Darby, R., "Size Safety-Relief Valves for Any Conditions", Chemical Engineering, 112, no. 9, pp 42-50, Sept, (2005)

Article *in* Chemical Engineering -New York- Mcgraw Hill Incorporated then Chemical Week Publishing Llc- · September 2005

1 author:

Ron Darby
Texas A&M University

Published in Chemical Enginering, **112**, no. 9, pp 42-50, Sept, (2005)

PROPERLY SIZE SAFETY RELIEF SYSTEMS FOR ANY CONDITIONS

Ron Darby
Professor Emeritus
Department of Chemical Engineering
Texas A&M University
College Station, TX 77843-3122
(979) 846-4175 Fax: (979) 846-4175
r-darby@tamu.edu

Overview

Safety relief valves and rupture disks are typically used to protect equipment from excessive overpressure. Typical scenarios that can result in such overpressure in excess of the vessel MAWP (maximum allowable working pressure) include external fire, blocked outlet line, power failure, loss of cooling water or steam, thermal expansion, excess inlet flow, accumulation of non-condensables, failure of check or control valve, exchanger tube rupture, runaway reaction, human error (e.g. open or close the wrong valve), etc. These and other scenarios are discussed in more detail by Wong [1].

Reliefs should be installed on all vessels other than steam generators, including reactors, storage tanks, towers, drums, etc. Other locations where reliefs are required are blocked in sections of liquid filled lines exposed to external heating, the discharge from positive displacement pumps, compressors and turbines, and vessel steam jackets. Storage vessels containing volatile liquids and a vapor space should be protected from both excessive pressures from external heat or flow input but also from the possibility of a vacuum from condensation of the vapor.

Relief valves are designed to open at a preset pressure and are sized to allow mass flow out of the vessel at a rate sufficient to remove excess energy from the vessel at least as fast as it is input to the vessel contents (from either external or internal sources, e.g. external heating, runaway reaction, etc.) to prevent further pressure build up. The valve will close when the pressure drops to a safe level, thus containing and protecting the bulk of the vessel contents. Since the capacity of a valve is limited, it cannot accommodate the extreme flow rate that might be required to protect against an extremely high energy input rate such as might result from a very energetic runaway reaction, a deflagration or explosion. Rupture disks are a less expensive alternative, especially for very large capacity requirements, but of course do not reclose to contain the vessel contents.

Proper design of a relief system requires not only determining the correct size for the valve or rupture disk, but also the proper size and selection of upstream and downstream piping and effluent handling systems. The procedure for the design for a safety relief system can vary from a relative simple, fairly routine, process for single phase (gas or liquid) flow to a complex procedure for two-phase flow requiring considerable expertise and procedures that depend on conditions and the nature and characteristics of the fluid being discharged. The details of this total process are beyond the scope of this article, and authoritative references should be consulted for further details and procedures (e.g. API [*2, 3*], CCPS [*4*], DIERS [*5*]). This article will summarize the overall considerations important in the design process, and will concentrate on the basic procedure for properly sizing the relief (either a SRV or rupture disk) under a variety of conditions for single and two-phase flows.

Required Relief Rate

The first step in the design process for valve sizing is to postulate one or more credible scenarios that could result in unacceptable overpressure, and determine the corresponding required discharge mass flow rate $\left(\dot{m}\right)$ that would be sufficient to prevent overpressure in excess of the vessel MAWP. The required relief mass flow rate $(\dot{m})$ is determined by an energy and mass balance on the vessel under the conditions of the specific postulated relief scenario, e.g. a runaway reaction, external fire, loss of cooling, blocked line, etc. The value of $\dot{m}$ is determined by the requirement that the rate of energy discharge from the vessel be equal to or greater than the maximum rate at which excess energy is input into the vessel under the assumed scenario. It is normal to postulate a "worst case" scenario and a "most credible" scenario, and base the design on the worst of the likely cases. This is, of course, a judgment call coupled with the probability of the scenario occurring.

For a runaway reaction involving volatile or gaseous components, data from an adiabatic calorimeter or detailed kinetic information are required to predict the required relief rate. Specialized techniques and/or equipment are needed for this, and the process should be left to the experts (see e.g. DIERS [*5*], CCPS [*4*], Darby [*6*]).

For storage vessels containing a volatile liquid, a common scenario is an external fire which heats the vessel and contents, resulting in superheating the liquid. If the vapor pressure builds up to a point which exceeds the vessel MAWP, the vessel could rupture resulting in a BLEVE (Boiling Liquid Expanding Vapor Explosion). The relief mass flow rate must be high enough that the rate of discharge of the total sensible and latent heat through the vent must equal or exceed the rate of heat energy transferred to the fluid through the vessel wall from the fire exposure. Since the liquid will typically be superheated, flashing will occur as the pressure drops through the vent; resulting in two-phase flow in the relief which must be accounted for in sizing the relief, as described below (the relief area required for two-phase flow is normally significantly larger than that which would be required for single phase flow). Methods for estimating the heat transfer rate from a fire to storage vessels are presented by NFPA [*7*] and API

[*8*]. For conditions not adequately covered by these documents, Das [*9*] has presented fundamental relations for determining the heat transfer rate.

Valve Sizing

The required orifice area[1] for a relief valve or rupture disk is determined from the formula

$$A = \frac{\dot{m}}{K_d G_o} \tag{1}$$

Here $\dot{m}$ is the required relief mass flow rate (mass/time) and G_o is the theoretical mass flux (mass/time area), calculated for flow through an ideal (isentropic) nozzle. The expression for G_o follows directly from application of the general steady state energy balance (Bernoulli) equation to the fluid (gas, liquid or two-phase) in the nozzle (see e.g. Darby [*10*]):

$$G_o = \rho_n \left(-2 \int_{P_1}^{P_n} \frac{dP}{\rho} \right)^{1/2} \tag{2}$$

where P_1 is the pressure at the entrance to the valve, P_n is the pressure at the nozzle exit, ρ is the fluid (or mixture) density at pressure P, and ρ_n is the fluid density at pressure P_n, the nozzle exit or throat.

K_d in Eqn (1) is the (dimensionless) discharge coefficient that accounts for the difference between the predicted ideal nozzle mass flux and the actual mass flux in the valve. This is determined by the valve manufacturer from measurements using (typically) single-phase air or water flows. Further assumptions must be made to determine the appropriate value of K_d to use for two-phase flow (this is discussed later).

1 Although the term "orifice" is commonly used to describe the minimum flow area constriction in the valve, the geometry more commonly resembles a nozzle and the area is determined by applying the equation for flow in an isentropic nozzle, as described in this article.

There are a finite number of standard valve nozzle (orifice) sizes to choose from and the calculated area A would not be expected to correspond exactly to one of these sizes. In practice, a 10% "safety factor" is automatically applied to the calculated area (per the ASME code), and then the standard size nozzle orifice area which is the closest to the resulting value on the high side is then selected.

It is important that the relief area be neither too large nor too small. An undersized vent would obviously not provide the required overpressure protection, whereas an oversized vent will result in excessive flow which can adversely affect the opening and closing characteristics of the relief valve resulting in impaired performance (e.g. unstable operation, or chatter) with possible severe damage to the valve. If the valve is oversized, the actual flow rate will be significantly greater than the required design rate $(\dot{m})$ so that if the associated piping is sized for the design rate it will be undersized for the actual rate. This means the pressure drops through the entrance and exit piping will be greater than expected, and these pressure drops can have serious adverse effects on the stability of the valve (see the section on Inlet and Discharge Piping).

Although the flow through a relief valve is an unsteady (time-dependent) process, it is customary to base the calculations on assumed steady state conditions corresponding to the expected flow rate at a pressure which is 110% of the relief set pressure (i.e. 10% overpressure). The relief set pressure is normally the vessel MAWP, although other relief pressures are allowed by the ASME code for various special cases (e.g. API [*2*]).

Nozzle models

The term "model" as applied to valve sizing is frequently misunderstood. For example, the commonly referenced "Homogeneous Equilibrium Model (HEM)" is not a "complete model" for calculating the nozzle mass flux, but simply a set of conditions and assumptions which constrain the calculations. The HEM implies that if the fluid through the valve is a two-phase gas-liquid mixture, it will be sufficiently well mixed that it can be described as a single-phase fluid with properties that are a suitable combination of those each fluid, and that the two phases are in both mechanical and thermodynamic equilibrium. These assumptions are necessary, but not sufficient, for calculating the nozzle mass flux because additional assumptions or conditions must be specified with regard to the properties of the fluid which are necessary to determine the mixture density as a function of pressure. It is evident from Eqn (2) that the calculated nozzle mass flux is determined specifically by the manner in which the fluid density depends on pressure over the range of pressures in the nozzle. The "homogeneous equilibrium assumption" is inherent in the derivation of Eqn (2), but the specific relation to be used for the $\rho(P)$ function, and the manner in which the integral is evaluated using this function, must also be specified for the "model" to be complete.

