TYPE 532: SPHERICAL STORAGE TANK WITH IMMERSED HEAT EXCHANGER

General Description

This subroutine models a spherical fluid-filled, constant volume storage tank with immersed heat exchangers as shown in Figure 1 below. The fluid in the storage tank interacts with the fluid in the heat exchangers (through heat transfer with the immersed heat exchangers), with the environment (through thermal losses from the top, bottom and edges) and with up to two flow streams that pass into and out of the storage tank. The tank is divided into isothermal temperature nodes (to model stratification observed in storage tanks) where the user controls the degree of stratification through the specification of the number of "nodes." Each constant-volume node is assumed to be isothermal and interacts thermally with the nodes above and below through several mechanisms; fluid conduction between nodes, and through fluid movement (either forced movement from inlet flow streams or natural destratification mixing due to temperature inversions in the tank). The user has the ability to specify one of four different immersed heat exchanger types (or no HX if desired); horizontal tube bank, vertical tube bank, serpentine tube, or coiled tube. Auxiliary heat may be provided to each isothermal node individually; through the use if INPUTs to the model. The model also considers temperature-dependent fluid properties for either pure water, an ethylene glycol and water solution, or a propylene glycol and water solution for both the tank and heat exchanger fluids.

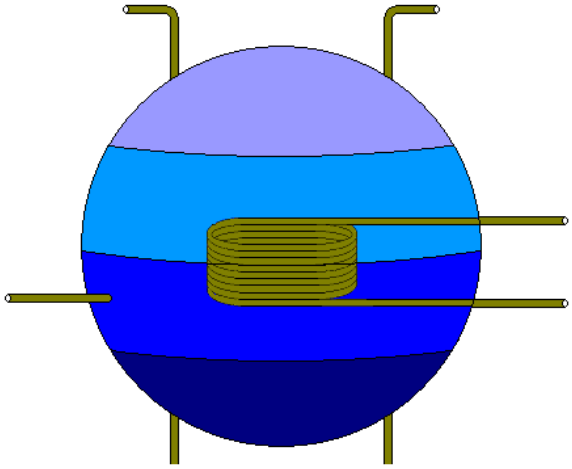


Figure 1: Schematic

Detailed Description

In this type of storage tank, heat can be transferred into and out of the storage tank through 3 unique fluid flow streams. Two of the fluid streams mix with the storage fluid while the third flow stream transfers heat to/from the storage tank through an immersed heat exchanger (the HX fluid does not mix with the storage fluid). The natural convection from the heat exchanger to the fluid in the storage tank can be a difficult problem to solve. Compounding this problem is the fact that the effects of the heat exchanger fluid mass must be considered for these types of systems. The problem breaks down into the required solution of two coupled differential equations:

$$dT_{Tank}/dt = (Q_{in,Tank} - Q_{out,Tank})/C_{Tank} \quad \text{Eq. 532.1}$$

$$dT_{HX}/dt = (Q_{in,HX} - Q_{out,HX})/C_{HX} \quad \text{Eq. 532.2}$$

where $Q_{in,Tank}$ and $Q_{out,Tank}$ are functions of the ambient temperature, the inlet fluid conditions and flow rates, and the heat exchanger temperature, and $Q_{in,HX}$ and $Q_{out,HX}$ are functions of the inlet fluid temperature and flow rate to the heat exchanger and the tank temperatures. While there are other available methods to solve coupled differential equations, we decided to solve the problem with an approximate analytical solution. The analytical solution has several inherent advantages over numerical solutions. First, the subroutine solves its own mathematical problem and does not have to rely on non-standard numerical recipes that must be attached to the subroutine. In this way, the subroutine can be imported into any FORTRAN compiler without problems. Secondly, some of the other solution methods (mainly the numerical solutions) are extremely dependent on the simulation timestep and may not converge under certain circumstances commonly encountered in domestic hot water systems (namely high flow rates for example). The analytical solution is timestep independent but does require an iterative solution inside the subroutine to solve the coupled differential equations. While solving two coupled differential equations iteratively can sometimes lead to convergence problems, this does not seem to be the case with this model under almost all operating scenarios.

To solve the differential equations analytically, the equations are placed into the form:

$$dT/dt = aT+b \qquad \text{Eq. 532.3}$$

where T is the dependent variable, t is time, a is a constant and b may be a function of time or the dependent variable. If b is a constant, than the solution of this differential equation can be readily solved. If b is not constant, then a reasonable approximation to the analytical solution can be found by assuming that b is constant over the timestep and equal to its average value over the timestep.

