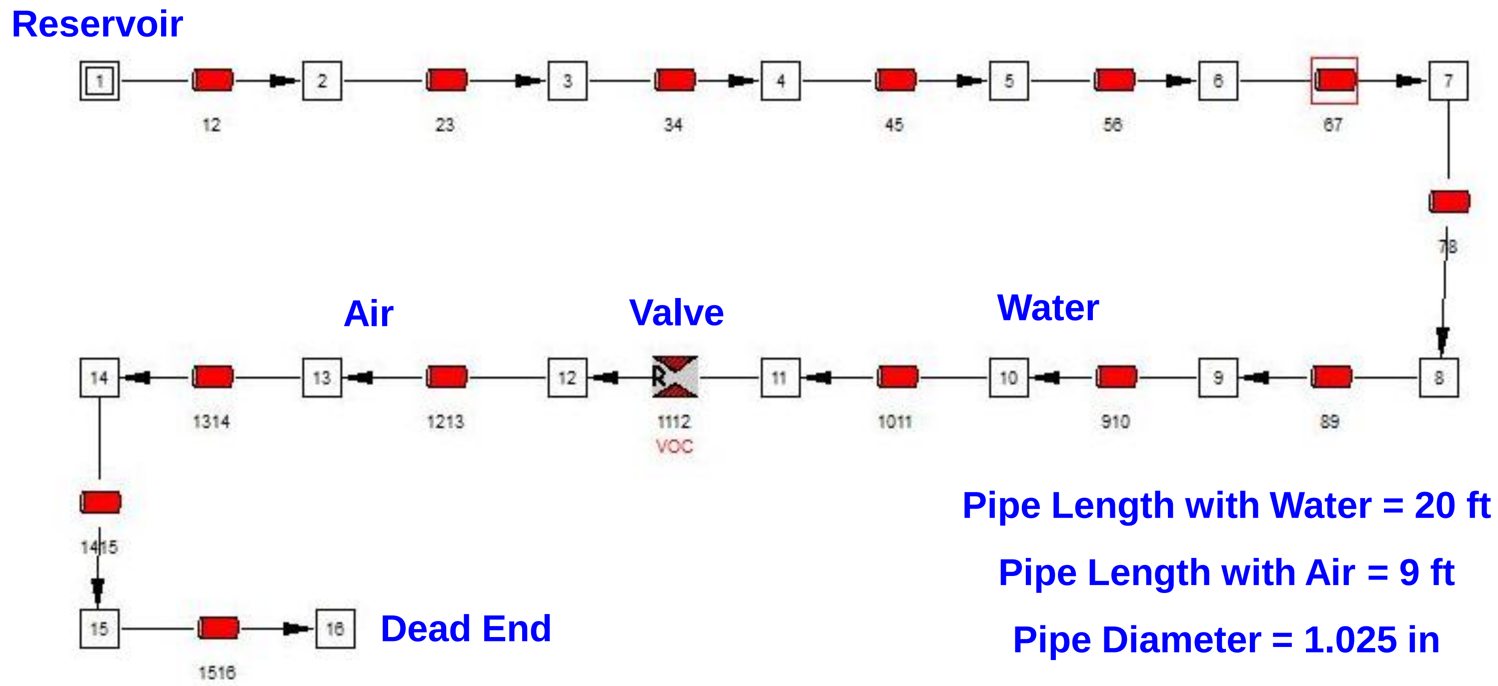
Ball Valve Opening History Parameters

$\alpha = L_g/L_T$

$P_R = p_R/p_0$

GFSSP Model (priming)

- Reservoir (Node 1) pressure range: 29.4 to 102.9 psia
- Initial Air pressure is atmospheric
- After Valve opens, Water rushes into the Air column and pressure rises



Pipe Length with Water = 20 ft

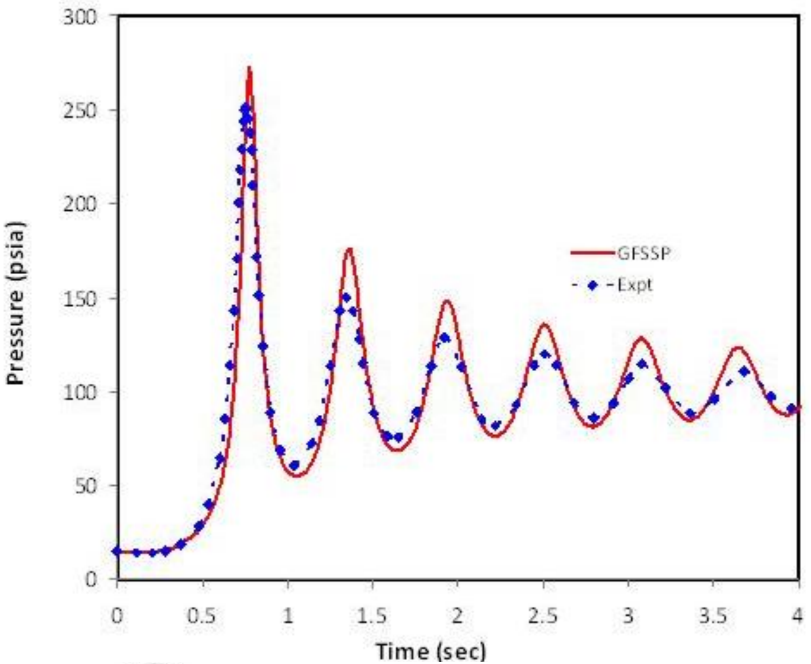
Pipe Length with Air = 9 ft

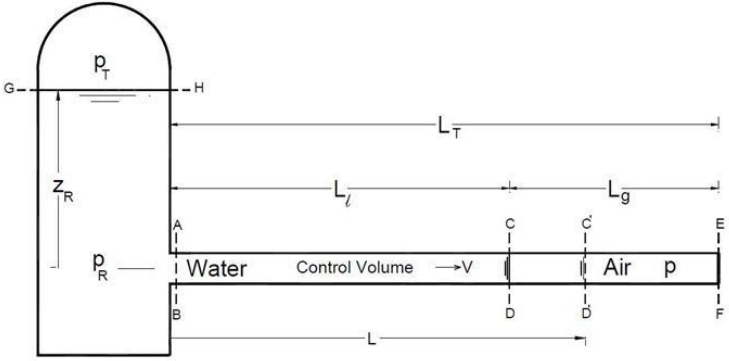
Pipe Diameter = 1.025 in

Results (priming)

- Pressure at Dead End (entrapped air pressure)

Ball Valve
Opening History
Parameters
$\alpha = L_g/L_T = 0.45$
$P_R = p_R/p_0 = 7$





Conclusions

- **GFSSP** has been used to compute fluid transient following rapid valve closure (waterhammer) and opening (priming)

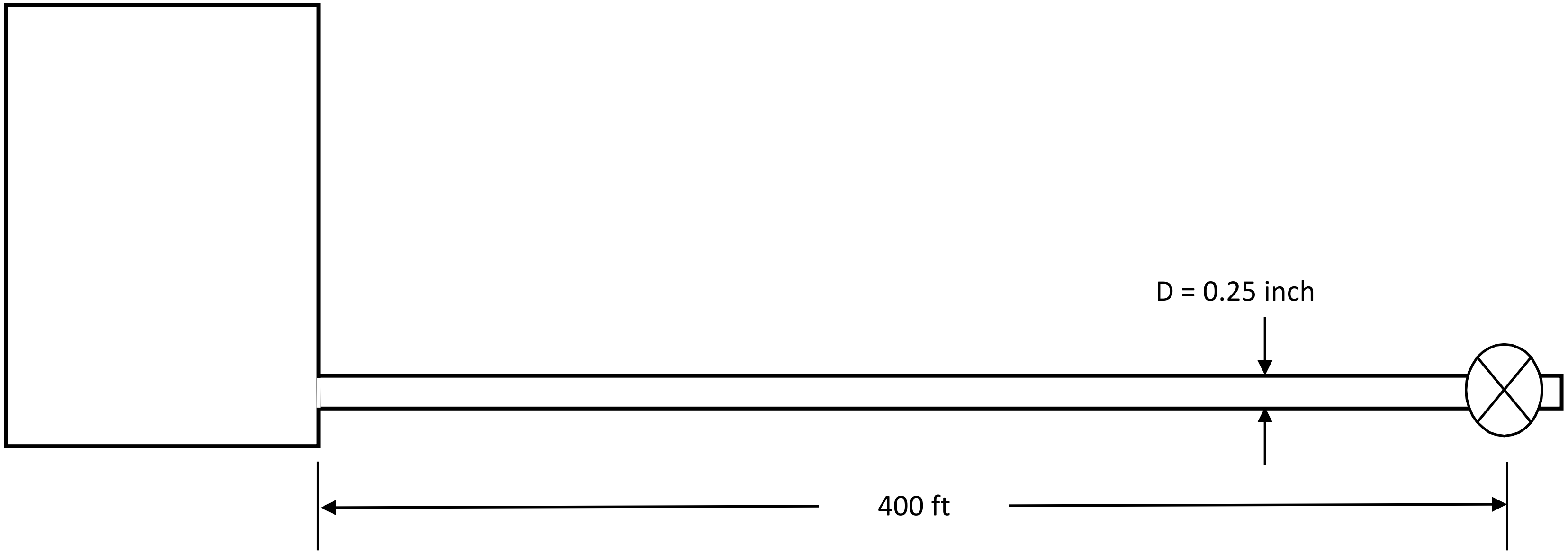
- **GFSSP** predictions have been compared with MOC solution and experimental data
  - Maximum pressure predictions - compare well
  - Oscillation (frequency) predictions - compare well
  - Discrepancies exist in damping rate - primarily due to rigid pipe assumption

- Demonstrations have been made
  - Two phase (Gas-Liquid) flow following valve closure
  - Condensation of liquid-vapor flow following valve closure
  - Sudden opening of valve in long pipeline

- Time step must satisfy Courant condition

Tutorial – 2

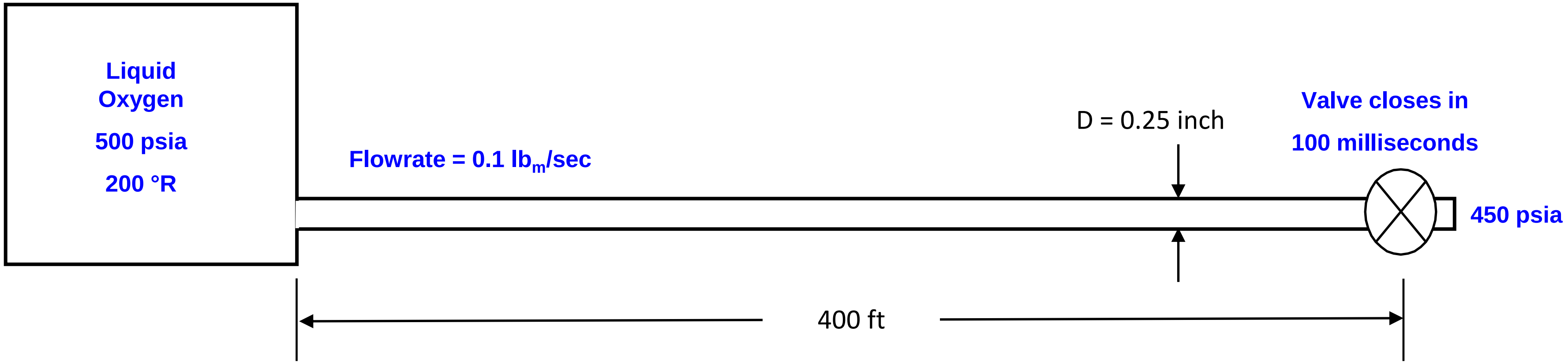
Simulation of Flow Transient Following Sudden Valve Closure



Fluid Transient Schematic

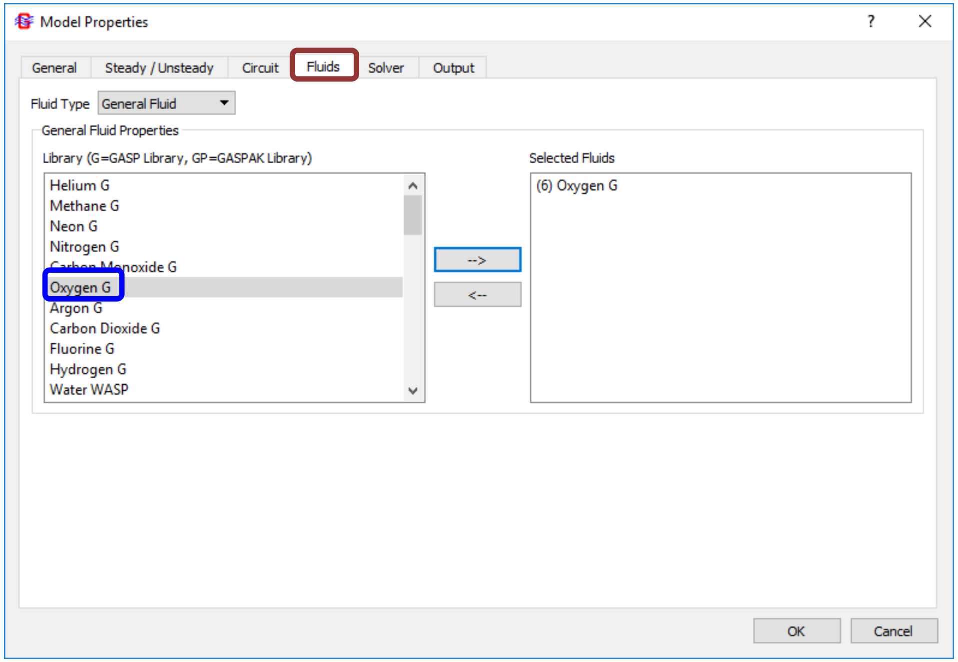
Problem Considered:

- Time dependent Pressure and Flow rate history during and after valve closure
- Speed of sound in LOx: a = 2462 ft/s
- Note that if valve closes in 0.1 sec, wave can only travel 246.2 ft, not far enough to reach the upstream end of the pipe.



Part 1: Build Steady State Model (1/3)

- Model File: Tut2.gfssp
- Input File: Tut2.DAT
- Output File: Tut2.OUT
- General Fluid: Oxygen (Fluids tab)
- Check: Save Information (Solver tab)
  - Save the steady state solution in the restart files



Fluid Type: General Fluid

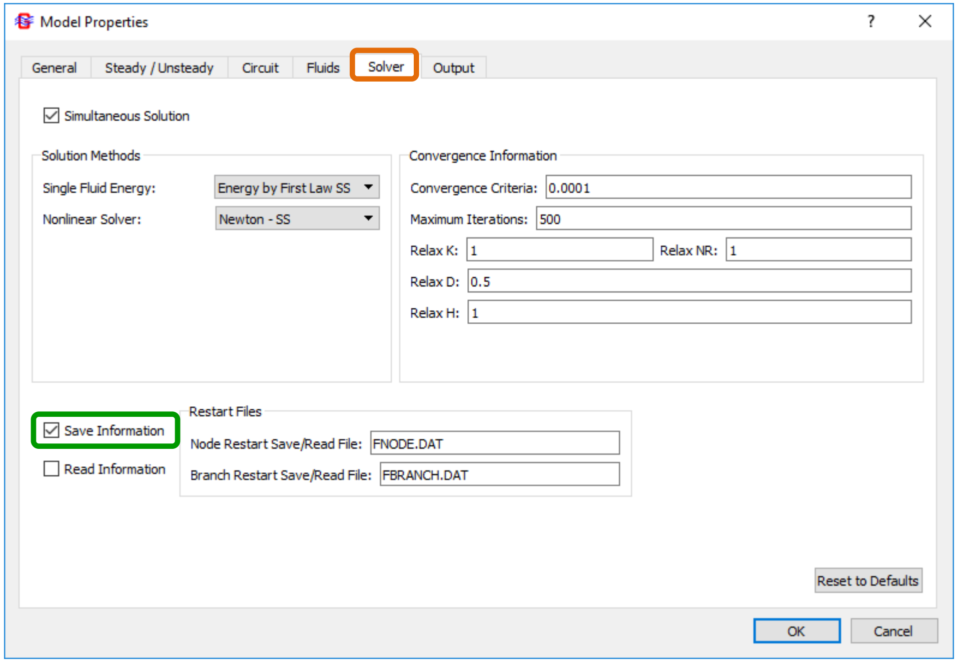
General Fluid Properties

Library (G=GASP Library, GP=GASPAK Library)

Helium G
Methane G
Neon G
Nitrogen G
Carbon Monoxide G
Oxygen G
Argon G
Carbon Dioxide G
Fluorine G
Hydrogen G
Water WASP

Selected Fluids

(6) Oxygen G



☑ Simultaneous Solution

Solution Methods

Single Fluid Energy: Energy by First Law SS

Nonlinear Solver: Newton - SS

Convergence Information

Convergence Criteria: 0.0001

Maximum Iterations: 500

Relax K: 1

Relax NR: 1

Relax D: 0.5

Relax H: 1

☑ Save Information

☐ Read Information

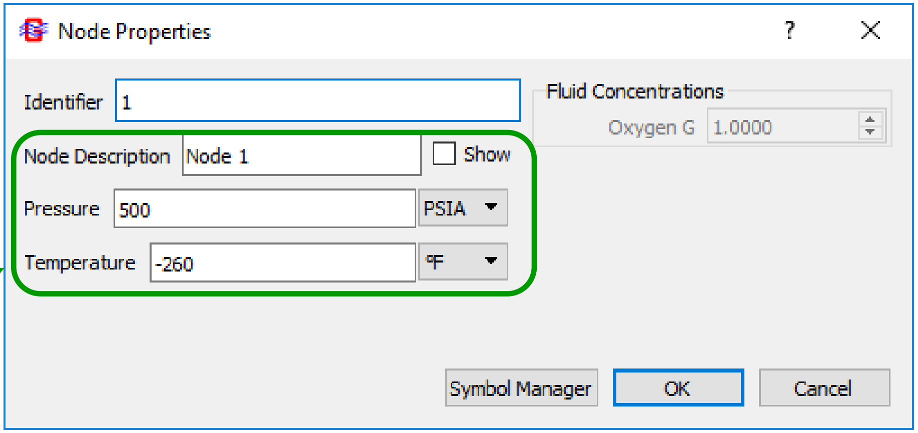
Restart Files

Node Restart Save/Read File: FNODE.DAT

Branch Restart Save/Read File: FBRANCH.DAT

Part 1: Build Steady State Model (2/3)

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



Identifier: 1

Fluid Concentrations: Oxygen G 1.0000

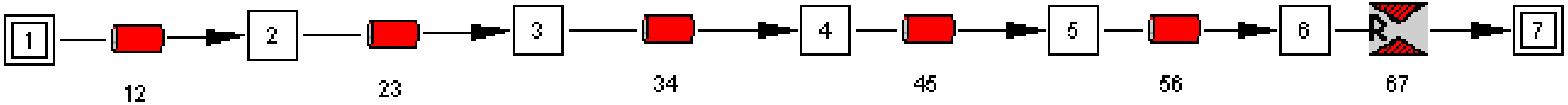
Node Description: Node 1 ☐ Show

Pressure: 500 PSIA

Temperature: -260 °F

LOX
P = 500 psia.
T = -260 °F

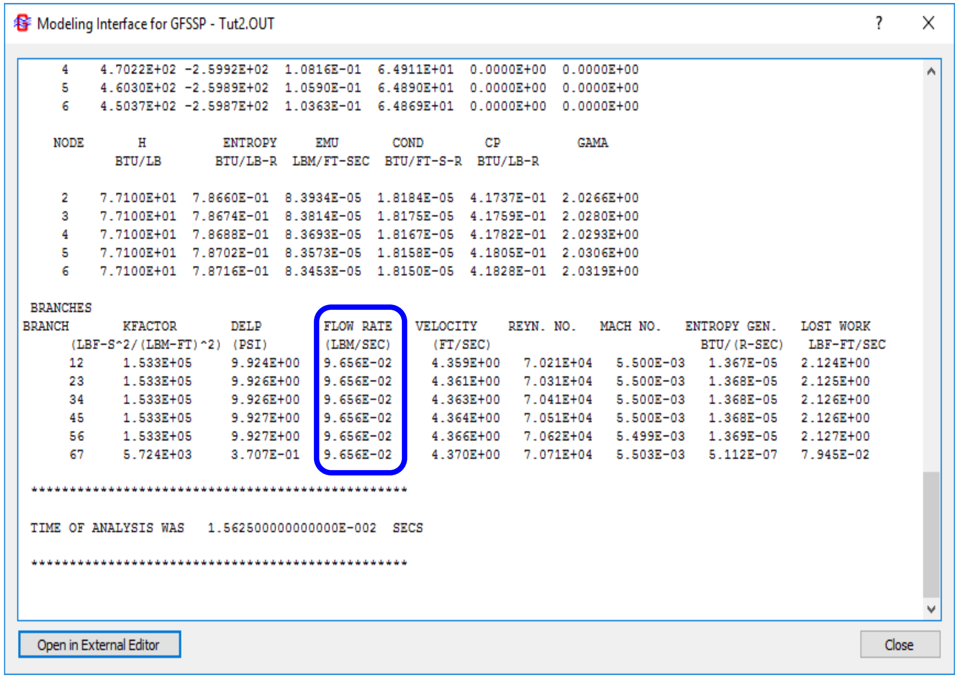
P = 450 psia.
T = -260 °F



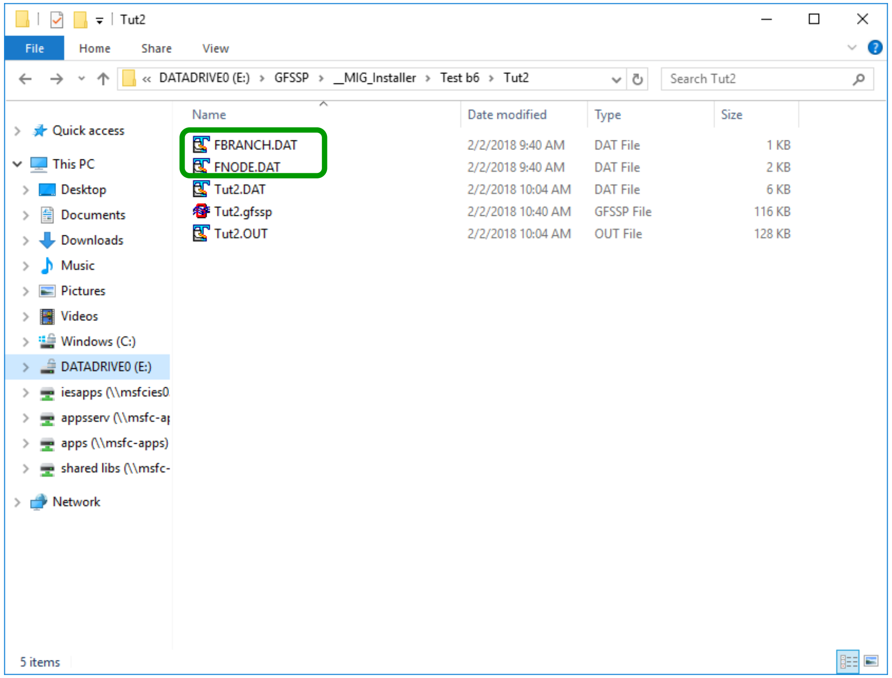
Pipe **L = 960 in.** **D = 0.25 in.**	**Pipe** **L = 960 in.** **D = 0.25 in.**	**Pipe** **L = 960 in.** **D = 0.25 in.**	**Pipe** **L = 960 in.** **D = 0.25 in.**	**Pipe** **L = 960 in.** **D = 0.25 in.**	**Valve** **C_L = 0.6** **A = 0.0491 in^2**

Part 1: Build Steady State Model (3/3)

- 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 files
- Note for later: RHO6 = 64.87 lb_m/ft^3 and V67 = 4.37 ft/s



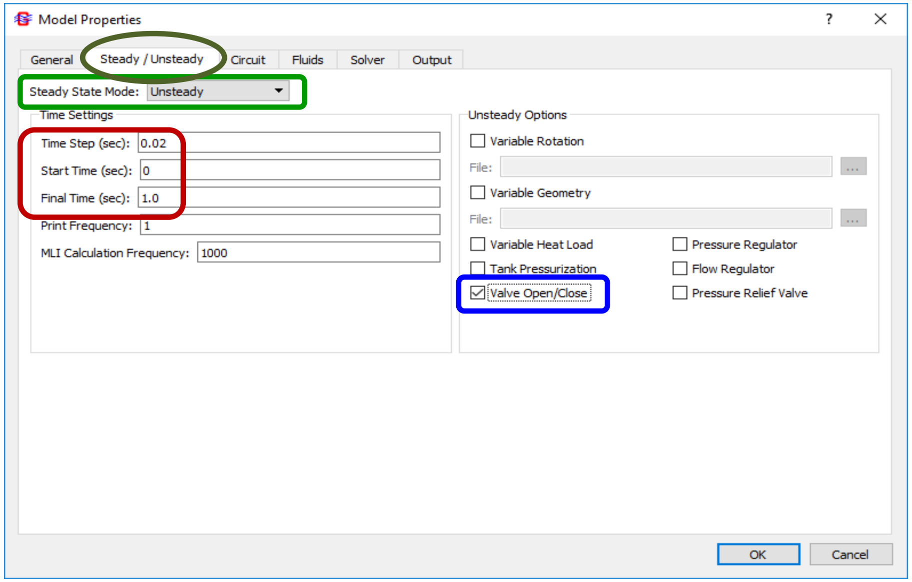
```
     4    4.7022E+02 -2.5992E+02  1.0816E-01  6.4911E+01  0.0000E+00  0.0000E+00
     5    4.6030E+02 -2.5989E+02  1.0590E-01  6.4890E+01  0.0000E+00  0.0000E+00
     6    4.5037E+02 -2.5987E+02  1.0363E-01  6.4869E+01  0.0000E+00  0.0000E+00

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

     2    7.7100E+01  7.8660E-01  8.3934E-05  1.8184E-05  4.1737E-01  2.0266E+00
     3    7.7100E+01  7.8674E-01  8.3814E-05  1.8175E-05  4.1759E-01  2.0280E+00
     4    7.7100E+01  7.8688E-01  8.3693E-05  1.8167E-05  4.1782E-01  2.0293E+00
     5    7.7100E+01  7.8702E-01  8.3573E-05  1.8158E-05  4.1805E-01  2.0306E+00
     6    7.7100E+01  7.8716E-01  8.3453E-05  1.8150E-05  4.1828E-01  2.0319E+00

 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     1.533E+05      9.924E+00   9.656E-02     4.359E+00   7.021E+04   5.500E-03   1.367E-05   2.124E+00
      23     1.533E+05      9.926E+00   9.656E-02     4.361E+00   7.031E+04   5.500E-03   1.368E-05   2.125E+00
      34     1.533E+05      9.926E+00   9.656E-02     4.363E+00   7.041E+04   5.500E-03   1.368E-05   2.126E+00
      45     1.533E+05      9.927E+00   9.656E-02     4.364E+00   7.051E+04   5.500E-03   1.368E-05   2.126E+00
      56     1.533E+05      9.927E+00   9.656E-02     4.366E+00   7.062E+04   5.499E-03   1.369E-05   2.127E+00
      67     5.724E+03      3.707E-01   9.656E-02     4.370E+00   7.071E+04   5.503E-03   5.112E-07   7.945E-02

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

 TIME OF ANALYSIS WAS   1.562500000000000E-002  SECS

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



Name	Date modified	Type	Size
FBRANCH.DAT	2/2/2018 9:40 AM	DAT File	1 KB
FNODE.DAT	2/2/2018 9:40 AM	DAT File	2 KB
Tut2.DAT	2/2/2018 10:04 AM	DAT File	6 KB
Tut2.gfssp	2/2/2018 10:40 AM	GFSSP File	116 KB
Tut2.OUT	2/2/2018 10:04 AM	OUT File	128 KB

Part 2: Build Transient Model (1/4)

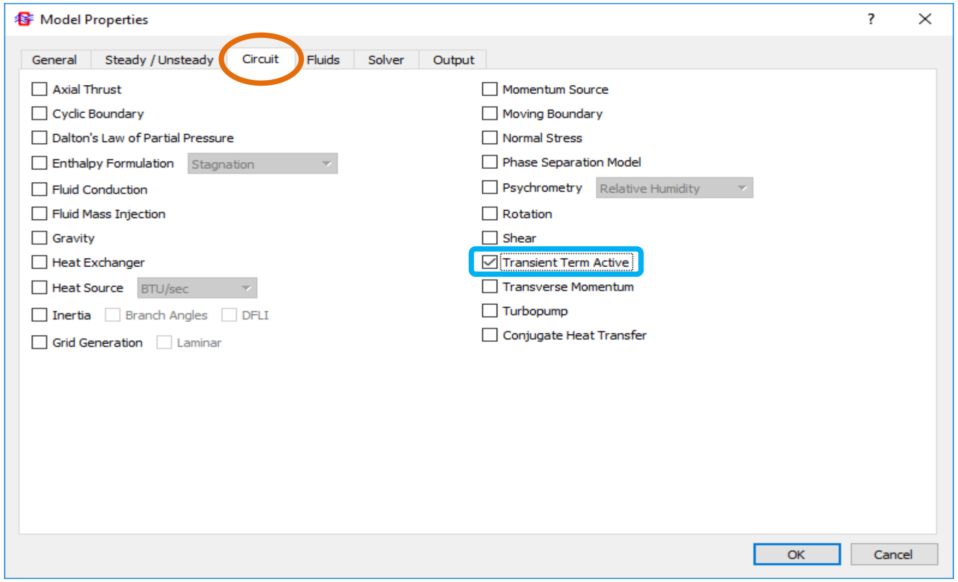
- Model Properties
  - Check: “Unsteady” (Steady/Unsteady tab; under Steady State Mode)
  - Convert model to transient (Steady/Unsteady tab)
    - Time step = 0.02 sec
    - Run time = 1.0 sec
  - Check: “Valve Open/Close” (Steady/Unsteady tab)
  - Check: “Transient Term Active” (Circuit tab)
    - Activates the transient term in the momentum equation
    - Usually negligible **except** in waterhammer problems



Steady State Mode: Unsteady

Time Settings
Time Step (sec): 0.02
Start Time (sec): 0
Final Time (sec): 1.0
Print Frequency: 1
MLI Calculation Frequency: 1000

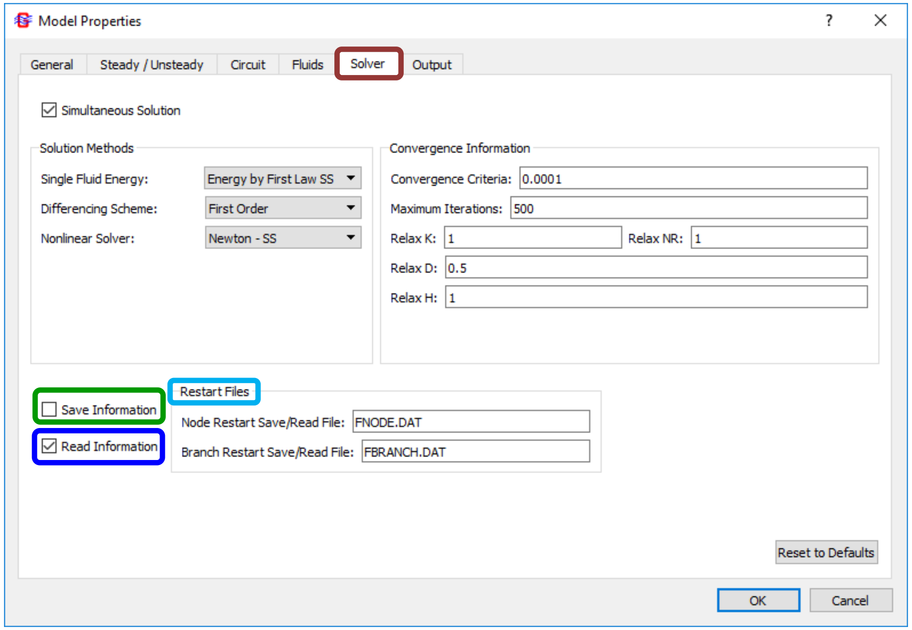
Unsteady Options
☐ Variable Rotation
File:
☐ Variable Geometry
File:
☐ Variable Heat Load
☐ Pressure Regulator
☐ Tank Pressurization
☐ Flow Regulator
☑ Valve Open/Close
☐ Pressure Relief Valve



☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☐ Heat Exchanger
☐ Heat Source BTU/sec
☐ Inertia ☐ Branch Angles ☐ DFLI
☐ Grid Generation ☐ Laminar
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry Relative Humidity
☐ Rotation
☐ Shear
☑ Transient Term Active
☐ Transverse Momentum
☐ Turbopump
☐ Conjugate Heat Transfer

Part 2: Build Transient Model (2/4)

- Model Properties
  - Uncheck: "Save Information" (Solver tab, under Restart Files)
  - Check: "Read Information" (Solver tab, under Restart Files)
  - Check: "Winplot Data" (Output tab)



☑ Simultaneous Solution

Solution Methods

Single Fluid Energy: Energy by First Law SS

Differencing Scheme: First Order

Nonlinear Solver: Newton - SS

Convergence Information

Convergence Criteria: 0.0001

Maximum Iterations: 500

Relax K: 1

Relax NR: 1

Relax D: 0.5

Relax H: 1

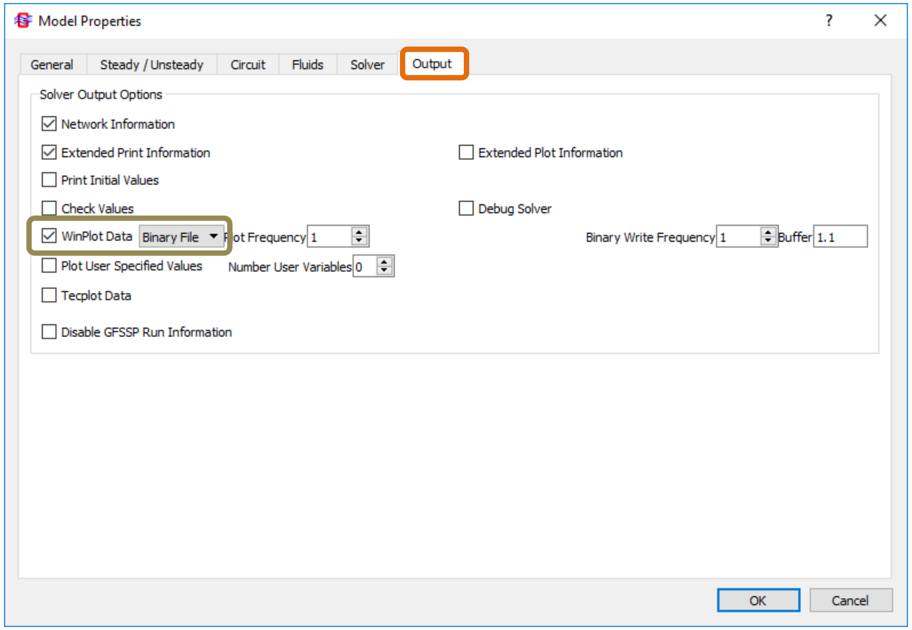
☐ Save Information

☑ Read Information

Restart Files

Node Restart Save/Read File: FNODE.DAT

Branch Restart Save/Read File: FBRANCH.DAT



Solver Output Options

☑ Network Information

☑ Extended Print Information

☐ Extended Plot Information

☐ Print Initial Values

☐ Check Values

☐ Debug Solver

☑ WinPlot Data Binary File

Plot Frequency 1

Binary Write Frequency 1

Buffer 1.1

☐ Plot User Specified Values

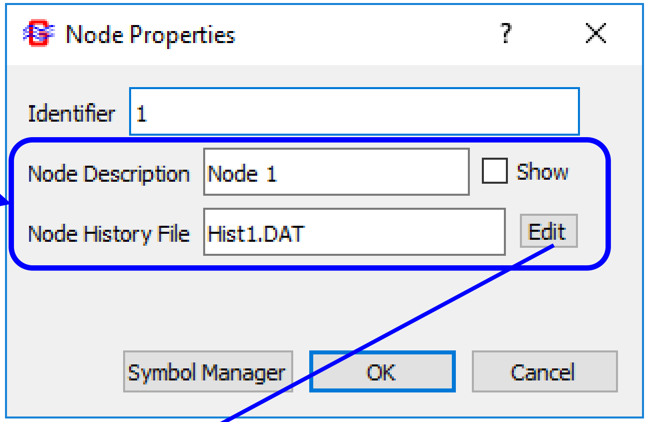
Number User Variables 0

☐ Tecplot Data

☐ Disable GFSSP Run Information

Part 2: Build Transient Model (3/4)

- Create history files
  - Boundary Node 1
    - P = 500 psia
    - T = -260 °F
  - Node 7
    - P = 450 psia
    - T = -260 °F

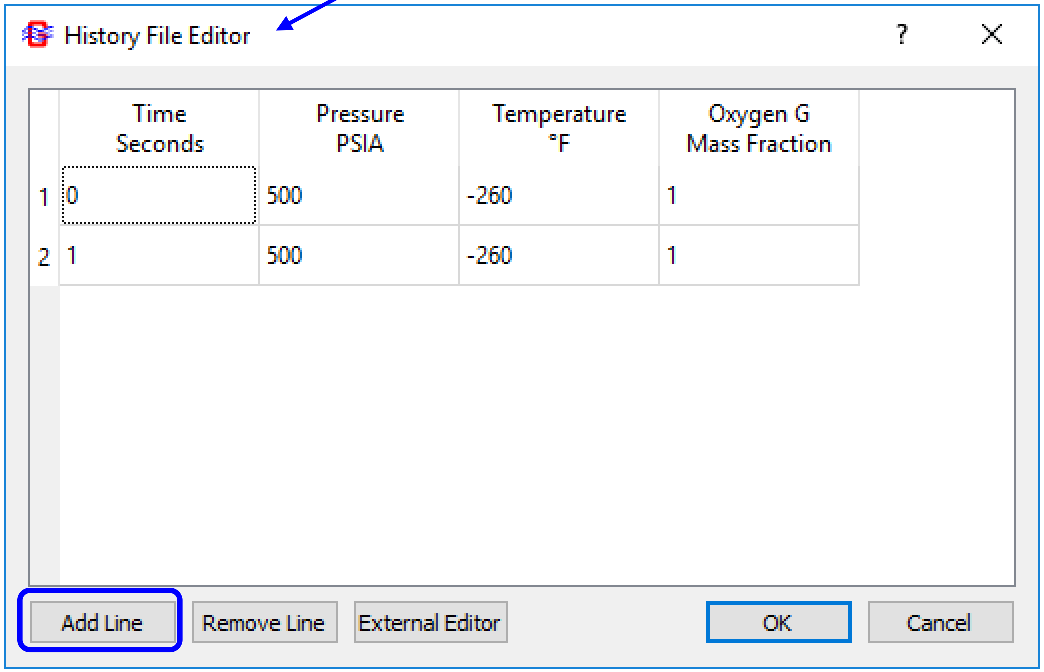


Node Properties

Identifier: 1

Node Description: Node 1 ☐ Show

Node History File: Hist1.DAT



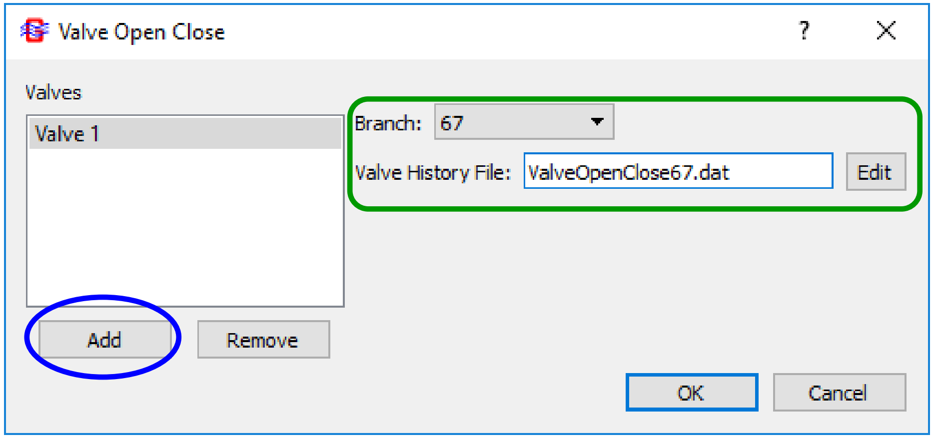
History File Editor

	Time Seconds	Pressure PSIA	Temperature °F	Oxygen G Mass Fraction
1	0	500	-260	1
2	1	500	-260	1

Part 2: Build Transient Model (4/4)

- Open the "Valve Open Close" dialog box from the Advanced menu
  - Click Add; Input name of the Valve History File in the dialog box: click Edit
  - Input Valve Closure History values (time, area)
    - Represents valve closing

- **GFSSP** will interpolate Branch 67 area for each time step



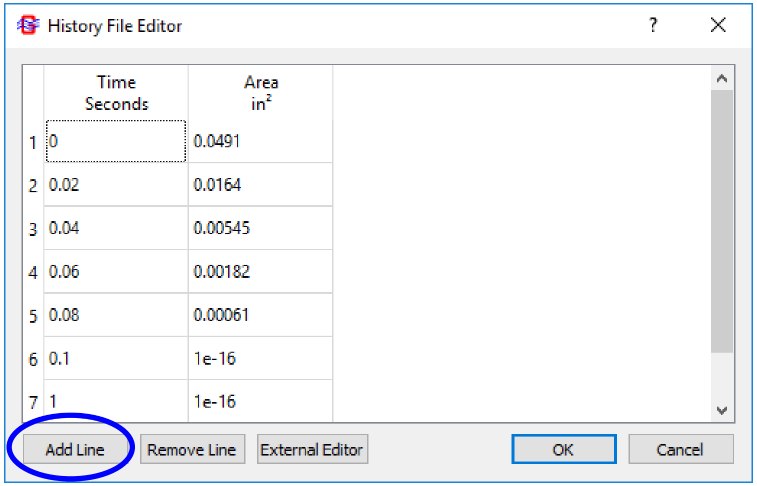
Valves

Valve 1

Branch: 67

Valve History File: ValveOpenClose67.dat

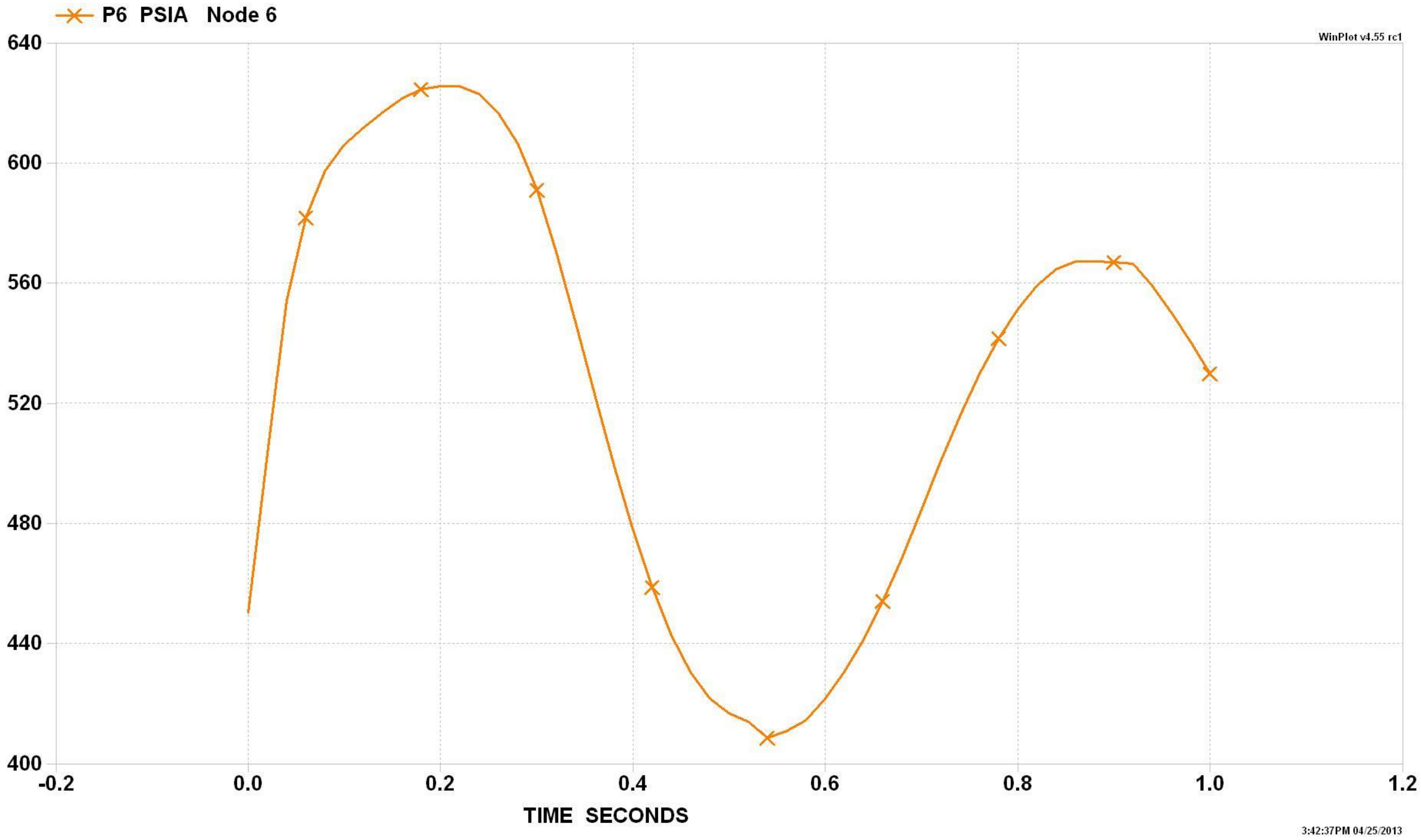
Valve Closure History



	Time Seconds	Area in^2
1	0	0.0491
2	0.02	0.0164
3	0.04	0.00545
4	0.06	0.00182
5	0.08	0.00061
6	0.1	1e-16
7	1	1e-16

Note: To prevent division by zero, a ***closed valve*** is given a very small area, such as 1×10^{-16}

Pressure History at Valve



Study of the Results

- Plot pressure and flowrate history
  - Peak pressure approximately 620 psia

- Estimate the predicted period of oscillation and compare with formula
  - Period of Oscillation = 4L/*a*
    - L = length of the pipe = 400 ft
    - *a* = Speed of sound = 2462 ft/sec (for LOX)

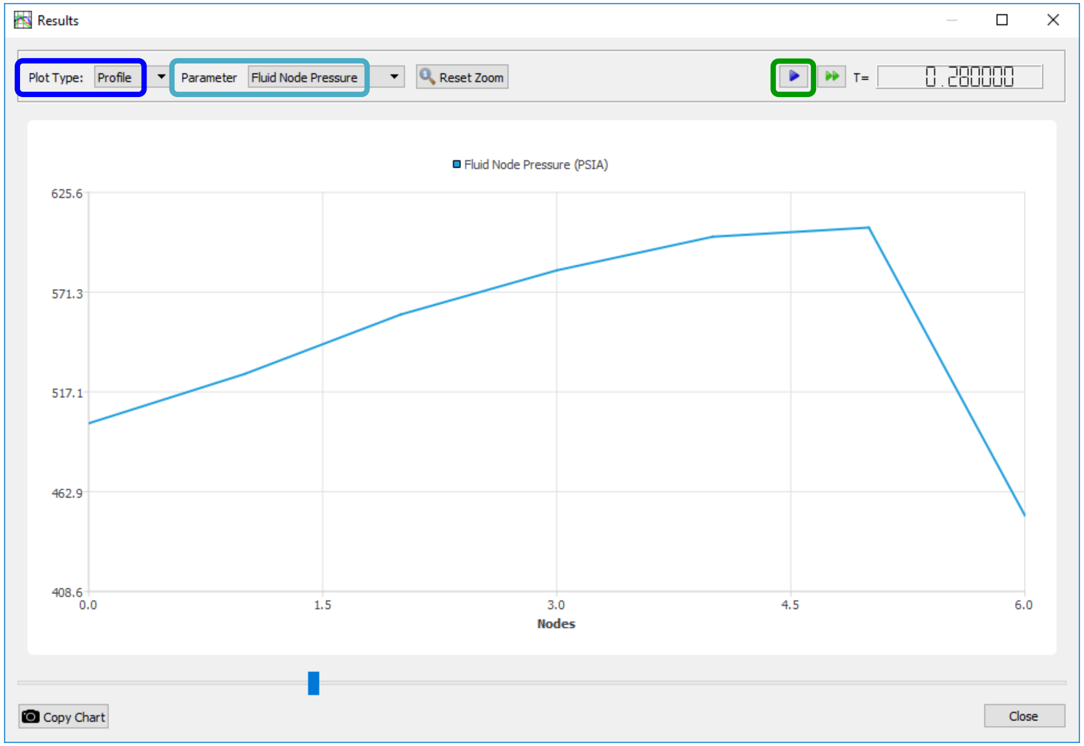
- Estimate worst case pressure rise for an instantaneous valve closure:
  - $\Delta P = \frac{\rho a V_{steady}}{g_c}$
    - g_c = 32.174 lb_m-ft/(s^2-lb_f)
    - Remember to convert from psf to psi (divide by 144)
    - Compare this value to the predicted pressure rise relative to the 500 psi supply pressure

- Plot density (RHO6) and compressibility (Z6) history
  - Note variation of compressibility with time

Animate the Pressures

- Open the **MIG** plotter by clicking Model / Plot Results
  - Change: Plot Type from Temporal to Profile
  - Change: Parameter to Fluid Node Pressure
  - Press: Play button to animate the Pressure vs. Node plots over time

- Animations are most useful for single row, evenly-spaced node models



Plot Type: Profile

Parameter Fluid Node Pressure

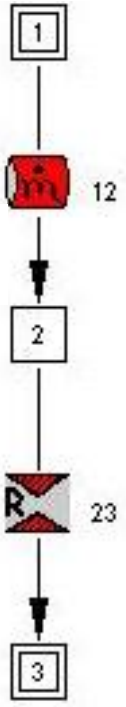
T= 0.280000

Challenge Problem 2

Draining a Water Heater

Given:

An insulated, electrically heated water heater contains 190 kg of liquid water at 60°C when a power outage occurs. If water is withdrawn from the heater tank at a constant rate of 0.2 kg/s, how long will it take for the temperature of the water in the tank to drop from 60°C to 35°C? Assume that cold water enters the tank at 10°C, and that the tank is well insulated.



Cold Water Source

T = 10°C

Flowrate = 0.2 kg/s

Area = 5 cm^2

Water Heater

$T_{initial}$ = 60°C

Volume = 0.193 m^3

Exit restriction

Area = 5 cm^2

C_L = 1.0

Challenge Problem 2

Draining a Water Heater

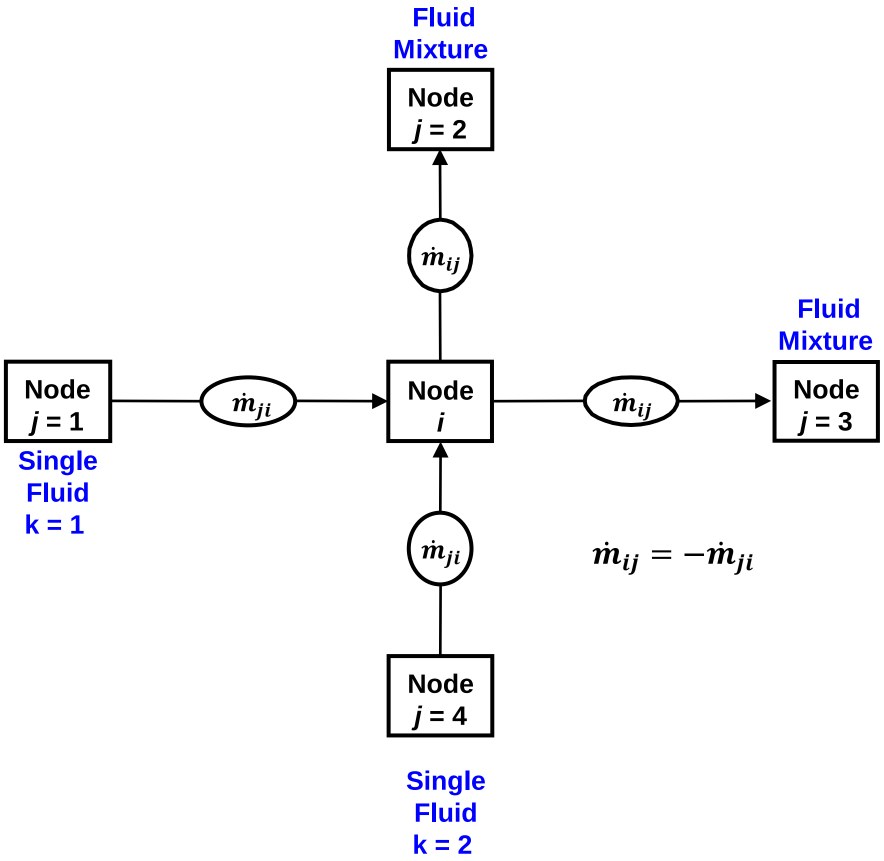
- Two ways to set up/run **GFSSP** using SI units
  - Click: File / Preferences, then “Default SI”
    - Close and restart MIG
  - Click: General tab of the Model Properties page
    - Select SI for History Files and Output
    - Manually input each entry in SI units into the dialog boxes

- Fixed Flow Branch Option
  - Can be used to specify the flow rate in this simple system
  - Assume reasonable pressure value at the boundaries (e.g. 101.3 kPa)
  - Acts as a pump
    - Raises the driving pressure enough to maintain the specified flow rate of 0.2 kg/s

- This problem is Example 7.3 of Introduction to Chemical Engineering Thermodynamics, 5th ed. by Smith et al.
  - Given answer: 658.5 seconds (assuming constant properties)
  - How does **GFSSP**’s answer compare?

Mathematical Formulation

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



Fluid Mixture

Node
***j* = 2**

$\dot{m}_{ij}$

Fluid Mixture

Node
***j* = 1**

Single Fluid
k = 1

$\dot{m}_{ji}$

Node
i

$\dot{m}_{ij}$

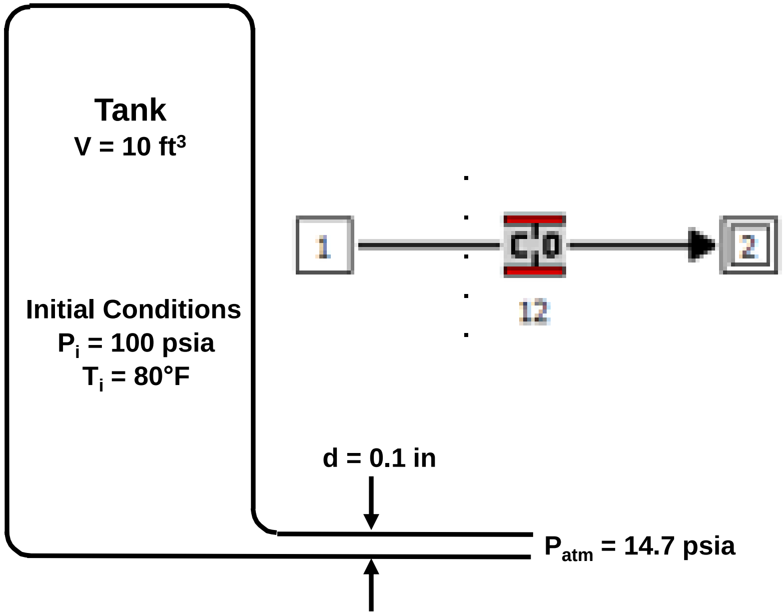
Node
***j* = 3**

$\dot{m}_{ji}$

$$\dot{m}_{ij} = -\dot{m}_{ji}$$

Node
***j* = 4**

Single Fluid
k = 2



Tank
V = 10 ft^3

Initial Conditions
P_i = 100 psia
T_i = 80°F

d = 0.1 in

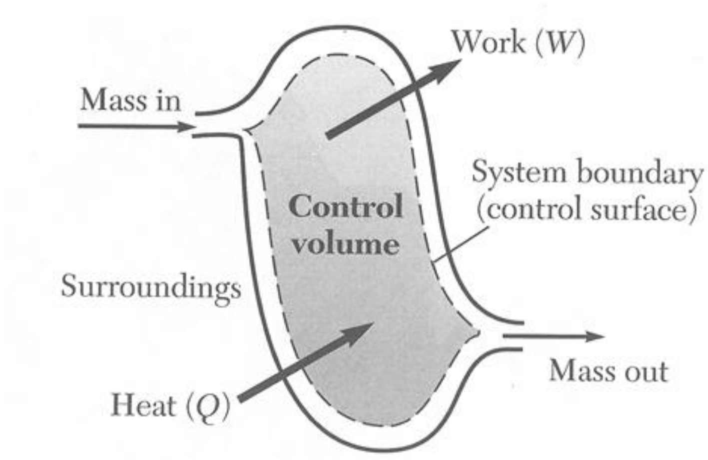
P_{atm} = 14.7 psia

Content

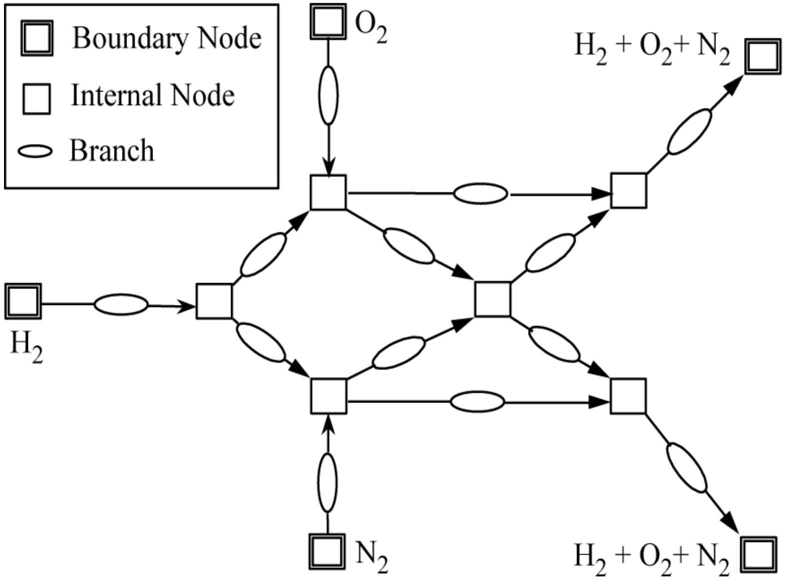
- Finite Volume Procedure Basics
- Mathematical Closure
- Governing Equations
- Solution Procedure

Finite Volume Procedure Basics (1/2)

- The Finite Volume Procedure for a fluid network is an extension of single control volume analysis of mass and energy conservation in classical thermodynamics.



Control Volume Analysis
in Classical Thermodynamics



Finite Volume Analysis
in Fluid Network

Finite Volume Procedure Basics (2/2)

- Development of governing equations
  - Conservation of mass, momentum, and energy of working fluid
  - Conservation of thermal energy of solid in contact with working fluid

- Use of accurate thermodynamic and thermo-physical properties of fluid and material properties in development of the governing equations

- Numerical Solution of the governing equations by an iterative method

Mathematical Closure (1/5)

Problem of a Steady State Flow Network

Given: Boundary Node Pressures and Temperatures

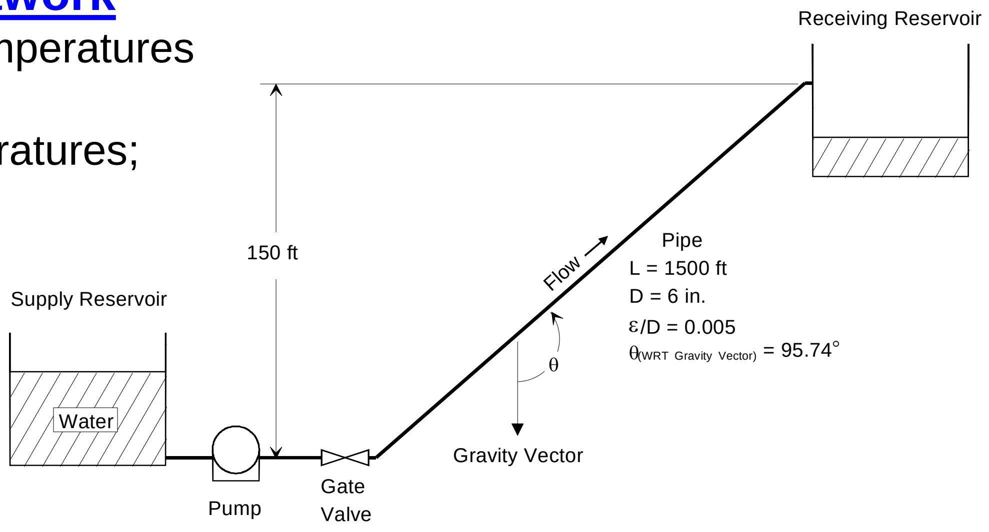
Find: Internal Node Pressures and Temperatures;
Flowrates in Branches

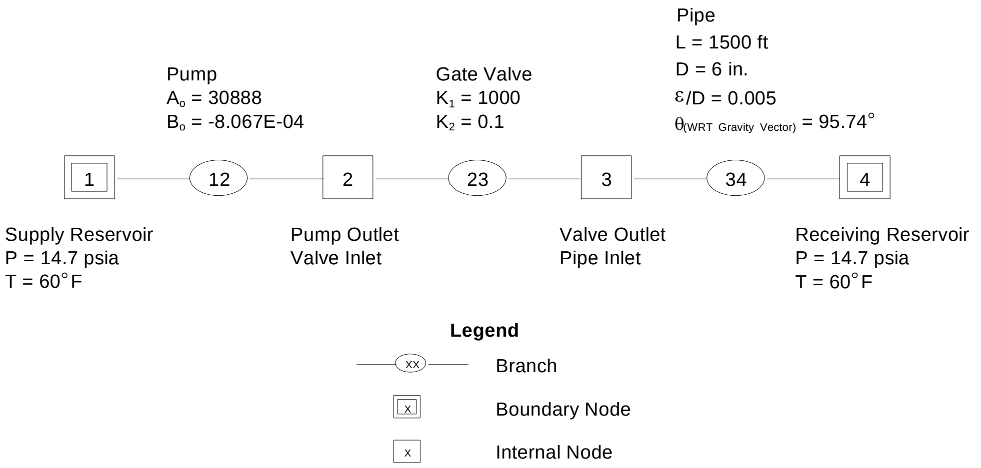
Primary Variables

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

Secondary Variables

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



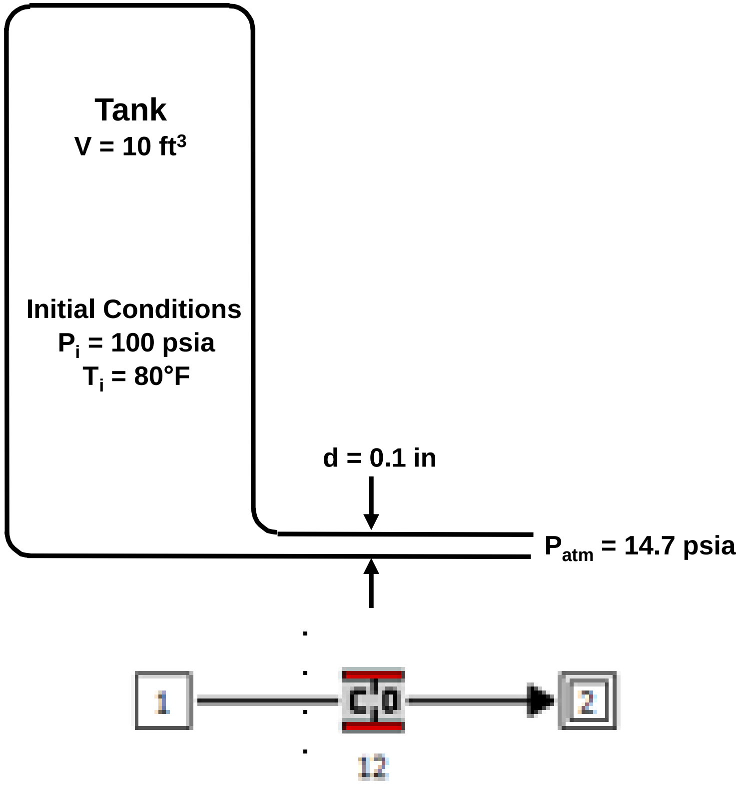


Mathematical Closure (2/5)

Problem of an Unsteady Flow Network

Given: Boundary Node Pressures and Temperatures; Initial Values at Internal Nodes

Find: Internal Node Pressures and Temperatures; Flowrates in Branches with Time



Primary Variables

$$p_1(\tau), T_1(\tau), m_1(\tau),\ \dot{m}_{12}(\tau)$$

Secondary Variables

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

Mathematical Closure (3/5)

Problem of an Unsteady Flow with Conjugate Heat Transfer

Given: Boundary Node Pressures and Temperatures; Initial Values at Internal Fluid Nodes and Solid Nodes

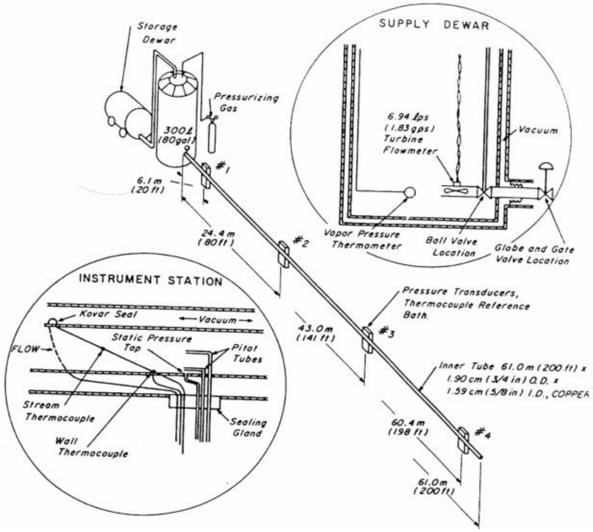
Find: Internal Node Pressures and Temperatures; Flowrates in Branches with Time

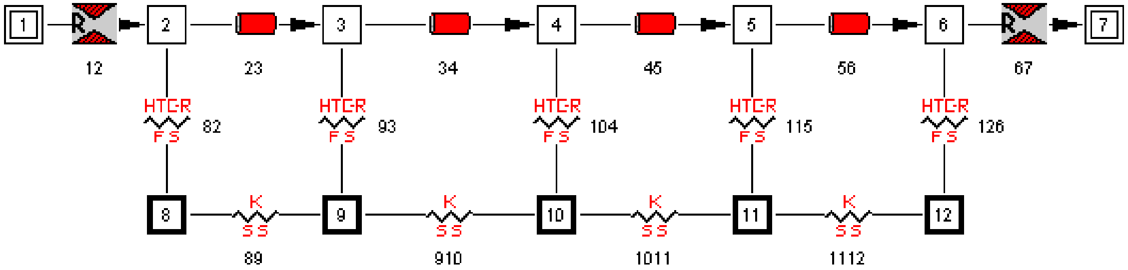
Primary Variables

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

Secondary Variables

$$\rho(\tau), \mu(\tau), k(\tau), h_c(\tau)$$





Mathematical Closure (4/5)

- Primary 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. Species Concentration	5. Conservation Equations for Mass Fraction of Species
6. Mass	6. Thermodynamic Equation of State

Mathematical Closure (5/5)

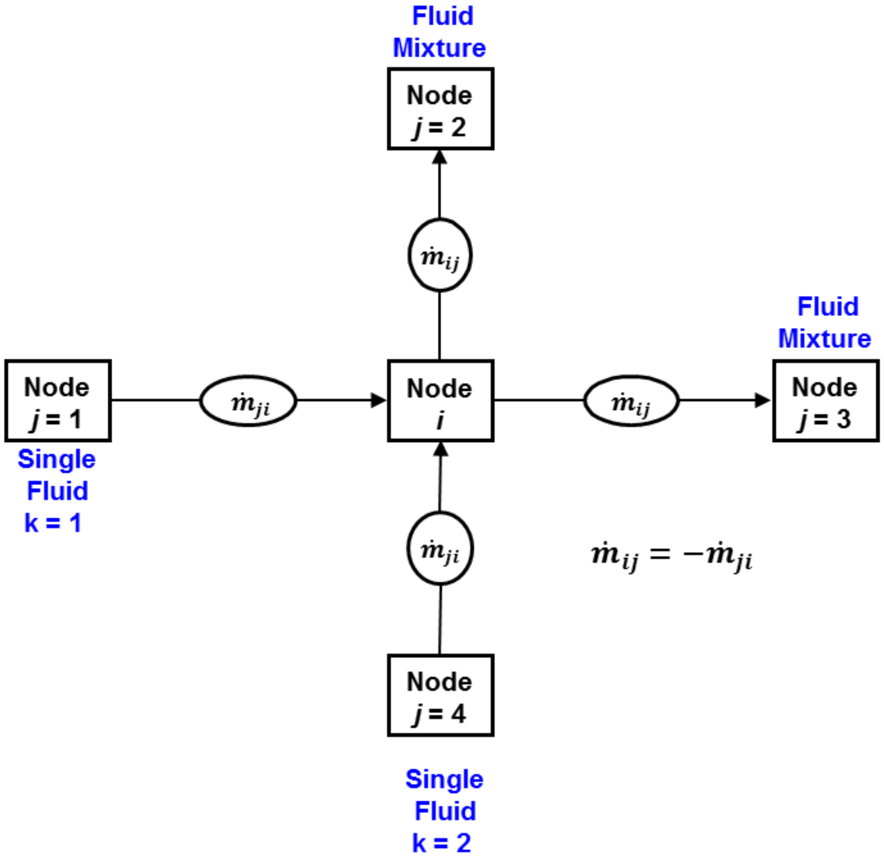
- Secondary Variables
  - Thermodynamic & Thermophysical Properties

Unknown Variables	Available Equations to Solve
Density (ρ)	Equilibrium Thermodynamic Relations [GASP/WASP & GASPAK]
Specific Heat (C_p)	
Viscosity (ν)	
Thermal Conductivity (k)	
Friction Factor (f)	Empirical Relations
Heat Transfer Coefficients (h_c)	

Mass Conservation Equation

- Mass Conservation Equation (for node *i*)

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



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

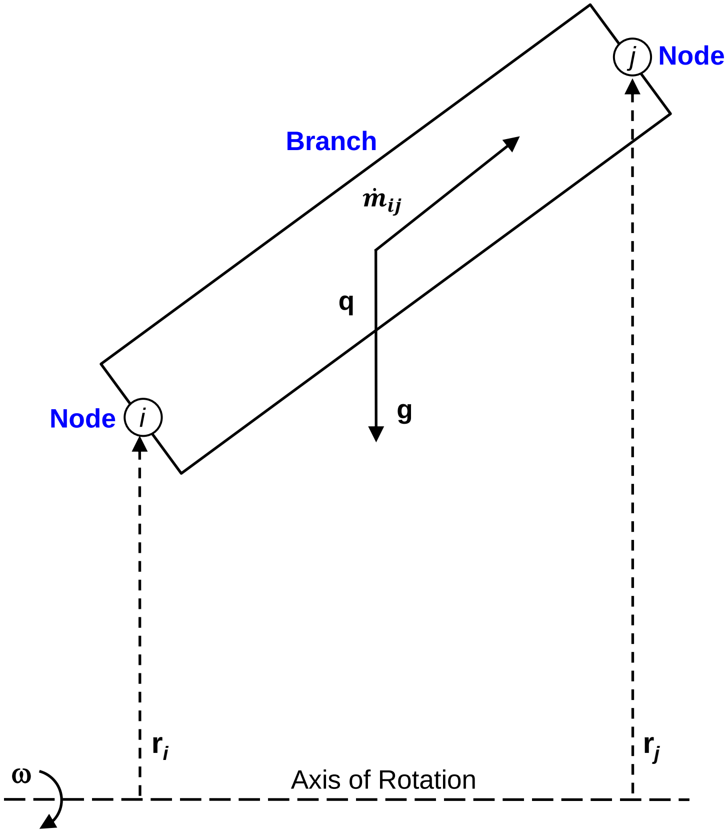
Momentum Conservation Equation (1/4)

- Momentum Conservation Equation
  - Represents Newton's Second Law of Motion

mass * acceleration **=** ***forces***

- Unsteady
- Longitudinal Inertia
- Transverse Inertia

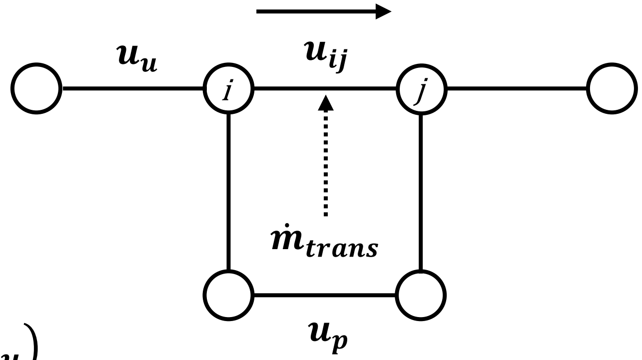
- Pressure
- Gravity
- Friction
- Centrifugal
- Shear Stress
- Moving Boundary
- Normal Stress
- External Force



Momentum Conservation Equation (2/4)

- 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

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

Momentum Conservation Equation (3/4)

- Force Terms in **GFSSP** (1/2)
  - Pressure

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

  - Gravity

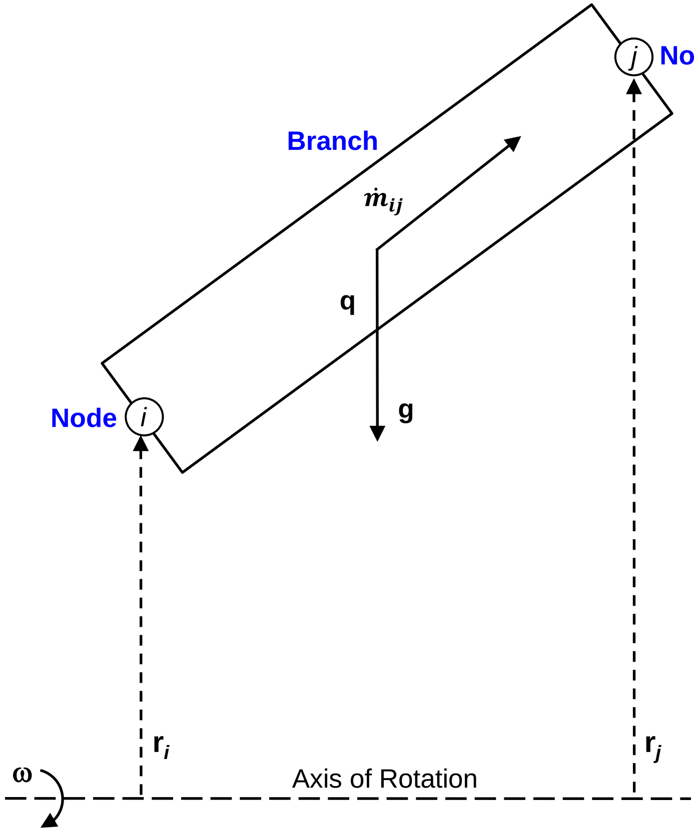
$$\frac{\rho g V \cos\Theta}{g_c}$$

  - Friction

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

  - Centrifugal

$$\frac{\rho\, K_{rot}^2 \omega^2 A\left(r_j^2 - r_i^2\right)}{g_c}$$



Momentum Conservation Equation (4/4)

- Force Terms in **GFSSP** (2/2)
  - Shear Stress

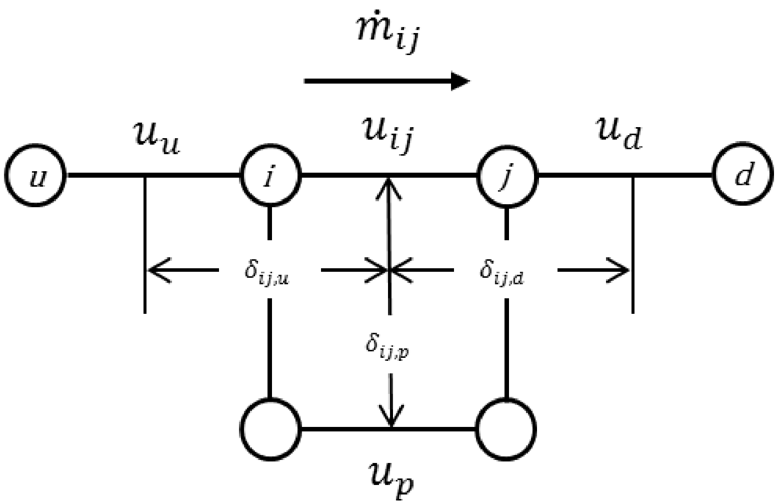
$$\mu \frac{u_p - u_{ij}}{g_c \delta_{ij,p}} A_s$$

  - Normal Stress

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

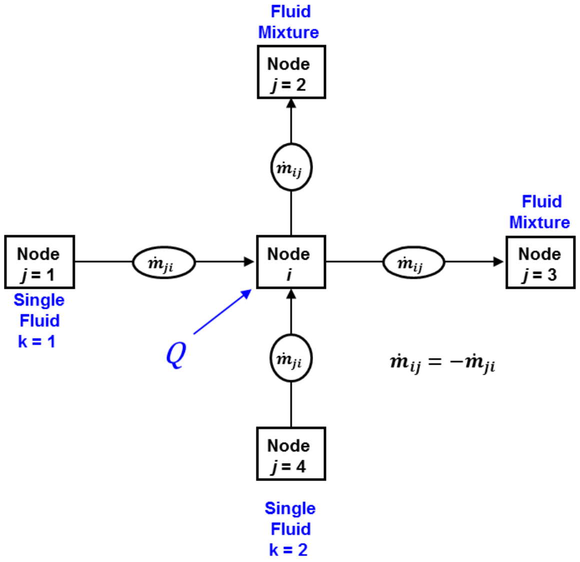
  - Moving Boundary

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



Governing Equations (1/8)

- Energy Conservation Equation (1/2)
  - Can be written in terms of Enthalpy (h) or Entropy (s)
  - Based on Upwind Scheme



- Enthalpy Equation

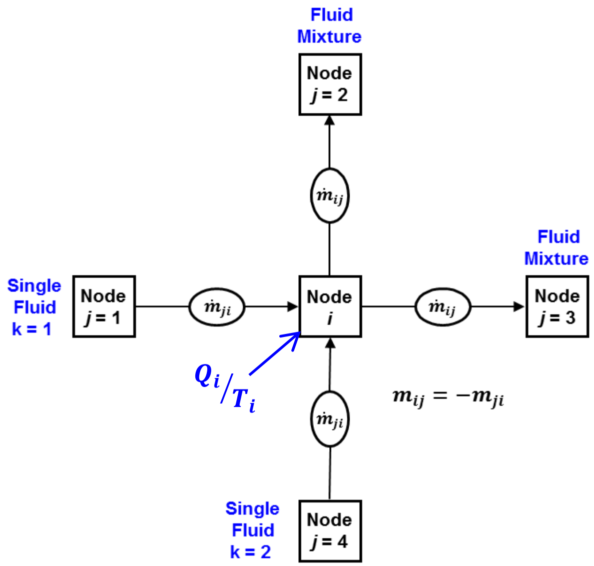
Rate of Increase of Internal Energy = Enthalpy Inflow - Enthalpy Outflow + Heat Source

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

Note: J = 778.17 ft-lb$_f$/Btu

Governing Equations (2/8)

- Energy Conservation Equation (2/2)



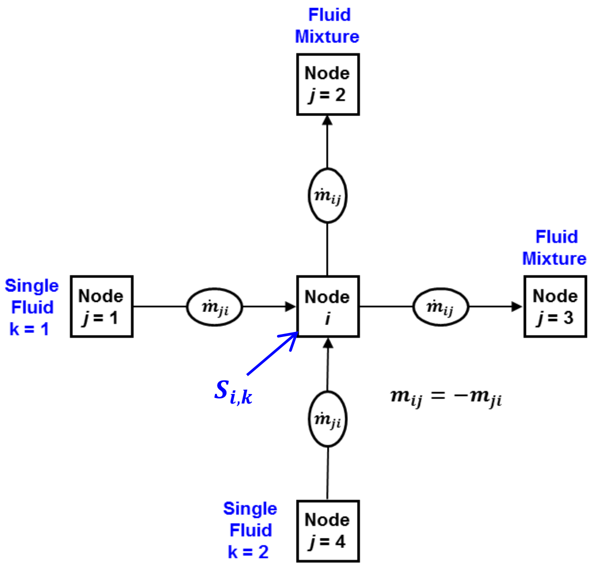
- Entropy Equation

Rate of Increase of Entropy = Entropy Inflow - Entropy Outflow + Entropy Generation + Entropy Source

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

Governing Equations (3/8)

- Fluid Species Conservation Equation



- Fluid Species Equation

Rate of Increase of Fluid Species = Fluid Species Inflow – Fluid Species Outflow + Fluid Species Source

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

Governing Equations (4/8)

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

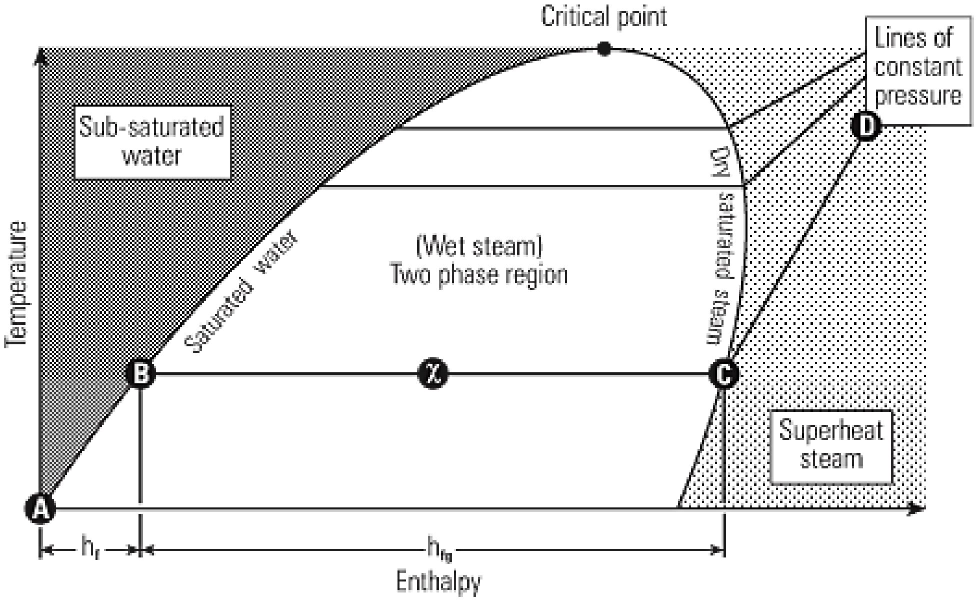
$$m = \frac{pV}{RTz}$$

  - z is the compressibility factor determined from higher order equation of state to give density (from property packages or interpolated from tables)

Governing Equations (5/8)

- Liquid-Vapor Mixtures
  - If the enthalpy at a given pressure is under the saturation dome
    - Fluid is considered to be a homogeneous liquid-vapor mixture with a quality
    - Qualilty is the vapor mass fraction

  - Saturated fluid properties will be quality-weighted averages of the liquid and vapor properties



Governing Equations (6/8)

- Mixture Property Relations (1/4)
  - Mixture Density
    - Amagat’s Law of Partial Volumes
    - GFSSP’s default
    - Suitable for liquids and most gas mixtures
    - Uses density evaluated by property package at the node pressure

$$\frac{1}{\rho_{mix}} = \sum \frac{x_k}{\rho_k}$$

Governing Equations (7/8)

- Mixture Property Relations (2/4)
  - Mixture Density
    - Dalton's Law of Partial Volumes
    - Activated on Circuit Options tab
    - Properties are evaluated at the partial pressure of the gas
    - Appropriate for gas mixtures where at least one gas would be a liquid if properties were evaluated at the total pressure of the mixture.

$$\rho_{mix} = \sum \rho_k$$

Governing Equations (8/8)

- Mixture Property Relations (3/4)
  - Example: When Dalton's Law of Partial Pressures would be appropriate
    - Mixture: 76 mol% O_2 and 24 mol% He, at 45 psia, -277.7 °F
    - At a total pressure of 45 psia, O_2 is a liquid
    - At a partial pressure of 34.2 psia, O_2 is a gas

Fluid	Amagat's Law ρ (lb_m/ft^3)	Dalton's Law ρ (lb_m/ft^3)
O_2	57.73	0.5946
He	0.0918	0.0221
Mixture	2.336 (Incorrect!)	0.6167

Solution Procedure (1/10)

- Successive Substitution (SS)
- Newton-Raphson (N-R)
- **S**imultaneous **A**djustment with **S**uccessive **S**ubstitution (SASS)
- Program Sequence
- Convergence
- Sparse Matrix Solver
- Time Step
- Relaxation Parameters
- Troubleshooting

Solution Procedure (2/10)

- Non-linear algebraic equations solution options
  - Successive Substitution (SS)
  - Newton-Raphson (N-R)

- **GFSSP** uses a Hybrid Method
  - SASS (**S**imultaneous **A**djustment with **S**uccessive **S**ubstitution)
  - Method is a combination of Successive Substitution and Newton-Raphson

Solution Procedure (3/10)

- Successive Substitution (SS) 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 – 4 until solution convergence

  - Advantages
    - ➢ Simple to program
    - ➢ Takes less computer memory

  - Disadvantages
    - ➢ Difficult to decide in what order to solve the equations to ensure convergence

Solution Procedure (4/10)

- Newton-Raphson (N-R) 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 the correction equations by Gaussian Elimination method
    5. Apply correction to each variable
    6. Iterate until corrections become very small

  - Advantages
    - No decision-making process involved to determine order in which equations must be solved

  - Disadvantages
    - Requires more computer memory
    - Difficult to program

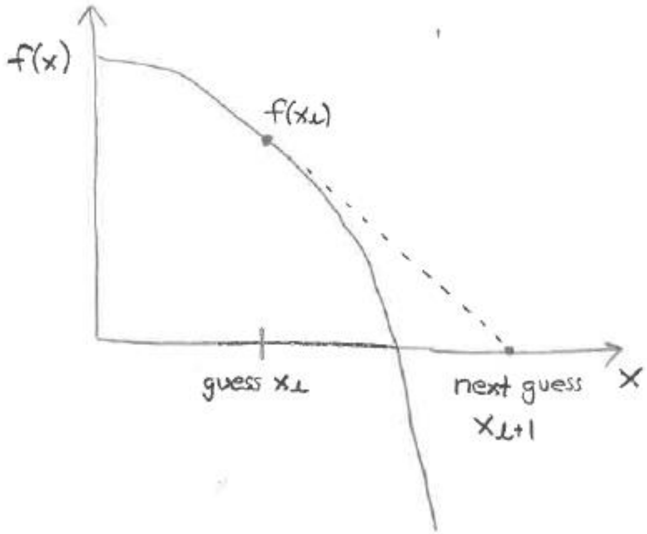
Solution Procedure (5/10)

- Using Newton’s method to find the root (zero) of a single equation.
  - Guess x_i
  - Calculate $f(x_i)$ and its derivative $f'(x_i)$
  - The next guess x_{i+1} is:

$$x_{i+1} = x_i - \frac{f(x_i)}{f'(x_i)}$$

  - Therefore, the correction Δx_{corr} is:

$$\Delta x_{corr} = x_{i+1} - x_i = -\frac{f(x_i)}{f'(x_i)}$$



$$f'(x_i) = \frac{0 - f(x_i)}{x_{i+1} - x_i}$$

$$\Downarrow$$

$$x_{i+1} = x_i - \frac{f(x_i)}{f'(x_i)}$$

Solution Procedure **(6/10)**

- Using Newton-Raphson method to find the roots of <u>multiple</u> equations.
  - Arrange conservation equations so that all terms are on one side.
    - These terms add up to a residual R.
    - Want to drive $R_1(x_1, x_2, x_3\ldots)$, $R_2(x_1, x_2, x_3\ldots)$,… to value of zero.
    - R can be conservation of mass, momentum, or equation of state.
  - Guess x_1, x_2, x_3,...
    - x_i could be a pressure, flow rate, or resident mass
  - Calculate current values of residuals R_1, R_2, R_3, … and place in vector R:

$$R = \begin{bmatrix} R_1 \\ R_2 \\ \ldots \\ R_N \end{bmatrix}$$

Solution Procedure (7/10)

- Using Newton-Raphson method to find the roots of multiple equations (cont.).
  - Use numerical differencing to calculate current values of partial derivatives in Jacobian matrix J.

$$J = \begin{bmatrix} \frac{\delta R_1}{\delta x_1} & \frac{\delta R_1}{\delta x_2} & \dots \\ \frac{\delta R_2}{\delta x_1} & \frac{\delta R_2}{\delta x_2} & \dots \\ \dots & \dots & \frac{\delta R_N}{\delta x_N} \end{bmatrix}$$

  - Invert the Jacobian matrix J, multiply by the vector of residuals R, and calculate a vector of corrections:

$$\Delta x_{corr} = \begin{bmatrix} \Delta x_1 \\ \Delta x_2 \\ \dots \\ \Delta x_N \end{bmatrix} = -J^{-1} R$$

Note similarity to single variable form:

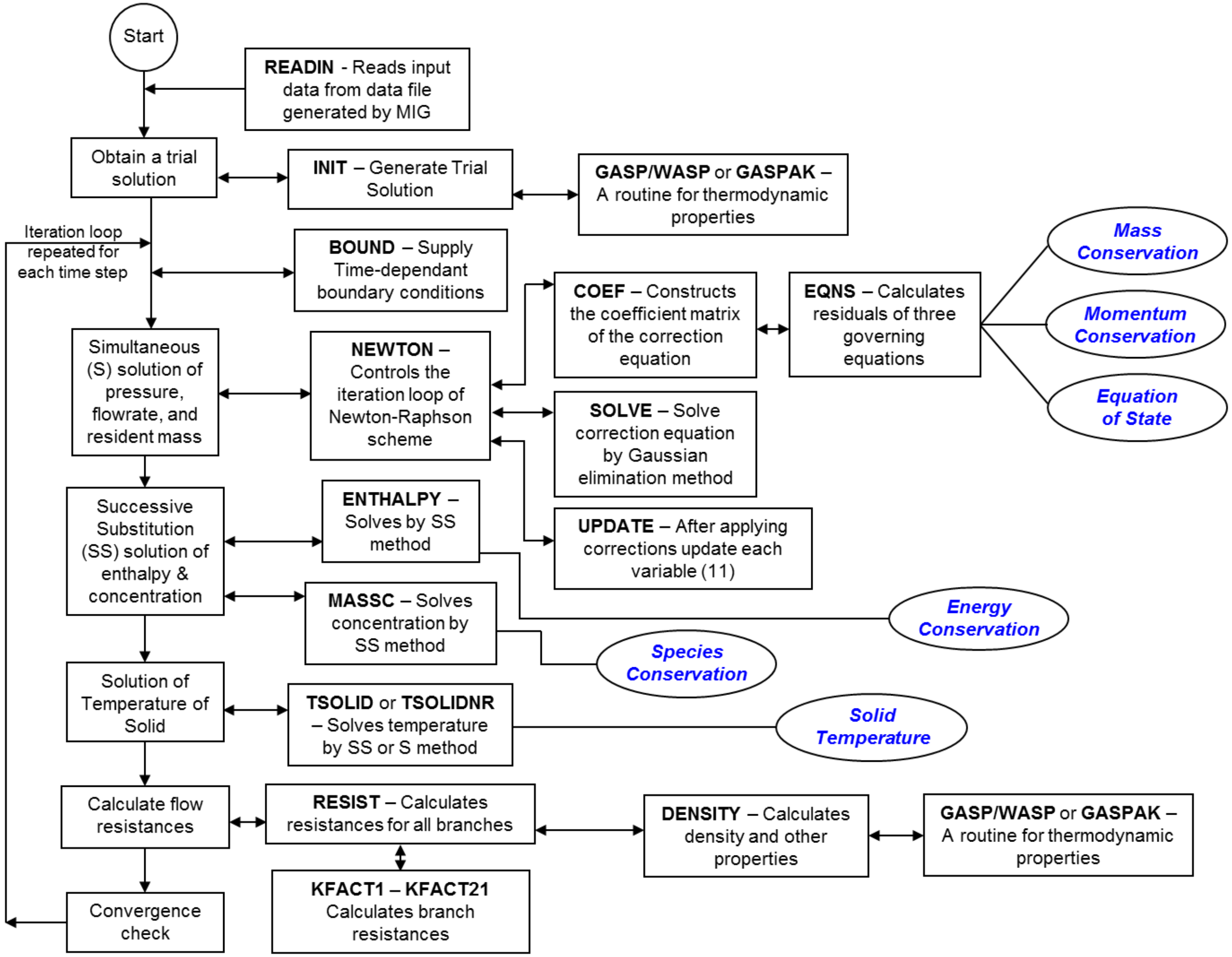
$$\Delta x_{corr} = -\frac{f(x_i)}{f'(x_i)}$$

Solution Procedure (8/10)

- SASS (**S**imultaneous **A**djustment with **S**uccessive **S**ubstitution) Scheme
  - Combination of Newton-Raphson (N-R) and Successive Substitution (SS) methods
    - NR method solves *mass conservation, momentum, and equation of state*
    - SS method solves *energy conservation and concentration equations*
  - Underlying principle for making such division
    - Equations which have strong influence on other equations are solved by the NR method
    - Equations which have less influence on other equations are solved by SS method
- SASS Advantages
  - Approach reduces code overhead
  - Maintains superior convergence characteristics

Solution Procedure (9/10)

- Flow Chart of Solution Algorithm



Solution Procedure (10/10)

- Solution of the governing equations involves following steps

1. Subdivide the flow domain into fluid nodes and branches
2. Subdivide the solid domain into solid nodes and conductors
3. Connect the solid and fluid nodes with solid to fluid conductors
4. Solve at each fluid node
    a. Mass and Energy Conservation equations to calculate Pressure (p) and Enthalpy (h)
    b. Equation of state to compute resident mass (m_R)
5. At each fluid branch, solve Momentum Conservation equations for calculate flow rate ($\dot{m}$)
6. From Pressure and Enthalpy, calculate fluid Temperature (T_F) and all other thermodynamic and thermophysical properties required in governing equations
7. At each solid node, solve Energy Conservation equation to calculate solid Temperature (T_S)
8. Repeat Steps 4 – 7 until convergence
9. Repeat Steps 4 – 8 for each time step

Convergence (1/5)

- Numerical solution can only be trusted when fully converged

- **GFSSP**'s convergence criterion
  - Based on difference in variable values between successive iterations (DIFMAX)
  - Normalized Residual Error is also monitored (RSDMAX)

- **GFSSP**'s solution scheme
  - Two options to control the iteration process
    - Simultaneous (SIMUL = TRUE)
    - Non-Simultaneous (SIMUL = FALSE)

Convergence (2/5)

- Simultaneous Option
  - Single Iteration Loop
    - Perform one iteration of the Mass, Momentum, and Equation of State by N-R scheme
    - Solve Energy and Species Conservation equations by SS Scheme
    - Solution is converged when the *normalized maximum correction*, Δ_{max} (DIFMAX), is less than the convergence criterion

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

where: N_E is the total number of equations solved by the NR scheme

Convergence (3/5)

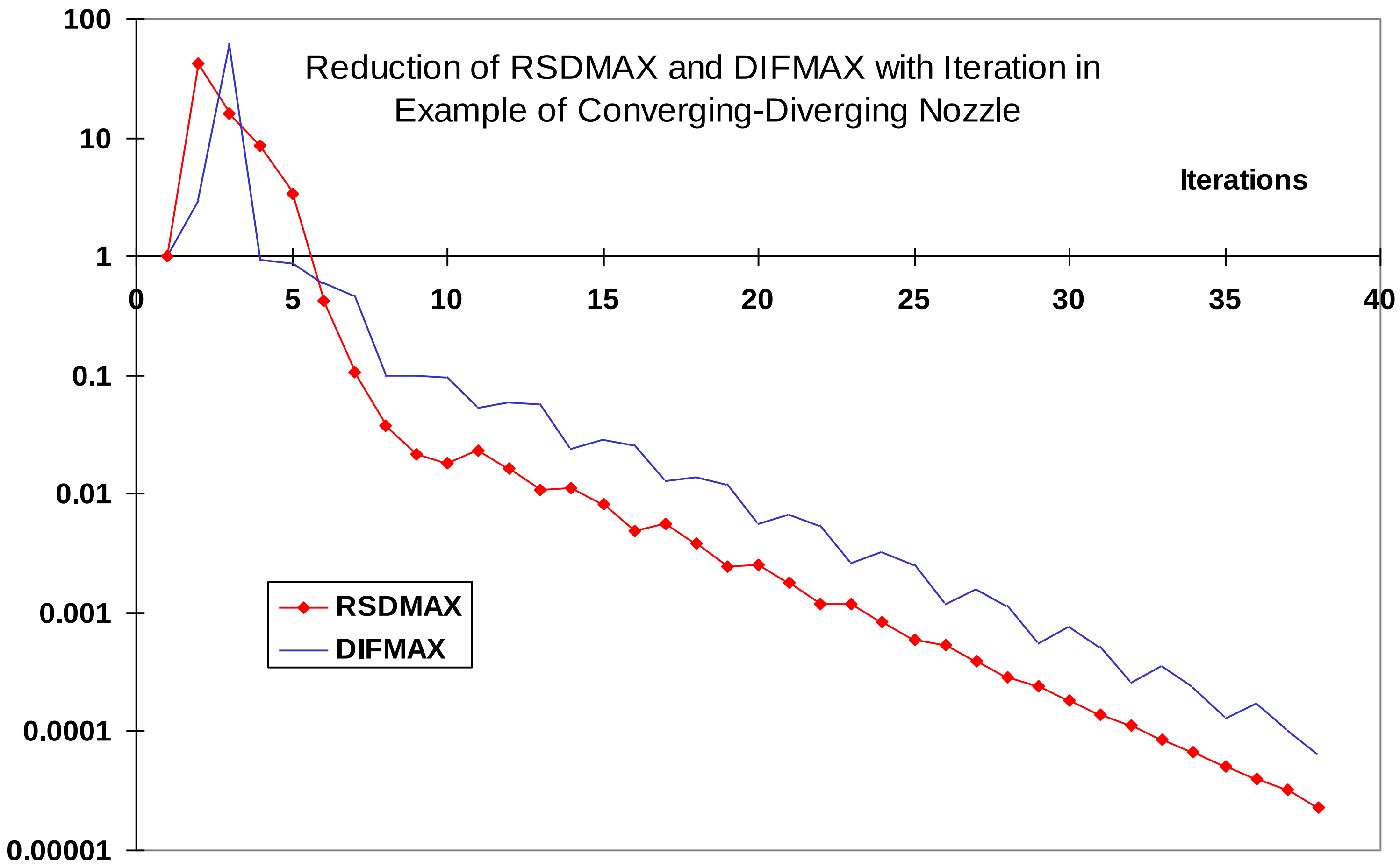
- Non-Simultaneous Option
  - Inner & Outer Iteration Loop
    - Mass, Momentum, and Equation of State are solved in inner iteration loop by N-R scheme
    - Energy and Species Conservation equations are solved in outer iteration loop by SS Scheme
    - Convergence of NR scheme is determined by Δ_{max}
    - Convergence of SS scheme is determined by $\overset{\circ}{\Delta}_{max}$

$$\overset{\circ}{\Delta}_{max} = MAX \left| \Delta_{Kf}, \Delta_{\rho}, \Delta_{h} \ or \ \Delta_{s} \right|$$

$$\Delta_{Kf} = MAX \left| \sum_{i=1}^{N_B} \frac{K_i'}{K_i} \right| \text{, etc.}$$

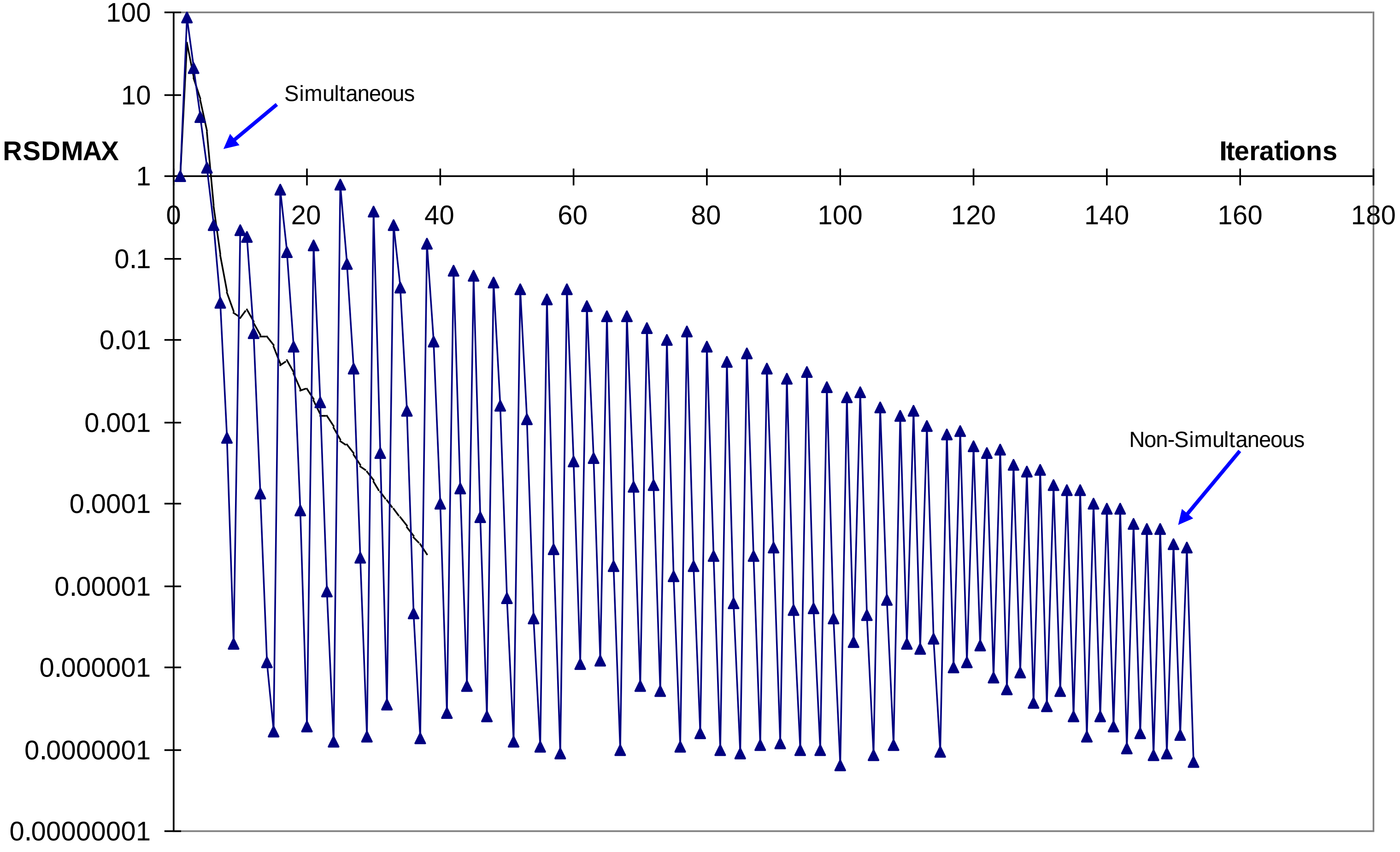
Convergence (4/5)

- Convergence Characteristics for Simultaneous Option



Convergence (5/5)

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



Sparse Matrix Solver (1/4)

- In numerical analysis, a sparse matrix is a matrix in which most of the elements are zero

- **GFSSP** uses matrix method (Gaussian Elimination) to solve the system of correction equations while using N-R method

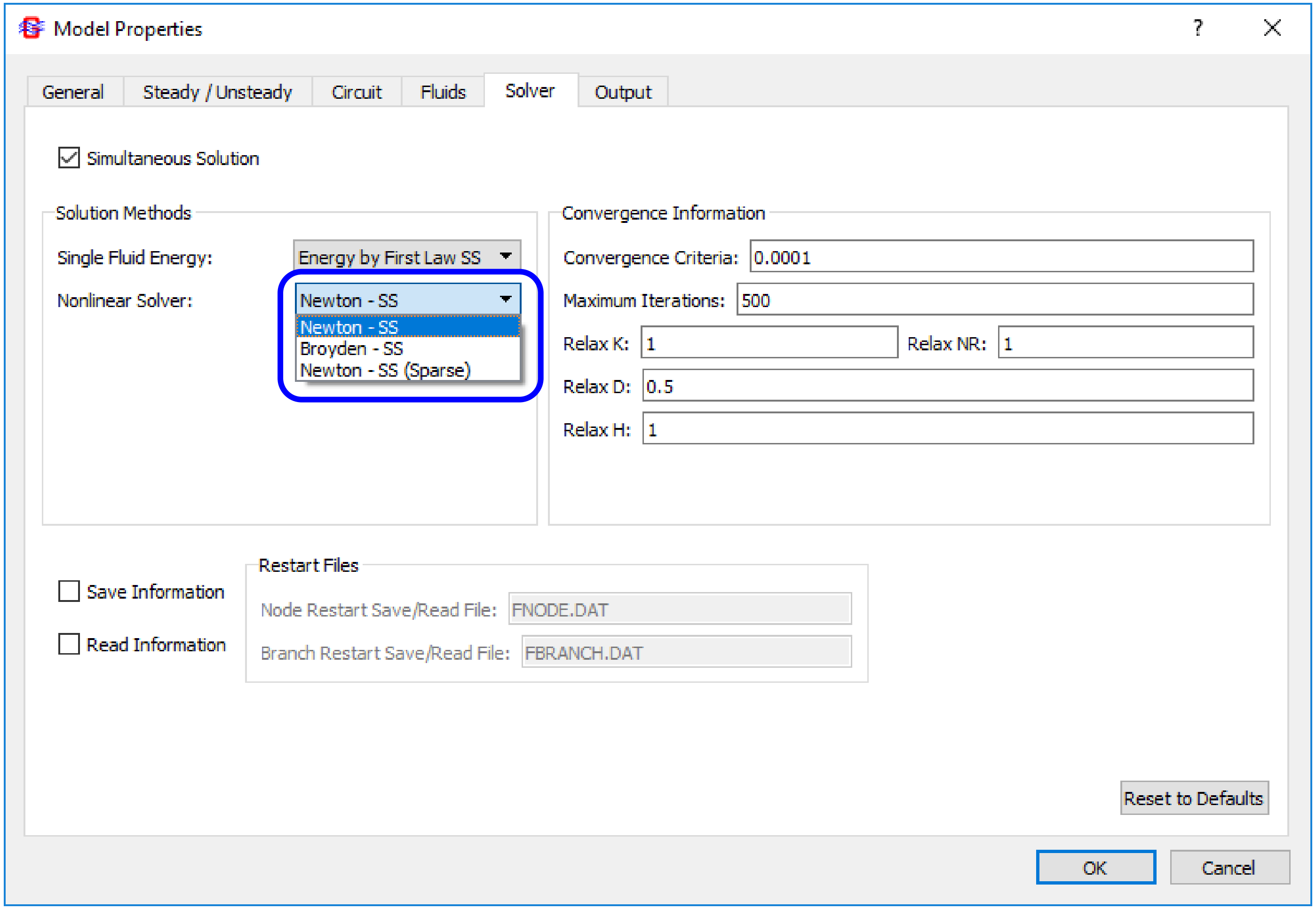
- For large network models, the matrices are usually very sparse

- There are iterative and direct methods of solving sparse matrices
  - **GFSSP** uses direct method

- Sparse matrix solver
  - Eliminates multiplications of zero elements
  - Saves processing time

Sparse Matrix Solver (2/4)

- New Solver Options



Model Properties

☑ Simultaneous Solution

Solution Methods

Single Fluid Energy: Energy by First Law SS

Nonlinear Solver: Newton - SS

Convergence Information

Convergence Criteria: 0.0001

Maximum Iterations: 500

Relax K: 1

Relax NR: 1

Relax D: 0.5

Relax H: 1

☐ Save Information

☐ Read Information

Restart Files

Node Restart Save/Read File: FNODE.DAT

Branch Restart Save/Read File: FBRANCH.DAT

Sparse Matrix Solver (3/4)

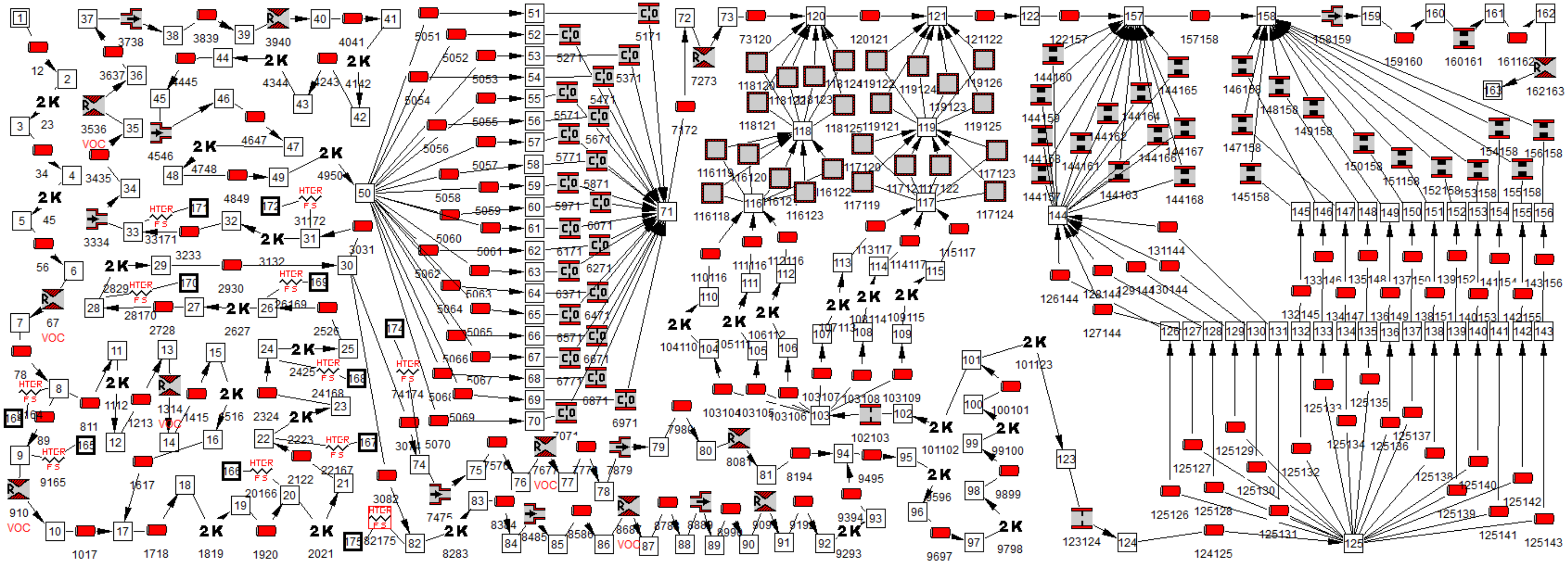
- Performance of Sparse Matrix Solver

Problem Description	No. of Internal Nodes	No. of Branches	No. of Time Steps	CPU Time (sec) (non-sparse)	CPU Time (sec) (sparse)	Processing Time Saved (%)
Tank Self-pressurization due to Boil-off (Example 29)	6	13	498500	8902	8677	3
Transfer Line Chilldown (Example 14)	31	32	16000	1166	1044	11
Tank Pressurization (Example 12)	59	64	892	2214	1108	50
Arc Jet Facility Model (LaRC/Hass)	161	238	600	66443	9687	85

Sparse Matrix Solver (4/4)

- Example: Arc Jet Facility Model (LaRC/Hass)

Sparse Matrix Solver reduces solution time by **85%**



Time Step (1/2)

- Explicit or Implicit methods can be used to solve systems of differential equations

- Explicit methods
  - Easier to program
  - Time step must be kept small for numerical stability
  - Not always easy to determine stable time step *a priori*

- Implicit methods
  - Numerically stable regardless of time step
  - May still require a small time step for solution accuracy

Time Step (2/2)

- **GFSSP** uses an implicit solver
  - In theory: Implicit solver is numerically stable at any time step
  - In practice: Very large time steps may not run
    - ➢ Negative temperatures and/or pressures will crash the property packages

- It is still the user's responsibility to verify solution time step independence
  - Example: Verify time step independence
    - ➢ Run model with a 0.1 second time step; solution converged
    - ➢ Re-run model with a time step of 0.05 seconds
    - ➢ If solutions are comparable, then time step independence is verified

Non-linearity & Under-relaxation

- Under-relaxation is necessary to solve non-linear equations
  - Example: Solve a simple non-linear equation $x^2 = 16$
    - Rewrite equation: $x_{new} = 16/x^*$
    - Suppose $x^* = 2$; then $x_{new} = 8$
    - If we substitute $x^* = 8$, we get $x_{new} = 2$
    - The solution will oscillate between 2 and 8 but will never reach the correct answer

  - Try under-relaxation with a value of 0.5 (α)
    - Iteration #1
      - Guess: $x^* = 2$
      - $x = 16/x^* = 16/2 = 8$
      - $x_{new} = (1-\alpha)x^* + \alpha x$
      - $x_{new} = (0.5)(2) + (0.5)(8) = 5$

    - Iteration #2
      - Guess: $x^* = x_{new} = 5$
      - $x = 16/x^* = 16/5 = 3.2$
      - $x_{new} = (0.5)(5) + (0.5)(3.2) = 4.1$

    - Iteration #3
      - Guess: $x^* = x_{new} = 4.1$
      - $x = 16/x^* = 16/4.1 = 3.902$
      - $x_{new} = (0.5)(4.1) + (0.5)(3.902) = 4.001$

Relaxation Parameters (1/4)

- **GFSSP** provides several relaxation parameters
  - Used to reduce the size of the corrections to the solution variables
  - Can prevent the solution “running away” to outrageous values

- Relaxation parameters can also increase the time needed for the solution to converge

- In general, explicit under-relaxation is employed
  - Relaxation parameter multiplies the calculated correction by the relaxation parameter before applying it

- Example
  - Set the relaxation parameter to 0.6
  - Only 60% of the calculated correction will be applied in each iteration

Relaxation Parameters (2/4)

- RELAXNR
  - Under-relaxes the Newton-Raphson solver
    - For the mass and momentum equations solving for pressures and flow rates
  - Generally the most effective relaxation parameter on the solution
- RELAXK
  - Under-relaxes the change in the factor K_f
    - Used in the friction term of the momentum equation
  - May be useful if the model has elements with large swings in K_f
    - For example, a valve opening and closing

Relaxation Parameters (3/4)

- RELAXH
  - Under-relaxes the change in enthalpy or entropy between iterations
  - If using 1st Law, enthalpy uses <u>inertial relaxation</u>
    - Weight (or inertia) is given to the enthalpy from the previous iteration
    - <u>Larger values of RELAXH will apply more relaxation</u>
      - This is different from the other relaxation parameters
  - If using 2nd Law, entropy is <u>explicitly</u> under-relaxed
    - Example: Setting RELAXH to 0.6 will apply 60% of the correction
  - The energy equation is fairly linear and usually well-behaved
    - Problems are most often caused by bad inputs (pressures and/or flow rates) from the solution of the mass and momentum equations
    - RELAXNR is more likely to fix the energy equation than RELAXH

Relaxation Parameters (4/4)

- RELAXD
  - Under-relaxes the change in fluid density between iterations
  - Set to 0.5 by default. Generally does not need further reduction
- RELAXHC
  - Under-relaxes the change in calculated convection coefficient between iterations
- RELAXTS
  - Under-relaxes the change in solid temperature between iterations

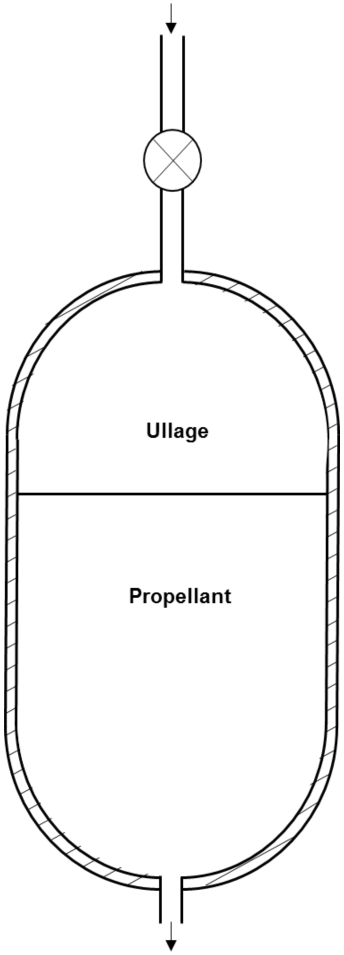
Troubleshooting

- Check that input parameters are correct and make sense
- Try under-relaxation (especially RELAXNR)
- Change time-step
- Tighten convergence criteria
- Try non-simultaneous solution
- If steady-state model won't converge
  - Convert it to a transient and let it run to steady-state
- If a model converges with less severe boundary conditions
  - Try using that solution in a restart file to provide an initial guess
- Contact the developers for help

Solution Procedure Summary

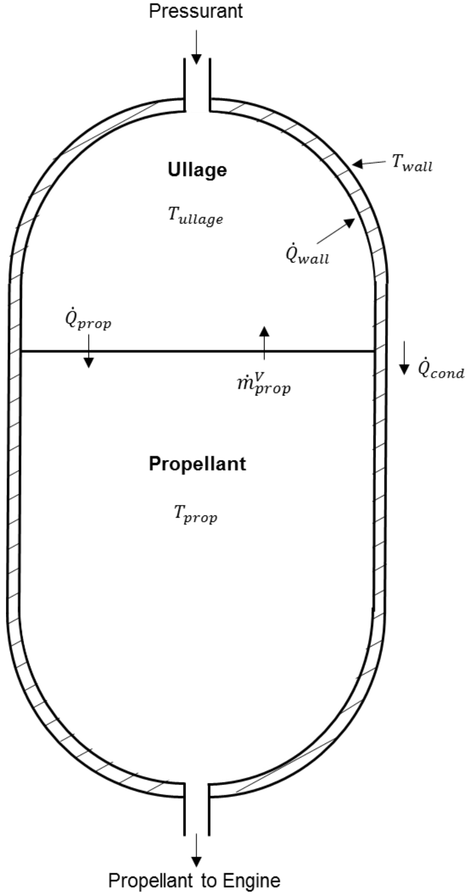
- Simultaneous option is more efficient than Non-Simultaneous option
- Non-Simultaneous option is recommended when Simultaneous option experiences numerical instability
- Non-linearity and strong coupling need under-relaxation
- Good initial guess help to overcome convergence problem
- A lack of realism in problem specification can lead to convergence problem
- Lack of realism
  - Unrealistic geometry and/or boundary conditions
  - Attempt to calculate properties beyond operating range

Tank Pressurization, Control Valves, and Relief Valves



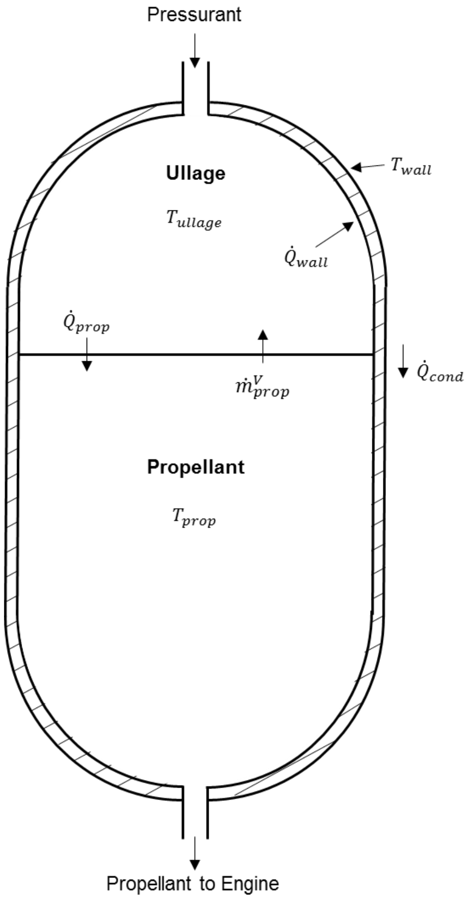
Tank Pressurization (1/19)

- Predict
  - Ullage conditions between the propellant and the tank wall
    - Includes heat transfer, and may include mass transfer
  - Propellant conditions leaving the tank



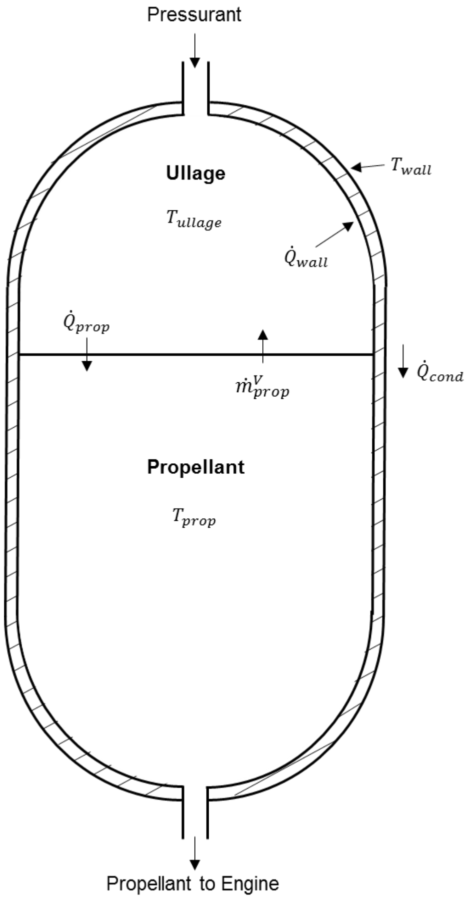
Tank Pressurization (2/19)

- Additional Physical Processes
  - Volume change in ullage and propellant
  - Gravitational head change in the tank
  - Heat transfer from pressurant to propellant ($\dot{Q}_{prop}$)
  - Heat transfer from pressurant to the tank wall ($\dot{Q}_{wall}$)
  - Heat conduction between the pressurant exposed tank surface and the propellant exposed tank surface ($\dot{Q}_{cond}$)
  - Mass transfer between the pressurant and propellant ($\dot{m}_{prop}^{V}$)
    - Optional, with user subroutine



Tank Pressurization (3/19)

- Assumptions
  - Liquid in tank remains at constant temperature
  - Ullage gas is modeled as one bulk temperature
    - No stratification
  - Tank walls are well insulated
    - Heat leak from outside is negligible compared to heat transfer from pressurant



Tank Pressurization (4/19)

- Mathematical Modeling of Physical Processes (1/4)
  - Change in Ullage and Propellant Volume

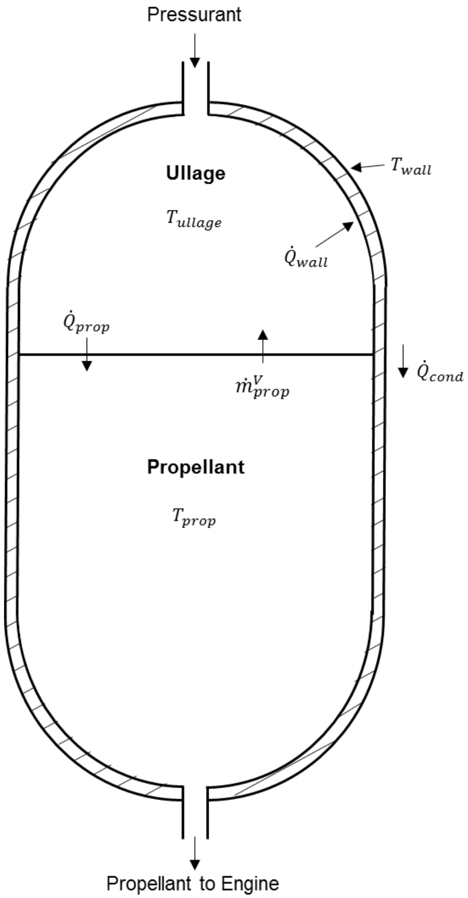
$$dV_{ullage} = \frac{\dot{m}_{prop}\Delta\tau}{\rho_{prop}} = -dV_{prop}$$

  - 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 (5/19)

- Mathematical Modeling of Physical Processes (2/4)
  - Change in Gravitational Head in the Tank

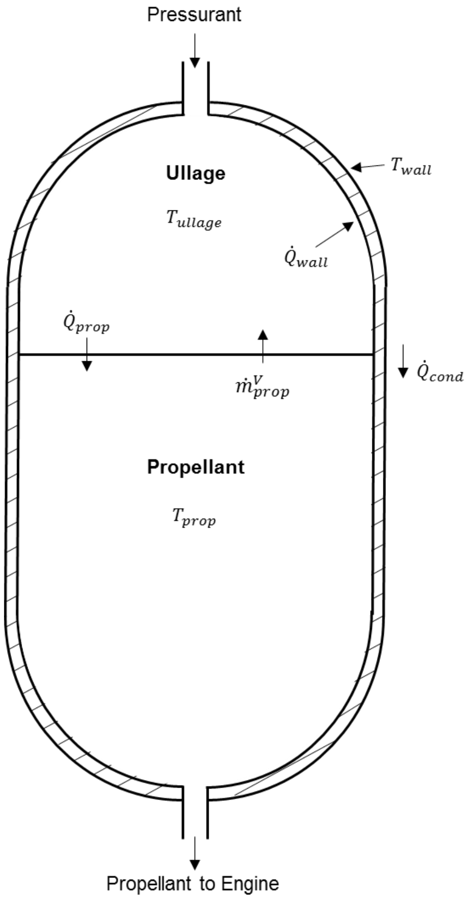
$$p_{tank\ bottom} = p_{ullage} + \frac{\rho_{prop} g H}{g_c}$$

  - Heat Transfer from Ullage to Propellant

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

  - Heat Transfer Coefficient (Natural Convection)

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



Tank Pressurization (6/19)

- Mathematical Modeling of Physical Processes (3/4)
  - Heat Transfer from Ullage to Wall

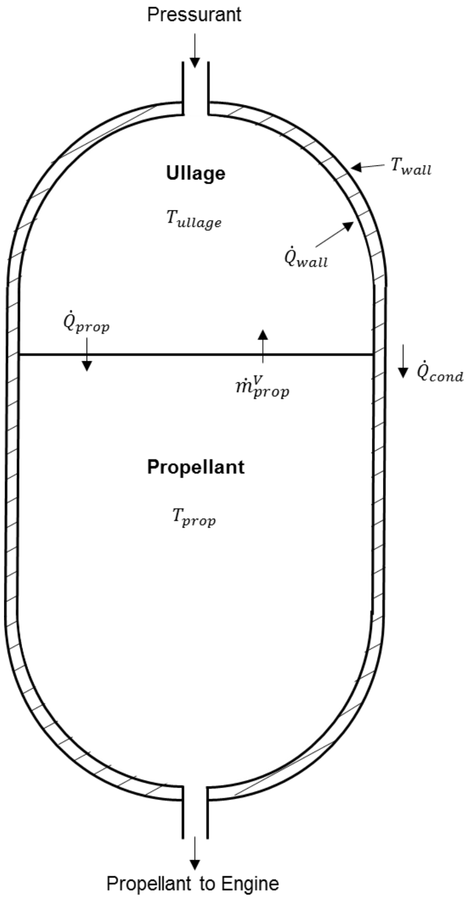
$$\dot{Q}_{wall} = [h_c A]_{U-W}\left(T_{ullage} - T_{wall}\right)$$

  - Tank Wall Conduction

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

  - Energy Balance on Tank Wall

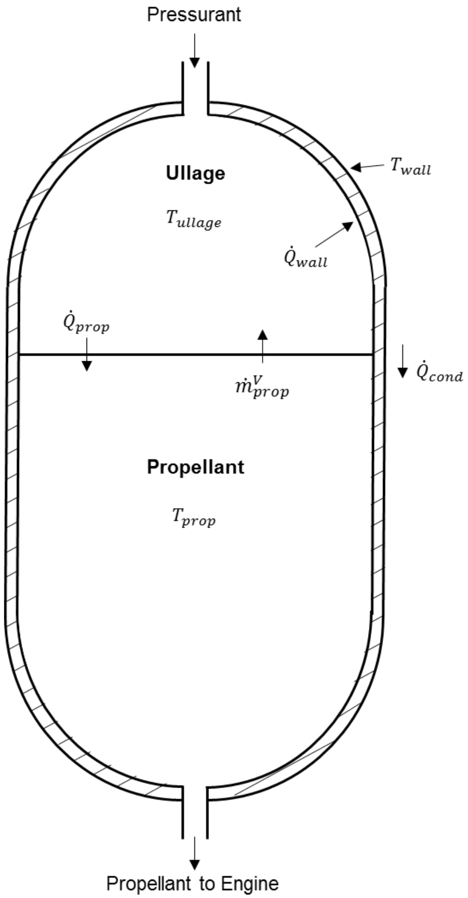
$$mC_p T_{wall}^{i-1} + \Delta m C_p T_{wall}^{liq} + \left(\dot{Q}_{wall} - \dot{Q}_{cond}\right)\Delta\tau = (m + \Delta m) C_p T_{wall}^{i}$$



Tank Pressurization (7/19)

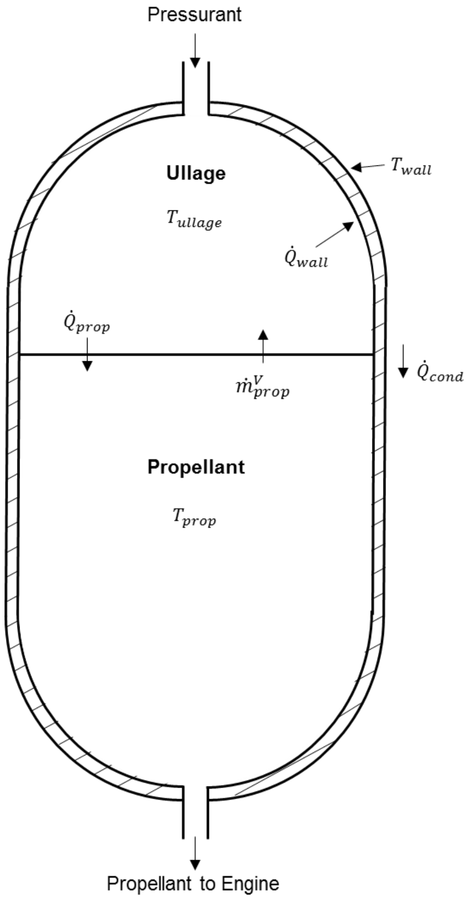
- Mathematical Modeling of Physical Processes (4/4)
  - Mass Transfer from Propellant to Ullage
    - With optional user subroutine
  - Heat of Vaporization (h_{fg}) and saturation temperature (T_{sat})
    - Determined at current ullage pressure by calling utility subroutine PROPS_PSAT

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



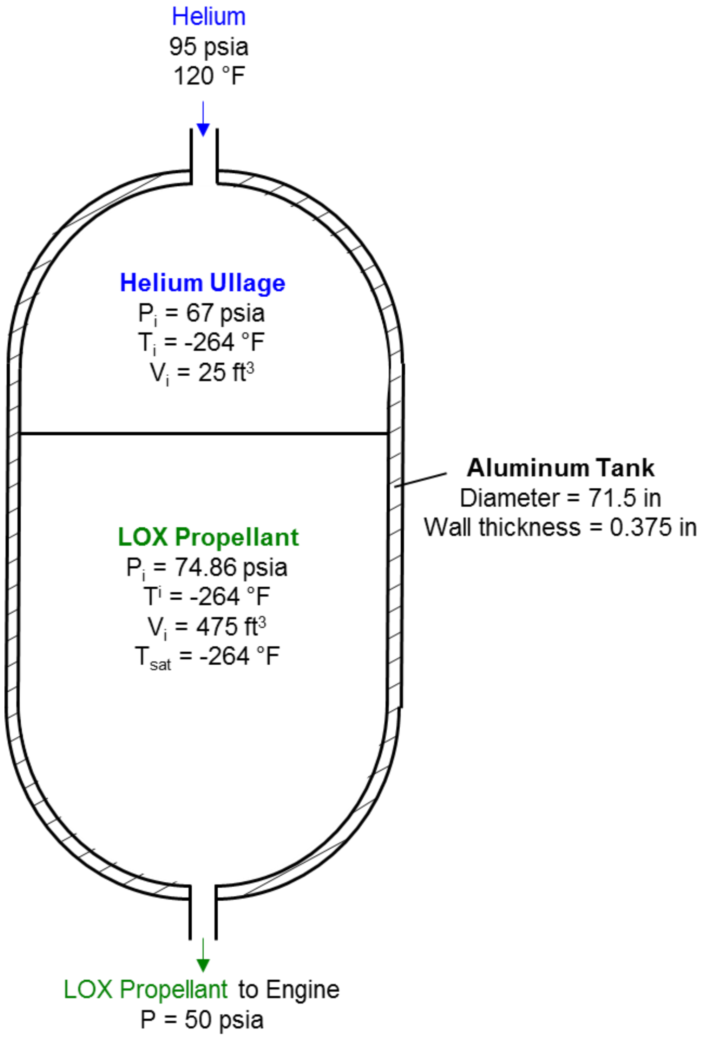
Tank Pressurization (8/19)

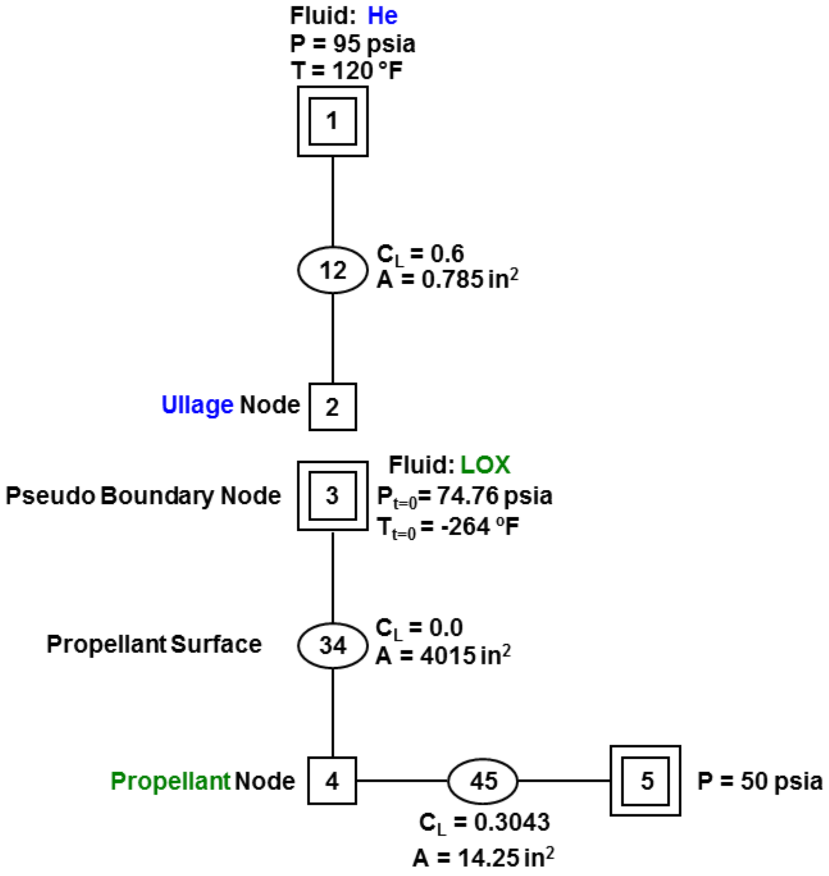
- Calculation Steps
  - Ullage and Propellant Volumes
  - Tank Bottom Pressure
  - Heat Transfer
    - Between pressurant and propellant ($\dot{Q}_{prop}$)
    - Between pressurant and wall ($\dot{Q}_{wall}$)
  - Wall Temperature
  - Mass Transfer from propellant to ullage ($\dot{m}^{V}_{prop}$)
    - Only with optional user subroutine



Tank Pressurization (9/19)

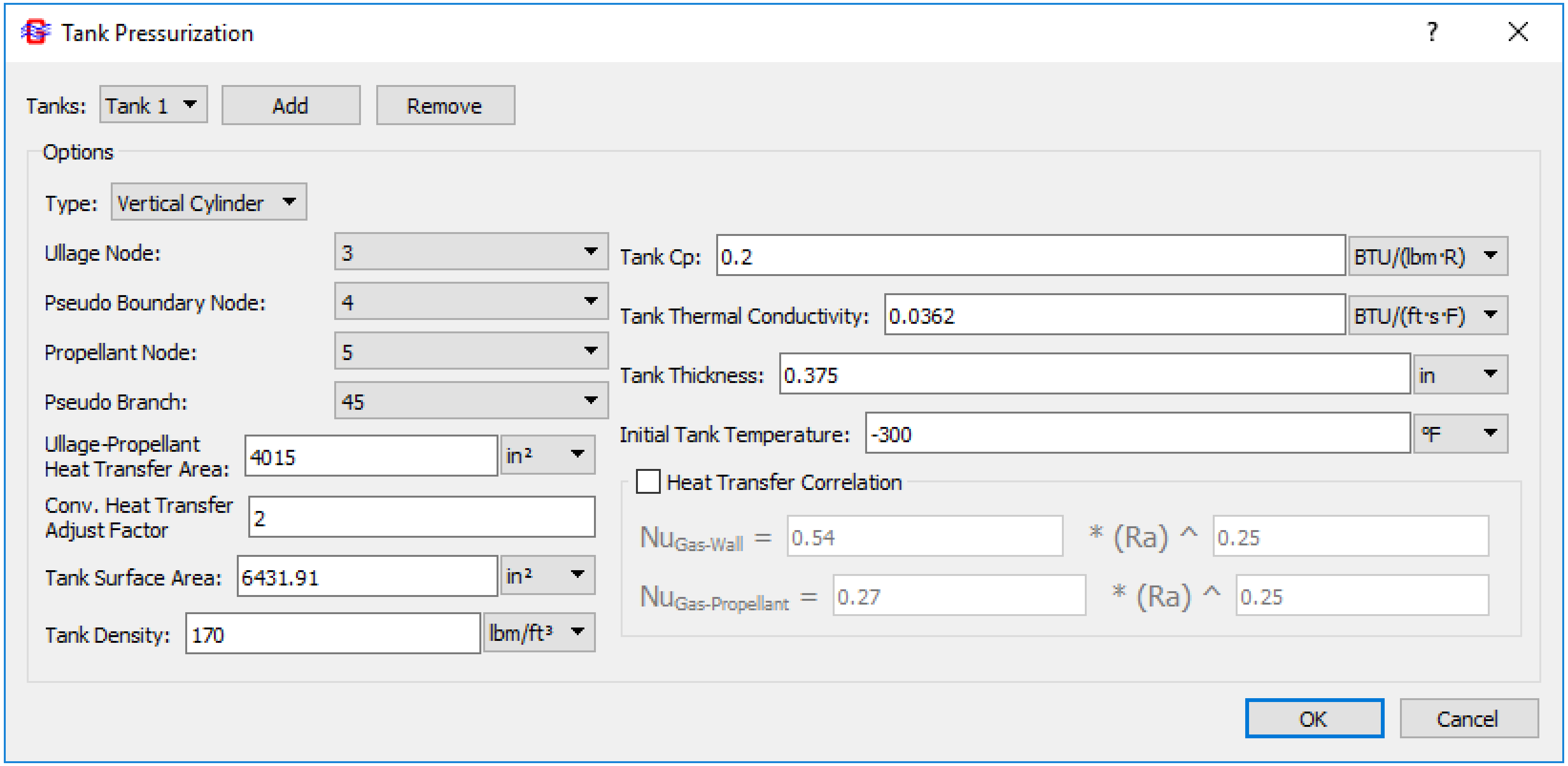
- Example 10 - Tank Schematic and **GFSSP** Model





Tank Pressurization (10/19)

- Additional Input for Pressurization Option



Tank Pressurization

Tanks: Tank 1

Options

Type: Vertical Cylinder

Ullage Node: 3

Pseudo Boundary Node: 4

Propellant Node: 5

Pseudo Branch: 45

Ullage-Propellant Heat Transfer Area: 4015 in^2

Conv. Heat Transfer Adjust Factor: 2

Tank Surface Area: 6431.91 in^2

Tank Density: 170 lbm/ft^3

Tank Cp: 0.2 BTU/(lbm·R)

Tank Thermal Conductivity: 0.0362 BTU/(ft·s·F)

Tank Thickness: 0.375 in

Initial Tank Temperature: -300 °F

☐ Heat Transfer Correlation

$Nu_{Gas\text{-}Wall}$ = 0.54 * (Ra) ^ 0.25

$Nu_{Gas\text{-}Propellant}$ = 0.27 * (Ra) ^ 0.25

Tank Pressurization (11/19)

- Example 10 - Pressurization Output

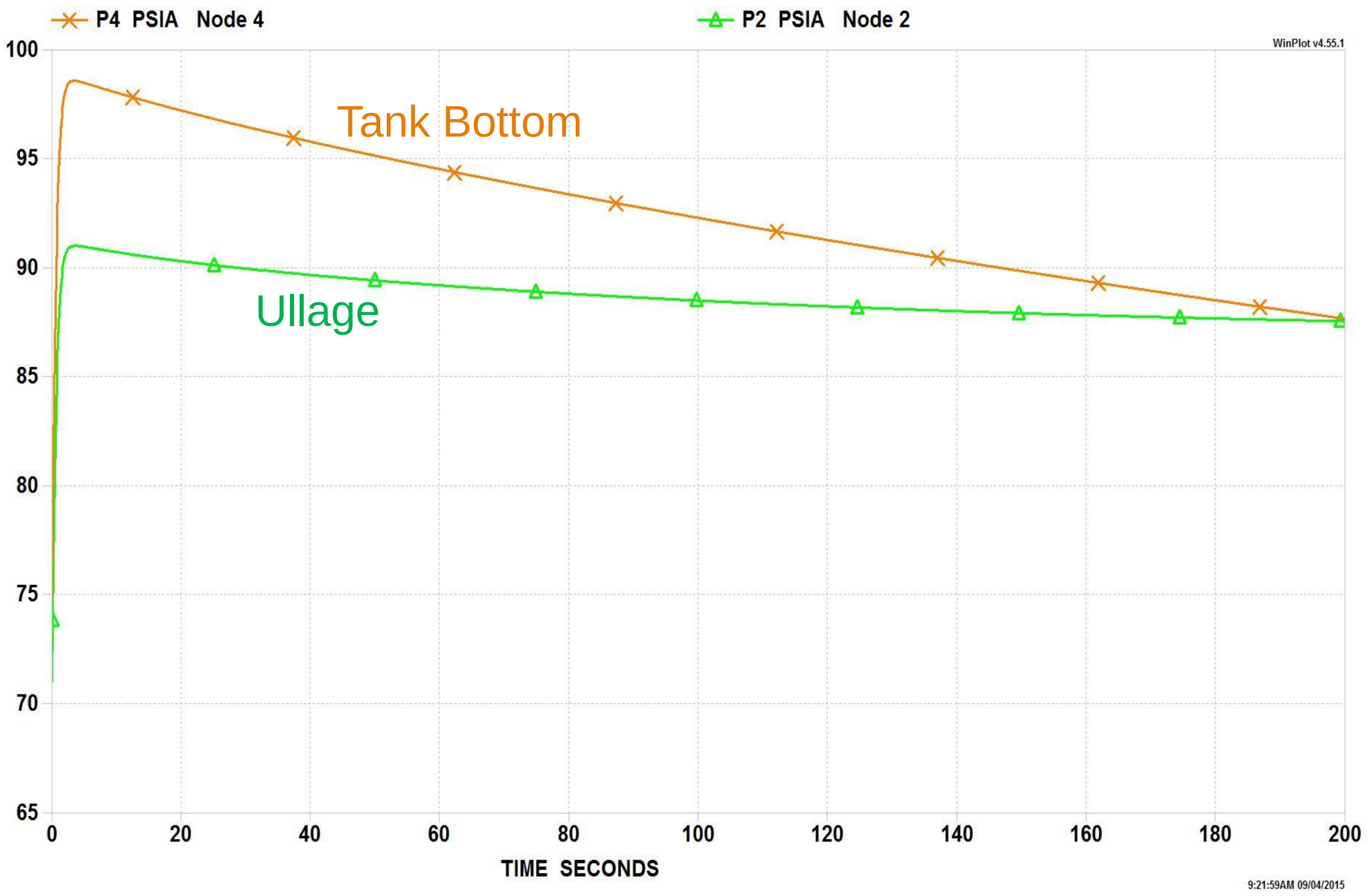
```
SOLUTION
INTERNAL NODES
    NODE     P(PSI)      TF(F)         Z         RHO        EM(LBM)    CONC
                                              (LBM/FT^3)
                                                                         HE        O2
     2    0.9138E+02 -0.1347E+03  0.1006E+01  0.1047E+00  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

