# THERMAL ANALYSIS OF THE TRANSHIELD 20 CONTAINER

*C. J. Fry and J. E. Gillard*

AEA Technology, Winfrith, Dorchester, Dorset, DT2 8DH, England

## SUMMARY

This paper describes the thermal analysis of the Transhield 20 container which has been performed in support of its safety approval.

## INTRODUCTION

The Transhield 20 is a type B(U)F container developed by AEA Technology for the transport of bulk quantities of alpha-active intermediate level waste. Externally the package is an 6.1m x 2.4m x 2.6m ISO container while internally it contains a 1.9m diameter stainless steel pressure vessel. The space between the external frame and the pressure vessel is mostly filled with cork.

The Transhield 20 is designed to transport up to thirty five 200l drums, held in five carousels. It can be adapted, however, to transport 500l drums or other large items. Because it is intended for the transport of materials with low heat generation, the container is designed to carry a maximum total heat load of only 280W. A fuller description of the Transhield 20 container is given by Gaffka and Lawrence [1998].

## EXPERIMENTAL TEST OF NORMAL TRANSPORT TEMPERATURES

The assessment of the temperature of the container during normal transport was performed by computer modelling validated by full-scale experimental test.

For the experimental test a full-scale, prototype container was placed inside a large plastic 'tent' within which controlled heating was provided so that the air temperature remained virtually constant at 16°C. This temperature, which corresponded to the maximum temperature expected in the surrounding room, was selected so that the minimum possible heat need be input into the enclosure. If a greater ambient temperature (such as 38°C) had been selected then it would have been necessary to provide more heat to the enclosure which would probably produce significant temperature stratification.

Fans can be provided to mix the air and produce an even air temperature but the air movement that these fans produce may then affect the rate of convective heat transfer from the container surface. In the experimental test a few small fans were used to eliminate any temperature stratification produced by the controlled air heating and the heat from the container itself, but the air flows that these fans produced was sufficiently small to avoid significantly affecting the convective heat transfer at the container surface.

The container, in the test, was loaded with a full complement of thirty five 200l drums in five carousels. These drums were all empty except for one which was filled with water and heated by a small immersion heater at a constant rate of 8W (the normal maximum heat load of each drum). The heat generated by the remaining drums was simulated by an electric heater at the base of each of the carousels.

The temperature of the container was measured by 27 thermocouples located on the inside of the pressure vessel, on the water-filled drum, at the door seal, on the container exterior and in the surrounding ambient air. The temperature measured by each thermocouple, together with the power supplied by the heaters, was recorded at 15 minute intervals by a data logger.

Two tests were performed. In the first the total power to the container was 280W, equivalent to 8W per drum, which is the maximum heat rating of the container. It was anticipated, however, that at this low heat load some thermocouples would not show any significant temperature rise above ambient. A second test was therefore performed at a much higher power (780W) in order to produce measurable temperature rises at all locations.

Because of the low heat input and its high thermal capacity the container heated up slowly. The first test was continued for six days but, even then, steady conditions had not quite been achieved and a slight extrapolation of the results was required. At the maximum heat rating of the container, 280W, the pressure vessel was 14°C above the temperature of the ambient air and the temperature of the water-filled drum was 1°C above the local air temperature. The surface of the container was only 0.4°C above the local air temperature.

## MODELLING OF TEMPERATURES DURING NORMAL TRANSPORT

The assessment of the temperature of the container during normal transport was performed by computer modelling using the TAU [Johnson, 1990] general purpose, finite element, heat transfer code. In order to represent all the potentially significant heat transfer mechanisms, a full 3-dimensional model of the container was required. Symmetry conditions, however, allow the model to be restricted to just one half of the container.

The model is shown in Figure 1. It represents the stainless steel pressure vessel and lid, the cork that surrounds it and the mild steel outer box. The thin stainless steel cladding around the cork was also modelled. For simplification, the box was represented as having flat, rather than corrugated, walls and the mild steel box sections which provide strength were also not included. These simplifications are not considered to have any significant effect upon heat transfer.

Before the computational model was used to determine the temperature of the Transhield 20 container under the conditions specified by the IAEA Regulations [1990], it was first validated by comparison with the experimental test described above. A calculation was first performed, at a total heat input of 280W, using reference values for the material properties of all the materials. The predicted temperature of the pressure vessel in this case was more than 10°C hotter than that measured in the experimental test.

