Int. J. Heat Mass Transfer. Vol. 33, No. 3, pp. 435–445, 1990
Printed in Great Britain

0017-9310/90 \$3.00+0.00

Natural convection in a vertical cylinder: comparison of COMMIX-1A predictions with experiment

J. F. LAFORTUNE and D. A. MENELEY
Department of Chemical Engineering, University of New Brunswick, P.O. Box 4400, Fredericton, New Brunswick, Canada E3B 5A3

(*Received* 30 *August* 1988 *and in final form* 16 *May* 1989)

Abstract—A natural convection experiment is conducted in water to study the effect of multiple vertical heating rods and of the cooling surface elevation on the total heat transfer characteristics of the assembly inside a vertical cylinder. The three-dimensional computer code COMMIX-1A is used to predict the temperature profiles and overall heat transfer coefficients. The agreement between experimental data and the numerical predictions is shown to be excellent.

1. INTRODUCTION

1.1. *Background*

SINGLE-phase natural convection heat transfer has received attention in the nuclear industry with respect to the search for an inherently safe reactor [1, 2]. Natural circulation in water and in sodium, depending on the type of reactor considered can provide fuel cooling under normal and abnormal conditions.

A new small (2 MWTh) reactor concept which combines high electrical conversion efficiency with inherent safety is currently being studied [3]. It uses small cylindrical assemblies (10 cm i.d. × 200 cm) containing both fuel and coolant (sodium) and it relies on single-phase natural convection to transport the heat from the bottom half of the unit, where the fuel is located, to the top half, where heat is removed by an external coolant. A schematic representation of the unit is shown in Fig. 1.

The design of the basic unit requires good knowledge of the general heat transfer properties of this type of assembly. A calculational tool is needed to carry out a numerical investigation and to determine its main characteristics. Therefore, the present work was initiated in an effort to understand natural convection inside a vertical cylinder and to identify an adequate computer code for further design studies.

1.2. *Previous work*

Natural convective heat transfer in enclosed spaces has been considered in response to various energy problems. It is important in the study of energy losses through thermal insulation as well as in safety related research in the nuclear industry, such as the transport of spent fuel in specially designed canisters. It also finds a number of applications in the field of alternate energy resources. The number of problems encountered equals the multitude of specific geometries and types of enclosures in which natural convection has been studied. Consequently, very few specific cases have been examined in detail and most of the work has looked at some very general configurations. Ostrach offers an extensive review of the work carried out prior to 1982 [4].

Schwab and de Witt were among the first to study heat transfer by natural convection from a single vertical rod to an outer concentric cylinder [5]. Their numerical model was used to predict the effect of geometrical ratio and Prandtl number on the Nusselt number. Later work on heat transfer in an annulus was primarily concerned with the insulating properties of the gap and involved a porous medium inside the

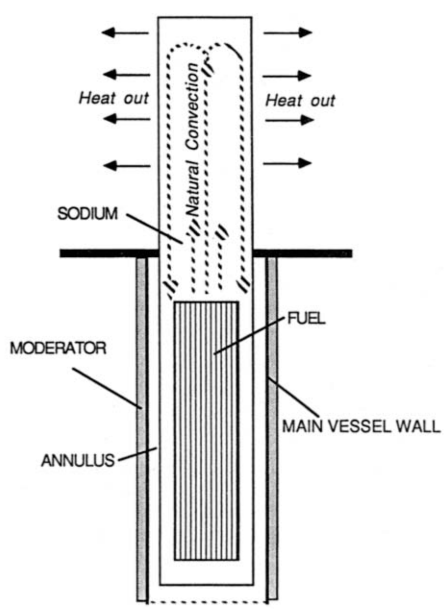

FIG. 1. The basic concept.

NOMENCLATURE

D	outer cylinder diameter
d	rod diameter
g	gravitational acceleration
h	heat transfer coefficient
H	L/l
K	D/d
k	thermal conductivity of the fluid
L	rod height
l	characteristic length, $(D-d)/2$
N	number of rows in a square rod array
Nu	Nusselt number, hD/k
P	pitch
Pr	Prandtl number, ν/α
q''	surface heat flux
Ra	Rayleigh number, $g\beta\Delta T l^3/\nu\alpha$
Ra_d	Rayleigh number with respect to d
Ra_D	Rayleigh number with respect to D
Re_x	Reynolds number, ux/ν
T_b	temperature of the bulk of the fluid
T_c	cool wall temperature
T_f	film temperature
T_h	hot wall temperature
T_w	wall temperature
ΔT	temperature difference, $T-T_c$
u	velocity component
x	distance along the thermal surface
Δx	radial mesh size
y	distance from the thermal surface.

