ADVANCED HYBRID MODELLING OF SEPARATORS FOR SAFE DESIGN IN OIL/GAS PRODUCTION PLANTS

James Marriott,
Process Systems Enterprise Limited, UK

The use of advanced process modelling for safe vessel design can result in a considerable reduction in capital expenditure while simultaneously improving plant safety.

This paper considers the requirements and challenges for transient modelling of separators in oil / gas production plants using an HP flare KO drum as an example. These devices must be able to withstand high two-phase flowrates that may enter, through several nozzles, at very cold temperatures for approximately 15 minutes. The large vessel size usually means that average wall temperatures are well above material embrittlement temperatures. However, effects such as cold jet impingement on vessel walls and liquid droplet entrainment can create local cold spots on the vessel walls that threaten its integrity.

This paper presents an efficient and tested solution that accurately describes the transient nature of the feeds to the drum and the resulting dynamic response of the vessel walls at all locations throughout the vessel. The model combines a representation of the complex fluid flow behaviour taking place in the vessel (computational fluid dynamics) and a rigorous thermodynamic description of the fluid (accounting for the disproportionate impact of liquid droplets on wall temperatures), to ensure that all relevant effects are considered in the analysis.

1. INTRODUCTION

Safety – in the context of this paper, maintaining vessel integrity during low-temperature operation – is paramount for all Oil & Gas operations. However, mitigating against the effects of low temperature can be very expensive in terms of capital cost associated with low-temperature alloys, or the reduced production resulting from, for example, the need to depressurise a flowline slowly in order to avoid low-temperature situations. It is thus very important to quantify integrity risks very accurately, in order to ensure safe operation but not over-design vessels unnecessarily.

Particular consideration must be given to transient operations in oil–gas processing plants. For example, during plant blowdown, separators such as high-pressure (HP) knock-out drums must be able to withstand two-phase inlet flows at high flowrates and very cold temperatures for periods of up to 15 minutes.

Such flows can create areas of localised low temperature in the metal walls of the vessel, which may exceed safe limits for embrittlement temperature. Flows at different temperatures may enter through several nozzles, creating temperature differentials between adjacent areas on the vessel wall, leading to unacceptable thermal stresses. Both types of hazard can threaten the integrity of the vessel as a whole.

However, the thermal inertia of the large metal mass is often considered to be sufficient to compensate for the effects of the inlet cold flows, preventing the most affected areas of the vessel from reaching unsafe temperatures. In order to make decisions about safety that simultaneously ensure safe design and operation and also minimise capital costs, it is increasingly necessary to quantify the effects of such events very accurately.

This paper shows examples of current approaches and their shortcomings, and describes a significantly more effective approach that combines hybrid Computational Fluid Dynamics (CFD) and dynamic vessel wall models to ensure both safe design and minimised capital expenditure.

2. CASE STUDY

A HP flare system knock-out drum is used as an illustrative example. The flare network flowsheet is shown in Figure 1 and a typical configuration for a flare network knock-out drum is shown in Figure 2.

During blowdown both warm and cold flows enter the top of the separator via nozzles N1 and N2 (respectively); vapour and liquid separate in the disengagement space and liquid flows through the perforated baffles towards outlets at either end of the vessel.

During periods of high flow – for example, during system blowdown – cold jets of incoming fluid are directed at the vessel lateral walls.

For vapour-only flows, the cooling effect is controlled by the relatively low heat transfer coefficient. However if entrained liquids enter at high velocity, their momentum carries the droplets directly onto the vessel wall.

In such cases the combination of the high heat transfer coefficient between liquid and metal and the evaporative cooling effect resulting from vaporisation as droplets hit the walls can create significant localised cooling.

Blowdown temperature curves for both wet (warm) and cold header flows into nozzles N1 and N2 respectively are shown in Figure 3.

The curves illustrate the two potential hazards described above:

- the temperature of the metal affected by the cold flow may drop to near or below embrittlement temperature
- the large relative temperature between the metal affected by the cold flow and the surrounding metal wall can give rise to thermal stresses.