Single-Phase Liquid Flow **-** For single-phase liquid flow, the nozzle mass flux integral (Eqn 2) is simple to evaluate since the fluid density is assumed independent of pressure. Thus, for liquids with a constant density, Eqn (2) reduces to

$$G_o = \sqrt{2\rho\left(P_o - P_n\right)} \tag{3}$$

This equation is valid for fully turbulent flow (e.g Reynolds numbers above about 100,000), for which the flow rate is independent of the fluid viscosity. For low Reynolds number (e.g. high viscosity) flows, the value given by Eqn (3) can be multiplied by a correction factor $\left(K_v\right)$ that reflects the dependence of G_o on Reynolds number as well as on the d/D = β ratio of the nozzle (Darby and Molavi [*11*]):

$$K_v = 0.975\sqrt{\frac{\beta^{0.1}}{\left[950(1-\beta)^{1.4}/N_{Re}\right]+0.9}} \tag{4}$$

where N_{Re} is the Reynolds number through the nozzle. For a two-phase mixture, a volumetric average values of density and viscosity is used. Here, $\beta = d/D$, the ratio of the nozzle diameter to valve inlet diameter. These equations also assume that the liquid is Newtonian. There are no known data for non-Newtonian flow in relief valves and there are no current models that account for such properties. However, in the absence of more specific information, it may be assumed that Eqn (4) can be applied to non-Newtonian viscous fluids if the Reynolds number is modified accordingly for the specific non-Newtonian rheological model (see e.g. Darby [*10*], Ch 7).

Single-phase gas flow – In the case of an ideal gas, the integral can be readily evaluated assuming isentropic flow for which P/ρ^k = constant, where k is the isentropic exponent (which for an ideal gas is the ratio of the specific heat at constant pressure to that at constant volume). However, the result depends upon whether or not the nozzle exit pressure (P_n) is at or below the value at which the speed of sound is reached in the nozzle (i.e. choked flow). The criterion for choked flow is $P_n \le P_c$, where $P_c = P_o\left[2/(k+1)\right]^{k/(k-1)}$. If the flow is choked the mass flux is given by

$$G_o = \sqrt{kP_o\rho_o}\left(\frac{2}{k+1}\right)^{(k+1)/2(k-1)} \tag{5}$$

which is independent of the downstream pressure. If the flow is not choked (i.e. sub-critical), $P_n > P_c$ and the mass flux depends on both the upstream and downstream pressures as follows:

$$G_o = \sqrt{\frac{2P_o\rho_o k}{k-1}\left[\left(\frac{P_n}{P_o}\right)^{2/k} - \left(\frac{P_n}{P_o}\right)^{(k+1)/k}\right]^{1/2}} \tag{6}$$

If $(P_o / P_n) \geq 2$ (approximately), the flow will probably be choked and Eqn (5) applies. Non-ideal gases can be treated using Eqn (2) along with actual property data or an appropriate equation of state to evaluate the gas density. Alternately, the above equations can be used if a "non-ideal k value" is used, and the density is divided by an appropriate value of the compressibility factor (z) evaluated at the choke conditions (see Shakleford [*12*] for a discussion of the suitability of using ideal vs non-ideal gas k values).

Two-phase flow

Thousands of relief valves in process plants are installed in vessels that operate under conditions that can result in two-phase flow through the relief valve, and the valve must be properly sized to accommodate such flows. Various conditions could result in flashing, condensing, or "frozen" (non-flashing) flow. Flashing flow occurs in nozzles/valves whenever the entering fluid is a saturated liquid, a sub-cooled liquid that reaches the saturation pressure within the nozzle, or a two-phase vapor-liquid mixture. Frozen two-phase flow may occur if the vessel initially contains both gas and a non-volatile liquid (e.g. a vessel with inert gas padding). Either frozen or flashing flow could result from a runaway reaction, for example. Retrograde condensation may also occur when the fluid in the vessel is a dense gas that condenses when the pressure drops.

Two-phase flow is considerably more complex than single-phase flow, and there are a number of additional factors that must be considered such as the flow regime, the nature of the interaction between the phases, the method of determining the properties of the two-phase mixture, and the method of incorporating these properties into evaluation of the mass flux.

If a vessel initially contains both liquid and gas or vapor, or a superheated liquid, the mass fraction of gas (i.e. the quality) in the two-phase mixture entering the relief device will depend upon the amount of gas/vapor generated within the liquid phase, the degree of mixing in this phase, the bubble rise velocity, the physical properties of the liquid, and the initial void fraction (i.e. the vapor space) in the vessel. The prediction of this initial quality can be a complex procedure, and the pertinent references should be consulted (e.g. CCPS [*4*].

Flow Regime: This refers to the distribution of the two phases in the flow field, which can be classified as distributed (such as stratified, wavy, slug, or bubbly) or homogeneous (i.e. well mixed). Because of the high velocities and high degree of turbulence in typical relief flows, the usual assumption is that the flow is well mixed and hence homogeneous within the relief device. This means that the two-phase mixture can be represented as a "pseudo single-phase" fluid, with properties that are a suitable average of the individual fluid properties. There are many ways that this average can be defined, but the most widely accepted is a volume-weighted average. On this basis, the density of the two-phase mixture is given by

$$\rho = \alpha\rho_G + (1-\alpha)\rho_L \tag{7}$$

where α is the volume fraction of the gas phase, given by

$$\alpha = \frac{x}{x + S(1-x)\rho_G / \rho_L} \tag{8}$$

Here x is the quality (i.e. mass fraction of the gas phase) and S is the slip ratio, i.e. the ratio of the gas velocity to the liquid velocity in the mixture (see below).

Mechanical Equilibrium: This implies that the two phases are flowing at the same velocity, i.e. no slip (S = 1). Slip occurs because the gas phase expands as the pressure drops and hence must speed up relative to the liquid phase. Slip becomes more important as the pressure gradient increases, and is most pronounced as the velocity approaches the speed of sound (choking). Although there are a variety of "models" in the literature for estimating slip as a function of fluid properties and flow conditions, it is often neglected under pressure relief conditions because of the high degree of turbulence and mixing. For flashing flows, slip effects are normally negligible, since the volumetric expansion due to flashing will overwhelm the expansion of the gas phase due to pressure drop alone. However, slip can be significant for frozen flows (e.g. air and cold water). For example, Jamerson and Fisher [*13*] and Darby et al. [*14*] found that a slip ratio (S) of 1.1 to 1.5 is consistent with various frozen flow data in nozzles. Most frozen flow data in the literature are for air-cold water mixtures, and there are little or no data for industrial fluids. Note that Eqn (8) shows that an increase in S results in a larger two-phase density and corresponding higher mass flux than would be predicted with no slip. Some models for the nozzle mass flux include provision for slip and some do not, as described later.

Thermodynamic Phase Equilibrium: It is commonly assumed that the gas or vapor phase is in local thermodynamic equilibrium with the liquid phase, which means that the properties of the mixture are a function only of the local temperature, pressure and composition. In other words, when the pressure in a liquid drops to the saturation (vapor) pressure, vaporization (flashing) will occur instantly if the system is in equilibrium. However, flashing is actually a rate process that takes a finite time (e.g. a few milliseconds) to develop fully. During this "relaxation time" a liquid can travel several inches (i.e. a corresponding "relaxation distance") in the nozzle of a valve under typical relief conditions. Under these conditions the amount of vapor generated (e.g. the quality) is much smaller than would occur under equilibrium conditions, and the mixture density and mass flux are correspondingly larger. Experimental data on a number of single component systems (e.g. Henry and Fauske [*15*]) have indicated that this "relaxation length" is of the order of 10 cm for typical relieving conditions, which means that flashing flow in nozzles shorter than 10 cm should be in non-equilibrium. Some nozzle flow models have provision for non-equilibrium effects and some do not, as discussed later.

***Thermodynamic Path*:** As the fluid flows through the nozzle, the pressure and temperature drop and the volume fraction of gas/vapor increases. For frozen flows, the mass flow rate of each phase remains constant throughout the flow path, although the phase volume fractions change because the gas expands. For a volatile liquid, the quality (i.e. the mass fraction of gas) will also change from point to point because of increasing evaporation as the pressure drops, and it is necessary to determine the local quality as a function of pressure in order to calculate the two-phase mixture density from Eqn (7). This is done by assuming the fluid follows a specific thermodynamic path as it traverses the nozzle, which may be isothermal, isentropic, or isenthalpic, and then determining the gas and liquid densities and the quality (or phase ratio) along this path. The usual assumption is that this path is isentropic, since the "isentropic nozzle equation" is used as the basis for the mass flux. On the other hand, a case can be made for assuming that the flow in the nozzle is isenthalpic and using an "enthalpy balance" to determine the local properties. In some cases (e.g. liquid flow), the isentropic, isenthalpic and isothermal paths are virtually identical. For example if the inlet conditions are subcooled or saturated, and are sufficiently far from the critical point, there is usually a negligible difference between the isentropic and isenthalpic paths. However, as the critical point is approached, or for low vapor/liquid ratios (low quality), the difference is more pronounced. Particularly in the vicinity of the thermodynamic critical point the differences may be quite significant. There are no definitive studies to show which assumption is the most appropriate, but the general consensus favors the isentropic path (which is inherent in the isentropic nozzle equation).

