AU0221633

Computer Analysis of Thermal Hydraulics for Nuclear Reactor Safety

JIYUAN TU, GUAN YEOH, GREG STORR and DAVID BEATTIE
Australian Nuclear Science and Technology Organisation
Private Mail Bag 1 Menai NSW 2234

SUMMARY. This paper gives an overview of ANSTO's capability and recent R&D activities in thermal hydraulic modelling for nuclear reactor safety analysis, particularly for our research reactor, HIFAR (HIgh Flux Australian Reactor) and its intended replacement, the Replacement Research Reactor (RRR).

1. Introduction.

Nuclear reactor safety continues to receive intensive attention the world over and at ANSTO. This includes research dealing with interrelated systems behaviour, human factors and human-machine interfacing, and research related to specific technological questions. Of these, the various aspects of the thermal hydraulics of cooling the reactor core remain paramount. Since full-scale experiments of hypothetical abnormal reactor events are not practical, computer analyses of the thermal hydraulics of such events play a unique role in nuclear reactor safety.

Several tools contribute to ANSTO's capability in thermal hydraulic modelling for nuclear reactor safety analysis, particularly for our research reactor HIFAR, and its intended replacement, the Replacement Research Reactor (RRR). Computer codes available at ANSTO include:

- RELAP (developed in US) -- A code for reactor system thermal-hydraulics analysis;
- CFX (UK) -- A general computational fluid dynamics code we have used for thermal-hydraulic analysis in reactor fuel elements; and
- HIZAPP (ANSTO) -- A code for coupling neutronics with thermal-hydraulics for reactor transient analysis.

In addition, since most of the model and code developments have been devoted to power reactors that operate at high-pressures, a number of R&D activities have been undertaken at ANSTO covering specific thermal-hydraulic behaviours that occur under low-pressure conditions in research reactors. Of these, the ANSTO's low-pressure subcooled boiling models will be discussed here.

2. RELAP modelling of HIFAR reactor.

Heat flux and power densities in a research reactor like HIFAR [1] are considerably higher than that for power reactors and fuel element melting points are comparatively lower. A consequence is a requirement for the shutdown of research reactors prior to events that would impair the core cooling significantly, so as to avoid melting of the fuel. Hence, the thermal-hydraulic response of any research reactor to various hypothetical events need to be carefully analysed. These analyses would enhance the understanding of the behaviour of the reactor and provide us the maximum time frame for the initiation of remedial actions or alternate safety measures, to maintain core integrity.

Thermal-hydraulic simulation of several events without scram for HIFAR was performed using the best-estimate RELAP5/MOD3.2 thermal-hydraulic computer code [2], in collaboration with Department of Nuclear Engineering at Texas A&M University [3]. RELAP5/MOD3 is a best-estimate thermal-hydraulic computer program for simulating hypothetical transients and accidents in water reactor coolant systems. The code models the coupled behavior of the reactor coolant systems and the core for hypothetical loss of coolant accidents (LOCAs) and operational transients. The RELAP5/MOD3.2 equation set simulates a two-fluid system using a non-equilibrium non-homogeneous six-equation representation. RELAP5 computer program can be applied to a wide range of reactor designs and transient/accident conditions. Except for certain reactivity-initiated events, the code is applicable to LOCAs; loss of flow accidents (LOFAs); loss of heat removal events and anticipated transients without scram (ATWS).

The RELAP nodalisation of the HIFAR primary cooling circuit is presented in Fig. 1. The function of the shutdown pumps in the primary circuit was accounted for by imposing a constant mass flow rate of 28 kg/s in the primary circuit nodalisation. The primary coolant pumps were modelled as standard Westinghouse pumps and their capacities were specified as a function of the pump speed. The heat exchangers were nodalised into three axial volumes each and appropriate surface area factors were assigned to account for the multiple tubes. The reactor core was nodalised with an inlet plenum, the fuel elements and an outlet plenum. The central fuel element was divided into 12 axial nodes and 11 radial nodes, with five annular fluid regions and six heat structures. The four fuel rings and the inner and the outer clad cover form the six heat structures and the flow area between these rings form the five fluid regions. Being the hot channel, a power of 800 KW was assigned to it which is conservative. The remaining twenty-four fuel elements were lumped into two groups of twelve elements each. The lumped fuel elements were nodalised into three axial nodes and ten radial nodes, with three heat structures and two annular regions. The three heat structures are the inner and outer aluminum clad and a collapsed single fuel ring.