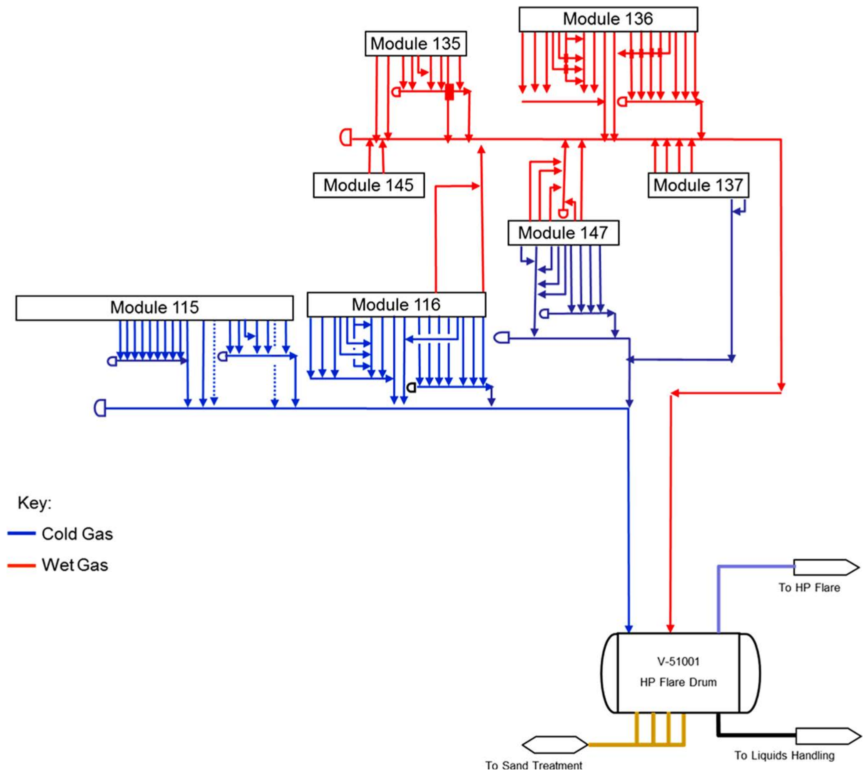

Figure 1. Example configuration: HP Flare system KO drum

3. MODEL-BASED APPROACHES – FROM SIMPLE TO DETAILED

Process modelling is an essential tool for analysing the thermodynamic and related thermal (i.e. temperature) effects occurring during events such as blowdown. A model-based approach solves the mathematical equations describing the physical behaviour of the process – mass and energy balances, thermodynamic and hydraulic relationships, geometry descriptions, component physical property calculations, and so on – to predict important design and operational information such as minimum wall temperatures.

This paper describes typical current techniques and their advantages and disadvantages, and proposes a new approach that has been industrially proven to address the shortcomings of current techniques on several projects.

3.1 LUMPED-PARAMETER MODEL APPROACH

Current commercial practice is to use a 'lumped' dynamic model of the vessel and wall. The lumped approach assumes well-mixed fluids in the vessel; it does not take into account spatial variation within the vessel contents or temperature variations along the vessel walls. The temperature differential across the wall thickness is calculated.

Despite the simplifying assumptions, models are not necessarily simplistic. For example, heat transfer between the vessel metal and the fluid is typically based on rigorous forced convection (e.g. Colborn or Gnielinski) or natural convection (e.g. Churchill and Chu) correlations.

The results of the blowdown into the HP flare drum are shown in Figure 4, which plots the single average fluid and vessel wall inner and outer temperatures over the course of the event. Figure 5 shows the temperatures of the metal wall.

It can clearly be seen from the separator geometry in Figure 2 that the when cold fluid flows through either nozzle N1 or N2 there is potential for localised cold spots to occur on the vessel wall, and that the temperature is not necessarily uniform throughout the vessel walls.

However this is not evident in the temperature plots in Figure 5. Here the lumped approach shows uniform temperatures for the entire vessel wall at different stages of a blowdown event (2, 420 and 780 seconds).

The reading of results from this calculation provides the conclusion that the system is safe. However, it may be worth checking the results using an approach that takes into account localised conditions.

