TYPE 541: ICS WITH LOAD-SIDE HEAT EXCHANGER

General Description

This subroutine models an integral collector storage (ICS) domestic hot water system with or without an immersed load-side heat exchanger. In this model, the ICS system is basically a liquid-filled rectangular "box" with a solar collector located directly above it. The fluid in the box is in contact with the collector's absorbing surface and heat is transferred to the fluid in the box and then to the fluid in the immersed heat exchanger (if present) when solar radiation is incident upon the collecting surface.

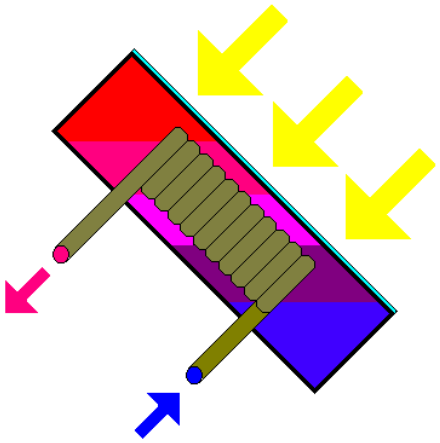


Figure 1 - ICS Schematic

Detailed Description

While the standard ICS system has been modeled in the past (most notably by Zollner at the University of Wisconsin in 1984), the load-side immersed heat exchanger addition has not been modeled to our knowledge. In this type of system, the ICS storage fluid is not mixed with the incoming fluid to be heated, but transfers heat from the storage fluid to an immersed heat exchanger which passes through the storage tank. The natural convection from the heat exchanger to the liquid in the ICS 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:

$$dT_{Tank}/dt = (Q_{in,Tank} - Q_{out,Tank})/C_{Tank}$$

$$dT_{HX}/dt = (Q_{in,HX} - Q_{out,HX})/C_{HX}$$

where $Q_{in,Tank}$ and $Q_{out,Tank}$ are functions of the ambient temperature, the incident solar radiation, and the heat exchanger temperature, and $Q_{in,HX}$ and $Q_{out,HX}$ are functions of the inlet fluid temperature to the heat exchanger and the tank temperature. While there are numerical 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, 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.

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

$$dT/dt = aT+b$$

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):

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

where

$$b_{ave} = b(T_{ave})$$

and

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

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 exchanger is 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.

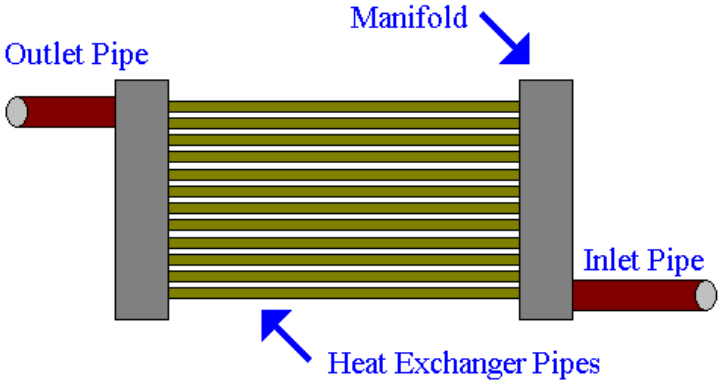


Figure 1 - Horizontal Tube Bank HX

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.

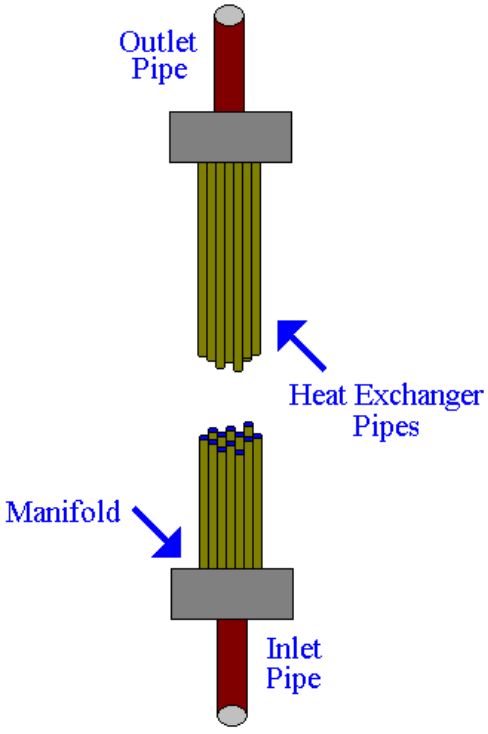


Figure 2 - Vertical Tube Bank HX

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.

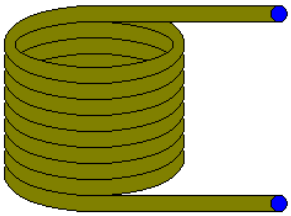


Figure 3 - Coiled Tube HX

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.

