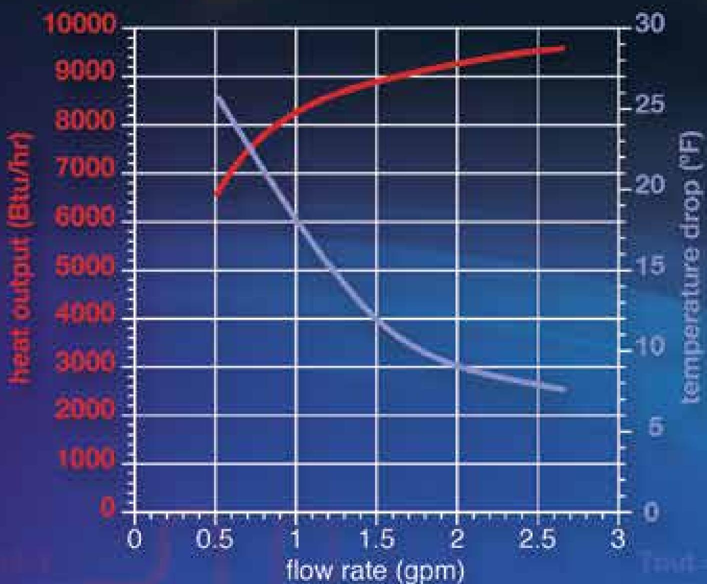
july 2018
23

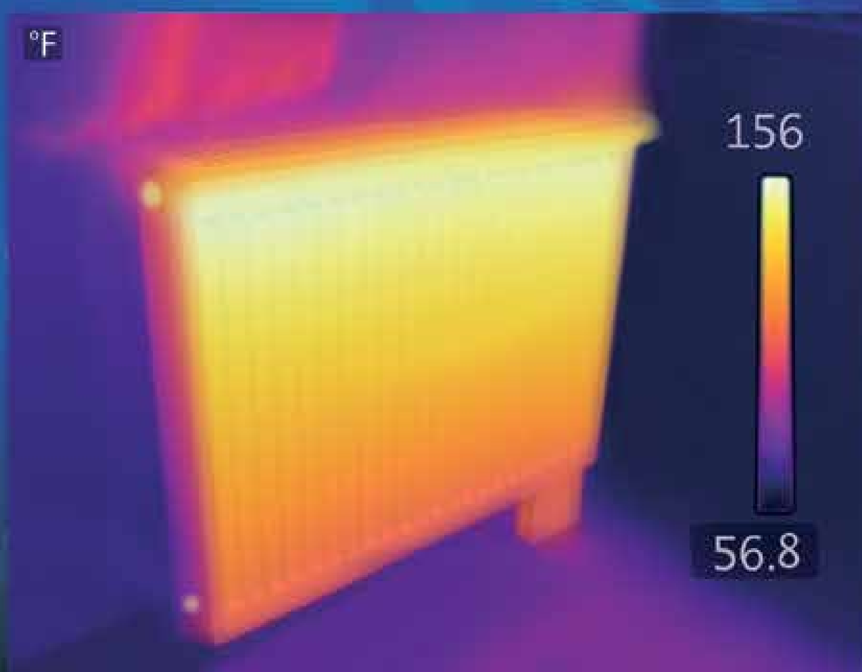
idronics™

JOURNAL OF DESIGN INNOVATION FOR HYDRONIC PROFESSIONALS

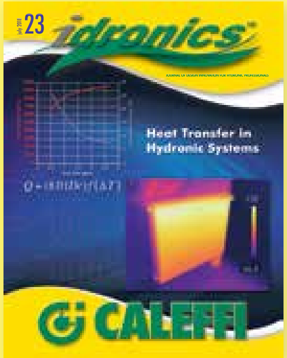


Heat Transfer in Hydronic Systems

$$Q = (8.01Dc)f\left(\Delta T\right)$$



CALEFFI



A Technical Journal
from
Caleffi Hydronic Solutions

CALEFFI NORTH AMERICA, INC
3883 W. Milwaukee Rd
Milwaukee, Wisconsin 53208 USA

Tel: 414-238-2360
FAX: 414-238-2366

E-mail: idronics@caleffi.com
Website: www.caleffi.us

To receive future idronics issues
FREE, register online www.caleffi.us

Printed: Milwaukee, Wisconsin USA

Caleffi North America, Inc.
3883 W. Milwaukee Rd
Milwaukee, Wisconsin 53208
T: 414.238.2360 F: 414.238.2366

Dear Plumbing and Hydronic Professional,

Is an underperforming heat emitter the result of insufficient flow rate, low fluid temperature, undersizing, improper placement, trapped air, or sludge deposits? It could be any or all of these.

An understanding of how heat moves and what factors enhance (or inhibit) that movement is vitally important to those who design, troubleshoot, or maintain hydronic systems. Conduction, convection, and radiation all play rolls, as do material selections and the placement of those materials within a system.The ability to estimate heat transfer rates using basic formulas is also important when designing systems.

This issue of idronics discusses the fundamentals of heat transfer as they apply to hydronic heating and cooling systems. It describes how the rate of heat movement in these systems is influenced by flow rate and temperature differences. It also describes practical details that enhance heat transfer in a wide variety of system applications.

We hope you enjoy this issue of idronics and encourage you to send us any feedback by e-mailing us at idronics@caleffi.com.

For prior issues please visit us at www.caleffi.us, and click on the icon. There you can download the PDF files. You can also register to receive hard copies of future issues.

Mark Olson

Mark Olson

General Manager & CEO

INDEX

Heat Transfer in Hydronic Systems

1. INTRODUCTION:

All hydronic heating and cooling systems move thermal energy (e.g., heat) from one location to another. In heating applications, this movement takes place across many components, starting at a heat source and exiting the hydronic system at the heat emitter(s). Between these “end points” of the system, several intermediate heat transfer processes must occur. Examples include heat moving from combustion gases to water flowing through a boiler, heat passing from one fluid to a different fluid through a heat exchanger, and heat passing from water inside a copper tube to aluminum fins attached to the outer surface of that tube and finally into room air.

It’s important for those who design or troubleshoot hydronic systems to understand the physical processes by which heat moves. That knowledge can help them quickly identify factors that may constrict heat movement. Examples of such constrictions include scale formation on a boiler’s heat exchanger, heat emitters with insufficient surface area, or a thick rug placed over a heated floor slab.

This issue of *idronics* discusses the fundamentals of heat transfer as they apply to hydronic heating and cooling systems. It also shows how the *rate* of heat movement in hydronic systems is influenced by flow rate and temperature change of water or water-based fluids.

Another concept called thermal equilibrium, which applies to all hydronic heating and cooling systems, will be described. The relationship between heat emitter surface area and required water temperature for a given rate of heat transfer is also be examined.

Heat transfer can be a complex subject from a theoretical standpoint. Mathematical models that predict the heat transfer performance of various heat emitters have been developed, and some are presented in the appendices. However, the primary objective of this issue is *not* to dive deeply into the mathematical theory of heat transfer. Rather, it’s to provide a solid understanding of fundamentals, along with many practical applications of those fundamentals. Such an understanding can help root out potential performance problems at the design stage, long before they reveal themselves as costly mistakes within installed systems. It can also help technicians quickly identify performance problems caused by heat transfer “bottlenecks,” rather than faulty equipment.

2. HEAT TRANSFER FUNDAMENTALS

Two fundamental concepts apply to all situations involving heat transfer:

1. Heat always moves from a material at some temperature to another material at a lower temperature.

2. The rate of heat transfer depends on the temperature difference between the two materials. The greater this difference, the higher the rate of heat transfer.

To gain a better detailed understanding of how a hydronic system works, heat transfer needs to be classified. A well-established scientific principal is that all heat transfer processes occur through one or more of the following modes:

• Conduction
• Convection
• Thermal radiation

CONDUCTION:
Conduction heat transfer takes place within solid materials. It’s the result of atomic vibrations within those materials. The atoms in all materials having temperatures above absolute zero (-458ºF) vibrate to some extent. The higher the material’s temperature, the more vigorous the atomic vibrations.

When heat is added to a solid material, the atomic vibrations become more energetic. These vibrations spread out across trillions of atoms that are bonded together, moving towards material at a lower temperature.

The property that determines how well a material transfers heat by conduction is called its *thermal conductivity*. The higher a material’s thermal conductivity value, the faster heat can pass through it, with all other conditions being equal. The thermal conductivity of a material is usually determined by testing. The thermal conductivity of many materials is listed in references such as the *ASHRAE Handbook of Fundamentals*.

Heat moving from the inner surface of a pipe to its outer surface is an example of conduction heat transfer. Heat passing from tubing embedded in a concrete slab to the surface of that slab is another example. The warmth felt on hands wrapped around a cup of hot coffee is the result of heat conduction through the cup.

Figure 2-1

Source: http://stillstandingmag.com/2014/02/epiphany/

The rate of heat transfer by conduction is directly proportional to both the temperature difference across the material and its thermal conductivity. It is inversely proportional to the thickness of a material. Thus, if one were to double the thickness of a material while maintaining the same temperature difference between its sides, the rate of heat transfer through the material would be cut in half.

Materials such as copper, aluminum, and steel have relatively high values of thermal conductivity, and thus, allow high rates of heat conduction. Copper and aluminum in particular are commonly used in applications where the ratio of the heat transfer rate by conduction, per pound of metal used, needs to be high. Other materials, such as wood, concrete and polyethylene, have lower values of thermal conductivity relative to metals. Materials such as expanded polystyrene and polyurethane foam have very low thermal conductivities, and are commonly used as thermal insulation to minimize conduction heat transfer.

The *rate* of conduction heat transfer depends on the thermal conductivity of the material, its thickness, the area across which heat is passing, and the temperature difference across it. The relationship between these quantities is given as Formula 2-1.

Formula 2-1:

$$Q = A\left(\frac{k}{\Delta x}\right)(\Delta T)$$

Where:
Q = rate of heat transfer through the material (Btu/hr)
k = thermal conductivity of the material (Btu/°F•hr•ft)
Δx = thickness of the material in the direction of heat flow (ft)
ΔT = temperature difference across the material (°F)
A = area heat flows across (ft^2)

Although the value of thermal conductivity for a given material is usually treated as a constant, it does vary slightly based on factors such as the material's temperature, age and moisture content. The thermal conductivity of certain foam insulations increases slightly as the foam ages. This happens because the gas used to inflate the cells in the foam is slowly replaced with air. The thermal conductivity of freshly sawn wood decreases as its moisture content decreases. Similarly, the thermal conductivity of dry soil can be substantially lower than wet soil. The latter is an important consideration when designing earth loops for geothermal heat pump systems that dissipate significant amounts of heat into the soil, causing it to dry.

The mathematical term (k/Δx) in Formula 2-1 is sometimes called a heat transfer coefficient, and is represented by the symbol (U). The reciprocal of the heat transfer coefficient U (e.g., 1/U) is called "thermal resistance," and is represented by (R).

Formula 2-1 can therefore be modified into Formula 2-2:

Formula 2-2:

$$Q = \frac{A}{R}(\Delta T)$$

Where:

Q = rate of heat transfer through the material (Btu/hr)
R = thermal resistance (or "R-value) of a material (°F•hr•ft^2/Btu)
ΔT = temperature difference across the material (°F)
A = area heat flows across (ft^2)

The R-value of a material can be determined based on its thermal conductivity (k), and its thickness (ΔX). The relationship is given as Formula 2-3.

Formula 2-3:

$$R = \frac{\Delta x}{k}$$

Where:
k = thermal conductivity of the material (Btu/°F•hr•ft)
Δx = thickness of the material in the direction of heat flow (ft)

For example: The thermal conductivity of a specific polyurethane foam is listed as
k = 0.015 (Btu/°F•hr•ft). Determine the R-value of a 2-inch-thick slab of this material.

Solution:

$$R=\frac{\Delta x}{k}=\frac{(2in)\left(\frac{1ft}{12in}\right)}{0.015\left(\frac{Btu}{ft\cdot^{o}F\cdot hr}\right)}=11.1\left(\frac{ft^{2}\cdot{}^{o}F\cdot hr}{Btu}\right)$$

Note that it was necessary to convert the thickness of the material from inches to feet so that the resulting R-value is expressed in traditional North American units of (°F•hr•ft^2/Btu).

The R-value of many common materials can be found in references such as the *ASHRAE Handbook of Fundamentals*. The R-value of insulation materials is usually listed on their packaging. The table in Figure 2-2 lists the R-value of some common materials on a per inch of thickness basis.

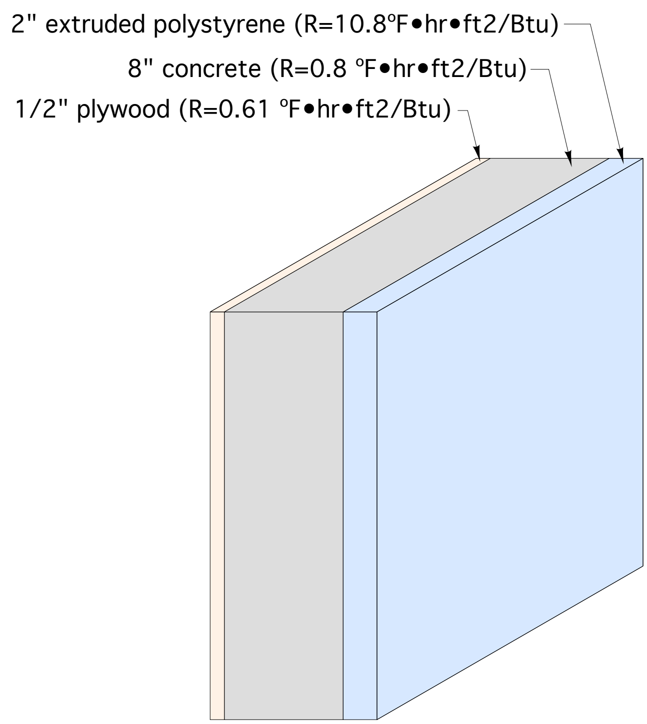
Figure 2-2

Material	R-value per inch of thickness (°F•hr•ft²/Btu)
fiberglass insulation	3.1
concrete	0.1
extruded polystyrene	5.4
plywood	1.24
copper	0.0051

When the R-value of a one-inch-thick layer of a material is known, the R-value of other thicknesses can be found by multiplying the R-value per inch of thickness, by thickness. For example, the R-value of a 2-inch-thick layer of extruded polystyrene would be 2 x 5.4 = 10.8 (°F•hr•ft^2/Btu), and the R-value of a ½" thick layer of plywood would be 0.5 x 1.24 = 0.62 (°F•hr•ft^2/Btu).

When several materials are combined in layers, the total R-value of the assembly is found by adding the R-values of those materials. An example is shown in Figure 2-3.

Figure 2-3



Total R-value of assembly =
0.61+0.8+10.8 = 12.21 (°F•hr•ft2/Btu)

Many situations in which heat is transferred by conduction involve multiple materials with different thermal conductivities. Heat passes from one material to another, depending on how those materials are arranged. Materials with high thermal conductivity create very little temperature drop as heat passes through them at a given rate (e.g., Btu/hr). Materials with lower thermal conductivity create higher temperature drops while passing heat at the same rate. The overall heat transfer rate of an assembly of materials is typically limited by the material with lower thermal conductivity.

One example of this is a concrete floor slab with embedded heating tubing. It's logical to assume that using a tubing material with high thermal conductivity would increase the slab's upward heat output. But by how much?

Consider a 4-inch-thick concrete slab with tubing embedded at 12-inch spacing. The slab has a layer of 3/8" hardwood flooring fully bonded to its surface. The tubing options are copper or PEX, both in ½" nominal tube size.

Figure 2-4

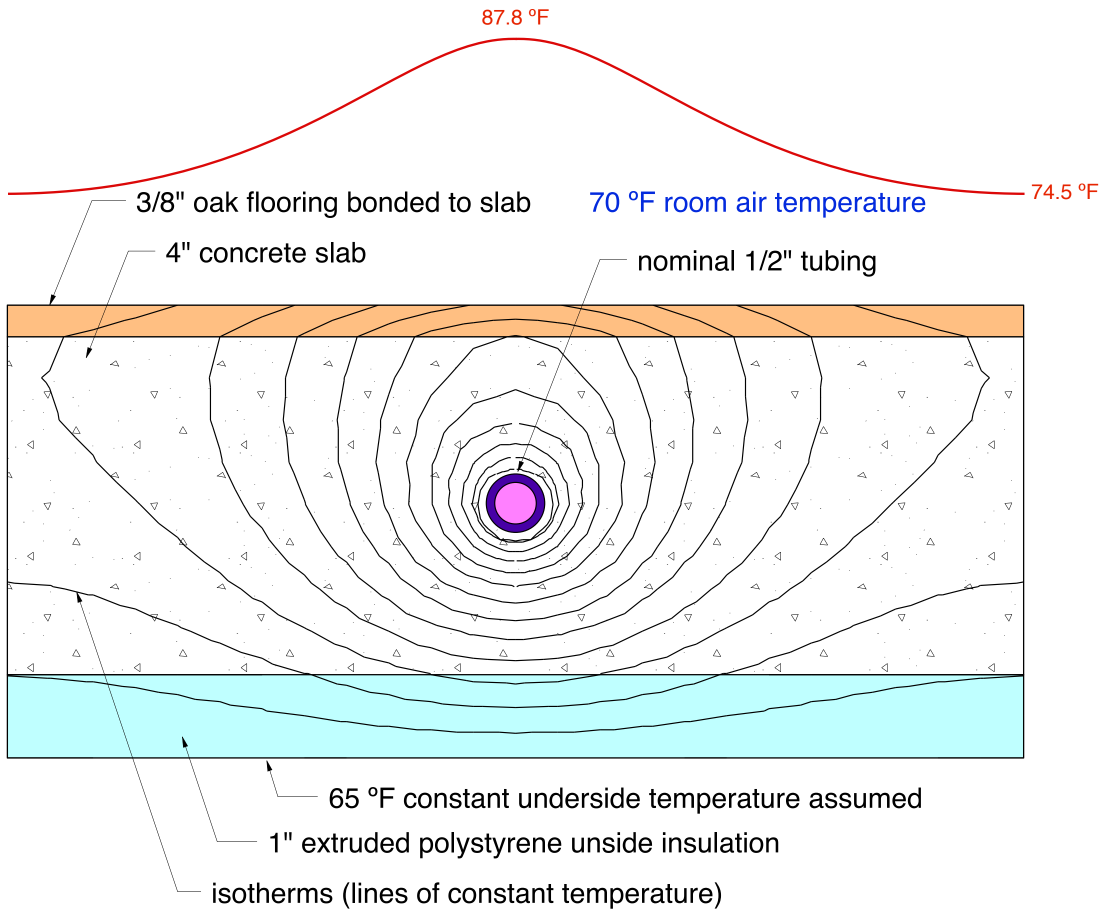
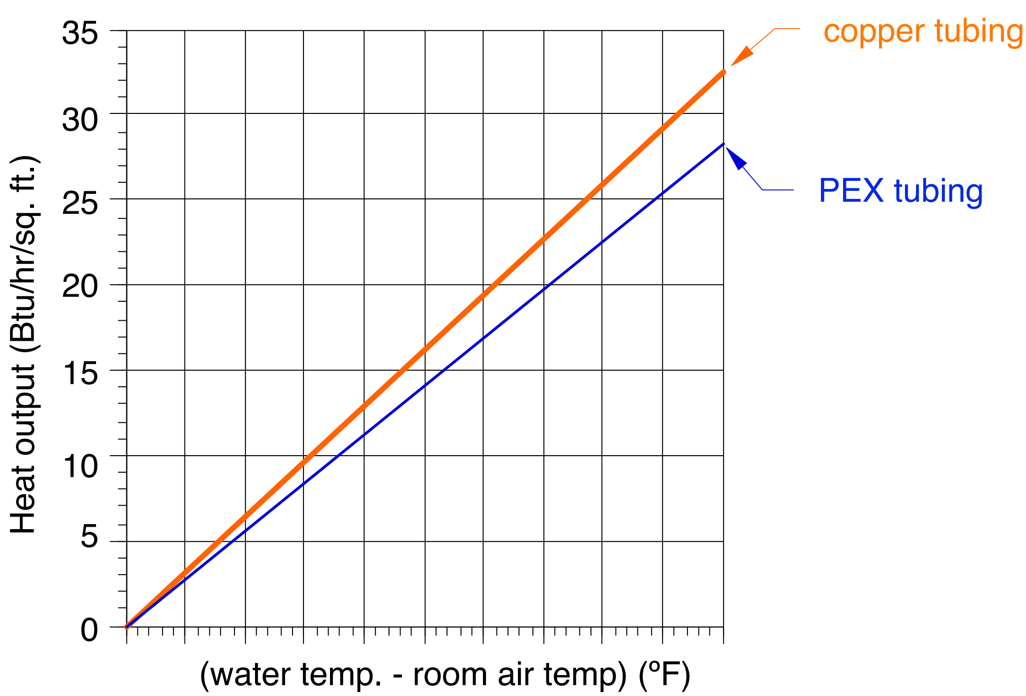


Figure 2-5



The thermal conductivity of the tubing options is:

Copper: k= 223 (Btu/hr•ft•°F)
PEX: k= 0.237 (Btu/hr•ft•°F)

The thermal conductivity of the copper is 941 times greater than that of the PEX tubing. Does this imply that the heat output of the floor slab would be 941 times greater if copper tubing were used instead of PEX?

It's a reasonable question, given the large difference in thermal conductivity. To find the answer one could build two otherwise identical slabs, one with the copper tubing and the other with PEX tubing, and perform accurate testing. An alternative is to simulate the same scenario using finite element analysis software.

Figure 2-4 shows the result of a finite element analysis of the slab described above.

The section of the floor shown spans 6 inches to the left and right of the tubing centerline. The 4-inch concrete slab (k= 0.833 Btu/ft•F•hr) rests on 1" thick extruded polystyrene (k= 0.0154 Btu/ft•F•hr). The wood flooring (k = 0.0833 Btu/ft•F•hr) is assumed to be fully bonded to the top of the slab.

The contour lines show in Figure 2-4 are called *isotherms*. Each one represents the locations within the material that are at the same temperature. The farther an isotherm is from the tube, (e.g., the source of heat input), the lower the temperature it represents. Heat flow is always perpendicular to any isotherm at any location. In this case, the isotherms can be thought of as "waves" of heat flowing outward from the tube. They show the combined effect of all the materials, including their shape, relative position and thermal conductivity.

The red curve above the floor section is a surface temperature profile. It's a plot of the surface temperature versus position across the floor section. The floor is warmest directly above the tube and coolest half way to the adjacent tube (assuming that any adjacent tubes contain water at the same temperature).

By analyzing the surface temperature profile, it's possible to estimate the rate of upward heat output from the floor. Figure 2-5 shows this upward heat output versus the difference between the average water temperature in the tubing and the room air temperature. The type of tubing used is the only variable.

As expected, the slab using the copper tubing has the higher heat output, but only about 16 percent higher than

Figure 2-6

Figure 2-7

Courtesy of Harvey Youker

the slab using the PEX tubing. This example demonstrates that even when some materials within an assembly have high thermal conductivity relative to the other materials, it's the *assembly* of all materials that dictates overall heat transfer rates.

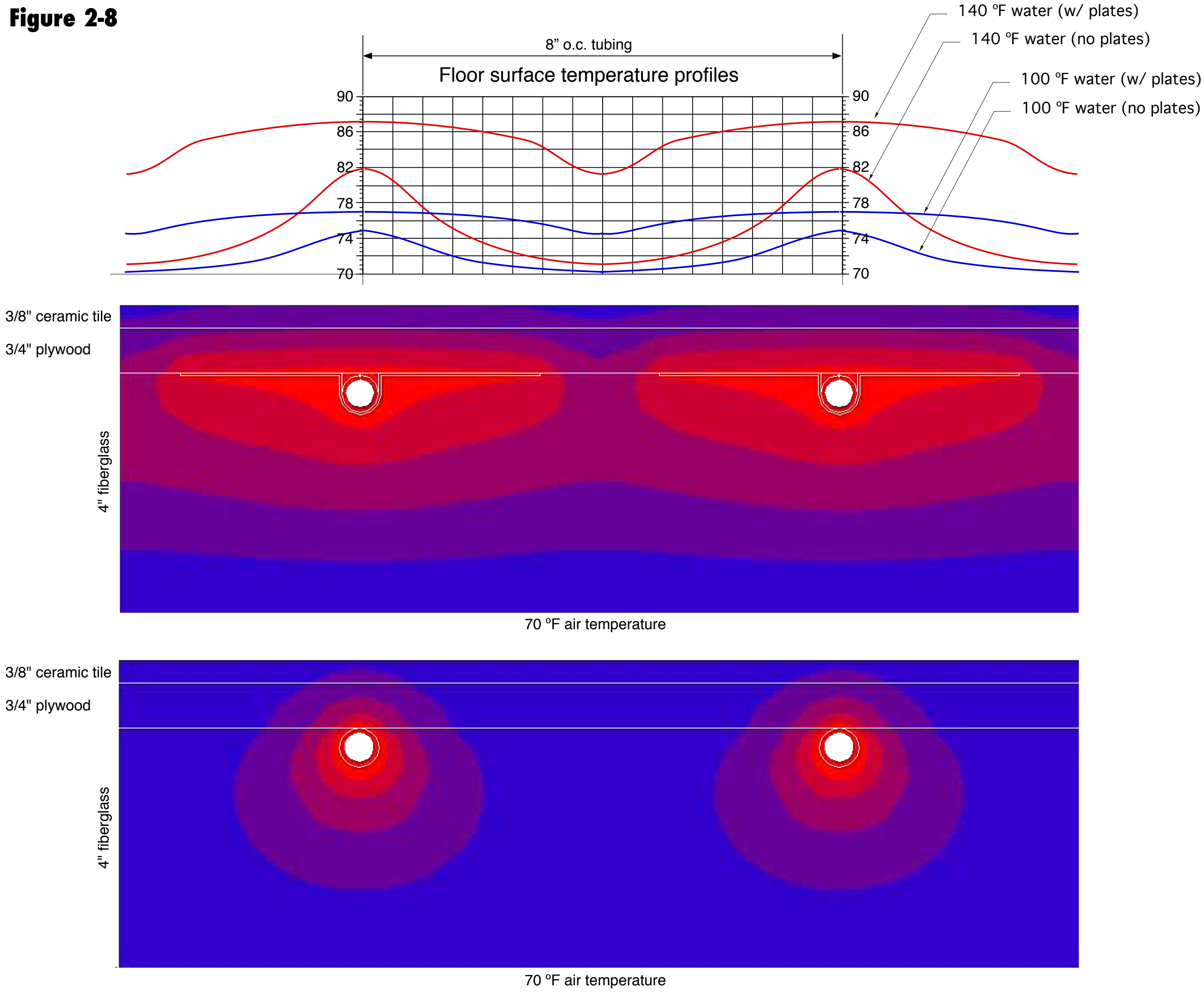
Another example that demonstrates conduction heat transfer is the use of formed aluminum heat transfer plates in combination with PEX or PEX-AL-PEX tubing in an "underside" tube & plate radiant floor construction.

Figure 2-6 shows an example of ½" PEX tubing stabled to the bottom of a plywood subfloor. The tubing is spaced approximately 8 inches apart. No heat transfer plates were installed.

Figure 2-7 shows a similar installation, but one where 6-inch-wide by 0.024-inch-thick pre-formed aluminum plates were installed to help conduct heat away from the tubing and spread it across the floor. The tube spacing is the same nominal 8 inches.

Figure 2-8 shows a thermal model of these two installations produced using finite element analysis. The

Figure 2-8

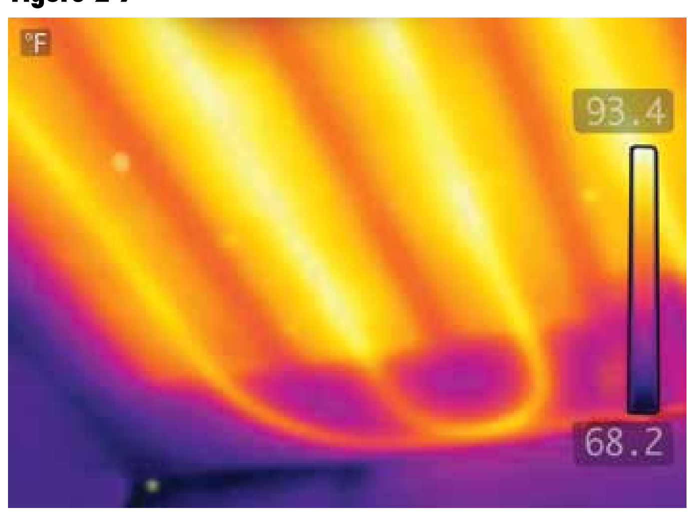


only variable between the two models is the presence of 6-inch-wide by 0.024-in.-thick pre-formed aluminum heat transfer plates in the upper graphic, and the absence of any plates in the lower graphic.

Analysis of the surface temperature profiles of both systems shows that, for the water temperatures simulated, the system using aluminum plates has almost *3 times more heat output* than the "plateless" system, all other factors being the same.

