C&R TECHNOLOGIES

Sinaps*Plus*®
Pipe Prebuilt Model

User Documentation
September 22, 2003; Rev. 4.6

C&R Technologies, Inc.
9 Red Fox Lane
Littleton, Colorado 80127-5710
Voice: (303) 971-0292
Fax: (303) 971-0035

info@crtech.com
www.crtech.com

1.0   Introduction

This document explains the use of the prebuilt Sinaps*Plus*® model of flow through a pipe (adiabatic or diabatic) given pressure drop or given mass flowrate. Various working fluids may be used, and the fluid may be incompressible, compressible, and may even boil or condense. This software and other supporting documentation are freely distributed. To run this model, you must:

1. Install Sinaps*Plus* Version 4.6 or later for your machine (PC, Sun, or HP).
2. Install the pipe prebuilt package in the location where Sinaps*Plus* was installed, per the instructions in the generic prebuilt document (available separately).

A basic understanding of SINDA/FLUINT and Sinaps*Plus* is necessary to utilize this model. Tutorials, training notes, and on-line user's manuals for SINDA/FLUINT and Sinaps*Plus* are available separately, as are generic descriptions of prebuilt models and their usage. These documents (along with Sinaps*Plus* and the prebuilt package) may be accessed via C&R's web page (http://www.crtech.com). Please contact Cindy Beer at (303) 567-4514 (cindy@crtech.com) for more information, or for copies of these documents and software for those lacking Internet access.

1.1   Prebuilt Models using SINDA/FLUINT and Sinaps*Plus*

This paper documents one of a series of prebuilt models ("prebuilts") offered by Cullimore and Ring Technologies (C&R) that provide useful analysis tools to the heat transfer and fluid flow engineering community. Prebuilt models are built using SINDA/FLUINT general-purpose thermal/fluid modeling software and Sinaps*Plus*, its graphical user interface.

Prebuilts may be used without a license to run either SINDA/FLUINT or SinapsPlus, nor do they need the Fortran compiler normally required to execute SINDA/FLUINT. The user will be able to change input parameters

(dimensions, thermophysical properties, boundary conditions), and generate new results. Using the parametric study option, the user will be able to evaluate pipe flow over a range of user specified parameters. As explained in separately available documentation on prebuilt models, unlicensed users cannot make changes to the thermal/fluid network itself, nor to the user logic.

1.2   About This Prebuilt

This model was generated in Sinaps*Plus* (SINDA Application Programming System), a graphical SINDA/FLUINT pre- and postprocessing system. It performs a steady-state analysis of a generic pipe of either circular or nearly circular cross section. This prebuilt can be used as a design tool to easily evaluate specific system and design requirements for any length and cross-sectional area of a pipe, with a wide variety of boundary conditions.

The user can quickly perform various parametric sizing analyses through the use of pre-defined calculator registers. Either the flowrate or the pressure difference across the pipe may be specified. Various working fluids may be used, and the fluid may be incompressible, compressible, and even two-phase. The pipe may be adiabatic, or may be heated or cooled via convection with the wall, which itself can be either heated directly or may convect to the environment.

This prebuilt model was validated with several steady-state analytical solutions.[*]

Future updates to this pipe model will include transient analyses and additional fluids.

1.3   Introduction to Pipe Flow Hydrodynamics and Heat Transfer

Pipe flow is common to a variety of engineering applications. Some important applications where this type of flow is present are: compact heat exchangers, coolant systems, civilian water distribution systems, chemical processing, and HVAC systems. While many of these flow situations are commonly associated with a single phase working fluid, two-phase flow systems are also very common.

The thermal-hydraulic process associated with a pipe is shown schematically in Figure 1. Here, a working fluid from an infinite reservoir flows into the entrance of the pipe. As the fluid travels along the length of the pipe, the fluid pressure drops due to friction with the pipe walls, or other friction-causing devices (bends, elbows, etc.), and possible due to accelerational forces if density gradients occur. The pressure of the working fluid many also change due to elevation changes of the pipe. The temperature (and perhaps quality) of the working may also change due to either an applied heat load or to heat transfer with the environment, and is affected by the thickness and conductivity of the pipe wall, and the flow velocity.

2.0   Purpose, Capabilities, and Limitations

2.1   Purpose

The primary purpose of this prebuilt is to allow a user to perform a parametric analysis of a simple pipe flow problem over a range of user-specified parameters. Output may be in the form of plots of pressure or temperature along the length of the pipe, temperatures at various locations along the length of pipe versus various user-

---

[*] Keller, J.R., and Vogel, M.R., "Validation of the SINDA/FLUINT Thermal Analyzer Code Using Several Analytical Solutions," SAE Paper 961452, Presented at the 26th Annual ICES Conference, Monterrey CA, July 1996.
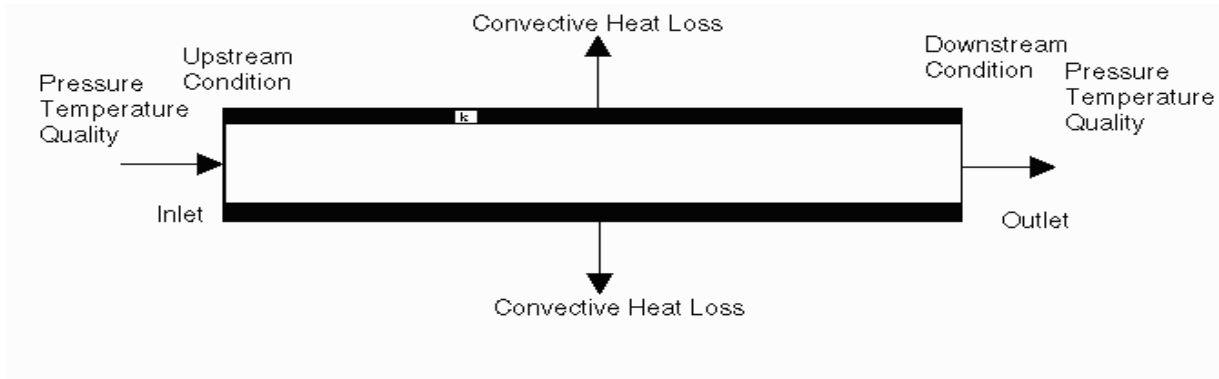
FIGURE 1. Schematic of Pipe Flow Hydrodynamics

specified input parameters, or coloring of the node network diagram by temperature or pressure. For example, the user will be able to see how the performance of a pipe changes as a function of environmental temperature, system dimensions, working fluids or material properties.

Another purpose of this prebuilt is to demonstrate simple modeling in SINDA/FLUINT using Sinaps*Plus*. It can also be used by licensed users as a starting point for generating more complex models of pipe flow networks.

2.2 Modeling Assumptions and Limitations

The assumptions for this pipe model (which are not necessarily assumptions or limitations in SINDA/FLUINT) are:

1. The model can use either metric (m, W, K) or English (ft, BTU/hr, R) units.
2. The pipe wall has constant thermophysical properties along its length (e.g., conductivity).
3. The working fluid is assumed to be a pure substance, or at least to behave like one thermodynamically (e.g., air, water/glycol). Certain assumptions (e.g., incompressibility) are implicit in the type of working fluid selected.
4. The cross sectional area is constant along the length.
5. The convective heat transfer coefficient to the environment is constant along the length.
6. The ends of the pipe wall are adiabatic
7. K-factors from bends etc. are lumped into one value and then divided evenly along the length of the pipe.
8. Only steady-state conditions are considered. *Specific heat and density inputs are currently ignored*.
9. Kinetic energy is neglected.
10. The default convection heat transfer and pressure drop correlations in FLUINT are adequate.
11. When the flow is two-phase, flow-regime-based pressure drop correlations are used, but slip flow is not modeled: the flow is assumed homogeneous.