***Physical Property Data*:** In order to calculate the two-phase density (and other properties) along the chosen path (i.e. isentropic), a database of thermo-physical properties of the fluids is required. The specific properties and the amount of data required depend on the particular model used, but at a minimum the mass fraction of the gas phase (quality) and the densities of each fluid phase are required as a function of pressure along the path (e.g. for the HDI model). For frozen flows, the liquid density is constant so the only property data required is a suitable equation of state for the gas (e.g. the ideal gas law), or appropriate data for the gas. Some models require enthalpies, entropies, densities, heats of vaporization and specific heats at one or more conditions. For flashing pure components, the required data are usually available in a thermo-physical property database or simulator. The Omega and HNE models require thermo-physical properties at only one state (e.g. the stagnation state), and employ an entropy or enthalpy balance to determine the vapor fraction (quality) of the two-phase flashing mixture. The API version of the Omega method for mixtures requires thermo-physical properties at two-states instead of evaluation of the Omega parameter at one state. For multi-component mixtures, additional property data or mixture models must be available and can be used with a flash routine to determine the vapor-liquid equilibrium properties (e.g. density and quality) of the two-phase multi-component mixture as a function of pressure.

Model assumptions – The assumptions made with regard to the above considerations constitute the "model" for the nozzle mass flux. The most common assumption is the HEM (Homogeneous Equilibrium Model), which implies that the two-phase mixture is homogeneous and the phases are in equilibrium (both mechanical and thermodynamic). Several versions of the HEM are in use, which differ in the specific assumptions and methods used to evaluate the two-phase density and the mass flux integral (Eqn 2). Some of these variations are described below.

The Omega method - This method (Leung [*16, 17*]) was derived for a single component fluid, and assumes that the density of the two-phase mixture can be represented by a linearized equation of state. It requires fluid properties at only one state (the saturation or stagnation state).

Factors that should be considered when using the Omega method are:

- The equations are based on an analytical evaluation of the mass flux integral, using an approximate linearized two-phase equation of state for the fluid density. The equations are fairly complex, so care is required to insure the calculations are correct.
- Fluid property data are required at only one state, simplifying the required amount of input property data. However, these thermodynamic and physical property data must be accurate, since small variations or errors in the thermodynamic properties can have a large effect on the resulting density values.
- The linearized equation of state may not give accurate two-phase density values vs pressure for some conditions since it extrapolates the two-phase density from the relief (stagnation pressure) state. The accuracy depends not only on the nozzle conditions and the nature of the fluid but also the range of pressures in the nozzle.
- The method tends to be unreliable in the vicinity of the critical point or for dense gases that condense when the pressure is reduced (retrograde condensation).
- It was derived for single component fluids and is not easily adapted to multi-component mixtures unless modified (see the API Method below) or unless the boiling range of the mixture is small. Consequently, it is inappropriate for mixtures with light gas components (e.g. hydrogen).
- Neither slip nor non-equilibrium effects are accounted for in the model.
- A special version of the basic model is required for slightly subcooled liquids.

***API method*:** The method presently recommended by API 520 [*2*] is the Omega method for single component fluids and multi-component mixtures with a normal boiling range less than 150^{o}F. The heat of vaporization is calculated as the difference between the vapor and liquid specific enthalpies of the mixture. For flashing mixtures with a normal boiling range greater than 150^{o}F, the ω parameter is determined from the calculated two-phase density of the mixture at two pressures (P_o and $P_9 = 0.9P_o$) and constant entropy. Factors to be considered when applying this method include:

- It is basically a two-point linear fit of the two-phase density at pressures P_o and P_9. This is better than the one-point Omega extrapolation but still may not give accurate results depending on the fluid, the conditions, and the pressure range involved (particularly near the critical point).

- The choke pressure is estimated using the "single point ω method", which could introduce some error or uncertainty.
- Since the two-phase density is calculated from a fluid property database at two points using the single component thermodynamic properties, it can be used for multi-component mixtures if an appropriate property database is available.
- A reliable property database must be used to determine the two-phase density and quality (x) at two separate pressures at constant entropy. For multi-component systems, this can be done using a flash routine coupled with an appropriate fluid database in a simulator. Accurate thermo-physical property (density) data are required since small variations or errors in the thermodynamic properties can have a large effect on the resulting density values
- Non-equilibrium effects (either thermodynamic or mechanical) are not included.

TPHEM – This model (the *Two-Phase Homogeneous Model*) is implemented using a computer routine that is available on a CD that accompanies the CCPS Guidelines book "Pressure Relief and Effluent Handling Systems" [*4*]. The mass flux integral (Eqn 2) is evaluated numerically by the program using input data for the densities of the liquid and gas/vapor and the mixture quality at two or three states at constant entropy from the stagnation pressure to the discharge pressure. The density data are fitted in the program by an empirical equation, which is used to interpolate the densities at intermediate pressures for evaluation of the integral. The user can choose from a variety of empirical equations for fitting the two-phase P, ρ data, with one, two or three parameters (Simpson [*18, 19*]).

The densities of the gas and the liquid and the quality (x) of the mixture at each of the two or three pressures along an isentropic path are input into the program. The single parameter density model is equivalent to the Omega method. The 2-parameter model is equivalent to the API method, with $P_2 = 0.9P_o$. For flashing of an initially sub-cooled liquid, the three pressures are the saturation pressure, the nozzle exit pressure, and one intermediate pressure. It is necessary to have an accurate property database for the fluids in order to determine the required input density data. The program output is the mass flux at the specified exit pressure (or vice versa). A variety of other output options are also available including viscous or non-viscous flow, pressure drop in straight pipe with or without fittings, etc. The choke pressure and corresponding mass flux are determined by initially specifying the stagnation pressure as the backpressure and then decreasing this pressure in increments until the mass flux reaches a maximum.

Some key characteristics of this method are:

- It is applicable to frozen or flashing flows, as well as subcooled or saturated liquids.
- The program does all of the calculations automatically, so it is quick and easy to implement.
- Two or three (P, ρ, x) data points are required along an isentropic path. Using more than one data point can improve the property estimates considerably over that of the Omega method in many cases. Accurate thermo-physical property (density) data are required since small variations or errors in the thermodynamic properties can have a large effect on the resulting density values.
- A wide variety of conditions, including pipe flow or nozzle flow for inviscid or viscous fluids can be run using various combinations of "switches" in the program, for calculating either the mass flux or the exit pressure.
- It has the capability of including a slip parameter or a non-equilibrium parameter, but there are no guidelines for selecting the values of these parameters.
- Multi-component systems can be handled using a flash routine to generate the required (P, ρ, x) data points if a suitable property database is available.
- Multiple runs are required in order to determine the choke pressure and maximum (choked) mass flux.
- The multiple combinations of program "switches" and options required to run the various cases can sometimes be confusing and requires care to ensure proper implementation.
- The results can be sensitive to the choice of conditions for the input data and the range of pressures required, especially near the critical point.
- Because of the density-pressure fitting equation, the choke point may not be accurately predicted.

HNE mode: This model (the *Homogeneous Non-Equilibrium* model) is based on an energy balance on a flashing liquid. It is an extension of the *Equilibrium Rate Model* (ERM), which employs an isenthalpic energy balance on a saturated liquid to determine the fraction that is flashed. The rapid generation of vapor from the flash is assumed to result in choked flow and the mass flux is evaluated from the definition of the speed of sound using the Clausius-Clapeyron equation to relate the vapor density to the vapor pressure of the flashing fluid and the thermodynamic properties (e.g. heat of vaporization, etc). The mass flux predicted by this model for a saturated flashing liquid is typically about 10% higher than corresponding values predicted by the HEM. However, for slightly sub-cooled liquids it has been observed that the actual mass flux may be as much as 300% greater than predicted by either model.