At any time (for **a** not equal to zero):

$$T_{final} = (T_{initial} + b_{ave}/a) * e^{(a\Delta t)} - b_{ave}/a \qquad \text{Eq. 532.4}$$

where

$$b_{ave} = b(T_{ave}) \qquad \text{Eq. 532.5}$$

and

$$T_{ave} = 1/a\Delta t*(T_{initial} + b_{ave}/a) * (e^{(a\Delta t)} - 1) - b_{ave}/a \qquad \text{Eq. 532.6}$$

With this assumption, the problem becomes straightforward to solve. Simply write the differential equation in the correct form, determine a and b_{ave} and solve for T_{final} and T_{ave}. Then recalculate b_{ave} and iterate until the temperatures converge.

While the assumption that b is constant over the timestep (and equal to its average value) is not technically correct (b for the tank is a function of the heat exchanger temperature for example), it is a reasonable approximation for the small timesteps we are using in the TRNSYS simulation (maximum timestep=1 hour).

This subroutine allows the user to designate the level of stratification for the storage tank by specifying a number of tank nodes (N). The tank is then divided into N horizontal nodes of equal volume (before the placement of the heat exchanger which reduces the volume (and hence the capacitance) of the node).

Each of the tank nodes is assumed to be isothermal. Node 1 is assumed to be at the top of the storage tank. The greater the number of nodes, the greater the stratification in the storage tank. Unreasonably increasing the number of nodes may slow down the simulation dramatically.

The heat exchangers are also divided into a user-defined number of isothermal nodes (N_{hx}), but these nodes are arranged along the flow direction of the heat exchanger (along the length of the pipe). Adjacent heat exchanger nodes interact thermally only via the flow stream, conduction is not considered between heat exchanger nodes.

For horizontal tube bank heat exchangers, each tube comprising the heat exchanger is uniformly divided into N_{hx} nodes along the horizontal direction. The user must supply the fraction of the total number of heat exchanger pipes that lie completely within each of the tank nodes.

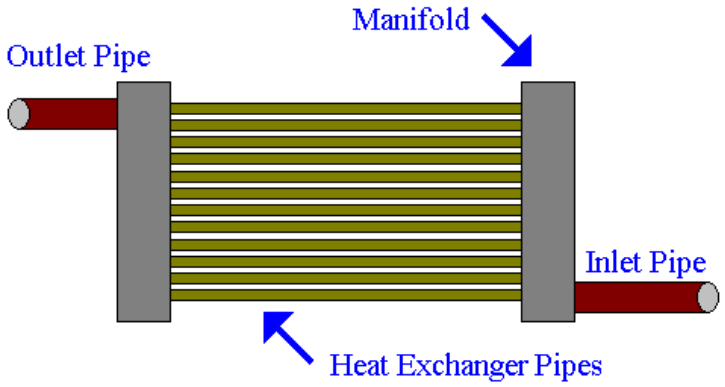


Figure 2 - Horizontal Tube Bank HX

For vertical tube bank heat exchangers, the tubes comprising the heat exchanger are divided into N_{hx} nodes along the vertical direction. For each heat exchanger node, the user must supply the fraction of the heat exchanger pipe length to be assigned to that node and the tank node in which this heat exchanger node is completely located. It is possible to have more than one heat exchanger node located within a tank node.

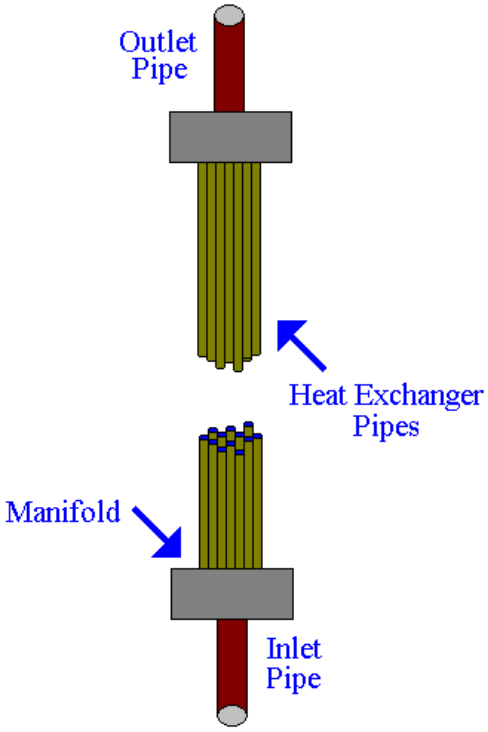


Figure 3 - Vertical Tube Bank HX

For coiled tube heat exchangers, the tubes comprising the heat exchanger (there may be more than one tube bundled together) are divided into N_{hx} nodes along the flow direction. For each heat exchanger node, the user must supply the fraction of the heat exchanger pipe length to be assigned to that node and the tank node in which this heat exchanger node is completely located. It is possible to have more than one heat exchanger node located within a tank node.

