Flask Fluid Flow Simulation Using CFD

W.E. Swindlehurst[1], E. Livesey[1], D. Worthington[2]

[1]*British Nuclear Fuels, plc, Risley, Warrington*
[2]*CHAM Ltd., Wimbledon, London, United Kingdom*

INTRODUCTION

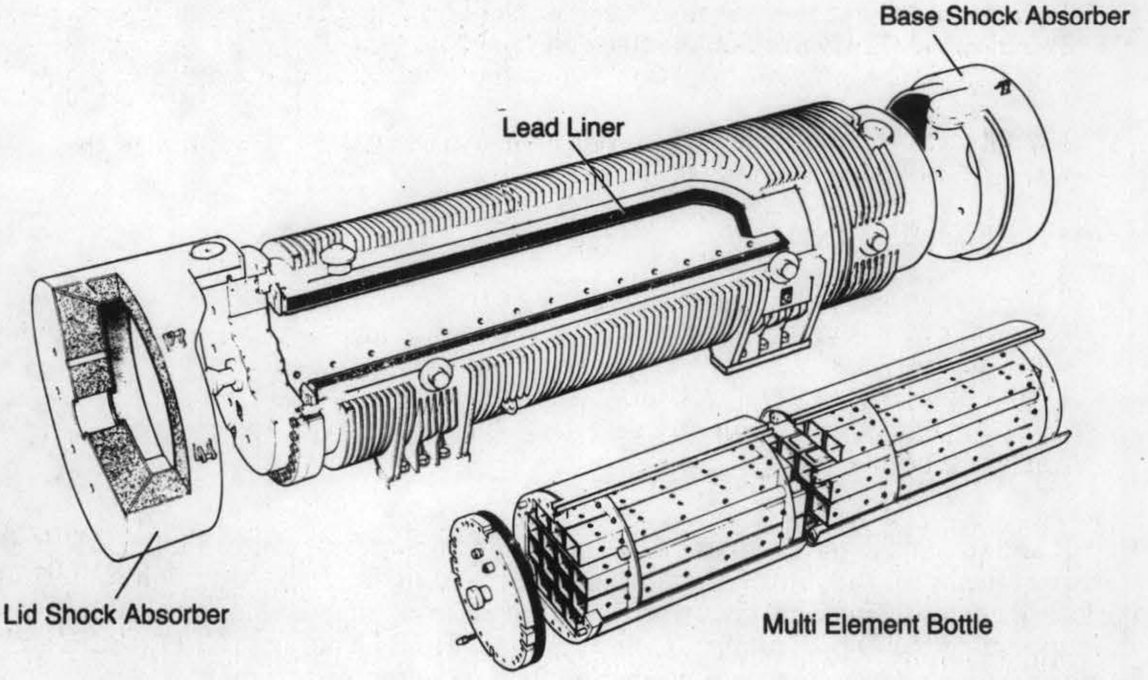
BNFL and its subsidiary Company, PNTL, design and operate waterfilled LWR fuel transport flasks for the international transport of irradiated fuel. Although some 150 flasks are currently in operation, new flask designs are being developed. As part of the supporting R&D programme, Computational Fluid Dynamics (CFD) codes are being investigated as a means of predicting fluid movements and temperatures within the complex internal geometry of flasks. The ability to simulate fluid flow is particularly important when convection heat transfer is significant. Although obviously relevant to water filled flasks, the technique is applicable to dry flask thermal assessments (where experience shows that convection heat transfer is often underestimated). Computational Fluid Dynamics has emerged in recent years as an important technique in engineering design and safety assessments. Cheaper computing and the development of general CFD codes allows complex engineering structures to be analysed. However, because of this complexity, it is essential that the application and associated modelling assumptions are critically reviewed. To assess the ability of a CFD code to model flask internals, the code PHOENICS has been used to model the fluid movements in a BNFL Excellox-type flask and the results compared with test data.