This model was extended by Henry and Fauske [*15*], and later by Fauske [*20*], to account for non-equilibrium effects resulting from delayed flashing by the rate processes involved. The model determines the gas mass flux and liquid mass flux separately using the respective single phase discharge coefficients K_{dG} and K_{dL}, and combines these in proportion to the respective phase mass fractions. Non-equilibrium is characterized by a delayed flashing parameter which is a function of the "relaxation length", $L_e = 10\ \text{cm}$. Non-equilibrium conditions were found to occur when $L < L_e$, and equilibrium occurs if $L > L_e$. Factors which should be considered when using the HNE model include:

- It is applicable to single-phase (liquid or gas), subcooled or saturated liquid, or two-phase mixtures. It is applicable to flashing but not condensing flow conditions.
- It predicts effects of non-equilibrium conditions (e.g. in short nozzles).
- It requires property data only at the saturation state. This minimizes the amount of input data required, but may result in lower accuracy relative to those methods that utilize data at more conditions. Accurate thermo-physical property data are required since small variations or errors in the thermodynamic properties can have a large effect on the resulting density values
- The calculations are simple and easy to perform
- The choke pressure is assumed to be the saturation pressure, but this is not always appropriate especially for low relief pressures and low subcooling. Better results may sometimes be obtained if the actual choke pressure is used instead of P_b in the model

equations, but this pressure has to be determined using another method (such as TPHEM or HDI).

- The assumption of an ideal gas phase is made for the gas phase, which can introduce errors particularly in the vicinity of the critical point.
- The model does not include any provision for slip.
- The relaxation flow length (10 cm) is based on a relatively small number of observations.

The HDI method: This method (the *Homogeneous Direct Integration* method, Darby et.al [*21*], Darby, [*22*]) involves generating multiple (P, ρ, x) data points over an isentropic range of pressures from P_o to P_n using a thermodynamic property database for a pure fluid, and a flash routine for a multi-component mixture. These data are used to evaluate the mass flux integral, Eqn (2), by direct numerical integration. This can be done easily on a spreadsheet by a simple trapezoidal rule, or a simply quadrature formula, as follows:

$$G_o = \rho_n \left(-2\int_{P_o}^{P_n} \frac{dP}{\rho} \right)^{1/2} \cong \rho_n \left[-2\sum_{P_o}^{P_n} \left(\frac{P_{i+1} - P_i}{\bar{\rho}_i} \right) \right]^{1/2} \tag{9}$$

Pressure increments of 1 psi are usually quite adequate to provide sufficiently accurate results. The choke point is determined by repeating the calculations at successively lower values of P_n, starting at P_o, until the mass flux reaches a maximum. If no maximum is reached before $P_n = P_b$ the flow is not choked. The method is perfectly general, and applies to any fluid, under any conditions (single-phase gas or liquid, or two-phase) for which property data are available.

This method can be extended to account for non-equilibrium effects for flashing flow in short nozzles (L < 10 cm) (i.e the HNDI or *Homogeneous Non-Equilibrium Direct Integration* model), as follows. The effect of non-equilibrium is to delay the development of flashing to a pressure below the normal equilibrium saturation pressure. That is, when the pressure reaches the saturation pressure the flashing process is not completely developed so that the quality (x) is actually lower than it would be under equilibrium flashing conditions $(\text{i.e. } x_e)$. Since the equilibrium two-phase density is related to the quality by

$$\frac{1}{\rho} = \frac{x}{\rho_G} + \frac{(1-x)}{\rho_L} \tag{26}$$

the density (and hence the mass flux) would be higher under non-equilibrium conditions than at equilibrium. Thus the effect of non-equilibrium can be accounted for by appropriately modifying the value of the quality, x. As indicated from the HNE model, observations have shown that for typical flashing flows in nozzles equilibrium is reached at a distance of about 10 cm along the nozzle, with non-equilibrium conditions prevailing for L < 10 cm. Thus, if we assume that x approaches the equilibrium quality x_e as L approaches 10 cm. Thus for $L \leq 10$ cm the effective quality at the nozzle throat can be estimated as

$$x = x_o + (x_e - x_o)\frac{L}{10} \tag{27}$$

where L is the nozzle length in cm, x_o is the initial quality of the fluid entering the relief device. For L > 10 cm, $x = x_e$. Considerations appropriate to this method include:

- The method is rigorous within the assumptions inherent in the ideal nozzle equation and the HEM assumptions, and the precision of the property data.
- It is universally applicable for all fluids under any/all conditions for which the property data are available.
- The procedure does not depend on whether the entering fluid is cold liquid, sub-cooled flashing liquid, a condensing vapor, or a two-phase mixture, or on whether or not the flow is choked.
- It is simple to understand and apply.
- It is easily applicable to multi-component systems, provided the mixture property data are available for performing the required flash calculations over the pressure range of interest. A process simulator using the property database can usually generate the required data.
- The calculation method is simple and direct, and is ideally suited to a spreadsheet solution.
- The method is more accurate than those above because no "model approximation" for the fluid properties is involved.

- The method can easily be applied to short (non-equilibrium) as well as long (equilibrium) nozzles
- Accurate thermodynamic and physical property data, $\rho(P)$, are required to give good results.
- A flash routine must be used for multicomponent mixtures to generate the (P, ρ, x) data required for the integration, and more data points must be computed.
- Slip effects can be readily incorporated into the method via Eqn (8) provided an appropriate value for the slip ratio (S) is known or can be predicted.

The discharge coefficient

The discharge coefficient (K_d) in Eqn (1) corrects for the difference between the flow predicted by the ideal isentropic nozzle model and that in an actual valve. Thus the values of K_d depend upon how accurately the theoretical isentropic nozzle "model" represents the real valve flow rate. Thus, the value of K_d depends upon both the nature (geometry) of the valve as well as the accuracy of the fluid property "model". Values of the gas phase coefficient K_{dG} are always closer to unity (i.e. a perfect model) than the liquid phase coefficient K_{dL} values. This is because the gas flow coefficients are measured under choked flow conditions, for which the isentropic ideal gas model is a much better representation of the actual flow. Conditions for liquid flow coefficients are obviously determined under non-choked flow conditions, for which the entire valve (not just the nozzle) influences the flow rate and therefore the "isentropic nozzle model" is much less adequate. Values of K_d for valves and rupture disks are determined by the manufacturer in a certified calibrated test facility using water or air (sometimes steam), and are updated annually in the "Red Book"[2]. The "Red Book"[2] value or "ASME K_d" is based on the actual area and should be used if the ASME relief valve orifice size (actual area) is used. Values of the single-phase

[2] Pressure Relief Device Certifications, National Board of Boiler Inspectors, (http://www.nationalboard.org/Redbook/redbook.html)

K_d **'s** are also given in API Standard 526 "Flanged Steel Pressure Relief Valves" [*23*] which are based on "standardized" nozzle (orifice) areas, as opposed to the actual area. Specifying the API "standardized" nozzle sizes (with the corresponding values of K_d) provides a uniform method for sizing valves independent of the specific vendor or valve dimensions. The K_d values published by vendors for use with the API standard orifice sizes should only be used with these sizes (API, [*2*]). In general, the API K_d's are about 10% higher than the ASME (Red Book) K_d values, and the API standard areas correspondingly smaller. The API values for spring-loaded relief valves are approximately 2% higher than the ASME valves. (the product $K_d A$ is approximately the same for either the ASME or the API values). Use the API K_d when API standard size relief orifice sizes are specified and the ASME K_d when the actual nozzle sizes are used.

For two-phase flow there are no validated databases or certified test facilities, so experimental values of K_d **a**re not available. The few suggestions available in the literature are based on a limited number of experimental observations. Some investigators suggest various averaging methods for the two-phase K_d, such as a volume-weighted average of the liquid and gas phase coefficients based on the relative volumes of liquid and gas. However, data on frozen air/water flows in various relief valves (Darby, [*22*]) indicate that when a rigorous mehtod like the HDI or HNDI is used, a value of $K_d = K_{dG}$ is appropriate when the flow is choked, and $K_d = K_{dL}$ if the flow is not choked. This is quite logical, because measured K_{dG} values are representative of choked flow conditions (for which the mass flux is independent of conditions downstream of the nozzle) and measured K_{dL} values are representative of non-choked conditions (where the mass flux is affected by the flow resistance in the body of the valve as well). At the point where the transition from choked to non-coked flow occurs the pressure is discontinuous and the flow resistance shifts from the nozzle only to the entire valve including the body resistance. This increased flow resistance causes a corresponding reduction in the mass flux, which is therefore also discontinuous at this point. This is also the reason that values of $K_{dG} > K_{dL}$, i.e. the choked flow condition under which K_{dG} is determined is more accurately represented by the isentropic nozzle model, which does not include the valve body effects that influence the value of K_{dL}. Since two-phase flashing flows choke much more readily than single-phase gas flows (e.g. choking can occur at pressures as high as 90% of the upstream pressure), it is very unusual to encounter subcritical (non-choked) conditions with two-phase flows. Thus the use of K_{dG} is generally appropriate for two-phase flows.