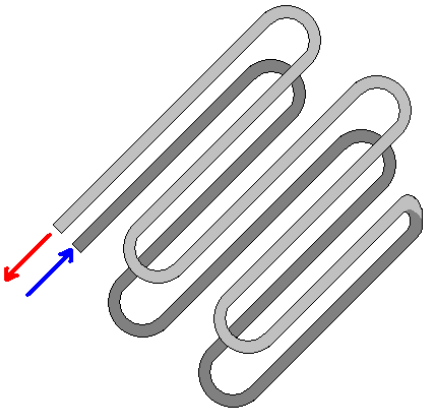


Figure 4 - Serpentine Tube HX

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:

$$n/UA = (1/h_o A_o) + R_w + (1/h_i A_i)$$

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:

$$h_o = N_{ud} * k/d_o$$

where:

- N_{ud} = Nusselt number based on tube diameter
- k = fluid thermal conductivity
- d_o = outer pipe diameter

and:

$$N_{ud} = C * (Ra)^n * (GF)^m$$

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:

$$R_w = \ln(r_o/r_i)/(2\pi L_{pipe} k_w)$$

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:

$$Re_{hx} = 4\ \dot{m}\ / (\pi\ d_i\ \mu)$$
$$Pr_{hx} = C_p * \mu / k$$

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

$$Re_{crit}=20000*(d_i/d_{coil})^{0.32}$$

where:

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

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

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

where:

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

and for turbulent flow conditions is:

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

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

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

otherwise

$$Nusselt_{hx} = 4.364$$

and for turbulent flow conditions is:

$$Nusselt_{hx} = 0.0214 \cdot (Re_{hx}^{0.8} - 100) \cdot Pr_{hx}^{0.4} \qquad (Pr_{hx} \leq 1.5)$$

$$Nusselt_{hx} = 0.012 \cdot (Re_{hx}^{0.87} - 280) \cdot Pr_{hx}^{0.4} \qquad (Pr_{hx} > 1.5)$$

The internal heat transfer coefficient is then:

$$h_i = Nusselt_{hx} * k / d_i$$

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.

The calculated value of UA for each heat exchanger node is then modified by a user-specified multiplier (X_{mult}) and a user-specifed additive factor (X_{add}) of the form:

$$UA_{j,k} = UA_{j,k} * X_{mult} + X_{add}/N_{HXnodes}$$

Finally, the heat transfer between the tank node (j) and heat exchanger node (k) can be calculated as:

$$Q_{hx,j \rightarrow k} = UA_{j,k} * (T_{tank,j} - T_{hx,k})$$

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

$$Q_{flow} = \dot{m} * C_p * (T_{hx,k} - T_{in})$$

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:

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

where:

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

Storage Tank:

The storage tank portion of the ICS model is basically a solar collector model coupled with a storage tank model. This ICS tank model accounts for the following component heat transfers:

- thermal losses to the environment through the collector top (top losses)
- thermal losses to the environment through the sides of the storage
- thermal losses to the environment through the bottom of the storage
- absorption of incident solar radiation at the collector surface
- heat exchange with flowing and stagnant fluid in the heat exchanger
- conduction between adjacent tank nodes
- additional thermal losses through a stagnation safety feature
- mixing between nodes to eliminate thermal instabilities
- mixing between nodes due to load flow through the storage tank

Collector Useful Energy Gain:

The storage tank interacts thermally with the ambient through thermal losses (or gains) to the ambient temperature through the collector surface and from the absorption of incident solar radiation. If the user wishes to consider additional radiation losses from the ICS storage tank to the night sky, the ambient temperature input should be connected to an equivalent sky temperature model output. The heat transfer through the collector surface of the storage for node i is:

$$Q_{top,i} = A_{top,i} * G_T * \tau\alpha * IAM - A_{top,i} * U_L * (T_{tank,i} - T_{amb}) - Q_{addloss} * A_{top,i}$$

where:

$A_{top,i}$ = collector area for current node
U_L = collector top loss coefficient
$T_{tank,i}$ = temperature of current tank node
T_{amb} = ambient or equivalent temperature
G_T = incident solar radiation on the collector surface (beam + diffuse)
$\tau\alpha$ = transmittance-absorptance product for the collector surface
IAM = incidence angle modifier

$Q_{addloss}$ = additional top losses per unit area

The user must enter $\tau\alpha$ and U_L as parameters to the model. These values can typically be found from the SRCC handbook of OG100 collector test ratings or from a collector test.

This model allows the user to select one of seven possible methods to determine the off-normal solar radiation effects (the incidence angle modifiers).

Mode 1:
In mode 1, no off-normal effects are considered.