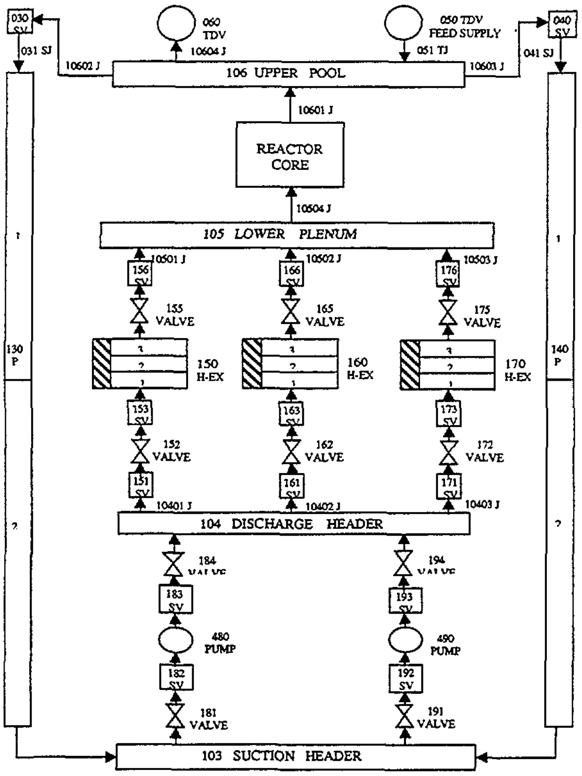


Figure 1 REALP5/MOD3.2 nodalisation of the reactor primary cooling system

Hypothetical events where the reactor protection system failed simultaneously along with decreased heat removal from the fuel elements were simulated. One such significant event would be the transients associated with the primary coolant circulation pumps. Hence, two events associated with the primary pumps were simulated. The first event simulated the non-availability of one of the two coolant pumps and the second event simulated the non-availability of one pump and 50% operation of the other pump. Another significant event would be transients associated with reduced secondary circuit flow in the heat exchangers. Under this category, three events were simulated. The first event simulated the removal capacity reduced by one-third, the second event simulated the heat removal capacity reduced by two-third, and the third event simulated circumstances where no heat was removed. It should be noted that these simulations neglected reactivity feedback effects. As result, predicted temperatures are conservatively high.

The summary of the results of the simulated transients is presented in Table I. The results of the simulated pump transients indicated that the mass flow rate would decrease to 270 Kg/s in the first transient (only one pump operating) and to 133 Kg/s in transient 2 (only one pump operating- at 50% capacity). With this assumptions, the maximum rise in average coolant temperatures in the primary circuit would be 5°C and the maximum rise in the central fuel element temperature would be 23°C. In the transients which simulated the reduced heat removal, the maximum rise in average coolant temperatures in the primary circuit would be 18.5°C and the maximum rise in the central fuel element temperature would be 6°C.

3. Safety analysis of HIFAR fuel element.

The prediction of the Onset of significant Net Vapour Generation (ONVG) during the subcooled flow boiling in reactor fuel elements is important, because accurate knowledge of void fraction distributions is vital for the prediction of two-phase friction and momentum pressure losses, flow stability limits, and reactivity effects in nuclear reactors. ONVG is of particular importance for low-pressure two-phase flows in channels of research reactor core. Since the pressure drop of a channel experiencing ONVG must remain the same as the no-boiling channels in a reactor core with common inlet and outlet headers, this will result in increased boiling, higher resistance and accelerational pressure drop, and hence a further flow reduction.

The final result is a reduction in flow -- a "flow excursion" -- to a new steady level at which vapour occupies most of the channel. This phenomenon is often referred to as a two-phase flow excursion instability, and it can lead to channel dryout and possibly burnout of a reactor fuel element.

