HTR2008-58161

A TECHNIQUE TO SIMULATE A TUBE BREAK IN A HIGH-PRESSURE GAS/COOLING WATER HEAT EXCHANGER

HJ van Antwerpen
M-Tech Industrial
Potchefstroom 2520, ZA

EJ Mulder
Post Graduate School for Nuclear Sciences and Engineering
North-West University, Potchefstroom 2520, ZA

ABSTRACT

The gas cycles of most High Temperature Gas-Cooled Reactors (HTR's) reject heat to water at some stage. In the helium/water heat exchangers of HTR's with direct Brayton cycles, the helium is usually at a much higher pressure than the water. If the pressure boundary between the helium and the water fails inside the heat exchanger, the effect on the rest of the water system has to be established in order to do a proper system design. This can be done most efficiently by using a system simulation code, however, very few system simulation codes has the capability to do gas/liquid interface tracking as required for this problem.

This study describes a calculation method with which a gas/liquid heat exchanger tube rupture can be calculated in a simulation code without interface tracking. The course of events after tube rupture is described and appropriate calculation models derived. A mathematical model for a pressure relief valve (PRV) was also created.

The calculation models were implemented in the system simulation software Flownex and used to study a tube rupture on a 5000kPa helium/water heat exchanger. The assembled calculation network solved stable and within reasonable time.

The simulation provided insight into the course of events following the tube break. It was shown that the acceleration of water out of the helium cooler, by choked-flow helium, caused the main pressure pulses during the event. The maximum pressure in the water loop occurs on the opposite side of the helium cooler due to constructive interference of the initial pressure wave with itself. It was also shown that by changing only pipe lengths, the system could become prone to severe oscillations after a tube rupture event.

NOMENCLATURE

A Cross-sectional area [m^2]

f Natural frequency [Hz]

k Spring constant [N/m]

L Length of water column [m]

m Mass flow rate [kg/s] or Pressure Relief Valve component mass [kg]

P Pressure [kPa]

Q Volume flow rate [m^3]

ρ Density [kg/m^3]

t Time [s]

V Velocity [m/s]

x Pressure Relief Valve plunger position [m]

SUPERSCRIPTS AND SUBSCRIPTS

N Value at new timestep

O Value at previous timestep

ave Average

atm Atmospheric

INTRODUCTION

Brayton power generation cycles that use high temperature gas-cooled reactors as heat sources, typically use gas/water coolers, where the gas is at a considerably higher pressure than the water.

Such coolers are subject to nuclear safety standards, as in some cases, the radionuclide-containing primary coolant flows through them. The design envelope for these heat exchangers and their cooling water loops include the unlikely event of a single complete tube break. In order to design the water system to cater for a tube break, the conditions during the event has to be calculated.

In the case of a tube break in a cooler, a non-condensable gas blows into an incompressible liquid system. Accurate simulation of this two-phase flow event would require fluid interface tracking, with simultaneous calculation of pressure wave propagation. A Computational

Fluid Dynamics (CFD) code might be considered for this calculation, but as pressure waves travel through the entire cooling water system, detail CFD is not appropriate. To capture the one-dimensional movement of the pressure waves and the system response, a system simulation or network code has to be used.

Previous work, where the gas was on the tube-side, confirmed that a one-dimensional approach to simulating such an event is appropriate and gives conservative results [4]. In the case considered here, the gas is on the shell-side and the liquid on the tube-side.

The system response is critical, as the maximum system pressure due to the tube break could occur far from the point of origin, due to constructive interference. However, very few system simulation codes have fluid interface tracking capability. Therefore, a method was developed to simulate a gas/water heat exchanger tube break without the use of interface tracking in the system simulation software Flownex.

The development and application of the calculation techniques is described according to a case study.

CASE STUDY

DESCRIPTION

As case study, a tube break on a shell-and-tube type heat exchanger in an intermediate cooling water loop was simulated. The helium pressure level is $5000kPa_a$, while the water side pressure level is $200kPa_a$. A process flow diagram of the loop, with pipe lengths, is shown in Figure 1.
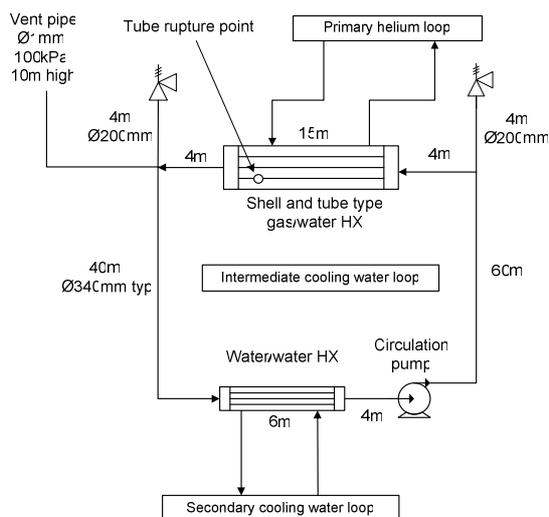
Figure 1. Process Flow Diagram of an intermediate cooling water loop on a High Temperature Gas-Cooled Reactor, showing pipe lengths as well.

The heat exchangers in the cooling loop have characteristics as described in Table 1. The vent pipe is only used to determine the pressure level ($200kPa_a$) in the cooling water loop. Its diameter is specified as 1mm so that is will have a negligible pressure relief effect during the transient event.

The details of the tube break event are determined by the tube geometry, the postulated failure mode and tube material properties and will not be treated in this investigation. The techniques developed in this study will apply to most kinds of helium/water pressure boundary failures, with only the time from zero to full opening and the helium flow resistance differing from case to case. The worst type of tube break was assumed, namely a complete

double-ended guillotine break, where the opening from the helium to the water side goes from 0mm to 14.85mm in 0.015 seconds.

