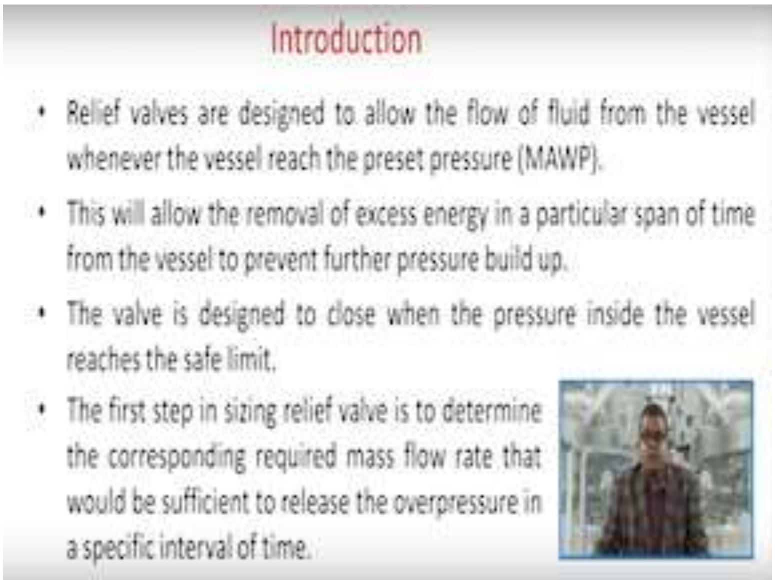
Chemical Process Safety
Professor Shishir Sinha
Department of Chemical Engineering
Indian Institute of Technology, Roorkee
Lecture 36: Relief Sizing

Welcome to this module of Relief Sizing. So earlier we have studied about what is the relief. What different kind of relief methodology available? We did the classification of relief and we have discussed the various relief scenarios. In this particular chapter we are going to discuss with the relief sizing.

(Refer Slide Time: 0:57)



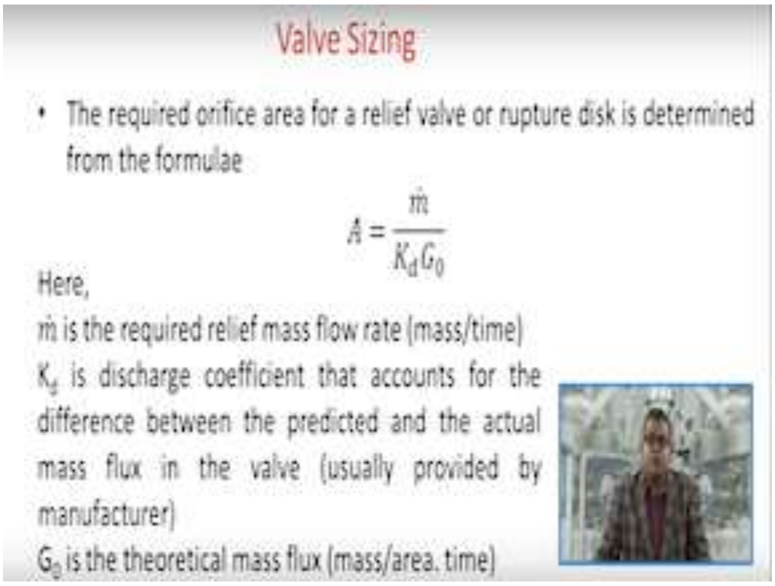
Introduction

- Relief valves are designed to allow the flow of fluid from the vessel whenever the vessel reach the preset pressure (MAWP).
- This will allow the removal of excess energy in a particular span of time from the vessel to prevent further pressure build up.
- The valve is designed to close when the pressure inside the vessel reaches the safe limit.
- The first step in sizing relief valve is to determine the corresponding required mass flow rate that would be sufficient to release the overpressure in a specific interval of time.

Now, basically relief valves, they are designed to allow the flow of fluid from the vessel whenever the vessel reach the preset pressure that is the (MAWP). As we have discussed already that is the maximum allowable working pressure. Now this allows that the removal of excess energy in a particle span of time from the vessel to prevent further pressure built up.

Now usually the valve are designed to close when the pressure inside the vessel reaches is the safe limit. So we have already the set pressure etc but the first step is sizing relief valve is to determine the corresponding required mass flow rate that would sufficient to release the overpressure in a specific interval of time.

(Refer Slide Time: 1:33)



Valve Sizing

- The required orifice area for a relief valve or rupture disk is determined from the formulae

$$A = \frac{\dot{m}}{K_d G_0}$$

Here,

$\dot{m}$ is the required relief mass flow rate (mass/time)

K_d is discharge coefficient that accounts for the difference between the predicted and the actual mass flux in the valve (usually provided by manufacturer)

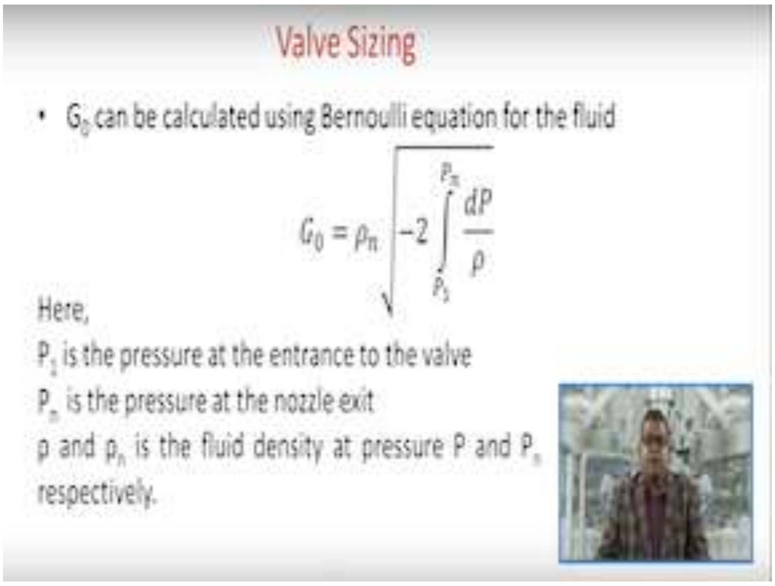
G_0 is the theoretical mass flux (mass/area. time)

Now the required orifice area for a relief valve or a ruptured disk is determined by this particular formula.

$$A = \frac{\dot{m}}{K_d G_0}$$

Where m_0 is the required relief mass flow rate and usual units are in terms of mass per time, K_d is the discharge coefficient that account for the difference between the predicted and actual mass flux in the valve and usually provided by the manufacturer. And G_0 is the theoretical mass flux having the unit of mass per unit area time.

(Refer Slide Time: 2:09)



Valve Sizing

- G_0 can be calculated using Bernoulli equation for the fluid

$$G_0 = \rho_n \sqrt{-2\int_{P_1}^{P_n} \frac{dP}{\rho}}$$

Here,

P_1 is the pressure at the entrance to the valve

P_n is the pressure at the nozzle exit

ρ and ρ_n is the fluid density at pressure P and P_n respectively.

So, whenever there is a need to calculate the G_0 it can be calculated by the Bernoulli's equation for the fluid and that is the

$$G_0 = \rho_n \sqrt{-2\int_{P_1}^{P_n} \frac{dP}{\rho}}$$

Where P_1 is the pressure at the entrance of the valve, and P_n is the pressure at the nozzle exit. So whenever you are using the nozzle, ρ and ρ_n is the fluid density at the pressure P and P_n respectively.

(Refer Slide Time: 2:43)

Valve Sizing

- Only standard sizes of valve nozzle are available commercially, and calculated area would not be expected to match those sizes.
- Practically 10% as a safety factor is applied to the calculated area and then the closest matching size is selected.
- As discussed previously, the relief area must neither be too large nor too small to omit the chances of excessive flow or chattering in case of large sized nozzle and compromised overprotection in case of small sized nozzle respectively.

So only standard sizes of all nozzles are available commercially and calculated area would not expected to match the sizes so practically 10 percent as a safety factor is applied to the calculated area then the closest matching size is selected. So we have discussed previously that the relief area must neither be too large nor too small to omit the chances of excessive flow or chattering in case of large size nozzle and compromised over protection in case of a small size nozzle respectively. So this is prima facie requirement.

(Refer Slide Time: 3:52)

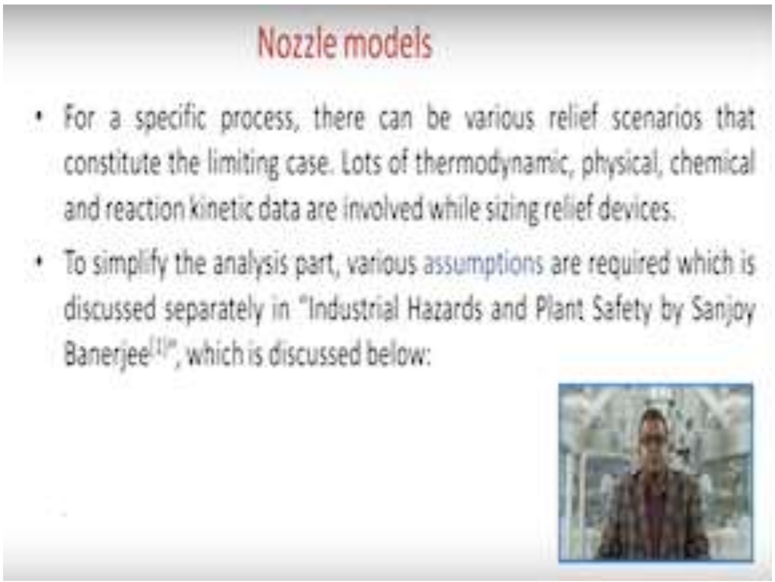
Valve Sizing

- Suppose, an oversized valve is selected then the actual flowrate due to larger area will be significantly greater the desired rate for which the system is designed.
- When the fluid flow through the piping system, due to higher flow rate there will be greater pressure drop along the entrance and exit point of the piping system. This excessive pressure drop can pose serious effects on the stability of the system.

The calculations are performed by assuming steady state flow through relief valve. The flowrate is taken 110% of the relief set pressure.

Now let us see the oversized valve is selected then the actual flow rate due to the large area will be significantly greater the desired rate for which the system is designed. So, when the fluid flow through the piping system, due to higher flow rate, there will be greater pressure drop along the entrance and exit point of the piping system. So this excessive pressure drop can pose a serious effect on the stability of the system. So usually those calculations, which we have discussed they are performed by steady state flow through relief valve and the flow rate is taken 110 percent of the relief set of pressure.