The following parameters can be changed by the user:

1. The dimension of the pipe (length, diameter, flow area, and wall thickness).
2. The wall material properties (conductivity, specific heat, density).

3. The external convective heat transfer coefficient associated with heat transfer to the environment.

4. The fluid boundary conditions (quality, pressure, temperature).

5. The working fluid.

6. The thermal boundary conditions (adiabatic, constant temperature, constant heat load).

7. The temperature of the wall, or perhaps the temperature of the environmental and a convection coefficient from the outside of the pipe to this environment.

8. The tilt of the pipe above horizontal.

9. The driving force: flow can be driven by either a constant mass flow rate or by a pressure differential, or by body forces.

3.0   Pipe Model Description

3.1   User Defined Calculator Registers

Although other inputs may be changed throughout the model, like most prebuilts, the pipe model is designed to use registers as a "control panel" for making global changes to key model parameters. Registers are accessed from the left menu of the "model control" tab in the "model control" window. These registers are used throughout the model in network definitions, logic blocks, and in the calculation of other registers. The values can easily be modified by the user to tailor the analysis to a specific design or application. The registers also provide a convenient mechanism to perform parametric analyses without excessive need for user logic. The register values can be modified within the program, and then update the parameters which used the registers in their definition.

Table 1 summarizes the registers along with their default values. A more complete discussion of these variables can be found in Section 7.0 on page 11.

The initial set of units is SI. To change to English (US Customary) units, refer to Section 3.1.1 on page 6.

Table 1: Register Summary - Pipe Model

Register Name	Initial Value	Units	Description	Change Notes
Diameter	0.02 m	m or ft	Inner diameter of the pipe	Must be positive, avoid unrealistically large or small values.
PLength	10.0 m	m or ft	Length of the pipe	Must be positive
Pup	100000 Pa	Pa or psi	Pressure of upstream reservoir	Must be within the fluid pressure range limits
Pdown	95000 Pa	Pa or psi	Pressure of downstream reservoir	Must be within the fluid temperature range limits
Xup	0.0	-	Quality of the upstream reservoir (0.0 for liquid, 1.0 for vapor)	Must be between 0 and 1.0, inclusive
Tup	300 K	K or R	Temperature of the upstream reservoir	Must be within the fluid temperature range limits
Tilt	0.0	Degrees	Angle of Tilt (zero for horizontal, 90 for vertical upflow, -90 for vertical down-flow, etc.)	

Table 1: Register Summary - Pipe Model

Register Name	Initial Value	Units	Description	Change Notes
MassSet	1	-	1=on (set mass flowrate) 0=off (use pressure difference)	Must also set FlowRate variable if 1
FlowRate	0.1 kg/s	kg/s or lb_m/hr	Specified mass flow rate	MassSet must be set to 1 to use this feature.
Iterate	1000	-	Number of iterations performed for steady-state solution	Must be positive.
gravity	9.806	m/s^2 or ft/hr^2	gravity acceleration	For English units, set to 32.174*3600.0^2. May set to other values for zero-g or microgravity.
PSTUDY	0	-	If nonzero, specifies a parametric study option	See Section 7.0 on page 11 for list of values that can be examined.
PMAX	100.0	varies	Maximum value for the parametric study	Must be positive.
PMIN	10.0	varies	Minimum value for the parametric study.	Must be less than PMAX and must be positive.
PNUM	10.0	varies	Number of divisions for parametric study	Must be greater than one.
IDFluid	718 (water)	-	ID number of working fluid	See Section 9.0 on page 22 for a list of possible fluids (or user-supplied fluid for licensed users)
WallCT	0	-	Sets the thermal model at a fixed temperature if unity	
WalRough	1.0E-5	-	Wall Roughness Factor	Must be positive. Set to zero to simulate smooth pipe.
FrLoss	0.0	-	K factor for bends, etc. This value is apportioned over the entire length of pipe.	Must be positive.
Finlet	0.0	-	K factor for inlet. Applies to first path, whether or not flowrate is positive.	Set to 1.0 if upstream pressure is stagnation (not static), and add any inlet losses.
Aflow	Defined Function	m^2 or ft^2	Flow area	For use in modeling noncircular cross sections.
Conduct	393 W/m-K	W/m-K or BTU/hr-ft-R	Thermal Conductivity of the pipe wall.	Must be positive
HeatLoad	0	W or BTU/hr	Heat Load applied to the surface of the pipe wall	May be negative or positive.
Hcoeff	100 W/m^2-K	W/m^2-K or btu/hr-ft^2-R	Convective heat transfer coefficient (or pipe wall insulation coef)	Set to near zero to turn off convection (1.0E-10)
Walthick	1.1*Diameter	m or ft	Pipe wall thickness	Must be positive.
Tamb	250 K	K or R	Ambient temperature for convective sink	Must be positive. Should be within fluid range limits.
Density	8900	kg/m^3 or lb_m/ft^3	Density of the wall material	Not used in steady-state analyses
Spheat	383		Specific heat of the wall material	Not used in steady-state analyses
Twint	290	K or R	Initial Temperature of the wall	Must be within fluid range limits.
CN	Defined Function	J/K or BTU/R	Program defined capacitance	See Table 2 for definition

Table 1: Register Summary - Pipe Model

Register Name	Initial Value	Units	Description	Change Notes
GR	Defined Function	W/K or BTU/hr-R	Program defined conductor for radial conduction	See Table 3 for definition
GH	Defined Function	W/K or BTU/hr-R	Program defined convection conductor	See Table 3 for definition
GA	Defined Function	W/K or BTU/hr-R	Program defined conductor for axial heat transfer	See Table 3 for definition
TiltR	Defined Function	Radians	Tilt Angle in Radians	-
Odiam	Defined Function	m or ft	Calculated outer diameter of pipe	Must be greater than Diameter
Apipe	Defined Function	m^2 or ft^2	Calculated pipe cross sectional area	
Numnode	10	-	Number of fluid lumps and thermal nodes	Must be positive. *Can only be changed by a licensed user.*

3.1.1   Changing Units

To use degrees C instead of K, go to Global Control Data and change ABSZRO to -273.15.

To change to English (US Customary) units, go to Global Control Data. Change UID (toggle button in the lower right hand corner of the form). The value of gravity (a register) should also be changed to 32.174*3600^2 or the built-in register GRAV. To use degrees F, change ABSZRO to -459.69. To use psig, change PATMOS to -14.696.

If units are changed, be sure that all inputs reflect the new unit system.

3.1.2   Changing Working Fluids

To change working fluids, specify a new fluid identifier as IDFLUID in the Register control panel. The new ID must correspond to one of those listed in Section 9.0 on page 22 (Table 7 or Table 9), or to 8000 (custom gas) or 9000 (custom liquid), as defined in Section 3.1.3 below.

When changing working fluids, the range limits (valid temperatures and pressures) will change, and previously valid inputs may no longer be valid or even sensible. See also Section 6.0 on page 10.

If the *licensed* user wants to use working fluids other than the standard SINDA/FLUINT fluids and the user-defined fluids that already exist in the prebuilt model, the user must develop or acquire the appropriate FPROP fluid data blocks. C&R Technologies keeps a library of fluids that are distributed on request.

3.1.3   Custom Working Fluids

If none of the existing fluids is desired, the user may use templates to define nonvolatile liquids or perfect non-condensible gases on the basis of a few simple properties (e.g., density, viscosity, etc.). To use these templates, set IDFLUID to either 8000 (for the gas) or 9000 (for the liquid), then edit the fluid properties found under Fluid Property Data-->Input/Edit on the "model control" tab in the "model control" window. See Section 9.3 on page 25 for details.