3.2 COMPUTATIONAL FLUID DYNAMICS (CFD) APPROACH

In order to address some of the questions arising from the lumped-parameter approach described above, additional studies are sometimes performed using Computational Fluid Dynamics (CFD) models. These take into account spatial

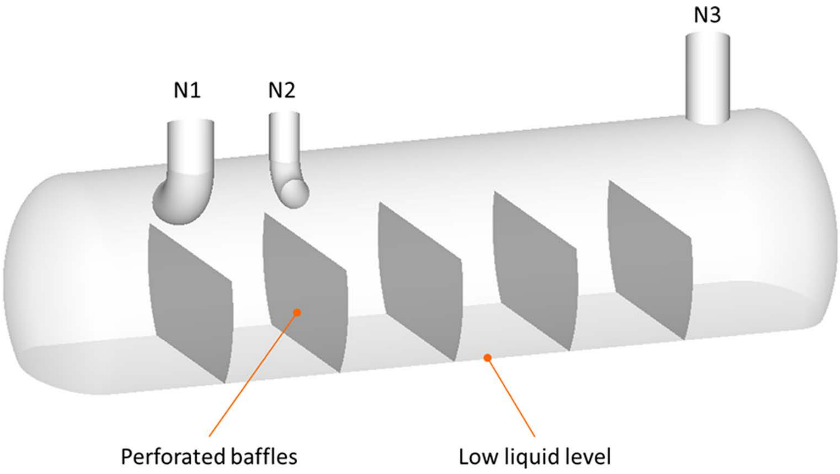

Figure 2. Typical HP Flare system KO drum configuration

variations by, for example, analysing the actual flows onto the vessel wall.

A CFD model of the separator describes the flow patterns of the incoming fluid, taking into account important geometry considerations such as the direction of the inlet flows, as well as the velocity of the flows and hydrodynamic effects such as turbulence. Computational limitations prevent modelling of the transient response of the vessel wall temperature.

Figure 6 shows the separator geometry and incoming flow directions for the same blowdown event in more detail.

The side view (upper figure) shows a relatively warm stream entering via the left-hand nozzle, and a smaller cold flow entering via the right-hand one. The top view (lower figure) shows the lateral direction of these flows as they leave the respective nozzles.

Figure 7 shows the corresponding streamlines for the flows generated by CFD analysis. It can be seen that both flows reach the vessel wall at significant velocity, impacting a relatively small area of the vessel wall.

Fluid temperature results determined from the CFD analysis are shown in Figure 8. The results are shown for steady-state snapshots calculations corresponding to the instantaneous conditions at the inlet nozzles from Figure 3. From these calculations, the fluid temperature at the coldest location, where the fluid strikes the vessel wall opposite nozzle N2, can be seen to drop to 210K after 200 seconds. The areas highlighted by the red circles in Figure 8 show the location where this occurs.

The calculated fluid temperatures are below 228K, the minimum allowable safe wall temperature – a potentially hazardous situation. Contrary to the analysis of 3.1,