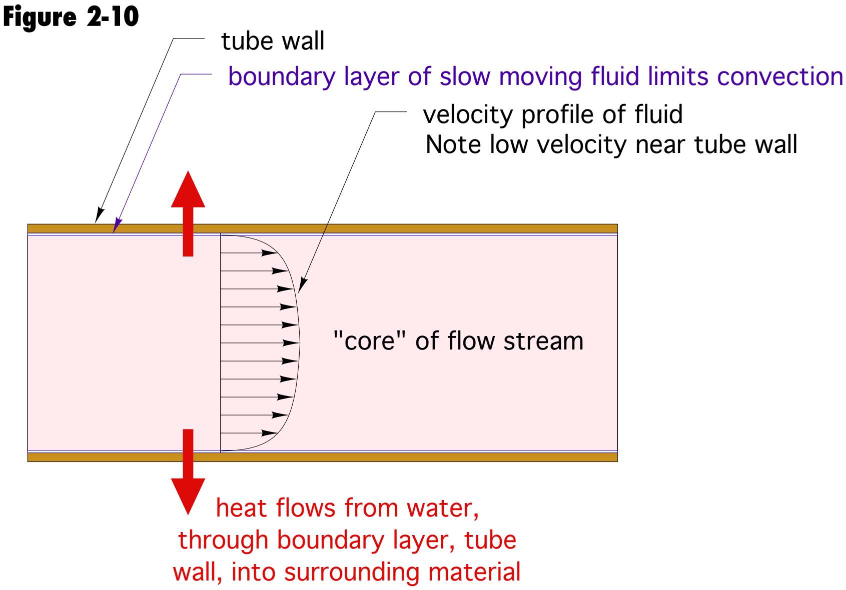
Figure 2-9 is a thermograph that also shows the importance of aluminum heat transfer plates in conducting heat away from tubing in a radiant *ceiling* heating application. This image is based on thermal radiation released from the lower surface of ½" drywall installed over the tubing and plates.

Figure 2-9



The location of the 6-inch-wide aluminum plates, placed on tubing spaced 8 inches apart, is easily seen as the bright yellow color, even though the plates and tubing are covered with ½" drywall. That fact that the drywall surface is significantly warmer under the plates compared to the gap between the plates indicates that the plates are effectively spreading heat, by conduction, away from the tubing. Contrast this with the "striping" seen where the tubing is present without the plates in the lower part of the image. As was the case with floor heating, the high thermal conductivity of the aluminum plates greatly enhances lateral heat diffusion across the surface of the radiant ceiling panel. The plates make the ceiling panel a more efficient heat emitter, allowing it to deliver heat to the room at a relatively low water temperature.

Figure 2-10



CONVECTION:

Convection heat transfer occurs as the result of fluid movement. The fluid can be a liquid or a gas. When heated, fluids expand. This lowers their density relative to surrounding cooler fluid. Lowered density increases buoyancy, which causes the warmer fluid to rise upward. Examples of the latter include warm air rising toward the ceiling in a room, and heated water rising to the upper portion of tank-type water heater. Both processes occur without circulators or blowers. As such, they are examples of "natural" convection.

Convection heat transfer is also responsible for moving heat between a fluid and a solid.

For example, consider water at 100ºF flowing along a solid surface that has a temperature of 120ºF. The cooler water molecules contacting the warmer surface absorb heat from that surface. These molecules are churned about as the water moves along. Molecules that have absorbed heat from the surface are constantly being swept away from that surface into the bulk of the water stream and replaced by cooler molecules. One can envision this form of convective heat transfer as heat being "scrubbed" off the surface by the flowing water.

The *speed* of the fluid moving over the surface significantly affects the *rate* of convective heat transfer.

Most people have experienced the increased "wind chill" effect as cold outside air blows past them, compared to how they would feel if standing in still air at the same temperature. The faster the air blows past them the greater the rate of convective heat transfer between their skin or clothing surfaces and the air stream. Although the person may *feel* as if the moving air is colder, it isn't. Instead, they're experiencing an increased *rate* of heat loss due to enhanced convective heat transfer. Achieving the same cooling sensation from calm air would require a much lower air temperature.

Convective heat transfer increases with increasing fluid speed. This happens because a layer of fluid called the "boundary layer," which clings to surfaces, gets thinner as the fluid's velocity increases (see Figure 2-10).

The thinner the boundary layer, the lower the thermal resistance between the bulk of the fluid stream and the surface. Less thermal resistance allows for higher rates of heat transfer between the fluid molecules in the bulk of the stream and the tubing wall.

The relationship between convective heat transfer and fluid speed past a surface can be seen in many hydronic systems. For example, the faster water flows through a finned-tube baseboard heat emitter, the higher it's heat output, with all other conditions being the same. This effect is evident in the heat output rating data listed by manufacturers of finned-tube baseboard. An example of such ratings, using numbers from a specific make and model of baseboard, is shown in Figure 2-11.

Figure 2-11

flow rate (gpm)	heat output (Btu/hr/ft) at stated water temperature (°F)							
flow rate (gpm)	110 °F	120 °F	130 °F	140 °F	150 °F	160 °F	170 °F	180 °F
1 gpm	150	200	260	310	370	430	490	550
4 gpm	**160**	**210**	**270**	**330**	**390**	**450**	**520**	**580**

At a given water temperature, the heat output at a flow rate of 4 gallons per minute (gpm) will always be slightly higher than at a flow rate of 1 gpm. This increase is due to a thinner boundary layer between the bulk of the fluid stream and the inner tube wall of the finned-tube element.

One way to rationalize the increase in heat output with increasing flow rate is to consider what happens to the *average water temperature* inside the heat emitter with changes in flow rate. Figure 2-12 shows an example.

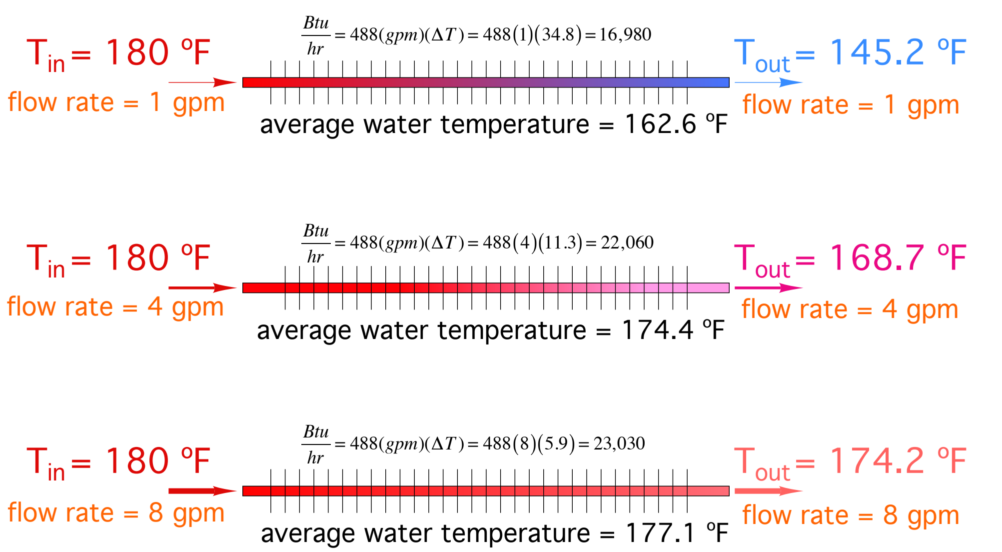
As flow rate increases, the temperature drop along the finned-tube element decreases. Since the inlet temperature remains constant, *decreasing temperature drop implies a higher average water temperature in the finned tube.* Higher average water temperature, in any heat emitter, results in higher heat output.

Never "judge" the rate of heat transfer from a heat emitter based solely on temperature drop. The rate of heat transfer is determined by the *multiplication* of flow rate times temperature drop. This is why the rate of heat transfer in Figure 2-12 increases as the flow rate increases, even though the temperature drop across the heat emitter is decreasing. This will be discussed more in section 3.

From the standpoint of heat transfer only, there is no such thing as the water moving "too fast" through a finned-tube element. This is also true for other heat emitters, such as the coil of an air handler, or a tubing circuit in a floor heating system. The faster the flow rate, the higher the rate of heat transfer.

Another example of increasing convective heat transfer with increasing flow rate can be found in heat output ratings for fan-coils. Figure 2-13 shows an example of how heat output increases with increasing water flow rate through the coil of the air handler, while the inlet water temperature and incoming air temperature remain constant.

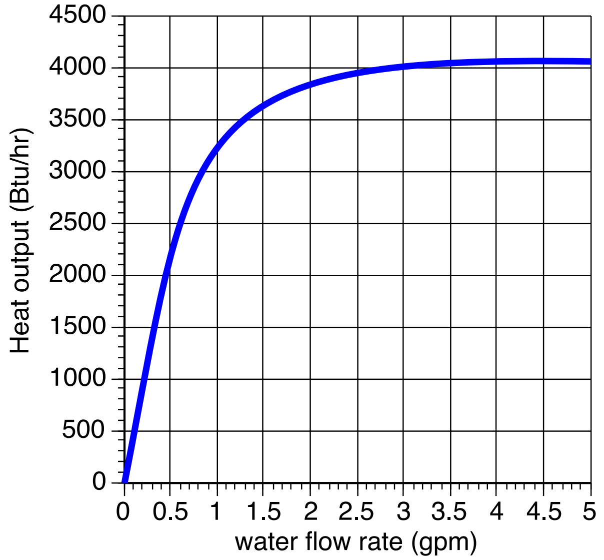
Figure 2-12



The heat output rate increases rapidly at low flow rates. It continues to increase as flow rate increases, but the *rate of increase* is much slower at higher flows.

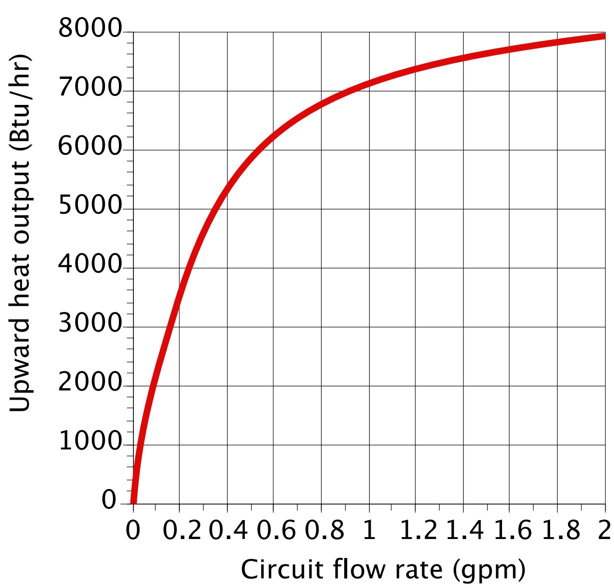
This demonstrates that there are practical limits on how much the heat output of a hydronic heat emitter can be increased based on increasing flow rates.

Figure 2-13



The head loss created by fluid moving through the heat emitter, in theory, increases with the *cube* of flow rate (e.g., if the flow rate is doubled, the head loss increases $2^3 = (8)$ times). Thus, attempting marginal gains in heat transfer by selecting circulators that can maintain high flow rates will significantly increase the installation and operating cost of the circulator. Velocity noise and erosion corrosion also become concerns at high flow rates. To avoid erosion corrosion, the flow velocity in smaller copper tubes should not exceed 5 feet per second.

Figure 2-14



Heat output also increases as the air flow rate through the air handler's coil increases. This happens because faster air flow reduces the resistance of the boundary layer between the bulk air stream and the surface of the coil.

The "non-linear" relationship between heat output and flow rate seen in Figure 2-13 is a characteristic of all hydronic heat emitters. Figure 2-14 shows the relationship for the upward heat output of a 250-foot-long circuit of 1/2-inch PEX tubing embedded at 12-inch spacing in a 4-inch bare concrete slab. The supply water temperature to this circuit is constant at 110ºF. The only thing being varied is flow rate.

As was true with the fan-coil, the gains in heat output are much more noticeable at lower flow rates. At 0.2 gallons per minute, which is only 10% of the maximum flow rate shown on the graph, the circuit releases about 44 percent of the maximum heat output. Increasing flow from 1 to 2 gpm only increases heat output about 11 percent.

NATURAL VS. FORCED CONVECTION

When fluid motion is caused by a circulator, a blower or any other powered device, the resulting convective heat transfer is called "forced convection." When the fluid motion is strictly the result of buoyancy differences within the fluid, the resulting convective heat transfer is called "natural convection."

The *room side* heat output of hydronic heat emitters such as finned-tube baseboard and wall cabinet heaters with internal coils (but no blower) is largely the result of natural convection. Such heat emitters are appropriately called "convectors."

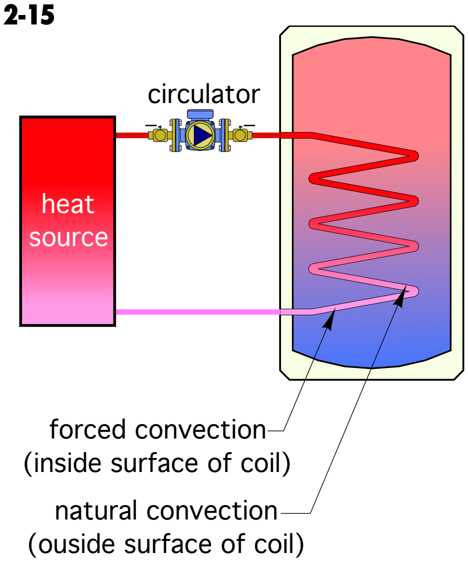
However, when a circulator is used to move water through these heat emitters, the convection occurring between the water and the inner surfaces of the heat emitter is forced convection.

Heat emitters that use fans or blowers to force air through a heat exchanger are usually called fan-coils or air handlers. The convective heat transfer that occurs on the room (air) side of these heat emitters is another example of forced convection.

Another situation in which both natural and forced convection are present is a heat exchanger coil immersed in a thermal storage tank, as shown in Figure 2-15.

Natural convection is usually a "weaker" form of heat transfer in comparison to forced convection. It's the result of slower fluid motion created by buoyancy

Figure 2-15



differences versus much faster motion created by a circulator. The slower fluid motion increases the thickness of the boundary layer, which creates greater resistance to heat transfer between the bulk of the fluid and the surface.

Forced convection versus natural convection explains why a small wall-mounted fan coil that's only 18 inches wide can provide the equivalent heat output of 10+ feet of finned-tube baseboard when both are operating at the same water supply temperature and flow rate. The rate of heat transfer from the finned-tube element in the baseboard is limited by natural convection heat transfer between its outer surfaces and the surrounding air.

The rate of convective heat transfer can be estimated using Formula 2-4.

Formula 2-4:

$$Q = hA(\Delta T)$$

Where:
Q = rate of heat transfer by convection (Btu/hr)
h = convection coefficient (Btu/hr•ft^2•ºF)
A = area over which fluid contacts a surface with which it exchanges heat (ft^2)
ΔT = temperature difference between bulk fluid stream and surface (ºF)

Although Formula 2-4 is relatively simple, determining the value of the convection coefficient (h) is often complex. The value of (h) can vary widely depending on the "geometry" of the surface relative to the fluid, the speed of the fluid, and the physical properties of the fluid (e.g. thermal conductivity, specific heat, density and viscosity). In many cases, the value of (h) needs to be determined experimentally. Heat transfer textbooks sometimes list values of h, or methods for determining (h) for very specific and often simplified situations. The table in Figure 2-16 shows how wide the range of (h) values can be for specific situations.

Figure 2-16

situation	range of convection coefficient (h) (Btu/hr•ft^2•ºF)
natural convection involving air	1-5
forced convection involving air	2-100
convection involving water	20-3000
convection involving boiling water	500-5000

THERMAL RADIATION:
Thermal radiation is probably the least understood form of heat transfer. To many, the word "radiation" denotes an undesirable or harmful effect associated with *nuclear* radiation. However thermal radiation and nuclear radiation are very different. Human skin releases thermal radiation similar to that emitted by a low-temperature hydronic radiant panel. A lightly clothed human body often releases over half of its metabolic heat production as thermal radiation. It's an entirely natural and harmless process.

Think of thermal radiation as *light*. More specifically, the thermal radiation emitted by components such as a panel radiator in a hydronic heating system is *infrared* light.

Human eyes cannot see infrared light. However, just like visible light, infrared light (a.k.a. thermal radiation) travels outward from its source in straight lines at the speed of light (186,000 miles per second). Although it cannot "bend" around corners, thermal radiant can be reflected by some surfaces. *It also travels equally well in any direction*, from some surface that emits it to another surface that absorbs it. This is why a heated ceiling can warm the objects and floor in the room below. Just as a ceiling-mounted lighting fixture can shine visible light onto objects below, thermal radiation can shine down from a warm ceiling and be absorbed by objects below.

The instant thermal radiation is absorbed by the surface of an object, it becomes heat that warms that object.

Consider the crucible of molten metal in a dark room shown in Figure 2-17.

Figure 2-17

Source: matrix.edu.au

The molten metal is likely in the temperature range of 2,500ºF. At that temperature, it emits both visible light (since it's obviously visible in an otherwise dark room) and thermal radiation. The latter would be immediately detected by any skin surface within a few feet of the crucible.

If the metal were allowed to cool, the color of visible light it emits would change from the bright yellowish/white seen in Figure 2-17 to deeper and deeper shades of red. The intensity of the visible light (e.g., its perceived brightness) would also decrease.

When the surface of the metal dropped below approximately 970ºF, it would no longer emit visible light. If the surrounding space was dark, the human eye could no longer detect the presence of the metal *by sight*. However, any exposed skin within a few feet of the cooling metal would confirm that it's still emitting profuse amounts of heat. *That sensation is caused by thermal radiation emitted by the surface of metal being absorbed by the surface of the exposed skin.*

The initial situation where the molten metal was emitting both visible light and thermal radiation changed to one where only thermal radiation was emitted. The change was caused by a shift in the wavelengths of the radiation emitted from the surface of the metal.

Unlike conduction or convection, thermal radiation needs no material (e.g., a fluid or solid) to transfer heat from one location to another. Thermal radiation cannot pass through a solid (as radiation). It can, to differing extents, pass through gases with minimal warming effect on those gases. Intense sunlight passing through very cold air in the upper layers of the earth's atmosphere is an example of the latter.

Consider a person standing a few feet away from a campfire on a cold day. If pointed toward the fire, their face probably feels warm, even though the air surrounding them is cold. This sensation is the result of thermal radiation emitted by the fire traveling through the cold air and being absorbed by their exposed skin. The air between the fire and the person absorbs very little of the energy being transferred from the flames to the person's face. Likewise, thermal radiation emitted from the warm surface of a heat emitter such as a panel radiator can pass through the air in a room without first heating that air. When the thermal radiation strikes another surface in the room, most of it is absorbed. At that instant, the energy carried by the thermal radiation becomes heat.

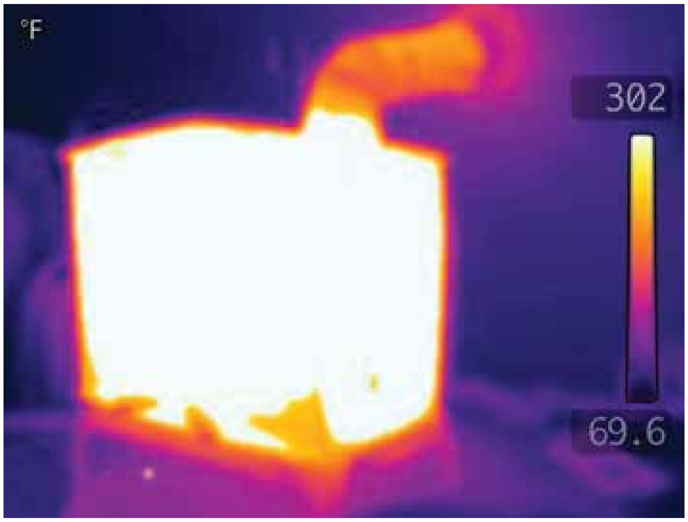
Every surface continually emits thermal radiation to any cooler surface within sight of it. The surface of a heat emitter that is warmer than our skin or clothing surfaces transfers heat to us by thermal radiation. Likewise, our skin and clothing continually give off thermal radiation to any surrounding surfaces at lower temperatures. A lightly clothed person standing next to a large cold window surface emits significant thermal radiation to that cold surface. This eventually leads to discomfort, even when the air temperature surrounding the person is in the normal comfort range of 68-72ºF.

Figure 2-18

When thermal radiation strikes an opaque surface, part of it is absorbed as heat and part is reflected away from the surface. The percentage of incoming radiation that is absorbed or reflected is determined by the optical characteristics of the surface and the wavelength of the radiation. Most interior building surfaces absorb the majority of thermal radiation that strikes them. The small percentage that is reflected typically strikes another surface within the room where most of it will be absorbed, and so on. Very little, if any, thermal radiation emitted by warm surfaces in a room escapes from the room.

Although the human eye cannot see thermal radiation, there are devices that can detect it, and display an image that uses colors to represent different surface temperatures. Such an image is called an infrared thermograph. Figure 2-19 shows an example of such an image for a hot wood stove.

Figure 2-19

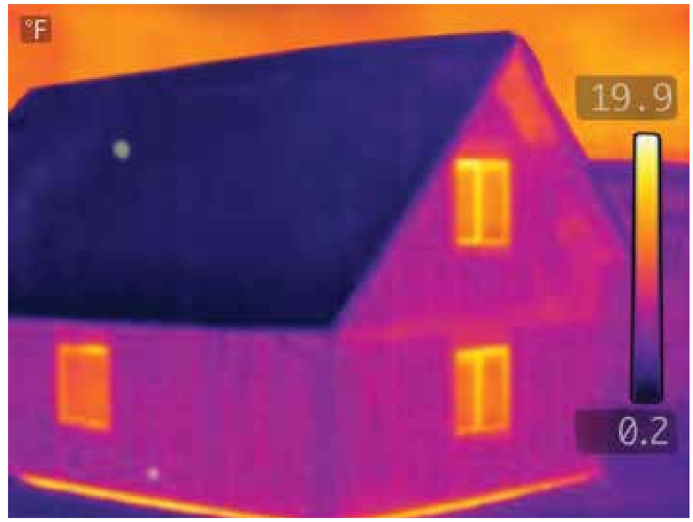


The thermal radiation coming from the wood stove is captured by the imaging system in a special camera and converted to visible colors. The correspondence between these colors and the surface temperatures is shown on the scale at the right of the image. This image shows that the sides and top of the stove are hot (around 300°F), while some of the materials behind the stove are at a normal room temperature of about 70°F.

Objects don't necessarily have to be "hot" to emit thermal radiation. The image in Figure 2-20 is an infrared thermograph of a small heated building on a cold winter night. The temperature scale shows that the hottest surface temperature detected is about 20°F.

A close look at Figure 2-20 shows the location of the wooden studs in the exterior walls. They show up because the exterior surface at the stud locations is slightly warmer than the exterior surface between the studs. The wooden studs have a higher thermal conductivity than the insulation material fitted between the studs, and thus create higher temperatures at the exterior surface of the building. The window areas show the highest exterior surface temperatures because they have lower thermal resistance than the insulated walls. The relatively dark color of the roof indicates a relatively cool surface. This suggests that the roof structure is well insulated, and thus there is minimal heat transfer from the interior space to the roof surface.

Figure 2-20



The rate thermal radiation transfers heat between two surfaces depends upon their temperatures, an optical property of each surface called emissivity, as well as the angle and distance between the surfaces.

The rate of heat exchange by radiation between two flat surfaces having the same area can be estimated using Formula 2-5.

Formula 2-5:

$$Q = \frac{sAF_{12}\left(T_1^4 - T_2^4\right)}{\left[\frac{1}{e_1} + \frac{1}{e_2} - 1\right]}$$

Where:
Q = rate of heat transfer from hotter to cooler surface by thermal radiation (Btu/hr)
s = Stefan Boltzmann constant = 0.1714×10^{-8} Btu/hr•ft^2•°R^4
F_{12} = shape factor between the two surfaces (unitless)
A = area of either surface (ft^2)
T_1 = *absolute* temperature of the hotter surface (°R)
T_2 = *absolute* temperature of the cooler surface (°R)
e_1 = emissivity of the hotter surface (unitless)
e_2 = emissivity of the cooler surface (unitless)

This formula is more complex that those used for estimating conduction and convection heat transfer.

The value of the "shape factor" (F_{12}) is a number between 0 and 1.0. It's determined based on the relative angle and distance between the two surfaces exchanging radiant heat. Heat transfer textbooks give specific methods for finding values of the shape factor (F_{12}) for different surfaces and orientations. For two parallel planes having infinite width and depth, the value of the shape factor (F_{12}) is 1.0.

e_1 and e_2 are the *emissivities* of surfaces 1 and 2. Emissivity is a surface property determined experimentally based on how well the surface emits thermal radiation. It must be a number between 0 and 1. A high value indicates that the surface is a good emitter, and vice versa. Emissivity values for various surfaces can be found in references such as heat transfer handbooks. Interestingly, the emissivity of a surface is not necessarily correlated with its color. A rough metal surface coated with *white* enamel paint has an emissivity of 0.91, and a flat black painted surface has an emissivity of 0.97. Freshly fallen snow can have an emissivity over 0.90. The emissivity of a polished copper surface is 0.023, while a heavily oxidized copper surface has an emissivity of 0.78. Most highly polished metal surfaces have low emissivities, and thus would not be good choices for the surface of a hydronic heat emitter that's expected to radiate heat into a room.

It's also important to understand that the temperatures (T1) and (T2) in Formula 2-5 must be *absolute temperatures*. Temperatures in ºF can be converted to absolute temperatures in degrees Rankine (ºR) by adding 458 degrees. Thus, 32ºF becomes 32 + 458 = 490ºR.

The mathematical result of the calculation (T_1^4-T_2^4) changes much more than the simple ∆T term used in the formulas for conduction and convection. For example, consider two surfaces exchanging radiant heat with temperatures of 100ºF and 80ºF. These temperatures would convert to 558ºR and 538ºR. The *difference* between these temperatures would be only 20ºR, the same as the difference between 100ºF and 80ºF. However, when these Rankine temperatures are used in Formula 2-5 the resulting number for the term (T_1^4-T_2^4) is 131,700,000ºR^4.

Due to mathematical complexities, as well as variability or uncertainty in properties such as surface emissivities, theoretical calculations of radiant heat transfer are often limited to relatively simple situations.

Other than situations where heat is exchanged in a vacuum, radiant heat transfer occurs in combination with convective heat transfer, since air is a fluid and is present between the objects exchanging heat. To simplify estimating overall heat transfer rates, some references provide hybrid heat transfer coefficients that *combine* the effects of radiant and convective heat transfer for typical situations where the variation in conditions is limited. An example is a hybrid heat transfer coefficient (m) for a heated floor, wall or ceiling panel that embodies the effect of both radiant and convective heat output. Figure 2-21 lists some values of hybrid heat transfer coefficients that embody both convective and radiant effects for commonly constructed interior building surfaces.

Figure 2-21

surface	combined radiative / convective coefficient (m) (Btu/hr/ft²/ºF)
radiant floor	2.0
radiant wall	1.8
radiant ceiling	1.6

These combined heat transfer coefficients greatly simplify the estimating of heat transfer in common (but limited) situations. These coefficients can be used in Formula 2-6.

Formula 2-6:

$$Q = mA(\Delta T)$$

Where:
Q = rate of heat transfer from both thermal radiation and convection (Btu/hr)
m = combined heat transfer coefficient (Btu/hr/ft^2/ºF)
A = area of the panel releasing heat to the room (ft^2)
∆T = difference between the *average* temperature of the *surface* releasing heat and room air temperature (ºF).

For example: If the *average surface temperature* of a heated floor was 80ºF, the floor area was 100 ft^2, and the room air temperature was 70ºF, the estimated heat output from the floor to the room would be:

$$Q = mA(\Delta T) = \left(2.0\frac{Btu}{\left(hr \cdot ft^2 \cdot {}^\circ F\right)}\right)\left(100\, ft^2\right)(80^\circ F - 70^\circ F) = 2{,}000\frac{Btu}{hr}$$

Keep in mind that this is an *estimate* based on a typical range of operating conditions and assumed common building materials. It is also based on *average surface temperature* rather than the water temperature circulating through tubing in the radiant surface. The average surface temperature of the panel, as well as the relationship between it and the water temperature in the tubing, are highly influenced by the material and "geometry" of how the radiant panel is constructed. Other references can be used to estimate the room side heat output of radiant floor, wall and ceiling panels based on specific panel construction and the average water temperature within the embedded tubing.

3. HOW FLOW AND TEMPERATURE CHANGE INFLUENCE HEAT EXCHANGE

All hydronic systems transport heat using streams of water. The water absorbs heat at a heat source, carries it through piping, fittings and other components, and eventually releases it to the space to be heated through one or more heat emitters. The flow rate of the water, as well as the temperature change it undergoes during the heat "delivery" process, affect the rate of heat transfer. This section describes the relationship between these factors, presents methods for estimating rates of heat transfer, and cautions against certain assumptions that are sometimes made about the fundamental relationship between flow, temperature change and rates of heat transfer.

SENSIBLE HEAT RATE EQUATION

Hydronic system designers often need to know the *rate* of heat transfer to or from a fluid flowing through a device such as a heat source or heat emitter. This can be determined using the sensible heat rate formula, given as Formula 3-1.

Formula 3-1:

$$q = (8.01Dc)f(\Delta T)$$

where:
Q = rate of heat transfer into or out of the water stream (Btu/hr)
8.01 = a constant based on the units used
D = density of the fluid (lb/ft^3)
c = specific heat of the fluid (Btu/lb/°F)
f = flow rate of fluid through the device (gpm)
ΔT = temperature change of the fluid through the device (°F)

When using Formula 3-1, the density and specific heat of the water (or water-based antifreeze solution) should be based on the *average* temperature of the liquid during the process by which the fluid is gaining or losing heat.