3.2   The Pipe Thermal Model - WALL

The thermal submodel WALL represents the pipe structural material, the applied heat load, and the convective heat transfer to the environments. A summary of the node descriptions is provided in Table 2 Table 3 summarizes the definitions and descriptions for the various conductors in the WALL submodel. While the user has the ability to modify the parameters in these equations, there should be no need to do so as long as the pipe remains circular.

To turn off convective heat transfer to the ambient environment, the user should set HCOEFF (convection) to a small but nonzero number (e.g., 1.0E-10). To do otherwise would either generate errors in the Sinaps*Plus* expressions, or would violate a SINDA rule that no node can be connected only by zero-value conductors.

Table 2: WALL Node Summary

NODE ID	TYPE	Description	Definition
1000	BOUNDARY	Convection Sink Temperature	
100 -109	DIFFUSION	Pipe Inner Wall Node	CN = Density*Specific Heat*Volume
200 -209	DIFFUSION	Pipe Outer Wall Node	CN = Density*Specific Heat*Volume

Table 3: WALL Conductor Summary

Conductor ID	Type	Definition	Description
100 - 108 200 - 208	L	GA = Apipe*conduct/(PLength/NumNode)/2.	Axial heat transfer along the length of the pipe
300 - 309	L	GH = hcoeff*Odiam*pi*PLength/NumNode	Convective heat transfer to the environment.
400 - 409	L	GR = 2*pi*conduct*(PLength/NumNode)/(ln(Odiam/Diameter))	Radial heat transfer in the pipe.

3.3   The Pipe Fluid Model - PIPE

The fluid submodel PIPE represents the fluid flow within the pipe wall and the various flow control systems. A summary of the FLUINT lump descriptions is provided in Table 5, while Table 5 summarizes the definitions and descriptions for the paths in the PIPE submodel. While the user has the ability to modify the parameters in these equations, it is not recommended that the user modify these standard text book definitions.

Table 4: PIPE Lump Summary

LUMP ID	TYPE	Description	Definition
1001	PLENUM	Upstream Source	
2001	PLENUM	Downstream Sink	
100 -109	TANK	Volume of liquid in Pipe	Volume = Aflow * (PLength) / Numnode
2000	JUNCTION	Pipe Outlet	

Table 5: PIPE Path Summary

PATH ID	Type	Definition	Description
101 - 109	TUBE	DH = DIAMETER TLEN = PLENGTH/NUMNODE	Main section of the pipe
100	TUBE	DH = DIAMETER TLEN = PLENGTH/NUMNODE/2	Inlet section of the pipe
110	TUBE	DH = DIAMETER TLEN = PLENGTH/NUMNODE/2	Outlet section of the pipe
2002	MFRSET	SMFR = FlowRate	Used to set a constant mass flow rate in the pipe

Since this model assumes an infinite fluid souce entering the pipe, the status option of the source plenum (1001) is set to "STAG" specifing the pressure as a stagnation pressure and invoking automatic calculation of entrance excelleration losses.

Convection between the fluid and the pipe wall is included in this prebuilt model by using the HTNC heat transfer tie option. The values of the heat transfer coefficients are determined by internal FLUINT routines.

4.0   Running SINDA/FLUINT

The executable for the prebuilt pipe model allows the user to perform steady state analyses for either a single condition or for a parametric study with one changing variable.

4.1   Generating New Results

To run this executable, change the desired calculator registers, or network inputs, and then select Run SINDA/ FLUINT from the "model control" tab in the "model control" window, then Preprocess and Run-->Rerun Existing Executable."

Unlicensed users: do not use the Clean-up option nor any other options within this submenu structure.

Upon completion of a run, the user should find that the model has generated four files. The file PIPE.OUT file is the standard SINDA/FLUINT output file. The user should check this file to ensure that there were no errors during the execution of the program. The file, PIPE.ECH, is an input check of the calculator registers. The file PIPE.DAT is used for the parametric study, and shows the parametric variable values which were changed for the run. Finally, the model writes out the save file, PIPE.SAV, which is used for postprocessing in Sinaps*Plus*.

4.2   Running Parametric Studies

The user may elect to perform a series of steady-state runs in which a selected parameter varies from PMIN to PMAX in PNUM steps. The parametric study option is invoked when the user sets the variable PSTUDY to a value from 1 to 18. The parameters that PSTUDY affects are listed below.

When PSTUDY is set to zero, the model will use the standard inputs, and perform only one steady state run. When the variable is set from one to eighteen, the parametric option is invoked and one variable can be altered

to study its effects on the system by performing multiple steady state runs. A list of the variables which are changed for different values of PSTUDY are listed below in Table 6.

Table 6: Parametric Study Options

Option Number (PSTUDY)	Description of Varied Property	Relevant Calculator Register
o	No parametric performed: single steady-state run only	
1	pipe diameter (Diameter), assuming a circular cross section and constant wall thickness	
2	length of the pipe (Length)	
3	upstream inlet pressure	Pup
4	downstream outlet pressure (only if MASSSET=0)	Pdown
5	quality at the inlet	Xup
6	fluid temperature at the inlet (delete "!" from PL definition if two-phase fluid)	Tup
7	tilt angle of the pipe	Tilt
8	mass flow rate in the pipe	FlowRate
9	thermal conductivity of the pipe wall	Conduct
10	heat load applied to the pipe walls	HeatLoad
11	convective heat transfer coefficient on the outside of the pipe (could also be used to model insulation performance)	Hcoef
12	thickness of the pipe walls	Walthick
13	ambient temperature for convective heat transfer	Tamb
14	density of the pipe wall material - *reserved for future*	Density
15	specific of the pipe wall material - *reserved for future*	Spheat
16	the wall roughness factor	Walrough
17	overall K-factor of the pipe	FrLoss
18	wall temperature (only if WALLCT = 1)	Twint

The internal subroutine PSWEEP is used to initiate and conduct the parametric analysis. By selecting the appropriate PSTUDY variable and setting the upper and lower limits (PMAX and PMIN) for calculation, PSWEEP will conduct a series of steady state parametrics between the values of PMIN and PMAX for over PNUM increments. PMIN, PMAX and PNUM are set by the user in the calculator registers. The PSWEEP subroutine is invoked in OPERATIONS.

5.0   Postprocessing

Once a run has been made and completed successfully, Sinaps*Plus* can be used to postprocess data in a variety of graphical methods. The user can postprocess the network based on temperature, pressure, quality, etc. To begin postprocessing, the user should go to the desired network diagram, and choose the desired SAVE file (Save File Info under PostProcessing) as explained in the Sinaps*Plus* user documentation and summarized in Section 8.0 on page 16. Some examples of postprocessing are presented in this section.

Figure 2 depicts postprocessing of the PIPE submodel lumps for the initial conditions listed in Table 1 (This is a single steady-state run). A color scale is added to the network and the nodes are displayed in color based on pressure. These color results clearly show the expected pressure decrease along the length of the pipe.