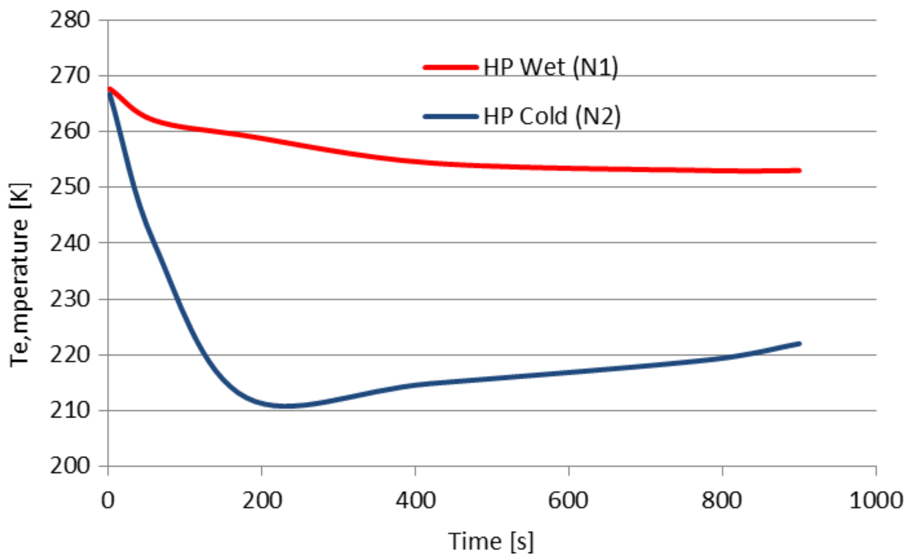

Figure 3. Temperatures of entering flows

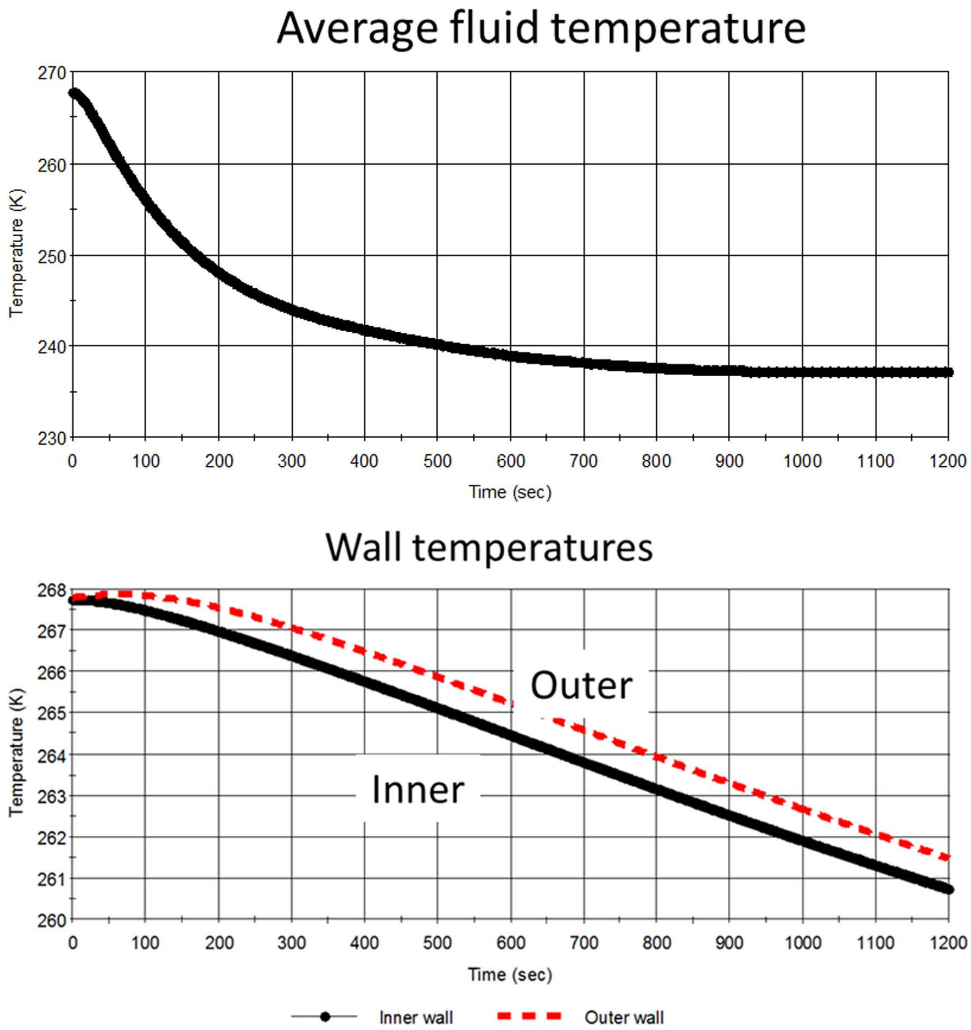

Figure 4. Temperature profiles over time for the lumped parameter model approach

if the wall temperature tracks the fluid temperature, then the conclusion is that the system could be unsafe. However, is this a valid conclusion? Does the thermal inertia of the metal mass in the vessel wall possibly compensate for the effect of the low temperatures of the vapour hitting the wall (as is commonly assumed)?