For *cold water only*, Formula 3-1 simplifies to Formula 3-2:

Formula 3-2

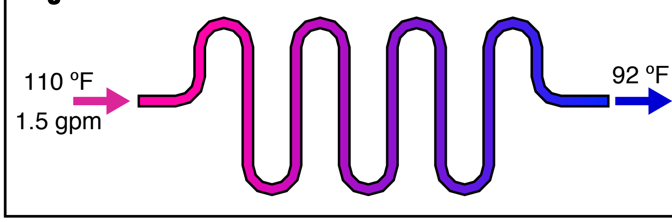
$$q = 500f(\Delta T)$$

where:
Q = rate of heat transfer into or out of the water stream (Btu/hr)
f = flow rate of water through the device (gpm)
500 = constant rounded off from 8.33 x 60
ΔT = temperature change of the water through the device (°F)

Formula 3-2 is technically only valid for cold water because the factor 500 is based on the density and specific heat of water at 60ºF. However, because the factor 500 is easy to remember, Formula 3-2 is often used for quick mental calculations of the rate of sensible heat transfer involving water. While Formula 3-2 is fine for initial estimates, Formula 3-1 is more accurate, especially for higher water temperatures, because it accounts for variations in both the density and specific heat of the fluid. Formula 3-1 can also be used for liquids (such as antifreeze solutions) that have different densities and specific heats relative to water.

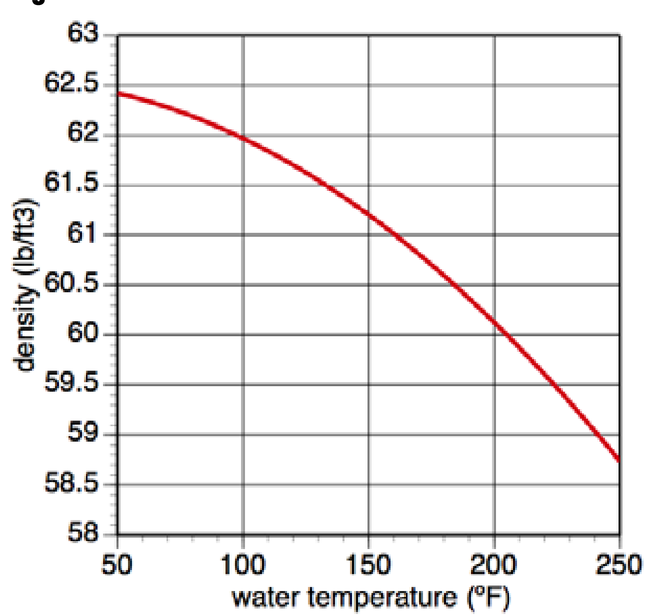
Example: Water flows into a radiant panel circuit at 110°F and leaves at 92°F. The flow rate is measured as 1.5 gpm, as shown in Figure 3-1. Calculate the rate of heat transfer from the water to the heat emitter using Formula 3-1:

Figure 3-1



Solution: To use Formula 3-1, the density of water at its average temperature of 101°F must first be estimated using Figure 3-2:

Figure 3-2



D = 61.96 lb/ft^3

The specific heat of water can be assumed to remain 1.0 Btu/lb/°F.

Putting these numbers into Formula 3-1 yields:

$$q = (8.01Dc)f(\Delta T) = (8.01 \times 61.96 \times 1.00) \times 1.5 \times (110 - 92) = 13{,}400\,Btu/hr$$

Formula 3-1 can be rearranged to determine the temperature drop or flow rate that would be required for a specific rate of heat transfer.

For example, what is the temperature drop required to deliver heat at a rate of 50,000 Btu/hr, using a distribution system operating with water at a flow rate of 4 gpm? Assume the average water temperature is 101°F

Solution: The density of water at 101°F was previously determined to be 61.96 lb/ft^3. After rearranging Formula 3-1, the numbers are entered and calculated.

$$\Delta T = \frac{q}{8.01cd(f)} = \frac{50{,}000\frac{Btu}{hr}}{\left(8.01\frac{ft^3 \cdot min}{gal \cdot hr}\right)\left(1\frac{Btu}{lb \cdot {}^\circ F}\right)\left(\frac{61.96lb}{ft^3}\right)(4\frac{gal}{min})} = 25.2^\circ F$$

It's important to realize that the 25.2°F temperature drop calculated in this example is *what would have to occur* for a water stream flowing at 4 gallons per minute to deliver 50,000 Btu/hr. Using Formula 3-1 does not "guarantee" that a water stream flowing at 4 gallons per minute with an average water temperature of 101°F *will* drop 25.2°F as it passes through a distribution system that was *intended* to deliver 50,000 Btu/hr. Whether it does or doesn't deliver this rate of heat transfer depends on system design, the actual (not assumed) room air temperature, the heat loss of the supply and return piping that connect the heat emitter(s) to the balance of the system, the condition of the internal surfaces of the heat emitters (e.g., are they scaled or corroded), and the condition of the external surfaces of the heat emitters (e.g., are they bent or covered with dust, pet hair, etc.).

Formula 3-1 should be used as a *"what is required"* rather than a *"what will be"* design tool.

For example, what would the temperature drop of a water stream flowing 0.5 gallons per minute have to be to deliver 50,000 Btu/hr to a load using water at an average temperature of 101°F?

Mathematically this is simple. Just put the given numbers into Formula 3-1 and calculate.

$$\Delta T = \frac{q}{8.01cd(f)} = \frac{50{,}000\frac{Btu}{hr}}{\left(8.01\frac{ft^3 \cdot min}{gal \cdot hr}\right)\left(1\frac{Btu}{lb \cdot {}^\circ F}\right)\left(\frac{61.96lb}{ft^3}\right)(0.5\frac{gal}{min})} = 201.5^\circ F$$

If the 4.0 gpm flow rate used in the previous example was reduced to 0.5 gpm, would the temperature drop of the circuit "automatically" change to 201.5°F to deliver the *intended* 50,000 Btu/hr heat output?

Absolutely not!

While it is possible to operate very specialized industrial hydronic systems with a temperature drop in the range of 200°F, under specific conditions, those systems are very different from typical hydronic space-heating systems. There is no way that a practical or cost-effective residential or light commercial hydronic heating system could operate with this calculated 201.5°F drop between the supply and return side of the distribution system.

Formula 3-1 provided a *"what is required"* value based solely on a mathematical relationship of the variables. It does not assure that the resulting numbers accurately describe a *"what will be"* situation in a real system. When *"what is required"* values determined by calculation seem very out of the ordinary, designers should carefully check their assumptions and their calculations. It's likely that a situation that seems too good to be true will be just that.

When all but one of the variables in Formula 3-1 are known *based on accurate physical measurements*, the remaining unknown can be calculated with confidence.

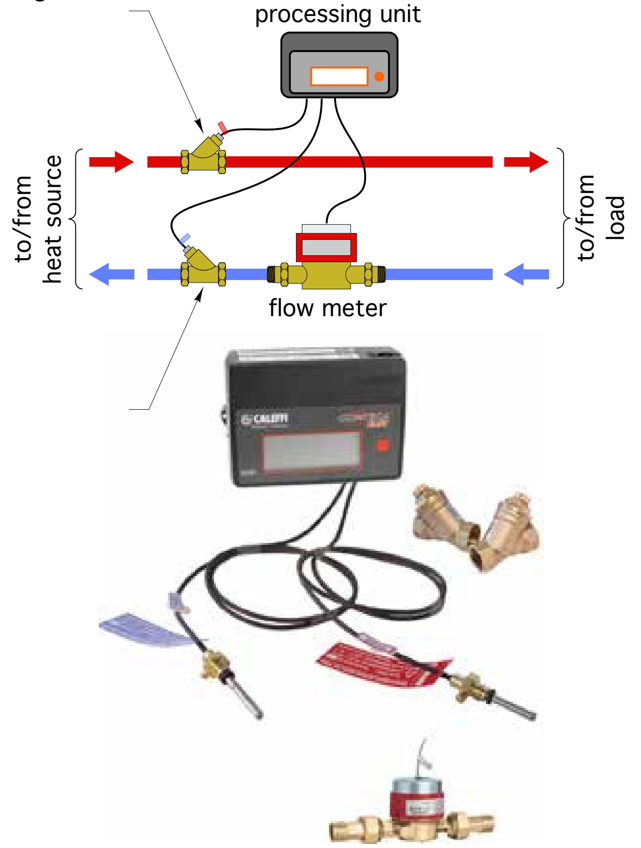
For example, if a stream of water with an *accurately measured* flow rate of 4.00 gpm passed through a hydronic distribution system, and *if* the *accurately measured* temperature drop from the supply to return of that distribution system was 25.19°F, and *if* the *accurately measured* average water temperature in the system was 101°F (e.g., so that the density is known to be 61.96 lb/ft^3), *then* the actual rate of heat delivery could be calculated using Formula 3-1.

$$q = 8.01cd(f)(\Delta T) = \left(8.01\frac{ft^3 \cdot min}{gal \cdot hr}\right)\left(1\frac{Btu}{lb \cdot {}^\circ F}\right)\left(\frac{61.96lb}{ft^3}\right)\left(4.00\frac{gal}{min}\right)(25.19^\circ F) = 50{,}007\frac{Btu}{hr} \approx 50{,}000\frac{Btu}{hr}$$

Using Formula 3-1 to calculate a rate of heat transfer in an operating system requires instrumentation for measuring flow rate and temperature change. Flow rate can be measured using a flow meter or a balancing valve equipped with an accurate flow indicator, such as shown in Figure 3-3.

Figure 3-3

Figure 3-4



Temperature drop can be measured with any accurate temperature indicator. Devices that simultaneously read two sensors and give a direct reading of ∆T are also available.

The instantaneous rate of heat transfer, as well as the total heat passing a given location in the system can also be measured using a heat meter, such as shown in Figure 3-4.

Heat meters combine an accurate flow meter with two calibrated temperature sensors. The latter are mounted in special fittings that allow a very precise measurement of ∆T.

The sensors and flow meter send data to a processing unit that calculates the instantaneous rate of heat transfer, based on Formula 3-1. The processing unit also integrates the heat transfer rate over time to determine the total amount of heat that has passed through the meter. The latter can be used to verify operation of the system, or as the basis of billing a customer for the total amount of heat used. Caleffi CONTECA heat meters are in conformance with the newly released ASTM E3137/ E3137M-17 heat metering standard.

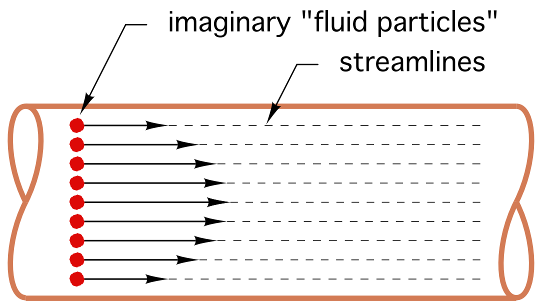
LAMINAR VS. TURBULENT FLOW

It's helpful to imagine liquids flowing through pipes as a stream containing millions of tiny "water particles." The path that any given water particle takes as it moves through the pipe is called a *streamline*.

In some situations, the streamlines of all the water particles are parallel to each other, like multiple lanes of traffic on a major highway. The water particles near the center of the pipe move faster than those near the inner pipe wall, but they don't "change lanes." This type of fluid movement, represented by Figure 3-5 is called laminar flow. It can be pictured 3-dimensionally as multiple concentric "layers" of liquid, each moving at a different speed, sliding past each other, as shown in Figure 3-6.

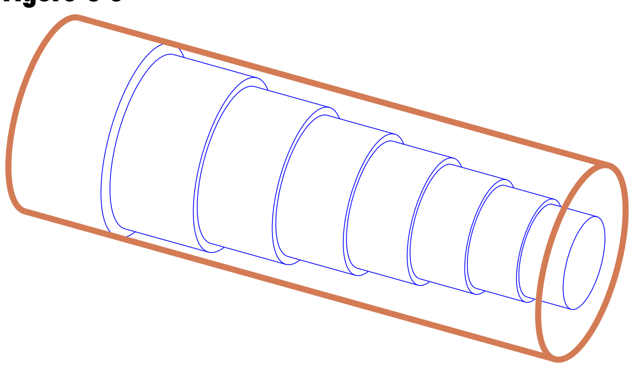
Laminar flow can be desirable or undesirable depending on where it occurs in hydronic systems, and the design goals for the system. If the goal is to *move fluid long distances with minimal pumping power,* laminar flow is desirable. However, if the goal is to *maximize heat transfer* from a flowing fluid to a tube wall or other solid surface, laminar flow is very undesirable.

Figure 3-5



Laminar flow streamlines

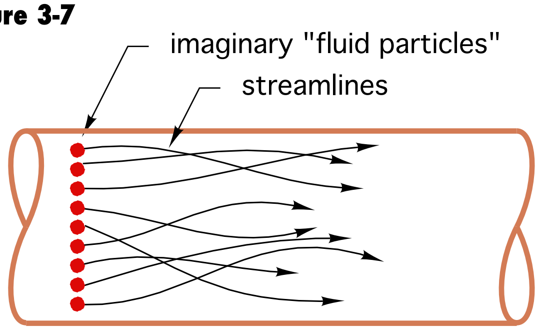
Figure 3-6



Because the water particles move in parallel streamlines during laminar flow, there is very little mixing between them. This increases the thickness of the boundary layer between the bulk of the fluid stream and the inner wall of the pipe. Thicker boundary layers decrease convective heat transfer.

Turbulent flow causes the streamlines of imaginary fluid particles to bend and twist as the fluid moves down the pipe, as shown in Figure 3-7.

Figure 3-7



turbulent flow streamlines

The erratically shaped streamlines present during turbulent flow create good mixing. A small quantity of fluid that's close to the tube wall one instant could be swept into the bulk of the fluid stream the next instant. This decreases the boundary layer thickness, and significantly improves convective heat transfer.

The convection coefficient associated with a relatively viscous liquid at low temperatures, such as a 25% solution of propylene glycol antifreeze at 30ºF, under laminar flow, can be only 7% of the convection coefficient of the same fluid under turbulent flow. This implies a very significant drop in heat transfer between a fluid and the inner wall of a tube if the flow is laminar versus turbulent. Such a change could have profound effect on the ability of a hydronic circuit, such as a ground loop heat exchanger supplying a geothermal heat pump, to transfer heat at the required rate.

Although turbulent flow improves convective heat transfer, it also increases head loss, and thus requires more pumping power to maintain a given flow rate relative to the power required for laminar flow. However, the increased heat transfer capabilities of turbulent flow are often more important than the penalty associated with the increased pumping power.

REYNOLDS NUMBER

It's possible to predict if flow through a pipe will be laminar or turbulent. It's based on calculating a dimensionless quantity called the Reynolds number of the fluid (abbreviated as Re#). If the calculated Re# is over 4000, the flow is turbulent. If it is below 4,000, the flow may be either laminar or turbulent. If the Re# is below 2,300, the flow will be laminar.

The Reynolds number for flow in a pipe can be calculated using Formula 3-3:

Formula 3-3:

$$\mathrm{Re\#} = \frac{vdD}{\mu}$$

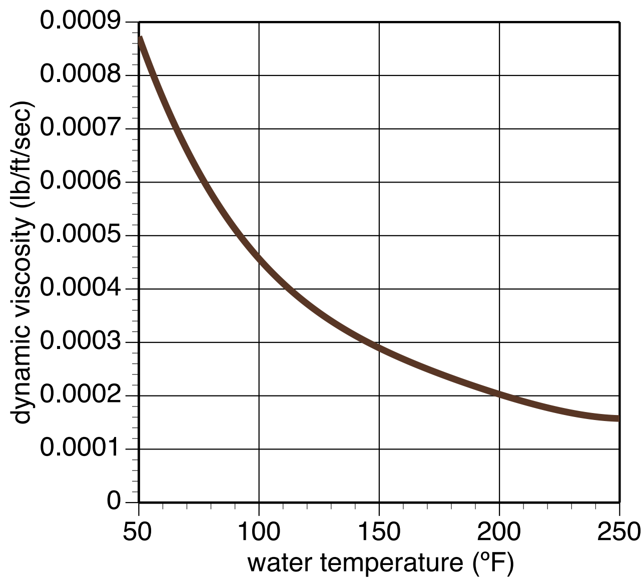
where:

v = average flow velocity of the fluid (ft/sec)
d = internal diameter of pipe (ft)
D = fluid's density (lb/ft^3)
μ = fluid's dynamic viscosity (lb/ft/sec)

For example, determine the Reynolds number of water at 140°F flowing at 5 gpm through a 3/4-inch type M copper tube. Is this flow laminar or turbulent?

Solution: To calculate the Re#, the density and dynamic viscosity of the water must be determined. The density of water can be found from Figure 3-2. The dynamic viscosity of water can be found from Figure 3-8.

Figure 3-8



At 140ºF, the density of water is 61.35 lb/ft^3. From Figure 3-8, the dynamic viscosity of water 140ºF is 0.00032 lb/ft/sec.

The inside diameter of a 3/4-inch type M copper tube is 0.811 inches. This must be converted to feet to match the stated units for Formula 3-3:

$$d = (0.811 \text{ in})\left(\frac{1 \text{ ft}}{12 \text{ in.}}\right) = 0.06758 \text{ ft}$$

The average flow velocity corresponding to a flow rate of 5 gpm can be found using Formula 3-4.

Formula 3-4:

$$v = \left(\frac{0.408}{d^2}\right)f$$

Where:
v = average fluid velocity (ft/sec)
d = exact inside diameter of pipe (inches)
f = flow rate through pipe (gpm)

For ¾" Type M copper tubing operating at a flow rate of 5 gpm, the average flow velocity is:

$$v = \left(\frac{0.408}{d^2}\right)f = \left(\frac{0.408}{(0.811)^2}\right)5 = 3.1\frac{ft}{\sec}$$

The Reynolds number can now be calculated.

$$\text{Re\#} = \frac{(3.106 \text{ ft/sec})(0.06758 \text{ ft})(61.35 \text{ lb/ft}^3)}{0.00032 \text{ lb/ft/sec}} = 40{,}242$$

This value is well above the threshold of 4,000, and therefore the flow is turbulent.

Notice that the units on the quantities used in the Re# formula cancel out completely. This serves as a check that the proper units are being used in Formula 3-3.

TEMPERATURE DROP IN A HYDRONIC SYSTEM

Formula 3-1 indicates that, with all other factors and conditions being equal, the *rate of heat delivery from a stream of water (or solution of antifreeze) is directly proportional to the temperature drop that stream undergoes as it passes through a hydronic distribution system.*

The ΔT at which a hydronic circuit operates is always determined, at any time, by the circuit's ability to release heat.

Consider a hydronic heating distribution system that's releasing heat at a rate of 100,000 Btu/hr into a space at 70ºF air temperature when supplied with 180ºF water at 9 gpm. The temperature drop of the circuit under these conditions is found using Formula 3-1:

$$\Delta T = \frac{q}{8.01cd(f)} = \frac{100{,}000\frac{Btu}{hr}}{\left(8.01\frac{ft^3 \cdot min}{gal \cdot hr}\right)\left(1\frac{Btu}{lb \cdot ^\circ F}\right)\left(\frac{60.7lb}{ft^3}\right)(9\frac{gal}{min})} = 22.85^\circ F$$

based on this calculation, the return water temperature from the distribution system would be:
180 - 22.85 = 157.15ºF

How would the output of this distribution system change if:

1. The room temperature was reduced from 70ºF to 65ºF with all other conditions being the same?

2. The supply water temperature was changed to 140ºF with all other conditions being the same.

Both changes can be estimated based on a fundamental tenant of hydronic heating, specifically:

The heat output of any hydronic distribution system will be approximately proportional to the difference between the supply water temperature and the room air temperature.

This implies that the following mathematical relationship can be written:

Formula 3-5

$$q = c \times (T_{supply} - T_{air})$$

Where:
q = rate of heat output from the circuit (Btu/hr)
c = a constant of proportionality that can be determined for each system
T_{supply} = supply water temperature to the circuit (ºF)
T_{air} = room air temperature (where heat is being released) (ºF)

If the circuit's heat output is known at a specific set of operating conditions, including a flow rate, supply water temperature, and room air temperature, the value of (c) in Formula 3-5 can be found. Substituting the previously stated conditions into Formula 3-5 yields:

$$100{,}000 = c \times (180 - 70)$$

$$c = \frac{100{,}000\,Btu}{(180^\circ F - 70^\circ F)} = 909\frac{Btu}{hr \cdot {}^\circ F}$$

For the distribution system being considered, it follows that the heat output formula is as follows:

$$q = c \times (T_{supply} - T_{air}) = 909 \times (T_{supply} - T_{air})$$

This formula can be treated as the "heat output formula" for this specific distribution system.

If the room air temperature was reduced from 70ºF to 65ºF, the estimated heat output of the distribution system would be approximately:

$$q = 909 \times (180 - 65) \cong 104{,}500\frac{Btu}{hr}$$

The circuit's heat output increased because the ∆T between its supply water temperature and the room air temperature increased. The new ∆T for the system would be:

$$\Delta T = \frac{q}{8.01cd(f)} = \frac{104{,}500\frac{Btu}{hr}}{\left(8.01\frac{ft^3 \cdot min}{gal \cdot hr}\right)\left(1\frac{Btu}{lb \cdot {}^\circ F}\right)\left(\frac{60.7lb}{ft^3}\right)(9\frac{gal}{min})} = 23.88^\circ F$$

Note that the ∆T of the distribution system changed based solely on a small change in the room air temperature.

If the supply water temperature was reduced from 180ºF to 140ºF, and all other conditions remained the same, the heat output of the distribution system would be:

$$q = 909 \times (140 - 70) \cong 63{,}630\frac{Btu}{hr}$$

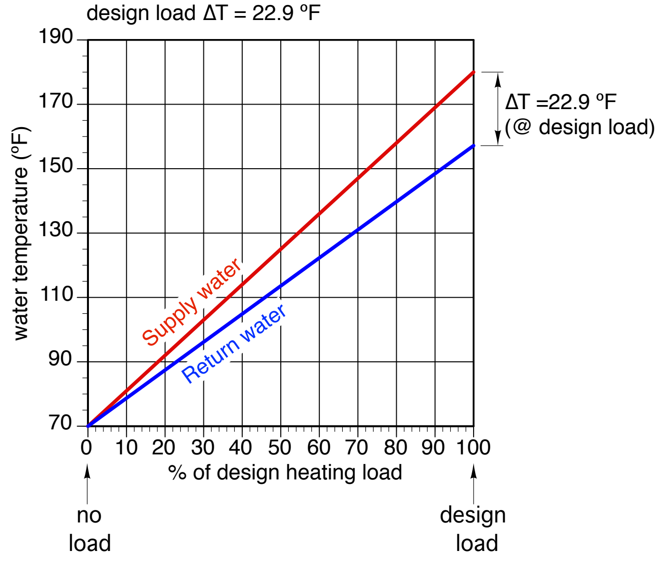
HOW SUPPLY TEMPERATURE AFFECTS ∆T

Because heat output is approximately proportional to the difference between the supply water temperature and the room air temperature, *and when the circuit flow rate is constant*, the ∆T between the supply and return of any hydronic system must decrease as the supply water temperature decreases.

Figure 3-9 shows the relationship between the supply and return water temperature for a specific distribution system that supplies a design output 100,000 Btu/hr into a space at 70ºF, using a supply water temperature of 180ºF and a flow rate of 9 gpm.

Figure 3-9

Supply & return water temperature based on constant flow rate
design load = 100,000 Btu/hr
flow rate = 9 gpm (water)
supply water temp. @ design load = 180 ºF
room air temperature = 70 ºF (constant)
design load ∆T = 22.9 ºF



Although the value of the supply water minus return water ∆T will be different for different hydronic systems using different heat emitters, the relationship between supply water temperature and return water temperature will be similar to that shown in Figure 3-9 (e.g., the ∆T will shrink from a maximum value at design load to 0 at no load).

CONSTRAINED ∆T

It is possible to vary the flow rate through a hydronic heating distribution system so that the temperature drop from the supply to the return remains constant. This method of control is called constrained ∆T. It's appropriate in systems that have *all* the following attributes:

1. Multiple heating zones controlled with valves.

2. Low thermal mass distribution systems.

3. Maintain a *constant supply water temperature, at the design load value*, whenever any zone is calling for heat.

Consider an accurately designed low thermal mass hydronic heating system that supplies design load heat output when all zones are active and the supply water temperature remains constant at the design load value. An example of such a system is shown in Figure 3-10.

If the heat source was not oversized for the design load, all zones would, in theory, remain on until the design load condition subsided (or other factors such as internal gains or intentional thermostat setbacks began influencing the zone loads).

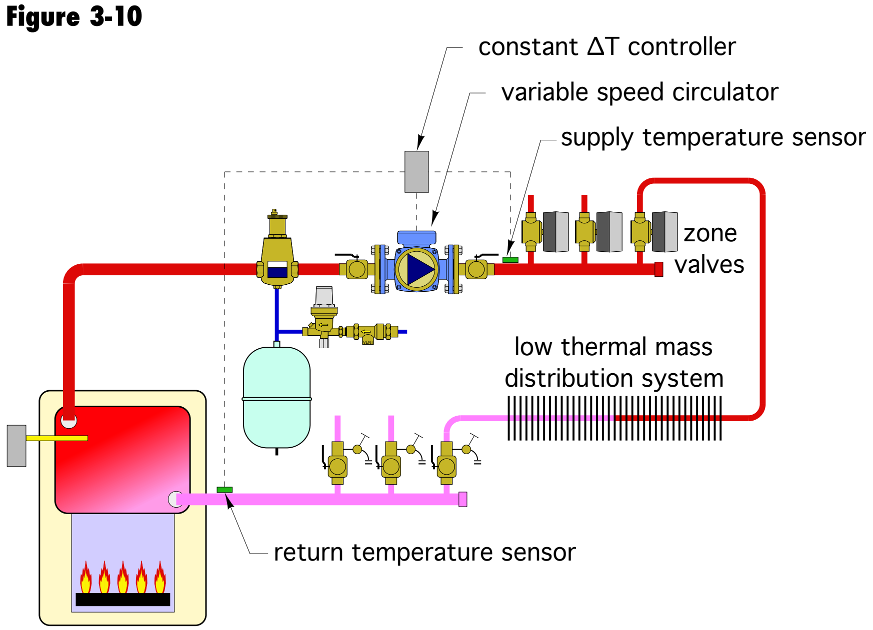
When design load is no longer present in one zone, and it turns off, less heat would be removed from the distribution system. This change would reveal itself as an *increase in return water temperature* (assuming that the supply water temperature is fixed). The ∆T of the circuit would decrease. A controller that senses this decrease could respond by reducing the speed of a circulator so that the design load ∆T was reestablished for the zones that remain active. This process would repeat when another zone turned off. This method of control reduces circulator energy use during partial load conditions.

When a zone turned on, *and the supply water temperature remains fixed at the design load value*, the return water temperature decreases because more heat is being removed from the distribution system. A controller could sense this increase in ∆T and respond by increasing circulator speed to reestablish the design load ∆T for all the active zones.

The requirement that the distribution system have low thermal mass implies that the temperature changes on the return side of the system would occur quickly as zones turn on and off. A high thermal mass distribution system, such as a heated concrete floor slab, could significantly delay these temperature changes due to heat being absorbed into or released from the thermal mass.

This method of control forces the active portion of the system to operate at design load conditions based on supply and return water temperatures. When a zone doesn't require design load heat input, the thermostat in that zone would have to completely stop flow.

Figure 3-10

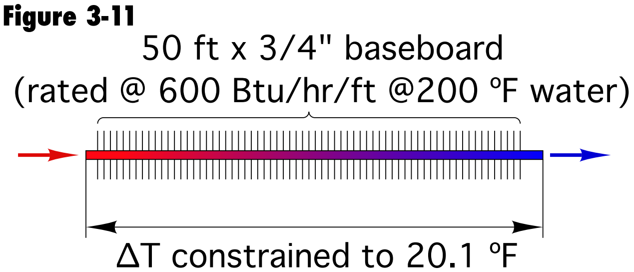


This type of system, in effect, directs "pulses" of heat into each zone whenever the associated zone thermostat calls for heat and opens the corresponding zone valve. *The rate of heat delivery during each pulse remains at the design load rate.* The duration of each pulse is the time that the zone valve is open. The heat transfer rate multiplied by the time duration of the heat input pulse determines the total heat added to the space during that time. This "pulsed" method of heat delivery has been used in millions of North American hydronic systems. It is generally acceptable if the thermostat differential is reasonable.

However, not all hydronic systems meet the three previously stated constraints. Many modern systems now use outdoor reset control to vary the water temperature supplied to the distribution system. When this method of water temperature control is combined with a controller that varies flow rate to maintain a *fixed* ∆T, the heat output from the distribution system decreases faster than it should based on outdoor reset control theory.

This can be illustrated by determining the flow rates that would be necessary to maintain a constant ∆T as the supply water temperature to the system decreases.

Figure 3-11

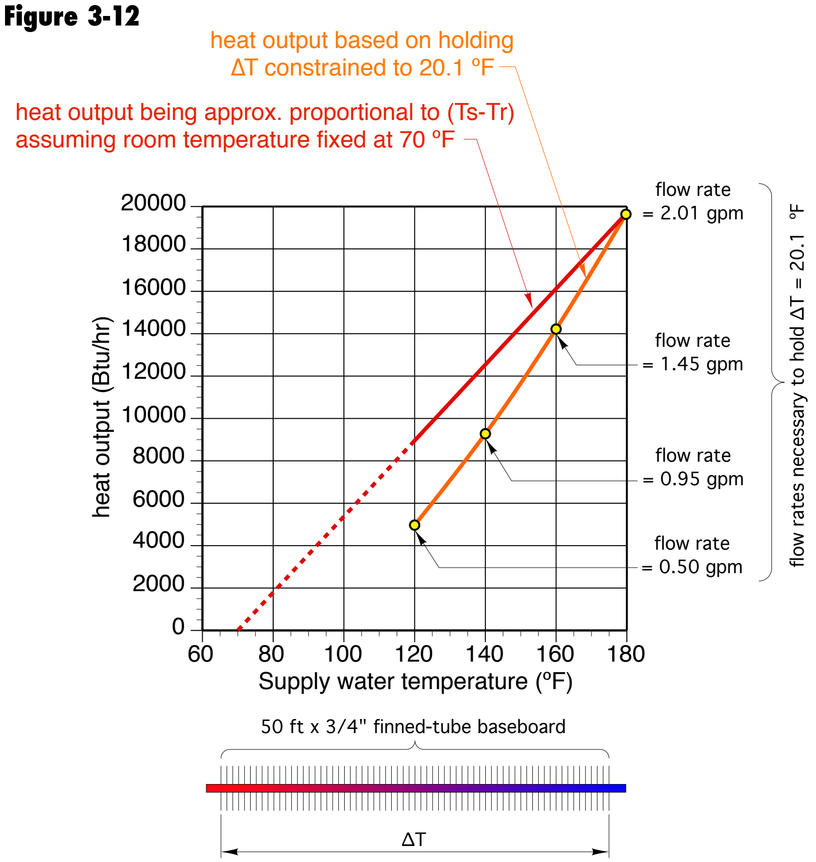


Consider a 50-foot-long finned-tube baseboard, as shown in Figure 3-11.