TABLE I

Summary of the Results of the Simulated Transients

EVENT	PRIMARY COOLANT FLOWRATE (Kg/Sec)	PRIMARY LOOP AVERAGE TEMPERATURE (°K)	CENTRAL FUEL ELEMENT MAX TEMPERATURE (°K)	CHF
ONE PUMP TRIPPED	270	324.16	394	NO
ONE PUMP TRIPPED OTHER PUMP 50 %	133	328.8	404	NO
ONE HX NOT AVAILABLE	398.2	328.5	383.8	NO
TWO HX'S NOT AVAILABLE	398.2	341.1	392	NO

With the progress in two-phase flow modelling using an advanced computer code, CFX [4], it has become possible to investigate such important phenomena that may occur in HIFAR reactor fuel elements. A HIFAR fuel element consists of 4 concentric fuel and clad cylinders, 5 coolant channels, a central element and the outer shell. The fourth coolant channel of the HIFAR fuel element has been studied using the present model as it removes more heat than the other channels. This channel is 600 mm high with inner and outer annular diameters of 41.71 mm and 45.09 mm respectively. A fraction of 13.75% of the element power is removed from the inner annular surface while a fraction of 15.85% is removed from the outer surface. A cosine power distribution profile is applied along the channel height. The inlet temperature is 40 °C.

Figure 2 shows predicted void fractions at the channel exit and pressure drop between the channel inlet and outlet with the variation of fuel element powers under nominal operating flow conditions. On the basis of a common definition of ONVG – 5% void fraction at the channel exit; we found that for the ONVG, the fuel element power peaked at about 2300 kW. This was found to be very close to the ONVG fuel element power of 2500 kW predicted by Anne and Beattie [4] using their analytical method. Boiling causes an increase in frictional drag due to the presence of bubbles, and also influences pressure drop through acceleration and buoyancy effects. As shown in figure 2(b), the channel pressure drop would increase with the increase of reactor power if it were not fixed to be the same as in the other channels, (see Fig.2b). Instead, the increase in impedance in one channel would cause flow to be diverted to other channels of the fuel element, so initiating an excursive instability [6].

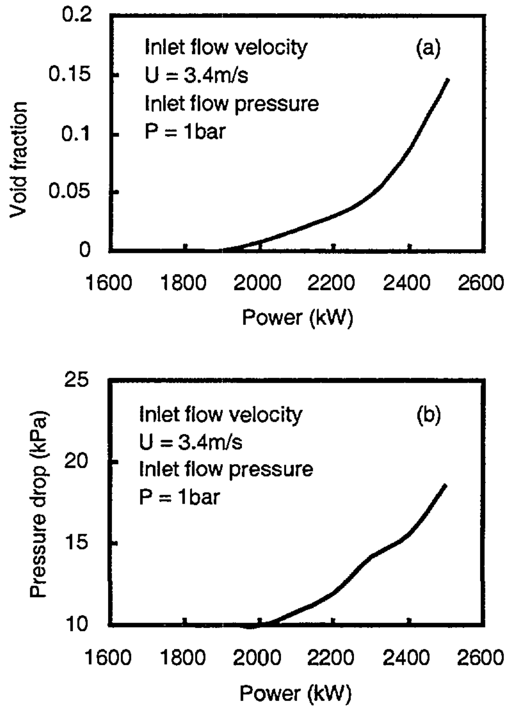
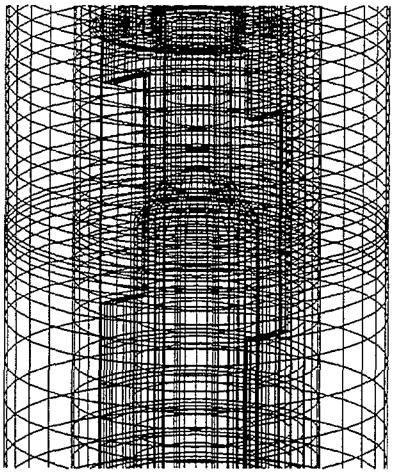


Figure 2 Predicted (a) void fraction at the channel exit and (b) pressure drop as a function of reactor powers

