Flow Path (FL) Package
Users' Guide

The Control Volume Hydrodynamics (CVH) and Flow Path (FL) packages are responsible for modeling the thermal-hydraulic behavior of liquid water, water vapor, and gases in MELCOR. Modeling is based on a control volume/flow path formulation and is described in detail in the Thermal Hydraulic (CVH and FL) Packages Reference Manual.

This Users' Guide describes input to the FL package, which is concerned with the geometry and characteristics of connections between control volumes, through which the control volume contents may be transported. FL input describes these connections, and defines the network through which hydrodynamic materials (water, its vapor, and noncondensible gases, residing in control volumes) may flow. No material resides in flow paths. When a flow of atmospheric materials (water vapor and noncondensible gases) enters a control volume below the surface of a water pool in that volume, the resulting mass and heat transfer will be calculated if requested by user input to the FL package. The phenomena of thermal equilibration and condensation of evaporation are treated as occurring within the flow path, using models from the SPARC90 code.

In addition to geometry, FL input includes all definition of flow resistance, including any frictional losses associated with the walls of control volumes and blockages calculated by the COR package. Special models including externally controlled flow areas (valves), forced flows, and momentum sources (pumps) are also defined by FL input in conjunction with functions defined by the Tabular Function (TF) and Control Function (CF) packages.

The geometry and contents of control volumes are defined by input to the CVH package, as described in the Control Volume Hydrodynamics (CVH) Package Users' Guide.

Contents

List of Figures

1. Flow Path Models

The Users' Guide presents only an overview of the modeling in the MELCOR Flow Path (FL) package. More detailed descriptions, with appropriate references, may be found in the Thermal Hydraulic (CVH and FL) Packages Reference Manual.

1.1 Flow Path Definition

Each *flow path* connects two control volumes. Each connection is referred to as a *junction*; the two junctions associated with a flow path may be at different elevations. One volume is referred to as the *from* volume and the other as the *to* volume, thus defining the direction of positive flow.

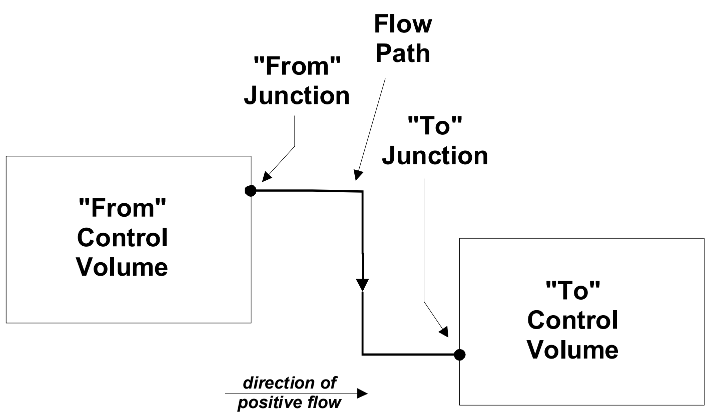

Figure 1.1 Flow Path Definition

There is no residence time for material flowing through a flow path. Therefore, there is no mass or energy associated with a flow path; all mass and energy reside in the control volumes. There is no heat or mass transfer between the pool and atmosphere materials flowing through the flow paths, nor is there heat transfer to or from structures since heat structures are not allowed to interact with flow paths.

A flow path may represent a pipe-like connection in a tank-and-tube model, or the open area of a separating surface (cell boundary) in a finite-difference-like model. The former case represents the limit of a control volume when residence effects are not important. In the interest of computational speed, MELCOR nodalizations are typically made relatively coarse, with only a modest number of control volumes. Therefore, it is common to reduce volumes of intermediate size but potentially high flow—such as relief valve discharge lines, pressurizer spray lines in a PWR, and jet pumps in a BWR—to simple flow paths in a MELCOR nodalization. The associated volume is typically included in one of the connected control volumes.

The same input parameters, differently chosen, are used to represent the two limiting types of flow paths (pipe-like and cell boundary), and a variety of intermediate cases. The primary difference is in the definition of the junctions of control volumes. The *junction elevations* represent the elevations of the central points of the connections to the respective control volumes; the *junction opening heights* represent the range of elevations over which material can be drawn *out of* the corresponding volume into the flow path.

For a pipe-like flow path, the junction elevations are typically unequal and the opening heights should be characteristic of the dimensions of the pipe. For a cell-boundary flow path, the junction elevations should be chosen to be equal, defined by the elevation of the mid-point of the cell boundary, and each opening height should be characteristic of the dimensions of the associated control volume.

1.1.1 Area and Length

Among the fundamental properties of a flow path are its area and its length. In many cases a flow path represents a geometry with varying flow area. In most cases, the area input as FLARA on the FLnnn01 record in FL input should be chosen as the minimum area along the path.

A flow path need not be fully open. The fraction F $(0.0 \le F \le 1.0)$ that is initially open may be defined by the variable FLOPO on the FLnnn01 record. F may also be modified as a function of time by a valve model in the flow path, as described in Section 1.6.1.

The CVH package calculates a velocity for each phase, pool and atmosphere, in each flow path. Only the open area, $F \bullet FLARA$, is used in the models: the volume flow $J \equiv F \bullet FLARA \bullet v$ is used in the advection of materials and in definition of wall friction (Section 1.2.2), and the mass flux based on the open area is the one compared with the critical mass flux in the choking model (Section 1.4). The individual variables F and FLARA may clearly be chosen in several ways. It is common—but not universal—in constructing FL package input to choose FLARA as the *maximum* area that will *ever* be open in the flow path. FLOPO will then be 1.0 for all flow paths not containing valves, and a fully open valve will correspond to an open fraction of 1.0.

The length specified as FLLEN on the FLnnn01 record in the FL input will be used as an *inertial length*. The inertia of a flow path is a measure of the average mass per unit area along its length. If the area is not constant, a rigorous approach is to chose the length input as FLLEN in conjunction with the flow path area, FLARA, to match this average:

$$\frac{FLLEN}{FLARA} = \int_{"fm"}^{"to"} \frac{dx}{A(x)} \tag{1.1}$$

Here *x* measures distance along a path from the center of the "from" volume to the center of the "to" volume, and *A(x)* is the flow area at *x*. In practice, this is rarely necessary.

FLLEN may also be used as the default length over which the force between pool and atmosphere acts. See Section 1.3.2 for details. It is *not* used in friction calculations; *segment lengths* are used instead, as discussed in Section 1.2.2.

1.1.2 Horizontal and Vertical Paths, Junction Openings

Flow paths are designated as *horizontal* or *vertical*, depending on the dominant direction of flow in the path, by user input on record FLnnn02; the default is *vertical*. The designation affects the definition of junction geometry and the default definition of the momentum exchange length in two-phase flow (Section 1.3.2).

The *nominal elevation* of each junction, defined by ZFM or ZTO on input record FLnnn00, represents the midpoint of the connection, and must lie within the range of elevations contained in the volume. The *junction opening height*, defines the range of elevations, H, over which material may be drawn *out of* a control volume through a flow path. By default, as discussed below, this range is calculated from a total junction height based on the flow path area and orientation; this total height may be over-ridden by user input of FLHGTF or FLHGTT on record FLnnn01. In either case, if the top and/or bottom of the junction opening, at elevation $Z \pm H/2$, lies outside of the control volume, the junction opening for a horizontal flow path will be truncated; for a vertical flow path, it will first be translated to preserve as much as possible of the original height H.

In MELCOR, the user may elect to specify the elevations of the top and bottom of the junction opening to either or both control volumes directly on FLnnn0F and/or FLnnn0T records. (These records may also be used to modify the junction opening on restart.) If these records are used, no adjustments will be made. The bottom of the opening must lie between the bottom of the control volume and the nominal junction elevation (inclusive), while the top must lie between the nominal junction elevation and the top of the control volume (inclusive).

The default values of junction opening heights (H in Figure 1.2) are the diameter of a circle of area FLARA for a horizontal path, and the radius of such a circle for a vertical path. These values are appropriate for a tank-and-tube model, as shown in Figure 1.2. For a horizontal flow path, the default represents the range of altitudes seen by a circular pipe; for a vertical flow path, the default represents a rough estimate of the extent to which the pool surface may be drawn down by flow through a vertical drain.