Imagine an experiment where the inlet water temperature to the baseboard will be varied from 180ºF down to 120ºF in increments of 20ºF. The flow rate at each inlet temperature will be adjusted to produce a nominal 20ºF temperature drop along the baseboard. The corresponding heat output of the baseboard will then be calculated. The results are shown in Figure 3-12.

As the supply water temperature decreases, with the ∆T constrained to 20.1ºF, the heat output of the baseboard, shown by the orange line, drops rapidly. For comparison, the red line shows how the baseboard's heat output would normally decrease in proportion to the difference between the supply water temperature and the room air temperature. This proportional relationship is the basis of operation for many outdoor reset controllers (standalone or integrated into boilers). At a supply water temperature of 120ºF, and with the ∆T constrained to 20.1ºF, the heat output of the baseboard is only about 55% of the output, based on the previously mentioned proportionality. *This suggests that heat output under the combined effects of 1) reduced supply water temperature based on outdoor reset, and 2) constrained ∆T, could result in inadequate comfort under partial load conditions.*

Figure 3-12



WHY 20?

One misconception that has long been entrenched in the North American hydronics market is that all hydronic systems should operate at a temperature drop of 20ºF. The origin of this number in the North American hydronics market is unknown. Perhaps it came from the concept that ***if*** a system using water operates with a 20ºF temperature drop, the rate of heat delivery would be approximately 10,000 Btu/hr per gpm of flow rate. While this is a close approximation to the rate of heat delivery, it all hinges on the word "if." More specifically, *if* the system operates with a 20ºF temperature drop, then each gpm of water flow rate would transport about 10,000 Btu/hr.

There is nothing sacrosanct about operating a hydronic system with a 20ºF temperature drop. Systems can operate very well using much lower and much higher ∆Ts.

For example, if a very high flow rate was maintained through a radiant floor tubing circuit, that circuit might operate at a ∆T as low as perhaps 4ºF. That would make the average water temperature in the circuit only 2ºF lower than the supply water temperature. It would also make the floor area served by the circuit relatively uniform in surface temperature. So, what's wrong with operating a circuit under this 4ºF ∆T? From the standpoint of heat transfer, and achieving relatively uniform floor surface temperatures, there's nothing wrong. However, maintaining the high flow rate necessary for the low ∆T would require a larger circulator that would use significantly more electrical energy compared to that required to operate the system at a higher ∆T.

Figure 3-13 shows the ∆Ts that would occur in a radiant floor heating circuit based on holding the inlet temperature constant at 110ºF, and varying the flow rate from 0.52 to 2.65 gpm.

At the 0.52 gpm flow rate, the circuit's ∆T is almost 26ºF. At the 2.65 gpm flow rate, the circuits flow rate would be 7.3ºF. The heat output of the circuit increases with increasing flow rate because the average water temperature in the circuit is increasing. However, the gains become smaller and smaller at higher flow rates.

It's also possible to operate hydronic circuits at temperature drops much higher than 20ºF. In Europe, panel radiator systems are sometimes designed for a 20ºC temperature drop under design load conditions. 20ºC converts to 36ºF. Using the higher ∆T allows the flow rates to each panel radiator to be reduced. If the ∆T was increased from 20 to 36ºF, the required flow rate would decrease to (20/36) = 56 percent of the flow rate required at 20ºF. This may allow smaller tubing to be used to supply the radiator. It may also allow a smaller circulator with a lower power motor to be used.

Figure 3-14 shows a thermograph of a panel radiator operating with an estimated 40ºF ∆T between its supply and return water temperature. The ∆T was estimated based on the temperature scale and colors on the supply and return piping.

Figure 3-13

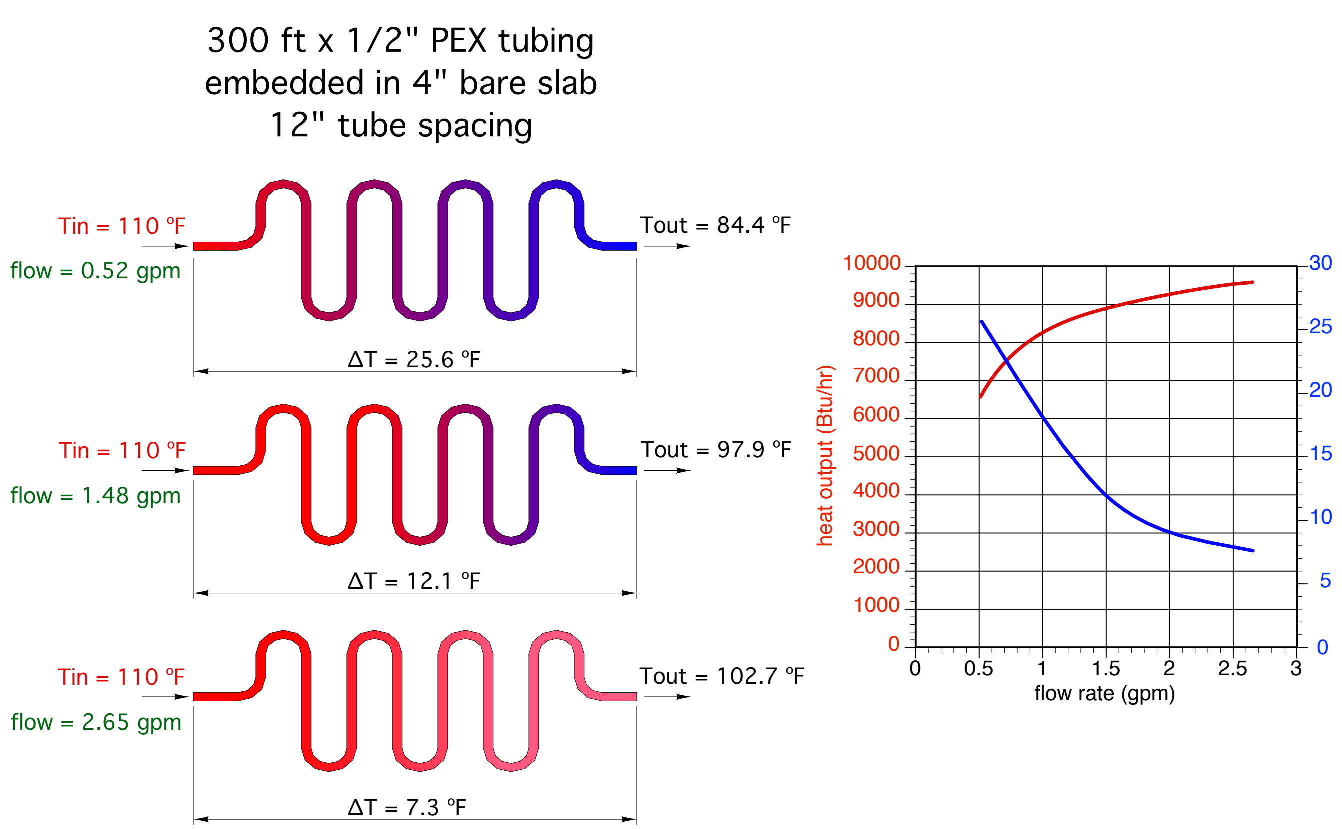
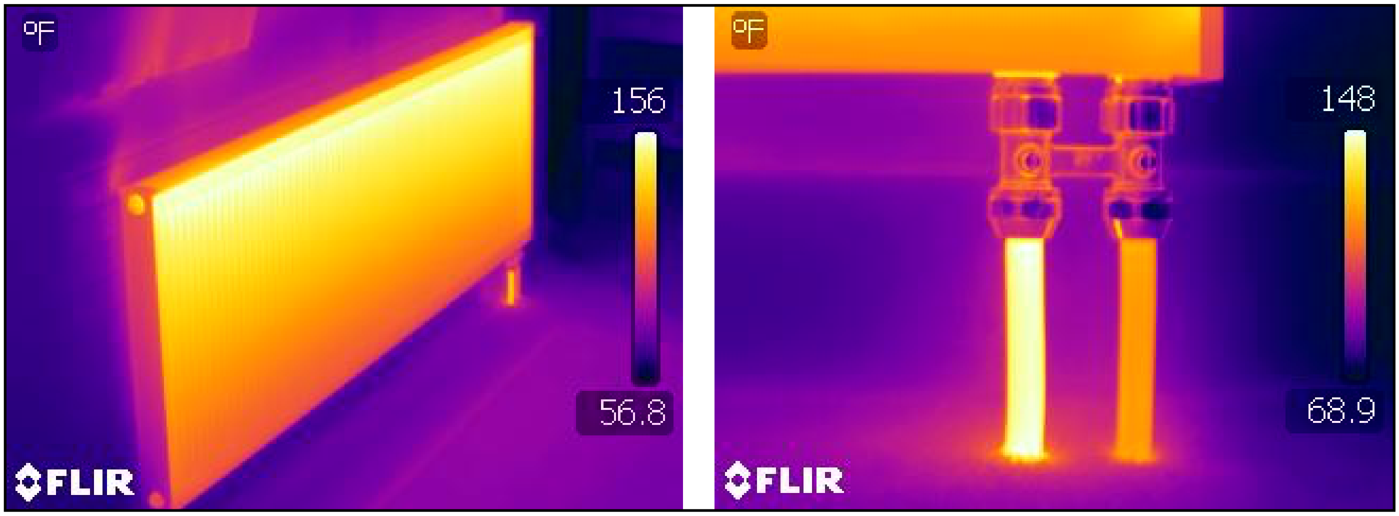


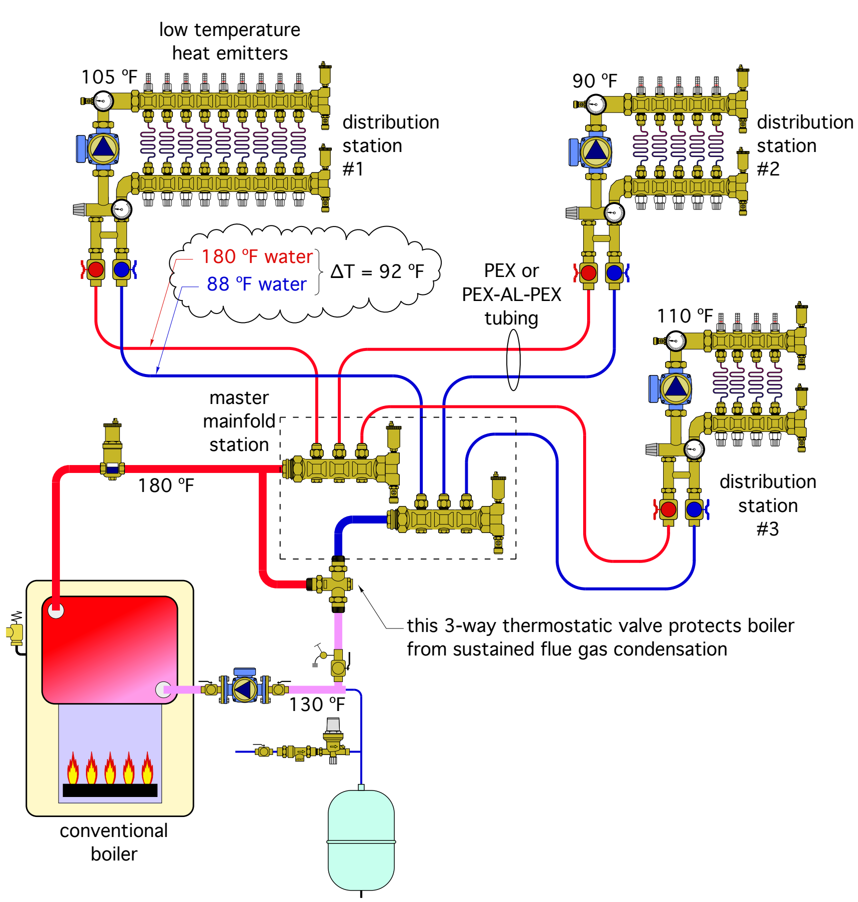
Figure 3-14



Notice the relatively even temperature gradient from the top to the bottom of the panel radiator. This indicates even flow distribution among the panels' vertical flow channels.

There are also applications in which *very high* ΔTs are possible in portions of a hydronic heating system. They typically involve the combination of a higher temperature heat source, such as a conventional boiler, and low-temperature heat emitters, such as radiant floor panels. Figure 3-15 shows an example.

Figure 3-15



Under design load conditions, this system supplies water at 180ºF to three Caleffi distribution stations. At distribution station #1, this hot water is mixed with return water from the radiant panel circuits at 88ºF, to achieve a supply water temperature of 105ºF. The radiant panel circuits served by the distribution station operate at a ΔT of 105 - 88 = 17ºF under design load conditions. The portion of the 88ºF water not used for mixing returns from the mixing manifold station.

Although the radiant panel circuits operate at a typical design load temperature drop, the ΔT for the tubing set carrying water between the boiler manifold and distribution station #1 operate at a ΔT of 180 - 88 = 92ºF. At this ΔT, each gallon per minute of water flow transports approximately 45,000 Btu/hr from the boiler manifold to the distribution station. This allows small diameter flexible tubing to carry a substantial rate of heat transfer. Low flow rates and small flexible tubing help reduce installation cost.

4: OPERATING TEMPERATURE OF HYDRONIC HEATING SYSTEMS

The first law of thermodynamics can be expressed as follows:

Energy input = Energy Output + Energy Stored

This principle of energy conservation is a widely used "starting point" when analyzing physical systems involving energy flow. It cannot be circumvented. Any analysis, method or product that claims to have "side-stepped" the first law of thermodynamics is flawed.

When this law is applied to hydronic heating systems, the energy flows can be categorized as thermal energy and electrical energy. Thermal energy (e.g., heat) is the dominant form of energy in hydronic systems. The electrical energy used in hydronic systems (other than those with electrically operated heat sources) is for operating the distribution system. *In well-designed hydronic systems,* electrical energy use is very small in comparison to thermal energy use. It is often treated as insignificant, and as such, is not included in the thermodynamic analysis of the system. In such cases, the first law can be limited to thermal energy flows in the system and stated as follows:

Heat generated by the heat source = heat released from the system + heat stored within the system.

The ability of hydronic heating systems to store heat varies significantly. Systems with high thermal mass components, such as a concrete floor slab with embedded tubing, or a large thermal storage tank holding several hundred gallons of water, can store large quantities of heat. Systems with low metal and water volume store minimal amounts of heat.

The way in which a hydronic heating system is *operated* also affects how the heat it stores behaves. For example, in a high thermal mass system operated with constant circulation and outdoor reset control of supply water temperature, the influence of the stored heat is minimal under normal conditions. This type of system makes very gradual changes in its rate of heat output. However, if the water flow through that system is turned on and off, and if the supply water temperature always remains close to the temperature required at design load conditions, the influence of the thermal mass in determining room temperature will be much greater. Wider swings in room temperature are likely.

Precise analysis of energy flows in high thermal mass hydronic systems must address transient system response. The word transient implies that temperatures and possibly flow rates within the system are changing significantly with time. This further implies that the system's thermal mass is absorbing or releasing heat. Transient system analysis is mathematically complex, and as such, is usually done through computer simulation.

A simplified analysis examines a system that is at or very close to steady state conditions. Steady state means that the heat stored in the system's thermal mass is neither increasing or decreasing. Under steady state conditions, the water temperatures and flow rates in various parts of the system are not changing with time. Steady state analysis produces "snapshots" of system performance, rather than showing how operating conditions change over time.

One of the most common "snapshots" of steady state performance are when the system is operating at its maximum heat conveyance rate. This is called "design load conditions." Designers use this condition to size equipment that can maintain specified comfort conditions when the building is losing heat at a maximum rate based on the statistically lowest outdoor temperature, and assuming all zones in the system are operating.

Because design load conditions assume steady state, the thermal mass of the system is not included in the analysis. The first law of thermodynamics now simplifies to:

Rate of heat input by the heat source = rate of heat release by the distribution system

The condition described by this relationship is called *thermal equilibrium*. It's a condition that all hydronic systems inherently try to establish and maintain. Assuming that controls in the system do not intervene, and flow rates remain steady, thermal equilibrium will eventually occur at any rate of heat input by the heat source. When it does, the temperatures in all parts of the system remain constant.

When a hydronic heating system begins operation from a cold start, the heat source is injecting heat to the system's fluid at a rate much higher than the rate of heat dissipation from the relatively cool distribution system. The fluid temperature in all portions of the system increases under this condition. Assuming there is no interference by system controls, this increase in temperature continues until the system "finds" it thermal equilibrium condition, where the rate of heat input exactly matches the rate of heat dissipation. If there are no changes to the heat input rate, or the ability of the distribution system to release heat, the system remains at thermal equilibrium indefinitely.

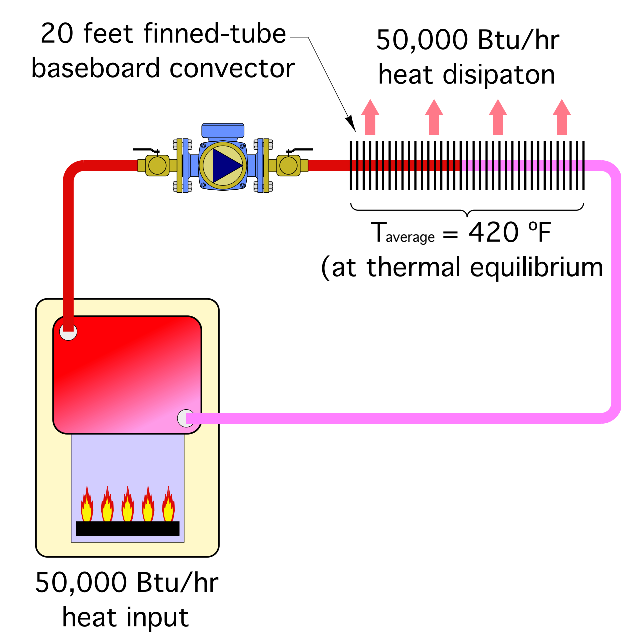
If the rate of heat output at the heat source changes, or there's a change in the distribution system, such as a zone circuit turning on or off, thermal equilibrium is temporarily disrupted. However, the system immediately seeks to reestablish thermal equilibrium under the new operating conditions. If the system's thermal mass is small, thermal equilibrium will be reestablished within a few minutes. Systems with high thermal mass take longer, in some cases several hours, to reestablish thermal equilibrium conditions.

When the rate of heat input to the system increases, so do the water temperatures within the distribution system. These temperature increases allow the distribution system to dissipate heat at a higher rate. When the rate of heat input decreases, so do the water temperatures within the system. The reduced temperature causes the distribution system to dissipate heat at a lower rate.

It's important to understand that *EVERY* hydronic system always seeks to operate at thermal equilibrium. The conditions necessary for this to occur may not produce the desired heat delivery. They may not allow the system to operate efficiently. *They may not even allow the system to operate safely*. Still, unless controllers within the system intervene, the system always strives to achieve thermal equilibrium.

Consider a hypothetical system with a 50,000 Btu/hr heat source and 20 feet of finned-tube baseboard, as shown in Figure 4-1.

Figure 4-1



Assume that there are no controllers in this system, no pressure relief valve, and that all components used have very high temperature and pressure ratings. Also, assume that the room air temperature is 70ºF.

What average water temperature would be necessary in the baseboard at thermal equilibrium?

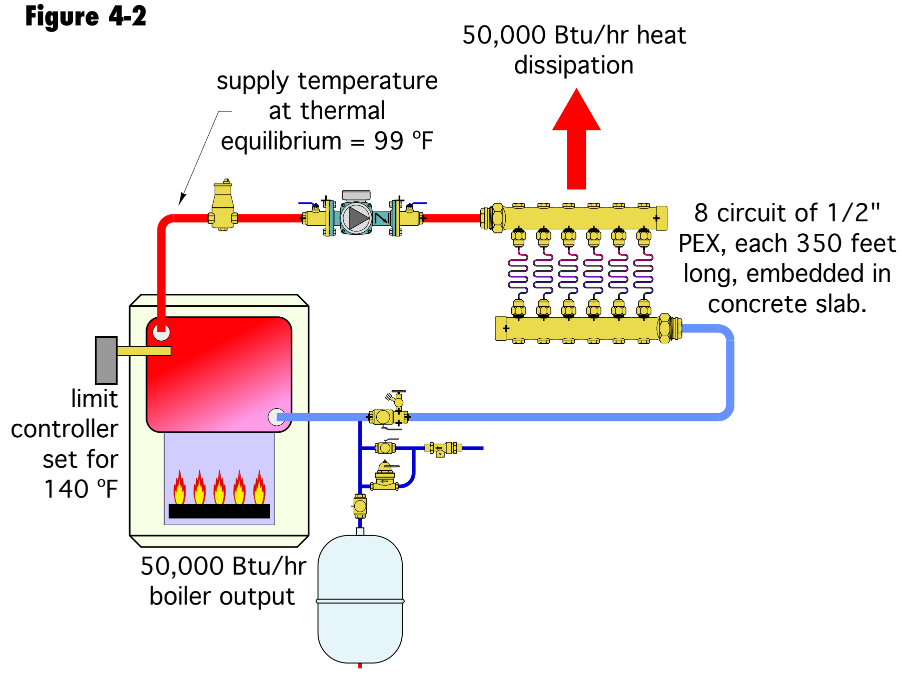
Based on a detailed performance model for the baseboard, the required *average* water temperature in the 20-foot-long finned-tube baseboard would be 420ºF! It is possible to maintain water in a liquid state at this temperature, but it requires a gauge pressure of 294 psi! This water temperature and its requisite pressure to maintain a liquid state is much higher than practical in any residential or light commercial hydronic system. Still, it represents a thermodynamic condition that the system is striving for. Fortunately, if this were a real system, one or more safety devices, such as a boiler high limit aquastat or pressure relief valve, would interrupt the process long before it reached such a dangerous condition.

Many hydronic systems operate in a similar manner. Each time the heat source is turned on, the system's water temperature begins climbing toward thermal equilibrium, but the operating controls on the system interrupt that progression when a specific temperature or pressure condition is attained at which the designer wants the heat source to stop producing heat. A common example is the high limit controller on a boiler turning off the burner when the water temperature in the boiler reaches a set "high limit" value, such as 190ºF. The system's circulator continues to operate, and thus heat continues to flow from the thermal mass of the boiler to the distribution system. After a few minutes, the water temperature inside the boiler has dropped to perhaps 170ºF due to heat dissipation, and the burner is turned back on. Each boiler cycle represents an attempt by the system to reach thermal equilibrium that is subsequently interrupted by the high limit controller, so that operating conditions remain safe and reasonably efficient.

Here's another example: Imagine a hydronic floor heating system that has eight parallel 350-foot circuits of ½" PEX tubing embedded in a bare concrete slab. The system is directly piped to a 50,000 Btu/hr boiler, as shown in Figure 4-2. The boiler's temperature limiting controller has been set by the installer for 140ºF because that's the temperature the installer thinks the boiler should supply to the distribution system. Also, assume that the load on the system remains constant at 50,000 Btu/hr.

When the system turns on, the boiler's outlet temperature climbs over a period of several hours and eventually

Figure 4-2



stabilizes at 99ºF. The burner remains in continuous operation, but the water temperature leaving the boiler doesn't rise above 99ºF. The installer thinks there's something wrong because the boiler is not reaching the temperature (140ºF) that was set on the boiler's high limit controller. After several more hours of continuous operation, the boiler's outlet temperature remains at 99ºF. The installer now thinks that the high limit controller is defective, removes it, and heads to the wholesaler for a replacement.

There's nothing wrong with the boiler's high limit controller! The system simply reached thermal equilibrium at a water temperature well below the boiler's high limit setting. The floor heating subsystem was capable of releasing 50,000 Btu/hr of heat when supplied with 99ºF water. Thermodynamically, there was no need for the water temperature to rise higher. Because the limit controller was set well above this temperature, it could not intervene, and thus could not affect the operating conditions. Its presence is irrelevant, as far as "controlling" this system.

PREDICTING THERMAL EQUILIBRIUM

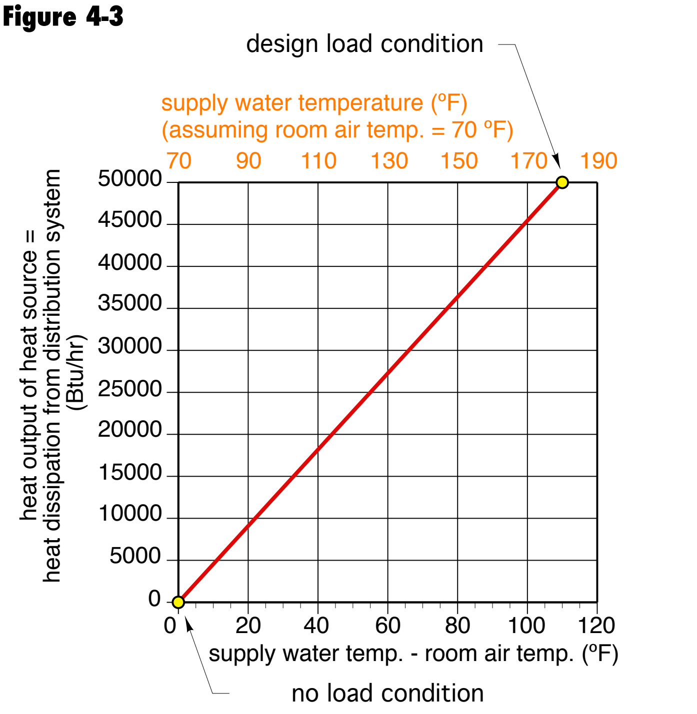
It's possible to estimate the supply water temperature at which a specified distribution system will reach thermal equilibrium. What's needed is an accurate relationship between the supply water temperature at design load, and the total heat output of the distribution system at that same condition. This information should be determined as part of any accurate design process.

Once a pairing of supply water temperature and corresponding heat output from the distribution system is established, a graph can be created based on the heat output of a distribution system being approximately proportional to the difference between supply water temperature and room air temperature.

Figure 4-3 shows this relationship for a hypothetical distribution system that's designed to release 50,000 Btu/hr into a space at 70ºF air temperature, when supplied with 180ºF water.

The vertical axis of Figure 4-3 indicates the system's heat output. The lower horizontal axis shows the difference between supply water temperature and room air temperature. This temperature difference can be thought of as the "driving ΔT" that

Figure 4-3



"pushes" heat from the water out though the heat emitters and into the room air.

The yellow dot at the top right of the graph represents design load conditions (e.g., a supply water temperature of 180°F, and a corresponding heat dissipation of 50,000 Btu/hr from the distribution system). The yellow dot in the lower left represents a "no load" condition. Supplying 70°F water to any distribution system contained within a space at 70°F air temperature would yield zero heat transfer. The red line represents the proportionality between these two extreme conditions.

The supply water temperature necessary to dissipate any rate of heat output between 0 and 50,000 Btu/hr can be estimated by finding that rate on the vertical axis, drawing a horizontal line to intersect the red line on the graph, and then a vertical line from this intersection to the lower horizontal axis. The number read from the horizontal axis is the ∆T between the necessary supply water temperature and the room air temperature.

For example, for the distribution system represented in Figure 4-3 to release 20,000 Btu/hr, the ∆T between supply water temperature and room air temperature needs to be approximately 44°F. If the room temperature was 70°F, the supply water temperature would need to be approximately 44 + 70 = 114°F. If the room temperature was to be 65°F, the necessary supply water temperature for the same rate of heat output would be 44 + 65 = 109°F.

SUPPLY WATER TEMPERATURE vs. HEAT EMITTER SURFACE AREA

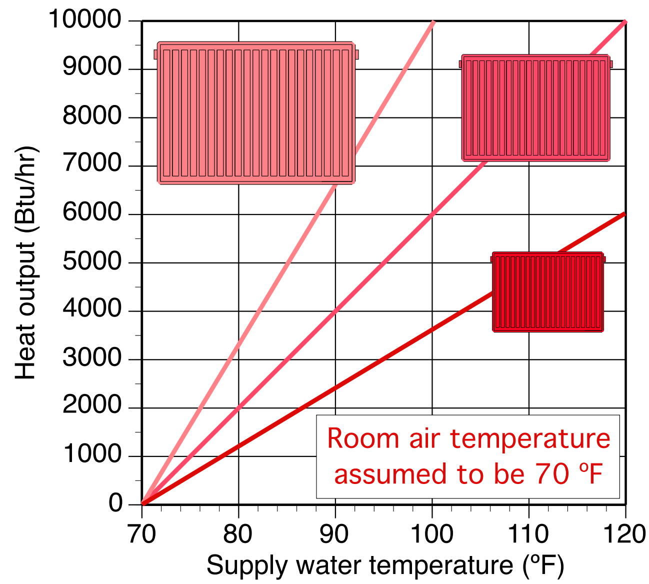
The slope of the line of heat output versus the difference between supply water temperature and room air temperature depends on the total surface area of the heat emitters in the distribution system. The greater the total surface area of heat emitter, the steeper the slope of the line, as shown in Figure 4-4.

