



A three-dimensional study of the heat and mass transfer within the irradiation space of the HIFAR fuel element

G. H. Yeoh & G. J. Storr

*Australian Nuclear Science & Technology Organisation (ANSTO),
Lucas Heights, Australia.*

Abstract

A three-dimensional mathematical model for the HIFAR fuel element is described. The thermal-hydraulics behaviour within the irradiation space in the fuel element has been investigated. Effective cooling of the target cans could be obtained from the current operational settings. Complex flow structures were observed in the liner and support tube nose cones area. Large temperature gradients were found across the UO_2 pellets and MgO powder. The predicted central temperature in the UO_2 pellets was 2550 K. Maximum surface temperatures on the outer wall of the target cans were found to be 370 K for the first can and 380 K for the second can respectively. These values were comparable with experimental measurements and within the normal operating limits.

1 Introduction

The High Flux Australian Reactor (HIFAR) is a tank-type reactor, moderated and cooled by heavy water circulating under atmospheric pressure. One of its major functions is to provide benefits to health care such as the production of medical isotopes for the pharmaceutical industry.

We report on the ongoing work directed toward a better understanding of the complex thermal-hydraulics behaviour in the space within a HIFAR fuel element where sealed target cans containing uranium oxide pellets are irradiated.

The ability of a Computational Fluid Dynamics (CFD) code to simulate the thermal-hydraulics behaviour was investigated. Modelling the process was challenging because of, the complicated geometry of the irradiation rig and target cans, and the requirement to accurately model the heat transfer which was coupled to the fluid flow through solid multi-material regions.

Flow structures and temperature profiles within the irradiation space of the fuel element were predicted. Temperatures were compared with measurements wherever available.

2 Description of HIFAR fuel element

Figure 1 shows the schematic view of a whole fuel element in HIFAR [1]. Heavy water enters the fuel element through the "skirt" and flows upwards. When the coolant reaches the fuel liner, a small fraction of the heavy water enters the liner via four small holes on the liner nose cone. The rest of the coolant flows through

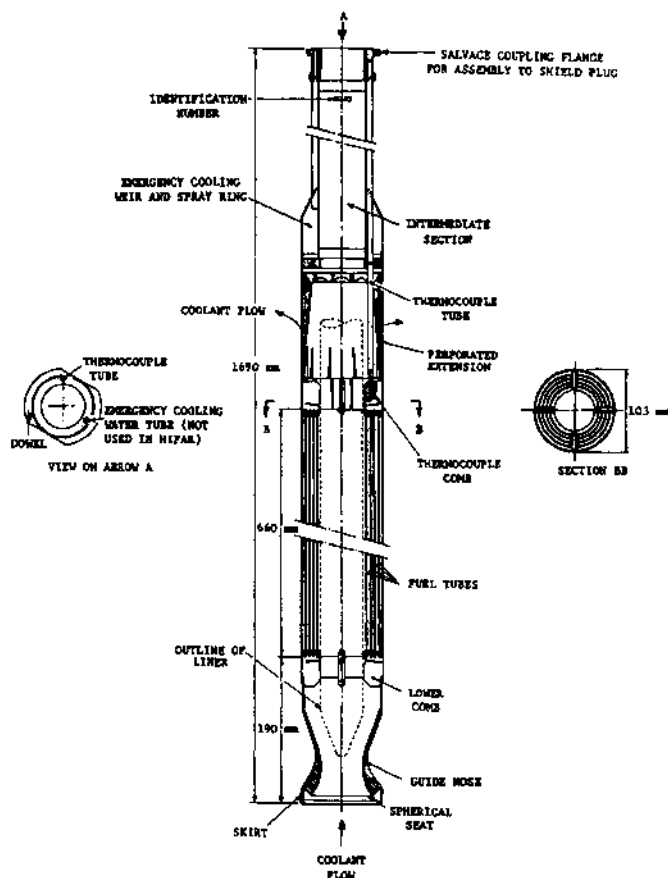


Figure 1: Schematic drawing of the HIFAR fuel element



the fuel tubes as can be seen from the schematic diagram. Details of the placement of the target cans and irradiation rig in the fuel liner [2] can be seen in Figure 2(a). The resulting low velocity flow inside the liner moves up through a vertical space containing an irradiation rig and the target cans via five holes on the support tube nose cone. Water flowing over the surfaces of the target cans removes the fission heating produced in the pellets.