Balanced bellows relief valves utilize a backpressure correction to account for the action of the bellows in compensating for the backpressure and enhancing the lift of the spring. This backpressure correction factor uses the gas correction factor for choked flow and the liquid correction factor for non-choked flows.

Comparison of Model Predictions

Darby et al. [*21*] compared most of the methods discussed herein for predicting the required relief mass flux for several fairly severe cases involving flashing and (retrograde) condensing ethylene at several different conditions. They found that most of the equilibrium models and the HNE model for nozzle lengths greater than 6 inches gave results that were up to 200% higher or lower than the HDI model, depending upon the value of the relief pressure relative to the saturation pressure, for conditions well away from the critical point. However, in the vicinity of the critical point the results varied by up to 6-700%, depending upon how close the relief pressure is to the saturation pressure (i.e. the degree of sub-cooling). These differences illustrate that the variation in the results that can arise from applying different models to the same case can be quite significant, although the trends shown here may not be typical of all conditions that may arise. Specifically, the Omega, API and HNE methods are not recommended in the vicinity of the critical point, but may give excellent results under other less stringent conditions, notably for single component simple fluids far from the critical point, over a small moderate pressure range.

Darby [*22*] compared the predictions of the HDI method with frozen air/water data (Lenzing, et al. [*24-27*]) in four different valves over a range of pressures (see Figs. 1 – 9), and the HNDI method for steam/water flashing data in one (Leser) valve (Lensing [*24, 25*]) over a range of pressures (Figs. 10 – 13). The specifications of the valves are given in Table 1, and the flow conditions of the tests are shown in Table 2. The manufacturer's gas flow coefficient, K_{dG}, was used in all cases when the flow was choked, and the reported liquid coefficients, K_{dL}, were used when the flow was not choked. Note that non-choked flow occurred only when the entering quality (x) of the mixture was less than 0.001. For the flashing flows, all data points corresponded to choked flow, and the HNDI method was used with an equilibrium relaxation length L_e of 40 mm. It should be noted that using different values of the discharge coefficient for choked and non-choked flow results in a discontinuity at the point corresponding to the transition point between the two (see e.g. Figure 1). This is realistic, since the actual flow resistance in choked flow is due only to the nozzle, and is hence lower than that for non-choked flow where the valve body resistance is also important. The discontinuity is not apparent in the other Figures, since there are no data points in the immediate vicinity of the choke/non-choke transition point.

Inlet and Discharge Line Sizing

It is necessary to size the inlet line from the vessel to the relief valve large enough that the irreversible friction loss in this line is less than 3% of the valve gauge set pressure. This "3%

rule" is specified by API 520 [*2*] in order to avoid a condition that results in rapid opening and closing of the valve ("chatter"), with potential damaging consequences. Although the basic nozzle equations are written in terms of the pressures just upstream and downstream of the valve nozzle, it is common practice to use the pressure in the protected equipment (stagnation pressure, P_o) as the valve inlet (upstream) pressure and the backpressure on the valve (P_B) as the downstream pressure. This ignores the pressure drop in the piping from the vessel to the valve. This assumption does not introduce a serious error when the inlet pressure drop is low compared to the set pressure, i.e. when the "3% rule" is satisfied.

Similarly, the irreversible friction loss in the discharge piping should be kept to less than 10% of the valve gauge set pressure, to avoid excessive built-up backpressure which can also adversely affect the chatter characteristics of the valve. This guideline applies to normal spring loaded relief valves, but different guidelines apply to balanced bellows and pilot operated valves (see API [*2*]).

Recommendations

The HDI/HNDI method is recommended as the calculation method of choice, for both single phase (gas or liquid) and two-phase flows. It is not subject to the many assumptions/restrictions that are inherent in the various other methods/models. These restrictions can be very limiting under certain circumstances, and the identification of these circumstances is difficult to determine rigorously. The HDI/HNDI method is not only more rigorous but simpler to apply than the other methods. Its only limitation is the availability of a thermodynamic data base or model which enables determining the two-phase mixture density as a function of pressure.

Example

To illustrate the procedure for sizing a pressure relief valve, the calculations required to size a valve for a vessel initially containing saturated water and a small fraction of vapor, will be shown.

The initial conditions in the vessel are: P_o = 116 psia (8 bar), x_o = 0.001 (quality, or mass fraction vapor). From steam tables, the entropy of this initial mixture is 0.48946 BTU/lbm. The valve discharge pressure $P_n = P_b$ is 14.64 psia.

The postulated relief scenario requires that a valve be sized to relieve the mixture in the vessel at a maximum rate of 10,042 kg/hr. A Leser valve will be used, with a K_{dG} coefficient of 0.77 and a nozzle length of 40 mm.

The procedure using a spreadsheet is as follows:

1. Starting at the vessel conditions (P_o=116 psia, x_o=0.001, s = 0.48946 BTU/lbm), decrease the pressure by 1 psi increments at constant entropy, and obtain the density of the two-phase mixture ρ_m from steam tables at each increment. Some steam tables tabulate this mixture directly; others will give the quality (x) and separate densities of the liquid and vapor phases, in which case Eqns (7 and 8) can be used to calculate the two-phase density (setting S = 1).
2. For each pressure increment i, calculate $\left(P_{i+1} - P_i\right)/\overline{\rho}_i$, where $\overline{\rho}_i$ is the average two-phase density over the increment.
3. Evaluate the summation in Eqn (9) at each interval from P_o to P_{i+1} (i.e. summing the terms from step 2 from the initial state to pressure P_{i+1})
4. Inserting the summation from Step 3 into Eqn (9), and multiplying by the discharge coefficient K_{dG} gives the nozzle mass flux G_n corresponding to each exit pressure $\left(P_{i+1}\right)$.
5. The procedure is repeated at successively lower values of pressure until either (a) the discharge pressure P_b is reached, or (b) the mass flux G_n goes through a maximum. If the maximum is reached before the discharge pressure is reached, this means that the flow is choked at the nozzle pressure corresponding to the point where the mass flux is maximum. If no maximum is reached before the discharge pressure is reached, then the flow is not choked.

This procedure is proper for equilibrium flows, e.g. for nozzles less than 10 cm long. However, in our example the nozzle length is only 40 mm (4 cm), so non-equilibrium would be expected. This is easy to account for, as follows:

At each pressure increment, using the values of quality $\left(x_e\right)$, the densities of the liquid and vapor from the steam tables, and the initial entering quality $\left(x_o\right)$, calculate the equivalent non-equilibrium quality x from Eqn (27) (using L = 4 cm). This non-equilibrium quality is used in Eqns (7 and 8) to compute the corresponding non-equilibrium two-phase density. This is then used to evaluate the summation in Eqn (9) and G_n as above.

The results are shown in the attached spreadsheet output. (Note: $G_n = G_o K_{dG} / 0.9$ in the table. The 0.9 is the recommended "safety factor", and is incorporated into the value of G_n). It is noted that assuming equilibrium flow, the calculated mass flux is 4,548 kg/sm^2 with choking at a nozzle pressure of 104 psia, whereas the non-equilibrium flow results in a mass flux of 6,714 $kg/s\,m^2$ and choking at a nozzle pressure of 96 psia.

In order to select the proper size valve (and nozzle), the required orifice area is calculated from the computed mass flux and the required relief rate, as follows:

$$A_n = \frac{\dot{m}}{G_n} = \frac{10,042\ kg/hr}{6,714 kg/\left(s\ m^2\right) 3600\ s/hr}\left(\frac{100\ cm/m}{2.54\ in/cm}\right)^2 = 0.644\ in^2$$

(Note: this value includes the API "safety factor" of 0.9). This value falls between the area of a G orifice (0.5674 in^2) and a H orifice (0.8874 in^2), so the H orifice would be selected. This means that the actual flow rate through the nozzle would be:

$$\dot{m}_{actual} = \left(10,042\ kg/hr\right)\frac{0.8874}{0.644} = 13,840\ kg/hr$$

which is the design flow rate that should be used for sizing the inlet and discharge lines.