(Refer Slide Time: 4:05)

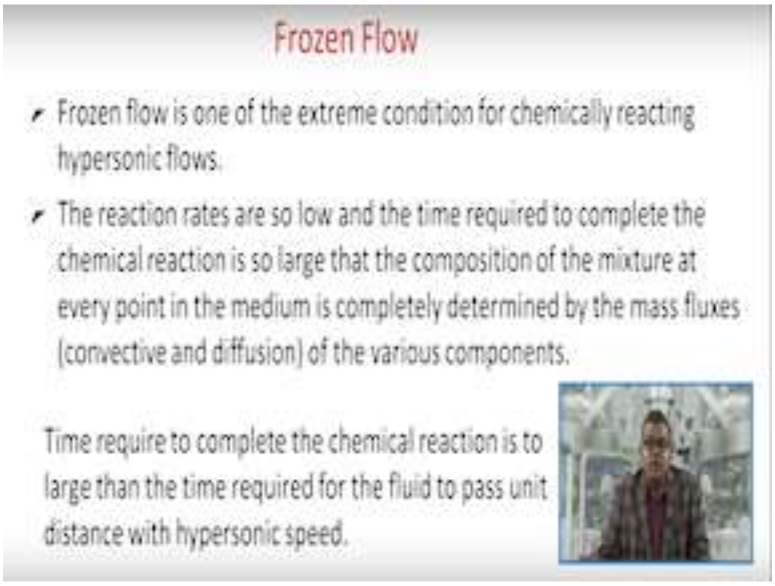


Nozzle models

- For a specific process, there can be various relief scenarios that constitute the limiting case. Lots of thermodynamic, physical, chemical and reaction kinetic data are involved while sizing relief devices.
- To simplify the analysis part, various assumptions are required which is discussed separately in "Industrial Hazards and Plant Safety by Sanjoy Banerjee[1]", which is discussed below:

Now, whenever we are discussing this particular aspect, we should give a due care to the nozzle model so for a specific processes there can be various relief scenarios that constitutes the limiting case. Lots of thermodynamics, physical, chemical and reaction kinetic data are involved while we go for these relief devices. To simplify the analysis part various other assumptions are required which is discussed separately in the given reference; that is “Industrial Hazard and Plant Safety” by Sanjoy Banerjee and we are going to discuss these things in sequential manner.

(Refer Slide Time: 4:57)



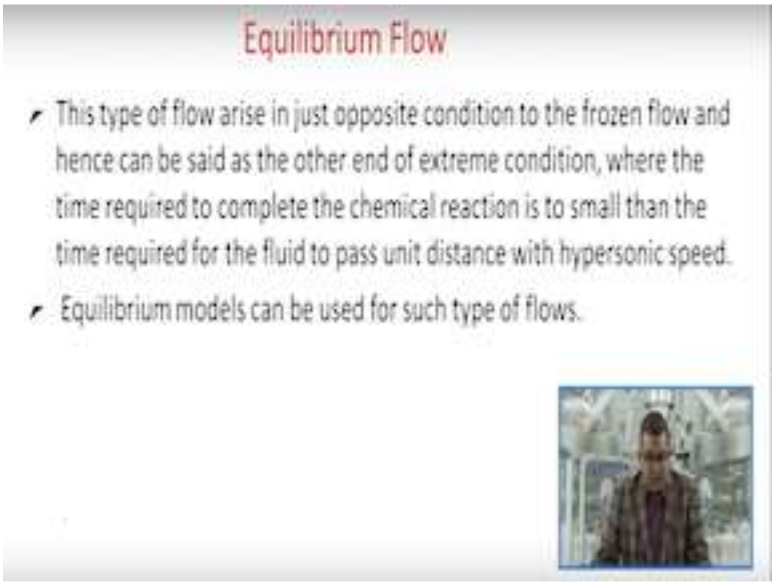
Frozen Flow

- Frozen flow is one of the extreme condition for chemically reacting hypersonic flows.
- The reaction rates are so low and the time required to complete the chemical reaction is so large that the composition of the mixture at every point in the medium is completely determined by the mass fluxes (convective and diffusion) of the various components.

Time require to complete the chemical reaction is to large than the time required for the fluid to pass unit distance with hypersonic speed.

So, first let us take the frozen flow. Now frozen flow is one of the extreme condition for chemically reacting hypersonic flows. Now the reaction rates are so low and the time required to complete the chemical reaction is so large that the composition of the mixture at every point in the medium is completely determined by the mass fluxes, both the convective as well as diffusive mass fluxes. Now the time required to complete the chemical reaction is to larger than the time required for the fluid to pass unit distance with the hypersonic speed.

(Refer Slide Time: 5:31)

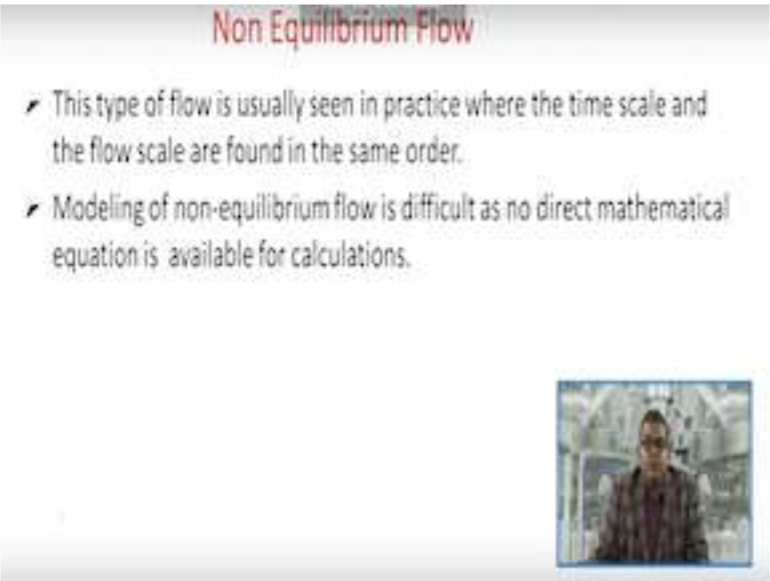


Equilibrium Flow

- This type of flow arise in just opposite condition to the frozen flow and hence can be said as the other end of extreme condition, where the time required to complete the chemical reaction is to small than the time required for the fluid to pass unit distance with hypersonic speed.
- Equilibrium models can be used for such type of flows.

Then we have to consider the equilibrium flow. This type of flow arise in the just opposite condition to the frozen flow and can be said as the other end of extreme condition where the time required to complete the chemical reaction is too small and the time required for the fluid to pass unit distance with hypersonic speed. So these equilibrium models can be used for such type of flow.

(Refer Slide Time: 6:01)

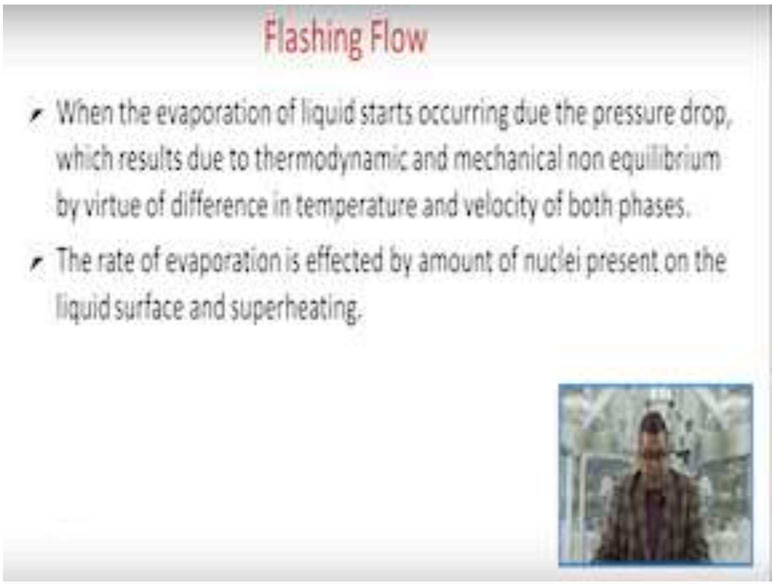


Non Equilibrium Flow

- This type of flow is usually seen in practice where the time scale and the flow scale are found in the same order.
- Modeling of non-equilibrium flow is difficult as no direct mathematical equation is available for calculations.

There are certain non-equilibrium flow, now this type of flow is usually seen in the practice where is the time scale and the flow of scale are found in the same order. So the modeling of non-equilibrium flow is difficult as no direct mathematical equation is available as on date for the calculation.

(Refer Slide Time: 6:21)

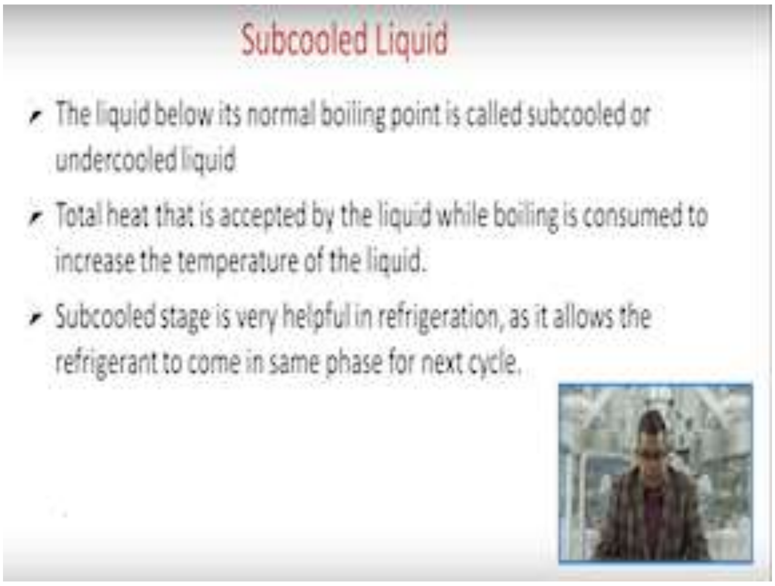


Flashing Flow

- When the evaporation of liquid starts occurring due the pressure drop, which results due to thermodynamic and mechanical non equilibrium by virtue of difference in temperature and velocity of both phases.
- The rate of evaporation is effected by amount of nuclei present on the liquid surface and superheating.

There is a flashing flow when the evaporation of liquid start occurring due to the pressure drop, sometimes we may have the higher pressure and the fluid moves from the higher pressure zone to all of sudden to a lower pressure zone so this result due to the thermodynamic and mechanical non-equilibrium by virtue of the difference in temperature and velocity of the both phases. The rate of evaporation is effected by amount of nuclei present on the liquid surface and superheating.

(Refer Slide Time: 6:55)



Subcooled Liquid

- The liquid below its normal boiling point is called subcooled or undercooled liquid
- Total heat that is accepted by the liquid while boiling is consumed to increase the temperature of the liquid.
- Subcooled stage is very helpful in refrigeration, as it allows the refrigerant to come in same phase for next cycle.

There is a concept of subcooled liquid, this liquid below its normal boiling point is called subcooled or undercooled liquid. This is a well-known thermodynamic phenomenon. So the total heat is accepted by liquid when boiling is consume to increase the temperature of the liquid. So subcooled state is very helpful in refrigeration as it allows the refrigerant to come in same phase for the next cycle. So, the energy efficiency in that particular case can be improvised.

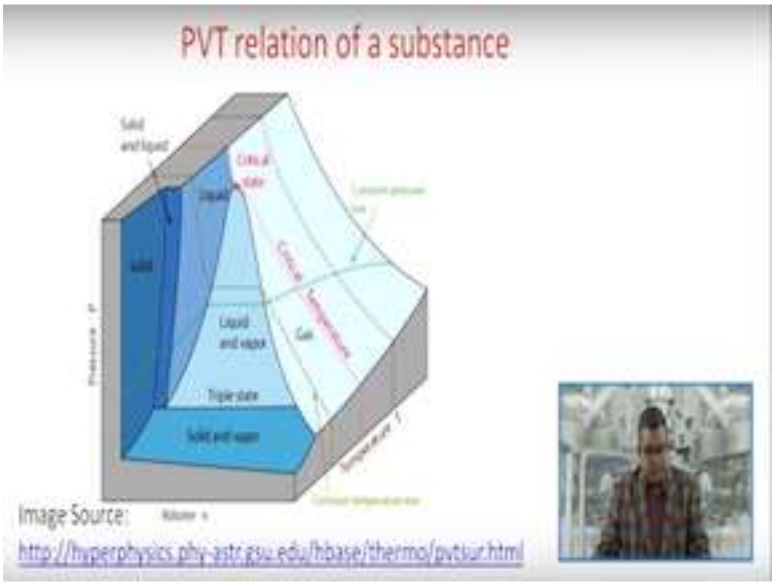
(Refer Slide Time: 7:32)

Saturated Liquid

- For a given pressure, when the liquid and vapour phase of the fluid are in equilibrium with each other then that state is called as saturation state.
- Small addition of energy after this point leads to gain the liquid a new stage known as superheated stage of the liquid, it turns into vapour at constant temperature.