3.3 HYBRID APPROACH – COUPLED CFD AND DYNAMIC PROCESS MODELS

The CFD-based approach described in Section 3.2 typically uses a 'pseudo-dynamic' simulation, where the 'transient' behaviour is determined by performing a sequence of steady-state calculations. The lack of detailed wall modelling

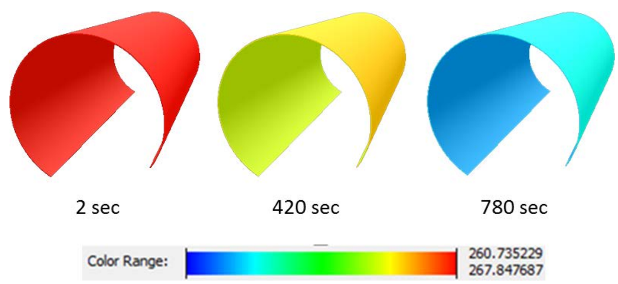

Figure 5. 'Lumped' approach results, showing uniform temperatures throughout the separator walls

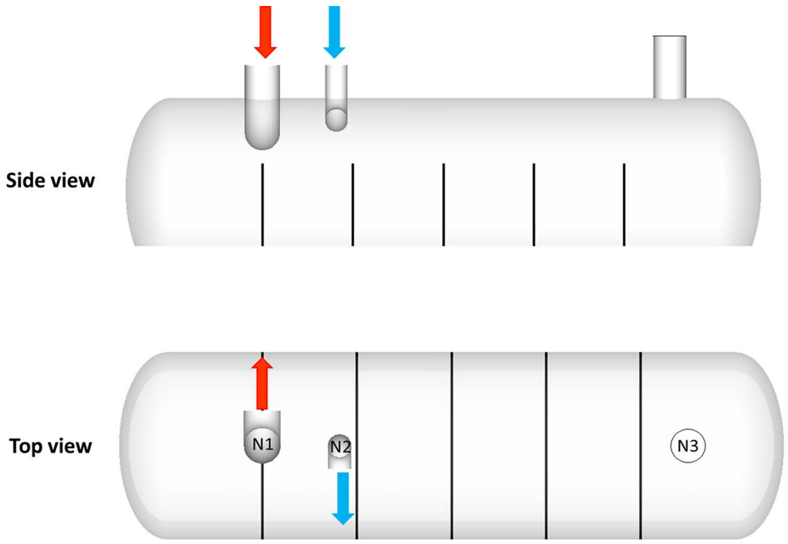

Figure 6. Oil-gas separator inlet flows through nozzles: side and top views

(including thermal inertial terms) means that the CFD-only simulation is missing important effects that could mitigate the cooling effect. This clearly has the potential to give rise to over-conservative design.

A hybrid approach combines a CFD model of the incoming fluid flow with a detailed, spatially-distributed dynamic model of the wall and fluid thermodynamics in PSE's gFLARE advanced process modelling software for safety systems. Information is exchanged between the two models via a proprietary interface.

The wall model is spatially distributed in 3 dimensions, to take into account variations through the wall and in axial and azimuthal (i.e. around the circumference) dimensions.

This approach has the potential to take into account all important effects. This is studied for two cases – gas-phase inlet flow only and gas with entrained liquid – below.

3.3.1 Hybrid Model with Gas Phase Flow

Here we consider the same blowdown scenario assuming gas-only inlet flows. The major difference with the previous example is that the thermal inertia of the wall is dynamically modelled, to take into account the compensating effect of heat flowing in from other parts of the wall.

The difference is immediately apparent from the temperature plots in Figure 9. Here it can be seen that:

(a) the wall temperature at the cold spot is much higher than the fluid temperature seen in section 3.2

(b) the lowest wall temperatures occur only toward the end of the blowdown rather than coinciding with the minimum inlet flow temperature at 200s.

The actual temperatures at the cold spot (the global minimum temperature) are shown in Figure 10. The global minimum inner wall temperature is around 240 K, well above the minimum allowable temperature.