P psia	T F	xe	RhoM lbm/ft^3	RhoL lbm/ft^3	RhoG lbm/ft^3	RhoAvg lbm/ft^3	DP/RhoM ft^2/s^2	SumDP/rho ft^2/s^2	**Gn** kg/sm^2
s =	0.48946								
115.9925	338.77	0.001	46.109	56.002	0.2597				
115	338.13	0.001756	40.596	56.024	0.25759	43.3525	-106.068	-106.068	2475.302
114	337.48	0.002521	36.149	56.047	0.25547	38.3725	-120.739	-226.807	3223.127
112	336.16	0.004062	29.479	56.094	0.25121	32.814	-282.383	-509.19	3938.271
110	334.83	0.005617	24.717	56.141	0.24696	27.098	-341.948	-851.138	4269.222
108	333.48	0.007188	21.147	56.188	0.2427	22.932	-404.069	-1255.21	4435.672
106	332.11	0.008775	18.373	56.236	0.23843	19.76	-468.933	-1724.14	4516.68
104	330.71	0.010378	16.155	56.285	0.23417	17.264	-536.73	-2260.87	4547.762
102	329.3	0.011998	14.324	56.334	0.2299	15.2395	-608.033	-2868.9	4542.294
100	327.87	0.013635	12.833	56.384	0.22563	13.5785	-682.411	-3551.31	4527.68
98	326.41	0.01529	11.558	56.434	0.22135	12.1955	-759.798	-4311.11	4492.936
96	324.92	0.016963	10.466	56.485	0.21708	11.012	-841.456	-5152.57	4447.803
94	323.42	0.018656	9.5211	56.537	0.2128	9.99355	-927.209	-6079.78	4395.253
92	321.88	0.020369	8.6956	56.589	0.20851	9.10835	-1017.32	-7097.1	4337.035
90	320.32	0.022103	7.9686	56.642	0.20422	8.3321	-1112.1	-8209.19	4274.5

NON-EQUILIBRIUM L = 40mm

x	alpha	RhoTPa	DP/RhoM ft^2/s^2	SumDP/rho	**Gn**
0.001	0.177535	46.1058			
0.001302	0.220934	43.70329	-105.2	-105.2	2653.844
0.001608	0.261118	41.47881	-111.68	-216.88	3616.503
0.002225	0.332376	37.53317	-246.84	-463.719	4785.153
0.002847	0.393581	34.14218	-271.356	-735.075	5480.372
0.003475	0.446712	31.19659	-296.977	-1032.05	5933.499
0.00411	0.493254	28.61499	-323.77	-1355.82	6238.037
0.004751	0.534331	26.3353	-351.797	-1707.62	6442.979
0.005399	0.57085	24.307	-381.152	-2088.77	6577.028
0.006054	0.603503	22.49228	-411.904	-2500.68	6659.095
0.006716	0.632872	20.85858	-444.166	-2944.84	6701.449
0.007385	0.659395	19.38221	-477.999	-3422.84	6713.509
0.008062	0.683488	18.04007	-513.561	-3936.4	6701.021
0.008748	0.705451	16.81531	-550.966	-4487.37	6668.893
0.009441	0.725542	15.69401	-590.332	-5077.7	6620.949

NOTATION

A	cross sectional area of the nozzle throat (orifice) in a valve, or open area of rupture disk, $\left(in^2\right)$ or $\left(mm^2\right)$
G_o	theoretical mass flux through an isentropic nozzle, $\left(lb_m / s\ in^2\right)$ or $\left(lb_m / hr\ ft^2\right)$ or $\left(kg / hr\ cm^2\right)$, etc.
G_n	actual mass flux through nozzle = $K_d G_o$, $\left(lb_m / s\ in^2\right)$ or $\left(lb_m / hr\ ft^2\right)$ or $\left(kg / hr\ cm^2\right)$, etc.
k	isentropic exponent for a gas, equal to c_p / c_v for ideal gas, (-)
K_d	relief valve discharge coefficient (-)
K_{dG}	gas phase discharge coefficient (-)
K_{dL}	liquid phase discharge coefficient (-)
K_v	viscosity correction factor for viscous fluids, (-)
L	nozzle length, 9in) or (cm), etc.
L_e	relaxation length for non-equilibrium flow = 10 cm.
$\dot{m}$	required relief mass flow rate, $\left(lb_m / s\right)$ or $\left(lb_m / hr\right)$ or $\left(kg / hr\right)$, etc.
N_{Re}	Reynolds number through the valve nozzle, using volumetric weighted fluid properties for mixtures.
P	pressure, (psia) or (Pa). etc.
P_i	pressure at interval i, (psia) or (Pa). etc.
P_o	pressure at valve entrance, (psia) or (Pa). etc.
P_n	pressure at the nozzle throat (exit), (psia) or (Pa). etc.
s	specific entropy (BTU/ lb_m) or $\left(Nm / kg\right)$
S	slip ratio (ratio of the gas phase velocity to the liquid phase velocity) (-)
x	quality, or mass fraction of gas phase, (-)
x_o	quality at nozzle entrance, (-)
x_e	equilibrium quality at pressure P, (-)
α	volume fraction fo the gas phase, (-)
β	d/D, ratio of the nozzle diameter to the valve inlet diameter, (-)
ρ	density of the fluid (mixture) in the nozzle at pressure P, $\left(lb_m / ft^3\right)$ or $\left(kg / m^3\right)$
ρ_G	gas phase density, $\left(lb_m / ft^3\right)$ or $\left(kg / m^3\right)$
ρ_L	liquid phase density, $\left(lb_m / ft^3\right)$ or $\left(kg / m^3\right)$
ρ_n	fluid density at the nozzle throat at pressure P_n, $\left(lb_m / ft^3\right)$ or $\left(kg / m^3\right)$
$\bar{\rho}_i$	average fluid density over interval i to i+1, (psia) or (Pa). etc.

REFERENCES

1. Wong, W.Y., "Protect Plants Against Overpressure",, *Chemical Engineering,* pp. 66 – 73, June, 2001
2. American Petroleum Institute, "Sizing, Selection, and Installation of Pressure-Relieving Devices in Refineries", ANSI/API RP 520, 7th Edition, Part 1 Sizing and Selection, Washington, D.C., January 2000
3. American Petroleum Institute, "Sizing, Selection, and Installation of Pressure-Relieving Devices in Refineries", ANSI/API RP 520, 5th Edition, Part 2 Installation, Washington, D.C., January 2003
4. Center for Chemical Process Safety, "Guidelines for Pressure Relief and Effluent Handling Systems", American Institute of Chemical Engineers, New York, NY, 1998
5. The Design Institute for Emergency Relief Systems (DIERS) Project Manual, "Emergency Relief System Design Using DIERS Tecnology", American Institute of Chemical Engineers, 1992
6. R. Darby, AExperiments for Runaway Reactions and Vent Sizing@, Center for Chemical Process Safety, SACHE Program, AIChE, (ISBN 0-8169-0856-7), November 2001
7. National Fire Protection Association, NFPA-30, "Flammable and Combustible Liquids Code", Fullment Center, 11 Tracy Drive, Avon, MA 02322-9910, 2000
8. American Petroleum Institute, "Guide for Pressure Relieving and Depressuring Systems", ANSI/API RP 521, Washington, D.C., 2005
9. Das, D.K., "Emergency Relief System Design: In Case of fire, Break Assumptions", *Chemical Engineering*, pp 34 – 40, February, 2004
10. Darby, R., "Chemical Engineering Fluid Mechanics", 2nd Ed.., Marcel Dekker, NY, 2001
11. Darby, R. and K. Molavi, "Viscosity Correction Factor for Emergency Relief Valves", *Process Safety Progress*, v. 16, no.2, pp 80-82, Summer, 1997
12. Shackleford, A., "Using the Ideal Gas Specific Heat Ratio for Relief-Valve Sizing", *Chemical Engineering*, pp. 54-59, November, 2003
13. Jamerson, S.C. and H.G. Fisher, "Using Constant Slip Ratios to Model Non-Flashing (Frozen) Two-Phase Flow through Nozzles", *Process Safety Progress*, 18 (2), Summer, 1999
14. Darby R, P.R. Meiler, and J. R. Stockton, "Flashing Flow in Nozzles, Pipes and Valves", *Chemical Engineering Progress*, pp 56-64, May, 2001
15. Henry, R E and H K Fauske, "The Two-Phase Critical of One-Component Mixtures in Nozzles, orifices, and Short Tubes", *Trans. American Society of Mechanical Engineers J., Heat Transfer*, pp 179-187, May , 1971
16. Leung, J. C., "Easily Size Relief Devices and Piping for Two-Phase Flow", *Chemical Engineering Progress*, pp 28-50, 1996
17. Leung, J. C., "The Omega Method for Discharge Rate Evaluation", pages 367 – 393, in *International Symposium on Runaway Reactions and Pressure Relief Design,* editors G.A.Melhem and H.G. Fisher, American Institute of Chemical Engineers, New York, NY 1995