Now saturated liquid for a given pressure when the liquid and vapor phases of the fluid are in equilibrium with each other then that state is called a saturation state and sometime it is quite evident in the steam table that you maybe encounter the saturated liquid concept. In a small addition of energy after the point led to gain the liquid a new state known as superheated stage of the liquid and it turns into the vapor at constant temperature.

(Refer Slide Time: 8:01)



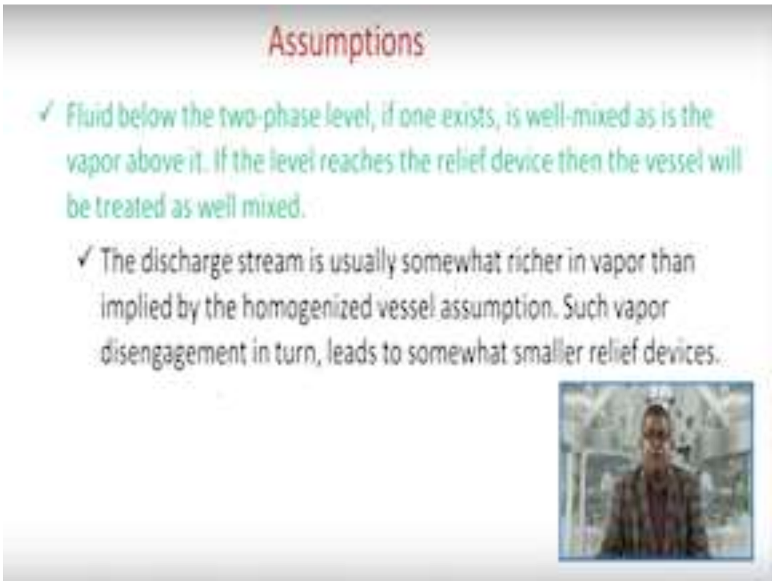
PVT relation of a substance

Image Source:
http://hyperphysics.phy-astr.gsu.edu/hbase/thermo/pvtsur.html

Now here this particular diagram clearly shows that the different aspects of all thermodynamic stages. Here the axis is the pressure and the volume sometime you may experience this is the solid and vapor phase. Call it phase and this reflects the temperature effect. So you may have the critical state, critical temperature whereas the several constant pressure line etc. So by slight variation in any of the parameter pressure, volume and temperature you may have changed in different type of like from sub-cooled to saturated, saturated to superheated etc.

Now remember this type of diagram they are extremely useful for the performing thermodynamic calculation because the pressure volume and temperature, these are only things which you can measure directly, which you can alter directly otherwise rest other things like internal energy, enthalpy, entropy, these are available in terms of correlation so you have a liberty to alter all these pressure, volume and temperature in due course of time.

(Refer Slide Time: 9:15)

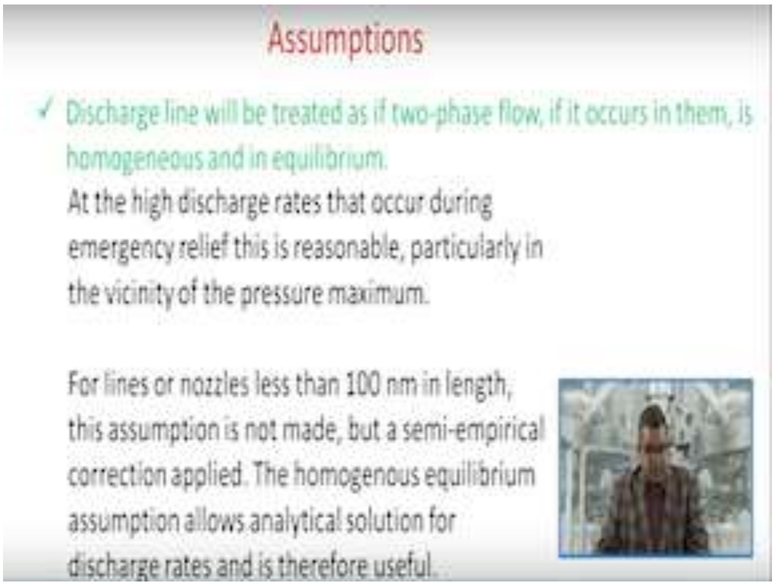


Assumptions

- ✓ Fluid below the two-phase level, if one exists, is well-mixed as is the vapor above it. If the level reaches the relief device then the vessel will be treated as well mixed.
  - ✓ The discharge stream is usually somewhat richer in vapor than implied by the homogenized vessel assumption. Such vapor disengagement in turn, leads to somewhat smaller relief devices.

So while we calculate or while we go for any kind of thermodynamic calculation we need to take several assumptions. Now, let us have a look at those assumptions that fluid below the two phase level, if one exist, is when mixed as is the vapor above it. Now, if level reaches the relieve device then the vessel will be treated as well mixed. Now, the discharge of stream is usually somewhat richer in vapor that implied by the homogenized vessel assumptions. Such vapor disengagement in turn lead to the sum what you can say the smaller relief devices.

(Refer Slide Time: 9:56)



Assumptions

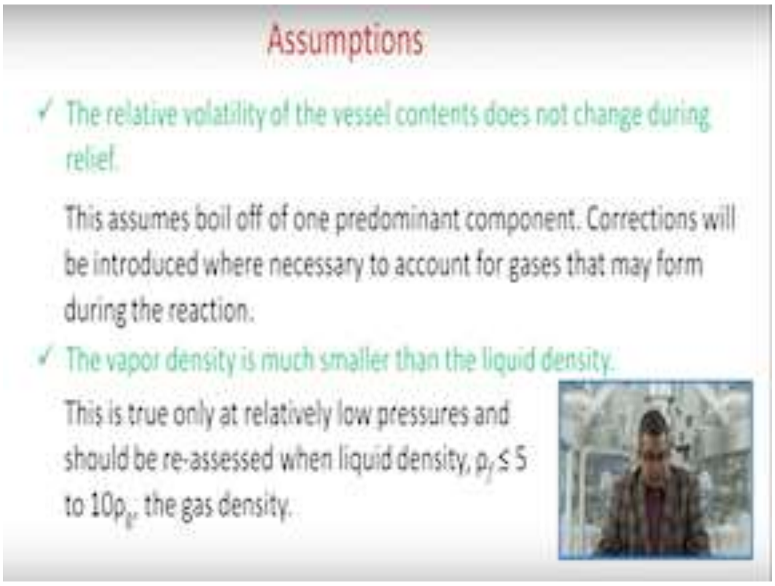
- ✓ Discharge line will be treated as if two-phase flow, if it occurs in them, is homogeneous and in equilibrium.

At the high discharge rates that occur during emergency relief this is reasonable, particularly in the vicinity of the pressure maximum.

For lines or nozzles less than 100 nm in length, this assumption is not made, but a semi-empirical correction applied. The homogenous equilibrium assumption allows analytical solution for discharge rates and is therefore useful.

Discharge line will be treated as if two-phase flow if it occurs in them, is homogenous and in equilibrium. So at high discharge rates that occur during the emergency relief this is reasonable particularly in the vicinity of the pressure maximum. So, for lines or nozzles less than 100 nano metres in length, this assumption is not made but a semi-empirical correction applied. The homogenous equilibrium assumption allows the analytical solution for discharge rate that is therefore useful.

(Refer Slide Time: 10:31)



Assumptions

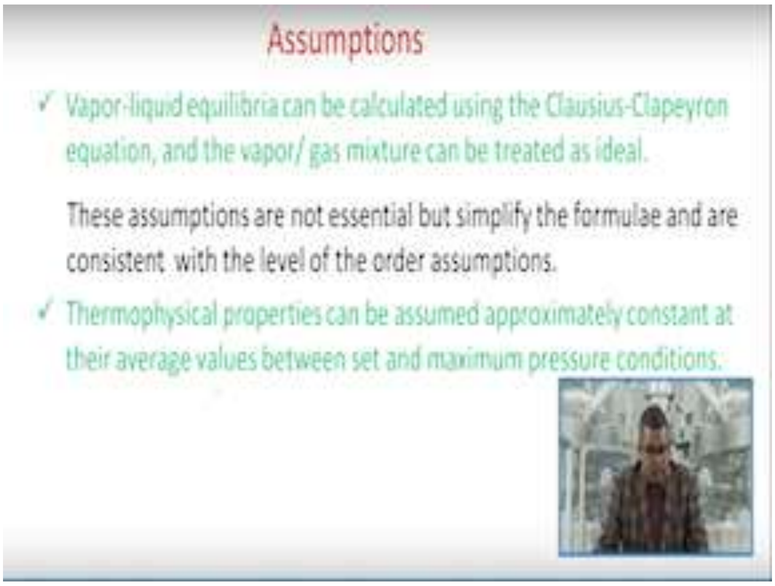
- ✓ The relative volatility of the vessel contents does not change during relief.

  This assumes boil off of one predominant component. Corrections will be introduced where necessary to account for gases that may form during the reaction.
- ✓ The vapor density is much smaller than the liquid density.

  This is true only at relatively low pressures and should be re-assessed when liquid density, $\rho_f \leq 5$ to $10\rho_g$, the gas density.

Another assumption is that the relative volatility of the vessel content does not change during the release. Now this assumes boil off of one pre-dominant component. Correction will be introduced where necessary to account for gases that my form during the reaction. Now another assumption is that vapor density is much smaller than the liquid density. This is true only at the relative low pressure and should be reassessed when liquid density ρ f is less than or equal to 5 to 10 ρ g, that is the gas density.

(Refer Slide Time: 11:14)



Assumptions

- ✓ Vapor-liquid equilibria can be calculated using the Clausius-Clapeyron equation, and the vapor/ gas mixture can be treated as ideal.

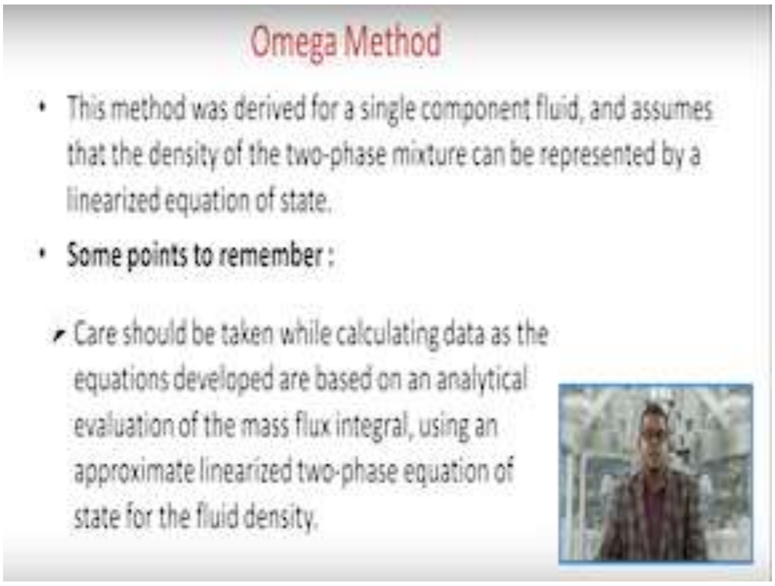
  These assumptions are not essential but simplify the formulae and are consistent with the level of the order assumptions.