3.3.2 Hybrid Model with Entrained Liquid

This case considers a similar scenario to 3.3.1 but the gas stream includes a small fraction of entrained liquids (less that 0.5% by mass of the feed) that have been picked up by the high-velocity gas.

Figure 7. CFD analysis of the entering flows showing direction and velocity

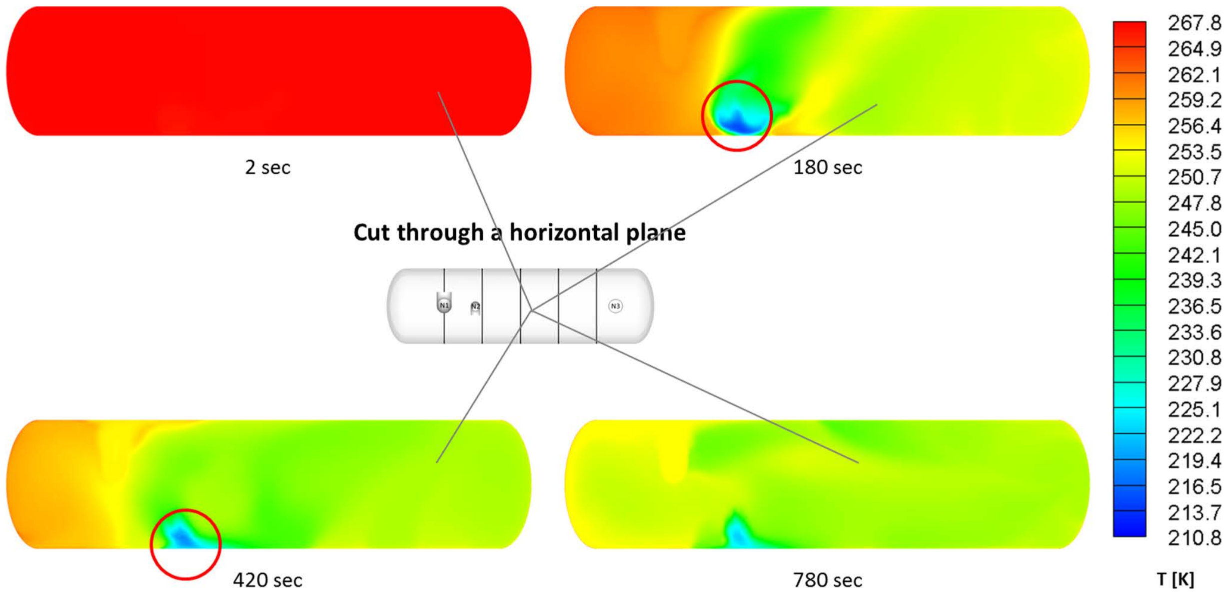

Figure 8. Temperatures of fluid from CFD analysis

Though the volume fraction of such entrained liquids is usually low, they can have disproportionate effect on wall cooling, because:

- the liquid / solid heat transfer coefficient is very much higher than the gas / solid coefficient, leading to a much higher heat transfer rate
- the partial vaporisation of the liquid at the wall provides additional cooling.

In addition, as entry velocities are very high it typically takes less than 0.1 seconds for cold fluid to reach the wall, delivering a relatively large amount of cold material rapidly.

The CFD model provides a description of the liquid droplet dissipation from the cold inlet nozzle and the velocity of liquid droplets towards the vessel wall, as in the CFD-only case above. The model also calculates the fluid-to-wall heat transfer coefficient from first principles for all points on the wall.

The gFLARE model contains a detailed wall model as in the previous example, plus a rigorous fluid thermodynamic description of the gas and entrained liquid droplets entering at each nozzle. The purpose of this is to rigorously account for the energy effects of phase change, including mass and heat transfer between liquid droplets and the bulk gas.

The results are shown in Figure 11, which plots the temperature at the global minimum temperature point over the course of the blowdown.