18. Simpson, L. L., "Estimate Two-Phase Flow in Safety Devices", Chem. Eng., pp 98-102, 1991
19. Simpson, L. L.. "Navigating the Two-Phase Maze", pages 394 – 415, in *International Symposium on Runaway Reactions and Pressure Relief Design,* editors G.A.Melhem and H.G. Fisher, American Institute of Chemical Engineers Press, 1995.
20. Fauske, H K, "Determine Two-Phase Flows During Releases", *Chemical Engineering Progress*, pp 55-58, February, 1999
21. Darby, R., F.E. Self and V.H. Edwards, "Properly Size Pressure-Relief Valves for Two-Phase Flow", *Chemical Engineering*, pp. 68 – 74, June, 2002
22. Darby, R., "On Two-Phase Frozen and Flashing Flows in Safety Relief Valves. Recommended Calculation Method and the Proper Use of the Discharge Coefficient", *J. of Loss Prevention in the Process Industries*, *17,* 255 -5259, 2004
23. American Petroleum Institute, "Flanged Steel Safety Relief Valves", API Standard 526, 3rd Edi., Washington, DC, February 1984
24. Lenzing, F., Cremers, J. and Friedel, L., "Prediction of the Maximum Full Lift Safety Relief Valve Two-Phase Flow Capacity", ISO TC 185 WG1 Meeting, Paper N106, Rome, October 29-31, 1997
25. Lenzing, T. and Friedel, L., "Full Lift Safety Valve Air/Water and Steam/Water Critical Mass Flow Rates", Presented at ISO TC 185 WG1 Meeting, Paper N94, Louvain-la-Neuve, September 3-4, 1996
26. Lenzing, F., Friedel, L., Cremers, J. and Alhusein, M., "Prediction of the Maximum Full Lift Safety Valve Two-Phase Flow Capacity", *J. of Loss Prev. in the Proc. Ind.*, *11*, 307-321, 1998
27. Lenzing. T., Schecker, J., Friedel, L., and Cremers, J. "Safety Relief Valve Critical Mass Flux as a Function of Fluid Properties and Valve Geometry", Presented at ISO TC 185 WG1 Meeting, Paper N103, Rome, October 29-31, 1997

TABLE I VALVE SPECIFICATIONS (Lenzing, et al, [*18, 20*]				
Valve	K_{dG}	K_{dL}	Orifice Dia. (mm)	Orifice Area $\left(in^2\right)$
B&R DN25/40 (Bopp & Reuther Si63)	0.86	0.66	20	0.4869
ARI DN25/40 (Albert Richter 901/902)	0.81	0.59	22.5	0.6163
Crosby 1 x 2 "E" (JLT/JBS)	0.962	0.729	13.5	0.2219
Leser DN25/40 (441)	0.77	0.51	23	0.6440

TABLE II FLOW CONDITIONS (Lenzing, et al, [*18, 20*]			
Fluid	Nom. Pressure (bar)	P_o (psia)	P_b (psia)
Air/Water	5	72.495	14.644
Air/Water	8	115.993	14.644
Air/Water	10	144.991	14.644
Steam/Water	5.4	78.295	14.644
Steam/Water	6.8	98.594	14.644
Steam/Water	8	115.993	14.644
Steam/Water	10.6	153.690	14.644