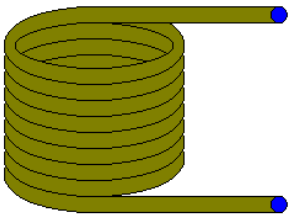


Figure 4 - Coiled Tube HX

For serpentine tube heat exchangers, the tubes comprising the heat exchanger (there may be more than one tube bundled together) are divided into N_{hx} nodes along the flow direction. For each heat exchanger node, the user must supply the fraction of the heat exchanger pipe length to be assigned to that node and the tank node in which this heat exchanger node is completely located. It is possible to have more than one heat exchanger node located within a tank node. With the serpentine tube HX, complex flow patterns through the tank may be specified and studied.

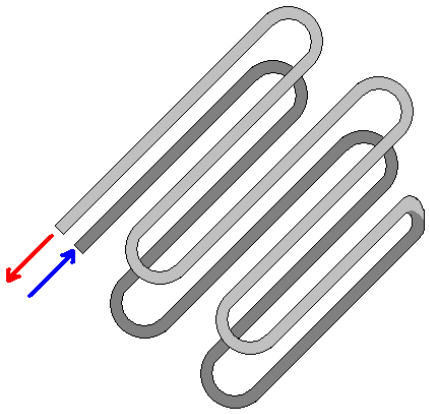


Figure 5 - Serpentine Tube HX

Heat Exchanger:

This model allows the user to designate which of 4 unique heat exchanger arrangements are immersed in the storage tank.

- vertical tube bank
- horizontal tube bank
- serpentine tube
- coiled tube

The outlet temperature at any time during the simulation for the heat exchanger flow stream is the average temperature of the heat exchanger node that contains the outlet (the last specified node) over the timestep. Header volumes (a user-supplied parameter) are added to the first and last heat exchanger nodes. The heat exchanger and storage tank interact thermally through natural convection heat transfer from the heat exchanger outer surface to the tank fluid (or in this case from the outer surface of the heat exchanger node to the tank fluid node). The overall heat transfer coefficient (UA) of the heat exchanger to tank heat transfer for any heat exchanger node can be expressed as:

$$n/UA = (1/h_o A_o) + R_w + (1/h_i A_i) \qquad \text{Eq. 532.7}$$

where:

n = number of identical tubes (pipes)
h_o = outer surface heat transfer coefficient (tank fluid side)
A_o = outer surface area of the HX node
R_w = tube wall resistance
h_i = inner surface heat transfer coefficient (in the tubes)
a_i = inner surface area of the HX node

and:

$$h_o = N_{ud} * k/d_o \qquad \text{Eq. 532.8}$$

where:

N_{ud} = Nusselt number based on tube diameter
k = fluid thermal conductivity

d_o = outer pipe diameter

and:

$$N_{ud} = C * (Ra)^n * (GF)^m \quad \text{Eq. 532.9}$$

where:

Ra = Rayleigh number for the tank fluid (based on fluid properties at $(T_{surf} + T_\infty)/2 \rightarrow$ this requires an iterative solution as Tsurf depends on the heat transfer)

T_{surf} = outer surface temperature of HX

T_∞ = temperature of the tank node containing this HX node

C, n, GF, and m = user supplied parameters

and:

$$R_w = \ln(r_o/r_i)/(2\pi L_{pipe} k_w) \quad \text{Eq. 532.10}$$

where:

r_o = outer pipe radius

r_i = inner pipe radius

L_{pipe} = length of pipe node

k_w = HX wall thermal conductivity

The inner surface heat transfer coefficient (h_i) is dependent on the type of heat exchanger as well as the fluid properties. The flow is assumed to be fully developed and the internal convection coefficient is based on the length of the tubes; and not on the length of the node. For all cases the Reynolds number (Re_{hx}) and Prandtl number (Pr_{hx})are calculated as:

$$Re_{hx} = 4\ \dot{m}\ / (\pi\ d_i\ \mu) \quad \text{Eq. 532.11}$$

$$Pr_{hx} = C_p * \mu / k \quad \text{Eq. 532.12}$$

where:

$\dot{m}$ = Flow rate through 1 tube of the HX

d_i = inner tube diameter

μ = fluid viscosity

k = fluid thermal conductivity

C_p = fluid specific heat

For all heat exchanger types with the exception of the coiled tube HX, the transition from laminar flow to turbulent flow occurs at a Reynolds number of 2300. For coiled tube heat exchangers, the critical Reynolds number for transition from laminar flow to turbulent flow is calculated as

$$Re_{crit}=20000*(d_i/d_{coil})^{0.32} \quad \text{Eq. 532.13}$$

where:

d_{coil} = diameter of the coiled tube heat exchanger

For coiled tube heat exchangers the Nusselt number correlation for laminar flow is:

$$Nusselt_{hx} = \left[\left(\frac{48}{11} + \frac{\frac{51}{11}}{1 + \left[\frac{\frac{1342}{Pr_{hx}}}{HE^2} \right]^2} \right)^3 + 1.816 \cdot \left(\frac{HE}{1 + \frac{1.15}{Pr_{hx}}} \right)^{1.5} \right]^{(1/3)} \quad \text{Eq. 532.14}$$

where:

$$HE = \frac{Re_{hx} \cdot \left[\frac{d_i}{d_{coil}} \right]^{0.5}}{1 + \left[\frac{Pitch_{coil}}{\pi \cdot d_i} \right]^2} \quad \text{Eq. 532.15}$$

and for turbulent flow conditions is:

$$Nusselt_{hx} = 0.023 \cdot Re_{hx}^{0.85} \cdot Pr_{hx}^{0.4} \cdot \left[\frac{d_i}{d_{coil}} \right]^{0.1} \quad \text{Eq. 532.16}$$

For serpentine heat exchangers, vertical tube bank heat exchangers and horizontal tube bank heat exchangers the Nusselt number correlation for laminar flow conditions is:

For cases where: $L_{tube} / d_i \leq 0.0425 * Re_{hx} * Pr_{hx}$

$$Nusselt_{hx} = \left[3.66^3 + 1.61^3 \cdot Re_{hx} \cdot Pr_{hx} \cdot \frac{d_i}{L_{tube}} \right]^{(1/3)} \quad \text{Eq. 532.17}$$

otherwise

$$Nusselt_{hx} = 4.364 \quad \text{Eq. 532.18}$$

and for turbulent flow conditions is:

$$Nusselt_{hx} = 0.0214 \cdot (Re_{hx}^{0.8} - 100) \cdot Pr_{hx}^{0.4} \quad (Pr_{hx} \leq 1.5) \quad \text{Eq. 532.19}$$

$$Nusselt_{hx} = 0.012 \cdot (Re_{hx}^{0.87} - 280) \cdot Pr_{hx}^{0.4} \quad (Pr_{hx} > 1.5) \quad \text{Eq. 532.20}$$

The internal heat transfer coefficient is then:

$$h_i = Nusselt_{hx} * k / d_i \quad \text{Eq. 532.21}$$

For cases where there is no flow through the heat exchanger, the Nusselt number is set to 1.0 for the calculation of the internal heat transfer coefficient. Finally, the heat transfer between the tank node (j) and heat exchanger node (k) can be calculated as:

$$Q_{hx,j\rightarrow k} = UA_{j,k} * (T_{tank,j} - T_{hx,k}) \quad \text{Eq. 532.22}$$

The other component of heat transfer for the heat exchanger node (besides the heat transfer to the storage tank fluid) is via fluid flow through the heat exchanger:

$$Q_{flow} = \dot{m} * C_p * (T_{hx,k} - T_{in}) \quad \text{Eq. 532.23}$$

where:

$\dot{m}$ = mass flow rate through the node (depends on HX flow rate and the number of tubes)
C_p = fluid specific heat
T_{in} = the inlet temperature to the heat exchanger node (this temperature may be the entering HX temperature or the temperature of the node upstream of this node depending on the location of the node)
$T_{hx,k}$ = the temperature of the heat exchanger node