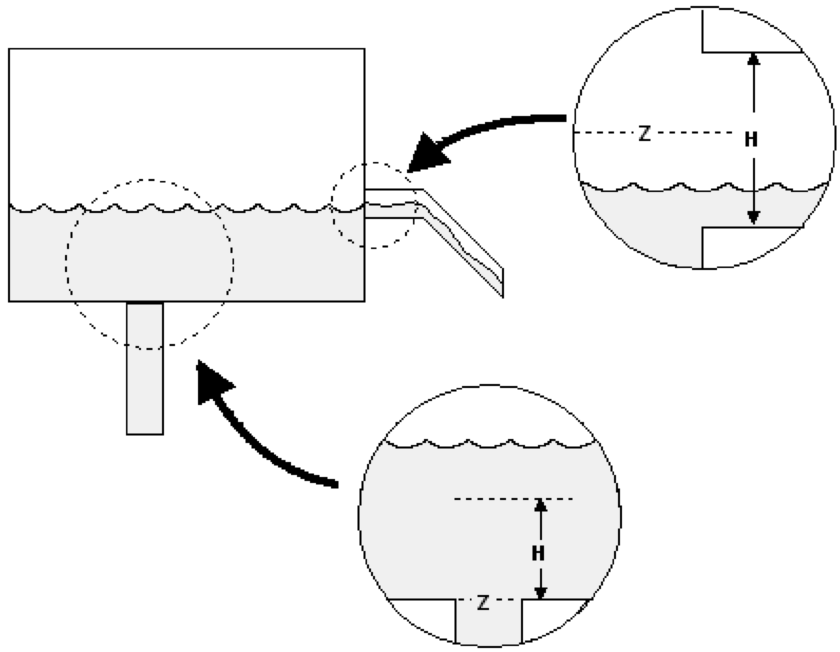

Figure 1.2 Junction Geometry in Tank-and-Tube Limit

The elevations of the top and bottom of each flow path, whether directly input or calculated, are included in the "FLOW PATH TIME INDEPENDENT EDIT" generated by MELGEN and reproduced at the beginning of each MELCOR run.

A flow path may also represent an open surface separating two control volumes. For a horizontal flow path, corresponding to a vertical surface, the opening height should be taken to include the entire open area. The elevation of the center of the opening should be used for *both* junction elevations, as shown in Figure 1.3. It is generally necessary to override the default opening height to ensure that the entire open area—and only that area—lies within the junction opening. For vertical flow through a horizontal separating surface, the junction elevations should again be chosen to be equal as shown in Figure 1.3. The opening height has no rigorous interpretation; it serves only to define the range of elevations from which material may be drawn, and some significant fraction of the volume height is appropriate. Because the flow equations include both gravitational (buoyancy) and momentum exchange (entrainment) forces, slip between the phases will tend to create and preserve stratification unless velocities are great enough to cause entrainment. Thus, extremely small junction openings are not necessary to preserve the stratification of pool and atmosphere in cases where it would be expected to occur. This is discussed further, in the context of the two-phase flow, in Section 1.3.1.

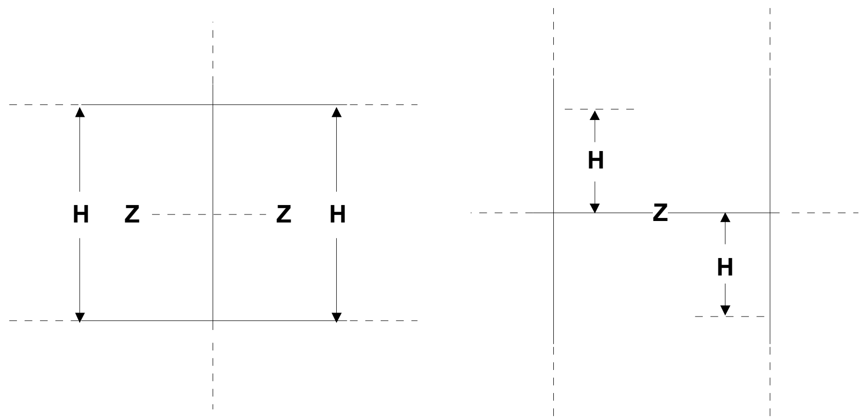

Figure 1.3 Junction Geometry in Separating-Surface Limit

1.1.3 Gravitational Head Terms

The equation for each flow includes a gravitational head term, calculated as the sum of three contributions:

(1) the head difference between the reference point for pressure in the *from* volume and the junction in that volume;

(2) the head difference associated with the change of elevation within the flow path; and

(3) the head difference from the junction in the *to* volume and the reference point in that volume.

The pressure in a MELCOR control volume is defined at the pool/atmosphere interface, taken as the bottom of the volume if there is no pool and the top of the volume if there is no atmosphere.

For a pipe-like flow path with junctions at different elevations, the second, within-path contribution is particularly significant. There is a basic limitation in its evaluation: the flow path has no contents and therefore no density or temperature of its own. A control volume density (or an average) must therefore be used for the within-path contribution to the head. This is adequate for those purposes where the net head results from the difference between pool and atmosphere density. In particular, it properly accounts for the additional head which must be overcome to depress a liquid surface and clear sparger vents or to raise a liquid surface to initiate overflow of a standpipe. It is *not* adequate for calculation of natural convection where the net head results from the change in density of one phase as a function of temperature.

Therefore, if a natural convection loop involving a number of control volumes is expected to be an important part of a MELCOR calculation, flow paths connecting points with different elevations should *not* be used to represent connections through which the circulating flow will pass.

1.2 Frictional Losses

All dissipative pressure drops between volumes are assumed to take place within the flow paths connecting them. Contributions from both form loss and wall friction are included. An additional frictional term calculated by the COR package may be included where appropriate to model the effect of blockage by core debris, as described in Section 1.6.6.

1.2.1 Form Loss Coefficients

The form loss calculation is based on user-input loss coefficients (K in Equation (1.2)), which may be different for forward and reverse flow. These coefficients are input on the FLnnn03 record, and are applied directly to the velocities calculated by the CVH and FL packages. The resulting pressure differential (in the direction of positive flow) is

$$\Delta P_{\varphi} = -\frac{1}{2} K \rho_{\varphi} \left| v_{\varphi} \right| v_{\varphi} \tag{1.2}$$

for phase ϕ, where $\phi = P$ or A to denote pool or atmosphere.

For a complicated pipe network, the individual contributions from multiple changes of area must be combined into a single loss coefficient, to be used in Equation (1.2).

1.2.2 Flow Path Segments and Wall Friction

A wall friction contribution is added to the form loss term. In evaluating this contribution, the flow path is treated as consisting of one or more segments in series; this allows an accurate representation of wall friction in complicated, pipe-like geometries.

The flow path velocity and open area are used with the segment area to calculate a distinct velocity within each segment, based on the assumption of incompressible flow. The segment velocity is then given by

$$v_s = FAv / A_s \tag{1.3}$$

(F is again the fraction of the nominal area that is open), so that the segment velocity depends only on the volumetric flow through the flow path, $J \equiv FAv$. The frictional pressure differential is evaluated as

$$\Delta P_{\varphi} = -\sum_{s} \left(2 f_s L_s / D_s\right) \rho_{\varphi} \left| v_{\varphi,s} \right| v_{\varphi,s} \tag{1.4}$$

where L_s and D_s are the length and hydraulic diameter of the segment. The Fanning friction coefficient, f_s, is calculated according to the method of Beattie and Whalley, using a mixture Reynolds number based on the segment velocity.

The segments should represent all important aspects of the flow path, from volume center to volume center. If wall losses within a volume are important, the geometry of the appropriate parts of the volume should therefore be included as segments in the flow path, because all frictional losses are assumed to occur within the flow paths.

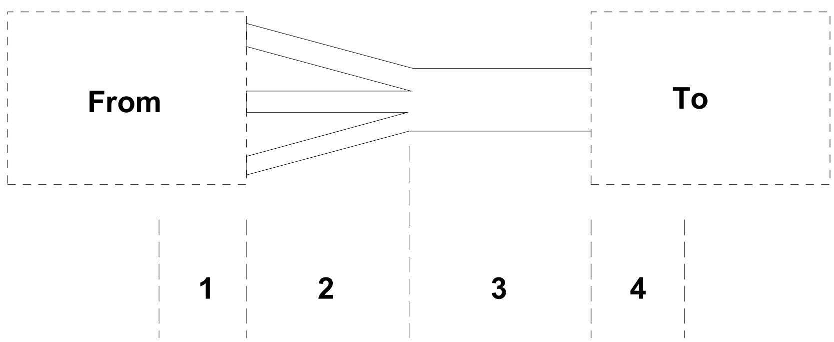

Figure 1.4 Flow Path Segments

Figure 1.4 illustrates the use of flow path segments. Volumes *from* and *to* are connected by a pipe manifold, which will be represented as a flow path. For the situation depicted, input for segment 1 should be based on the flow area, hydraulic diameter, and half length of volume *from* and input for segment 4 on the same properties for the *to* volume. Input for segment 2 should use the length and hydraulic diameter of a single tube and the total area, which is three times the area of a single tube. Appropriate tube properties should be used for segment 3. Note that no material resides in the flow path; if the volume of the manifold is significant, it should be accounted for by increasing the volumes of *from* and/or *to*.