The worst case in terms of water side pressure increase will be when the flow resistance into the water system is a minimum. Therefore, the tube break is assumed to occur at the gas side of heat exchanger tube sheet, as indicated in Figure 1. This means that with a 300mm tube sheet thickness, the 15m tube is divided into a 0.3m piece and a 14.7m piece.

Table 1. Heat exchanger characteristics. Helium is on the shell-side and water on the tube-side of the He/water heat exchanger.

He/water HX	Helium inlet temperature	115℃
	Water inlet temperature	24℃
	Water outlet temperature	65℃
	Water mass flow	175 kg/s
	Heat transfer	30 MW
	Number of tubes	1050
	Tube inner diameter	14.85mm
	Tube wall thickness	3.32mm
	Tube length	15m
Water/water HX	Intermediate Water inlet temperature	65℃
	Intermediate Water outlet temperature	24℃
	Number of tubes	500
	Tube inner diameter	14.85mm
	Tube wall thickness	3.32mm
	Tube length	15m

COURSE OF EVENTS AFTER TUBE BREAK

According to the dominating phenomena, the event can broadly be divided into three phases, as shown in Figure 2.
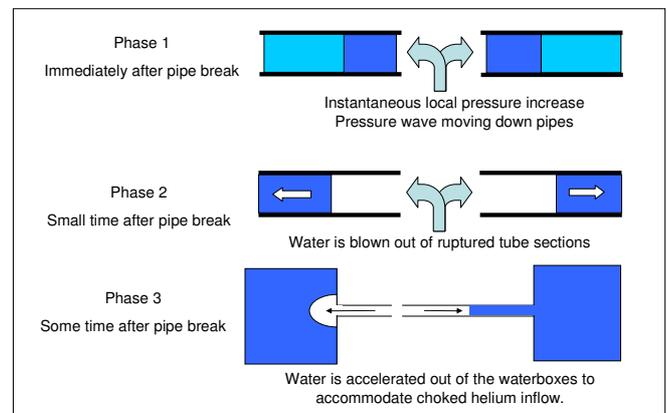
Figure 2. Schematic explanation of the events after tube break.

At the time of the tube break, the water in the tubes is still moving at steady-state velocity but as the water comes in contact with the high-pressure helium, a high-pressure wave is created that moves at sonic velocity down both sections of the ruptured tube. This pressure wave is damped out quickly by the area transition from the 14.85mm tube to the waterbox volumes behind the tube sheets. This is considered the first phase of the event.

During the second phase, the water columns in the ruptured tube sections are accelerated and forced into the upstream and downstream water boxes, with the shortest tube section being emptied first. Implicit in this description is the assumption that the water velocity will be far greater than the free-surface wave speed of the water, so that the

helium/water interface moves as a sharply defined front through the tube.

The most significant part of the event is the third phase, where the helium flow becomes choked at the tube outlet into the waterbox and water is accelerated out of the heat exchanger waterbox to accommodate the helium inflow. Once the water outflow rate has increased to equal the helium volume flow rate, no additional pressure waves are created.

It will be shown that this acceleration of water out of the heat exchanger causes a pressure buildup in the heat exchanger water box. This pressure pulse then propagates at sonic speed throughout the entire water system and interferes constructively with itself at the farthest end of the loop to reach the maximum pressure of the whole event. After this, the pressure wave is dissipated as it travels along the water loop. The pressure relief valves open and the inlet and outlet flow rates reaches an equilibrium value after some time. In the large (340 mm) water pipes, where the helium volume flow rate could be slower than the free-surface wave speed of water, the helium will form a bubble at the top of the pipe. As soon as the helium bubble reaches a pressure relief valve, the valve will discharge gas instead of liquid at a much higher flow rate so that the pressure in the system is expected to decrease rapidly.

CALCULATION MODELS

Resolution of all the phenomena described above requires a system simulation code that has fluid interface tracking capability, such as the Volume Of Fluids method ([1]: p355). As pointed out, very few such codes are available, which is why the method described in this paper was developed. The minimum requirement for a system simulation code to use the methods described in this paper, is the capability of resolving pressure waves in an incompressible medium.

To calculate pressure wave propagation through a pipe network, Flownex solves the transient compressible and incompressible conservation equations for mass, momentum and energy for an unstructured network layout. The conservation equations used in Flownex, as well as the implicit solution technique to calculate fast transients are described by Greyvenstein [2].

The pipe network discretisation in Flownex is derived from the CFD staggered-grid variable arrangement. In Flownex, mass and energy conservation is solved on control volumes which are referred to as "nodes", while momentum conservation is solved on the "cell faces" or "elements" between nodes. Various elements are available in Flownex to represent heat flow or fluid flow between control volumes or nodes. The type of element determines the method used to calculate the resistance to flow, i.e. pressure drop, or the heat resistance in case of heat transfer elements. This means that scalar-valued variables (such as temperature and pressure) are calculated at nodes while vector-valued variables (such as velocity or heat flow) are calculated on elements.

Flownex has the capability of solving additional user-coded mathematical models by means of the Equation Element. The Equation Element links various variables in a network and does user-specified calculations between solution timesteps.