In addition to coloring the nodes and lumps to evaluate the results, the results may also be plotted on X-Y plots or Bar Plots. Figure 3 shows the lump pressure along the length of the pipe for various values of pipe lengths. This figure was generated by setting PSTUDY to 2, PMAX to 10 and PMIN to 1.0 (the length changes from 1m to 10m), while the rest of the variables were held to the initial calculator register values. These results show that as the length increases, the pressure drop along the length of the pipe increases. This occurs because the frictional area increases.
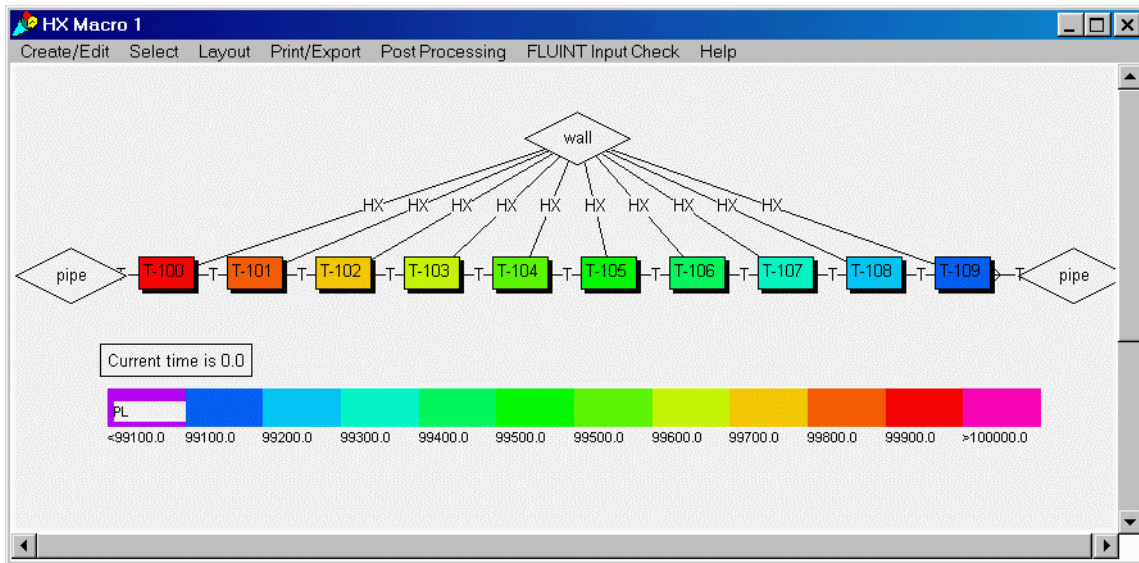

FIGURE 2. Predicted Lump Pressures

Hint: to postprocess pressure drop given fixed mass flowrate, postprocess the DeltaP value across the MFR-SET conductor in the main PIPE diagram window. Otherwise, most informative fluid variables are accessible in the HX macro.

6.0 Usage Cautions

Not all inputs are legal, and of those that are legal, not all are sensible or lead to a sensible solution. While the code makes certain attempts to constrain or limit problematic inputs, ultimately it is the user's responsibility to make sure that legal and sensible inputs have been made.

The following cautions are issued to help the user to avoid potential errors and convergence problems with the pipe model:

1. When pipe lengths are long, diameters are small, and/or the mass flow rate of the MFRSET is high, the pressure drop of the fluid will be large and may drop below the minimum fluid pressure (sometimes its saturation temperature). This problem is especially common for two-phase and compressible (gas) flows. The user is cautioned to use realistic values.

2. If the inlet represents an large area and if the mass flow rate of the MFRSET is high and the tube length is short, the flow will not be fully developed and the pressure drop and heat transfer rates will be under-predicted.
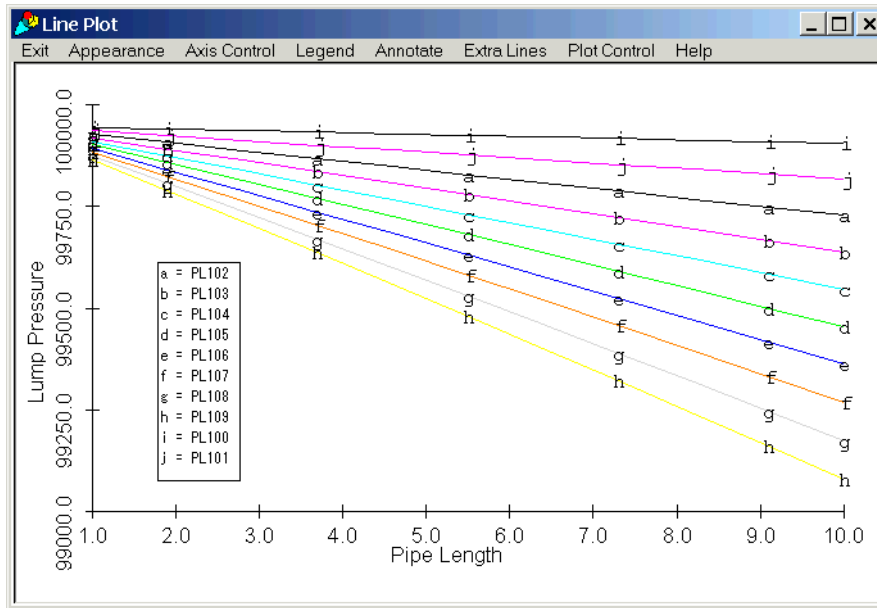
FIGURE 3. Predicted Lump Pressures for Various Pipe Lengths.

3. When tilting the pipe in a gravity field and the flowrate is not specified, the hydrostatic pressure should be taken into account when defining Pup and Pdown.

7.0   Detailed Discussion of the Variables

Below is a listing of all the calculator register values. Similar descriptions may be found in the comment field of the named calculator register form.

Diameter

Diameter is the diameter of the pipe. This variable is important in determining the velocity of the fluid, which in turn affects the pressure loss along the length of the pipe. The diameter is also important in determining the conduction through the pipe walls and the convective heat losses to the environment.

This value must be greater than zero.

If using the English system this value should be in feet, whereas if using the SI system this value should be in meters.

For noncircular ducts, set Aflow below and set Diameter to 4.0*Aflow/P where P is the wetted perimeter. Also modify the formula for GR, GA, GH etc. if wall heat transfer is included.

PLength

Length is the length of the pipe. This variable is important in determining the pressure loss along the length of the pipe. The length is also important in determining the conduction through the pipe walls and the convection to the environment.

This value must be greater than zero.

If using the English system this value should be in feet, whereas if using the SI system this value should be in meters.

Pup

This term is the upstream STATIC pressure at the pipe inlet. When MassSet is zero, the upstream pressure is important in determining the mass flowrate in the pipe. Otherwise, it is important only in setting the inlet thermodynamic state.

This value must be within the fluid range limits.

If using the English system this value should be in psia, whereas if using the SI system this value should be in Pascals.

Pdown

This term is the downstream STATIC pressure at the pipe inlet. When MassSet is zero, the downstream pressure is important in determining the mass flowrate in the pipe. Otherwise, it is largely ignored.

This value must be within the fluid range limits.

If using the English system this value should be in psia, whereas if using the SI system this value should be in Pascals.

Xup

This variable represents the upstream quality at the pipe inlet. This term can play an important role in determining the pressure drop along the length of the pipe.

This value must be between 0 and 1. For an all liquid system the, the quality must be set to 0.0, while for an all vapor system, the quality should be set to 1.0.

Tup

Tup is the upstream temperature at the pipe inlet. This term is important in determining the heat loss to the environment. For two-phase systems, this value may be overwritten if it conflicts with Pup and Xup. To use Tup and overwrite Pup instead, remove the "!" from the definition of the PL in the upstream plenum.

This value must be positive and above the saturation temperature.

If using the English system this value should be in R, whereas if using the SI system this value should be in K.

Tilt

This variable represents the angle of tilt for the pipe in degrees. An angle of zero degrees is a horizontal pipe, while a tilt angle of 90 degrees represents a vertical pipe flowing upwards (for positive FlowRate or large Pup-Pdown). This term may be important in determining the pressure drop or gain in a system.