- ✓ Thermophysical properties can be assumed approximately constant at their average values between set and maximum pressure conditions.

Now another assumption that vapor-liquid equilibria can be calculated using the Clausius-Clapeyron equation and the vapor/gas mixture can be treated as ideal. Then Clausius-Clapeyron equation is one of the most fundamental equations in thermodynamics. Now, these assumptions are 0 essential but simplify the formula and are consistent with the level of the order of assumptions.

Now, thermo-physical properties can be assumed approximately constant at their average values between set and maximum pressure conditions, this is another assumption. So various models have been discussed by Ron and Darby and you can have a look at those model and the reference is given at this particular model module, so the few, but few of them are discussed in the next slide.

(Refer Slide Time: 12:03)



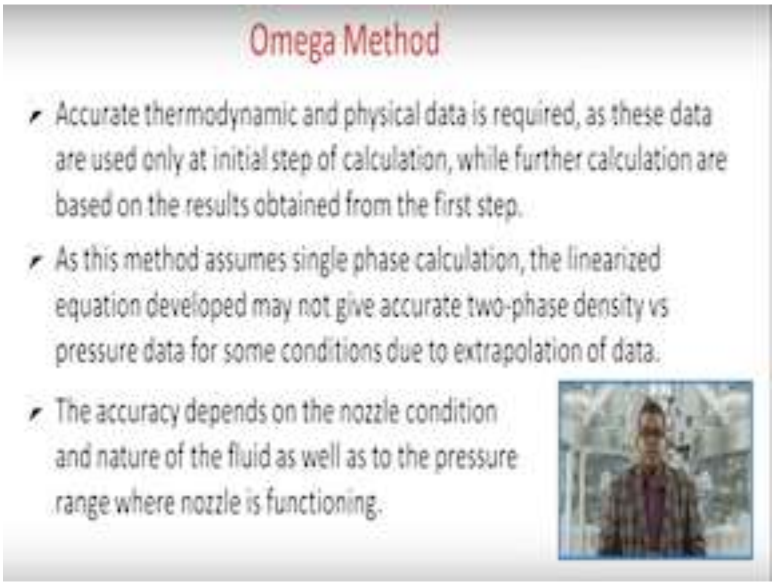
Omega Method

- This method 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.
- **Some points to remember :**
  - Care should be taken while calculating data as the equations developed are based on an analytical evaluation of the mass flux integral, using an approximate linearized two-phase equation of state for the fluid density.

Now the first methodology is the Omega method. This method 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. Equation of a state they are much popular in thermodynamic because they represent the correlation of measurable to non-measurable quantity.

Now, there are some point needs to be remembered: One is that the care should be taken while calculating data as equation developed are based on the analytical evaluation of the mass flux integral using an approximate linearize two-phase equation of the state for the fluid density.

(Refer Slide Time: 12:53)

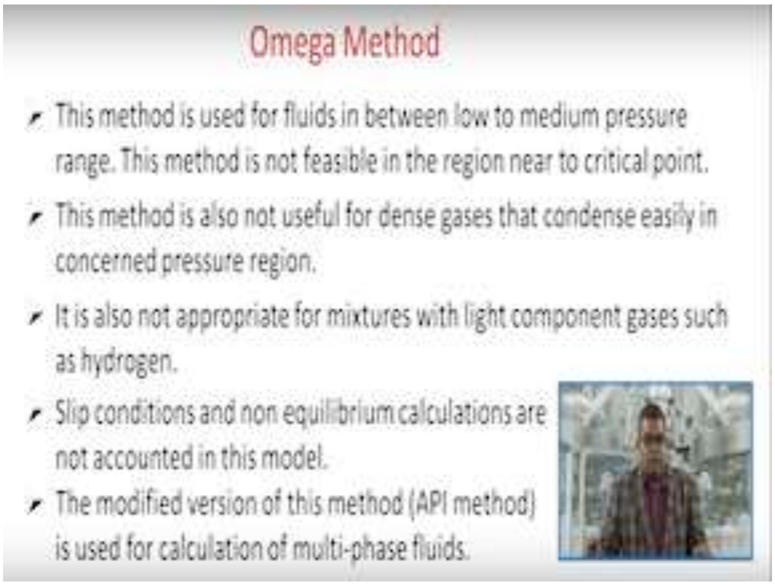


Omega Method

- Accurate thermodynamic and physical data is required, as these data are used only at initial step of calculation, while further calculation are based on the results obtained from the first step.
- As this method assumes single phase calculation, the linearized equation developed may not give accurate two-phase density vs pressure data for some conditions due to extrapolation of data.
- The accuracy depends on the nozzle condition and nature of the fluid as well as to the pressure range where nozzle is functioning.

Another is that accurate thermodynamic and physical data is required. As these data are used only at initial step of calculation while further calculation are based on the result obtained from the first step which we have discussed in the previous slide. Now as this method assumes a single-phase calculation the linearized equation developed may 0 give accurate two-phased density versus pressure data for some conditions due to the extra pollution of data. So this is again a very important point. Now the accuracy depends on the nozzle condition and nature of the fluid as well as to the pressure range where nozzle is functioning.

(Refer Slide Time: 13:28)



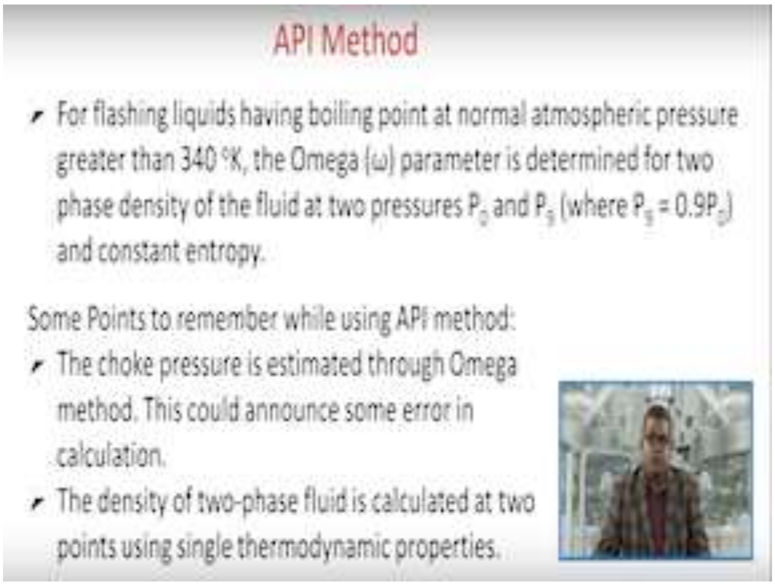
Omega Method

- This method is used for fluids in between low to medium pressure range. This method is not feasible in the region near to critical point.
- This method is also not useful for dense gases that condense easily in concerned pressure region.
- It is also not appropriate for mixtures with light component gases such as hydrogen.
- Slip conditions and non equilibrium calculations are not accounted in this model.
- The modified version of this method (API method) is used for calculation of multi-phase fluids.

Now, this method is used for fluids in between low to medium pressure range. Now, this method is not at all feasible in the region near to the critical point. With this particular point we have discussed in the previous diagram. Now, this method is also 0 useful for dense gases that condense easily in constant pressure region so we have discussed the pressure region in that particular diagram.

No it is also not appropriate for mixtures with light component gases such as hydrogen etc. The slip conditions and non-equilibrium calculations are 0 accounted in that particular model. So appropriate correction need to be incorporated while we go for higher calculation. Now this, the modified version of this method, this particular method that is the omega method, API method is used for the calculation of multiphase fluids.

(Refer Slide Time: 14:30)



API Method

- For flashing liquids having boiling point at normal atmospheric pressure greater than 340 °K, the Omega (ω) parameter is determined for two phase density of the fluid at two pressures P_0 and P_9 (where $P_9 = 0.9P_0$) and constant entropy.

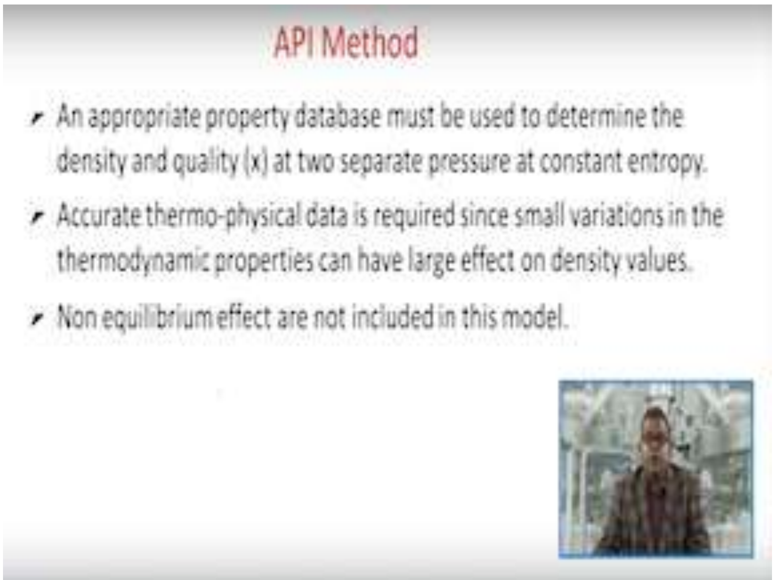
Some Points to remember while using API method:

- The choke pressure is estimated through Omega method. This could announce some error in calculation.
- The density of two-phase fluid is calculated at two points using single thermodynamic properties.

Now, let us have a discussion about the API, the API stands for American Petroleum Institute, so API method. For flashing liquid having the boiling point at normal atmospheric pressure greater than 340 kelvin, the omega parameter is determined for two-phased density of the fluid at two pressure P_0 and P9. Now where P9 is equal to $0.9P_0$ and the constant entropy. So entropy remains constant throughout.

Now, there are some point need to remember while we use this API method. So number one is that the choke pressure is estimated through omega method and this could announce some error in the calculation. So, while we go for the precision and then we need to due consideration for those errors. Now, second point is that the density of two-phase fluid is calculated at two point using single thermodynamic properties.

(Refer Slide Time: 15:38)



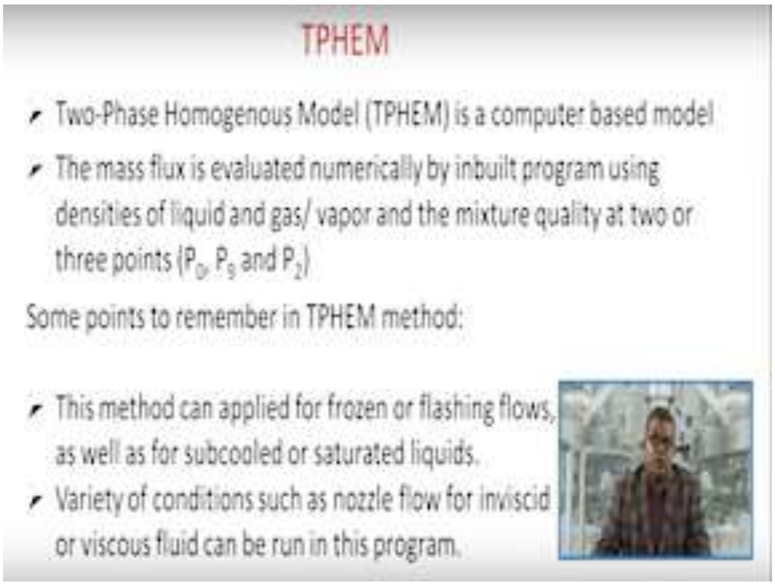
API Method

- An appropriate property database must be used to determine the density and quality (x) at two separate pressure at constant entropy.
- Accurate thermo-physical data is required since small variations in the thermodynamic properties can have large effect on density values.
- Non equilibrium effect are not included in this model.