Mode 2: 1st-Order IAM Effects
In this mode, the user must supply the 1st-order coefficient (b_0) of the standard incidence angle modifier function:

$$IAM = 1 - b_0*S$$

where $S = 1/\cos(\theta) - 1$, and θ is the incidence angle

Values for this parameter can be found in the SRCC handbook of solar collectors. IAM's for beam, sky diffuse, and ground diffuse are calculated as described above and combined to provide an overall IAM. The effective angles for sky and ground diffuse are calculated using the relations of Brandemuehl (refer to the TRNSYS Type 1 documentation for more information).

Mode 3: 2nd -Order IAM Effects
In this mode, the user must supply the 1st-order coefficient (b_0) and the 2nd-order coefficient (b_1)of the standard incidence angle modifier function:

$$IAM = 1 - b_0*S - b_1*S^2$$

where $S = 1/\cos(\theta) - 1$, and θ is the incidence angle

Values for these parameters can be found in the SRCC handbook of solar collectors. IAM's for beam, sky diffuse, and ground diffuse are calculated as described above and combined to provide an overall IAM. The effective angles for sky and ground diffuse are calculated using the relations of Brandemuehl (refer to the TRNSYS Type 1 documentation for more information).

Mode 4: Single Axis IAM's from a data file

In this mode, the user must supply a data file containing the IAM's as a function of the incidence angle.

θ_1 θ_2 θ_3 . . . . . θ_N
$IAM_{\theta 1}$
.
.

.
$IAM_{\theta N}$

where:

θ_i = incidence angle at which the IAM is supplied (in increasing order)
$IAM_{\theta i}$ = incidence angle modifier at the given incidence angle

The model calls the DATA subroutine with the given incidence angle and linearly interpolates to find the IAM. IAM's for beam, sky diffuse, and ground diffuse are calculated as described above and combined to provide an overall IAM. The effective angles for sky and ground diffuse are calculated using the relations of Brandemuehl (refer to the TRNSYS Type 1 documentation for more information).

Mode 5: IAM's Based on Cover and Absorber Properties

In this mode, the user must supply several parameters describing the properties of the collector covers and absorber plate. The model then uses the TALF function in TRNSYS to calculate the theoretical IAM's as a function of the incidence angle. IAM's for beam, sky diffuse, and ground diffuse are calculated as described above and combined to provide an overall IAM. The effective angles for sky and ground diffuse are calculated using the relations of Brandemuehl (refer to the TRNSYS Type 1 documentation for more information).

Mode 6: Multiplicative Bi-Axial IAM's

In this mode, the user must supply a file of IAM's as a function of two separate collector angles; the longitudinal angle and the transverse angle. The total IAM will then be the product of these two angular IAM's. This assumption works fairly well for optically non-symmetric collectors such as evacuated tubes and CPC's. A nice description of this approach can be found in the TRNSYS Type 1 documentation. The file format is given below:

θ_1 θ_2 θ_3 . . . . . θ_N
IAM_{L1} IAM_{T1}
.
.
.
IAM_{LN} IAM_{TN}

where:

θ_i = angle at which the IAM is supplied (in increasing order)
IAM_{L1} = incidence angle modifier at the specified collector longitudinal angle
IAM_{T1} = incidence angle modifier at the specified collector transverse angle

Mode 7: Bi-Axial IAM's

In this mode, the user must supply a file containing an array of IAM's as a function of two separate collector angles; the longitudinal angle and the transverse angle. The model will calculate the transverse and longitudinal angles based on the incidence angle and the collector slope. The model will then call the DATA subroutine to linearly interpolate between points in two directions to find the IAM at the required combination. Unlike previous modes that assumed the IAM was the product of the IAM at the transverse angle and the IAM at the longitudinal angle, this model allows the user to input the IAM exactly as it should be for the desired combination of angles.

The user must also supply a file of bi-axial incidence angle modifier data in the format:

$T_1\ T_2\ T_3\ .\ .\ .\ .\ .\ T_N$
$L_1\ L_2\ L_3\ .\ .\ .\ .\ .\ L_N$
$IAM_{T1,L1}\ \ IAM_{T1,L2}\ .\ .\ .\ .\ IAM_{T1,LN}$
$IAM_{T2,L1}\ \ IAM_{T2,L2}\ .\ .\ .\ .\ IAM_{T2,LN}$
.
.
.
$IAM_{TN,L1}\ \ IAM_{TN,L2}\ .\ .\ .\ .\ IAM_{TN,LN}$

where:

T_i = transverse collector angle at which the IAM is supplied (in increasing order)
L_i = longitudinal collector angle at which the IAM is supplied (in increasing order)
$IAM_{Ti,Li}$ = incidence angle modifier at the combination of transverse and longitudinal angles specified

Edge and Bottom Losses:

The storage tank also interacts thermally with the ambient through heat losses (or gains) to the ambient temperature through the tank edges and bottom. The heat transfer from the edges and the bottom of the storage for node i is:

$$Q_{loss,i} = [\ (A_{edge,i} * U_{edge}) + (A_{bottom,i} * U_{bottom})\ ] * (T_{tank,i} - T_{env})$$

where:

$A_{edge,i}$ = storage edge area for current node (nodes remain horizontal regardless of collector slope)
$A_{bottom,i}$ = storage bottom area for current node (nodes remain horizontal regardless of collector slope)

U_{edge} = collector edge loss coefficient
U_{bottom} = collector bottom loss coefficient
$T_{tank,i}$ = temperature of current tank node
T_{env} = tank environment temperature (can be different from the ambient temperature specified for the top losses - this can be used to account for mounting of the collector within the roof trusses

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 conduction in the heat exchanger wall). Conduction between nodes can be turned off by the user by setting the additional conductivity term less than zero. The formulation of the conductivity heat transfer is:

$$Q_{cond,i} = k_i * A_i * (T_i - T_{i+1}) / L_i + k_{i-1} * A_{i-1} * (T_i - T_{i-1}) / L_{i-1}$$

where

k_i = thermal conductivity of fluid in node (evaluated at average temperature between this node and the node below)
A_i = interface area between this node and the one below it
T_i = temperature of this node
T_{i+1} = temperature of the node below the current node
L_i = vertical distance between the centroid of this node and the centroid of the node below
k_{i-1} = thermal conductivity of fluid in node (evaluated at average temperature between this node and the node above)
A_{i-1} = interface area between this node and the one above
T_{i-1} = temperature of the node directly above the current node
L_{i-1} = vertical distance between the centroid of this node and the centroid of the node above

The tank model assumes equal volume isothermal nodes (before the heat exchanger placement). These nodes remain horizontal, regardless of the slope of the collector. To calculate the conduction distance between nodes (L_i), the mass centroid of each node is found. The conduction distance is then the distance between the centroids of the adjacent nodes.

Safety Device:

This tank model is equipped with an over-temperature safety device to provide additional thermal losses from the storage to a heat sink. The overall loss coefficients (UA's) for each of the tank nodes are provided to the model as inputs and should be connected to a controlling device which senses the tank temperature. An example of this feature would be to use the TYPE 2 differential controller with hysteresis to watch the average tank temperature. The controller would then turn on the additional UA when the tank reached a critical temperature, and keep the additional losses on until the tank falls below the specified reset temperature. At this point, the additional losses are turned off. In many cases, this will lead to cycling of the additional losses during periods of high solar gain and low heat exchanger flow rate.

The formulation of the safety device heat transfer is:

$$Q_{safe,i} = UA_{safe,i} * (T_i - T_{safe})$$

where

$UA_{safe,i}$ = overall heat transfer coefficient input for the additional thermal losses between the node and the sink temperature.
T_i = temperature of node
T_{safe} = sink temperature for aditional thermal losses

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 incorrectly. The two most common tank models in TRNSYS calculate the temperatures of the nodes, then 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 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:

$$Q_{mix,i} = m_i * Cp_i * (T_i - T_{i+1}) + m_{i-1} * Cp_{i-1} * (T_i - T_{i-1})$$

where

m_i = mixing flow rate between this node and the one below
Cp_i = specific heat of fluid (at average temperature of adjacent nodes)
T_i = temperature of current node
T_{i+1} = temperature of node below
m_{i-1} = mixing flow rate between this node and the one above
Cp_{i-1} = specific heat of fluid (at average temperature of adjacent nodes)
T_i = temperature of current node
T_{i+1} = temperature of node above

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.

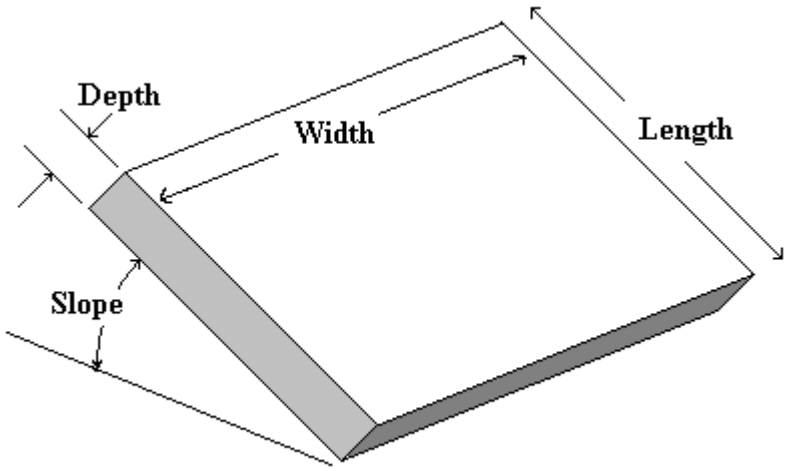


Figure 2 - Schematic of ICS Orientations