Steeper sloped lines imply greater heat output rates at lower supply water temperature.

Figure 4-4 assumes that panel radiators are used as the heat emitters. The larger the panel radiator's surface area, the greater it's heat output at a given supply water temperature.

When a modern heat source such as a modulating/condensing boiler or hydronic heat pump is used, a critical design objective is to keep the required supply water temperature to the distribution system as low as possible. Doing so improves the thermal efficiency of the heat source, and thus minimizes the primary energy required.

Figure 4-4



The greater the total surface area of heat emitters in a system, the lower the required supply water temperature.

LOWERING WATER TEMPERATURE OF EXISTING SYSTEMS

Many hydronic heating systems in pre-1980 North American buildings were designed when energy was relatively inexpensive. Boilers burning fossil fuels were the dominant heat source, and their efficiency was of lesser concern than it is today. Designers typically specified heat emitters, such as finned-tube baseboard, that required high supply water temperatures (180+°F) at design load conditions. High water temperatures allowed the heat emitters to be relatively small, which reduced installation cost. Very few hydronic heating systems were designed around low-temperature heat emitters.

Today, these "legacy" distribution systems present a challenge when an older heat source needs to be replaced or supplemented by a modern lower temperature heat source, such as a mod/con boiler or a hydronic heat pump.

Although a mod/con boiler can produce the high water temperatures required by older distribution systems, they do so with a significant loss in thermal efficiency. High-temperature distribution systems do not create conditions that allow flue gases to condense inside the boiler. Without flue gas condensation, a supposedly high-performance mod/con boiler will yield minimal efficiency gains over a lower cost conventional boiler.

Most hydronic heat pumps cannot reliably produce water temperatures over 125ºF. Those that can reach higher temperatures often do so with a major decrease in their coefficient of performance.

In summary, nearly all contemporary hydronic heat sources perform better when matched with a distribution system that operates at a lower water temperature. What follows are concepts and methods for lowering the water temperature of existing higher temperature hydronic distribution systems.

The two fundamentals ways to reduce the supply water temperature of any hydronic heating system are:

1. Reduce the design load of the building envelope through improvements such as added insulation, better windows and reduced air leakage.

2. Add heat emitters to the existing system.

Combinations of these two approaches are also possible.

Building envelope improvements reduce the design heating load of the building. After such improvements are made, the original hydronic distribution system can meet the reduced design load while operating at lower supply water temperatures.

The change in supply water temperature depends on the change in design heating load. The new supply water temperature can be determined based on the same concepts used for outdoor reset control. It can be calculated using Formula 4-1:

Formula 4-1:

$$T_{new} = T_{in} + \left(\frac{Q_{new}}{Q_{existing}}\right) \times (T_{De} - T_{in})$$

Where:
T_{new} = supply water temperature at design load after building envelope improvements (ºF)
T_{in} = desired indoor air temperature (ºF)
Q_{new} = design heating load after building envelope improvements (Btu/hr)
$Q_{existing}$ = existing design heating load (before improvements) (Btu/hr)
T_{De} = existing supply water temperature at design load (before improvements) (ºF)

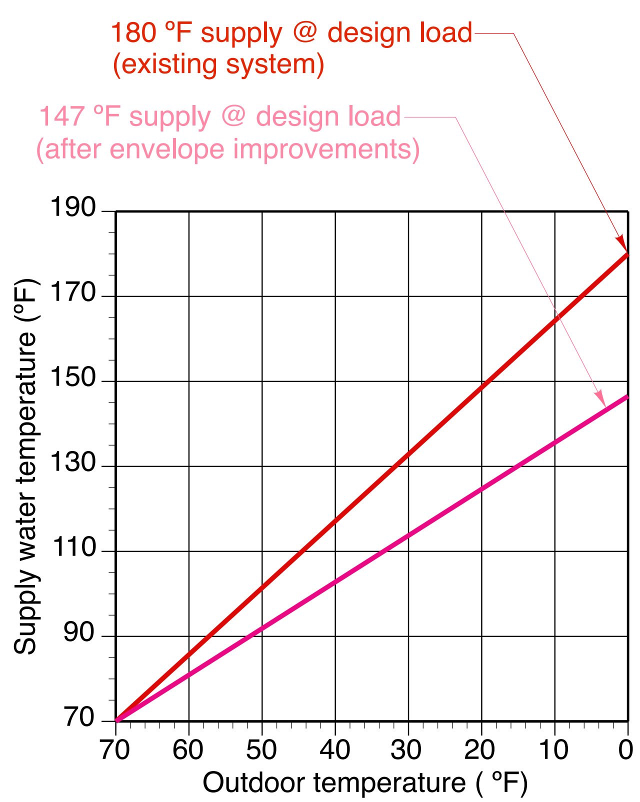
For example, assume an existing building has a design heating load of 100,000 Btu/hr, based on maintaining an interior temperature of 70ºF. The existing hydronic distribution system uses standard finned-tube baseboard and requires a supply water temperature of 180ºF at design load conditions. Also, assume that improvements to the building envelope have reduced the design load from 100,000 Btu/hr to 70,000 Btu/hr. The new supply water temperature to the existing distribution system under design load conditions can be estimated using Formula 4-1:

$$T_{new} = 70 + \left(\frac{70{,}000}{100{,}000}\right) \times (180 - 70) = 147^{\circ} F$$

A graph of the supply water temperature versus outdoor temperature for both the existing building (e.g., with design load of 100,000 Btu/hr), and after the envelope improvements which reduced the design load to 70,000 Btu/hr, is shown in Figure 4-5.

In this example, reducing the design heating load from 100,000 Btu/hr to 70,000 Btu/hr reduced the required supply water temperature from 180ºF to 147ºF.

Figure 4-5



ADDING HEAT EMITTERS TO LOWER SUPPLY WATER TEMPERATURE

If reducing the building's design heating load is not an option, or does not lower the required supply water temperature to the desired value, it will be necessary to add heat emitters to the system.

A wide variety of heat emitters could be added. For example, an existing high-temperature baseboard system could have more baseboard added to it, assuming sufficient wall space is available. Another option might be to change out some existing baseboard emitters with new baseboard emitters that have higher heat output ratings. It may also be possible to add a different type of heat emitter to the system. An example would be adding panel radiators to an existing baseboard system. Another would be adding some areas of radiant panel heating (floor, wall or ceiling) to the existing system.

The choice of which type of heat emitter to add will depend on several factors, including:

1. Availability of different makes/models of heat emitters.
2. Cost of the new heat emitters.
3. How difficult it is to integrate the new heat emitters into the building.
4. Aesthetic preferences.
5. Floor coverings (in the case of radiant floor panels).
6. Surface temperature limitations (in the case of radiant panels).
7. The specific supply water temperature that is to supply design load output in the renovated distribution system.

Each project has to be evaluated individually based on these factors.

ADDING MORE FINNED-TUBE BASEBOARD

One way to lower the required supply water temperature of an existing baseboard system is to add more baseboard.

The following procedure can be used to calculate the amount of finned-tube baseboard to be added to reduce the supply water temperature at design load to a pre-determined value. *It assumes that the baseboard being added is the same make and model as the existing baseboard. It also assumes that the existing baseboard is a standard residential-grade product with nominal 2.25" square aluminum fins with an I=B=R rated output of approximately 600 Btu/hr/ft at 200ºF water temperature.*

Step 1: Accurately determine the building's design heat load.

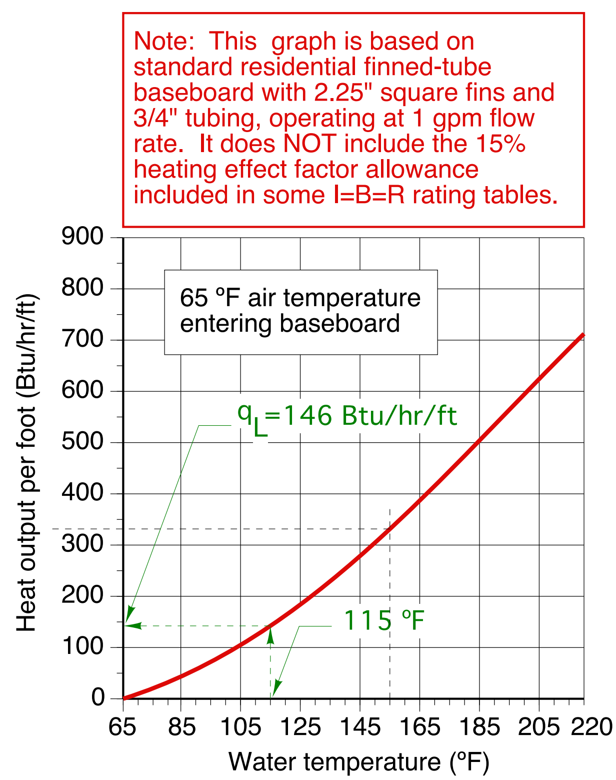
Step 2: Determine the total length of *finned tube* in the existing distribution system. Do not include the length of tubing that doesn't have fins on it. The existing finned tube length will be designated as L_e.

Step 3: Determine the desired (lower) supply water temperature for which the system is to supply design load output.

Step 4: Estimate the lower *average* circuit water temperature by subtracting 5 to 10ºF from the supply water temperature determined in Step 3. In circuits with more than 2 gallons per minute flow rate through the baseboard, assume the average water temperature will be 5ºF *below* the supply water temperature. In circuits with less than 2 gallons per minute flow rate through the baseboard, assume the average water temperature will be 10ºF below the lower supply water temperature.

Step 5: Find the new average circuit water temperature on the horizontal axis of the graph in Figure 4-6. Draw a vertical line up from this point until it intersects the red curve. Draw a horizontal line from this intersection to the vertical axis of the graph, and read the heat output of the finned tube at the lower average circuit water temperature. This number is designated as q_L. The green lines and numbers in Figure 4-6 show how q_L is determined for an average circuit water temperature of 115ºF.

Figure 4-6



Step 6: Determine the length of baseboard to be added using Formula 4-2.

Formula 4-2:

$$L_{added} = \frac{\text{design load}}{q_L} - L_e$$

Where:

L_{added} = length of finned tube of same make/model baseboard to be added (feet)
design load = design heating load of building (Btu/hr)
q_L = output of baseboard at the lower average circuit water temperature (Btu/hr/ft)
L_e = total existing length of baseboard in system (feet)

For example: Assume a building has a calculated design load of 40,000 Btu/hr, and its distribution system contains 120 feet of standard residential finned-tube baseboard. The goal is to reduce the supply water temperature to 120ºF at design conditions, using more of the same baseboard. Assume the temperature drop of the distribution system is 10ºF. Determine the amount of baseboard that must be added:

Solution:

Step 1: The design load has been calculated as 40,000 Btu/hr.

Step 2: The total amount of finned tube in the system is 120 feet.

Step 3: The lower *supply* water temperature at design load will be 120ºF.

Step 4: The lower *average* circuit water temperature will be 120 - (10/2) = 115ºF.

Step 5: The output of the finned tube at an average circuit water temperature of 115ºF is determined from Figure 4-6 as 146 Btu/hr/ft.

Step 6: The required additional length of baseboard is now calculated using Formula 4-2:

$$L_{added} = \frac{\text{design load}}{q_L} - L_e = \left[\frac{40{,}000\,\frac{Btu}{hr}}{146\,\frac{Btu}{hr \cdot ft}} - 120 \right] = 154\,ft$$

Although it might be possible to add 154 feet of baseboard to the system, it would require lots of available wall space. In most buildings, adding this much baseboard is not a practical solution, especially in kitchens or bathrooms. Alternatives include using baseboard with higher heat output or using other types of heat emitters to achieve the necessary design load output.

One option is to consider adding "high output" finned-tube baseboard rather than standard baseboard. Figure 4-7 shows the heat available from one model of high output baseboard (shown as the blue curve), and for comparison, standard residential baseboard (shown as the red curve).

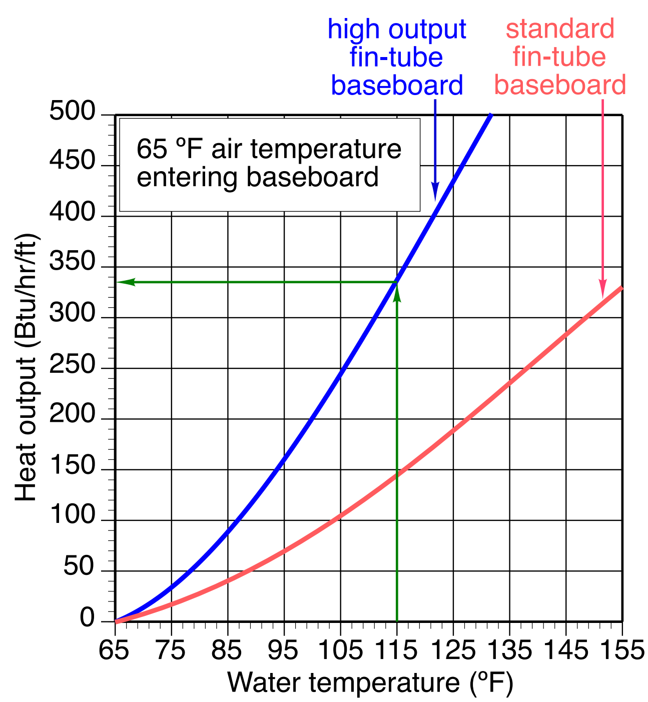
The steps of the previous procedure can be modified to determine the amount of high output finned-tube baseboard that is required to reduce the supply water temperature to the system under design load.

Steps 1-4: Same

Step 5: Determine the output of high-output baseboard at the average circuit water temperature using Figure 4-7 (or manufacturer's literature for a specific make and model).

Step 6: The required length of high output baseboard to add to the system is found using Formula 4-3.

Figure 4-7



Formula 4-3:

$$L_{ho} = \frac{\text{design load-}(q_L)(L_e)}{q_{ho}}$$

Where:

L_{ho} = length of high-output finned-tube baseboard to be added (feet)

design load = design heating load of building (Btu/hr)

q_L = output of existing baseboard at the lower average water temperature (Btu/hr/ft)

L_e = total existing length of baseboard in system (feet)

q_{ho} = output of high-output baseboard at the lower average water temperature (Btu/hr/ft)

For example: Assume a building has a calculated design load of 40,000 Btu/hr, and its distribution system contains 120 feet of standard residential finned-tube baseboard. The goal is to reduce the supply water temperature at design load to 120ºF. Additional high-output baseboard will be added to allow this lower water temperature operation. Assume that the temperature drop of the distribution system at design load is 10ºF, and this existing baseboard has the same output as in the previous example (146 Btu/hr/ft at average circuit water temperature of 115ºF). Determine the amount of high-output baseboard required based on the thermal performance of the high-output baseboard shown in Figure 4-7.

Solution:

Step 1: The design load has been calculated as 40,000 Btu/hr.

Step 2: The total amount of existing finned tube in the system is 120 feet.

Step 3: The new lower *supply* water temperature at design load will be 120ºF.

Step 4: The new lower *average* circuit water temperature will be 120 - 5 = 115ºF.

Step 5: The output of high-output finned tube at an average water temperature of 115ºF is determined from Figure 4-7 as 335 Btu/hr/ft.

Step 6: The required length of high-output baseboard to add to the system is found using Formula 4-3.

$$L_{ho} = \frac{\text{design load-}(q_L)(L_e)}{q_{ho}} = \frac{40{,}000\text{-}(146)(120)}{335} = 67\,ft$$

Although this is a reduction compared to the 154 feet of added baseboard required in the previous example, it is still a substantial length. The building must be carefully evaluated to see if this additional length of baseboard can be accommodated.

If the added length of high-output baseboard cannot be accommodated, another option is to raise the supply water temperature constraint from 120 to 130ºF under design load conditions. This would reduce the amount of added high-output baseboard in the previous example to 40 feet.

OTHER HEAT EMITTER OPTIONS

If the amount of finned tube that must be added is beyond what can be accommodated, consider other added heat emitter options. They include panel radiators, fan-coils or areas of radiant floor, radiant wall or radiant ceiling panels. In each case, the selection of these new heat emitters should be based on a selected supply water temperature at design load, along with a "credit" for the existing heat emitters in the system operating at the lower supply water temperature. The fundamental concept is given by Formula 4-4.

Formula 4-4:

$$Q_n = \text{design load - Q}$$

Where:

Q_n = required heat output of the new heat emitters at lower supply water temperature (Btu/hr)

design load = the design heating load of the building (Btu/hr)

Q_e = heat output of existing heat emitters at the lower supply water temperature (Btu/hr)

Once the value of Q_n is determined, designers can use tables or graphs from manufacturers to determine the heat output of specific heat emitters based on the average water temperature within them. Remember that the average water temperature will typically be 5 to 10ºF lower than the supply water temperature.

The goal is to select a grouping of new heat emitters with a total heat output that's approximately equal to the value of Q_n in Formula 4-4.

Assume a building has a calculated design load of 40,000 Btu/hr, and its distribution system contains 120 feet of standard residential finned-tube baseboard. The goal is to reduce the supply water temperature to 120ºF under design load conditions. Panel radiators are available in 24" x 72" size that can release 4,233 Btu/hr when operated at an average water temperature of 115ºF in rooms with 70ºF interior temperature. How many of these radiators are necessary to meet the design load?

Solution: First, use Formula 4-4 to determine the output required of the new radiators.

$$Q_n = \text{design load - } Q_e = \text{design load - } (q_L)(L_e) = 40{,}000\text{- } (146)(120) = 22{,}480\,Btu/hr$$

The number of radiators needed is then found as follows:

$$\frac{22{,}480\,Btu/hr}{4233\dfrac{Btu/hr}{radiator}} = 5.3 \text{ radiators}$$

The designer could either add 6 of these panel radiators, or choose a slightly higher supply water temperature and use 5 radiators.

Another option is to use panel radiators of different sizes, provided that their total output at the lower supply water temperature could meet the value of Q_n. In this example, the 6 new radiators would be combined with the 120 feet of existing baseboard to provide the 40,000 Btu/hr design load at a supply temperature of 120ºF.

A similar calculation could be made for fan-coils, air handlers or other heat emitters.

In the case of radiant panels, the designer needs to determine the output of each square foot of panel based on the lower average circuit temperature, and the specific construction of the panel. The total required panel area is found by dividing this number into the value of Q_n.

PIPING FOR SUPPLEMENTAL HEAT EMITTERS

There are several factors that could influence how the supplemental heat emitters are piped into the system. They include:

1. Are the existing heat emitters piped in a series circuit?
2. Are the existing heat emitters piped as individual parallel circuits?
3. Where is the piping between existing heat emitters easiest to access?
4. Are there multiple heat emitters within a given space?
5. What are the flow resistance characteristic of the supplemental heat emitters relative to those of the existing heat emitters?
6. What type of control will be used to regulate heat output to each zone of the distribution system?

There is no one best approach. Every situation must be evaluated individually while weighing these factors to determine the best fit for that project.

CONVERTING SERIES LOOPS TO PARALLEL BRANCHES

Many residential hydronic systems have finned-tube baseboards connected in series or "split-series" circuits, as shown in Figures 4-8 and 4-9.

When several heat emitters will to be added to a system using series or split-series connected baseboards, they should NOT be simply cut into the series circuit. Doing so could substantially increase the flow resistance of that circuit, which reduces flow, assuming the same circulator is used. Adding heat emitters in series also increases the temperature drop of the circuit. This reduces the heat output of heat emitters near the end of the circuit, especially when the supply temperature to that circuit is lowered.

One approach is to make strategic cuts into the series circuit where it is easiest to access, and reconnect the segments back into a parallel distribution system.

Figure 4-8

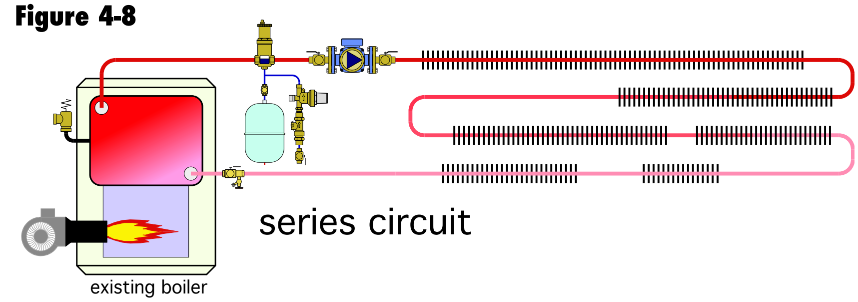
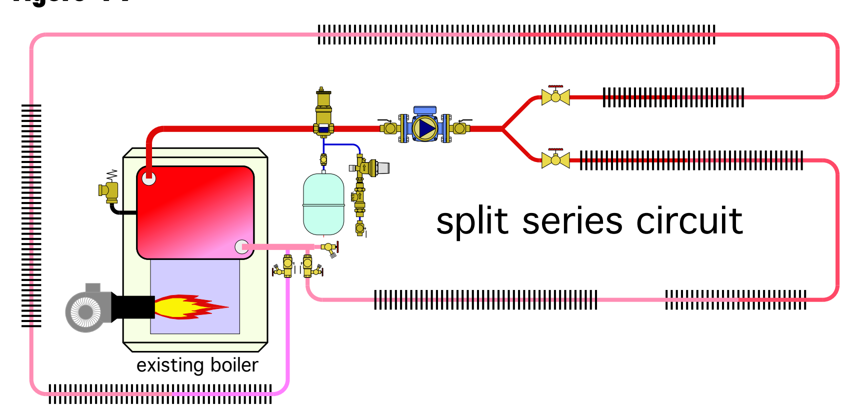


Figure 4-9



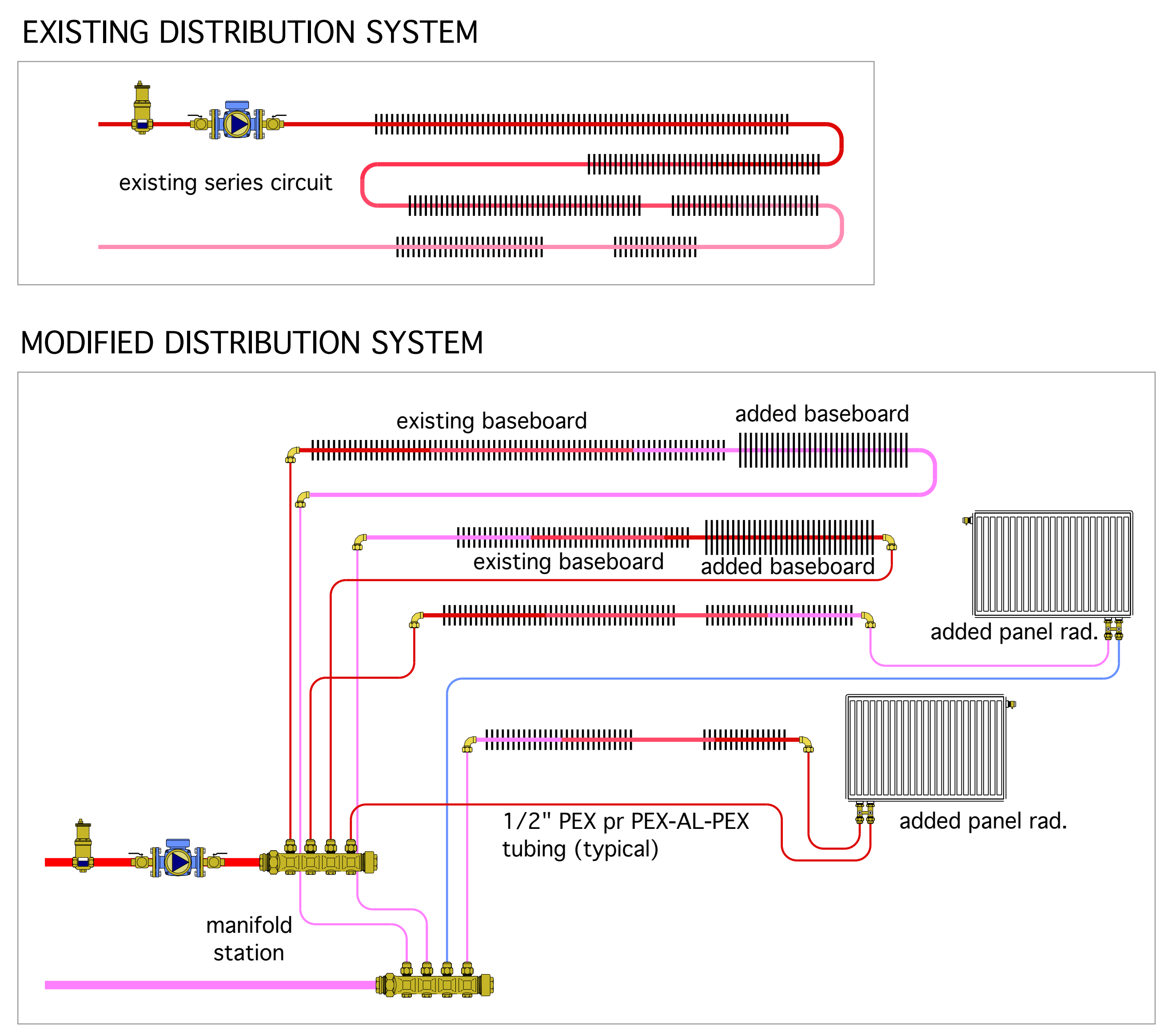
These cuts could make each room a separate parallel circuit. They might also make a group of two rooms into a new zone.

One of the easiest ways to divide an existing series loop or split-series distribution system into multiple parallel circuits is by creating a homerun distribution system. Each heat emitter, or grouping of an existing heat emitter and a supplemental heat emitter, is supplied by a separate circuit of PEX or PEX-AL-PEX tubing. This tubing is easy to route through cavities or along framing. The homerun circuits begin and end at a manifold station. The concept is shown in Figure 4-10.

In this example the existing series circuit was divided into four branch circuits. Supplemental heat emitters were added to each of these branch circuits. Two of the branch circuits received additional finned-tube baseboard, and the other two received panel radiators. These heat emitter selections illustrate that multiple types of supplemental heat emitters can be used depending on available wall space, budget and aesthetic preferences.

In some branch circuits, the supplemental heat emitters were added upstream of the existing baseboard. In others, they were added downstream of the existing baseboard. The choice depends on the available wall space and

Figure 4-10



placement of the existing baseboard within each room. Designers should estimate the water temperature in the circuit at the location where the supplemental heat emitter will be placed, and size it accordingly.

The ¾” copper tubing in the existing circuit was cut at locations that preserved a reasonable amount of existing tubing, but also allow convenient transition to ½” PEX or PEX-AL-PEX tubing. Adapter fittings for transitioning from ¾” copper to ½” PEX or PEX-AL-PEX tubing are readily available. The ½” PEX or PEX-AL-PEX supplies and returns are routed back to a manifold station. That manifold station should be equipped with individual circuit-balancing valves that allow the flow rate through each of the new branch circuits to be adjusted.

All four branch circuits in Figure 4-10 operate simultaneously (e.g., they are not configured as individual zones). As such, this distribution system presents a constant flow resistance. Due to the parallel versus series configuration, that flow resistance of the modified distribution system is likely to be lower than that of the original series circuit. This should be verified by calculating the head loss or pressure drop of the branch with the greatest flow resistance using standard hydronic pipe analysis methods. If the head loss and total flow rate through the modified distribution system is significantly lower than that of the original series circuit, consider a replacement circulator that operates with reduced power input.

A significant benefit of a parallel distribution system is that all branches receive water at approximately the same temperature. This is likely to boost the heat output of some existing baseboards that were previously located near the end of the series circuit.

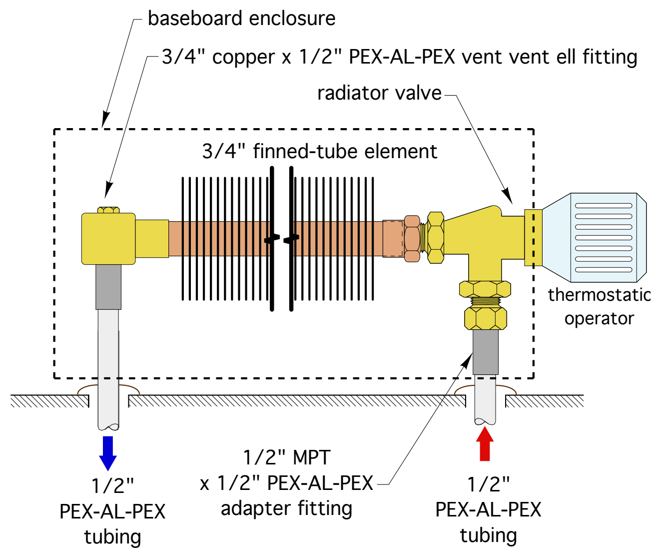
CREATING NEW ZONES

Another advantage of parallel distribution systems is the ease of creating multiple zones. If the existing series circuit is converted to multiple branches, each of those branches could be equipped with a thermostatic radiator valve. These non-electric valves automatically modulate to vary the flow rate in each branch in response to the room temperature. As room temperature begins to drop, the thermostatic valve opens to increase flow through that branch circuit to boost heat output, and vice versa.

Thermostatic radiator valves are available in several configurations. One is known as an “angle pattern” supply valve. It can be mounted on the inlet of a finned-tube baseboard element, as shown in Figure 4-11.