EXPERIMENTAL DATA

Data from an extensive programme of experimental work on the thermal behaviour of a water-cooled LWR flask (EXL3A design) were used for model validation. This programme of work has been reported earlier (Burgess et al 1983) and is summarised briefly below. The flask used in these experiments is shown in detail in Figure 1, together with a sectional diagram in Figure 2 showing the main components. The flask itself comprises two regions:

(i) an inner thin-walled container referred to as a Multi-Element Bottle, or MEB. Inside the MEB, the fuel elements are held within a support frame which, in this instance, consisted of an assembly of 14 square-sectioned channels. The MEB is sealed and partially filled with water to provide a low thermal-resistance pathway from the fuel to the MEB wall. To enhance heat

Fig 1. Excellox 3A Irradiated Fuel Transport Flask



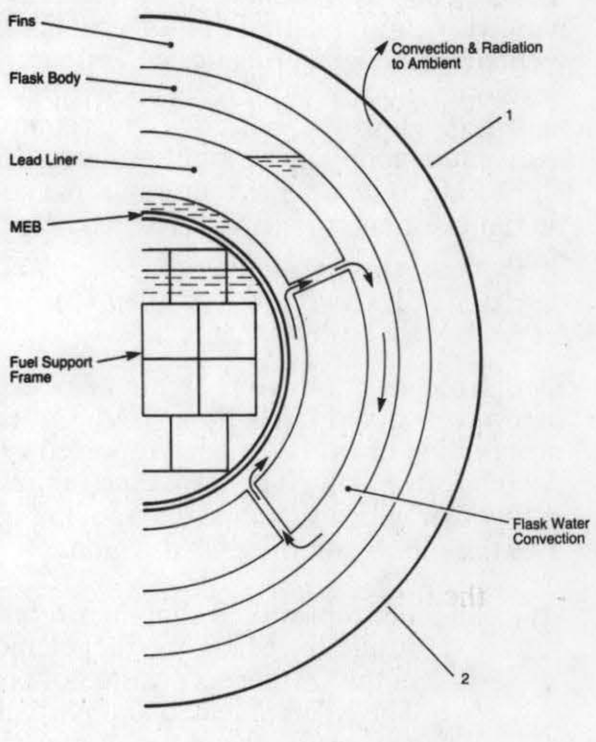
transfer, the fuel support frame channels contain holes at regular intervals along their lengths to allow water circulation within the MEB;

(ii) a cylindrical lead-liner surrounds the MEB and, finally, an outer finned steel shell provides the structural integrity and containment boundary for the flask.

The MEB/liner and liner/flask body interfaces are again water-filled with an associated ullage space. Holes in the liner allow water to circulate from the MEB outer surface to the flask body.

Each of the 14 BWR fuel assemblies was repesented by a 7 x 7 array of steel tubes. The decay heat load in each element was simulated by a spiral heater cable.

Fig 2. Excellox Flask Cross-Section



The MEB internals, MEB/body cavity and body/fin region, were extensively instrumented. Over 140 thermocouples were used to measure surface and water temperatures.

Within the MEB, thermocouples were located on the central cross-section while those outside were distributed along the full length of the flask. In addition, air temperatures were measured below and above the flask and at mid-fin depth.

Equilibrium temperature and pressure measurements were taken over the range 15 to 35 kW total heater power.

THE MODELLING

The CFD code PHOENICS (Rosten and Spalding 1987) is a finite volume computer program designed to provide a general purpose capability for simulating fluid flow, heat transfer and chemical reaction processes. The physical region of interest is divided, using a grid system, into discrete volumes. Equations describing the conservation of mass, momentum and energy are then solved numerically.

The starting point for the analysis is a set of simultaneous differential equations governing the water flows in the flask. These relate to the conservation of momentum in each of the co-ordinate directions considered, the conservation of matter and the conservation of thermal energy. Because the flask is cylindrical and generally axisymmetric, two-dimensional flow was assumed for this initial study. A section normal to the axis of the flask was considered sufficient to represent the main flow features, illustrated in Figure 2.

In the analysis of the flask, the PHOENICS code solved six conservation equations, predicting the two-dimensional flow, pressure and temperature fields.