LIST OF ASSUMPTIONS

1. The gas side of the ruptured tube is assumed to have unrestricted inflow, i.e. the surrounding pipes or fins do not obstruct the flow in any way. This assumption only influences the flow resistance from the gas to the liquid side and has no impact on the development of the simulation model.
2. It is assumed that heat transfer plays a negligible role on the outcome of the event. Therefore the heat exchanger can be replaced with pipe elements for this simulation. This will greatly increase the speed of the simulation.
3. Resulting from the assumption that heat transfer plays a negligible role in this simulation, the specific heat of water is taken as constant at 4.18kJ/kgK, instead of using lookup tables for enthalpy.
4. The deformation of the heat exchanger water boxes is not modelled.
5. Once into the water network, no further volume change of the helium is calculated as the helium volume is represented by water volume.
6. The axial restraint parameter used in the water sonic velocity calculation was calculated as 0.91, for steel pipes fixed and supported at both ends.
7. Interaction between the pump and the pressure wave is not modelled in any detail. The pump is specified with a fixed pressure rise/flow characteristic.
8. Fluid/structure interaction is not investigated in this study. It is, however, an important are of further development for this calculation model.
9. No phase changes in the water, such as column separation cavitation, are modelled due to the difficulty of modelling such events. The rationale is that when negative pressures are calculated in a system, its design is not yet mature. The design should be updated to a point where no negative pressure occur. At this stage, it will no longer be necessary to calculate column separation cavitation.

FIRST PHASE: PRESSURE WAVE

Considering the pipe rupture event in more detail, the first phase of the event can be accurately calculated in Flownex by suddenly increasing the pressure at the node representing the rupture point.

SECOND PHASE: ACCELERATION AND EJECTION OF WATER FROM TUBES

For the second phase of the event, where water is accelerated and ejected from the ruptured tube sections, a more detailed approach is required. In order to resolve the water inertia effect, an equation for the balance of forces on the water column is solved. For this calculation model, the solution method and implementation is described in the following paragraphs.

The calculation model for the third phase will not model this second phase accurately, as the third phase assumes that the ruptured tubes have already been blown empty. It must also be noted that water is expelled only from the ruptured tube in the heat exchanger while all the other heat exchanger tubes are still filled with water.

Due to the water in the ruptured tube section blowing out into the waterbox, the accelerating column shortens with time. The rate at which its length L shortens, equals its velocity so that

$$\frac{dL}{dt} = -V \qquad (1)$$

It is assumed that the inertia of water inside the waterbox has no effect on the inertia of the water inside the ruptured tube. Thus, accelerating mass m is calculated only as

$$m = \rho A L \qquad (2)$$

where ρ is density and A is flow cross-sectional area. It is also assumed that inertial forces dominate this phase of the event so that pipe wall friction and waterbox outlet pressure loss are neglected. The rate of change of the velocity is then found from momentum conservation as

$$\rho A L \frac{dV}{dt} = P_1 A - P_2 A \qquad (3)$$

where P_1 and P_2 are respectively the helium and waterbox pressures. In the calculation, the helium pressure is assumed to rise instantly to 5000 kPa$_a$. The geometry for this problem, as well as the physical meaning of some of the variables is illustrated in Figure 3.
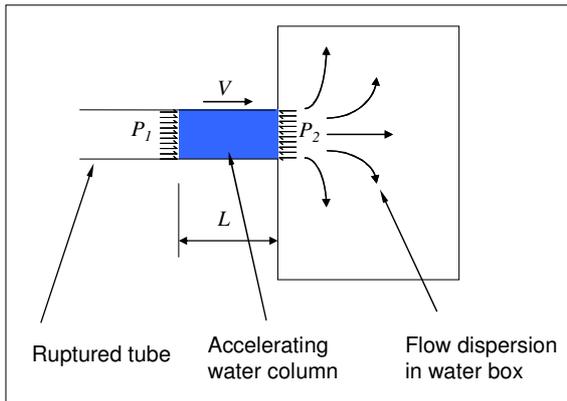
Figure 3. Geometry of the flow problem described by Eqs.(1) and (3).

The assumptions of negligible pipe wall friction, instant helium pressure increase at the rupture and neglecting outlet/waterbox water inertia will lead to faster water column ejection times, which is conservative as that will result in a higher waterbox pressure.

SECOND PHASE: BOUNDARY VALUES
As stated above, P_1 is assumed to rise to 5000 kPa at the moment of the double-ended guillotine tube rupture initiation. If only a small leak occurs, say a 3mm diameter hole, the upstream pressure P_1 can be calculated from the outlet of an orifice representing the hole. The calculation procedure for such a choked flow problem will be described in the next paragraph on the third phase.

The downstream pressure P_2 is the waterbox pressure that has to be determined dynamically as it responds the volume inflow from the ruptured tube.

SECOND PHASE: SOLUTION METHOD
The differential terms in Eqs.(1) and (3) are replaced with the following first-order accurate finite difference approximations:

$$\frac{dL}{dt} \approx \frac{L^N - L^O}{\Delta t} \quad \text{and} \quad \frac{dV}{dt} \approx \frac{V^N - V^O}{\Delta t} \qquad (4)$$

to obtain

$$\frac{L^N - L^O}{\Delta t} = -V^O \qquad (5)$$

and

$$\rho L_{ave} \frac{V^N - V^O}{\Delta t} = P_1 - P_2 \qquad (6)$$

where the superscripts N and O respectively denotes new and old timesteps. L_{ave} is the average length between the old and new timesteps.

Eqs.(5) and (6) are rearranged to obtain the following recursive equations:

$$L^N = L^O - \Delta t V^O \qquad (7)$$

$$V^N = V^O + \Delta t \frac{P_1^O - P_2^O}{\rho \frac{(L^N + L^O)}{2}} \qquad (8)$$

The length equation (7) is calculated first so that the new length L^N is available for the velocity equation (8). The equations are specifically written as explicit recursive equations so that they can be implemented in a Flownex Equation Element.