The third point is that an appropriate property database must be used to determine the density and quality that is represented by X at two separate pressure at constant entropy. The ultimate point is that accurate thermo physical data is required since small variation in the thermodynamic properties can have large effect on the density value.

Now this is again very crucial the reason is that sometimes small value variation in any of the thermodynamic property may have a large impact on the density. So we need to be very careful while using this particular aspect. The last point is that non-equilibrium effects are not included in this particular model. So, we have to give a due consideration for all those points while calculating this particular aspect.

(Refer Slide Time: 16:31)



TPHEM

- Two-Phase Homogenous Model (TPHEM) is a computer based model
- The mass flux is evaluated numerically by inbuilt program using densities of liquid and gas/ vapor and the mixture quality at two or three points (P_0, P_9 and P_2)

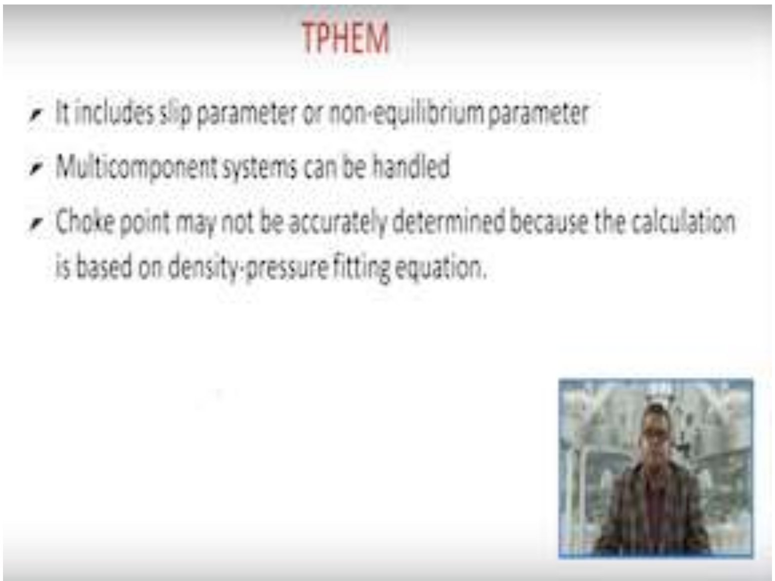
Some points to remember in TPHEM method:

- This method can applied for frozen or flashing flows, as well as for subcooled or saturated liquids.
- Variety of conditions such as nozzle flow for inviscid or viscous fluid can be run in this program.

Now next is the two-phase homogeneous model referred as TPHEM. Now this two-phase homogenous model is a computer based model and the mass flux is evaluated numerically by inbuilt program using density of liquid and gas/vapor and the mixture quality at two or three different points that is P_0, P9 and P2.

There are some point which we need to be remember while using the TPHEM method. Number one is that this model can applied for frozen or flashing flows, as well as for the sub-cooled or saturated liquids. Now variety of conditions such as the nozzle flow for in viscid or viscous fluid can be done in the program.

(Refer Slide Time: 17:29)

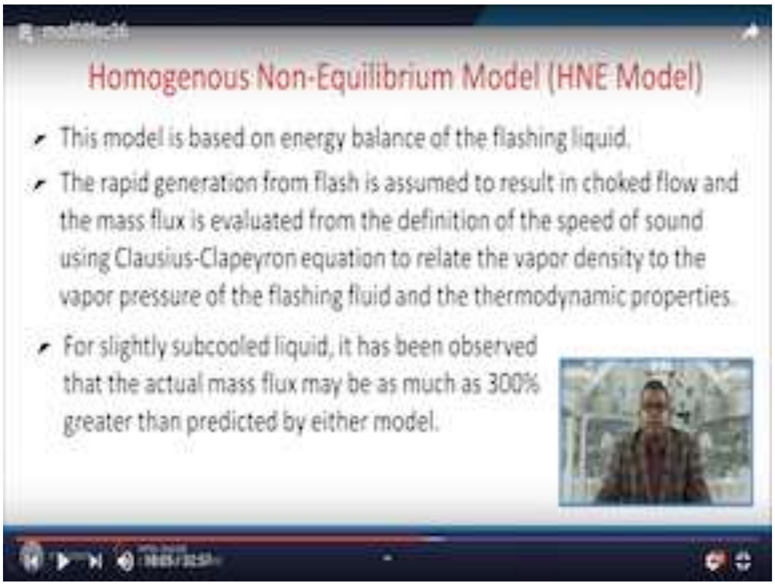


TPHEM

- It includes slip parameter or non-equilibrium parameter
- Multicomponent systems can be handled
- Choke point may not be accurately determined because the calculation is based on density-pressure fitting equation.

The third point is that it includes slip parameter or non-equilibrium parameter. Fourth is that multicomponent system can be handled in this particular program and the last one is that choke point may not be accurately determined because the calculation is based on density pressure fitting equations. So, once we have this particular equation then we have to give due consideration of this last point.

(Refer Slide Time: 18:05)



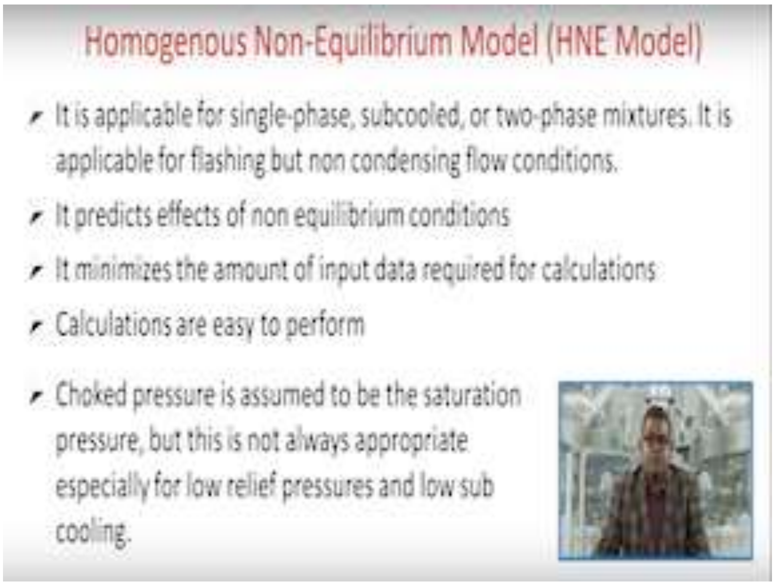
Homogenous Non-Equilibrium Model (HNE Model)

- This model is based on energy balance of the flashing liquid.
- The rapid generation from flash is assumed to result in choked flow and the mass flux is evaluated from the definition of the speed of sound using Clausius-Clapeyron equation to relate the vapor density to the vapor pressure of the flashing fluid and the thermodynamic properties.
- For slightly subcooled liquid, it has been observed that the actual mass flux may be as much as 300% greater than predicted by either model.

Another model is the homogeneous non-equilibrium model that is HNE model. Now this particular model is based on energy balance of the flashing liquid. We have encountered, we have discussed the flashing concept in the previous slides. Now the rapid generation from 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.

So whatever thermodynamic properties are required we can use this particular model. For slightly sub-cool liquid it has been observed that the actual mass flux maybe assumed as 300 percent greater than the predicted by any of the model which we have discussed previously.

(Refer Slide Time: 18:58)



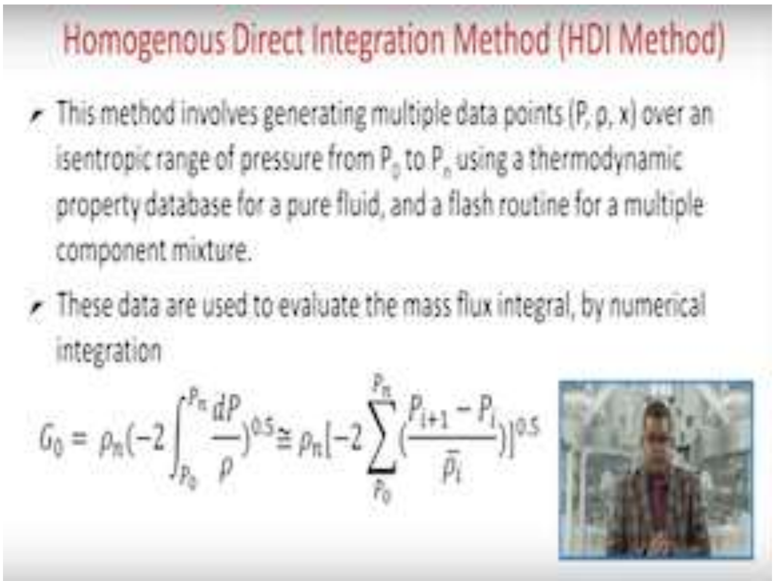
Homogenous Non-Equilibrium Model (HNE Model)

- It is applicable for single-phase, subcooled, or two-phase mixtures. It is applicable for flashing but non condensing flow conditions.
- It predicts effects of non equilibrium conditions
- It minimizes the amount of input data required for calculations
- Calculations are easy to perform
- Choked pressure is assumed to be the saturation pressure, but this is not always appropriate especially for low relief pressures and low sub cooling.

So it is applicable for single phase, sub-cooled or two–phase mixture. Now it is applicable for flashing but non-condensing flow conditions so be particular about this, the use of this particular model. Now, it predicts effects of non-equilibrium conditions. Usually it minimizes the amount of input data required for calculations.

Usually calculations are extremely easy to perform. The choked pressure is assumed to be the saturation pressure, but this is not always appropriate especially for low relief pressure and low sub-cooling. So while using this particular model you must know that what is the concept of low sub-cooling and low relief pressure.

(Refer Slide Time: 20:03)



Homogenous Direct Integration Method (HDI Method)

- This method involves generating multiple data points (P, ρ, x) over an isentropic range of pressure from P_0 to P_n using a thermodynamic property database for a pure fluid, and a flash routine for a multiple component mixture.
- These data are used to evaluate the mass flux integral, by numerical integration

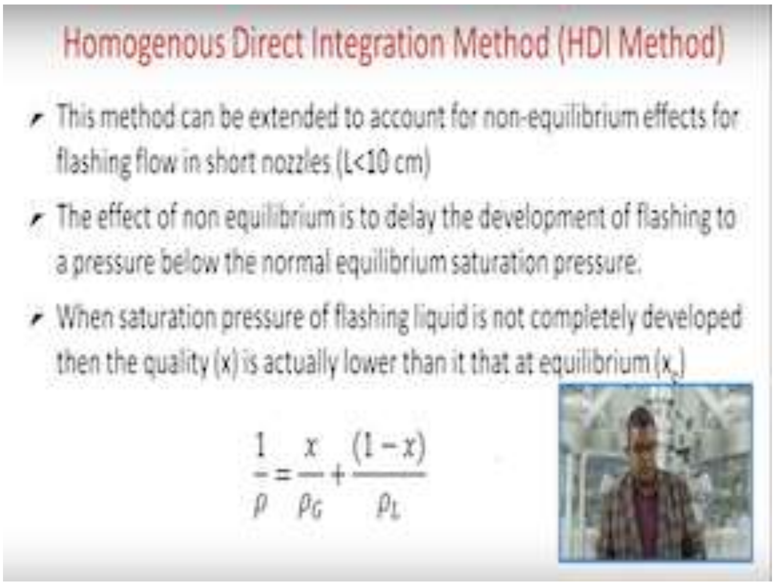
$$G_0 = \rho_n(-2\int_{P_0}^{P_n}\frac{dP}{\rho})^{0.5} \cong \rho_n[-2\sum_{P_0}^{P_n}(\frac{P_{i+1}-P_i}{\bar{\rho_i}})]^{0.5}$$

Now another method is Homogeneous Direct Integration method sometimes referred as HDI method. Now this method involves generating multiple data points, P, ρ, X that is a dryness factor, over an isentropic range of pressure from P_0 to Pn using a thermodynamic property database for a pure fluid and a flash routine for a multiple component mixture.