The complex geometry of the flask was represented by two models - the predominantly rectangular geometry of the MEB by a cartesian finite volume grid and the cylindrical features of the flask body and lead liner by a cylindrical polar grid. The meshes are shown in Figures 3 and 4.

Reductions in available flow areas arise from the geometry, eg the flow around the inside boundary of the MEB shell and the flow through the circular convection holes in the lead liner and fuel support frame. This phenomenon was represented by changing individual cell properties to represent the proportion of each cell volume and face area available to the fluid.

Other aspects of the model were:

- a constant heat transfer coefficient was used to represent convection from the external finned surface of the flask to ambient air;
- the fuel was not modelled explicitly - the energy input was specified as a uniform power per unit volume and an empirical correlation used for momentum loss through the pin array;
- the ullage spaces were treated as quasi-solid regions with enhanced conductiv-

Fig 3. MEB Mesh

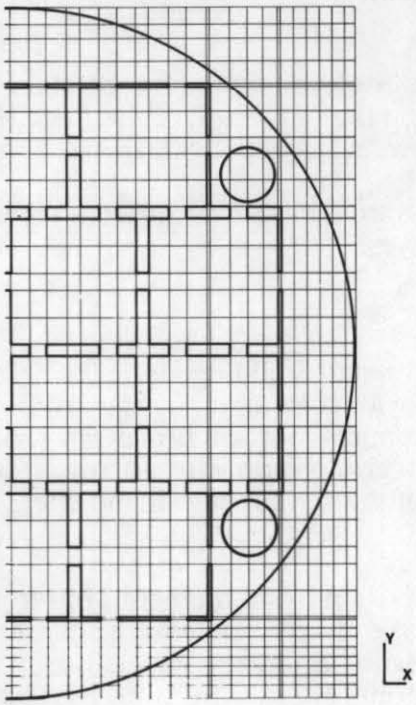
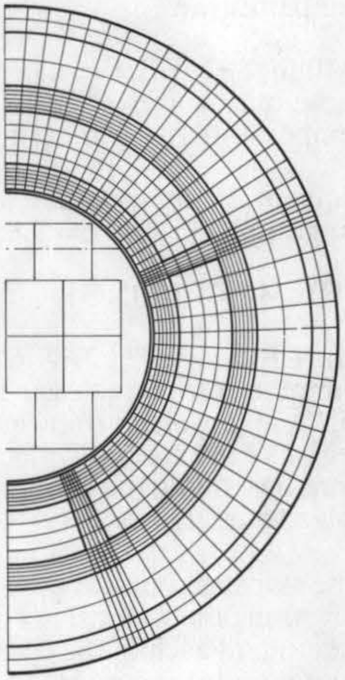


Fig 4. Flask Mesh



ity to represent the effects of convection and two phase heat transfer. This simplification was used at this stage to avoid having to solve convection and mass transfer in the ullage spaces which were considered secondary effects;

- from an initial assessment of the flows, it was assumed that the flows are laminar in the MEB and turbulent in the flask cavity. The two parameter k-e turbulence model was used.