The segmentation of a flow path affects frictional losses only; the segment lengths and areas are not used by any other models.

1.3 Two-Phase Flow

The cross-sectional area of a flow path may be shared by flows of pool and atmosphere materials. If both are present, their velocities will not, in general be equal, and countercurrent flow is possible. The open flow area is partitioned between them in accordance with a calculated void fraction, α, which should, strictly speaking, be referred to as an "atmosphere fraction."

Separate equations are solved for the two velocities. A *momentum exchange* term is included to model the forces acting between the two flows. A balance among this term, gravitational terms, and friction terms is responsible for the prediction of unequal velocities in quasi-steady flow.

1.3.1 Void Fraction

The void fraction in a MELCOR flow path is defined as the fraction of the open area of that path that is occupied by atmosphere flow. For concurrent flow of pool and atmosphere, it is taken as the fraction of the upstream junction opening that is occupied by atmosphere. For countercurrent flow, a weighted average of the area fractions at the two junctions is used.

This effectively prevents flow of pool *out of* a control volume unless the pool surface is above the bottom of the junction opening in that volume. Similarly, atmosphere is prevented from flowing unless the pool surface is below the top of the opening. Junction opening heights must be specified to ensure that such trapping of pool or atmosphere will occur only where it is physically appropriate.

MELCOR calculations sometimes show a small water pool in the upper of a pair of vertically stacked control volumes, with an atmosphere in the volume below. Users sometimes specify extremely small opening heights for the vertical flow path that defines a separating surface between them in an attempt to produce near-perfect separation of the phases. We have not found this approach to be particularly effective, and have seen evidence that it contributes to numerical problems and slow running of MELCOR. Use of somewhat larger opening heights frequently leads to a more orderly draining of the pool into the lower control volume.

1.3.2 Momentum Exchange Length

As noted above, the flow equations include both gravitational separation (buoyancy) and momentum exchange (entrainment) forces. A simple model is used for the momentum exchange; it was chosen to reproduce the Wallis flooding curve in the appropriate limit, and is described in the CVH/FL Reference Manual. The extent of coupling or separation between the flows is strongly influenced by the relative magnitudes of these forces, which are proportional to the distances over which they act. For horizontal flow paths, the default length over which momentum exchange occurs is taken as the inertial length of the flow path. For vertical flow paths, it is taken as the difference in elevation between the lowest point (including the junction opening) and the highest point in the flow path. The different treatment of vertical paths is intended to make the two forces act over the same distance for vertical paths which represent the boundaries of vertically stacked control volumes.

These defaults are appropriate for simple vertical pipes, and not unreasonable for horizontal pipes. However, they can lead to significant overprediction of coupling between the phases—and underprediction of countercurrent flow—in a number of cases where a flowpath includes a number of separated channels. Examples include the entrance or exit paths for a reactor core, and perhaps steam generator tube bundles, where the two phases may flow through distinct, noninteracting portions of the actual geometry.

The default length may be replaced by a user-input length XL2PF on the optional FLnnn05 input record. Increasing the length will tend to tighten coupling between the flows, reducing both the relative velocities and the limits on countercurrent flow; decreasing the length will have the opposite effect. Reducing the length for core flow paths will tend to reduce the "levitation" of water sometimes predicted by MELCOR in the upper plenum and core regions after lower regions have voided. A momentum exchange length of zero can be used to model a "perfect" separator in steam generator secondaries.

1.4 Choked Flow

The critical mass flux for pool flow in MELCOR is based on the RETRAN model; it uses the Moody model for saturated water and the Henry-Fauske model for subcooled water, with a small interpolation region between them. A further interpolation to simple orifice flow is imposed at large subcooling. The critical mass flux for atmosphere flow is based on sonic flow at the minimum section. An interpolation scheme motivated by the Moody model is used for mixed flows. The mass flux is evaluated based on the open area of the flow path (the fraction open times the nominal area).

The possibility of choked flow is automatically modeled in all flow paths. No user input is required, but the default forward and reverse discharge coefficients of 1.0 may be modified, if desired. The choking model restricts the mass flux, through a flow path (based on the open area) to be no greater than the critical value, multiplied by the discharge coefficient corresponding to the current flow direction. A discharge coefficient less than 1.0 may be used to account for *vena contracta* effects. The possibility of choking in a given flow path may be effectively eliminated by specifying a value substantially greater than 1.0 for the associated discharge coefficient.

If choking is detected, pool and atmosphere velocities are set equal. For more details, see the Thermal Hydraulics (CVH and FL) Packages Reference Manual.

1.5 Bubble (SPARC) Physics

If a flow of atmospheric material enters a control volume below the surface of the pool in that volume, it must pass through the pool to reach its final destination. By default, any interactions during this passage are ignored and the atmospheric material is simply added to the control volume atmosphere.

The user may specify that interactions between the pool and the rising gases be modeled by setting one or both of the flags IBUBF and/or IBUBBT on the FLnnn02 input record. Selection of the interaction model may be made independently for the two flow directions. If it is active, the RadioNuclide (RN) package will perform a pool scrubbing calculation using the SPARC90 model to evaluate the removal of fission products from rising gases for those flow paths and flow directions—*and only those paths and directions*—for which the model is activated on the FLnnn02 record. No additional input (other than activation of the RN package) is required, and RN cannot be made to perform a pool scrubbing calculation for a flow path where SPARC modeling was not selected in the FL package input.

The condition of the transmitted gases and the receiving pool is modified by mass and heat transfer processes. The interaction process is viewed as involving the breakup of the injected gas stream into a swarm of bubbles, thermal equilibration of the gases with the pool, and saturation of the bubbles with water vapor at local conditions. It is treated parametrically. If there are no noncondensible gases in the flow, the bubbles may be completely condensed. Details of the model are contained in Section 6.1 of the CVH/FL Reference Manual. The efficiency of the process is controlled by sensitivity coefficients in the array C4405.

The interaction is treated as occurring within the flow path. The bubbles are *not* considered to be resident in the pool for the purposes of calculating the pool composition and do *not* contribute to pool swelling.

1.6 Other Models

Several additional models are available for use in flow paths. These can be used to modify the fraction of the flow path area that is open, define momentum sources in the path, specify the velocity in the path, modify the treatment of two-phase flow, include the effects of momentum flux, or model blockage of the path by materials such as core debris.

1.6.1 Valves

The user may include a valve in a flow path to control the fraction of the area of the path that is open. The open area of the flow path is defined as the fully open area (FLARA) multiplied by the fraction open. Valves do not modify the areas of flow path segments and therefore do not affect the dependence of wall friction on volumetric flow.

For the specific case of an ideal check valve that allows flow in one direction only, an alternate approach is available in MELCOR, where the definition of flow path types has been extended to include *one-way* flow paths. This approach avoids the limitations of the numerically explicit nature of the general valve model, including the numerical difficulties that may arise when a user specifies a very large reverse form loss coefficient in an attempt to restrict any calculated reverse flow through a check valve that was open at the start of an advancement. If, however, it is essential to model the differential pressure necessary to open a check valve, valve inertia, or hysteresis, the general approach described below must be used.

Control functions are used to define the fraction open; values returned will be bounded internally to lie between zero and one before they are used. When the fraction open is zero, the flow path is closed. When the fraction open is one, the flow path is fully open and the open area is equal to user input flow path area FLARA. Two approaches are permitted. In simple cases, the user can specify a single control function which defines the fraction of the area that is open. In particular, a relief valve is typically modeled in MELCOR by defining the open fraction as a hysteresis function of the pressure differential between the volume connected by the flow path. (See the example control function input in Section 6.3 of the CF Package Users' Guide.)

In more complicated situations, a trip control function can be specified that is used to switch among three definitions of that fraction as follows:

(1) If the trip control function is zero, the trip is "off" and the area fraction is unchanged from its last value;

(2) if the trip control function is positive, the trip is "on-forward" and the area fraction is that defined by an "on-forward" control function; and

(3) if the trip control function is negative, the trip is "on-reverse" and the area fraction is that defined by an "on-reverse" control function.