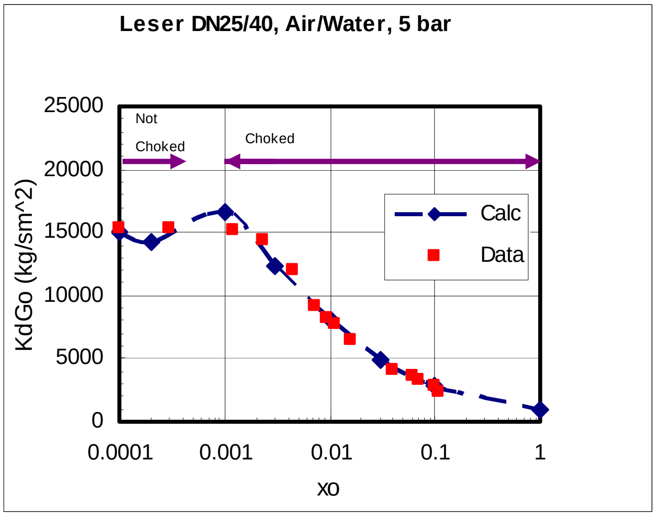


Figure 1

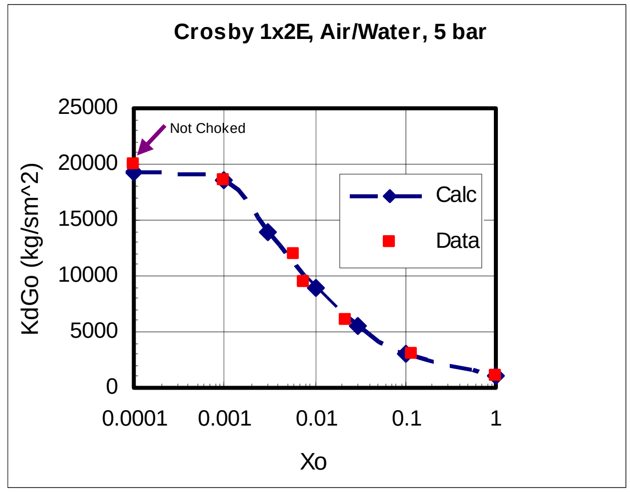


Figure 2

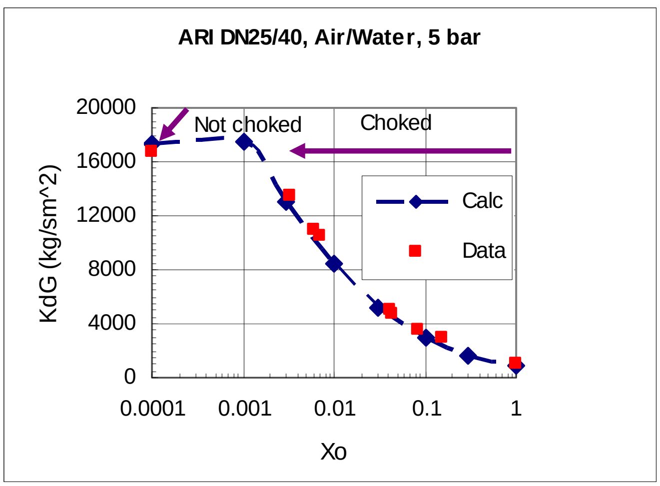


Figure 3

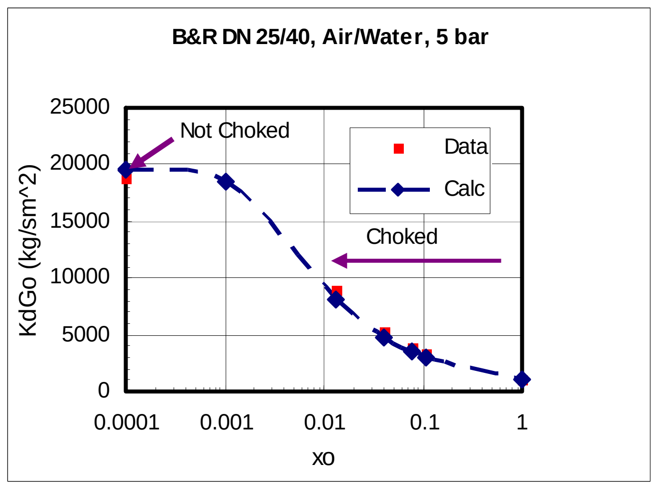


Figure 4

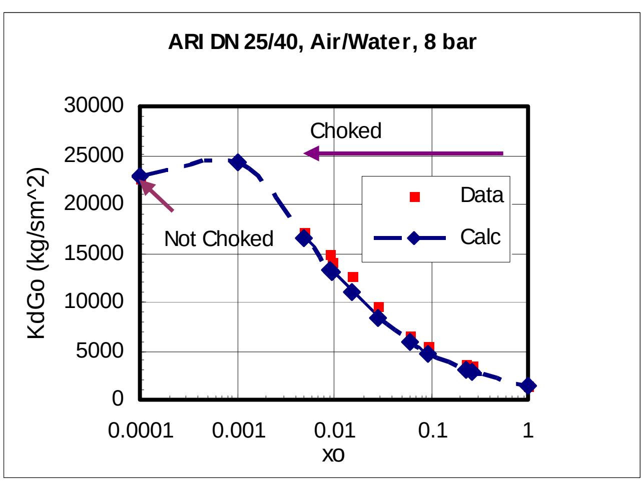


Figure 5

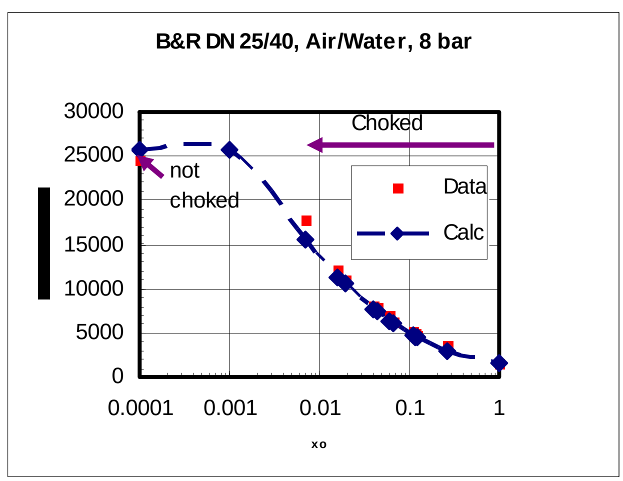
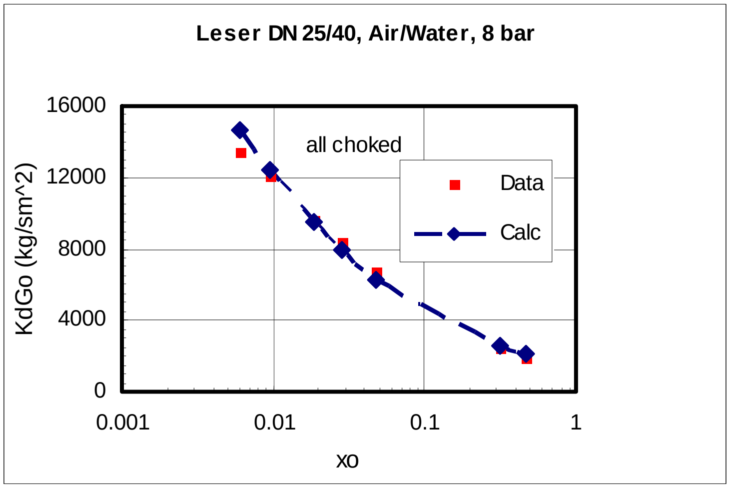


Figure 6



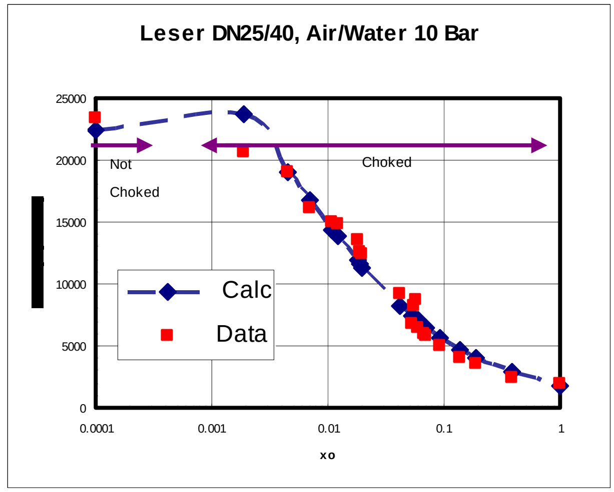


Figure 7

Figure 8

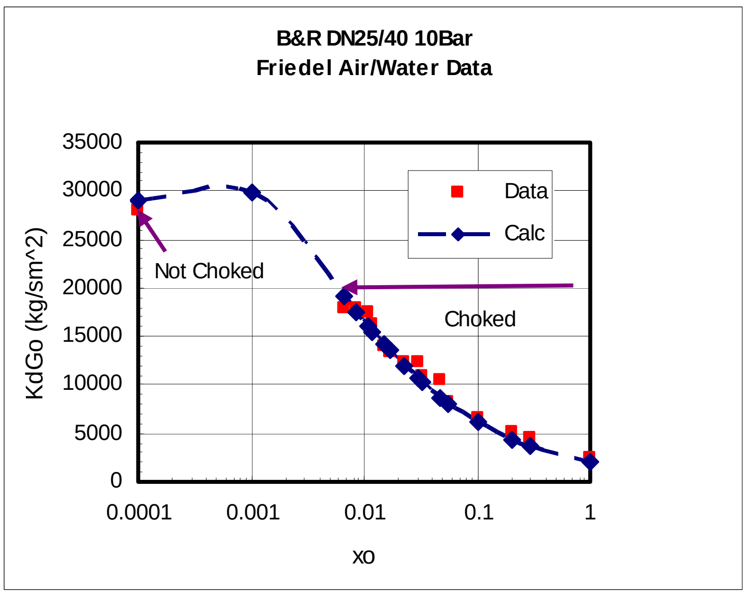


Figure 9

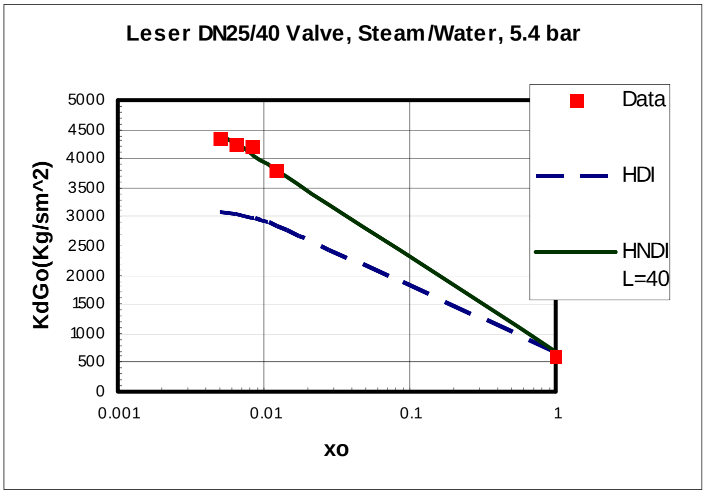


Fig. 10

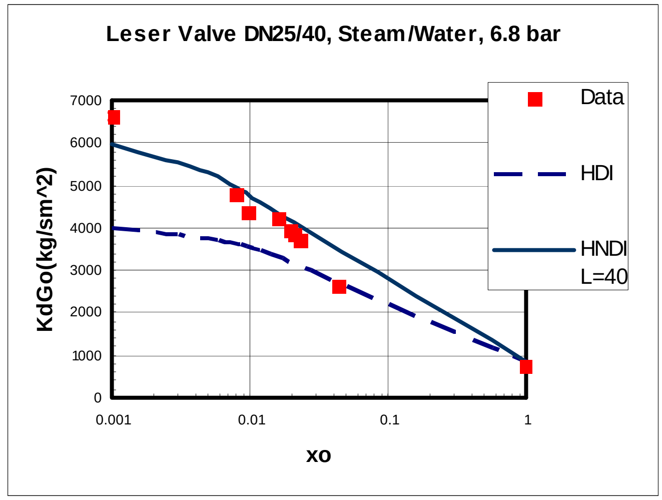


Fig. 11

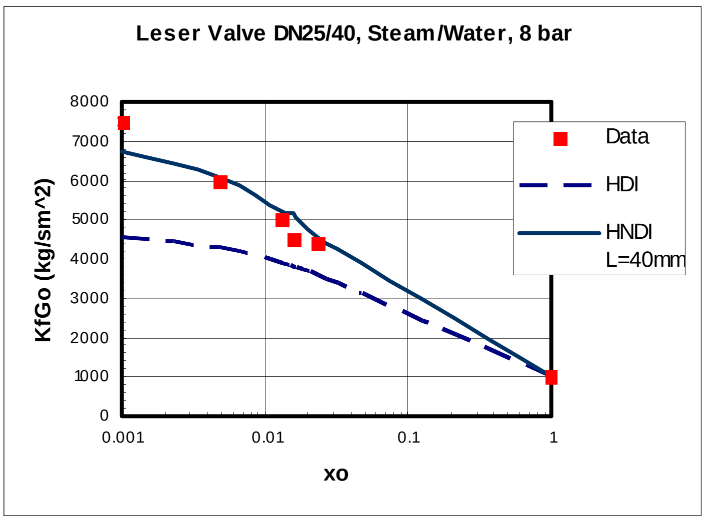


Fig. 12

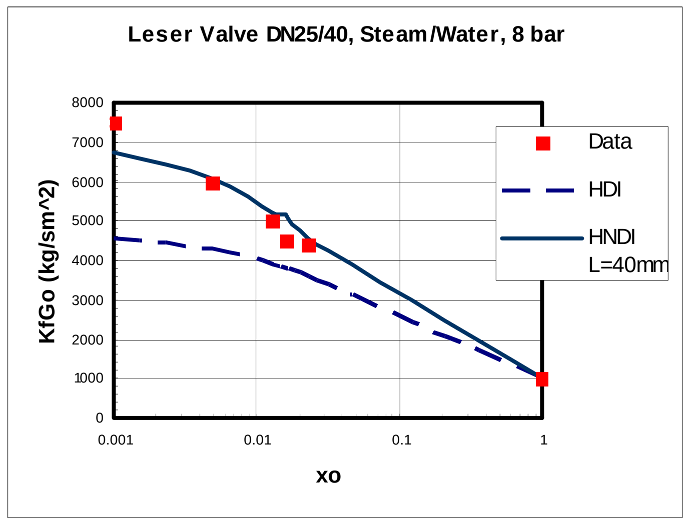


Fig. 13