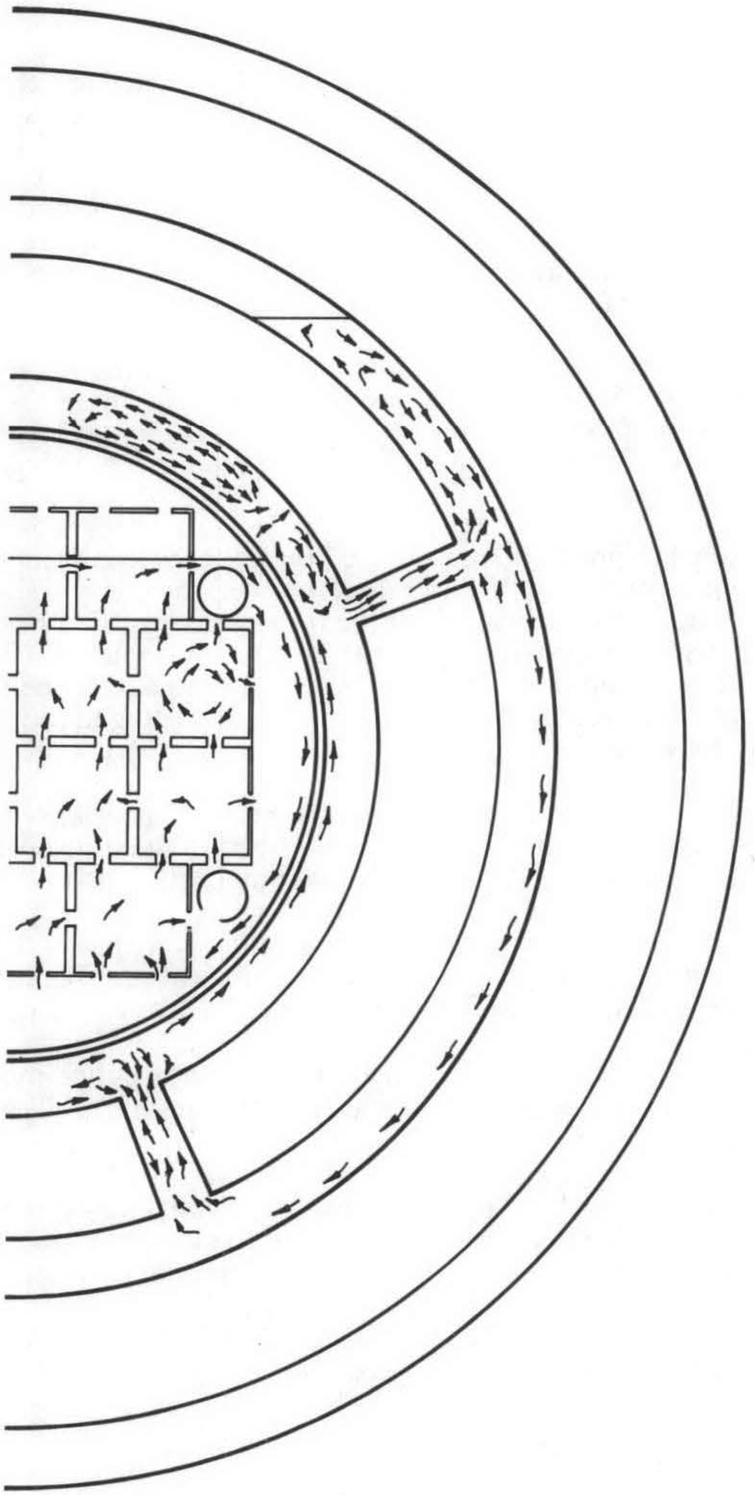
The iterative calculation was initialised by obtaining a solution in the flask model based on a uniform heat flux at the inner boundary. The resultant inner boundary temperatures then became the outer boundary condition for the MEB model. The MEB model was then run to determine a heat flux distribution at the outer shell and this in turn was used as the inner boundary condition during a re-run of the flask model. The iteration process was applied until convergence was obtained for both MEB and flask models, matching the temperature and heat flux distributions at the common boundary.

MODELLING PREDICTIONS AND COMPARISON WITH TEST DATA

The predicted water flows for a 15 kW heat load are illustrated in Figure 5, which is a simplified version of the computational graphical output.

Within the MEB, it can be seen that flows are as would be expected. Flow is generally up through the fuel compartments and down the inside surface of the MEB shell. Inflow occurs through the convection holes in the lower half of the

Fig 5. Computed Water Flows



compartment array and outflow through the upper with a recirculation loop predicted in the upper outermost channels. Some water passes from the outer compartments on the bottom row through the outer circulation holes, the water then being entrained with the flow down the MEB shell. Temperatures increase with height within the fuel compartments.

In the flask cavity, a clear thermosyphon loop is predicted to occur between the inner and outer annuli, although the details of the flow patterns are complex. There are two recirculatory flow cells in the upper part of the MEB/lead liner annulus and some evidence of recirculation around the upper convection port. As the flow through this port passes into the lead liner/flask body annulus, it turns upwards and forms a recirculation region below the water free surface. Flow from this free surface travels down the flask body into the lower regions of the annulus. There are no large scale recirculations in the lower region.