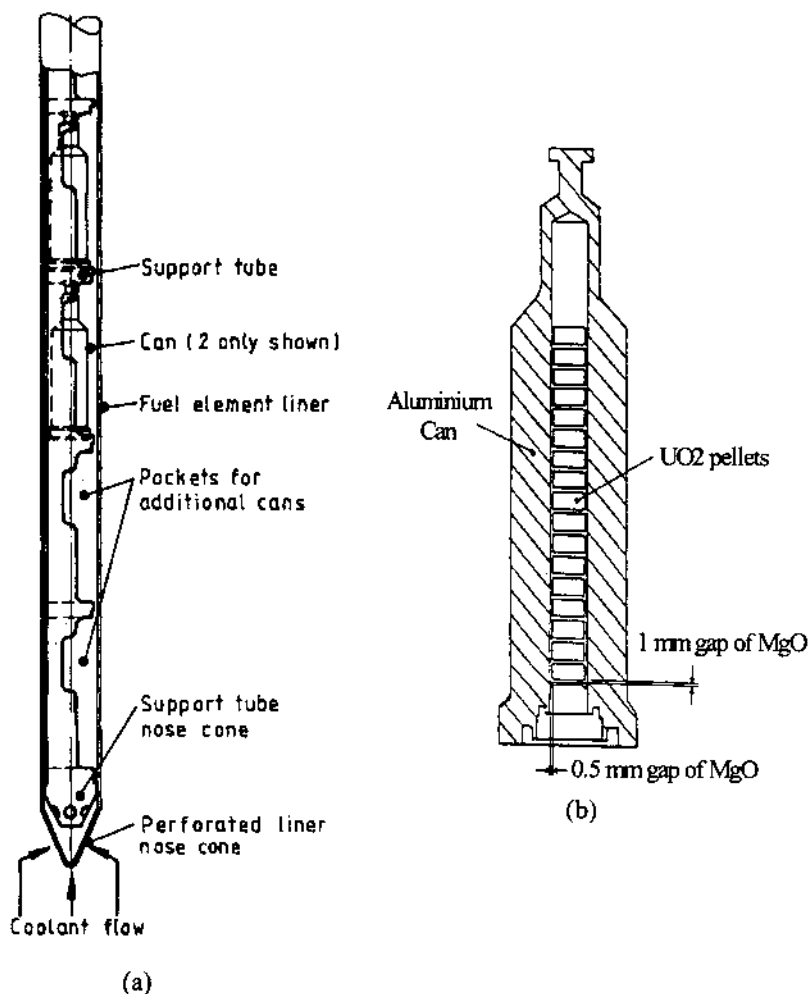


Figure 2: Schematic drawing of the (a) irradiation rig and placement of target cans in the liner and (b) UO₂ pellets arrangement in a target can.

As the pellets generate large amount of heat during the irradiation process, and in order to effectively dissipate the heat into the coolant, the components that include the target can, rig and liner are generally made of Aluminium (Al). For the



346 *Advanced Computational Methods in Heat Transfer VI*

current production of medical isotopes (Mo99) with a U_{235} enrichment of 2.2 %, 17 Uranium Oxide (UO_2) pellets and Magnesium Oxide (MgO) powder are arranged in the centre core of the target can [2] as shown in Figure 2(b). There is a horizontal gap of 1 mm filled with MgO powder in between the individual UO_2 pellets. MgO powder also separates the UO_2 pellets from the can walls with a vertical gap of 0.5 mm. It is noted that above the 17th pellet, the MgO powder does not fill completely the whole vertical space but rather the area is left filled with air.

3 Model description

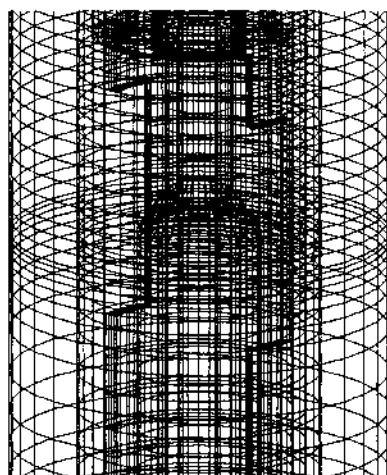
3.1 Governing equations

The heat and mass transfer system of the HIFAR fuel element considered the time-averaged conservation equations for mass, momentum and energy. In order to achieve closure of the turbulent flow, the standard two-equation turbulence k - ϵ model was used. The model solved two-differential equations for the turbulent kinetic energy (k) and its dissipation rate (ϵ). The effect of buoyancy was included in the governing equations to account for regions in the fluid with very low velocities.