From the velocity V^N, the mass flow is calculated and assigned to the Flownex pipe element representing the ruptured tube. The resulting waterbox pressure is fed back into the velocity calculation by means of P_2 so that with small timesteps the interaction is captured accurately.

A test spreadsheet calculation of Eqs.(7) and (8) has been done, where it was found that the short (0.3m) tube section is blown empty within 0.006 seconds and the long section blows empty within 0.27 seconds. The actual blowout times is expected to be longer than this, due to the downstream waterbox pressure rise, but that effect has to be confirmed with the integrated network. Velocity and water column length is shown in Figure 4.
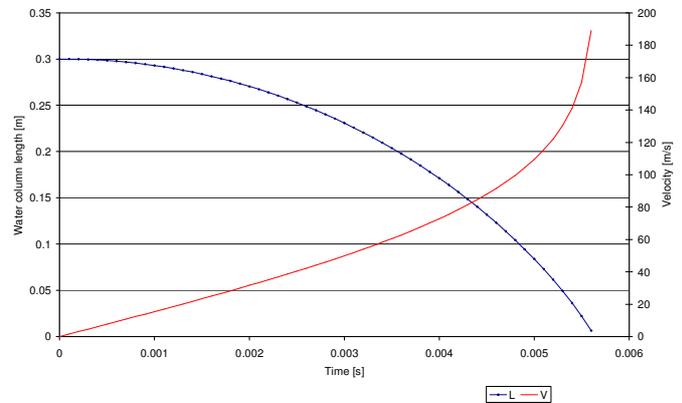
Figure 4. Water column length L and velocity V as function of time for constant pressure difference. The timestep length was 0.0001 seconds.

A sensitivity study showed that timestep independence is reached at 0.0001 second timesteps for the short (0.3m) tube section, while only 0.01 second timesteps are required by the long (14.7m) tube section.

As soon as the water column is blown out the tube, the choked flow model described in the next paragraph is applied to calculate helium mass flow. The transition is signalled in the Equation Element by an if-statement that checks whether the remaining helium column length is shorter than the tube length.

THIRD PHASE: CHOKED HELIUM FLOW

For the third phase, a modelling methodology was followed in which there is a water network and a gas network with the same geometry. The pressure response of the water system P_{water} is transferred to the gas network and a new gas volume flow Q_{helium} is calculated. A corresponding water volume inflow is applied to (forced into) the waterbox to determine its pressure response. In this way, gas friction and choking is calculated accurately for the upstream and downstream pressures. The principle of this simulation method is illustrated in Figure 5.
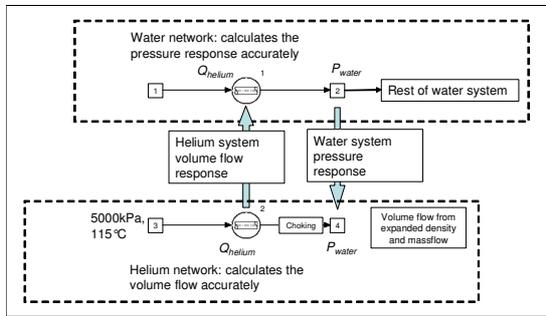

Figure 5. Modelling methodology for coupling the helium and water networks.

During one transient timestep, the upstream and downstream pressures of the helium network are fixed and a flow rate is calculated. Similarly, the volume inflow into the water network is fixed during a timestep and the resulting pressure response is calculated. Between the timesteps, the downstream pressure of the gas network is set equal to the pressure calculated in the water network during the previous timestep. Similarly, the volume flow into the water network is set equal to the volume flow calculated in the gas network during the previous timestep.

When the timesteps are adequately small, a relatively good coupling between gas and water system behaviour can be obtained.

In Flownex, the ruptured tube was modelled as two separate pipes, an upstream and a downstream section, while the rest of the intercooler tubes were modelled with a single pipe that has the number of parallel pipes specified as shown in Figure 6. The pipe elements representing the ruptured tube were copied and pasted alongside the water network to form the gas network, as shown in Figure 6.

PRESSURE RELIEF VALVE

The Pressure Relief Valves (PRVs) on the water loop was simulated by solving the force balance, or momentum conservation for the valve plunger. The method followed here is similar to that of previous researchers, such as Nielacny [3], who have also created a PRV model for simulation of a water network. The force balance on the valve plunger is given by the following equation:
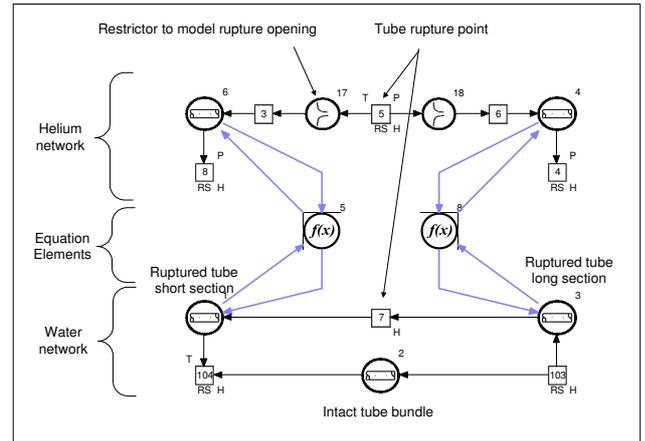

Figure 6. Heat exchanger model to take tube rupture into account.

$$m\frac{d^2 x}{dt^2} = (P - P_{atm})A - k(x + x_0)$$
$$\text{if } x = 0: \quad A = A_{hole} \qquad (9)$$
$$\text{if } x > 0: \quad A = A_{plunger}$$

where m is the plunger mass, x is the plunger position, P is the system absolute pressure and P_{atm} the outside atmospheric pressure. A is the plunger area that is subjected to the pressure difference. k is the PRV spring constant and x_0 is the spring preloading. The area subjected to differential pressure determines the moment the PRV opens. To solve Eq.(9), the following finite difference approximations for the differential terms are introduced:

$$\frac{d^2 x}{dt^2} \approx \frac{x^{nn} - 2x^n + x^o}{\Delta t^2} \qquad (10)$$

with x^{nn} the new value of x at the newest timestep, x^n the value of x at the previous timestep and x^o the value if x two timesteps ago. Δt is the timestep length.