Figures 6 and 7 give a comparison of the predicted and measured temperatures along the 45° and 135° radii (radial positions 1 and 2 in Figure 2). Temperatures in the upper half of the model tend to be lower than the test results but generally agree within 5°C. Predictions in the lower half are poorer, giving higher temperatures than measured.

The modelling has produced reasonable flow patterns, confirming flow paths which were previously assumed to exist. However, the comparison with measured temperature data indicates that the resulting convection heat transfer is underpredicted. The second stage of the investigation will be to determine the sensitivity of the results to the modelling assumptions and refine the simulation. Features of the present model which may be contributing to the overprediction of temperatures are:

- the model results are grid dependent. It is therefore necessary to refine the mesh, particularly in areas of high gradients in the transport variables (such as adjacent to the MEB and flask inner surfaces);

- examination of the results indicates that the characteristics of the flows within the MEB vary with location. There are areas of possible transition or turbulent flow - the MEB was analysed using a laminar simulation;

- the correlations used for convection at the solid surfaces and momentum losses through the fuel need to be verified (for the specific application) or refined;

- there is evidence in the test data of three-dimensional flow effects. The present modelling has been restricted to two-dimensional simulation which may not be sufficient to represent essential flow details.

When the model has been refined and verified against the 15 kW test data, it will be run to simulate the higher power tests.

CONCLUDING REMARKS

A CFD model has been developed that simulates the fluid flow and heat transfer that occurs in an Excellox irradiated fuel transport flask. These water-filled