The a and b terms for the differential equation (dT/dt = aT + b) for the heat exchanger heat transfer for the node can be expressed as:

$$a_{hx,k} = (\dot{m} * C_p - UA_{j,k}) / Cap_k \quad \text{Eq. 532.24}$$
$$b_{hx,k} = [(-\dot{m} * C_p * T_{in}) + (UA_{j,k} * T_{tank,j})] / Cap_k \quad \text{Eq. 532.25}$$

where:

Cap_k = mass of fluid in HX node * specific heat of HX fluid

Storage Tank:

The storage tank model accounts for the following component heat transfers:

- thermal losses to the environment through the sides of the storage tank
- thermal losses to the environment through a gas flue passing through the tank
- heat exchange with flowing and stagnant fluid in the heat exchanger
- conduction between adjacent tank nodes
- mixing between nodes to eliminate thermal instabilities
- mixing between nodes due to load flow through the storage tank
- auxiliary heat input to the tank
- miscellaneous heat gains/losses to/from the tank

Side Losses:

The storage tank interacts thermally with its environment through heat losses (or gains) to the tank from the side area of the storage tank. The heat transfer from the sides of the storage for tank node j is:

$$Q_{loss,edges,j} = (A_{edges,j} * U_{edges}) * (T_{tank,j} - T_{env,edge}) \quad \text{Eq. 532.26}$$

where:

$A_{edge,j}$ = tank top surface area for thermal losses (distributed equally amongst all the nodes)
U_{edge} = storage tank edge heat loss coefficient
$T_{tank,j}$ = temperature of the tank node
$T_{env,top}$ = tank environment temperature for losses through the edges of the storage

Gas Flue Losses:

The storage tank also interacts thermally with its environment through heat losses from a gas flue which passes through the storage volume. The user specifies the overall heat loss coeffeicient from the tank node to the gas flue as parameter and provides the temperature of the gas flue as as INPUT to the model. **The losses from the gas flue are set to zero if auxiliary energy is being added to the tank.** The heat transfer from the storage to the gas flue for tank node j is:

$$Q_{loss,flue,j} = UA_{flue,j} * (T_{tank,j} - T_{flue}) * (1 - \gamma_{aux}) \quad \text{Eq. 532.27}$$

where:

$UA_{flue,j}$ = overall heat loss coefficient from node j to the gas flue
$T_{tank,j}$ = temperature of the tank node
T_{flue} = the temperature of the gas flue (an INPUT)
γ_{aux} = on/off indicator for whether auxiliary heat is being added to the storage tank

Conduction Effects:

The nodes in the storage tank of this model can interact thermally via conduction between nodes. An additional term must be supplied by the user to account for conduction in the storage tank wall (and maybe even to account conduction in the heat exchanger wall). Conduction between nodes can be turned off by the user by setting the additional conductivity term (a parameter) less than zero. The formulation of the conductivity heat transfer from tank node j is:

$$Q_{cond,j} = k_j * A_j * (T_j - T_{j+1}) / L_{cond,j} + k_{j-1} * A_{j-1} * (T_j - T_{j-1}) / L_{cond,j-1} \quad \text{Eq. 532.28}$$

where

T_j = temperature of this node
T_{j+1} = temperature of the node directly below the current node
T_{j-1} = temperature of the node directly above the current node

k_j = thermal conductivity of fluid in node j (evaluated at average temperature between this node and the node below)

k_{j-1} = thermal conductivity of fluid in node j (evaluated at average temperature between this node and the node above)

A_j = conduction interface area between this node and the one below it (may be reduced by the presence of a heat exchanger tube passing through the boundary between the tank nodes)

A_{j-1} = conduction interface area between this node and the one above it (may be reduced by the presence of a heat exchanger tube passing through the boundary between the tank nodes)

$L_{cond,j}$ = vertical distance between the centroid of this node and the centroid of the node below

$L_{cond,j-1}$ = vertical distance between the centroid of this node and the centroid of the node above

Inlet Flow Effects:

This model allows the user to have multiple flow streams which pass into and out of the storage tank. There are several methods which dictate where the enetering water is placed in the storage tank.