A review was made of potential heat transfer mechanisms from the pressure vessel which may not be represented in the TAU model. No significant mechanisms were identified and it was therefore concluded that the most probable cause of the discrepancy was that the assumed thermal conductivity of the cork was in error. A check of the density of the cork in the container revealed it to be 16% greater than the reference value and the thermal conductivity is known to increase with density. This therefore supported the conclusion that the actual thermal conductivity of the cork was greater than had been initially assumed.

The calculation was repeated with the thermal conductivity of the cork increased by 40% and this produced good agreement with the results of the experimental test with most of the measured temperatures being predicted correctly to within ±1°C. It should be noted that an increase in thermal conductivity of the cork will be pessimistic with regards to temperature predicted during the fire test.

The boundary conditions in the model were modified to represent the transport conditions specified in the IAEA Regulations for a type B(U) container and the steady state calculation repeated. In the absence of any solar insolation the maximum accessible surface temperature was predicted to be 39°C. With the heat from solar insolation included the maximum surface temperature increased to 69°C and the temperature of the pressure vessel was predicted to be 72°C.

## MODELLING OF TEMPERATURES DURING THE THERMAL TEST

The Transhield 20 is a very large container and it had been predicted by Fry [1995] that producing adequate flame cover around such a size of container in a practical pool fire test would be very difficult. It was therefore decided to assess the effect of the thermal test on the container by calculation.

The thermal, or fire, test on the Transhield 20 container was modelled using the same TAU finite element model as used and validated for the normal transport temperature assessment. A 25mm air gap was, however, included between the cork and the outer cladding to represent the shrinking and charring of the cork during the fire. The container was modelled as being empty as this is both a simplifying and pessimistic assumption.

The IAEA Regulations specify that the container which is subjected to the thermal test must first be subjected to various impact tests. When performing the assessment of the thermal test by calculation the damage generated by the impact tests must therefore be represented.

The impact tests are described fully by Gaffka and Lawrence. The 0.3m drops onto a corner and base produced negligible damage and even the 9m drop onto one end produced no significant damage worthy of being represented in the thermal model. The 1m drop onto a punch, however, generated a hole through the steel cladding at one end and right through the cork so that the inner steel pressure vessel was exposed. In the subsequent thermal test this damage would expose the inner pressure vessel directly to the heat of the fire and must therefore be represented. A representation of the hole through the outer cladding and cork was therefore included in the TAU model.

During the fire the exposed area of pressure vessel will become a 'hot spot'. On the inside of the container the heat from the 'hot spot' will be transferred by radiation and convection to other areas of the pressure vessel. The radiation heat transfer was easily included in the TAU model but the convection heat transfer was more problematic.

If the heat were assumed to be transferred to only a small region at the top of the pressure vessel, then this might be argued to be an optimistic assumption. However, if the heat were assumed to be transferred to a large area of the pressure vessel then this would significantly reduce the average heat flux from this source and could again be argued to be optimistic. No correlations for convective heat transfer in this geometry were identified in published literature. It was therefore decided to determine the magnitude and distribution of the convective heat transfer inside the pressure vessel by use of a Computational Fluid Dynamics model.

## CALCULATION OF INTERNAL CONVECTION USING CFD

A model representing the inside of the pressure vessel was generated using the CFD code CFX-4 [1995]. Because of symmetry considerations only one half of the vessel was represented. The model contained 88,000 cells. The vessel was assumed to be empty of any internal structure or contents and to be filled with air. Heat transfer by radiation was not represented as this analysis was only concerned with convection.

To represent the 'hot spot' at the end of the pressure vessel, a circle of 0.3m diameter was given a fixed temperature of 400°C, the temperature that it was estimated it would reach during the thermal test. The size of this 'hot spot' was greater than the exposed area of pressure vessel so as to allow for the effect of conduction around the pressure vessel. The body and lid of the pressure vessel were given a fixed temperature of 38°C. Only a steady state calculation was performed.

The results of the CFD calculation showed that a plume of hot air would rise up from the 'hot spot' to the top of the pressure vessel and then travel along it, gradually cooling. The air in the upper half of the pressure vessel would become stratified while that in the lower half, below the level of the 'hot spot' would remain unheated.

Although the 'hot spot' was at 400°C, the temperature of the air in the hot plume was typically only 70°C. By the time the plume had reached the region of the seals round the door at the far end of the pressure vessel the maximum air temperature was predicted to be only 42°C, just 4°C above the assumed temperature of the vessel walls. The total heat flux from the 'hot spot' was predicted to be 82W.