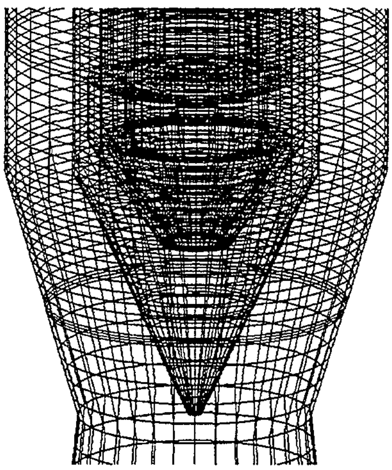
4. CFX modelling of HIFAR fuel rigs

A computational fluid dynamics (CFD) code, CFX, is used to simulate the thermal-hydraulics behaviour within a HIFAR fuel element where sealed target cans containing uranium oxide pellets are irradiated. Modelling of this process is 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.

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 4(b) respectively. Multi-block construction approach 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.

Figures 4 shows the velocity vectors around the liner and support tube nose cones plotted on a plane of x = 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. 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.

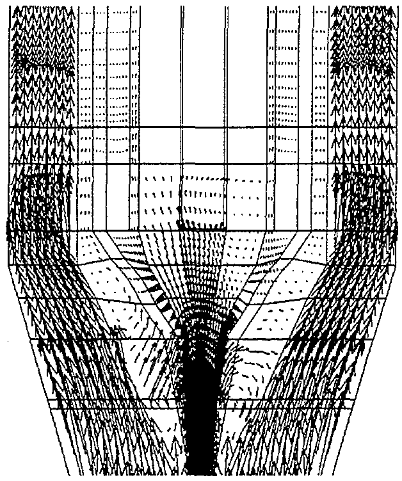


Figure 4. Flow structures of the heavy water on the outside of the liner and its entry into the liner and support tube nose cones. Here a length of 1 cm of the velocity vector corresponds to a velocity of 3.23 ms^{-1}.

5. HIZAPP and its application for HIFAR

At ANSTO, the ZAPP code [7] has been used to carry out computer simulation of power transients in HIFAR. The code has been extensively tested against the SPERT benchmark experiments [8, 9] and excellent agreement has been obtained. ZAPP solves the point reactor kinetics equation coupled with one-dimensional heat conduction equations that describe heat transfer from fuel tube to coolant. The original model included a method of modelling the increased rate of heat transfer with the onset of nucleate boiling.

The standard ZAPP code can be regarded as having been extensively and successfully benchmarked against SPERT data. The next step was to develop a HIFAR-specific version of the code called HIZAPP. This code uses traditional correlations to describe heat transfer from fuel to coolant. Such an approach has the advantage of employing heat transfer coefficients that recognise the influence of flow rate and thus more easily accommodate flow rate changes during the transients of interest. Another advantage is that such correlations are widely used in the nuclear industry and enjoy a wide level of acceptance.

In HIZAPP, heat transfer across interfaces other than the aluminium-coolant ones continues to be treated with the conduction model. However, at the aluminium-coolant interfaces, heat transfer correlations are used to calculate the heat flux at the surfaces from predictions of the solid wall temperatures and of the bulk coolant temperatures.

The correlations currently used cover non-boiling heat transfer at turbulent and laminar flows, and boiling heat transfer as a function of coolant flow-rate.

The HIZAPP code has been validated against steady-state predictions of temperature in a HIFAR fuel element, and for calculations of loss-of-flow transients in the FRJ-2 reactor. The code has been used to study a hypothetical beyond design basis accident the Loss of Flow Anticipated Transient Without Scram (ATWS) in HIFAR. A revised set of boiling heat transfer correlations were incorporated into the code for this study.

An example of the cases studied using HIZAPP is given in Figure 5 showing the evolution of clad and coolant temperature over the first 7 seconds of the transient. The highest rated fuel element power is taken as 640 kW which is typical for HIFAR full power operation. The transient begins with failure of the main circulators that run down to zero flow in a little more than 3.2 seconds. The protection system is assumed to fail so that neither the CCAs nor the safety rods drop into the core.