Fixed Inlets and Outlets ("Inlet Flow Mode" = 1):
The user must specify the inlet and outlet locations (nodes) for each of the inlet and outlet streams. The temperatures at the outlets at any time are simply the average temperatures of the nodes containing the outlets over the simulation timestep. The user provides the inlet temperature and flow rate through each of the inlets. The inlet and outlet locations are also fixed throughout the simulation – they cannot be changed. The outlet flow rate from any outlet is simply the inlet flow rate from its paired inlet (for example 100 kg/h into inlet 1 causes 100 kg/h to exit the tank through outlet 1). An example is shown below of the path of the flow streams with the following conditions:

Stream 1 = 200 kg/h with Inlet in Node 4 and Outlet in Node 1
Stream 2 = 75 kg/h with Inlet in Node 1 and Outlet in Node 3

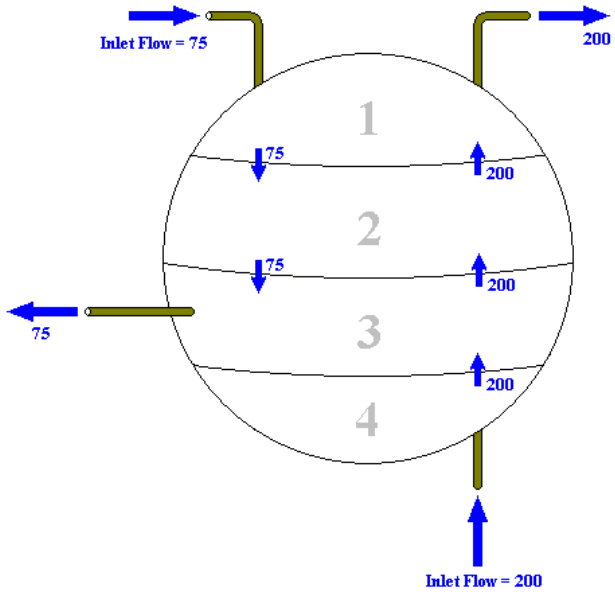


Figure 6 - Example Flows

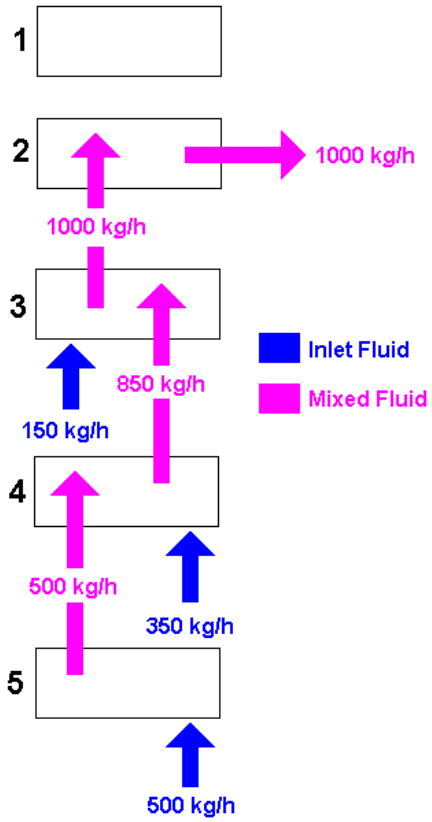
In this tank model, the inlet fluid into a node is completely mixed with the storage tank fluid at every timestep before this fluid moves on to the next node (the fluid flow path is directly from the inlet node to the outlet mode). For example, water at 10 C and at a flow rate of 200 kg/h enters node 4 (outlet = node 1) of a storage tank which has an initial temperature of 30 C. The 10 C inlet flow stream mixes with the 30 C water in the tank node causing the tank node to fall to 25 C (for example). 200 kg/h of 25 C water then crosses the boundary into node 3 where it mixes with the fluid in this boundary before moving on towards node 2 and eventually out through the outlet in node 1.

As mentioned earlier, each inlet to the tank is assumed to be paired with an outlet from the tank. In other words, you cannot have two outlets from the tank and only one inlet into the tank or two inlets to the tank and only one outlet from the tank. However, by the use of a Type 11 mixing valve (2 inlets 1 outlet) or a Type 11 flow diverter (1 inlet 2 outlets) in combination with this tank model you can effectively simulate these odd paired systems. For example, if you wished to simulate 100 kg/h of water entering the bottom of the storage tank (node 5) through an inlet port and 100 kg/h of water entering the middle of the tank (node 3) from a different fluid stream with 200 kg/h leaving through the top of the tank (node 1) in one outlet you could specify the following parameters:

"Inlet Flow Mode" = 1 : Indicates the user will specify the inlet and outlet locations
"Entry node for Inlet 1" = 5 : Indicates the first inlet location is at the bottom of the tank
"Exit node for Outlet 1" = 1 : Indicates the first outlet location is at the top of the tank
"Entry node for Inlet 2" = 3 : Indicates the seond inlet location is at the middle of the tank
"Exit node for Outlet 2" = 1 : Indicates the second outlet location is at the top of the tank

A Type 11 flow mixer is then used to mix the two outlet streams from the tank (they are both at the same temperature) and you have now effectively modeled a two inlet, 1 outlet storage tank.