See the CF Package Users' Guide for more information on trip control functions.

1.6.2 Pumps and Fans

Pumps can be included in flow paths. They are modeled as introducing a pressure "boost" which is ordinarily a function of the volumetric flow through the path. In defining a pump, the flow should be thought of as the independent variable and the pressure head delivered by the pump as the dependent variable; the actual flow on any timestep is calculated from the balance of this head (as a function of flow) against static, frictional, and acceleration pressure differentials in the rest of the flow circuit.

There are two types of pumps available in MELCOR. The first, referred to as "FANA," was originally intended to model a fan that impels the atmosphere through a flow path. It can also be used to represent a constant speed coolant pump, although in a very simple form. The second, called "QUICK-CF," simply uses a control function to define the pressure head, allowing the user complete freedom but also giving him complete responsibility for all details.

Both models are numerically explicit; that is, the pressure head is based on conditions at the start of the MELCOR system timestep, and remains unchanged throughout the step—even if the CVH package is subcyling.

The FANA model assumes that the fan can be adequately represented by a parabolic pressure head *vs* volumetric flow rate equation. The velocity of the pool is set equal to that of the atmosphere when the pump is on, so that the pump will move pool water through the flow path if the void fraction in the flow path is less than 1.0. The major limitation in the model is the absence of any direct dependence of the pressure head on the density of the flowing material.

The user must define the maximum pressure head, DPMAX, and the volumetric flow, VPZERO, at which the head is zero. The user may optionally define a non-zero volumetric flow, VPMAX, corresponding to the point of maximum head generation; otherwise, maximum head is assumed to occur at zero flow. If DPMAX and VPZERO are both positive, the fan produces a pressure boost in the direction of positive flow, corresponding to a fan blowing from the "from" volume to the "to" volume. The pressure head is then given by

$$\frac{DP}{DPMAX} = \begin{cases} 1.0 & VDOT \le VPMAX \\ \sqrt{\dfrac{VPZERO - VDOT}{VPZERO - VPMAX}} & VPMAX \le VDOT \le VPZERO \\ 0.0 & VPZERO \le VDOT \end{cases} \qquad (1.5)$$

where DP and VDOT are the pressure head and volumetric flow, respectively. A fan which is oriented in the opposite direction, producing its head in the direction of negative flow, can be modeled by simply defining DPMAX and VPZERO to be negative, and reversing the sign of VPMAX, if it is input. In this case, the effect of the pump is maximum for VDOT $\ge$ VPMAX, and is zero for VDOT $\le$ VPZERO < 0.

There is no provision to control the speed of the fan, but it can be turned on and off, if desired, by defining either a control function or a tabular function of time which will be zero when the fan is to be off, and non-zero when it is to be on. By default, the fan is always on; when it is, DP is simply set to zero.

Note that this model is not a reasonable representation of a real pump far outside the range of flows given by VPMAX ≤ VDOT ≤ VPZERO.

The QUICK-CF pump model relies on a user-defined control function to evaluate the pressure boost (in Pa) for the pump. All control logic must be included in the control function. Section 6.7 of the CF Package Users' Guide provides example input for a homologous pump similar to TRAC and RELAP.

1.6.3 Time Dependent Flow Paths

The velocities in a flow path can be specified by the user through control functions or tabular functions. The specified velocity is used for both the atmosphere and the pool. The void fraction is calculated using the standard model.

1.6.4 Pool-First and Other Flow Path Options

Several modifications may be made to the treatment of two-phase flow described in Section 1.3. These are activated by choice of the KFLGFL flag on input record FLnnn02, the same flag that defines the vertical or horizontal orientation of the path.

The velocities of the pool and of the atmosphere in a flow path may be forced to be equal rather than calculated independently.

The partition of the flow area between pool and atmosphere (defined by the flow path void fraction) is ordinarily based on current geometry. Alternatively, it may be defined such that either pool or atmosphere is preferentially transmitted. These options, referred to as *pool-first* and *atmosphere-first*, respectively, should be used with great care (if at all), as they often produce unexpected results. The preferential flow is accomplished by overriding the normal definition of the flow path void fraction if the preferred phase is available to flow through the flow path, setting it to 0.0 for a pool-first path and 1.0 for an atmosphere-first path. If the preferred phase is not available within the junction opening, the other phase is permitted to flow in the normal manner.

1.6.5 Momentum Flux Terms

Momentum flux terms can be included as an option in certain designated flow paths, using information supplied by the user to calculate them. These terms may be important in paths where large flow accelerations may occur, when multidimensional flow patterns occur, or where frictional and form loss terms are very small. In order to obtain a meaningful velocity difference for the momentum flux terms in the direction of flow, the user is required to specify appropriate upstream and downstream flow areas. As currently implemented, the momentum flux terms are calculated in a numerically explicit manner, and are only one-dimensional in nature; i.e., they do not currently include the cross-derivative term v_y $(\partial v_x / \partial y)$, where y is perpendicular to x, the direction of flow. The current model treatment will therefore be incomplete in situations where velocity gradients transverse to the direction of flow are important.

Although the momentum flux terms can be included, other terms in the CVH equations on the order of the velocity squared (the kinetic energy terms) are still omitted.

1.6.6 Flow Blockage Controlled by Other Packages

Phenomena modeled by other packages in MELCOR can result in changes to the geometry—area and flow resistance—of flow paths modeled by the MELCOR hydrodynamic packages. Although such effects can be captured through use of control functions to define effective valves and friction coefficients, this approach places a large burden on the user. Among the most important of these for reactor safety calculations are changes resulting from degradation of the reactor core. In MELCOR, the blockage of flow by relocation of core debris and the opening of flow paths by failure of channel boxes in a Boiling Water Reactor (BWR) can be easily included in the modeling of a flow path. The user is required only to specify the flow orientation and the core cell or cells that contribute to the geometry of the path. When particulate debris is involved, the calculation is based on the Ergun equation for flow in a porous medium.

1.7 Flow Paths to and from Core Volumes

The core package employs a model, referred to as the "dT/dz" model, to infer an axial temperature variation in a control volume containing core cells. (The details of this model are presented in the COR Reference Manual.) It uses the mass flow into the bottom of the control volume as a boundary condition and, as currently coded, assumes that fluid flows up the core. It further assumes that this corresponds to the direction of positive flow, which places a constraint on the definition of flow path orientations: the user must define the "to" and "from" control volumes for core flow paths such that the sign of the velocity is positive when fluid flows up the core. This is illustrated in Figure 1.5.

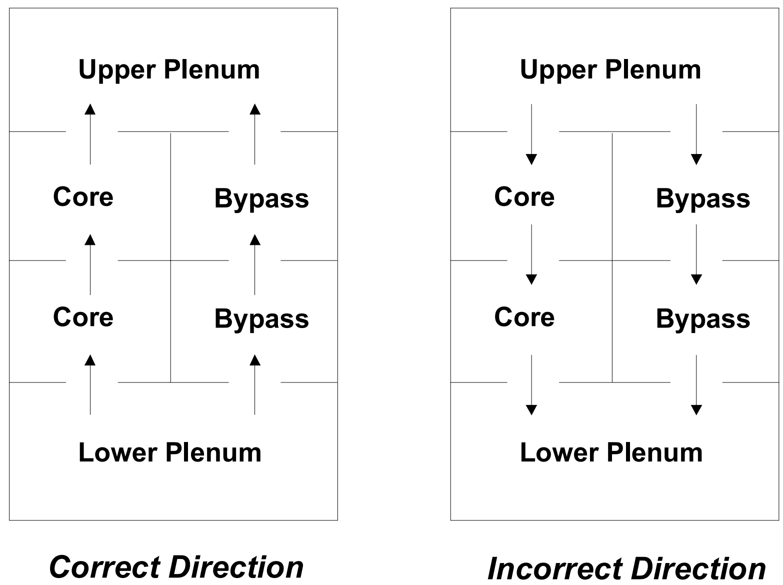

Figure 1.5 Flow Paths for To and From Core Volumes

1.8 Modeling Breaks and Failures

All types of failures which lead to the opening of additional paths for fluid flow are modeled by using flow paths containing valves which are defined to open when a failure criterion is reached. This includes pipe breaks, melt-through or catastrophic failure of the pressure vessel, failure of rupture disks and blowout panels, and failure of containment. Valve models are based on control functions, as described in Section 1.6.1. The flexibility of the MELCOR CF Package, described in the CF Package Users' Guide, makes it possible to define extremely general failure criteria through user input to the CF package in addition to simple pressure or temperature thresholds.