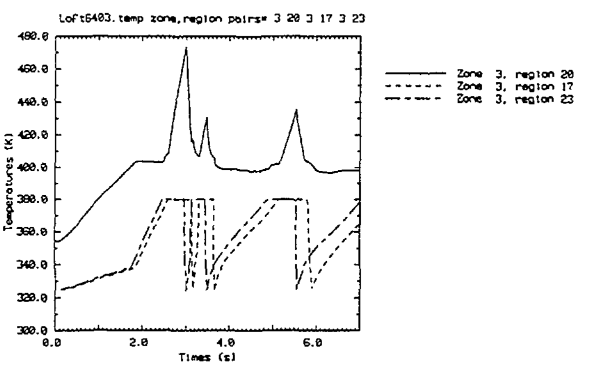


Figure 5. Temperatures of fuel tube #3 and surrounding coolant channels in a 640 kW fuel element in HIZAPP's HIFAR model.

For this transient, Fuel element temperatures remain stable for long time periods, and the maximum temperatures in the fuel tubes always occur in the first few seconds of the transient. It is the temperature and void coefficients of reactivity which provide protection during the transient.

After the initiation of the transient and as the main circulators run down the D_2O temperatures in the coolant channels increase causing the power to decrease due to the insertion of negative reactivity. The fuel and coolant temperatures continue to rise and after about 2 seconds nucleate boiling starts in the 640 kW elements. Flow is suspended in those coolant channels and bulk boiling begins at about 2.7 seconds. At this time heat transfer begins to decrease as the transition regime is crossed, and fuel temperatures in the adjacent tube show a rapid increase. This temperature rise peaks at about 3.0 seconds and is suddenly reversed when heat transfer rises rapidly as the coolant channel is re-flooded.

6. Analytic computations

ANSTO's code capabilities described above have been supported by complementary and supplementary analytic computations, typically performed with a PC spreadsheet. In addition, solutions obtained from analytic computations are on occasions preferred to those obtained from the application of larger computer codes.

Complementary analyses are those required to provide input data to, or to complete subsections of, larger codes. Analysis has also provided thermal boundary conditions for HEATRAN computational examination of cooling of radioisotope production cans, and of stored used fuel elements undergoing a hypothetical loss of coolant flow event.

Supplementary analytic calculations use simple theoretical equations or accepted correlations to confirm specific predictions from larger codes and so validate these codes. Such validation checks demonstrated that RELAP and CFX boiling models were valid at high pressure, but not at the low pressures of interest to ANSTO [10]. This stimulated the analytic boiling work described in the next section. Such supplementary calculations will be performed to validate code predictions for the replacement research reactor.

Analytic approaches have also provided alternative solutions to those from larger computer codes if the analytic approach is simpler and/or quicker and is also adequate for the purposes. Simplified analyses have provided adequate predictions of radioisotope rig powers and heat fluxes for various hypothesised circumstances (increasing the number of production cans, etc). RISO fuel elements proposed for use in HIFAR do not have the HIFAR emergency core cooling influx holes. An analytic examination provided and optimum means of adding such holes, and also provided information on the extent that the minor element design differences affected the emergency cooling of the different element fuel tubes [11]. A supplementary analysis also demonstrated that radiation and conduction would adequately remove heat from a fuel tube even in the unrealistic event that required emergency cooling did not reach one of the element fuel tubes [11]. Analytic solutions also provided an adequate examination of the capacity of HIFAR's top shield cooling circuit to remove decay heat in the event of a hypothetical loss of coolant flow event together with an absence of emergency core cooling [12].

7. R&D in modelling subcooled flow boiling

Modelling of a subcooled boiling flow is important because an accurate knowledge of the void fraction distribution in reactor cores is required to properly perform various safety analyses. Most available boiling models were developed for and tested at the high-pressure conditions of a power reactor. Many reactor safety analysis codes such as RELAP5 [1] and CATHARE [13], which use such models, cannot satisfactorily predict void fraction distributions in low pressure subcooled boiling flows [14]. This has limited the use of the RELAP5 code for low-pressure research reactor applications. The research and model development activities being conducted at ANSTO are primarily driven by the need for analysis of a low-pressure research reactor, HIFAR (HIgh Flux Australian Reactor) and its eventual replacement (RRR).