This value must be between 0 and 90 degrees.

MassSet

If MassSet is 1, the flowrate in the pipe will be specified by FlowRate (below). Otherwise, the difference between Pup and Pdown will drive the flow.

FlowRate

The term sets the mass flow rate when MassSet is on (set to 1). When this situation occurs, the downstream pressure is irrelevant.

In the English system, this value should be set to lb_m/hr, while in the SI system it should be set to kg/s.

Iterate

This is the maximum number of iterations that the code will perform while trying to reach a steady state solution. This term sets NLOOPS in the SINDA/FLUINT Model. The output file should be checked to see if the maximum number was reached before convergence. If NLOOPS was reached before convergence, the model should be reevaluated for logical inputs, and the number of NLOOPS should be increased.

Gravity

Acceleration of gravity in m/s^2 or **ft/hr^2**. For English units, set to 32.174*3600.0^2. May set to other values for zero-g or microgravity.

Pstudy

This variable invokes the parametric study feature of the prebuilt model. When pstudy is set to set to zero, the model will use the standard inputs. When the variable is set to one through eighteen, the parametric option is invoked and one variable can be altered.

Pmax

This is the maximum value to be used in the parametric study.

Pmin

This is the minimum value to be used in the parametric study.

Pnum

This is the number of steps to be used in the parametric study.

IDFluid

This variable allows the user to choose different fluids to study. A list of fluids can be found in Section 9.0 on page 22. Future updates will include non-standard additional fluids.

WALLCT

When the user wants to examine pipe flow in a system with a constant temperature wall, this variable should be set to 1. The user must also set the variable Twint to the desired temperature.

WalRough

This variable represents the wall roughness factor. This value is important in determining the pressure loss in the pipe when turbulent flow exists. A smooth or frictionless pipe has a wall roughness factor of zero.

This value must be greater than or equal to zero. For a smooth pipe, set WalRough to zero.

Frloss

This term represents an overall K-factor for the pipe to account for bends, elbows and other friction producing items. The use of a K-factor will reduce the pressure in the pipe. The K-factor is apportioned over the length of the pipe.

This value must be greater than or equal to zero.

Finlet

Inlet loss. Applies to the first (inlet) path only, whether or not flowrate reverses. Set to 1.0 if upstream pressure is stagnation (not static), and add any inlet losses (due to sharp edges etc.)

Aflow

Flow area. Only change if noncircular, and then change also GH, GA, and GR below.

This value must be greater than or equal to zero.

Conduct

The variable Conduct is used to describe the thermal conductivity of the pipe wall. A low value of conductivity will simulate an insulated pipe. In the English system the term should have units of BTU/hr-ft-F. In the SI system, this term should have units of W/m-K.

This value must be positive.

Some suggested values of conductivity are listed below.

Copper (pure)	393 W/m-K	227 BTU/hr-ft-F
Aluminum (pure)	221 W/m-K	128 BTU/hr-ft-F
Steel (typical)	36 W/m-K	20.8 BTU/hr-ft-F

HeatLoad

This variable represents a heat load applied to the outer surface of the pipe walls and can be used to represent a heater applied to the outer surface of the pipe.

Its units should be BTU/hr in the English system and Watts in the SI system.

Hcoeff

This variable describes the convective heat transfer coefficient for the outer surface of the pipe. This term is important in determining the amount of heat gained from or lost to the environment (Tamb), and hence may also be used to model insulation performance. It is assumed constant over the entire length of the pipe. It units are BTU/hr-ft2-R (English) or W/m2-K (metric).

This value must be nonnegative.

To turn off convection to the environment set hcoeff to near zero, say 1.0E-10.

Walthick

This terms represents the thickness of the pipe wall. The variable is important in determining the heat loss to the environment.

This value must be greater than zero.

Tamb

This is the temperature of the ambient environment for convective heat transfer. This temperature can be either is Rankin (English) or Kelvin (Metric).

This value must be positive.

Density

The variable Density represents the density of the pipe wall material. This term is used with the specific heat in the calculation of transient phenomenon. Units for density are either lb/ft3 (English) or Kg/m3 (metric).

This value must be positive.

Suggested values of density:

Copper (pure)	8900 kg/m3	556 lb/ft3
Aluminum (pure)	2740 kg/m3	171 lb/ft3
Steel (typical)	7830 kg/m3	489 lb/ft3

This value is not currently used. It is reserved for future transient calculation options.

Spheat

This term represents the specific heat of the pipe wall material. Its units should be Btu/lb-F in the English system, while it in the SI system its units should be J/kg-K. This term is used in the calculation of transient phenomenon.

This value must be positive.

Suggested values of specific heat:

Copper (pure)	383 J/kg-K	0.165 BTU/lb-F
Aluminum (pure)	896 J/kg-K	0.385 BTU/lb-F
Steel (typical)	486 J/kg-K	0.209 BTU/lb-F

This value is not currently used. It is reserved for future transient calculation options.

Twint

This is the initial temperature of the pipe wall. If the WALLCT variable is activated, this is the set temperature of the walls. This temperature can be either is Rankin (English) or Kelvin (Metric).

This value must be greater than the minimum fluid temperature.

Numnode

This is the number of segments into which the pipe is subdivided.

THIS VALUE CANNOT BE CHANGED BY THE UNLICENSED USER. It is supplied for the convenience of licensed users. Changing facilitates the task of changing the pipe resolution, but the user must still add/delete nodes and conductors in the diagram window.

8.0　Selected Post-Processing Sections (Excerpts from the Sinaps*Plus* Manual)

After SINDA/FLUINT has been executed, the results can be visualized using the Sinaps*Plus* post-processing features. Using SAVE files and manipulating display options is the domain of the **post processing** option of the main menu. Once **post processing** is chosen, the user is led into a menu where various options may be utilized. This section is generic for all Sinaps*Plus* postprocessing, not just for this prebuilt

8.1　Save Files

This section describes the selection and usage of save files that are produced by SINDA/FLUINT.

8.1.1　Selecting the Save File

Before any postprocessing can be performed, a save file must be specified. When the user picks save file info from the post processing menu, a data form will appear. At the top of the form, the user chooses the source machine from the available machine types (SUN, HP, PC). Use the *select* button to select the type of machine which produced the save file (e.g., if SINDA/FLUINT was run on a SUN and that was where the save file was created - select SUN).

In the middle of the form is a field in which the user enters the name of the save file, including possible subdirectory information. If the user is unsure of the name or location of the save file, the button near the bottom of the form, Find Save File, may be used to invoke a file selector window.

If SINDA/FLUINT has been rerun without changing the save file name, then the user must so indicate to Sinaps*Plus* via the reopen button. Otherwise, the save file from the previous postprocessing session will be used.

8.2　Color

The color options provide a means to color nodes, lumps, conductors, paths and ties in a network diagram by data values that are found on a specified SINDA save file, or that can be calculated using those values. The user has control over the color scaling, and is able to step through or *animate* the save file and monitor changes propagating through the network.

8.2.1　Setting a Time

Before any color displays may be created, the user must first specify the desired time or record in the save file that contains the data. This is accomplished by selecting the Set Starting Time/Record option in the Post Processing pull down. When this option is selected, a scrolling list of records will be displayed. Pressing and

releasing the *select* mouse button over a highlighted time/record will select that time. When colorization is later performed, a comment box will appear indicating the current time being postprocessed.

8.2.2   The Color Bar

When a set of objects are colored, a color bar will appear. The characteristics of this color bar may be set before or after it is displayed. The user has control over whether or not the bar uses autoscaling, and if not, what the minimum and maximum values should be. There are toggles for a vertical or horizontal bar and toggles for color or grey scale. The user may also control the number of colors that are used by the bar. For certain directional values such as heat rates, flow rates, delta pressures, and delta temperatures, the user can signal that only the absolute value of requested data is to be displayed via the abs. value button.