Input to the CF package includes user-specified bounds on the permitted values of the control functions. These limits may be changed on restart of MELCOR. This makes it possible to include in FL input *all* the failure paths and failure criteria that will be required to investigate a variety of scenarios. In any given MELCOR run, all but one can be inactivated by limiting the unwanted failure areas to zero through limits on the associated control functions.

For related scenarios, the transient response of the system will be identical until failures occur. If restart dumps are generated at the times when failures occur, the invariant portions of the calculation need not be repeated. Multiple branches may be followed by restarting MELCOR from the common point with one or another of the failure modes enabled by appropriate modification of control function bounds through CF package input.

2. User Input

FL input to MELGEN defines the geometry of flow paths connecting control volumes, any initial velocities in these flow paths, and various options in the modeling. It also defines momentum sources (pumps) and control logic (valves) through reference to tabular and control functions defined in the TF and CF packages.

FL input to MELCOR is limited to modification of the momentum exchange length for two-phase flow. However, control function data may be modified on a restart, giving the user significant access to control logic affecting pumps and valves. It is common to define input decks to include representations of several possible failures, with all but one disabled through limits imposed on values of control functions through CF package input. This allows several scenarios to be investigated by using modifications to CF input to branch calculations from intermediate restarts without rerunning the earlier, invariant portion of the transient.

2.1 MELGEN User Input

The user input for the flow path package is described below. One set of the following records is required for each flow path. Up to 999 flow paths may be defined. Practically, the number is limited only by the available memory on the computer. Unless otherwise stated, if the field variable starts with I through N, it is an integer. Unless otherwise stated, if the field variable starts with A through H or O through Z, it is a real number.

FLnnn00 – Flow path name, *to* and *from* control volume data
$1 \le nnn \le 999$, nnn is the user-assigned flow path number
Required

This record defines the flow path name. It defines the control volumes that the flow path connects. The flow is positive when it flows from the *from* volume to the *to* volume. The altitude of the connection with *from* and *to* control volumes is also defined.

(1) FLNAME - User defined flow path name
(type = character*16)

(2) KCVFM - *From* control volume number
(type = integer, default = none, units = dimensionless)

(3) KCVTO - *To* control volume number
(type = integer, default = none, units = dimensionless)

(4) ZFM - Altitude of *from* junction
(type = real, default = none, units = m)

(5) ZTO - Altitude of *to* junction
(type = real, default = none, units = m)

FLnnn01 – Flow path geometry
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
Required

This record defines the flow path geometry. Only the first three fields need be present. If record FLnnn0F and/or FLnnn0T are present, data on those records will define the junction opening, and the values of FLHGTF and/or FLHGTT, whether input or default, will not be used.

(1) FLARA - Flow path area
(type = real, default = none, units = m^2)

(2) FLLEN - Flow path length
(type = real, default = none, units = m)

(3) FLOPO - Fraction of flow path open
(type = real, default = none, units = dimensionless)

(4) FLHGTF - *From* junction flow path opening height
(type = real, default = diameter of circle with area FLARA for a horizontal flow path = radius of circle with area FLARA for a vertical flow path, units = m)

(5) FLHGTT - *To* junction flow path opening height
(type = real, default = diameter of circle with area FLARA for a horizontal flow path, = radius of circle with area FLARA for a vertical flow path, units = m)

FLnnn02 – Flow path junction switches
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
Optional

This record defines the type of flow path. A flow path may be horizontal or vertical. The algorithm used to define the equivalent opening height is different for the two cases. A pool-first flow path will only flow pool water until all available pool water is exhausted. When there is no pool available, it will flow atmosphere. The atmosphere-first flow path is equivalent with the pool and atmosphere roles reversed. The normal horizontal or vertical flow paths will flow a mixture of pool and atmosphere consistent with the pool elevation, velocities and flow path opening elevation and height. It is also possible to specify that the pool and atmosphere must have the same velocity.

A flow path may also be defined as a *one-way* path, allowing flow in one direction only, to simulate the behavior of an idealized check valve. This is done by adding a leading digit of 1 for forward-flow-only, or 2 for reverse-flow-only, to the variable KFLGFL that defines the type of flow path.

A bubble rise model may be specified that includes thermodynamic interactions and, if the RN package is active, RN scrubbing. The calculation of heat and mass transfer is performed for *any* acceptable non-zero value of IBUBF or IBUBT. The *specific* non-zero value is significant only if the RN package is active.

(1) KFLGFL - Type of flow path flag
(type = integer, default = 0, units = dimensionless)
= 0 normal vertical flow path
= 1 atmosphere-first vertical flow path
= 2 pool-first vertical flow path
= 3 normal horizontal flow path
= 4 atmosphere-first horizontal flow path
= 5 pool-first horizontal flow path
= 6 pool velocity = atmosphere velocity, vertical flow
= 7 pool velocity = atmosphere velocity, horizontal flow
= 10 – 17 same as 0 – 7, but only forward flow is permitted
= 20 – 27 same as 0 – 7, but only reverse flow is permitted

(2) KACTFL - Active/inactive flow path flag
(type = integer, default = 0, units = dimensionless)
= 0 active
= 1 inactive ** do not use. Input is accepted, but code does not function correctly for flow paths specified as inactive **

(3) IBUBF - *From* junction bubble rise model switch
(type = integer, default = 0, units = dimensionless)
= 0 no bubble rise physics
= 1 SPARC model with RN aerosol and iodine vapor scrubbing
= -1 SPARC model with no RN scrubbing
= -2 SPARC model with scrubbing of RN aerosols only
= -3 SPARC model with scrubbing of RN iodine vapor only

(4) IBUBT - *To* junction bubble rise model switch
(type = integer, default = 0, units = dimensionless)

= 0 no bubble rise physics
= 1 SPARC model with RN aerosol and iodine vapor scrubbing
= -1 SPARC model with no RN scrubbing
= -2 SPARC model with scrubbing of RN aerosols only
= -3 SPARC model with scrubbing of RN iodine vapor only

FLnnn03 – User specified loss coefficients
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
Optional

The user can specify forward and reverse loss coefficients and choked flow discharge coefficients. If this record is input, then at least two values must be input. The first two are interpreted as the loss coefficients. The third and fourth values are the forward and reverse choked flow discharge coefficients.

The loss coefficients determine the form loss pressure drop across a flow path; the forward coefficient is used when the velocity is positive and the reverse coefficient is used when the velocity is negative. For additional information refer to Section 1.2.1 of this Users' Guide and Section 5.4 in the CVH/FL Reference Manual

In the test for choking, critical flow is determined from a correlation and then multiplied by a discharge coefficient. The coefficient may be different for forward flow (positive velocity) and reverse flow (negative velocity). A minimum velocity, below which the test is bypassed, can be defined by sensitivity coefficient C4402(1).

(1) FRICFO - Forward loss coefficient
(type = real, default = 1., units = dimensionless)

(2) FRICRO - Reverse loss coefficient
(type = real, default = 1., units = dimensionless)

(3) CDCHKF - Choked flow forward discharge coefficient
(type = real, default = 1., units = dimensionless)

(4) CDCHKR - Choked flow reverse discharge coefficient
(type = real, default = 1., units = dimensionless)

FLnnn04 – Initial atmosphere and pool velocities
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
Optional

The initial pool and atmosphere velocities may be defined by the user. If this record is present, both velocities must be input; if it is absent, both initial velocities will be taken as 0.0.

(1) VLFLAO - Atmosphere velocity
(type = real, default = none, units = m/s)

(2) VLFLPO - Pool velocity
(type = real, default = none, units = m/s)

FLnnn05 – Length for pool/atmosphere momentum exchange
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
Optional

This record defines the length over which momentum exchange occurs when pool and atmosphere flows share the flow path. The value used affects entrainment and flooding by controlling the balance between the resulting force between the two flows (which tends to couple them and reduce slip) and buoyancy (which tends to separate pool from atmosphere). A larger value tends to promote coupling between pool and atmosphere, while a smaller one tends to enhance their separation.

The default for horizontal flow paths is the inertial length defined on record FLnnn01. The default for vertical flow paths is the distance between the lowest point in the flow path (including the junction opening) and the highest point, as defined by the junction elevations on record FLnnn00 and the opening heights from record FLnnn01 (or default). See Section 1.3.2 of the Users' Guide and Section 5.5 of the CVH/FL Reference Manual for more information.