Substituting Eq.(10) into Eq.(9) and rearranging gives the following set of recursive equations for x^{nn}

$$\text{if } x = 0: \quad A = A_{hole}$$
$$\text{if } x > 0: \quad A = A_{plunger}$$
$$x^{nn1} = 2x^n - x^o + \frac{\Delta t^2}{m}\left((P - P_{atm})A - k(x^n + x_0)\right)$$
$$\text{if } x^{nn1} > x^n: \qquad x^{nn} = x^{nn1} \quad \text{Default}$$
$$\text{if } x^{nn1} > x_{max}: \qquad x^{nn} = x_{max} \quad \text{Opening position limit}$$
$$\text{if } x^{nn1} < 0: \qquad x^{nn} = 0 \quad \text{Closing position limit}$$

x_{max} is the maximum plunger travel. These equations were implemented in a Flownex Equation Element. The input values were obtained by solving the following set of simultaneous equations:

P_{open} = 1100 kPa   Opening pressure

P_{max} = 1200 kPa   Steady-state pressure at maximum opening

P_{atm} = 100 kPa   Atmospheric pressure

Δx = 0.25m   Maximum opening distance

A_{hole} = 1   Reference area

t = 0.05s   Characteristic time/opening response time

$$\left(P_{open} - P_{atm}\right) A_{plunger} = k \left(x_0 + \Delta x\right)$$

$$\left(P_{max} - P_{atm}\right) A_{hole} = k \left(x_0 + \Delta x\right)$$

$$\left(P_{open} - P_{atm}\right) A_{hole} = k x_0$$

$$\left(P_{close} - P_{atm}\right) A_{plunger} = k x_0$$

$$t = \frac{1}{f}$$

$$f = \sqrt{\frac{k}{m}}$$

(11)

Note that in Eq.(11), the plunger effective mass was solved from the spring constant and the characteristic opening time of 0.05s. f is the natural frequency of the spring/plunger assembly in the valve.

The plunger/hole area ratio is determined by the different opening and closing pressures, while the spring constant is determined by the fully-open plunger travel and pressure. This means that the pressure relief setpoint is not set explicitly, but rather m and k are calculated with Eq.(11) so that the opening pressure is implicit in the preload.

The values for m and k are not actual values, but in conjunction with the other values calculated from Eq.(11), they give realistic transient response times.

A Birkett Size "T" PRV was used. As described in Eq.(11), properties for that valve model was calculated to be fully open at an overpressure of 10 percent, so that the flow rate corresponds to data for the valve. From the data in the Birkett catalogue, the type "T" PRV flow rate at 10 percent overpressure (i.e. dP_{PRV} = 1100kPa or P_{max} =1200 kPa$_a$) was estimated as 513 l/s. This was modeled in Flownex as a Restrictor with Discharge coefficient (RD) element. For a 250mm diameter restrictor, a discharge coefficient of 0.225 gives the required flow rate of 513l/s at a pressure difference of 1100kPa, i.e. the discharge coefficient of 0.225 represents the size or flow capacity characteristic of the restrictor, while its diameter is used as working variable in the Flownex model.

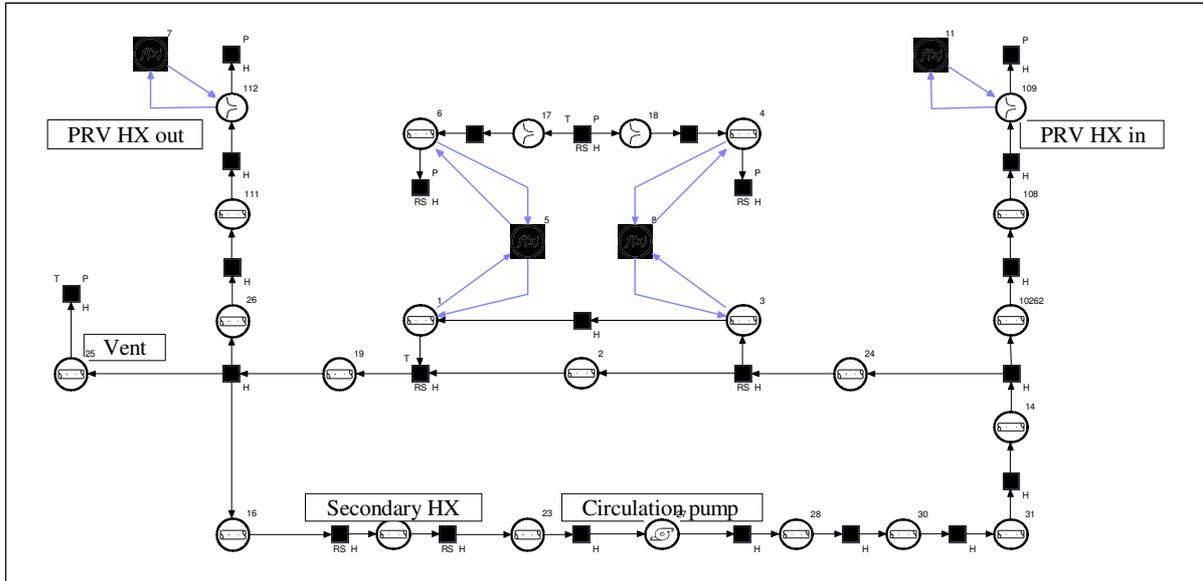The assembled Flownex model is shown in Figure 7.