Note: *The displayed scale numbers represent the edges and not the centers of each data range*. When autoscaling is off, values out of range are colored or shaded according to the ends of each bar (roughly, purple and pink), which were chosen to catch the user's attention.

The data for new option is used to set the characteristics of the next color bar to be created. The edit old option allows the use to select a color bar and then edit the characteristics of that bar, although double-clicking the color bar is a faster means of modifying an existing color bar. The network will then be recolored to reflect the changes in the color bar. Color bars may also be resized with the resize option, whereby the user indicated the desired dimensions (and location) via an areal drag.

The reset option is used to remove all color bars from the graph, and the remove option can be used to delete them individually. Color bars may also be selected and moved like any other icon, using either the menu bar or the <shift> *select* keys. (The latter provides a convenient means of bringing the color bar over to a particular colored icon in order to compare colors.)

Up to three color bars may be displayed simultaneously on a single network diagram: one for nodes/lumps, one for conductors/paths, and a third one in FLUINT models for ties.

8.2.3   Selecting Objects to Color, and Values by Which to Color

Object selection for post processing is performed by the same methods used elsewhere in Sinaps*Plus*. Furthermore, operations not applicable to certain selected items will be ignored by those items (e.g., colorization of nodes will not affect any inadvertently selected conductors, etc.). The user will then be shown a select menu to obtain the parameter to be read (T, C, TL etc.), and the network will be colored. As with other post processing options, many values are derived (e.g., delta temperature or pressures) and have no standard representation in SINDA/FLUINT.

After selecting the items and the operations for the first time, a box will appear under the cursor for placement. This box describes the current time being postprocessed. Next, a color bar will appear for placement, and the selected network items will be colored as requested.

8.2.4   Stepping Through a Save File

Once one or more groups of objects have been colored, the user may step through the save file. This is accomplished with the next time, current time, previous time and animate options under the Color/Thickness Control menu. If multiple groups of objects have been colored (e.g., nodes and conductors), stepping through the save file will update all of them. (This option also updates any thicken-by-value operations that have been requested, as will be described below in Section 8.3.)

There are two options for updating the display that are controlled by editing the color bars. If autoscaling is on, then stepping through a save file will update the scaling numbers of the color bar. If autoscaling is turned off, the colors will change on the nodes/lump and conductors/paths etc. As guidance, note that the color bar default of *autoscaling is usually undesirable when stepping through or animating results*. Otherwise, the scale changes while the color of the selected icons remains roughly invariant. On the other hand, when using fixed scaling the user must apply foresight in selecting the scale limits to avoid ranges that are too limited.

The animate option will bring up a data form that allows the user to specify how many steps to take through the save file and whether Sinaps*Plus* should wait for a mouse click between steps or step continuously. The user also has control over whether the steps are forward or backward through the file and if save file records should be skipped between the steps. The user may step by time or by record number.

8.3   Thickness

Conductors, paths and ties may be thickened by value. This operation is analogous to the previously described color operations, except that the objects are thickened instead of (and perhaps in addition to) being colored. The user is encouraged to explore this option, since thickening often results in presentations that are more intuitively understood than colorization.

Objects may be both colored and thickened independently. For example, conductors may be simultaneously colored by conductance and thickened by heat rate, while nodes are colored by temperature. This usage visually provides the same information as does QMAP.

When a group of objects are thickened, a thickness scale will appear that will follow the cursor until dropped in its final location. The numbers on the thickness scales represent edges, rather than centers of each range. Caution should be used when interpreting data if negative values are present, since the thickness scale is one dimensional. An **abs. value** option is available if helpful in such circumstances.

The thickness scale are editable in a manner analogous to that of color bars, and they contain analogous suboptions whose descriptions will not be repeated here. Analogous to the number of colors in a color scale, the user may select the number of lines in a thickness scale. The user should note that turning off autoscaling for thickness scales can sometimes result in exceptionally thick lines, and that for this reason an upper limit (approximately one inch) is placed on the maximum possible thickness for any tie, path, or conductor.

8.4   Plotting

Sinaps*Plus* provides a number of different plotting options, including X-Y plots, bar plots and polar plots. Any information found on the SINDA save file may be displayed with these options.

8.4.1   X-Y Plots

Several types of X-Y plots may be created: plots of variables versus time (or save file points or snapshots, in the case of sets of parametric steady-state), plots of variables versus icon position, plots of variable verses loop count and plots of variable verses registers. In the first case, the user need not first specify the starting time since the entire save file will be plotted by default.

As with colorization options, once a group of objects is selected, a query form will appear to determine which information to plot. This selection form will be appropriate to the type of object selected (e.g., nodes will query for T, Q and C, conductors for G).

Sinaps*Plus* will then produce an independent X-Y plot window which the user may place and size like any other Sinaps*Plus* window. (Resizing the window automatically resizes the plot.) Once the plot is created, a number of options exist within the plot window.

The various plot options that will be described below allow the user to toggle a grid, color the lines, add or edit labels for the X and Y axes, save the plot (to be accessed later from the edit old plot option), and print.

8.4.1.1   Legends

Legends may be turned on or off, moved and edited. When the legend is turned on or moved, it will follow the cursor until the *select* button is pressed (like placing a node). The on/move/refresh option brings up a new legend, and allows its placement. The old legend will remain until the new one is placed, at which time the old legend will be erased. Legends may also be turned off (the default state).

If edit is chosen, a text editing window will appear that contains the legend text. Once saved, the new legend will appear for placement.

8.4.1.2   Annotations and Extra Lines

Annotations (comments) are added in a manner analogous to that used to create comments in a network diagram. If annotate is chosen a window will appear to obtain the annotation text. Once this text is accepted, the text will appear on the plot and will follow the cursor until the *select* button is pressed.

The extra lines choice will cause a select cursor to appear. Place the cursor over the start point of the line and press the *select* button. A line will then appear that will follow the cursor until the *select* button is pressed again.

Legend annotation and extra line positions are window dependent. If you resize the window it is likely that these items will no longer be in the appropriate position.

Note that the usual mode of select-then-operate is not available for editing plots. The user must first select the operation from the pull-down menu bar, or from the pop-up *operate* menu, and a select cursor will appear to prompt the user to choose the desired element (e.g., annotation, extra line, etc.)

8.4.1.3   Appearance (Grids and Coloring Lines)

If desired, a grid can be toggled on or off. This grid follows the resolution of the axes, which can be customized separately (Section 8.4.1.4).

If more than one value is plotted and it becomes difficult to distinguish between lines, the user may choose to color the data lines. When this option is chosen, the program assigns a distinct color to each line. An internal list of approximately 8 distinct colors is used, and plotting more than that number of data values in one plot will result in at least two lines sharing the same color.

8.4.1.4   Axis Control (Limits and Expressions)

The axis limits may be changed so that all or part of a set of curves is visible. If axis limits is chosen, a data form will appear to obtain the new minimum, maximum and step size for each axis. Only data points falling within the chosen range will be shown, potentially truncating some lines.

In addition to scaling specifications, an expression may be applied as a filter for the data being plotted along each axis. In other words, mathematical operations can be specified that operate on the data being plotted in order to change units for display purposes, etc. This expression may be consist of any equation that is recog-

nized by the Sinaps*Plus* calculator but should include the name of the axis (X or Y). For example to change the X axis from hours to seconds enter "x*3600.0" or to change the Y axis from degrees F to degrees C enter "(y-32.0)/1.8".