Greek symbols

α	thermal diffusivity
β	thermal expansion coefficient of the fluid
δ	velocity boundary layer thickness
δ_T	thermal boundary layer thickness
δ^*	optimum mesh size near a thermal surface
ε_{DD}	experimental error due to the thermometer display
ε_{Os}	experimental error due to natural oscillations
ε_T	error on T
$\varepsilon_{\Delta T}$	error on ΔT
σ_{TC}	thermocouple standard deviation.

annulus [6]. Charmchi and Sparrow [7] looked at the problem of different cylinder heights and sizes. Their numerical simulation of a small inner cylinder located at various positions within a concentric cylinder of a greater height showed that the position of the inner cylinder affects the flow pattern, but does not significantly change the average Nusselt number. At high Rayleigh numbers, most of the action in the fluid was shown to take place near the boundaries of the system. Sparrow and Charmchi experimentally studied the effect of inner cylinder eccentricity [8]. Heat transfer correlations were provided by Keyhani *et al.* for the case of cylinders of equal heights in a concentric configuration [9]. In the boundary regime ($Ra \geqslant 10\,000$), the results were found to be well correlated by

$$Nu = 0.163\,Ra^{0.322}. \tag{1}$$

Keyhani *et al.* extended their work to consider the case of several heater rods within one cylinder [10]. They used two square arrays (3×3 and 5×5) of vertical rods with different pitches and used both air and water as the working fluids. Correlations were found for individual rods and for different configurations. All the results obtained in water were reasonably well described by one correlation

$$Nu_d = 0.162\,Ra_d^{0.322}. \tag{2}$$

A generalized correlation was also obtained to account for the various geometrical parameters. In the boundary layer regime, the following expression was found to be in good agreement with the experimental data:

$$Nu = 0.188K^{0.442}H^{-0.238}(P/d)^{0.045N+0.541}Ra^{0.322}. \tag{3}$$

This correlation shows that Nu increases with P or N. Since P and N are often related for a given fuel bundle size and coolant volume ratio in a nuclear core, with P varying inversely with N, this result has important consequences on the actual fuel bundle and heat removal design.

Extrapolation of the results obtained thus far in air and in water to liquid metals in general and to liquid sodium specifically is difficult due to the difference in Prandtl numbers, which are very low for liquid metals. Most of the work carried out on natural convection in liquid metals involves either simple geometries or very specific ones. Wiles and Welty [11] looked at vertical cylinders immersed in a large pool of mercury and concentrated on the laminar regime. Sheriff and Davies [12] considered the case of a vertical plate in a sodium pool. Colwell and Welty [13] looked at two parallel vertical plates forming a channel and found that the heat transfer was enhanced as the gap is reduced, indicating that multiple heat sources in close proximity may enhance heat transfer. However, there seems to be an optimum distance between heat sources beyond which heat transfer is actually impaired. A correlation found by Viskanta *et al.* [14] for rectangular enclosures bears some resemblance to that obtained by Keyhani *et al.* [9] with the exception of an additional Prandtl number dependency

$$Nu = 0.16Ra^{0.31}Pr^{0.14}. \tag{4}$$

This would indicate that a correction term in Pr can be applied to previously prescribed correlations to enable extrapolations to liquid metals.

In general, for equal Ra, Nu increases with Pr. This was confirmed by Kubair and Simha [15] for mercury in a vertical annulus. The effect is most marked for $Pr > 1$, as demonstrated by MacGregor and Emery [16].

The limited data on multiple-heat sources suggests that more work is needed to accurately estimate this effect. No data are available on the effect of cooling coil elevation. Furthermore, the correlations suggested are limited in scope. The information which can be deduced from previous work is insufficient for design of the fuel–sodium unit.

1.3. *Objectives*

The main objective of the work presented was to identify a design tool for the thermal–hydraulic study of the small reactor proposed. To achieve this goal, this part of the research aimed specifically at:

(i) obtaining qualitative information on the effects of multiple heaters and elevated cooling coil;

(ii) assessing the capability of a numerical program to predict heat transfer by natural convection in a vertical cylinder.

Experiments were conducted in water with up to seven heaters. The data obtained are compared with predictions by the three-dimensional computer code COMMIX-1A [17]. The results are then examined to evaluate the potential use of the COMMIX-1A code for the design phase.