The predicted heat flux to the pressure vessel at the seal region, as a function of angle around the vessel, is shown in Figure 2. The maximum of only 13W/m$^2$ occurs at the top of the vessel, as would be expected. A heat flux of this magnitude would heat up the 12.5mm thick stainless steel vessel at a rate of only 1°C per hour. This demonstrates that natural convection heat transfer from the 'hot spot' is not a significant heat transfer mechanism.

This convective heat transfer inside the pressure vessel was included in the TAU model by representing the air inside the vessel as a solid, of high thermal conductivity (so that it has a uniform temperature), to which heat is transferred from the vessel walls using a specified heat transfer coefficient. For the 'hot spot' at the end of the vessel an established correlation for natural convection coefficient from a vertical plate was used. This gave a heat flux some 20%

greater than that predicted by the CFD model. For the remainder of the vessel a 'top hat' function for heat transfer coefficient, as a function of height, was applied, as shown in Figure 2. This was considered to be a pessimistic representation.

## RESULTS OF THE THERMAL TEST CALCULATION

The thermal test calculation applied the boundary conditions representing a fire as specified in the IAEA Regulations. Thus a 30 minute 800°C fire was simulated. The calculated steady state temperature distribution for normal transport, with insolation included, was taken as the starting point for the transient calculation. The effect of solar insolation was also included during the cool-down phase. An internal heat load of 280W was assumed throughout the calculation.

The calculated transient temperatures at various locations are shown in Figure 3. The lid seals are predicted to experience a maximum temperature of 94°C, well below the high temperature limit for continuous operation of 120°C for the EPDM material. The pressure vessel, half way down its length, is predicted to reach a peak temperature of only 80°C.

The 'hot spot' at the end of the pressure vessel is predicted to reach a peak temperature of 147°C before starting to cool as soon as the heat from the fire is removed. Although this temperature of the 'hot spot' is significantly below that assumed in the CFD model the results of that calculation are still considered to be valid.

## CONCLUSIONS

The thermal performance of the Transhield 20 container has been analysed. This analysis involved a combination of calculations using a finite element model, calculations using a Computational Fluid Dynamics model and experimental trials on a full-scale prototype container.

The Transhield 20 container was demonstrated to easily meet all the requirements, for both normal transport and during a thermal test, stipulated by the IAEA Regulations.

## REFERENCES

CFX 4.1 User Guide. CFDS, AEA Technology, Harwell, 1995.

Fry CJ. The Use of CFD for Modelling Pool Fires. PATRAM '95, Las Vegas, USA, December 1995.

Gaffka AP & Lawrence G. Design & Approval Testing of the Transhield-20 Type-B ISO Container for Drummed PCM & Other Waste. PATRAM '98, Paris, France, May 1998.

Johnson D & Collier WD. TAU: A Computer Program for the Analysis of Temperature in Two- and Three- Dimensional Structures using the UNCLE Finite Element Scheme. AEA-RF-0010(R), 1990.

Regulations for the Safe Transport of Radioactive Material. 1985 Edition (As Amended 1990). Safety Series No 6, IAEA, Vienna, 1990.