QULPRP (Btu/s)
QULWAL (Btu/s)
QCOND (Btu/s)
TNKTM (°R)
VOLPROP (ft^3)
VOLULG (ft^3)

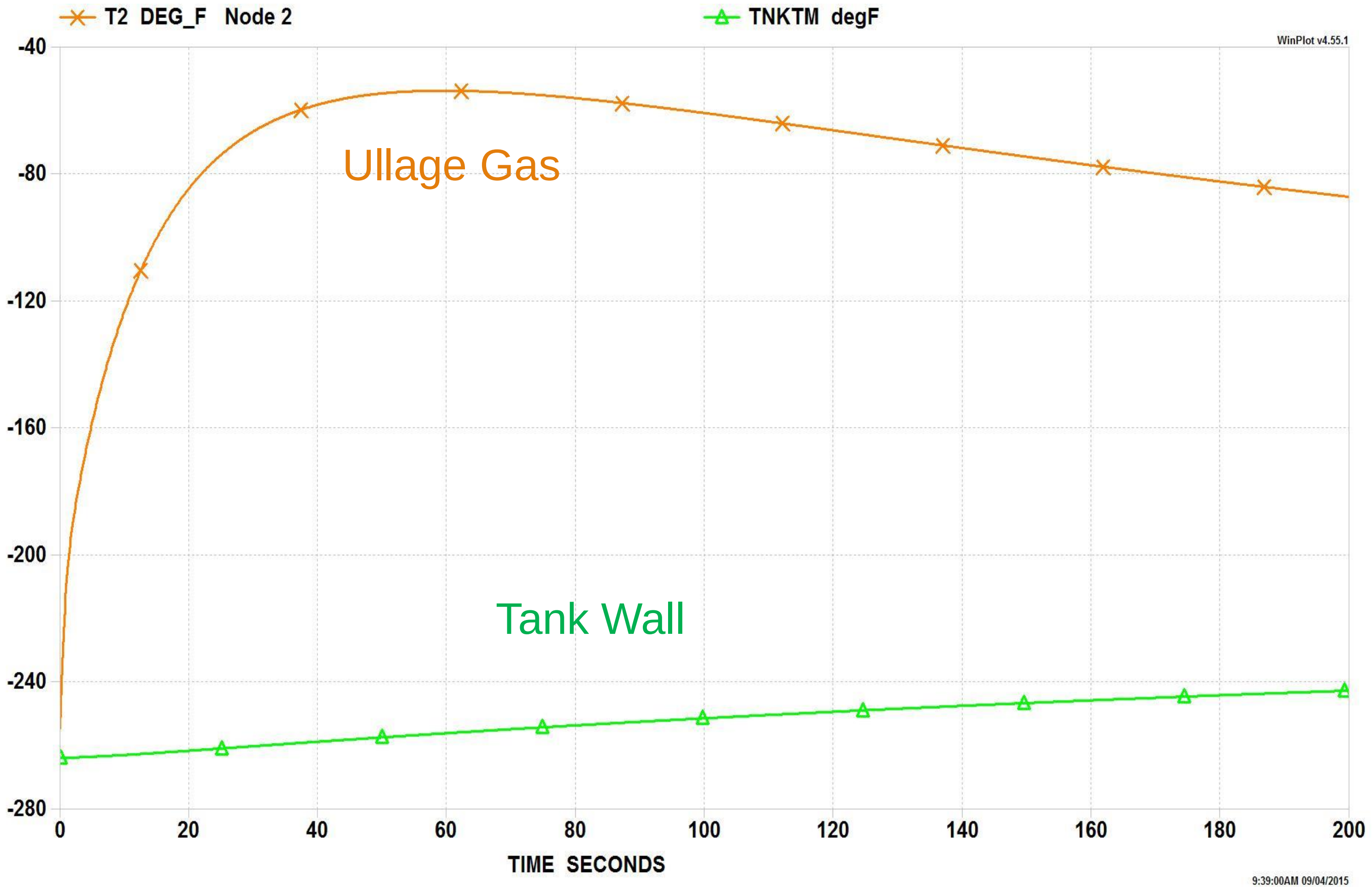
Tank Pressurization (12/19)

- Example 10 - Ullage and Tank Bottom Pressure History



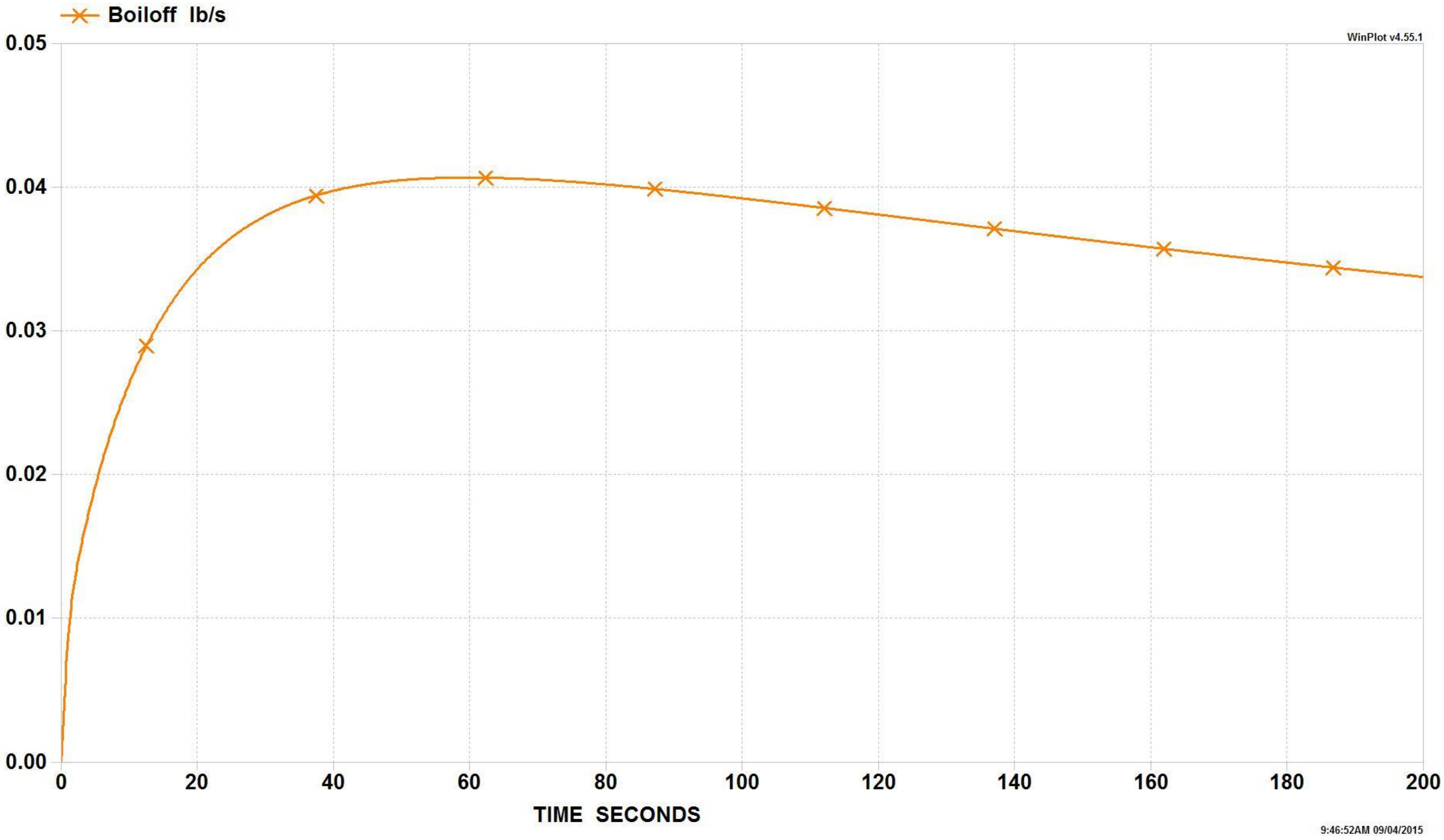
Tank Pressurization (13/19)

- Example 10 - Ullage and Tank Wall Temperature History



Tank Pressurization (14/19)

- Example 10 – Propellant to Ullage Mass Transfer Rate History



Tank Pressurization (15/19)

- Collapse Factor Correlation
  - Ratio of *actual* pressurant consumption to an *ideal* pressurant consumption
  - Ideal consumption assumes **no** heat or mass transfer
  - Calculated by the Epstein Correlation

where:

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

$$w_p^0 = \rho_G^0 \Delta V \qquad C = \frac{\left(\rho c_p^0 t\right)_w}{\left(\rho c_p\right)_G^0 D_{eq}} \frac{T_s}{T_0} \qquad S = \frac{h_c \theta_T}{\left(\rho c_p\right)_G^0 D_{eq}} \frac{T_S}{T_0} \qquad Q = \frac{\dot{q} \theta_T}{\left(\rho c_p\right)_G^0 D_{eq} T_0}$$

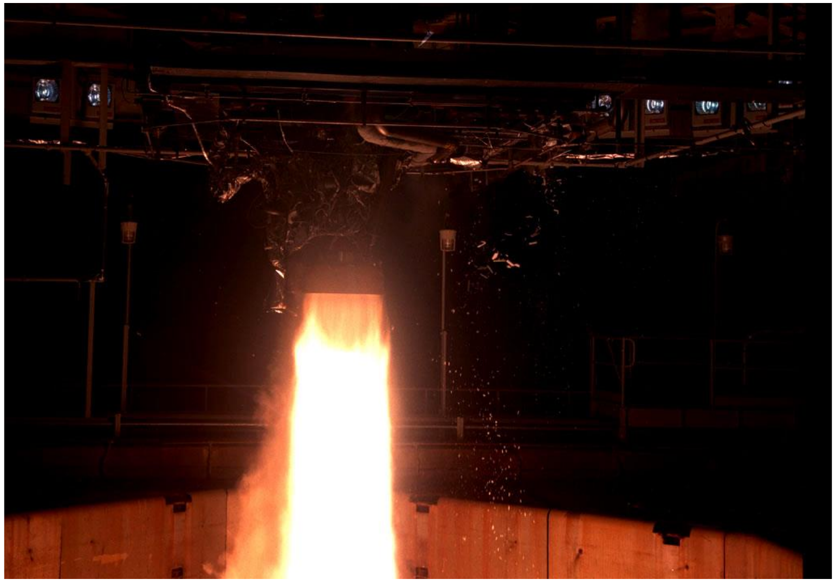
C	ratio of wall to gas thermal capacitance
p_1 - p_8	fitted constants (dependent on propellant)
Q	ratio of ambient heat input to effective thermal capacitance of gas
S	modified Stanton number
T_0	pressurant inlet temperature
T_s	propellant saturation temperature at initial tank pressure

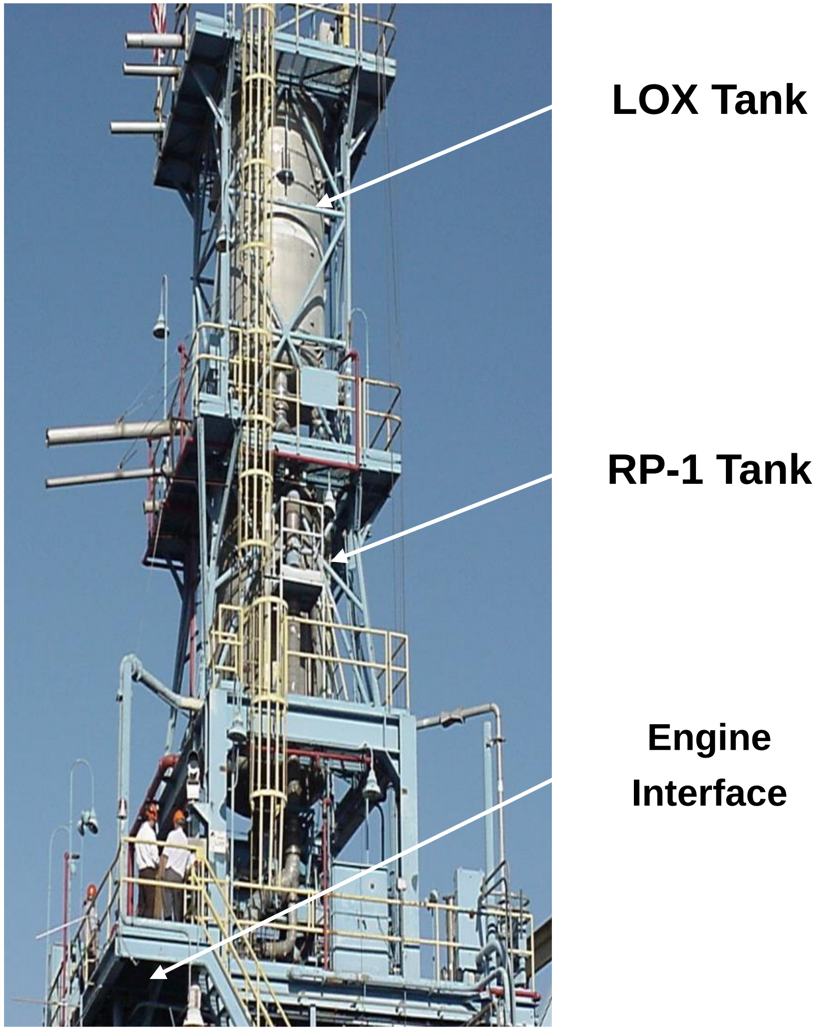
Tank Pressurization (16/19)

- Pressurization Model Validation
  - **GFSSP** Collapse Factor Prediction: **1.46**
  - Epstein Correlation Collapse Factor Prediction: **1.51**
    - **GFSSP** Prediction Discrepancy: **-3.3%**

Tank Pressurization (17/19)

- Applications

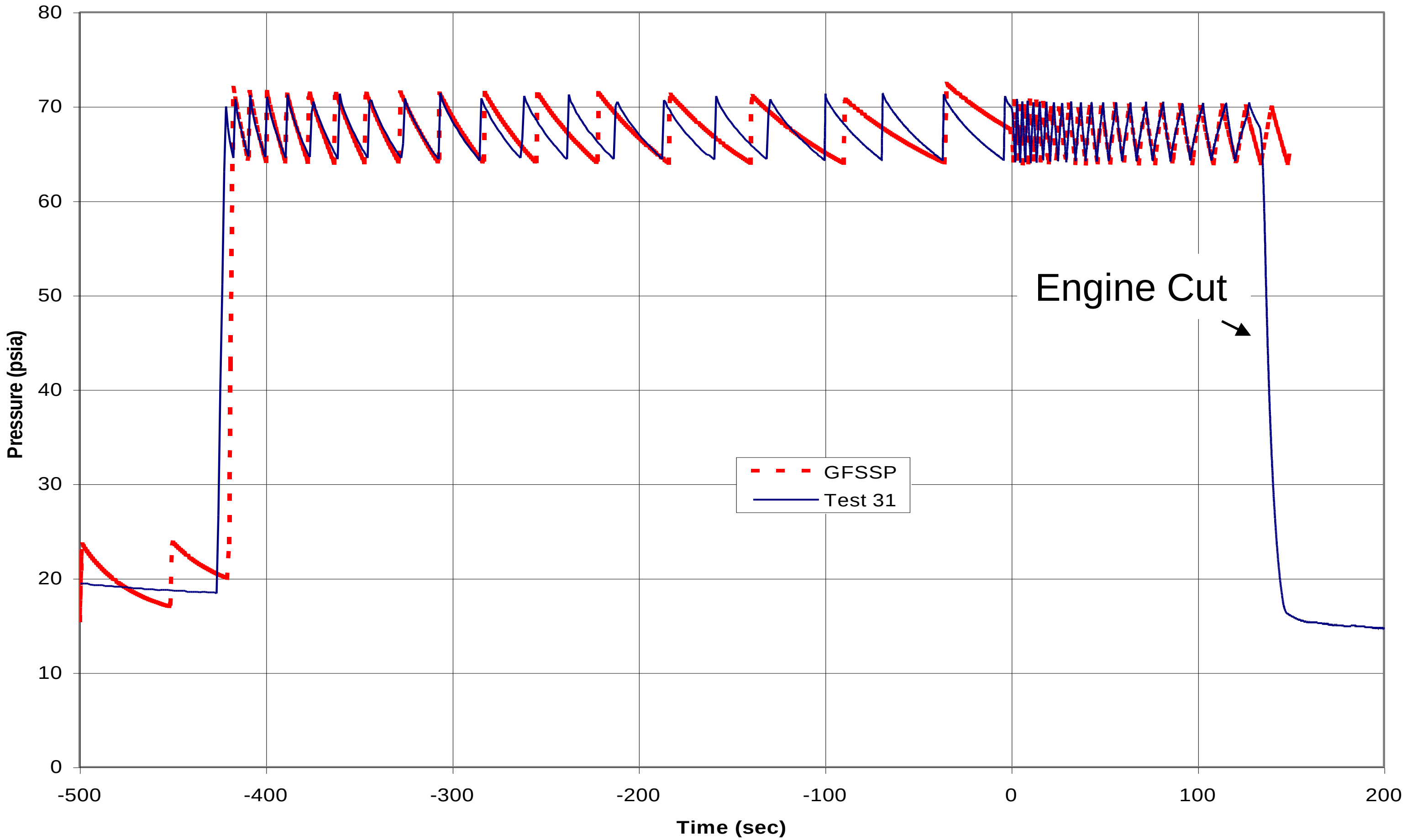




Tank Pressurization (18/19)

Marshall Space Flight Center
GFSSP Training Course

- Comparison of **GFSSP** Predicted LOX Ullage Pressure with Test Data



Tank Pressurization Summary (19/19)

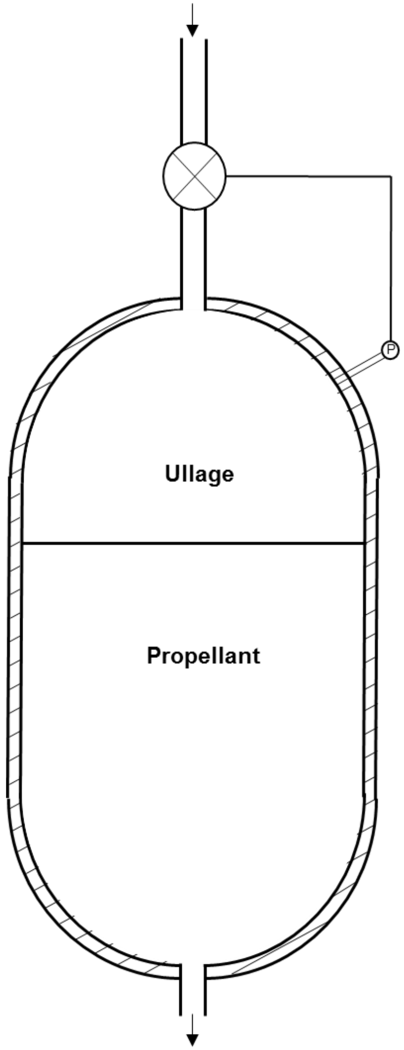
- **GFSSP**’s transient capability option
  - Extended to model the pressurization of a propellant tank
- User-activatated
  - Inputs additional tank information
- Code predicts the history of ullage and propellant conditions
- **GFSSP** Example 10
  - Demonstrates use of Tank Pressurization option
  - Describes verification of numerical prediction

Control Valve (1/9)

- Pressure monitored at arbitrary point downstream of valve

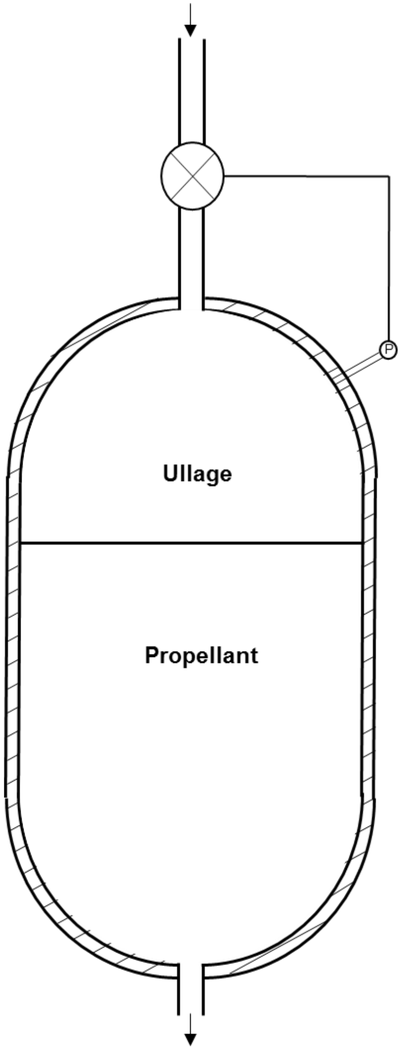
- Valve maintains pressure within user-specified tolerance
  - Closes when pressure exceeds maximum value
  - Opens when pressure drops below minimum value

- Flow Resistance Factor
  - Calculated using same equations as Branch Option 2 (Restriction)



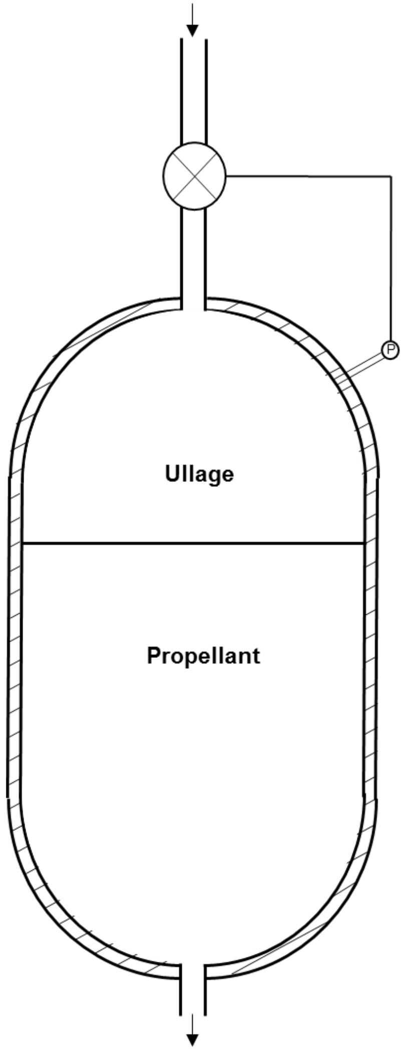
Control Valve (2/9)

- 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



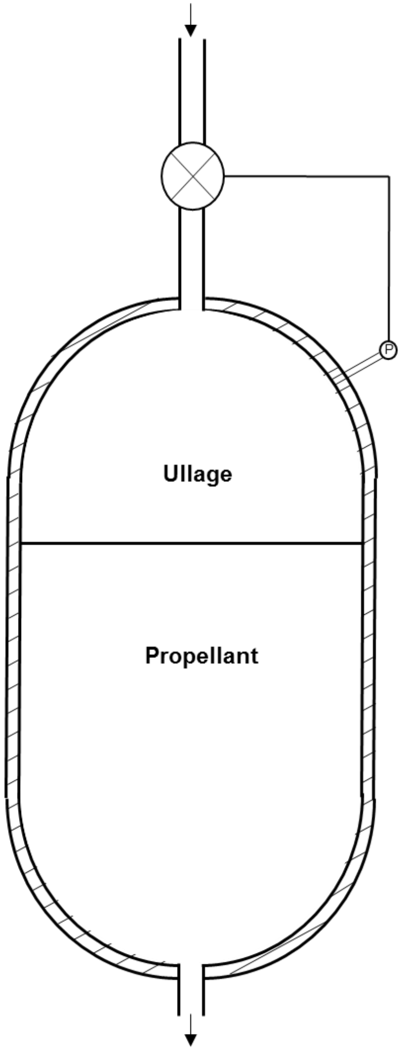
Control Valve (3/9)

- Branch Inputs -1
  - Sub-option
  - Flow Coefficient (C_L)
  - Area (A)
  - Control Node
  - Valve Initial Position
  - Pressure Tolerance File Name



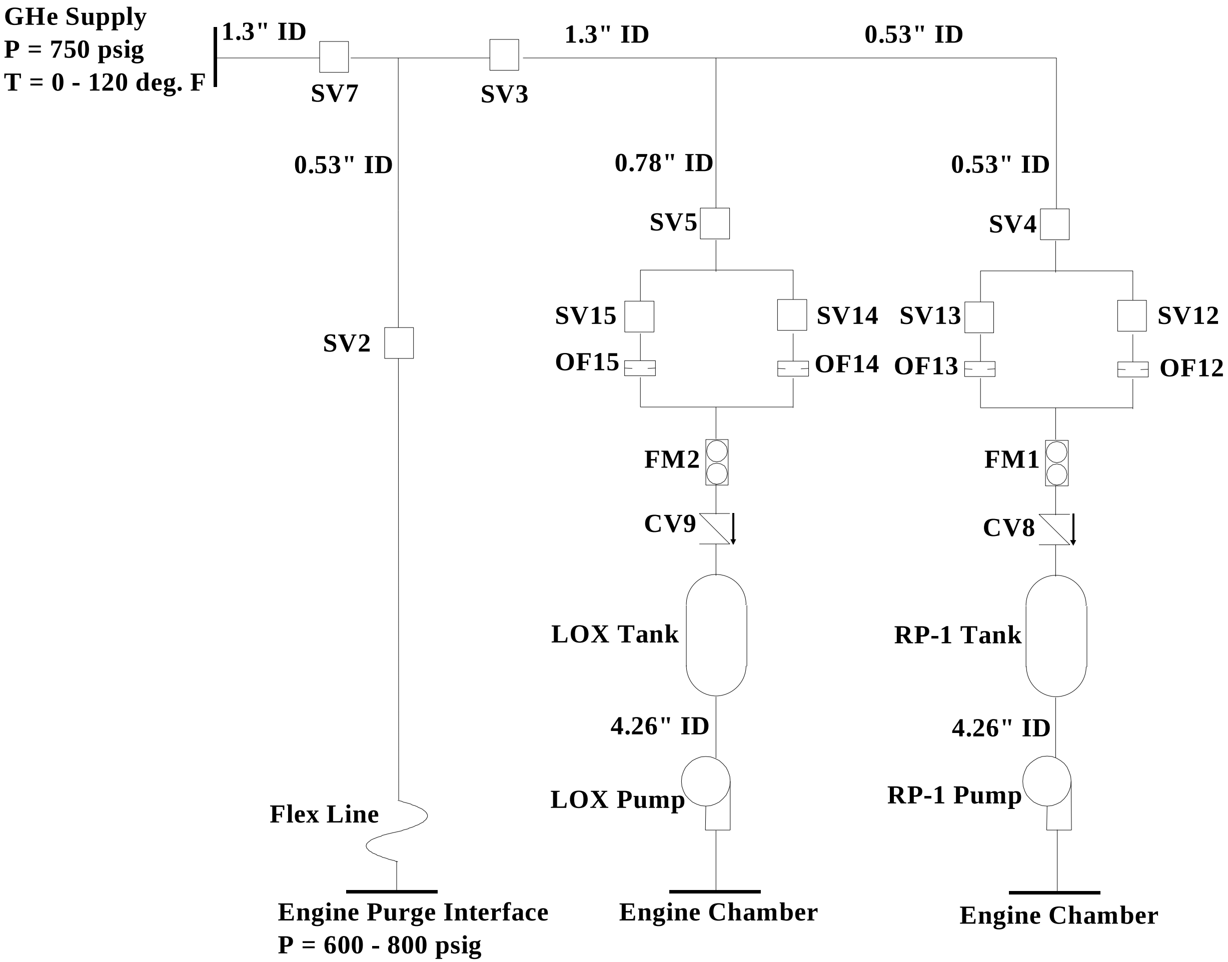
Control Valve (4/9)

- 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



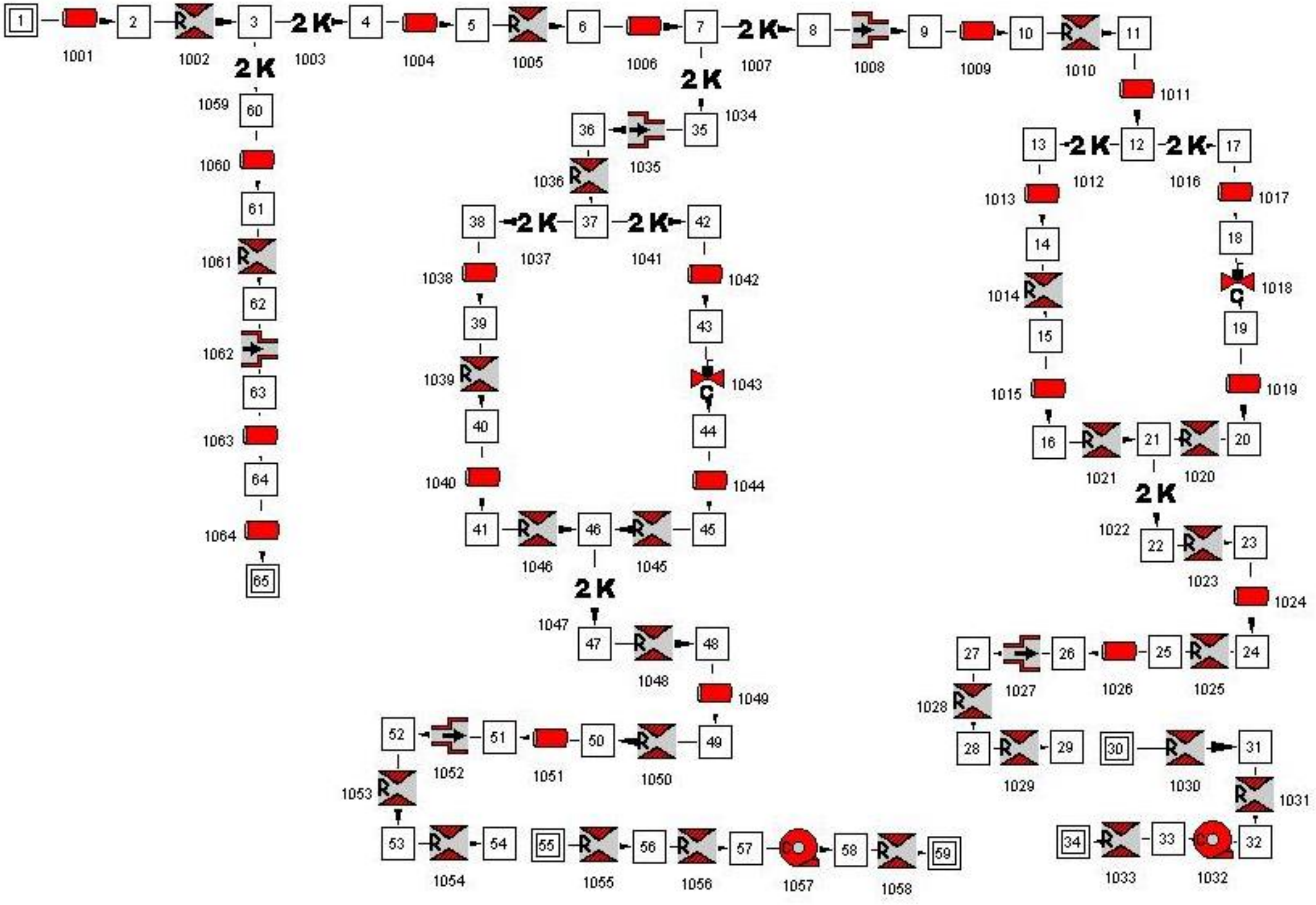
Control Valve (5/9)

- Example 12 - Schematic



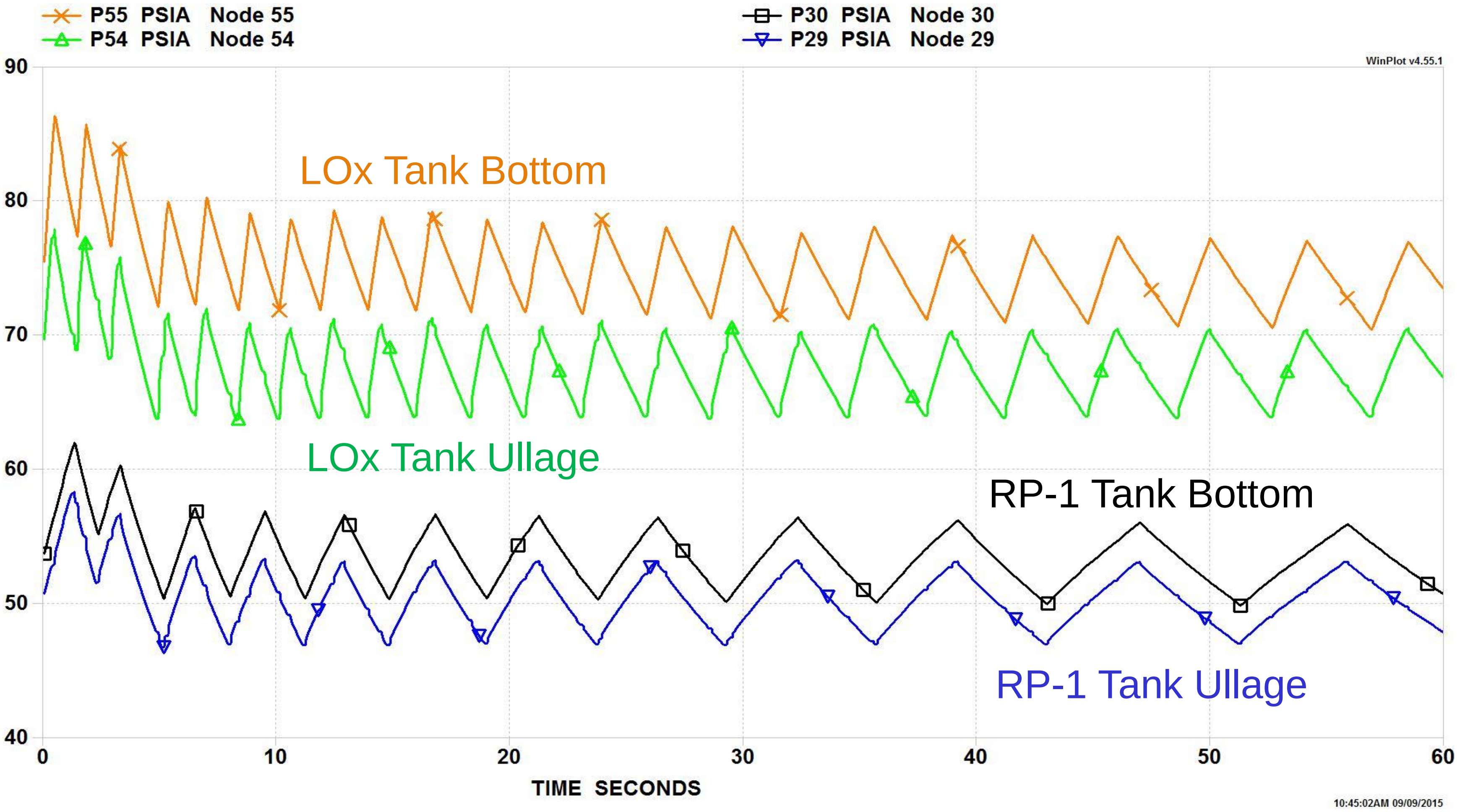
Control Valve (6/9)

- Example 12 – **MIG** Canvas



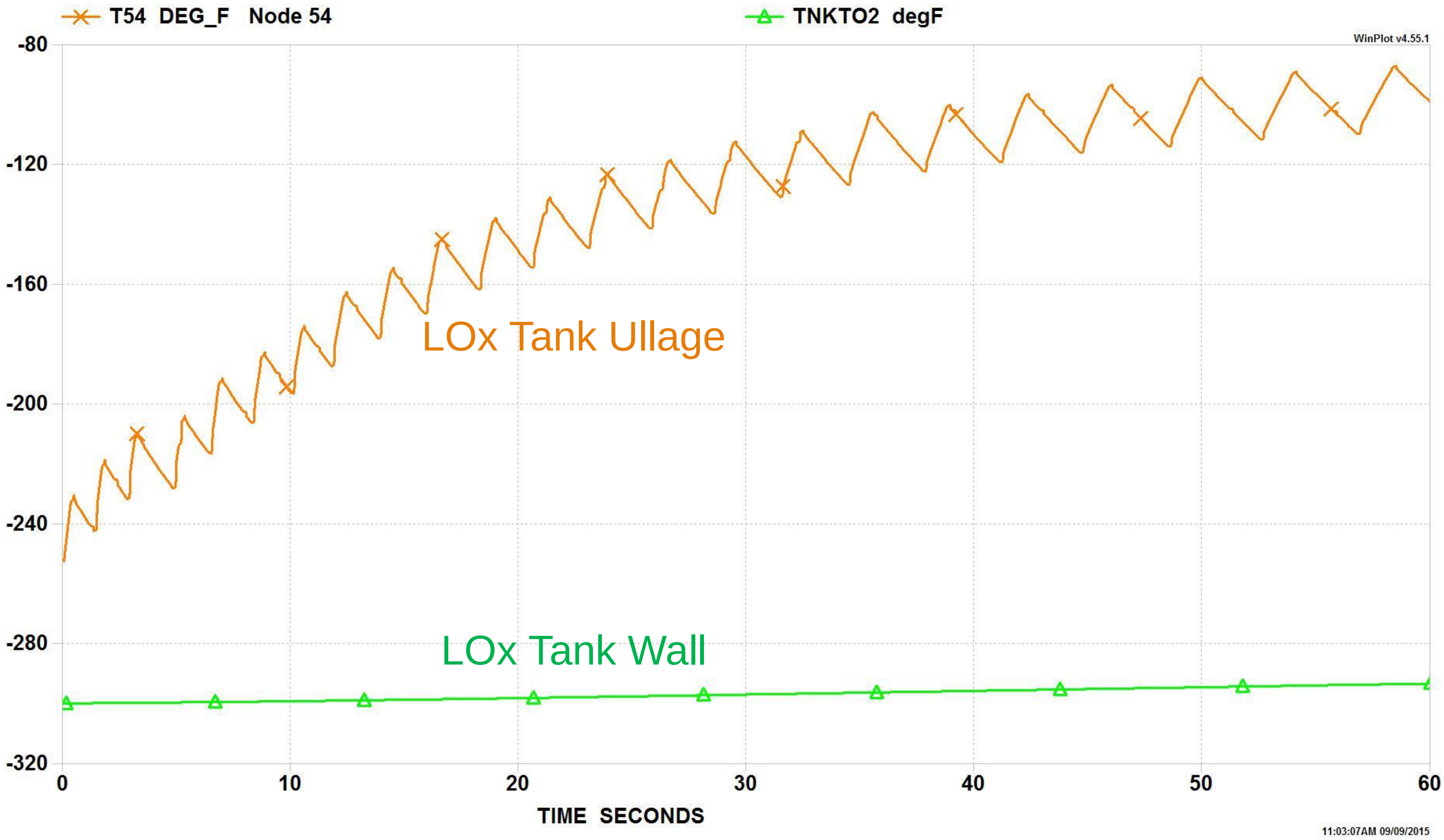
Control Valve (7/9)

- Example 12 – Propellant Tank Pressure History



Control Valve (8/9)

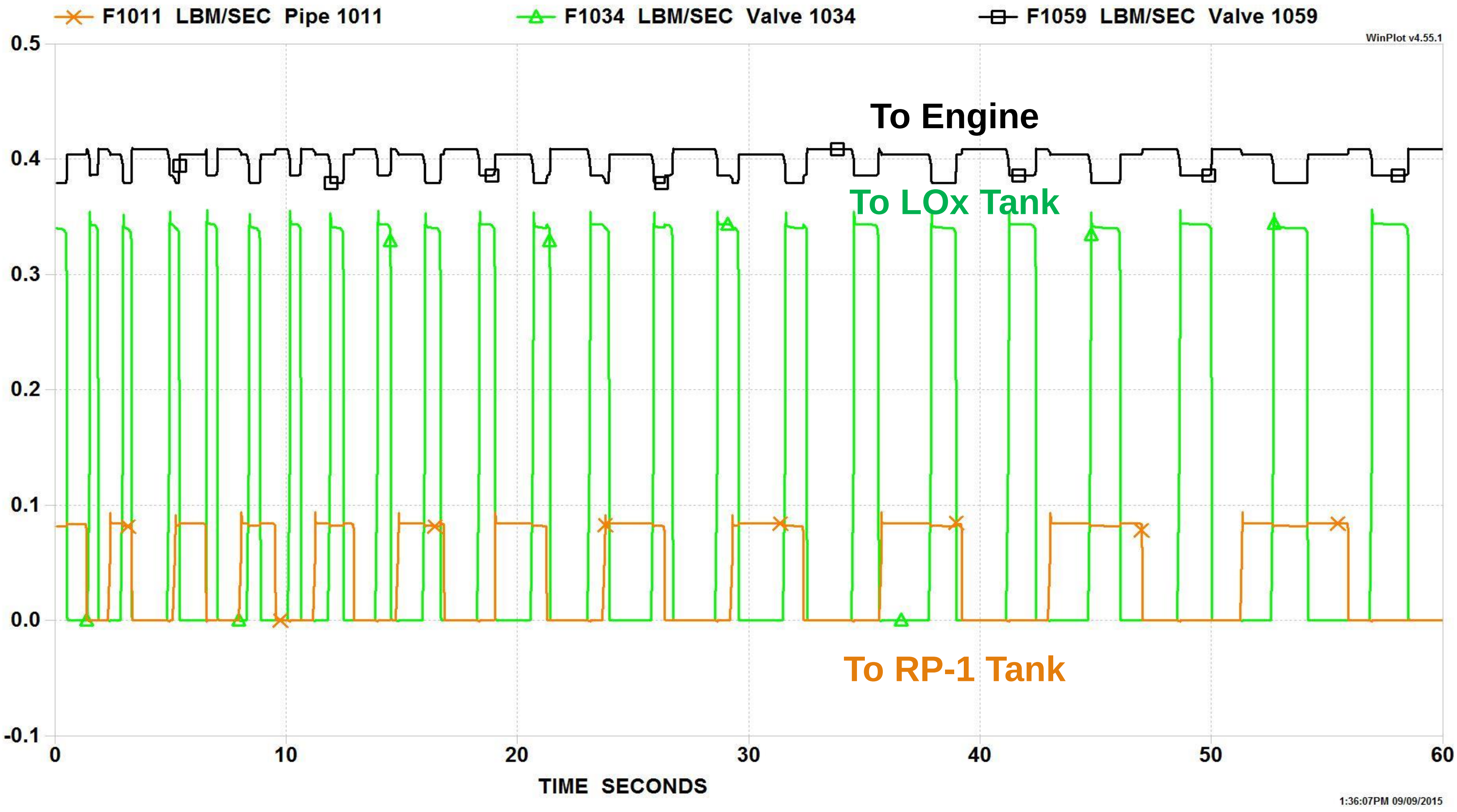
- Example 12 – LOX Tank Temperature History



Control Valve (9/9)

Marshall Space Flight Center
GFSSP Training Course

- Example 12 – Helium Flow Rate History



Control Valve Summary

- Control Valve
  - Monitors pressure of a target node
  - Opens/Closes as needed

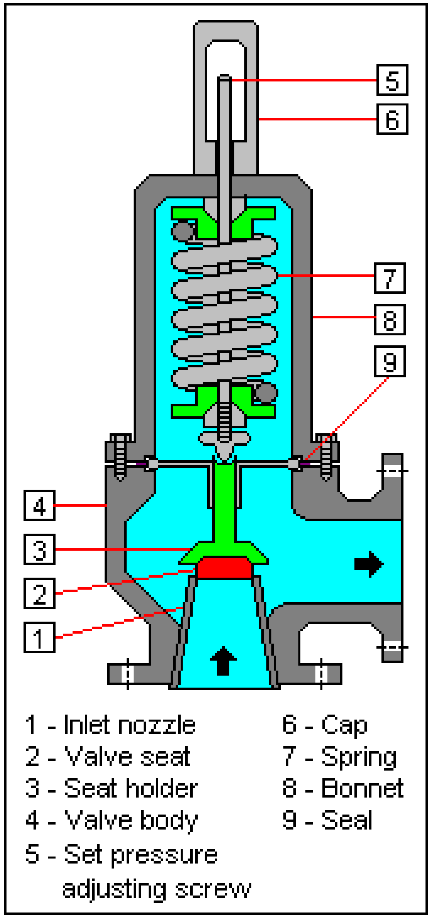
- Valid only for transient models

- User provides
  - Flow and operational characteristics of the valve

- **GFSSP** Example 12
  - Demonstrates the operation of the Control Valve option

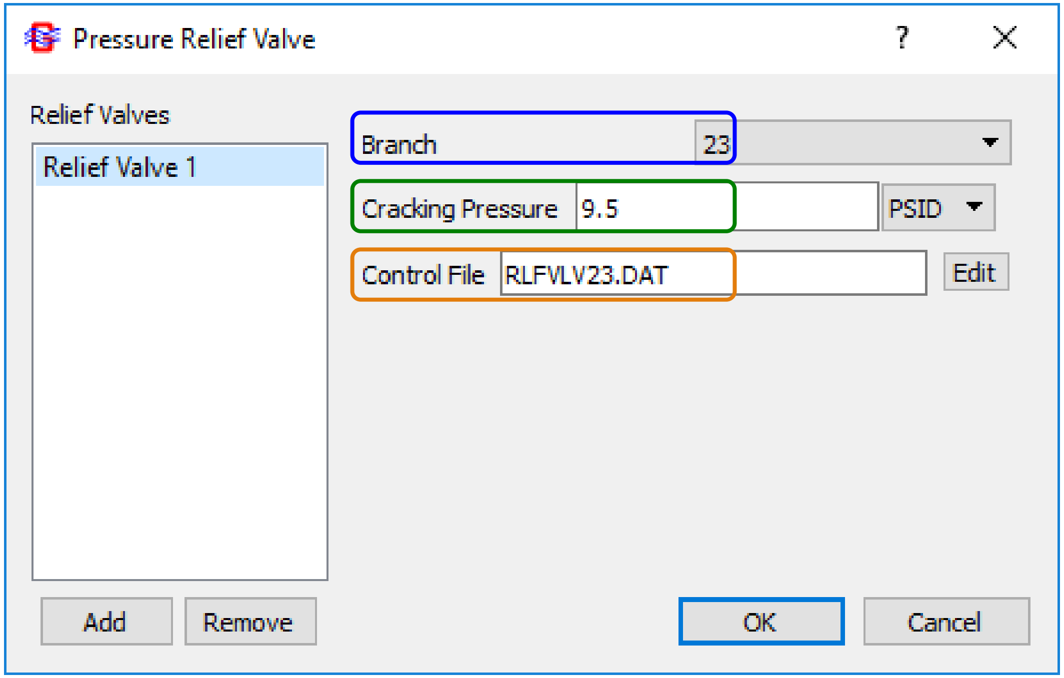
Relief Valve (1/6)

- Distinct from Control Valve
- Monitors pressure differential across valve branch
- Valve opens when pressure differential exceeds cracking pressure
- Can also be used as a check valve if small cracking pressure is used
- Relief valve is an Advanced Option that may be linked to:
  - Restriction
  - Compressible orifice
  - Valve with Cv



Relief Valve (2/6)

- Relief Valve Inputs
  - Branch ID number
  - Valve cracking pressure differential (psid)
  - Control File
    - Determines valve branch flow resistance as function of pressure differential



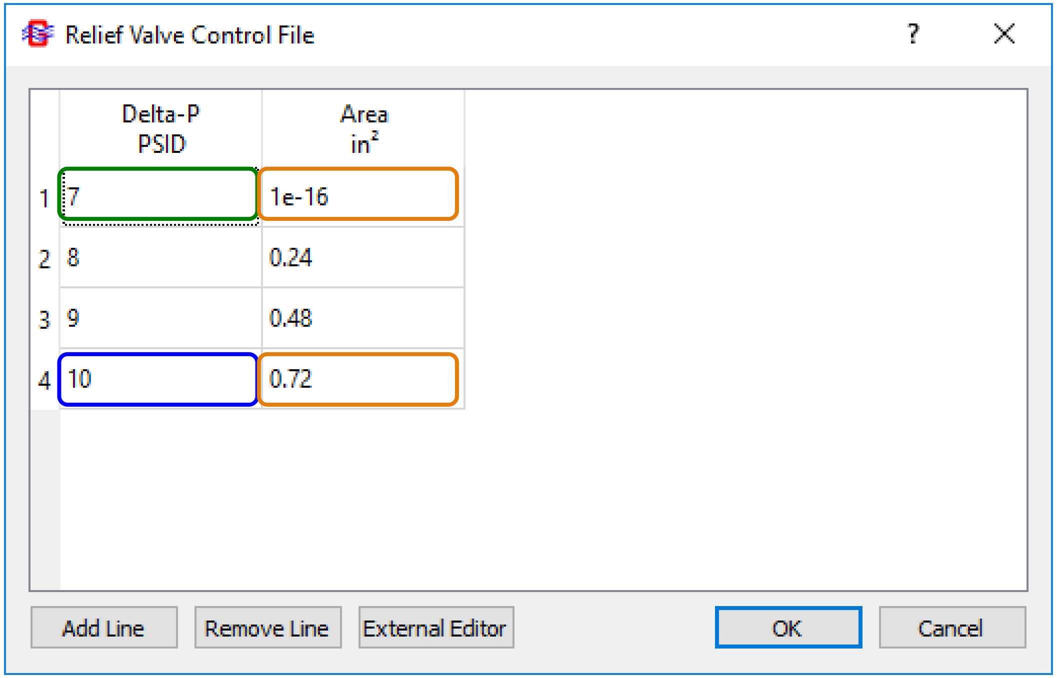
Branch: 23

Cracking Pressure: 9.5 PSID

Control File: RLFVLV23.DAT

Relief Valve (3/6)

- Relief Valve Control File
  - Reseating pressure (psid)
  - Fully-open pressure (psid)
  - Area (in^2) or Cv

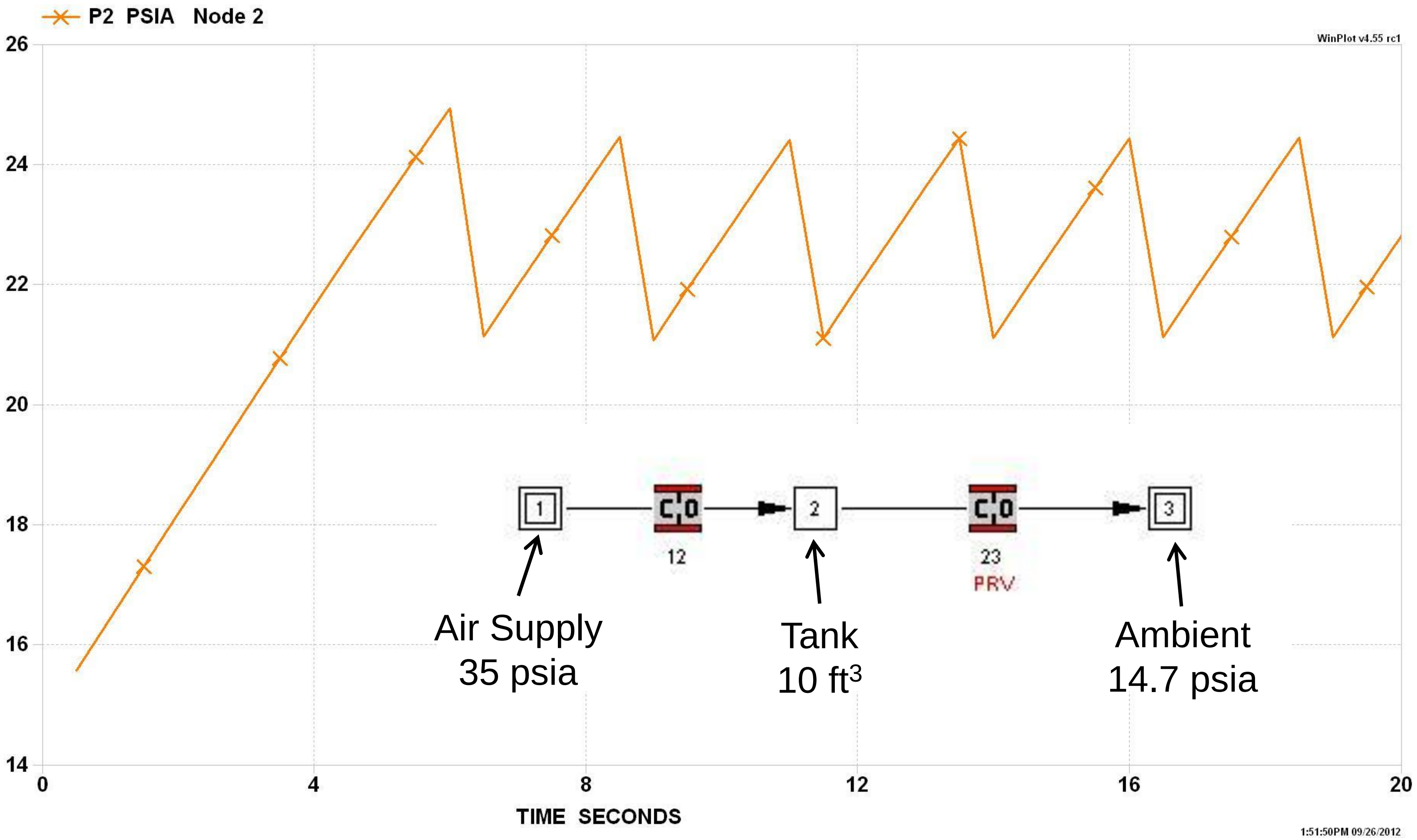


Relief Valve Control File

	Delta-P PSID	Area in^2
1	7	1e-16
2	8	0.24
3	9	0.48
4	10	0.72

Relief Valve (4/6)

- Example 24 – Tank Pressure History



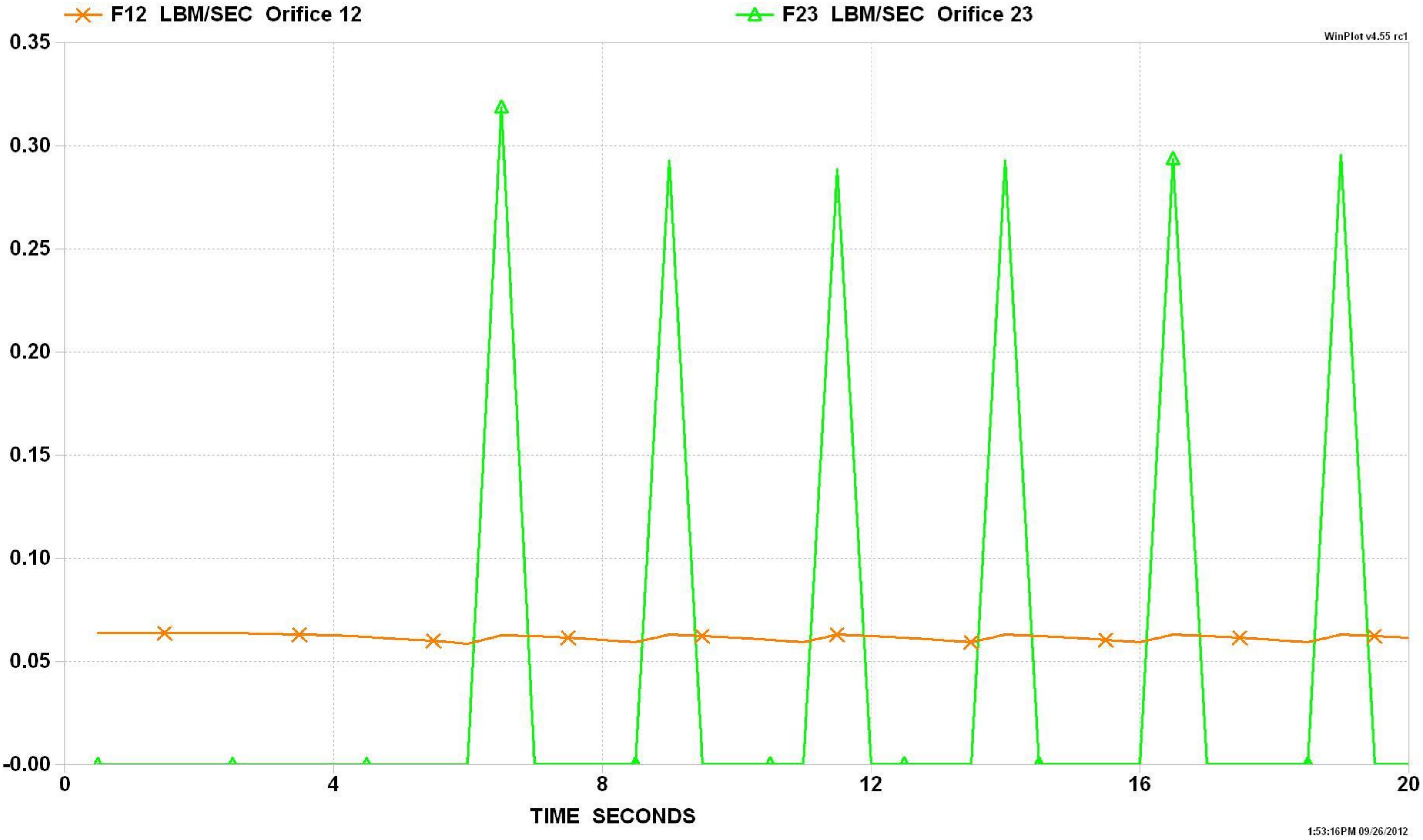
Air Supply
35 psia

Tank
10 ft^3

Ambient
14.7 psia

Relief Valve (5/6)

- Example 24 – Flow Rate History



Relief Valve Summary (6/6)

- GFSSP Advanced Option
  - Models behavior of a Relief Valve

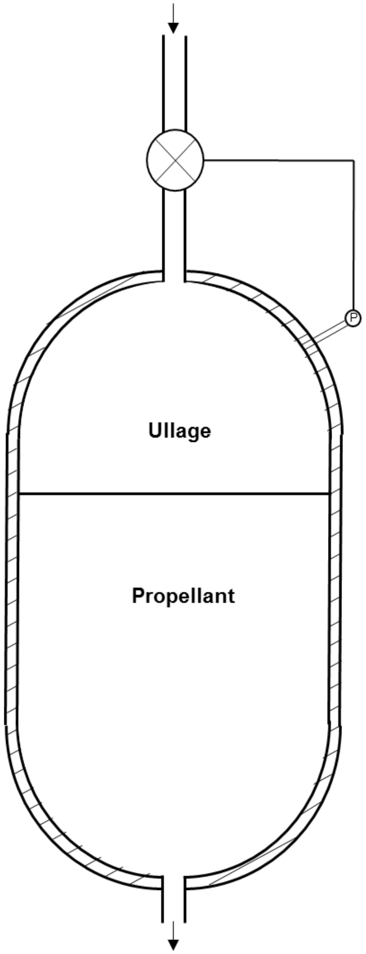
- Valid only for transient models

- User provides
  - Cracking Pressure
  - Flow resistance characteristics

- **GFSSP** Example 24
  - Demonstrates the operation of the Relief Valve option

Tutorial – 3

Valve-Controlled Pressurization of a Propellant Tank



Fluid Transient Schematic

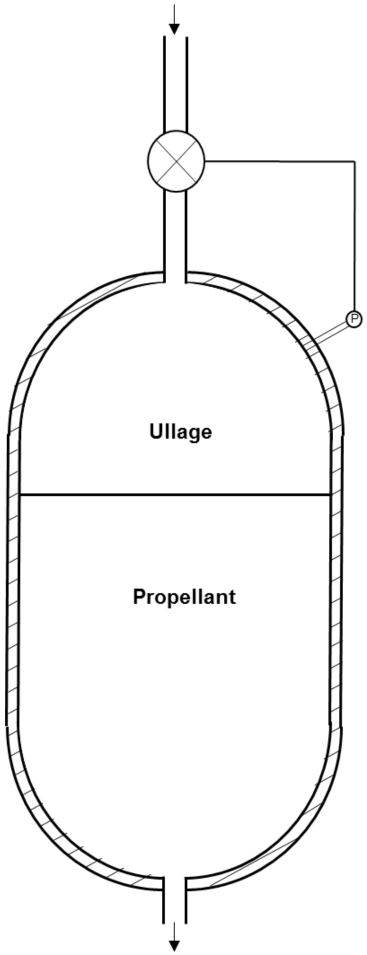
Problem Elements

- Control tank pressure within a specified tolerance

- Use control valve branch option

- Use tank pressurization advanced option

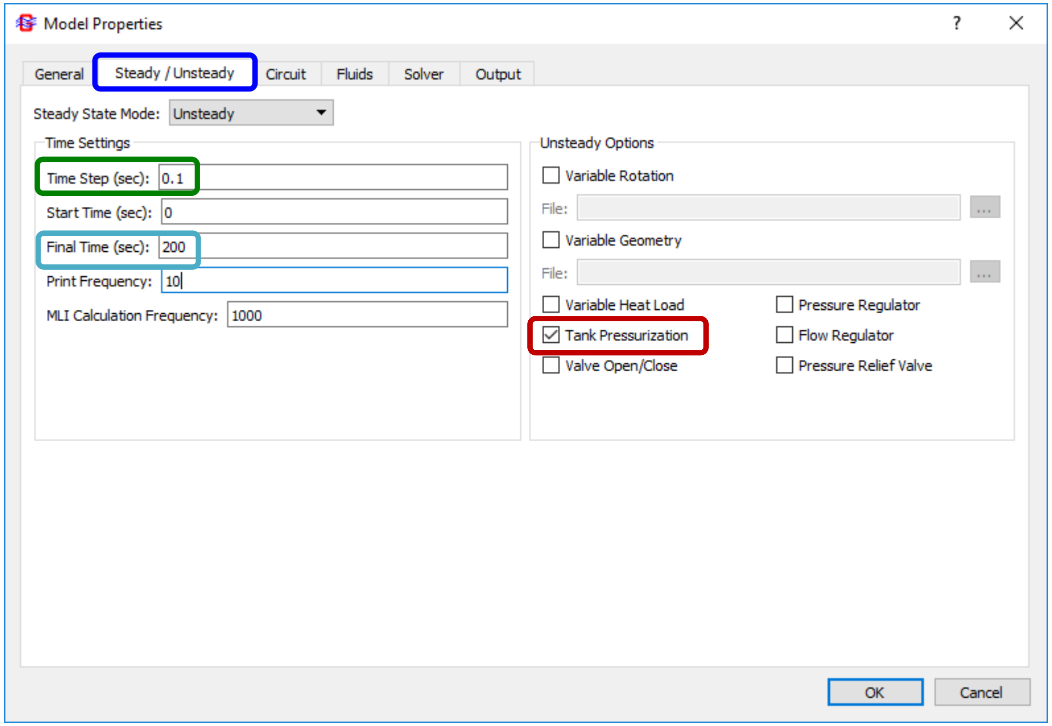
- Use 2 fluids (oxygen and helium)



Set Up Options (1/2)

- General
  - Model File: Tut3.gfssp
  - Input File: Tut3.DAT
  - Output File: Tut3.OUT

- Steady / Unsteady
  - Time step = 0.1 s
  - Final time = 200 s
  - Check Tank Pressurization



Steady State Mode: Unsteady

Time Step (sec): 0.1

Start Time (sec): 0

Final Time (sec): 200

Print Frequency: 10

MLI Calculation Frequency: 1000

☐ Variable Rotation

File:

☐ Variable Geometry

File:

☐ Variable Heat Load

☑ Tank Pressurization

☐ Valve Open/Close

☐ Pressure Regulator

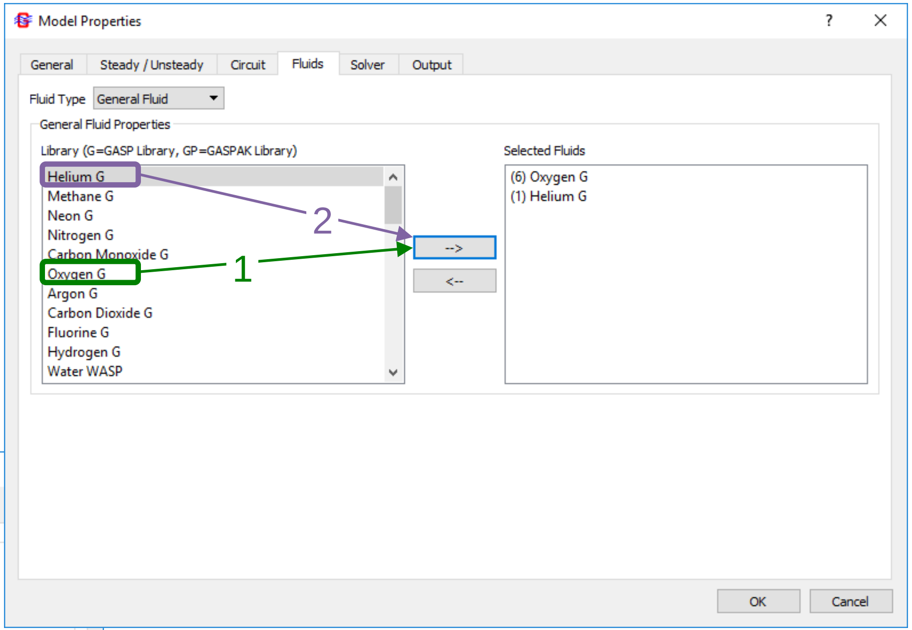
☐ Flow Regulator

☐ Pressure Relief Valve

Set Up Options (2/2)

- Fluids
  - Select Oxygen (first)
  - Select Helium (second)

- Output
  - Select Winplot binary output



Model Properties

Fluid Type: General Fluid

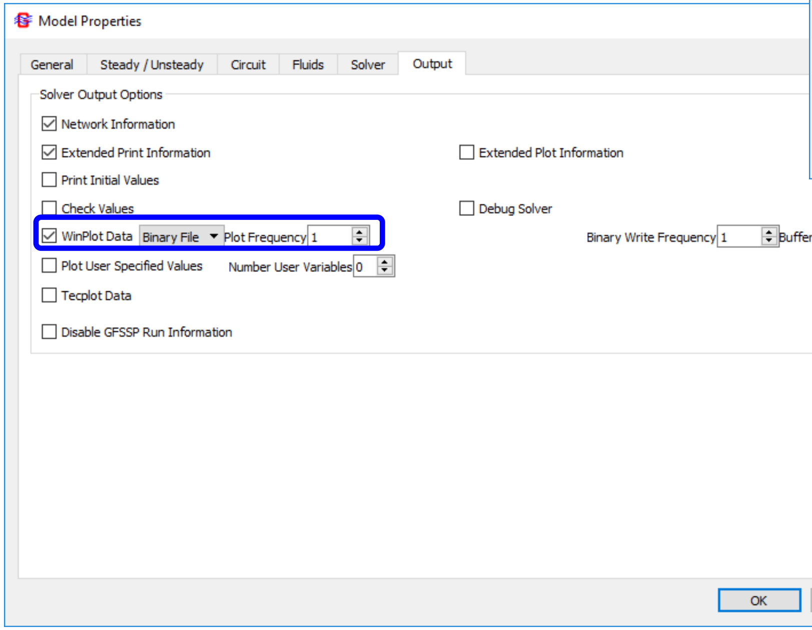
General Fluid Properties

Library (G=GASP Library, GP=GASPAK Library)

Helium G
Methane G
Neon G
Nitrogen G
Carbon Monoxide G
Oxygen G
Argon G
Carbon Dioxide G
Fluorine G
Hydrogen G
Water WASP

Selected Fluids

(6) Oxygen G
(1) Helium G



Model Properties

Solver Output Options

☑ Network Information

☑ Extended Print Information ☐ Extended Plot Information

☐ Print Initial Values

☐ Check Values ☐ Debug Solver

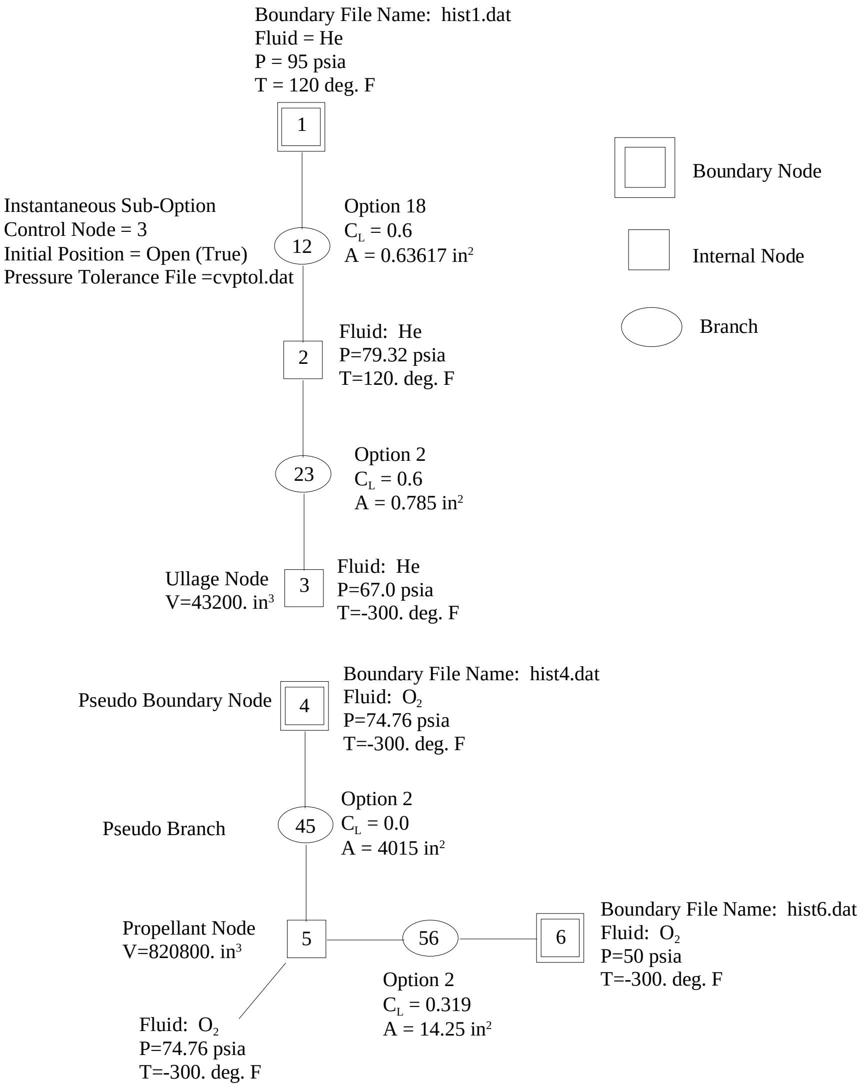
☑ WinPlot Data: Binary File, Plot Frequency: 1 — Binary Write Frequency: 1, Buffer: 1.1

☐ Plot User Specified Values — Number User Variables: 0

☐ Tecplot Data

☐ Disable GFSSP Run Information

Build Model on Canvas



Boundary File Name: hist1.dat
Fluid = He
P = 95 psia
T = 120 deg. F

Instantaneous Sub-Option
Control Node = 3
Initial Position = Open (True)
Pressure Tolerance File =cvptol.dat

Option 18
C_L = 0.6
A = 0.63617 in^2

Fluid: He
P=79.32 psia
T=120. deg. F

Option 2
C_L = 0.6
A = 0.785 in^2

Ullage Node
V=43200. in^3

Fluid: He
P=67.0 psia
T=-300. deg. F

Pseudo Boundary Node

Boundary File Name: hist4.dat
Fluid: O_2
P=74.76 psia
T=-300. deg. F

Pseudo Branch

Option 2
C_L = 0.0
A = 4015 in^2

Propellant Node
V=820800. in^3

Fluid: O_2
P=74.76 psia
T=-300. deg. F

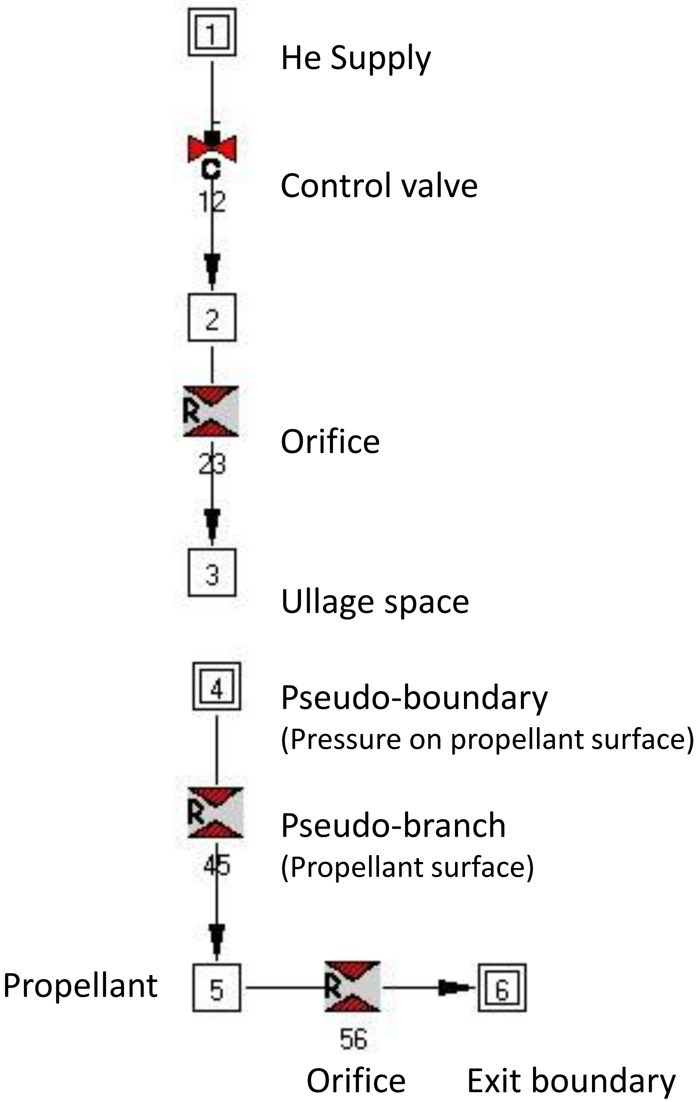
Option 2
C_L = 0.319
A = 14.25 in^2

Boundary File Name: hist6.dat
Fluid: O_2
P=50 psia
T=-300. deg. F

Boundary Node

Internal Node

Branch



He Supply

Control valve

Orifice

Ullage space

Pseudo-boundary
(Pressure on propellant surface)

Pseudo-branch
(Propellant surface)

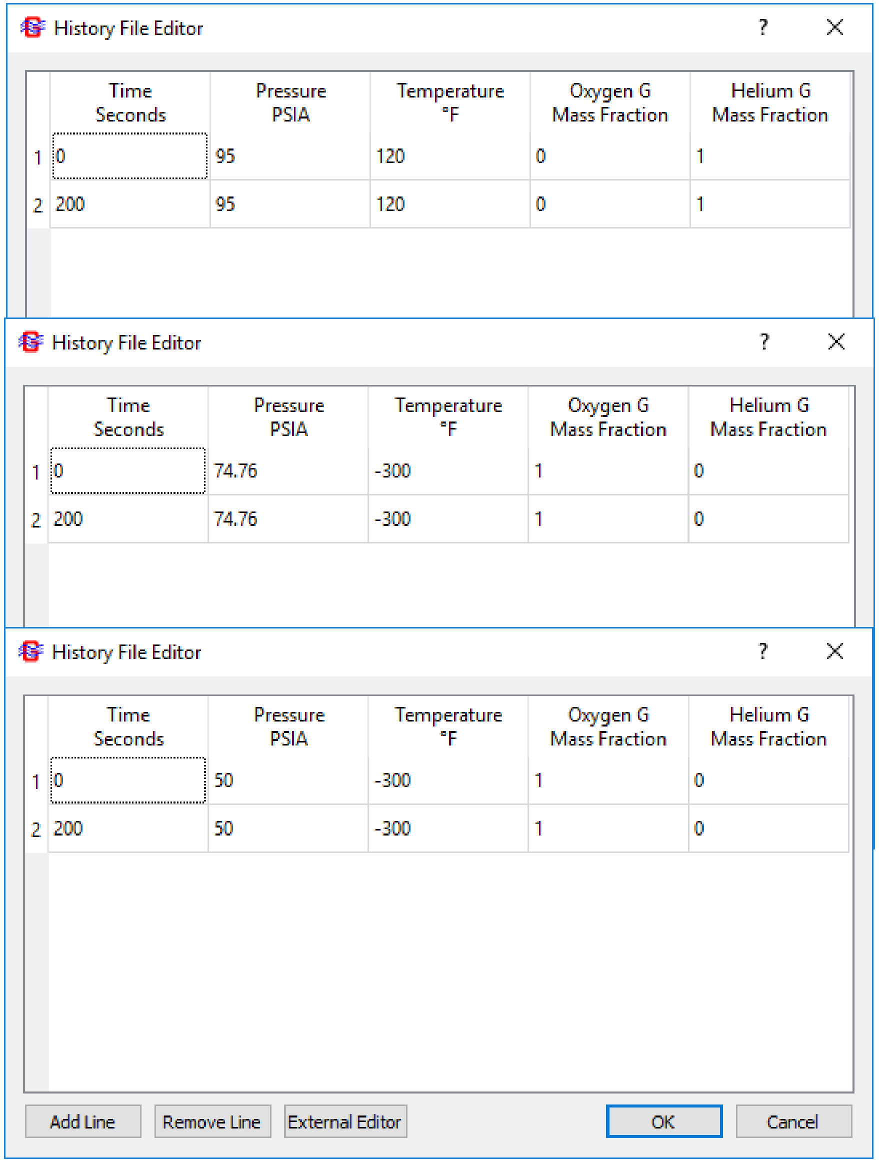
Propellant

Orifice

Exit boundary

Set Up Boundary Nodes

- Node 1 is the helium supply
  - P = 95 psia; T = 120 °F
  - LO_2 mass fraction = 0.0
  - He mass fraction = 1.0
- Node 4 is a pseudo-boundary node
  - It separates the He from the LO_2
  - History file is required
    - ➢ Pressure will be overwritten by Node 3 ullage pressure plus propellant head
  - P = 74.76 psia; T = -300 °F
  - LO_2 mass fraction = 1.0
  - He mass fraction = 0.0
- Node 6 is the LO_2 exit boundary
  - P = 50 psia; T = -300 °F
  - LO_2 mass fraction = 1.0
  - He mass fraction = 0.0



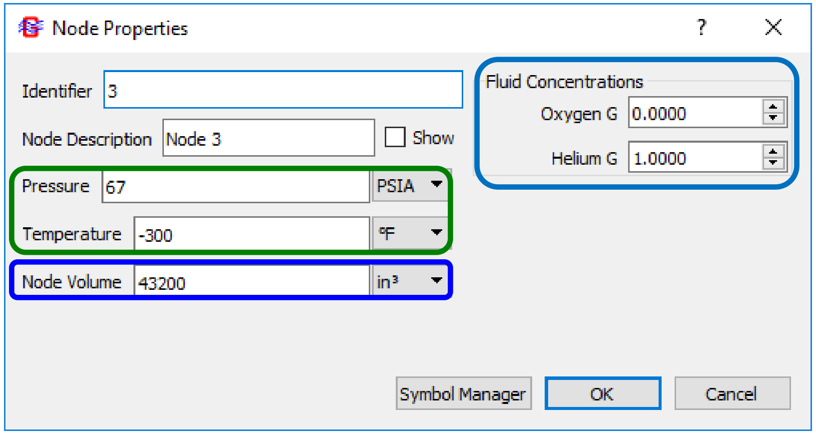
	Time Seconds	Pressure PSIA	Temperature °F	Oxygen G Mass Fraction	Helium G Mass Fraction
1	0	95	120	0	1
2	200	95	120	0	1

	Time Seconds	Pressure PSIA	Temperature °F	Oxygen G Mass Fraction	Helium G Mass Fraction
1	0	74.76	-300	1	0
2	200	74.76	-300	1	0

	Time Seconds	Pressure PSIA	Temperature °F	Oxygen G Mass Fraction	Helium G Mass Fraction
1	0	50	-300	1	0
2	200	50	-300	1	0

Set Up Internal Nodes

- Node 3 represents the ullage space
  - Initial P = 67 psia; T = -300.0 °F
  - Initial Volume = 43200 in^3
  - He fraction = 1.0
  - LO_2 fraction = 0.0
- Node 5 represents the propellant space
  - Initial P = 74.76 psia; T = -300.0 °F
  - Initial Volume = 820800 in^3
  - LO_2 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
  - LO_2 fraction = 0.0



Node Properties

Identifier: 3

Node Description: Node 3 ☐ Show

Pressure: 67 PSIA

Temperature: -300 °F

Node Volume: 43200 in³

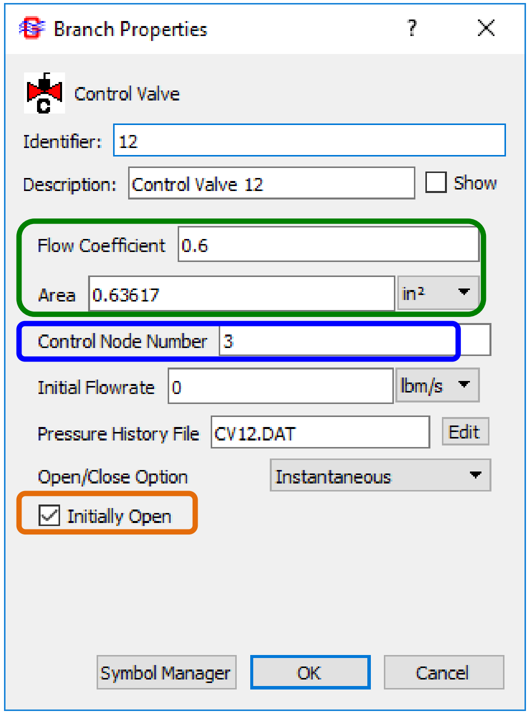
Fluid Concentrations

Oxygen G: 0.0000

Helium G: 1.0000

Set Up Branches

- Branch 12 - an Instantaneous Control Valve
  - C_L = 0.6; A = 0.63617 in^2
  - Controlled by pressure in Node 3
    - ➢ 70 psia – close
    - ➢ 64 psia – open
  - Valve is initially open
  - Requires a history file
- Branch 23 - 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



Identifier: 12

Description: Control Valve 12 ☐ Show

Flow Coefficient 0.6

Area 0.63617 in²

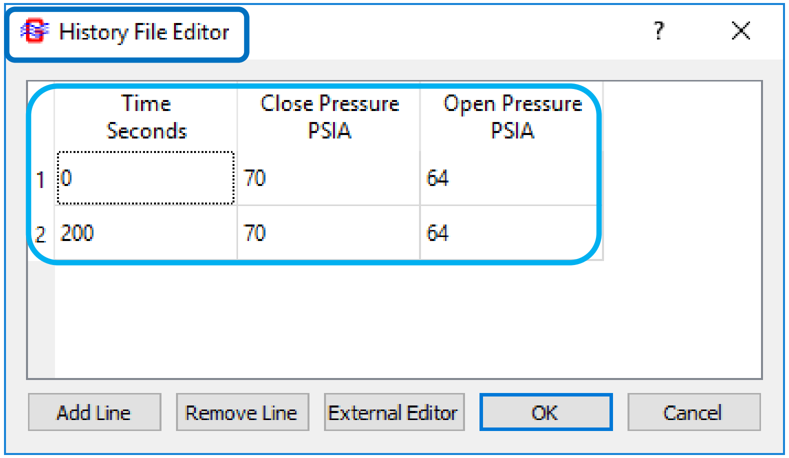
Control Node Number 3

Initial Flowrate 0 lbm/s

Pressure History File CV12.DAT

Open/Close Option Instantaneous

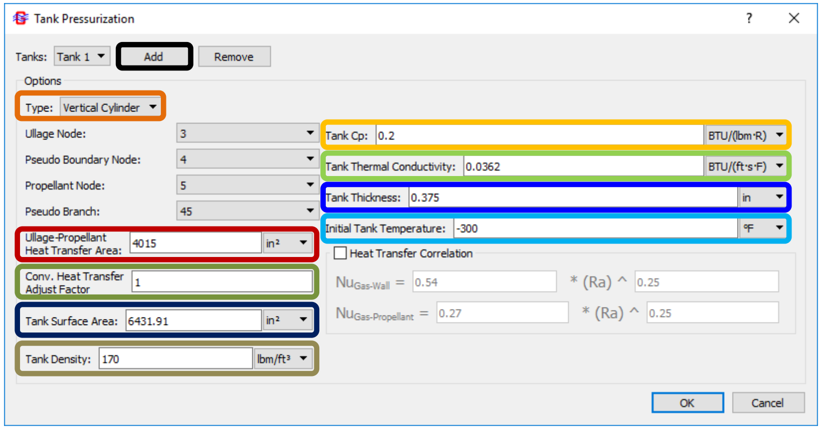
☑ Initially Open



	Time Seconds	Close Pressure PSIA	Open Pressure PSIA
1	0	70	64
2	200	70	64

Tank Pressurization Option

- Open Tank Pressurization dialog from Advanced menu
  - Click “Add”
  - Type: Vertical Cylinder (aluminum tank)
  - Ull.-Prop. Heat Transfer Area: 4015 in^2
  - Conv. Heat Transfer Adj. Factor: 1.0
  - Tank Surface Area*: 6431.91 in^2
  - Density: 170. lb$_m$/ft^3
  - Specific Heat: 0.2 Btu/lb$_m$-R
  - Thermal Conductivity: 0.0362 Btu/ft-s-°F
  - Wall Thickness: 0.375 in.
  - T$_{tank}$: -300 °F
  - Use default convection correlation coefficients



Tanks: Tank 1

Options

Type: Vertical Cylinder

Ullage Node: 3

Pseudo Boundary Node: 4

Propellant Node: 5

Pseudo Branch: 45

Ullage-Propellant Heat Transfer Area: 4015 in²

Conv. Heat Transfer Adjust Factor: 1

Tank Surface Area: 6431.91 in²

Tank Density: 170 lbm/ft³

Tank Cp: 0.2 BTU/(lbm·R)

Tank Thermal Conductivity: 0.0362 BTU/(ft·s·F)

Tank Thickness: 0.375 in

Initial Tank Temperature: -300 ºF

☐ Heat Transfer Correlation

$Nu_{Gas-Wall}$ = 0.54 * (Ra) ^ 0.25

$Nu_{Gas-Propellant}$ = 0.27 * (Ra) ^ 0.25

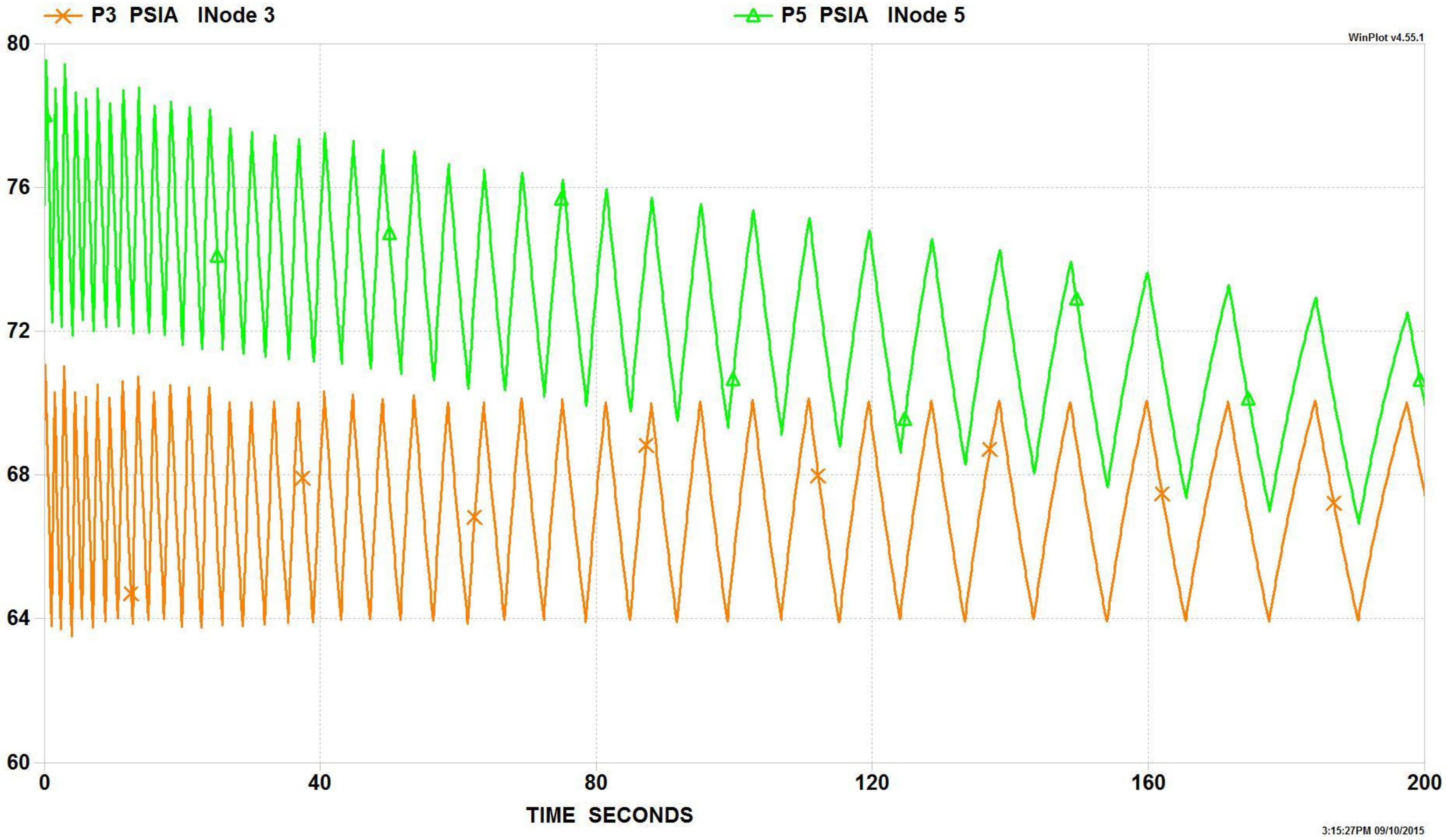
*Tank wall surface area initially exposed to ullage. It will automatically increase as the tank drains.