Figure 7. Flownex model of the intermediate cooling water loop in Figure 1.

The event specification for this simulation is relatively simple as it involves only specification of the openings to the ruptured tube and timestep changes as described in Table 2.

Table 2. Transient event and timestep specification.

Time	timestep	Event
0	5.00E-05	Orifices representing rupture points start opening. Inner bundle and outer bundle downstream node temperatures are unfixed.
5.00E-05	5.00E-05	One timestep after opening started, the pressure of the water node at the rupture point is set to 5000 kPa, and the temperature fixed at 115C, the temperature of the gas. This is done as a method to observe the extent of gas propagation in the water network, as the water temperature is now proportional to the helium volume fraction.

| Time	timestep	Event
0.003	0.0001	Timestep changed
0.05	5.00E-05	Orifices representing rupture points fixed at full opening, equal to pipe inner diameter.
0.1	0.0005	Timestep changed
0.4	0.0001	Timestep changed

RESULTS

The first phase of the event, where the pressure increase is communicated with the rest of the system, was found to be of no significance, as the pressure wave is damped out at the area transition from the ruptured tube to the heat exchanger water box.

The second phase, where the water column is ejected from the ruptured tube section, was found to end at 0.0055s for the short section (as shown in Figure 8) and 0.279s for

the long section, which is in good agreement with the expected values of 0.006s and 0.27s. The Equation Elements for the respective tube sections automatically detects the phase of the event and applies the correct calculation model.

Figure 9 shows that, surprisingly, the peak waterbox pressure is at the opposite end of the short tube section. Furthermore, with a peak waterbox pressure of 1700kPa, the maximum pressure in the system (2750kPa) is reached at a point 45.5m from the PRV junction, 14.5m downstream of the circulation pump. This is due to constructive interference of two pressure waves travelling in opposite directions around the cooling water loop.