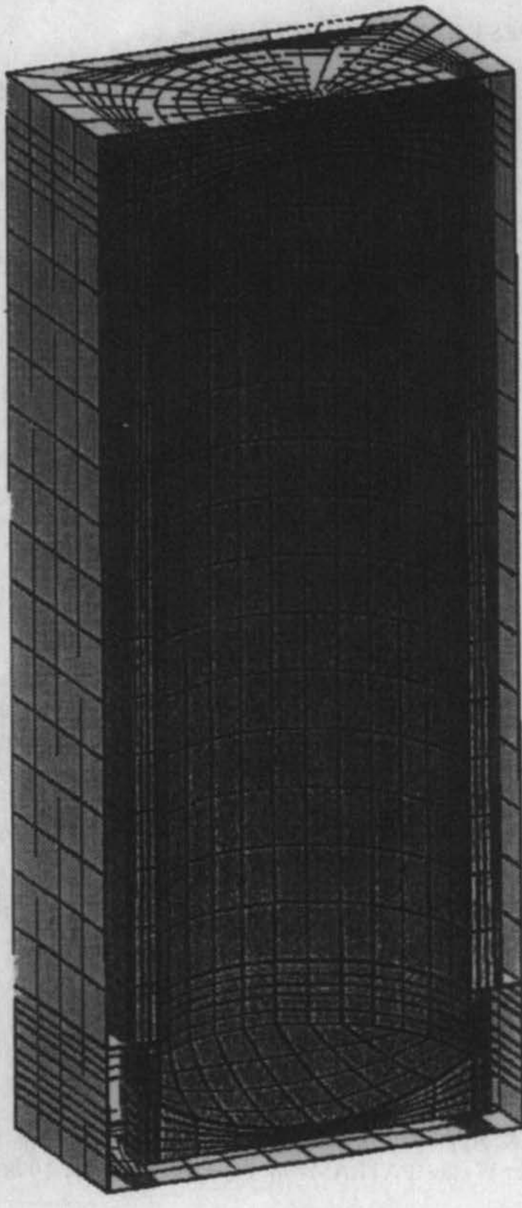

FIG.1 THE TAU MODEL

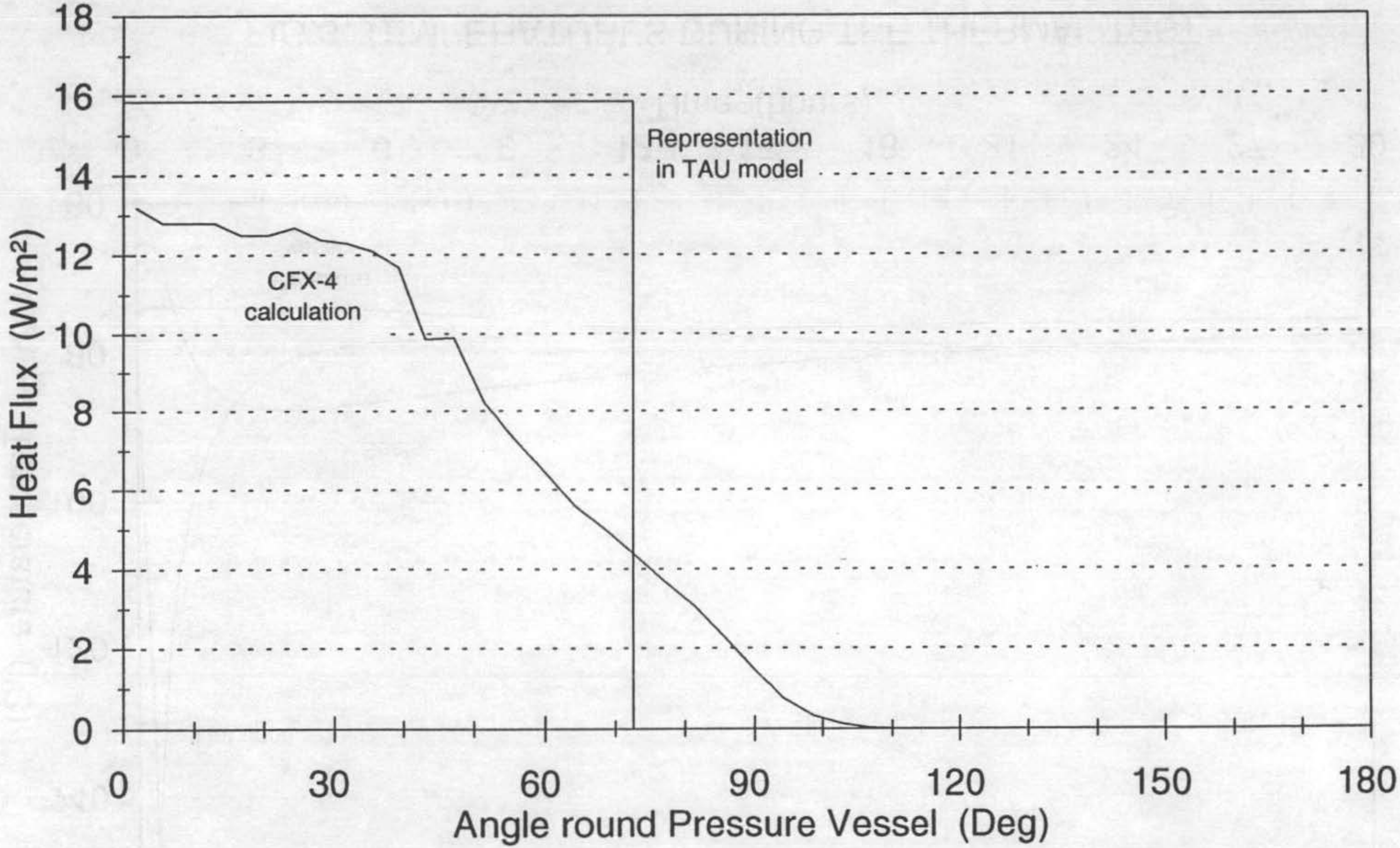