Study of the Results

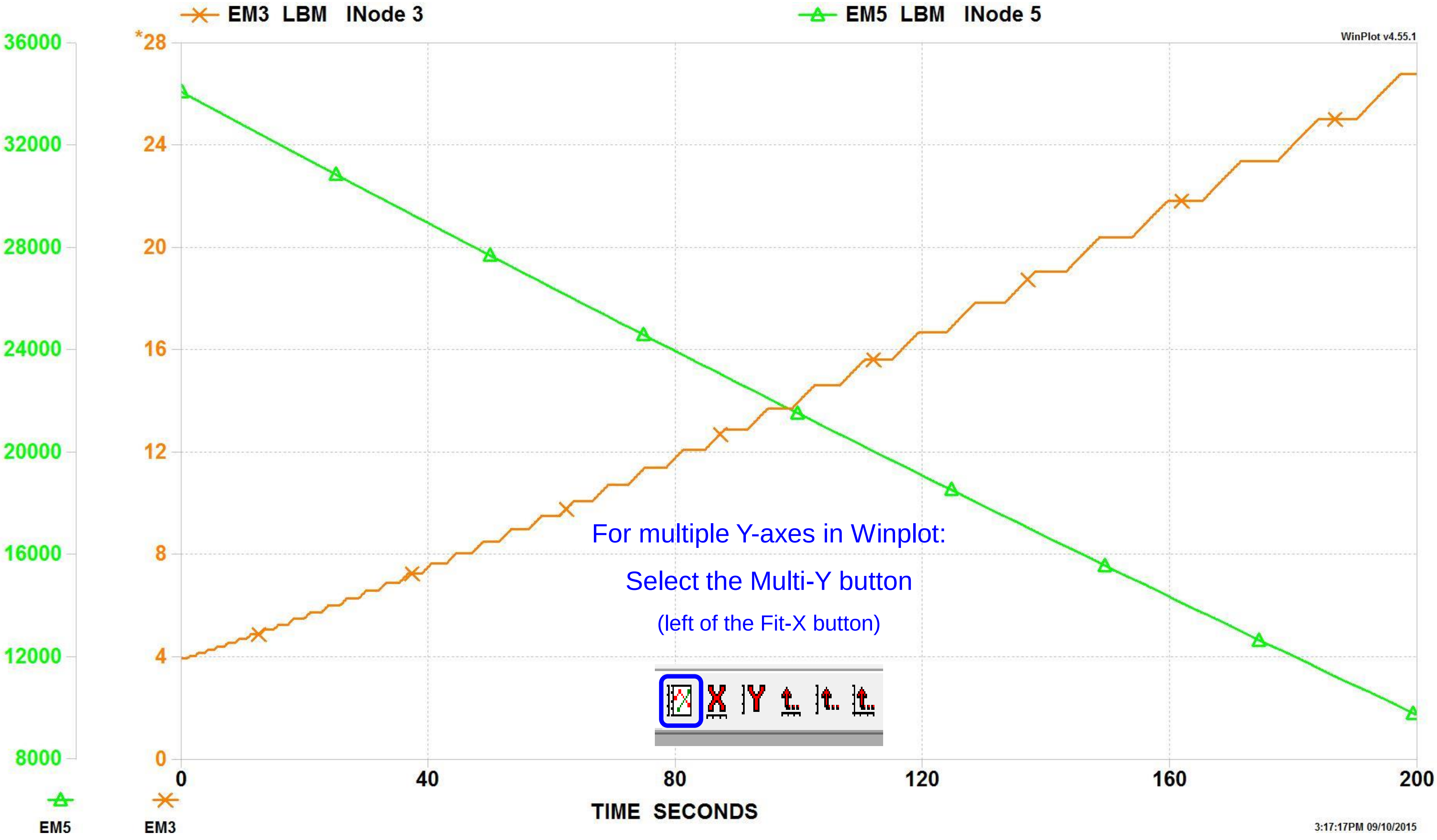
- Study *tut3.out* and *plot files* to note the following facts:
  - Ullage pressure is maintained between 64 and 70 psia by the control valve
  - Difference between ullage pressure and tank bottom pressure due to gravitational head
  - Tank bottom pressure decreases as propellant is expelled from the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

- If you finish early:
  - Re-run the model with increased heat transfer
    - Set the Heat Transfer Adjustment Factor to 2
  - What effect does this have on the valve cycling time and the final helium mass in the ullage node?

Tank Pressure History



Tank Mass History



For multiple Y-axes in Winplot:

Select the Multi-Y button

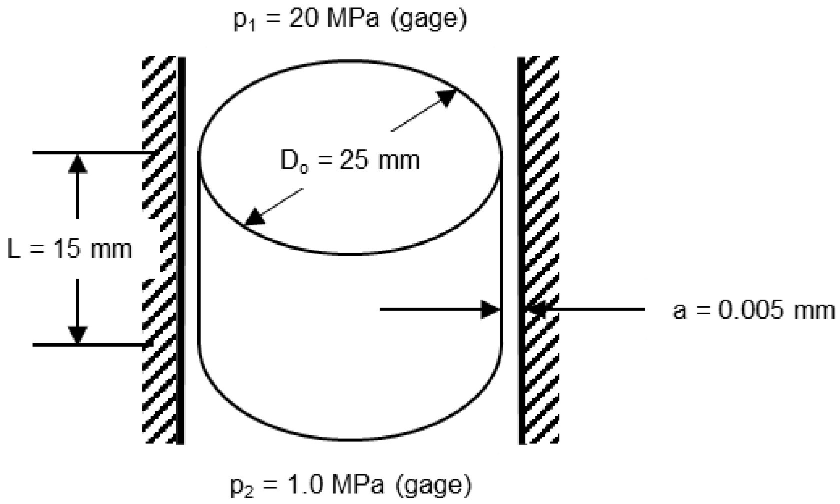
(left of the Fit-X button)

Challenge Problem 3 (1/2)

Leakage Flow Past a Piston

Given:

A hydraulic system operates at a pressure of 20 MPa. The hydraulic fluid is SAE 10W oil (density = 920 kg/m^3, viscosity at 55 °C = 0.018 N-s/m^2). A control valve consists of a piston 25 mm in diameter, fitted to a cylinder with a mean radial clearance of 5 microns.



Determine: The leakage flow rate if the pressure on the low-pressure side of the piston is 1.0 MPa. The piston is 15 mm long.

Challenge Problem 3 (2/2)
Leakage Flow Past a Piston

- There are two ways to work with SI units
  - Click File / Preferences, then “Default SI”, close and restart **MIG**
  - On the General tab of the Model Properties page, select SI for History Files and Output, then manually change each entry to SI units as you enter them into the dialog boxes.

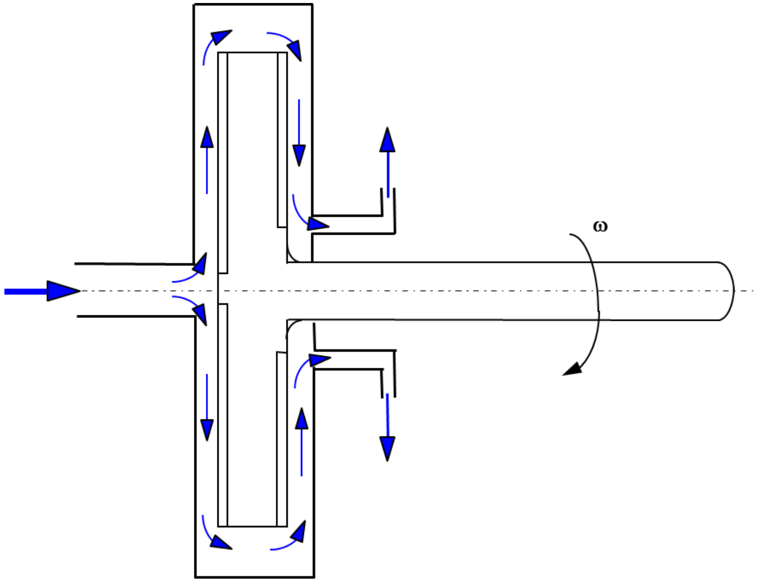
- The Face Seal branch option can be used to model laminar flow through a tight clearance. Note that it asks for radius, not diameter.

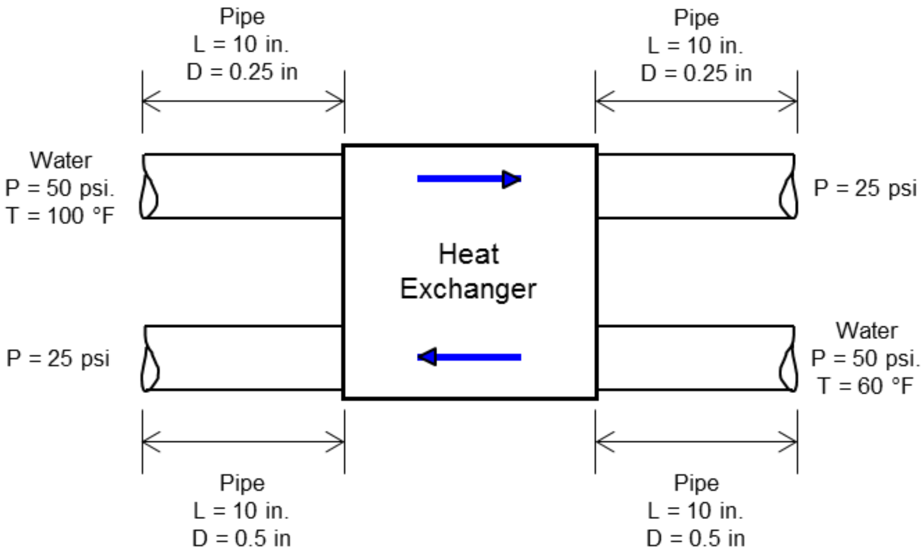
- The Concentric Annulus branch option will also work in this model.

- **GFSSP** requires at least one internal node between the two boundaries. The single flow resistance can be broken up into two identical branches, each with half the cylinder length.

- This problem is Example 8.1 in *Introduction to Fluid Mechanics*, 4th ed., by Fox and McDonald
  - The velocity given in the text is 0.147 m/s, and the volumetric flow rate (mass flow rate / density) is 57.6 mm^3/s.
  - How does **GFSSP**’s answer compare?

Rotating Flow, Turbopump, and Heat Exchanger





Content

- Centrifugal Force
  - Example 6: Radial Flow on a Rotating Radial Disk

- Axial Thrust
  - Example 21: Axial Thrust Calculation in a Turbopump
  - FASTRAC Turbopump

- Turbopump Option
  - Example 11: Power Balancing of a Turbopump Assembly

- Heat Exchanger
  - Example 5: Simulation of a Flow System Involving a Heat Exchanger
  - Example 20: Simulation of a Lithium Loop Model

- 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 ---------------------------

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

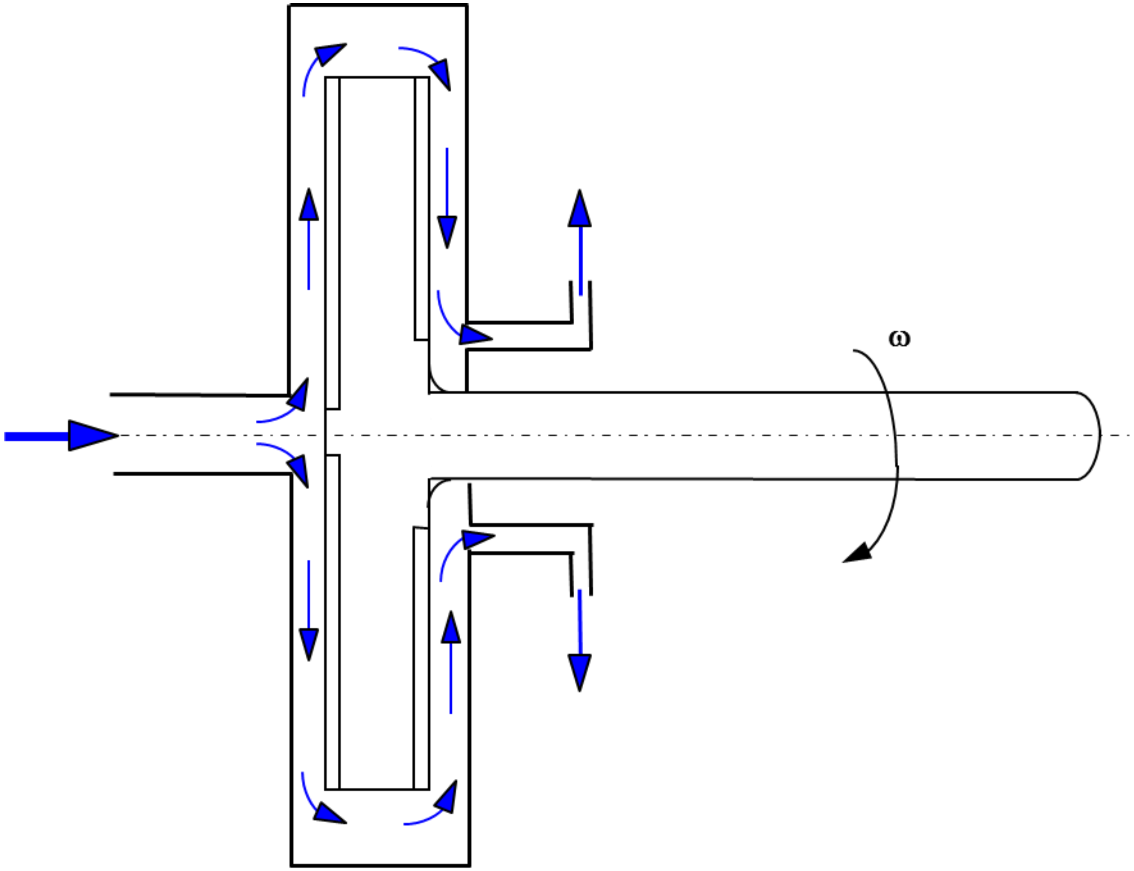
-- Pressure -- -- Gravity -- -- Friction -- **-- Centrifugal --** -- Shear--

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

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

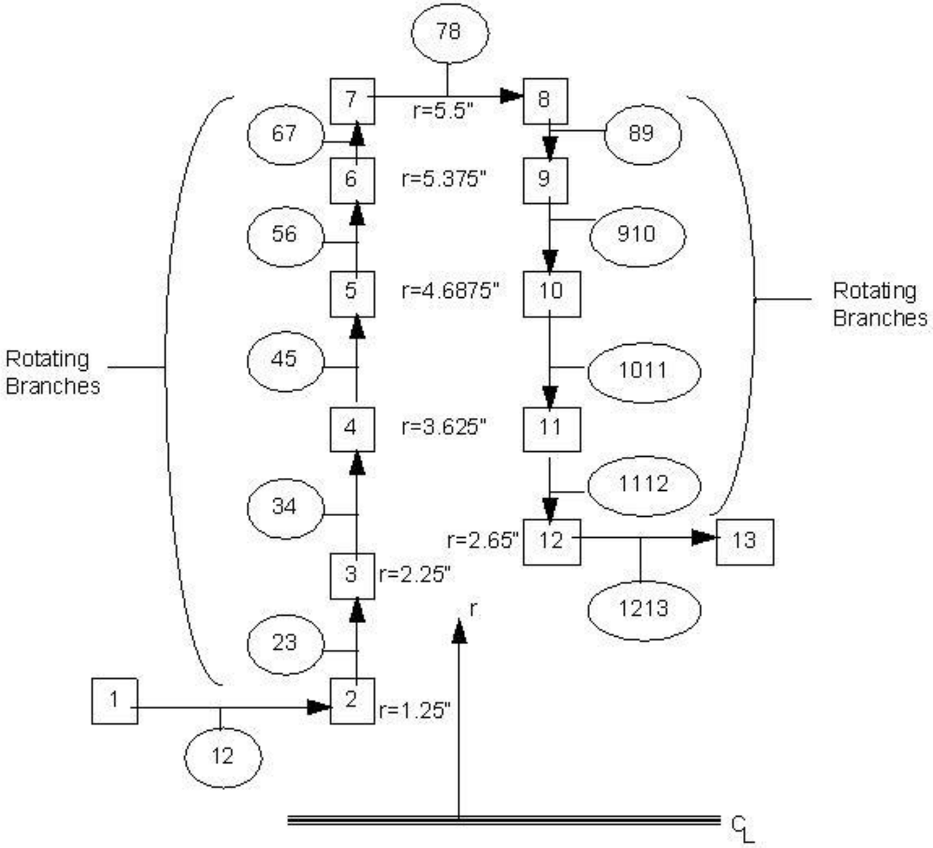
Ex6 – Radial Flow on a Rotating Radial Disk (1/4)

- Features
  - Rotating Flows
  - Comparison with Textbook Solution

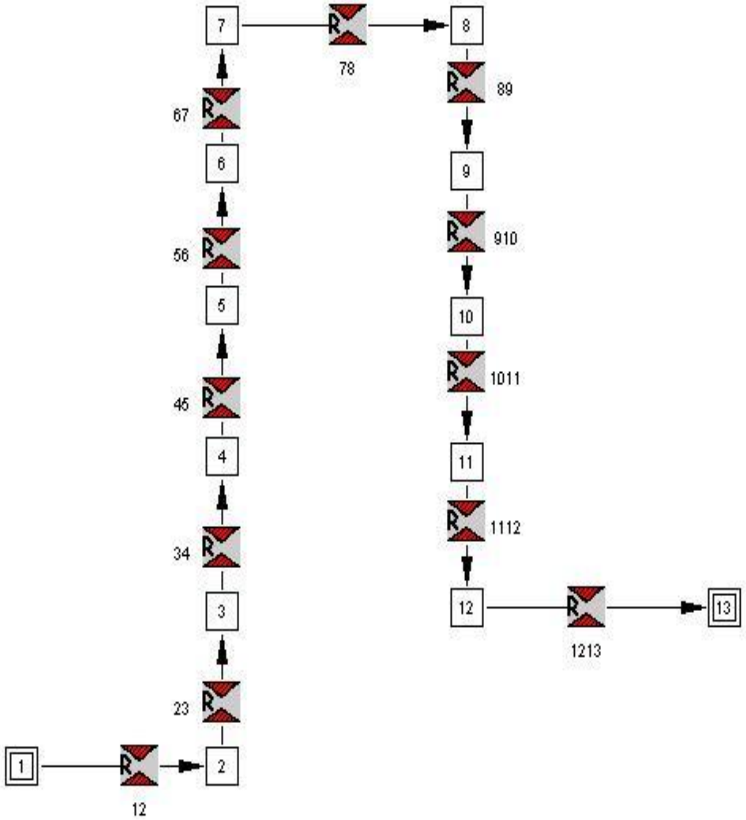


Ex6 – Radial Flow on a Rotating Radial Disk (2/4)

Detailed Schematic

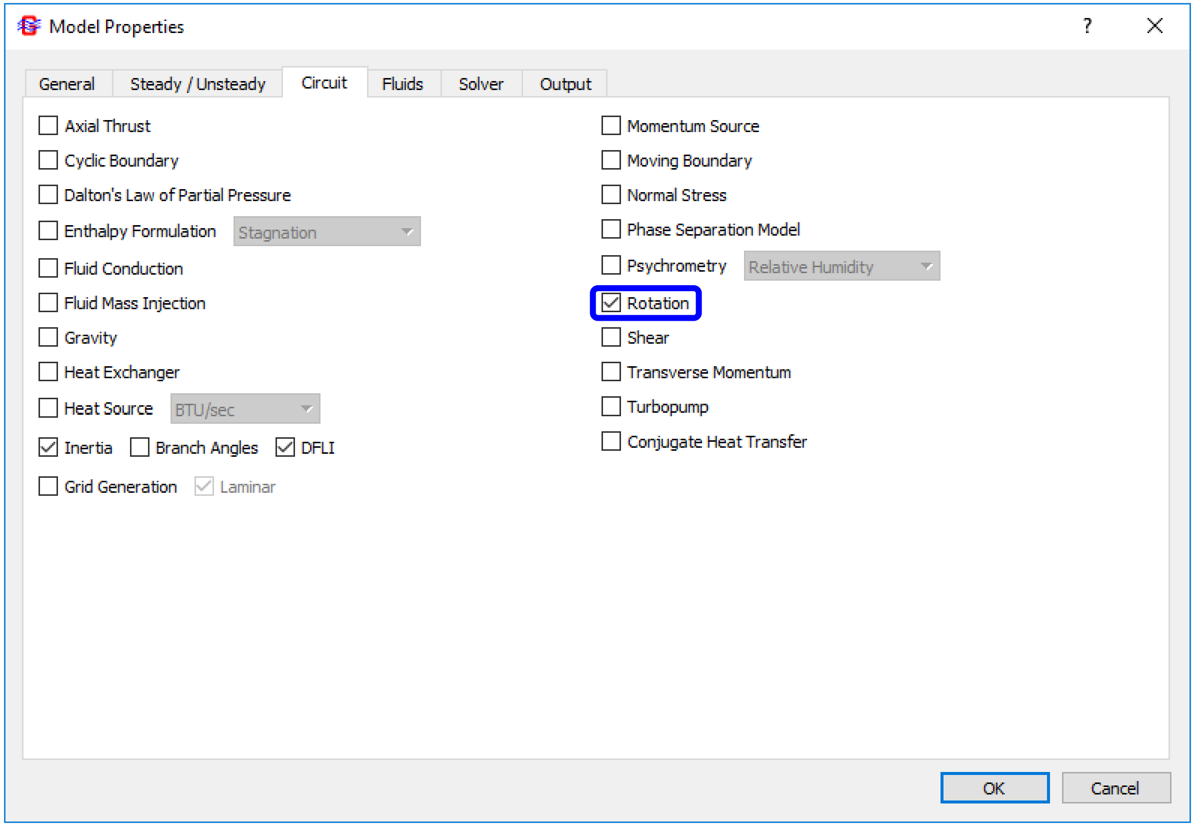


MIG Model



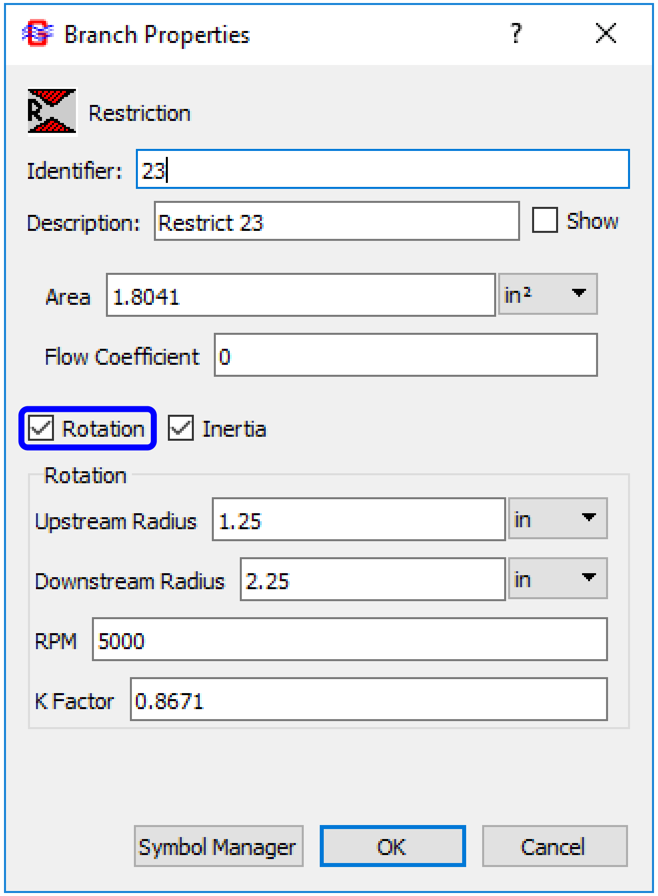
Ex6 – Radial Flow on a Rotating Radial Disk (3/4)

- Activation of Rotational term in **MIG**



☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation: Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☐ Heat Exchanger
☐ Heat Source: BTU/sec
☑ Inertia ☐ Branch Angles ☑ DFLI
☐ Grid Generation ☑ Laminar

☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry: Relative Humidity
☑ Rotation
☐ Shear
☐ Transverse Momentum
☐ Turbopump
☐ Conjugate Heat Transfer

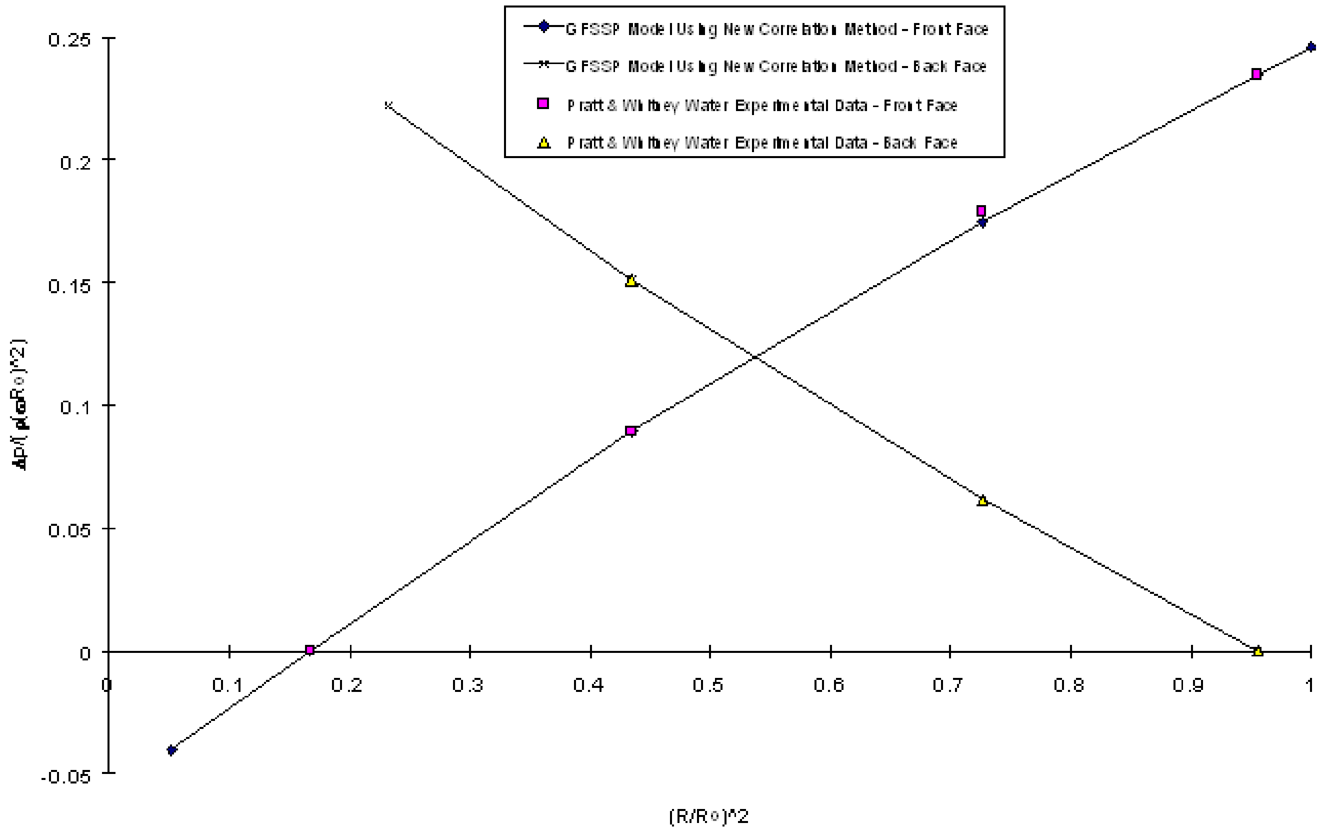


Identifier: 23
Description: Restrict 23 ☐ Show
Area 1.8041 in²
Flow Coefficient 0
☑ Rotation ☑ Inertia

Rotation
Upstream Radius 1.25 in
Downstream Radius 2.25 in
RPM 5000
K Factor 0.8671

Ex6 – Radial Flow on a Rotating Radial Disk (4/4)

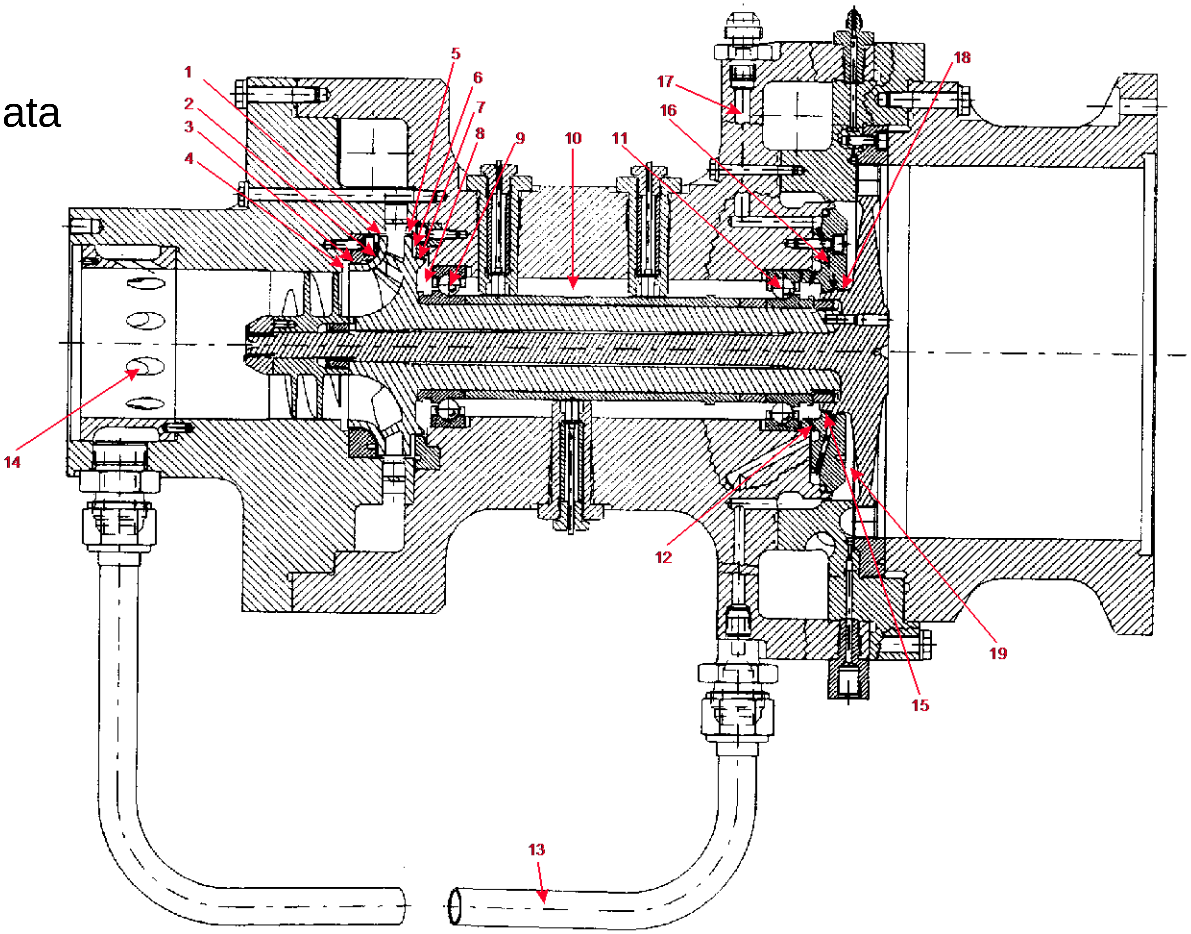
- Comparison of **GFSSP** Model Results with Experimental Data



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

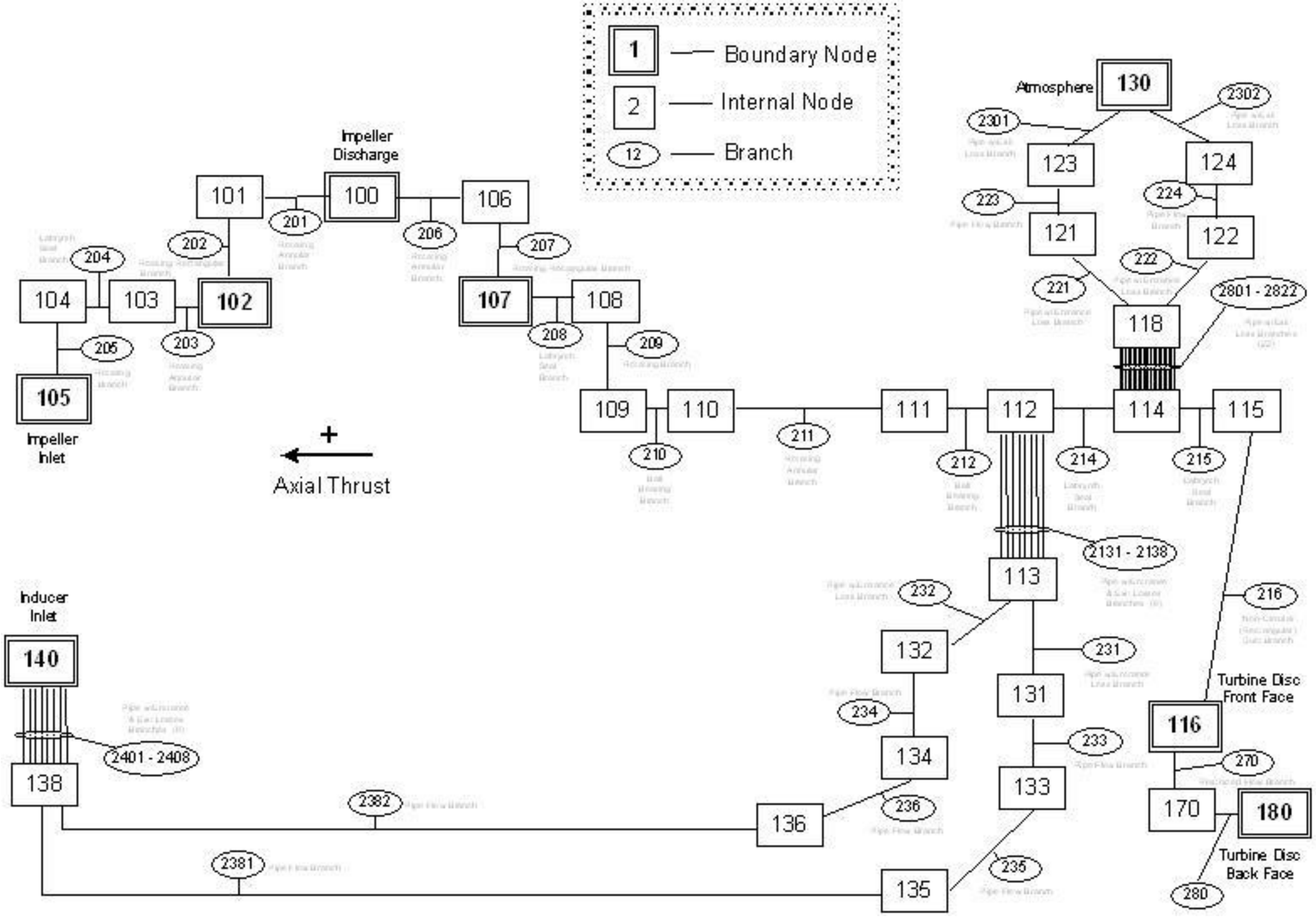
Ex21 – Axial Thrust Calculation in a Turbopump (1/5)

- Features
  - Axial Thrust
  - Rotating Flow
  - Parallel Tube
  - Comparison with test data



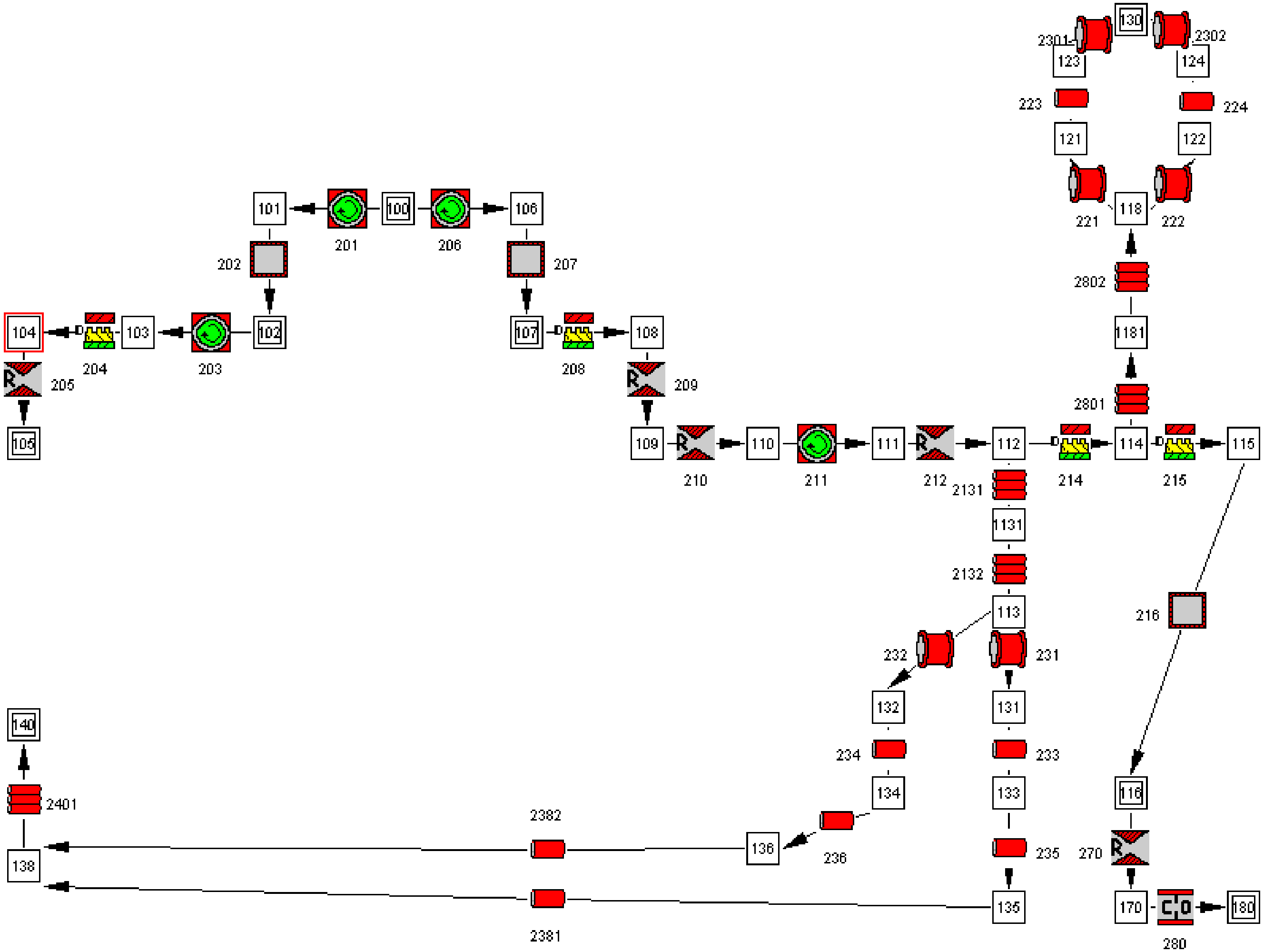
Ex21 – Axial Thrust Calculation in a Turbopump (2/5)

- Simplex Turbopump Detailed Model



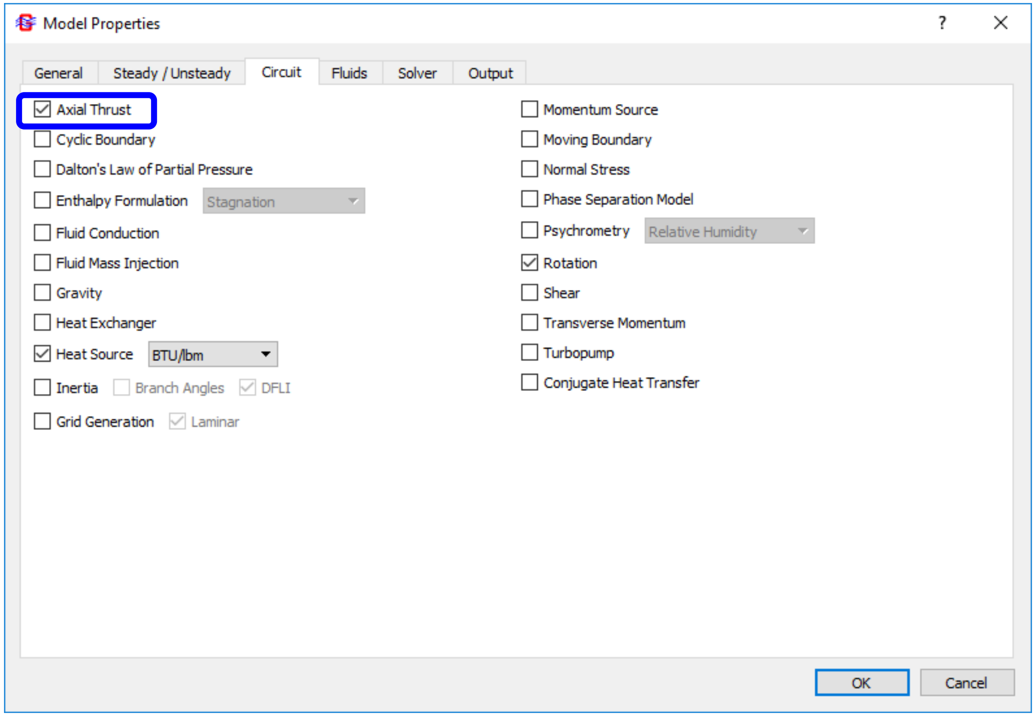
Ex21 – Axial Thrust Calculation in a Turbopump (3/5)

- Simplex Turbopump **MIG** Model



Ex21 – Axial Thrust Calculation in a Turbopump (4/5)

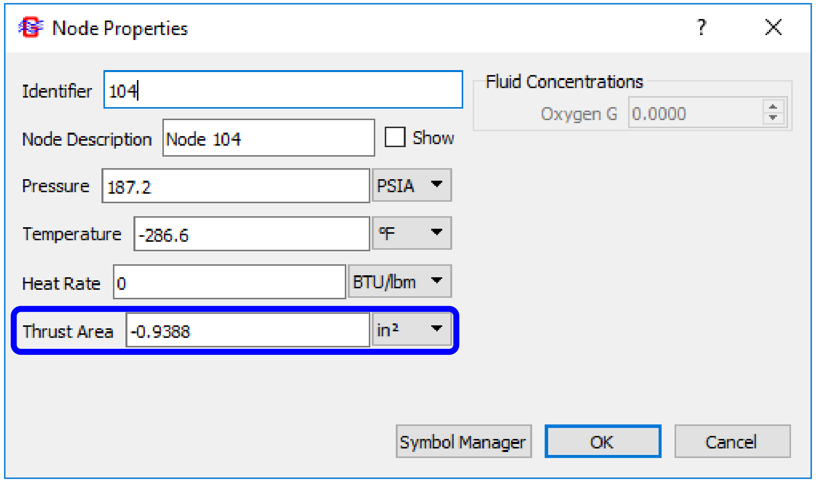
- Activation of Axial Thrust in **MIG**



Model Properties

☑ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☐ Heat Exchanger
☑ Heat Source BTU/lbm
☐ Inertia ☐ Branch Angles ☑ DFLI
☐ Grid Generation ☑ Laminar

☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry Relative Humidity
☑ Rotation
☐ Shear
☐ Transverse Momentum
☐ Turbopump
☐ Conjugate Heat Transfer



Node Properties

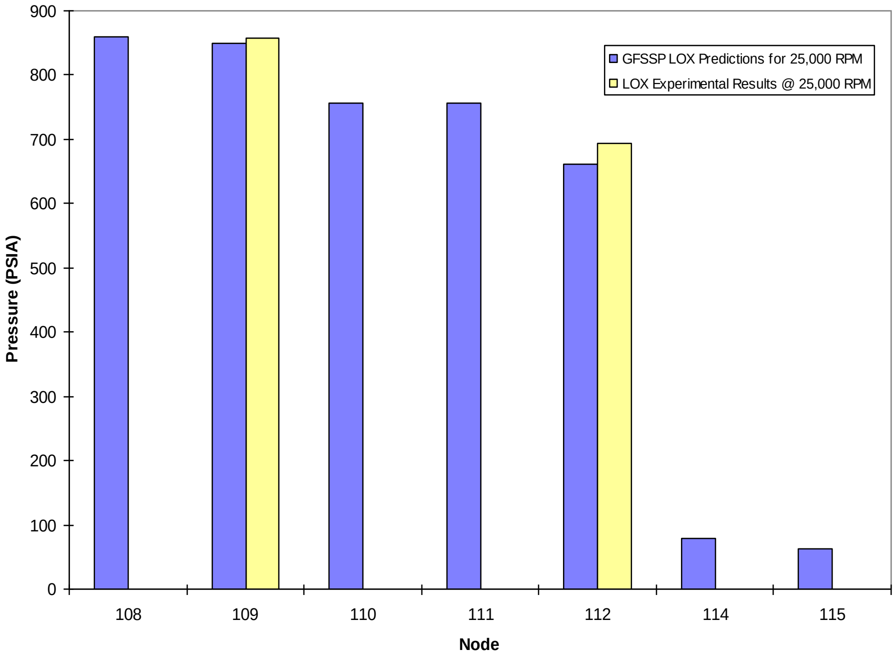
Identifier 104
Node Description Node 104 ☐ Show
Pressure 187.2 PSIA
Temperature -286.6 ºF
Heat Rate 0 BTU/lbm
Thrust Area -0.9388 in²

Fluid Concentrations
Oxygen G 0.0000

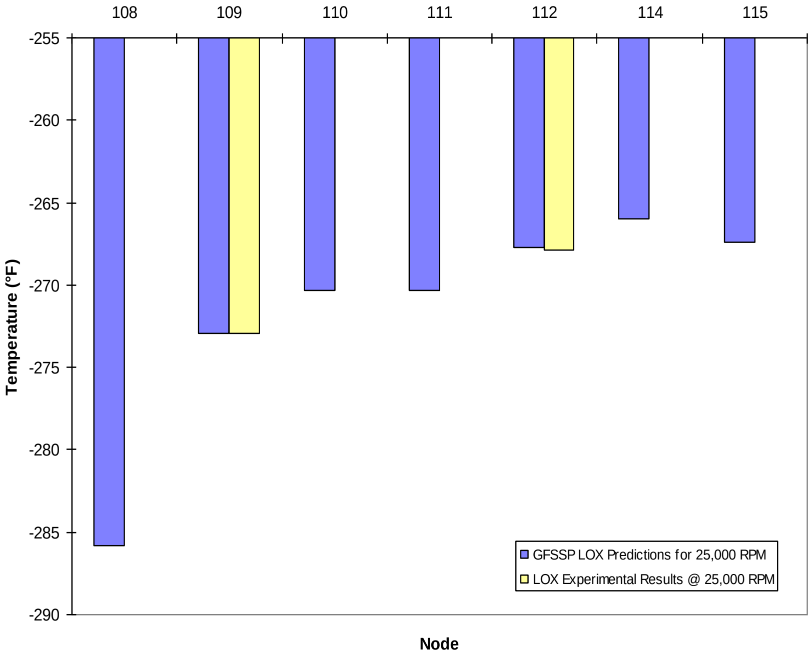
Ex21 – Axial Thrust Calculation in a Turbopump (5/5)

- Comparisons with Experimental Data

Pressure Predictions
Compared to Experimental Data

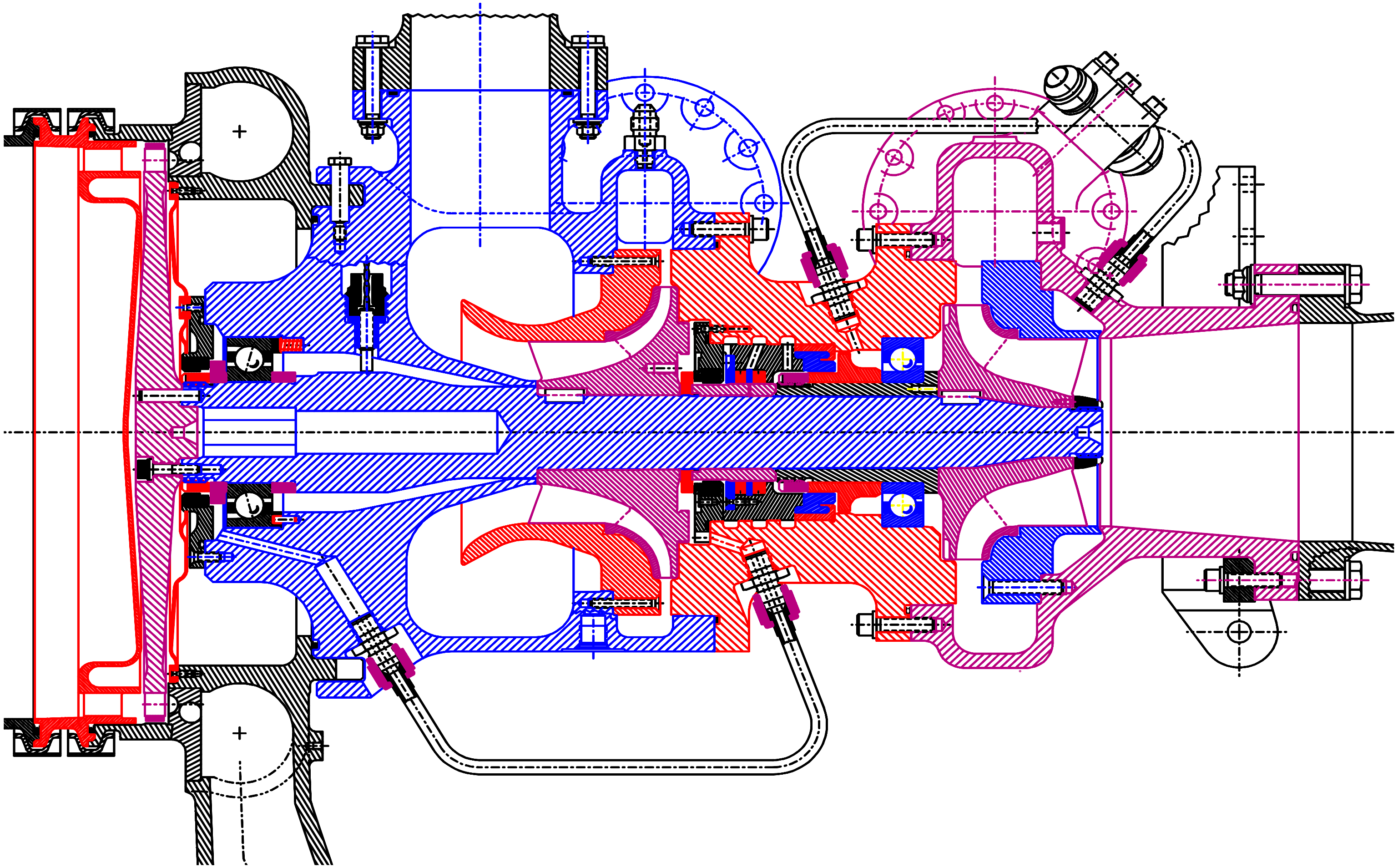


Temperature Predictions
Compared to Experimental Data



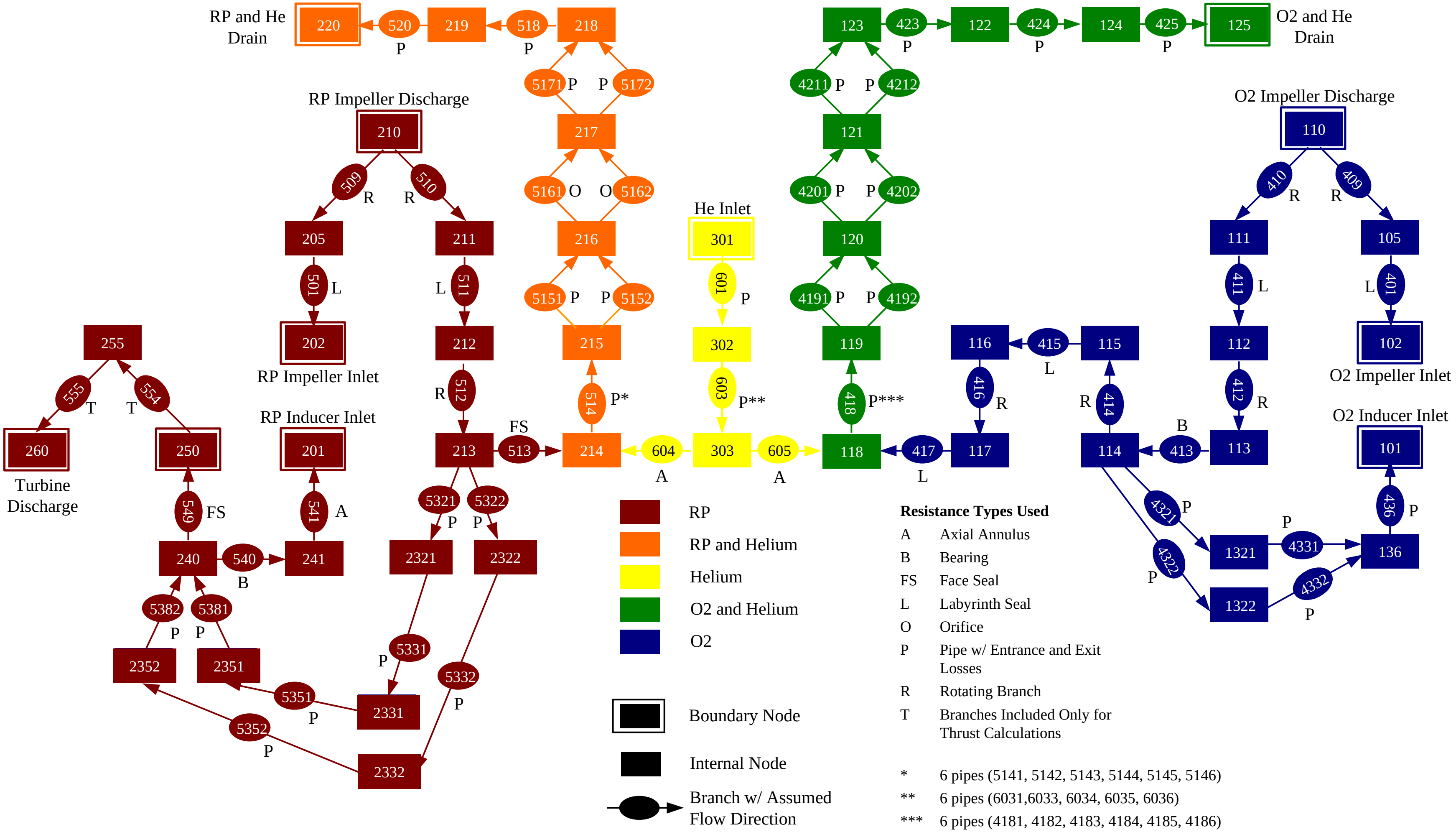
Schallhorn, Paul, Majumdar, Alok, Van Hooser, Katherine, and Marsh, Matthew, “Flow Simulation in Secondary Flow Passages of a Rocket Engine Turbopump”, Paper No. AIAA 98-3684, 34th AIAA/ASME/SAE/ASEE, Joint Propulsion Conference and Exhibit, July 13-15, 1998, Cleveland, OH

FASTRAC Turbopump (1/4)



FASTRAC Turbopump (2/4)

- **GFSSP** Model of the Fastrac Turbopump



🟥 RP

🟧 RP and Helium

🟨 Helium

🟩 O2 and Helium

🟦 O2

▣ Boundary Node

■ Internal Node

—●→ Branch w/ Assumed Flow Direction

Resistance Types Used

A	Axial Annulus
B	Bearing
FS	Face Seal
L	Labyrinth Seal
O	Orifice
P	Pipe w/ Entrance and Exit Losses
R	Rotating Branch
T	Branches Included Only for Thrust Calculations

*	6 pipes (5141, 5142, 5143, 5144, 5145, 5146)
**	6 pipes (6031,6033, 6034, 6035, 6036)
***	6 pipes (4181, 4182, 4183, 4184, 4185, 4186)

FASTRAC Turbopump (3/4)

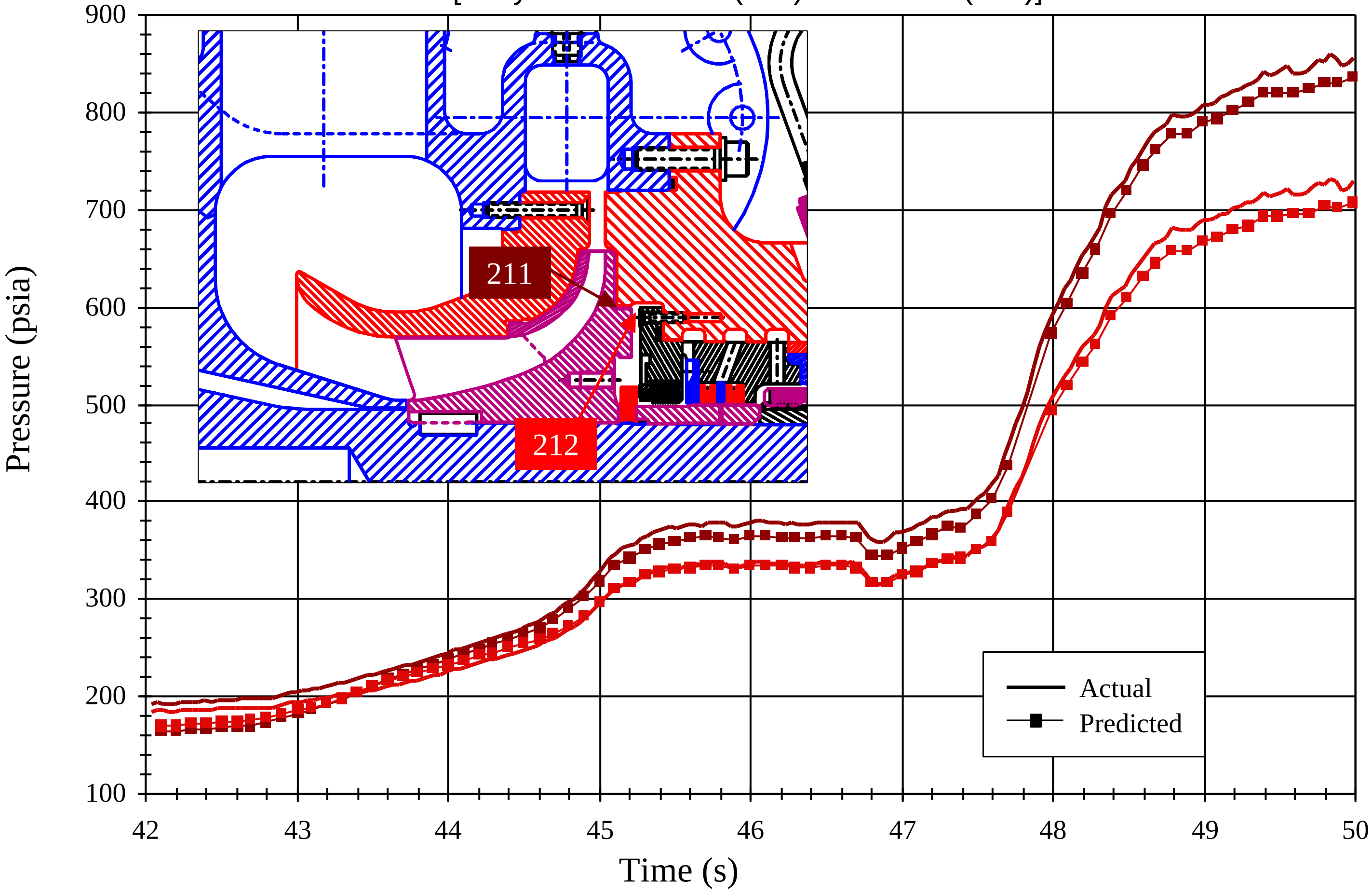
- Turbopump Test to 20000 RPM with Gas Generator

FASTRAC Turbopump (4/4)

- FASTRAC Turbopump Model Results

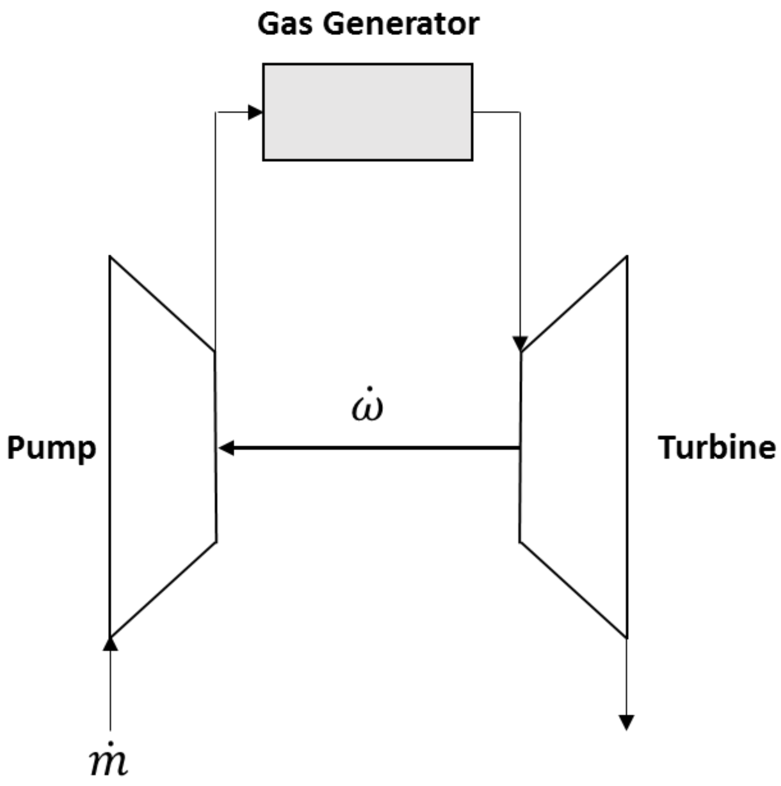
Pressure History Comparison at RP-1 Impeller Back Face

[Labyrinth Seal Inlet (**211**) and Outlet (**212**)]



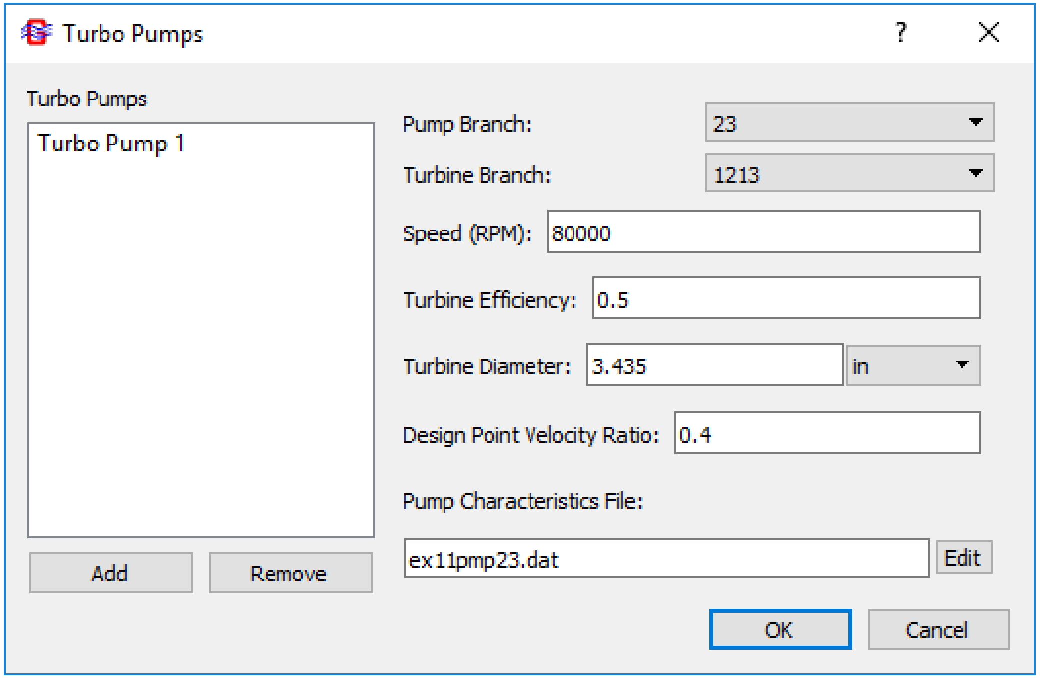
Turbopump Option (1/6)

- Objectives
  - Calculate the flowrate in a turbopump for given pump performance characteristics and speed
  - Calculate the power developed by the turbine to drive the pump



Turbopump Option (2/6)

- Number of Turbopump Assemblies
- Branches representing pump and turbine
- Rotational Speed(s)
- Pump Performance Characteristics
- Velocity Ratio and Efficiency of Turbine at Design Point



Turbo Pumps

Turbo Pump 1

Pump Branch: 23

Turbine Branch: 1213

Speed (RPM): 80000

Turbine Efficiency: 0.5

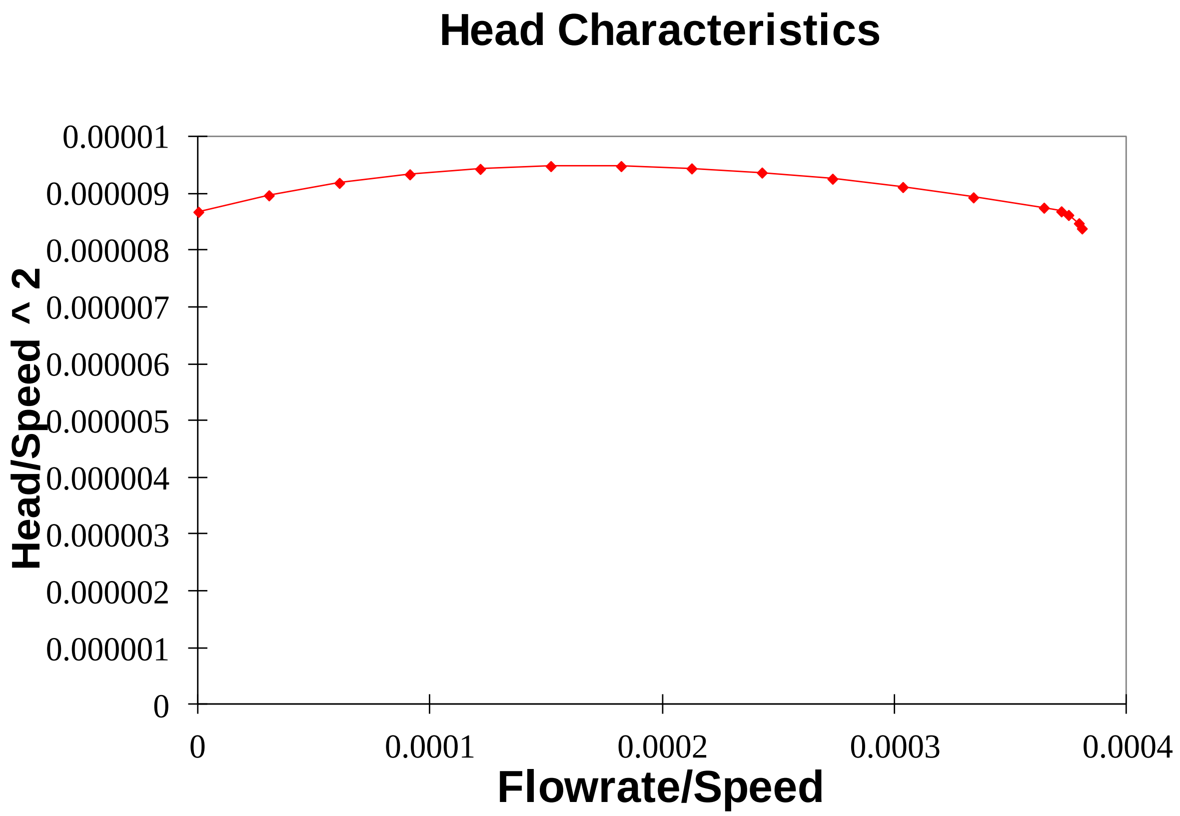
Turbine Diameter: 3.435 in

Design Point Velocity Ratio: 0.4

Pump Characteristics File: ex11pmp23.dat

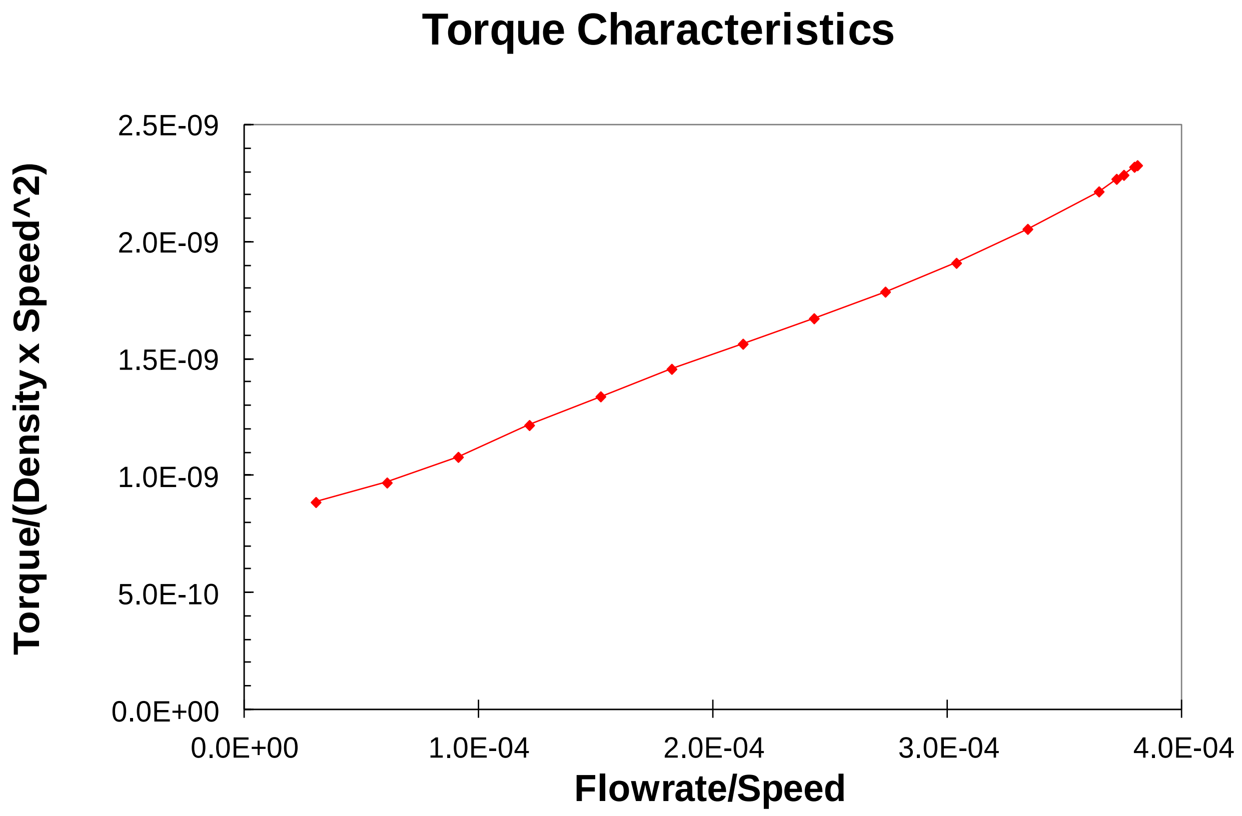
Turbopump Option (3/6)

- **GFSSP** requires Head Characteristics in the following format
  - Flowrate/Speed [GPM/RPM]
  - Head/Speed^2 [ft/RPM^2]



Turbopump Option (4/6)

- **GFSSP** requires Torque Characteristics in the following format
  - Flowrate/Speed [GPM/RPM]
  - Torque/(Density x Speed^2) [lb_f-in/(lb_m/ft^3 x RPM^2)]



Turbopump Option (5/6)

- Turbopump Model Algorithm
    - For a given flowrate
        - Calculate pressure rise across pump
        - Calculate required torque from the characteristics
    - Use this pressure rise as source in the momentum equation
    - Estimate the horsepower turbine must develop to drive the pump
    - Calculate turbine pressure ratio from turbine performance relation
    - Use this pressure drop as sink in the momentum equation

Turbopump Option (6/6)

- Turbine Performance Relationships
  - Horsepower Ratio

$$HP = \frac{2\pi\ NT}{3.96E + 05}$$

  - Pressure

$$\dot{m} = \frac{550\ HP}{\eta_T J c_p T_{T1}\left[1 - \left(\frac{1}{PR}\right)^{\gamma-1/\gamma}\right]}$$

  - Efficiency

$$\eta_T = \left((\eta_D/\varphi_D - 4)\ \varphi/\varphi_D\right)\varphi$$

where
$\varphi = U/C_0$ (Velocity Ratio)

$C_0 = \sqrt{2g_c J C_p T_{T1}\left(1 - \left(1 - \frac{1}{PR}\right)^{\gamma-1/\gamma}\right)}$ (Isentropic Spouting Velocity)

$U = \frac{D\Omega}{2}$ (Blade Speed)

Pump Characteristics

- Curve File Overview

$\frac{\text{Torque}}{(\text{Density x Speed}^2)}$

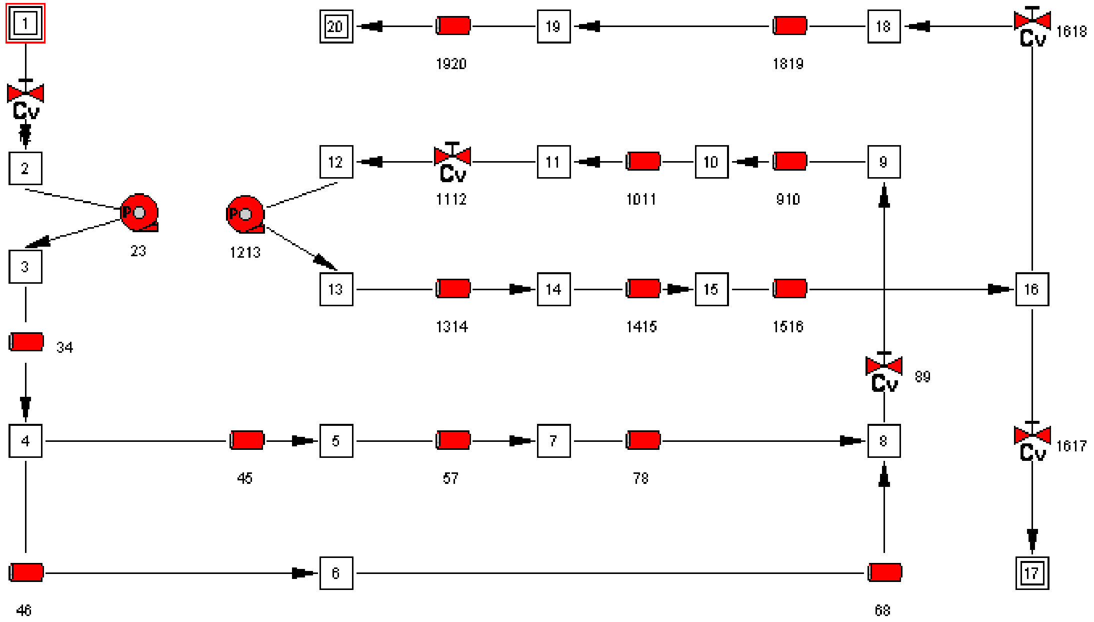
Number of Data Lines

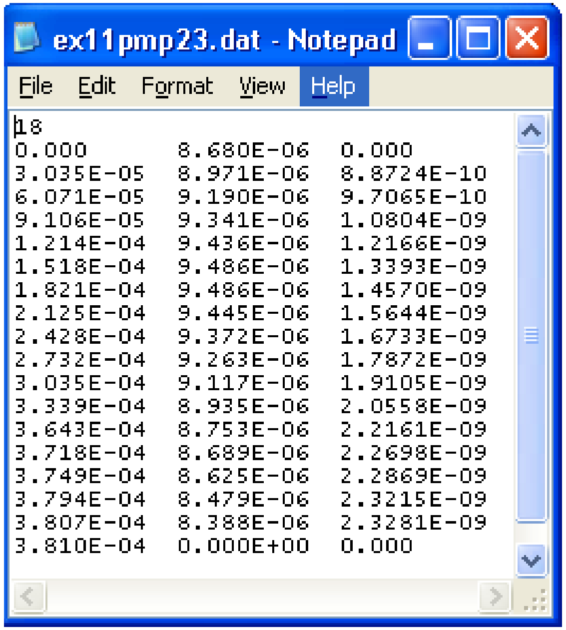
```
18
0.0000000  8.680E-06  0.00000000
3.035E-05  8.971E-06  8.8724E-10
6.071E-05  9.190E-06  9.7065E-10
9.106E-05  9.341E-06  1.0804E-09
1.214E-04  9.436E-06  1.2166E-09
1.518E-04  9.486E-06  1.3393E-09
1.821E-04  9.486E-06  1.4570E-09
2.125E-04  9.445E-06  1.5644E-09
2.428E-04  9.372E-06  1.6733E-09
```

$\frac{\text{Flowrate}}{\text{Speed}}$

$\frac{\text{Head}}{\text{Speed}^2}$

Turbopump Option – Example 11





```
18
0.000        8.680E-06    0.000
3.035E-05    8.971E-06    8.8724E-10
6.071E-05    9.190E-06    9.7065E-10
9.106E-05    9.341E-06    1.0804E-09
1.214E-04    9.436E-06    1.2166E-09
1.518E-04    9.486E-06    1.3393E-09
1.821E-04    9.486E-06    1.4570E-09
2.125E-04    9.445E-06    1.5644E-09
2.428E-04    9.372E-06    1.6733E-09
2.732E-04    9.263E-06    1.7872E-09
3.035E-04    9.117E-06    1.9105E-09
3.339E-04    8.935E-06    2.0558E-09
3.643E-04    8.753E-06    2.2161E-09
3.718E-04    8.689E-06    2.2698E-09
3.749E-04    8.625E-06    2.2869E-09
3.794E-04    8.479E-06    2.3215E-09
3.807E-04    8.388E-06    2.3281E-09
3.810E-04    0.000E+00    0.000
```

```
1920      0.843E+06     0.294E+03    0.224E+00     0.161E+04    0.198E+07    0.474E+00    0.213E+00
0.574E+05
     1
 IBRPMP  IBRTRB   SPEED(RPM)   ETATRB     PSITR     TORQUE(LB-IN)    HPOWER
    23   1213    0.800E+05    0.580E+00   0.275E+00    0.513E+02    0.651E+02

  SOLUTION SATISFIED CONVERGENCE CRITERION OF     0.100E-03 IN     114 ITERATIONS
TAU =     0.100000E+09ISTEP =                1
```

Turbopump Option Summary

- **GFSSP** has the capability to model a turbopump assembly as one component in a larger system model.

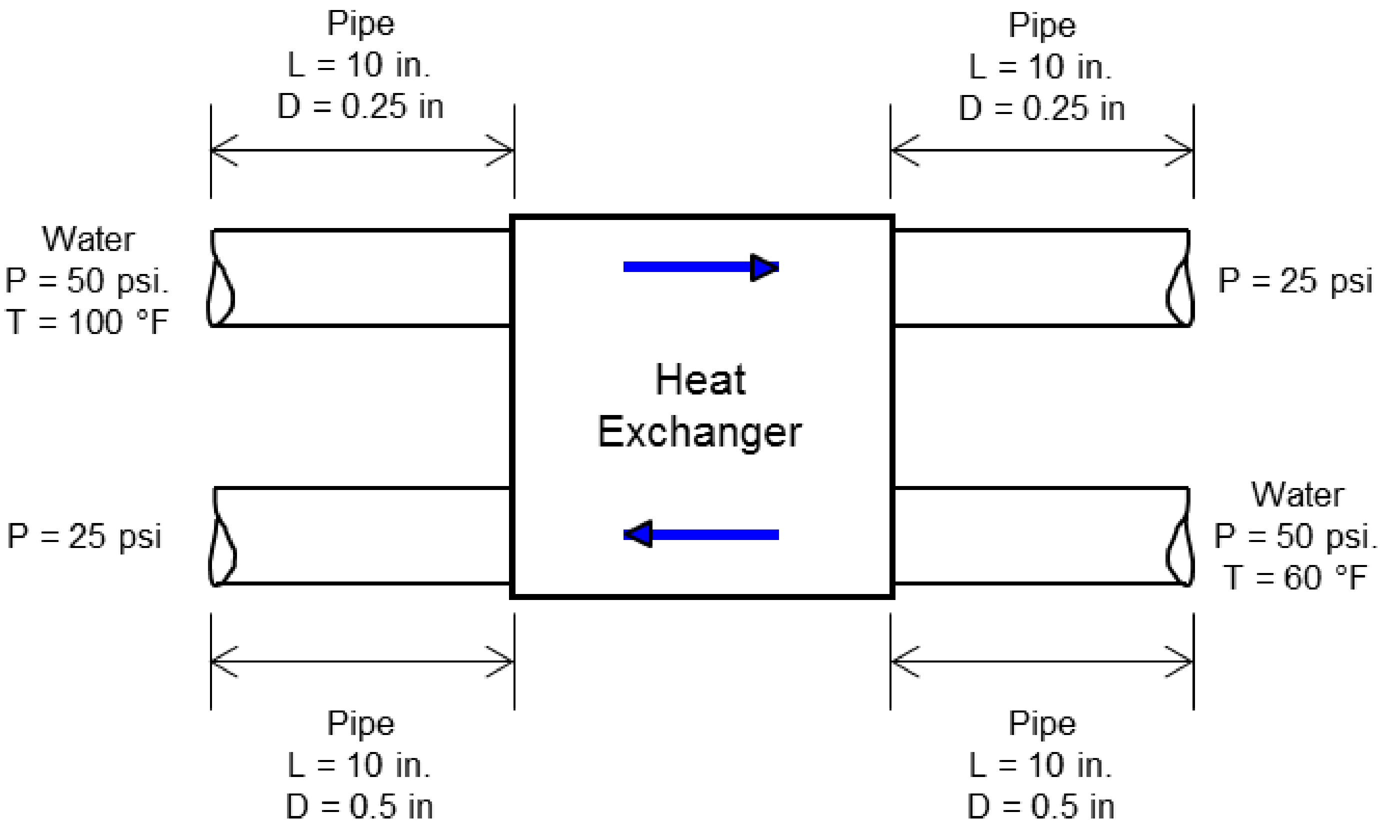
- Turbopump option
  - Allows two components of a flow circuit to exchange mechanical power

- User is required
  - To activate this option
  - Supply additional information of the turbopump assembly

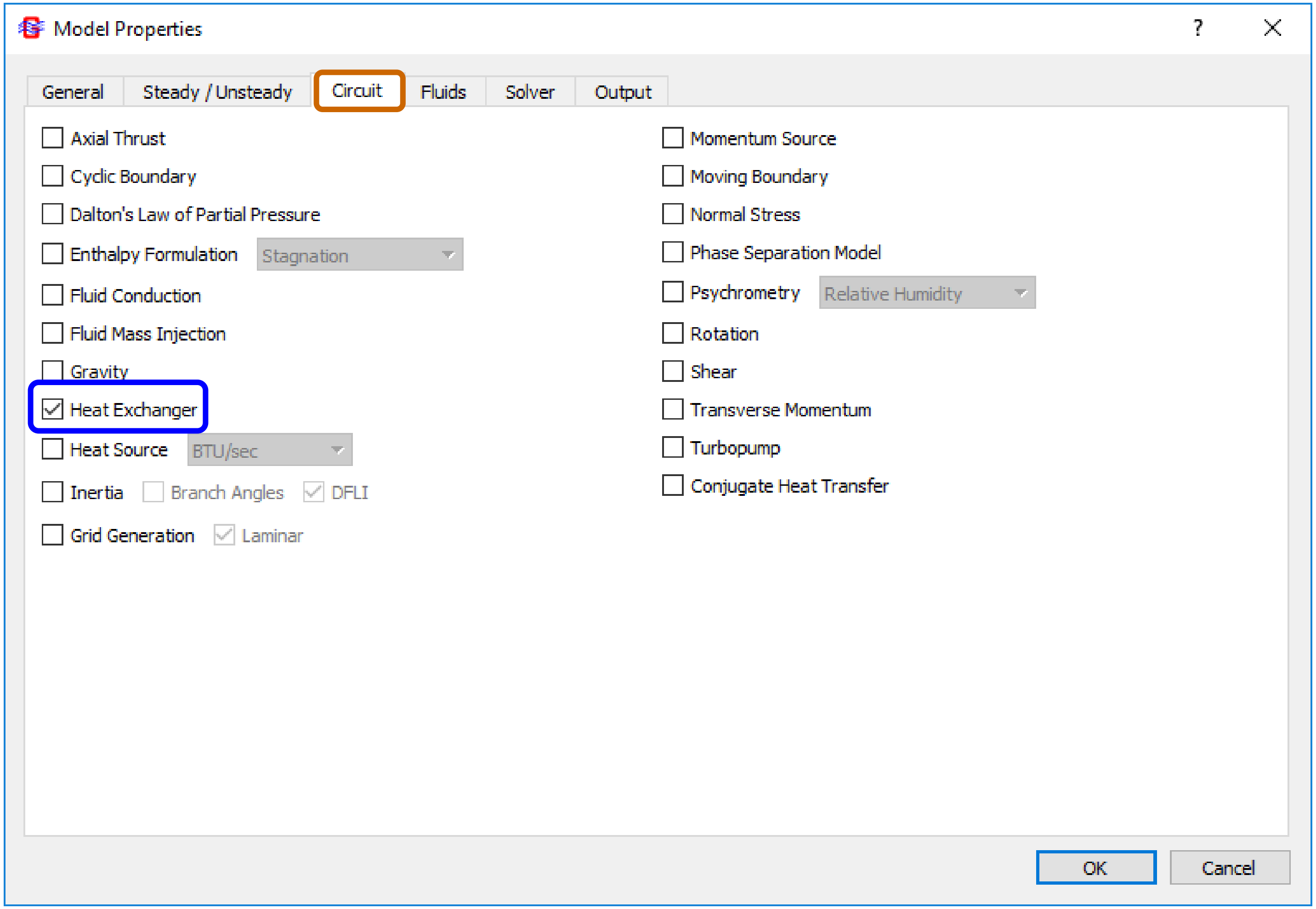
- **GFSSP** predicts (for a given design and operating conditions)
  - Flowrate
  - Pressure differential
  - Mechanical power

Ex5 – Simulation of a Flow System Involving a Heat Exchanger (1/3)

- Features
  - Heat Exchanger Option
  - Comparison with Textbook Solution



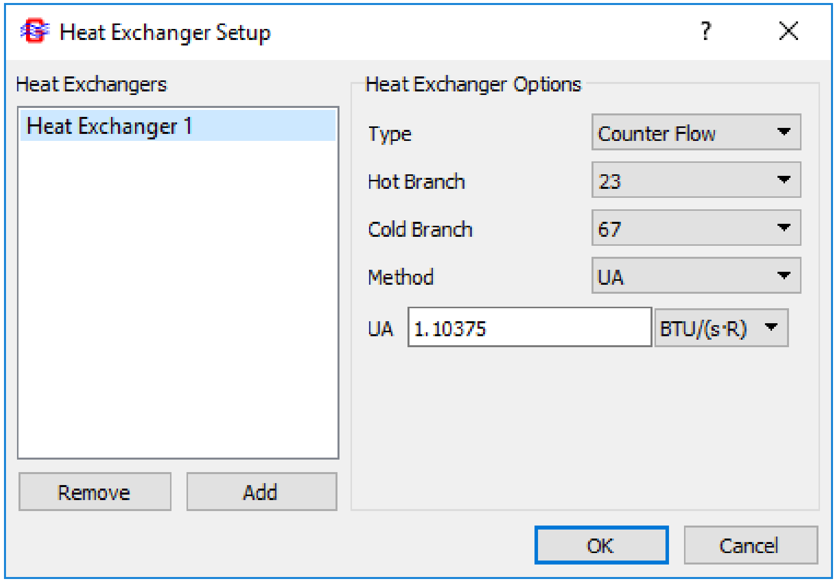
Ex5 – Simulation of a Flow System Involving a Heat Exchanger (2/3)



☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation — Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☑ Heat Exchanger
☐ Heat Source — BTU/sec
☐ Inertia ☐ Branch Angles ☑ DFLI
☐ Grid Generation ☑ Laminar
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry — Relative Humidity
☐ Rotation
☐ Shear
☐ Transverse Momentum
☐ Turbopump
☐ Conjugate Heat Transfer

Ex5 – Simulation of a Flow System Involving a Heat Exchanger (3/3)

- Heat Exchanger Option



Heat Exchanger Setup

Heat Exchangers

Heat Exchanger 1

Heat Exchanger Options

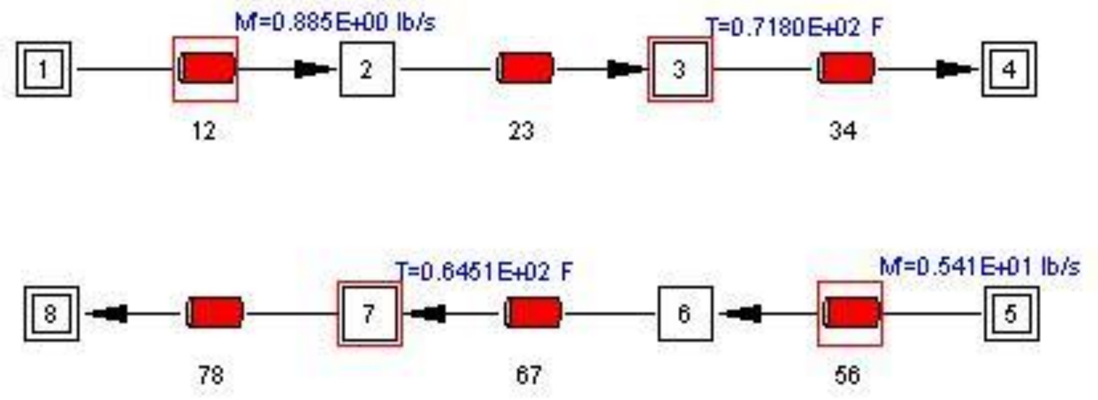
Type: Counter Flow

Hot Branch: 23

Cold Branch: 67

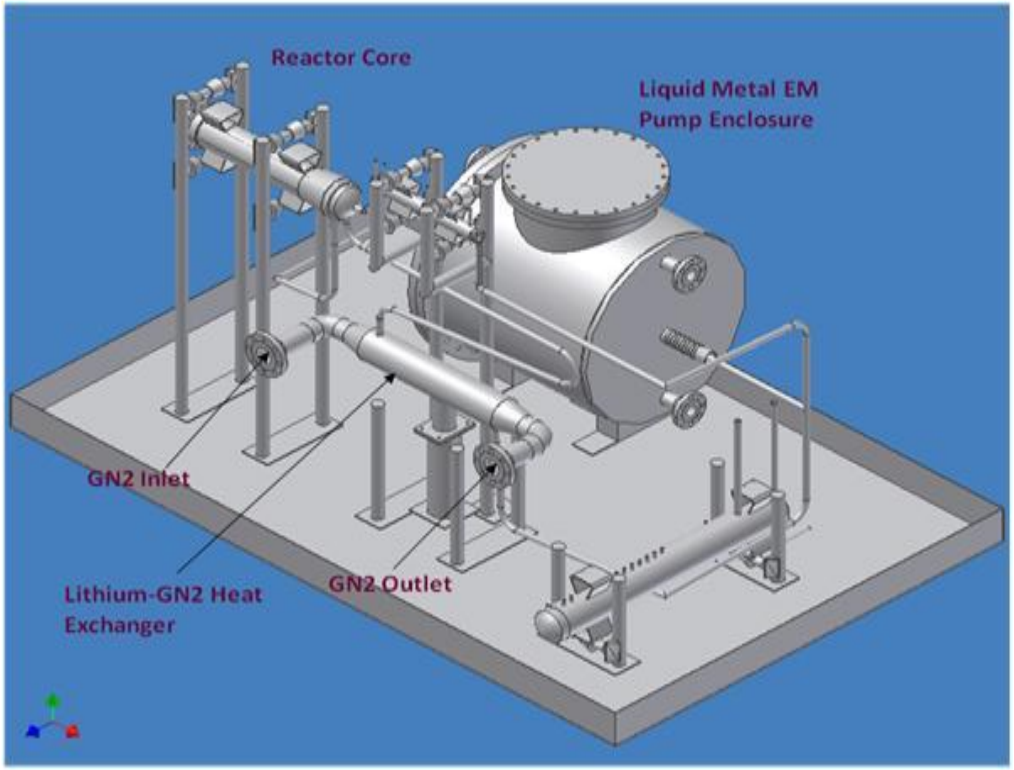
Method: UA

UA: 1.10375 BTU/(s·R)



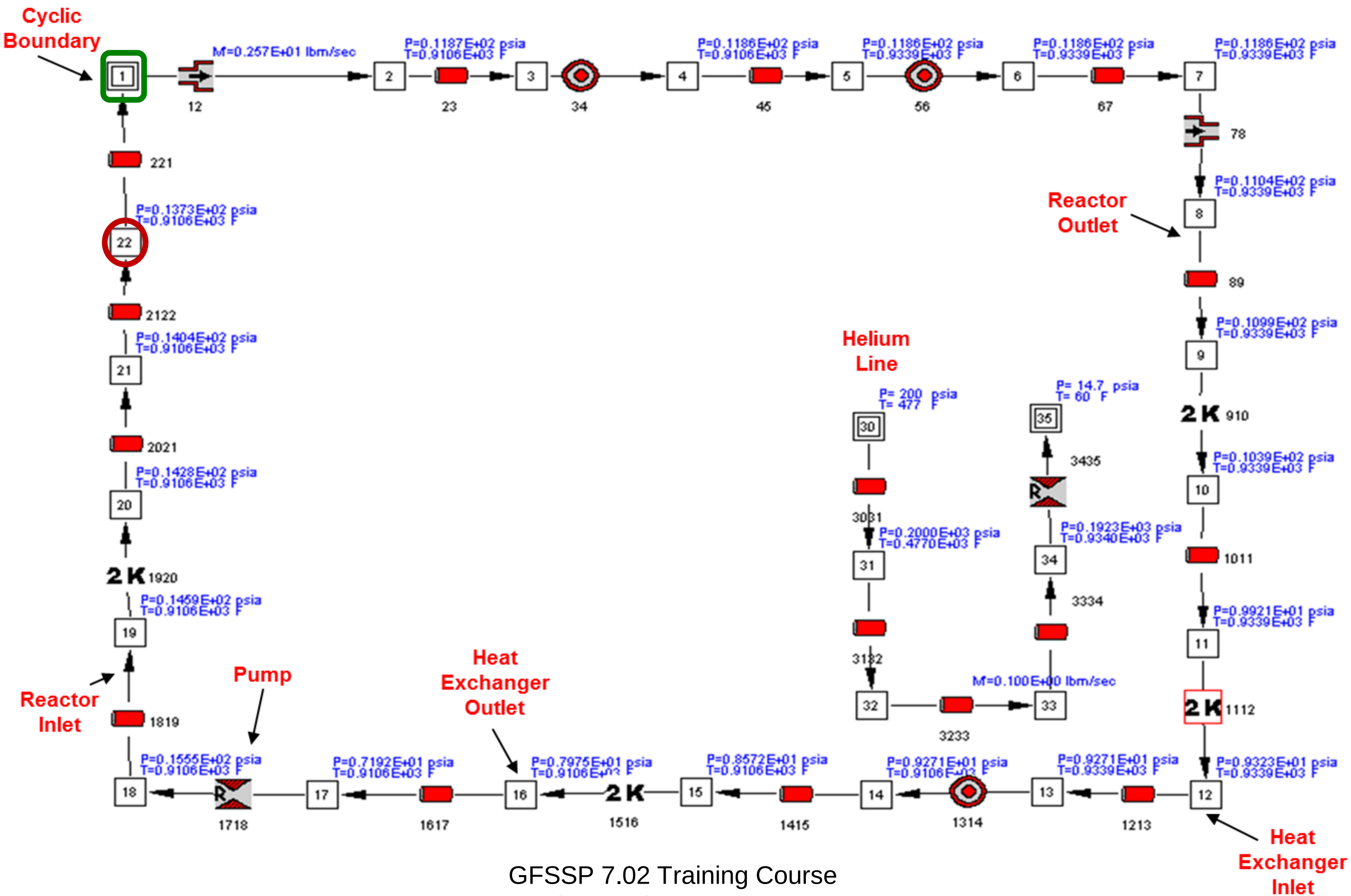
Ex20 – Simulation of a Lithium Loop Model (1/2)

- Features
  - Closed Loop with Cyclic Boundary
  - Use of User-specified Property
  - Heat Exchanger
  - User Subroutine to model Electro-Magnetic Pump



Ex20 – Simulation of a Lithium Loop Model (2/2)

- Closed Circuit Modeling
  - Cyclic Boundary Condition needs to be satisfied at Node 1
    - Implies Temperature at Node 22 must be equal to Temperature at Node 1
    - Must be achieved by iteration



Summary

- Rotational Flow and Internal Flow in Turbopump capabilities in **GFSSP**
  - Application was illustrated with Examples 6 and 21
  - Model predictions were compared with Test Data
    - Comparisons were satisfactory

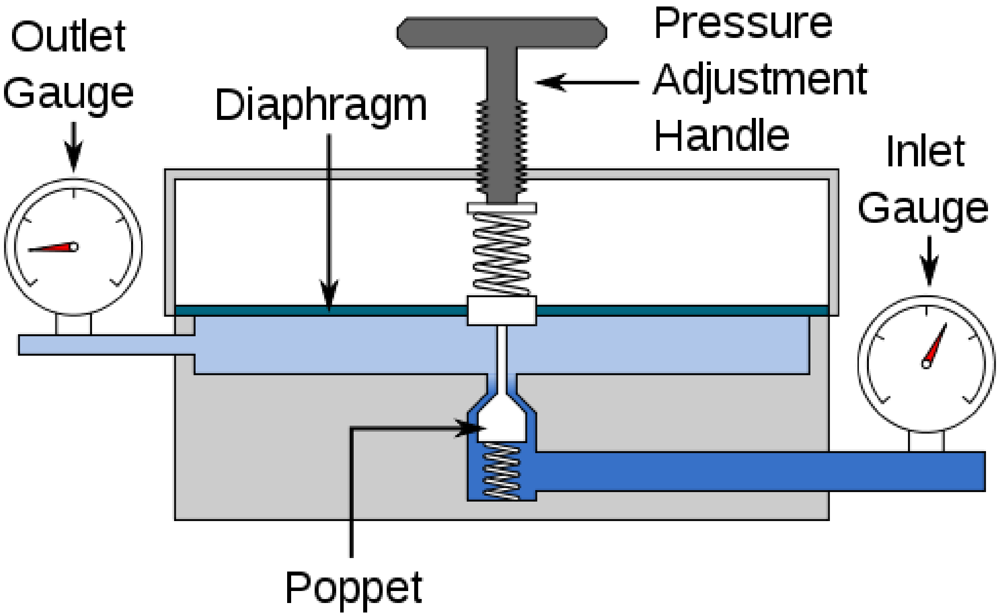
- Activation of Rotational Term and Axial Thrust Calculation in **MIG**

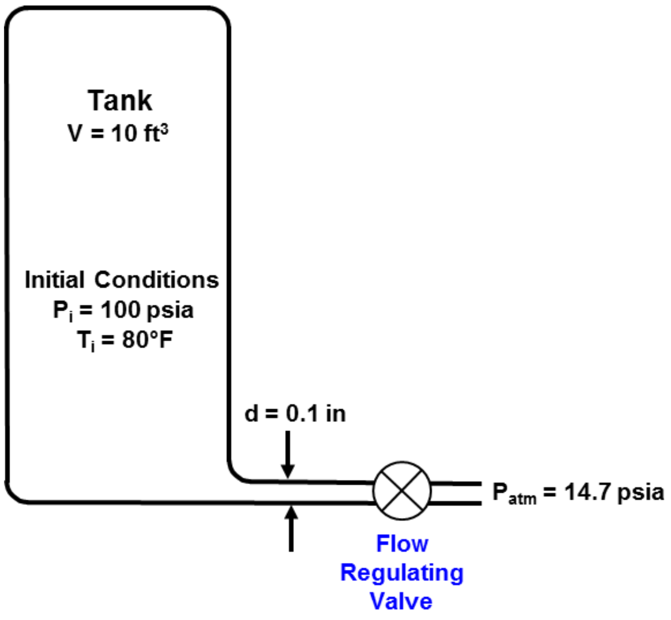
- References 27 and 38 provide more details of these models

- Example 11 illustrates the Turbopump option.
  - Used to model turbopump as a component in a larger system.
  - Transfers momentum between two branches, representing turbine and pump

- **GFSSP** can be used to model a Heat Exchanger in a flow circuit
  - Application was illustrated with Examples 5 and 20
  - Transfers heat between two nodes on the hot and cold sides of the flow circuit.