The pressure history for the rest of the event is shown in Figure 10. Figure 10 shows that the maximum pressure oscillations occur at the same location as the initial maximum pressure. Some other, smaller oscillations caused by the PRVs are superimposed on the large oscillations. Figure 10 also shows that an equilibrium pressure is reached at a value just above the PRV opening pressure of 1100kPa.
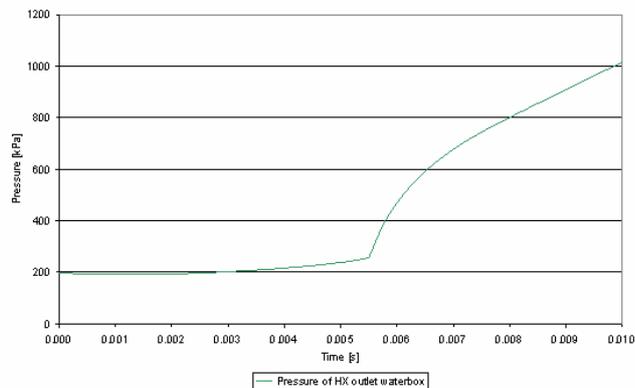
Figure 8. Transition from second to third phases for the short tube section at 0.0055s, i.e. the tube is blown empty of water and choked helium flow starts at 0.0055s
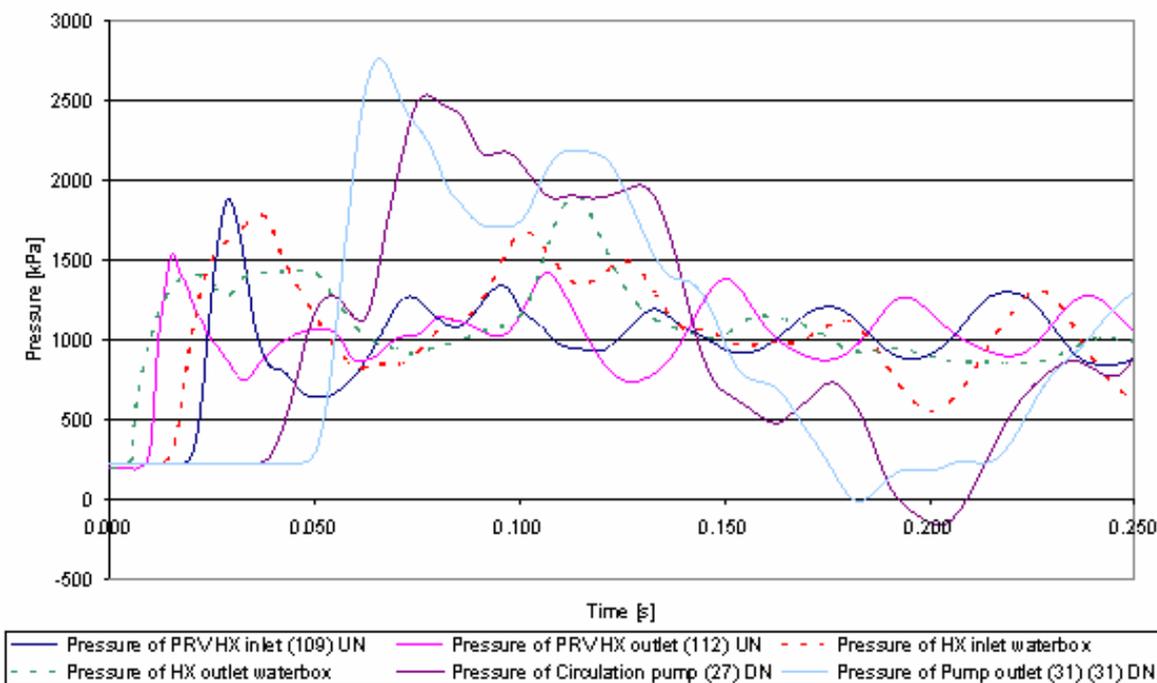
Figure 9. Pressure history for the first 0.25s after the tube rupture for selected points in the network.
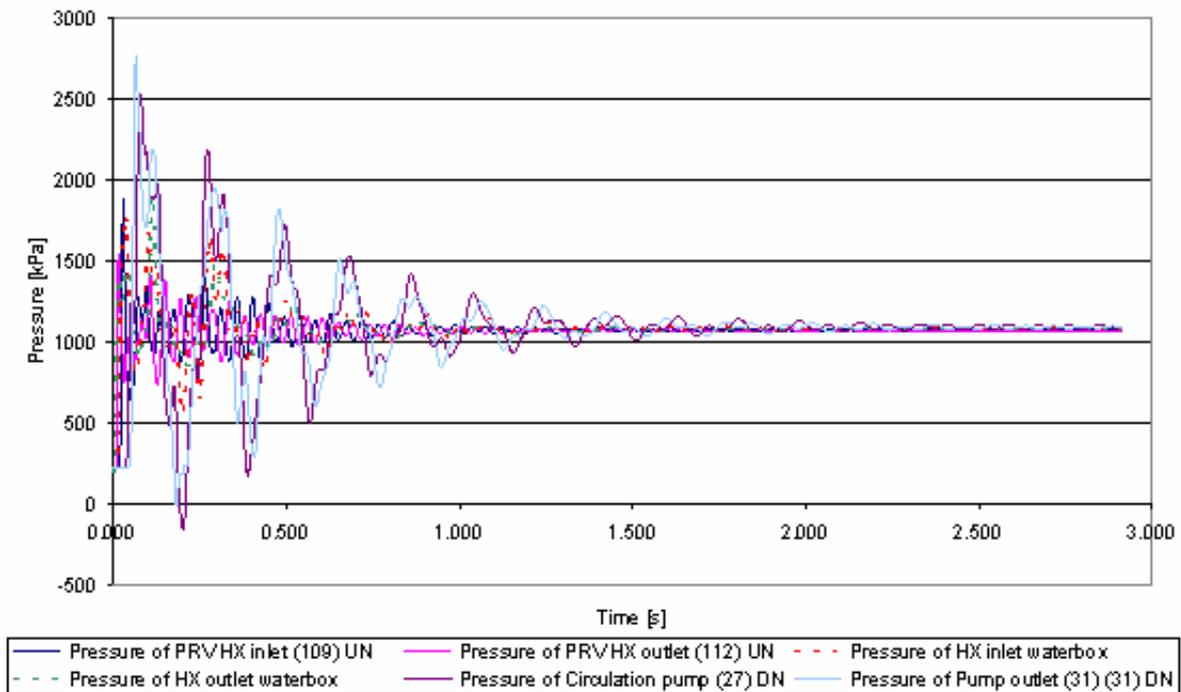
Figure 10. Pressure history up to three seconds after tube rupture.

The interaction of the PRV with the system is shown in Figure 11. Due to the short duration of the pressure pulses, the PRV never reaches its maximum opening.
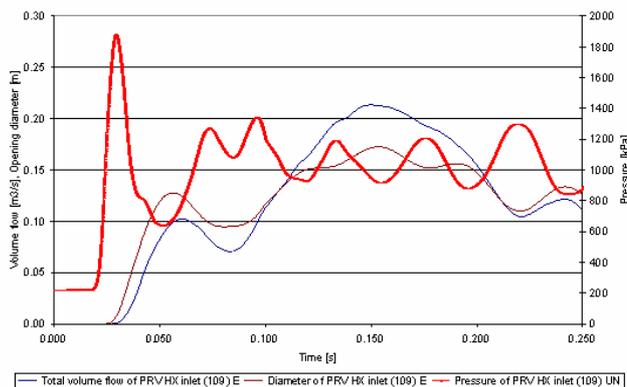

Figure 11. Pressure, opening diameter and flow rate for the PRV at the heat exchanger inlet.

In order to ensure the accuracy of the numerical solution, a grid dependence study was done. In the reference case, the pipe increment length was 0.5m while the test case had 0.25m increments. The difference in maximum pressure (in the section after the circulation pump) was calculated and normalised with respect to the peak pressure of 2700kPa. As the maximum normalised difference over the full three seconds simulated time was smaller than 0.7 percent, the reference discretisation of 0.5m per increment is considered adequate.

Both simulation models solved up to three seconds simulated time within about 25 minutes on an Intel Core2 Duo 2.2GHz laptop computer. The solution speed is highly dependent on the amount of interaction from the graphical user interface during solution, so that grid refinement is not necessarily on the critical path for reducing solution time. 25 minutes is regarded as a reasonable solution time for this problem.