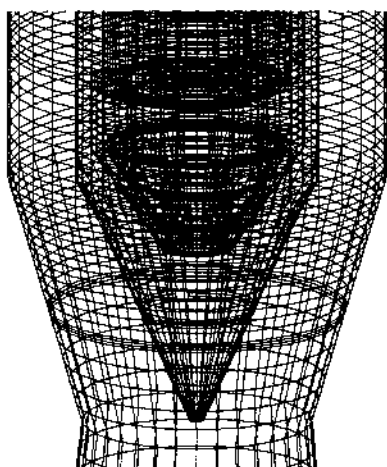
3.2 Grid generation for HIFAR fuel element

The grid generation for the HIFAR fuel element posed a challenging task because of its inherent intricate details. We have restricted our present investigation by considering two target cans residing on the irradiation rig inside the liner. The first can sat on the platform just above the support tube nose cone with the second can placed in the next pocket. The geometry beyond the second can was simplified by arbitrarily attaching a cylindrical pipe of 1.5 m high. This allowed the coolant to achieve fully developed flow conditions at the extended boundary. It would have been a momentous task in attempting to model the whole 17 individual UO_2 pellets separated with MgO powder. In the current simulation, we have considered the heat source as a solid cylindrical block in representing the UO_2 pellets. Nevertheless, the presence of MgO powder separating the pellets from the can walls and air gap was retained to accurately model the heat transfer characteristics as the fission heat dissipated away from the pellets.

Figure 3(a) illustrates the computational mesh generated for the target can residing in the pocket of the irradiation rig. The mesh created for the perforated liner and support tube nose cone regions can be seen in Figure 3(b) respectively. The commercial CFD code CFX4.2 [3] was used, employing the multi-block construction approach that resulted in a total of 1524 blocks and a mesh of 173493 control volumes for the geometrical structure. Curvilinear Body-Fitted Coordinates were used for the geometry generation.



(a)



(b)

Figure 3: Computational mesh generated for (a) the target can residing in the pocket of the irradiation rig and (b) the perforated liner and support tube nose cones.

It is noted that we have concentrated on more of the geometrical details inside the liner. We have not modelled the fuel tubes outside of the liner as for the current analysis we were only interested in the flow and heat transfer behaviour in the vicinity of the target cans and the flow entering in the liner around the nose cone area.

3.3 Boundary conditions

During the normal operating limits of HIFAR, it is known that a mass flow rate of 16 kg s^{-1} is established for a single fuel element. Based on this value, normal velocity was calculated and applied at the inlet. The temperature of the heavy



water entering the fuel element is at 318 K. This was used as the inlet boundary condition for the temperature. Turbulence kinetic energy κ and its dissipation rate ϵ were obtained from empirical functions of:

$$0.002 u_{in}^2 \quad ; \quad \frac{\kappa^{1.5}}{0.3 l_e} \quad (1)$$

where u_{in} is the normal velocity and l_e is the hydraulic diameter respectively.

At the outlet, mass flow boundaries were applied. It has been found from experiments [2] that 1% of the total flow of 16 kg s⁻¹ enters the liner while the rest of the heavy water flows outside of the liner. CFX4.2 allows the specification of fractional mass flow rates and the option was used for the current simulation. For the rest of the variables, fully developed boundary conditions were employed.

Solid boundary conditions were implemented for the target, irradiation rig, liner and fuel element walls. No-slip condition was employed for the velocities. All the wall temperatures were calculated using energy balance except for the fuel element walls where adiabatic condition was applied. The boundary conditions for turbulence kinetic energy κ has its normal derivative at the wall equal to zero. The dissipation rate ϵ was obtained through the relation

$$\epsilon = \frac{C_\mu^{3/4} \kappa^{3/2}}{\kappa_c \Delta} \quad (2)$$

where C_μ is the standard turbulence constant, κ_c is the Von Karman constant and Δ is the distance of the nearest grid point from the wall boundary.