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

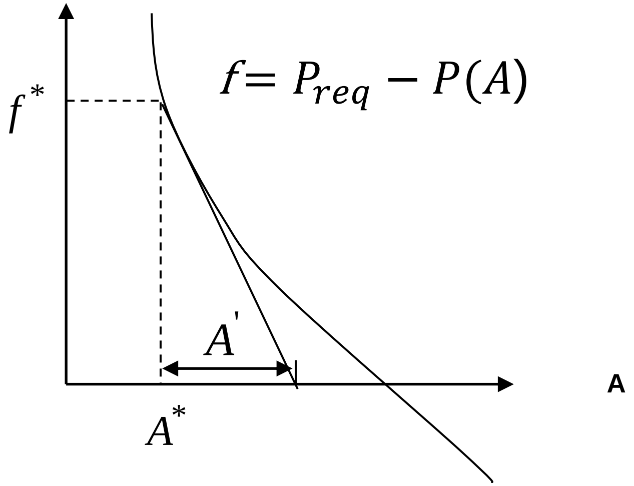
Iterative Algorithm

- The required flow area is determined by Newton’s Method:
  1. Assume an Area: A^*
  2. Compute the deviation: $f(A^*)$
  3. Estimate the gradient:

$$\Delta P = P_{up} - P_{dn} = \frac{\dot{m}^2}{2g_c\rho_{up}C_L^2A^2}$$

$$f = P_{req} - P_{dn} = P_{req} - P_{up} + \frac{\dot{m}^2}{2g_c\rho_{up}C_L^2A^2}$$

$$f' = \frac{df}{dA} \approx \frac{-\dot{m}^2}{g_c\rho_{up}C_L^2A^3}$$



  4. Estimate the correction in the Area:

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

  5. Compute the new Area (A)

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

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

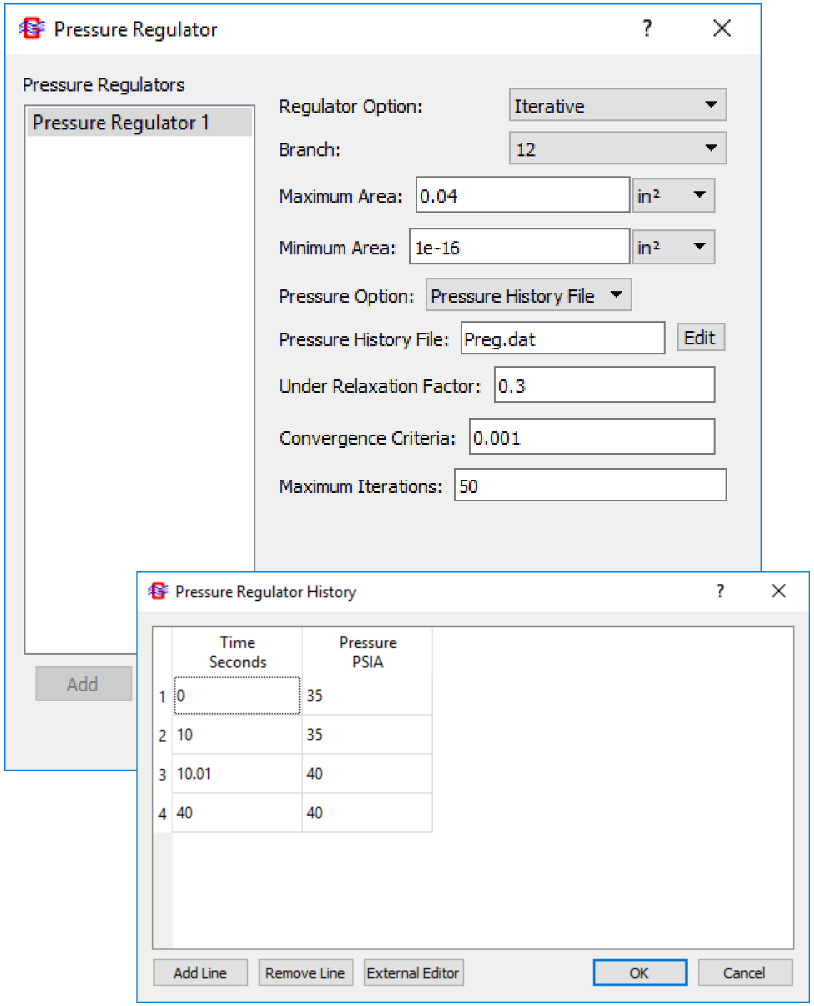
Pressure Regulator - Option 1

- Iterative Algorithm

- Purpose
  - To control pressure at a given node by adjusting the flow area of the upstream branch

- Implementation
  - Step 1
    - Steady/Unsteady
      - Pressure Regulator
  - Step 2
    - Advanced
      - Pressure Regulator

- Application
  - Example 16 – Simulation of a Pressure Regulator downstream of a pressurized tank



Pressure Regulator 1

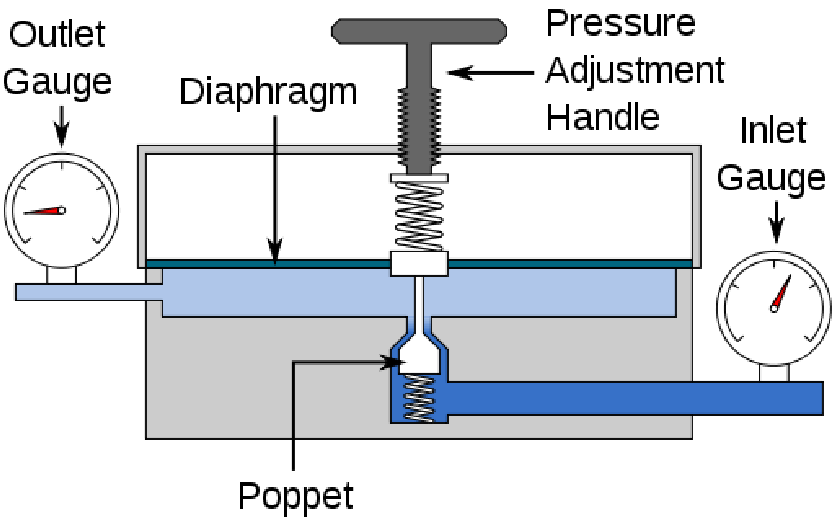
Regulator Option: Iterative
Branch: 12
Maximum Area: 0.04 in²
Minimum Area: 1e-16 in²
Pressure Option: Pressure History File
Pressure History File: Preg.dat
Under Relaxation Factor: 0.3
Convergence Criteria: 0.001
Maximum Iterations: 50

	Time Seconds	Pressure PSIA
1	0	35
2	10	35
3	10.01	40
4	40	40

Pressure Regulator - Option 2 **(1/2)**

- 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
    - Backward differencing algorithm (Schallhorn-Majumdar)
    - Forward looking algorithm (Schallhorn-Hass)

- Schallhorn-Hass Algorithm has been implemented in **GFSSP** as Option 2



Pressure Regulator - Option 2 **(2/2)**

- Forward Looking Algorithm
  - Previous time step result is not used
  - Area is calculated from the following expression

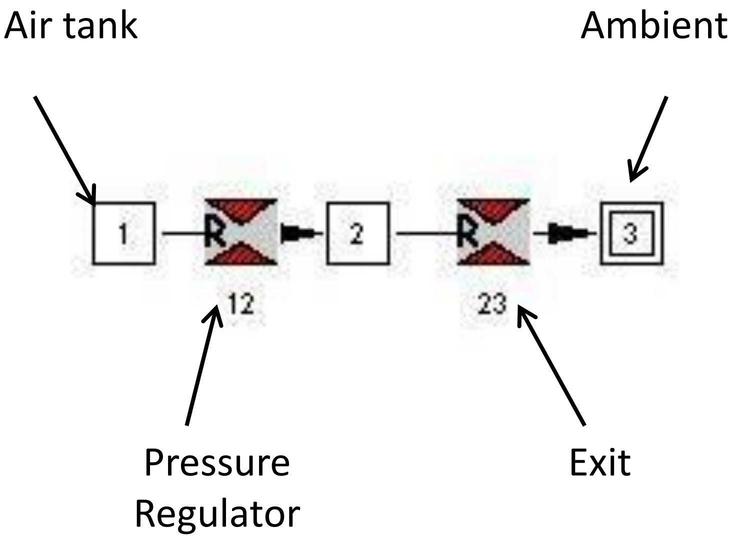
$$A^{*}_{\tau+\Delta\tau} = \begin{cases} \min\left(\left[A_{\tau} + \eta_{relax}\left(A_{new} - A_{\tau}\right)\right]A_{max}\right) \\ \max\left(\left[A_{\tau} + \eta_{relax}\left(A_{new} - A_{\tau}\right)\right]0\right) \end{cases}$$

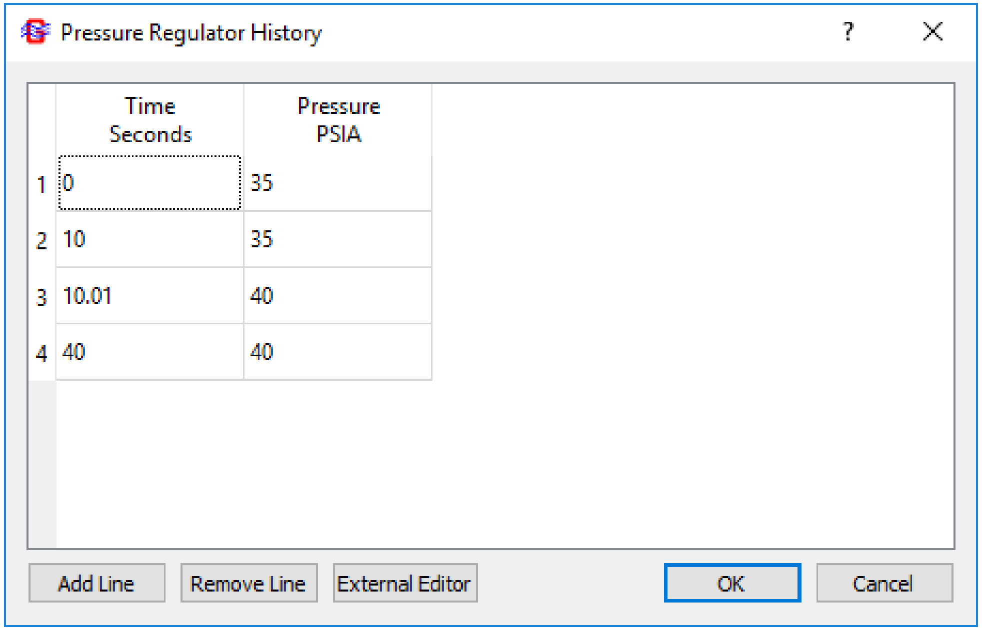
where,

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

Forward-Looking Algorithm (1/2)

- Application Results



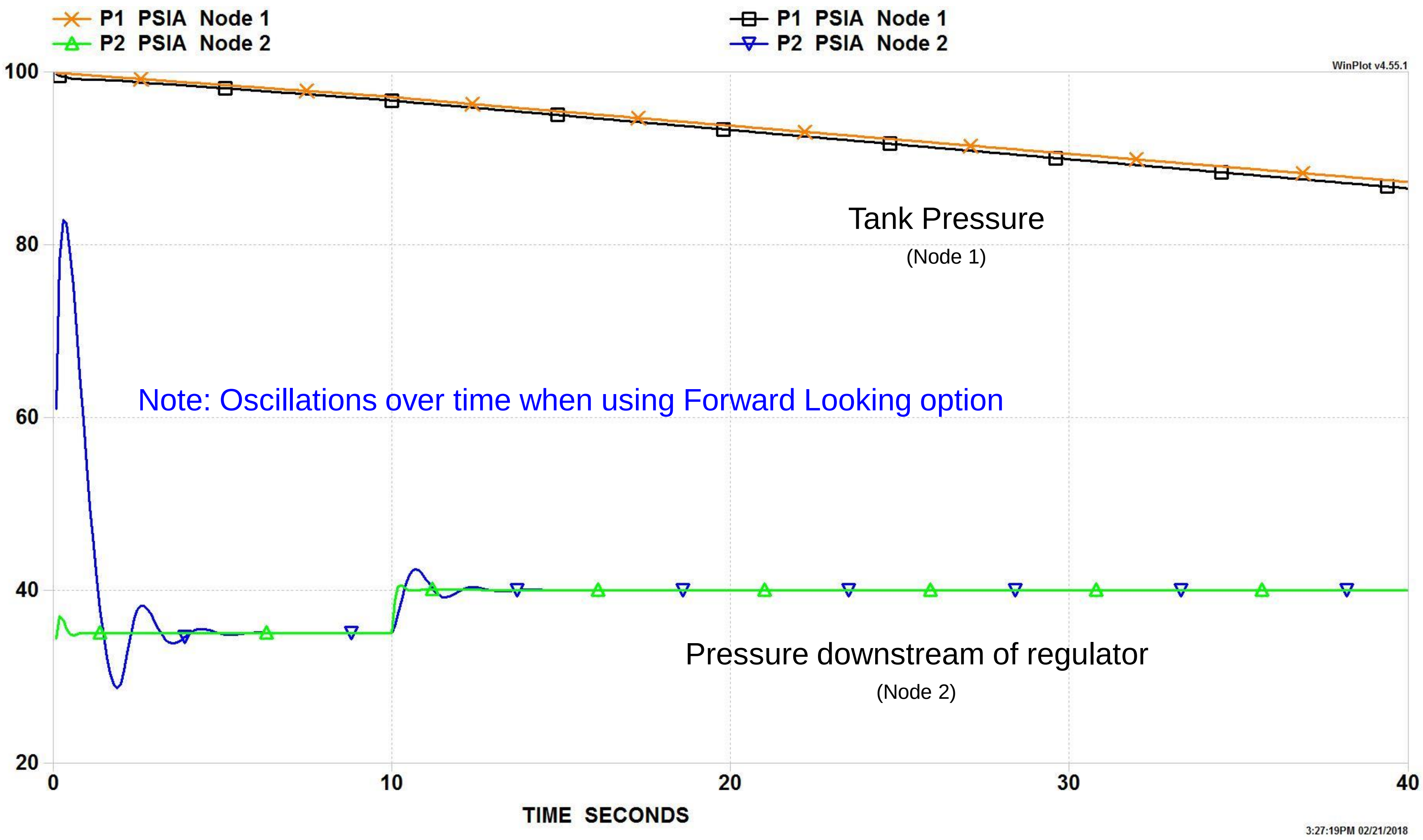


	Time Seconds	Pressure PSIA
1	0	35
2	10	35
3	10.01	40
4	40	40

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

Forward-Looking Algorithm (2/2)

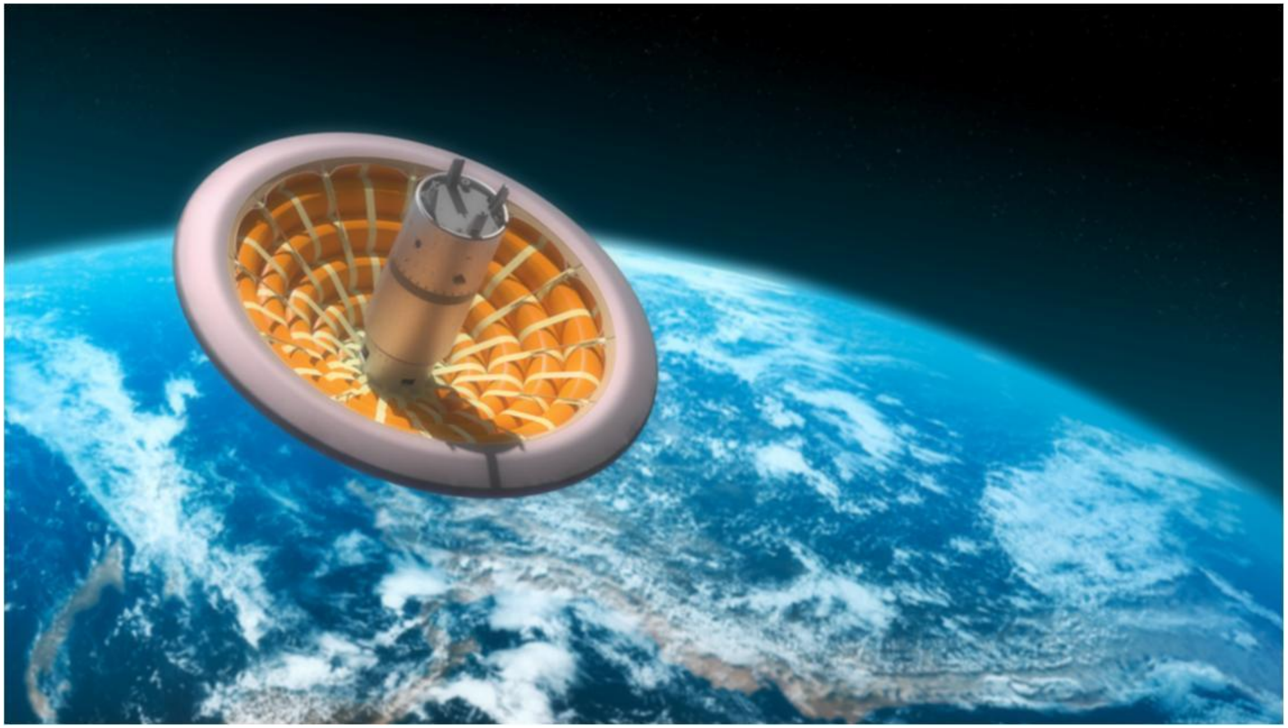
- Comparison between Forward-Looking Marching and Iterative Algorithm



Note: Oscillations over time when using Forward Looking option

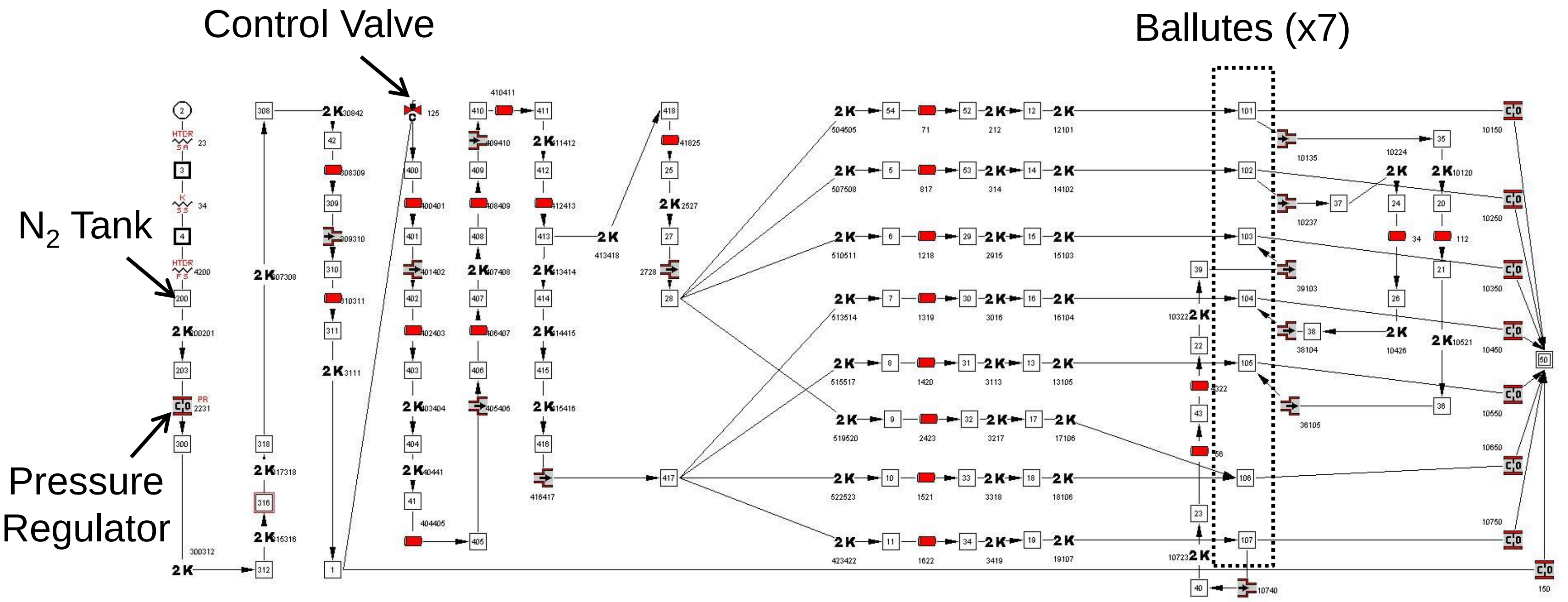
Applications (1/5)

Inflatable Re-Entry Vehicle (IRVE3)



Applications (2/5)

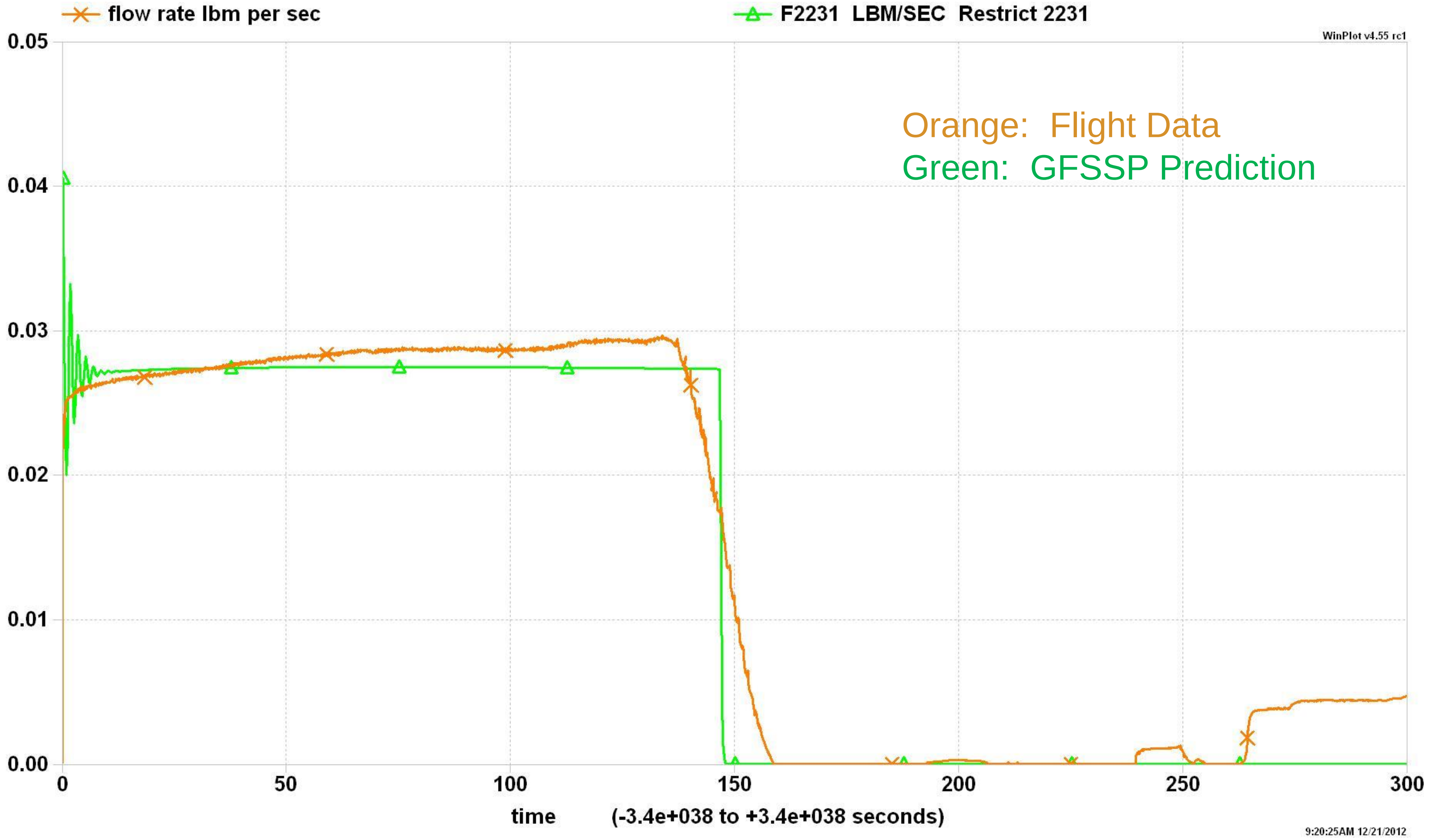
- **GFSSP** IRVE3 model



Applications (3/5)

Marshall Space Flight Center
GFSSP Training Course

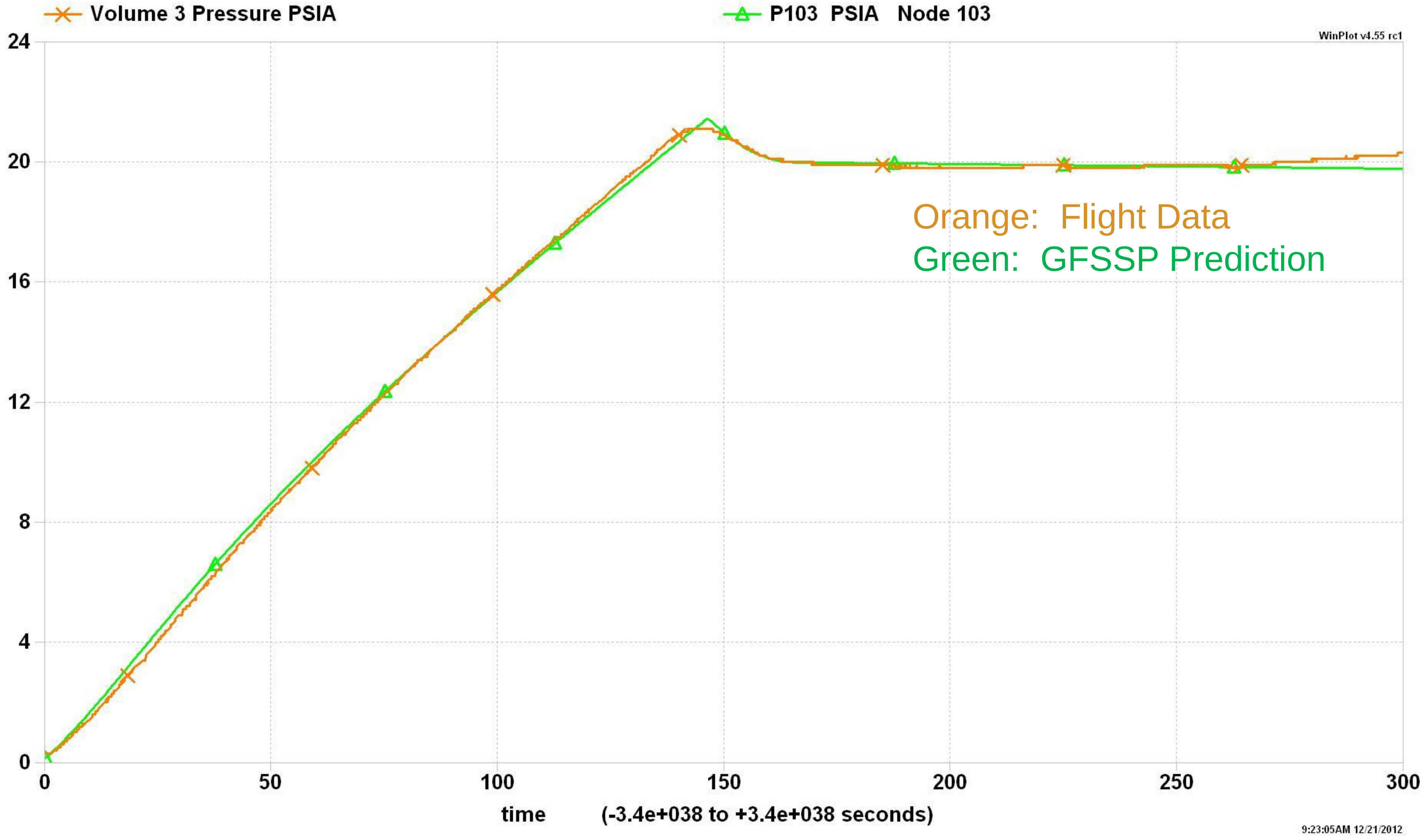
- Flow Rate



Orange: Flight Data
Green: GFSSP Prediction

Applications (4/5)

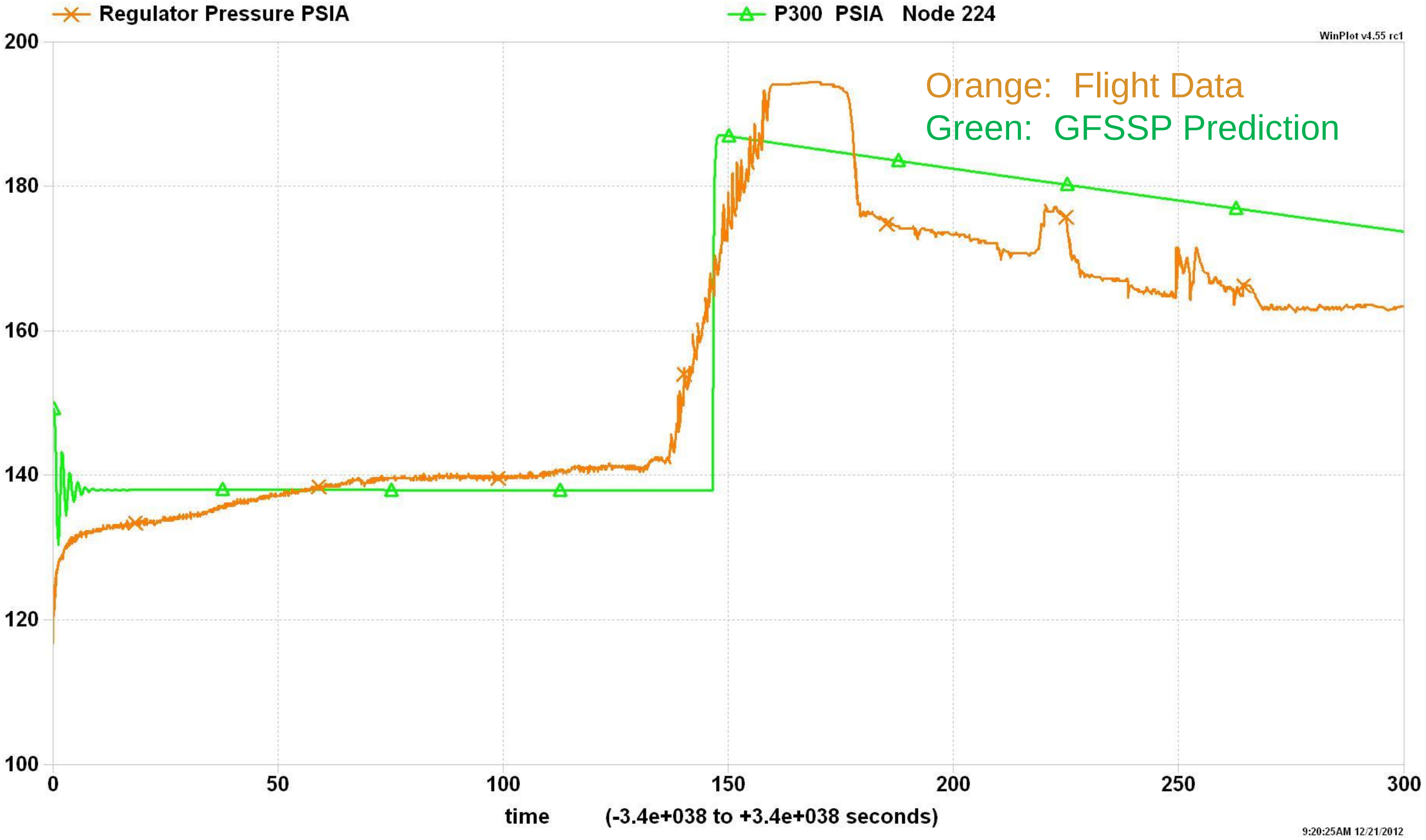
- Pressure in Ballute 3



Orange: Flight Data
Green: GFSSP Prediction

Applications (5/5)

- Regulator Pressure



Orange: Flight Data
Green: GFSSP Prediction

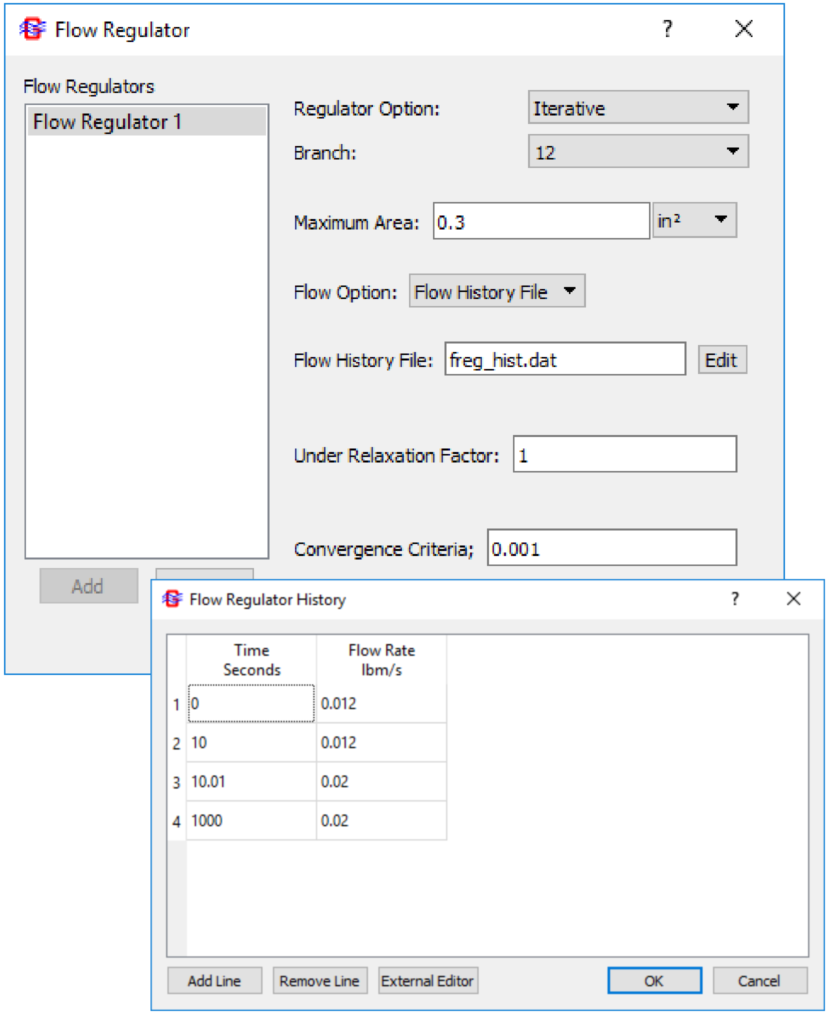
Modeling Flow Regulator

- **GFSSP** has two built-in options (algorithms) to model a Flow Regulator
  - 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

  - 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
    - ➢ Steady/Unsteady Tab
      - Flow Regulator
  - Step 2
    - ➢ Advanced
      - Flow Regulator
- Application
  - Example 17: Simulation of a Flow Regulator Downstream of a Pressurized Tank



Flow Regulator 1

Regulator Option: Iterative

Branch: 12

Maximum Area: 0.3 in²

Flow Option: Flow History File

Flow History File: freg_hist.dat

Under Relaxation Factor: 1

Convergence Criteria; 0.001

	Time Seconds	Flow Rate lbm/s
1	0	0.012
2	10	0.012
3	10.01	0.02
4	1000	0.02

Flow Regulator – Option 2

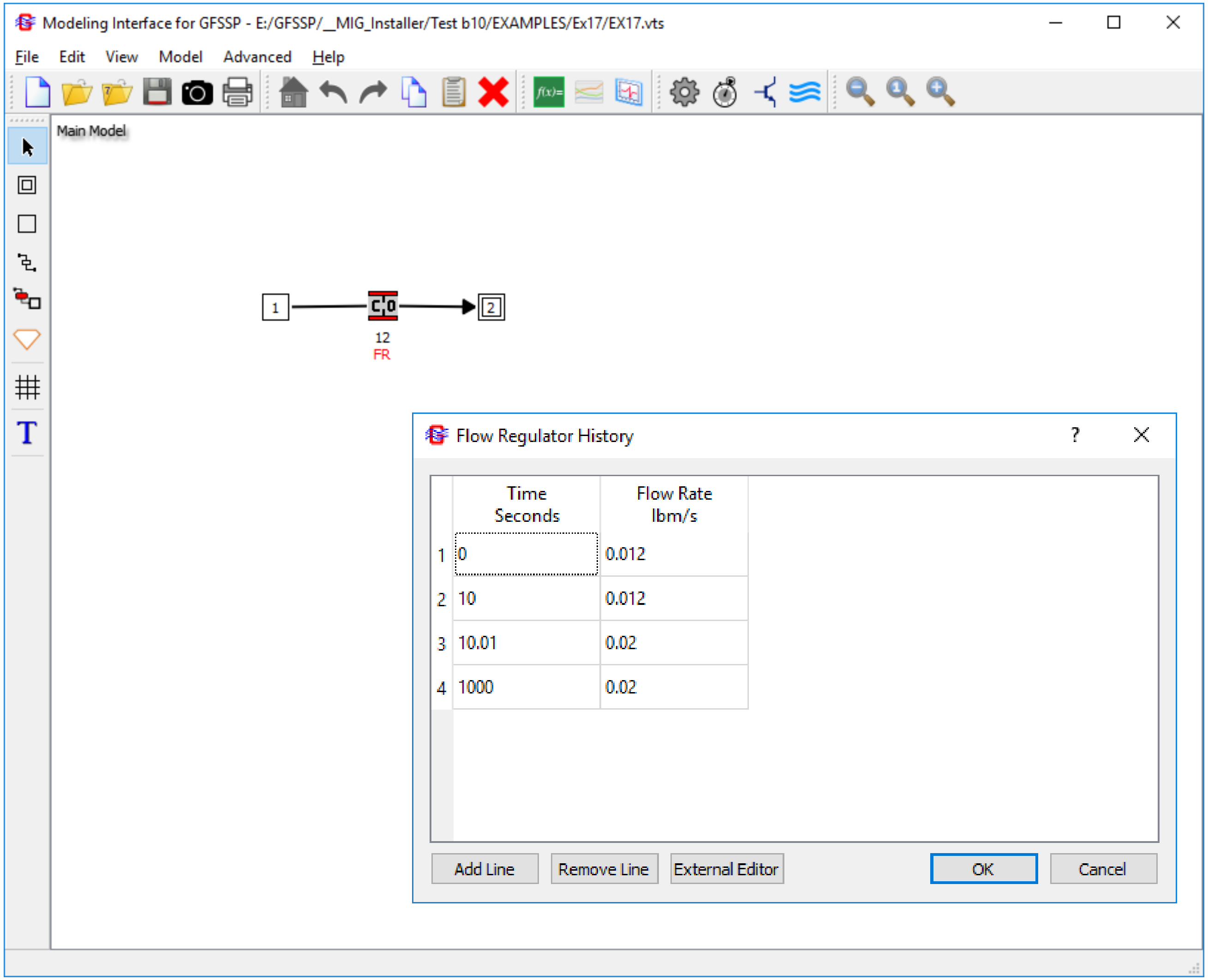
- Time-Marching Algorithm
  - Area is adjusted only once at the beginning of each time step
  - Adjustment of area is calculated based on the functional derivative dF/dA and the difference between the calculated and desired flow rate
  - CAUTION: If other elements of the model have significant effect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

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

where

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

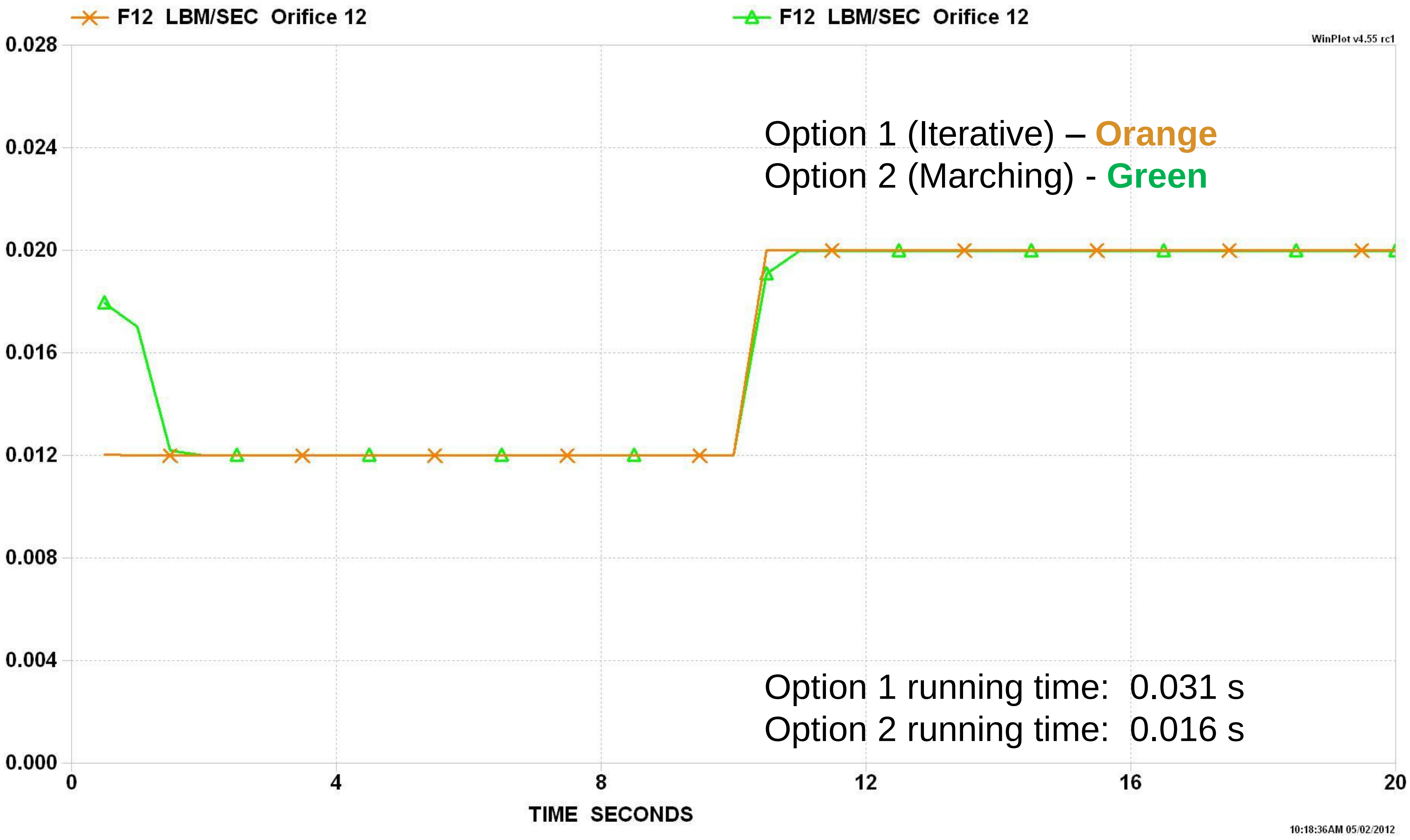
Flow Regulator – Example 17 (1/2)



	Time Seconds	Flow Rate lbm/s
1	0	0.012
2	10	0.012
3	10.01	0.02
4	1000	0.02

Flow Regulator – Example 17 (2/2)

- Comparison between Iterative and Time-Marching Algorithms



Option 1 (Iterative) – **Orange**
Option 2 (Marching) - **Green**

Option 1 running time: 0.031 s
Option 2 running time: 0.016 s

Summary

- Pressure & Flow Regulator Options have been made available to include in any unsteady flow simulation

- Pressure Regulator has two options
  - Iterative (Option 1)
  - Marching (Option 2)
    - ➢ Option 2 has the flexibility of using multiple regulators and runs faster

- Flow Regulator also has two options
  - Iterative (Option 1)
  - 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 tutorial, you will:

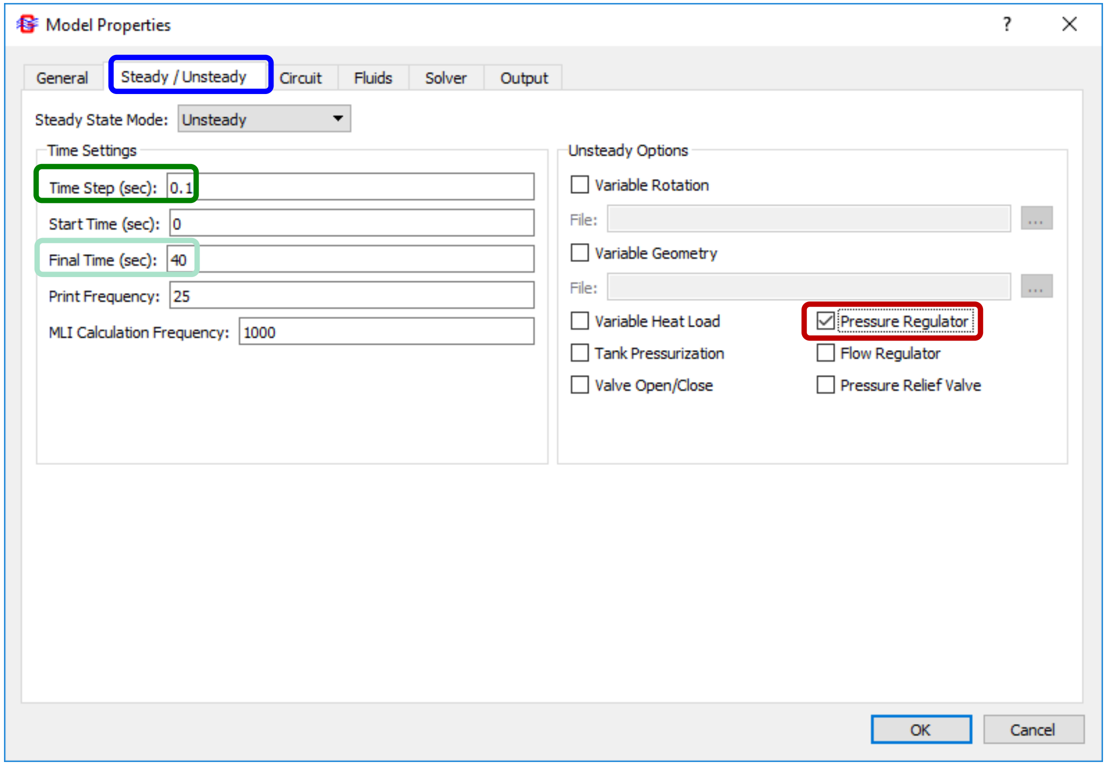
- Use **GFSSP’**s built-in Pressure Regulator options to model the regulated blowdown of a tank of compressed air

- Learn the difference between the two Pressure Regulator options

Set Up Options (1/2)

- General
  - Model File: Tut4.gfssp
  - Input File: Tut4.DAT
  - Output File: Tut4.OUT

- Steady / Unsteady
  - Time step = 0.1 s
  - Final time = 40 s
  - Check Pressure Regulator



Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

Steady State Mode: Unsteady

Time Settings

Time Step (sec): 0.1

Start Time (sec): 0

Final Time (sec): 40

Print Frequency: 25

MLI Calculation Frequency: 1000

Unsteady Options

☐ Variable Rotation

File:

☐ Variable Geometry

File:

☐ Variable Heat Load

☑ Pressure Regulator

☐ Tank Pressurization

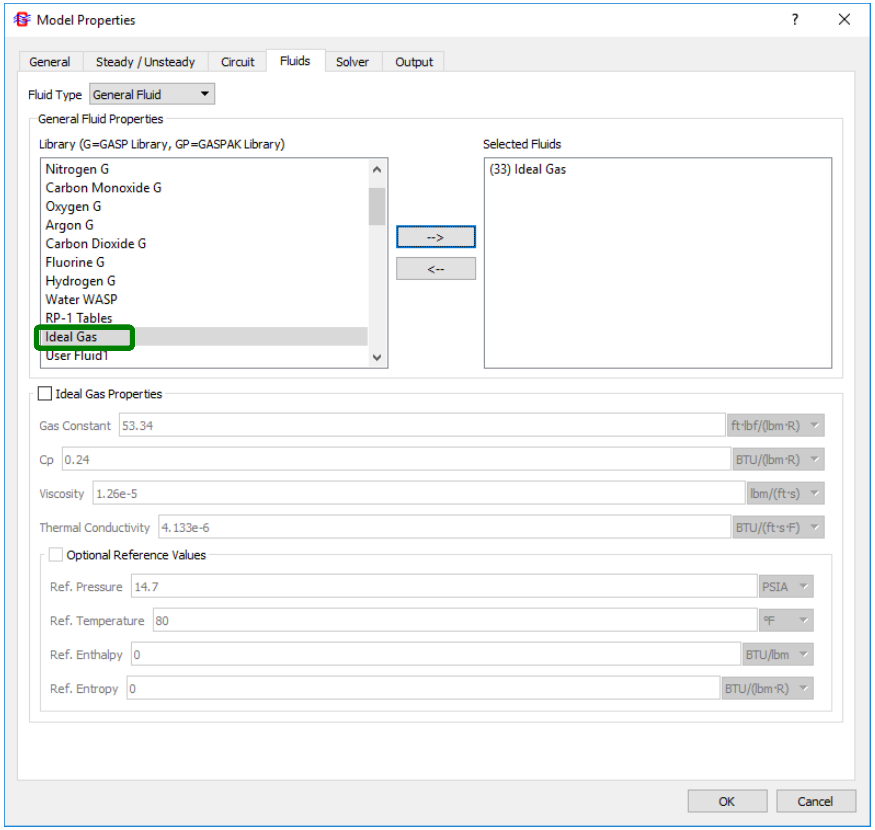
☐ Flow Regulator

☐ Valve Open/Close

☐ Pressure Relief Valve

Set Up Options (2/2)

- Fluids
  - Select Ideal Gas
  - Defaults to Air properties
- Output
  - Select Winplot binary output



Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

Fluid Type: General Fluid

General Fluid Properties

Library (G=GASP Library, GP=GASPAK Library)

- Nitrogen G
- Carbon Monoxide G
- Oxygen G
- Argon G
- Carbon Dioxide G
- Fluorine G
- Hydrogen G
- Water WASP
- RP-1 Tables
- Ideal Gas
- User Fluid1

Selected Fluids: (33) Ideal Gas

☐ Ideal Gas Properties

Gas Constant: 53.34 ft·lbf/(lbm·R)

Cp: 0.24 BTU/(lbm·R)

Viscosity: 1.26e-5 lbm/(ft·s)

Thermal Conductivity: 4.133e-6 BTU/(ft·s·F)

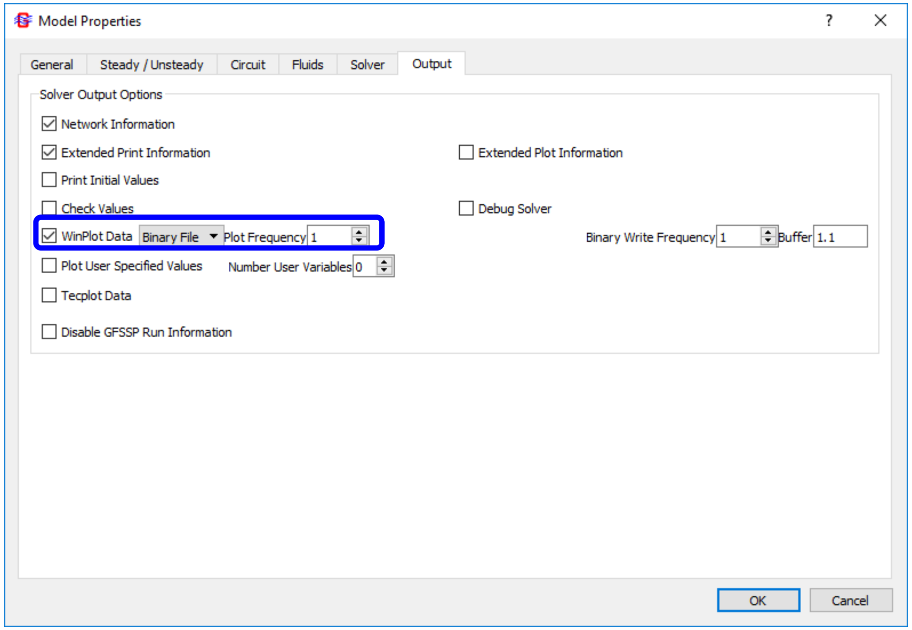
☐ Optional Reference Values

Ref. Pressure: 14.7 PSIA

Ref. Temperature: 80 ºF

Ref. Enthalpy: 0 BTU/lbm

Ref. Entropy: 0 BTU/(lbm·R)



Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

Solver Output Options

☑ Network Information

☑ Extended Print Information ☐ Extended Plot Information

☐ Print Initial Values

☐ Check Values ☐ Debug Solver

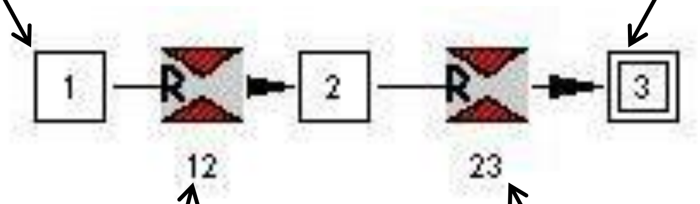
☑ WinPlot Data: Binary File, Plot Frequency: 1 Binary Write Frequency: 1 Buffer: 1.1

☐ Plot User Specified Values Number User Variables: 0

☐ Tecplot Data

☐ Disable GFSSP Run Information

Build Model on Canvas



Air tank

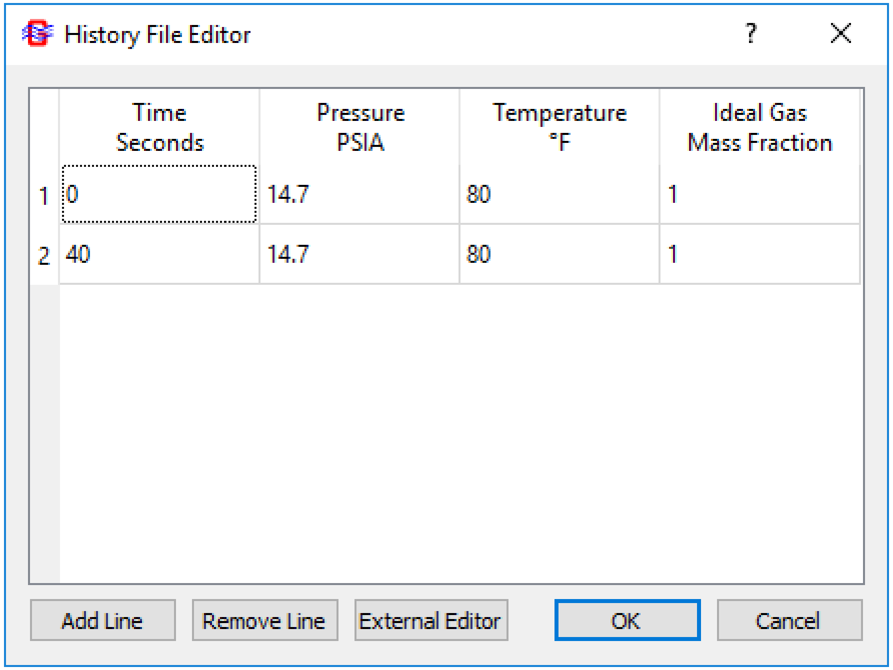
Ambient

Pressure Regulator

Exit

Set Up Transient Boundary Conditions

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

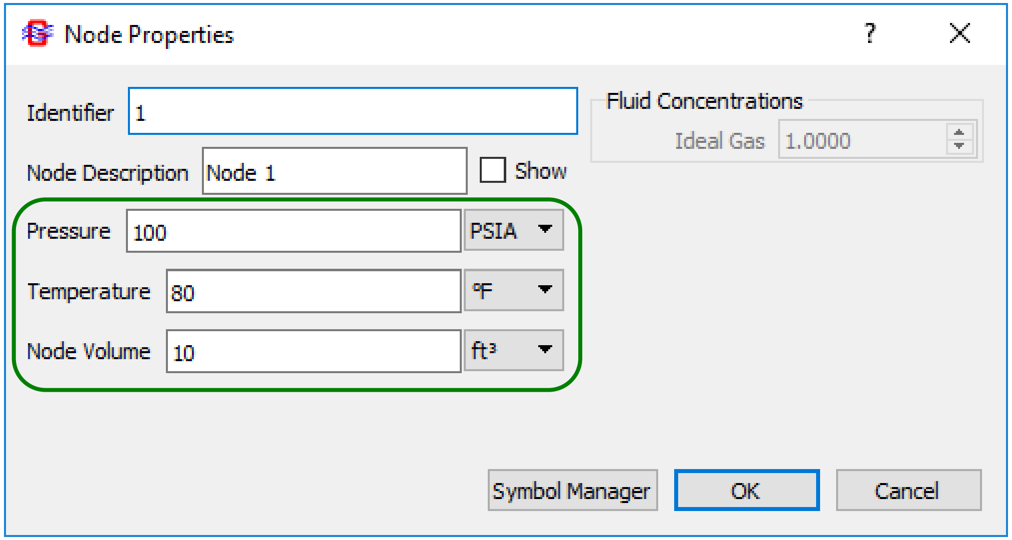


	Time Seconds	Pressure PSIA	Temperature °F	Ideal Gas Mass Fraction
1	0	14.7	80	1
2	40	14.7	80	1

Set Up Internal Nodes

- Node 1
  - Initial P = 100 psia
  - Initial T = 80.0 °F
  - Volume = 10 ft^3

- Node 2 (represents the volume downstream of the regulator)
  - Initial P = 14.7 psia
  - Initial T = 80.0 °F
  - Volume = 100 in^3



Identifier 1

Node Description Node 1 ☐ Show

Ideal Gas 1.0000

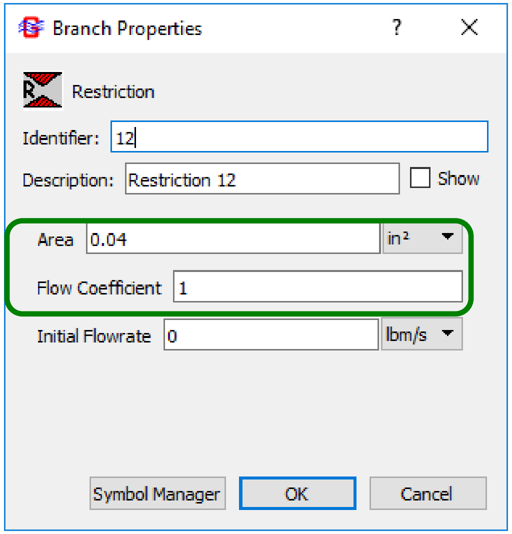
Pressure 100 PSIA

Temperature 80 °F

Node Volume 10 ft³

Set Up Fluid Branches

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



Branch Properties

Restriction

Identifier: 12

Description: Restriction 12

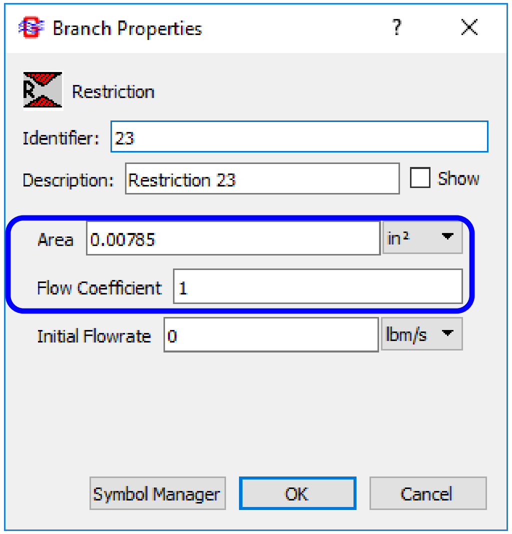
☐ Show

Area 0.04 in²

Flow Coefficient 1

Initial Flowrate 0 lbm/s

- Branch 23: Exit
  - A = 0.00785 in^2
  - C_L = 1.0



Branch Properties

Restriction

Identifier: 23

Description: Restriction 23

☐ Show

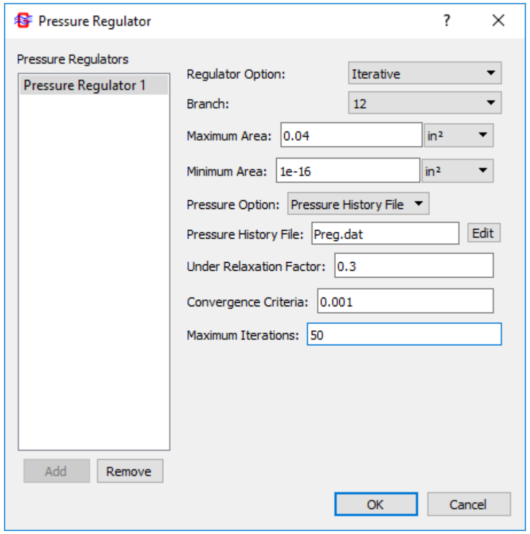
Area 0.00785 in²

Flow Coefficient 1

Initial Flowrate 0 lbm/s

Set Up Pressure Regulator – Option 1

- Select Advanced/Pressure Regulator
- Click “Add”
- Fill in the dialog boxes
- Create a pressure history data file: Preg.dat
- For each time step, **GFSSP** will adjust the area of Branch 12 to maintain the desired pressure in the downstream node.



Pressure Regulators: Pressure Regulator 1

Regulator Option: Iterative

Branch: 12

Maximum Area: 0.04 in²

Minimum Area: 1e-16 in²

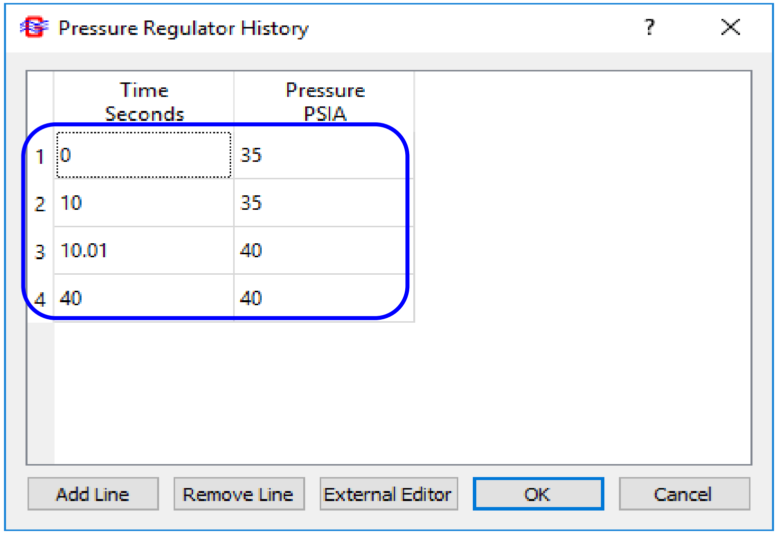
Pressure Option: Pressure History File

Pressure History File: Preg.dat

Under Relaxation Factor: 0.3

Convergence Criteria: 0.001

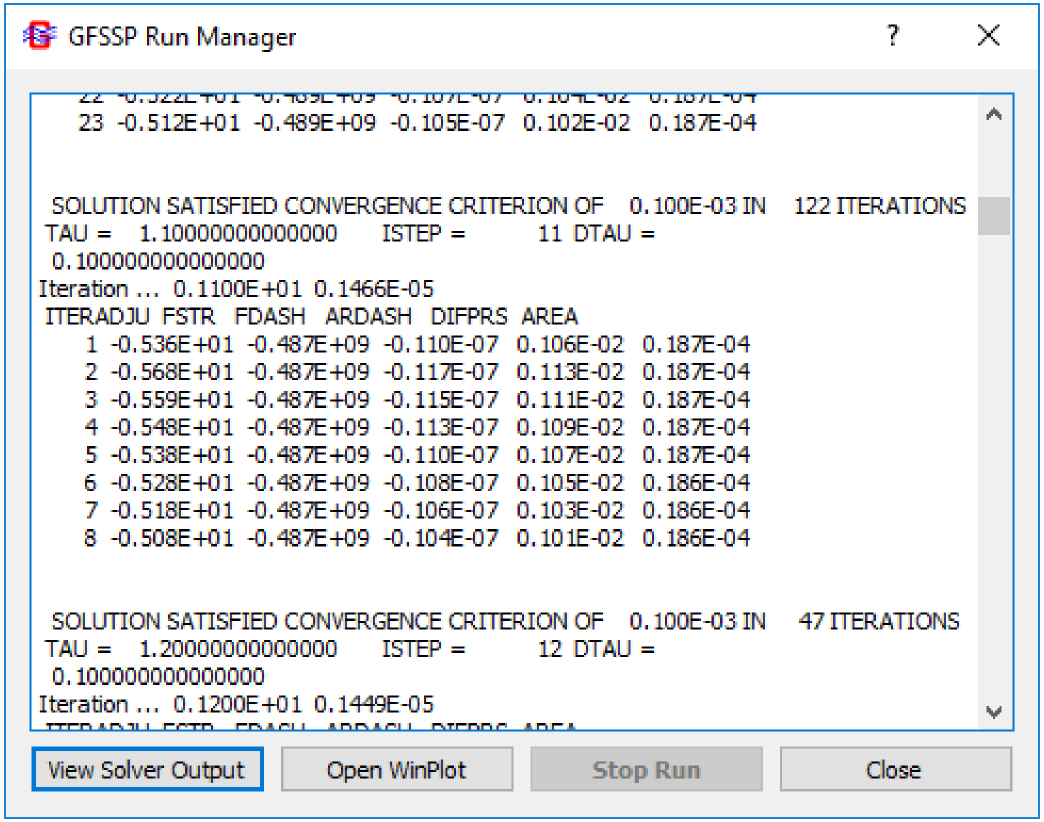
Maximum Iterations: 50



	Time Seconds	Pressure PSIA
1	0	35
2	10	35
3	10.01	40
4	40	40

Results of Pressure Regulator – Option 1 (1/4)

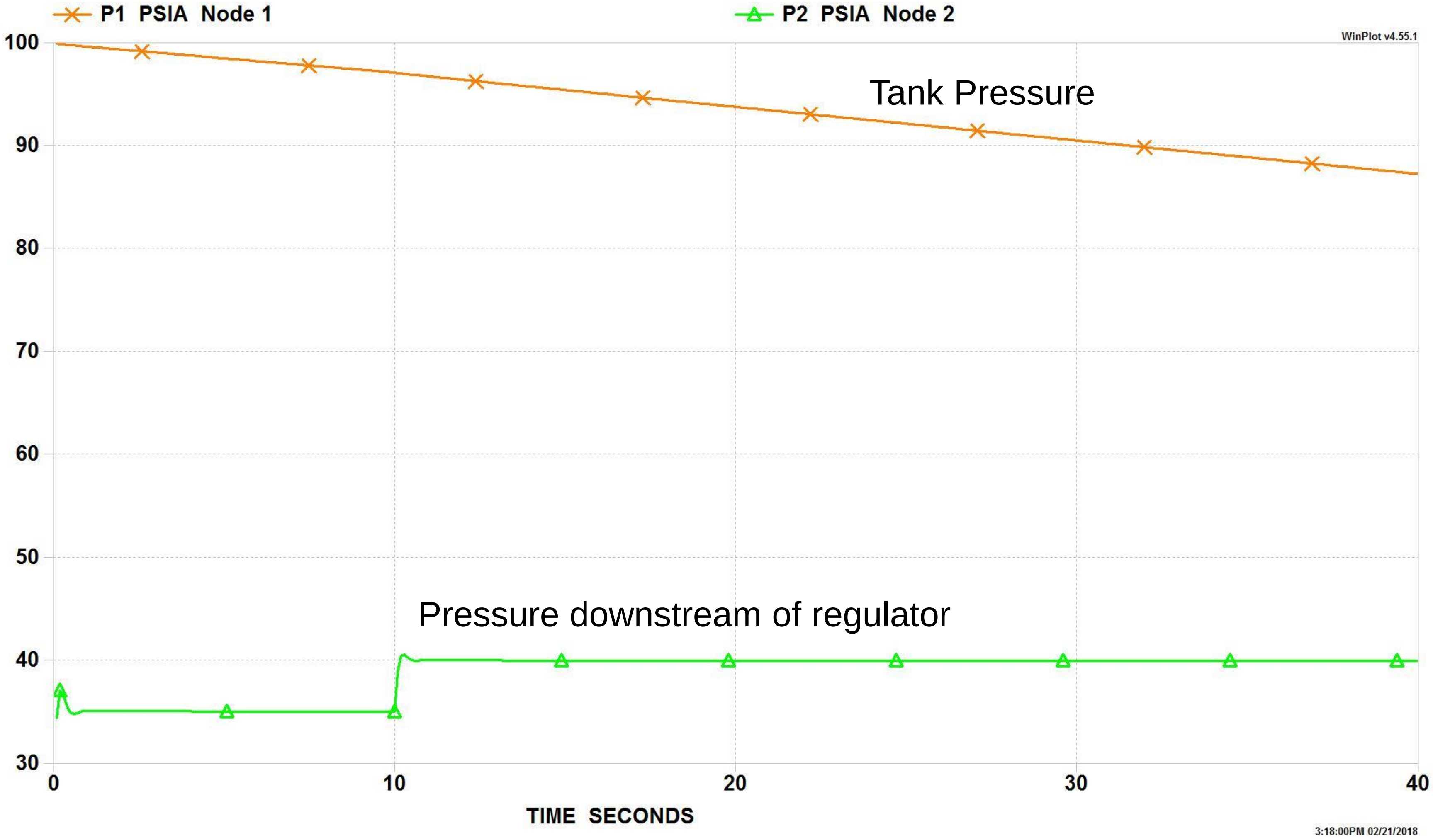
- Run the model
- Note that in each time step **GFSSP** is adjusting the area of Branch 12 to meet the desired pressure.
  - What effect do you think this has on run time?



```
    23  -0.512E+01  -0.489E+09  -0.105E-07   0.102E-02   0.187E-04


 SOLUTION SATISFIED CONVERGENCE CRITERION OF    0.100E-03 IN    122 ITERATIONS
 TAU =    1.10000000000000       ISTEP =          11  DTAU =
  0.100000000000000
Iteration ...  0.1100E+01  0.1466E-05
 ITERADJU  FSTR   FDASH   ARDASH   DIFPRS  AREA
     1  -0.536E+01  -0.487E+09  -0.110E-07   0.106E-02   0.187E-04
     2  -0.568E+01  -0.487E+09  -0.117E-07   0.113E-02   0.187E-04
     3  -0.559E+01  -0.487E+09  -0.115E-07   0.111E-02   0.187E-04
     4  -0.548E+01  -0.487E+09  -0.113E-07   0.109E-02   0.187E-04
     5  -0.538E+01  -0.487E+09  -0.110E-07   0.107E-02   0.187E-04
     6  -0.528E+01  -0.487E+09  -0.108E-07   0.105E-02   0.186E-04
     7  -0.518E+01  -0.487E+09  -0.106E-07   0.103E-02   0.186E-04
     8  -0.508E+01  -0.487E+09  -0.104E-07   0.101E-02   0.186E-04


 SOLUTION SATISFIED CONVERGENCE CRITERION OF    0.100E-03 IN     47 ITERATIONS
 TAU =    1.20000000000000       ISTEP =          12  DTAU =
  0.100000000000000
Iteration ...  0.1200E+01  0.1449E-05
```

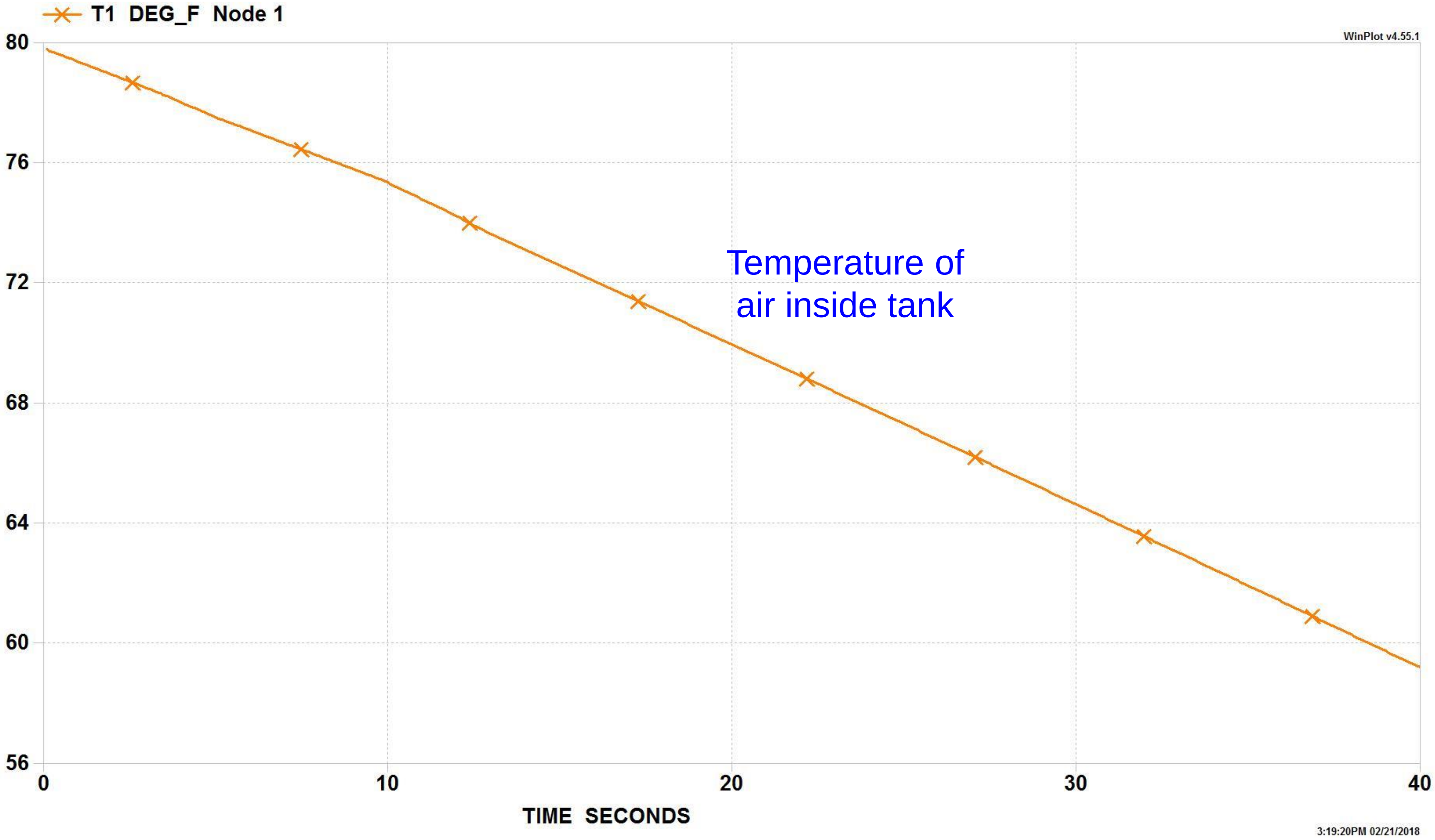
Results of Pressure Regulator – Option 1 (2/4)

- Pressure History



Results of Pressure Regulator – Option 1 (3/4)

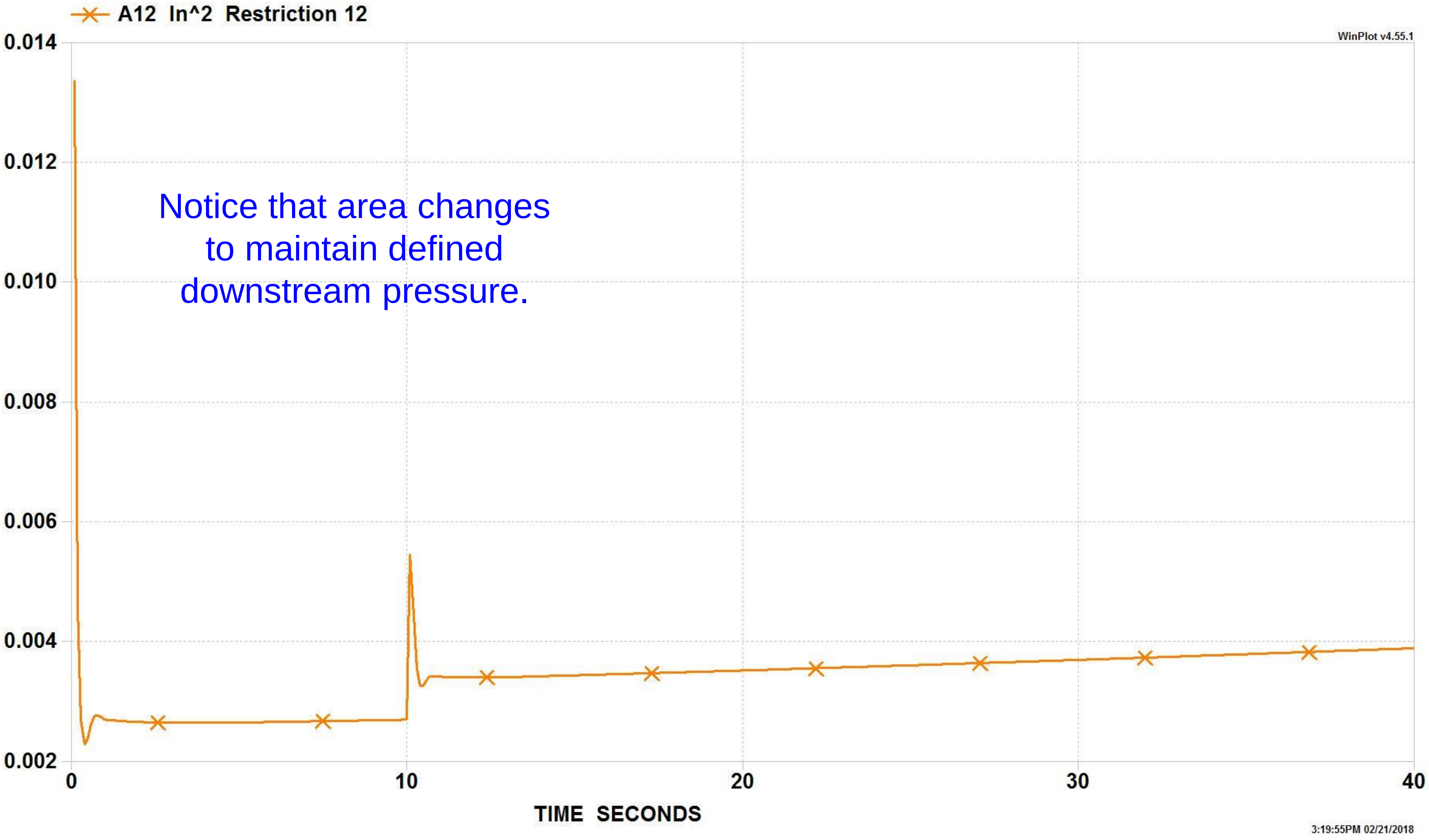
- Temperature History



Temperature of air inside tank

Results of Pressure Regulator – Option 1 (4/4)

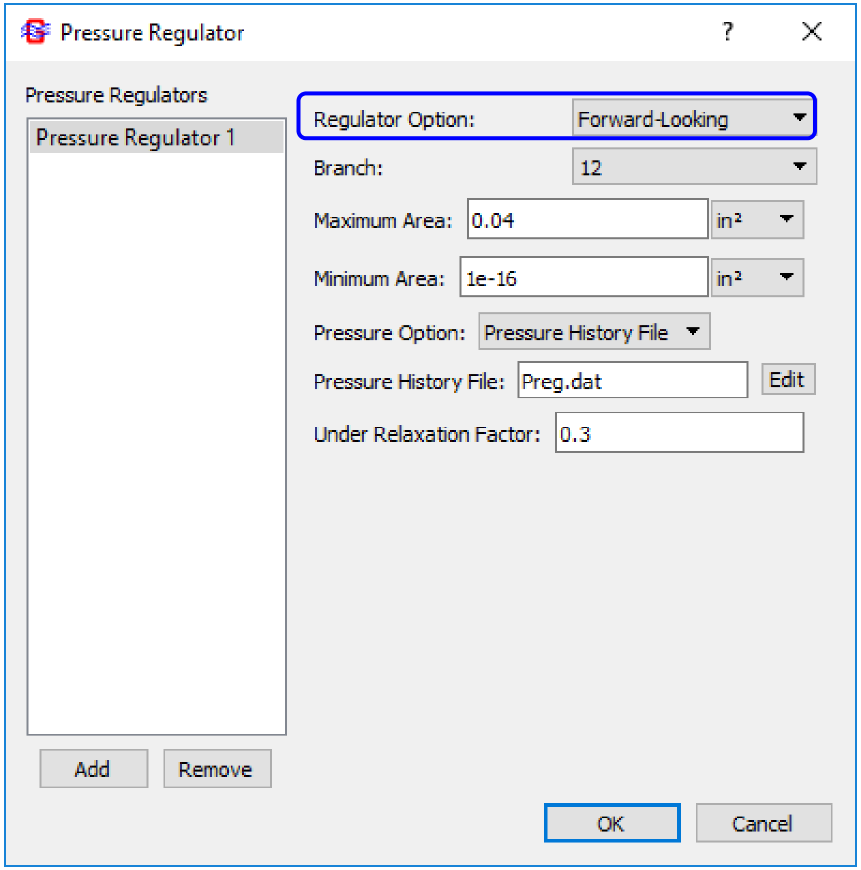
- Pressure Regulator Area History



Notice that area changes to maintain defined downstream pressure.

Set Up Pressure Regulator – Option 2 (1/2)

- Model the Forward-Looking option
  - Go to Advanced/Pressure Regulator
  - Select Forward-Looking regulator option
  - Fill in the dialog boxes



Pressure Regulator

Pressure Regulators

Pressure Regulator 1

Regulator Option: Forward-Looking

Branch: 12

Maximum Area: 0.04 in²

Minimum Area: 1e-16 in²

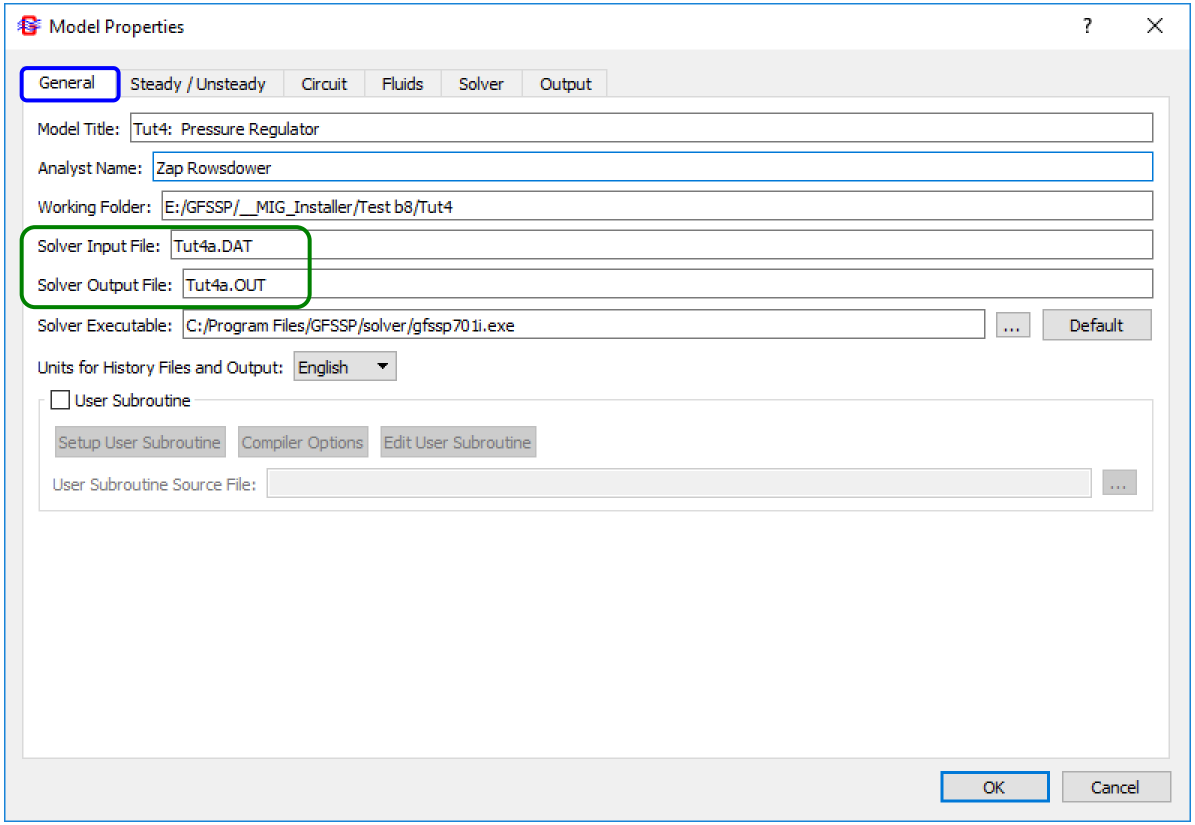
Pressure Option: Pressure History File

Pressure History File: Preg.dat

Under Relaxation Factor: 0.3

Set Up Pressure Regulator – Option 2 (2/2)

- Rename **GFSSP** files
  - Prevents overwriting of first Pressure Regulator results
- Under General tab
  - Rename Input File: Tut4a.DAT
  - Rename Output File: Tut4a.OUT



Model Properties

Model Title: Tut4: Pressure Regulator

Analyst Name: Zap Rowsdower

Working Folder: E:/GFSSP/__MIG_Installer/Test b8/Tut4

Solver Input File: Tut4a.DAT

Solver Output File: Tut4a.OUT

Solver Executable: C:/Program Files/GFSSP/solver/gfssp701i.exe

Units for History Files and Output: English

☐ User Subroutine

User Subroutine Source File:

Results of Pressure Regulator – Option 2 (1/2)

- 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.
  - **GFSSP**'s 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.

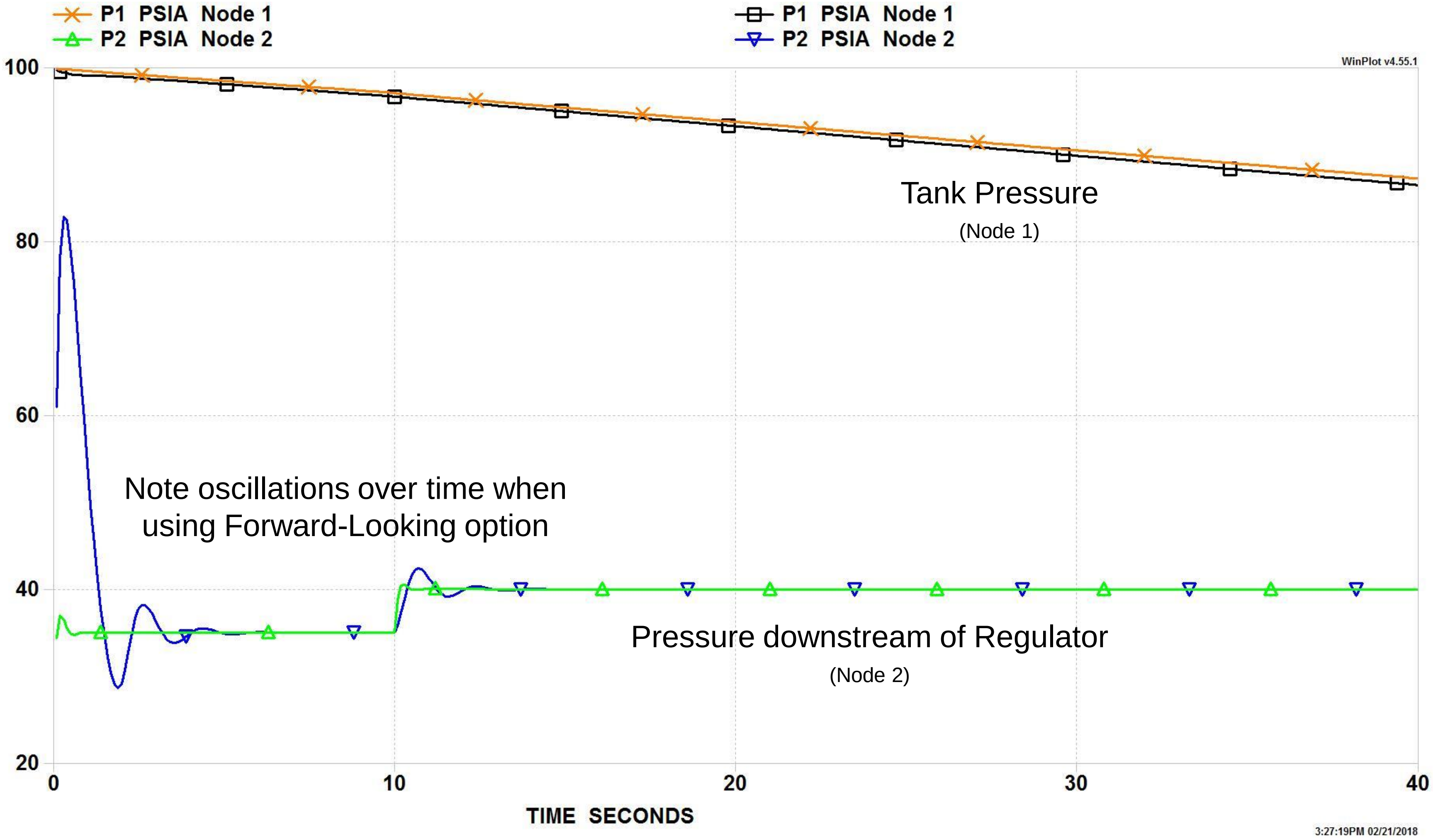
$$A_{new} = A_\tau \left(\frac{p_{reg}}{p_\tau}\right)^3 \left(e^{\left(\frac{p_{reg}}{p_\tau}-1\right)}\right)$$

- Plot the new Option 2 results (Tut4a.WPL) over the Option 1 results (Tut4.WPL)

- Time permitting, try rerunning Option 2 with a different relaxation factor and note its effect on the pressure oscillations.

Results of Pressure Regulator – Option 2 (2/2)

- Pressure History



Tank Pressure

(Node 1)

Note oscillations over time when using Forward-Looking option

Pressure downstream of Regulator

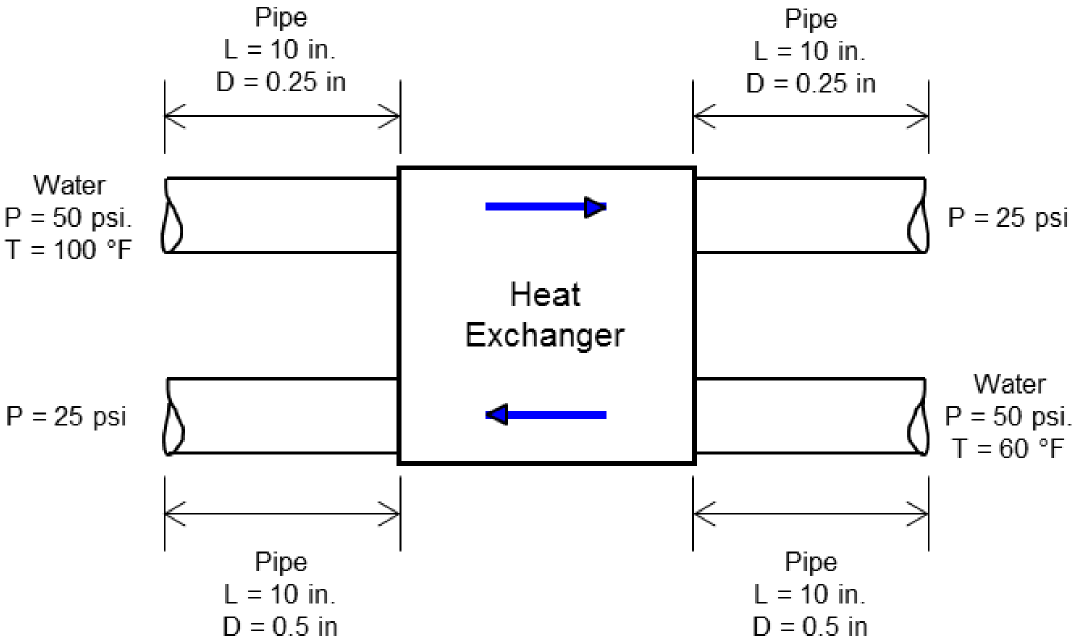
(Node 2)

Challenge Problem 4 (1/2)

Simulation of a Flow System Involving a Heat Exchanger

Given:

Hot and cold water streams enter the system shown below at 50 psia and exit at 25 psia. The hot water enters at 100 °F; the cold, 60 °F. In addition to the 10-inch inlet and exit pipes, the counterflow heat exchanger may be modeled as 10-inch pipes with diameter of 0.25 inches (hot side) and 0.50 inches (cold side). The heat exchanger effectiveness is known to be 0.7, and the pipes are assumed to be smooth.

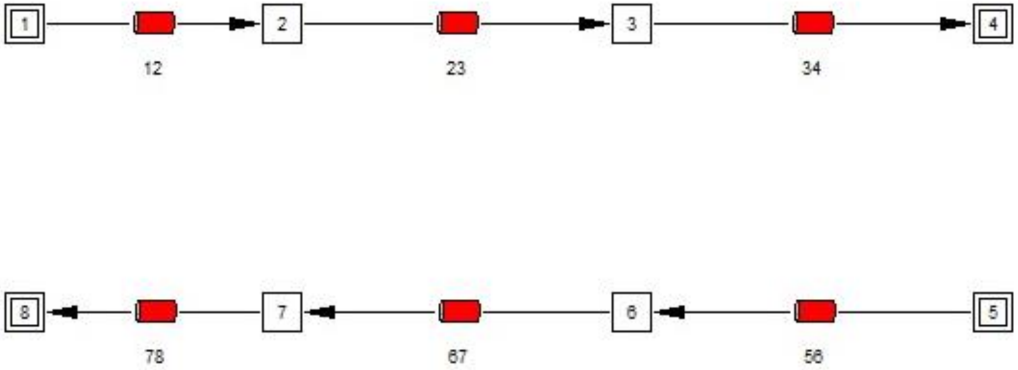


Determine the mass flow rates and exit temperatures of the two streams.

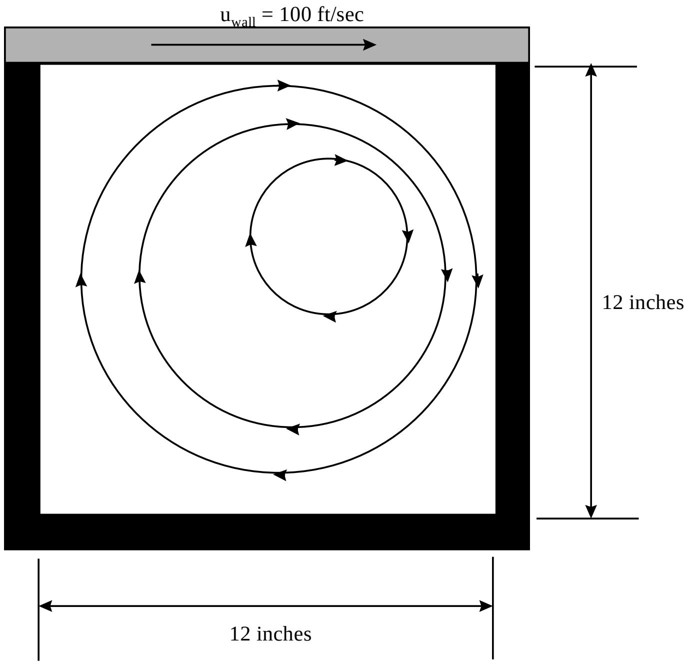
Challenge Problem 4 (2/2)

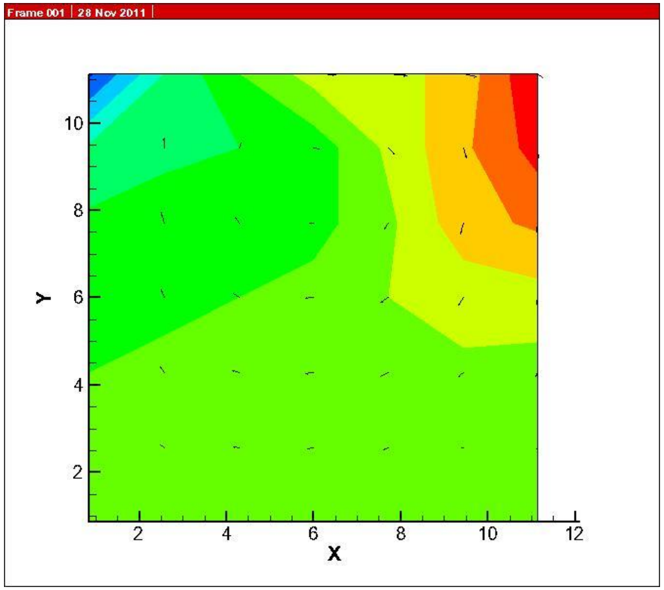
Simulation of a Flow System Involving a Heat Exchanger

- Hints:
  - Because **GFSSP**’s energy equation uses an upwind scheme, exit boundary temperatures are dummy values
  - The heat exchanger option is activated on the Circuit tab; the dialog box is accessed from the Advanced menu
  - When the heat exchanger effectiveness is known, the product of the overall heat transfer coefficient and the area (UA) does not need to be specified.
  - Answers: Hot: 0.885 lb/s; 72.1 °F Cold: 5.41 lb/s; 64.6 °F



Multi-Dimensional Flow Modeling and Psychrometric Properties





Multi-D Terms in Momentum Equation

- Momentum Conservation Equation

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

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

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

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

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

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

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

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

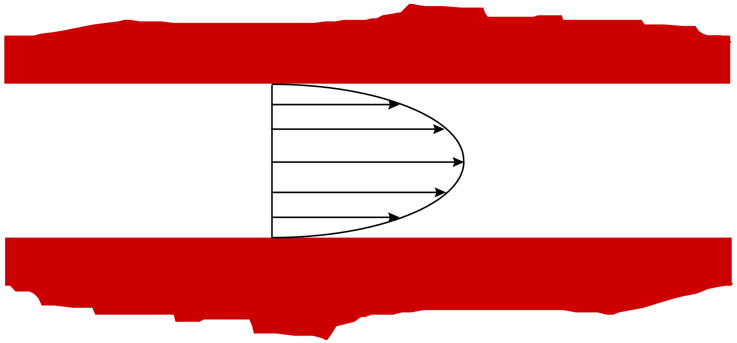
Validation of GFSSP Prediction

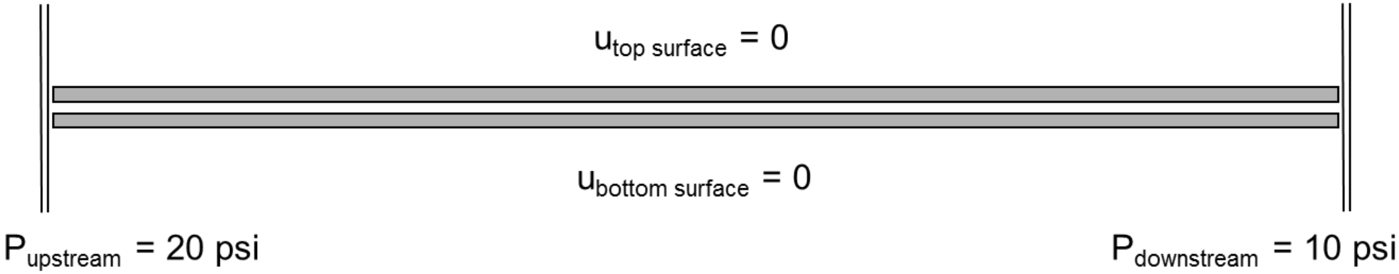
- Three classical fluid dynamics problems have been considered for validation of **GFSSP** Prediction
    - Poiseulle Flow
        - Shear dominated flow between two stationary flat plates
    - Couette Flow
        - Shear driven flow between one moving flat plate and one stationary flat plate
    - Driven Cavity Flow
        - 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 (1/2)

- Analytical Solution: $u = 0.005(y - y^2)$

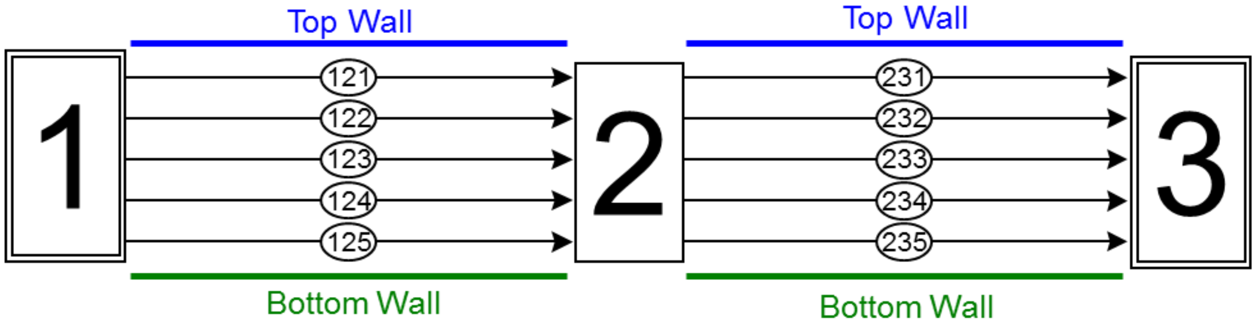


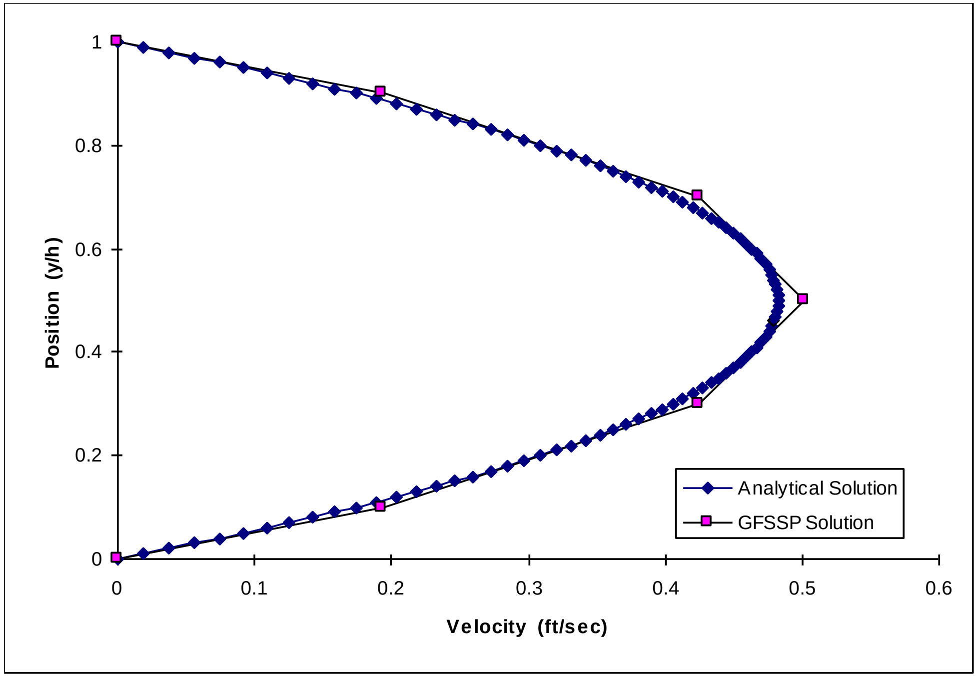


Length = 1000 in.
Distance between Plates = 1 in.
Fluid Density = 12 lb/ft^3
Fluid Viscosity = 1 lb/ft-sec

Poiseulle Flow (2/2)

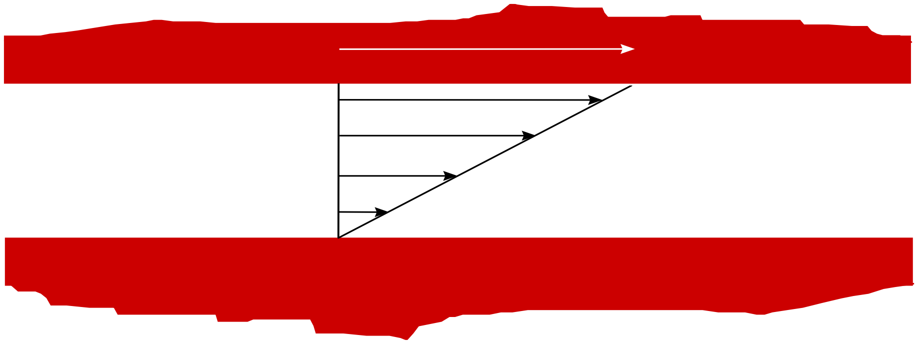
- **GFSSP** Model

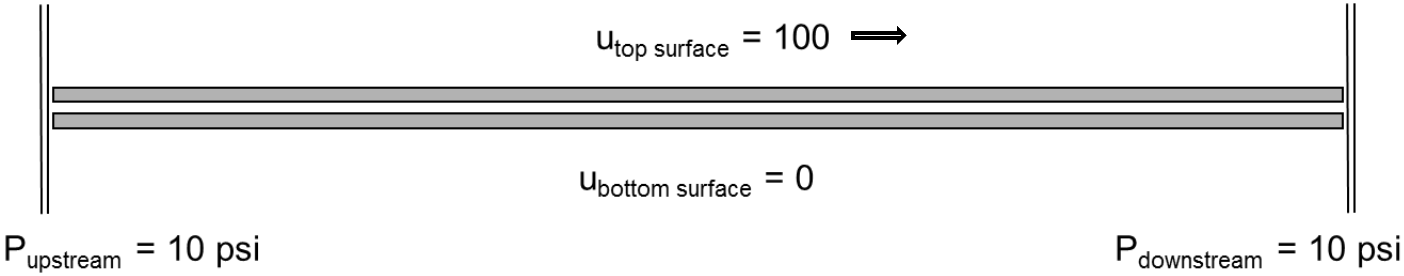




Couette Flow **(1/2)**

- Analytical Solution: $u = 100y$

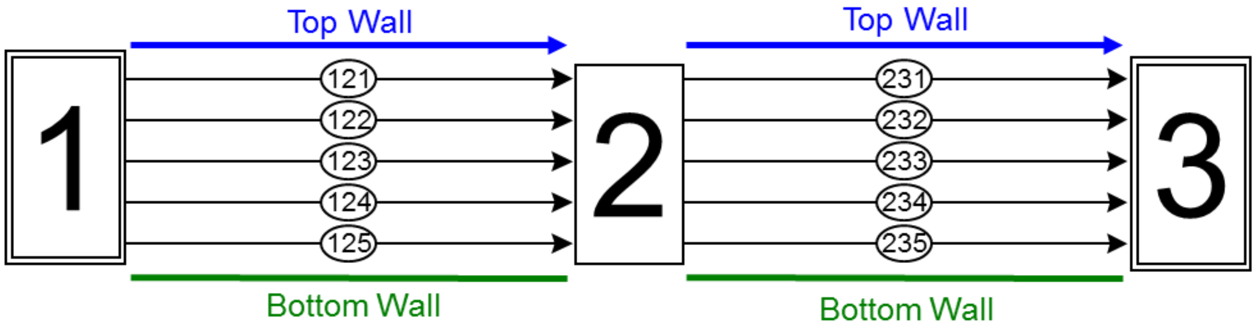


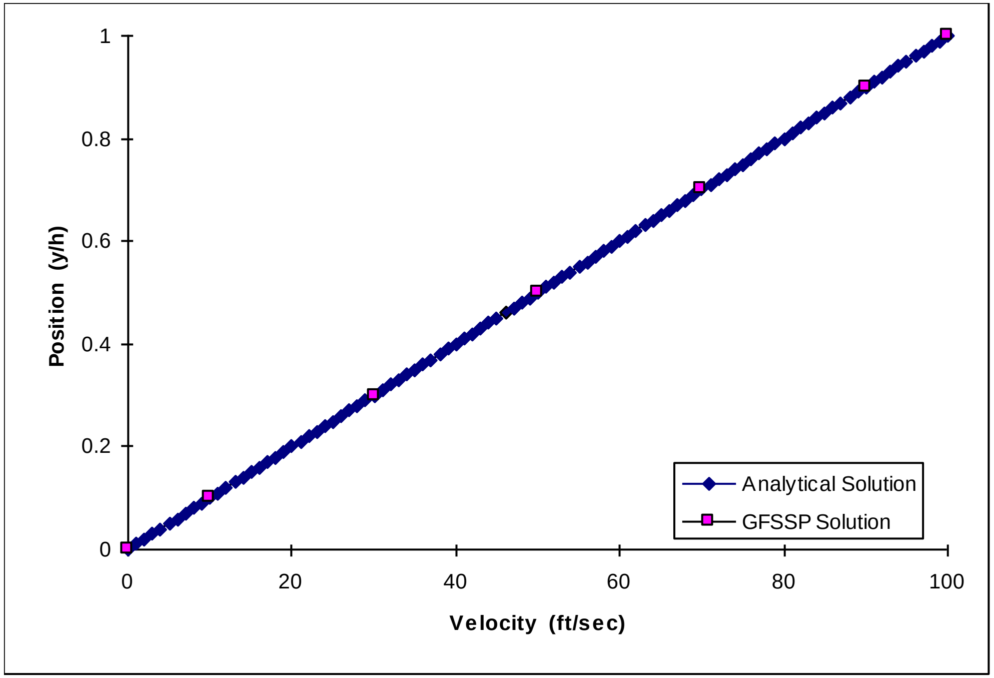


Length = 1000 in.
Distance between Plates = 1 in.
Fluid Density = 12 lb/ft^3
Fluid Viscosity = 1 lb/ft-sec

Couette Flow (2/2)

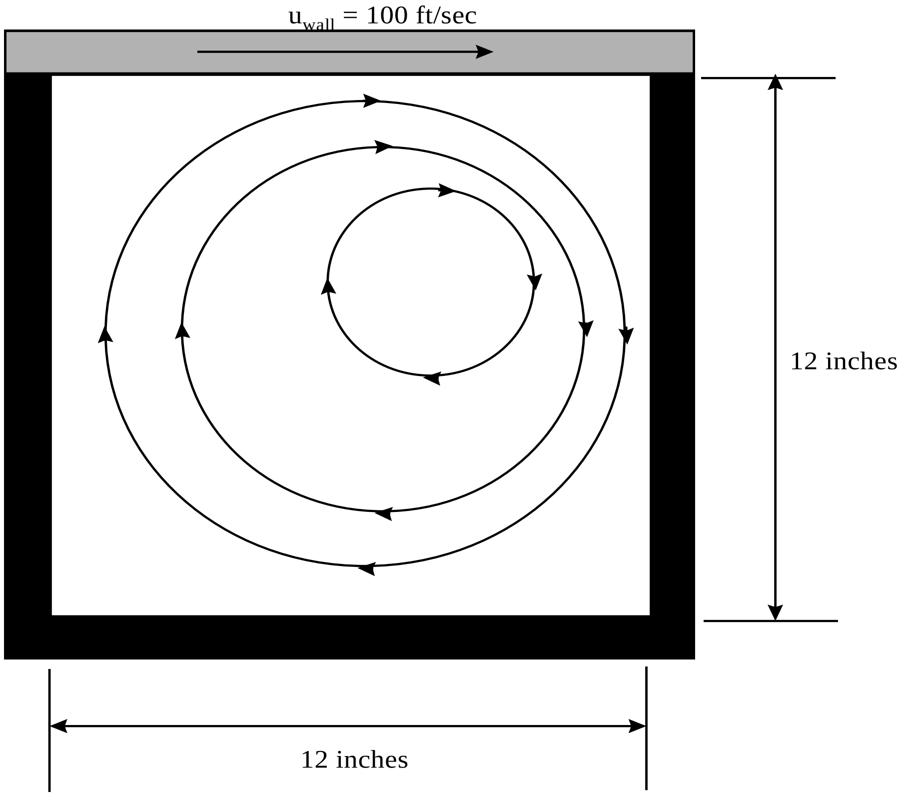
- **GFSSP** Model

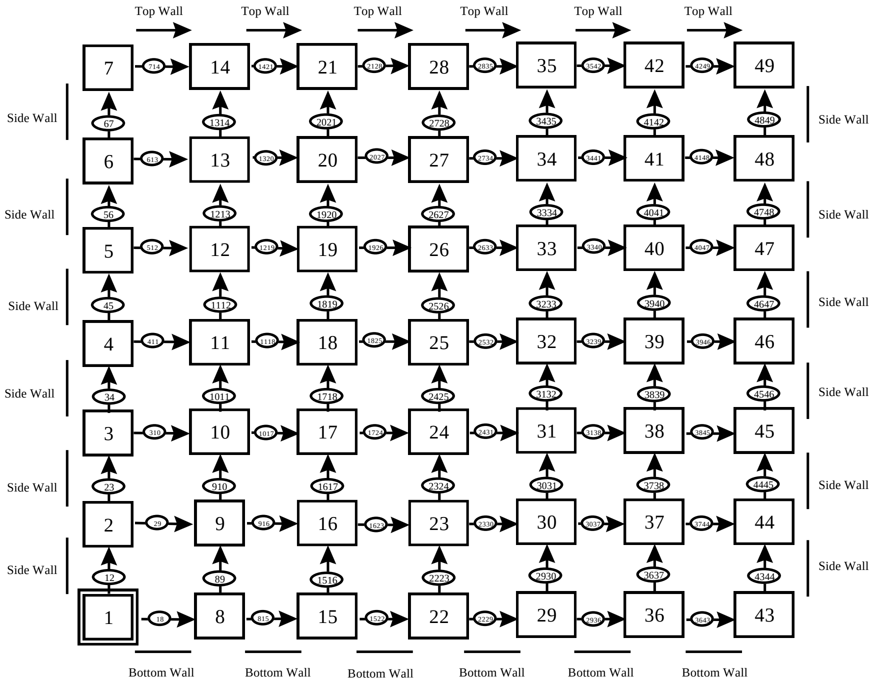




Ex25: 2-D Recirculating Flow in a Driven Cavity (2/3)

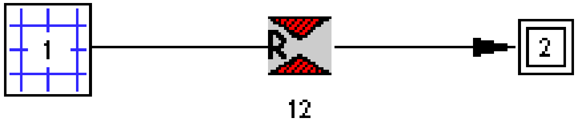
- Fluid inputs
  - Density = 1 lb/ft^3
  - Viscosity = 1 lb/ft-sec
  - Reynolds Number = 100

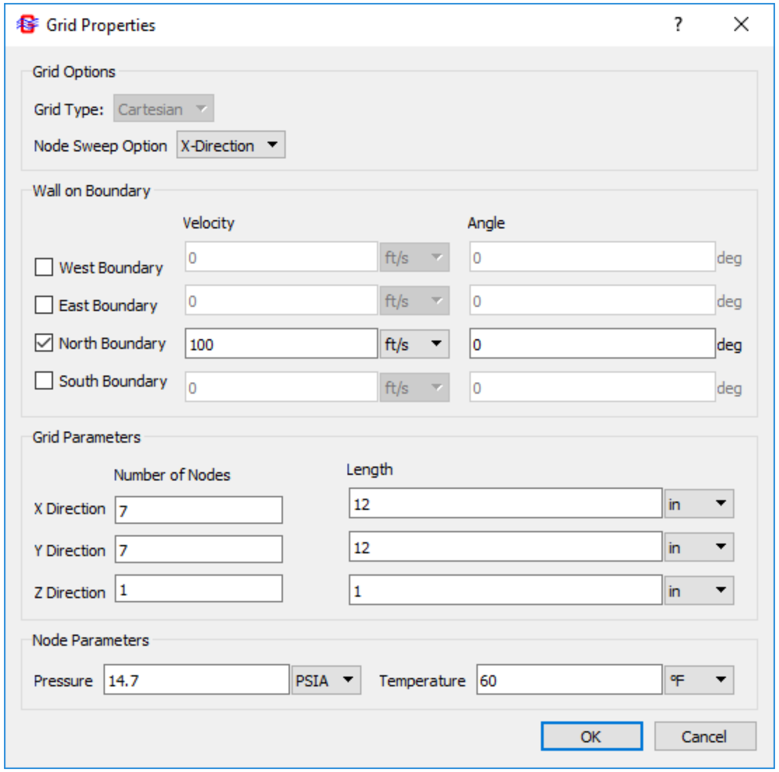




Ex25: 2-D Recirculating Flow in a Driven Cavity (2/3)

- Linear Cartesian Grid Generation and Display of 2-D cartesian grid





Grid Properties

Grid Options

Grid Type: Cartesian

Node Sweep Option: X-Direction

Wall on Boundary

	Velocity		Angle	
☐ West Boundary	0	ft/s	0	deg
☐ East Boundary	0	ft/s	0	deg
☑ North Boundary	100	ft/s	0	deg
☐ South Boundary	0	ft/s	0	deg

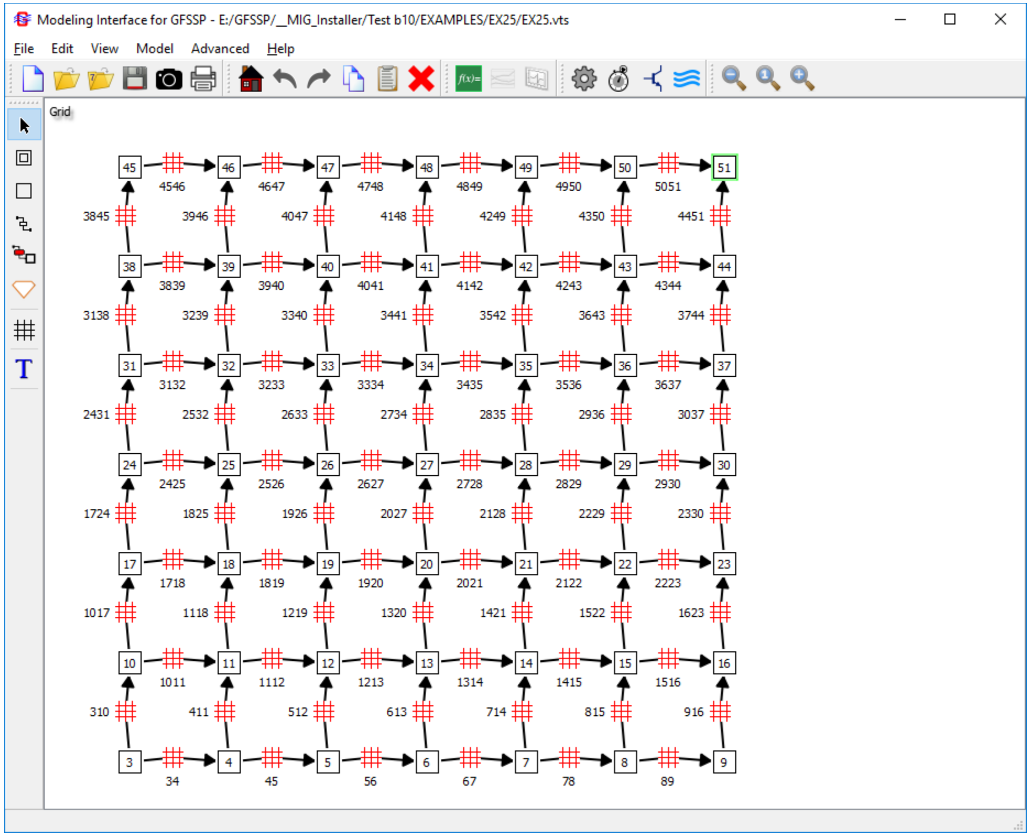
Grid Parameters

	Number of Nodes	Length	
X Direction	7	12	in
Y Direction	7	12	in
Z Direction	1	1	in

Node Parameters

Pressure: 14.7 PSIA

Temperature: 60 °F

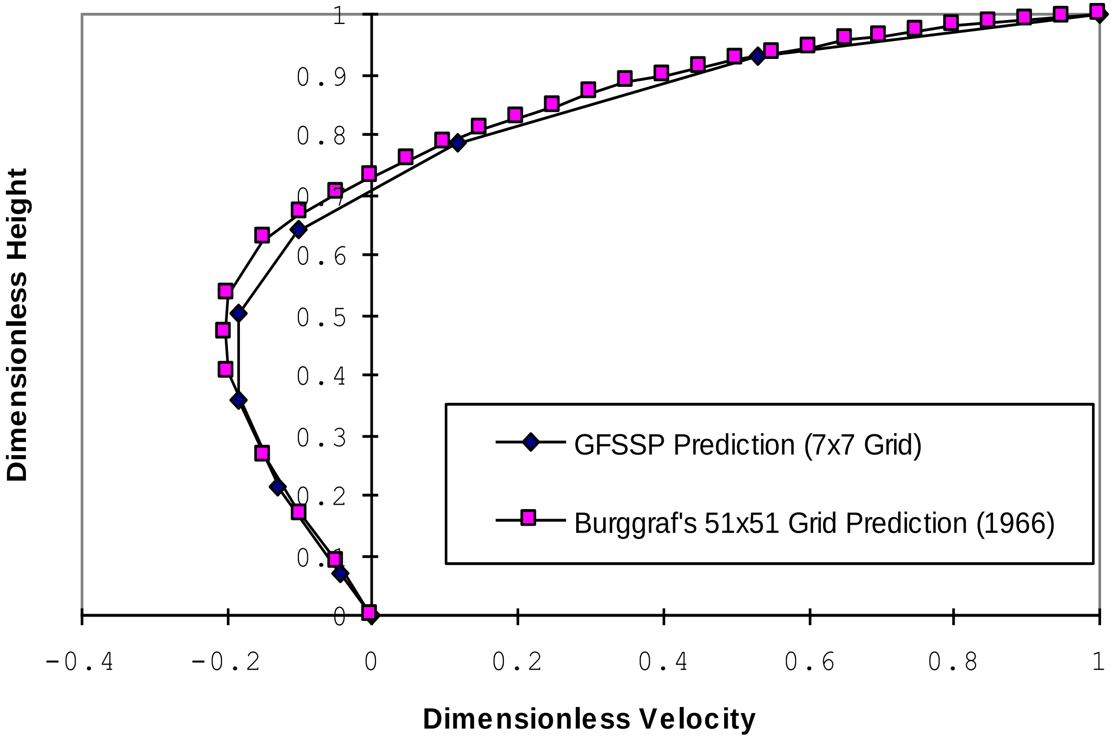


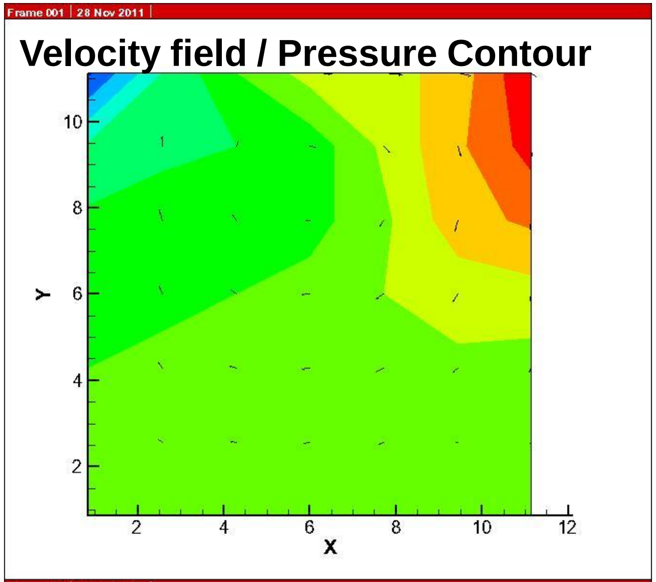
Modeling Interface for GFSSP - E:/GFSSP/__MIG_Installer/Test b10/EXAMPLES/EX25/EX25.vts

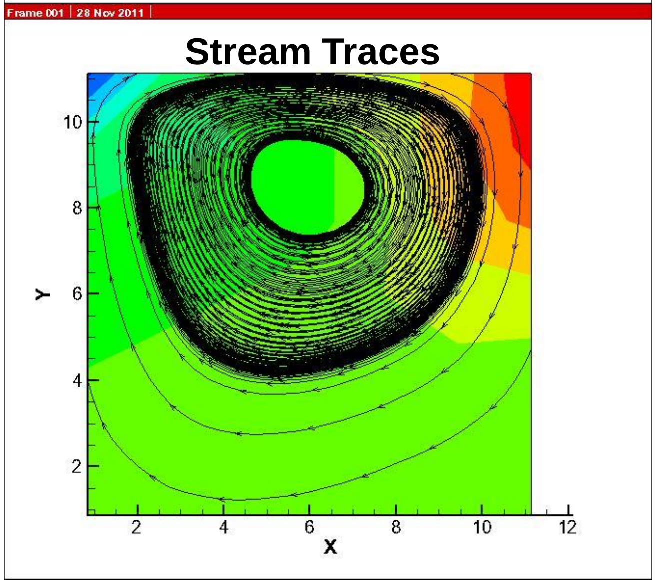
Ex25: 2-D Recirculating Flow in a Driven Cavity (3/3)

Marshall Space Flight Center
GFSSP Training Course

- Comparison with Benchmark Solution







Multi-dimensional Summary

- **GFSSP'**s Numerical Algorithm 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

- Extended formulation has been validated by comparing the numerical prediction with three benchmark solutions:
  - Poiseulle Flow
  - Couette Flow
  - Flow in a Driven Cavity

- Future work will include Heat Transfer & Turbulent Flow

Psychrometric Properties

- Definition of Psychrometric Property
- Subroutines for Psychrometric Property Calculation
- Control parameter for Psychrometric Option
- Example 31: Modeling Psychrometrics of Air-Water Vapor Mixture

Definition of Psychrometric Properties

- Dalton's Law of Partial Pressure

$$p = p_a + p_v$$

- Humidity Ratio

$$\omega = \frac{m_v}{m_a} = \frac{0.622 p_v}{p - p_v}$$

- Carrier Equation

$$\frac{(p - p_{wb})(T_{DB} - T_{WB})}{2831 \; - 1.43 T_{WB}} = P_{WB} - P_V$$

- Relative Humidity

$$\varphi = \frac{m_v}{m_g} = \frac{p_v}{p_g}$$

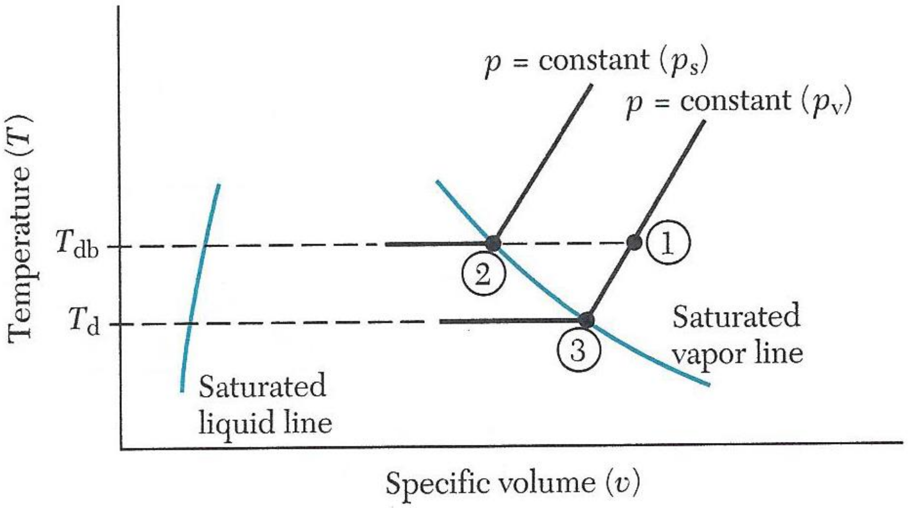
- Dew Point Temperature (at p_v)

$$T_d = \; T_{sat}$$

- Vapor Pressure Relation for Water:

$$ln(p_{sat}) = A + \frac{B}{T_{sat}} + Cln(T_{sat}) + DT_{sat}$$

where:
$A = 99.4824;\ B = -7894.6011;\ C = -11.9783;\ D = 0.01101$



Psychrometric Property Calculation

- New Subroutines in **GFSSP**
  - PSAT(T,P)
    - Calculates saturation pressure of water at a given temperature
  - TSATT(P,T,TGUESS)
    - Calculates saturation temperature of water from vapor pressure relation by N-R Method
  - TWBCAR(TDB,PDP,PAMB,TWB)
    - Calculates wet-bulb temperature from the Carrier Equation by N-R Method
  - CARIER(TWB,TDB,PWB,PAMB,PDP)
    - Calculates the pressure at the dew point temperature

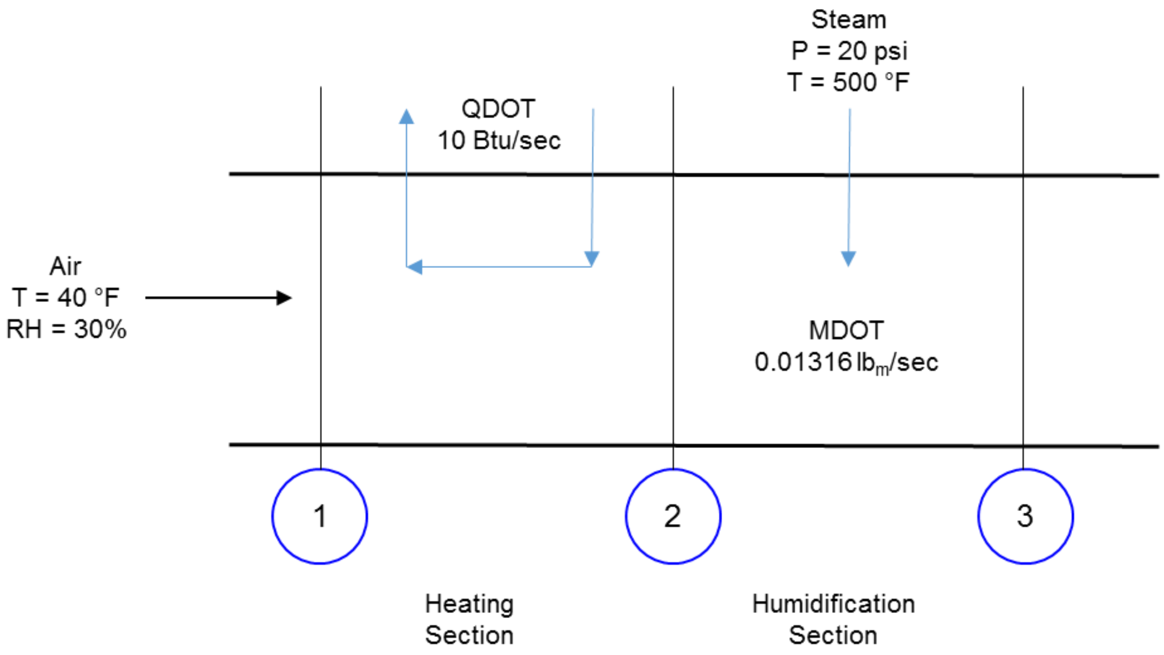
Psychrometric Option Control Parameter

- Control Parameter : IOPTPSY
  - IOPTPSY = 0: Psychometric Property Inactive
  - IOPTPSY = 1: Input Relative Humidity (PHI)
  - IOPTPSY = 2: Input Wetbulb Temperature (TWB)
  - IOPTPSY = 3: Input Humidity Ratio (OMEGA)
- Activation of Psychrometric Option
  - **GFSSP** reads either PHI, TWB or OMEGA for both steady-state and transient models
  - Boundary History File requires one of the three properties in addition to pressure, temperature and concentration
- Uses GASPAK Option for Air
- Uses GASP/WASP Option for Water

Ex31: Modeling Psychrometrics of Air-Water Vapor Mixture (1/6)

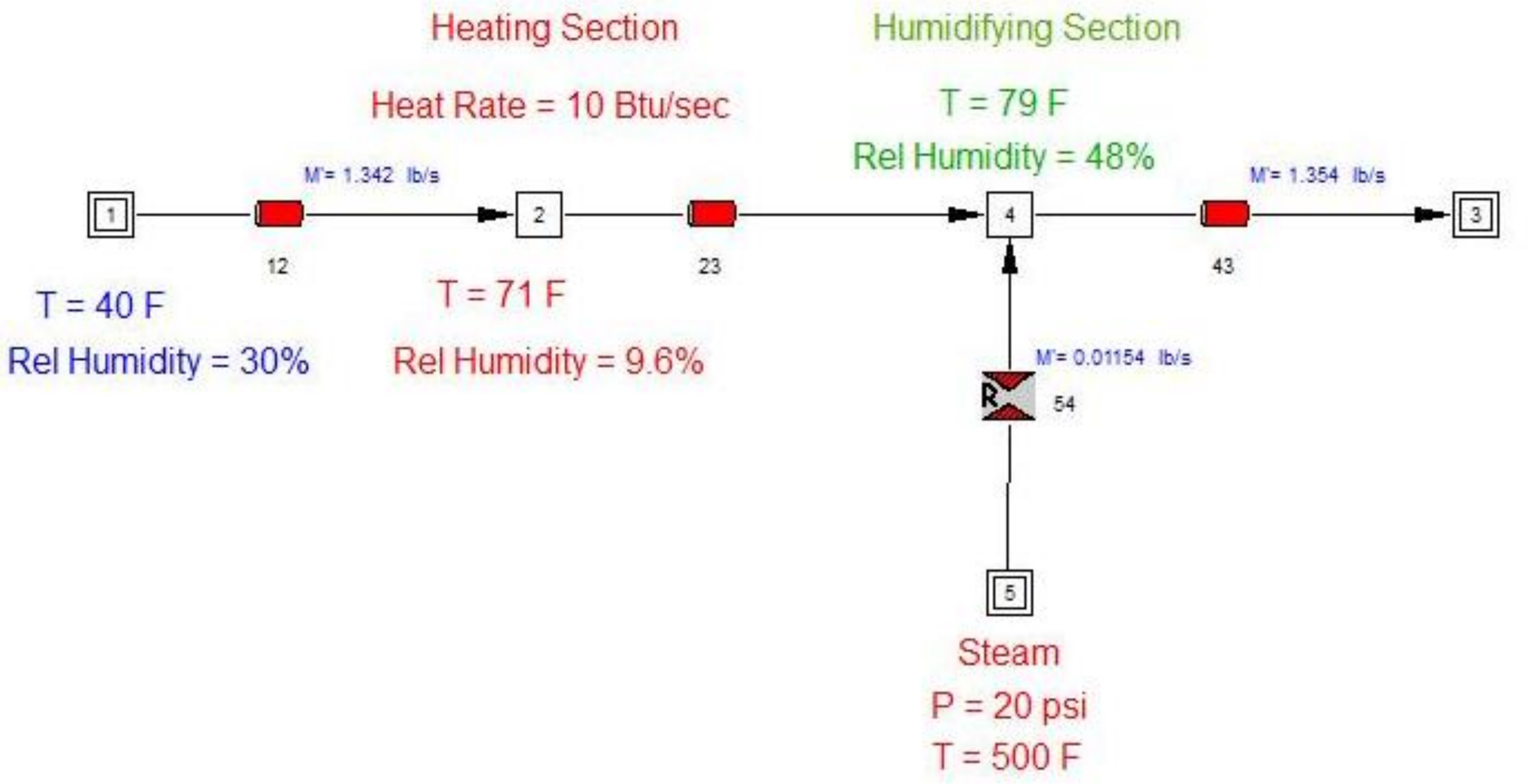
- Cold and dry air enters into an air-conditioning system
  - Air is first heated and then humidified

- **GFSSP** model purpose
  - Calculate the temperature and relative humidity of the air at the exit of the air conditioner



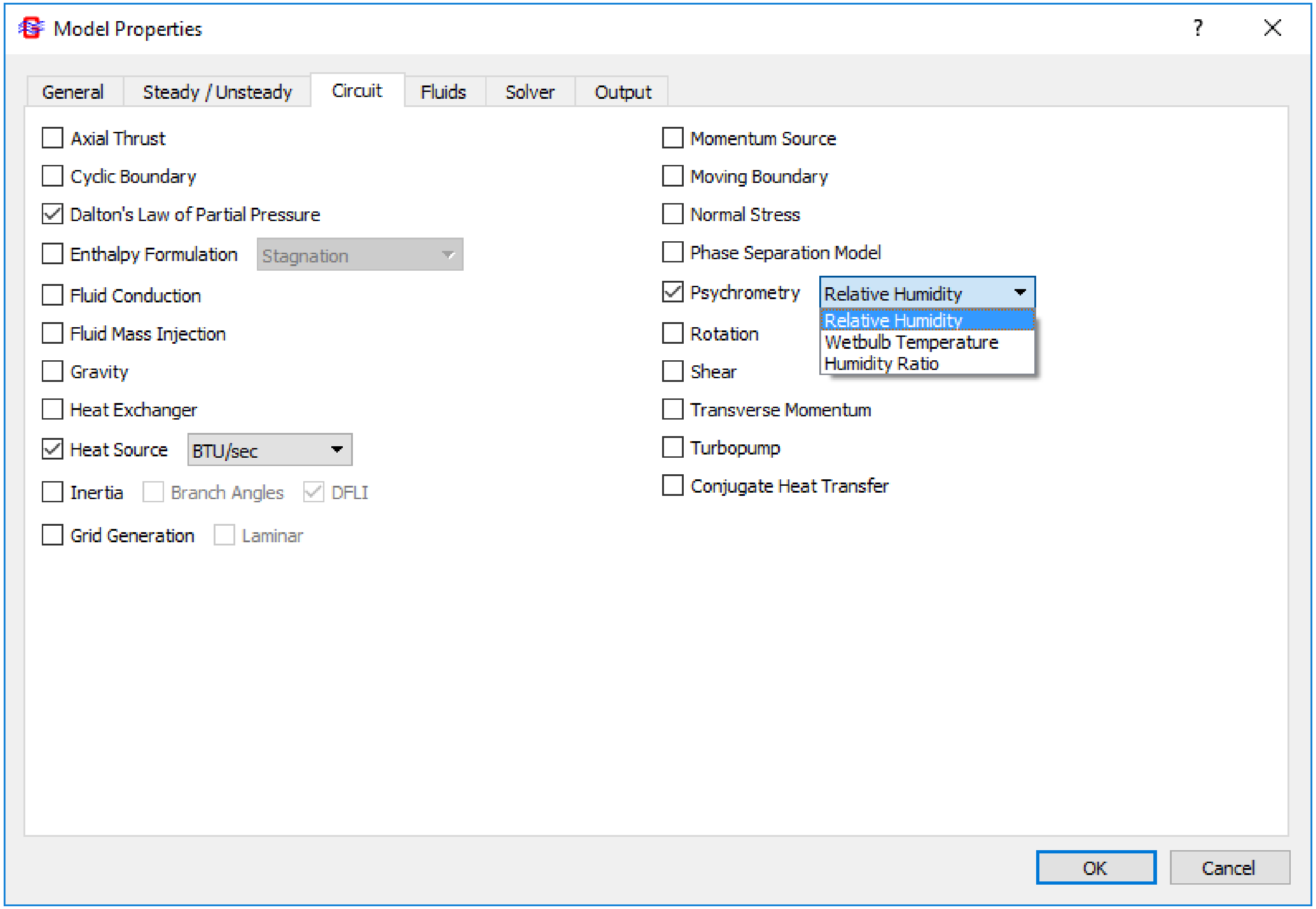
Ex31: Modeling Psychrometrics of Air-Water Vapor Mixture (2/6)

- **GFSSP** Model



Ex31: Modeling Psychrometrics of Air-Water Vapor Mixture (3/6)

- Activation of Psychrometry in Circuit Option



Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

☐ Axial Thrust
☐ Cyclic Boundary
☑ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation: Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☐ Heat Exchanger
☑ Heat Source: BTU/sec
☐ Inertia ☐ Branch Angles ☑ DFLI
☐ Grid Generation ☐ Laminar

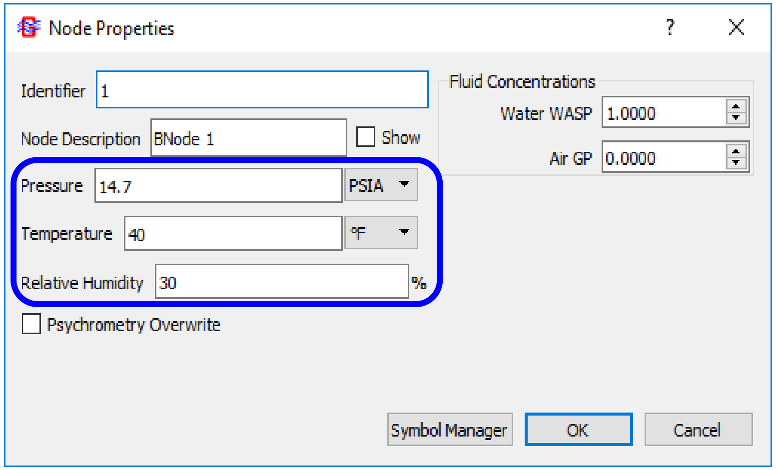
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☑ Psychrometry: Relative Humidity
- Relative Humidity
- Wetbulb Temperature
- Humidity Ratio

☐ Rotation
☐ Shear
☐ Transverse Momentum
☐ Turbopump
☐ Conjugate Heat Transfer

OK | Cancel

Ex31: Modeling Psychrometrics of Air-Water Vapor Mixture (4/6)

- Boundary Node Properties for Psychrometrics
  - Boundary Node 1
    - Based on 30% RH at this P/T
    - Mass Fractions for air and water calculated from the input RH. Fluid Concentrations are ignored.
  - Boundary Node 5
    - Select Psychrometry Overwrite
    - Allows users to specify 100% water as the Fluid Concentration. Input RH is ignored.