(1) XL2PF - Length for pool/atmosphere momentum exchange
(type = real, default as discussed above, units = m)

FLnnn06 – Flow path connection to EDF
1 ≤ nnn ≤ 999, nnn is the flow path number
Optional

This record directs that the cumulative flows of masses and enthalpies through the flow path be recorded to an external data file. The resulting file can then be used to define mass and energy sources to the CHV package in a subsequent calculation.

If a file is written, each record will contain NUMMAT cumulative masses, the cumulative flow of enthalpy associated with pool material, and the cumulative flow of enthalpy associated with atmosphere materials. Enthalpies are defined with respect to the normal MELCOR reference points.

(1) IP2EDF - User-assigned number of associated EDF FILE
(type = integer, default = none, units = dimensionless)

IP2EDF must be the number of a valid "PUSH" file containing exactly NUMMAT+2 channels. See the EDF Users' Guide for further input requirements, including file names and record frequencies.

FLnnn0F – Junction limits, *from* volume
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
Optional

This record defines the junction opening for the *from* volume by directly specifying the range of junction elevations for that volume. If this record is present, the value of FLHGTF from record FLnnn01, whether input or default, will not be used.

(1) ZBJFM - Elevation of bottom of junction opening for the *from* volume. Must lie between bottom of *from* volume and nominal junction elevation (inclusive).
(type = real, default = none, units = m)

(2) ZTJFM - Elevation of top of junction opening for the *from* volume. Must lie between nominal junction elevation and top of *from* volume (inclusive).
(type = real, default = none, units = m)

FLnnn0T – Junction limits, *to* volume
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
Optional

This record defines the junction opening for the *to* volume by directly specifying the range of junction elevations for that volume. If this record is present, the value of FLHGTT from record FLnnn01, whether input or default, will not be used.

(1) ZBJTO - Elevation of bottom of junction opening for the *to* volume. Must lie between bottom of *to* volume and nominal junction elevation (inclusive).
(type = real, default = none, units = m)

(2) ZTJTO - Elevation of top of junction opening for the *to* volume. Must lie between nominal junction elevation and top of *to* volume (inclusive).
(type = real, default = none, units = m)

FLnnnBk – Data for blockage of flow by another package

1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
0 ≤ k ≤ Z, k is a continuation character
Optional

Blockage of flow in response to change of geometry in another package (reduction of flow area, redefinition of friction) will be calculated if data are entered on this record. If a dataset *is* entered, no other control of the flow area is possible, and inclusion of a valve (FLnnnVk) record is not permitted. Only one blockage dataset may be entered for a flow path.

(1) PKG - Package defining the blockage
(type = character, default = none, units = dimensionless)

Specification of the defining package is included to allow later generalization, but at this time PKG = COR is the only available option.

Defining Package = COR

The COR flow blockage model can automatically adjust the area and flow resistance of specified flow paths to include the effects of blockage by core debris as modeled by the COR package. It can also model the opening of a flow area between the channel and bypass of a BWR when the separating channel box fails. The user must specify which core cells are to be associated with the flow path; the additional data required are defined below. Fields 2 – 4 are required, field 5 is optional.

(1) PKG - COR
(type = character, default = none, units = dimensionless)

(2) OPTION - Flow geometry to be modeled in this path.
= AXIAL Axial flow geometry. For a BWR, the channel and bypass regions are treated as combined, as in MELCOR 1.8.5
= AXIAL-C Axial flow geometry, considering only the channel region for a BWR.
= AXIAL-B Axial flow geometry, considering only the bypass region for a BWR.
= RADIAL Radial flow geometry. Not recommended for a BWR, because the channel and bypass regions are treated as combined (as in MELCOR 1.8.5).
= RADIAL-B Radial flow geometry, considering only the bypass region for a BWR.

= CHANNEL-BOX Connection between channel and bypass of a BWR that opens when the channel box fails.
(type = character, default = none, units = dimensionless)

(3) ICORC1
(4) ICORC2 - ICORC1 and ICORC2 define the limiting core ring and axial levels to be associated with the flow path by defining two "corner" core cells. Their order is not significant: 103 205, 203 105, 205 103, and 105 203 all specify levels 3 – 5 of rings 1 – 2. Each core cell in the range *must* be associated with either the *from* or the *to* control volume for the flow path as defined on the FLnnn00 record.
(type = character, default = none, units = dimensionless)

(5) FLMPTY - Form loss coefficient for "empty" geometry, to be added to the value from the Ergun correlation.
(type = real, default = 1.0, units = dimensionless)

FLnnnSk – Piping segment parameters
$1 \leq$ nnn $\leq$ 999, nnn is the user-assigned flow path number
$0 \leq$ k $\leq$ Z, k is a continuation parameter
Required

The user must specify several parameters for each segment; at least one segment is required. Some of the parameters are optional. Each record must contain all the data for a segment; the data for a segment cannot be split across records.

(1) SAREA - Segment flow area
(type = real, default = none, units = m^2)

(2) SLEN - Segment length
(type = real, default = none, units = m)

(3) SHYD - Segment hydraulic diameter. The conventional definition is given by 4 times the flow area divided by the wetted perimeter.
(type = real, default = none, units = m)

(4) SRGH - Surface roughness. Optional.
(type = real, default = 5.E-5, units = m)

(5) SLAM - Laminar flow coefficient. Optional. If negative, then control function number –SLAM is used to define the laminar flow coefficient.
(type = real, default = value of sensitivity coefficient C4404(13), whose default value is 16.0, units = dimensionless)

(6) ISFLT - This field is required if and only if an enhanced filter is defined on RN2FLTXXYY records with 21 ≤ YY ≤ 45, in which case ISFLT must be the user number, XX, of the enhanced filter modeled in this segment. See the RN User's Guide for more information.
(type = integer, default = 0, units = dimensionless)

FLnnnVk – Valve input

1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
0 ≤ k ≤ Z, k is a continuation character
Optional

Only one valve may be input for a given flow path. If more than one FLnnnVk record is included, only the one with the lowest value of k will be processed.

Valves may be used to open and close flow paths during the course of a calculation. The flow path area may take on any value between 0 (fully closed) and FLARA (fully open). The fraction open is defined by one or more control functions. It is defined directly by a single control function if no trip is used (although three fields must still be present on the record).

If a trip is specified, a trip control function is used to define "on-forward", "off" and "on-reverse" states. RELAP and TRAC users take note: A MELCOR trip is not what you are used to. The value of the trip control function may be positive ("on-forward"), negative ("on-reverse"), or zero ("off"). Different control functions are used to define the open fraction for "on-forward" and for "on-reverse" states. If the trip is "off", the valve opening remains unchanged from the previous timestep. See Section 1.5 of this document and the CF Users' Guide for more information.

(1) NVTRIP - If positive then NVTRIP is the trip control function number. If negative then a trip is not used and the fraction open is defined by the "on-forward" control function.
(type = integer, default = none, units = dimensionless)

(2) NVFONF - Control function used to defined the fraction open of the flow path for an "on-forward" state of the trip. If no trip is defined, this control function will always define the fraction open.
(type = integer, default = none, units = dimensionless)

(3) NVFONR - Control function used to define the fraction open of the flow path for an "on-reverse" state of the trip. If no trip is defined, this must be a valid control function number (i.e., one defined in MELGEN input), but the value of the control function will not be used.
(type = integer, default = none, units = dimensionless)

FLnnnPk – Pump input data

1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
0 ≤ k ≤ Z, k is a continuation character
Optional

Only one pump may be input for a given flow path. The first field on the first FLnnnPk record for a given flow path is interpreted as the pump type.

(1) PTYPE - Pump type
(type = character, default = none, units = dimensionless)

The additional data required to define a pump depend on the pump type. These data, which may comprise both real and integer values, are entered following the pump type and on continuation records if desired. They are interpreted according to the pump type flag. Because of the variable number and order of data, error checking is not exhaustive; the user is strongly advised to inspect the output of MELGEN in order to verify that the input was correctly interpreted.

PUMP TYPE = FANA

The FANA pump model represents a simple fan, in its normal operating range. It can, however, be used to approximate a constant-velocity coolant pump by appropriate choice of parameters. The input parameters are illustrated in the CVH/FL Reference Manual. Coupling to the momentum equation is explicit in time, and instabilities may arise for large timesteps. The problem may usually be mitigated by increasing the inertial length of the flow path, FLLEN, on record FLnnn01.