3.4 Computational procedure

The governing equations to be solved by matrix solution techniques were formed by integrating the system of equations using the finite volume technique over small control volumes. Block Stone solver was used for all the equations except for the pressure correction where the Algebraic Multi-Grid was used to accelerate convergence. The velocity-pressure linkage was handled through the SIMPLE algorithm. The non-linear partial differential equations were marched to steady state. Convergence was deemed to have been reached when the normalised residual mass was less than 5.0×10^{-4} .

It was obtained from calculation considering the neutron fluxes [4] that the total heat generated for each can was 3805 W. Of the total heat produced, about 89.5 % is generated from the UO₂ pellets. The reduced power of 0.895×3805 W was therefore used as the input data for the volumetric heat source for each of the UO₂ cylindrical solid block during the calculations.

Data from Touloukian et al. [5] have shown that the thermal conductivity of Al did not exhibit much variation with temperature - between 300 K - 2000 K; and was therefore be taken to be constant (220 Wm⁻¹K⁻¹) for the solid material heat transfer calculations. On the contrary, for the same range of temperatures, there was large variability in the thermal conductivities of UO₂ and MgO with temperature. The UO₂ conductivity varied from 7.24 Wm⁻¹K⁻¹ (419 K) to 2.26



$\text{Wm}^{-1}\text{K}^{-1}$ (1800 K) while MgO conductivity varied from $0.389 \text{ Wm}^{-1}\text{K}^{-1}$ (373 K) to $0.801 \text{ Wm}^{-1}\text{K}^{-1}$ (1101 K) respectively. This large variation would significantly influence the heat transfer characteristics for the fission heat generated in the UO_2 pellets and the heat conducted through the MgO powder. The strong dependence of the thermal conductivities on temperature was accounted in the analysis.

4 Results and discussion

Figures 4(a) and 4(b) show the velocity vectors around the liner and support tube nose cones with the former plotted on a plane of $x = 0$ while the latter on a plane of $z = 0$. A clear distinction of the high velocity flow outside of the liner and low velocity flow inside of the liner could be readily observed from the two figures. The design of the perforated holes on the liner nose cone was seen to significantly restrict the fluid flow that contributed to the low velocity flow with majority of the flow entering through the bottom hole rather than the three side holes. This was evidenced by the magnitude of the velocity vector through one of the side holes being smaller than the velocity vector through the bottom hole (see Figure 4(a)). Multiple vortices of recirculating flows could be observed just above the bottom hole of the liner nose cone and the entry of the coolant through the support tube nose cone holes. Nevertheless, as the heavy water moved up through the vertical space containing the target can, the fluid flow was gradually becoming more uniform. This behaviour was also reflected for the upper regions of the target can.

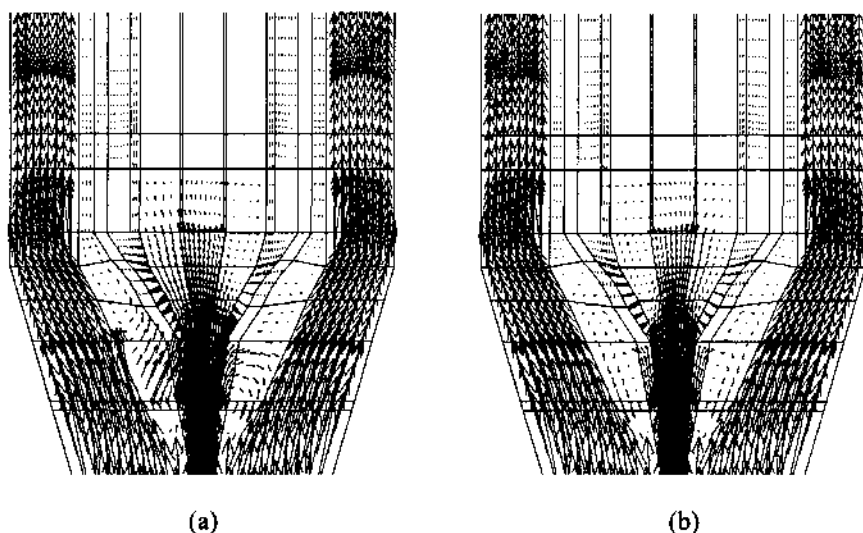


Figure 4: Flow structures of the heavy water on the outside of the liner and its entry into the liner and support tube nose cones. It is noted that a length of 1 cm of the velocity vector corresponds to a velocity of 3.23 ms^{-1} .