Node Properties

Identifier: 1

Node Description: BNode 1 ☐ Show

Pressure: 14.7 PSIA

Temperature: 40 °F

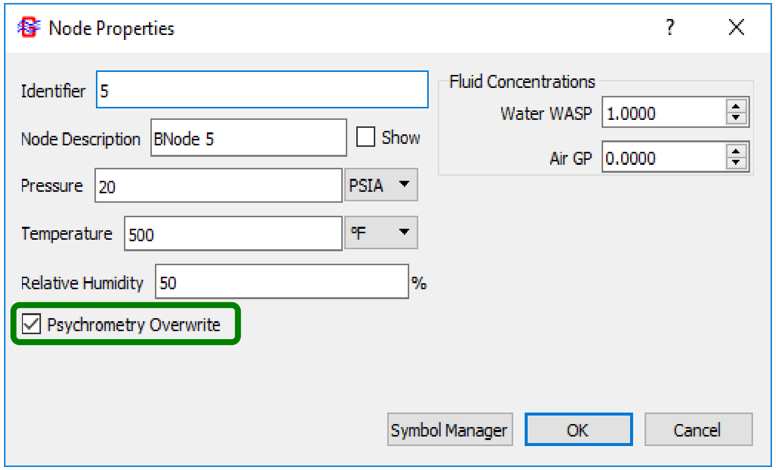
Relative Humidity: 30 %

☐ Psychrometry Overwrite

Fluid Concentrations

Water WASP: 1.0000

Air GP: 0.0000



Node Properties

Identifier: 5

Node Description: BNode 5 ☐ Show

Pressure: 20 PSIA

Temperature: 500 °F

Relative Humidity: 50 %

☑ Psychrometry Overwrite

Fluid Concentrations

Water WASP: 1.0000

Air GP: 0.0000

Ex31: Modeling Psychrometrics of Air-Water Vapor Mixture (5/6)

- Comparison of GFSSP prediction with hand calculation
  - Energy conservation between (1) and (2)

$$\dot{Q} + \dot{m}_a h_1 = \dot{m}_a + h_2$$

$$h_2 = \frac{\dot{Q}}{\dot{m}_a} + h_1 = \frac{10}{1.34} + 3.595 = \mathbf{11.06\ Btu/lb}$$

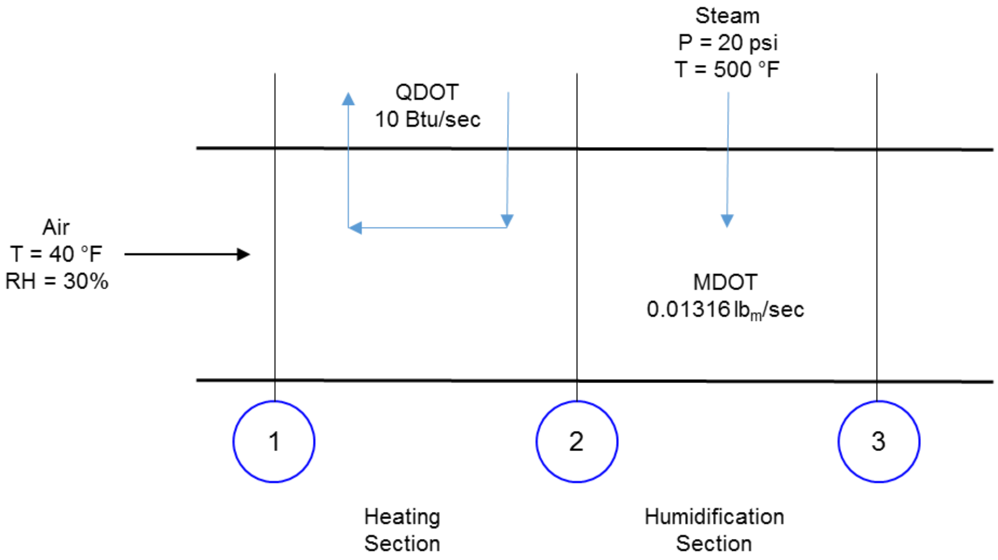
  - **GFSSP** calculates

$$T_2 = 71\,°F$$

$$\varphi_2 = 9.6\%$$

  - PSYCHRO calculates

$$h_2 = \mathbf{11.002\ Btu/lb}$$

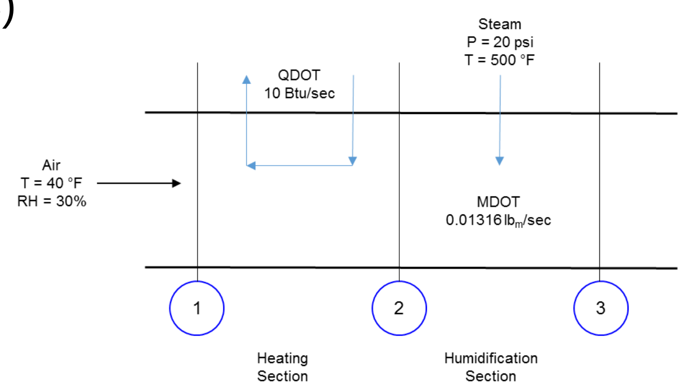


Ex31: Modeling Psychrometrics of Air-Water Vapor Mixture (6/6)

- Comparison of **GFSSP** prediction with hand calculation
  - Humidfier Mass Conservation between (2) and (3)

$$\dot{m}_a \omega_2 + \dot{m}_{H2O} = \dot{m}_a \omega_3$$

$$\omega_3 = \omega_2 + \frac{\dot{m}_{H2O}}{\dot{m}_a} = 0.0015 + \frac{0.01154}{1.34} = \mathbf{0.0101}$$



  - **GFSSP** calculates

$$\omega_3 = \mathbf{0.0102}$$

  - Energy Conservation (2) and (3)

$$h_3 = h_2 + \frac{\dot{m}_{H2O}}{\dot{m}_a} h_v = 11.02 + \frac{0.01154}{1.34}(1287.3) = \mathbf{22.10\ Btu/lb}$$

h_v is the enthalpy of steam used for humidification

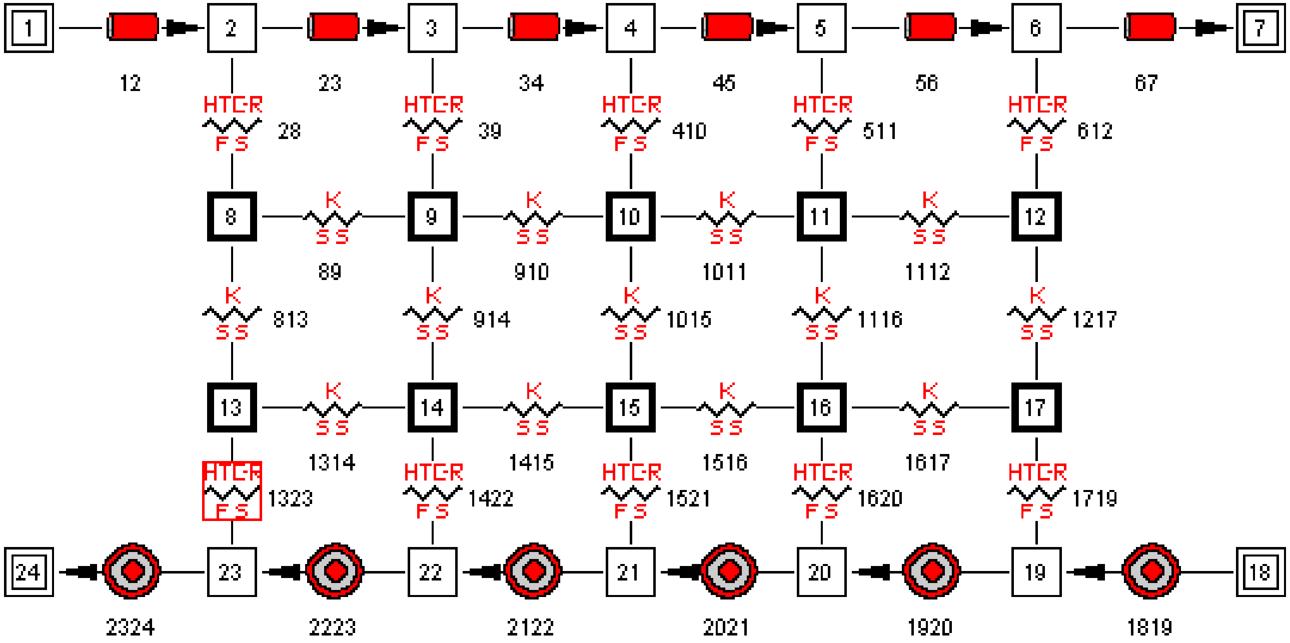
  - **GFSSP** calculates

$$T_3 = 79\ °F, \varphi_3 = 48\%, \omega_3 = \mathbf{0.0102}$$

  - PSYCHRO calculates

$$h_3 = \mathbf{22.13\ Btu/lb}, \omega_3 = \mathbf{0.01017}$$

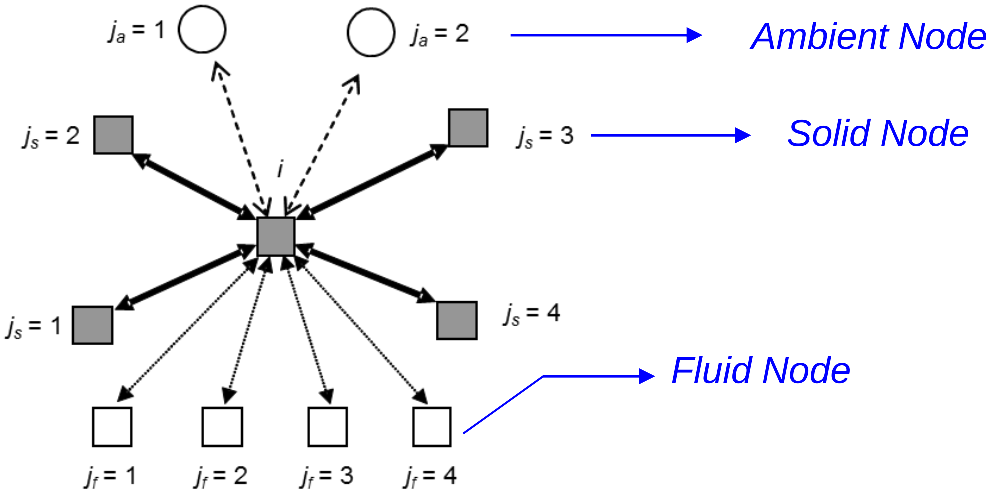
Conjugate Heat Transfer – Modeling Heat Transfer Between Solid and Fluid



Conjugate Heat Transfer

- Why do we need it?
  - Fluid flow and heat transfer are strongly coupled in many applications
  - Typical examples in Propulsion Systems
    - 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

Solid Energy Equation



Conservation Equation

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

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

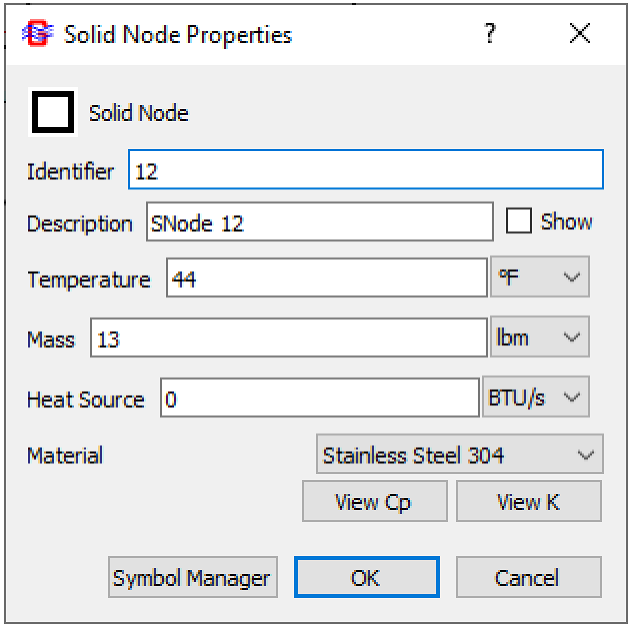
$$\dot{q}_{sf}=h_{ij_f}A_{ij_f}\left(T_f^{j_f}-T_s^i\right)$$

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

Successive Substitution Form

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

Solid Node Input (1/2)



Solid Node Properties

Solid Node

Identifier: 12

Description: SNode 12 ☐ Show

Temperature: 44 ºF

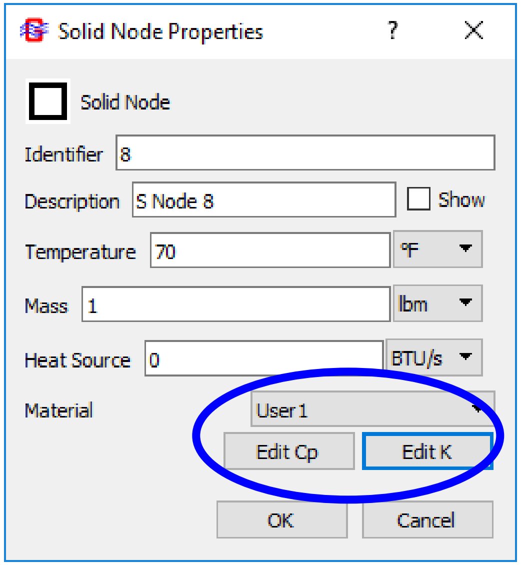
Mass: 13 lbm

Heat Source: 0 BTU/s

Material: Stainless Steel 304

Solid Node Input (2/2)

- Material Properties
  - **GFSSP** installation directory contains temperature-dependent properties (k and Cp) for 40 common materials
  - CAUTION: Not all library materials contain properties at cryogenic temperatures.
- Up to 5 user-defined material properties may be defined in short text files
  - User1k.prp and user1cp.prp
    - ➢ User2k.prp and user2cp.prp
    - ➢ Etc.
- Units
  - T (°R)
  - K (BTU/s-ft-°R)
  - Cp (BTU/lb$_m$-°R)



☐ Solid Node

Identifier: 8

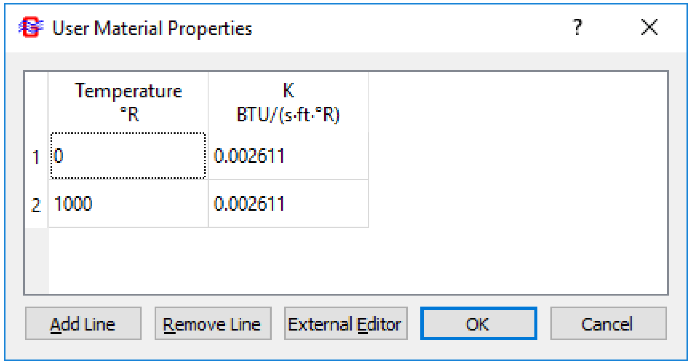
Description: S Node 8 ☐ Show

Temperature: 70 °F

Mass: 1 lbm

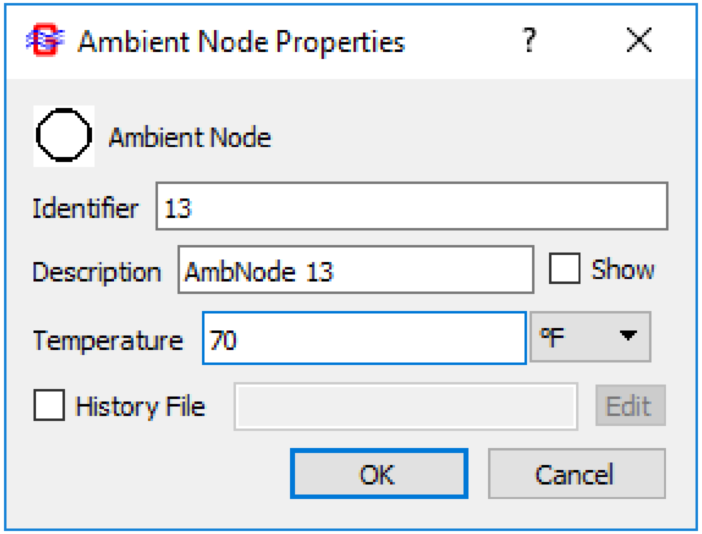
Heat Source: 0 BTU/s

Material: User 1



	Temperature °R	K BTU/(s·ft·°R)
1	0	0.002611
2	1000	0.002611

Ambient Node Input



Ambient Node

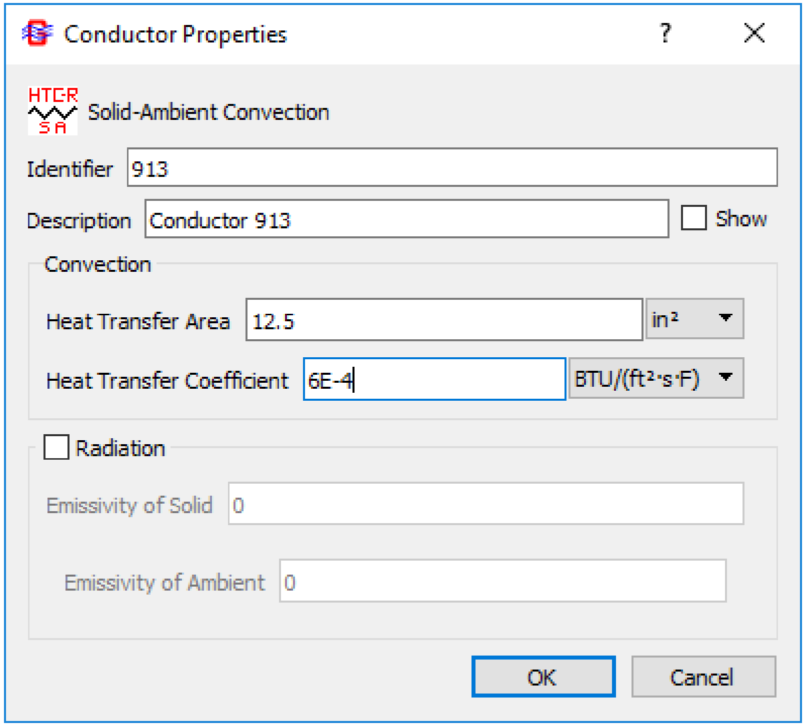
Identifier: 13

Description: AmbNode 13 ☐ Show

Temperature: 70 °F

☐ History File

Ambient to Solid Conductor



Conductor Properties

Solid-Ambient Convection

Identifier: 913

Description: Conductor 913 ☐ Show

Convection

Heat Transfer Area: 12.5 in²

Heat Transfer Coefficient: 6E-4 BTU/(ft²·s·F)

☐ Radiation

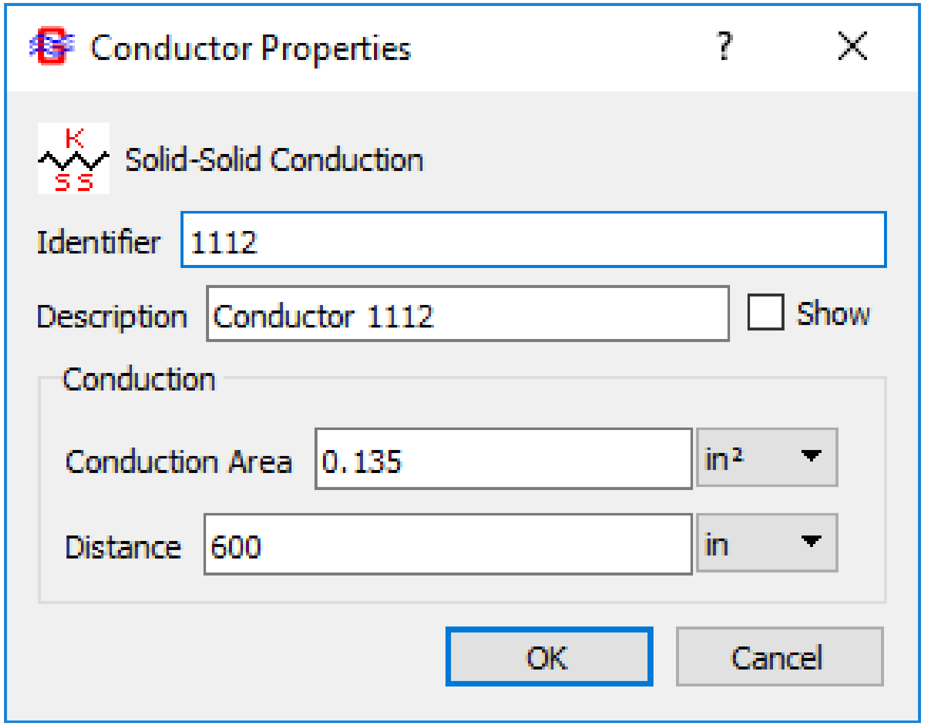
Emissivity of Solid: 0

Emissivity of Ambient: 0

- Radiation option for S-A Conductor
  - View factor = 1.0
  - ε must be > 0.0 to avoid division by zero error.
  - When ε_{amb} = 1.0, simplifies to equation for small object surrounded by a large ambient.

$$q_{s-amb} = \frac{\sigma A(T_s^4 - T_{amb}^4)}{\frac{1}{\varepsilon_s} + \frac{1}{\varepsilon_{amb}} - 1}$$

Solid to Solid Conduction Conductor (1/2)



Identifier 1112

Description Conductor 1112 ☐ Show

Conduction

Conduction Area 0.135 in²

Distance 600 in

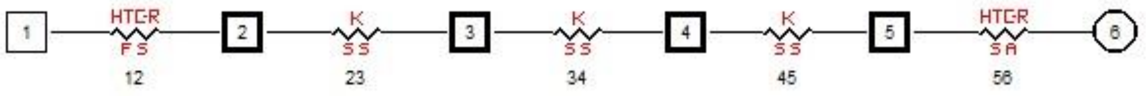
Solid to Solid Conduction Conductor (2/2)

- Mixing Materials
  - When a solid-to-solid conductor connects two different materials, the effective conductivity is the harmonic mean of the two conductivities.

$$k_{AB} = \frac{2k_A k_B}{k_A + k_B}$$

  - This relationship holds true only if the length of the solid-to-solid conductor is ½ material A and ½ material B.

- Example: A tank wall is 0.25 inches thick and covered with 1.0 inch of insulating foam.



Wall (0.20”)

Foam (0.95”)

Interface (0.1”)
(0.05” + 0.05”)

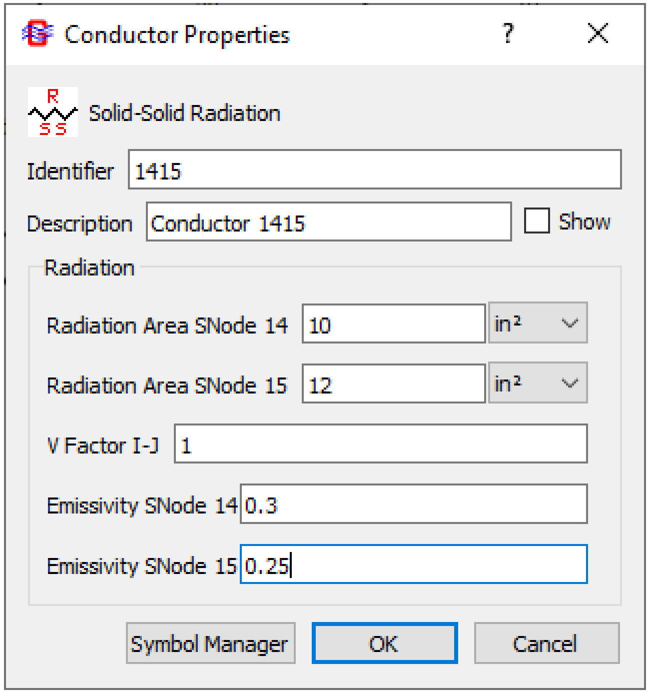
Solid to Solid Radiation Conductor

- Assumes two diffuse, gray surfaces that form an enclosure

- Example: a pipe surrounded by a vacuum jacket (F_{ij} = 1.0)

$$q_{ij} = \frac{\sigma(T_i^4 - T_j^4)}{\frac{1-\epsilon_i}{\varepsilon_i A_i} + \frac{1}{A_i F_{ij}} + \frac{1-\epsilon_j}{\varepsilon_j A_j}}$$

- Ith node is the first solid node selected.
- Jth node is the second solid node.
- Hint: when modeling a vacuum jacket, make the inside pipe Solid Node I, and the outside pipe Solid Node J. Then the view factor F_{ij} = 1.0.


Conductor Properties

Solid-Solid Radiation

Identifier: 1415

Description: Conductor 1415 ☐ Show

Radiation

Radiation Area SNode 14: 10 in²

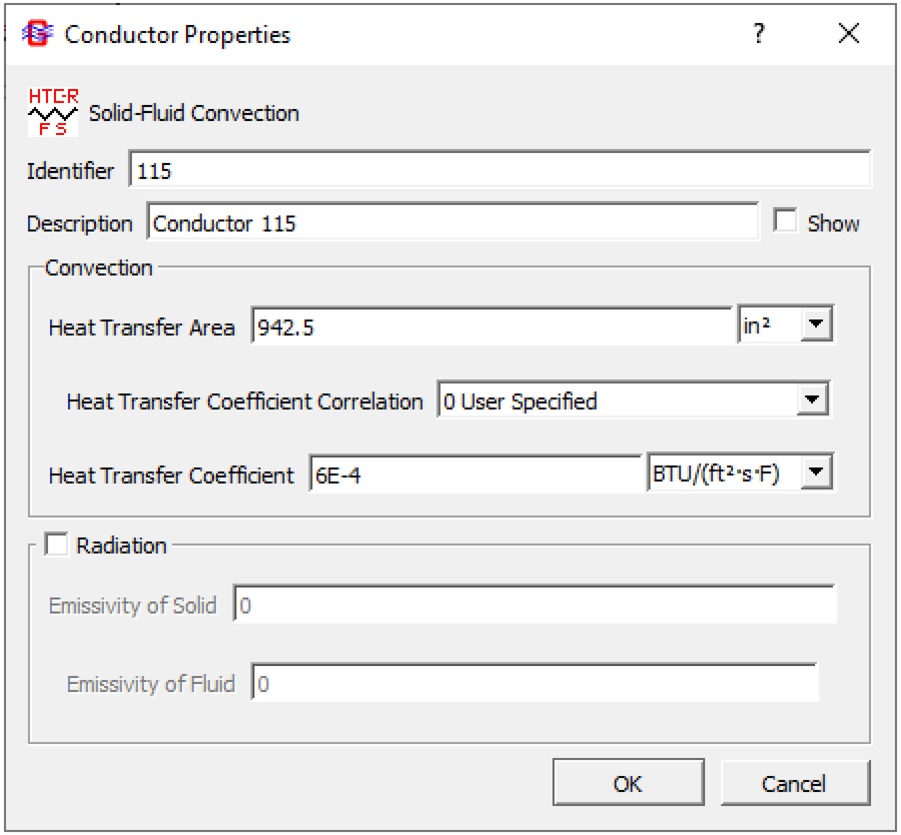
Radiation Area SNode 15: 12 in²

V Factor I-J: 1

Emissivity SNode 14: 0.3

Emissivity SNode 15: 0.25

Solid to Fluid Conductor



Conductor Properties

Solid-Fluid Convection

Identifier: 115

Description: Conductor 115 ☐ Show

Convection

Heat Transfer Area: 942.5 in²

Heat Transfer Coefficient Correlation: 0 User Specified

Heat Transfer Coefficient: 6E-4 BTU/(ft²·s·F)

☐ Radiation

Emissivity of Solid: 0

Emissivity of Fluid: 0

Solid to Fluid Conductor (1/8)

- Heat Transfer Coefficient
  - User-specified to a constant value set in MIG (Option 0)

  - Calculated by a correlation defined in a Fortran user subroutine

  - Calculated by built-in correlations for Forced Convection in a Pipe
    1. Dittus-Boelter
    2. Miropolskii
    3. Sieder-Tate
    4. Petukhov
    5. Gnielinski

  - Calculated by built-in correlations for Natural Convection to a Vertical Plate
    6. Empirical
    7. Churchill-Chu

Solid to Fluid Conductor (2/8)

- Dittus-Boelter (Option 1)
  - Properties evaluated at fluid node temperature.
  - Difference between fluid and wall temperatures should be less than 10 °F for liquids, less than 100 °F for gases.
  - Uses Colburn formulation where Prandtl exponent is always 1/3.
  - Valid range:
    - $0.7 \leq \mathrm{Pr} \leq 160$
    - $\mathrm{Re} \geq 10{,}000$

$$Nu = \frac{hD}{k} \quad \Longrightarrow \quad h = \frac{Nu\,k}{D}$$

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

$$\mathrm{Re} = \frac{\rho u D}{\mu} \qquad \mathrm{Pr} = \frac{C_p \mu}{k}$$

Solid to Fluid Conductor (3/8)

- Miropolskii (Option 2)
  - Film-boiling correlation for two-phase flow
  - Switches to Dittus-Boelter for single-phase flow
  - Suitable for chilldown problems, which are mostly film-boiling
  - Not accurate for nucleate boiling regime

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

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

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

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

Solid to Fluid Conductor (4/8)

- Sieder-Tate (Option 3)
  - Preferred over Dittus-Boelter when there are large temperature differences between fluid and wall.
  - Valid range:
    - $0.7 \leq \mathrm{Pr} \leq 16{,}700$
    - $\mathrm{Re} \geq 10{,}000$

$$Nu = 0.027\mathrm{Re}^{0.8}\mathrm{Pr}^{0.33}\left(\frac{\mu}{\mu_w}\right)^{0.14}$$

Solid to Fluid Conductor (5/8)

- Petukhov (Option 4)
  - May be more accurate (10% vs. 25%) than Dittus-Boelter or Sieder-Tate
  - Valid range:
    - $0.5 \leq Pr \leq 2{,}000$
    - $10{,}000 \leq Re \leq 5{,}000{,}000$

$$Nu = \frac{\left(\frac{f}{8}\right) RePr}{1.07 + 12.7\left(\frac{f}{8}\right)^{0.5} \left(Pr^{2/3} - 1\right)}$$

Solid to Fluid Conductor (6/8)

- Gnielinski (Option 5)
  - Useful for smaller Reynolds numbers
  - Valid range:
    - 0.5 ≤ Pr ≤ 2,000
    - 3,000 ≤ Re ≤ 5,000,000

$$Nu = \frac{\left(\frac{f}{8}\right)(Re - 1000)Pr}{1 + 12.7\left(\frac{f}{8}\right)^{0.5}\left(Pr^{2/3} - 1\right)}$$

Solid to Fluid Conductor (7/8)

- Empirical Natural Convection (Option 6)
  - Requires user to enter characteristic length, L
  - Properties evaluated at film temperature: $T_{film} = 0.5(T_w+T_f)$
  - In a mixture model, properties from fluid with greatest mass fraction node are used.
  - Valid range:
    - $10^4 \leq Ra \leq 10^{13}$

$$\mathrm{Ra} = \frac{g\beta\left|\left(T_w - T_f\right)\right|L^3\rho^2 c_p}{\mu k}$$

$$Nu = cRa^n$$

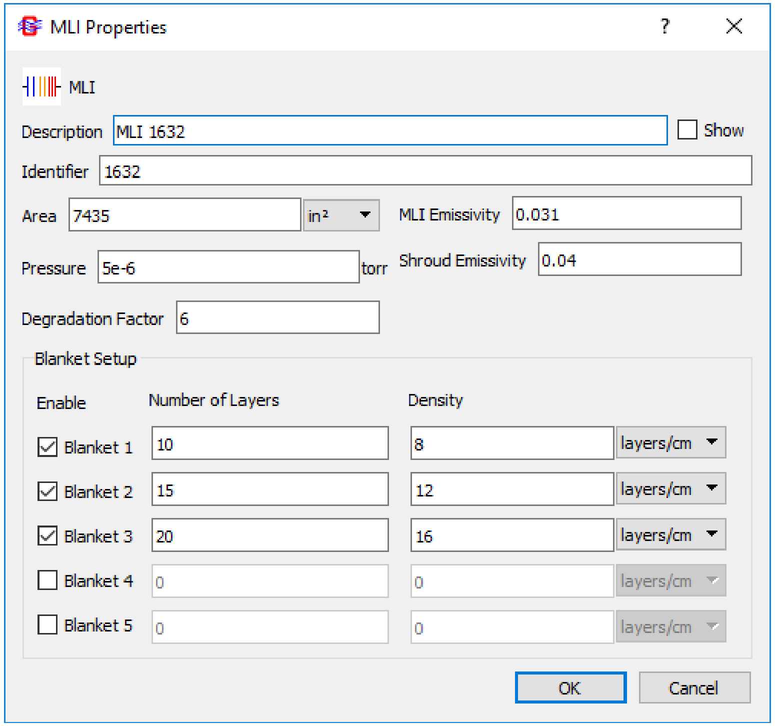
Region	c	n
Laminar, $Ra < 10^9$	0.59	0.25
Turbulent, $Ra > 10^9$	0.13	0.33

Solid to Fluid Conductor (8/8)

- Churchill-Chu (Option 7)
  - Requires user to enter characteristic length
  - Properties evaluated at film temperature
  - In a mixture model, properties from fluid with greatest mass fraction node are used.

$$Nu = \left\{ 0.825 + \frac{0.387 Ra^{1/6}}{\left[1 + \left(\frac{0.492}{Pr} \right)^{9/16} \right]^{8/27}} \right\}^2$$

MLI Conductor



MLI Properties

MLI

Description: MLI 1632 ☐ Show

Identifier: 1632

Area: 7435 in²

MLI Emissivity: 0.031

Pressure: 5e-6 torr

Shroud Emissivity: 0.04

Degradation Factor: 6

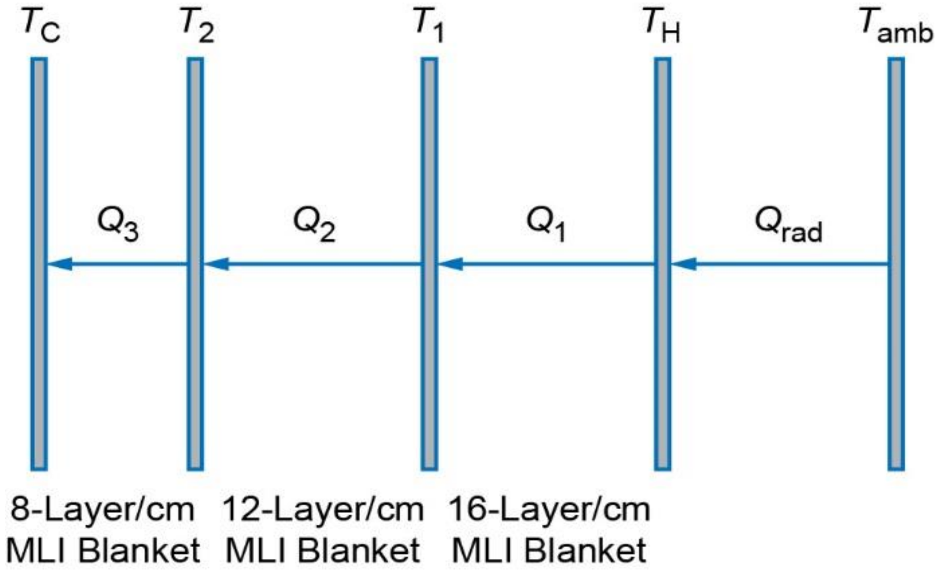
Blanket Setup

Enable	Number of Layers	Density	
☑ Blanket 1	10	8	layers/cm
☑ Blanket 2	15	12	layers/cm
☑ Blanket 3	20	16	layers/cm
☐ Blanket 4	0	0	layers/cm
☐ Blanket 5	0	0	layers/cm

Inside blanket first

Outside blanket last

MLI Modeling Methodology



Law of energy conservation

$$Q_{rad} = Q_1 = Q_2 = Q_3$$

$$Q_2(T_1, T_2) - Q_3(T_2, T_c) = 0$$

$$Q_1(T_H, T_1) - Q_2(T_1, T_2) = 0$$

$$Q_{rad}(T_{amb}, T_H) - Q_1(T_H, T_1) = 0$$

MLI Heat Transfer (1/2)

- Heat transfer through the MLI calculated by the Modified Lockheed equation

$$q = \left[\frac{C_s\left(0.017 + 7E - 6 * \left(800 - T_{avg}\right) + 2.28E - 2\ * ln\left(T_{avg}\right)\right)(N^*)^{2.63}(T_h - T_c)}{N_s} + \frac{C_r\varepsilon\left(T_h^{4.67} - T_c^{4.67}\right)}{N_s} + \frac{C_g P\left(T_h^{0.52} - T_c^{0.52}\right)}{N_s}\right]$$

q = heat flux through MLI (W/m^2)
T_{avg} = average of hot and cold boundary temperatures (K)
N^* = MLI layer density (layers/cm)
T_h = hot boundary temperature (K)
T_c = cold boundary temperature (K)
N_s = number of MLI layers
ε = MLI layer emissivity ($\varepsilon = 0.031$)
P = interstitial gas pressure (torr)

$C_s = 2.4 \times 10^{-4}$
$C_r = 4.944 \times 10^{-10}$
$C_g = 14600$

MLI Heat Transfer (2/2)

- Radiative heat transfer from the shroud to the outer layer of MLI

$$q_{rad} = \frac{\sigma\left(T_{amb}^4 - T_{outer}^4\right)}{\frac{1}{\varepsilon_{MLI}} + \frac{1}{\varepsilon_{shrd}} - 1}$$

  - Expression assumes radiation between closely spaced parallel planes
  - For other situations, user may wish to modify input shroud emissivity, for example:

- Concentric cylinders (where r_{outer} and r_{shrd} are not similar):

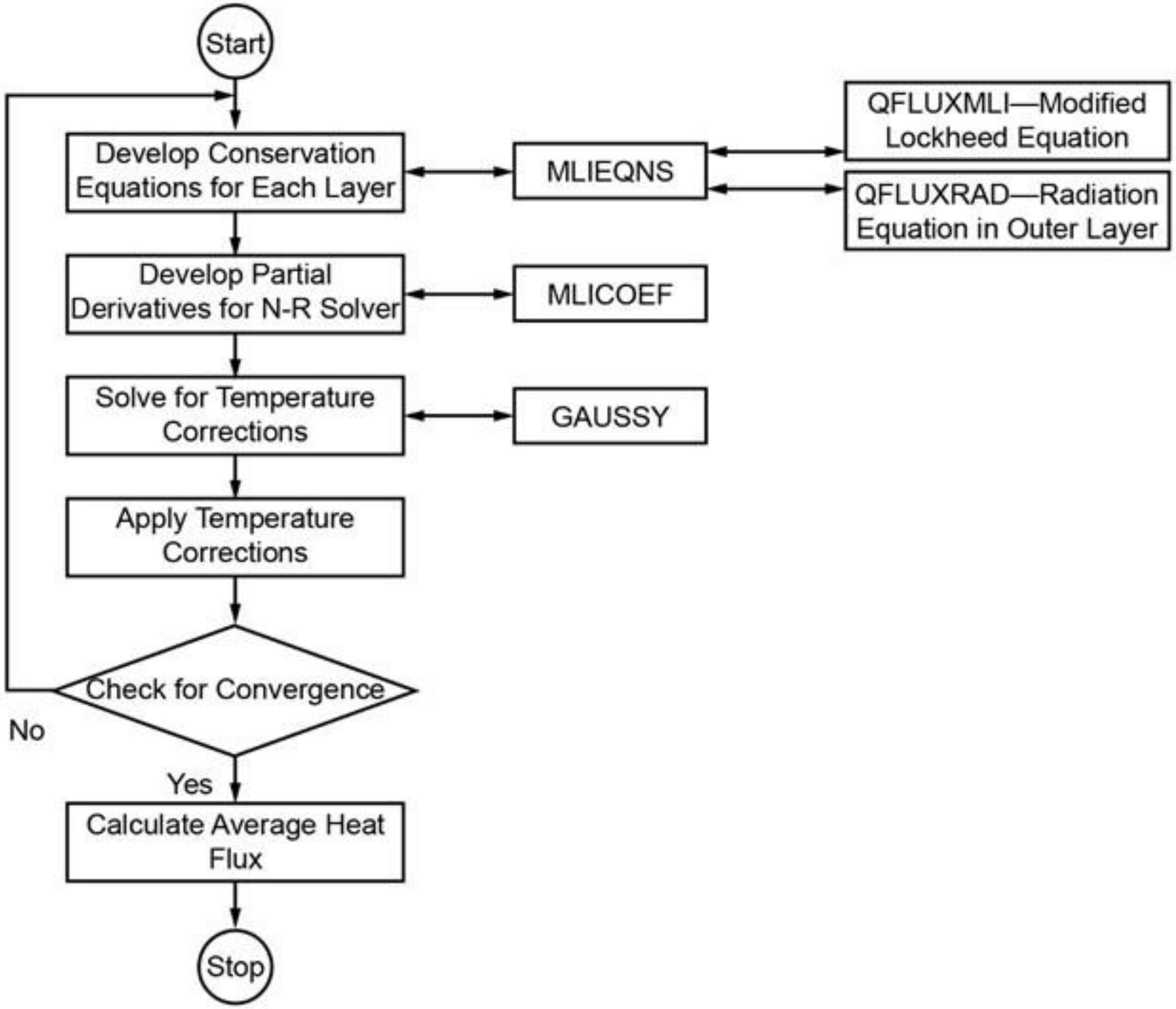
$$q_{rad} = \frac{\sigma\left(T_{amb}^4 - T_{outer}^4\right)}{\frac{1}{\varepsilon_{MLI}} + \frac{1-\varepsilon_{shrd}}{\varepsilon_{shrd}}\left(\frac{r_{outer}}{r_{shrd}}\right)} \quad \Longrightarrow \quad \varepsilon_{shrd-mod} = \left[1 + \frac{1-\varepsilon_{shrd}}{\varepsilon_{shrd}}\left(\frac{r_{outer}}{r_{shrd}}\right)\right]^{-1}$$

- Small object in large cavity (set $\varepsilon_{shrd} = 1$)

$$q_{rad} = \sigma\varepsilon_{MLI}\left(T_{amb}^4 - T_{outer}^4\right)$$

MLI Modeling Methodology

- Flowchart of MLI_HEAT_RATE Subroutine



Start

Develop Conservation Equations for Each Layer

Develop Partial Derivatives for N-R Solver

Solve for Temperature Corrections

Apply Temperature Corrections

Check for Convergence

No

Yes

Calculate Average Heat Flux

Stop

MLIEQNS

MLICOEF

GAUSSY

QFLUXMLI—Modified Lockheed Equation

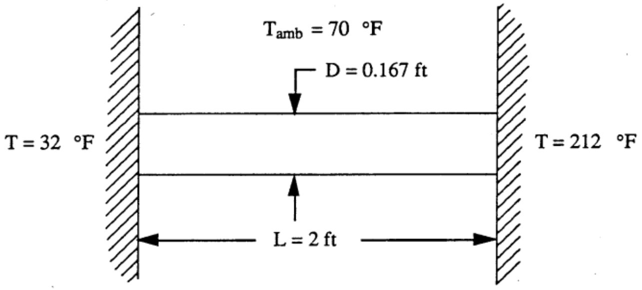
QFLUXRAD—Radiation Equation in Outer Layer

Applications

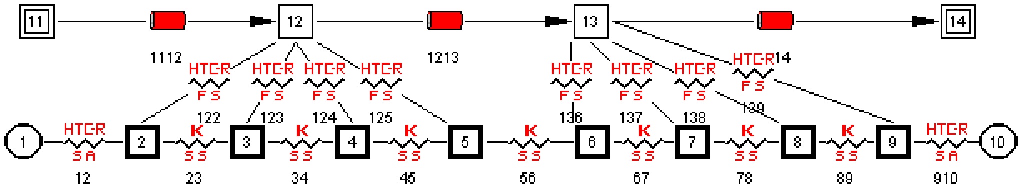
- Textbook Problem (Ex13)
- Cryogenic Transfer Line (Ex14)
- Propellant Loading
- Pressurization of Space Shuttle’s LH2 Tank
- Heat Leak through MLI to a Cryogenic Tank (Ex29)

Ex13: Verification of Conjugate Heat Transfer Results

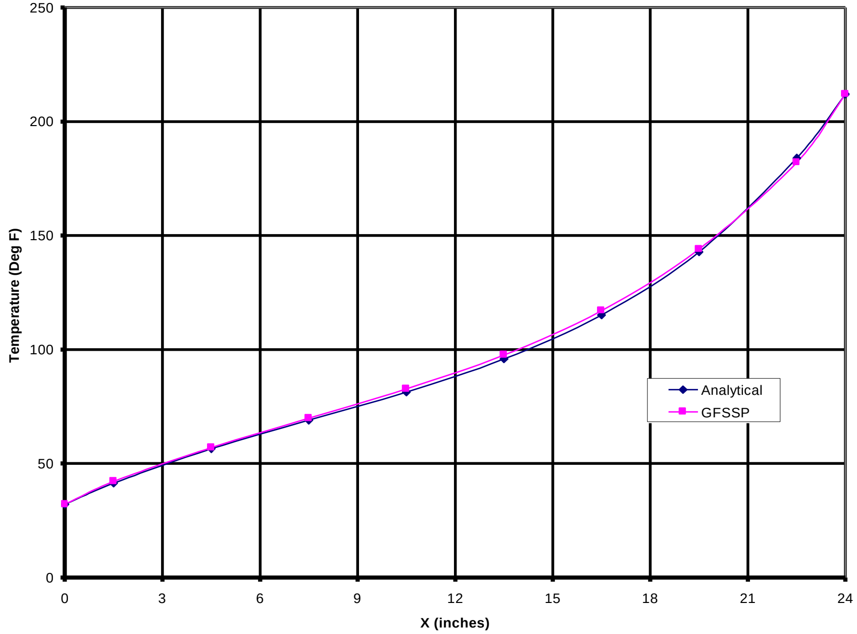
Problem Considered



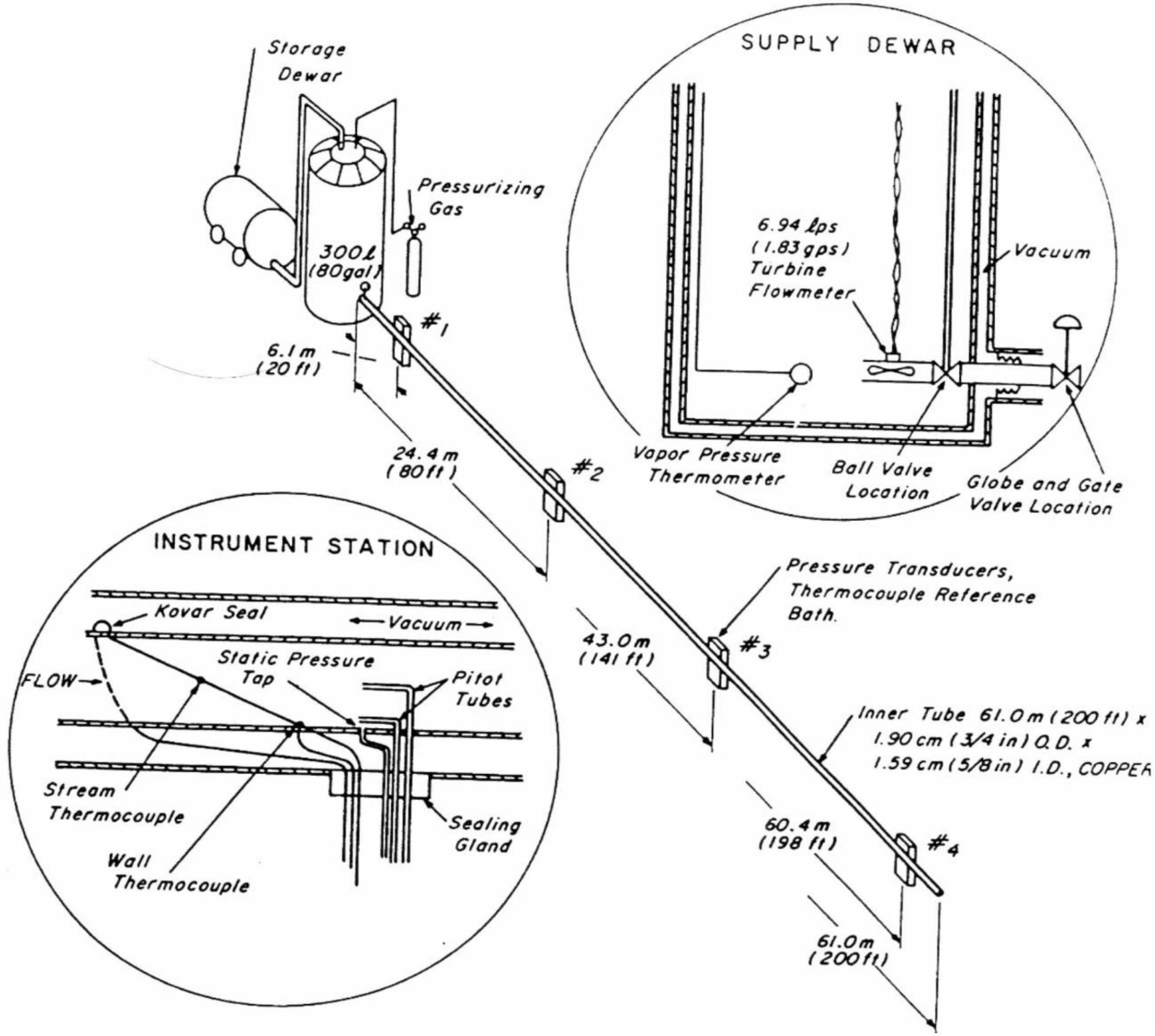
GFSSP Model



Comparison with Analytical Solution

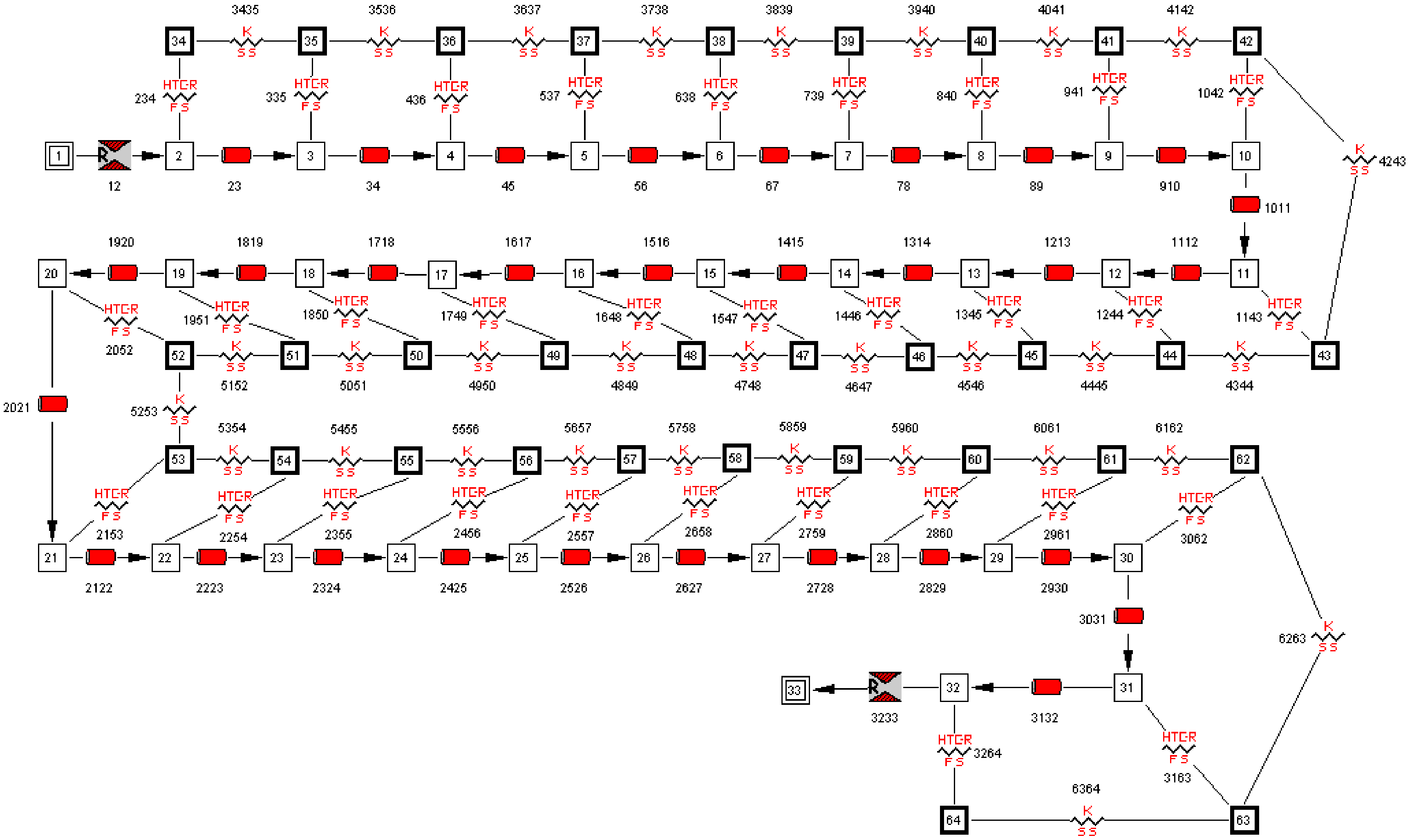


NBS Test Set-up of Cryogenic Transfer Line



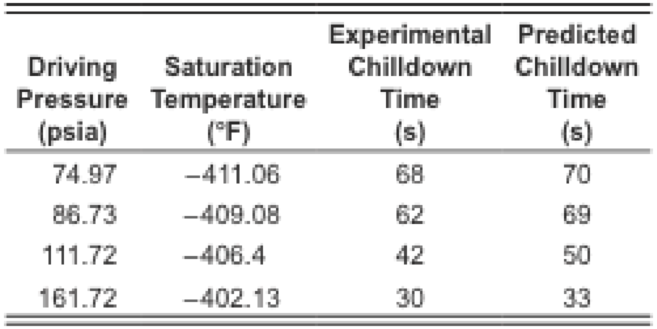
Ex14: GFSSP Model of Cryogenic Transfer Line

Marshall Space Flight Center
GFSSP Training Course



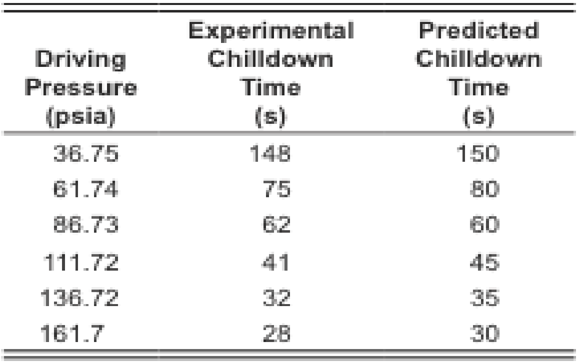
Ex14: Comparison with Test Data

Saturated LH_2 chilldown time for various driving pressures



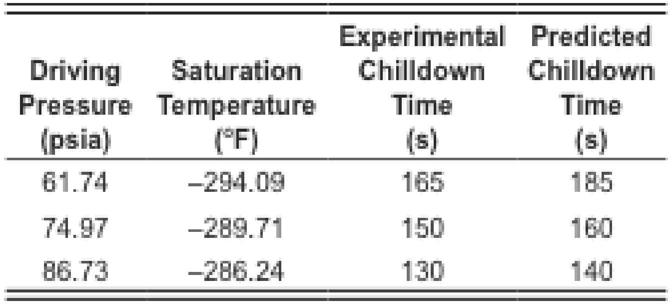
Driving Pressure (psia)	Saturation Temperature (°F)	Experimental Chilldown Time (s)	Predicted Chilldown Time (s)
74.97	–411.06	68	70
86.73	–409.08	62	69
111.72	–406.4	42	50
161.72	–402.13	30	33

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



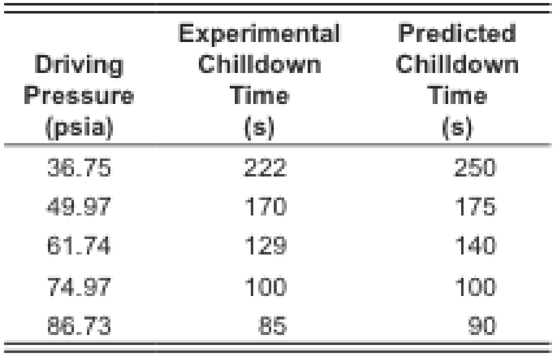
Driving Pressure (psia)	Experimental Chilldown Time (s)	Predicted Chilldown Time (s)
36.75	148	150
61.74	75	80
86.73	62	60
111.72	41	45
136.72	32	35
161.7	28	30

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

Ex14: Comparison of Temperature Histories

- Subcooled LH_2 for various driving pressures

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

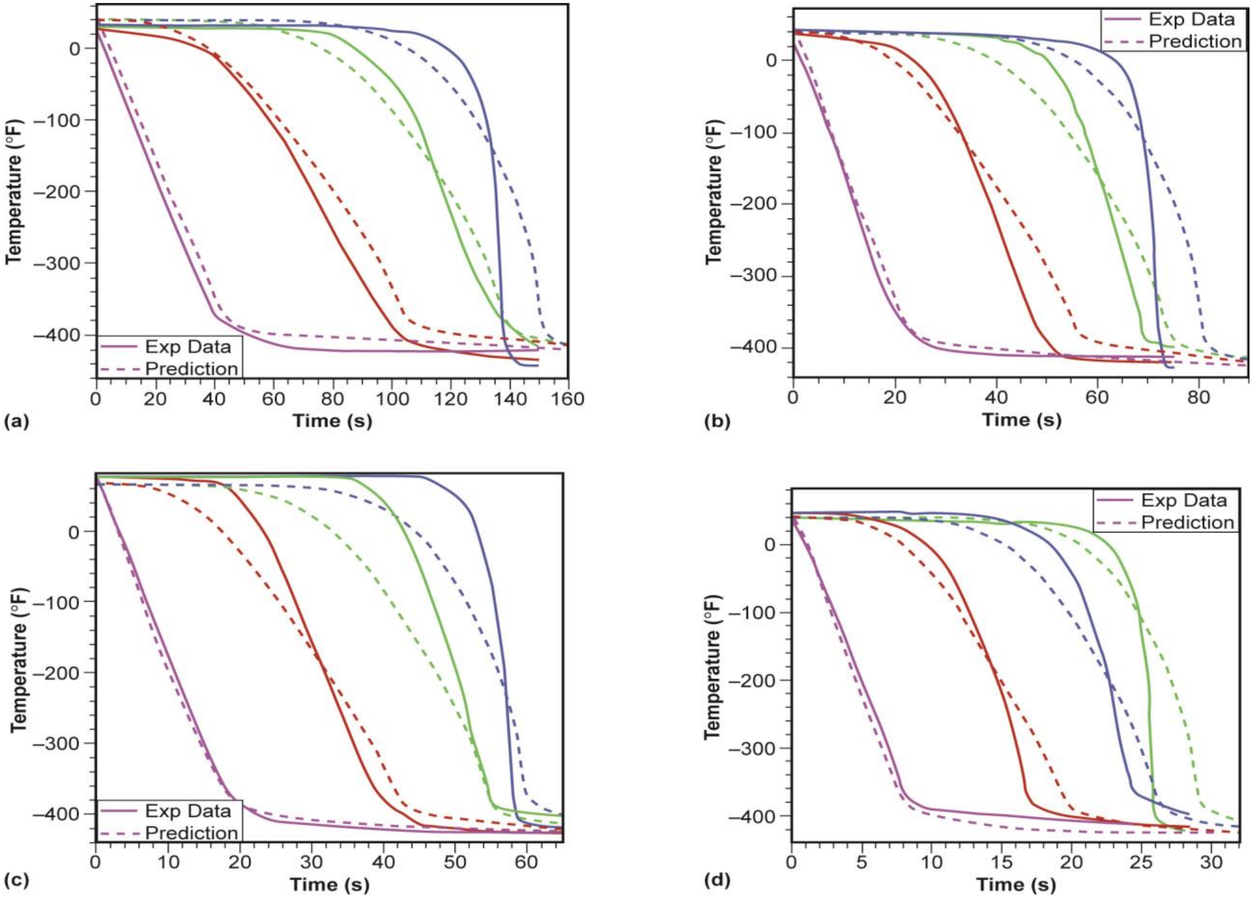
Station #2 (red)
—80 ft from tank inlet

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

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

p = 36.74 psia

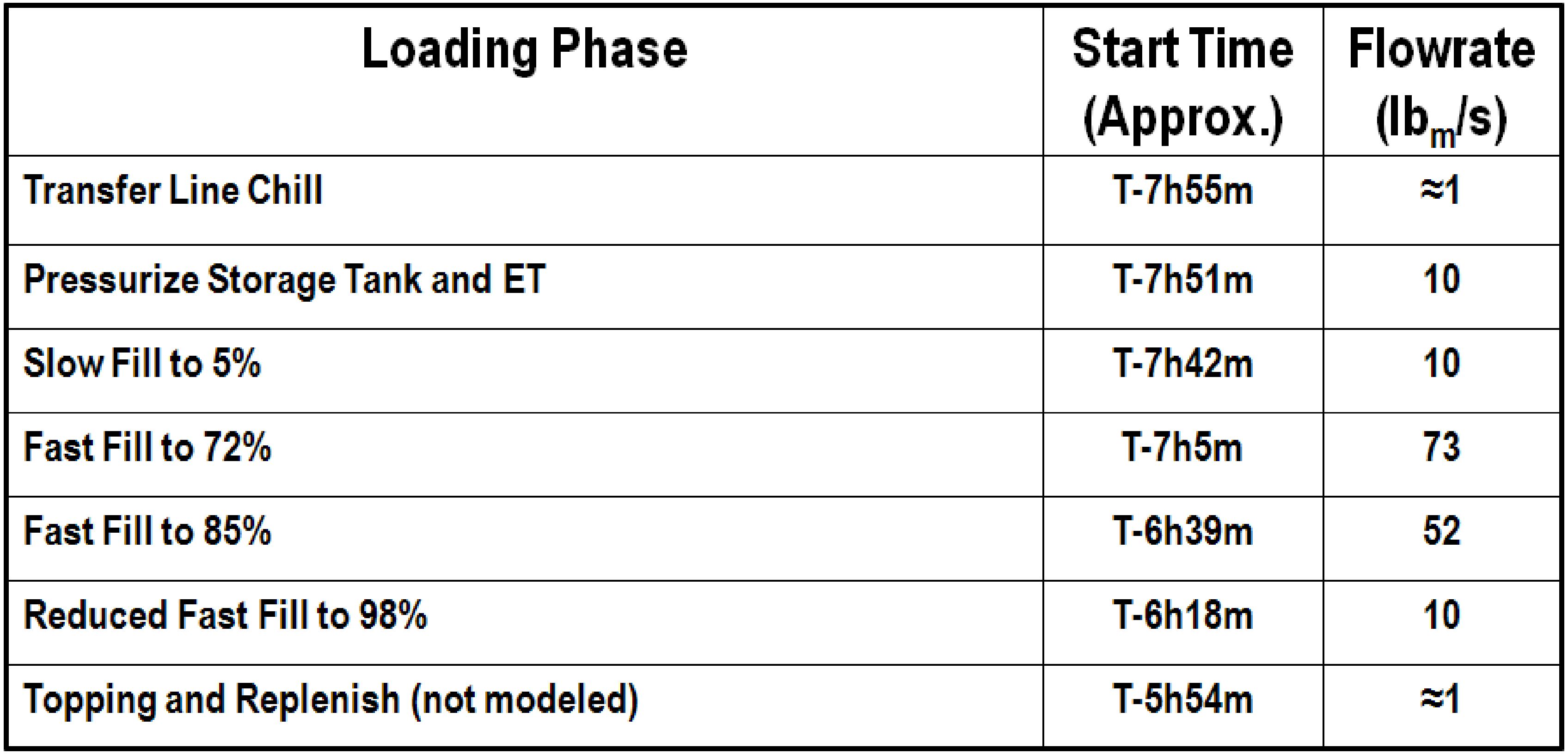
p = 61.72 psia



p = 86.7 psia

p = 161 psia

Propellant Loading - Shuttle ET LH$_2$ (1/6)



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

Propellant Loading - Shuttle ET LH_2 (2/6)

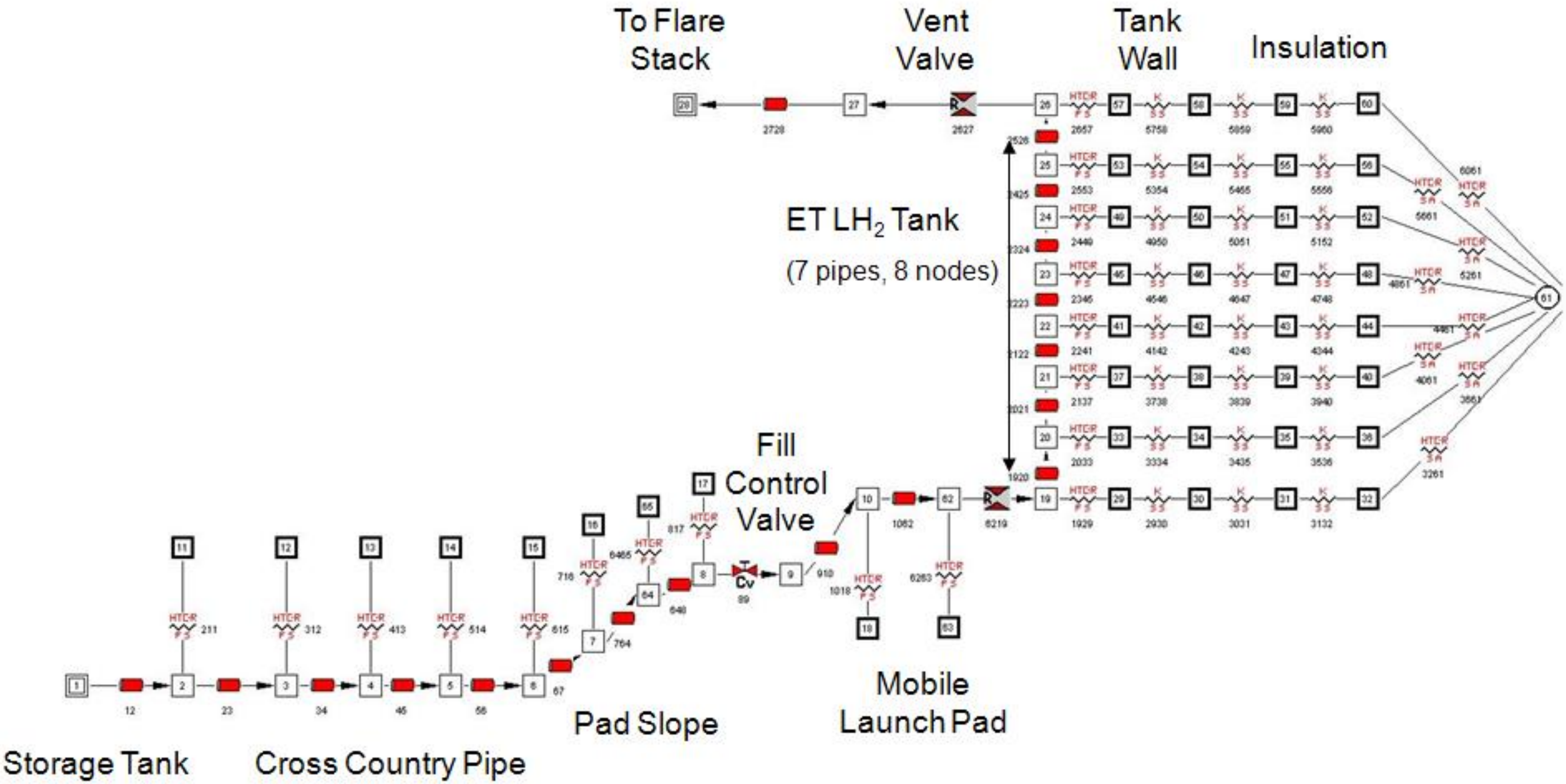
- KSC LH_2 Facility Properties
  - Cross-country Pipeline
    - ¼ mile of 10” Invar pipe, vacuum-jacketed
    - 26400 lb_m
    - Dz = 79 ft
  - Mobile Launch Platform
    - 334 ft of 8” and 10” stainless steel pipe, vacuum-jacketed
    - 6100 lb_m
    - Dz = 43 ft

Propellant Loading - Shuttle ET LH_2 (3/6)

- ET LH_2 Tank Properties
  - Tank Mass: 23600 lb_m
  - LH_2 mass: 227600 lb_m
  - Length: 97 ft
  - Diameter: 27.6 ft
  - Insulation: 2078 lb_m
    - ~1.0” NCFI on barrel and aft dome
    - ~0.75” BX-265 on forward dome
  - Surface area: 8550 ft^2
  - Vent: $C_dA = f(DP) \sim 18\ in^2$
    - Open during facility line chilldown
    - Cycles open and closed during slow/fast fill to maintain 24-27 psig

Propellant Loading - Shuttle ET LH_2 (4/6)

- ET LH_2 **GFSSP** Model



Propellant Loading - Shuttle ET LH_2 (5/6)

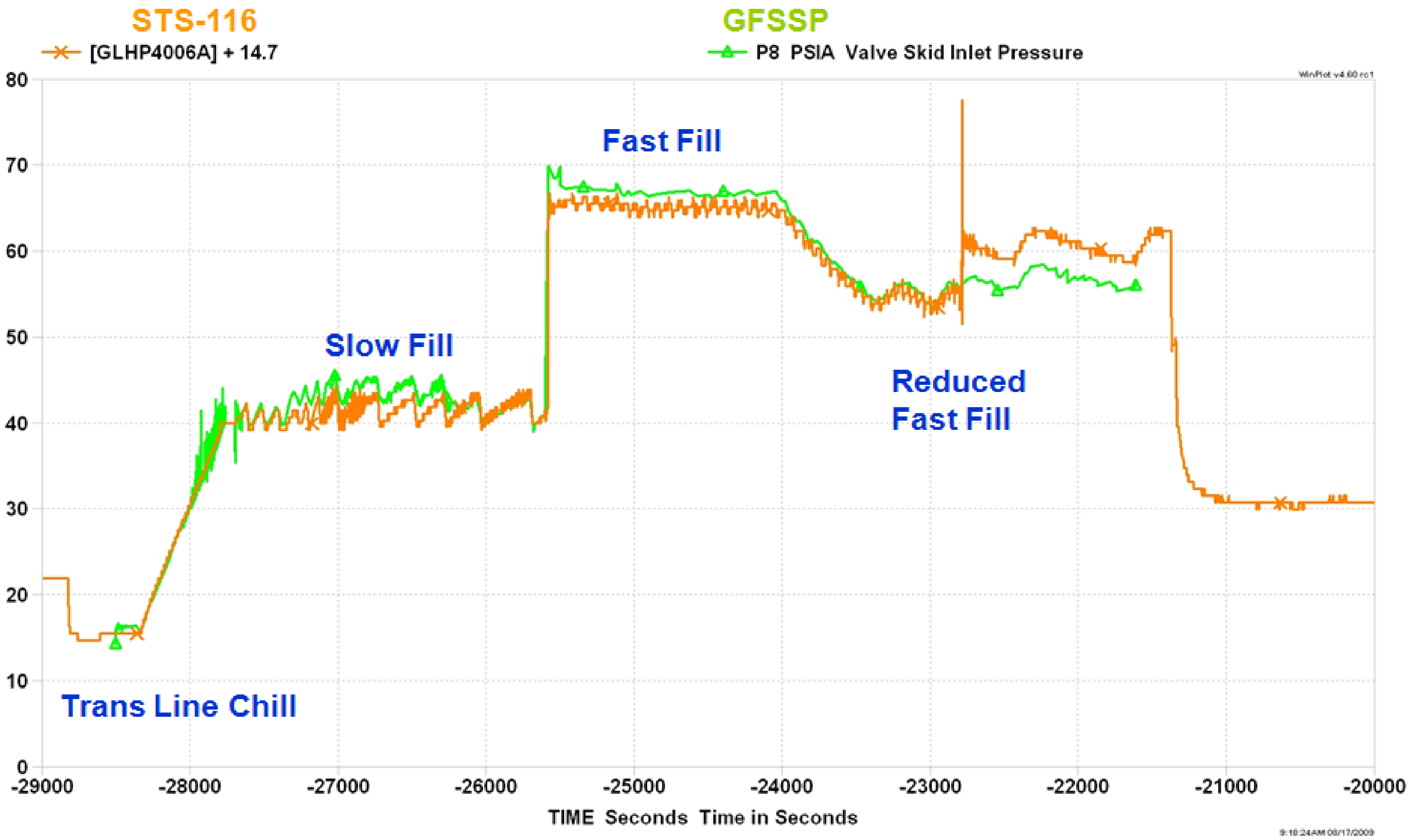
- Comparison

Condition	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

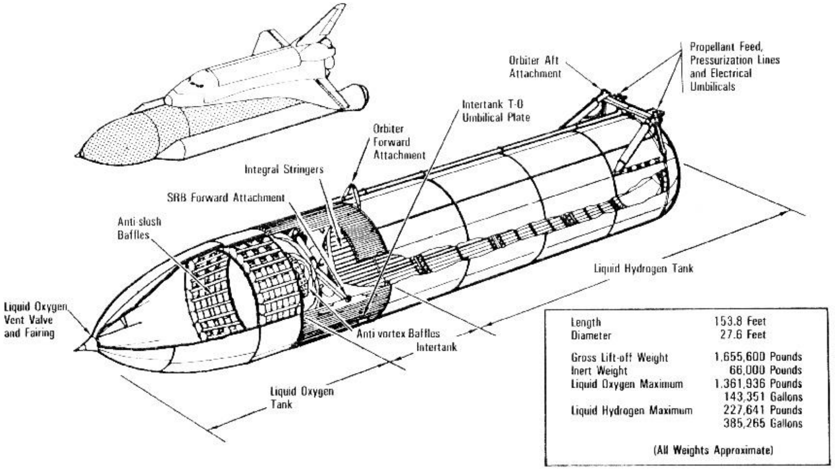
* Not measured – estimate from ET System Definition Handbook

Propellant Loading - Shuttle ET LH_2 (6/6)

- Pressure at Valve Skid



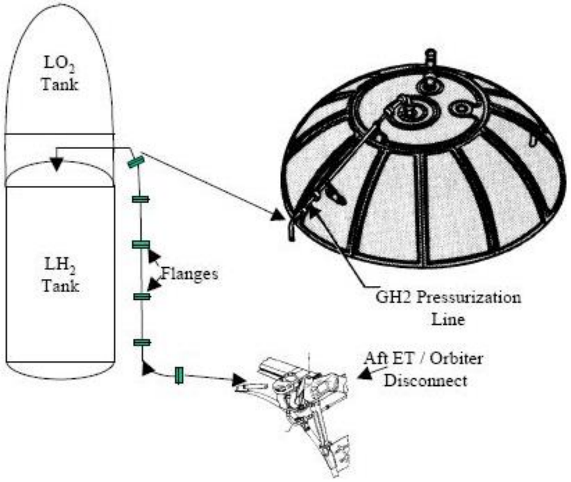
Pressurization of Space Shuttle's LH_2 Tank (1/4)

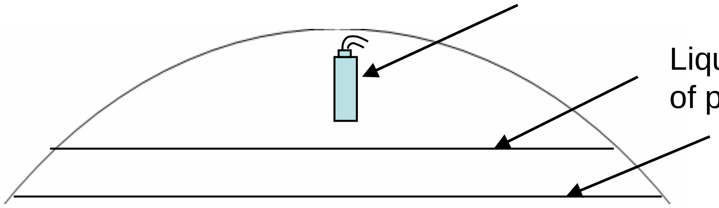


Length	153.8 Feet
Diameter	27.6 Feet
Gross Lift-off Weight	1,655,600 Pounds
Inert Weight	66,000 Pounds
Liquid Oxygen Maximum	1,361,936 Pounds 143,351 Gallons
Liquid Hydrogen Maximum	227,641 Pounds 385,265 Gallons

(All Weights Approximate)

Lightweight External Tank





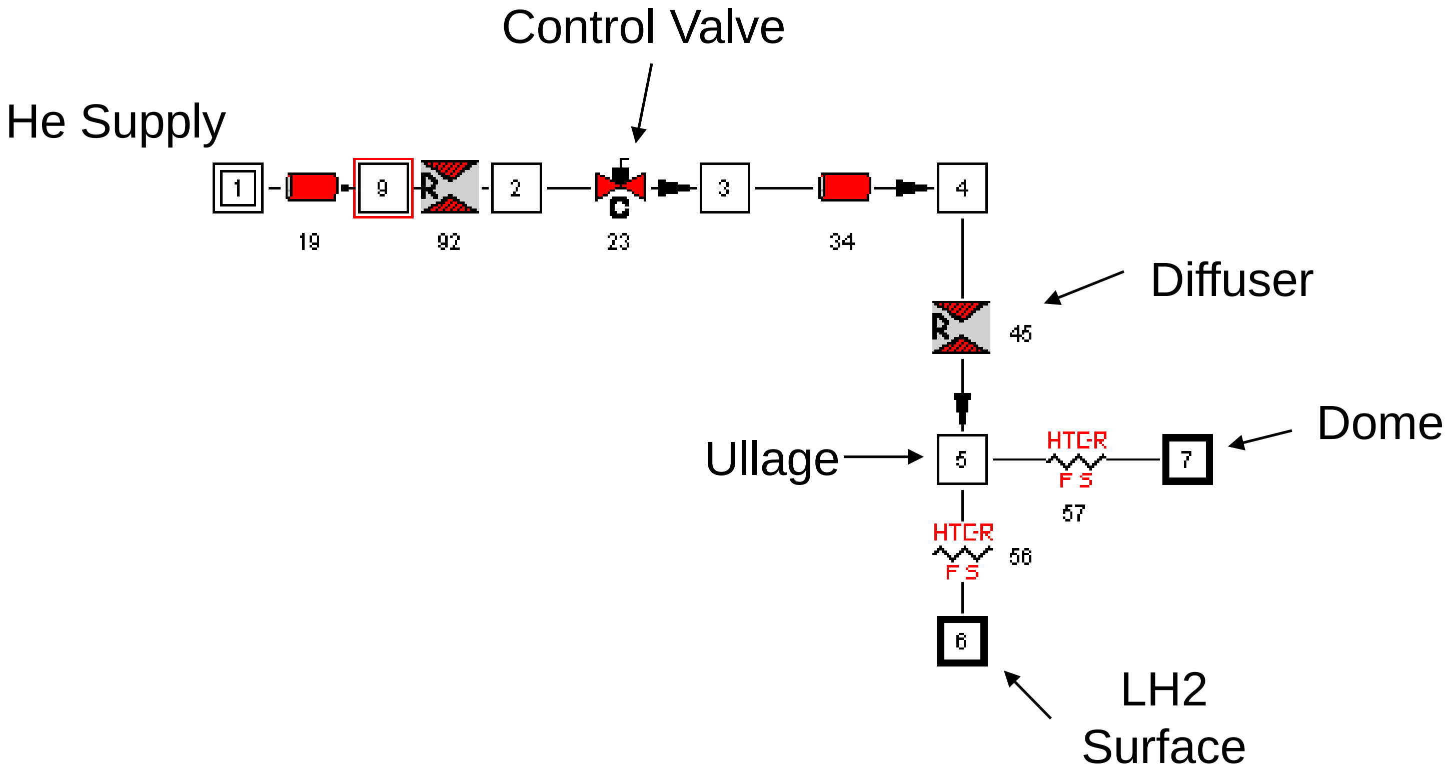
Diffuser

Liquid level at start of prepress

Liquid level at end of prepress

Pressurization of Space Shuttle's LH_2 Tank (2/4)

- **GFSSP** Model


Control Valve

He Supply

Diffuser

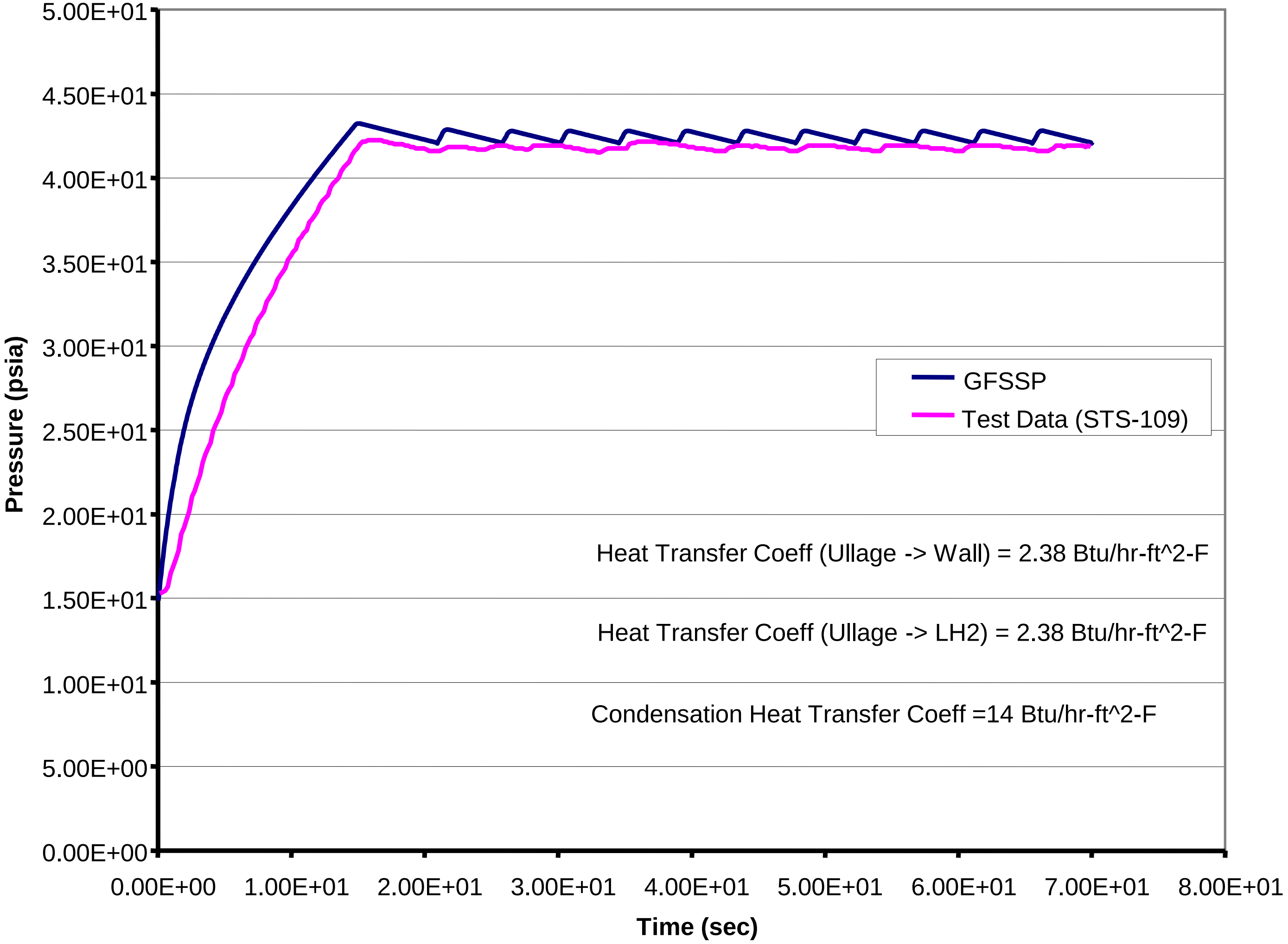
Dome

Ullage

LH2 Surface

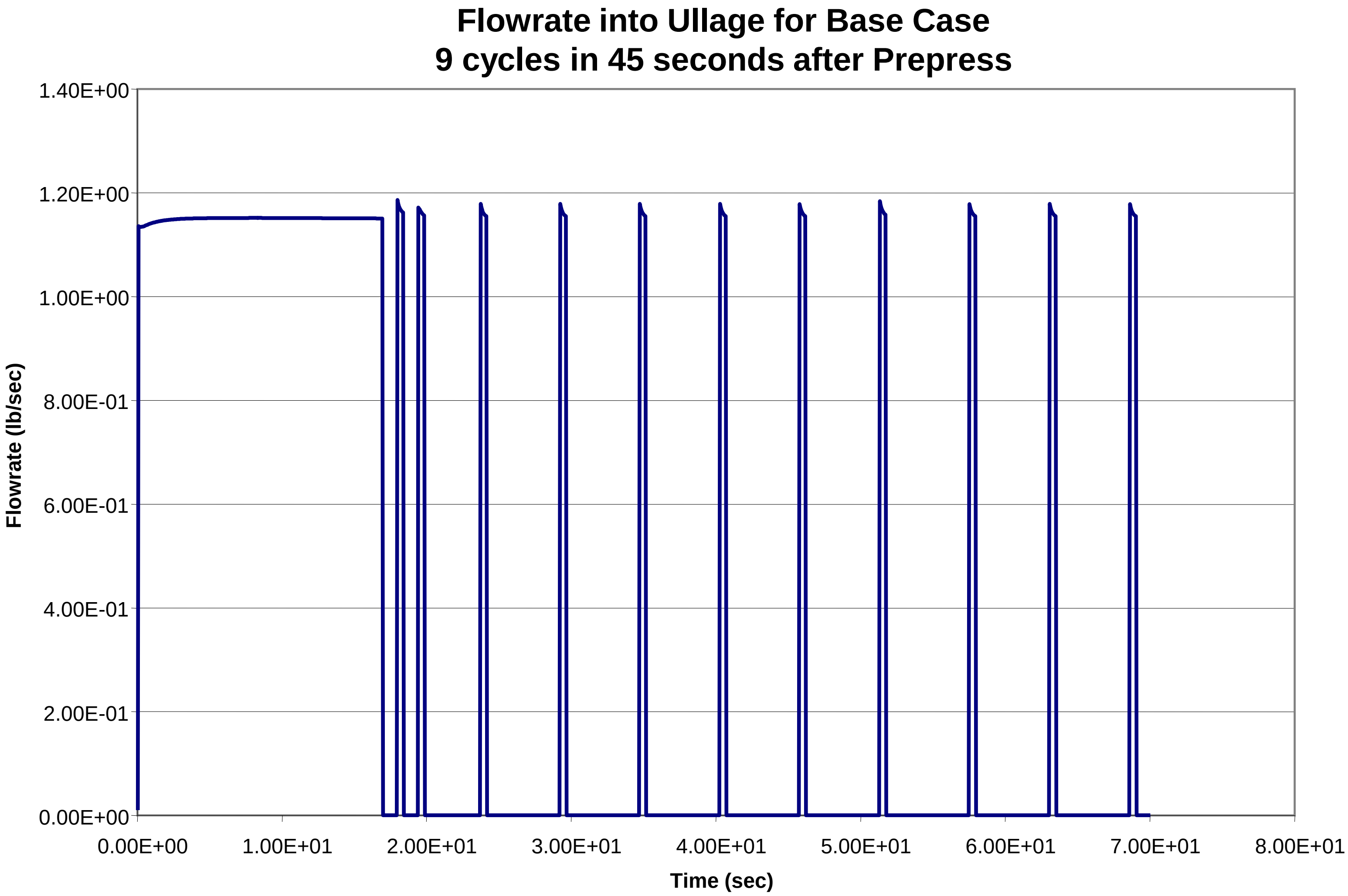
Pressurization of Space Shuttle's LH_2 Tank (3/4)

- Ullage Pressure History in LH2 Tank (STS-109)

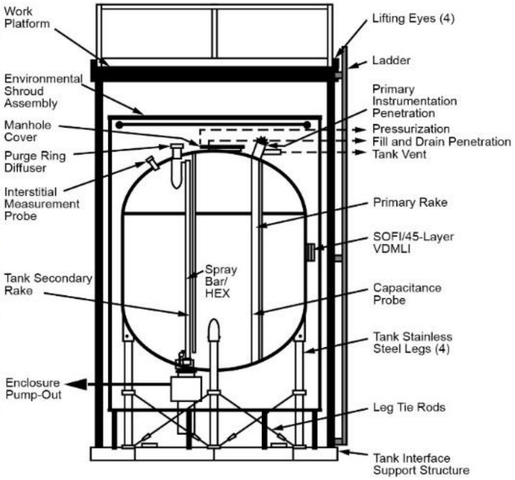


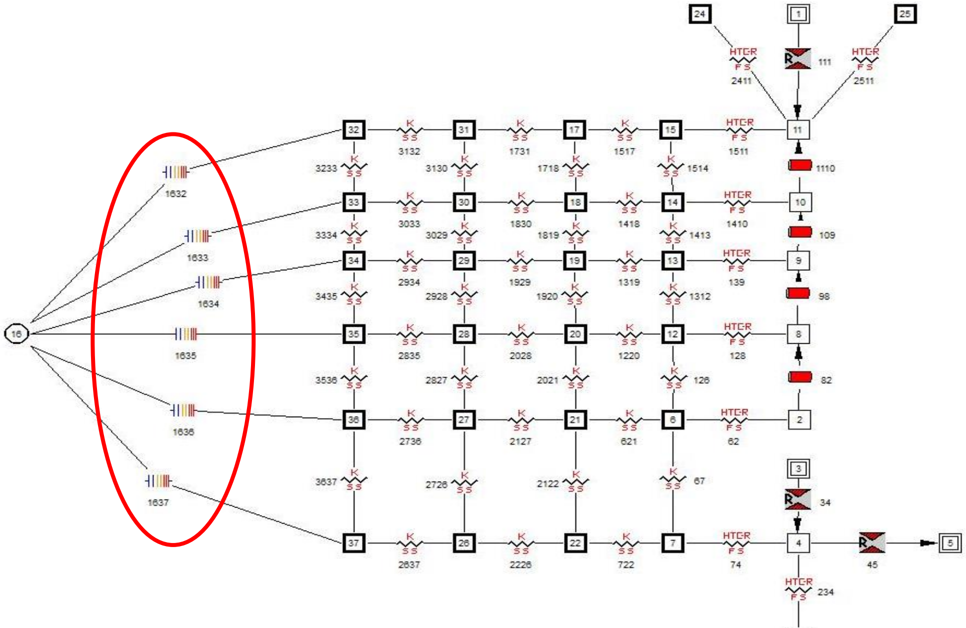
Pressurization of Space Shuttle's LH_2 Tank (4/4)

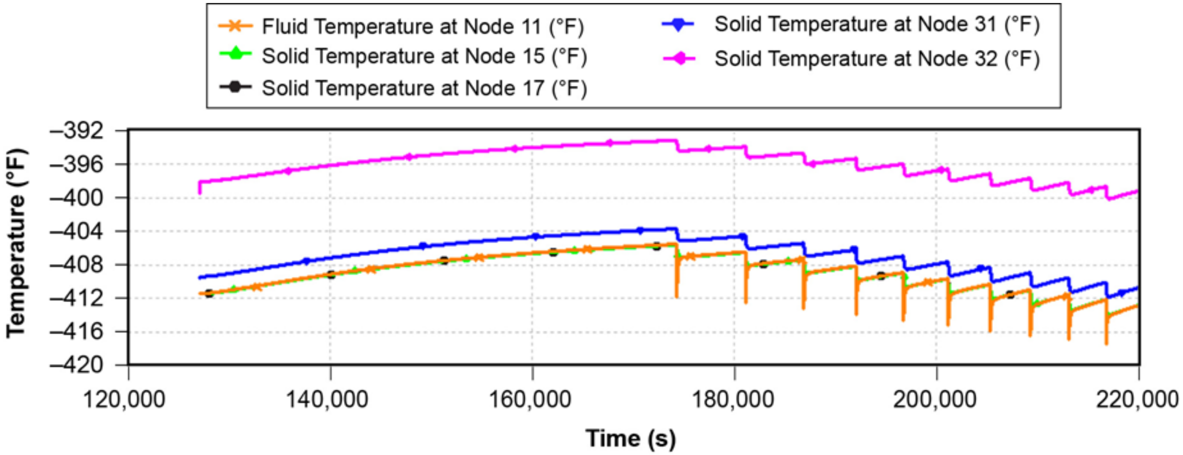
- Helium flowrate history



Ex29: Application of MLI Conductor in Modeling Cryogenic Tank (1/2)

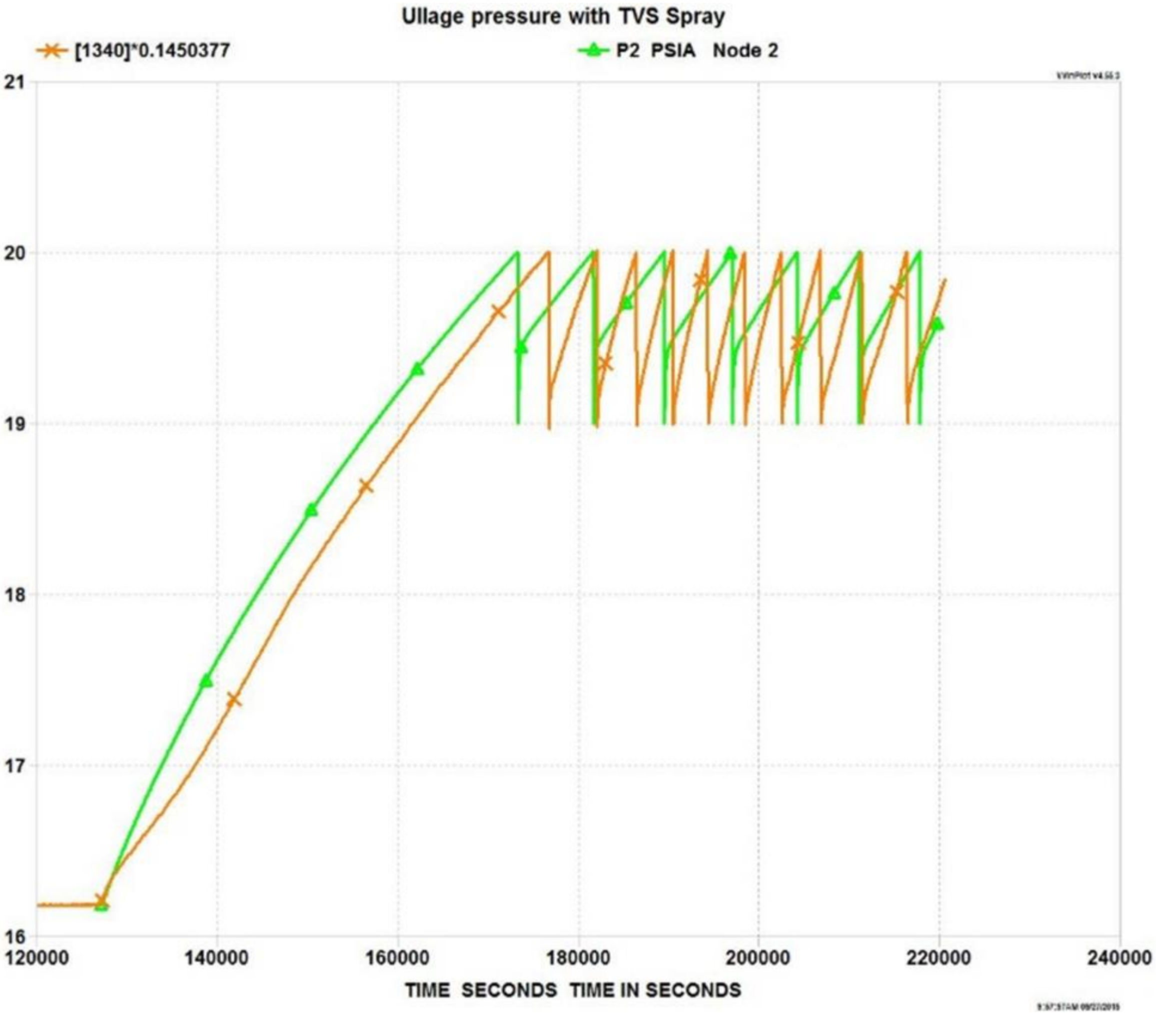






Ex29: Application of MLI Conductor in Modeling Cryogenic Tank (2/2)

- Pressure history



Summary

- **GFSSP** 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 **MIG** allows user to construct, run, and view results for network consisting of fluid and solid nodes

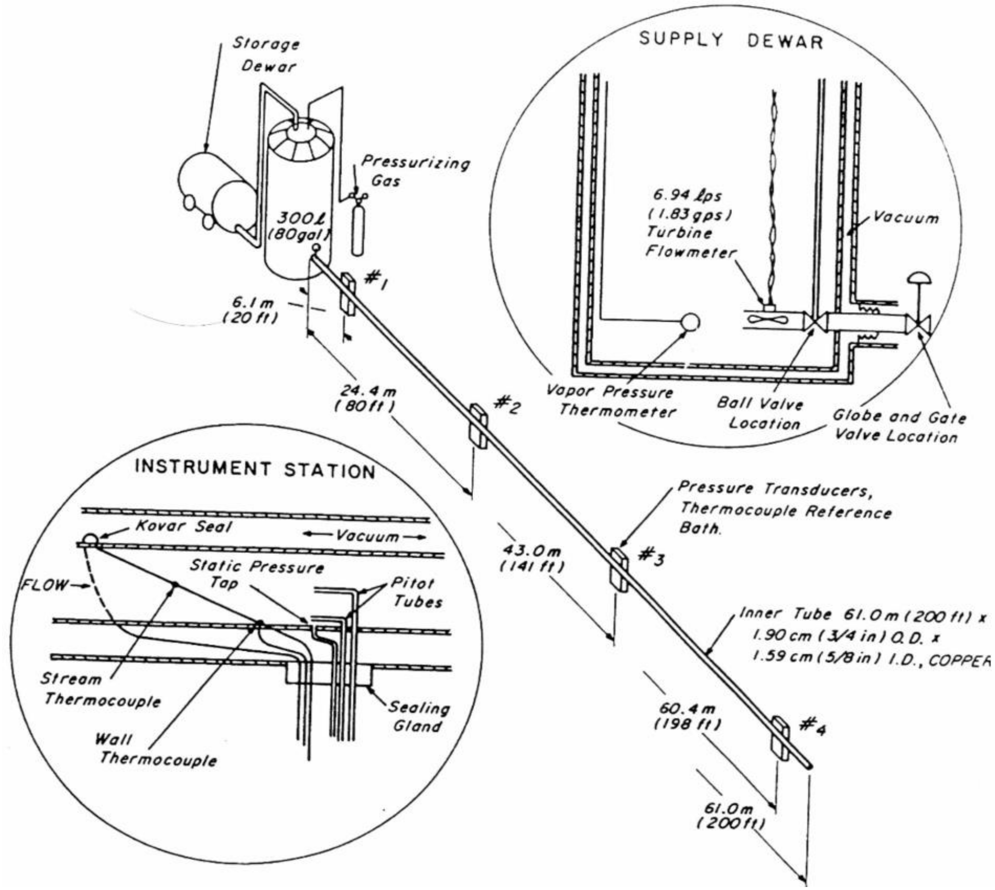
- For Heat Transfer Coefficients, simple forced convection pipe flow and natural convection vertical wall correlations are provided.
  - Other correlations can be implemented through User Subroutine

- **GFSSP**'s CHT capability has been validated by comparing with test data

- Examples 13, 14, and 29 illustrate the use of Conjugate Heat Transfer applications