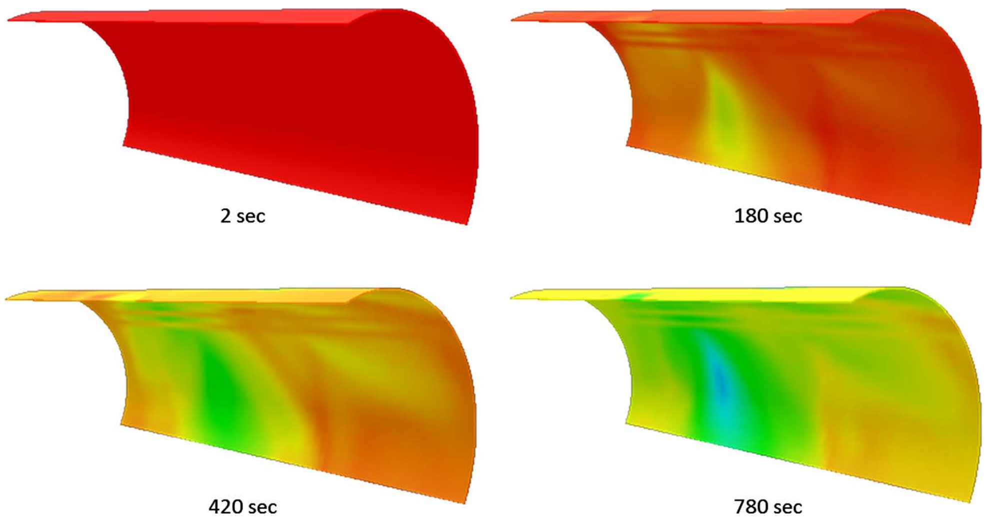

Figure 9. Wall temperatures from hybrid analysis, for gas phase flow

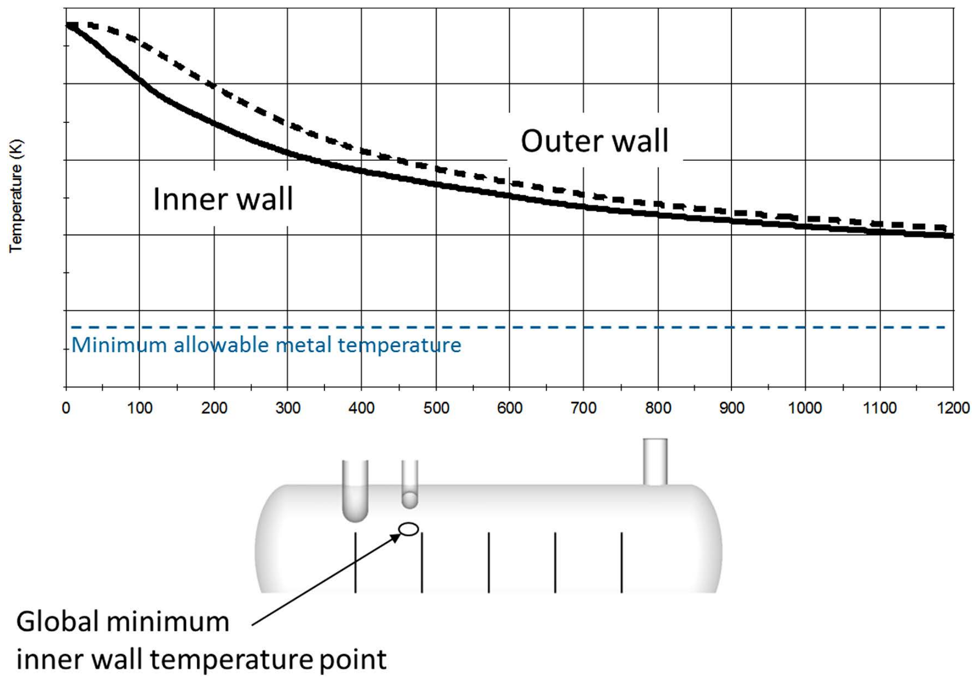

Figure 10. Wall temperatures from hybrid analysis

It can immediately be seen (black lines) that the local wall temperature rapidly drops below safe operational limits and remains there for some time.

Thus the same system is demonstrably unsafe when there is liquid entrainment.

4. CONCLUSIONS

This paper describes the application of models of increasing level of detail to the same blowdown scenario.