These expressions do not apply to the range limits (in the upper part of the form), since those values are based on the raw data. The original data (i.e., that contained on the save file) was produced in units of hours, but the plot shows units of seconds along the x axis. Thus, the axis limits in the top part of the form are defined in terms of hours, but the expression in the bottom field converts the display units to seconds.

The "X Axis Points" option in the Axis Control pull down menu allows the user to select which parameter is to be plotted on the X axis.

8.4.1.5 Plot Control

Under the Plot Control pulldown are several options which can greatly enhance the presentation of the SINDA/FLUINT predictions and the control the appearance and usefulness of plots. These features allow the plotting of results from different submodels and from different save files in addition to manipulating plot items and plotting from multiple submodels. It is important to note that a plot must first be created before the plot control features can be invoked.

8.4.1.6 File Ops: Saving, Printing, and Exporting Data

Plots may be saved independently of models and images via the File Ops --> Save Plot option. The resulting machine-independent binary file may be used for archival purposes, or can be read into any model desktop. Although the plot can then be changed and edited, the underlying data upon which the plot is based will remain unchanged as later post processing operations are performed, including save file changes. (To access new data, a new plot must be created.) The purpose of this option is to enable users to exchange plots, to prepare and save report graphics independently of other model changes.

A print option also exists under File Ops, which operates analogously to the network diagram print option. As part of the print option the user will be prompted for a plot title. In addition the user will have a choice of portrait or landscape modes as well as control over the plot scaling.

An important option under File Ops is Save Points, which allows the user to export files containing ASCII tables of the data underlying the plot. These tables can then be imported into third party plotting software or spreadsheets.

8.4.1.7 Plotting vs. Location

In addition to X-Y plotting by time, Sinaps*Plus* provides for plotting vs. *icon* location. To plot by location, select group of nodes or lumps are chosen and then choose a data value (temperature etc.) to plot. A plot of temperature versus the location of the nodes and lumps in the diagram will be produced.

Since there is no actual geometry in Sinaps*Plus*, *the location displayed is the relative screen location of the object in units of pixels*. However, if the user has evenly spaced the icons, then the Sinaps*Plus* depiction may be used to represent a spatial relationship. The axis scaling expressions (Section 8.4.1.4) can then be used to convert to real dimensions. The legend lists the screen locations to aid in the development of the rescale expression.

For example, if 10 nodes were known to represent 33.4 feet of a rod, and the legend in the plot showed the x axis limit to be 1032 pixels, then the following expression can be used to convert the display diagram from pixels to feet: "x*33.4/1032."

The reader should note that certain gradient information is also accessible by coloring and/or thickening paths, ties, and conductors by delta temperature or delta pressure.

8.5   Bar Plots

These plotting options work in the same manner as X-Y plots; refer to the prior subsection for a description. One of the few differences is that bar plots have a slightly different axis limits form since there is only one axis (the "y" or independent variable axis) that may be scaled.

Bar plots also provide a unique and very useful node/lump balance option that is similar to the SINDA/FLU-INT NODMAP and LMPMAP routines. This type of plot may only be performed on a single node or lump. A bar plot can be produced that shows the energy into and out of the node on a per conductor/path basis. For lumps, the user has a choice of mass flowrate, energy flowrate, tie conductance or tie heat rate as the plotting criteria. The legend in such plots contains further details about the node or lump in question.

Lump balance plotting can only be performed if the save file record was created using the 'ALL' option in the SAVE or RESAVE argument. Otherwise, Sinaps*Plus* will report an error reading the save file since all required information could not be found in that file. Also, all relevant network elements (nodes, conductors, etc.) should be depicted within Sinaps*Plus*, and should not be contained within INCLUDE files. Otherwise, misleading plots may be produced.

Since SAVE files produced by SINDA/FLUINT Version 2.6 and earlier contain no information regarding which subset of submodels are currently active (i.e., named on the current BUILD statement), Sinaps*Plus* considers all submodels to be active at all times under such circumstances. For node and lump balance plots where the BUILD or BUILDF configuration changes, this may cause spurious heat and mass flows to appear to elements in other models that are not currently active. Heat flows for active paths and conductors will be correct, but the net term may be in error. This difficulty does not exist when using SINDA/FLUINT Version 3.0 and later.

8.6   Text Output

In addition to plots and colorizations and thickening, the user may request ASCII text output for requested post processing operations. This option allows the user to request data for the current time, or for all times. When invoked, a pop-up text edit window will appear containing the requested data.

8.7   Submodel Initialization

Initialize Submodel operates analogously to the model-level option. However, in this option the save file already selected for postprocessing will be used for the initialization, which will be confined to the current submodel only. The user must use the set starting time/record option to determine which data is to be read.

Warning: Using this option will destroy any expressions (including calculator register references) entered into data fields. SIV options and other similar time- and temperature-varying options that are calculated at run time are not affected by this operation. **Initialize Submodel should not be used by unlicensed users**.

9.0   Available Fluids and Range Limits

In addition to 20 internally defined liquid, gas, or two-phase refrigerants, licensed users may add their own reusable fluid descriptions of perfect gases, simple liquids, simple two-phase fluids, or even arbitrarily complex two-phase fluids and real gases. In this prebuilt, 12 such "user-defined" fluids have been added for the sake of the unlicensed user. Furthermore, 2 additional fluids have been added as place-holders for unlicensed users to specify their own simple liquid or gas properties. Thus, the unlicensed user has three choices:

1. Use a standard library fluid description, selected from Table 7 (with range limits listed in Table 8).

2. Use a preexisting user-defined fluid, selected from Table 9.

3. Use a "place holder" user-defined perfect gas (IDFLUID = 8000) or simple incompressible liquid (IDFLUID = 9000), and edit the properties under Fluid Property Data-->Input/Edit.

These three choices are detailed in the following subsections.

9.1   Standard Library Fluids

Table 7 lists the standard library fluids that are immediately available in FLUINT. All fluids are referenced by their ASHRAE refrigerant number.

Users are cautioned that the accuracy of the standard library fluid properties is approximately 5% on average, and can be significantly worse at extreme temperatures for some fluids. More accurate descriptions of ammonia and water (as well as many cryogenic fluids) are available upon request for those licensed users requiring greater accuracy.

Table 8 lists the pressure and temperature values that delimit the valid range for the standard library fluids. ("Valid" does not imply accurate, merely "legal" or not nonsensical.) Excursions beyond these limits will produce and error message and FLUINT will reset the wandering lump to within limits. Note that such excursions can be disregarded within steady-state runs as long as they are temporary.

Note that the temperature and pressure cannot be simultaneously above their critical values. If the pressure is supercritical, the fluid is assumed to be liquid. If the temperature is supercritical, the fluid is assumed to be a gas. (These restrictions apply only to the standard library fluids.) The highest vapor temperature is arbitrary and is often unrealistically high, but allows a wide range for controlling excursions. The user should avoid analyses that rely on accurate properties at such high values of superheat.