(1) PTYPE - FANA
(type = character, default = none, units = dimensionless)

(2) DPMAX - Maximum pressure head
(type = real, default = none, units = Pa)

(3) VPZERO - Volumetric flow rate at zero pressure head. The pressure head is zero for volumetric flow rates greater than or equal to VPZERO.
(type = real, default = none, units = m^3/s)

(4) VPMAX - Volumetric flow rate at maximum pressure head. The pressure head is also set to DPMAX for flows less than VPMAX.
(type = real, default = 0, units = m^3/s)

(5) ITRIP - Pump trip flag. If ITRIP is zero (default), the pump is always on; otherwise, the pump is on and the pressure head is calculated

only if the value of a trip is nonzero. If ITRIP is positive, the value of control function number ITRIP is used for the trip value. If ITRIP is negative, tabular function number –ITRIP is used for the trip value. The tabular function independent argument is time.
(type = integer, default = 0, units = dimensionless)

PUMP TYPE = QUICK-CF

The QUICK-CF pump type allows the user to define the pump head through a control function. This function might be as simple as a tabular function of velocity or of time. The complexity of the model is limited only by the ingenuity and patience of the user; an example of a (partial) homologous pump model is shown in the CF User's Guide. <u>Coupling to the momentum equation is explicit in time, and instabilities may arise for large timesteps. The problem may usually be mitigated by increasing the inertial length of the flow path, FLLEN, on record FLnnn01.</u>

(1) PTYPE - QUICK-CF
(type = character, default = none, units = dimensionless)

(2) IPUMCF - Number of control function defining the pressure head; the values from this control function should have units of Pascals.
(type = integer, default = none, units = dimensionless)

FLnnnTk – Time dependent flow path
1 ≤ nnn ≤ 999, nnn is the user-assigned flow path number
0 ≤ k ≤ Z, k is a continuation character
Optional

The velocity through the flow path may be defined using tabular or control functions. The pool and atmosphere velocities are identical. Only one time dependent flow path dataset may be entered for a flow path.

(1) NTFLAG - Time dependent flow path type flag
(type = integer, default = none, units = dimensionless)
= 1 Use tabular function number NFUN to define velocity versus time.
= 2 Use control function number NFUN to define velocity versus time.

(2) NFUN - Tabular or control function number to define the velocity versus time. The interpretation is dependent on the value of NTFLAG.
(type = integer, default = none, units = dimensionless)

FLnnnMk – Momentum flux input data

$1 \leq$ nnn $\leq$ 999, nnn is the user-assigned flow path number
$0 \leq$ k $\leq$ Z, k is a continuation character
Optional

Momentum flux will be calculated for flow path nnn if data are entered on this record. Only one dataset may be entered for a flow path.

(1) NFLFM - Upstream flow path number. If set to 0, an upstream flow velocity of zero will be used to calculate momentum flux.
(type = integer, default = none, units = dimensionless)

(2) NFLTO - Downstream flow path number. If set to 0, a downstream flow velocity of zero will be used to calculate momentum flux.
(type = integer, default = none, units = dimensionless)

(3) XCVAFM - Upstream (*from*) control volume flow area appropriate for flow path nnn (i.e., in the direction of flow)
(type = real, default = none, units = m^2)

(4) XCVATO - Downstream (*to*) control volume flow area appropriate for flow path nnn (i.e., in the direction of flow)
(type = real, default = none, units = m^2)

2.2 MELCOR User Input

Records FLnnn05, Flnnn0F, and FLnnn0T may be included in MELCOR input.

Record FLnnn05 defines the length over which momentum exchange occurs between pool and atmosphere flows sharing a flow path. The value used affects entrainment and flooding by controlling the balance between momentum exchange (which tends to couple the two flows and reduce slip) and buoyancy (which tends to separate pool from atmosphere).

Records FLnnn0F and FLnnn0T define the junction openings in the *from* and *to* volumes, respectively, i.e., the range of elevations within those volumes "seen" by the flowpath for purposes of allowing pool and/or atmosphere to flow. In some calculations, use of a restricted opening height may be necessary to prevent the flow of inappropriate phases during relatively quiescent segments of a calculation when gravitational separation should occur. On the other hand, extremely slow running of MELCOR might occur during relatively dynamic segments when entrainment would be expected (see discussion in Sections 1.1.2 and 1.3.1.) Also, the consequences of an initial choice may not become apparent until late in a long calculation. Therefore, MELCOR allows redefinition of the junction opening on restart of a calculation. Although the default definition in MELGEN of the momentum exchange length for a vertical flow path involves the junction opening heights, inclusion of these records in MELCOR does *not* lead to automatic redefinition of that length.

In addition to these limited modifications to the flow path models and database permitted in MELCOR input, several characteristics of control functions—including bounds on their values—may be modified on restart. This allows limited access to the definitions of pumps and valves. It may be used, for example, to select among a number of scenarios involving different failure paths, *provided that all required paths were provided in the original MELGEN input.*

3. Sensitivity Coefficients

Sensitivity coefficients for thermal hydraulic models are described in the CVH Package Users' Guide.

4. Plot Variables and Control Function Arguments

The flow path package variables that may be used for plot variables and control function arguments are listed and described below. Within slashes (/ /), a 'p' indicates a plot variable and a 'c' indicates a control function argument.

FL-EFLOW.x.n	/cp/	Enthalpy flow rate of pool (x = 'P') or atmosphere (x = 'A') flow through flow path n, with respect to the normal MELCOR reference point (units = W)
FL-FRUNBLK.n	/p/	Fraction of flow path that is unblocked by core debris. This variable is available only for flowpaths where the flow blockage model has been invoked by inclusion of a FLnnnBk record as part of MELGEN input; for flowpaths whose area is controlled by an ordinary valve, the open fraction may be plotted as the value of the related control function, CFVALU.m (units = dimensionless)
FL-I-EFLOW.x.n	/cp/	Integral of enthalpy associated with pool (x = 'P') or atmosphere (x = 'A') flow through flow path n, with respect to the normal MELCOR reference point (units = J)
FL-I-H2O-MFLOW.n	/p/	Integral of mass of water (pool + fog + vapor) flowing through flow path n (units = kg)

FL-I-MFLOW.m.n	/cp/	Integral of mass of material m flowing through flow path n (units = kg)
FL-MFLOW.n	/p/	Total mass flow rate (all hydrodynamic materials) through flow path n (units = kg/s)
FL-MFLOW.m.n	/cp/	Mass flow rate of material m through flow path n (units = kg/s)
FL-V-N-OC.n	/p/	Number of times valve number n has opened or closed. Any area fraction greater than 0.0 is considered "open." (units = dimensionless)
FL-VELLIQ.n	/cp/	Velocity of pool through flow path n (units = m/s)
FL-VELVAP.n	/cp/	Velocity of atmosphere through flow path n (units = m/s)
FL-VOID.n	/cp/	Void fraction in flow path n (units = dimensionless)

5. Example Input

This section gives several examples of input to the FL package. Anything followed by an asterisk is a comment.

In the first example, flow path 12 defines a door between control volumes 10 and 20; positive flow is from volume 10 to volume 20, as a simple opening 1 m wide and 2 m high. The elevation of the floor is 0 m. Both junction altitudes are taken at the center of the door (1 m); both junction opening heights are taken as 2 m to include the full height of the door. The inertial length is estimated as 1 m, representing a length of a few times the thickness of the wall; the segment length is an estimate of the wall thickness, and the hydraulic diameter is 4 times the area (2 m^2) divided by the wetted perimeter (6 m). Form loss is probably more important than wall friction; the loss coefficient is estimated to be 2.0 in both directions.

```
FL01200   'Door'     10   20   1.0  1.0  * Center alt = 1 m
*            A    L  Open H-fm H-to      * Junction openings
FL01201   2.0  1.0  1.0  2.0  2.0        * see full 2 m height
FL01202   3                              * Normal, horiz
FL01203   2.0  2.0            * Forward and reverse loss coeffs
FL012S1   2.0  0.2  1.33      * Segment L, A, and hyd diam
```

In the second example, flow path 125, represents the vertical connection of the core (control volume 120) to the upper plenum (control volume 150) in a BWR; the direction of positive flow is from core to upper plenum. The elevation of the plane separating them is

10 m. The form loss coefficients are 9.5 for forward flow and 10.5 for reverse flow. From the center of the core to the center of the upper plenum, fluid must travel 2 m (half the height of the core) through a flow area of 4 m^2 and a hydraulic diameter of 0.01 m, and 1.6 m through the more-open upper plenum with an area of 8 m^2 and a hydraulic diameter of 0.8 m. The nominal flow area is taken as 4 m^2. The inertial length was calculated from L/ 4 m^2 = 2 m / 4 m^2 + 1.6 m / 8 m^2, which yields L = 2.8 m; the precise value used is not critical.

```
FL12500   'Core to UP'   120  150  10.0 10.0 * CV120 to CV150
FL12501   8.0  2.8  1.0       * A, L, Open fraction
FL12502   0    0    0    0    * Vert, Active, no SPARC + or -
FL12503   9.5  10.5           * Forward, Reverse form losses
*          A    L   Dhyd
FL125S1   4.0  2.0  0.01      * Segment for half Core
FL125S2   8.0  1.6  0.8       * Segment for half UP
```

Record FL12502 specifies default values, and could be omitted. Because no junction opening heights were included on the FL12501 record, the radius of a circle with area 4 m^2 (1.13 m) will be used in each volume. This is a reasonable fraction of the height of each volume, making use of the default reasonable.