The standard approach, using dynamic lumped-parameter models of the KO Drum vessel, as typically

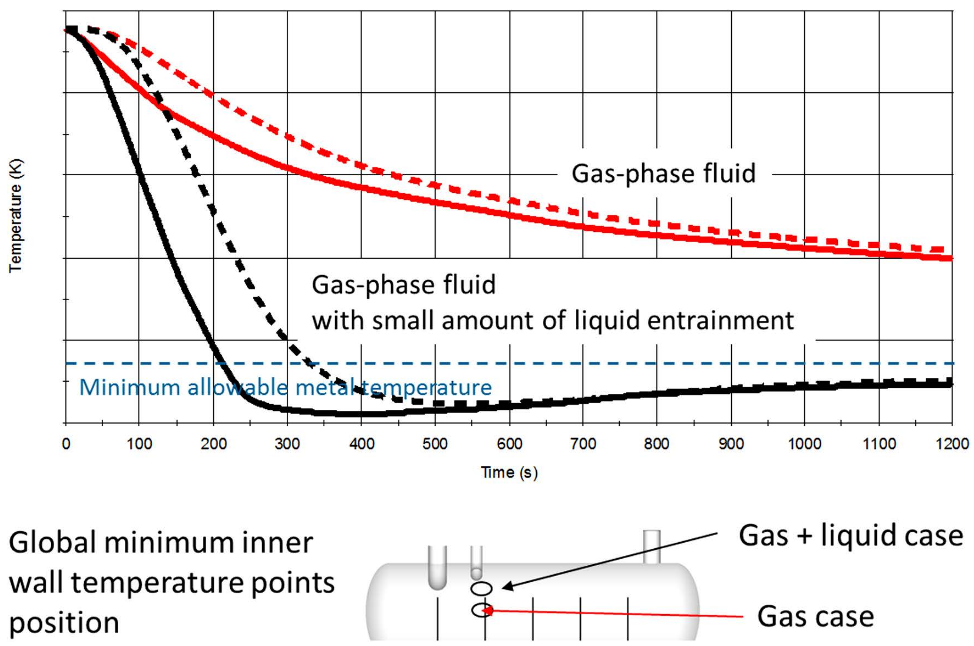

Figure 11. Hybrid model separator global minimum wall temperatures over time for gas-only (red) and entrained liquid (black) cases. Dotted and solid lines represent outer and inner wall temperatures respectively

implemented within dynamic process simulation packages, with no spatial variation taken into account, indicates that the example system is safe.

On the other hand using a much more rigorous CFD-based analysis, taking into account spatial variations, indicates that **if** the wall temperature perfectly tracks the adjacent fluid temperature then the operation is unsafe. However the CFD approach does not take important thermal inertia effects into account, so its results are generally not considered applicable.

A newly developed hybrid approach combining the spatial analysis of the CFD approach with correspondingly detailed dynamic wall models (to account for thermal inertia) and thermodynamic models (to account for mass transfer etc.) allows accurate analysis of many different scenarios that take all key effects into account.

The results from the hybrid approach, with its increased level of physical detail, shows that for the case of gas phase inlet flows the original design is indeed safe. The differences is that this is now demonstrated by rigorous analysis rather than through the use of simplistic assumptions of case 1, which may give rise to erroneous conclusions in other situations.

However, in cases where direct liquid droplet impingement on the vessel wall is likely to occur, a significant low temperature risk is seen and the vessel design would need to make provision for this. The model provides a sufficiently accurate tool to allow alternative vessel designs and operation modes to be assessed via sensitivity studies. In this case, it can be shown that redirecting the nozzles so that cold gas/liquid flow is not sent towards the wall increases the temperature of the walls around the nozzles by 12°C, resulting in a safe vessel design.

The conclusion is that high-fidelity dynamic modelling that takes into account spatial variations within the vessel is essential to ensure safety without expensive over-design. The hybrid approach, by accurately describing the dynamic response of the vessel walls at all locations throughout the vessel, can give accurate results for a wide range of scenarios. The approach has been implemented successfully on a number of projects.