Table 7:   Standard Library Fluids

Number	Description
11	Refrigerant 11
12	Refrigerant 12
13	Refrigerant 13
14	Refrigerant 14
21	Refrigerant 21
22	Refrigerant 22
23	Refrigerant 23
113	Refrigerant 113
114	Refrigerant 114
216	Dichlorohexafluoropropane
290	Propane
318	Octafluorocyclobutane
500	73.8% R-12, 26.2% R-152a
502	75% R-22, 25% R-12
503	40.1% R-23, 59.9% R-13
505	78% R-12, 22% R-31
506	55.1% R-31, 44.9% R-114
717	Ammonia
718	Water
1270	Propylene

Table 8: Range Limits of Standard Library Fluids

FLUID ASHRAE NO.	LOWEST TEMP. (R)	HIGHEST LIQUID TEMP. (R)	HIGHEST VAPOR TEMP. (R)	LOWEST PRESSURE (psia)	HIGHEST VAPOR PRESSURE (psia)
11	300.00	848.07	2544.2	1.61787E-03	639.50
12	300.00	693.30	2079.9	8.99569E-02	596.90
13	271.80	543.60	1630.8	0.83375	561.30
14	204.75	409.50	1228.5	0.77399	543.16
21	300.00	800.00	2438.7	3.20782E-03	670.49
22	300.00	664.50	1993.5	0.16170	721.84
23	269.17	538.33	1615.0	0.60119	701.42
113	428.70	871.00	2631.0	0.28871	443.78
114	322.50	753.95	2261.9	2.98006E-02	473.18
216	325.00	815.67	2447.0	1.85549E-03	399.46
290	300.00	665.95	1997.9	0.23209	617.40
318	417.10	699.27	2097.8	2.5725	403.60
500	300.00	681.59	2044.8	0.12214	641.90
502	300.00	639.56	1918.7	0.26453	591.00
503	263.39	518.00	1580.3	0.76153	563.66
505	300.00	703.66	2111.0	8.80965E-02	685.59
506	300.00	747.40	2242.2	2.46063E-02	749.37
717	351.80	730.08	2190.2	0.87683	1636.4
718	491.70	1150.0	3495.3	8.76988E-02	2878.2
1270	300.00	657.09	1971.3	0.31651	667.01

FLUID ASHRAE NO.	LOWEST TEMP. (K)	HIGHEST LIQUID TEMP. (K)	HIGHEST VAPOR TEMP. (K)	LOWEST PRESSURE (Pa)	HIGHEST VAPOR PRESSURE (Pa)
11	166.67	471.15	1413.5	11.155	4.40921E+06
12	166.67	385.17	1155.5	620.23	4.11547E+06
13	151.00	302.00	906.00	5748.5	3.87001E+06
14	113.75	227.50	682.50	5336.5	3.74494E+06
21	166.67	444.44	1354.8	22.117	4.62287E+06
22	166.67	369.17	1107.5	1114.9	4.97692E+06
23	149.54	299.07	897.22	4145.0	4.83612E+06
113	238.17	483.89	1461.7	1990.6	3.05973E+06
114	179.17	418.86	1256.6	205.47	3.26249E+06
216	180.56	453.15	1359.5	12.793	2.75415E+06
290	166.67	369.97	1109.9	1600.2	4.25684E+06
318	231.72	388.48	1165.5	17737.	2.78272E+06
500	166.67	378.66	1136.0	842.14	4.42575E+06
502	166.67	355.31	1065.9	1823.9	4.07480E+06
503	146.33	287.78	877.95	5250.6	3.88631E+06
505	166.67	390.92	1172.8	607.40	4.72697E+06
506	166.67	415.22	1245.7	169.65	5.16675E+06
717	195.44	405.60	1216.8	6045.5	1.12824E+07
718	273.17	638.89	1941.8	604.66	1.98443E+07
1270	166.67	365.05	1095.2	2182.2	4.59889E+06

9.2   User Defined Fluids (Pre-defined in This Prebuilt)

Twelve fluids have been added to the standard library for the convenience of unlicensed users. These include perfect (but not necessarily calorically perfect) noncondensible gas descriptions of water, air, and common cryogens, simple incompressible nonvolatile liquid descriptions of water and common cryogens, and a full two-phase description of refrigerant 134a. The user may choose one of these fluids simply by setting IDFLUID equal to the fluid number listed in Table 9.

Table 9: Predefined Alternative Fluids and Range Limits

FLUID NO.	Name	Type	LOWEST TEMP. (K)	HIGHEST TEMP. (K)	LOWEST TEMP. (R)	HIGHEST TEMP. (R)
8729	Air	Gas	90	1000	162	1800
8740	Argon	Gas	100	500	180	900
8702	Hydrogen	Gas	14	1000	25.2	1800
8728	Nitrogen	Gas	65	1000	117	1800
8732	Oxygen	Gas	80	700	144	1260
8718	Water	Gas	273.15	1366.5	491.7	2459.7
8000	Placeholder	Gas	(see Section 9.3 on page 25)			
9740	Argon	Liquid	83.8	150	46.6	270
9702	Hydrogen	Liquid	14	33	25.2	59.4
9728	Nitrogen	Liquid	65	126.2	117	227.2
9732	Oxygen	Liquid	57.6	154.5	103.7	278.1
9718	Water	Liquid	273.15	647.6	491.7	1165.7
9000	Placeholder	Liquid	(see Section 9.3 on page 25)			

Because of their simplicity, the valid ranges of user-defined single-phase fluids are large. Pressures for liquid (9000 series) may assume any value including negative values. Pressures for gases (8000 series) must be positive, but can be high. The valid temperature range for user-defined fluids is determined by the FPROP DATA inputs (under **Fluid Property Data-->Input/Edit**), whether explicitly or implicitly. The unlicensed user can explicitly influence temperature limits using the TMIN and TMAX input keywords, but should avoid changing other inputs for these fluids.

The R134a description is a two-phase fluid description that is has a somewhat smaller range than most of the internal library fluid descriptions, being limited to a pressure less than critical for two-phase and vapor states, and limited to a temperature larger than the freezing point. (Licensed users: other R134a descriptions and data are available, as are R123 descriptions.)

Note that the unit system defined on each FPROP DATA page need not correspond to the user's unit system, and hence nothing need be changed in this section when changing to English units.

9.3   Customized Gases and Liquids

In order to enable the unlicensed user to easily add their own simple liquid and vapor descriptions, two templates for a perfect gas (IDFLUID=8000) and a simple liquid (IDFLUID=9000) have been prepared in this prebuilt.

To define a new gas, select IDFLUID=8000, then go to Fluid Property Data-->Input/Edit on the Pipe Control Panel. The first sheet is a template, wherein you may replace values for RGAS (gas constant), TMIN and TMAX (range limits in absolute degrees), V (dynamic viscosity), K (thermal conductivity), and CP (specific heat). While the template uses SI units, if you have data available in English units you may select that button in the top part of the form. (The universal gas constant in those units is 1543.3). Save the changes in this form before starting the analysis.

To define a new liquid, select IDFLUID=9000, then go to Fluid Property Data-->Input/Edit on the Pipe Control Panel. The *second* sheet (arrived at by hitting Alt-N or Save-->Save and go to Next Page) is a template, wherein you may replace values for TMIN and TMAX (range limits in absolute degrees), V (dynamic viscosity), K (thermal conductivity), CP (specific heat), and D (density). While the template uses English units, if you have data available in SI units you may select that button in the top part of the form. Save the changes in this form before starting the analysis.

If you are an unlicensed user, you will have to repeat these changes in later sessions since the unlicensed user is unable to save image files to disk (via the Save Everything command). Therefore, *it is a good idea to save the lower portion of the FPROP DATA form to a file, such that this information can be more quickly restored in future runs.* The Utilities-->File Editor option in the main launcher can help with such tasks.

10.0   For More Information

For questions about this or other prebuilt models or about the use or availability of SINDA/FLUINT and Sinaps*Plus*, contact:

> Cullimore and Ring Technologies, Inc.
> 9 Red Fox Lane
> Littleton, Colorado 80127-5710
> Voice: (303) 971-0292
> Fax: (303) 971-0035

If you have access to the internet, you may contact us via e-mail at info@crtech.com. Also our web site is available at www.crtech.com. This site contains demonstration versions, training materials, newsletters, manuals, and other announcements.