Fractional Inlets with Fixed Outlets ("Inlet Flow Mode" = 2):
The user must specify the outlet locations (nodes) for each of the streams. The temperatures at the outlets at any time are simply the average temperatures of the nodes containing the outlets over the simulation timestep. The user provides the inlet temperature and flow rate through each of the inlets. The inlet and outlet locations are also fixed throughout the simulation – they cannot be changed. The outlet flow rate from any outlet is simply the inlet flow rate from its paired inlet (for example 100 kg/h into inlet 1 causes 100 kg/h to exit the tank through outlet 1). The user also has the ability in this model to dictate which tank nodes receive the inlet fluid flow. This feature is useful to simulate cases where some of the inlet flow is not completely mixed within a node before moving on to the next node. For example, in high flow rate inlets, some of the inlet fluid may effectively "jet" past the inlet node and into several nodes beyond the inlet node. For this inlet fluid mode, the user must specify the fraction of unmixed inlet flow rate that ends up in each of the tank nodes as parameters to the model. For example let us consider the case where 1000 kg/h of cold mains water enters a warm storage tank (5 nodes) with high enough velocity to cause some of the fluid to "jet" into the two nodes above (nodes 4 and 3). Let us assume that 50% of the inlet fluid enters the bottom node and mixes with the fluid in that node and 35% of the inlet fluid mixes with the fluid in the node above the inlet node, and the remaining 15% of the inlet fluid mixes with the node 2 above the inlet (node 3). If the outlet from the tank is in node 2 we have the following flow pattern established for the model:



Temperature Seeking Inlets with Fixed Outlets ("Inlet Flow Mode" = 3):
The user must specify the outlet locations (nodes) for each of the streams. The temperatures at the outlets at any time are simply the average temperatures of the nodes containing the outlets over the simulation timestep. The user provides the inlet temperature and flow rate through each of the inlets. The inlet locations are then found by finding the tank node closest in temperature to the inlet temperature and directing the entire inlet flow stream into this tank node. The outlet flow rate from any outlet is simply the inlet flow rate from its paired inlet (for example 100 kg/h into inlet 1 causes 100 kg/h to exit the tank through outlet 1). In this model, the inlet fluid into a node is completely mixed with the storage tank fluid at every timestep before this fluid moves on to the next node (the fluid flow path is directly from the inlet node to the outlet mode). For example, water at 10 C and at a flow rate of 200 kg/h enters node 4 (outlet = node 1) of a storage tank which has an initial temperature of 30 C. The 10 C inlet flow stream mixes with the 30 C water in the tank node causing the tank node to fall to 25 C (for example). 200 kg/h of 25 C water then crosses the boundary into node 3 where it mixes with the fluid in this boundary before moving on towards node 2 and eventually out through the outlet in node 1. This inlet flow mode is very similar to the first inlet mode except the inlet location is not specified by the user but rather found from the corresponding temperatures.

The heat transfer due to flow into a tank node is then expressed as:

$$Q_{flow,i,j} = \dot{m}_{in} * frac_{in,j} * C_p * T_{in} + \dot{m}_{in,j-1} * C_p * T_{j-1} + \dot{m}_{in,j+1} * C_p * T_{j+1} - \dot{m}_{push} * C_p * T_j \quad \text{Eq. 532.29}$$

where:

$\dot{m}_{in}$ = inlet mass flow rate into the tank

$\dot{m}_{in,j-1}$ = mass flow rate of fluid into node from node above
$\dot{m}_{in,j+1}$ = mass flow rate of fluid into node from node below
$frac_{in,j}$ = fraction of inlet fluid flow that enters this tank node
C_p = specific heat of the fluid
T_{in} = the temperature of the inlet fluid entering the storage tank
T_j = temperature of the tank node
T_{j-1} = the temperature of the tank node abovbe the current node
T_{j+1} = the temperature of the tank node below the current node

Mixing Effects:

At times, the nodes in the storage tank may become thermally unstable (a node has a higher temperature than the node above). If this happens, the model allows the user to "mix" the unstable nodes at a user-defined rate. While many of the tank models allow this instability mixing, most of the models calculate the effects using a simplifying assumption that speeds up the calculations but introduces some energy balance problems. The two most common tank models in TRNSYS calculate the temperatures of the nodes, then completely mix any nodes that are unstable at the end of the timestep to avoid problems. While this "averaging" of the tank temperatures (and hence an infinite flow mix between the nodes) is not a bad assumption, the energy balance is not correct as the tank losses, and other heat flows, are calculated with the tank temperatures before the mixing takes place. This model allows the user to utilize this method of mixing by specifying a negative value for the mixing flow rate input.