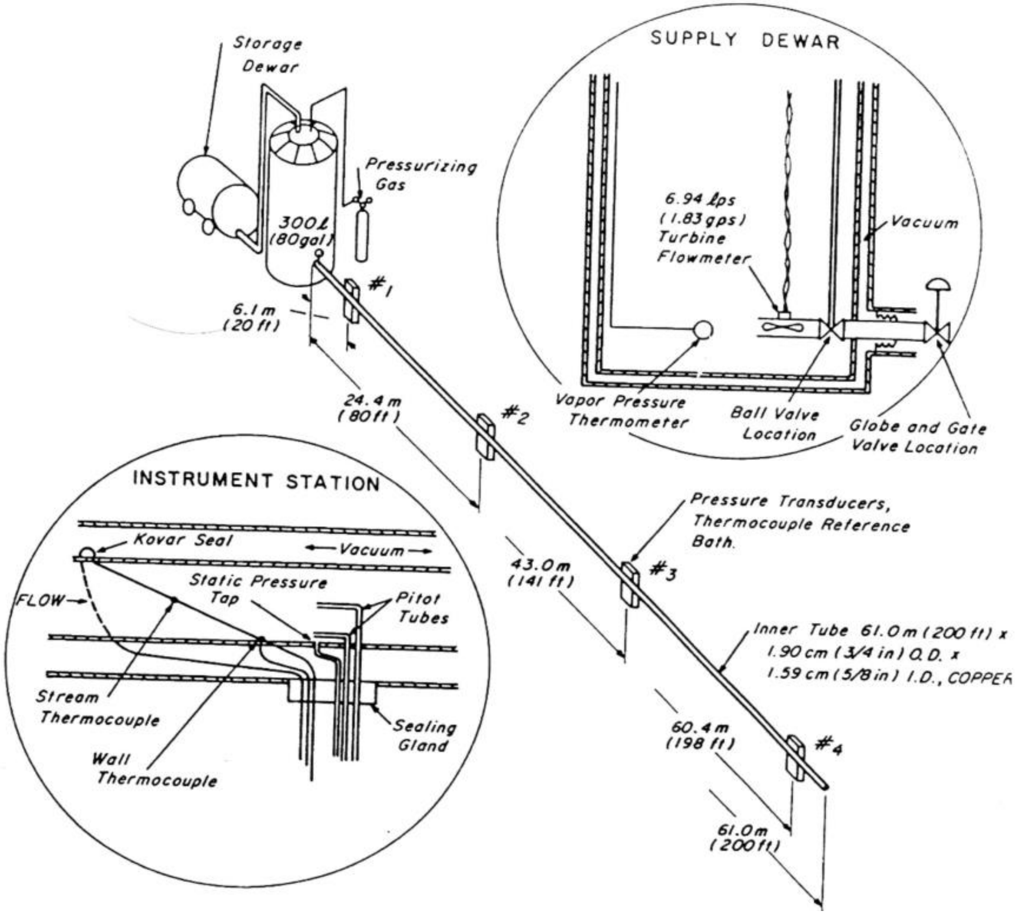
Tutorial – 5

Chilldown of Cryogenic Transfer Line



Chilldown of Transfer Line Schematic

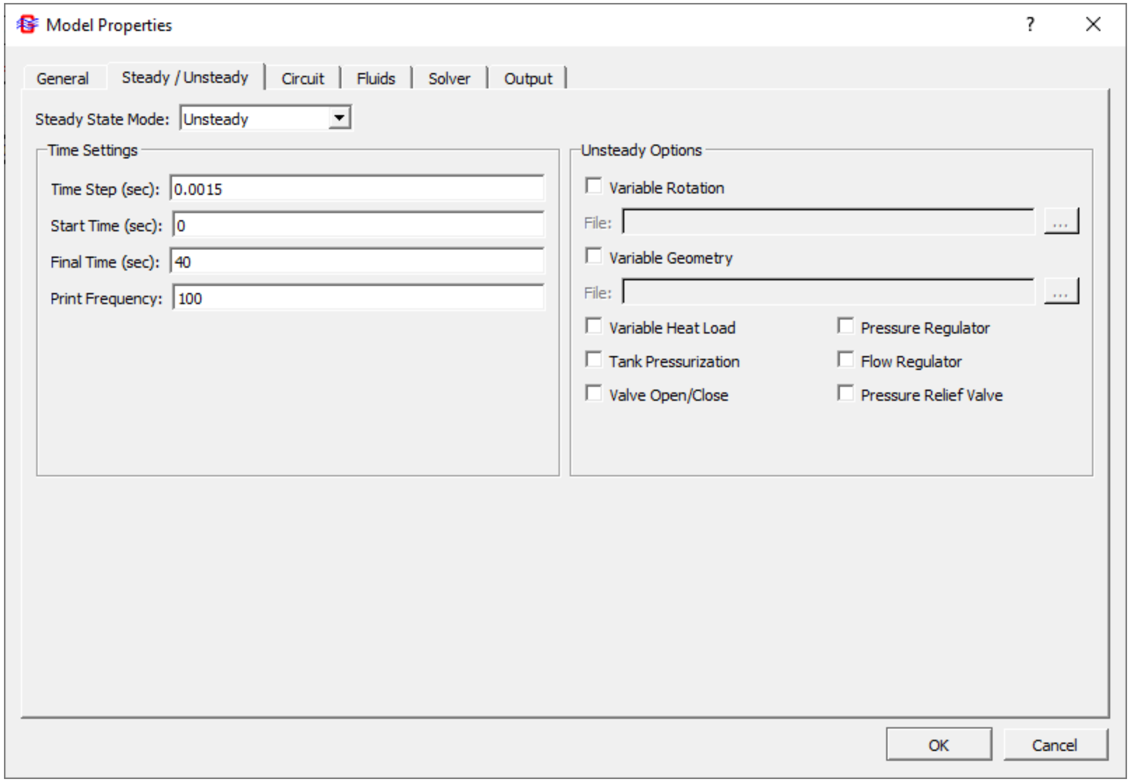
- Problem considered:
  - Time-dependent Pressure, Temperature and Flow Rate history during chilldown



Set Up Options (1/3)

- General
  - Model File: Tut5.gfssp
  - Input File: Tut5.dat
  - Output File: Tut5.out

- Unsteady Options
  - Time Step: 0.0015 s
  - Final Time: 40.0 s



Model Properties

Steady State Mode: Unsteady

Time Settings

Time Step (sec): 0.0015

Start Time (sec): 0

Final Time (sec): 40

Print Frequency: 100

Unsteady Options

☐ Variable Rotation

File:

☐ Variable Geometry

File:

☐ Variable Heat Load

☐ Pressure Regulator

☐ Tank Pressurization

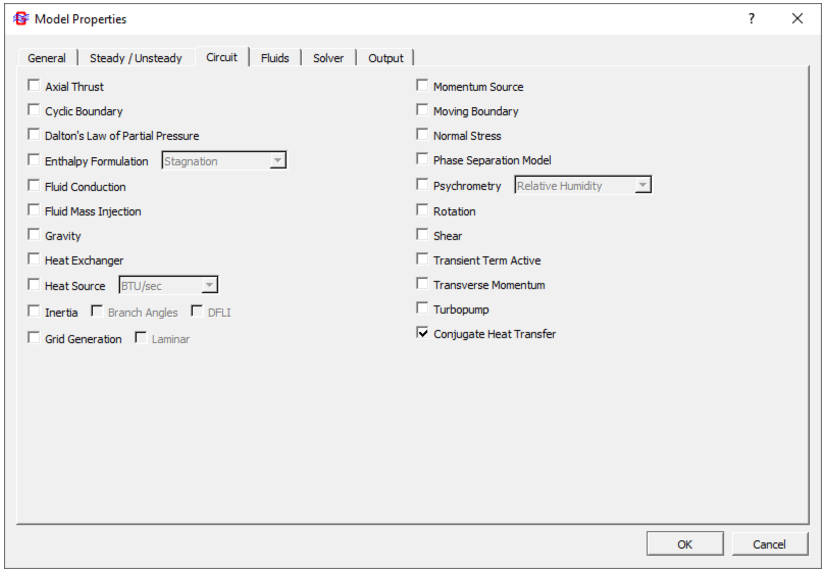
☐ Flow Regulator

☐ Valve Open/Close

☐ Pressure Relief Valve

Set Up Options (2/3)

- Circuit tab
  - Conjugate Heat Transfer

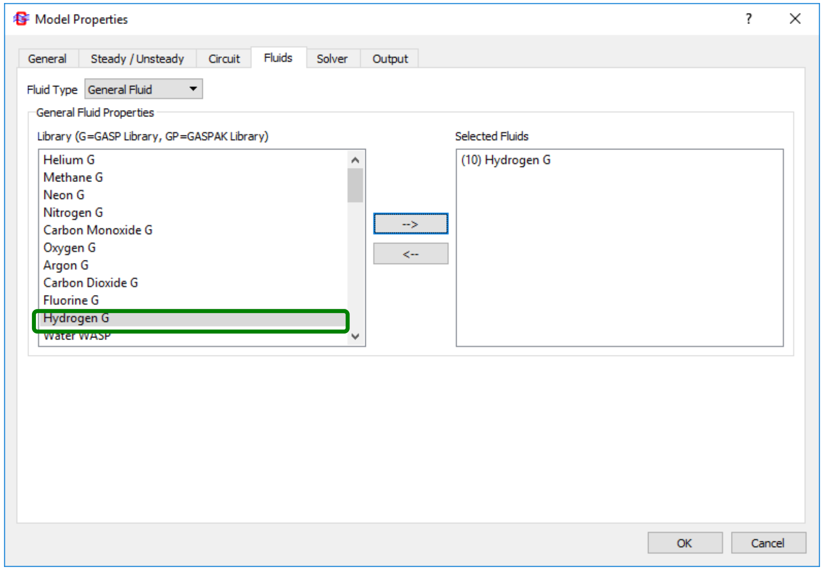


Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation: Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☐ Gravity
☐ Heat Exchanger
☐ Heat Source: BTU/sec
☐ Inertia ☐ Branch Angles ☐ DFLI
☐ Grid Generation ☐ Laminar
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry: Relative Humidity
☐ Rotation
☐ Shear
☐ Transient Term Active
☐ Transverse Momentum
☐ Turbopump
☑ Conjugate Heat Transfer

- Fluid: Hydrogen



Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

Fluid Type: General Fluid

General Fluid Properties

Library (G=GASP Library, GP=GASPAK Library)

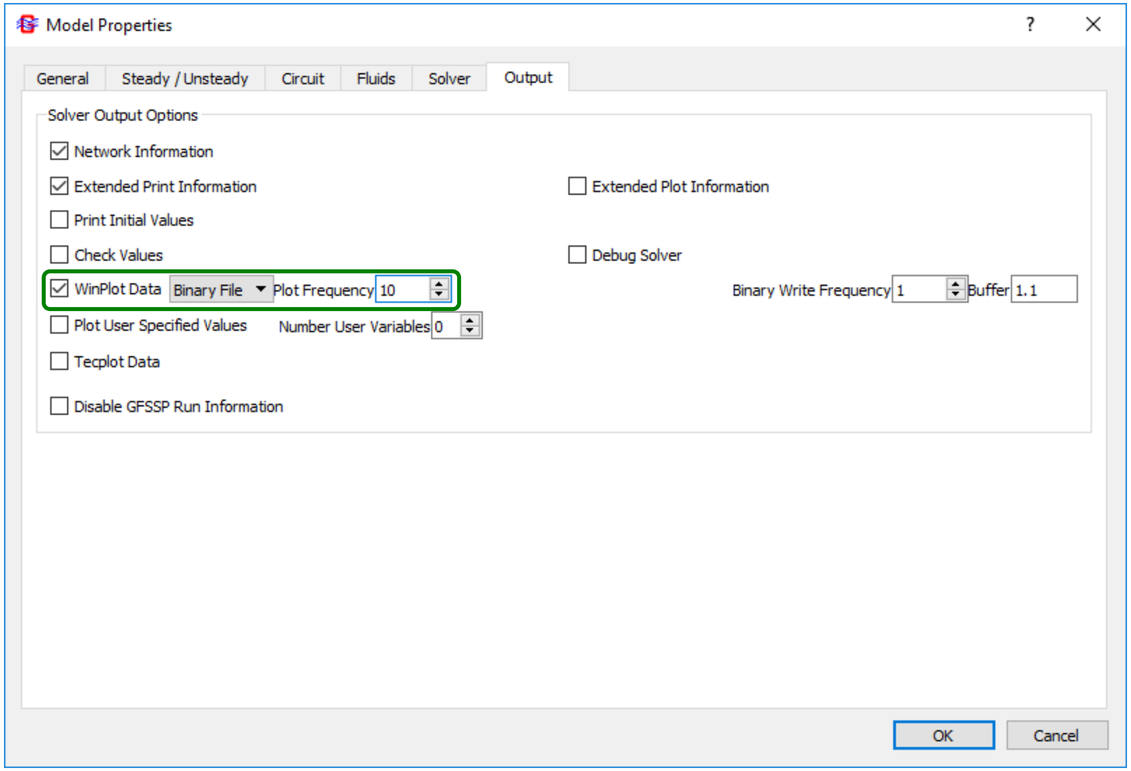
Helium G
Methane G
Neon G
Nitrogen G
Carbon Monoxide G
Oxygen G
Argon G
Carbon Dioxide G
Fluorine G
Hydrogen G
Water WASP

Selected Fluids

(10) Hydrogen G

Set Up Options (3/3)

- Output tab
  - Check: Winplot Data / Binary File
  - Set Plot Frequency to 10 (to avoid large Winplot file)



Model Properties

Solver Output Options

☑ Network Information

☑ Extended Print Information

☐ Extended Plot Information

☐ Print Initial Values

☐ Check Values

☐ Debug Solver

☑ WinPlot Data: Binary File; Plot Frequency: 10

Binary Write Frequency: 1; Buffer: 1.1

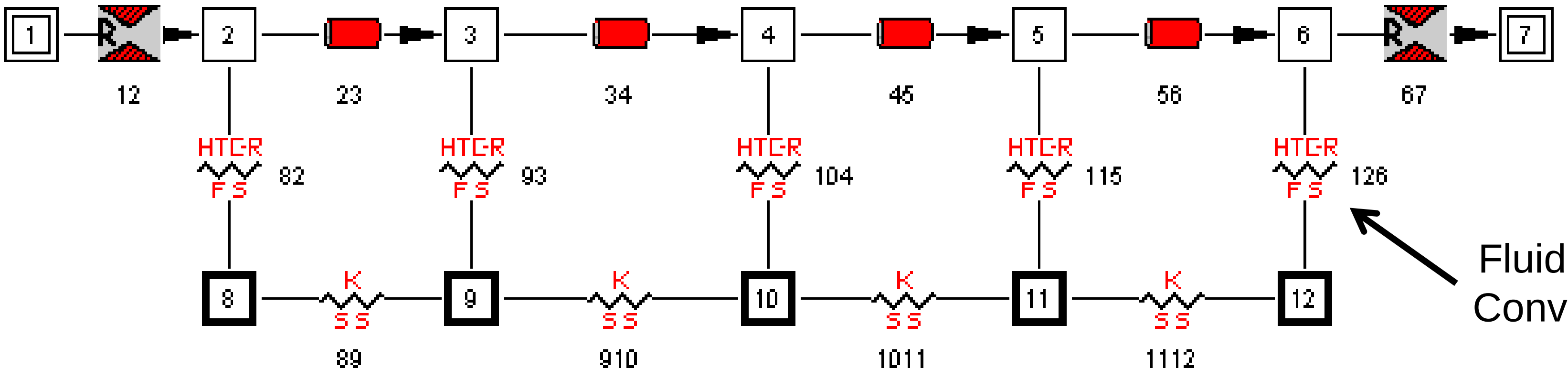
☐ Plot User Specified Values

Number User Variables: 0

☐ Tecplot Data

☐ Disable GFSSP Run Information

Build Model on Canvas



Fluid-Solid Convection

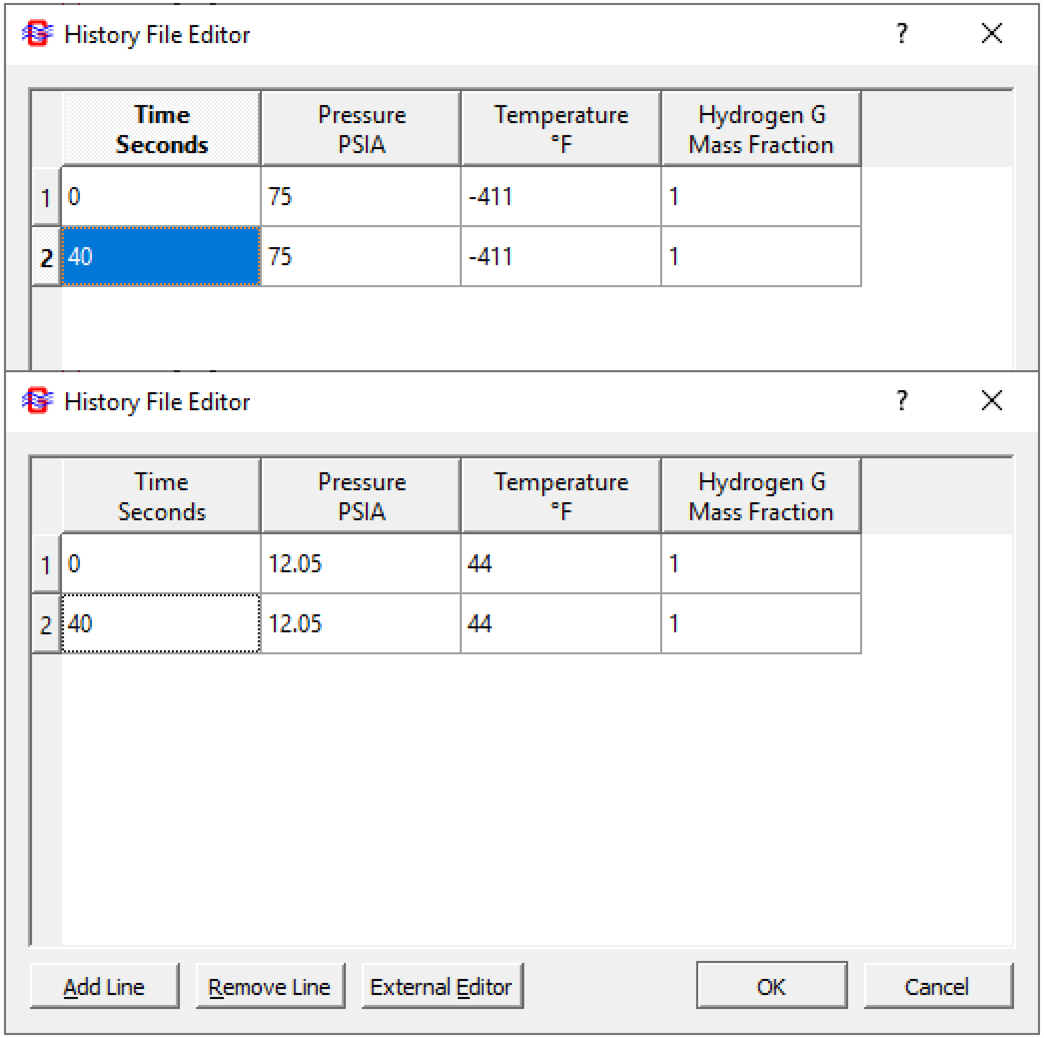
Solid Node

Solid-Solid Conduction

Now is a good time to save your **Tut5.gfssp** 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

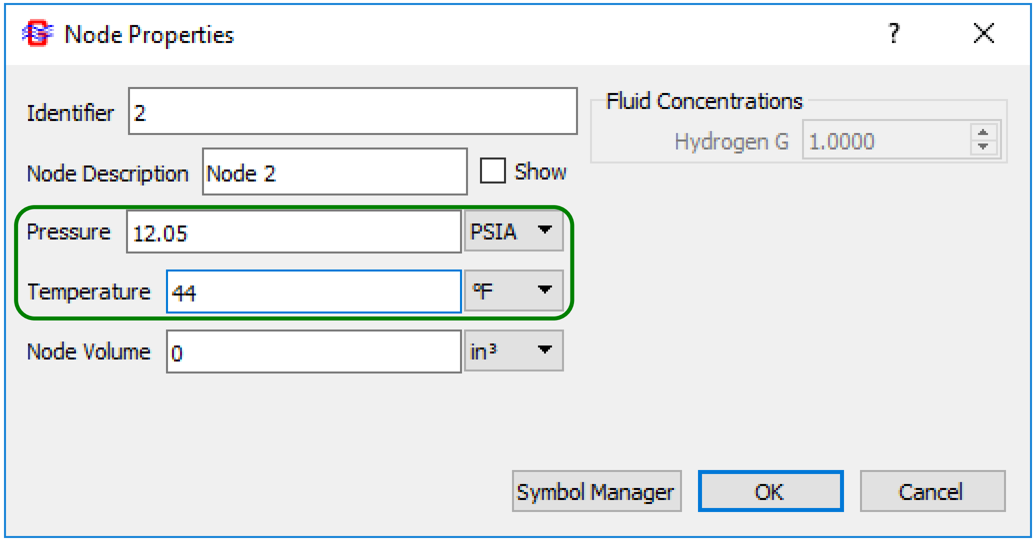


	Time Seconds	Pressure PSIA	Temperature °F	Hydrogen G Mass Fraction
1	0	75	-411	1
2	40	75	-411	1

	Time Seconds	Pressure PSIA	Temperature °F	Hydrogen G Mass Fraction
1	0	12.05	44	1
2	40	12.05	44	1

Set up Internal Node Initial Conditions

- Nodes 2 - 6
  - P = 12.05 psia
  - T = 44 °F
  - Volume not required – GFSSP will calculate from pipe dimensions
    - Hint: Copy/Paste Node 2 properties to Nodes 3 - 6



Identifier 2

Node Description Node 2 ☐ Show

Pressure 12.05 PSIA

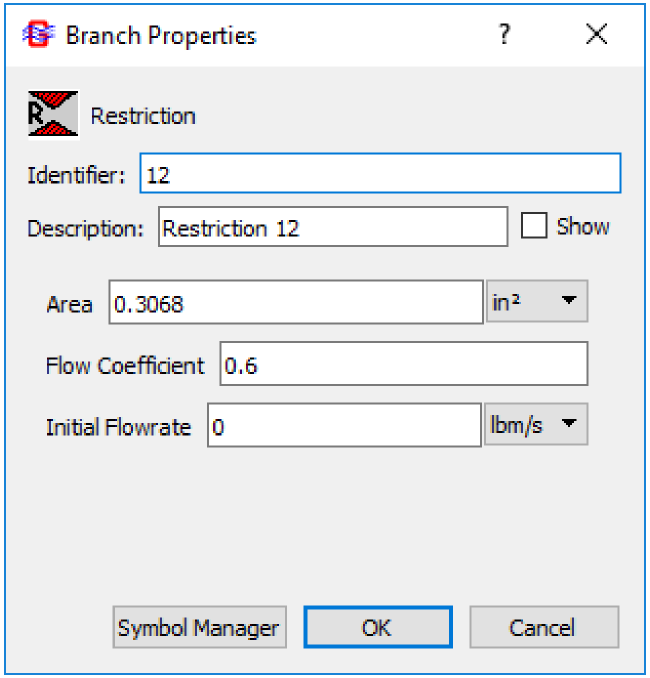
Temperature 44 °F

Node Volume 0 in³

Hydrogen G 1.0000

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
- Branches 23, 34, 45, 56: Pipes
  - L = 200 ft / 4 = 50 ft = 600 in
  - D = 0.625 in
  - Smooth pipe: ε = 0



Identifier: 12

Description: Restriction 12 ☐ Show

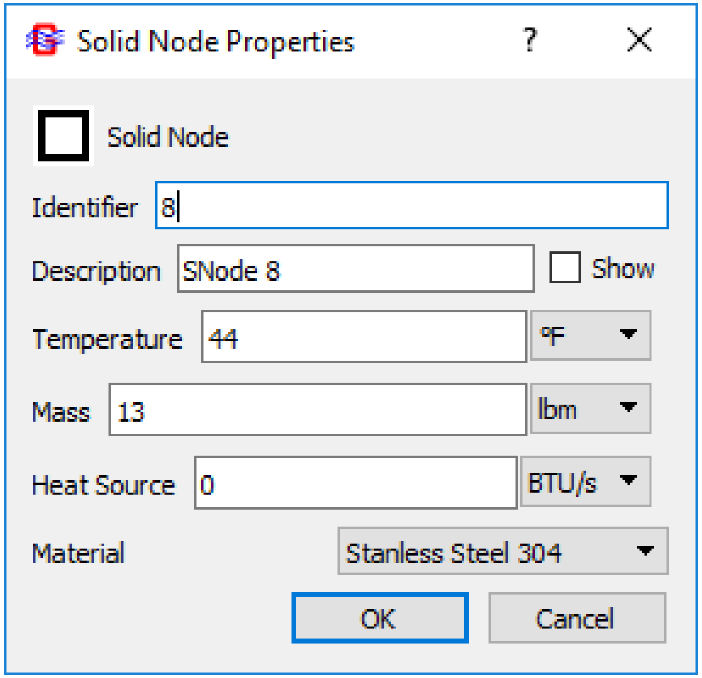
Area 0.3068 in²

Flow Coefficient 0.6

Initial Flowrate 0 lbm/s

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
  - Stainless Steel 304
    - Hint: Copy/Paste Solid Node 8 properties to Solid Nodes 9 -12



Identifier: 8

Description: SNode 8 ☐ Show

Temperature: 44 °F

Mass: 13 lbm

Heat Source: 0 BTU/s

Material: Stanless Steel 304

Set Up Conductors

- Fluid-Solid Convection
  - Total Wetted Area:

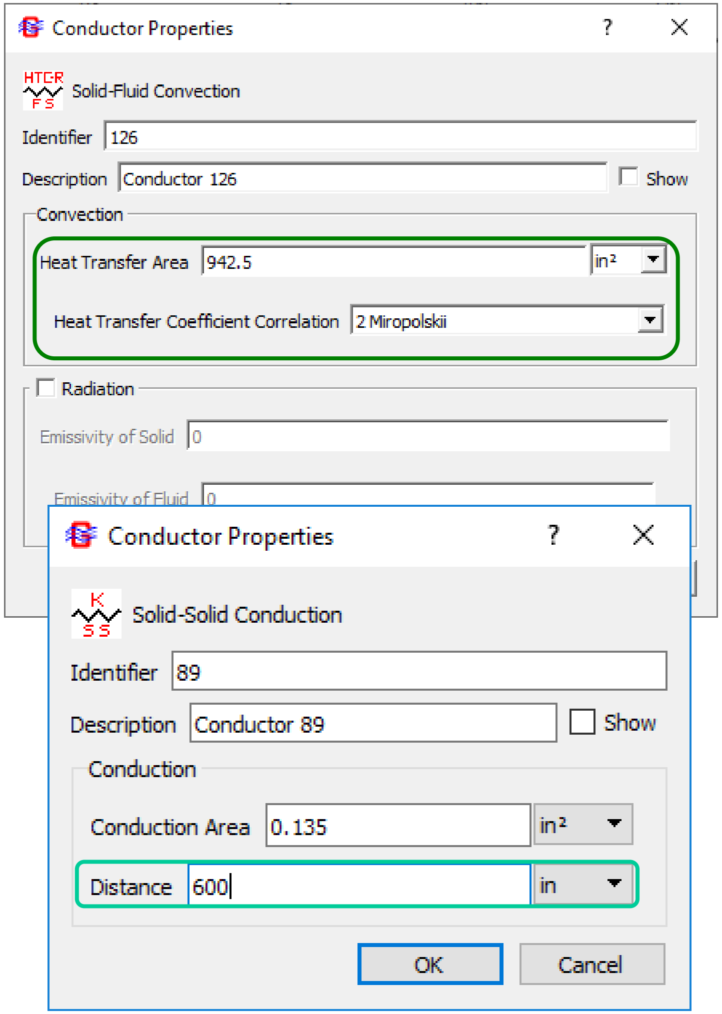
$$A = \pi D L = \pi(0.625 \text{ in.})(2400 \text{ in.}) = 4712 \text{ in}^2$$

  - Area per convector: 942.5 in^2
  - Miropolskii film boiling correlation

- Solid-Solid Conduction
  - Cross-Sectional Area:

$$A = \frac{\pi}{4}(OD^2 - ID^2)$$
$$= \frac{\pi}{4}[(0.75 \text{ in})^2 - (0.625 \text{ in})^2] = 0.135 \text{ in}^2$$

  - Length per conductor: 50 ft = 600 in
    - ➢ Hint: Copy/Paste also works for Fluid-to-Solid and Solid-to-Solid Conductors.



Conductor Properties

Solid-Fluid Convection

Identifier 126

Description Conductor 126 ☐ Show

Convection

Heat Transfer Area 942.5 in²

Heat Transfer Coefficient Correlation 2 Miropolskii

☐ Radiation

Emissivity of Solid 0

Emissivity of Fluid 0

Conductor Properties

Solid-Solid Conduction

Identifier 89

Description Conductor 89 ☐ Show

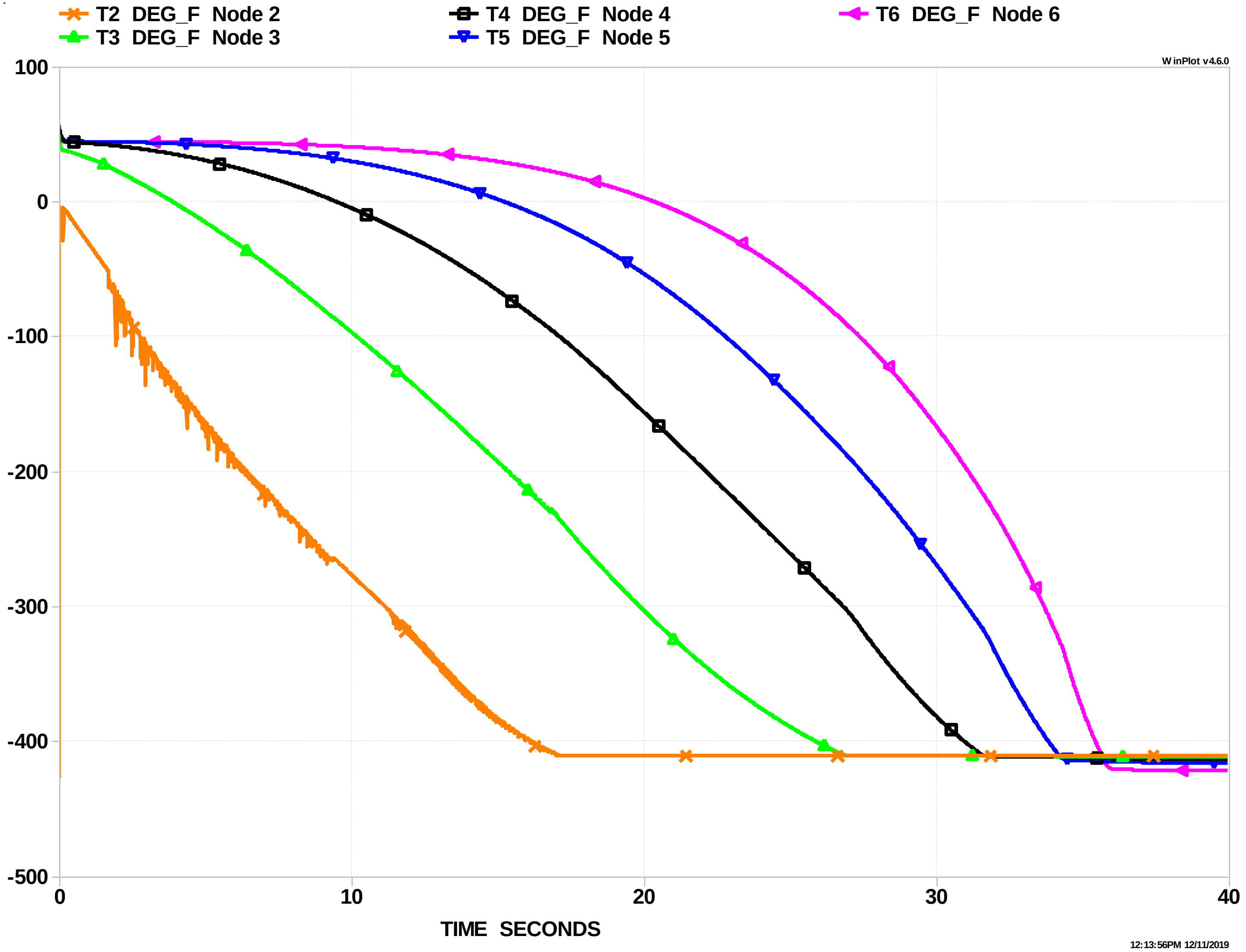
Conduction

Conduction Area 0.135 in²

Distance 600 in

Results (1/3)

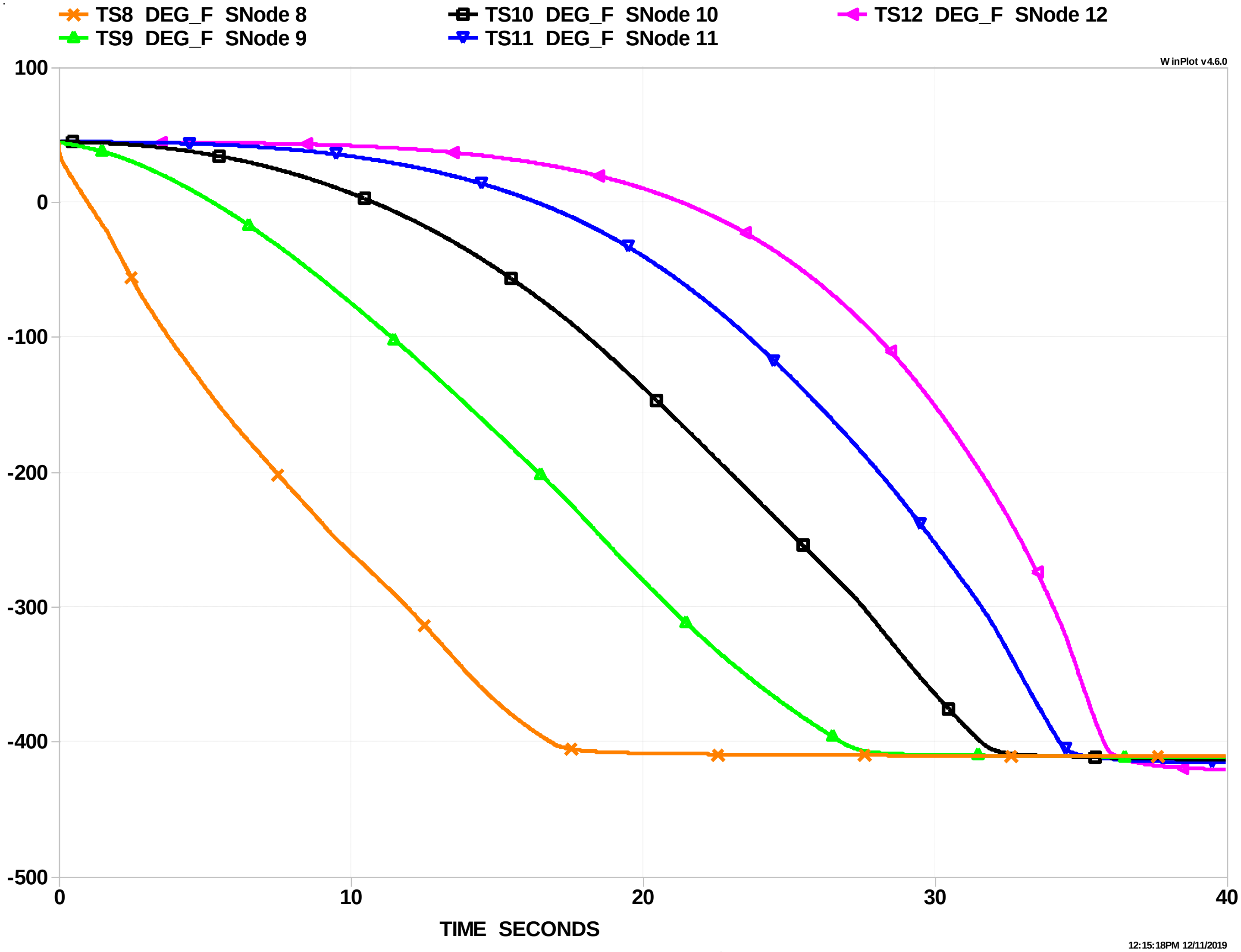
- Fluid Temperature



GFSSP 7.02 -- Tutorial 5

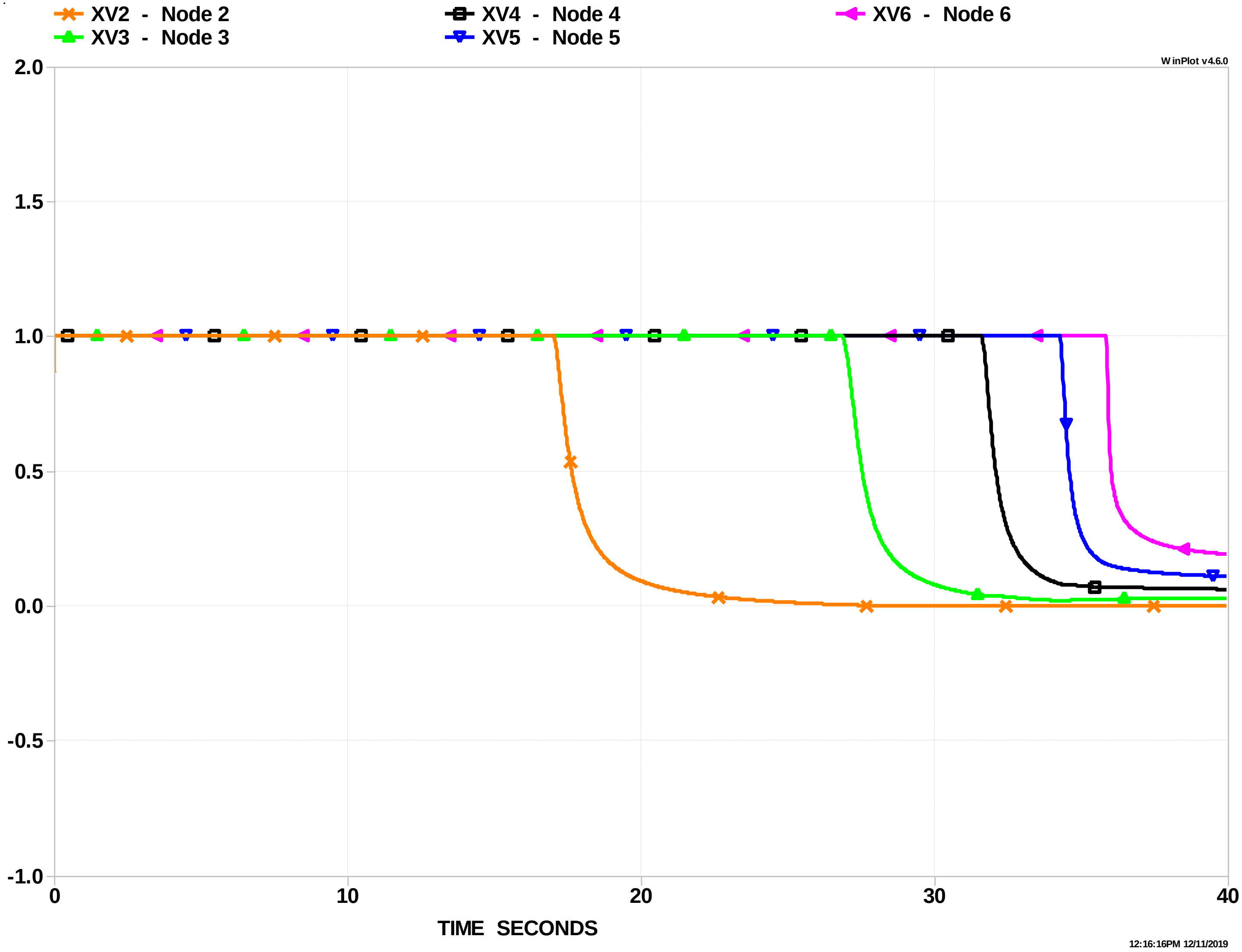
Results (2/3)

- Solid Temperature



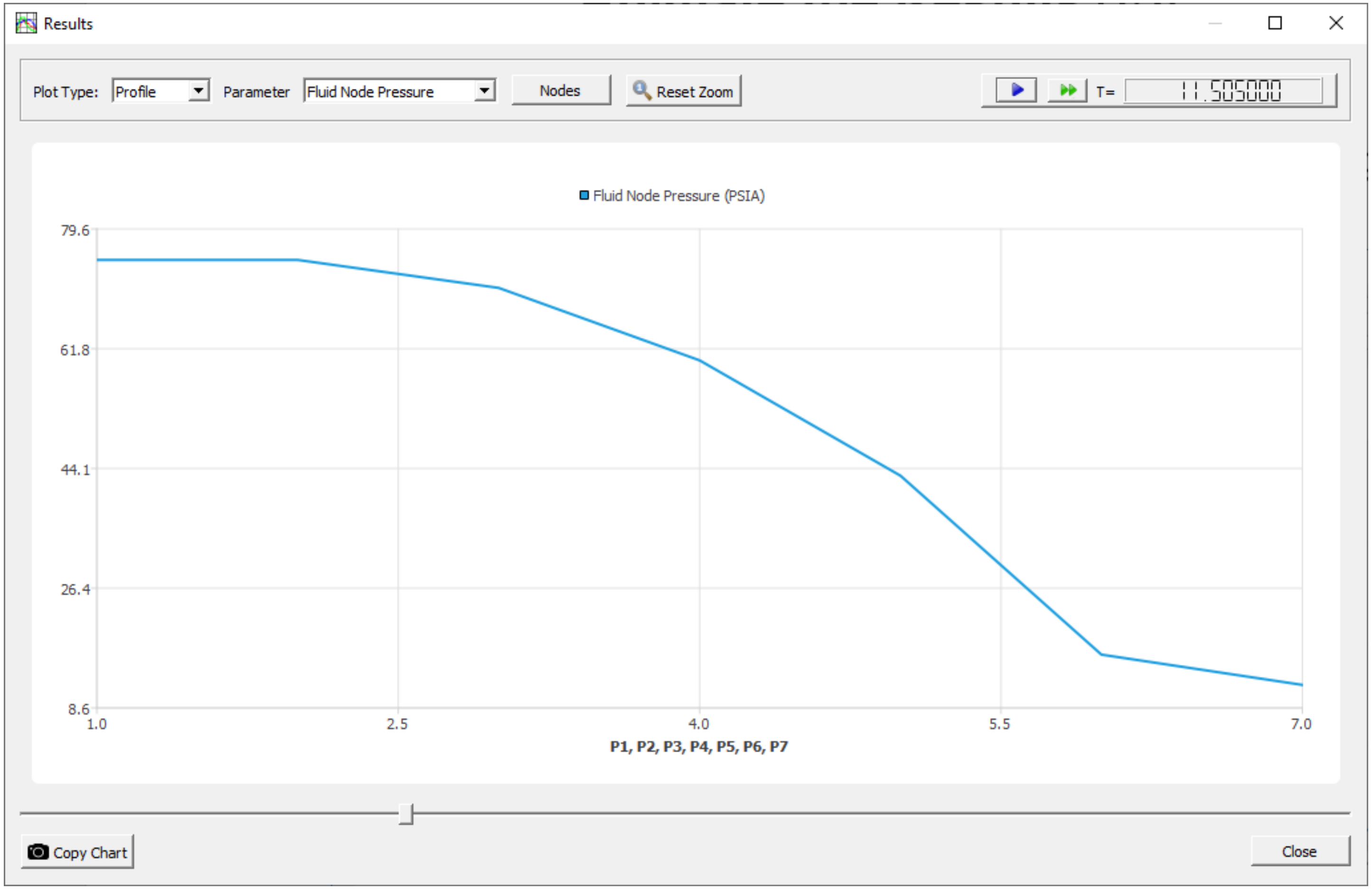
Results (3/3)

- Quality (Vapor Mass Fraction)



Animate the Results (1/2)

- Select Model / Plot Results to start the **MIG** plotter
- Change Plot Type to Profile and animate the fluid pressure changing over time.



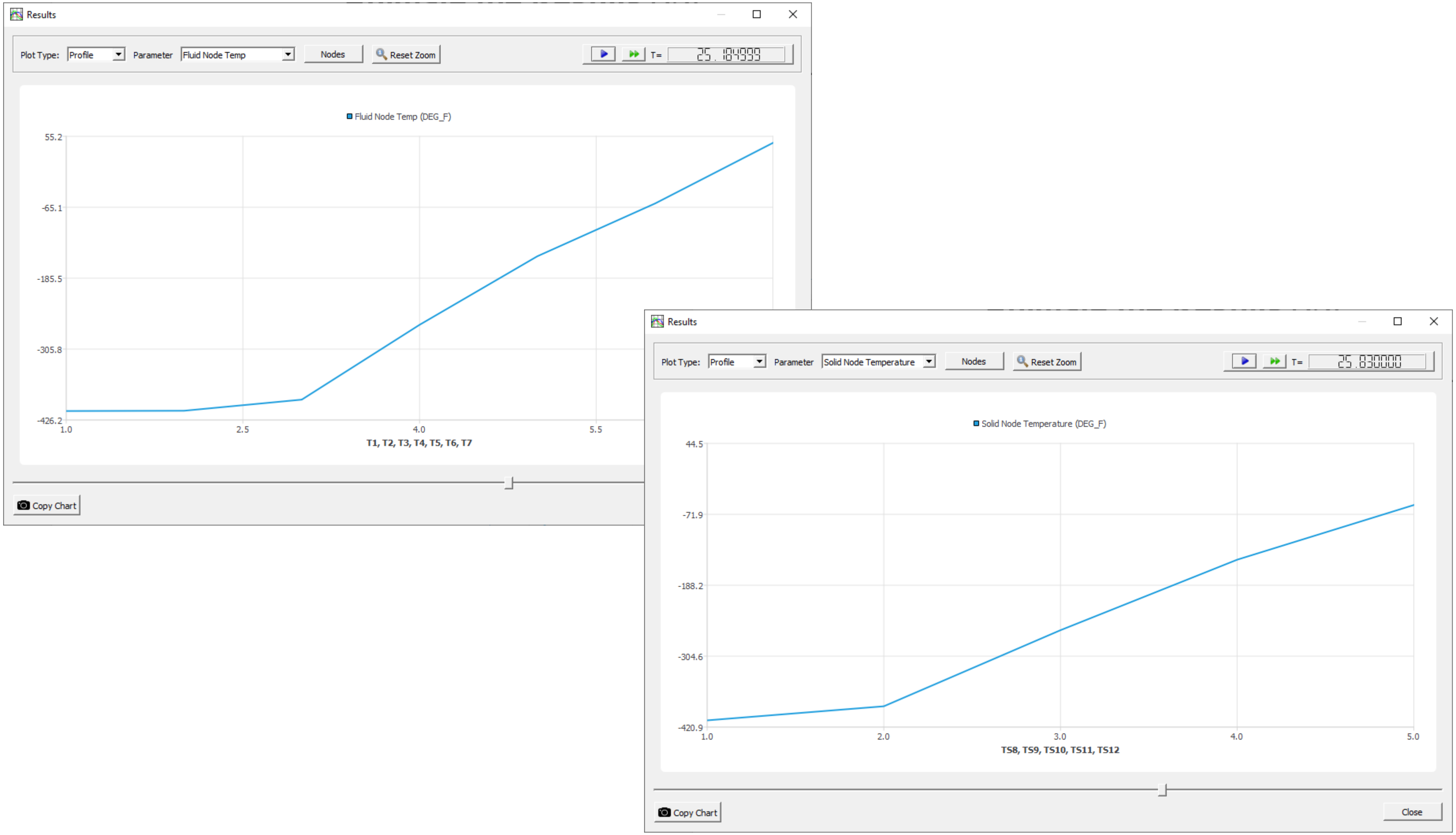
Plot Type: Profile

Parameter: Fluid Node Pressure

T= 11.505000

Animate the Results (2/2)

- Also animate Fluid Node Temperature and Solid Node Temperature



Results

Plot Type: Profile

Parameter: Fluid Node Temp

T= 25.184999

Results

Plot Type: Profile

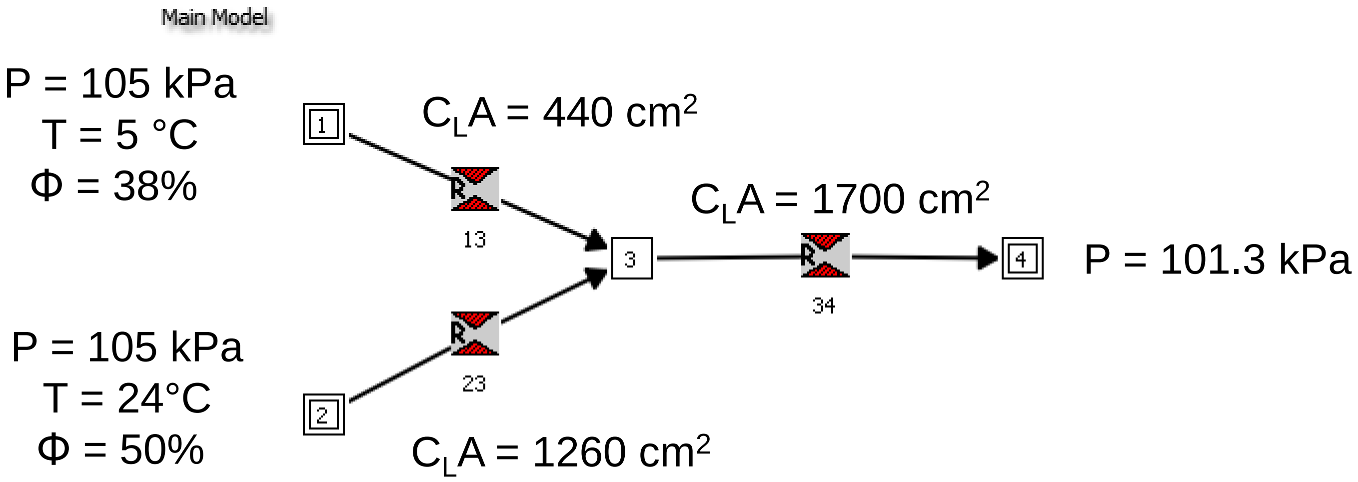
Parameter: Solid Node Temperature

T= 25.830000

Challenge Problem 5

Psychrometric Mixing

- A stream of moist air at 5°C and 38% relative humidity has a mass flow rate of 3.0 kg/s.
- A second stream of moist air at 24°C and 50% relative humidity flows at 8.4 kg/s.
- If the two streams mix adiabatically, predict the mixture temperature and humidity ratio (kg vapor / kg dry air).



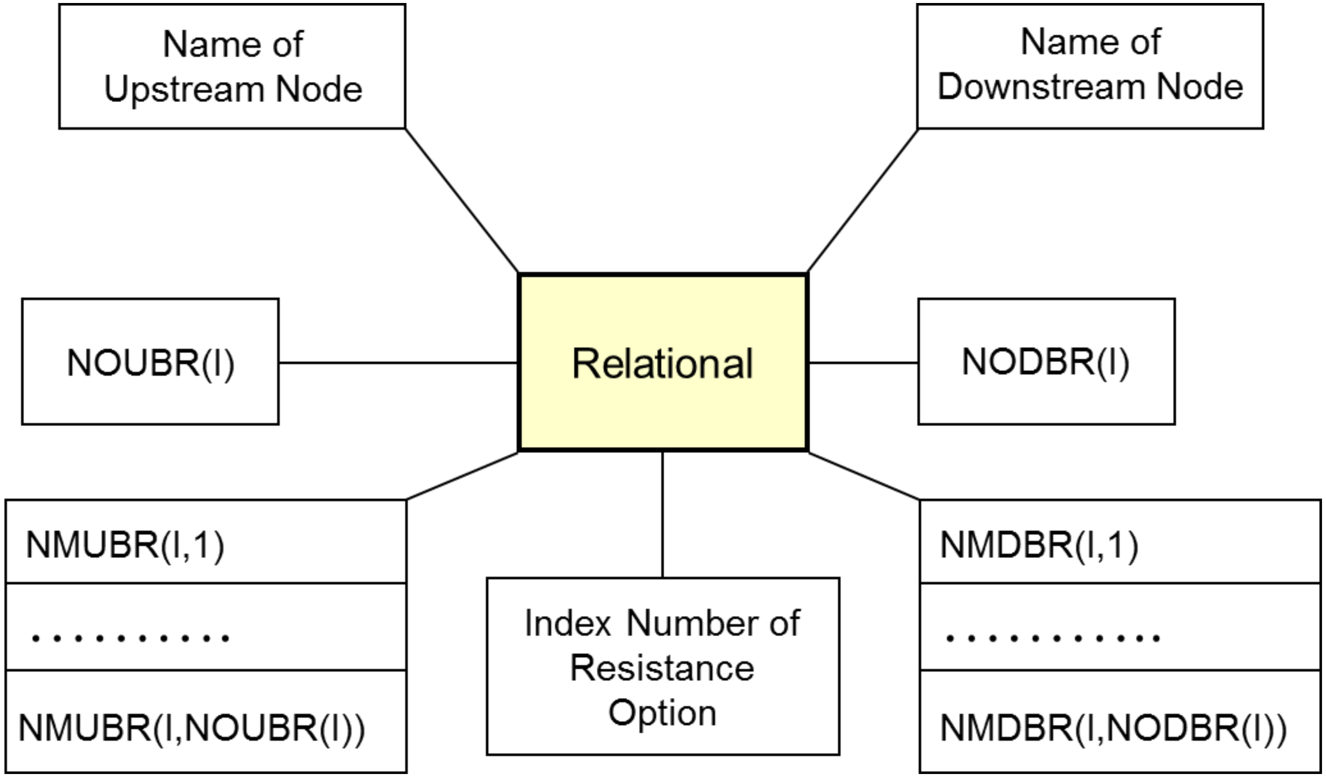
Challenge Problem 5
Psychrometric Mixing

- On the Circuit tab, activate Dalton's Law and Psychrometry

- There are two fluids
  - Water (WASP)
  - Air (GASPAK)

- Problem 5 is Example 12.17 from "Fundamentals of Engineering Thermodynamics", 3rd Ed., by Moran and Shapiro
  - Textbook answers
    - T_3 = 19 °C
    - ω_3 = 0.007 kg vapor / kg dry air

Data Structure



Name of Upstream Node

Name of Downstream Node

NOUBR(I)

Relational

NODBR(I)

NMUBR(I,1)

. . . .

NMUBR(I,NOUBR(I))

Index Number of Resistance Option

NMDBR(I,1)

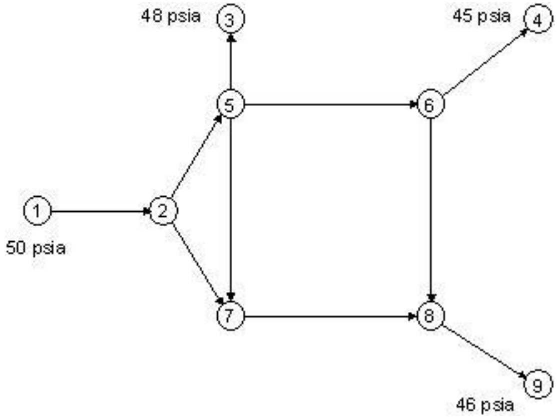
. . . .

NMDBR(I,NODBR(I))

Importance of Data Structure

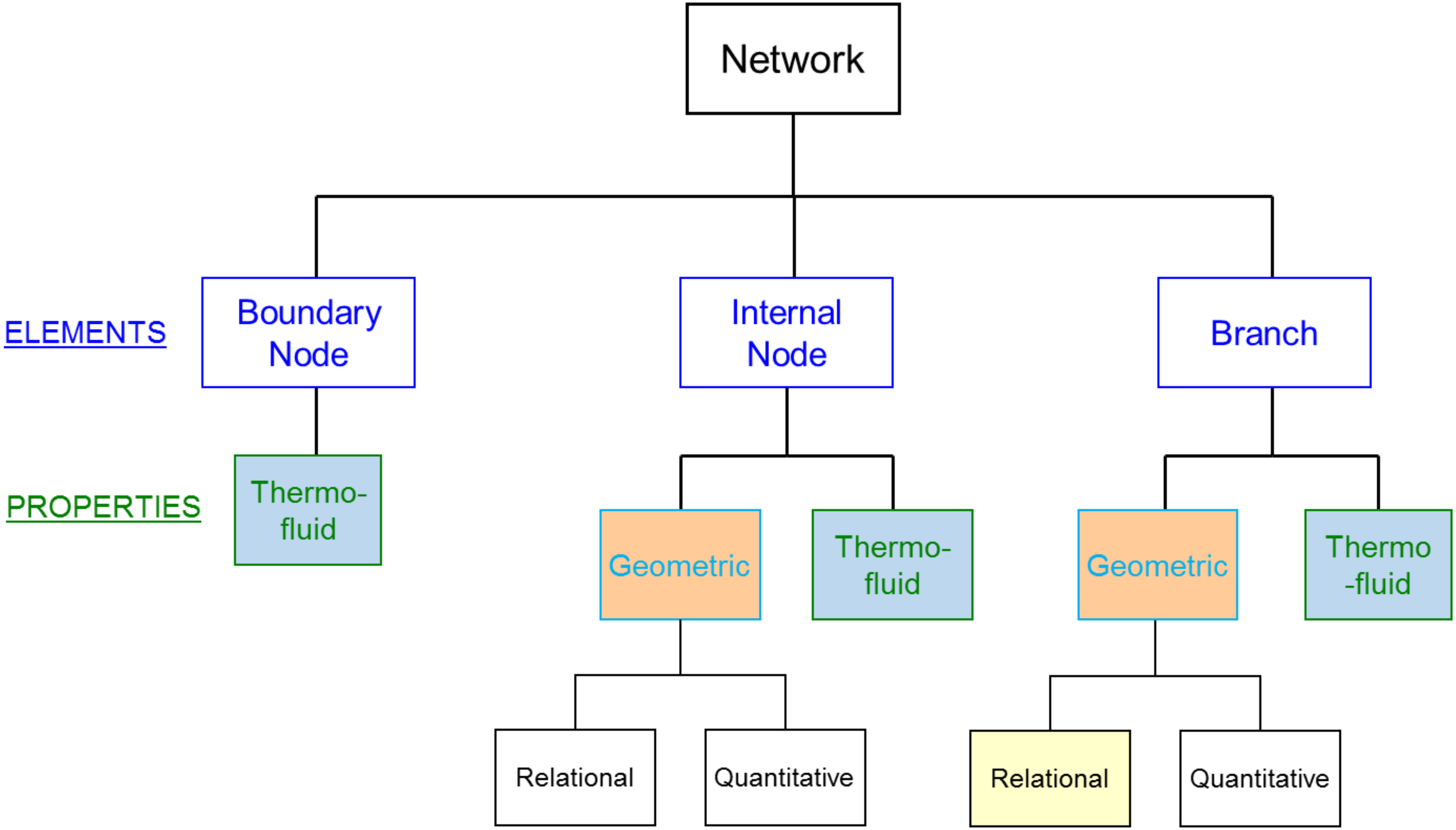
- In a Structured System
  - Array of nodes can be constructed in different coordinate direction
  - In 1-D flow network, each node has two neighbors
  - In 2-D flow network, each node has four neighbors
  - In 3-D flow network, each node has six neighbors

- In a Flow Network
  - Layout of nodes is not structured
  - No origin and coordinate direction to build the array of nodes
  - In a typical flow network a node can have “n” number of neighbors
    - “n” neighbors require unique data structure to define a flow network



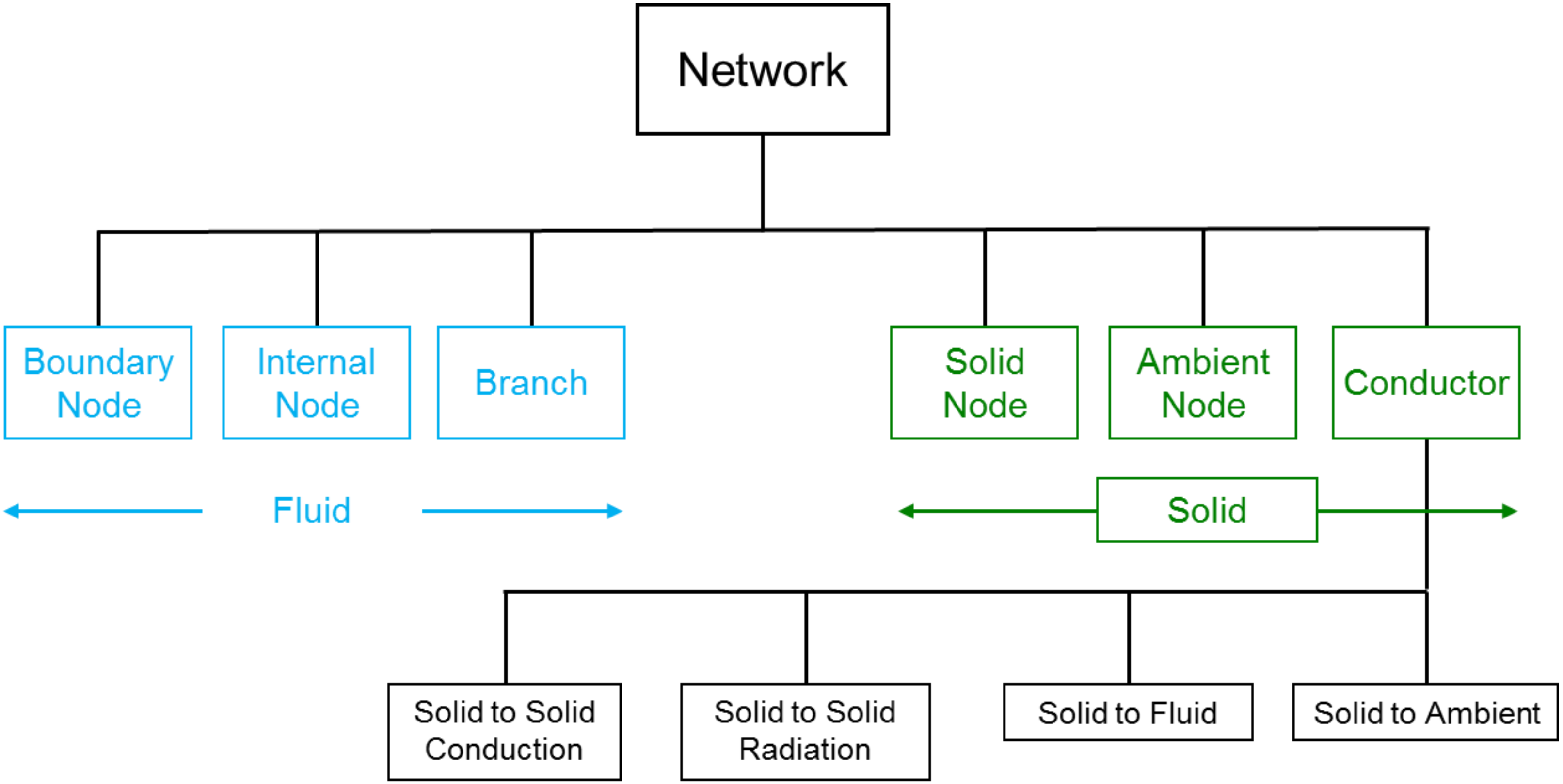
Data Structure for Flow Analysis

- Network Elements and Properties



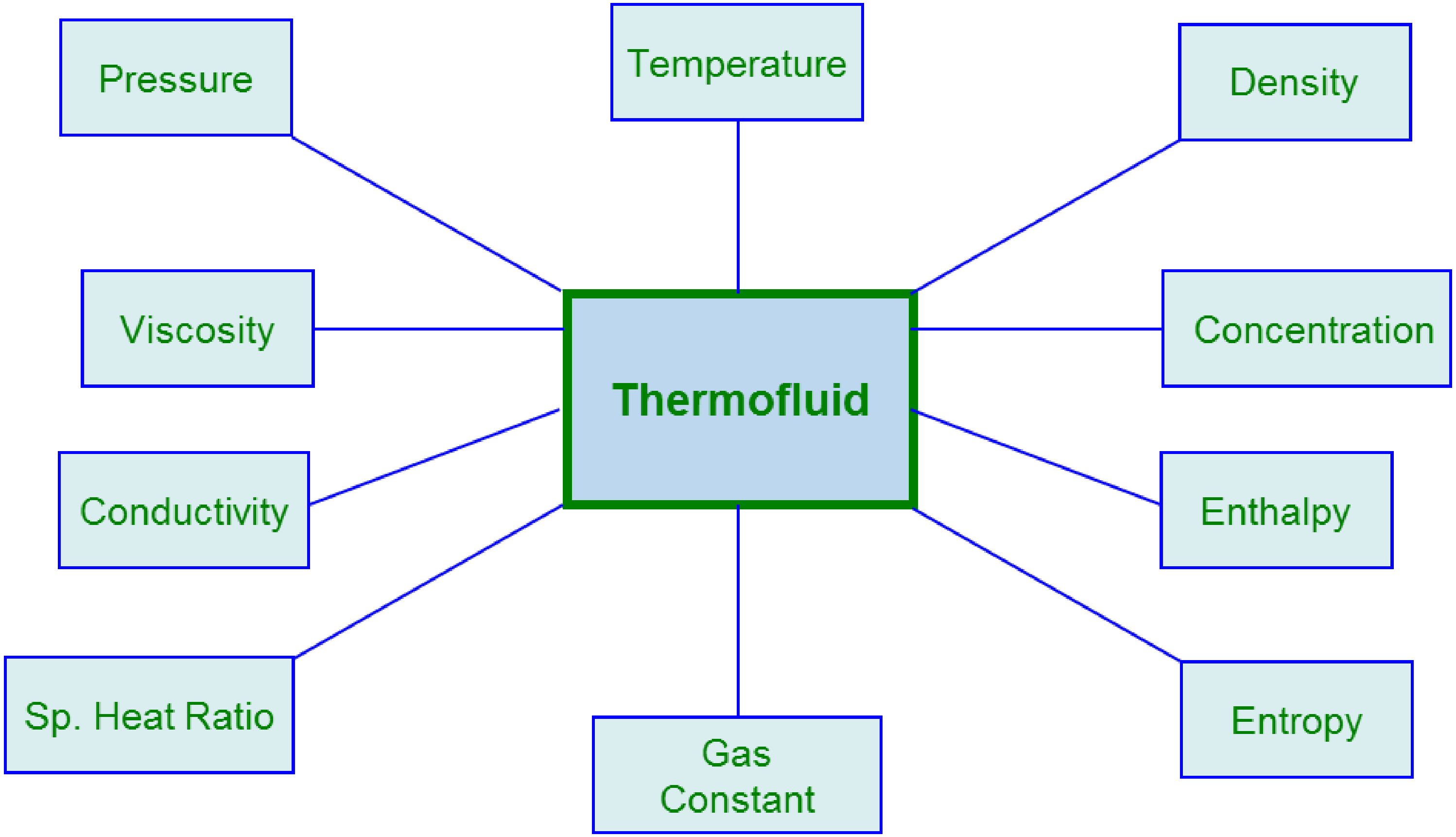
Extended Data Structure

- Network Elements for Conjugate Heat Transfer



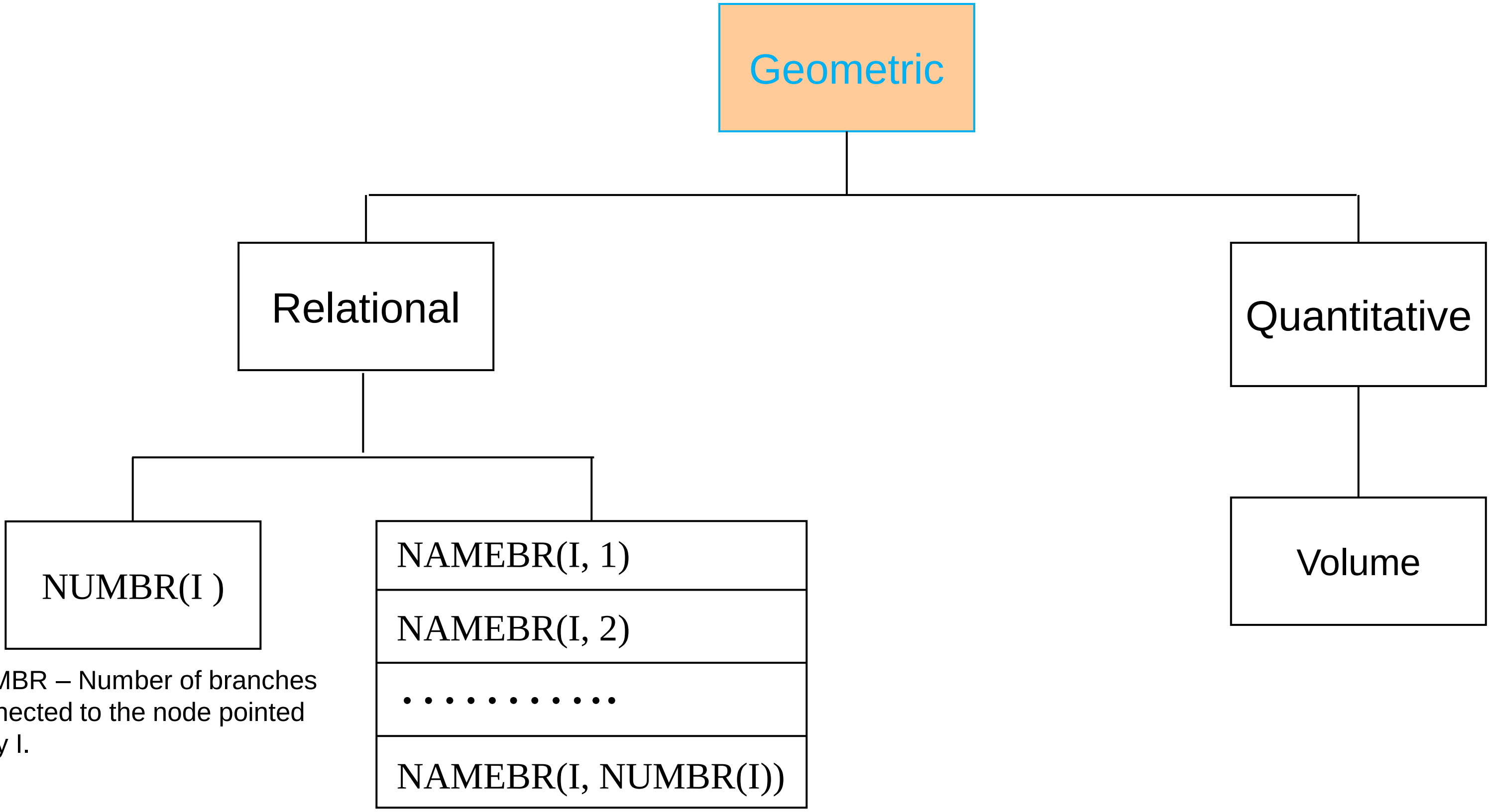
Internal & Boundary Nodes

- Thermofluid Properties



Internal Nodes

- Geometric Properties



Geometric

Relational

Quantitative

NUMBR(I)

NAMEBR(I, 1)
NAMEBR(I, 2)
. . . .
NAMEBR(I, NUMBR(I))

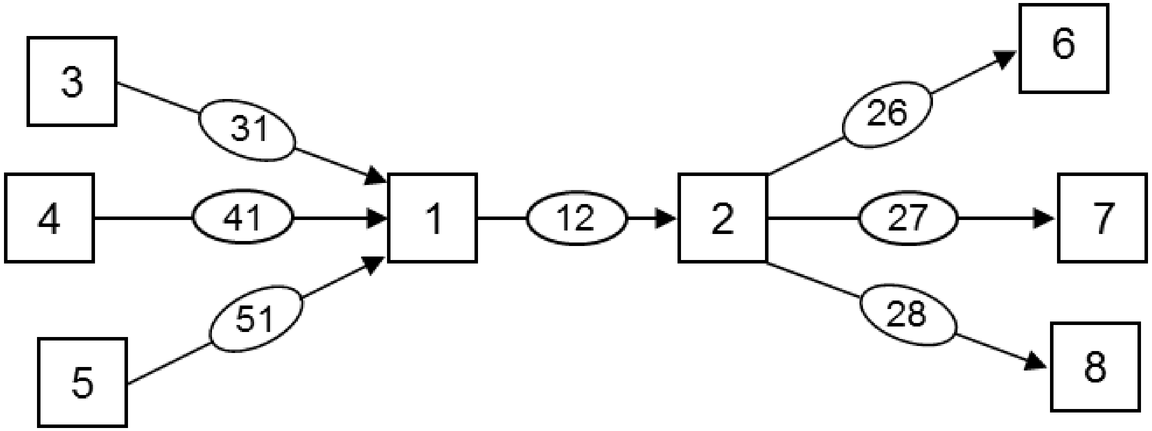
Volume

NUMBR – Number of branches connected to the node pointed to by I.

NAMEBR – Names of the branches connected to the node pointed to by I.

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

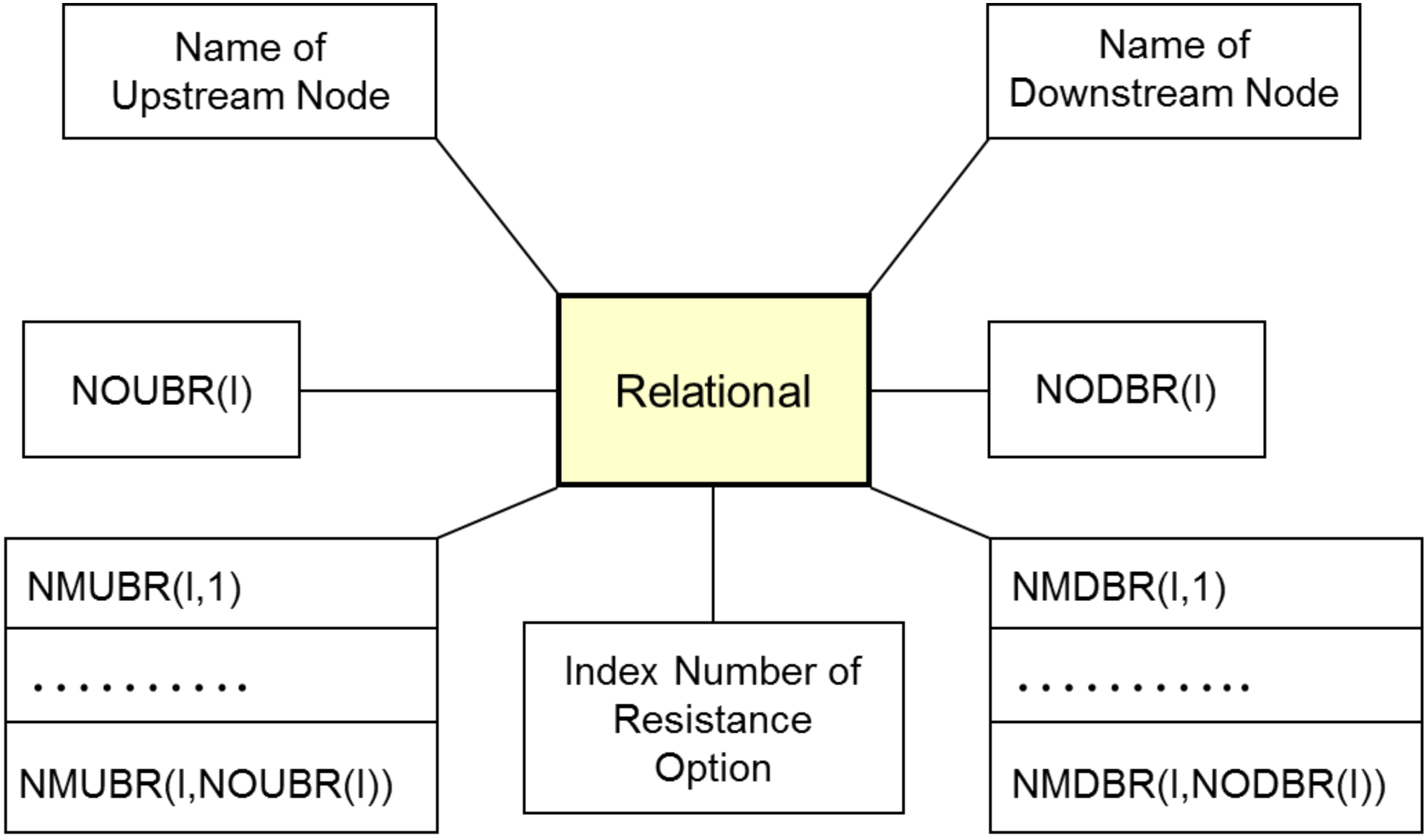
NAMEBR(I,2) = 41

NAMEBR(I,3) = 51

NAMEBR(I,4) = 12

Branch Properties

- Geometric - Relational



Name of Upstream Node

Name of Downstream Node

NOUBR(I)

Relational

NODBR(I)

NMUBR(I,1)

. . . .

NMUBR(I,NOUBR(I))

Index Number of Resistance Option

NMDBR(I,1)

. . . .

NMDBR(I,NODBR(I))

NOUBR – Number of Upstream Branches

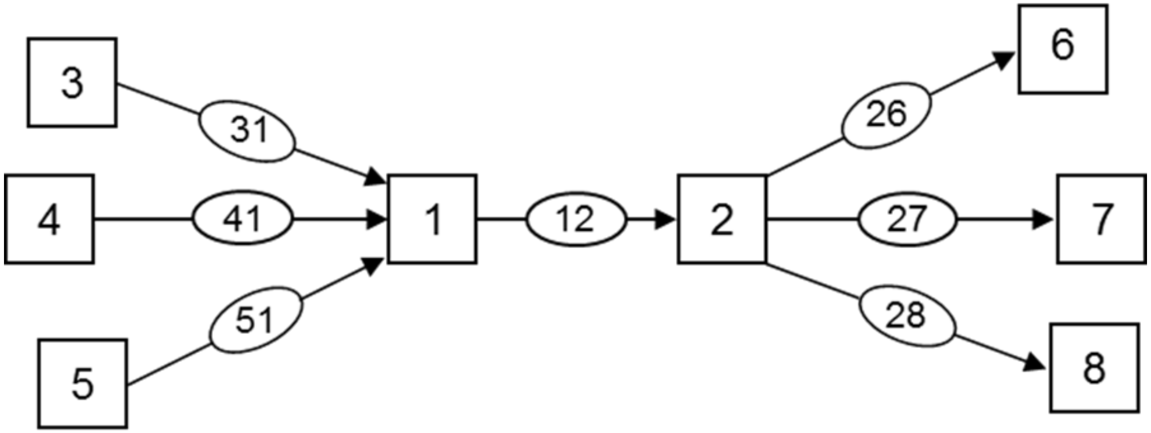
NMUBR – Name of Upstream Branches

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:

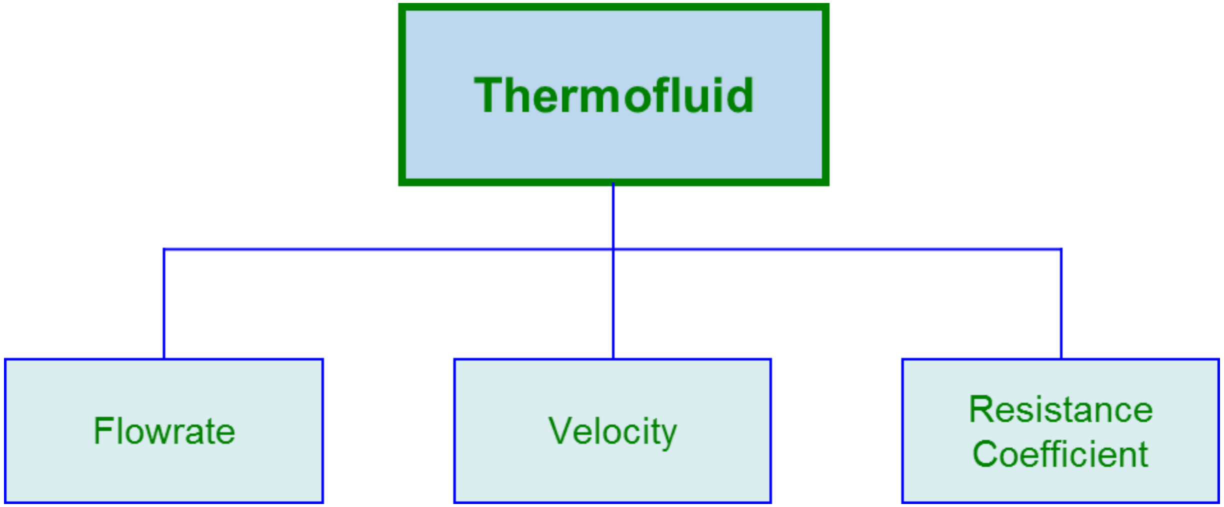
NMDBR(I,1) = 26

NMDBR(I,2) = 27

NMDBR(I,3) = 28

Branch Properties

- Thermofluid



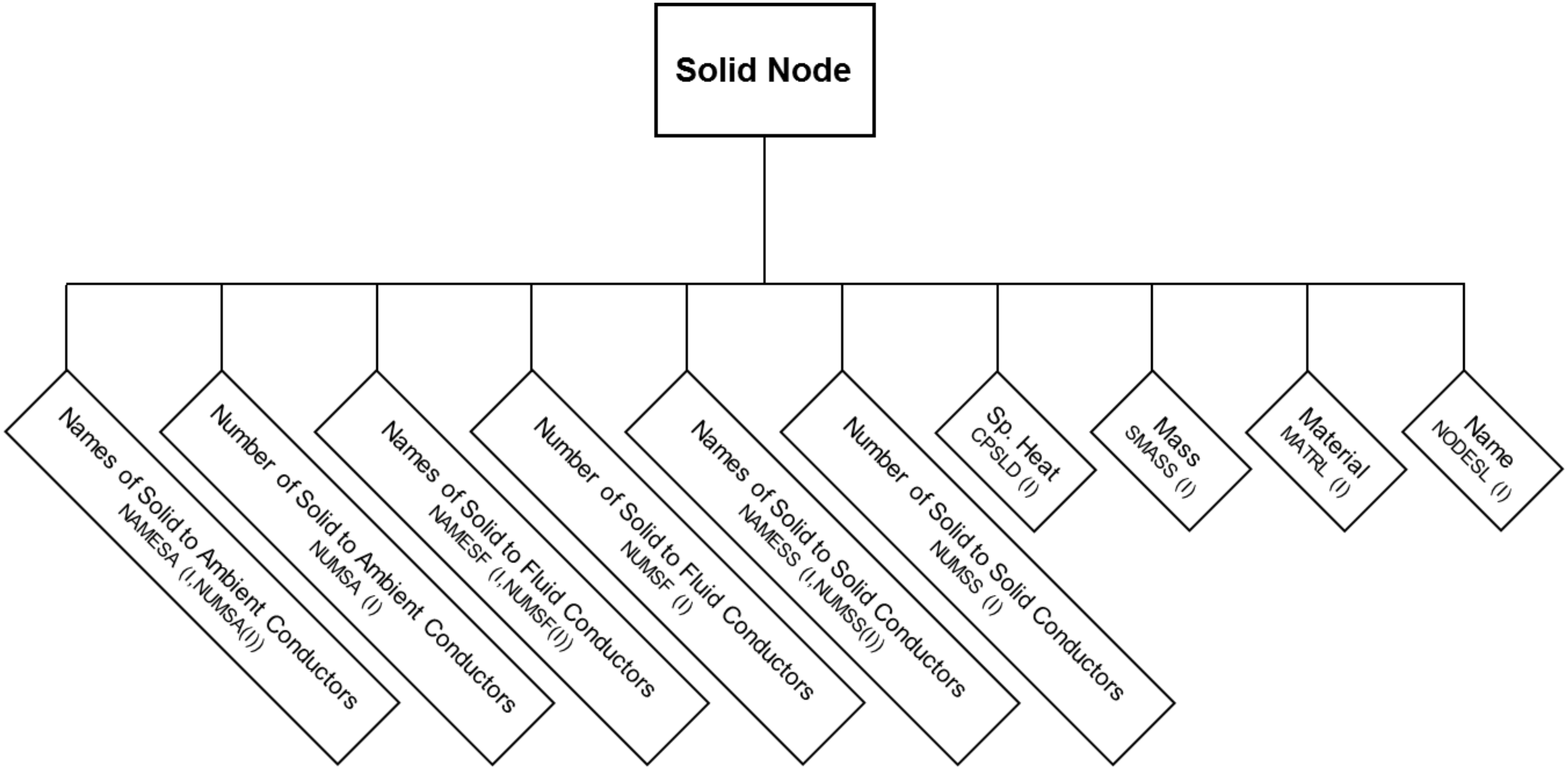
Thermofluid

Flowrate

Velocity

Resistance Coefficient

Solid Node Properties



Solid Node

Names of Solid to Ambient Conductors
NAMESA (I,NUMSA(I))

Number of Solid to Ambient Conductors
NUMSA (I)

Names of Solid to Fluid Conductors
NAMESF (I,NUMSF(I))

Number of Solid to Fluid Conductors
NUMSF (I)

Names of Solid to Solid Conductors
NAMESS (I,NUMSS(I))

Number of Solid to Solid Conductors
NUMSS (I)

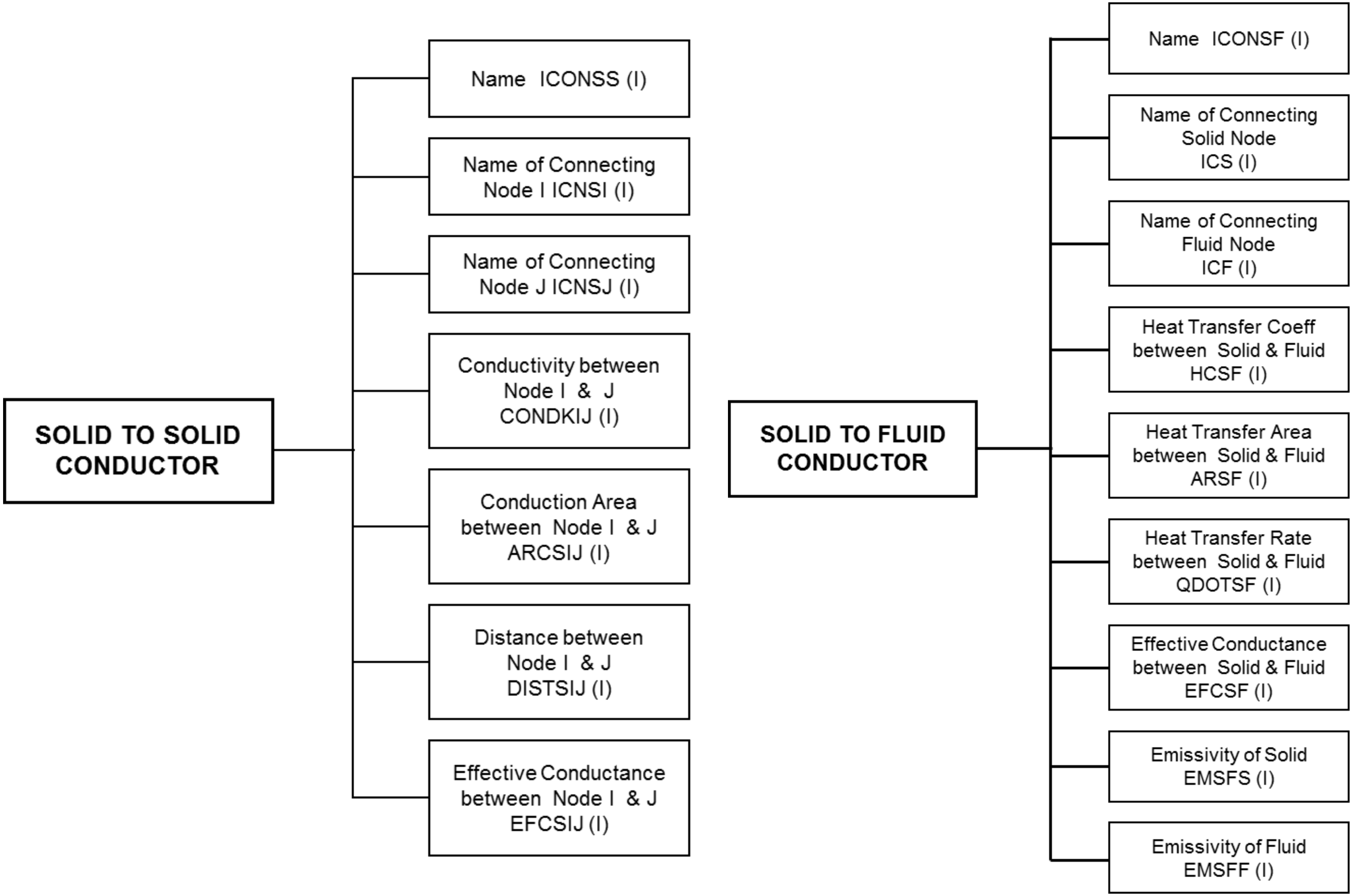
Sp. Heat
CPSLD (I)

Mass
SMASS (I)

Material
MATRL (I)

Name
NODESL (I)

Conductor Properties



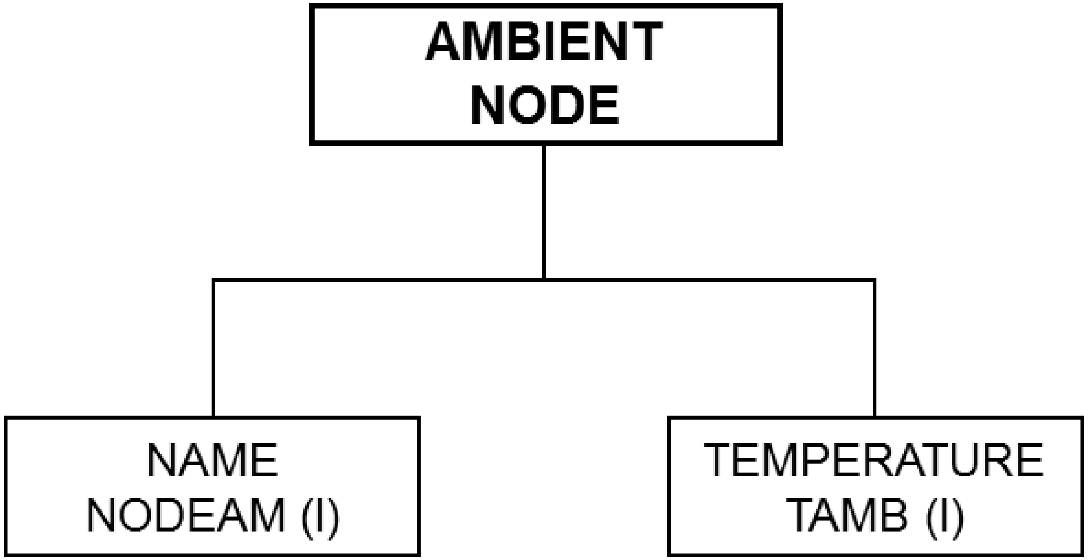
SOLID TO SOLID CONDUCTOR

- Name ICONSS (I)
- Name of Connecting Node I ICNSI (I)
- Name of Connecting Node J ICNSJ (I)
- Conductivity between Node I & J CONDKIJ (I)
- Conduction Area between Node I & J ARCSIJ (I)
- Distance between Node I & J DISTSIJ (I)
- Effective Conductance between Node I & J EFCSIJ (I)

SOLID TO FLUID CONDUCTOR

- Name ICONSF (I)
- Name of Connecting Solid Node ICS (I)
- Name of Connecting Fluid Node ICF (I)
- Heat Transfer Coeff between Solid & Fluid HCSF (I)
- Heat Transfer Area between Solid & Fluid ARSF (I)
- Heat Transfer Rate between Solid & Fluid QDOTSF (I)
- Effective Conductance between Solid & Fluid EFCSF (I)
- Emissivity of Solid EMSFS (I)
- Emissivity of Fluid EMSFF (I)

Ambient Node

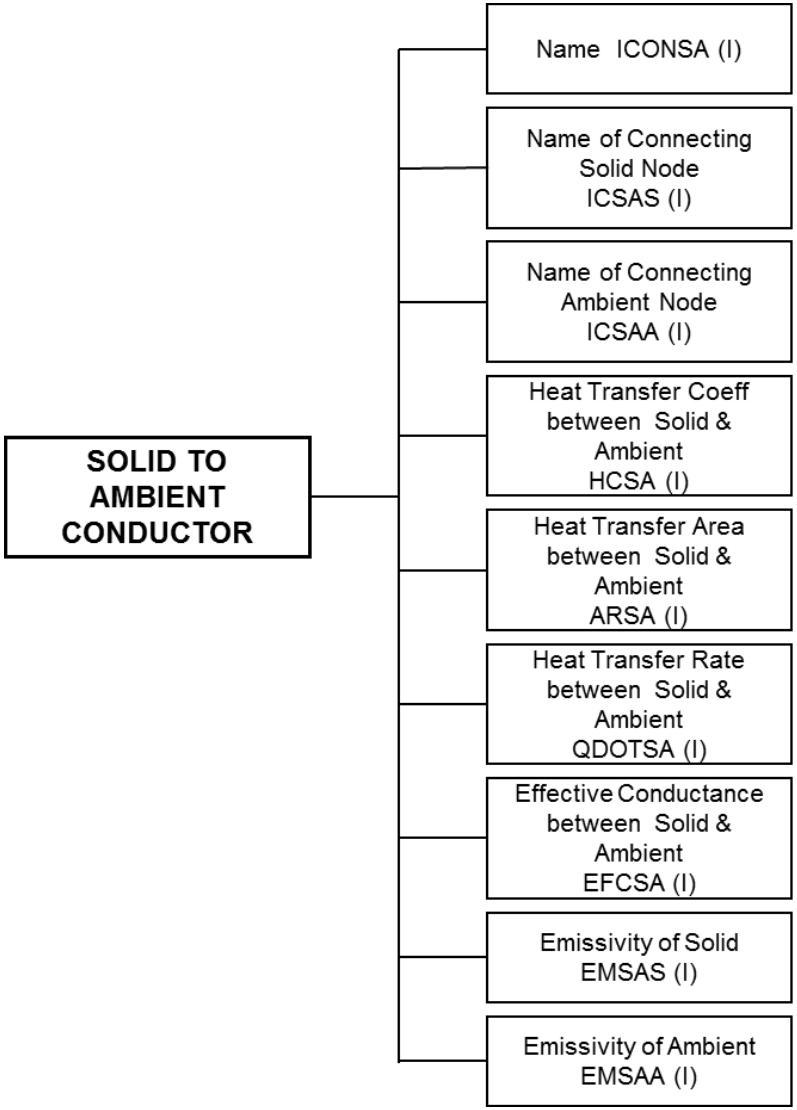


AMBIENT NODE

NAME
NODEAM (I)

TEMPERATURE
TAMB (I)

Solid to Ambient Conductor



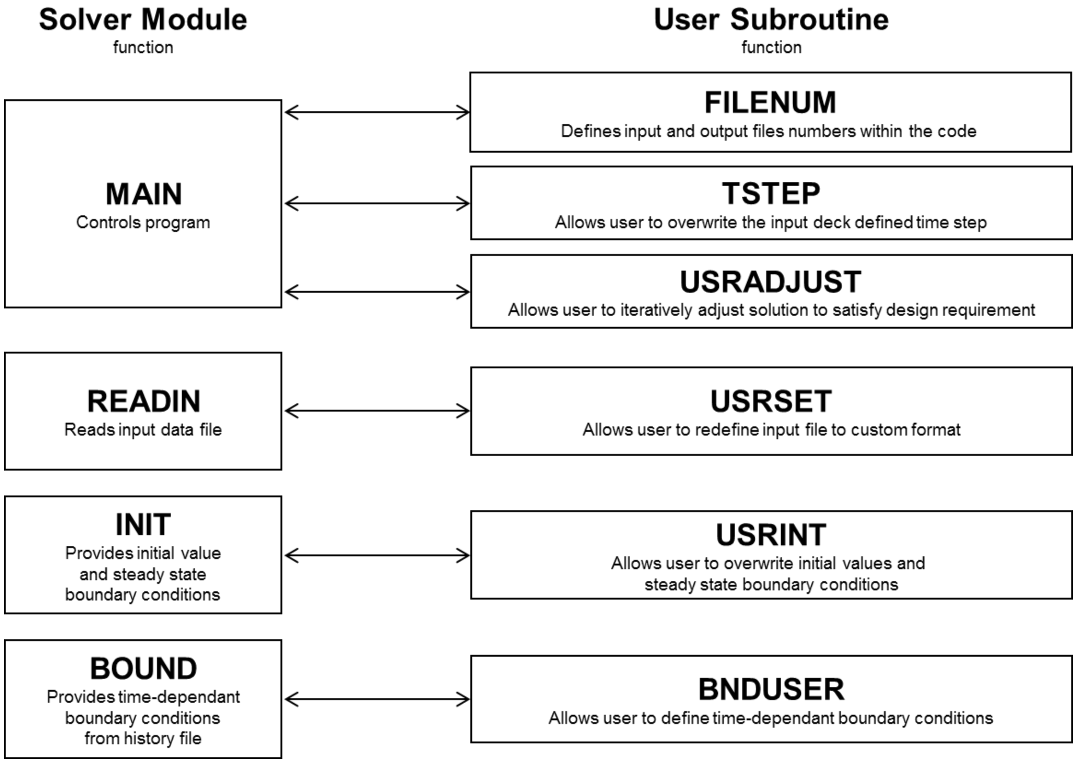
SOLID TO AMBIENT CONDUCTOR

- Name ICONSA (I)
- Name of Connecting Solid Node ICSAS (I)
- Name of Connecting Ambient Node ICSAA (I)
- Heat Transfer Coeff between Solid & Ambient HCSA (I)
- Heat Transfer Area between Solid & Ambient ARSA (I)
- Heat Transfer Rate between Solid & Ambient QDOTSA (I)
- Effective Conductance between Solid & Ambient EFCSA (I)
- Emissivity of Solid EMSAS (I)
- Emissivity of Ambient EMSAA (I)

Summary

- **GFSSP**’s Data Structure allows one to build any network system
  - Only limit is the dimension of the array - which can be increased, if needed
- **GFSSP** provides current allocations for
  - Nodes
    - Fluid: 300
    - Solid: 100
    - Ambient: 100
  - Branches:
    - Fluid branches: 500
    - Solid to Solid; Solid to Fluid; Solid to Ambient Conductor: 100 each
    - 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 Regulators: 10
- Knowledge of **GFSSP**’s Data Structure will be required for development of User Subroutines

User Subroutine



Solver Module
function

User Subroutine
function

MAIN
Controls program

FILENUM
Defines input and output files numbers within the code

TSTEP
Allows user to overwrite the input deck defined time step

USRADJUST
Allows user to iteratively adjust solution to satisfy design requirement

READIN
Reads input data file

USRSET
Allows user to redefine input file to custom format

INIT
Provides initial value and steady state boundary conditions

USRINT
Allows user to overwrite initial values and steady state boundary conditions

BOUND
Provides time-dependant boundary conditions from history file

BNDUSER
Allows user to define time-dependant boundary conditions

Background

- 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 user subroutines work?
  - A series of subroutines are called from various locations of the solver
  - Subroutines do not have any code but must include the GFSSP_GLOBAL module
  - Users can write FORTRAN code to develop any new physical model, in any particular Node or Branch

- What do users need to do?
  - Compile a user subroutine file containing all user routines into a Dynamic Link Library (*.DLL)
  - Run the main GFSSP executable with the DLL. (MIG handles this process automatically.)

Quick Fortran Review (1/3)

- Case insensitive: `DIAM` and `diam` are the same variable

- Implicit data typing:
  - `INTEGER` (variable starts with I, J, K, L, M, or N)
  - `REAL` (variable starts with any other letter)
  - “`SANTA` is `REAL` unless declared otherwise.”

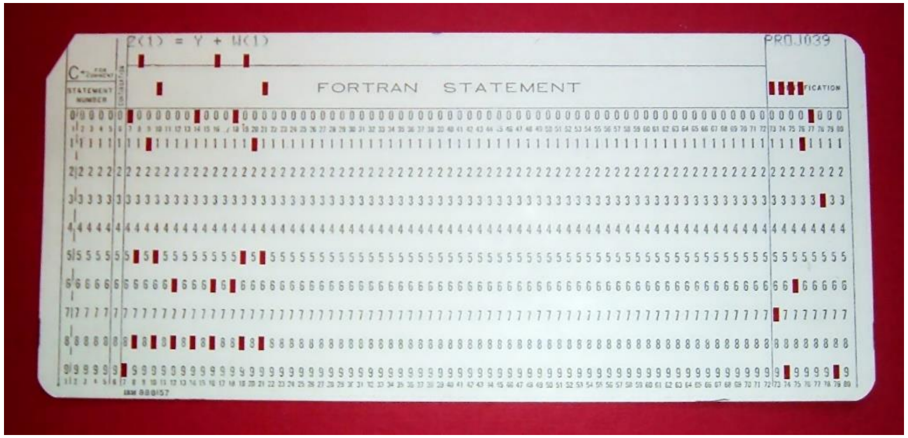
- Other data types must be explicitly declared
  - `LOGICAL`
  - `CHARACTER`
  - Arrays of any data type: `REAL FIDDLER(3)`
  - Declarations must be made at the start of each program unit

- Variables may be initialized at compile-time
  - Classic style, on two lines:
    ```
    REAL WEIGHT_LBS
     DATA WEIGHT_LBS /98.0/
    ```
  - F90 style, on one line: `REAL :: WEIGHT_LBS = 98.0`

Quick Fortran Review (2/3)

- Fixed-format (*.for files)
  - Classic style, based on 80-column punch cards
  - Column 1: `C` indicates a comment
  - Columns 1-5: An integer (1 – 99999) indicates a label number
  - Column 6: Any character (usually `+` or `&`) indicates line continuation
  - Columns 7-72: Actual code

```
C  This is sample fixed-format Fortran

      X = 4.0
      Y = X**2 + SQRT(X) /
     +    (X - 1.0)
      PRINT 5, ‘Y = ’, Y
    5 FORMAT(A, ES10.3)

C23456789012345678901234567890
```



Quick Fortran Review (3/3)

- Free-format (*.f90 files)
  - `!` indicates a comment
  - `&` indicates the next line is a continuation
  - Code may be written in any column, but regular indenting practices are recommended

```
        !  This is sample free-format Fortran

          X = 4.0
   Y = X**2 + SQRT(X) /    &
              (X - 1.0)
  PRINT 5, ‘Y = ’, Y
5 FORMAT(A, ES10.3)
```

Description of User Subroutines (1/3)

- Twenty-three User Subroutines are provided
- Most commonly used are:
  - **BNDUSER**: Variable boundary condition during transient run
  - **KFUSER**: New resistance option
  - **SORCEQ**: External Heat Source in Fluid Node
  - **SORCETS**: External Heat Source in Solid Node
  - **USRHCF**: New Heat Transfer Correlation
  - **USRADJUST**: Solution adjustment to satisfy design requirement
  - **KFADJUST**: Adjust resistance factor (K_f) if necessary

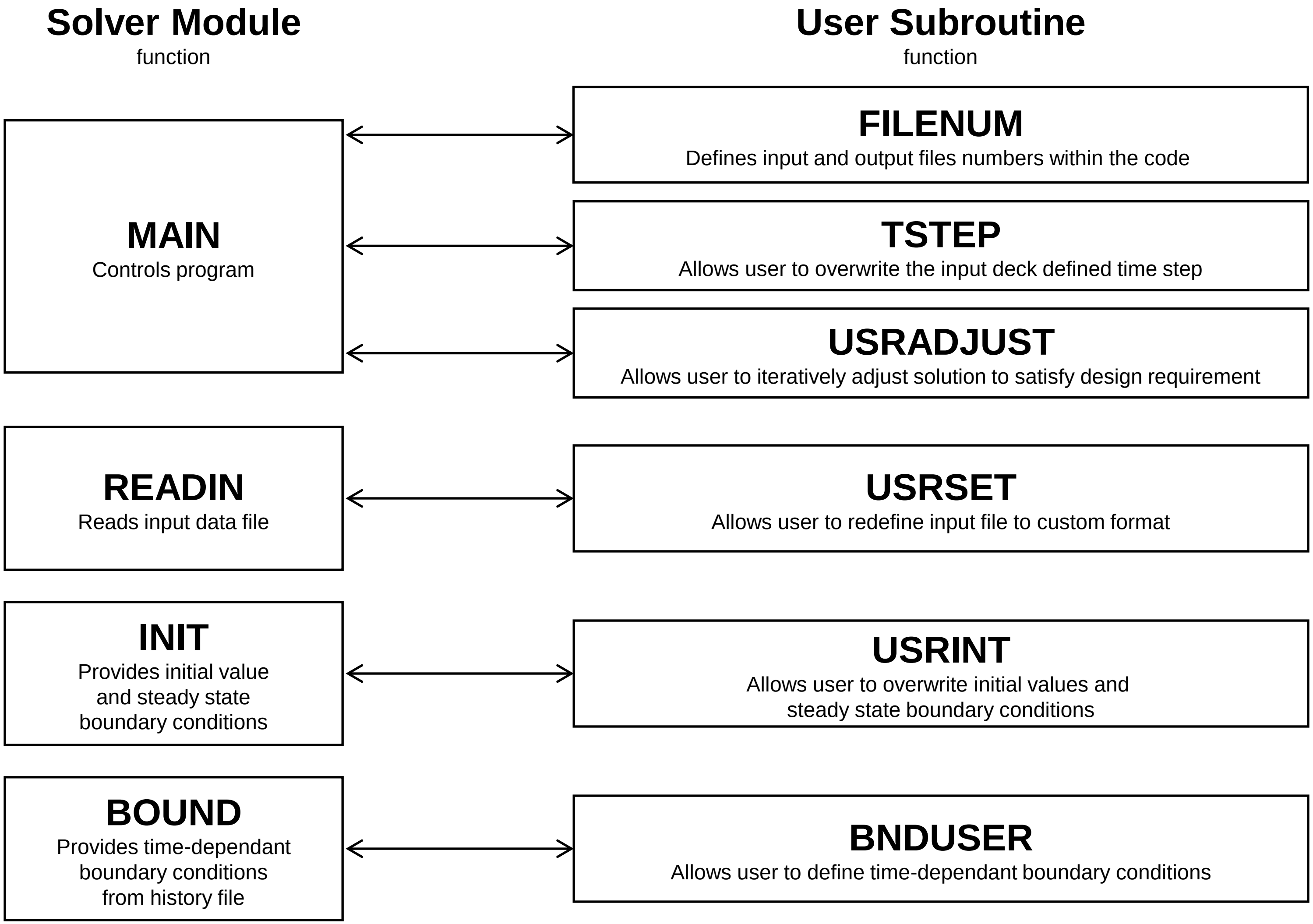
Description of User Subroutines (2/3)

- Less Commonly Used (1/2)
  - **SORCEM**: External Mass Source
  - **SORCEF**: External Force
  - **SORCEC**: External Concentration source
  - **PRPUSER**: Overwrite fluid properties; call other fluid packages such as REFPROP
  - **TSTEP**: Variable time step during a transient run
  - **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

Description of User Subroutines (3/3)

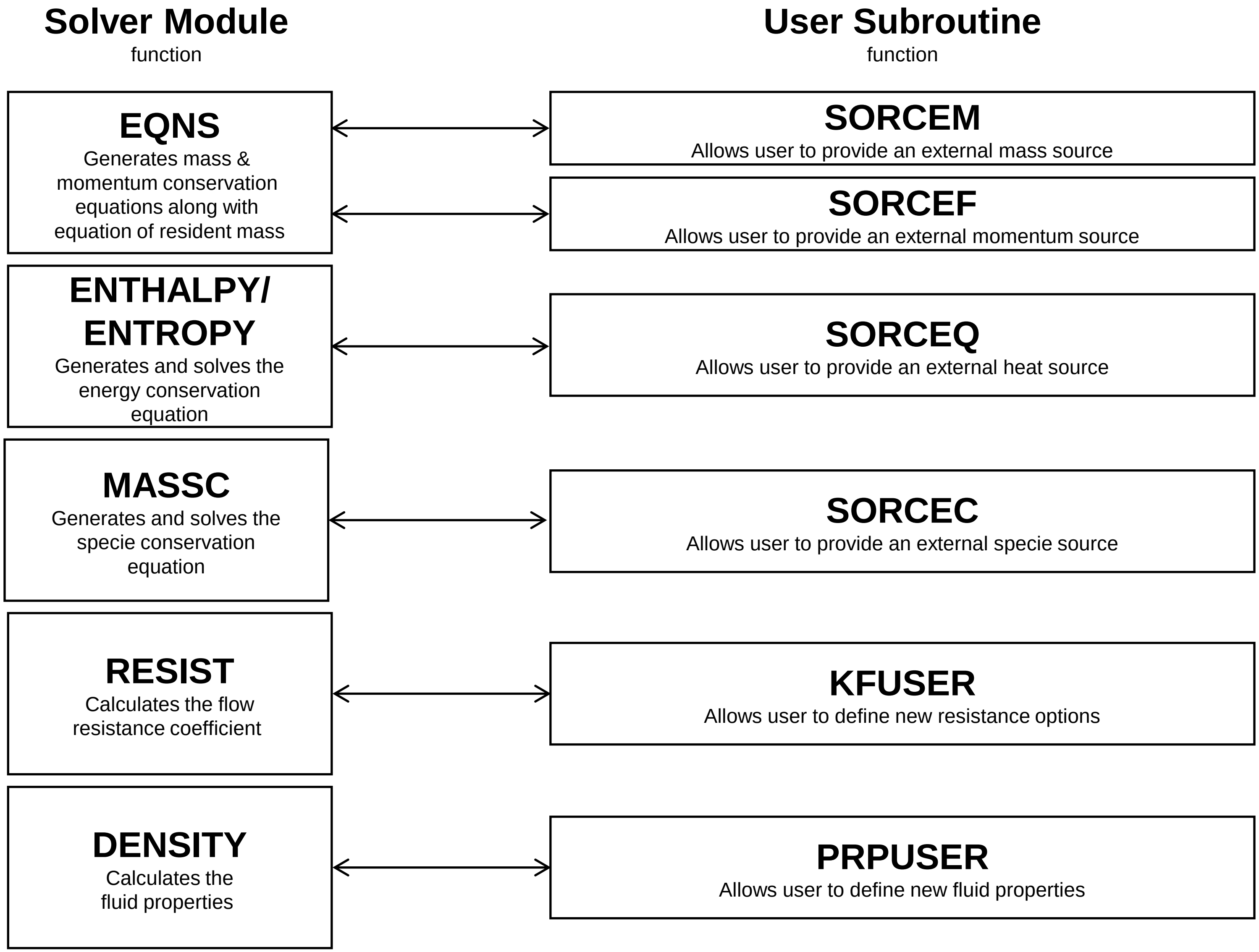
- Less Commonly Used (2/2)
  - **USRSET**: User can supply all the necessary information by writing their own code
  - **PRPADJUST**: Adjust Thermodynamic or Thermophysical Property
  - **TADJUST**: Adjust Temperature, if necessary
  - **PADJUST**: Adjust Pressure, if necessary
  - **FLADJUST**: Adjust 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 (1/3)



Solver Module function	**User Subroutine** function
MAIN Controls program	**FILENUM** Defines input and output files numbers within the code
	TSTEP Allows user to overwrite the input deck defined time step
	USRADJUST Allows user to iteratively adjust solution to satisfy design requirement
READIN Reads input data file	**USRSET** Allows user to redefine input file to custom format
INIT Provides initial value and steady state boundary conditions	**USRINT** Allows user to overwrite initial values and steady state boundary conditions
BOUND Provides time-dependant boundary conditions from history file	**BNDUSER** Allows user to define time-dependant boundary conditions

Solver-User Subroutine Interaction (2/3)



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

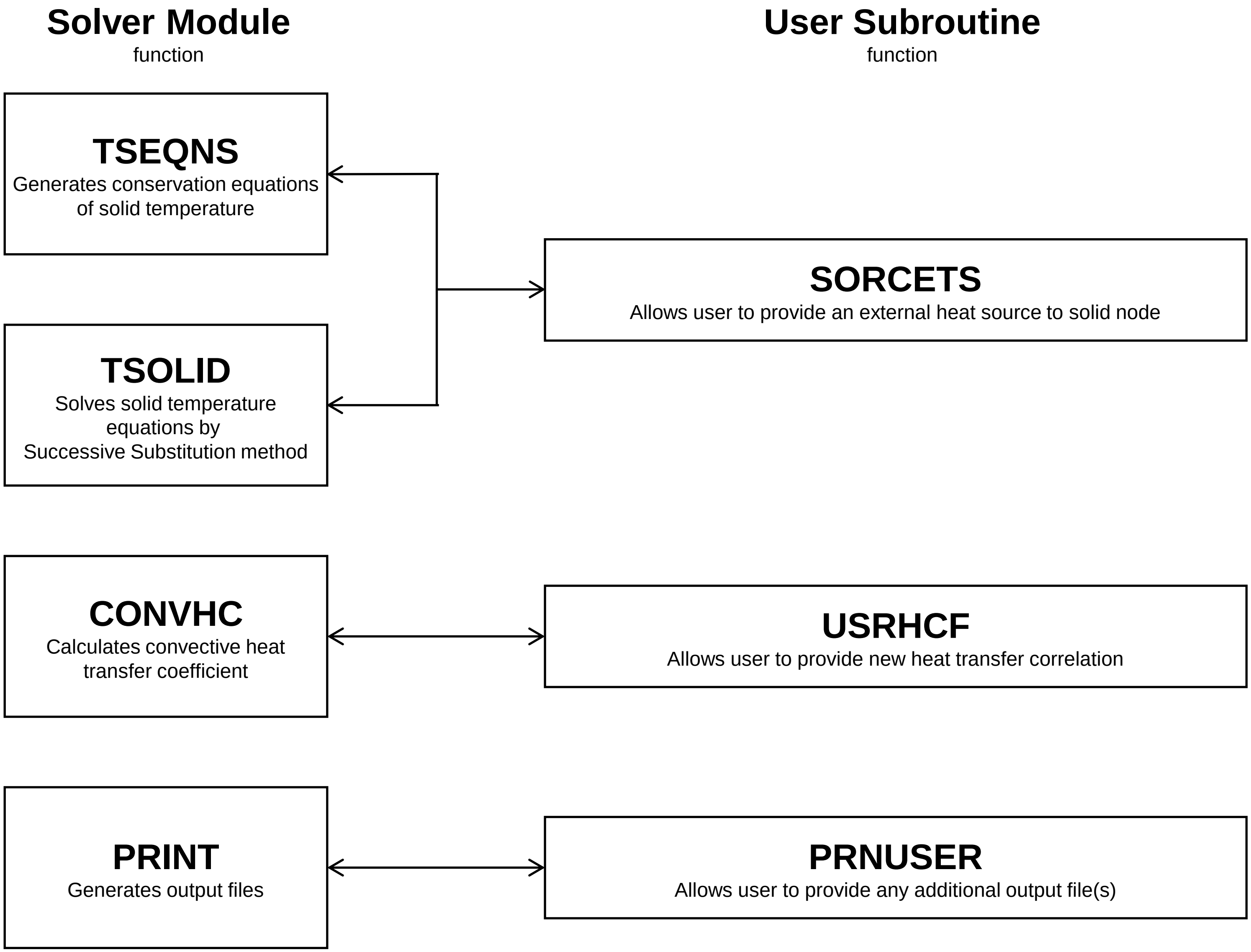
RESIST
Calculates the flow resistance coefficient

KFUSER
Allows user to define new resistance options

DENSITY
Calculates the fluid properties

PRPUSER
Allows user to define new fluid properties

Solver-User Subroutine Interaction (3/3)



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 an external heat source to solid node

CONVHC
Calculates convective heat transfer coefficient

USRHCF
Allows user to provide new heat transfer correlation

PRINT
Generates output files

PRNUSER
Allows user to provide any additional output file(s)

Indexing Subroutine

- **SUBROUTINE INDEXI** determines the pointer to a Node or Branch

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

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)

SUBROUTINE INDEXI Usage

Node Number	100	200	300	400	500
IPN	1	2	3	4	5
P	5125.5	4785.23	3876.45	2557.85	1668.25
TF	560.0	555.25	525.34	500.25	480.0

Example:
Address location of Node Number 400

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

-OR-

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

In this example:

IPN = 4
P(IPN) = 2557.85
TF(IPN) = 500.25

Indexing Subroutines

- **SUBROUTINE INDEXA (NUMBER, NODEAM, NAMB, IPAN)**
  - Determines the pointer of Ambient Node

- **SUBROUTINE INDEXS (NUMBER, NODESL, NSOLIDX, IPSN)**
  - Determines the pointer of Solid Node

- **SUBROUTINE INDEXSSC (NUMBER, ICONSS, NSSC, ICSS)**
  - Determines the pointer of Solid to Solid Conductor

- **SUBROUTINE INDEXSFC (NUMBER, ICONSF, NSFC, ICSF)**
  - Determines the pointer of Solid to Fluid Conductor

- **SUBROUTINE INDEXSAC (NUMBER, ICONSA, NSAC, ICSA)**
  - Determines the pointer of Solid to Ambient Conductor

- **SUBROUTINE INDEXSSRC (NUMBER, ICONSSR, NSSR, ICSSR)**
  - Determines the pointer of Solid to Solid Radiation Conductor

- CALL statements to indexing subroutines can be inserted into the code by right-clicking in the MIG Fortran editor. No need to memorize these statements!