The thermostatic actuator on the valve projects through a hole in the end cap of the baseboard. Heated water enters the port of the valve facing the floor, makes a

Figure 4-11



90° turn as it passes through the valve, and flows into the baseboard element. The other end of the finned-tube element can be equipped with a transition adapter (straight or 90º) to convert from ¾” copper to ½” PEX or PEX-AL-PEX.

Another type of thermostatic radiator valve allows the valve body and actuator to be mounted within the baseboard enclosure, while the adjustment knob is mounted at normal thermostat height on the wall. The adjustment knob connects to the valve actuator using a capillary tube, as seen in Figure 4-12.

Figure 4-12

Figure 4-13 MODIFIED DISTRIBUTION SYSTEM

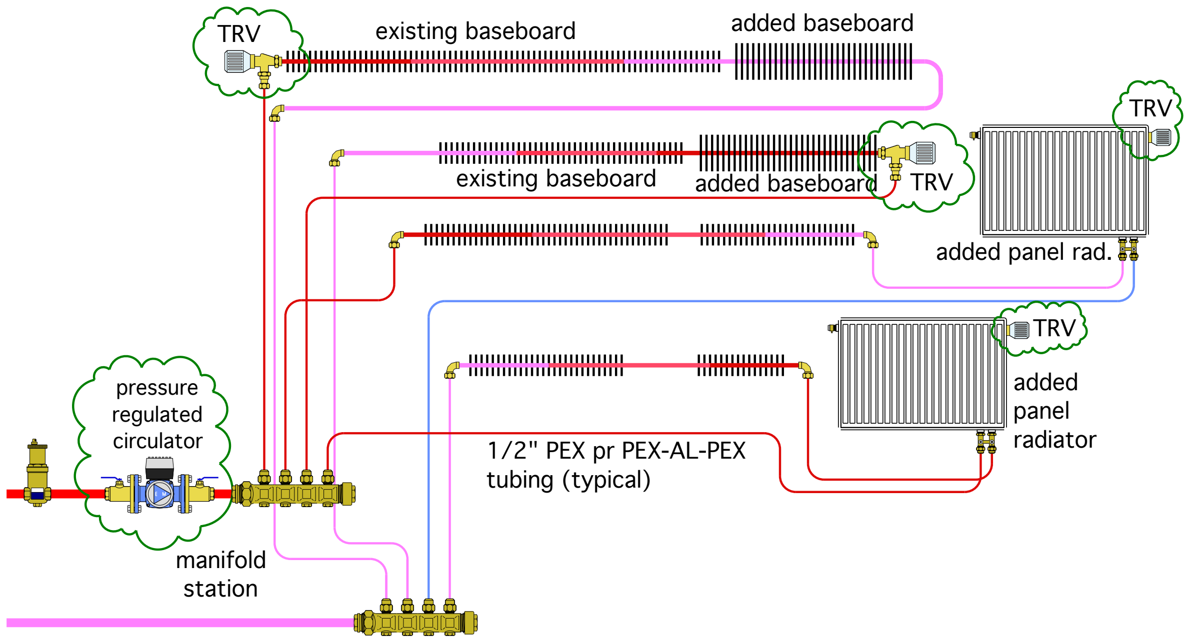
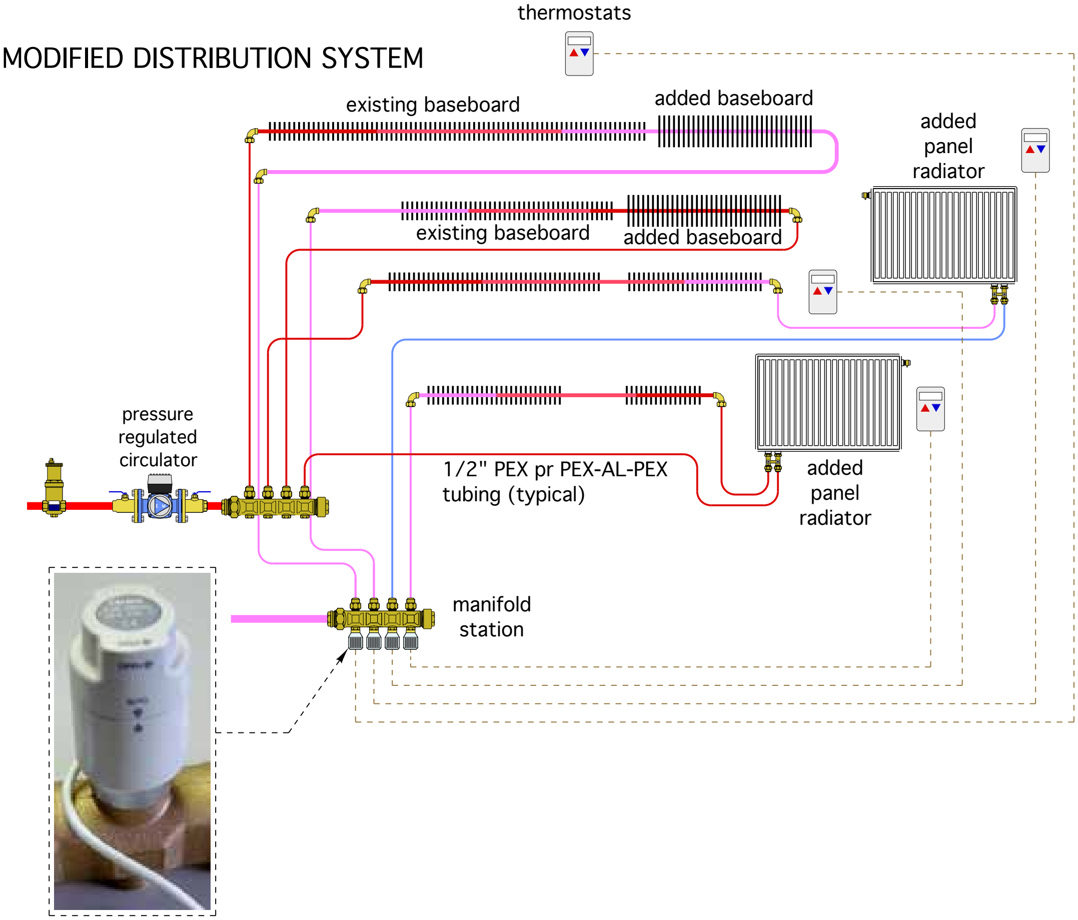


Figure 4-14 MODIFIED DISTRIBUTION SYSTEM



It’s also possible to use panel radiators with built-in thermostatic radiator valves.

Figure 4-13 shows how the distribution system can be modified using thermostatic radiator valves to create a distribution system with four independently controlled zones. This adds flexibility for adjusting interior comfort conditions far beyond that of the original series loop system.

Another possibility is to install a low-voltage (24 VAC) manifold valve actuator on each circuit valve at the manifold station. These actuators are wired to four new thermostats, one for each zone. This option is shown in Figure 4-14.

The systems shown in Figures 4-13 and 4-14 use valves for zoning. In Figure 4-13, non-electric thermostatic radiator valves regulate the flow through each parallel branch circuit. In Figure 4-14, low-voltage manifold actuators are attached to valves within the return manifold. In both cases, the original circulator has been replaced with a variable-speed pressure-regulated circulator. These circulators automatically adjust their speed and power input as the valves open, close or modulate flow. This helps stabilize the differential pressure across the manifold station, and maintain consistent flow rates within each zone circuit, regardless of what other zones are operating. These modern circulators also have significantly lower electrical energy consumption relative to standard wet rotor circulators with permanent split capacitor (PSC) motors.

DESIGN GUIDELINES

The modifications shown to convert a series baseboard circuit into parallel branch circuits are just a few of many possibilities. Each conversion situation must consider the exact layout of the existing heat emitters, and the practicality of modifying the system into parallel branches. Designers should follow these guidelines.

1. Always determine what type of supplemental heat emitter will be used in each room, and where it will be located *before* modifying the piping.

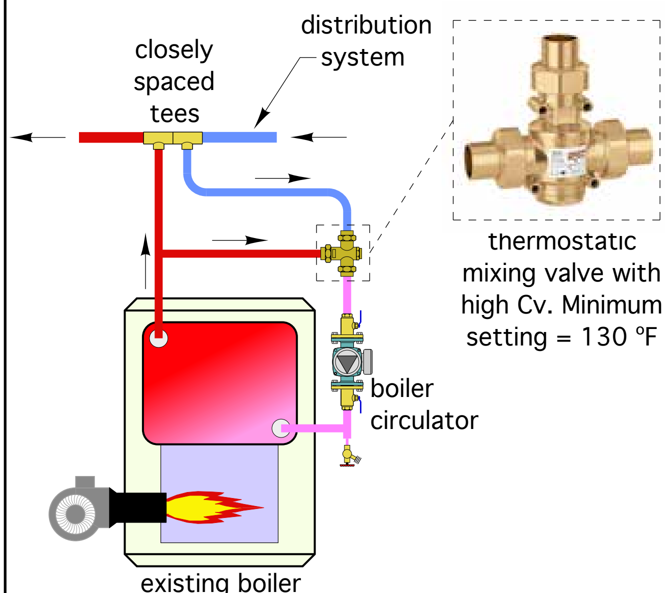
2. From the standpoint of cost and installation time, it’s best to use as much of the existing (and accessible) piping as possible.

3. Always consider the benefit versus cost of creating new zones when modifying the existing system. For example, if two bedrooms are typically maintained at the same temperature, and the existing system has accomplished this, it’s likely best to keep these two bedrooms together on the same zone after adding the necessary supplemental heat emitters. However, if the piping modifications to do this are comparable in cost/time to creating two independent zones, then the latter is arguably a better choice.

4. Once all the supplemental heat emitters have been selected, and the proposed modifications to the distribution system have been sketched, always run a flow and head loss analysis for the modified system. This is used to confirm sufficient flow to each branch, and to determine a suitable circulator for the modified system.

5. If an existing conventional boiler will be used as the heat source for the modified system, which now operates at significantly lower water temperatures, be sure the boiler is protected against sustained flue gas condensation by installing a thermostatic mixing valve near the boiler, as shown in Figure 4-15.

Figure 4-15



5. HEAT EXCHANGER FUNDAMENTALS

Many hydronic heating and cooling systems need to move heat from one fluid to another without mixing those fluids. One example is heat moving from hot “system water” that passes through a cast iron boiler into cooler domestic water that is being heated for showers, washing, etc. Another example is heat moving from an antifreeze solution that flows through a solar collector to water within a thermal storage tank. The fluids exchanging heat may be at significantly different temperatures and pressures, and must be fully isolated from each other. Such processes require a heat exchanger.

There are many types of heat exchangers. Nearly all of them separate the fluids exchanging heat using metal surfaces. Common materials used to separate the fluids include copper, cupronickel, stainless steel, carbon steel and titanium/stainless steel alloys. These metals all have relatively high thermal conductivity, and thus present minimal conduction resistance.

EXTERNAL vs. IMMERSION HEAT EXCHANGERS

The types of heat exchangers commonly used in hydronic systems can be categorized as external or immersion.

External heat exchangers are standalone devices with a minimum of four piping connections. They have an inlet and outlet connection for each of the two fluids that pass through the heat exchanger.

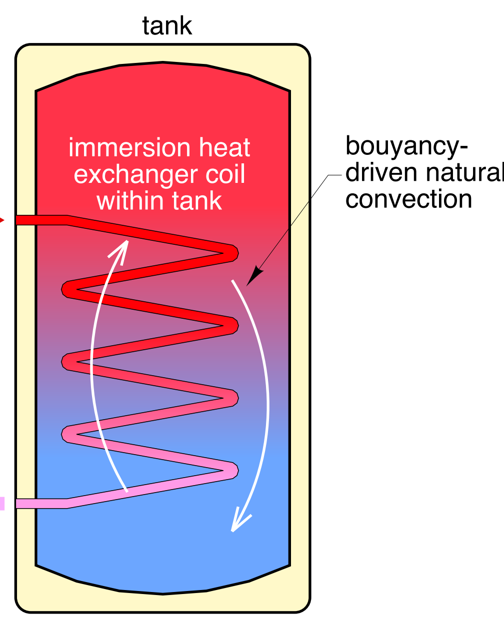
An immersion heat exchanger (also known as an “internal” heat exchanger) usually takes the form of a helical coil made of copper, stainless steel or carbon steel. It’s permanently mounted inside a tank, with only the inlet and outlet connections accessible outside that tank. The outer surface of an immersion heat exchanger is surrounded by the fluid within the tank.

The flow through each side of an external heat exchanger is typically driven by a circulator. This implies that forced convection heat transfer occurs between each fluid and the surfaces separating the fluids. Recall that forced convection results in higher rates of heat transfer in comparison to natural convection.

The flow through the *internal side* of an immersion heat exchanger is typically driven by a circulator, and thus heat transfer between this inner surface and the fluid is by forced convection. However, flow over the *external side* of an immersion heat exchanger is usually created by buoyancy differences in the fluid surrounding the heat exchanger. The heat exchanged between the outer surface and the surrounding fluid is by natural convection. The rate of heat transfer from an immersion heat exchanger is almost always limited by the natural convection at its outer surface.

Figure 5-1 illustrates a typical immersion heat exchanger coil within a tank.

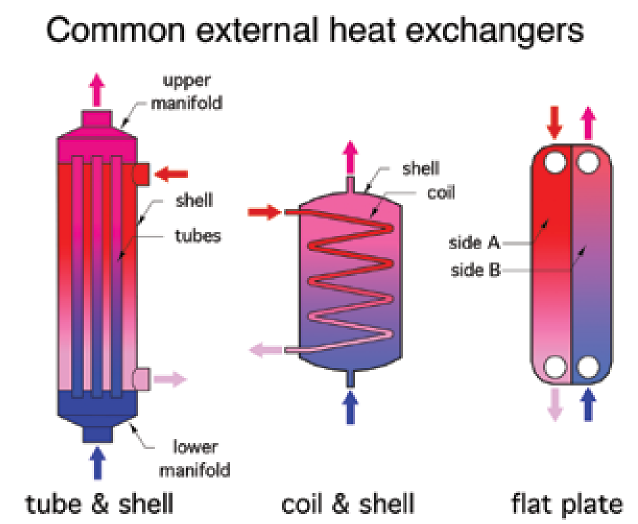
Figure 5-1



This heat exchanger geometry is commonly used for indirect domestic water heaters that heat cold domestic water in the tank by passing hot water from a boiler or other heat source through the immersion coil.

Figure 5-2 illustrates the types of the external heat exchangers most commonly used in hydronic heating and cooling systems.

Figure 5-2



All external heat exchanger designs create two pressure-tight cavities that separate the fluids exchanging heat. They also create an internal surface across which heat moves by conduction through the metal separating the two fluids.

FLAT PLATE HEAT EXCHANGERS

Figure 5-3

Courtesy of GEA PHE Systems

Flat plate heat exchangers are the most commonly used heat exchangers in modern hydronic systems. In smaller sizes, these heat exchangers are made by stacking specially formed stainless steel plates, as shown in Figure 5-3.

The narrow spaces between the pre-formed plates form channels through which each fluid passes. The surface patterns on the plates create turbulence that enhances convection. These patterns also create a flow distribution manifold at each of the four piping connections. If the channels between the plates were numbered in the same order in which the plates are stacked, one fluid flows through channels 1, 3, 5, 7, etc., while the other fluid flows through channels 2, 4, 6, 8, etc.

The plates provide a very large surface area between the two fluids, while also creating a compact device with low fluid volume. The combination of high surface area and low volume allows flat plate heat exchangers to respond very quickly to changes in fluid temperatures or flow rates. It also allows flat plate heat exchangers to be significantly smaller than other types of external heat exchangers of comparable thermal performance.

In small to medium sizes, the stainless steel plates are brazed together at their perimeters to form a "brazed plate" heat exchanger. Once brazed, the plates are permanently sealed in place, and cannot be disassembled for cleaning. Brazed plate heat exchangers are usually made in one of three common plate sizes (3" x 8", 5" x 12", or 10" x 20"). Other plate sizes such as 5" x 20" are sometimes available. The thermal and hydraulic performance of a brazed plate heat exchanger is determined by the number of plates used. For example, a 5" x 12" x 40 plate heat exchanger has twice as many plates as a 5" x 12" x 20 plate unit. The 40-plate heat exchanger will be capable of approximately twice the rate of heat transfer when operated at the same inlet fluid temperatures and temperature drops. The larger heat exchanger will also have a lower head loss than the smaller exchanger when operating at the same flow rate.

Small brazed plate heat exchangers, such as 3" x 8" x 20 plates, are often used in residential domestic water-heating applications. Larger brazed plate heat exchangers, such as 10" x 20" x 80 plates, are capable of transferring heat at several hundred thousand Btu/hr, and are used in applications such as snowmelting or district heating for small commercial buildings.

The specifications for brazed plate heat exchangers often list a certain rate of heat transfer. It's very important to realize that this rating is based on *specific entering fluid temperatures and flow rates*. For example, a certain heat exchanger may be listed as capable of transferring 150,000 Btu/hr based on one fluid entering at 180ºF and a flow rate of 15 gpm, and the other fluid entering at 100ºF and at 15 gpm. However, any difference between these rating conditions and the actual operating conditions of the heat exchanger in a given application could have a major impact on heat transfer rate. In general, the greater the difference between the entering fluid temperatures, and the higher the flow rates, the higher the rate of heat exchange. This will be discussed later in this section.

Figure 5-4

Courtesy of B&G

Large flat plate heat exchangers are also fabricated as a stack of stainless steel plates. However, the plates are not brazed together. Instead they are stacked together with elastomeric gaskets in between. The plate stack is then mechanically compressed using large threaded steel shafts and nuts. This compression causes the gaskets to form the perimeter seal as well as internal seals between adjacent plates. The stack of plates and gaskets are capped at each end by thick steel plates capable of withstanding high pressures if necessary. The internal plates and end plates are held in alignment by a structural steel frame. This type of heat exchanger is called a "plate & frame" heat exchanger. An example is shown in Figure 5-4.

One benefit of plate & frame heat exchangers is that the frame is often built so that additional plates can be added to the stack if ever needed. An example would be when the rate of heat exchange has to be maintained using a new heat source that supplies lower temperature hot water to the heat exchanger. Another would be when the rate of heat exchange needs to be increased to accommodate expansion of an existing heating system. It's also possible, to disassemble a plate & frame heat exchanger if required for cleaning or maintenance.

Figure 5-5 shows the rear side of a large plate & frame heat exchanger used in a district heating application. Notice that the threaded steel rods that hold the plate stack together extend well beyond the rear end plate. If more plates are to be added, the nuts on the treaded shafts are removed, allowing the end plate to slide backwards on the steel frame. The additional plates and their associated gaskets are placed into the stack. The rear end plate is then moved back into position, and the nuts retightened to a specified torque, resealing the larger plate stack.

Figure 5-5

FLOW DIRECTION THROUGH HEAT EXCHANGERS

In heat exchangers that operate with liquids, there are two possible flow configurations:

• Both liquids pass in the same direction through the heat exchanger.
• The liquids flow in opposite directions through the heat exchanger.

When the two liquids move in the *same direction,* the configuration is called *"parallel flow."* When the two fluids move in *opposite directions,* the configuration is called *"counterflow."*

Figure 5-6

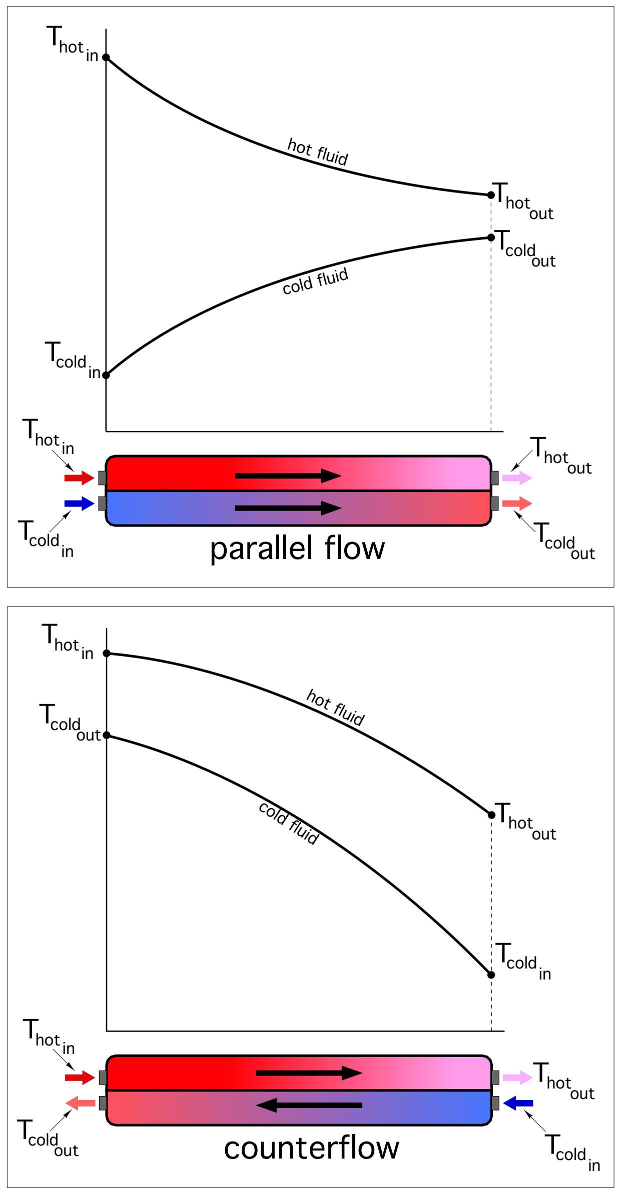


Figure 5-6 show how the temperature of the two liquid streams would change as they flow through a heat exchanger piped for parallel flow versus one piped for counterflow.

In a parallel flow configuration, the temperature of the cooler liquid absorbing heat can never meet or exceed the temperature of the exiting hot liquid. All other factors being equal, the rate of heat exchange is limited by the temperature of the exiting hot fluid stream.

When a counterflow configuration is used, the leaving temperature of the stream absorbing heat can be higher than the leaving temperature of the stream providing heat.

The rate of heat exchange will always be higher when a heat exchanger is configured for counterflow operation.

Counterflow should also be used with immersion heat exchangers. Figure 5-7 shows how the flow through an immersion coil heat exchanger that's adding heat to the tank (on left) is piped so that flow is in the opposite direction of natural convection currents inside the tank (e.g., flow is from top to bottom of coil).

Another immersion heat exchanger that removes heat from the right side of the tank is also piped for counterflow (e.g., flow is from bottom to top of coil).

RATE OF HEAT EXCHANGE

The rate of heat transfer between two fluids passing through a heat exchanger is dependent on several factors, including:

- The internal surface area that separates the two fluids.
- The thermal conductivity and thickness of the plate material.
- The convection coefficients developed between each fluid and the internal surfaces.
- The flow rate on each side of the heat exchanger.
- The specific heat, density and viscosity of each fluid.
- The insulation (if any) surrounding the heat exchanger.
- The direction of flow of one fluid relative to another.
- The "log mean temperature difference" between the two fluids.

Recall from previous discussions that the rate of conduction heat transfer is directly proportional to the temperature difference between the outer surfaces of a material through which heat is passing. The higher this temperature difference, the greater the rate of heat transfer by conduction.

Similarly, the rate of convective heat transfer is directly proportional to the temperature difference between the temperature in the bulk of the fluid stream (e.g., away from the boundary layer), and the temperature of the solid material the fluid is passing over. It's also dependent on the convection coefficient between the fluid and surface.

These relationships suggest that the rate of heat transfer between two fluids passing through any heat exchanger would depend on the temperature difference between the two fluids. However, the temperature difference between the two streams is not the same at all locations on the heat exchanger. Each fluid is either losing or gaining heat as it passes through the heat exchanger, and thus the temperature difference between the fluids varies depending on where it is measured within the heat exchanger.

Heat transfer theory defines a specific temperature difference based on the fluid temperatures at all four connections of the heat exchanger. That difference is called the *log mean temperature difference*, and is defined by Formula 5-1.

Figure 5-7

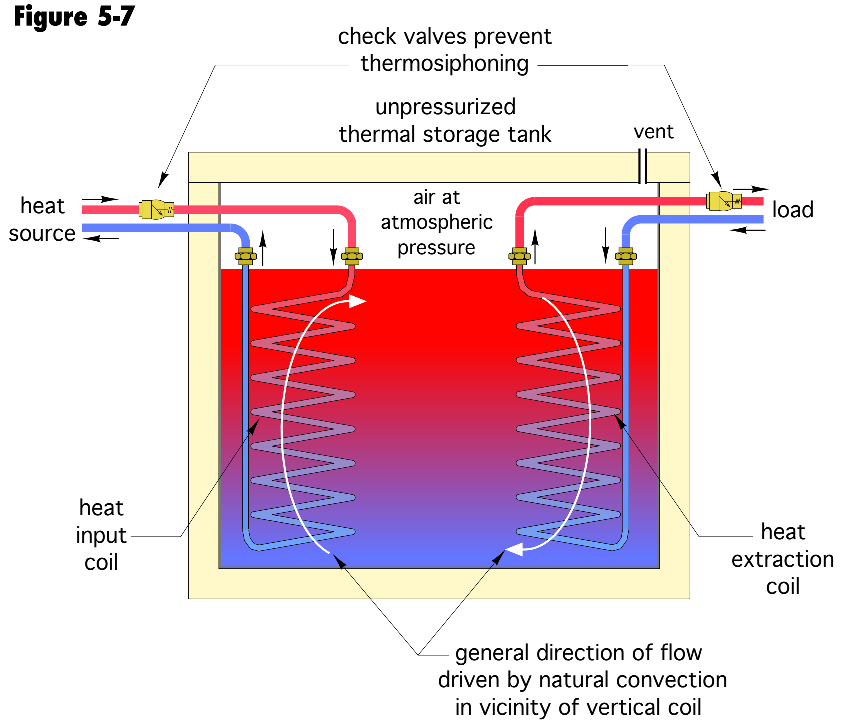


Figure 5-8

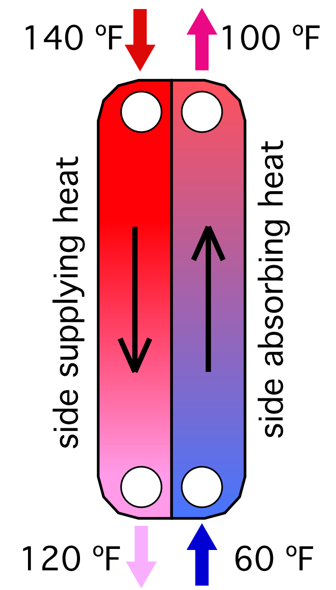
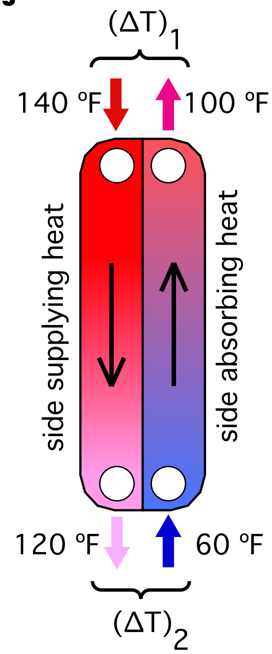


Figure 5-9



$$LMTD = \frac{(\Delta T)_1 - (\Delta T)_2}{\ln\left[\frac{(\Delta T)_1}{(\Delta T)_2}\right]}$$

$$LMTD = \frac{(\Delta T)_1 - (\Delta T)_2}{\ln\left[\frac{(\Delta T)_1}{(\Delta T)_2}\right]} = \frac{40-60}{\ln\left[\frac{40}{60}\right]} = \frac{-20}{-0.405} = 49.3^o F$$

Formula 5-1:

$$LMTD = \frac{(T_{ho} - T_{ci}) - (T_{hi} - T_{co})}{\ln\left[\frac{(T_{ho} - T_{ci})}{(T_{hi} - T_{co})}\right]}$$

Where:
LMTD = log mean temperature difference (ºF)
T_{ho} = temperature of the fluid supplying heat at the *outlet* of the heat exchanger (ºF)
T_{hi} = temperature of the fluid supplying heat at the *inlet* of the heat exchanger (ºF)
T_{co} = temperature of the fluid absorbing heat at the *outlet* of the heat exchanger (ºF)
T_{ci} = temperature of the fluid absorbing heat at the *inlet* of the heat exchanger (ºF)
ln [ ] = the natural logarithm of the quantity within brackets.

Example: Calculate the log mean temperature difference for the heat exchanger shown in Figure 5-8.

$$LMTD = \frac{(T_{ho} - T_{ci}) - (T_{hi} - T_{co})}{\ln\left[\frac{(T_{ho} - T_{ci})}{(T_{hi} - T_{co})}\right]} = \frac{(140-100)-(120-60)}{\ln\left[\frac{(140-100)}{(120-60)}\right]} = \frac{40-60}{\ln\left[\frac{40}{60}\right]} = \frac{-20}{-0.405} = 49.3^o F$$

Formula 5-1 can be used for *either* parallel flow or counterflow heat exchanger configurations.

The LMTD can also be thought of as the temperature difference between the fluids at one end of the heat exchanger, minus the temperature difference between the fluid at the other end, divided by the natural logarithm of the ratio of these two temperature differences. This is illustrated in Figure 5-9.

If $(\Delta T)_1$ and $(\Delta T)_2$ are the same, the LMTD, as it would be calculated using Formula 5-1 or in Figure 5-9, is undefined mathematically. In these cases, the LMTD = $(\Delta T)_1 = (\Delta T)_2$.

The LMTD of a heat exchanger depends on the relative flow directions of the two fluids passing through the heat exchanger. Figure 5-10 shows two identical heat exchangers, one piped for parallel flow and the other piped for counterflow. The LMTD for each situation has been calculated using Formula 5-1.