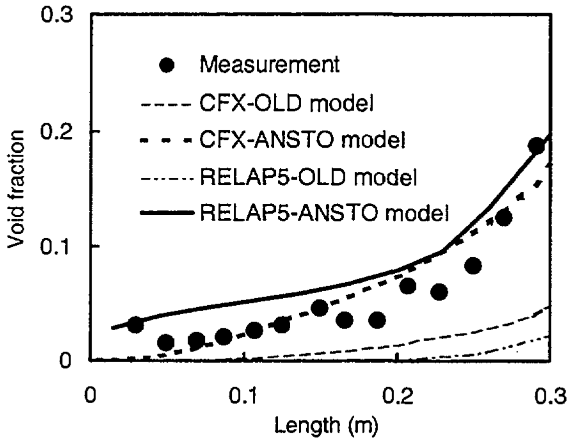


Figure 6 Comparison of void fraction between measurement and prediction using various models

Research has been undertaken to investigate the subcooled flow boiling mechanisms under low-pressure conditions. A general CFD code, CFX, is used to test new ideas and models through a better understanding of boiling mechanisms in relation to turbulent flow and heat transfer. By comparison of numerical prediction with experimental data, a new boiling model is validated to be capable of accurately predicting void fraction distribution in low-pressure subcooled boiling flow [15]. When this new model is introduced into the reactor safety analysis code, RELAP5, the code capability for low-pressure research reactor analysis has significantly been improved [16]. Figure 6 demonstrates a substantial improvement in prediction of void fraction distribution using the new model developed at ANSTO for a low pressure boiling experiment. The improved CFD code, CFX and reactor safety analysis code, RELAP5, are currently being used for HIFAR analysis and independent thermal-hydraulic calculations of replacement research reactor.

References:

[1] RELAP5/MOD3.2 CODE MANUAL, User's Guidelines, VOL. 5, REV. 1, NUREG/CR-5535, INEL-95/0174 (1995).

[2] J. W. CONNOLY and B. E. CLANCY, ANSTO Report ANSTO/E711 (1993).

[3] S. HARI, Y.A. HASSAN AND J. Y. TU, *Journal of Nuclear Technology*, Vol.130, pp. 296-309 (2000).

[4] CFX-4.2 Solver Manual, 1997. CFX, AEA Technology, Didcot, Oxon, UK.

[5] R. ANNE AND D. BEATTIE, in the *Proceedings of the sixth Australia Heat and Mass Transfer Conference*, pp. 503-509 (1996).

[6] J. Y. TU AND G.H YEOH, in the Proceedings of The 7th Australian Heat and Mass Transfer Conference, Townsville, July (2000).

[7] Clancy, B.E. - ZAPP - A Computer Program for Simulation of Reactor Power Transients. Report AAEC/E568 (1983).

[8] Clancy, B.E., Connolly, J.W. & Harrington, B.V. - An analysis of power transients observed in SPERT I reactors, Part I. Report AAEC/E345 (1975).

[9] Connolly, J.W. & Harrington, B.V. - An analysis of power transients observed in the SPERT II D2O moderated close packed core. Report AAEC/E418 (1977).

[10] R D Anne, D R H Beattie and J Y Tu, Thermal-Hydraulic Code Validation for HIFAR Fuel Element Safety Analysis, Proc. 13th Australasian Fluid Mechanics Conference (1998)

[11] Beattie, D R H, On the Incorporation of Wolters EEC Holes into Riso Fuel Elements Intended for HIFAR, ANSTO report NTD/TN 250 (2001).

[12] Beattie, D R H, Shield Cooling Removal of Decay Heat for Loss of Primary Coolant Flow, ANSTO report NTD/TN 228 (1999).

[13] CATHARE 2 code, CATHRE 5th International Seminar, CEA-DRN (1997).

[14] Tu, J.Y., The influence of bubble size on void fraction distribution in subcooled flow boiling at low pressure. *Int. Comm. Heat Mass Transfer*, **26**, pp.607-616 (1999).

[15] Tu, J. Y. and Yeoh, G. On Modelling of Low-Pressure Subcooled Flow Boiling, *International Journal of Heat and Mass Transfer*, accepted for publication (2001).

[16] Yeoh, G. and Tu, J. Y., Implementation of a Two-Phase Boiling Model into the RELAP5/MOD2 Computer Code for Predicting Void Distribution in Low-Pressure Subcooled Boiling Flows, *Journal of Nuclear Science and Engineering*, to be submitted for publication (2001).