The effect of the distance between the heat exchanger water boxes and the PRVs was investigated by changing the total distance from water box to PRV from eight meters (as in Figure 1) to two meters. The calculated pressure history is shown in Figure 12.
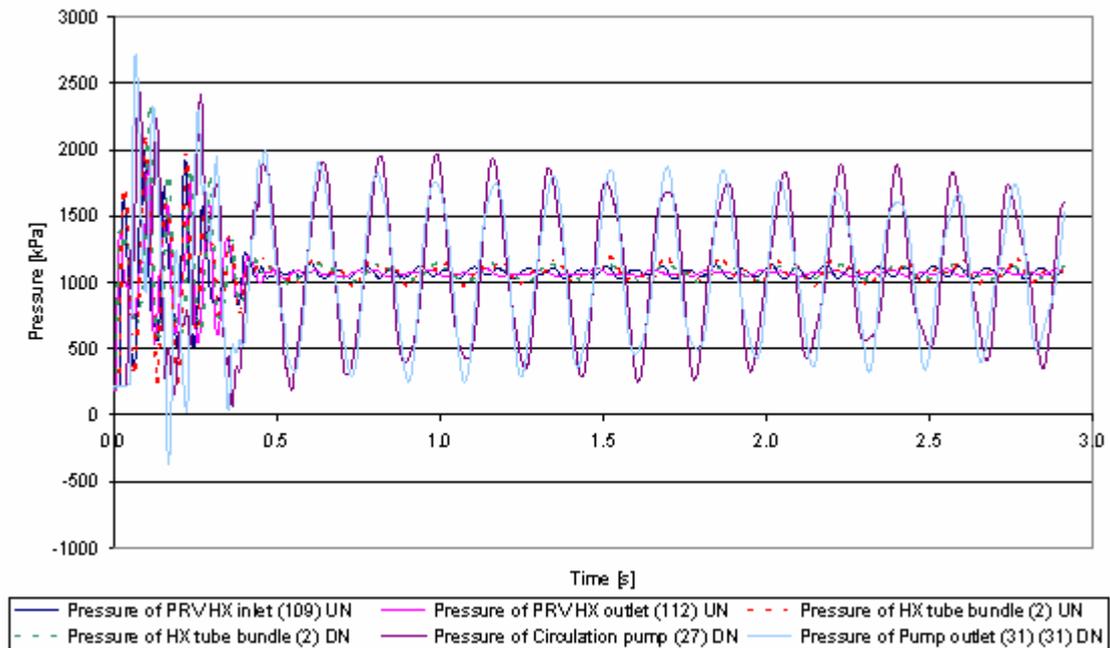
As shown in Figure 12, shortening the distance between the heat exchanger and the PRV causes extremely large pressure oscillations in case of a tube rupture. This is very important, as severe pressure oscillations could cause fatigue failure of other components. This result also shows that an apparently simple water network has many modes of vibration. It highlights the importance of being able to simulate the system as a whole and take into account the possible component interactions.

MITIGATION MEASURES

Several measures to mitigate the effect of the intense pressure wave and the resulting oscillations still have to be investigated. In order to do this, calculation models have to be developed for two additional components, namely an accumulator and a gas-charged PRV.

Some preliminary simulations with accumulators (using very simple accumulator models) close to the heat exchanger waterbox gave significant reduction of the initial pressure wave, as well as damping of the ensuing oscillations.

For the case presented in this paper, the inertia of an ordinary spring-type PRV plays a significant role in the system response. Another type of PRV of which the design could even be integrated into the heat exchanger waterbox, is a gas-charged rubber diaphragm type PRV. Its benefits are that it has negligible inertia compared to a spring-type PRV and its relief pressure can be set remotely by means of its charge pressure.

Figure 12. Pressure history, showing the effect of shortening the distance between the heat exchanger water boxes and the PRVs.

Another option to mitigate the effect of the helium ingress is to install PRVs directly onto the waterbox. However, this is not always feasible on a HTR as the heat exchanger could be situated inside an area where routine PRV maintenance is prevented by radiation levels during reactor operation. This issue is also design-specific for each system.

CONCLUSIONS AND RECOMMENDATIONS

The calculation model for helium ingress into the water system provided a tool to simulate gas/water heat exchanger tube rupture. A mathematical model for a pressure relief valve (PRV) was also created. The assembled network solved stable and within reasonable time.

The simulation provided insight into the course of events following the tube break. It was shown that the acceleration of water out of the helium cooler, by choked-flow helium, caused the main pressure pulses during the event. The maximum pressure in the water loop occurs on the opposite side of the helium cooler due to constructive interference of the initial pressure wave with itself. It was also shown that by changing only pipe lengths, the system became prone to severe pressure oscillations after a tube rupture.

The calculation model presented in this paper enabled simulation of a tube rupture event without fluid interface tracking.

Using this method, further investigations into the effect of break location, additional components and geometrical uncertainty analyses have to be performed

ACKNOWLEDGEMENTS

REFERENCES

[1] Ferziger, J.H., Peric, M (1999) Computational Methods for Fluid Dynamics. Second revised edition. Springer Verlag, Berlin. Page 355

[2] Greyvenstein, G.P. (2002) An implicit method for the analysis of transient flows in pipe networks. International Journal for Numerical Methods in Engineering, Vol. 53, pp 1127-1143.

[3] Nielacny, M. (2002) An Analytical Model Of The Water Hammer Effect Considering A Spring Safety Valve. 5th International Conference on Hydro Science & Engineering. Warsaw, Poland.

[4] Thyer, A.M., Wilday, A.J., Bankes, G. (2001) The Experimental Study and Simulation of Tube Rupture in Shell-And-Tube Heat Exchangers. Hazards XVI: Analysing the Past, Planning the Future By M. F. Pantony, Conference Proceedings, Institution of Chemical Engineers (Great Britain). North Western Branch. A three-day symposium organised by the Institution of Chemical Engineers (North West Branch) and held at UMIST, Manchester, UK, 6-8 November, 2001.