FIG.2 CONVECTIVE HEAT FLUX AROUND THE VESSEL

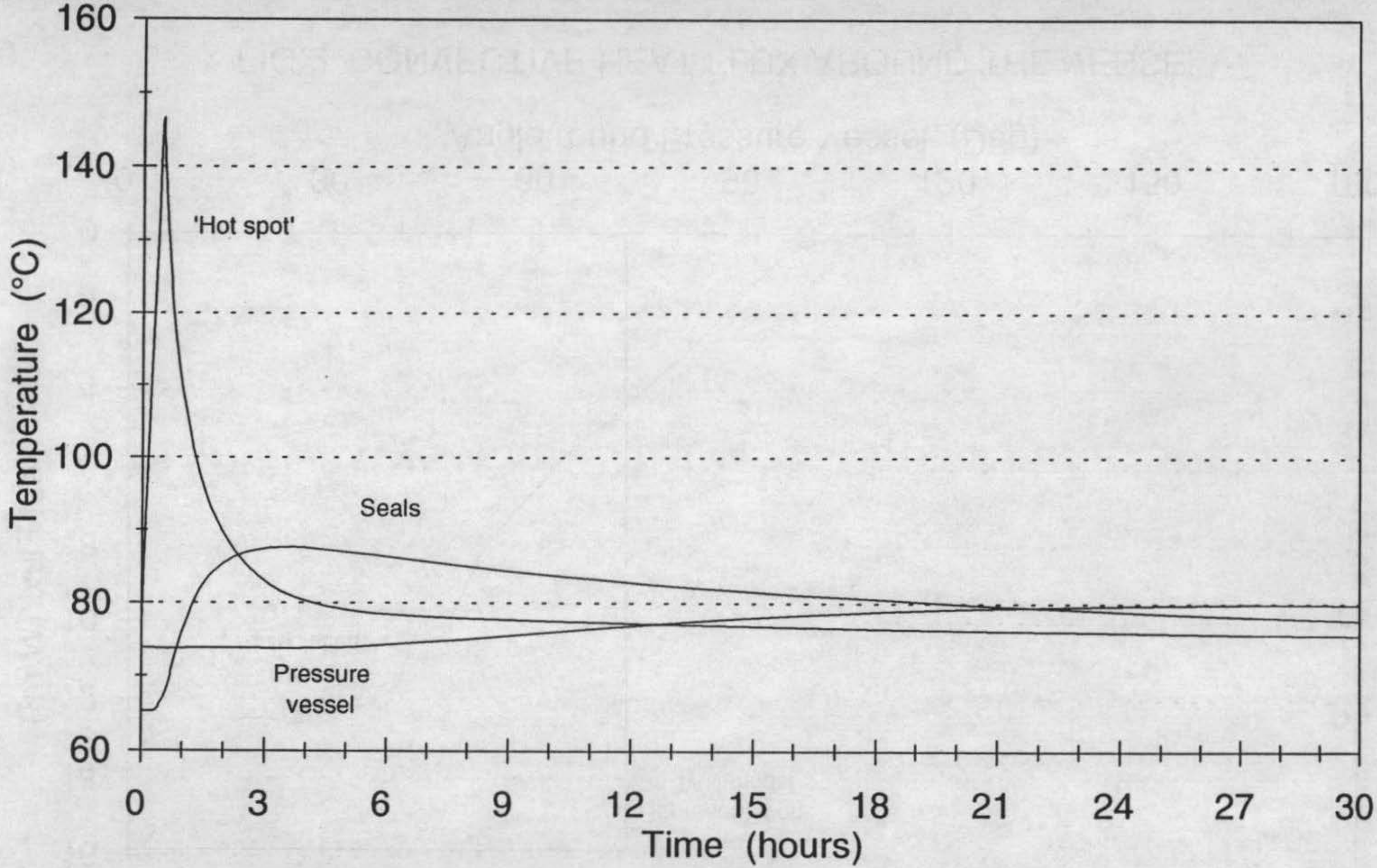


FIG.3 TEMPERATURES DURING THE THERMAL TEST