So these data are used to evaluate the mass flux integral by numerical integration of this particular equation. Now,

$$G_0 = \rho_n(-2\int_{P_0}^{P_n}\frac{dP}{\rho})^{0.5} \cong \rho_n[-2\sum_{P_0}^{P_n}(\frac{P_{i+1}-P_i}{\bar{\rho}_i})]^{0.5}$$

(Refer Slide Time: 21:04)



Homogenous Direct Integration Method (HDI Method)

- This method can be extended to account for non-equilibrium effects for flashing flow in short nozzles (L<10 cm)
- The effect of non equilibrium is to delay the development of flashing to a pressure below the normal equilibrium saturation pressure.
- When saturation pressure of flashing liquid is not completely developed then the quality (x) is actually lower than it that at equilibrium (x_e)

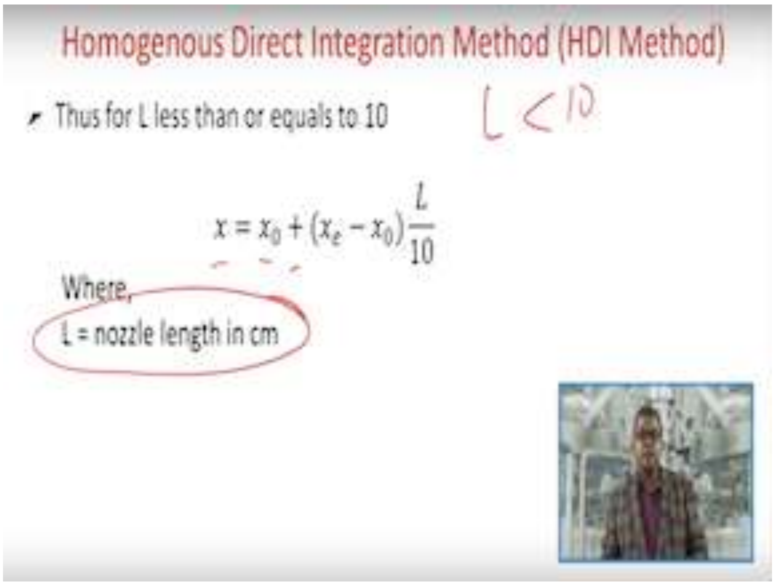
$$\frac{1}{\rho} = \frac{x}{\rho_G} + \frac{(1-x)}{\rho_L}$$

Now this method can be extended to account for non-equilibrium effect for flashing flow in short nozzle, so for flashing flow we can take L is less than 10 cm that is length. The effect of non-equilibrium is to delay the development of flashing to pressure below the normal equilibrium saturation pressure, so when saturation pressure of flashing liquid is not completely developed then the quality factor that if X is actually lower than that at the equilibrium that is X_e.

So we can represent this aspect by the following equation that is

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

(Refer Slide Time: 22:08)



Homogenous Direct Integration Method (HDI Method)

- Thus for L less than or equals to 10

$L < 10$

$$x = x_0 + (x_e - x_0)\frac{L}{10}$$

Where,

L = nozzle length in cm

So if you take that L is less than 10 then

$$x = x_0 + (x_e - x_0)\frac{L}{10}$$

Where L is having the nozzle length in centimeters.

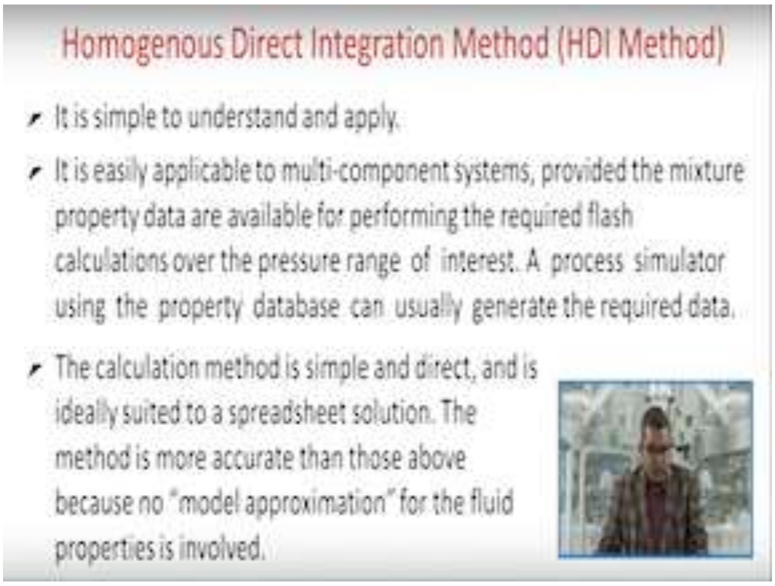
(Refer Slide Time: 22:16)

Homogenous Direct Integration Method (HDI Method)

- 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.

This model is rigorous within the assumption inherent in the ideal nozzle equation and the HEM assumptions and the precision of the property data, which you can have from various handbooks. Now, it is universally applicable for all fluids under any or all conditions, which are applicable for which we are having the property data available. Now, the procedure, this particular procedure does not depend on whether the entering fluid is cold liquid, sub-cooled flashing liquid, or condensing vapor or a two-phase mixture whether or not the flow is choked.

(Refer Slide Time: 22:57)



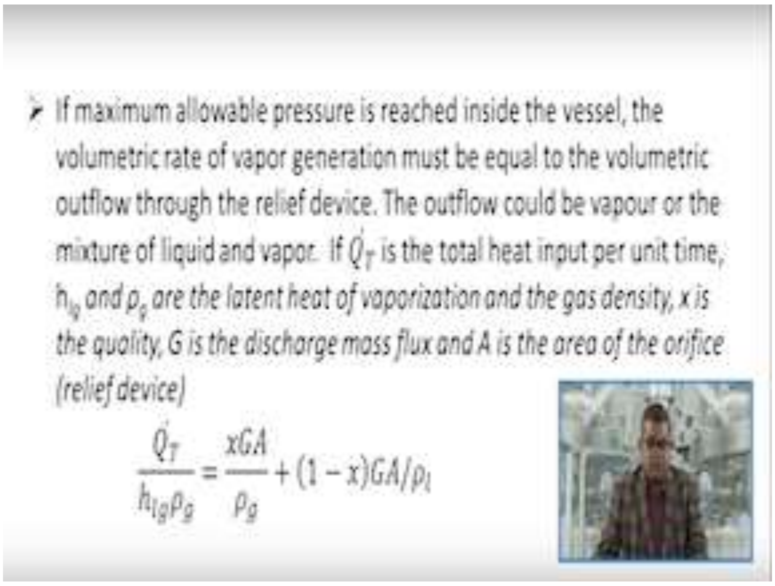
Homogenous Direct Integration Method (HDI Method)

- 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.

So it is simple to understand and apply, obviously which is evident from the previous calculation. Now, it is easily applicable to multicomponent system provided the mixture property data are available for performing the required flash calculation over the pressure range of interest, whatever it is given.

Now a process simulator using the property database can be usually generate data. The calculation method is usually a simple and direct and is ideally suited to the spreadsheet solution, so the method is more accurate than those above because no model approximation for the fluid properties is involved.

(Refer Slide Time: 23:48)



- If maximum allowable pressure is reached inside the vessel, the volumetric rate of vapor generation must be equal to the volumetric outflow through the relief device. The outflow could be vapour or the mixture of liquid and vapor. If $\dot{Q}_T$ is the total heat input per unit time, h_{lg} *and* ρ_g *are the latent heat of vaporization and the gas density, x is the quality, G is the discharge mass flux and A is the area of the orifice (relief device)*

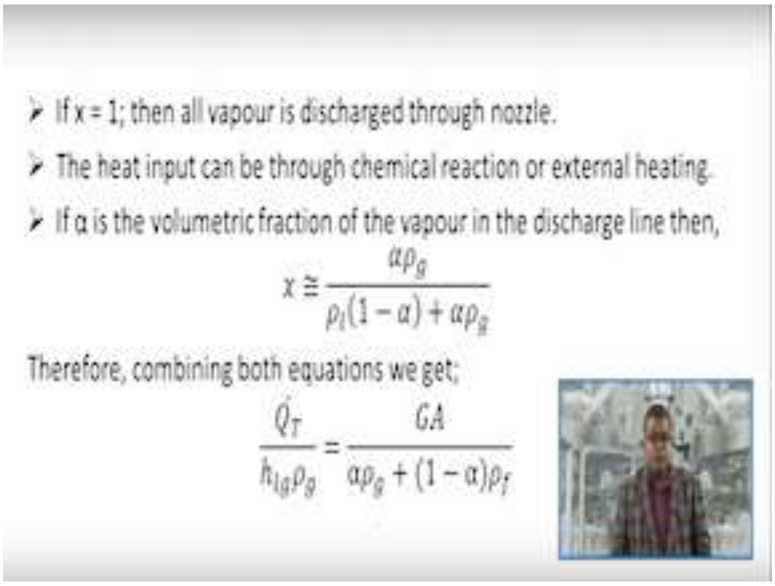
$$\frac{\dot{Q}_T}{h_{lg}\rho_g} = \frac{xGA}{\rho_g} + (1-x)GA/\rho_l$$

Let us have a look about the sizing calculations. Now, sizing calculation we use this certain set of equations. We are going through all those equations, which are important. Now, if maximum allowable pressure is reached inside the vessel the volumetric rate of vapor generation must be equal to the volumetric outflow through the relief device; that is the rule of thumb. Now the outlook could be vapor or a mixture of liquid and vapor.