If desired, the height of the junction openings could be reduced by including the desired values on the FL12501 record. This could also be done by specifying the desired limits directly on FL1250F and FL1250T records; for example inclusion of the records

```
FL1250F   9.5  10.0      *'From' opening, 9.5-10.0 m
FL1250T   10.0 11.0      *'To' opening, 10.0-11.0 m
```

would explicitly define the junction openings to see the range of elevations from 9.5 to 10.0 m for forward flow out of volume 120 and 10.0 to 11.0 for reverse flow out of volume 150.

The default momentum exchange length for this flow path might limit countercurrent flow of pool from the upper plenum (volume 150) and atmosphere from the core (volume 120) sufficiently to result in "levitation" of water in the upper plenum under conditions of vigorous boiling in the core. Inclusion of a record such as

```
FL12505   0.2            * Momentum exchange length
```

would substantially increase the limits of countercurrent flow and reduce the levitation of water in the upper plenum.

Addition of the record

```
FL12504   0.0  4.0       * Initial atmos, pool velocity
```

will define an initial pool velocity of 4 m/s from core to upper plenum in the flow path.

Jet pumps in a BWR are frequently modeled as pipe-like flow paths. The partial input

```
FL15100   'Jet Pump'     110  100  9.0  4.0  * From DC to LP
FL15101   0.63 5.0  1.0            *A, L, Open fraction
. . .
FL151S1   0.63 5.0  0.2  5.E-6     *Dhyd for one of 20, smooth
```

represents 20 jet pumps in parallel, each of 0.2 m diameter, with a total area of 0.63 m^2, connecting the downcomer (control volume 110) at 9 m elevation to the lower plenum (control volume 100) at 4 m elevation. Surface roughness is 5 μm rather than the default of 50 μm. The volume of the pumps is usually included in the lower plenum.

Increased flow resistance and blockage of flow paths by core debris may be modeled by use of FLnnnBk records. For example, if flow path 112 connects the lower plenum to the channel region of the core of a BWR, use of the input

```
FL11200   'LP to CHNL'   110  120  4.0  4.0  * CV110 to CV120
. . .
FL112B0   COR  AXIAL-C   104  306  * Channel flow blockage by COR
```

will include the resistance of debris in the channel in rings 1 through 3 of axial levels 4 through 6 of the core in computing flows through flow path 112. The input will be rejected, and MELGEN will not generate an initial restart, unless the channel regions of all core cells involved lie in one of the control volumes (110 or 120) connected by the flow path.

Similar input allows modeling of the opening of a flow path between the channel and bypass regions of a core. If flow path 123 connects the channel and bypass regions, use of the input

```
FL12300   'CH to BP'     120  130  6.0  6.0  * CV120 to CV130
. . .
FL123B0   COR  CHANNEL-BOX    104  306  * Channel box failure
```

will base the open area and flow resistance of flow path 123 on the state of the channel box in rings 1 through 3 of axial levels 4 through 6 of the core. The input will be rejected, and MELGEN will not generate an initial restart, unless the channel and bypass regions are distinguished in all core cells involved. Further, the channel regions of all the cells must lie in one of the control volumes (120 or 130) connected by the flow path, and the bypass regions of all the cells in the other.

Finally, the partial input

```
FL19900   'Pipe'    123  456  20.0 -10.0
FL19901   0.002     30.0 1.0  * A, L, Open fraction
. . .
*         A    L    Dhyd
```

```
FL199S1   .002  0.5 0.05
FL199S3   0.05 29.5 0.25
```

represents a vertical pipe, 30 m long, connecting control volume 123 at 20 m elevation to control volume 456 at –10 m elevation. A short segment has an area of 0.002 m^2, corresponding to a diameter of 50 mm; the remainder has an area of 0.05 m^2, corresponding to a diameter of 0.25 m. Addition of the record

```
FL199V0   -1   20   20   * no trip, open fraction from CF 20
```

includes a simple (untripped) valve in the flow path. The fraction of its area that is open will be obtained from the value of control function 20; a value of 1.0 corresponds to opening to the full 0.002 m^2 nominal area.

If flow path 199 contains a check valve, restricting flow through the flow path to (say) the forward direction, it is preferable to model it as a "one way" flow path rather than by defining a valve. This reduces the number of control functions that must be written, and avoids the small reverse flows that can occur before the CF package can detect the reverse flow and close the valve. (This is a general consequence of the numerically explicit nature of all MELCOR control function models.) The required input record is

```
FL19902   10             * Vertical, one-way forward
```

In this case, no FL199V0 record is required (or permitted).

Alternatively, addition of the record

```
FL199P0   QUICK-CF  200  * Control function pump
```

includes a pump (momentum source) in the flow path. The pump boost in pressure will be obtained from the value of control function 200. Section 6.7 of the CF Package Users' Guide provides partial input for an appropriate control function to represent a homologous pump model similar to that in TRAC and RELAP.

6. FL Package Output

Each printed edit generated by the FL package in MELGEN and MELCOR contains a snapshot description of the state of all flow paths, with the output organized in tabular form. The integrated (cumulative) flows are included. Many column headings are abbreviated, but most are relatively clear—particularly when it is understood that "LIQ" refers to pool and "VAP" to atmosphere, that "ALPHA" is the fraction of the open area of a flow path occupied by *atmosphere*, and that "P HEAD" refers to *pumps*. For each valve, the number of times its state has changed from fully closed to at least partially open or back is recorded as "NUMBER ON OR OFF." (A complete cycle, closed to open to closed is counted as 2.)

The "FLOW PATH TIME INDEPENDENT EDIT" is generated by the FL package in MELGEN and as part of the first edit for each MELCOR run. This edit list properties of all flow paths, with the input flags, pump data, and valve data interpreted in English. The segment data associated with each flow path are listed, as are the flow paths connected to each control volume.

Users are strongly advised to check the initial edit generated by MELGEN—as well as the contents of the diagnostic file (MEGDIA)—before proceeding with a calculation. This is particularly important if FLnnnBk input records were included in the input to model flow blockage effects in the COR package. When this is done, some of the values input to FL will be redefined for consistency with the geometry of COR. Any such values changed will be flagged by an asterisk ("*") in the edit, with the notation

    * MARKS DATA MODIFIED BY BLOCKAGE OR CHANNEL BOX MODEL

7. Diagnostics and Error Messages

Diagnostic messages will be written by MELGEN to report errors or inconsistencies in input. Typical errors include errors in record format and failure to supply all required input records. Inconsistencies between input to different packages are also identified; each of the volumes connected by a flow path must be defined, and must include the specified elevation of the flow path junction; any tabular functions or control functions referred to by pump, valve, or time-dependent flow path input must be properly defined.

Errors may propagate, resulting in messages that are apparently unrelated to the actual input records. In these cases, it is often necessary to rerun MELGEN with the identified errors corrected; the other error messages should then be eliminated or clarified.

No restart file will be written until all errors identified during input processing have been corrected. This does *not*, of course, assure that the accepted input properly describes the physical system that the user intended to model. The analyst should always examine the initial edit produced by MELGEN before proceeding to run MELCOR. We have found that an incorrect definition of control logic for a pump and valve model will frequently not be discovered until the logic is exercised—typically after the investment of significant computer resources. In many cases, the entire calculation must be rerun. A short preliminary run to test the logic, perhaps with simplified input or artificially modified initial conditions is often a good investment.

All time advancement calculations are performed by the CVH package. Errors related to time advancement in MELCOR are described in Section 7.1 of the CVH Package Users' Guide.