Figure 5 shows the shaded contours of the temperature distribution during the simulated fission process. The central temperature in the UO_2 pellets was measured in the experiments [6] by the insertion of an Iron (Fe, melting temperature – 1808 K) wire into a hole drilled through the centre of (3 or 4) pellets in the target can. The central temperature was measured at around 2035 K. The predicted central temperature in the UO_2 pellets during the simulation was 2550 K. Based on the crude experimental measurement, the calculated value agreed very well with the measured data.

The target can with the most efficient cooling was observed when it was positioned in the most bottom pocket. The temperature of the fluid in this region was 320 K. This showed only an increase of 2 K from the ambient fluid temperature of 318 K. In the vicinity of the second can, the fluid temperature increased to 328 K.

The temperature drop across the UO_2 pellets, MgO powder and Al can was similar for both of the target cans. Across the UO_2 pellets, a temperature gradient of 950 K was predicted. Across the MgO powder, the simulation yielded a gradient of 545 K. These large temperature gradients can be seen from the temperature distribution in Figure 5. Across the thickness of the Al can, a difference of 20 K was obtained as would be expected due to the high conductivity of Al.

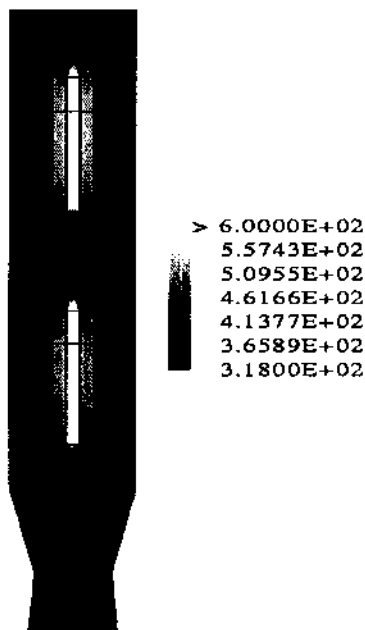


Figure 5: Temperature distribution of the dissipated heat during fission process.

Uniform radial surface temperatures on the outer wall of the first can were obtained in the simulations. The maximum temperature was calculated to be at



370 K. Similar behaviour was also observed for the second can with a maximum surface temperature of 380 K (above the boiling temperature). Nevertheless, the highest fluid temperature on the adjacent grid point of 0.3 mm away from the can boundary remained well below the boiling temperature of the heavy water at approximately 335 K.

An earlier experiment [7] carried out with an instrumented rig with thermocouples attached to two out of four cans on the HIFAR fuel element gave can surface temperatures of 377 K and 385 K respectively. It should be noted that the thermocouples attached to the surface of the two cans were positioned in the third and fourth pockets, in other words, above the current second can location. Although no comparison could be made directly with experimental data on the exact locations, the calculated surface temperatures were consistent with the range measured. Future modelling work on the HIFAR fuel element will be carried out to include the additional two cans on the existing two. Surface temperatures of the third and fourth cans will be compared with the data measured.

5 Conclusion

The formulation and details of a three-dimensional model to study the thermal-hydraulics behaviour within the irradiation space of the HIFAR fuel element have been described. Computational mesh for the HIFAR fuel element geometrical structure was successfully created using the multi-block construction. The model was capable of calculating complex flow of the heavy water inside the liner containing the irradiation rig and target cans. The model adequately represented the heat transfer characteristics, as predicted temperatures were consistent with experimental measurements.

References

- [1] Australian Nuclear Science Technology Organisation (ANSTO), *HIFAR Descriptive Manual*, HIFAR Safety Document Section 3, 1972.
- [2] Chapman, A.G. and Hargreaves, N.D., *Out-of-pile Burnout Experiments in a Full-Scale Simulation of an Irradiation Rig in a HIFAR Hollow Fuel Element Facility*, ANSTO report: AAEC/E632, 1986.
- [3] CFDS – AEA Technology, *CFX User's Guide*, Version 4.2, 1995.
- [4] Private communication with Graham Robinson, Nuclear Technology Division, ANSTO.
- [5] Touloukian, Y.S., Powell, R.W., Ho, C.Y., Klemens, P.G., *Thermophysical Properties of Matter*, 2, 1970.
- [6] Private communication with David Hurwood, Nuclear Technology Division, ANSTO.
- [7] Sirkka, P.T., *Instrumented Hifar Fuel Element Cakestand Rig*, ANSTO report, 1982.