Now, if Q_T is the total heat input per unit time and H_{lg} and ρ_g are the latent heat of a vaporization, lg stands from liquid to gaseous system and ρ_g is the gas density, x is the quality and sometime it is referred as dryness factor. Now G is the discharge mass flux and A is the area of the orifice that is attributed to the relief device. Then

$$\frac{\dot{Q}_T}{h_{lg}\rho_g} = \frac{xGA}{\rho_g} + (1-x)GA/\rho_l$$

(Refer Slide Time: 24:59)



- If x = 1; then all vapour is discharged through nozzle.
- The heat input can be through chemical reaction or external heating.
- If α is the volumetric fraction of the vapour in the discharge line then,

$$x \cong \frac{\alpha\rho_g}{\rho_l(1-\alpha)+\alpha\rho_g}$$

Therefore, combining both equations we get;

$$\frac{\dot{Q}_T}{h_{lg}\rho_g} = \frac{GA}{\alpha\rho_g+(1-\alpha)\rho_f}$$

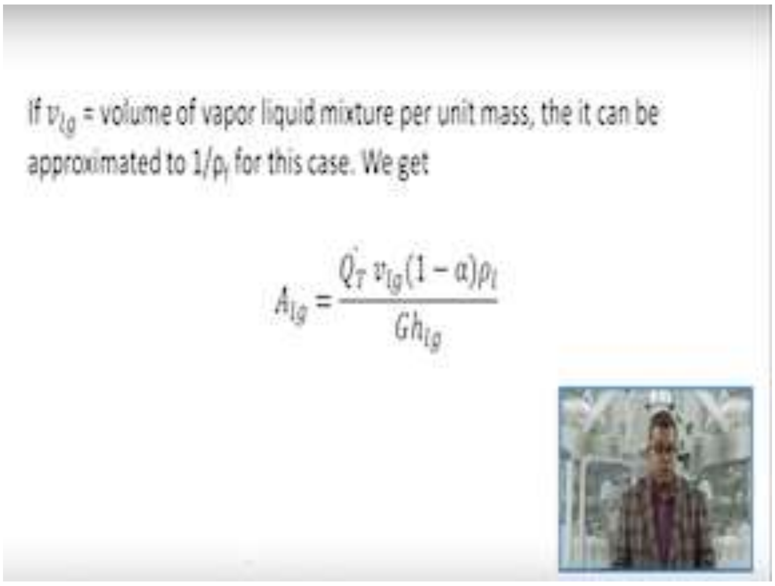
Now, if let us take that x is equal to 1 that is all vapor is this chat through the nose and that is there is no liquid fraction etc. The heat input can be through chemical reaction or external heating, this is again given or sometimes we have to take the assumption. Now, if alpha is the volumetric fraction of the vapor in the discharge line then

$$x \cong \frac{\alpha\rho_g}{\rho_l(1-\alpha)+\alpha\rho_g}$$

So, therefore, if we combine both the equation we get

$$\frac{\dot{Q}_T}{h_{lg}\rho_g} = \frac{GA}{\alpha\rho_g+(1-\alpha)\rho_f}$$

(Refer Slide Time: 25:46)



If v_{lg} = volume of vapor liquid mixture per unit mass, the it can be approximated to $1/\rho_l$ for this case. We get

$$A_{lg} = \frac{\dot{Q}_T\, v_{lg}(1-\alpha)\rho_l}{G h_{lg}}$$

So, upon simplification

$$A = \frac{\dot{Q_T}\,[\alpha\rho_g + (1-\alpha)\rho_l]}{Gh_{lg}\rho_g}$$

Now if you take alpha is equal to 1 that is for all vapor discharge then

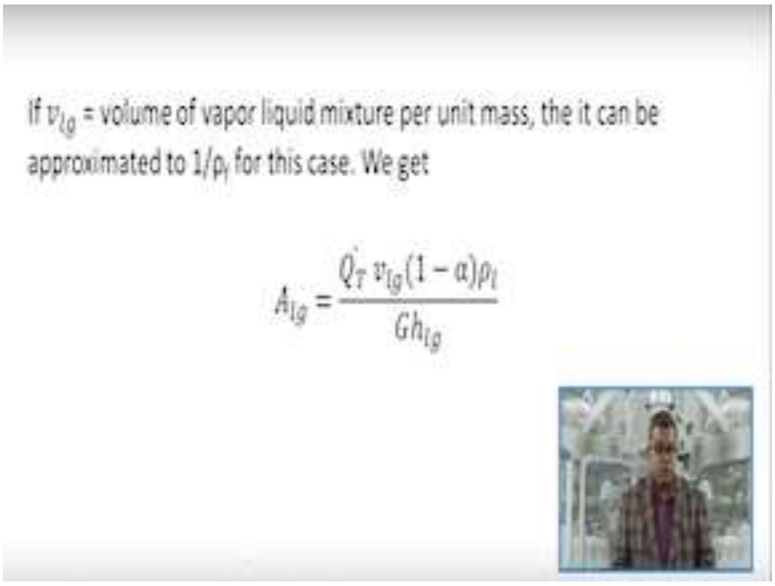
$$A_g = \frac{\dot{Q}_T}{Gh_{lg}}$$

that is enthalpy from liquid to gaseous phase. Now for two-phase discharge this equation can be simplified and if we take up early one assumption that is ρ_1 is greater than greater than ρ_g, then $\alpha\rho_g$ can be summarized to 0

then this equation comes out to be

$$A_{lg} = \frac{\dot{Q_T}\,(1-\alpha)\rho_l}{Gh_{fg}\rho_g}$$

(Refer Slide Time: 26:46)



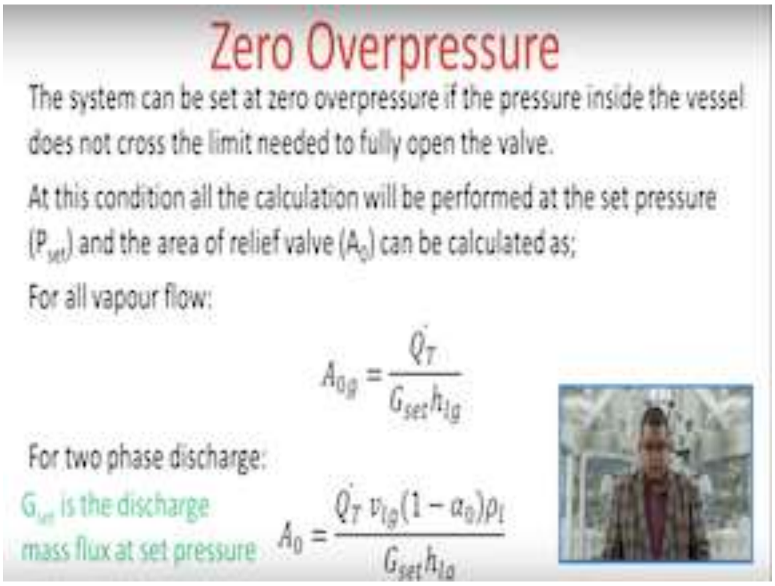
If v_{lg} = volume of vapor liquid mixture per unit mass, the it can be approximated to $1/\rho_l$ for this case. We get

$$A_{lg} = \frac{\dot{Q_T}\,v_{lg}(1-\alpha)\rho_l}{Gh_{lg}}$$

So, if we take V_{lg} is equal to the volume of vapor liquid mixture per unit mass then it can be approximated to $1/\rho_f$ for this particular case. So we get

$$A_{lg} = \frac{\dot{Q_T}\,v_{lg}(1-\alpha)\rho_l}{Gh_{lg}}$$

(Refer Slide Time: 27:10)



Zero Overpressure

The system can be set at zero overpressure if the pressure inside the vessel does not cross the limit needed to fully open the valve.

At this condition all the calculation will be performed at the set pressure (P_{set}) and the area of relief valve (A_0) can be calculated as;

For all vapour flow:

$$A_{0g} = \frac{\dot{Q}_T}{G_{set} h_{lg}}$$

For two phase discharge:

G_{set} is the discharge mass flux at set pressure

$$A_0 = \frac{\dot{Q}_T \, v_{lg}(1-\alpha_0)\rho_l}{G_{set} h_{lg}}$$

Now, let us understand the concept of Zero Over Pressure. In this particular aspect of the system can be set at zero over pressure if the pressure inside the vessel does not cross the limit needed to the fully open valve. So at this condition all the calculation will be performed at the set pressure and area of relief valve A_0.

This can be calculated as if you take the condition of all vapor flow then

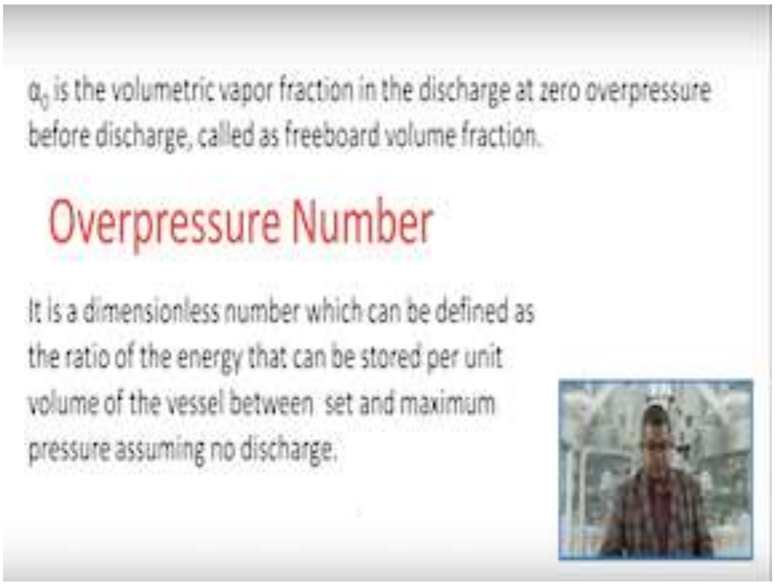
$$A_{0g} = \frac{\dot{Q}_T}{G_{set} h_{lg}}$$

So we can calculate this one. Now, if you take the condition of two-phased discharge then

$$A_0 = \frac{\dot{Q}_T \, v_{lg}(1-\alpha_0)\rho_l}{G_{set} h_{lg}}$$

Where G_{set} is the discharge mas flux at set pressure.

(Refer Slide Time: 28:08)



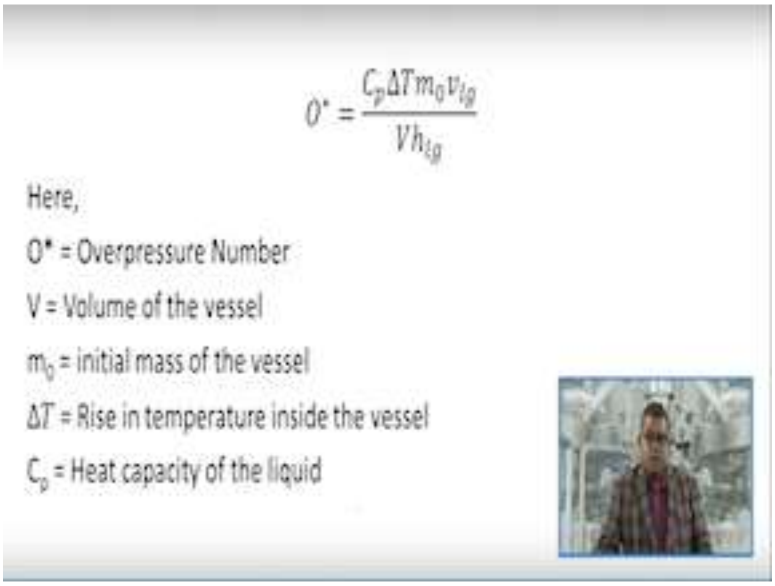
α_0 is the volumetric vapor fraction in the discharge at zero overpressure before discharge, called as freeboard volume fraction.

Overpressure Number

It is a dimensionless number which can be defined as the ratio of the energy that can be stored per unit volume of the vessel between set and maximum pressure assuming no discharge.

α_0 is the volumetric vapor fraction in the discharge at zero overpressure before discharge and this is called as the freeboard volume fraction. Now, there is overpressure number, it is the dimensional less number which can be defined as the ratio of the energy that can be stored per unit volume of the vessel between set and maximum pressure assuming no discharge. We may assume that there is no discharge at all.