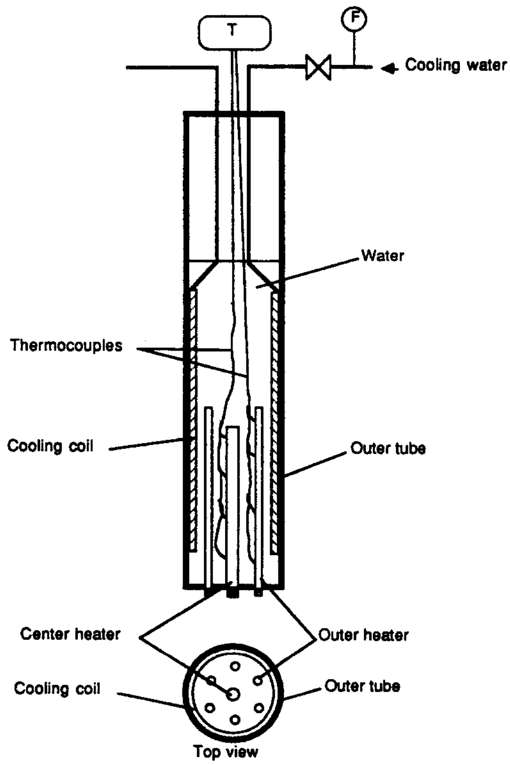

FIG. 2. Experimental setup.

2. EXPERIMENTS

The experiment was formulated to provide data applicable to one module of the proposed small reactor concept. The basic geometry is therefore that of a right circular cylinder representing the subunit's wall with internal heating elements to model part of the nuclear heat sources.

2.1. *Experimental setup*

The experimental apparatus consists of a glass tube, cooling coil, heater rods and related instrumentation. The layout is shown in Fig. 2.

The vertical tube is a 0.146 m i.d. × 1.22 m high open cylinder made of 19 mm thick glass. It is mounted on a stand and kept off the ground to allow the bottom insertion of the heaters. The bottom of the tube is sealed with a 12 mm thick stainless steel plate in which eight holes are bored for the insertion of seven threaded stainless steel cartridge heaters and one drain pipe.

The heat sink (or cooling wall) consists of 9.5 mm o.d. copper tubing coiled tightly to form a 335 mm high assembly which fits snugly against the glass tube's inner wall. Heat removal is provided by cooling water circulating inside the tube. During the experiments, the flow of cooling water is kept high enough to prevent a temperature drop of more than 2°C along the whole length, thereby actually ensuring that the cooling wall is maintained at an approximately constant temperature. The coil can be moved vertically within the glass tube.

Seven heaters are used in the experiments. Initially only the center rod was inserted in the tube; the other holes were plugged. Two types of rods are used. The center heater is a 2 kW cartridge of 25.4 mm o.d. × 185 mm. The six outer heaters are 500 W cartridges of 12.7 mm o.d. × 232 mm length with uniform heat flux.

Four thermocouples are soldered on the side of the center rod. Four more are affixed on the side of one of the outer heaters. Only one outer rod is instrumented based on the assumption that the temperatures on the other outer heaters are the same. Two more thermocouples are located at the inlet and outlet of the cooling coil and were used for the initial calibration of the system to demonstrate that no significant amount of heat loss occurs. The thermocouples are all type T and of a thickness of 26 ga. The temperatures were read with a thermally compensated digital thermometer with a display margin of ±0.2°C.

Power is supplied by a 240 V–8 A Variac. The electrical circuit enables power to be supplied evenly to any combination of one or more heaters. The power is read with a wattmeter, accurate to within 5 W.

2.2. *Experimental procedure*

Steady-state measurements were taken for the following cases:

(i) single rod configuration, up to 500 W;
(ii) seven-rod assembly, with power to the center rod only, up to 500 W;
(iii) seven-rod assembly, with power to the outer rods only, up to 1200 W total;
(iv) seven-rod assembly, with power to all seven rods, up to 1200 W total;
(v) same as (iv) with the cooling coil elevated to 20 cm.

In the first four cases, the cooling coil rests on the bottom of the tube. The tube is filled with water up to the top of the coil. The remainder of the tube contains ambient air and is sealed at the top by a plexiglass plate. The first experiments were conducted at very low flow rates in the coil. Measurements were made to ensure that all of the heat was actually transmitted to the cooling wall. Further experiments are conducted with high flow rates.

The feed water to the cooling coil was tap water with an average temperature of 10–11°C fluctuating daily. At the beginning of each experiment, time is required for the temperature inside the system to stabilize. Temperature readings are taken when temperatures reach an equilibrium. This generally requires 5–10 min, depending on the power level and on the number of heaters. Random verifications over a period exceeding 12 h proved that no significant changes occur after the initial 5–10 min delay.