Utility Subroutines and Functions

- **SUBROUTINE INTERPOL**
  - Linearly interpolates a **YVALUE** given an **XVALUE** and **XY data**
  - Does not extrapolate; returns the first or last y-value, as needed

SUBROUTINE INTERPOL (XVALUE, N, XARRAY, YARRAY, YVALUE)

Input Variables:

XVALUE: x value at which to interpolate y
N: number of points in **XARRAY** and **YARRAY**
XARRAY: array of x values, in increasing order
YARRAY: array of y values corresponding to **XARRAY**

Output Variable:

YVALUE: Interpolated y value

- Functions to convert units are listed in Appendix 5 of the User Manual
  - Example

REAL FUNCTION KW_BTUS (VALUE)
Converts VALUE (kW) to (BTU/s)

Fluid Property Subroutines (1/5)

- **SUBROUTINE PROPS_PT**

```
 CALL PROPS_PT(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H,
+               Z_CP, Z_CV, Z_S, Z_GAMMA, Z_MU
+               Z_K, I_KR, Z_XV)
```

  - Input
    - I_NFLUID: Fluid ID code (see next slide)
    - Z_P: Pressure
    - Z_T: Temperature
  - Output
    - Z_RHO: Density
    - Z_H: Enthalpy
    - Z_CP: Specific heat (constant pressure)
    - Z_CV: Specific heat (constant volume)
    - Z_S: Entropy
    - Z_GAMMA: Ratio of specific heats
    - Z_MU: Viscosity
    - Z_K: Thermal conductivity
    - I_KR: Fluid phase code (0 unknown; 1 saturated; 2 liquid; 3 gas)
    - Z_XV: Quality (vapor mass fraction)
    - If fluid P/T is exactly saturated, Z_RHO = 0.0. Users are encouraged to include an IF statement to check for this condition after each call.

Fluid Property Subroutines (2/5)

ID Number	SOURCE / FLUID	ID Number	SOURCE / FLUID	ID Number	SOURCE / FLUID
1	GASP He	51	GASPAK He	69	GASPAK Kr
2	GASP CH_4	52	GASPAK CH_4	70	GASPAK Propane
3	GASP Ne	53	GASPAK Ne	71	GASPAK Xe
4	GASP N_2	54	GASPAK N_2	72	GASPAK R-11
5	GASP CO	55	GASPAK CO	73	GASPAK R-12
6	GASP O_2	56	GASPAK O_2	74	GASPAK R-22
7	GASP Ar	57	GASPAK Ar	75	GASPAK R-32
8	GASP CO_2	58	GASPAK CO_2	76	GASPAK R-123
9	GASP F_2	59	GASPAK H_2 (para)	77	GASPAK R-124
10	GASP H_2 (para)	60	GASPAK H_2 (normal)	78	GASPAK R-125
11	WASP H_2O	61	GASPAK H_2O	79	GASPAK R-134A
12	RP-1 Tables	62	GASPAK RP-1 (liq)	80	GASPAK R-152A
		63	GASPAK Isobutane	81	GASPAK N_2F_3
33	Ideal Gas	64	GASPAK Butane	82	GASPAK NH_3
		65	GASPAK Deuterium	84	GASPAK H_2O_2
37	User Fluid 1	66	GASPAK Ethane	86	GASPAK Air
38	User Fluid 2	67	GASPAK Ethylene		
39	User Fluid 3	68	GASPAK H_2S		

Fluid Property Subroutines (3/5)

Property	English Units
Pressure (P)	Psf
Temperature (T)	°R
Conductivity (k)	BTU/ft-s-R
Density (r)	lb/ft^3
Viscosity (μ)	lb/ft-s
Specific Heat Ratio (γ)	Dimensionless
Enthalpy (H)	BTU/lb
Entropy (S)	BTU/lb-R
Specific Heat (Cp)	BTU/lb-R
Specific Heat (Cv)	BTU/lb-R

Fluid Property Subroutines (4/5)

- **SUBROUTINE PROPS_PH, SUBROUTINE PROPS_PS**

```
 CALL PROPS_PH(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H, Z_CP, Z_CV,
+        Z_S, Z_GAMMA, Z_MU, Z_K, I_KR, Z_XV,
+        Z_RHOL, Z_HL, Z_CPL, Z_CVL, Z_SL, Z_GAMMAL, Z_MUL, Z_KL,
+        Z_RHOV, Z_HV, Z_CPV, Z_CVV, Z_SV, Z_GAMMAV, Z_MUV, Z_KV)

 CALL PROPS_PS(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H, Z_CP, Z_CV,
+        Z_S, Z_GAMMA, Z_MU, Z_K, I_KR, Z_XV,
+        Z_RHOL, Z_HL, Z_CPL, Z_CVL, Z_SL, Z_GAMMAL, Z_MUL, Z_KL,
+        Z_RHOV, Z_HV, Z_CPV, Z_CVV, Z_SV, Z_GAMMAV, Z_MUV, Z_KV)
```

- Input
  - I_NFLUID: Fluid ID code
  - Z_P: Pressure
  - Z_H or Z_S: Enthalpy or Entropy

- Output
  - Similar to PROPS_PT
  - If the fluid is saturated (I_KR = 1)
    - Properties suffixed in “L” or “V” are the properties of the pure liquid or vapor
    - Other properties are quality-weighted averages of the two-phase mixture

Fluid Property Subroutines (5/5)

- **SUBROUTINE PROPS_PSATX, PROPS_TSATX**

```
CALL PROPS_PSATX(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H, Z_CP, Z_CV,
    +        Z_S, Z_GAMMA, Z_MU, Z_K, I_KR, Z_XV,
    +        Z_RHOL, Z_HL, Z_CPL, Z_CVL, Z_SL, Z_GAMMAL, Z_MUL, Z_KL,
    +        Z_RHOV, Z_HV, Z_CPV, Z_CVV, Z_SV, Z_GAMMAV, Z_MUV, Z_KV)
CALL PROPS_TSATX(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H, Z_CP, Z_CV,
    +        Z_S, Z_GAMMA, Z_MU, Z_K, I_KR, Z_XV,
    +        Z_RHOL, Z_HL, Z_CPL, Z_CVL, Z_SL, Z_GAMMAL, Z_MUL, Z_KL,
    +        Z_RHOV, Z_HV, Z_CPV, Z_CVV, Z_SV, Z_GAMMAV, Z_MUV, Z_KV)
```

  - Input
    - ➢ I_NFLUID: Fluid ID code
    - ➢ Z_P or Z_T: Saturation pressure or saturation temperature
    - ➢ Z_XV: Quality

  - Output
    - ➢ Z_T or Z_P: Saturation temperature or pressure

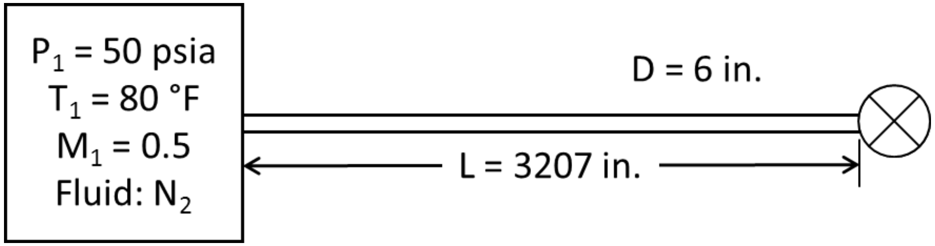
  - Subroutines for saturation properties
    - ➢ For GASP/WASP and GASPAK fluids only
    - ➢ If the input is greater than P_{crit} or T_{crit}, then Z_RHO = 0.0.

User Subroutine Applications

- Example 18 - Simulation of a Subsonic Fanno Flow
- User-Prescribed Heat Transfer Coefficient
- Thermostatically Controlled Heater
- Fixing the Temperature of an Internal Node
- User-Defined Branch Resistance
- User-Defined Plot Variables

Ex18: Simulation of a Subsonic Fanno Flow (1/4)

- Problem:



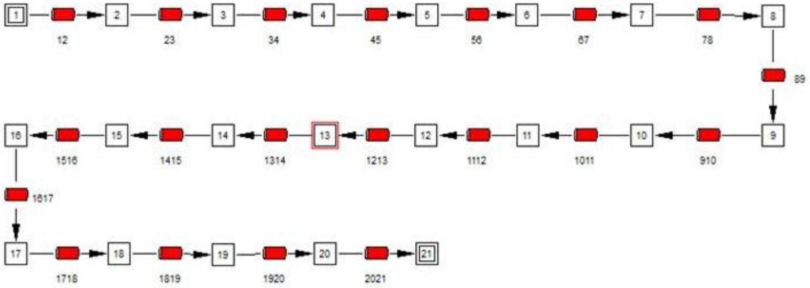
- To compare with textbook solution, friction factor must be constant
  - **GFSSP** always solves for friction factor based on Reynolds number

- Solution:
  - Use subroutine KFADJUST to recalculate pipe resistance factor K_f (assuming a constant friction factor)

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

Ex18: Simulation of a Subsonic Fanno Flow (2/4)

- **MIG** Schematic



- User Subroutine **KFADJUST**

```
C***********************************************************************
      SUBROUTINE KFADJUST(I,RHOU,EMUU,RHOUL,EMUUL,RHOUV,EMUUV,ISATU,
     &                    AKNEW)
C     PURPOSE: ADJUST RESISTANCE IN A BRANCH
C***********************************************************************
      USE GFSSP_GLOBAL
C**********************************************************************
C     ADD CODE HERE
      IF (IOPT(I) .EQ. 1) THEN
         PIPEL = BRPR1(I)
         PIPED = BRPR2(I)
         F = 0.002
         AKNEW = 8.0 * F * PIPEL / (RHOU * PI**2 * PIPED**5 * GC)
      END IF
      RETURN
      END
```

Ex18: Simulation of a Subsonic Fanno Flow (3/4)

```
C***********************************************************************
      SUBROUTINE KFADJUST(I,RHOU,EMUU,RHOUL,EMUUL,RHOUV,EMUUV,ISATU,
     &                    AKNEW)
```

Argument I is the pointer to the current branch.
Output AKNEW is the new value of Kf.

```
C     PURPOSE: ADJUST RESISTANCE IN A BRANCH
C***********************************************************************
      USE GFSSP_GLOBAL
```

Using the GFSSP_GLOBAL module gives us access to all the program variables listed in Appendix 4 of the User Manual.

```
C**********************************************************************
C     ADD CODE HERE
      IF (IOPT(I) .EQ. 1) THEN
```

If the current branch option is a pipe (option 1), then…

```
         PIPEL = BRPR1(I)
         PIPED = BRPR2(I)
```

Pipe length and diameter are stored in branch parameter arrays.

```
         F = 0.002
```

Set constant friction factor of 0.002

Recalculate KF for this branch

```
         AKNEW = 8.0 * F * PIPEL / (RHOU * PI**2 * PIPED**5 * GC)

      END IF
      RETURN
      END
```

Note that PI and GC are already program constants.
Upstream density was passed as argument RHOU.

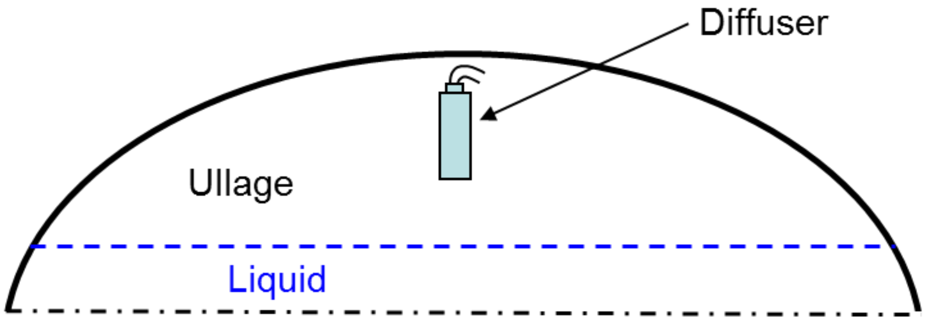
Ex18: Simulation of a Subsonic Fanno Flow (4/4)

- Branch Parameter Arrays
  - Users have access to branch parameters through BRPR arrays
  - Full table found in Chapter 4 of User Manual

Branch Option	BRPR1	BRPR2	BRPR3	BRPR4	BRPR5	BRPR6
1. Pipe	Length	Diameter	ε/D			
2. Restriction	C_L					
3. Non-Circular Duct	Length	Height	Width	Type (1-4)		
4. Pipe with Entrance and Exit Losses	Length	Diameter	ε/D	K_i	K_e	
5. Thin Sharp Orifice	D_1	D_2				
6. Thick Orifice	Length	D_1	D_2			
7. Square Reduction	D_1	D_2				
8. Square Expansion	D_1	D_2				
9. Rotating Annular Duct	Length	r_o	r_i	RPM		
10. Rotating Radial Duct	Length	Diameter	RPM			
11. Labyrinth Seal	Radius	Clearance, c	Pitch, m	Number of teeth, n	Multiplier, α	
12. Parallel Plates (Face Seal)	Radius	Clearance, c	Length			

User-Prescribed Heat Transfer Coefficient

- Problem: Heat transfer coefficient between ullage gas and tank dome is not a constant value.
  - Must be evaluated by natural convection correlations based on ullage properties



- Solution: Use subroutine **USRHCF** to calculate heat transfer coefficient

$$Nu = 0.15(Gr\,\mathrm{Pr})^{0.33} \qquad Nu = \frac{hL}{k}$$

$$Gr = \frac{L^3 \rho^2 g \beta \Delta T}{\mu^2} \qquad \mathrm{Pr} = \frac{C_p \mu}{k}$$

User-Prescribed Heat Transfer Coefficient

```
C**********************************************************
      SUBROUTINE USRHCF(NUMBER,HCF)
```

Argument NUMBER is the pointer to the current solid-to-fluid conductor. Output HCF is the heat transfer coefficient.

```
C     PURPOSE: PROVIDE HEAT TRANSFER COEFFICIENT
C**********************************************************

      USE GFSSP_GLOBAL
```

Using the GFSSP_GLOBAL module gives us access to program variables such as the node properties used below.

```
C**********************************************************
      DATA HL /15.0/
      DATA C1, C2 /0.15, 0.33/
```

Set characteristic length HL and correlation constants C1 and C2

```
      NUMF = ICF(NUMBER)
      CALL INDEXI(NUMF, NODE, NNODES, IPN)
      NUMS = ICS(NUMBER)
      CALL INDEXS(NUMS, NODESL, NSOLIDX, IPSN)
```

Get fluid and solid node numbers (NUMF, NUMS) and their pointers (IPN, IPSN)

```
      BETA = 1.0 / 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)
      XNU = C1 * (GR * PRNDTL)**C2
      HCF = XNU * CONDF(IPN) / HL
      RETURN
      END
C**********************************************************
```

Calculate heat transfer coefficient HCF

Having the fluid and solid node pointers (IPN, IPSN) allows us to access fluid and solid node properties such as temperature (TF, TS), density (RHO), viscosity (EMU), Cp (CPNODE), and k (CONDF).

Thermostatically Controlled Heater (1/2)

- Problem: Simulate a thermostatically controlled heat source in a fluid node
  - **GFSSP**'s built-in heat source options are constant-value or time-varying, but not temperature-varying

- Solution: Use subroutine BNDUSER
  - Apply a heat source to a node based on its temperature

Thermostatically Controlled Heater (2/2)

```
      SUBROUTINE BNDUSER
C     PURPOSE: MODIFY BOUNDARY CONDITIONS


C***********************************************************************
      USE GFSSP_GLOBAL
C**********************************************************************
C     ADD CODE HERE

C   Declarations

      DATA HEATPOWER /1.0/, THEATOFF /120.0/, THEATON /110.0/

C   In every time step, check the temperature of the heated node
C   and apply heat if temperature is less than THEATON, or
C   set heat to zero if temperature is above THEATOFF.

      CALL INDEXI(5, NODE, NNODES, IPN)

      TFAHRENHEIT = TF(IPN) - 459.67

      IF (TFAHRENHEIT .LT. THEATON) THEN
         HSORCE(IPN) = HEATPOWER
      ELSE IF (TFAHRENHEIT .GT. THEATOFF) THEN
         HSORCE(IPN) = 0.0
      END IF

      RETURN
      END
```

Declare a 1.0 BTU/s heater with temperature limits of 110 and 120 °F.

Get pointer to Node 5

Convert temperature of Node 5 from °R to °F.

If temperature is too low, apply a heat source. If too high, set heat source to zero. (At temperatures in between, HSORCE will retain its value from the previous time step.)

Fixing the Temperature of an Internal Node (1/2)

- Problem: A user has a large system model that includes a heat exchanger. The exit temperature of the heat exchanger is expected to be 400°F. The user doesn't want to model the heat exchanger in detail.

- Solution: Use subroutine SORCEQ to apply a large imaginary flow at 400°F into and out of a node.

$$h_{node} = \frac{\sum \dot{m}_{in} h_{in} + Q}{\sum \dot{m}_{out} + \text{TERMD}}$$

- What if Q were 10^{30} lb/s of flow with enthalpy corresponding to 400°F, and TERMD were 10^{30} lb/s of flow out of the node?

Fixing the Temperature of an Internal Node (2/2)

```
      SUBROUTINE SORCEQ(IPN,TERMD)
C*************************************************************************
      USE GFSSP_GLOBAL
C************************************************************************
C ADD CODE HERE

      IF (NODE(IPN) .EQ. 9) THEN      ! If it's node 9

         I_NFLUID = 1        ! ID number of helium
         Z_P = P(IPN)        ! Current pressure of node, psf
         Z_T = 400.0 + 459.67        ! Desired constant temperature, deg R
         CALL PROPS_PT(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H,
     +                 Z_CP, Z_CV, Z_S, Z_GAMMA, Z_MU,
     +                 Z_K, I_KR, Z_XV)
         SORCEH(IPN) = 1.0E30 * Z_H      ! Imaginary inlet flow
         TERMD = 1.0E30                  ! Imaginary outlet flow
      END IF

      RETURN
      END
```

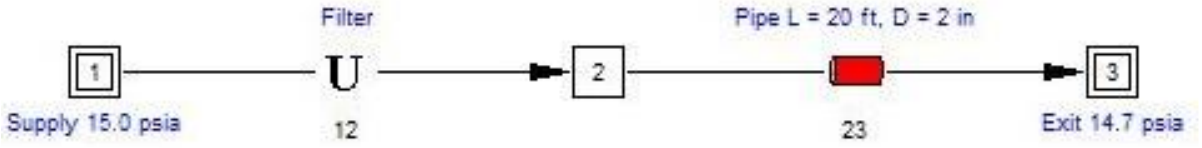
Subroutine **SORCEQ** is called by the energy equation for each internal node pointed to by **IPN**.

Heat source **SORCEH(IPN)** can be a simple external source (BTU/s), or an imaginary flow rate (lb/s) multiplied by a specific enthalpy (BTU/lb).

Optional **TERMD** represents an imaginary mass flow out of the node (lb/s)

User-Defined Branch Resistance (1/3)

- Problem: The user wishes to model pressure drop in a filter. Test data relating pressure drop to flow rate are available.



- Solution: Use the User-Defined Branch Option in MIG along with subroutine KFUSER

User-Defined Branch Resistance (2/3)

- Test Data and K_f

Flow Rate (lb_m/s)	ΔP (lb_f/ft^2)	K_f
1.0	10	10
2.0	20	5
3.0	30	3.33
4.0	40	2.5

- **GFSSP**’s momentum equation expresses friction losses in terms of flow rate

$$\Delta P = K_f \dot{m}^2$$

User-Defined Branch Resistance (3/3)

```
C***********************************************************************
      SUBROUTINE KFUSER(I,RHOU,EMUU,XVU,RHOUL,EMUUL,AKNEW)
C     PURPOSE: ADD A NEW RESISTANCE OPTION
C***********************************************************************
      USE GFSSP_GLOBAL
C**********************************************************************
C     ADD CODE HERE

C  Declarations

      REAL FILTERMDOT(4), FILTERKF(4)
      DATA FILTERMDOT /1.0, 2.0, 3.0, 4.0/
      DATA FILTERKF /10.0, 5.0, 3.33, 2.5/
```

Filter data (Kf vs. flow rate) stored in arrays.

```
C  Executable code.

      FILTERFLOW = FLOWR(I)
```

Get the flow rate in this user-defined branch in the current iteration.

```
      CALL INTERPOL(FILTERFLOW, 4, FILTERMDOT, FILTERKF, AKNEW)
```

Interpolate KF for this branch

```
      RETURN
      END
```

User-Defined Plot Variables (1/2)

- Problem: User wants to plot pressurization option heat transfer rates in Winplot.

- Solution:
  - Set up User-Defined Plot Variables
  - Sends extra variables to the Winplot file

User-Defined Plot Variables (2/2)

```
C*********************************************************************
      SUBROUTINE BNDUSER
C     PURPOSE: MODIFY BOUNDARY CONDITIONS
C*********************************************************************
      USE GFSSP_GLOBAL
C********************************************************************
C     ADD CODE HERE

C   Turn on user-variables to send to Winplot

      USRVAR = .TRUE.
      USRVARSNUM = 2
      USRPVARNAME(1) = 'QULPRP'
      USRPVARUNIT(1) = 'BTU/s'
      USRPVARNAME(2) = 'QULWAL'
      USRPVARUNIT(2) = 'BTU/s'

C   Copy data from pressurization option to user variable

      USRPVAR(1) = QULPRP(1)
      USRPVAR(2) = QULWAL(1)

      RETURN
      END
```

Declare two user variables and set their Winplot names and units

Copy the values of the pressurization option heat transfer rates to the user variables.

Summary

- User Subroutines
    - Adds 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 time step, etc.

- Checklist for User Subroutines
    - Identify subroutines that require modifications
    - Select **GFSSP** variables to be modified
    - Make use of **GFSSP** provided User Variables in your coding

Demo 3: A Deflating Bicycle Tire

- Source: Introduction to Fluid Mechanics, 4th ed., by Fox and McDonald, Problem 13.39

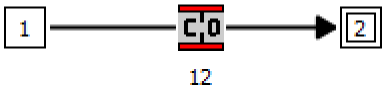
- Given: Air escapes from a high-pressure bicycle tire through a hole with diameter d = 0.254 mm. The initial pressure in the tire is P = 620 kPa (gage). Assume that the temperature remains constant at 27 °C. The internal volume of the tire is approximately 4.26×10^{-4} m^3 and is constant.

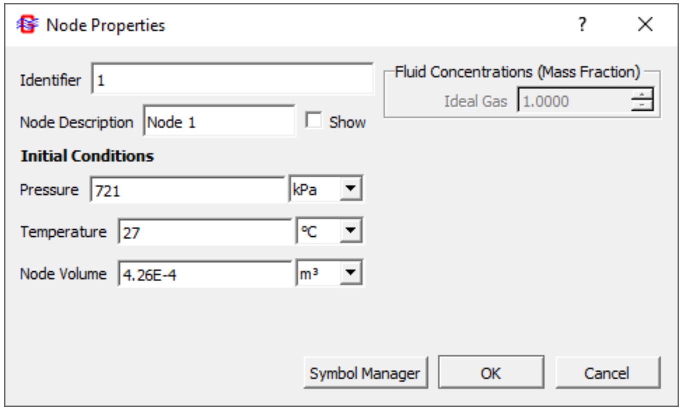
- Estimate: Time needed for the pressure in the tire to drop to 310 kPa (gage)

- Compute: Change in specific entropy of the air in the tire during this process

Demo 3: Build Model on Canvas

- Exact method: Write a short user subroutine to fix the temperature of Node 1.




Node Properties

Identifier: 1

Node Description: Node 1 ☐ Show

Fluid Concentrations (Mass Fraction)

Ideal Gas: 1.0000

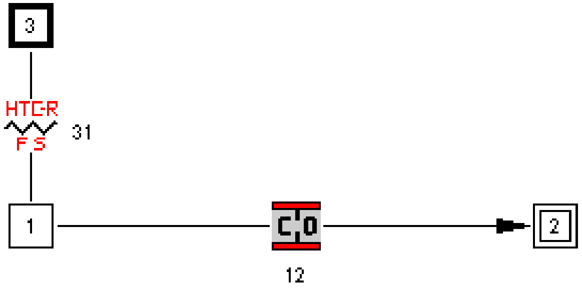
Initial Conditions

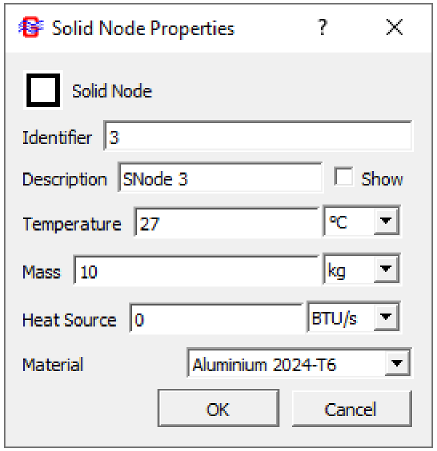
Pressure: 721 kPa

Temperature: 27 °C

Node Volume: 4.26E-4 m³

- Approximate method: Activate Conjugate Heat Transfer and connect Internal Node to Solid Node with large mass and initial temperature of 27 C.




Solid Node Properties

Solid Node

Identifier: 3

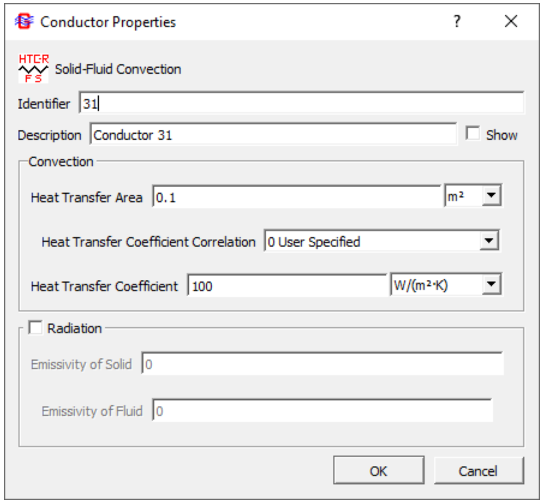
Description: SNode 3 ☐ Show

Temperature: 27 °C

Mass: 10 kg

Heat Source: 0 BTU/s

Material: Aluminium 2024-T6


Conductor Properties

Solid-Fluid Convection

Identifier: 31

Description: Conductor 31 ☐ Show

Convection

Heat Transfer Area: 0.1 m²

Heat Transfer Coefficient Correlation: 0 User Specified

Heat Transfer Coefficient: 100 W/(m²·K)

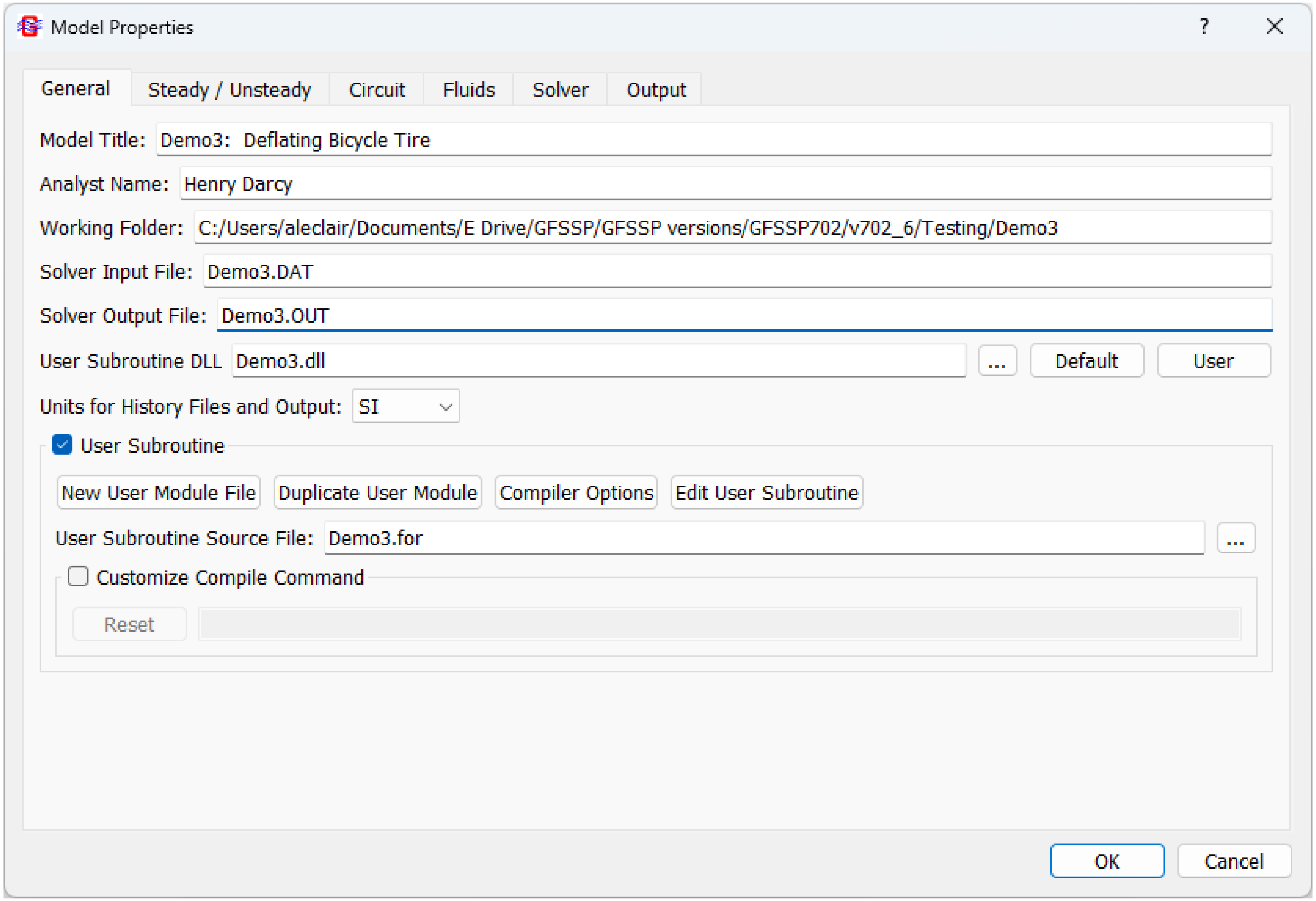
☐ Radiation

Emissivity of Solid: 0

Emissivity of Fluid: 0

Demo 3: Set Up User Subroutine

- Set up user subroutine (Demo3.for or Demo3.f90)



Model Title: Demo3: Deflating Bicycle Tire

Analyst Name: Henry Darcy

Working Folder: C:/Users/aleclair/Documents/E Drive/GFSSP/GFSSP versions/GFSSP702/v702_6/Testing/Demo3

Solver Input File: Demo3.DAT

Solver Output File: Demo3.OUT

User Subroutine DLL Demo3.dll

Units for History Files and Output: SI

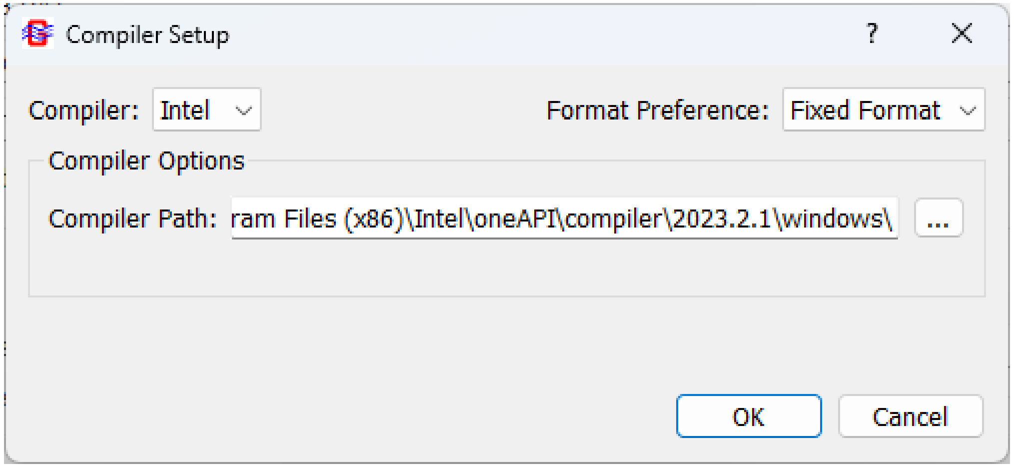
☑ User Subroutine

User Subroutine Source File: Demo3.for

☐ Customize Compile Command

Demo 3: Point MIG to Intel Compiler

- Click Compiler Options on the General tab of Model Properties
- Select Intel compiler.
- Depending on your version of Intel Fortran, your Compiler Path may look different from that shown below.



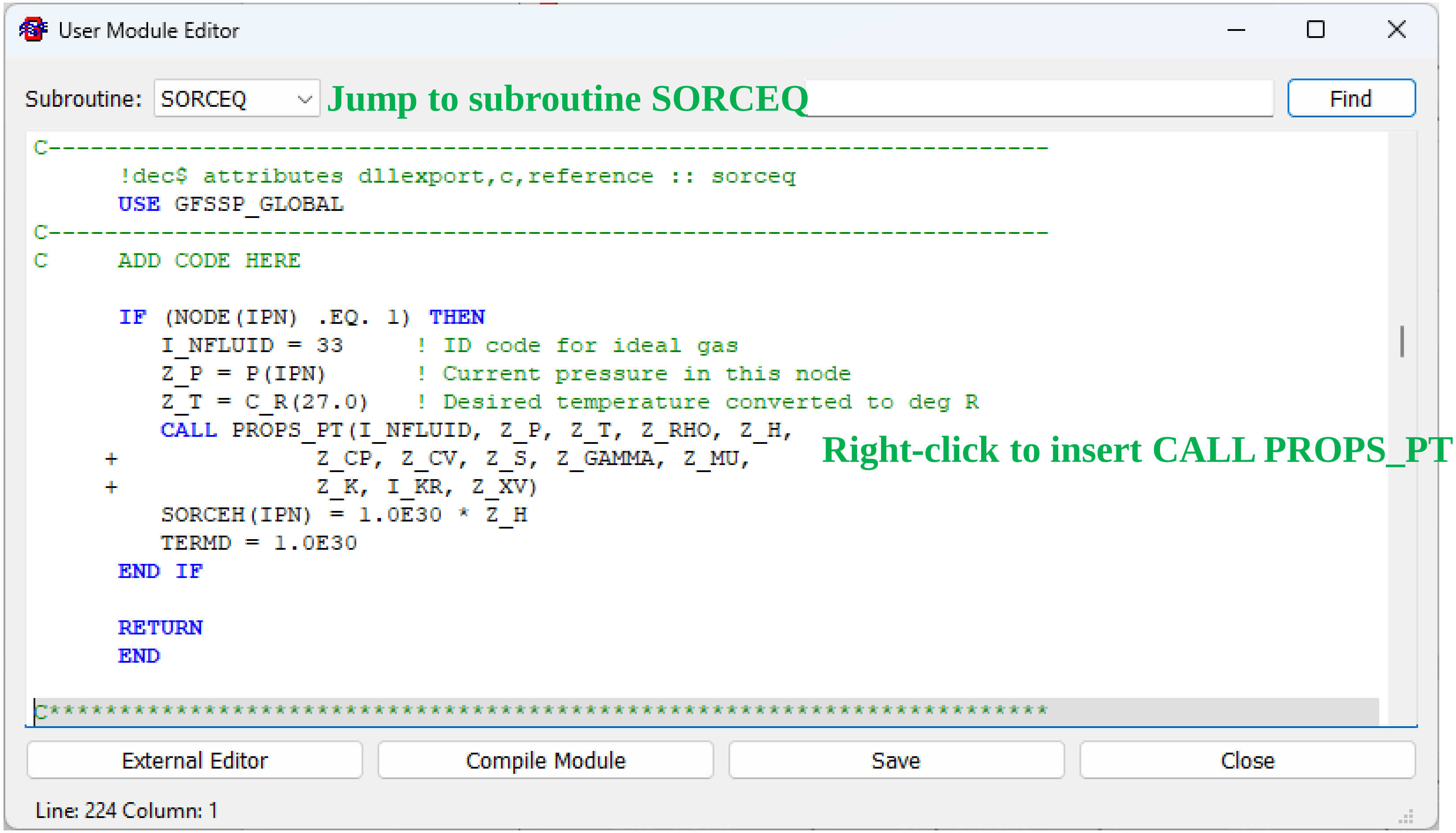
Compiler: Intel

Format Preference: Fixed Format

Compiler Options

Compiler Path: ram Files (x86)\Intel\oneAPI\compiler\2023.2.1\windows\

Demo 3: Edit User Subroutine



Subroutine: SORCEQ

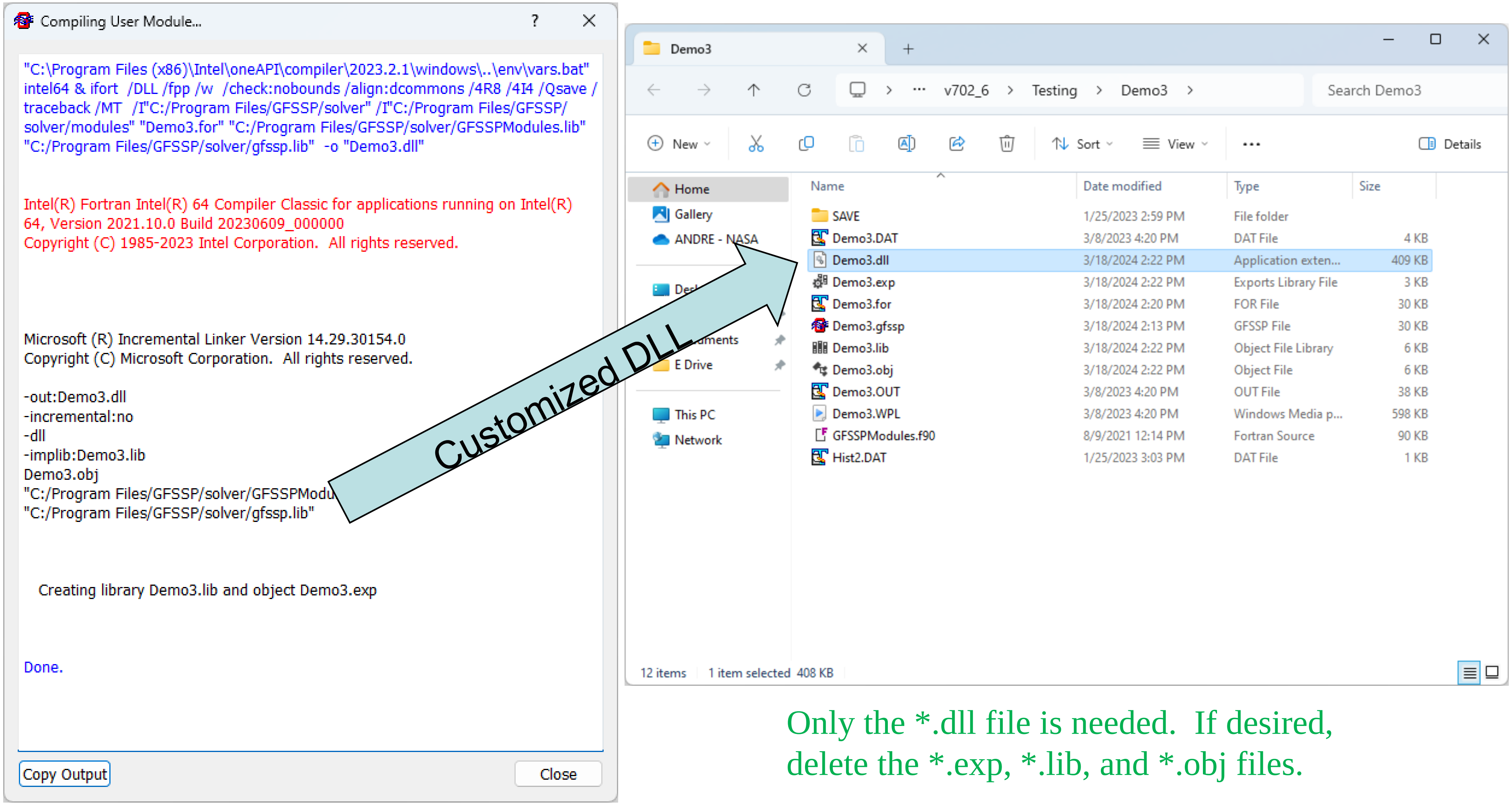
Jump to subroutine SORCEQ

```
C-----------------------------------------------------------------------
      !dec$ attributes dllexport,c,reference :: sorceq
      USE GFSSP_GLOBAL
C-----------------------------------------------------------------------
C     ADD CODE HERE

      IF (NODE(IPN) .EQ. 1) THEN
         I_NFLUID = 33      ! ID code for ideal gas
         Z_P = P(IPN)       ! Current pressure in this node
         Z_T = C_R(27.0)    ! Desired temperature converted to deg R
         CALL PROPS_PT(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H,
     +              Z_CP, Z_CV, Z_S, Z_GAMMA, Z_MU,
     +              Z_K, I_KR, Z_XV)
         SORCEH(IPN) = 1.0E30 * Z_H
         TERMD = 1.0E30
      END IF

      RETURN
      END

C***********************************************************************
```

Right-click to insert CALL PROPS_PT

Line: 224 Column: 1

Demo 3: Compile User Subroutine



```
"C:\Program Files (x86)\Intel\oneAPI\compiler\2023.2.1\windows\..\env\vars.bat"
intel64 & ifort  /DLL /fpp /w  /check:nobounds /align:dcommons /4R8 /4I4 /Qsave /
traceback /MT  /I"C:/Program Files/GFSSP/solver" /I"C:/Program Files/GFSSP/
solver/modules" "Demo3.for" "C:/Program Files/GFSSP/solver/GFSSPModules.lib"
"C:/Program Files/GFSSP/solver/gfssp.lib"  -o "Demo3.dll"


Intel(R) Fortran Intel(R) 64 Compiler Classic for applications running on Intel(R)
64, Version 2021.10.0 Build 20230609_000000
Copyright (C) 1985-2023 Intel Corporation.  All rights reserved.



Microsoft (R) Incremental Linker Version 14.29.30154.0
Copyright (C) Microsoft Corporation.  All rights reserved.

-out:Demo3.dll
-incremental:no
-dll
-implib:Demo3.lib
Demo3.obj
"C:/Program Files/GFSSP/solver/GFSSPModu
"C:/Program Files/GFSSP/solver/gfssp.lib"


   Creating library Demo3.lib and object Demo3.exp


Done.
```

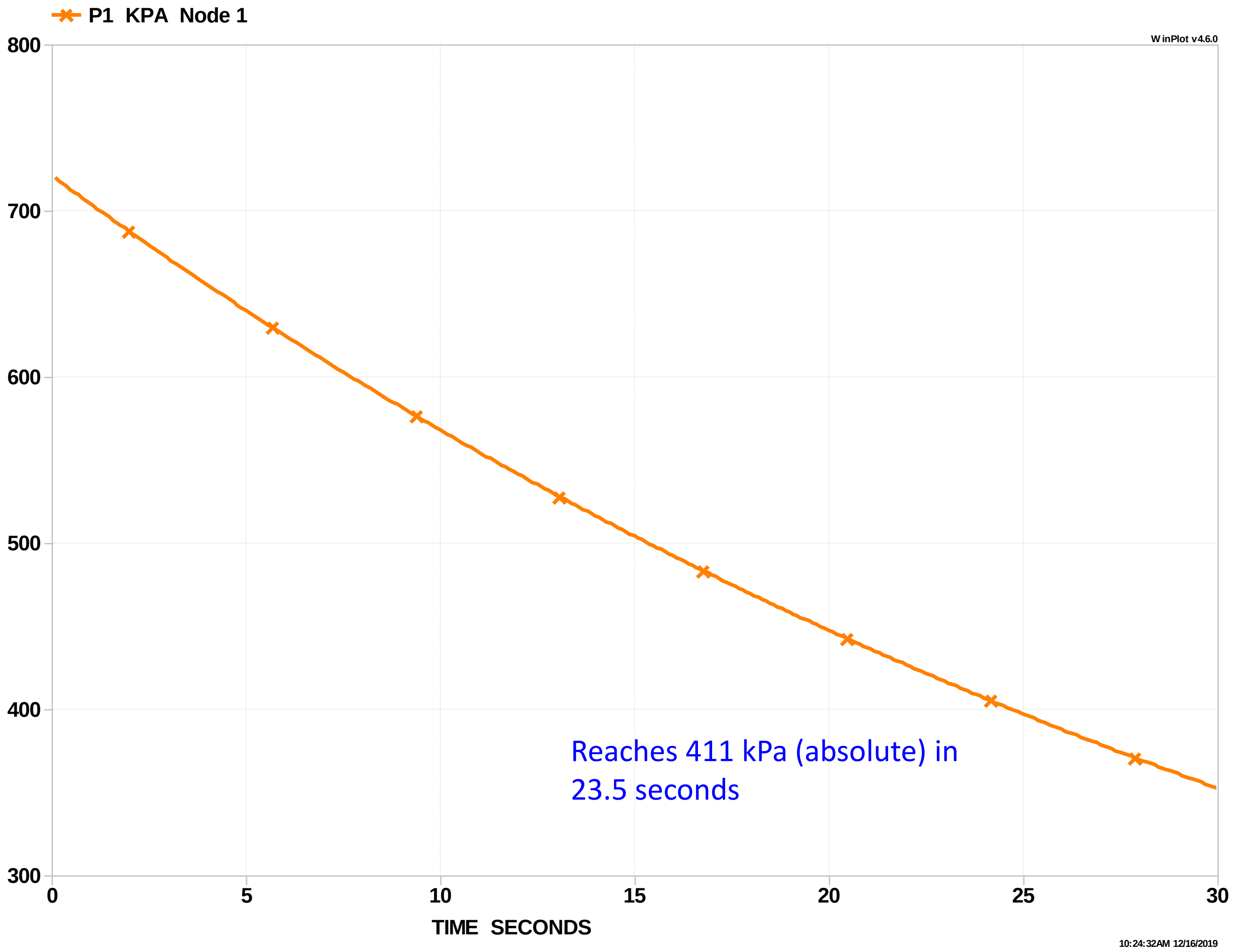
Name	Date modified	Type	Size
SAVE	1/25/2023 2:59 PM	File folder	
Demo3.DAT	3/8/2023 4:20 PM	DAT File	4 KB
Demo3.dll	3/18/2024 2:22 PM	Application exten...	409 KB
Demo3.exp	3/18/2024 2:22 PM	Exports Library File	3 KB
Demo3.for	3/18/2024 2:20 PM	FOR File	30 KB
Demo3.gfssp	3/18/2024 2:13 PM	GFSSP File	30 KB
Demo3.lib	3/18/2024 2:22 PM	Object File Library	6 KB
Demo3.obj	3/18/2024 2:22 PM	Object File	6 KB
Demo3.OUT	3/8/2023 4:20 PM	OUT File	38 KB
Demo3.WPL	3/8/2023 4:20 PM	Windows Media p...	598 KB
GFSSPModules.f90	8/9/2021 12:14 PM	Fortran Source	90 KB
Hist2.DAT	1/25/2023 3:03 PM	DAT File	1 KB

12 items   1 item selected 408 KB

Only the *.dll file is needed. If desired, delete the *.exp, *.lib, and *.obj files.

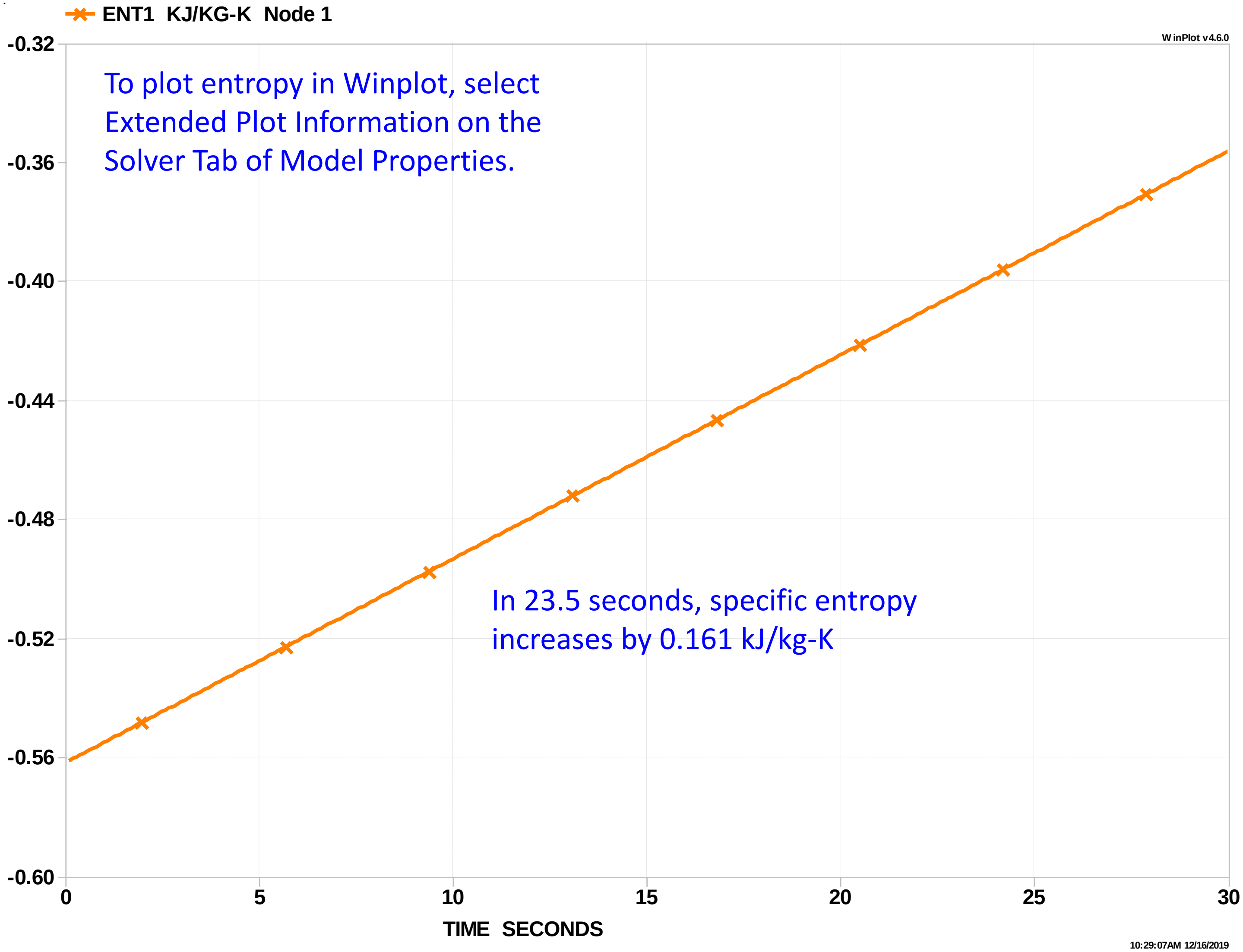
Demo 3: Plot Pressure

Marshall Space Flight Center
GFSSP Training Course



Reaches 411 kPa (absolute) in 23.5 seconds

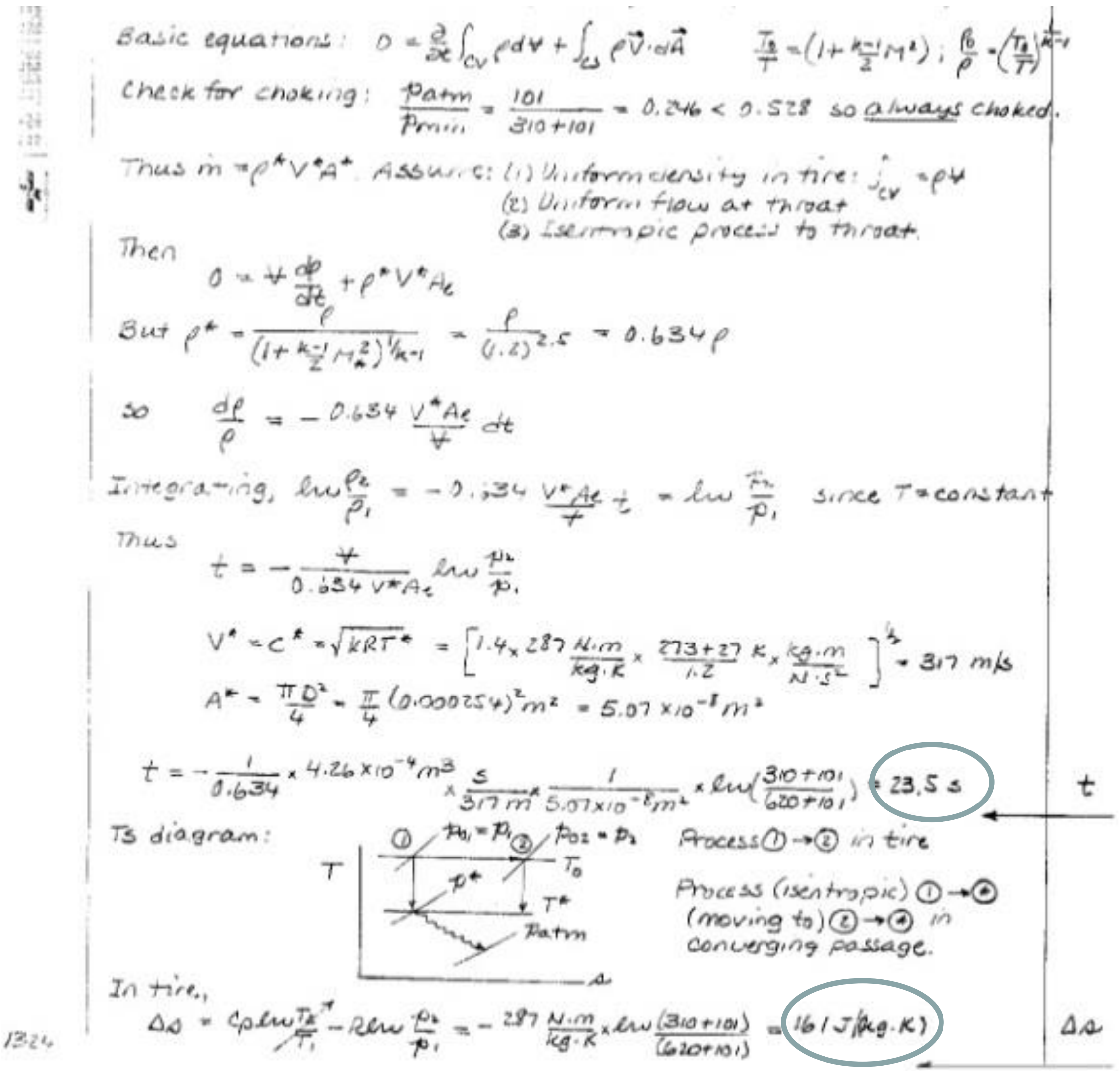
Demo 3: Plot Specific Entropy



To plot entropy in Winplot, select Extended Plot Information on the Solver Tab of Model Properties.

In 23.5 seconds, specific entropy increases by 0.161 kJ/kg-K

Demo 3: Hand Calculations



Basic equations: $0 = \frac{\partial}{\partial t}\int_{CV} \rho \, d\forall + \int_{CS} \rho \vec{V} \cdot d\vec{A}$ $\qquad \frac{T_0}{T} = \left(1 + \frac{k-1}{2} M^2\right); \ \frac{\rho_0}{\rho} = \left(\frac{T_0}{T}\right)^{\frac{1}{k-1}}$

Check for choking: $\frac{p_{atm}}{p_{min}} = \frac{101}{310+101} = 0.246 < 0.528$ so always choked.

Thus $\dot{m} = \rho^* V^* A^*$. Assume: (1) Uniform density in tire: $\int_{CV} = \rho \forall$

(2) Uniform flow at throat

(3) Isentropic process to throat.

Then

$$0 = \forall \frac{d\rho}{dt} + \rho^* V^* A_e$$

But $\rho^* = \frac{\rho}{\left(1 + \frac{k-1}{2} M_*^2\right)^{1/k-1}} = \frac{\rho}{(1.2)^{2.5}} = 0.634 \rho$

so $\frac{d\rho}{\rho} = -0.634 \frac{V^* A_e}{\forall} dt$

Integrating, $\ln \frac{\rho_2}{\rho_1} = -0.634 \frac{V^* A_e}{\forall} t = \ln \frac{p_2}{p_1}$ since T = constant

Thus

$$t = -\frac{\forall}{0.634 V^* A_e} \ln \frac{p_2}{p_1}$$

$$V^* = c^* = \sqrt{kRT^*} = \left[1.4 \times 287 \frac{N \cdot m}{kg \cdot K} \times \frac{273+27}{1.2} K \times \frac{kg \cdot m}{N \cdot s^2}\right]^{\frac{1}{2}} = 317 \ m/s$$

$$A^* = \frac{\pi D^2}{4} = \frac{\pi}{4}(0.000254)^2 m^2 = 5.07 \times 10^{-8} m^2$$

$$t = -\frac{1}{0.634} \times 4.26 \times 10^{-4} m^3 \times \frac{s}{317 \ m} \times \frac{1}{5.07 \times 10^{-8} m^2} \times \ln\left(\frac{310+101}{620+101}\right) = 23.5 \ s$$

t

TS diagram:

Process ① → ② in tire

Process (isentropic) ① → ④

(moving to) ② → ④ in

converging passage.

In tire,

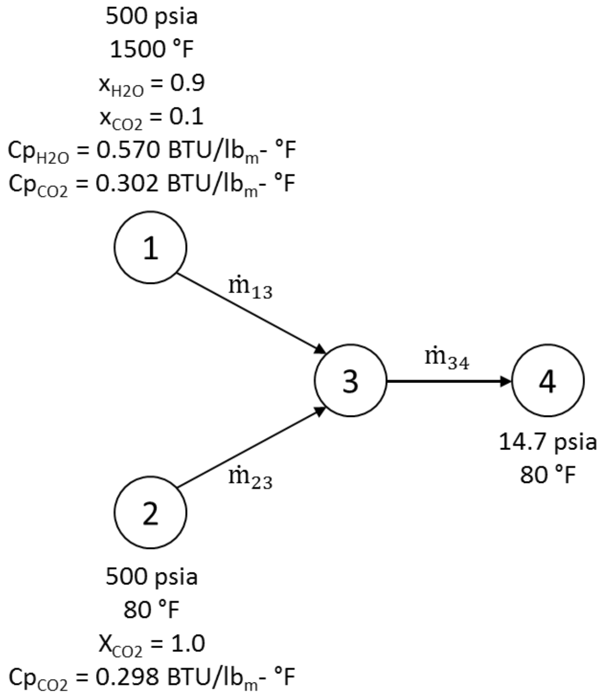
$$\Delta s = c_p \ln \cancel{\frac{T_2}{T_1}} - R \ln \frac{p_2}{p_1} = -287 \frac{N \cdot m}{kg \cdot K} \times \ln \frac{(310+101)}{(620+101)} = 161 \ J/(kg \cdot K)$$

Δs

1324

http://www.slideshare.net/SubodhKumar27/solution-manual-fluid-mechanics-fox-mcdonald

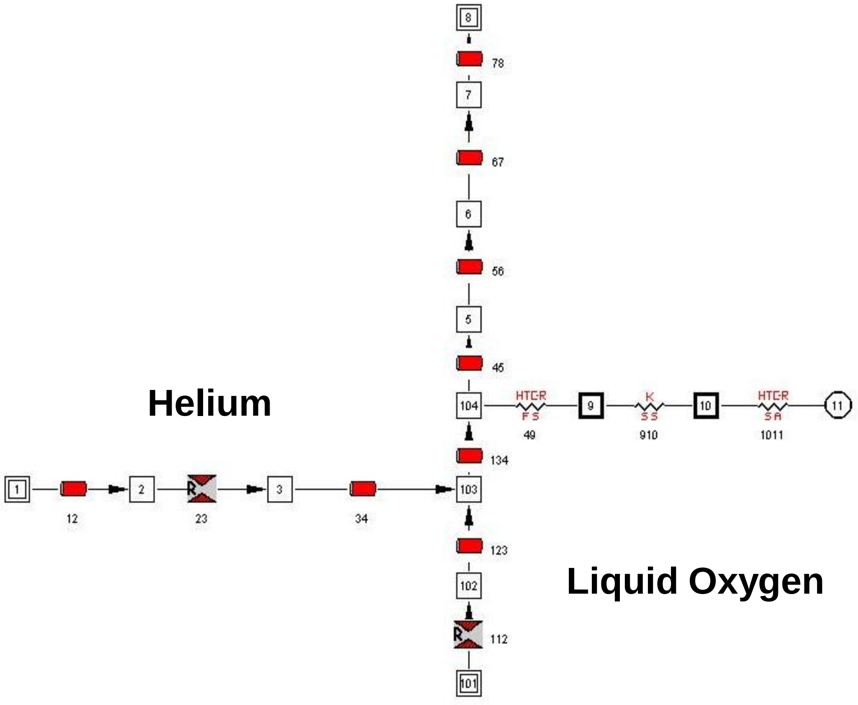
Fluid Mixture & Two-Phase Flows



500 psia
1500 °F
x_{H2O} = 0.9
x_{CO2} = 0.1
Cp_{H2O} = 0.570 BTU/lb_m- °F
Cp_{CO2} = 0.302 BTU/lb_m- °F

14.7 psia
80 °F

500 psia
80 °F
X_{CO2} = 1.0
Cp_{CO2} = 0.298 BTU/lb_m- °F



Helium

Liquid Oxygen

Content

- Temperature / Specific Heat Formulation
- Enthalpy 1 Formulation
- Enthalpy 2 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

Temperature / Specific Heat Formulation

- **GFSSP**'s default calculation of mixture temperature
- Modified Energy Conservation Equation using the specific heats of the individual species instead of enthalpy:

$$(\mathrm{T}_i)_{\tau+\Delta\tau} = \frac{\sum_{j=1}^{j=n}\sum_{k=1}^{k=\,n_f} \mathrm{Cp}_k x_k \mathrm{T}_j \ \mathrm{MAX}\left[-\dot{m}_{ij},0\right] + \left(\frac{\mathrm{Cv}_i m_i \mathrm{T}_i}{\Delta\tau}\right) + \ Q_i}{\sum_{j=1}^{j=n}\sum_{k=1}^{k=\,n_f} \mathrm{Cp}_k x_k \ \mathrm{MAX}\left[\dot{m}_{ij},0\right] + \ \left(\frac{(Cv_i m_i)_{\tau+\Delta\tau}}{\Delta\tau}\right)}$$

- Limitations
  - Cannot handle phase change of mixture (because there is no heat of vaporization, h_{fg}).
  - Assumes specific heats are relatively constant.

Enthalpy 1 Formulation (1/6)

- **GFSSP** Enthalpy 1 Option
  - Sums the enthalpies of the individual species to arrive at a Total Enthalpy of the Node:

$$(\mathrm{h}_i)_{\tau+\Delta\tau} = \frac{\sum_{j=1}^{j=n}\sum_{k=1}^{k=n_f} x_{j,k}\mathrm{h}_{j,k}\ \mathrm{MAX}\left[-\dot{m}_{ij},0\right] + \frac{(m_i\mathrm{u}_i)_\tau}{\Delta\tau} + Q_i}{\sum_{j=1}^{j=n}\sum_{k=1}^{k=n_f} x_{j,k}\ \mathrm{MAX}\left[\dot{m}_{ij},0\right] + \frac{(m_i)_{\tau+\Delta\tau}}{\Delta\tau}}$$

  - Using Enthalpy avoids the problem of non-constant C_p.

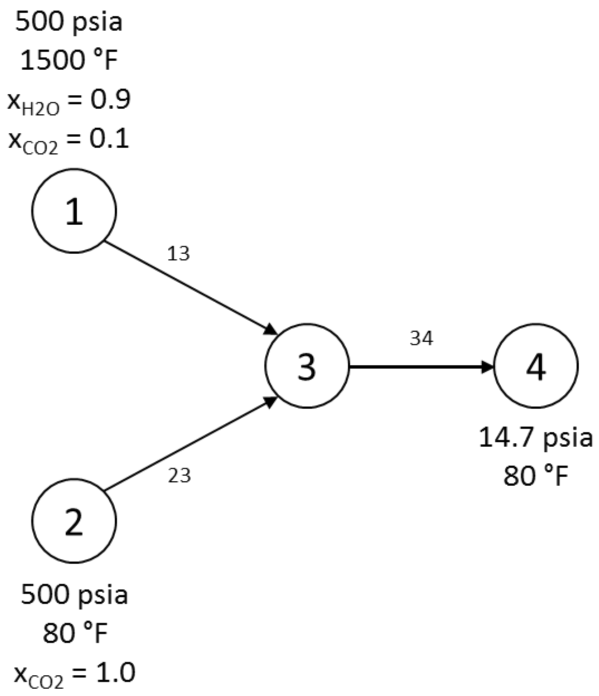
  - Using Enthalpy accounts for the heat of vaporization, h_{fg}.

  - BUT, the individual fluid species have different reference points.
    - ➢ Not possible to determine the mixture temperature directly.
    - ➢ Iterative procedure is used to find the mixture temperature.
    - ➢ The procedure also checks for phase-change by checking to see if the Total Enthalpy of the Node is bounded by the calculated mixture enthalpy at the saturation temperature of each fluid. If so, it then sets the node temperature to the saturation temperature and interpolates the quality of the saturated fluid.

Enthalpy 1 Formulation (2/6)

- Example 4: Simulation of the Mixing of Combustion Gases and a Cold Gas Stream (1/3)

- Features
  - Fluid Mixture
  - Comparison with Textbook Solution



Enthalpy 1 Formulation (3/6)

- Example 4: Simulation of the Mixing of Combustion Gases and a Cold Gas Stream (2/3)
- Mixture Temperature Option

$$x_{H2O}\dot{m}_{13}\mathrm{Cp}_{H2O}\mathrm{T}_1 + x_{CO2}\dot{m}_{13}\mathrm{Cp}_{CO2}\mathrm{T}_1 + \dot{m}_{23}\mathrm{Cp}_{CO2}\mathrm{T}_2 = \dot{m}_{34}\mathrm{Cp}_{mix}\mathrm{T}_3$$

$$\mathrm{T}_3 = \frac{x_{H2O}\dot{m}_{13}\mathrm{Cp}_{H2O}\mathrm{T}_1 + x_{CO2}\dot{m}_{13}\mathrm{Cp}_{CO2}\mathrm{T}_1 + \dot{m}_{23}\mathrm{Cp}_{CO2}\mathrm{T}_2}{\dot{m}_{34}\mathrm{Cp}_{mix}}$$

$$= \frac{(0.9)\left(1.15\frac{\mathrm{lb_m}}{\mathrm{sec}}\right)\left(0.570\frac{\mathrm{BTU}}{\mathrm{lb_m}-°\mathrm{R}}\right)(1960\ °\mathrm{R}) + (0.1)\left(1.15\frac{\mathrm{lb_m}}{\mathrm{sec}}\right)\left(0.302\frac{\mathrm{BTU}}{\mathrm{lb_m}-°\mathrm{R}}\right)(1960\ °\mathrm{R}) + (3.70\frac{\mathrm{lb_m}}{\mathrm{sec}})(0.298\frac{\mathrm{BTU}}{\mathrm{lb_m}-°\mathrm{R}})(540\ °\mathrm{R})}{(4.85\frac{\mathrm{lb_m}}{\mathrm{sec}})\left[(0.786)\left(0.266\frac{\mathrm{BTU}}{\mathrm{lb_m}-°\mathrm{R}}\right) + (0.214)\left(0.569\frac{\mathrm{BTU}}{\mathrm{lb_m}-°\mathrm{R}}\right)\right]}$$

$$\mathrm{T}_3 = 1134\ °\mathrm{R}\ (674\ °\mathrm{F})$$

Enthalpy 1 Formulation (4/6)

- Example 4: Simulation of the Mixing of Combustion Gases and a Cold Gas Stream (3/3)

- Enthalpy 1 Option (Total Enthalpy of the Node)

$$x_{H2O}\dot{m}_{13}\mathrm{h}_{H2O} + x_{CO2}\dot{m}_{13}\mathrm{h}_{CO2} + \dot{m}_{23}\mathrm{h}_{CO2} = \dot{m}_{34}\mathrm{h}_{node}$$

$$\mathrm{h}_{node} = \frac{x_{H2O}\dot{m}_{13}\mathrm{h}_{H2O} + x_{CO2}\dot{m}_{13}\mathrm{h}_{CO2} + \dot{m}_{23}\mathrm{h}_{CO2}}{\dot{m}_{34}}$$

$$= \frac{(0.9)\left(1.176\frac{\mathrm{lb_m}}{\mathrm{sec}}\right)\left(1800\frac{\mathrm{BTU}}{\mathrm{lb_m}}\right) + (0.1)\left(1.176\frac{\mathrm{lb_m}}{\mathrm{sec}}\right)\left(722\frac{\mathrm{BTU}}{\mathrm{lb_m}}\right) + (3.776\frac{\mathrm{lb_m}}{\mathrm{sec}})(332\frac{\mathrm{BTU}}{\mathrm{lb_m}})}{(4.952\frac{\mathrm{lb_m}}{\mathrm{sec}})}$$

$$\mathrm{h}_{node} = 655\frac{\mathrm{BTU}}{\mathrm{lb_m}}$$

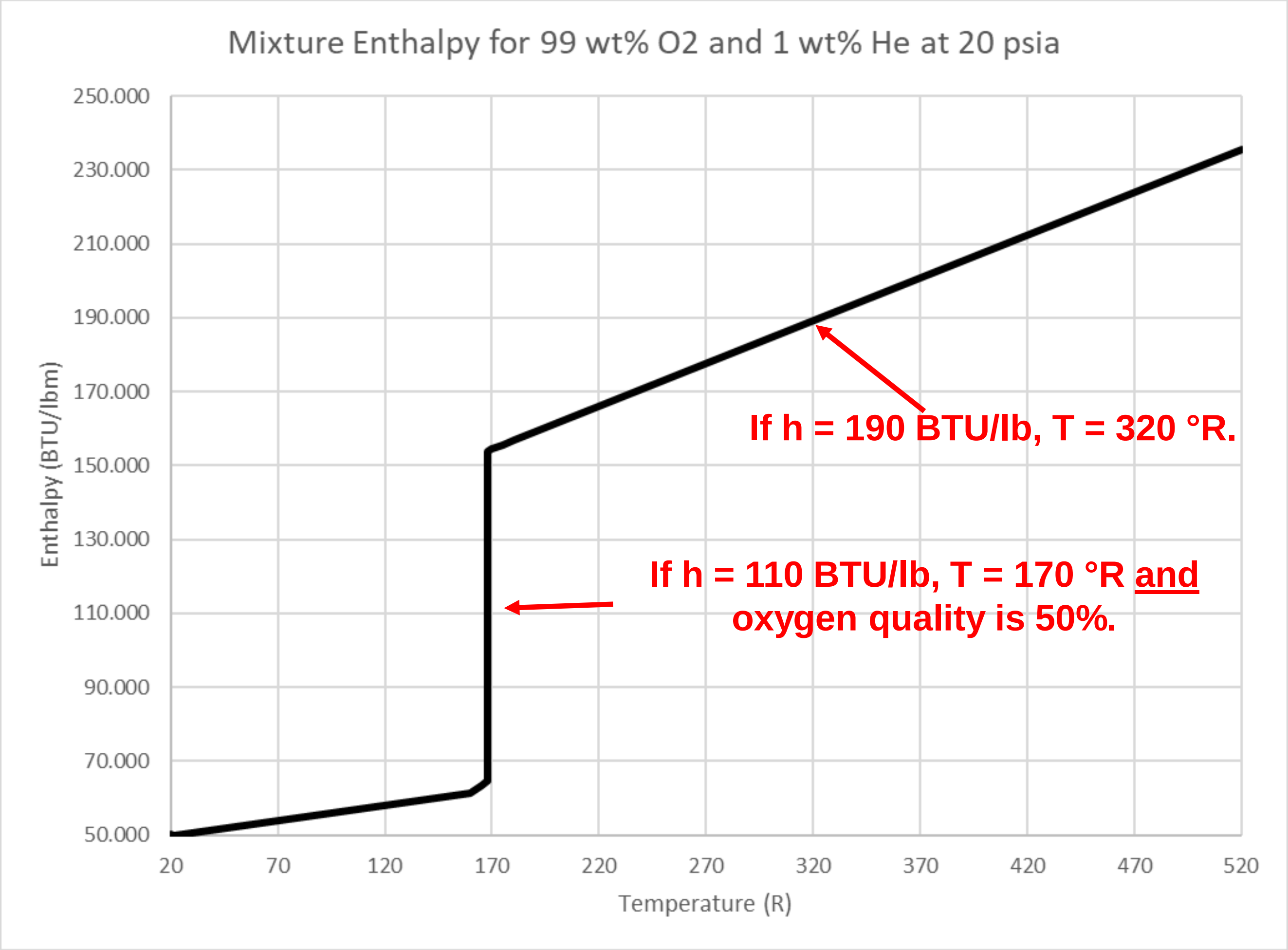
- Iterating on temperature results in enthalpies at 633 °F:
  - h_{H2O} = 1320 BTU/$\mathrm{lb_m}$
  - h_{CO2} = 474 BTU/$\mathrm{lb_m}$

$$\mathrm{h}_{node} = x_{H2O}\mathrm{h}_{H2O} + x_{CO2}\mathrm{h}_{CO2}$$

$$= (0.214)\left(1320\frac{\mathrm{BTU}}{\mathrm{lb_m}}\right) + (0.786)\left(474\frac{\mathrm{BTU}}{\mathrm{lb_m}}\right)$$

$$\mathrm{h}_{node} = 655\frac{\mathrm{BTU}}{\mathrm{lb_m}}$$

Enthalpy 1 Formulation (5/6)

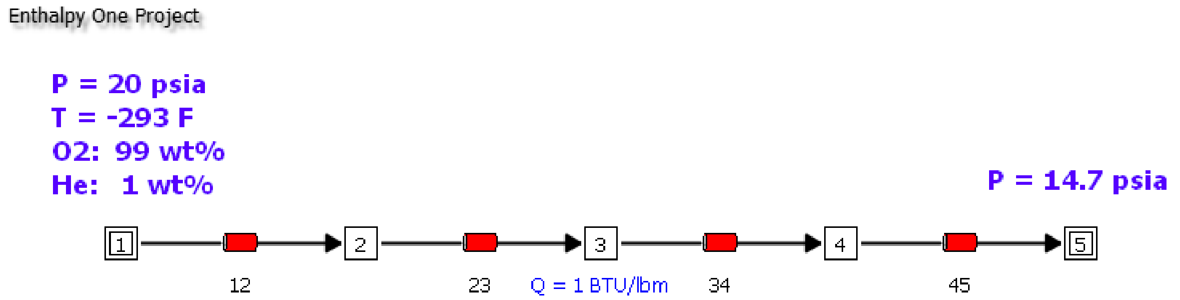


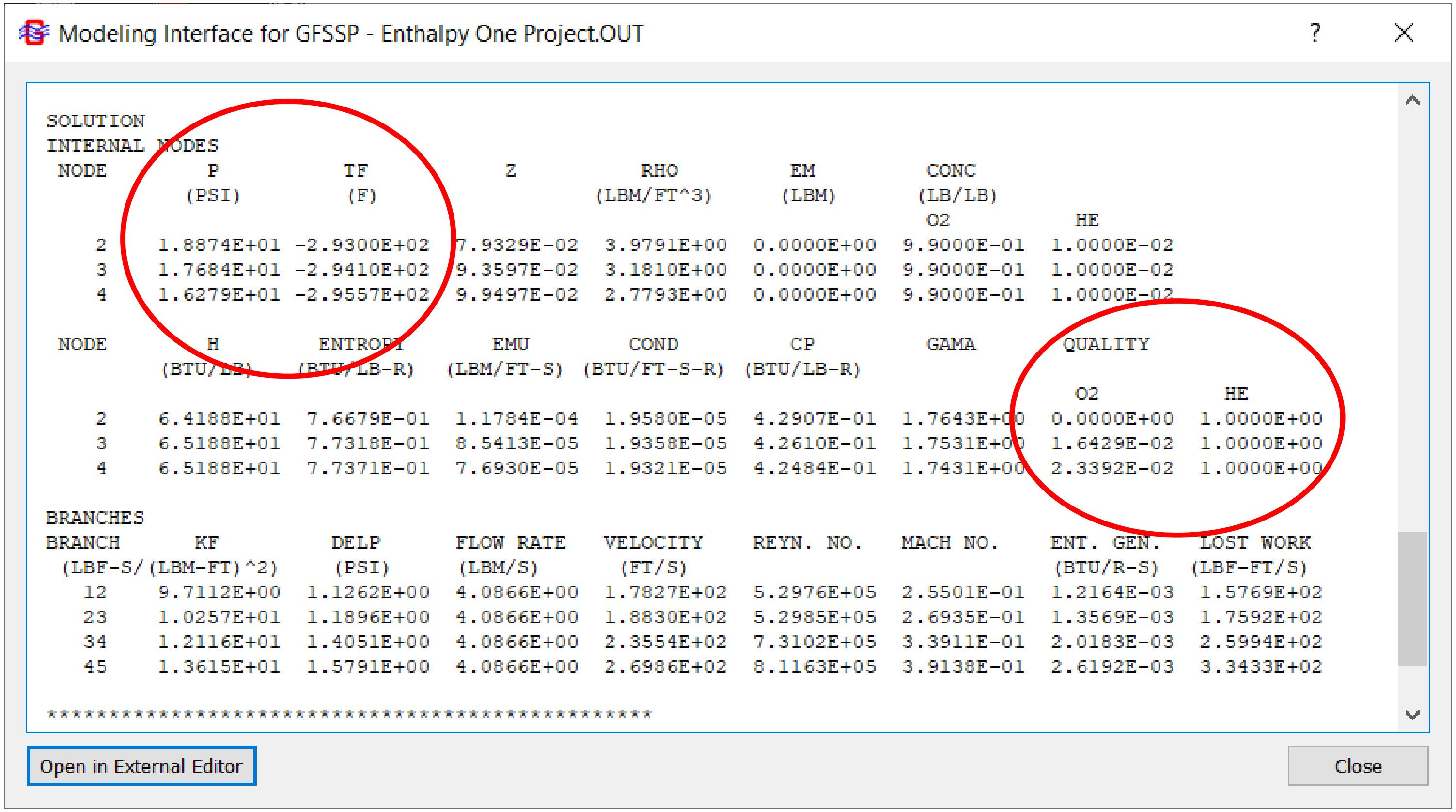
If h = 190 BTU/lb, T = 320 °R.

If h = 110 BTU/lb, T = 170 °R and oxygen quality is 50%.

The Enthalpy 1 option checks the saturation temperature of each fluid prior to iterating on temperature.

Enthalpy 1 Formulation (6/6)





```
 SOLUTION
 INTERNAL NODES
  NODE        P            TF          Z          RHO         EM         CONC
            (PSI)          (F)                (LBM/FT^3)     (LBM)      (LB/LB)
                                                                          O2          HE
     2    1.8874E+01 -2.9300E+02 7.9329E-02  3.9791E+00  0.0000E+00  9.9000E-01  1.0000E-02
     3    1.7684E+01 -2.9410E+02 9.3597E-02  3.1810E+00  0.0000E+00  9.9000E-01  1.0000E-02
     4    1.6279E+01 -2.9557E+02 9.9497E-02  2.7793E+00  0.0000E+00  9.9000E-01  1.0000E-02

  NODE        H          ENTROPY       EMU        COND         CP          GAMA        QUALITY
           (BTU/LB)   (BTU/LB-R)  (LBM/FT-S)  (BTU/FT-S-R)  (BTU/LB-R)

                                                                                      O2          HE
     2    6.4188E+01  7.6679E-01  1.1784E-04  1.9580E-05  4.2907E-01  1.7643E+00  0.0000E+00  1.0000E+00
     3    6.5188E+01  7.7318E-01  8.5413E-05  1.9358E-05  4.2610E-01  1.7531E+00  1.6429E-02  1.0000E+00
     4    6.5188E+01  7.7371E-01  7.6930E-05  1.9321E-05  4.2484E-01  1.7431E+00  2.3392E-02  1.0000E+00

 BRANCHES
 BRANCH       KF           DELP       FLOW RATE    VELOCITY    REYN. NO.   MACH NO.    ENT. GEN.   LOST WORK
  (LBF-S/(LBM-FT)^2)      (PSI)       (LBM/S)      (FT/S)                              (BTU/R-S)   (LBF-FT/S)
    12    9.7112E+00  1.1262E+00  4.0866E+00  1.7827E+02  5.2976E+05  2.5501E-01  1.2164E-03  1.5769E+02
    23    1.0257E+01  1.1896E+00  4.0866E+00  1.8830E+02  5.2985E+05  2.6935E-01  1.3569E-03  1.7592E+02
    34    1.2116E+01  1.4051E+00  4.0866E+00  2.3554E+02  7.3102E+05  3.3911E-01  2.0183E-03  2.5994E+02
    45    1.3615E+01  1.5791E+00  4.0866E+00  2.6986E+02  8.1163E+05  3.9138E-01  2.6192E-03  3.3433E+02

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

Enthalpy 2 Formulation (1/4)

- The Enthalpy 2 option was added to allow for one of the fluids in the mixture to be two-phase. (This option was developed before saturation-checking was added to the Enthalpy 1 option.)

- Liquid Propulsion Applications
  - Situations where one of the constituents is saturated
    - Mixture of liquid and vapor is in equilibrium
    - Example: A mixture of helium, LO2, and GO2 exists during purging of liquid oxygen by ambient helium

- In the Enthalpy 2 option, Separate Energy Equations are solved for each species. There is no heat transfer between the species unless coded in a user subroutine.
  - An average temperature is calculated for the node, but this is for output only. The temperatures of the individual species will likely be different. The properties of the individual species are evaluated at the species enthalpy, not at the average node temperature.
  - This contrasts with the updated Enthalpy 1 option with saturation-checking. The Enthalpy 1 option assumes complete heat transfer between the separate species, so that all species are at the same temperature.

Enthalpy 2 Formulation (2/4)

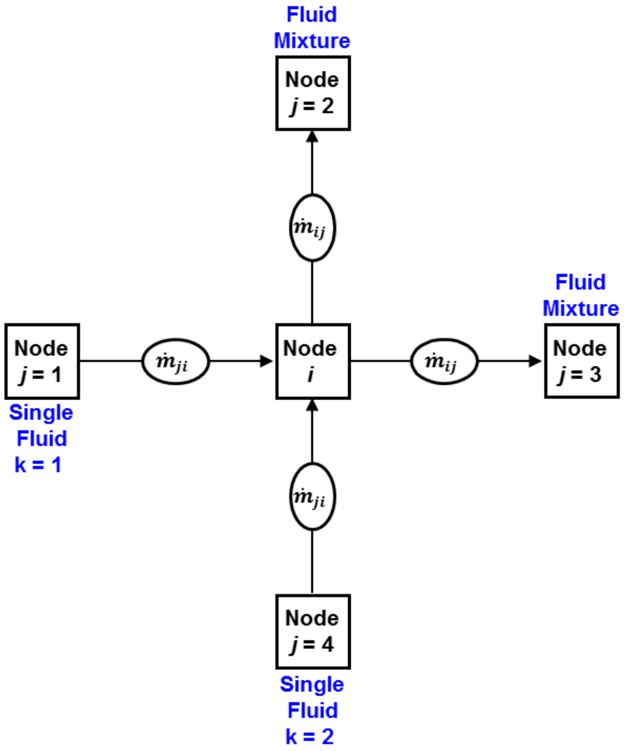
- Separate Energy Equation for Individual Species (**SEEIS**)

$$\frac{\left(m_i h_{ik} - \frac{p}{\rho_k J}\right)_{\tau+\Delta\tau} - \left(m_i h_{ik} - \frac{p}{\rho_k J}\right)_{\tau}}{\Delta\tau} = \sum_{j=1}^{j=n}\left\{\mathrm{MAX}\left[-\dot{m}_{ij},0\right]h_{jk} - \mathrm{MAX}\left[\dot{m}_{ij},0\right]h_{ik}\right\} + Q_{ik} + \left\{\pm Q^{HES}_{1\to2}\right\}$$

Transient Term

Advection Term

Source Term



GFSSP 7.02 -- Fluid Mixture

Enthalpy 2 Formulation (3/4)

- Thermodynamic Properties
  - Temperature and other properties of individual species
  - Calculated from node pressure and enthalpy of the species

$$T_{ik} = f(p_i, h_{ik})$$
$$\rho_{ik} = f(p_i, h_{ik})$$
$$\mu_{ik} = f(p_i, h_{ik})$$
$$K_{ik} = f(p_i, h_{ik})$$
$$Cp_{ik} = f(p_i, h_{ik})$$

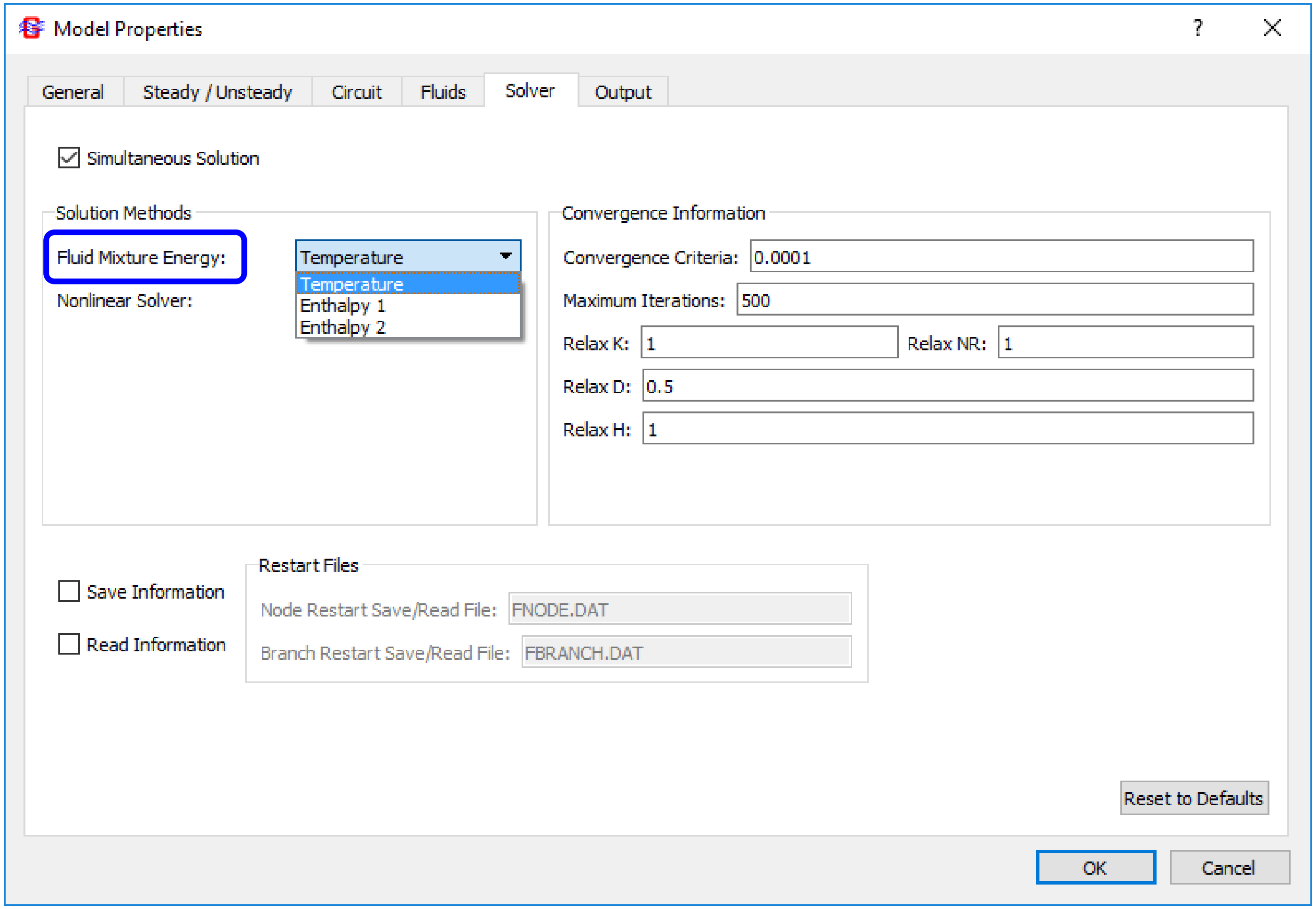
- Nodal Properties
  - Calculated by averaging the properties of species
    - ➢ Note: $\overline{c}_{ik}$ is the molar concentration of species *k* for Node *i*

$$\frac{1}{\rho_i} = \sum_{k=1}^{n_f} \frac{c_{ik}}{\rho_{ik}}$$

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

Enthalpy 2 Formulation (4/4)

- Fluid Mixture Energy Option



Model Properties

☑ Simultaneous Solution

Solution Methods

Fluid Mixture Energy: Temperature

Nonlinear Solver:

Convergence Information

Convergence Criteria: 0.0001

Maximum Iterations: 500

Relax K: 1

Relax NR: 1

Relax D: 0.5

Relax H: 1

☐ Save Information

☐ Read Information

Restart Files

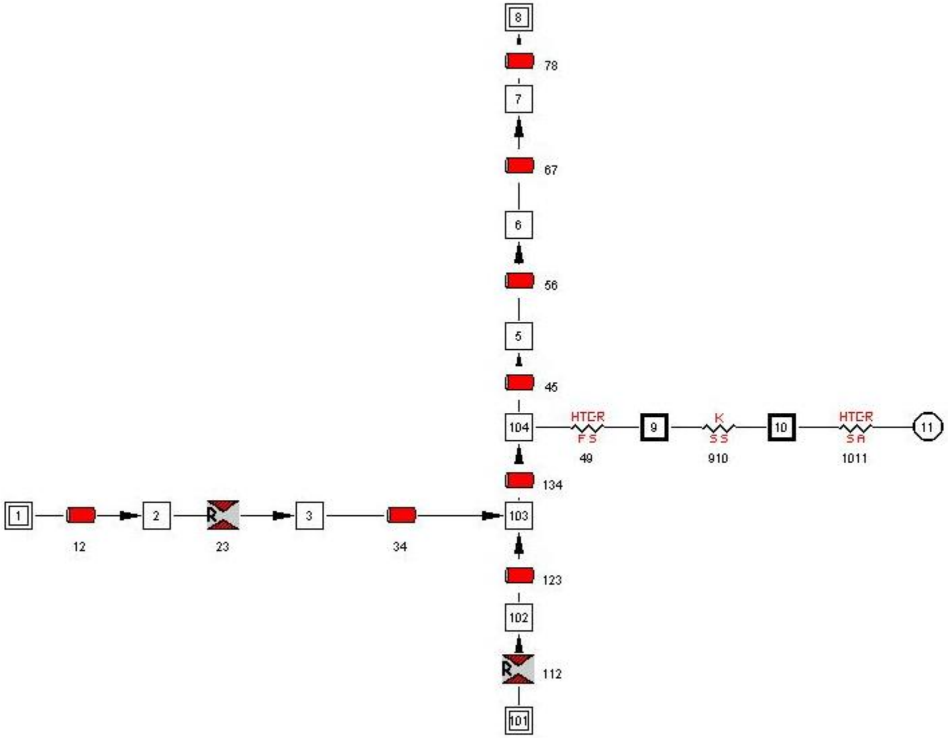
Node Restart Save/Read File: FNODE.DAT

Branch Restart Save/Read File: FBRANCH.DAT

Application: Example 23

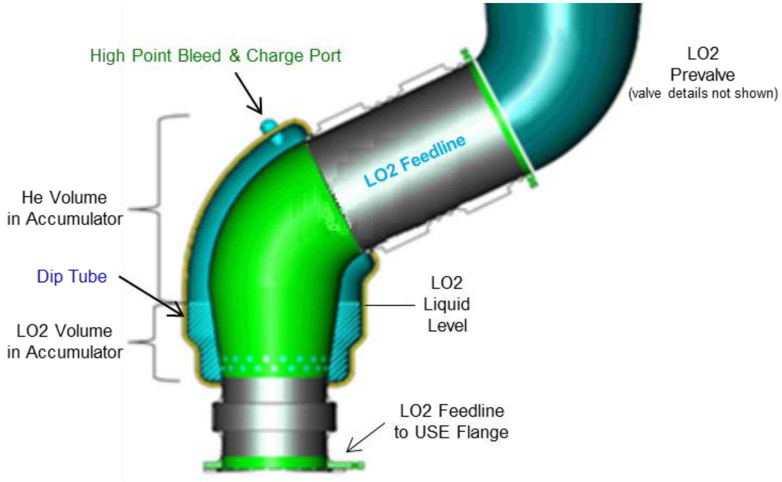
- Example 23: Helium-Assisted Buoyancy-Driven Flow in a Vertical Pipe Carrying Liquid Oxygen with Ambient Heat Leak

- Features
  - Phase Change in Fluid Mixture
  - Buoyancy-driven Flow
  - Conjugate Heat Transfer



Application: Charging of POGO Accumulator (1/6)

- Problem Considered
  - An annular-shaped POGO Accumulator is wrapped around a LO2 Feedline to a rocket engine turbopump. The lower portion of the Accumulator communicates to the feedline through a series of holes. The Accumulator has a Dip Tube located a few inches above the communication port and is connected to a dump line. The Accumulator also has a High Point Bleed & Charge Port.

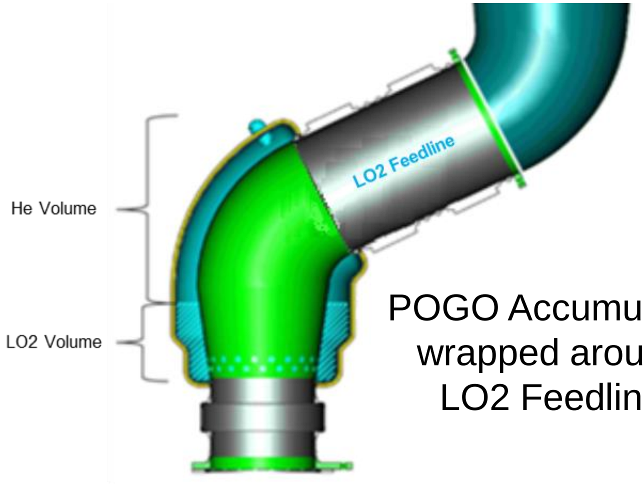


  - The Accumulator is initially filled with LO2. Helium enters into the accumulator through High Point Bleed & Charge Port and displaces LO2 to the feedline and the dump line. Once the LO2 level drops to the location of Dip Tube, a stable helium bubble is retained in the Accumulator that provides desired compliance.
  - Objective: Predict the charging process history as well as the steady operation of the Accumulator during engine run.

Application: Charging of POGO Accumulator (2/6)

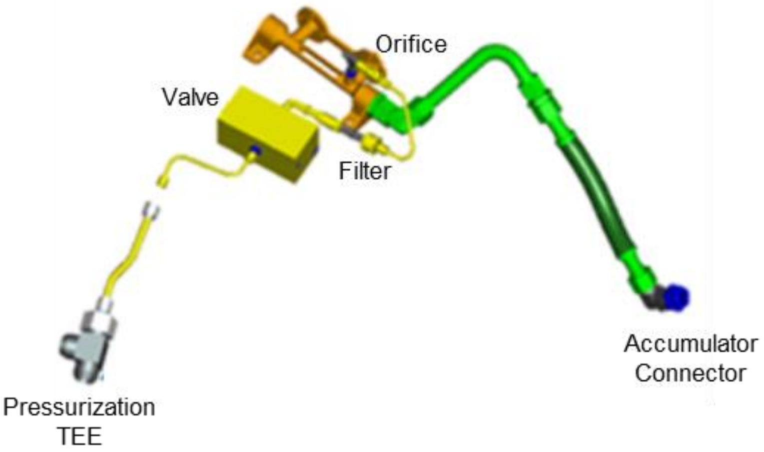
Marshall Space Flight Center
GFSSP Training Course

- POGO Accumulator with Charge Line and Dump Line

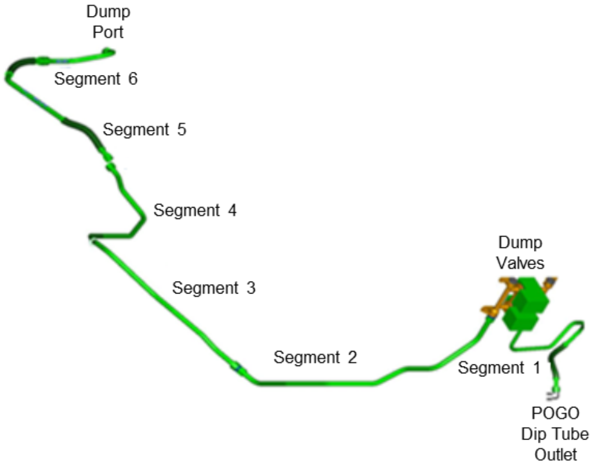


POGO Accumulator wrapped around LO2 Feedline

Charge System

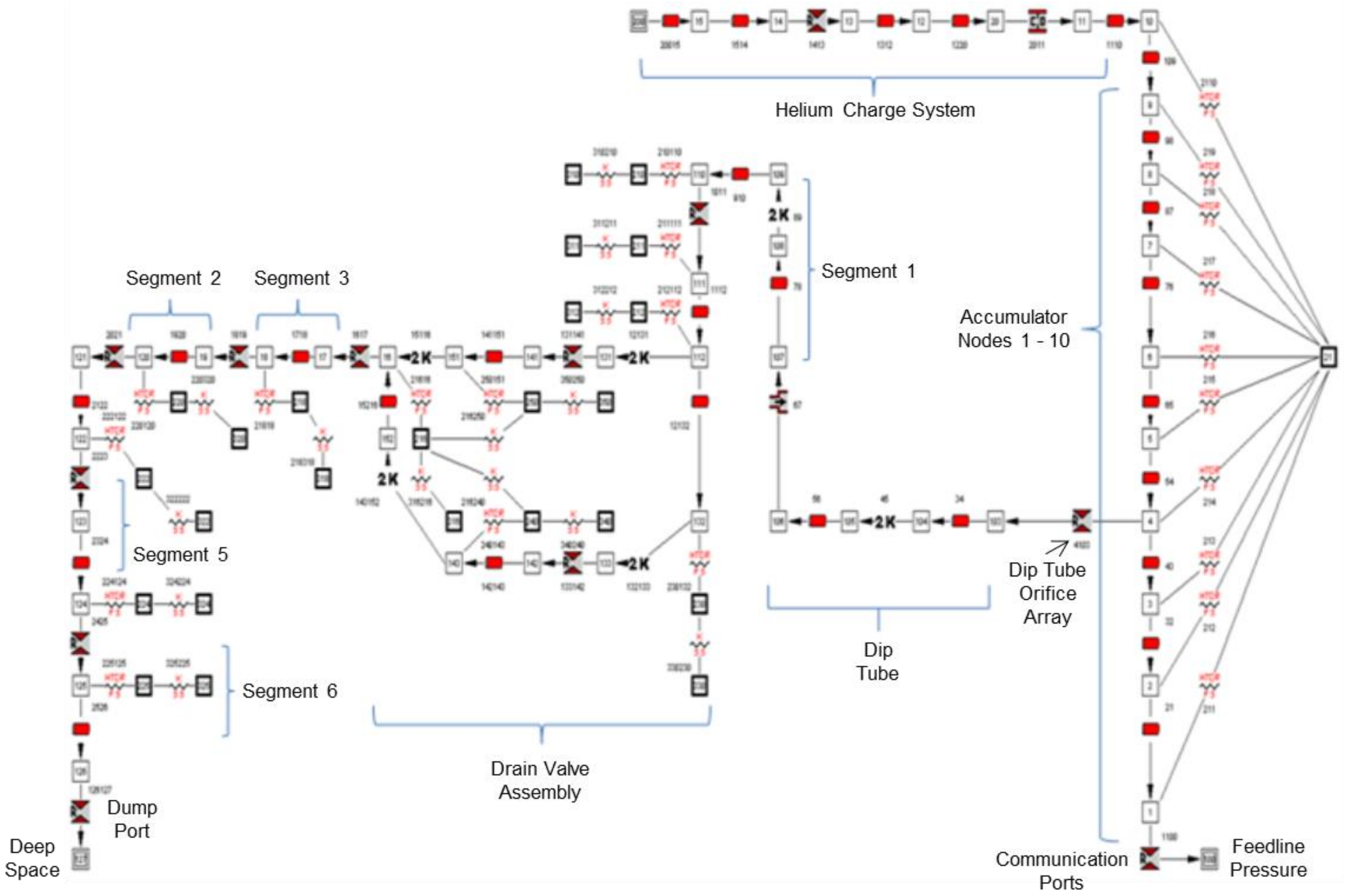


Dump System



Application: Charging of POGO Accumulator (3/6)

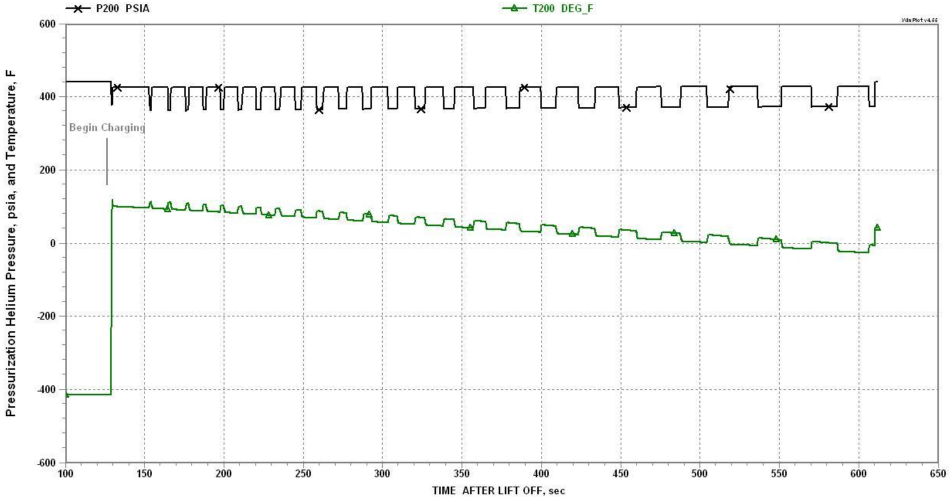
- **GFSSP** POGO Accumulator & Drain Line model



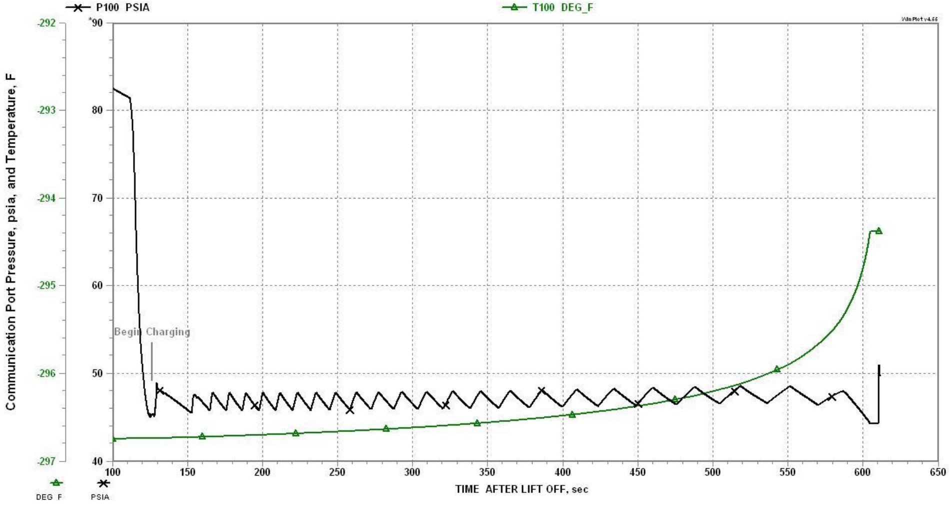
Application: Charging of POGO Accumulator (4/6)

- Boundary Conditions

Pressure and Temperature History at the Helium Supply Line



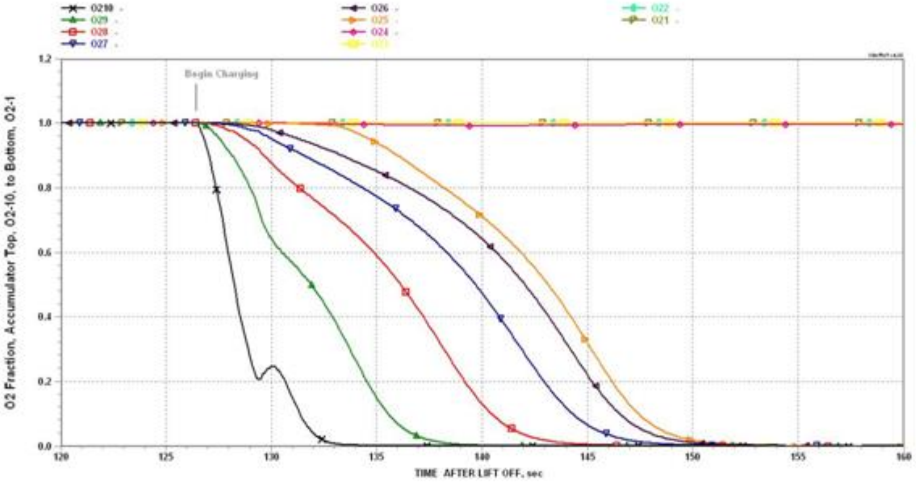
Pressure and Temperature History at the Communication Ports



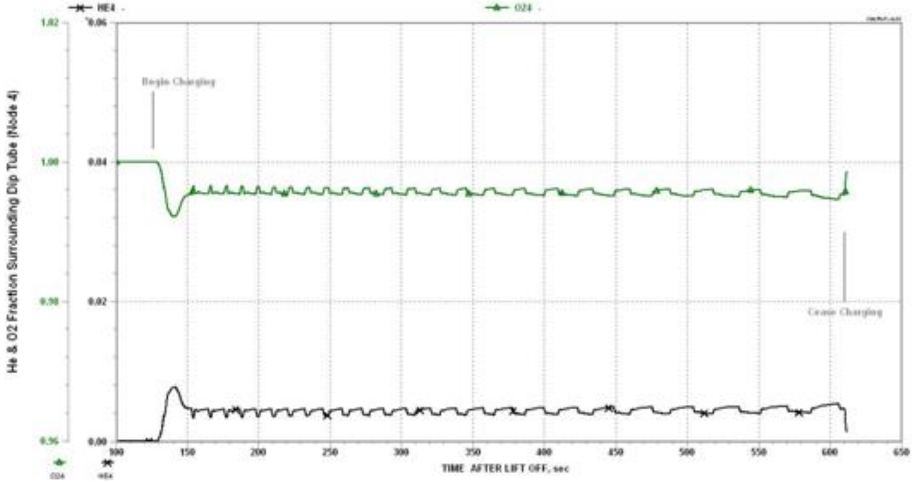
Application: Charging of POGO Accumulator (5/6)

- Charging of Helium and Draining of He-LO$_2$ Mixture

Displacement of Oxygen from the Accumulator During Charging

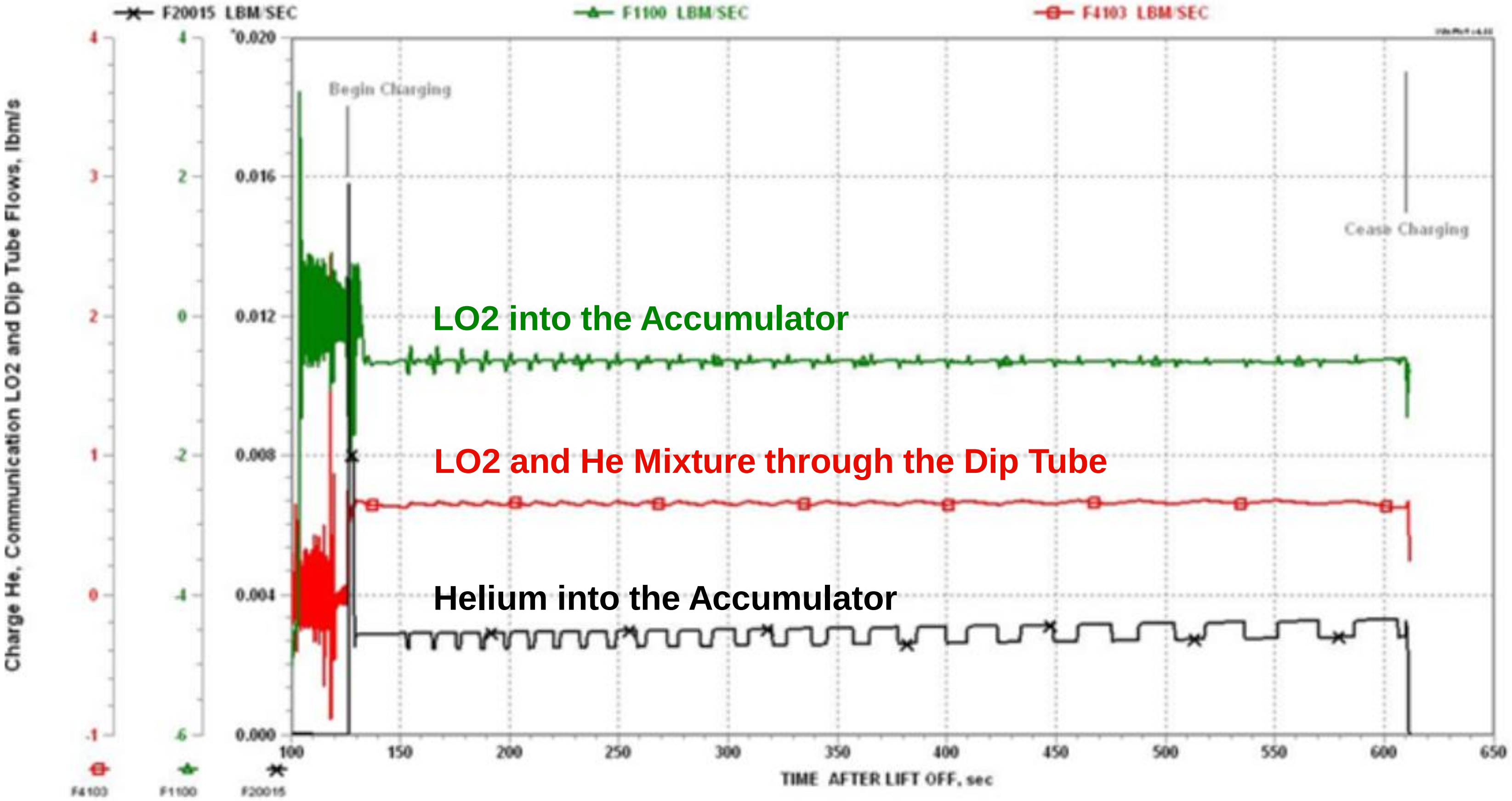


Concentrations of Helium and Oxygen in the Dip Tube



Application: Charging of POGO Accumulator (6/6)

- Flowrates



LO2 into the Accumulator

LO2 and He Mixture through the Dip Tube

Helium into the Accumulator

Summary

- Model with Multiple Species
  - Requires trade-offs
  - Requires reformulation of the energy equation

- Mixture Temperature Option
  - Simple to implement
    - Does not permit Phase Change or large variation in Cp

- Enthalpy 1 Option
  - Requires extra steps of iteration on temperature and saturation-checking
    - Gets around Cp variation and allows phase change

- Enthalpy 2 Option
  - Allows Phase Change
  - Species have individual temperatures
    - Used to calculate an average node temperature
  - User must decide whether and how to handle inter-species heat transfer

Tutorial – 6

Modeling an Oxygen Recirculation Line

Tutorial Objectives

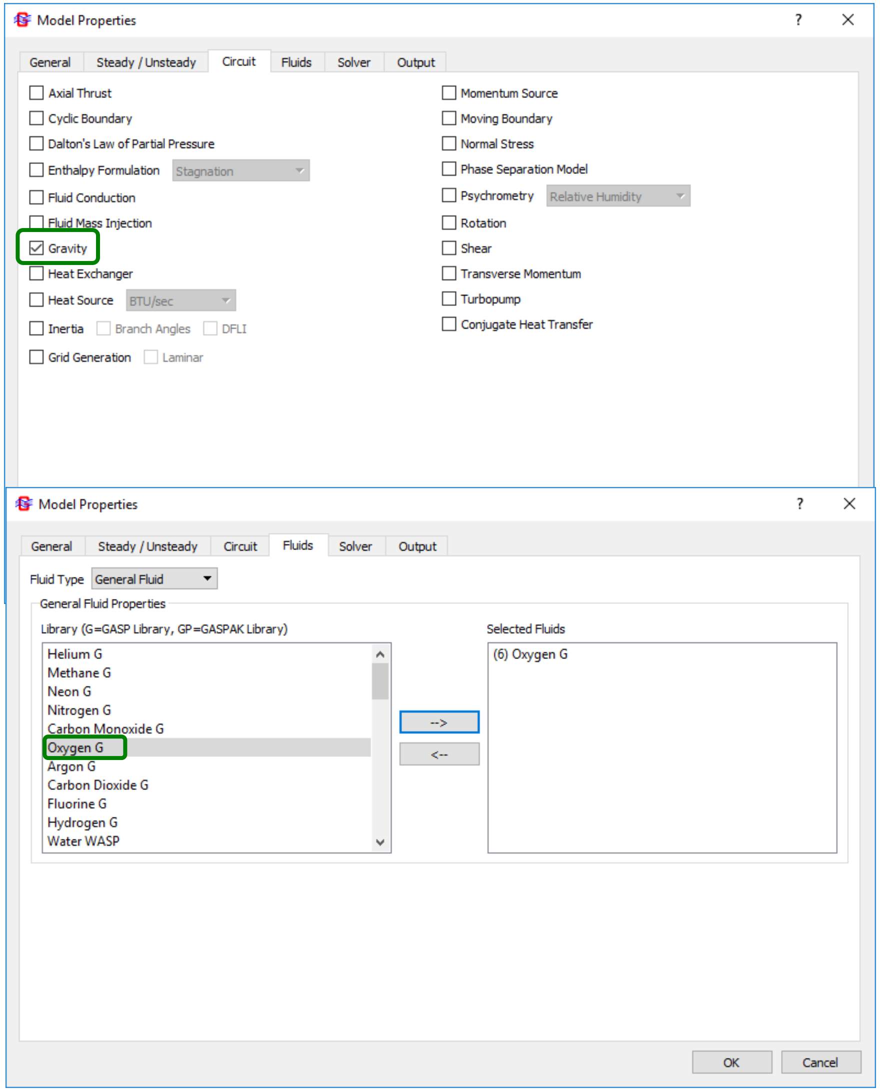
(1) Model LOx sitting stagnant in a vertical recirculation line

(2) Evaluate the effect of heat transfer on the flowrate

(3) Add a helium injector to the recirculation line

Part 1: LOx Sitting Stagnant

- General
  - Model File: Tut6.gfssp
  - Input File: Tut6.dat
  - Output File: Tut6.out
- Steady State
- Circuit Options
  - Gravity
- Fluid Options
  - Liquid Oxygen



Model Properties

☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation: Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☑ Gravity
☐ Heat Exchanger
☐ Heat Source: BTU/sec
☐ Inertia ☐ Branch Angles ☐ DFLI
☐ Grid Generation ☐ Laminar
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry: Relative Humidity
☐ Rotation
☐ Shear
☐ Transverse Momentum
☐ Turbopump
☐ Conjugate Heat Transfer

Model Properties

Fluid Type: General Fluid

General Fluid Properties

Library (G=GASP Library, GP=GASPAK Library)

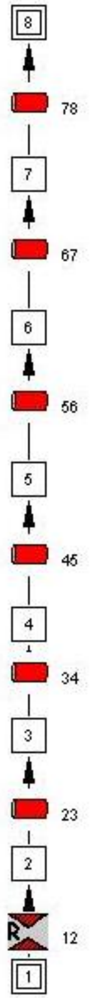
Helium G
Methane G
Neon G
Nitrogen G
Carbon Monoxide G
Oxygen G
Argon G
Carbon Dioxide G
Fluorine G
Hydrogen G
Water WASP

Selected Fluids

(6) Oxygen G

Part 1: Build Model on Canvas

- Recirculation line
  - 6' vertical smooth pipe
  - 1.87" diameter



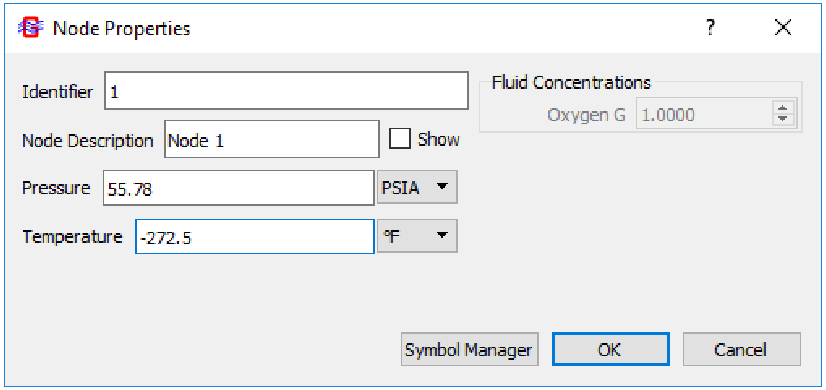
Now is a good time to save your **Tut6.gfssp** file

Part 1: 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

- Note: $\Delta P_{1,8}$ = 2.78 psia
  - Corresponds to the hydrostatic head of 6 ft of LOx
  - LOx is approximately 1 °F subcooled at this pressure



Identifier: 1

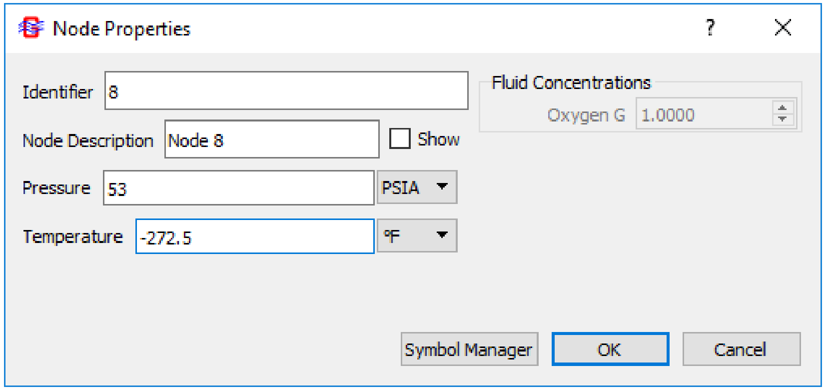
Node Description: Node 1 ☐ Show

Pressure: 55.78 PSIA

Temperature: -272.5 °F

Fluid Concentrations

Oxygen G 1.0000



Identifier: 8

Node Description: Node 8 ☐ Show

Pressure: 53 PSIA

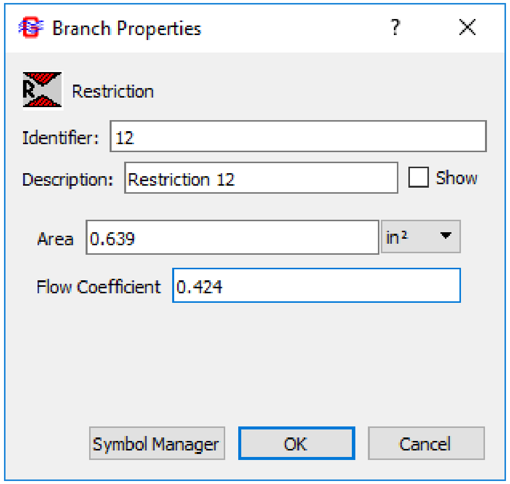
Temperature: -272.5 °F

Fluid Concentrations

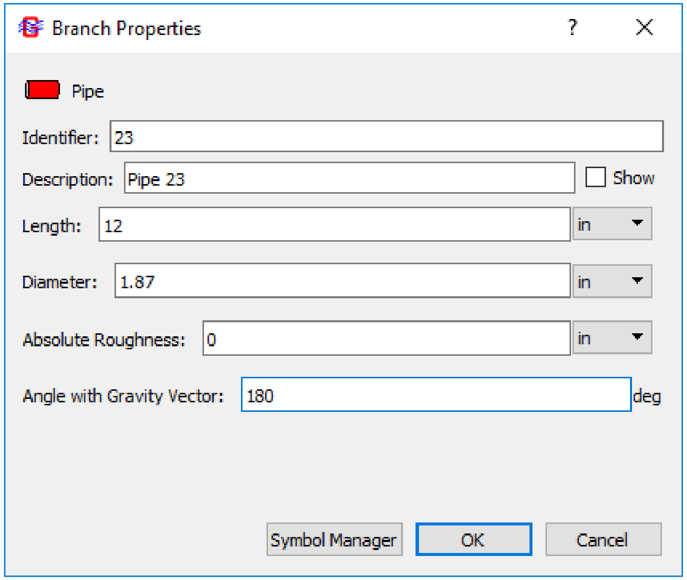
Oxygen G 1.0000

Part 1: 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)

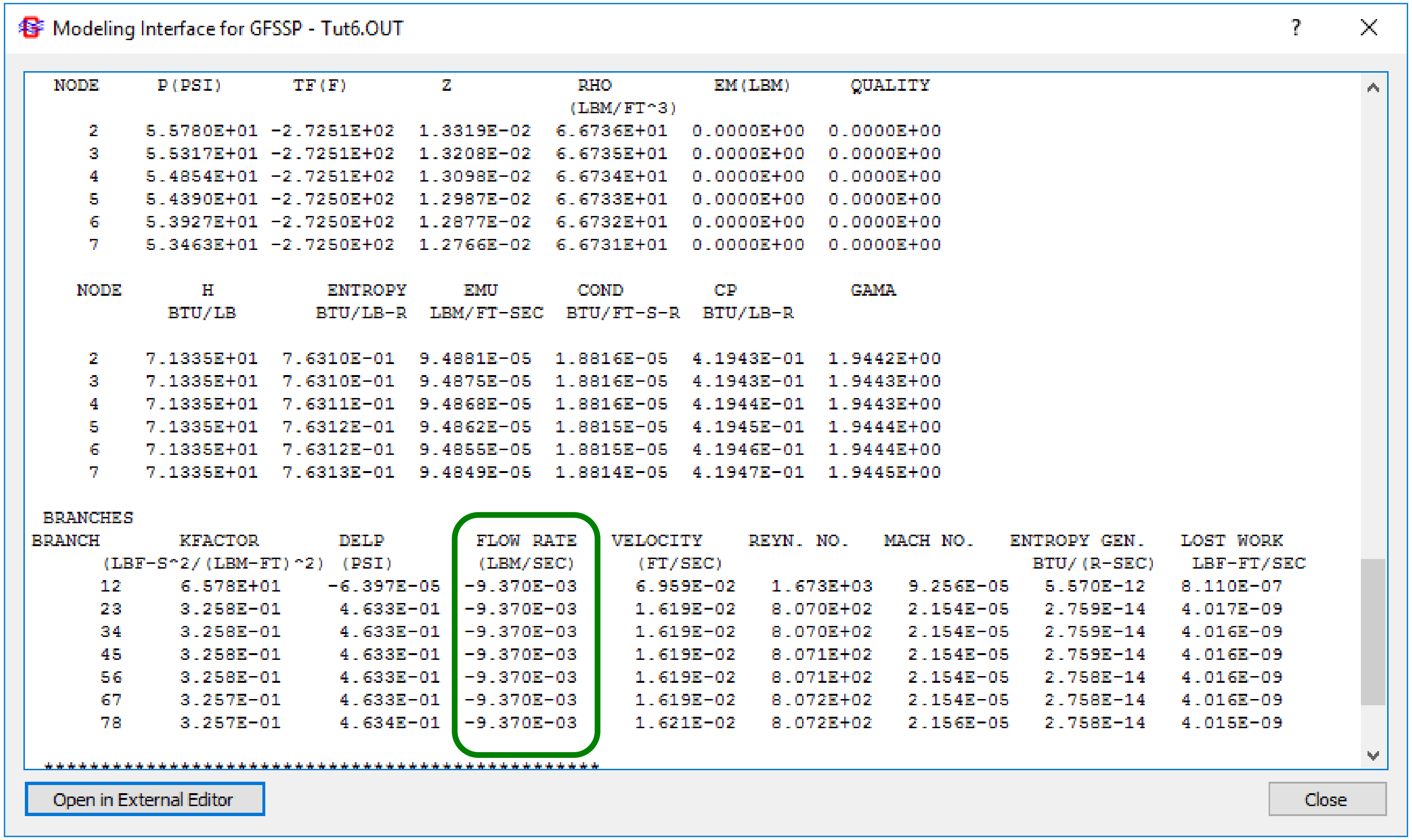


Identifier: 12
Description: Restriction 12 ☐ Show
Area 0.639 in²
Flow Coefficient 0.424



Identifier: 23
Description: Pipe 23 ☐ Show
Length: 12 in
Diameter: 1.87 in
Absolute Roughness: 0 in
Angle with Gravity Vector: 180 deg

Part 1: Result of Stagnant LOx Model

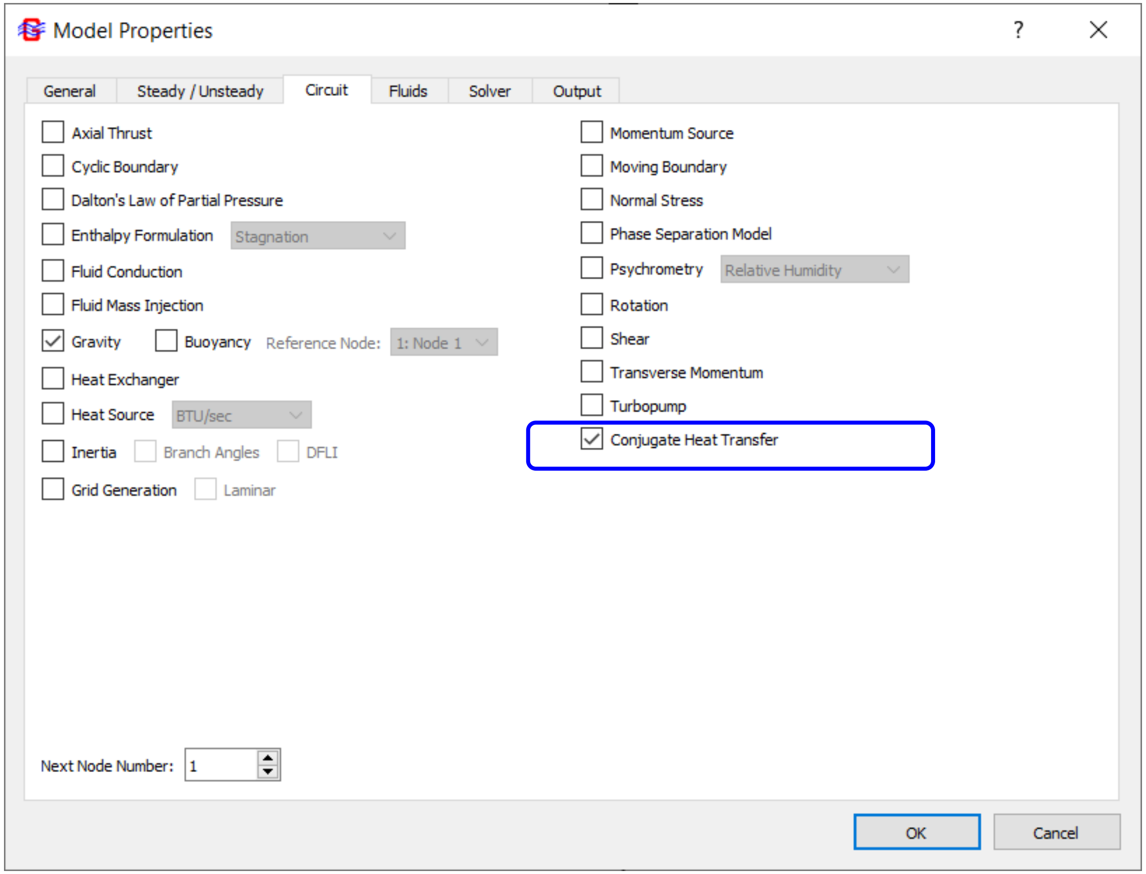


```
  NODE     P(PSI)      TF(F)         Z            RHO         EM(LBM)      QUALITY
                                               (LBM/FT^3)
     2    5.5780E+01 -2.7251E+02  1.3319E-02  6.6736E+01  0.0000E+00  0.0000E+00
     3    5.5317E+01 -2.7251E+02  1.3208E-02  6.6735E+01  0.0000E+00  0.0000E+00
     4    5.4854E+01 -2.7251E+02  1.3098E-02  6.6734E+01  0.0000E+00  0.0000E+00
     5    5.4390E+01 -2.7250E+02  1.2987E-02  6.6733E+01  0.0000E+00  0.0000E+00
     6    5.3927E+01 -2.7250E+02  1.2877E-02  6.6732E+01  0.0000E+00  0.0000E+00
     7    5.3463E+01 -2.7250E+02  1.2766E-02  6.6731E+01  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    7.1335E+01  7.6310E-01  9.4881E-05  1.8816E-05  4.1943E-01  1.9442E+00
     3    7.1335E+01  7.6310E-01  9.4875E-05  1.8816E-05  4.1943E-01  1.9443E+00
     4    7.1335E+01  7.6311E-01  9.4868E-05  1.8816E-05  4.1944E-01  1.9443E+00
     5    7.1335E+01  7.6312E-01  9.4862E-05  1.8815E-05  4.1945E-01  1.9444E+00
     6    7.1335E+01  7.6312E-01  9.4855E-05  1.8815E-05  4.1946E-01  1.9444E+00
     7    7.1335E+01  7.6313E-01  9.4849E-05  1.8814E-05  4.1947E-01  1.9445E+00

 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       6.578E+01   -6.397E-05  -9.370E-03    6.959E-02   1.673E+03   9.256E-05   5.570E-12   8.110E-07
      23       3.258E-01    4.633E-01  -9.370E-03    1.619E-02   8.070E+02   2.154E-05   2.759E-14   4.017E-09
      34       3.258E-01    4.633E-01  -9.370E-03    1.619E-02   8.070E+02   2.154E-05   2.759E-14   4.016E-09
      45       3.258E-01    4.633E-01  -9.370E-03    1.619E-02   8.071E+02   2.154E-05   2.759E-14   4.016E-09
      56       3.258E-01    4.633E-01  -9.370E-03    1.619E-02   8.071E+02   2.154E-05   2.758E-14   4.016E-09
      67       3.257E-01    4.633E-01  -9.370E-03    1.619E-02   8.072E+02   2.154E-05   2.758E-14   4.016E-09
      78       3.257E-01    4.634E-01  -9.370E-03    1.621E-02   8.072E+02   2.156E-05   2.758E-14   4.015E-09

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

Flow rate is nearly zero, as we would expect given that pressure drop between boundary nodes is approximately equal to the hydrostatic head.