(Refer Slide Time: 28:38)



$$O^* = \frac{C_p \Delta T m_0 v_{lg}}{V h_{lg}}$$

Here,

O^* = Overpressure Number

V = Volume of the vessel

m_0 = initial mass of the vessel

ΔT = Rise in temperature inside the vessel

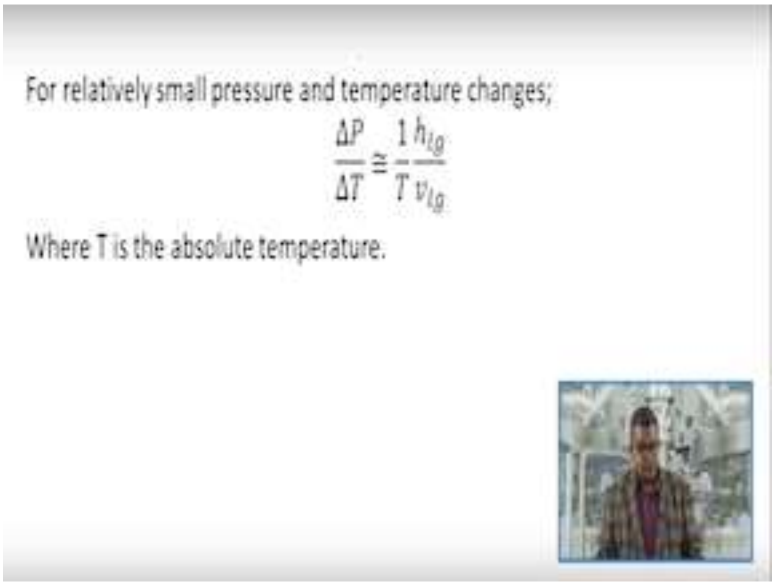
C_p = Heat capacity of the liquid

So, it is

$$O^* = \frac{C_p \Delta T m_0 v_{lg}}{V h_{lg}}$$

So where this (O^*)is the overpressure number, V is the volume of the vessel, m_0 is the initial mass of the vessel, ΔT is the rise in the temperature inside the vessel and Cp is the heat capacity of the liquid.

(Refer Slide Time: 29:07)



For relatively small pressure and temperature changes;

$$\frac{\Delta P}{\Delta T} \cong \frac{1}{T}\frac{h_{lg}}{v_{lg}}$$

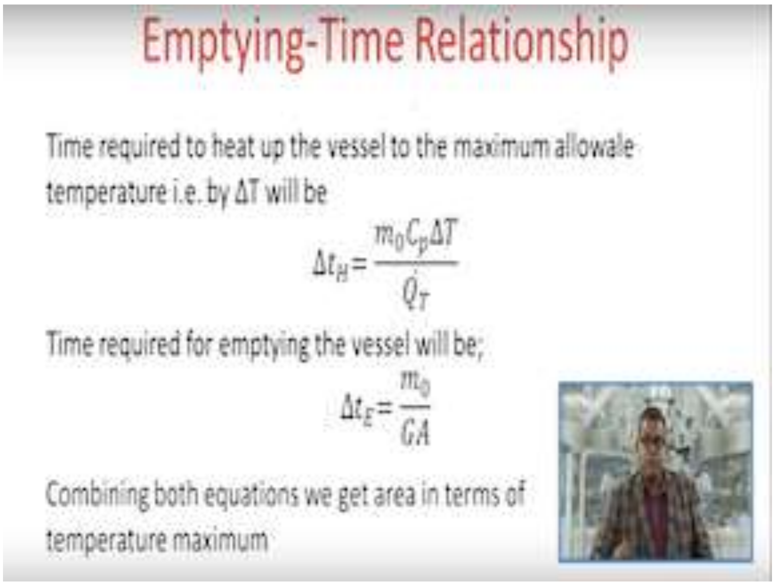
Where T is the absolute temperature.

So if you take the relatively small pressure and temperature change then this equation reduces to

$$\frac{\Delta P}{\Delta T} \cong \frac{1}{T}\frac{h_{lg}}{v_{lg}}$$

Where T is the absolute temperature. It is in correlation with the Clausius-Clapeyron equation.

(Refer Slide Time: 29:29)



Emptying-Time Relationship

Time required to heat up the vessel to the maximum allowale temperature i.e. by ΔT will be

$$\Delta t_H = \frac{m_0 C_p \Delta T}{\dot{Q}_T}$$

Time required for emptying the vessel will be;

$$\Delta t_E = \frac{m_0}{GA}$$

Combining both equations we get area in terms of temperature maximum

Now, let us have a look about the Emptying Time Relationship. Now, the time required to heat of the vessel to the maximum allowable temperature that is by ΔT will be

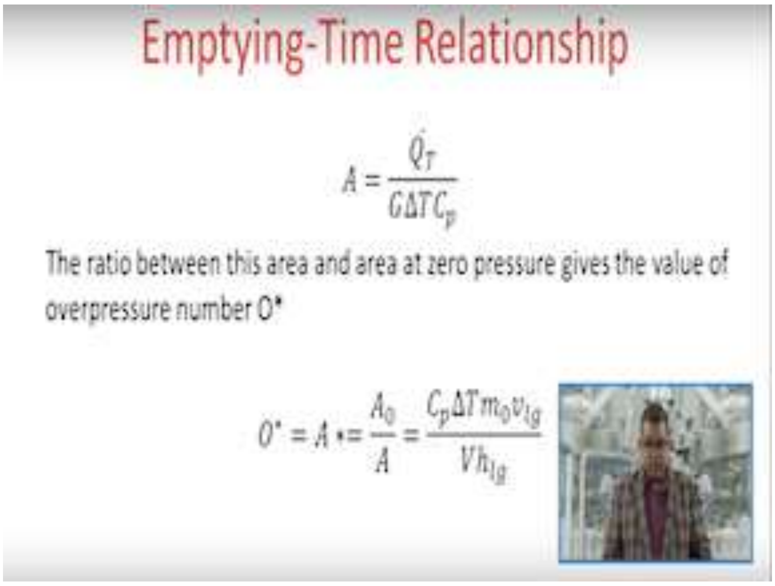
$$\Delta t_H = \frac{m_0 C_p \Delta T}{\dot{Q}_T}$$

The time required, this is the time required for emptying the vessel would be

$$\Delta t_E = \frac{m_0}{GA}$$

If we combine both the equations we get area in terms of temperature maximum.

(Refer Slide Time: 30:04)



Emptying-Time Relationship

$$A = \frac{\dot{Q}_T}{G \Delta T C_p}$$

The ratio between this area and area at zero pressure gives the value of overpressure number O*

$$O^* = A * = \frac{A_0}{A} = \frac{C_p \Delta T m_0 v_{lg}}{V h_{lg}}$$

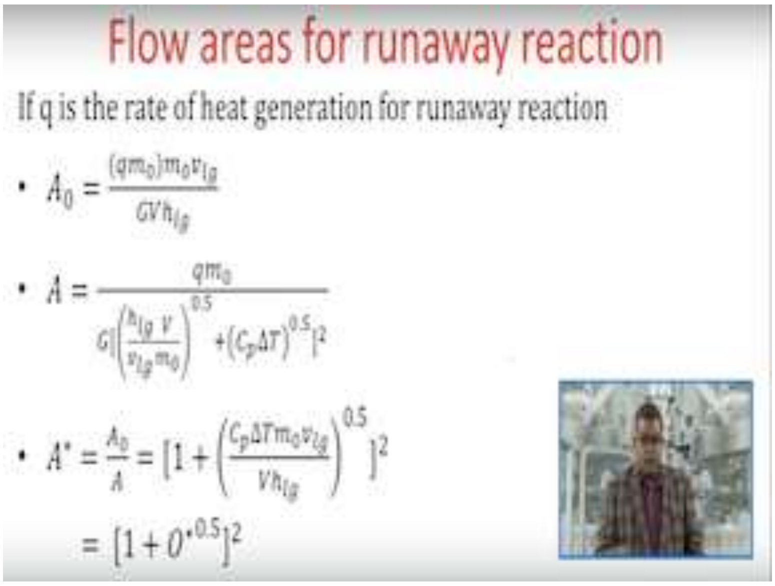
And that is

$$A = \frac{\dot{Q}_T}{G\Delta T C_p}$$

So the ratio between this area and the area at zero pressure gives the value of overpressure number. So we can redefine the overpressure number that is

$$O^* = A *= \frac{A_0}{A} = \frac{C_p \Delta T m_0 v_{lg}}{V h_{lg}}$$

(Refer Slide Time: 30:32)



Flow areas for runaway reaction

If q is the rate of heat generation for runaway reaction

- $A_0 = \frac{(qm_0)m_0 v_{lg}}{GVh_{lg}}$
- $A = \frac{qm_0}{G[\left(\frac{h_{lg}\,V}{v_{lg}m_0}\right)^{0.5} + (C_p\Delta T)^{0.5}]^2}$
- $A^* = \frac{A_0}{A} = [1 + \left(\frac{C_p\Delta T m_0 v_{lg}}{Vh_{lg}}\right)^{0.5}]^2$

$= [1 + O^{*0.5}]^2$

Now, let us take the flow area of runaway reaction. Now, if q is the rate of heat generation for runaway reaction then

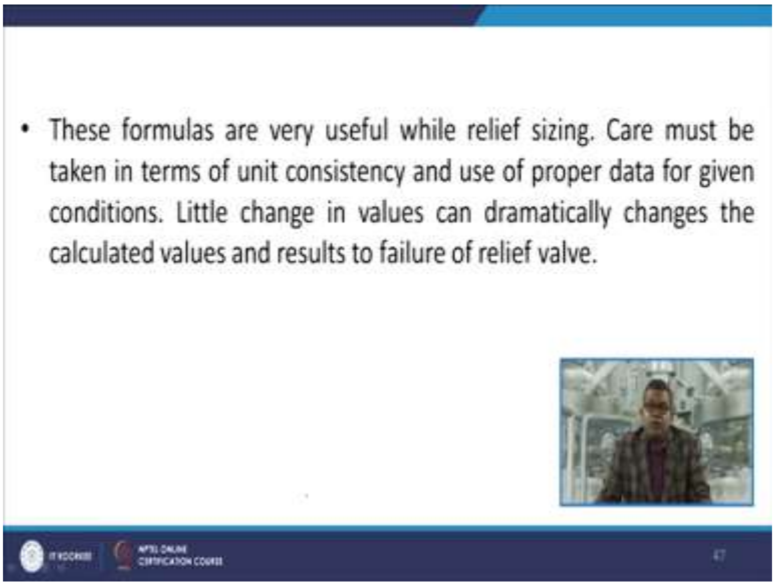
$$A_0 = \frac{(qm_0)m_0 v_{lg}}{GVh_{lg}}$$

$$A = \frac{qm_0}{G[\left(\frac{h_{lg}}{v_{lg}}\frac{V}{m_0}\right)^{0.5} + \left(C_p \Delta T\right)^{0.5}]^2}$$

$$A^* = \frac{A_0}{A} = [1 + \left(\frac{C_p \Delta T m_0 v_{lg}}{V h_{lg}}\right)^{0.5}]^2$$

$$= [1 + O^{*0.5}]^2$$

(Refer Slide Time: 31:24)



- These formulas are very useful while relief sizing. Care must be taken in terms of unit consistency and use of proper data for given conditions. Little change in values can dramatically changes the calculated values and results to failure of relief valve.

So these formula are very useful while relief sizing and care must be taken in terms of unit consistency and use of proper data for given condition. Now little change in the value can drastically change the calculated values and results to failure of the relief work and this may be extremely catastrophic. So first thing must be in mind that we must adhere with the consistency of the unit and we must use the proper data for selection.

(Refer Slide Time: 32:16)

References

1. Banerjee, S. Industrial Hazards and Plant Safety; Taylor & Francis, 2003.
2. Darby, R. Size Safety-Relief Valves for Any Conditions. Chem. Eng. 2005, 112 (9).
3. Sizing, Selection, and Installation of Pressure-Relieving Devices in Refineries Part I-Sizing and Selection API RECOMMENDED PRACTICE 520 SEVENTH EDITION, JANUARY 2000.

Now, in this particular module we had a discussion about the relief sizing. Different type of model we have discussed and different type of equation we have developed for the proper relief sizing. Now, if you wish to have a further reading there are references, which are enlisted, in this particular slide. Thank you very much.