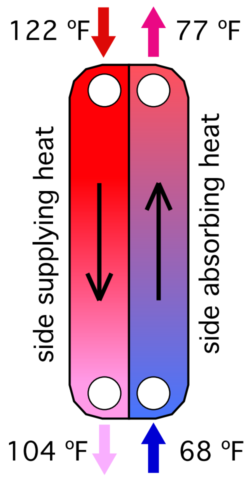
It can be shown that the LMTD will always be greater when the two fluids exchanging heat pass in opposite directions through the heat exchanger. Thus, to attain the highest rate of heat exchange, heat exchangers should always be piped for counterflow.

Once the LMTD is calculated, the rate of heat exchange across the heat exchanger can (theoretically) be calculated using Formula 5-2:

Formula 5-2:

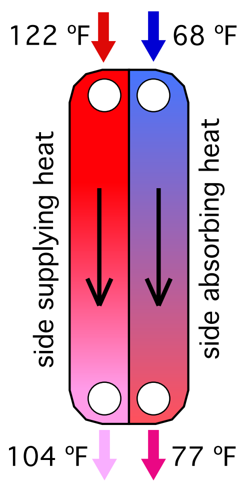
$$Q = UA(LMTD)$$

Figure 5-10



counterflow

$$LMTD=\frac{(T_{ho}-T_{ci})-(T_{hi}-T_{co})}{\ln\left[\frac{(T_{ho}-T_{ci})}{(T_{hi}-T_{co})}\right]}=\frac{(122-77)-(104-68)}{\ln\left[\frac{(122-77)}{(104-68)}\right]}=\frac{45-36}{\ln\left[\frac{45}{36}\right]}=\frac{9}{0.223}=40.3^{\circ}F$$



parallel flow

$$LMTD=\frac{(T_{ho}-T_{ci})-(T_{hi}-T_{co})}{\ln\left[\frac{(T_{ho}-T_{ci})}{(T_{hi}-T_{co})}\right]}=\frac{(122-68)-(104-77)}{\ln\left[\frac{(122-68)}{(104-77)}\right]}=\frac{54-27}{\ln\left[\frac{54}{27}\right]}=\frac{27}{0.693}=39.0^{\circ}F$$

Where:
Q = rate of heat transfer through heat exchanger (Btu/hr)
U = overall heat transfer coefficient of the heat exchanger (Btu/hr/ft^2/°F)
A = area of surface(s) separating fluids inside the heat exchanger (ft^2)
LMTD = log mean temperature difference across the heat exchanger (°F)

The value of "U" depends on the convection coefficients for each fluid, as well as the thickness, shape and thermal conductivity of the surfaces separating the fluids. It is defined by Formula 5-3.

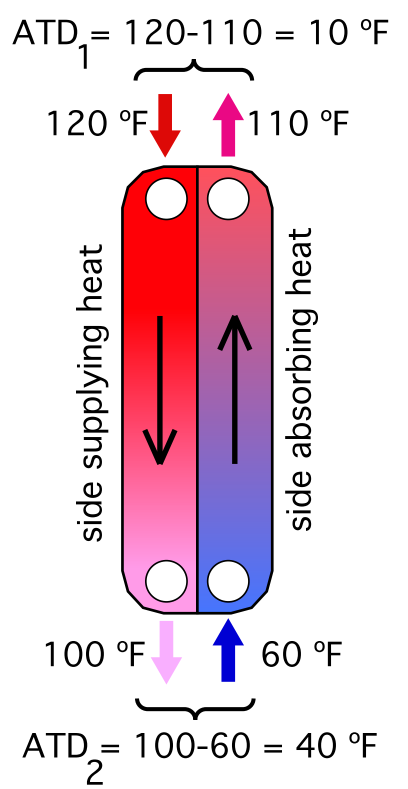
Formula 5-3:

$$U=\frac{1}{\left[\frac{1}{h_c}+\frac{d}{k}+\frac{1}{h_h}\right]}$$

Where:
U = overall heat transfer coefficient of the heat exchanger (Btu/hr/ft^2/°F)
h_c = convection coefficient for the fluid absorbing heat (Btu/hr•ft^2•°F)
h_h = convection coefficient for the fluid releasing heat (Btu/hr•ft^2•°F)
d = thickness of wall separating the fluids (ft)
k = thermal conductivity of wall separating the fluids (Btu/hr•ft•°F)

The values of the convection coefficients h_h and h_c can vary widely depending on the fluid, its density, specific heat and viscosity, as well as the fluid's flow rate. These conditions need to be evaluated on both sides of the heat exchanger. The theoretical determination of the convection coefficients is mathematically complex. It is usually based on empirical correlations of operating

Figure 5-11



conditions that are expressed in the form of *dimensionless numbers,* such as the Reynolds number, Nusselt number and Prandtl number. Many heat transfer textbooks summarize methods for estimating convection coefficients for heat exchangers of specific types and geometries.

Typical values of U in water-to-water heat exchangers are in the range of 100 to 300 (Btu/hr/ft^2/°F).

APPROACH TEMPERATURE DIFFERENCE

Another term that's commonly used in describing the operating conditions of a heat exchanger is called the "approach temperature difference" (ATD). Since all heat exchangers have two inlets and two outlets, there are technically *two* approach temperature differences, as shown in Figure 5-11.

Customarily, the approach temperature difference refers to the incoming temperature of the fluid supplying the heat minus the leaving temperature of the fluid absorbing the heat. For the heat exchanger shown in Figure 5-11, this would be 10°F.

Approach temperature difference can be thought of as a "thermal penalty" that's present simply because heat has to move through a heat exchanger. A temperature difference must be present between the two sides of any heat exchanger to create heat transfer. A hypothetical heat exchanger with infinite internal surface area and no external heat loss would have an approach temperature difference of 0°F, and thus induce no thermal penalty. While this is not possible for any "real" heat exchanger, approach temperature differences on the order of 2 to 3°F are achievable in some situations.

Figure 5-12a

Figure 5-12b

Note: To download a PDF file version of this report, navigate to the Print/Save page.

Model: FG5X12-120(1-1/4"MPT)			
Selection ID	NUD6E5E3Q	Model size	5x13
Application	Snow melt	Nominal surface (ft²)	45.2
Load (Btu/h)	260,000	Dimensions	5.1W x 13.3H x 11.0D
Log mean temp. diff. (°F)	12.3	Plate construction	Single wall
Overall HTC (Btu/h·ft²·°F)	469	Net weight (lb)	39.9
Oversurface percent	0.6		

Design Conditions	Side A - Liquid	Side B - Liquid
Fluid type	**Water**	**Propylene glycol**
Fluid conc.		50
Fluid mass flow rate (lb/min)	289.2	249.0
Entering fluid temp. (°F)	140.0	110.0
Leaving fluid temp. (°F)	125.0	130.0
Fluid flow rate (GPM)	35.2	29.1
Fluid fouling factor (h·ft²·°F/Btu)	0.00010	0.00010
Model Parameters		
Number of channels	59	60
Velocity (ft/s)	0.57	0.46
Pressure drop (psi)	1.7	1.4
Heat transfer coef. (Btu/h·ft²·°F)	1,942	706
Internal volume (ft³)	0.177	0.180

Ratings at Varying Conditions

Percent difference	**-15%**	**-7½%**	**0%**	**7½%**	**15%**
Pressure drop (psi) (Side A)	1.3	1.5	1.7	2.0	2.2
Pressure drop (psi) (Side B)	1.0	1.2	1.4	1.5	1.8
Load (Btu/h)	**221,000**	**240,500**	**260,000**	**279,500**	**299,000**
Fluid flow rate (GPM) (Side A)	29.9	32.5	35.2	37.8	40.4
Fluid mass flow rate (lb/min) (Side A)	245.8	267.5	289.2	310.9	332.6
Fluid flow rate (GPM) (Side B)	24.7	26.9	29.1	31.3	33.5
Fluid mass flow rate (lb/min) (Side B)	211.6	230.3	249.0	267.6	286.3
Entering fluid temp. (°F) (Side A)	140.0	140.0	140.0	140.0	140.0
Entering fluid temp. (°F) (Side B)	110.0	110.0	110.0	110.0	110.0
Leaving fluid temp. (°F) (Side A)	125.0	125.0	125.0	125.0	125.0
Leaving fluid temp. (°F) (Side B)	130.0	130.0	130.0	130.0	130.0
Oversurface percent	7.3	3.8	0.6	-2.3	-4.9

Disclaimer

This software and the generated calculations provided herein are estimates only and should be treated as such. GEA PHE Systems North America, Inc. always strives to give complete and accurate information, but cannot provide any guarantees. This software and its output are provided "as is" and any express or implied warranties, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose are disclaimed. In no event shall GEA PHE Systems North America, Inc. be liable for any direct, indirect, incidental, special, exemplary, or consequential damages (including, but not limited to, procurement of substitute goods or services; loss of use, data, or profits; or business interruption) however caused and on any theory of liability, whether in contract, strict liability, or tort (including negligence or otherwise) arising in any way out of the use of this software, even if advised of the possibility of such damage.

As the heat exchanger size increases, and the incoming fluid temperatures and flow rates remain constant, the approach temperature difference decreases. However, as the approach temperature difference decreases (desirable), the size and cost of the heat exchanger increases (undesirable). Heat exchangers are sometimes sized based on achieving a *maximum* approach temperature difference for a specific application.

SIZING HEAT EXCHANGERS

Although it is possible to theoretically estimate the thermal performance of heat exchangers, the calculations involved are complex and time consuming. Today, many heat exchanger manufacturers offer sizing and selection software, either for downloading or to be used online. An example of the user interface from one such software offering is shown in Figure 5-12.

Sizing and selection software allows designers to enter values for flow rates, temperatures, type of fluids being used and required rates of heat transfer. The software uses these inputs to select one or more specific heat exchanger configurations that can meet or slightly exceed the required thermal performance. The software also provides pressure drop or head loss information, which is used for the hydraulic design of the circuit containing the heat exchanger.

FOULING FACTORS FOR HEAT EXCHANGERS

The listed thermal and hydraulic performance of heat exchangers is usually based on the assumption that the heat exchanger is free of any internal scaling or debris. A relatively thin accumulation of dirt or scaling on the surfaces that separate the two fluids can significantly lower the heat exchanger's thermal performance. For optimal performance and long life, it's important to keep the surfaces of heat exchangers clean.

The effect of scale accumulation is sometimes expressed as a fouling factor, which is defined by Formula 5-4:

Formula 5-4:

$$R_f = \frac{1}{U_{dirty}} - \frac{1}{U_{clean}}$$

Where:
R_f = fouling factor (°F•hr•ft^2/Btu)
U_{dirty} = overall heat transfer coefficient for the surfaces that are fouled (Btu/hr/ft2/°F)
U_{clean} = overall heat transfer coefficient for the surface that is clean (Btu/hr/ft2/°F)

The fouling factor can be thought of as an additional "R-value" added to the surfaces of the material separating the two fluids. The greater the thermal resistance of any accumulated dirt or scale, the lower the overall heat transfer rate through the heat exchanger, with all other factors being equal. The value of R_f is determined experimentally by measuring the rate of heat transfer through a heat exchanger that has been fouled based on specific operating conditions and service time, versus the rate of heat exchange through an identical new heat exchanger operating with the same fluids, inlet flow rates and temperatures.

Fouling factors are generally small numbers in the range of 0.0005 to 0.005 (hr•ft^2•°F/Btu). Values for fouling factors can be obtained in the following reference: *[Kakac,S. and H. Liu, Heat Exchangers - Selection, Rating and Thermal Design, CRC Press, (1998)]*. Some sizing and selection software allows different fouling factors to be used on each side of the heat exchanger's internal surface.

KEEPING HEAT EXCHANGERS CLEAN

Fouling factors can be reduced by keeping the internal surfaces of a heat exchanger free of scale and debris.

Scale formation is generally caused by calcium or magnesium salts dissolved in water. So-called "hard" water can have a high concentration of these mineral salts. The ability of these mineral salts to come out of solution and attach to internal heat exchanger surfaces increases with increasing water temperature.

Fouling can also be caused by residual soldering flux or cutting oil left in the system. The chemical breakdown of glycol-based antifreeze solutions can also cause films to form on heat exchanger surfaces. Corrosion due to oxidation of ferrous metals also creates sludge that can lodge within heat exchangers.

One of the best ways to minimize the potential scale formation due to mineral salts is to demineralize the water used in hydronic heating or cooling systems. Demineralization can be done by passing the untreated water through a bed of resin beads that absorb the ions formed by the mineral salts. For hydronic system applications, demineralization can reduce the total dissolved solids content of the water into an ideal range of 10-30 parts per million (PPM).

idronics #18 (Water Quality in Hydronic Systems) provides a full discussion of demineralization techniques and hardware.

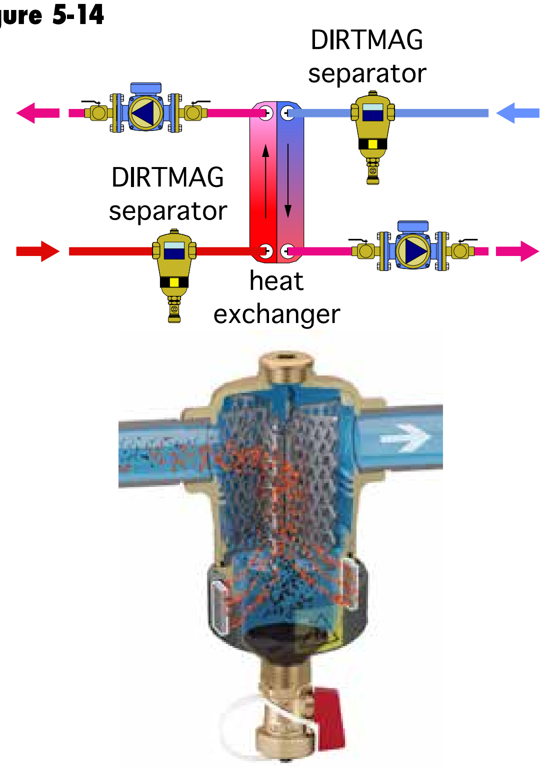
Dirt or other debris that lodges within heat exchangers can also reduce heat transfer. In extreme cases, it can render common heat exchangers, which cannot be disassembled for cleaning, useless, leaving no alternative but to replace the heat exchanger. Figure 5-13 shows an example of a heavily fouled plate that was removed from a plate & frame heat exchanger.

Figure 5-13

Dirt and iron oxides such as magnetite can be minimized in hydronic systems through use of low velocity zone dirt separators or magnetic separators. The latter are capable of separating both non-magnetic and magnetic particles from a flow stream.

Best practice is to provide a dirt separator on each flow stream entering a heat exchanger, as shown in Figure 5-14.

Figure 5-14



Caleffi DirtMag separators are recommended if either circuit passing through the heat exchanger contains components made of carbon steel or cast iron, both of which can be a source of magnetite. DirtMag separators are also recommended if a circulator with a permanent magnet motor (e.g., ECM) is used in either circuit. Figure 5-15 shows a Caleffi Dirtmag separator with the magnet removed, and being "blown down" to eliminate iron oxide (magnetite) from the system.

Figure 5-15

idronics #15 (Separation in Hydronic Systems) provides a full discussion of dirt separation techniques and hardware.

Another useful detail for heat exchangers that handle domestic water (which may contain dissolved mineral salts) is to install a combination isolation/flushing valve on the piping that carries domestic water into and out of the heat exchanger. Figure 5-16 shows a typical detail.

The combination isolation/flush valves allow the domestic water side of the heat exchanger to be isolated from the remainder of the system. The hose bibs on the side ports of these valves can be connected to a flushing assembly that circulates a mild acid solution through the heat exchanger to dissolve scaling caused by precipitation of mineral salts. Once the scale has been dissolved and removed, fresh water can be passed through the heat exchanger to remove any remaining cleaning solution. The isolation valves are then reopened to put the heat exchanger back in service.

In addition to external or immersion heat exchangers that may be in the system, the heat exchanger within the heat source, be it a boiler, heat pump or other source, should be kept as clean as possible to maximize heat transfer to system water and maintain high efficiency.

Figure 5-16

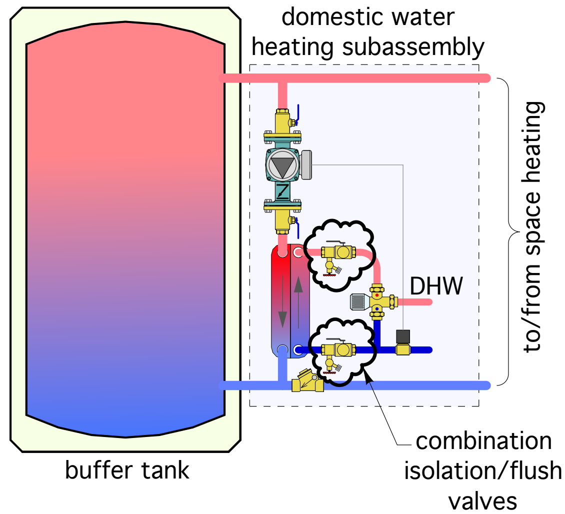


Figure 5-17

Courtesy of Apollo Unlimited

Figure 5-17 shows a severely corroded boiler heat exchanger. The scaling is the result of poor water quality in combination with repeated thermal expansion and contraction of the heat exchanger surfaces as the boiler operates. This scale can significant reduce boiler efficiency.

Figure 5-18

Courtesy of Apollo Unlimited

Systems equipped with high-performance air and dirt separators, and operating with properly demineralized water, provide a strong defense against loss of boiler heat exchanger performance due to scaling over the life of the system.

Heat emitters are also heat exchangers in a generic sense. Dust, or physical damaged to the fins on baseboard elements, radiators, or the coil of an air handler will decrease heat transfer. Figure 5-18 shows the finned-tube element in a wall convector with an accumulation of dust, food waste, wrappers, gum, and other debris. In addition to being a hygiene issue, the rate of heat emission from this baseboard will only be a fraction of its intended output.

Annual inspections of heating systems should include an examination of all accessible heat exchanger surfaces. Fins that are bent can usually be straighten using a fin comb. Dust and pet hair can be removed using a brush and shop vac.

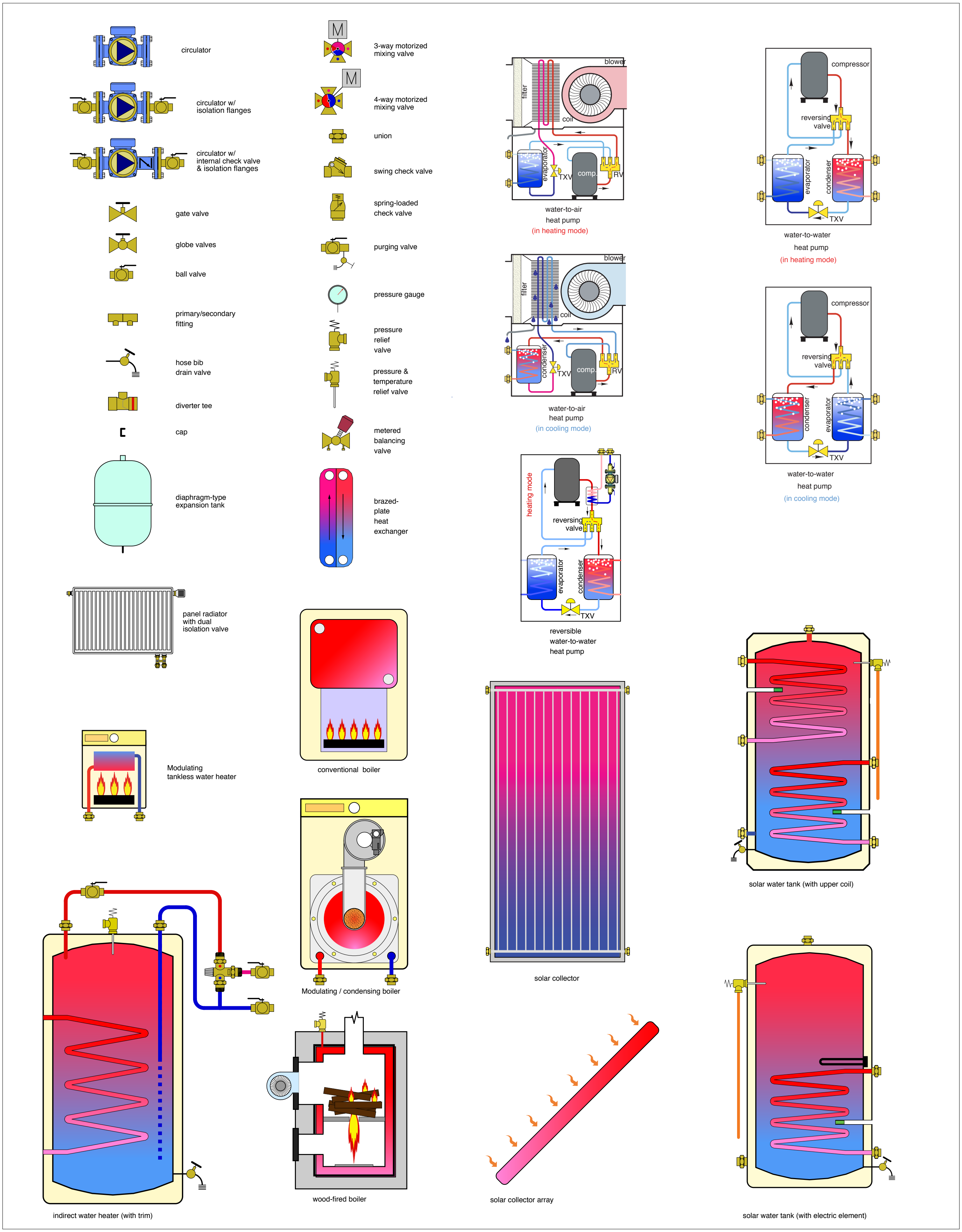
SUMMARY:

The transfer of heat at predictable rates and in many different locations is critical to the proper operation of hydronic heating or cooling systems. Understanding how heat is transferred by conduction, convection and radiation, and having reasonable methods of predicting the rate of heat transfer, can identify potential “bottlenecks” in the path from a heat source to one or more heat emitters, or between a source of chilled water and one or more terminal units.

Many of the performance problems associated with improperly designed, installed or commissioned hydronic systems can be traced to inadequate heat transfer in at least one area of the system. *It could be* a boiler with a fouled heat exchanger, an undersized heat emitter or insufficient flow rate caused by an improper circulator selection. *It could be* a thick carpet placed over a heated floor slab, a finned-tube element within a baseboard that’s covered with pet hair or a heat exchanger that’s piped for parallel flow rather than counterflow. *It could be* an assumption that natural convection is as effective as forced convection, or that the characteristics of the fluid circulating through the system have no significant effect on the rate of heat transfer.

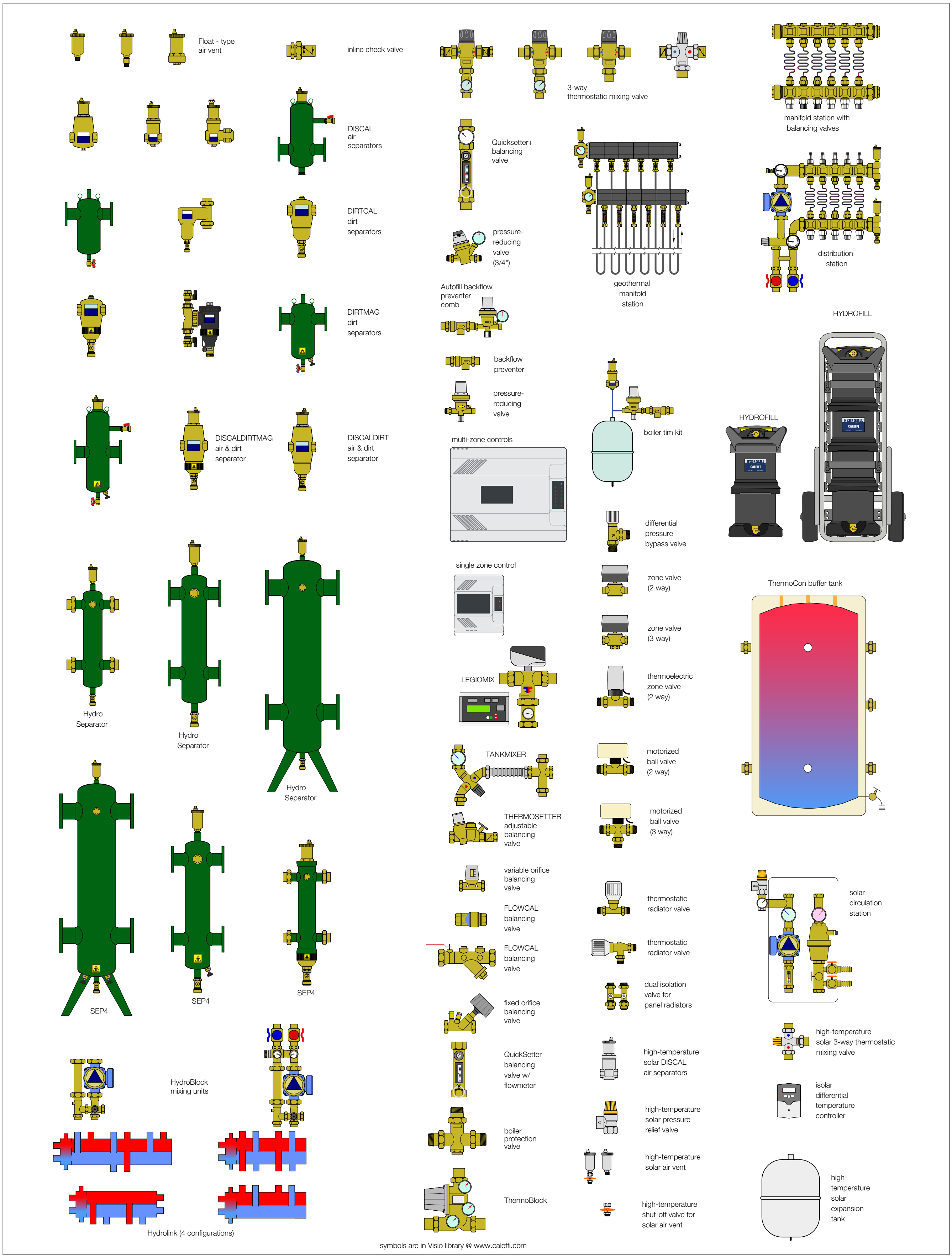
The list of “it could be” scenarios extends to hundreds of possibilities in which inadequate heat transfer is the underlying problem. Designers with a solid understanding of heat transfer fundamentals, and the ability to estimate heat transfer rates using the formulas and methods discussed in this issue of *idronics*, are far less likely to encounter these “it could be” scenarios.

GENERIC COMPONENTS



APPENDIX A: PIPING SYMBOL LEGEND

CALEFFI COMPONENTS



APPENDIX B: THERMAL MODEL FOR A FINNED-TUBE BASEBOARD

This appendix provides a detailed analytical model for the thermal performance of a finned-tube baseboard heat emitter. The model allows the temperature of the water passing through the baseboard to be determined at any location along the element, based on the inlet water temperature, room air temperature, fluid and flow rate. When the length of the baseboard is known, the fluid outlet temperature can be calculated. Once the fluid outlet temperature is determined, it can be combined with the inlet temperature and flow rate to determine the total heat output of the heat emitter.

The fluid temperature at a given location along a finned-tube baseboard can be found using Formula B-1:

Formula B-1:

$$T_f = T_{room} + \left\{ \left[\frac{(0.0000504)B}{(Dc)f^{0.96}} \right] L + (T_{inlet} - T_{room})^{-0.4172} \right\}^{-2.3969319}$$

Where:
T_f = fluid temperature at some location L along length of finned-tube element (°F)
T_{room} = room air temperature entering the baseboard (°F)
T_{inlet} = fluid temperature at inlet of baseboard (°F)
D = density of fluid within baseboard (lb/ft^3)
c = specific heat of fluid within baseboard (Btu/lb/°F)
f = fluid flow rate through the baseboard (gpm)
L = position along baseboard beginning from inlet (ft)
B = heat output of the baseboard at 200°F average water temperature, 1 gpm, from manufacturer's literature* (Btu/hr/ft)

The values 0.04, -0.4172, -2.3969319 and 1.4172 are all exponents.

Once the outlet temperature for a baseboard is determined, Formula B-2 can use it along with the inlet temperature and flow rate to determine the total heat output.

Formula B-2:

$$Q = (8.01Dc) f (T_{inlet} - T_{outlet})$$

Where:
T_{outlet} = outlet temperature of fluid leaving baseboard (°F)
T_{inlet} = fluid temperature at inlet of baseboard (°F)
D = density of fluid within baseboard (lb/ft^3)
c = specific heat of fluid within baseboard (Btu/lb/°F)
f = fluid flow rate through the baseboard (gpm)

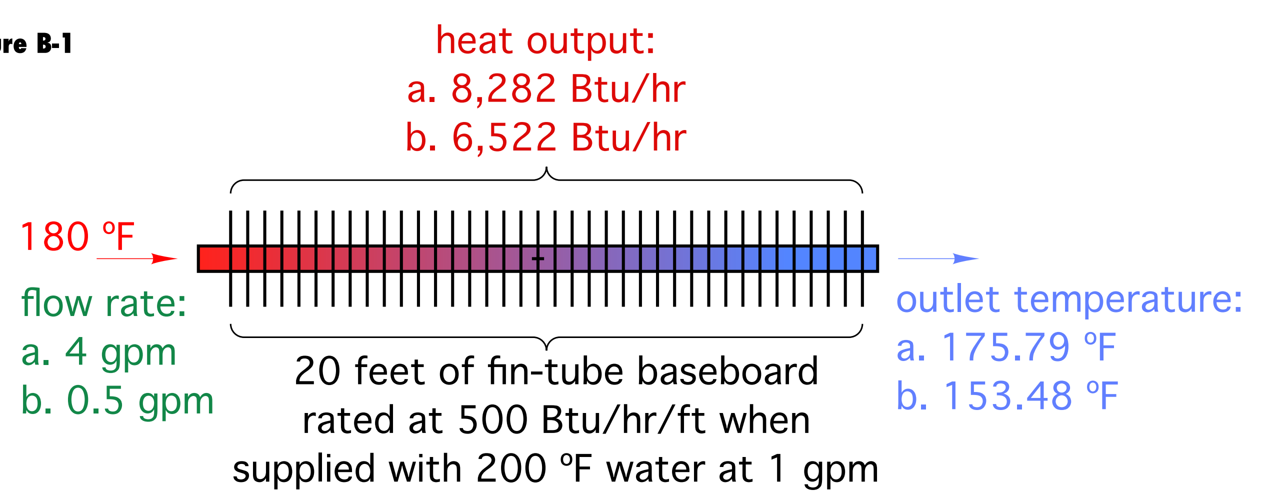
For example: Assume a finned-tube baseboard is 20 feet long, located in a room with a floor-level air temperature of 65°F, and supplied with water at 4 gpm and 180°F. The baseboard's manufacturer rates its heat output at 500 Btu/hr/ft when operating with 200°F water and a flow rate of 1 gpm. Determine the heat output of this baseboard at the stated conditions, and compare it to the output if the flow rate were reduced to 0.5 gpm.