Part 2: Add Heat Transfer to Model

- Activate Conjugate Heat Transfer



☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☑ Gravity ☐ Buoyancy Reference Node: 1: Node 1
☐ Heat Exchanger
☐ Heat Source BTU/sec
☐ Inertia ☐ Branch Angles ☐ DFLI
☐ Grid Generation ☐ Laminar

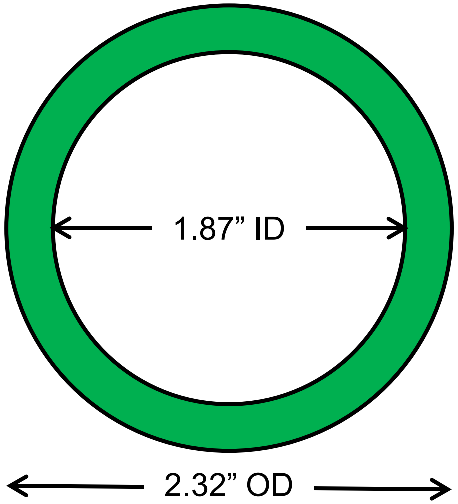
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry Relative Humidity
☐ Rotation
☐ Shear
☐ Transverse Momentum
☐ Turbopump
☑ Conjugate Heat Transfer

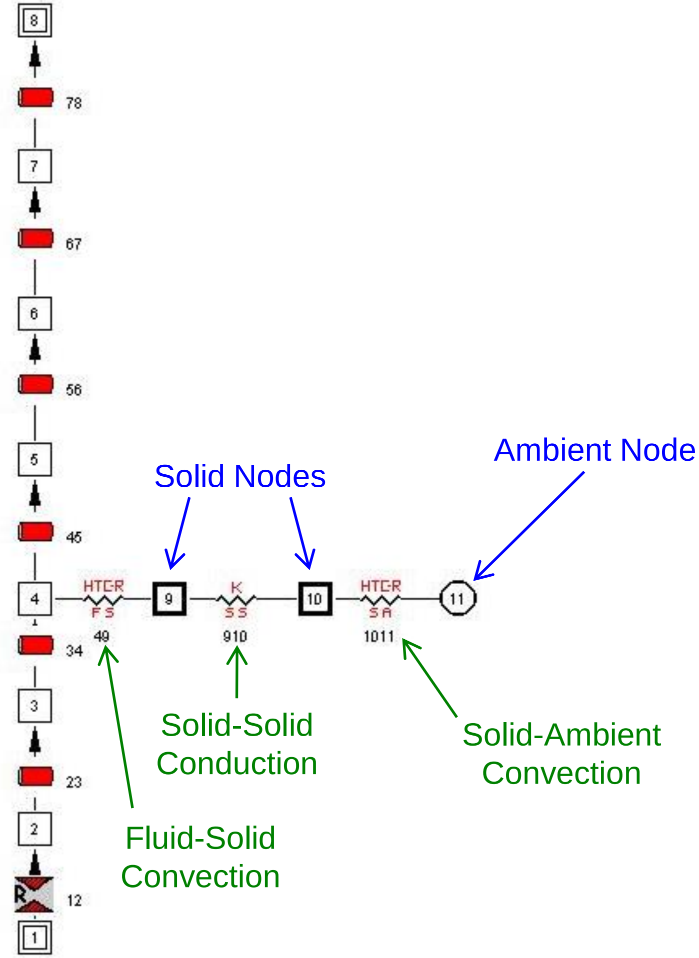
Next Node Number: 1

Part 2: Update Model on Canvas

- Allow 1’ of the pipe to be exposed to ambient.

- Remainder of pipe is well insulated.





Solid Nodes

Ambient Node

Solid-Solid Conduction

Solid-Ambient Convection

Fluid-Solid Convection

Part 2: Set Up Solid Nodes

- Exposed pipe section
  - Mass: 5.26 lb_m

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

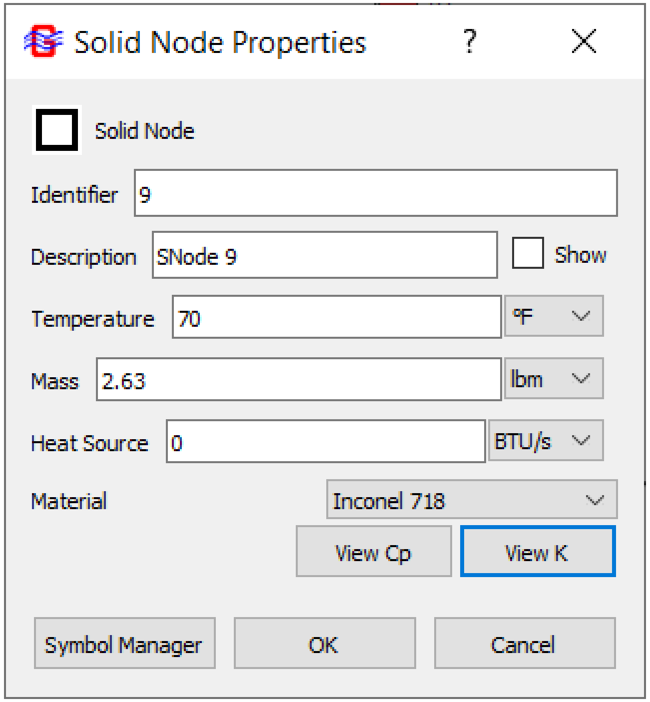
$$= \left(0.296\frac{lb_m}{in^3}\right)\left[\frac{\pi}{4}\left((2.32\ in)^2 - \ (1.87\ in)^2\right)(12\ in)\right]$$

$$m = 5.26\ lb$$

  - Inconel 718

- Nodes 9, 10
  - Guess: T = 70 °F
  - Mass: 5.26 lb_m / 2 = 2.63 lb_m
  - Inconel 718

- Ambient Node 11
  - T_{amb} = 70 °F



Solid Node Properties

Solid Node

Identifier: 9

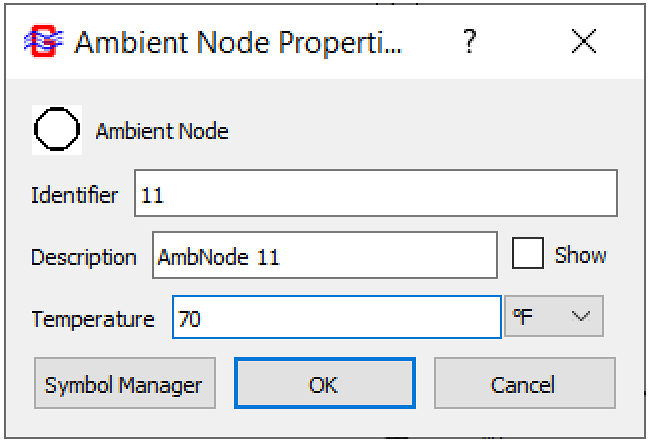
Description: SNode 9 ☐ Show

Temperature: 70 °F

Mass: 2.63 lbm

Heat Source: 0 BTU/s

Material: Inconel 718



Ambient Node Properti...

Ambient Node

Identifier: 11

Description: AmbNode 11 ☐ Show

Temperature: 70 °F

Part 2: Set Up Conductors

- Solid-Fluid Convection
  - Wetted Area:

$$A = \pi DL = \pi(1.87\text{ in})(12\text{ in}) = \mathbf{70.5\ in^2}$$

  - Select Dittus-Boelter correlation

- Solid-Solid Conduction
  - “Average” Area:

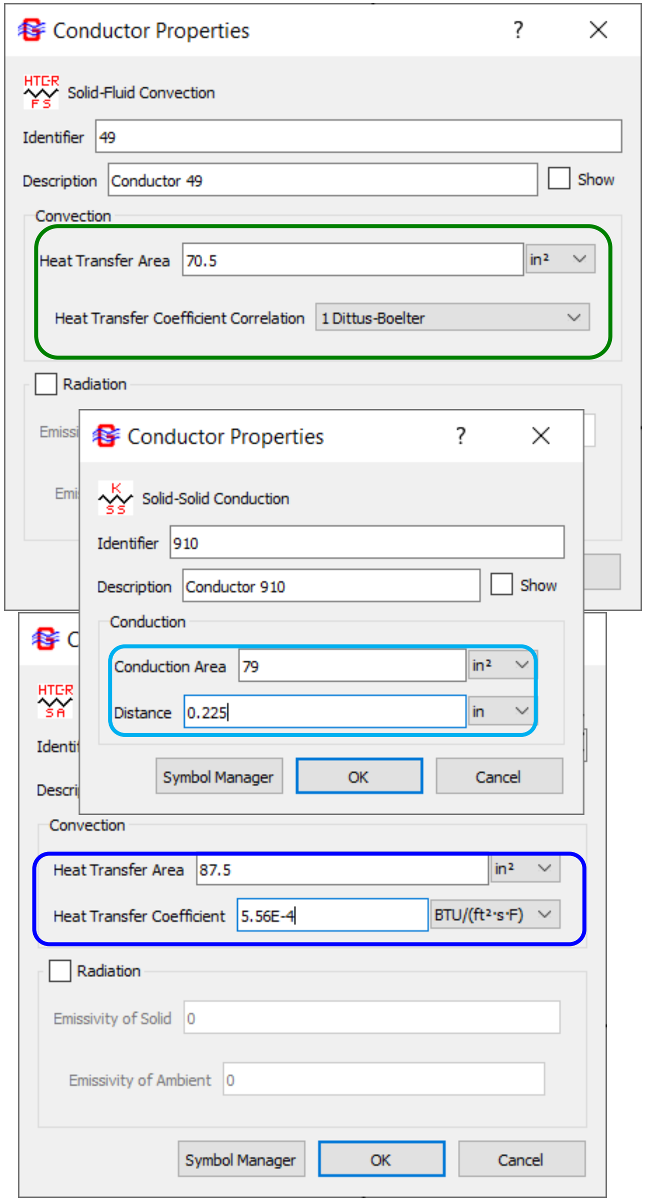
$$A = \pi D_{avg}L = \pi(2.095\text{ in})(12\text{ in}) = \mathbf{79.0\ in^2}$$

  - Distance (pipe wall thickness): **0.225 in**

- Solid-Ambient Convection
  - Exposed Area:

$$A = \pi DL = \pi(2.32\text{ in})(12\text{ in}) = \mathbf{87.5\ in^2}$$

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



Conductor Properties

Solid-Fluid Convection

Identifier: 49

Description: Conductor 49 ☐ Show

Convection

Heat Transfer Area: 70.5 in²

Heat Transfer Coefficient Correlation: 1 Dittus-Boelter

☐ Radiation

Conductor Properties

Solid-Solid Conduction

Identifier: 910

Description: Conductor 910 ☐ Show

Conduction

Conduction Area: 79 in²

Distance: 0.225 in

Symbol Manager | OK | Cancel

Convection

Heat Transfer Area: 87.5 in²

Heat Transfer Coefficient: 5.56E-4 BTU/(ft²·s·F)

☐ Radiation

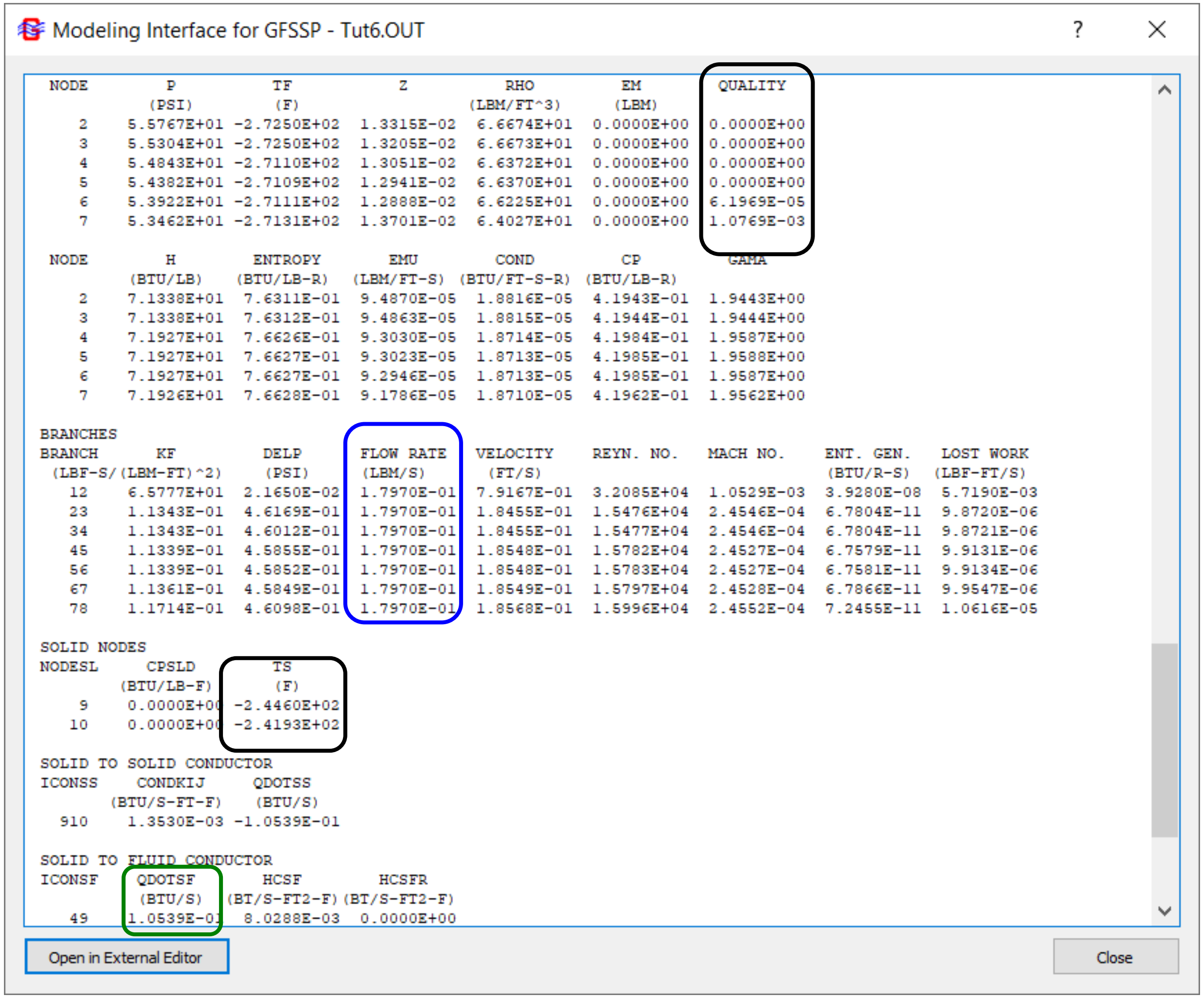
Emissivity of Solid: 0

Emissivity of Ambient: 0

Symbol Manager | OK | Cancel

Part 2: Result of LOx Model with Heat Input

- Rerun the model
  - Addition of 0.1054 BTU/s of heat has increased flow rate to **0.1797 lb/s**



```
 NODE        P          TF            Z          RHO         EM       QUALITY
           (PSI)        (F)                   (LBM/FT^3)    (LBM)
    2    5.5767E+01 -2.7250E+02  1.3315E-02  6.6674E+01  0.0000E+00  0.0000E+00
    3    5.5304E+01 -2.7250E+02  1.3205E-02  6.6673E+01  0.0000E+00  0.0000E+00
    4    5.4843E+01 -2.7110E+02  1.3051E-02  6.6372E+01  0.0000E+00  0.0000E+00
    5    5.4382E+01 -2.7109E+02  1.2941E-02  6.6370E+01  0.0000E+00  0.0000E+00
    6    5.3922E+01 -2.7111E+02  1.2888E-02  6.6225E+01  0.0000E+00  6.1969E-05
    7    5.3462E+01 -2.7131E+02  1.3701E-02  6.4027E+01  0.0000E+00  1.0769E-03

 NODE        H        ENTROPY       EMU        COND         CP         GAMA
         (BTU/LB)   (BTU/LB-R)  (LBM/FT-S) (BTU/FT-S-R) (BTU/LB-R)
    2    7.1338E+01  7.6311E-01  9.4870E-05  1.8816E-05  4.1943E-01  1.9443E+00
    3    7.1338E+01  7.6312E-01  9.4863E-05  1.8815E-05  4.1944E-01  1.9444E+00
    4    7.1927E+01  7.6626E-01  9.3030E-05  1.8714E-05  4.1984E-01  1.9587E+00
    5    7.1927E+01  7.6627E-01  9.3023E-05  1.8713E-05  4.1985E-01  1.9588E+00
    6    7.1927E+01  7.6627E-01  9.2946E-05  1.8713E-05  4.1985E-01  1.9587E+00
    7    7.1926E+01  7.6628E-01  9.1786E-05  1.8710E-05  4.1962E-01  1.9562E+00

BRANCHES
BRANCH      KF         DELP      FLOW RATE   VELOCITY    REYN. NO.   MACH NO.    ENT. GEN.   LOST WORK
 (LBF-S/(LBM-FT)^2)    (PSI)     (LBM/S)      (FT/S)                             (BTU/R-S)  (LBF-FT/S)
   12    6.5777E+01  2.1650E-02  1.7970E-01  7.9167E-01  3.2085E+04  1.0529E-03  3.9280E-08  5.7190E-03
   23    1.1343E-01  4.6169E-01  1.7970E-01  1.8455E-01  1.5476E+04  2.4546E-04  6.7804E-11  9.8720E-06
   34    1.1343E-01  4.6012E-01  1.7970E-01  1.8455E-01  1.5477E+04  2.4546E-04  6.7804E-11  9.8721E-06
   45    1.1339E-01  4.5855E-01  1.7970E-01  1.8548E-01  1.5782E+04  2.4527E-04  6.7579E-11  9.9131E-06
   56    1.1339E-01  4.5852E-01  1.7970E-01  1.8548E-01  1.5783E+04  2.4527E-04  6.7581E-11  9.9134E-06
   67    1.1361E-01  4.5849E-01  1.7970E-01  1.8549E-01  1.5797E+04  2.4528E-04  6.7866E-11  9.9547E-06
   78    1.1714E-01  4.6098E-01  1.7970E-01  1.8568E-01  1.5996E+04  2.4552E-04  7.2455E-11  1.0616E-05

SOLID NODES
NODESL     CPSLD        TS
        (BTU/LB-F)      (F)
    9    0.0000E+00 -2.4460E+02
   10    0.0000E+00 -2.4193E+02

SOLID TO SOLID CONDUCTOR
ICONSS    CONDKIJ     QDOTSS
       (BTU/S-FT-F)   (BTU/S)
  910    1.3530E-03 -1.0539E-01

SOLID TO FLUID CONDUCTOR
ICONSF    QDOTSF       HCSF        HCSFR
          (BTU/S)  (BT/S-FT2-F)(BT/S-FT2-F)
   49    1.0539E-01  8.0288E-03  0.0000E+00
```

Part 2: Efficiency of Heat Leak as Pump

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

Flow rate **P1 – P8**

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

  - Heat Input

$$\dot{Q} = ?\,\frac{\mathrm{BTU}}{\mathrm{s}}$$

  - Efficiency

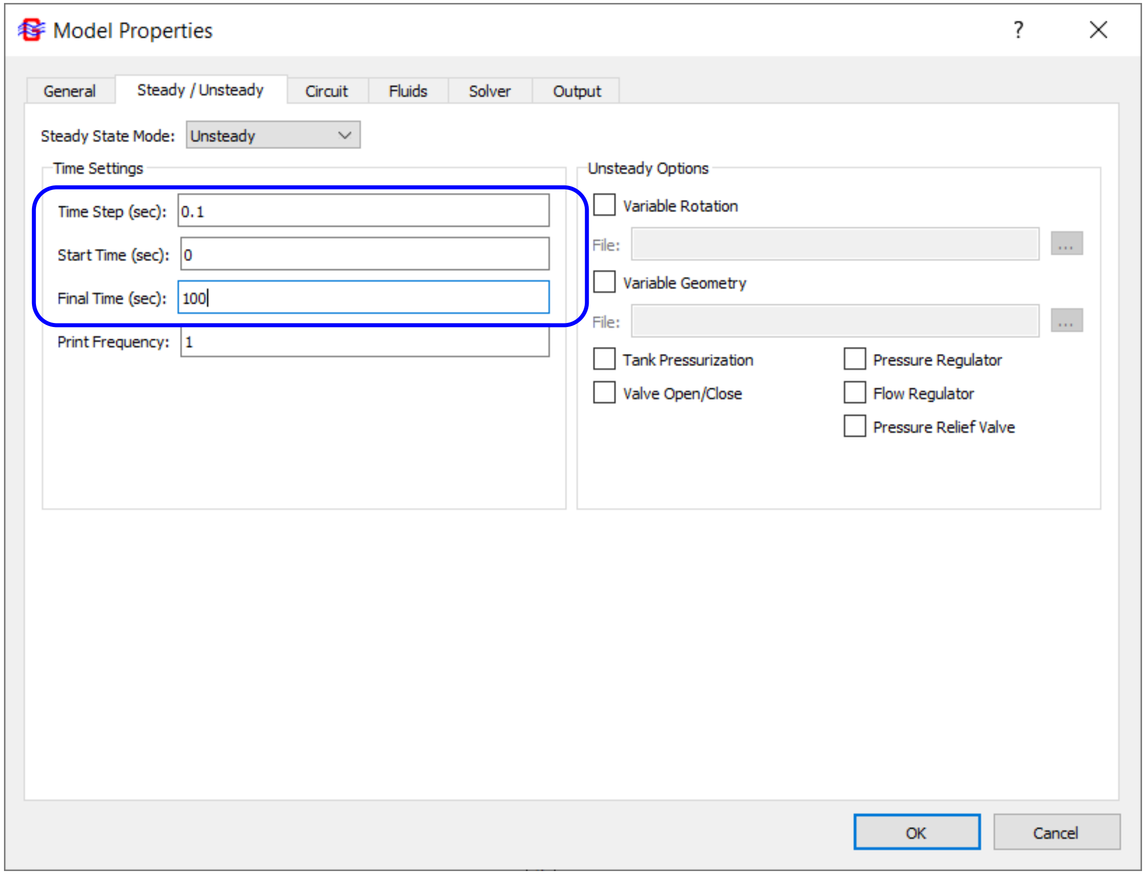
$$\eta = \frac{\dot{W}}{\dot{Q}} = ?$$

  - Carnot Efficiency

$$\eta_{ideal} = 1 - \frac{\mathrm{T_C}(°\mathrm{R})}{\mathrm{T_H}(°\mathrm{R})} = ?$$

Part 3: Add a Helium Injector

- Steady/Unsteady
  - Time step: 0.1 sec
  - Final Time: 100 sec
  - NOTE: Although we are after a steady-state solution, in numerically challenging problems it is often easier to run a transient model until it reaches steady-state.



Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

Steady State Mode: Unsteady

Time Settings

Time Step (sec): 0.1

Start Time (sec): 0

Final Time (sec): 100

Print Frequency: 1

Unsteady Options

☐ Variable Rotation

File:

☐ Variable Geometry

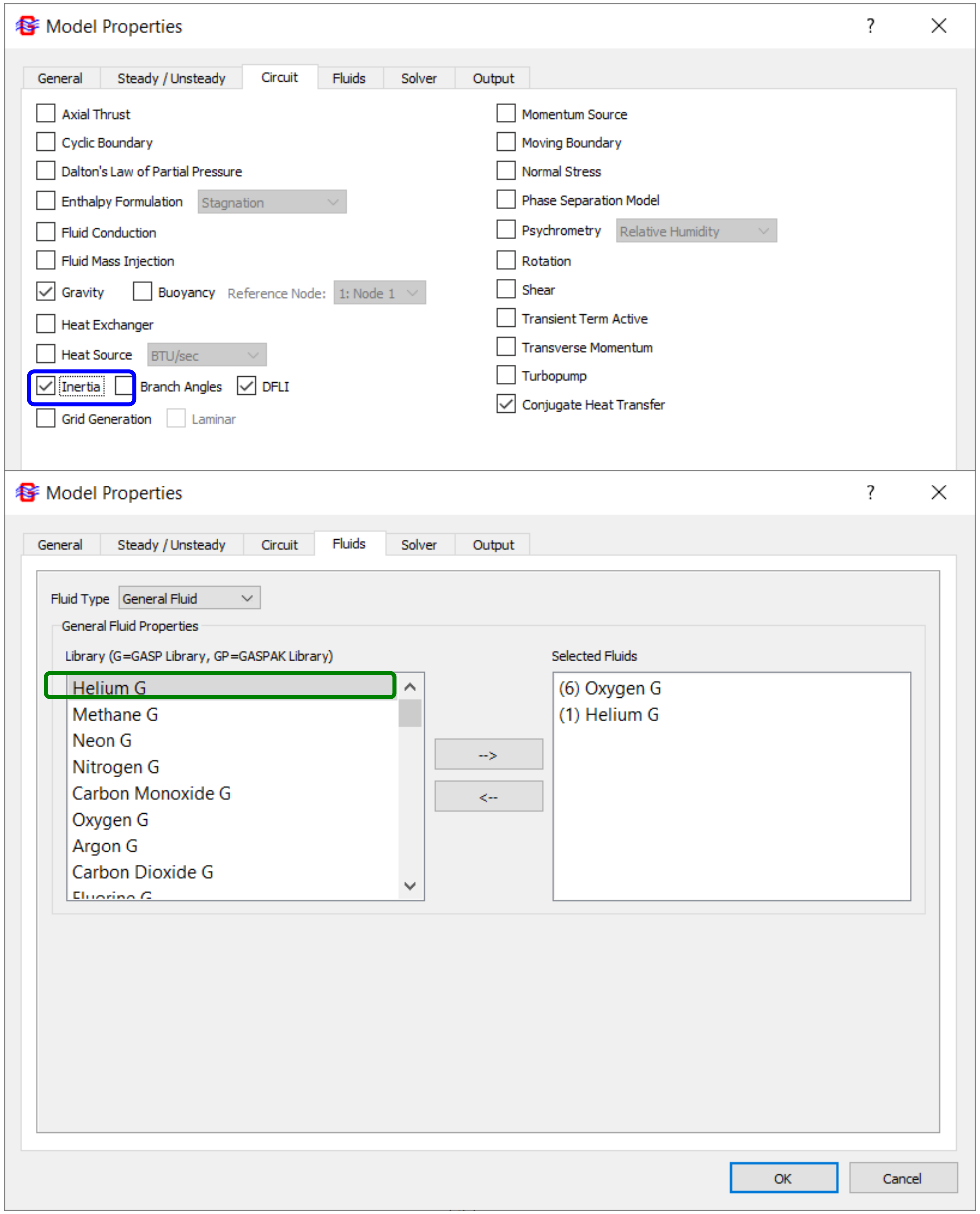
File:

☐ Tank Pressurization
☐ Valve Open/Close
☐ Pressure Regulator
☐ Flow Regulator
☐ Pressure Relief Valve

Part 3: Add Inertia Term and Helium Gas

- Circuit options
  - Inertia

- Fluid
  - Helium



Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

☐ Axial Thrust
☐ Cyclic Boundary
☐ Dalton's Law of Partial Pressure
☐ Enthalpy Formulation Stagnation
☐ Fluid Conduction
☐ Fluid Mass Injection
☑ Gravity ☐ Buoyancy Reference Node: 1: Node 1
☐ Heat Exchanger
☐ Heat Source BTU/sec
☑ Inertia ☐ Branch Angles ☑ DFLI
☐ Grid Generation ☐ Laminar
☐ Momentum Source
☐ Moving Boundary
☐ Normal Stress
☐ Phase Separation Model
☐ Psychrometry Relative Humidity
☐ Rotation
☐ Shear
☐ Transient Term Active
☐ Transverse Momentum
☐ Turbopump
☑ Conjugate Heat Transfer

Model Properties

General | Steady / Unsteady | Circuit | Fluids | Solver | Output

Fluid Type: General Fluid

General Fluid Properties

Library (G=GASP Library, GP=GASPAK Library)

Helium G
Methane G
Neon G
Nitrogen G
Carbon Monoxide G
Oxygen G
Argon G
Carbon Dioxide G
Fluorine G

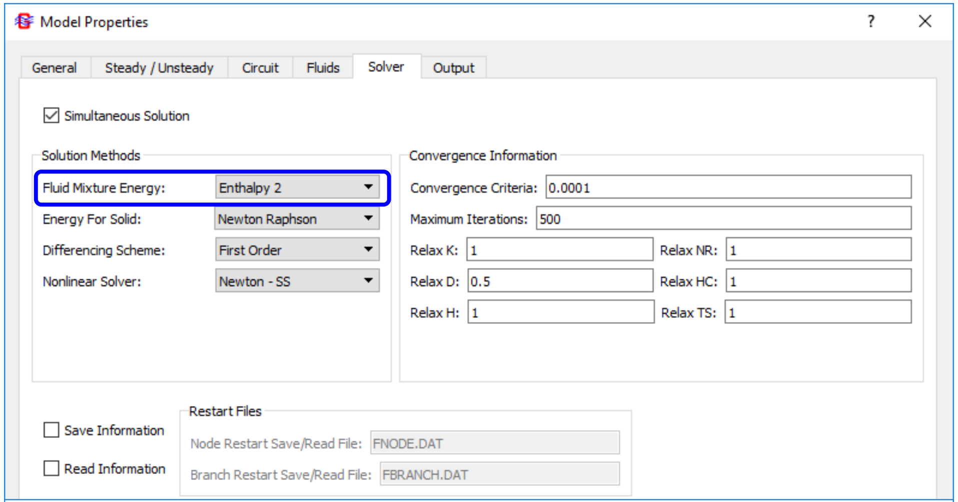
Selected Fluids

(6) Oxygen G
(1) Helium G

Part 3: Activate Mixture Enthalpy 2 Option

- Solver
  - Fluid Mixture Energy
    - Enthalpy 2

- Output
  - Winplot Data
    - Binary output



☑ Simultaneous Solution

Solution Methods

Fluid Mixture Energy: Enthalpy 2

Energy For Solid: Newton Raphson

Differencing Scheme: First Order

Nonlinear Solver: Newton - SS

Convergence Information

Convergence Criteria: 0.0001

Maximum Iterations: 500

Relax K: 1

Relax NR: 1

Relax D: 0.5

Relax HC: 1

Relax H: 1

Relax TS: 1

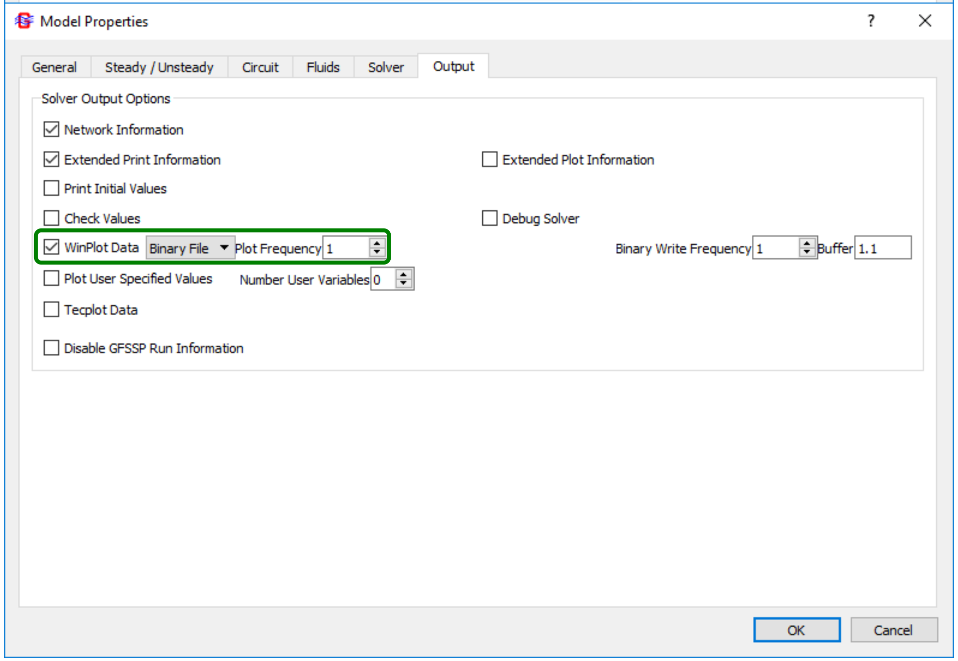
☐ Save Information

☐ Read Information

Restart Files

Node Restart Save/Read File: FNODE.DAT

Branch Restart Save/Read File: FBRANCH.DAT



Solver Output Options

☑ Network Information

☑ Extended Print Information

☐ Extended Plot Information

☐ Print Initial Values

☐ Check Values

☐ Debug Solver

☑ WinPlot Data Binary File Plot Frequency 1

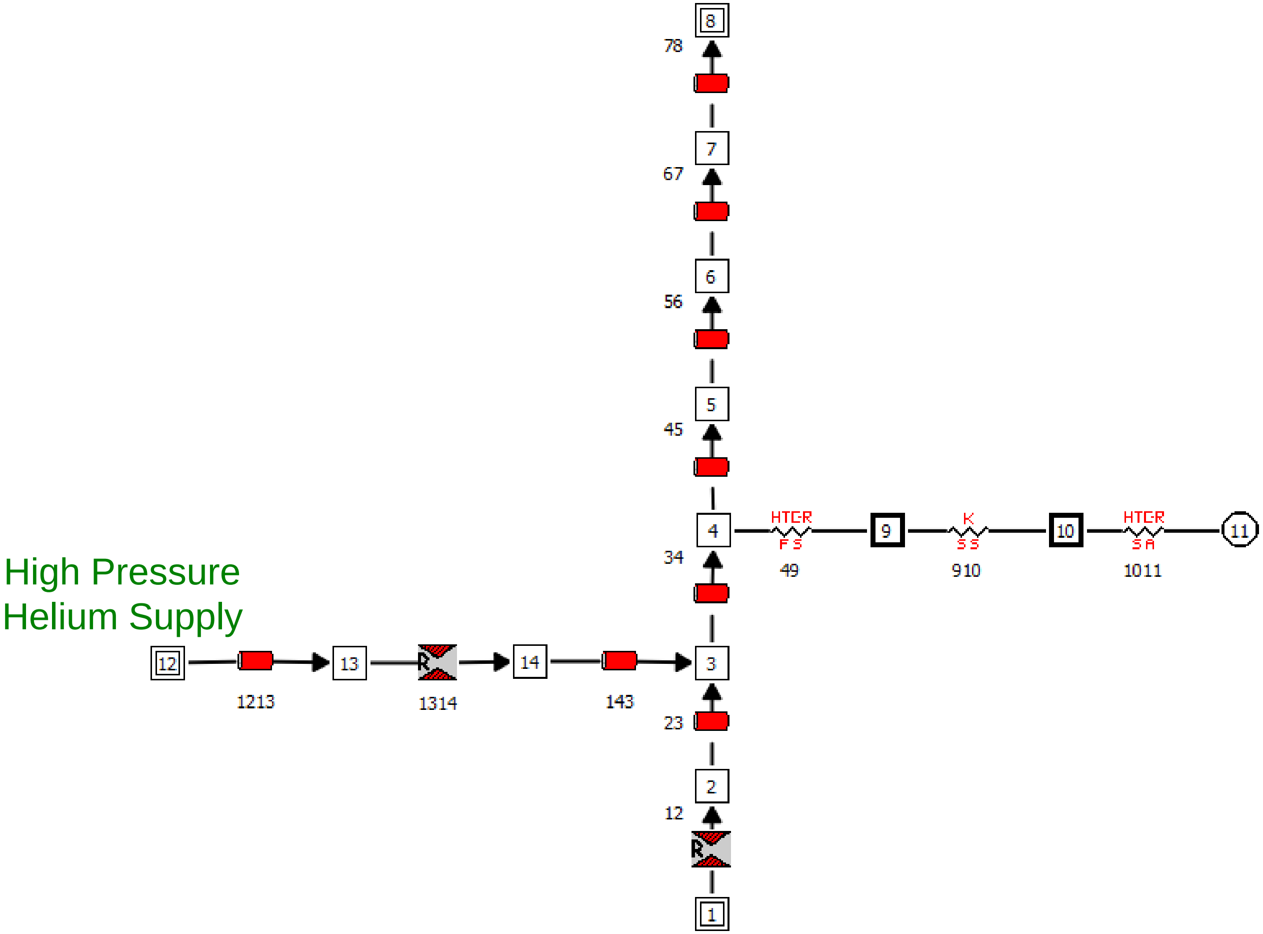
Binary Write Frequency 1 Buffer 1.1

☐ Plot User Specified Values Number User Variables 0

☐ Tecplot Data

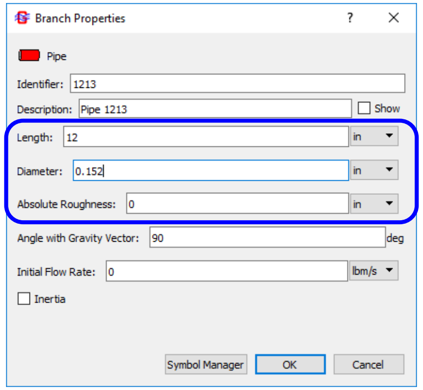
☐ Disable GFSSP Run Information

Part 3: Add Helium Injector Branches



Part 3: Set Up Helium Injector Branches

- Branch 1213: Pipe
  - L = 12 in
  - D = 0.152 in
  - Smooth pipe: ε = 0
- Branch 1314: Restriction
  - A = 0.0012566 in^2
  - C_L = 0.6
  - Check: Inertia box
- Branch 143: Pipe
  - L = 28 in
  - D = 0.152 in
  - Smooth pipe: ε = 0


Branch Properties

Pipe

Identifier: 1213

Description: Pipe 1213 ☐ Show

Length: 12 in

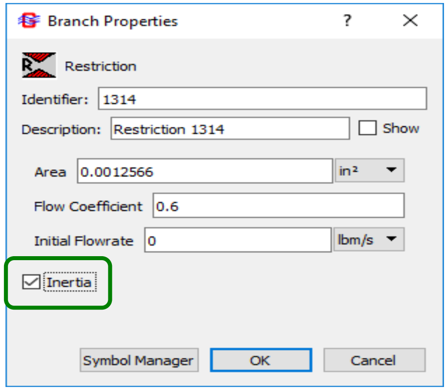
Diameter: 0.152 in

Absolute Roughness: 0 in

Angle with Gravity Vector: 90 deg

Initial Flow Rate: 0 lbm/s

☐ Inertia


Branch Properties

Restriction

Identifier: 1314

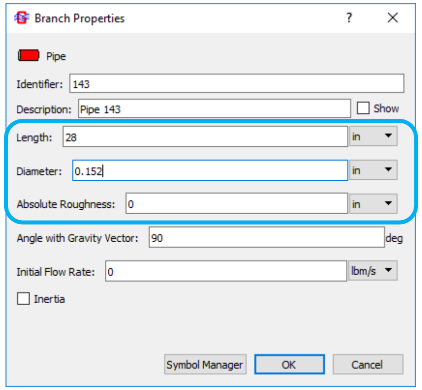
Description: Restriction 1314 ☐ Show

Area 0.0012566 in²

Flow Coefficient 0.6

Initial Flowrate 0 lbm/s

☑ Inertia


Branch Properties

Pipe

Identifier: 143

Description: Pipe 143 ☐ Show

Length: 28 in

Diameter: 0.152 in

Absolute Roughness: 0 in

Angle with Gravity Vector: 90 deg

Initial Flow Rate: 0 lbm/s

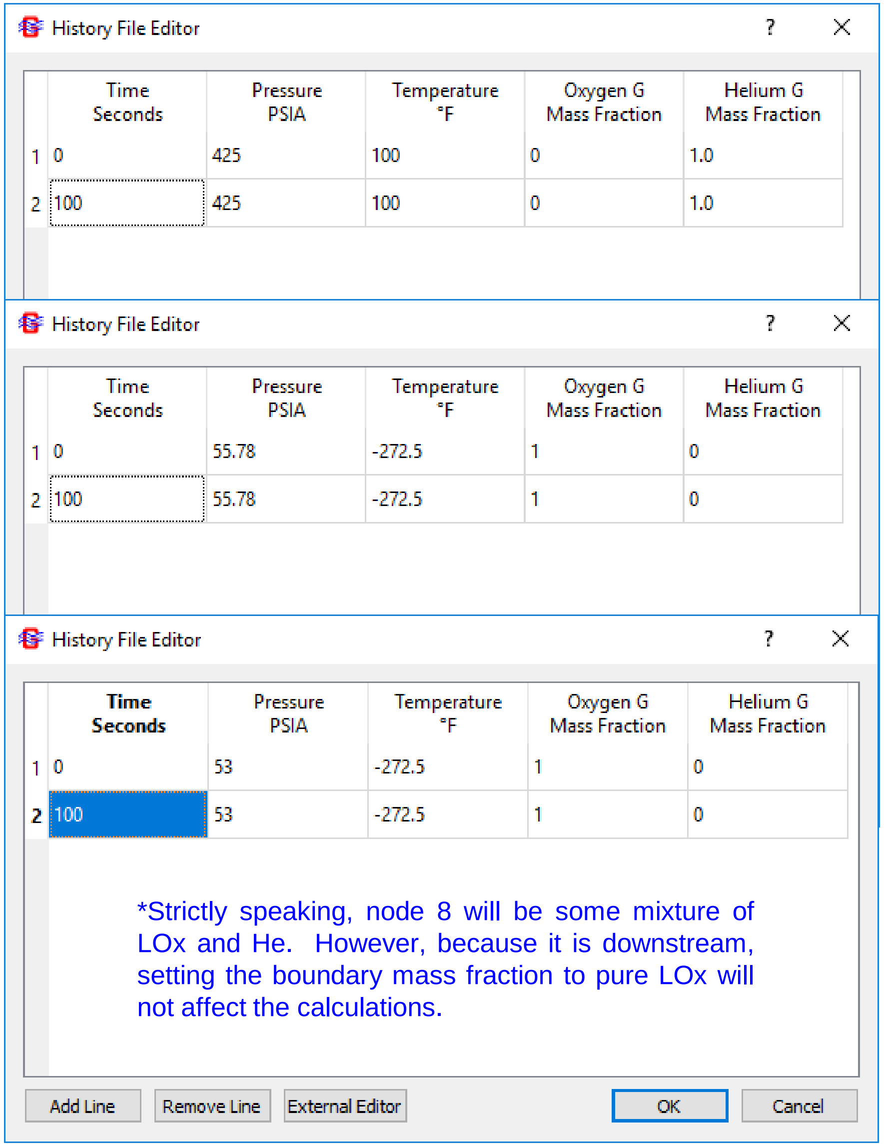
☐ Inertia

Part 3: Transient Boundary Conditions

- Node 12
  - P = 425 psia
  - T = 100 °F
  - He mass fraction: 1.0
  - LOx mass fraction: 0.0

- Node 1
  - 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*



	Time Seconds	Pressure PSIA	Temperature °F	Oxygen G Mass Fraction	Helium G Mass Fraction
1	0	425	100	0	1.0
2	100	425	100	0	1.0

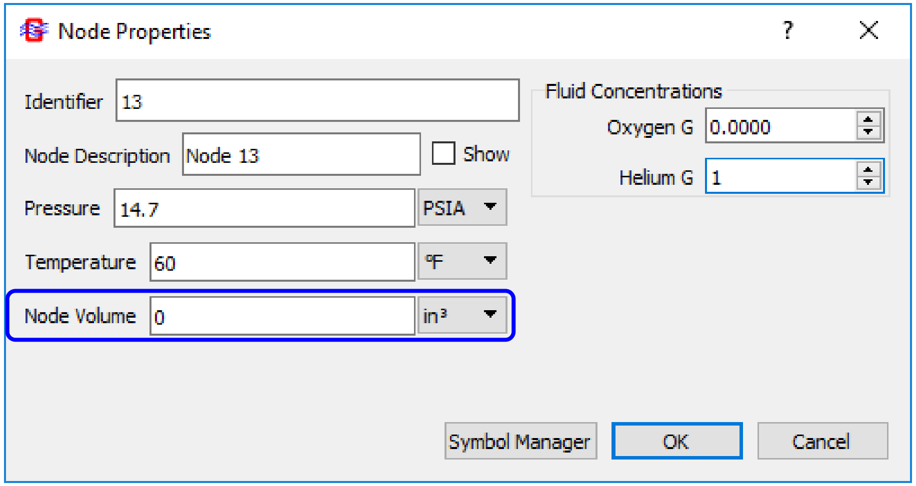
	Time Seconds	Pressure PSIA	Temperature °F	Oxygen G Mass Fraction	Helium G Mass Fraction
1	0	55.78	-272.5	1	0
2	100	55.78	-272.5	1	0

	Time Seconds	Pressure PSIA	Temperature °F	Oxygen G Mass Fraction	Helium G Mass Fraction
1	0	53	-272.5	1	0
2	100	53	-272.5	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.

Part 3: Internal Node Initial Conditions

- Nodes 13 and 14
  - P = 14.7 psia
  - T = 60 °F
  - He mass fraction: 1.0
  - LOx mass fraction: 0.0



Node Properties

Identifier: 13

Node Description: Node 13 ☐ Show

Pressure: 14.7 PSIA

Temperature: 60 ºF

Node Volume: 0 in³

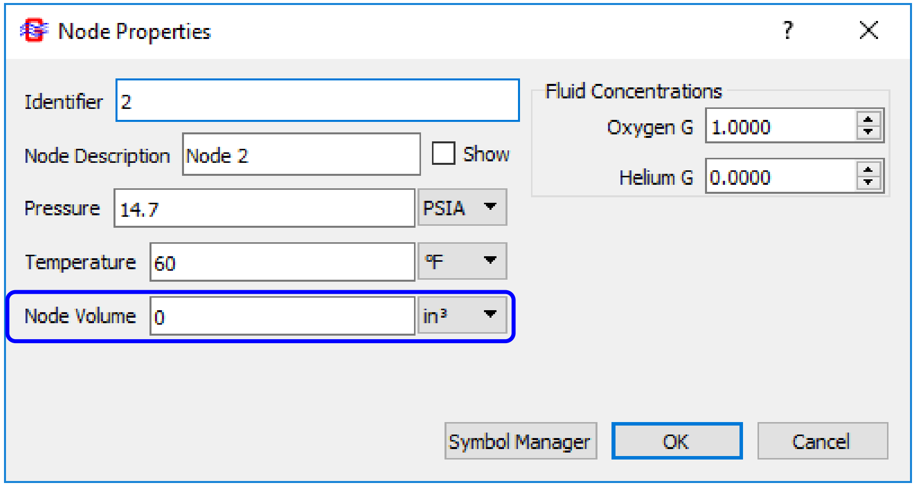
Fluid Concentrations

Oxygen G: 0.0000

Helium G: 1

- Nodes 2-7
  - 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.



Node Properties

Identifier: 2

Node Description: Node 2 ☐ Show

Pressure: 14.7 PSIA

Temperature: 60 ºF

Node Volume: 0 in³

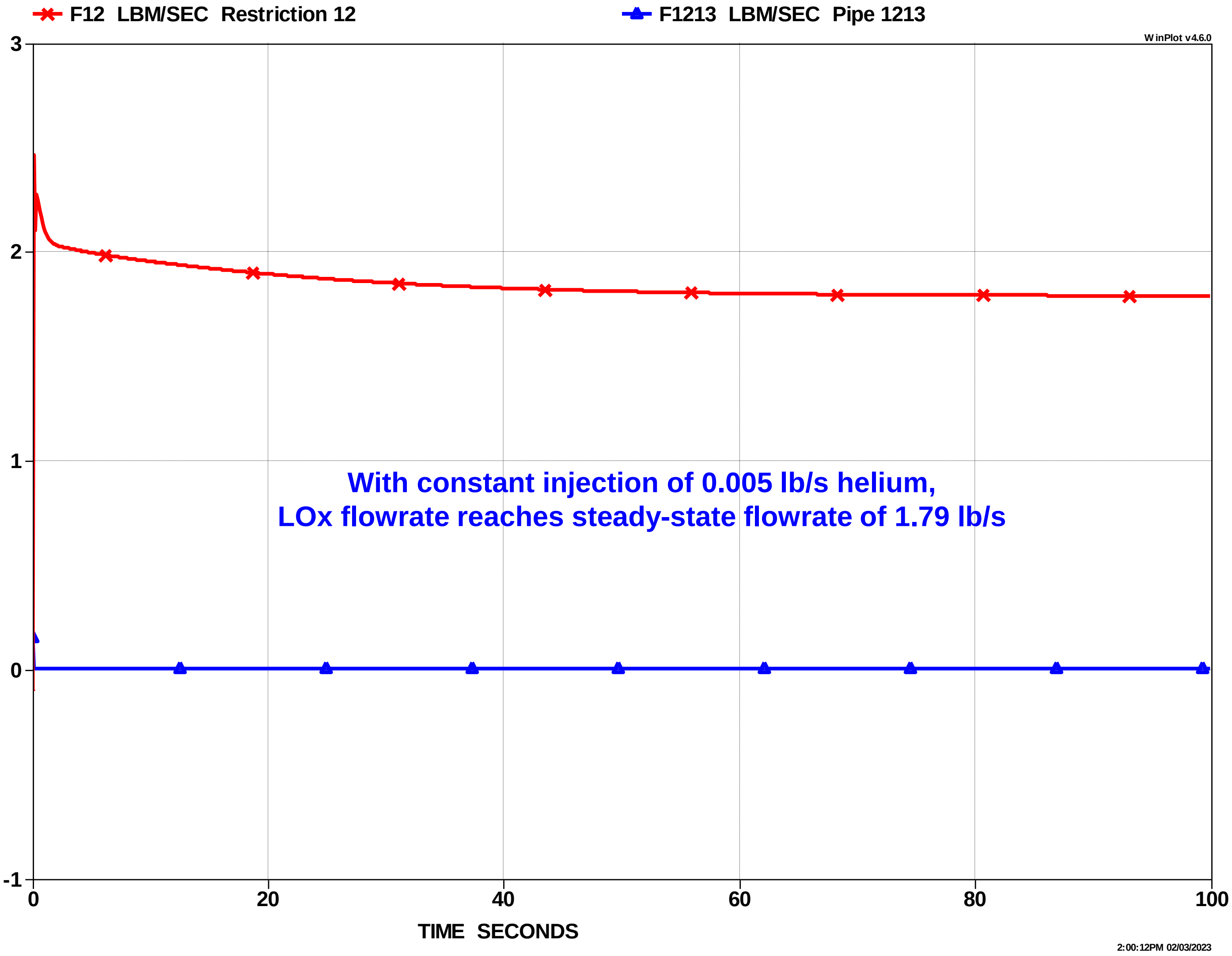
Fluid Concentrations

Oxygen G: 1.0000

Helium G: 0.0000

Part 3: Results

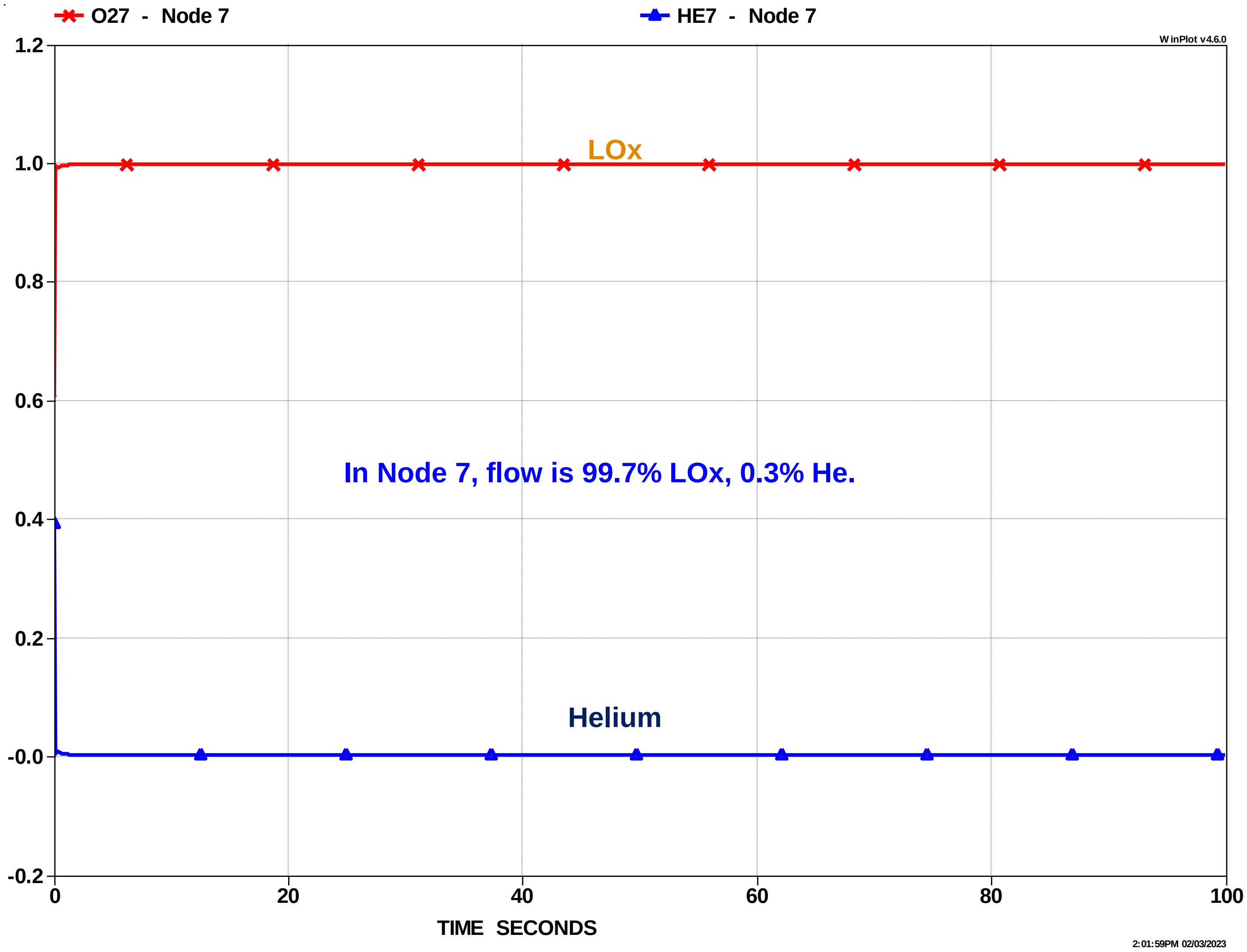
- Flowrates



With constant injection of 0.005 lb/s helium,
LOx flowrate reaches steady-state flowrate of 1.79 lb/s

Part 3: Results

- Mass Fractions

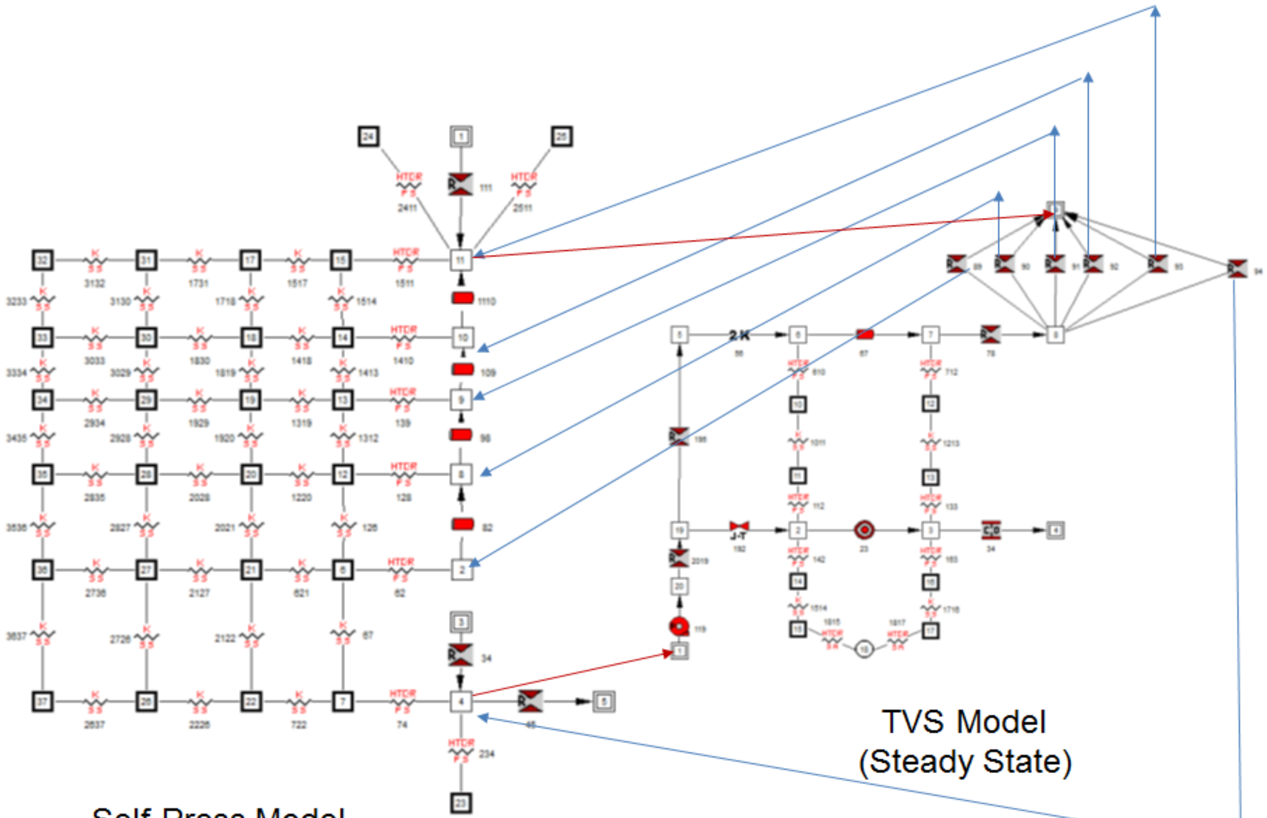


In Node 7, flow is 99.7% LOx, 0.3% He.

Part 3: Study of the Results

Model	Calculated Flowrate (lb_m/s)
Stagnant	**<0.010**
Heat Leak (0.106 BTU/s)	**0.1797**
Heat Leak and He Injection	**1.79**

Model Integration, Other Examples, and Future Developments



TVS Model
(Steady State)

Self-Press Model
(Transient)

Self-Press Model provides boundary condition to TVS Model

TVS Model provides flowrate and enthalpy of spray to Self-Press Model

CONTENT

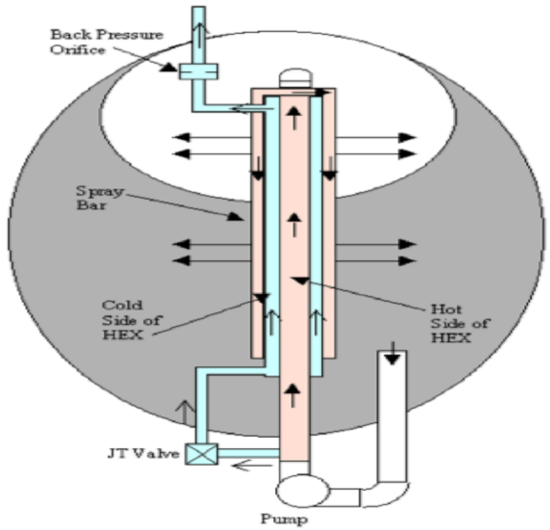
- Model Integration
- Other Example Problems
- Future Developments
  - Cavitating Venturi
  - v703
- Open Forum

Model Integration

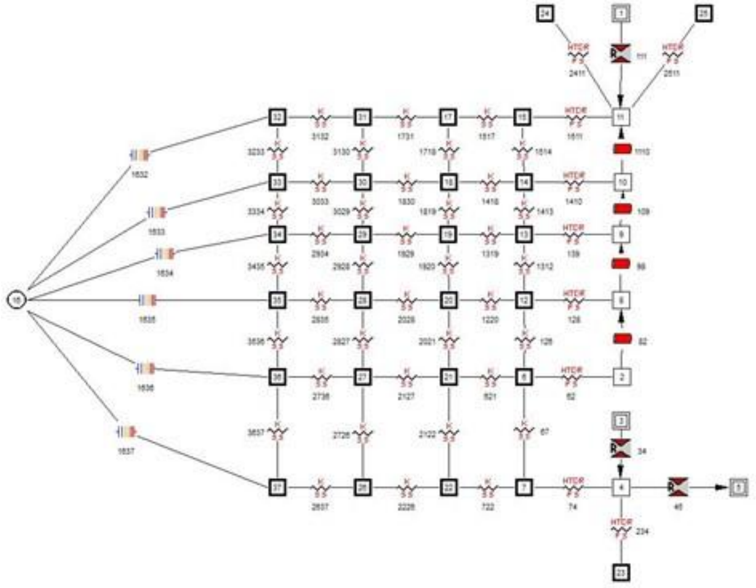
- There are two ways GFSSP can handle model integration
  - Copying and pasting nodes and branches from one model to another.
  - Integration of two models by executing another model from the existing one and data transfer between the models
- This presentation describes the second approach
  - Example 29 (Self-Pressurization of a Cryogenic Propellant Tank Due to Boil-off) demonstrates integration of two GFSSP Models (Tank Self-Press Model and Thermodynamic Vent System Model)
- The advantages of the second approach:
  - A transient system model can execute a steady-state component model
  - A “coupled” integrated model will be numerically more robust than one large integrated model
  - Works around the energy equation trade-offs that normally need to be accepted with mixture models.

Demonstration of Model Integration

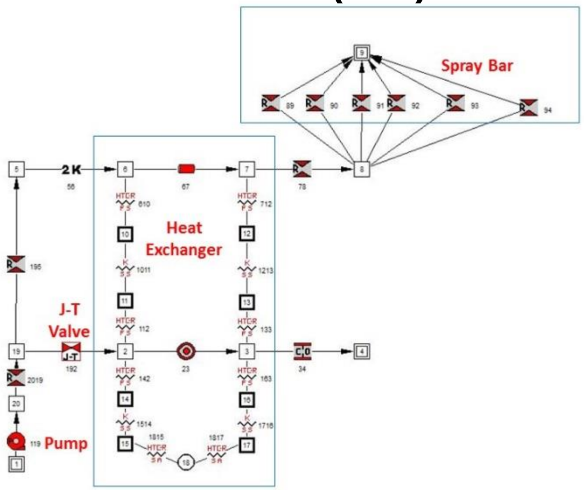
Example 29 - Self-Pressurization of a Cryogenic Propellant Tank Due to Boil-off



GFSSP Model of Self-Pressurization

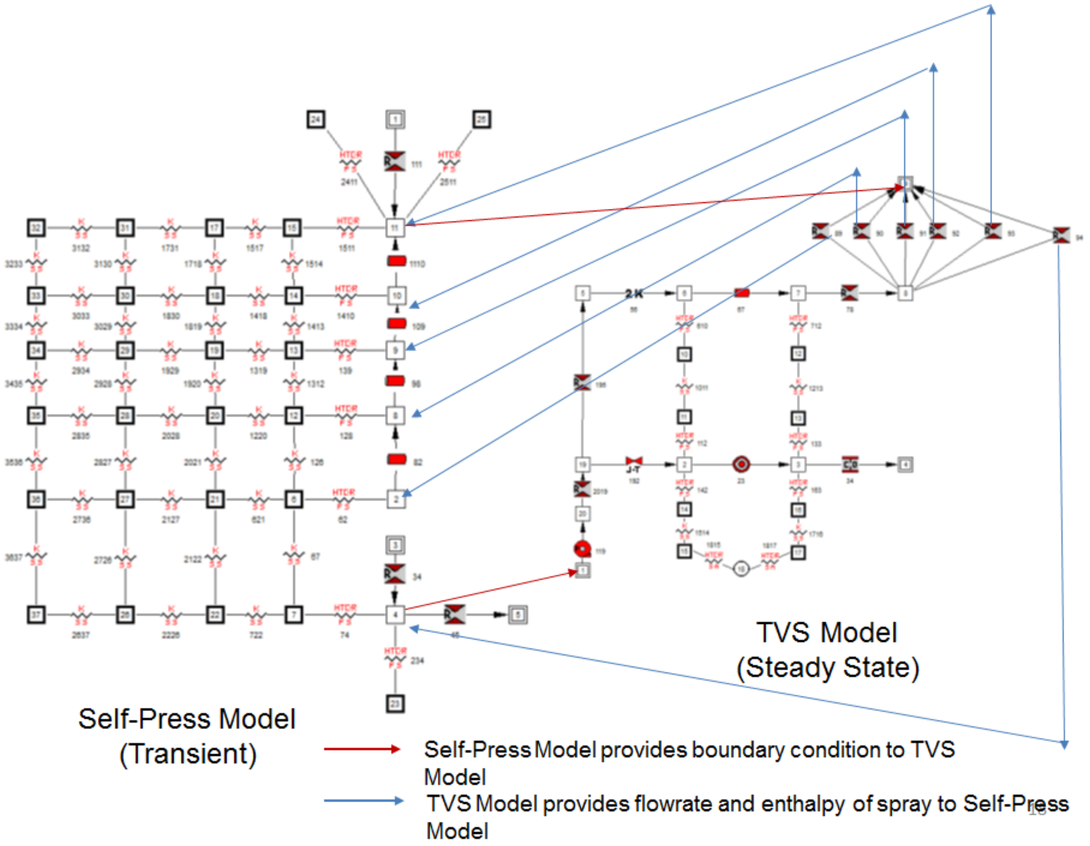


GFSSP Model of Thermodynamic Vent System (TVS)



- Self-Pressurization Model is transient
- TVS Model is steady-state
- Self-Pressurization model executes TVS model from its User Subroutine

Integrated Model



TVS Model
(Steady State)

Self-Press Model
(Transient)

→ Self-Press Model provides boundary condition to TVS Model

→ TVS Model provides flowrate and enthalpy of spray to Self-Press Model

- When ullage pressure in the Self-Press Model exceeds 20 psia, the Self-Press Model sends ullage pressure, liquid pressure, and liquid temperature to TVS model.
- The TVS Model returns the predicted flow rate and enthalpy of the sub-cooled spray back to the Self-Press model.
- When the ullage pressure in the Self-Press Model drops below 19 psia, it stops calling the TVS model until ullage pressure rises above 20 psia again.

User Subroutine for Self-Press Model

```
 CHARACTER*256 SYSC   ! Declare system command to run TVS model

 IF (SPRAY) THEN

C     SET TIME STEP TO LOWER VALUE

      DTAU = 0.01

C     GET POINTERS FOR ULLAGE NODES
      DO I = 1,6
        CALL INDEXI(NODESPR(I),NODE,NNODES,IPSPR(I))
      ENDDO

C     WRITE PRESSURE & TEMPERATURE FOR LIQUID AND ULLAGE NODE FOR TVS MODEL

      OPEN (UNIT = NUSR1, FILE = 'TO_TVS.DAT',STATUS = 'REPLACE',
     &      ACTION = 'WRITE')
      WRITE(NUSR1,*) P(IPN4),TF(IPN4),P(IPN2)
      CLOSE(NUSR1)

C     RUN TVS MODEL (Statement broken up to fit in 72 columns.)

      SYSC = '"C:\Program Files\GFSSP\solver\GFSSP.exe" TVS.DAT '
      SYSC = TRIM(SYSC) // ' TVS.DLL > DUMMY.TXT'
      CALL SYSTEM(SYSC)

C     READ DATA FROM TVS MODEL

      OPEN (UNIT = NUSR2, FILE = 'FROM_TVS.DAT', STATUS = 'OLD',
     &      ACTION = 'READ')

      READ(NUSR2,*) (SPRAYFL(I),SPRAYHL(I), I = 1,6)
      CLOSE (NUSR2)
```

TO_TVS.DAT

```
  P(IPN4)        TF(IPN4)       P(IPN2)
   (PSF)         (DEG R)         (PSF)
2735.537       37.582541      2736.116
```

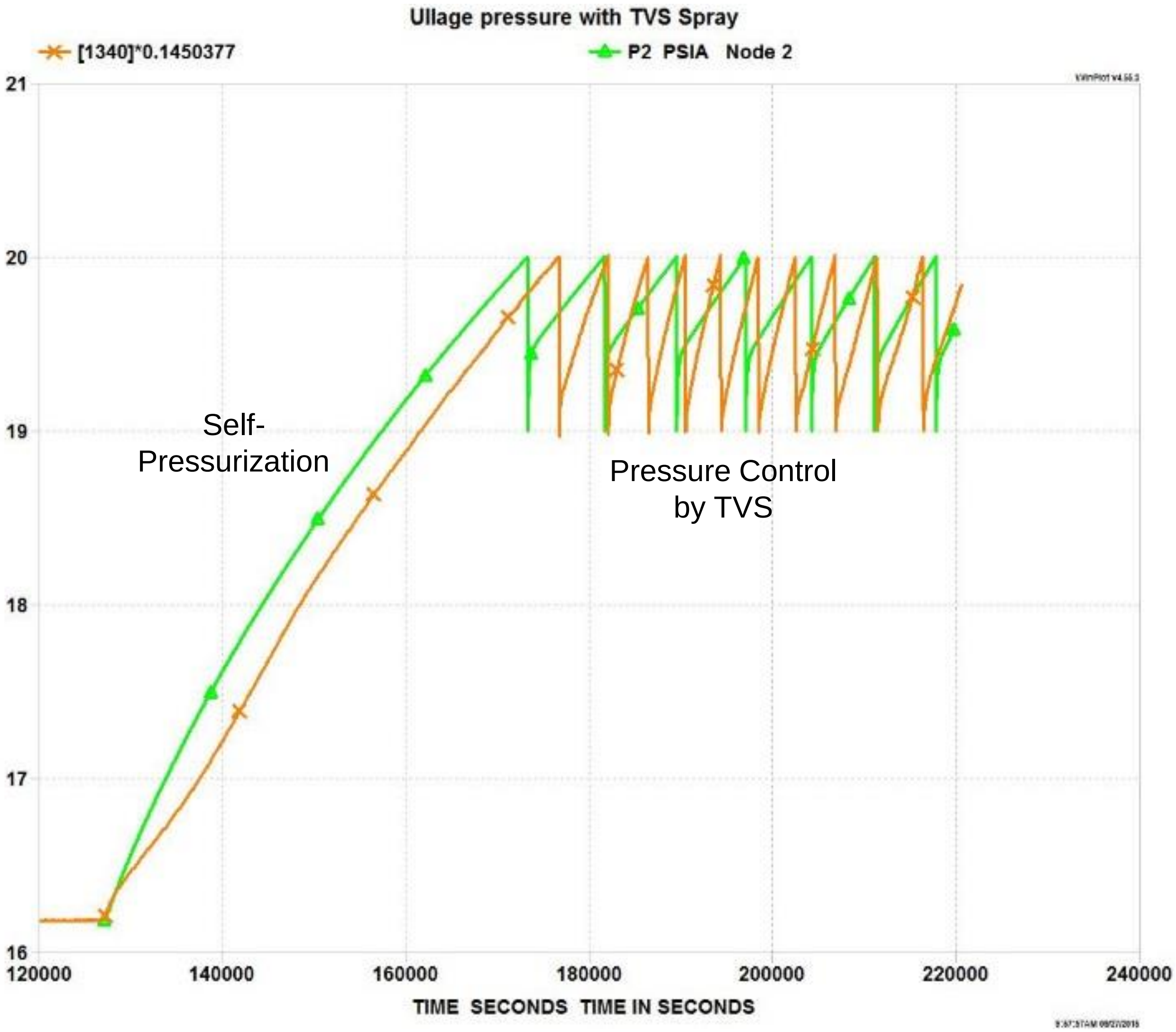
FROM_TVS.DAT

```
SPRAYFL(I)    SPRAYHL(I)
  LB/SEC        BTU/LB
0.32358E-01  -0.10786E+03
0.32358E-01  -0.10786E+03
0.32358E-01  -0.10786E+03
0.32358E-01  -0.10786E+03
0.32358E-01  -0.10786E+03
0.16190E+00  -0.10786E+03
```

If writing Free Format (*.f90), the call to the TVS model can be written on a single line:

```
CALL SYSTEM(‘ “C:\Program Files\GFSSP\solver\GFSSP.exe" TVS.DAT TVS.DLL > DUMMY.TXT’)
```

Demonstration of Model Integration of Tank Model & TVS Model (Example 29)



Self-
Pressurization

Pressure Control
by TVS

Concluding Remarks on Model Integration

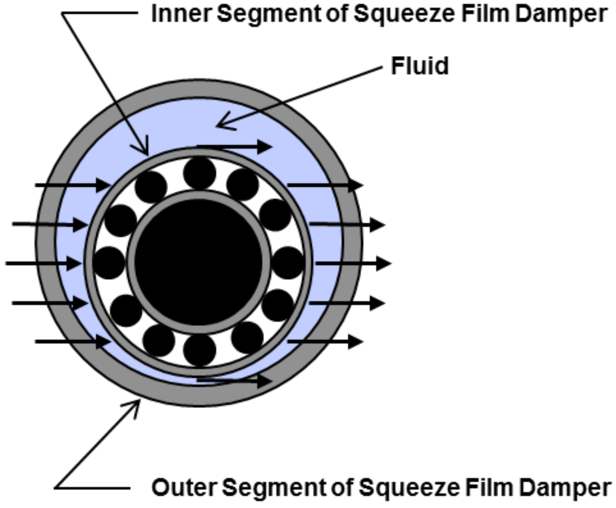
- Model integration is an effective way to model a large and complex system
- Example 29 demonstrates use of GFSSP for model integration
- There are many such applications where model integration is necessary
- Necessary user support will be provided to develop integrated model

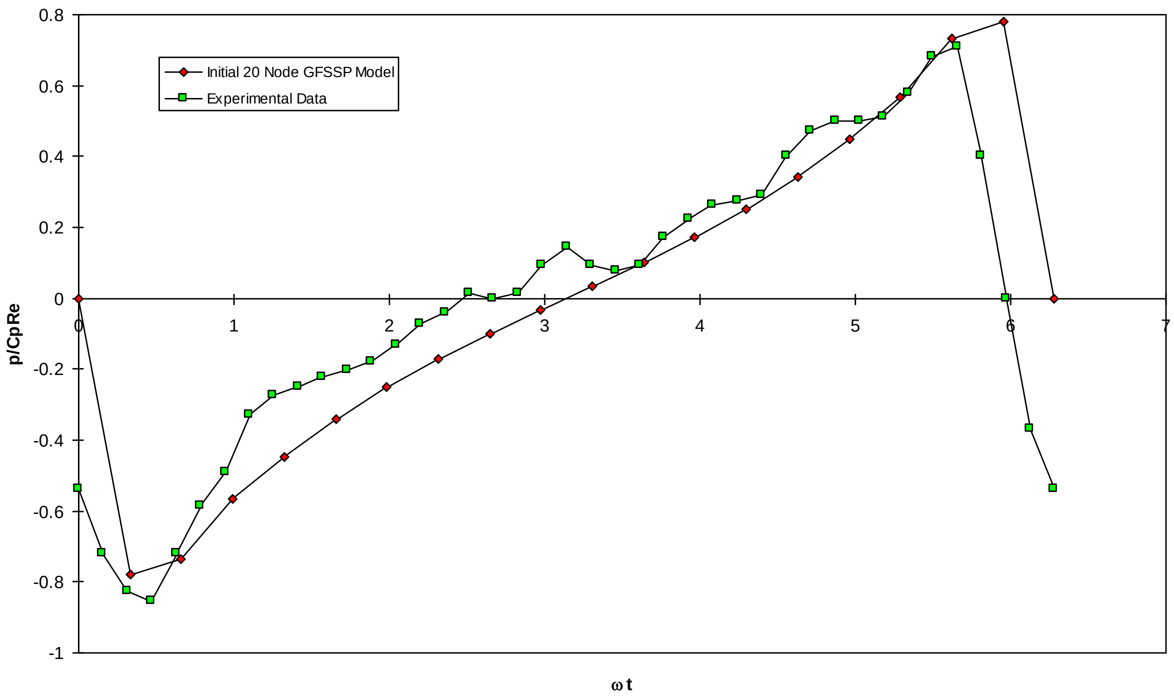
Other Examples

- In this class, we have looked at 27 of GFSSP’s 32 Example models.
- Here we briefly touch on the remaining 5 Examples.

Ex7: Flow in a Long Bearing Squeeze Film Damper

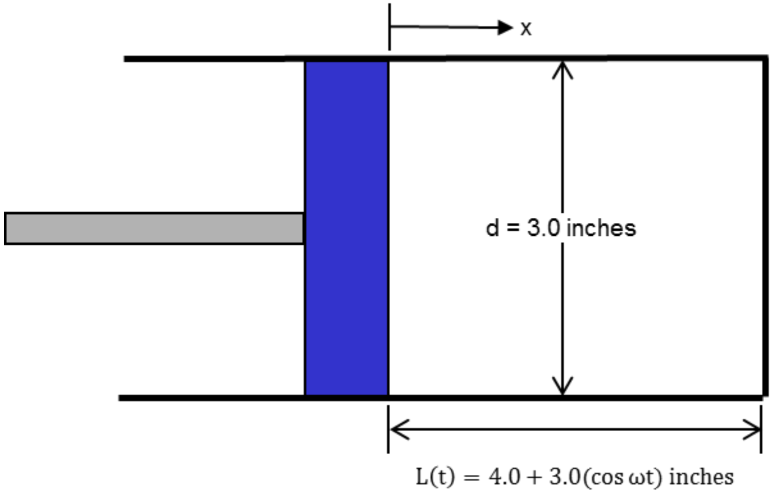
- **Ex7 Features**
  - **Moving Boundary**
  - **Comparison with Test Data**

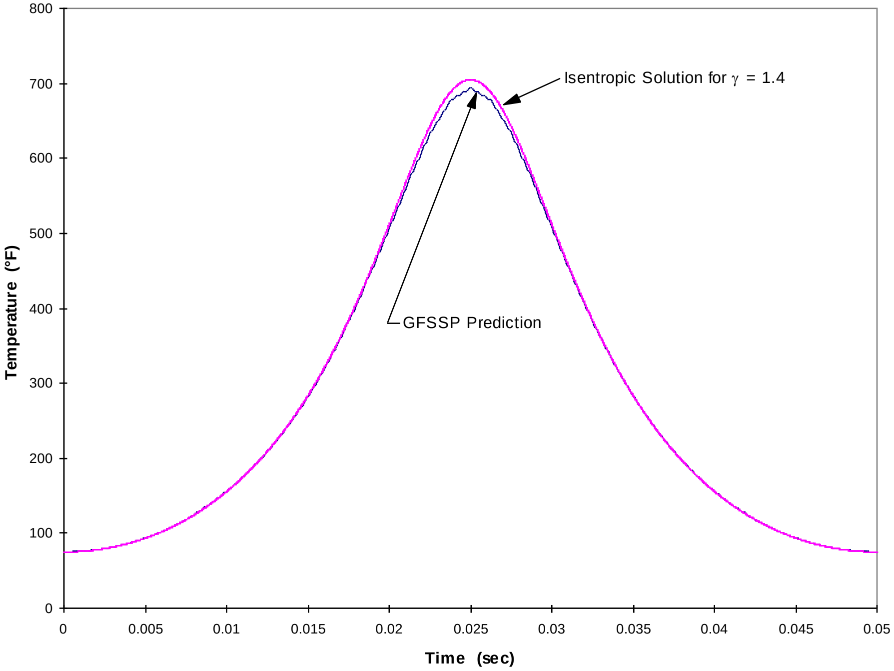




Ex9: A Reciprocating Piston-Cylinder

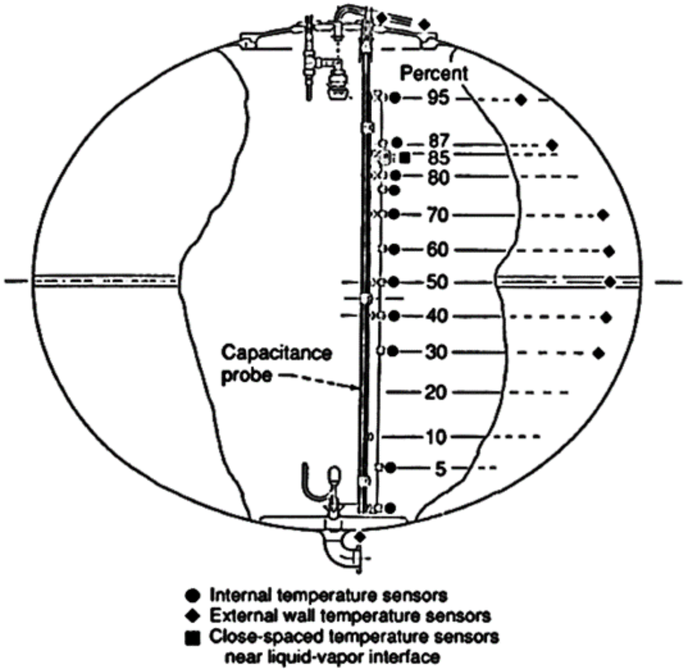
- **Ex9 Features**
    - **Variable Geometry**
    - **Moving Boundary**
    - **Comparison with Analytical Solution**

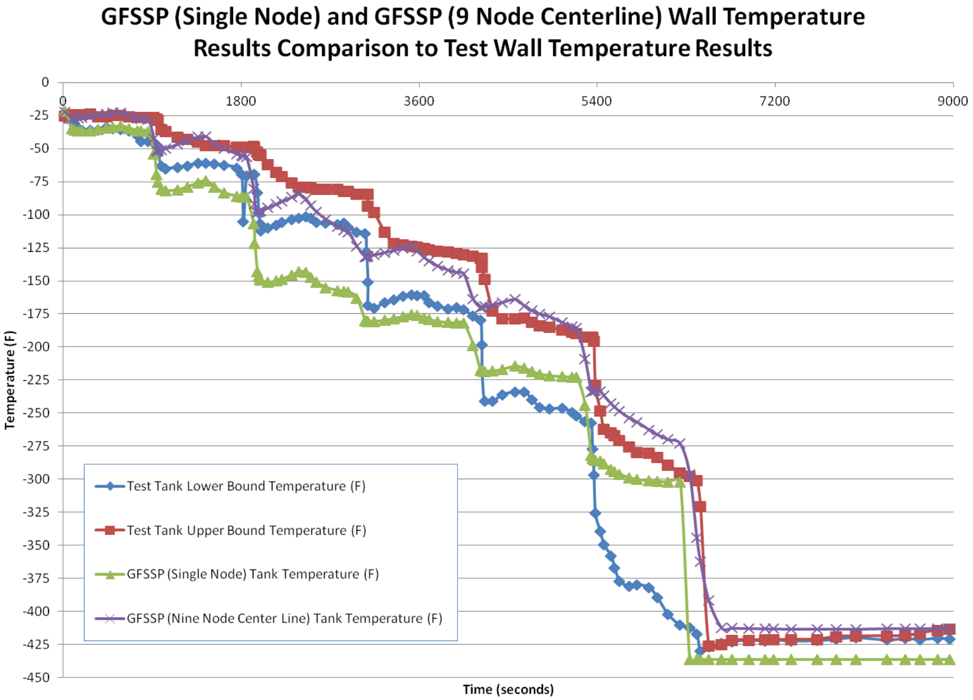




Ex28: No-Vent Tank Chill & Fill Model

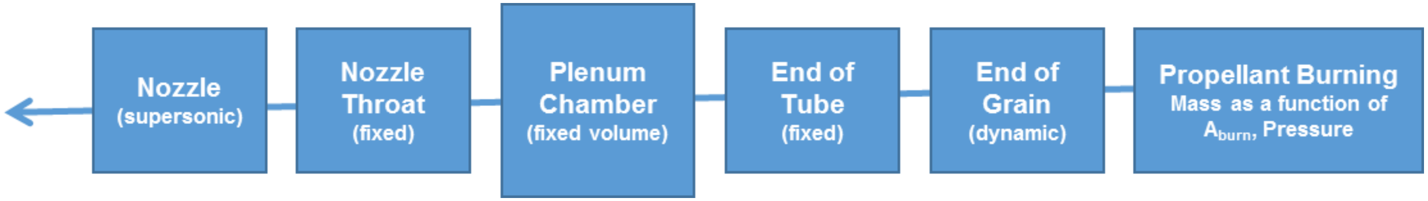
- **Ex28 Features**
  - **Conjugate Heat Transfer with Phase Change**
  - **Fixed Flowrate Option**
  - **Loading of Tank**





Ex30: Modeling Solid Propellant Rocket Motor Ballistic

- Ex30 Features
  - Propellant Burning as Mass & Energy Source
  - Flow in Rocket Nozzle
  - Thrust calculation



Nozzle
(supersonic)

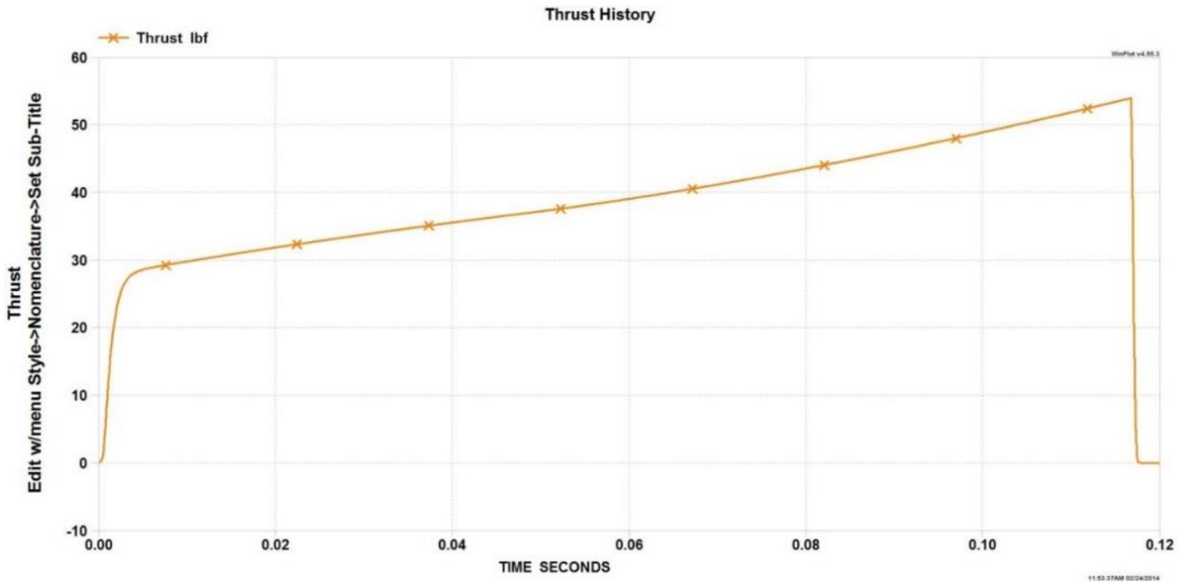
Nozzle Throat
(fixed)

Plenum Chamber
(fixed volume)

End of Tube
(fixed)

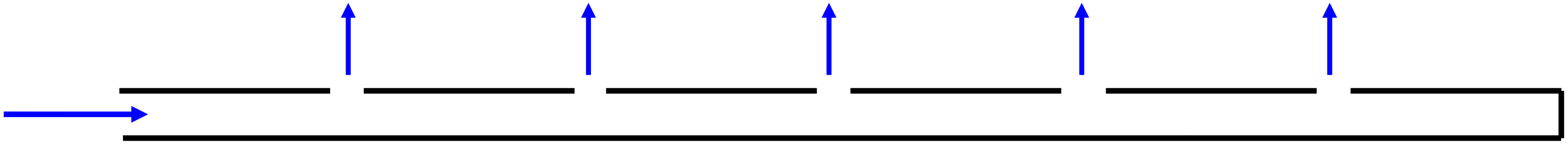
End of Grain
(dynamic)

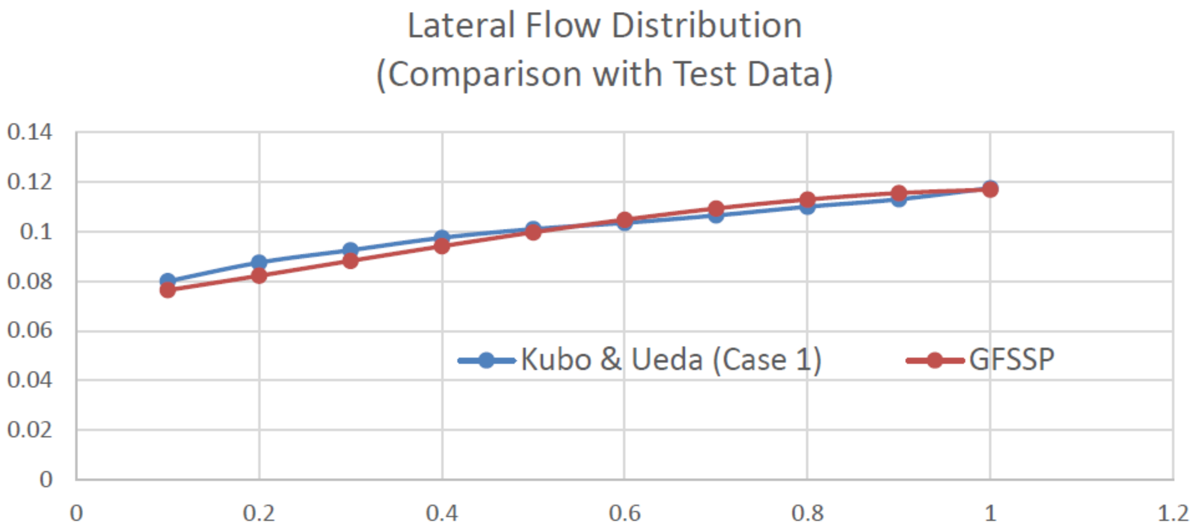
Propellant Burning
Mass as a function of A_{burn}, Pressure



Ex32: Flow Distribution in Manifold

- **Ex32 Features**
  - **Longitudinal Inertia in Momentum Equation**
    - ➢ **Calculates Flow Distribution in a Dividing Flow Manifold**





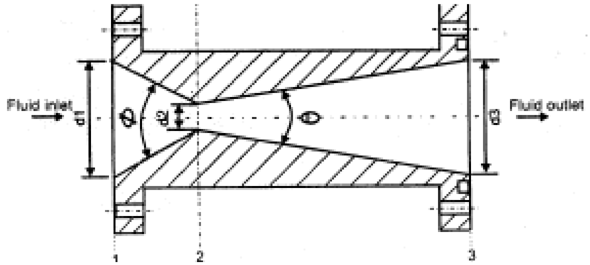
Future Developments

- Cavitating Venturi
- v703

Modeling Cavitating Venturi

What is a Cavitating Venturi ?

A venturi operating with a throat pressure equal to the vapor pressure of the fluid corresponding to the inlet temperature is called a ‘cavitating venturi’

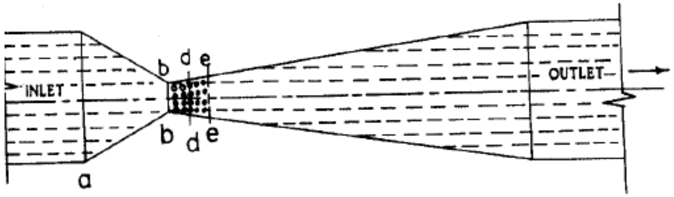


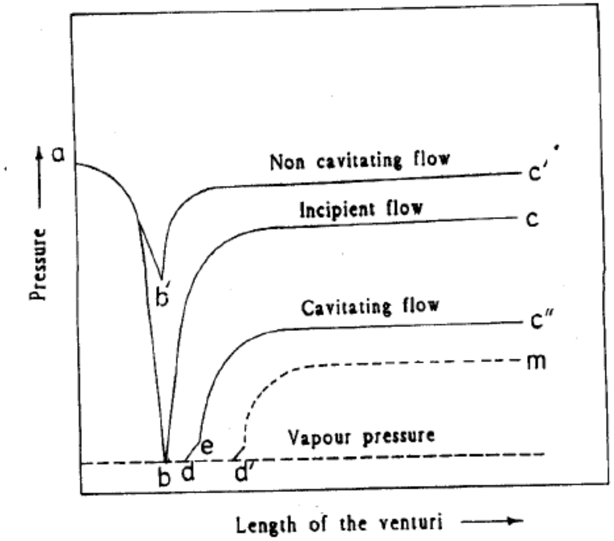
Use of Cavitating Venturi

- Propellant flow and mixture ratio in the combustion chamber is controlled by a cavitating venturi
- It maintains constant propellant flowrate for fixed inlet conditions (pressure and temperature) for a wide range of outlet pressure
- During ignition, it maintains constant flowrate while pressure in the combustion chamber is building from ambient condition
- During stable combustion, it maintains a constant flowrate while pressure fluctuates due to combustion instability

Flow Characteristics

- Pressure decreases in the converging section and increases in the diverging section
- With decrease of downstream, pressure at throat reaches vapor pressure (incipient cavitation)
- With further reduction of downstream pressure, two phase condition extends (cavitating flow)
- Vapor bubble collapses further downstream and flow becomes single phase





Flow Rate Calculation

- Flow rate through a cavitating venturi is calculated from the following equation:

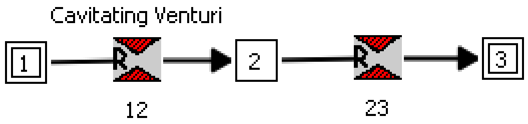
$$\dot{m} = C_d A_t \sqrt{2\rho g_c (p_{in} - p_{sat})}$$

- In subroutine SORCEF, the momentum equation of a restriction option has been replaced by the above equation
- It may be noted that a cavitating venturi will not cavitate if:

A) Inlet temperature is above critical temperature

B) The ratio of downstream to upstream pressure is greater than a critical ratio close to 0.8

Comparison of Predicted Choked Flowrate with Bernoulli Model (Fluid : Hydrogen)



P_{INLET} (PSIA)	T_{INLET} (R)	P_{SAT} (PSIA)	ρ_{INLET} (LBM/FT^3)	A_{THROAT} (IN^2)	C_D	$\dot{m}$ (GFSSP) (LB/S	$\dot{m}$ (BERNOULLI) (LB/S)
413	46.3	55.28	4.265	0.0113	0.9	0.2654	0.266
601	46.3	55.41	4.375	0.0113	0.9	0.3322	0.332
977	46.3	55.41	4.558	0.0113	0.9	0.4407	0.441
1381	46.3	55.41	4.715	0.0113	0.9	0.5375	0.537

Comparison of predicted choked flowrate with experimental data of Ghassemi et al (Fluid : Water)

P_{INLET} (Bar)	P_{OUTLET} (Bar)	Venturi Dia (mm)	C_D	$\dot{m}$ (Test) (KG/S	$\dot{m}$ (GFSSP) (KG/S)
20	0.774	5	0.94	1.2	1.167
20	13.821	5	0.94	1.2	1.167
15	0.894	2.5	0.94	0.28	0.253
15	8.126	2.5	0.94	0.28	0.253

In GFSSP, Cavitating Venturi can be modeled with a restriction option modified by user subroutine SORCEF

Use of Subroutine SORCEF to model Cavitating Venturi

Subroutine SORCEF performs the following functions:

1. Define Critical pressure ratio of downstream to upstream pressure for choked flow (PCRIT), Flow coefficient for choked flow (CLCAV), and Flow coefficient for non-choked flow (CLNOCAV)
2. Identify the cavitating branch in the flow network to deactivate the restriction option and introduce flowrate equation of cavitating venturi
3. Obtain the saturation pressure at upstream temperature
4. Check Critical Pressure Ratio to determine if flow is choked or not
5. If flow is not choked, momentum equation of restriction option is restored.
6. If flow is choked, flowrate equation of cavitating venturi is introduced

Listing of Subroutine SORCEF

```
C ADD CODE HERE
      LOGICAL FIRST
      DATA FIRST/.TRUE./
      DATA PRCRIT/0.8/
      DATA CLCAV/0.94/
      DATA CLNOCAV/0.6/

C CAVITATING VENTURI FLOWRATE EQUATION HAS BEEN INCORPRATED IN BRANCH 12

      IF (IBRANCH(I).EQ. 12) THEN

C OBTAIN SATURATION PRESSURE AT UPSTREAM TEMPERATURE

         NUMUP = IBRUN(I)
         NUMDN = IBRDN(I)
         CALL INDEXI(NUMUP, NODE, NNODES, IPUP)
         CALL INDEXI(NUMDN, NODE, NNODES, IPDN)
         PU = P(IPUP)
         PD = P(IPDN)
         PRATIO = PD / PU
         RHOU = RHO(IPUP)
         I_NFLUID = NFLUID(1)
         Z_T = TF(IPUP)
         Z_XV = 0.0
         CALL PROPS_TSATX(I_NFLUID, Z_P, Z_T, Z_RHO, Z_H, Z_CP, Z_CV,
     +                    Z_S, Z_GAMMA, Z_MU, Z_K, I_KR, Z_XV,
     +      Z_RHOL, Z_HL, Z_CPL, Z_CVL, Z_SL, Z_GAMMAL, Z_MUL, Z_KL,
     +      Z_RHOV, Z_HV, Z_CPV, Z_CVV, Z_SV, Z_GAMMAV, Z_MUV, Z_KV)
```

Define Parameters

Check for cavitating venturi branch

Obtain Saturation Pressure

```
         IF (Z_RHO .EQ. 0.0 .OR. PRATIO .GE. PRCRIT) THEN
            IF (FIRST) THEN
               IF (Z_RHO .EQ. 0.0) PRINT *, 'Inlet temperature in ',
     +           'cavitating venturi is above critical temperature'
               IF (PRATIO .GT. PRCRIT) PRINT *, 'Pressure ratio ',
     +           'across cavitating venturi is greater than critical'
               PRINT *, 'Cavitating venturi is treated as ',
     +                  'restriction option'
               FIRST = .FALSE.
            END IF

C  IF PRATIO >= PRCRIT, TREAT BRANCH 12 AS RESTRICTION OPTION

            TERM1 = 0.0
            TERM2 = (PU - PD) * AREA(I)
            BRPR1(I) = CLNOCAV
            CALL KFACT2(BRPR1(I), AREA(I), RHOU, AK(I))
            TERM4 = AK(I) * AREA(I) FLOWR(I) * ABS(FLOWR(I))

         ELSE

C IF PRATIO < PRCRIT, TREAT BRANCH 12 AS CAVITATING VENTURI

            TERM1 = FLOWR(I)
            PTHROAT = Z_P
            BRPR1(I) = CLCAV
            FACT1 = BRPR1(I) * AREA(I)
            FACT2 = SQRT(2.0 * RHOU * GC * (PU - PTHROAT))
            TERM2 = FACT1*FACT2
            TERM4 = 0.0

         END IF
      END IF

      RETURN
      END
```

Check for Critical Pressure Ratio and Critical Temperature

Use Restriction Option for Non-cavitating venturi

Use flowrate equation for cavitating venturi

Concluding Remarks on Modeling Cavitating Venturi

- Cavitating Venturi option will be made available in future GFSSP release
- In version 702, cavitating venturi can be modeled through user subroutine
- Predicted flow rates for choked flow in a cavitating venturi compare well with test data
- Users may need to find appropriate flow coefficients for a non-cavitating venturi
- Work is in progress to better model a non-cavitating venturi

Future Developments Planned for v703

- Add Cavitating Venturi branch option
- Fluid properties from Refprop DLL?
- Add roughness to Non-Circular Ducts?
- Multiple iterative (Type 1) regulators in one model?
- Add boiling correlations to library?

Thank You !

Questions for Open Forum?