This model also allows the user a second method of calculating the mixing effect. In the second method the model calculates the elapsed time into the current timestep at which adjacent tank nodes would become unstable and then starts to mix the nodes at a user-defined flow rate until the nodes stabilize. In this manner (as separate differential equations for each unique mixing period are calculated by the model), the overall energy balance is calculated correctly. Once two nodes begin to mix, they keep mixing until the temperatures stabilize or until the end of the current timestep; whichever occurs later.

The formulation of the mixing heat transfer between nodes is:

$$Q_{mix,j} = \dot{m}_j * Cp_j * (T_j - T_{j+1}) + \dot{m}_{j-1} * Cp_{j-1} * (T_j - T_{j-1}) \qquad \text{Eq. 532.30}$$

where

$\dot{m}_j$ = mixing flow rate between this node and the one below
$\dot{m}_{j-1}$ = mixing flow rate between this node and the one above
Cp_j = average specific heat of fluid between this node and the one below
Cp_{j-1} = average specific heat of fluid between this node and the one above
T_j = temperature of current node
T_{j+1} = temperature of node below current node
T_{j-1} = temperature of node above current node

Care should be taken when specifying large mixing flows between nodes as small temperature differences between node calculation iterations can lead to relatively large energy balance errors. The model reports the energy balance closure at each timestep so the user should check the results upon changing the mixing flow rate.

Auxiliary Heating Effects:

This storage tank relies on external controls to add (or possibly remove) heat from the storage tank. This feature can be used to model auxiliary heating effects (electrical element, combustion heating, etc.). The user supplies heat input rates for each of the nodes in the storage tank through INPUTs to the model. For example, an aquastat model may sense the top node temperature of the storage tank and send a control signal to an auxiliary heater model that adds heat to the bottom node of the storage tank until the top node reaches its setpoint; at which time the control signal from the aquastat is disabled and input energy to the tank is ceased.

Miscellaneous Heat Inputs/Losses:

This model allows the user to specify the magnitude and location of miscellaneous heat flows into (positive value) and out of the storage tank. This feature was added to account for things such as a pilot light in a gas storage tank or other unaccounted for gains or losses.

Solving the Tank Equations

The differential equations for the tank nodes can be written as:

$$dT_{Tank,j}/dt = (Q_{in,Tank,j} - Q_{out,Tank,j})/C_{Tank,j} \qquad \text{Eq. 532.31}$$

or expanded to:

$$dT_{Tank,j}/dt = (Q_{aux,j} - Q_{hx,j\rightarrow k} - Q_{loss,top,j} - Q_{loss,bottom,j} - Q_{loss,edges,j} - Q_{cond,j} - Q_{flow,i,j} - Q_{mix,j}) / C_{Tank,j} \qquad \text{Eq. 532.32}$$

These equations are then placed in the form:

$$dT/dt = aT+b \qquad \text{Eq. 532.33}$$

and a and b values are found for each node. The b term holds the temperatures of other tank nodes and heat exchanger nodes. These temperatures in the b term are assumed to be constant for the solution of the nodal differential equations at their <u>average</u> value over the timestep. The nodal differential equation is then solved (as described earlier), new final and average nodal temperatures are calculated and the entire process is repeated until a converged solution is obtained. Then the heat exchanger nodal calculations are repeated until all tank and heat exchanger nodes are converged.

As described earlier, mixing may initiate during the timestep if temperature inversions are observed between adjacent nodes. If a temperature inversion occurs, the time into the timestep at which the nodal temperatures become unstable is calculated. The results at this point are stored and a new solution is calculated with mixing occurring between the two offending nodes. Temperature inversions are again checked and a new time at which inversions occur may be found. This process is repeated until no new temperature inversions are calculated at which time the results from the model are reported. As in all

TRNSYS components, the OUTPUTs from the model are assumed to be average values over the timestep.