The outlet temperature of the baseboard can be determined using Formula B-1:

$$T_f = T_{room} + \left\{ \left[\frac{(0.0000504)B}{(Dc)f^{0.96}} \right] L + (T_{inlet} - T_{room})^{-0.4172} \right\}^{-2.3969319}$$

$$T_f = 65 + \left\{ \left[\frac{(0.0000504)500}{(61.4 \times 1.00)4^{0.96}} \right] 20 + (180 - 65)^{-0.4172} \right\}^{-2.3969319} = 175.79°F$$

Figure B-1



The total heat released from the baseboard can now be calculated using Formula B-2:

$$Q = (8.01Dc)f(T_{inlet} - T_{outlet})$$
$$Q = (8.01 \times 61.4 \times 1.00)4(180 - 175.79) = 8{,}282\,Btu / hr$$

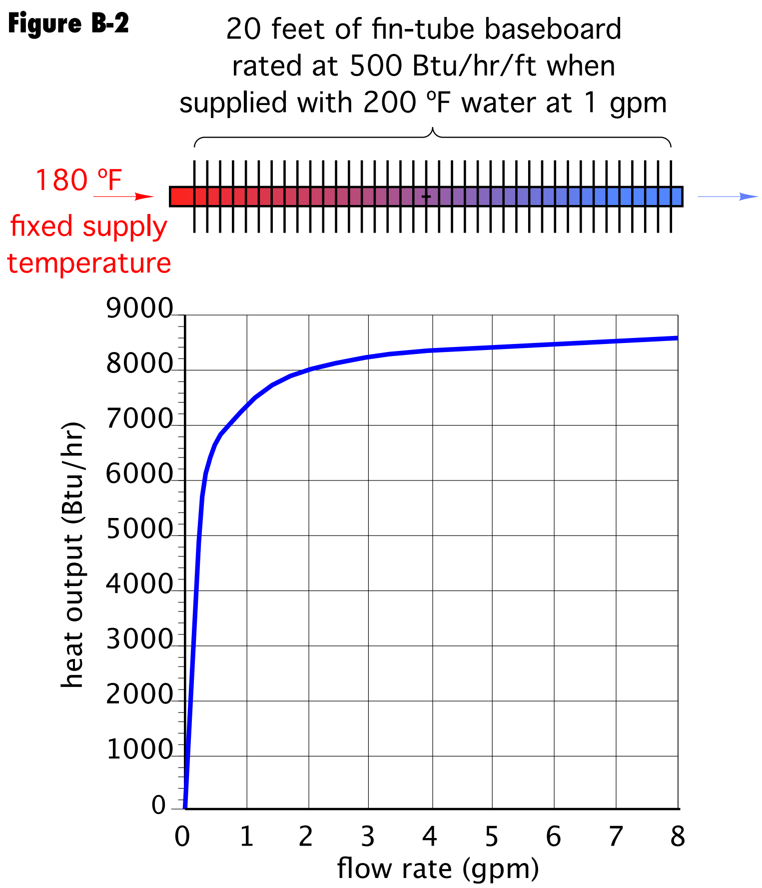
Using the same formulas at the reduced flow rate of 0.5 gpm yields an outlet temperature of 153.48ºF, and a total heat output of 6,522 Btu/hr. A comparison of the two operating conditions is shown in Figure B-1.

By repeating these calculations, it is possible to plot heat output of this baseboard as a function of flow rate. Figure B-2 shows the results over a wider range of flow rate.

This graphs shows a rapid increase in heat transfer at low flow rates, and a relatively slight gain in heat transfer as flow rates rise above approximately 2 gpm.

A similar analytical model for the thermal performance of a radiant panel circuit is given in Appendix C.

Figure B-2



APPENDIX C: THERMAL MODEL FOR A RADIANT PANEL CIRCUIT

Formula C-1 can be used to determine the outlet temperature of a specific radiant panel based on its construction and the conditions under which it operates:

Formula C-1:

$$T_{out} = T_{room} + \left(T_{in} - T_{room}\right) \times e^{-\left(\frac{al}{f(8.01Dc)}\right)}$$

Where:

T_{out} = fluid outlet temperature for baseboard (ºF)
T_{in} = fluid inlet temperature to baseboard (ºF)
T_{room} = temperature of room air entering baseboard (ºF)
a = thermal output parameter for a specific panel construction
l = length of radiant panel circuit (ft)
c = specific heat of fluid flowing through circuit based on its average temperature (Btu/lb/ºF)
D = density of fluid in circuit based on its average temperature (lb/ft^3)
f = flow rate through circuit (gpm)
e = 2.71828 (base of natural logarithm system)

Once the outlet temperature for the panel is determined, it can be combined with the inlet temperature and flow rate to determine the total heat output. Use Formula C-2 for this calculation.

Formula C-2:

$$Q = \left(8.01Dc\right) f \left(T_{inlet} - T_{outlet}\right)$$

Where:

T_{outlet} = outlet temperature of fluid leaving radiant panel (ºF)
T_{inlet} = fluid temperature at inlet of radiant panel (ºF)
D = density of fluid within baseboard (lb/ft^3)
c = specific heat of fluid within baseboard (Btu/lb/ºF)
f = fluid flow rate through the baseboard (gpm)

The value of “a” in Formula C-1 is based on the exact construction of the radiant panel, including tube spacing, tubing embedment method, finish floor coverings and underside insulation. This value can be determined based on a known relationship between heat output and a corresponding circuit water temperature and room air temperature. The relationship is given as Formula C-3.

Formula C-3

$$a = \left(\frac{s}{12}\right)\left(\frac{q_{up} + q_{down}}{T_w - T_{room}}\right)$$

Where:
s = tubing spacing (inches)
q_{up} = upward heat flux from panel (Btu/hr/ft^2)
q_{down} = downward heat flux from panel (Btu/hr/ft^2)
T_w = water temperature in circuit corresponding to the above heat outputs (ºF)
T_{room} = room air temperature corresponding to the above heat outputs (ºF)

For example, a radiant floor panel with 12” tube spacing creates an average upward heat output of 25 Btu/hr/ft^2, with a corresponding downward heat loss of 2 Btu/hr/ft^2. The average water temperature in the radiant panel circuit at these conditions is 110ºF, and the room air temperature above the panel is 70ºF. Determine the value of “a” for use in Formula C-3.

Solution:

$$a = \left(\frac{s}{12}\right)\left(\frac{q_{up} + q_{down}}{T_w - T_{room}}\right) = \left(\frac{12}{12}\right)\left(\frac{25+2}{110-70}\right) = 0.675$$

Assume a radiant floor panel consists of a single 300-foot-long circuit of ½” PEX tubing, spaced 12 inches apart. When operating at an average water temperature of 110ºF, this panel’s average upward heat output is 25 Btu/hr into a room at 70ºF, and its downward heat output is 2.5 Btu/hr. Further assume that the radiant panel circuit is supplied with water at a temperature of 120ºF and flow rate of 0.8 gpm. Determine:

a. The “a” value of the panel
b. The total heat transfer from the radiant panel
c. The total upward heat transfer from the radiant panel

Solution: The value of “a” can be determined from the given data using Formula C-3:

$$a = \left(\frac{s}{12}\right)\left(\frac{q_{up} + q_{down}}{T_w - T_{room}}\right) = \left(\frac{12}{12}\right)\left(\frac{25+2.5}{110-70}\right) = 0.688$$

It is necessary to determine the density and specific heat of the water at its average temperature before using Formula C-1. However, the average water temperature of the circuit is not known at this point. As an approximation, assume a temperature drop along the circuit of 20ºF. Under this assumption, the average water temperature would be 120 - (20/2) = 110ºF. The density of water at 110ºF is 61.8 lb/ft^3. The specific heat of water at this temperature is 1.00 Btu/lb/ºF. This data, along with the other stated and calculated numbers, is now used in Formula C-1 to determine the circuit's outlet temperature.

$$T_{out} = T_{room} + \left(T_{in} - T_{room}\right) \times e^{-\left(\frac{al}{f(8.01Dc)}\right)}$$

$$T_{out} = 70 + (120 - 70) \times e^{-\left(\frac{0.688 \times 300}{0.8(8.01 \times 61.8 \times 1)}\right)}$$

$$T_{out} = 70 + (50) \times e^{-(0.521)}$$

$$T_{out} = 70 + (50) \times 0.5938$$

$$T_{out} = 99.69^{\circ} F$$

The total heat output of the circuit can now be determined using Formula C-2:

$$Q_{total} = (8.01Dc) f \left(T_{inlet} - T_{outlet}\right)$$

$$Q_{total} = (8.01 \times 61.8 \times 1.00) 0.8 (120 - 99.69)$$

$$Q_{total} = 8043 Btu / hr$$

Finally, the total *upward* heat transfer can be estimated by multiplying the total heat transfer by the ratio of upward heat flux to total heat flux.

$$Q_{up} = Q_{total}\left(\frac{q_{up}}{q_{up} + q_{down}}\right) = 8043\left(\frac{25}{25 + 2.5}\right) = 7,312 Btu / hr$$

QuickSetter™
Balancing valve with flow meter

132 series

Function

The 132 series balancing valve accurately sets the flow rate of heating and cooling transfer fluid supplied to fan coils and terminal units. Proper hydronic system balancing ensures that the system operates according to design specifications, providing satisfactory thermal comfort with low energy consumption. The flow meter is housed in a bypass circuit on the valve body and can be shut off during normal operation. The flow meter permits fast and easy circuit balancing without added differential pressure gauges and reference charts. The threaded version is furnished with a hot pre-formed insulation shell to optimize thermal performance for both hot and cold water systems.

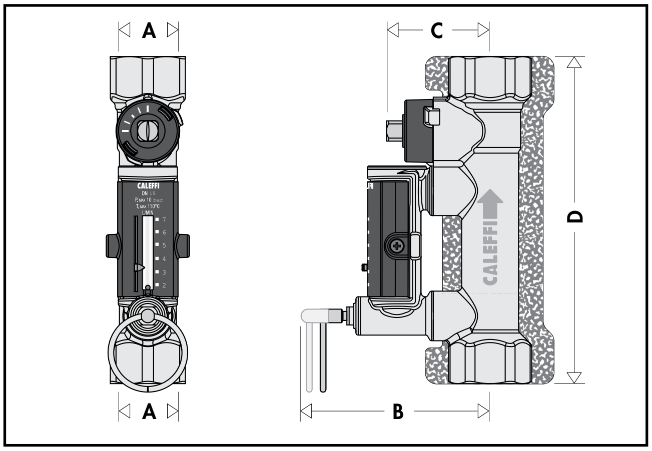
Patent pending.

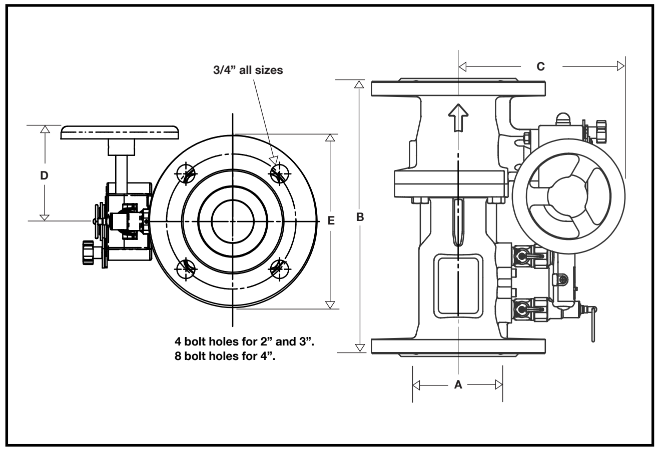
Product range

132 series Balancing valve with flow meter, threaded .... connections ½”, ¾”, 1”, 1 ¼”, 1 ½”, and 2" NPT female
132 series Balancing valve with flow meter, flanged .... connections 2 ½”, 3”, 4” ANSI 125

Series	132 Threaded	132 Flanged
Materials		
Valve		
Body:	brass	cast iron
Ball:	brass	brass
Ball control stem:	brass, chrome-plated	brass, chrome-plated
Ball seal seat:	PTFE	R-PTFE
Control stem guide:	PSU	PTFE
Seals:	peroxide-cured EPDM	peroxide-cured EPDM
Flow meter		
Body:	brass	brass
Bypass valve stem:	brass, chrome-plated	brass, chrome-plated
Springs:	stainless steel	stainless steel
Seals:	peroxide-cured EPDM	peroxide-cured EPDM
Flow meter float and indicator cover:	PSU	PSU
Performance		
Suitable Fluids:	water, glycol solutions	water, glycol solutions
Max. percentage of glycol:	50%	50%
Max. working pressure:	150 psi (10 bar)	150 psi (10 bar)
Working temperature range:	14 - 230° F (-10 - 110° C)	14 - 230° F (-10 - 110° C)
Flow rate range unit of measurement:	gpm	gpm
Accuracy:	±10%	±10%
Control stem angle of rotation:	90°	90°
Control stem adjustment wrench:	½" - 1¼": 9 mm 1½" - 2": 12 mm	with 5½" diameter handwheel
Connections:	½" - 2": NPT female	2½", 3" and 4": ANSI B16.1 125 CLASS RF flanged
Flow rate correction factor:	20% - 30% glycol solutions: 0.9 40% - 50% glycol solutions: 0.8	20% - 30% glycol solutions: 0.9 40% - 50% glycol solutions: 0.8
Insulation		
Material:	closed cell expanded PE-X	
Thickness:	25/64 inch (10 mm)	
Density:	- inner part: 1.9 lb/ft^3 (30 kg/m^3) - outer part: 3.1 lb/ft^3 (50 kg/m^3)	
Thermal conductivity (DIN 52612):	- at 32°F (0°C): 0.263 BTU·in/hr·ft2·°F (0.038 W/(m·K)) - at 104°F (40°C): 0.312 BTU·in/hr·ft2·°F (0.045 W/(m·K))	
Coefficient of resistance to water vapor (DIN 52615):	< 1,300	
Working temperature range:	32 - 212° F (0 - 100° C)	
Reaction to fire (DIN 4102):	class B2	

Dimensions





Code	A	B	C	D	Wt (lb/kg)
132432A	½”	3 5/16"	1 13/16"	5 ¾"	2.0/0.9
132552A	¾”	3 5/16"	1 13/16"	5 ¾"	1.8/0.8
132662A	1”	3 3/8"	1 7/8"	6 ¼"	2.4/1.1
132772A	1¼”	3 ½"	2"	6 ½"	2.8/1.3
132882A	1½”	3 5/8"	2 ¼"	6 ¾"	3.4/1.5
132992A	2”	3 ¾"	2 ½"	7"	4.4/2.0

Code	A	B	C	D	E	Bolt circle dia	Wt (lb/kg)
132060A	2 ½”	11 7/16"	6 31/32"	3 7/8"	7"	5 ½"	32/15
132080A	3"	12 7/32"	7 9/32"	3 7/8"	7 ½"	6"	40/18
132100A	4"	13 25/32"	7 29/32"	3 7/8"	9"	7½"	57/26

Flow rate ranges

Code	Connection	Flow rate (GPM)	Full open Cv
132432A	½” NPT	½ – 1¾	1.0
132552A	¾” NPT	2.0 – 7.0	6.3
132662A	1” NPT	3.0 – 10.0	8.3
132772A	1¼” NPT	5.0 – 19.0	15.2
132882A	1½” NPT	8.0 – 32.0	32.3
132992A	2” NPT	12.0 – 50.0	53.7

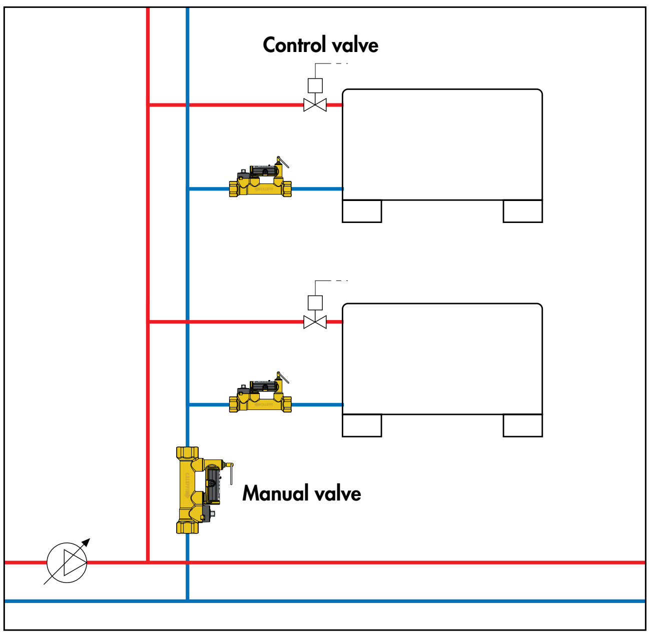
Flow rate ranges

Code	Connection	Flow rate (GPM)	Full open Cv
132060A	2 ½” flange	30 - 105	87
132080A	3” flange	38 - 148	164
132100A	4” flange	55 - 210	242

Advantages of balanced circuits

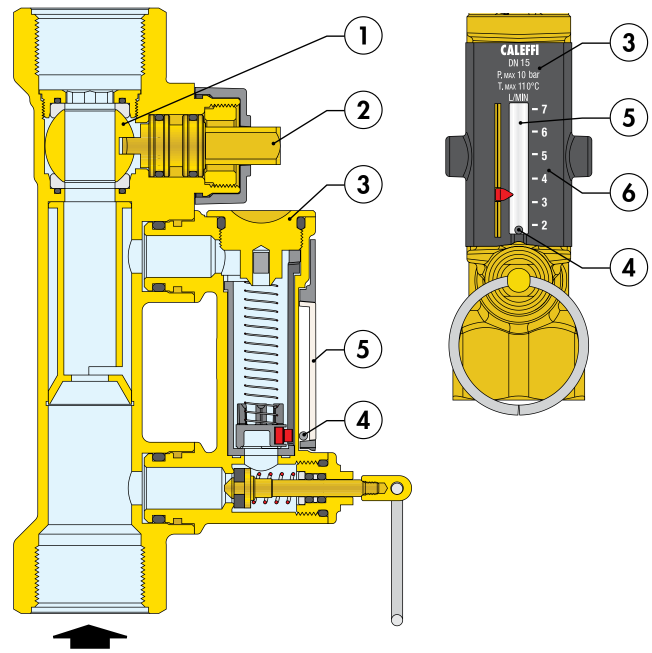
Balanced circuits have the following principal benefits:

1. The system emitters operate properly in heating, cooling and dehumidification, saving energy and providing greater comfort.
2. The zone circuit pumps operate at maximum efficiency, reducing the risk of overheating and excessive wear.
3. High fluid velocities which can result in noise and abrasion are avoided.
4. The differential pressures acting on the circuit control valves are reduced preventing faulty operation.



Operating principle

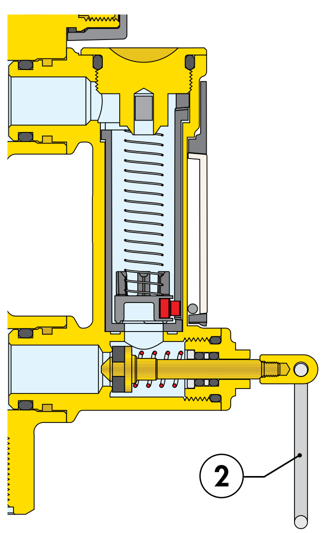
The balancing valve is a hydraulic device that controls the flow rate of the heating/cooling transfer fluid.
The control mechanism is a ball valve (1), operated by a control stem (2). The flow rate is manually and properly set by use of the convenient onboard flow meter (3) housed in a bypass circuit on the valve body. This circuit is automatically shut off during normal operation. The flow rate is indicated by a metal ball (4) sliding inside a transparent channel (5) with an integral graduated scale (6).



Construction details

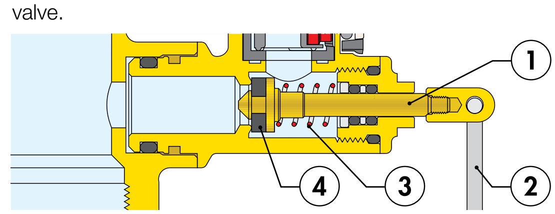
Flow meter

When activated, the flow rate is indicated on the flow meter housed in a bypass circuit on the valve body. When finished reading the flow rate, the flow meter is automatically shut off, isolating it during normal operation.
Use of a flow meter greatly simplifies the process of system balancing since the flow rate can be measured and controlled at any time without differential pressure gauges or reference charts. The onboard flow meter eliminates the need to calculate valve settings during system setup. Additionally, the unique onboard flow meter offers unprecedented time and cost savings by eliminating the long and difficult procedure of calculating pre-settings associated with using traditional balancing devices.
The by-pass circuit easily detaches for cleaning.



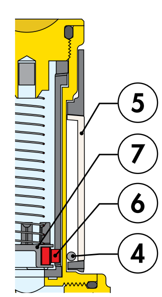
Flow meter bypass valve

The bypass valve (1) opens and closes the circuit between the flow meter and the valve. The bypass valve is easily opened by pulling the operating ring (2), and is automatically closed by the internal return spring (3) when finished reading the flow rate. The spring and the EPDM seal (4) provide a reliable seal to isolate the flow meter during normal operation, protecting potential debris from interfering with spring/magnetic disc mechanism.
The operating ring (2) material has low thermal conductivity to avoid burns if the flow meter is opened while hot fluid is passing through the valve.



Ball/magnet indicator

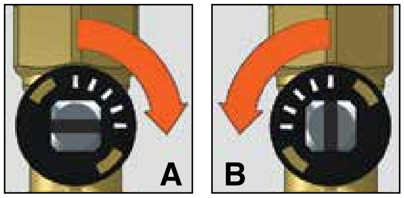
The metal ball (4) that indicates the flow rate is not in direct contact with the heating/cooling transfer fluid passing through the flow meter.
This is an effective and innovative measuring system in which the ball slides up and down inside a transparent channel (5) that is isolated from the fluid flowing through the body of the flow meter. The ball is moved by a magnet (6) connected to a float (7). In this way the flow rate indication system **remains perfectly clean and provides reliable readings over time**.



Complete closing and opening of the valve

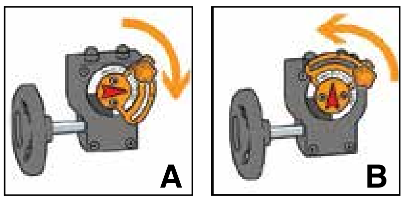
The valve can be completely closed and opened. A slot on the control stem indicates the valve position. When the control stem is turned fully clockwise (the slot is perpendicular to the axis of the valve), the valve is fully closed (A). When the control stem is turned fully counter-clockwise (the slot is parallel to the axis of the valve), the valve is fully open (B).

Completely closed **Completely open**



For the flanged version, rotate the adjusting handwheel 90° for the complete opening and closing of the valve as shown in (A) and (B). When at the desired position, lock the adjustment screw.

Completely closed **Completely open**

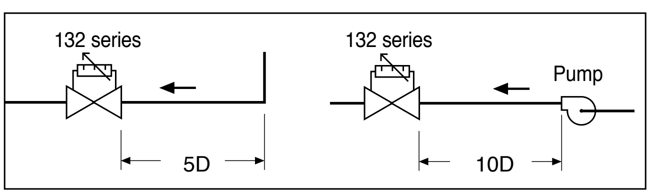


Insulation

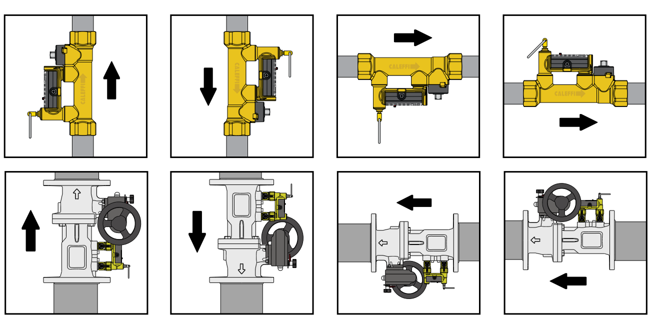
The 132 series threaded version,is supplied with a hot pre-formed insulating shell. This system ensures perfect heat insulation and keeps out water vapor from the environment. Additionally, this type of insulation is ideal in cold water circuits as it prevents condensation from forming on the surface of the valve body.

Installation

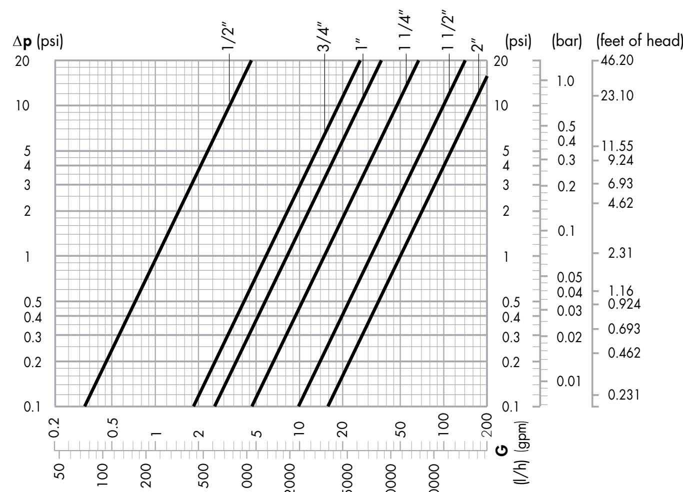
Install the balancing valve in a location that ensures free access to the flow meter shutoff valve, control stem and flow rate indicator. To ensure accurate flow measurement, straight sections of pipe installed as shown is recommended.



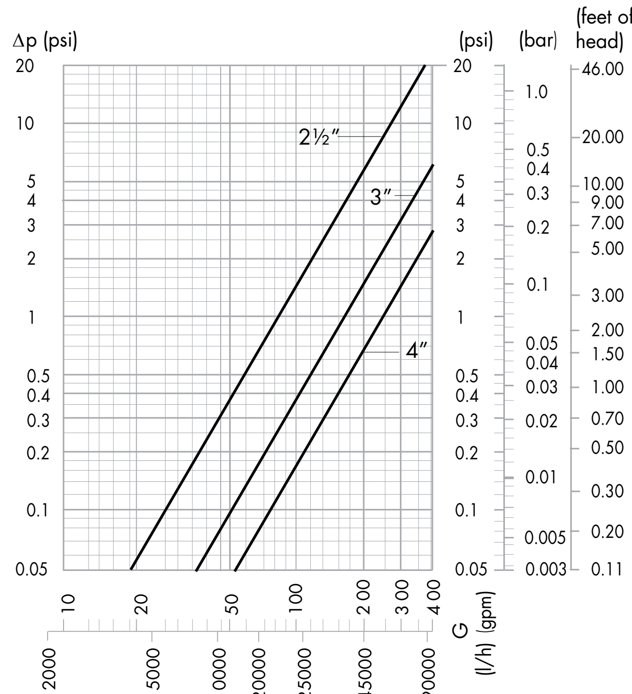
The valve can be installed in any position with respect to the flow direction shown on the valve body. Additionally, the valve can be installed either horizontally or vertically.



Hydraulic characteristics at 100% open



Code	Connection	Flow rate (GPM)	Full open Cv
132432A	½" NPT	½ – 1¾	1.0
132552A	¾" NPT	2.0 – 7.0	6.3
132662A	1" NPT	3.0 – 10.0	8.3
132772A	1¼" NPT	5.0 – 19.0	15.2
132882A	1½" NPT	8.0 – 32.0	32.3
132992A	2" NPT	12.0 – 50.0	53.7



Code	Connection	Flow rate (GPM)	Full open Cv
132060A	2 ½" flange	30 - 105	87
132080A	3" flange	38 - 148	164
132100A	4" flange	55 - 210	242

QuickSetter™

Pull - Adjust - Release

- Industry unique design; simply pull, adjust, and release for quick and accurate circuit balancing.
- No need for differential pressure tools or cross-reference charts.
- Built-in flow meter with memory pointer provides direct readout in GPM.
- Flow window scale never clouds; indicator ball travels within transparent isolated glass channel.
- Full range of pipe sizes from 1/2" to 2" NPT and 2-1/2" to 4" flanged to fit most commercial projects.
- Low lead models for plumbing applications are perfect for DHW recirculation.

Components for today's modern hydronic systems

Heating & Cooling

www.caleffi.com - Milwaukee, WI USA

CALEFFI
Hydronic Solutions