Fig 6. Comparison of Temperatures at Radial Position 1

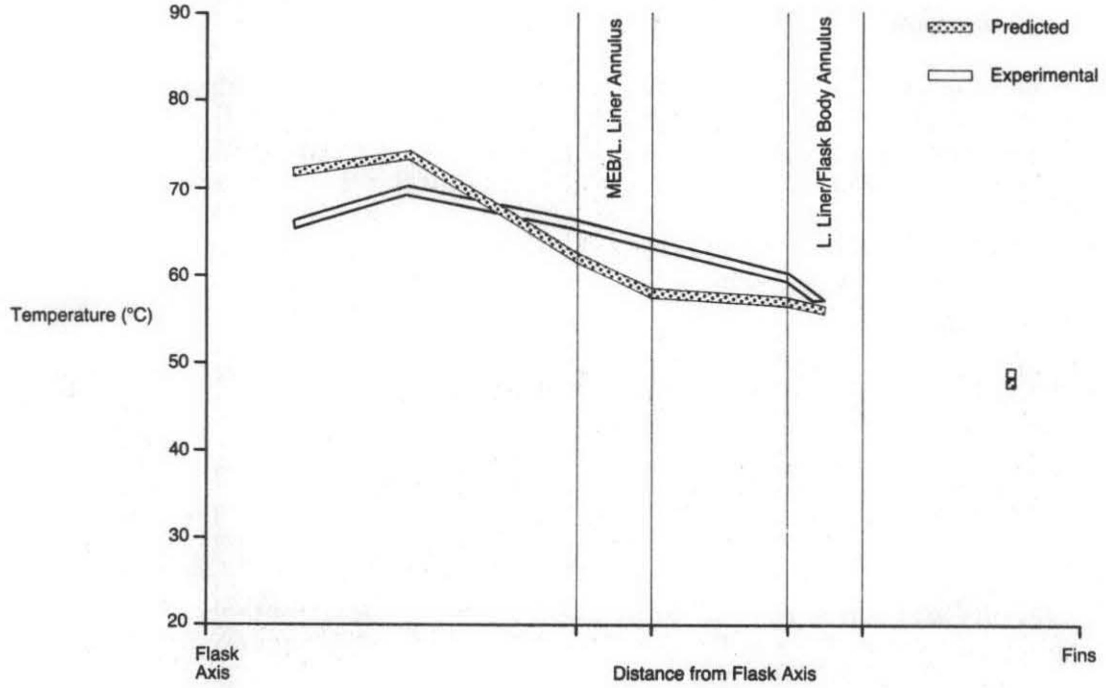
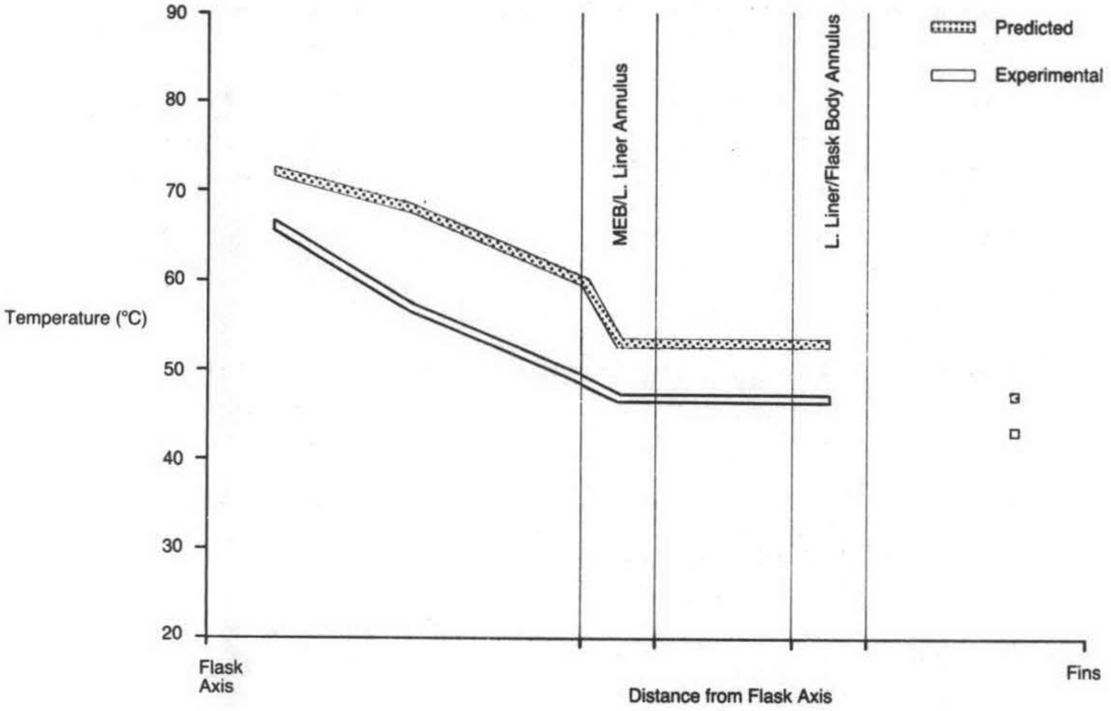


Fig 7. Comparison of Temperatures at Radial Position 2



flasks have a complex internal geometry and presented a major challenge to the code. In this first stage of study, a number of simplifications have necessarily been made to make the problem tractable. Even so, the predicted flow patterns and agreement of temperatures with test data indicate that the appropriate physical phenomena are being simulated.

Although the present study is based on a water-filled flask, the technique is equally applicable to dry flasks and fuel storage where gas convection can be an important component of the heat transfer. Thermal tests on a dry Excellox flask (Middleton and Livesey 1986), conducted as part of the BNFL experimental programme (Livesey and Swindlehurst 1988), will provide validation data for a revised dry flask version of the present CFD model.

REFERENCES

Burgess M I, Spiller G T, Livesey E. 'Thermal Trials on a Water Cooled LWR Flask', PATRAM 83, paper 19.

Livesey E, Swindlehurst W E. 'Experiments Conducted in Support of Transport Flask Heat Transfer Assessments'. International Conferences on Transportation for the Nuclear Industry, Stratford-on-Avon, England, Paper 46 (1988).

Middleton J E, Livesey E. 'Full Scale Experiments to Determine the Thermal Response Following Loss of Coolant Water from a Flask Containing Irradiated LWR Fuel'. PATRAM 86, paper 286.

Rosten H I, Spalding D B. 'The PHOENICS Beginners Guide and User Manual', CHAM Limited, London, England. CHAM TR/100, (1987).