2.3. *Data processing and experimental error*

The pertinent measurements made are the temperature profiles and the power output. All temperature data were referenced to the cool wall average temperature, T_c. The average temperature difference, ΔT, was used in the determination of the overall heat transfer coefficient. When results are presented in a dimensionless format, relating the Nusselt number to the Rayleigh number, fluid properties are computed at a reference temperature which is the average of the two surface temperatures [18]. This is different from the film temperature approach or the modified film temperature approach (19); in an enclosed space, it becomes necessary to account for the coupling between the hot and cool streams.

The error on the measurements is calculated from a combination of digital thermometer sensitivity (ε_{DD}), thermocouple accuracy (σ_{TC}) and, in some cases, natural oscillations of the temperature readings (ε_{Os}). The total error on each temperature measurement, ε_T, can be expressed as

$$\varepsilon_T = \varepsilon_{DD} + \varepsilon_{Os} + 3\sigma_{TC}$$

where $\varepsilon_{DD} = 0.2°C$, $\sigma_{TC} \cong 0.2°C$ and $\varepsilon_{Os} = 0$ most of the time. The temperature difference can then be calculated within a 95% confidence interval as

$$\Delta T = (T_h - T_c) \pm \varepsilon_{\Delta T}$$

where $\varepsilon_{\Delta T} = (2\varepsilon_T)^{1/2} = 1.1°C$.

Other sources of error such as conduction on the rod and thermocouple wire are expected to be small compared to ε_T. Flow perturbation due to the presence of the wires in the fluid may be more important but was not quantified.

3. COMMIX-1A

COMMIX-1A is a general purpose porous-medium three-dimensional thermal-hydraulics computer code designed for transient and steady-state analyses of simple and complex systems [17, 20]. The current version of COMMIX-1A allows the treatment of single- or multi-component systems in the presence of up to three fluids, including pumps and thermal structures. Advanced iterative schemes are included in the code. The development of COMMIX-1A started in 1976; its original objective was to analyze component mixing problems in single-phase flows. More recently, the code has been expanded to study the entire flow fields in a pool-type LMR. The version of COMMIX utilized in this work is a code the improvement and verification of which is a continuing process; it is currently being used for advanced design studies in the U.S. [21–23].

A detailed list of the code's original features can be obtained from ref. [17]. The following list of options is restricted to those aspects of COMMIX-1A which have a direct impact on the present simulation. COMMIX-1A treats each case as a transient problem which converges if a steady-state solution is sought. The user can select built-in property packages for either water or sodium. Solid structures may be placed within the fluid; these 'thermal structures' are connected to the fluid at their surface. They may conduct heat in one dimension and may contain heat sources. A porous body model combined with a directional surface permeability capability allows the code to analyze complex configurations. The code includes a one-equation turbulence model. Semi- and fully-implicit schemes can be used to solve the momentum and energy equations. In lieu of rigorous free-surface treatment, COMMIX-1A requires the user to include a cell (called the expansion cell) containing one surface through which fluid can escape or enter at a rate corresponding to the bulk density variation in the system. This modelling mechanism has no impact upon steady-state solutions.

4. THE CALCULATIONAL MODELS

4.1. *General description*

Because of the different heights of heaters and cooling coil, the problem at hand must be treated as a full three-dimensional case. Only in the case of the single heater can a two-dimensional representation suffice to fully describe the geometry. In the other cases, the angular symmetry allows the problem to be adequately described by a 1/6 slice in the azimuthal direction.

A 1/6 angular model is used. A tight mesh is used near the thermal boundaries in an attempt to model the increased action in those regions.

The main cylinder containing the heaters is treated as a cylindrical boundary with a no-slip flow condition; the cooling coil is treated as a constant temperature boundary while the rest is assumed to be adiabatic. The inner rods can be described either as thermal structures immersed in the fluid or as boundaries. The first option involves the specification of volumetric heat sources and initialization of the heater region with adequate porosity and permeabilities. Because the heating elements are actually solid, the second option is chosen; the heaters are described as a surface boundary with a specified heat flux. This option does not provide the thermal history within the heater. Although this may be important during a transient, it is irrelevant in the present steady-state analysis.

The heat transfer from the adjacent cells to the cooling coil is calculated from conduction theory by assuming a linear temperature profile between the cell center and the wall. Similarly, the temperatures of the hot walls are extrapolated linearly from the adjacent cell centers to match the defined heat transfer obtained by pure conduction. The next section will deal further with the question of heat transfer mechanisms, which is closely related to the source and sink models chosen.

Although COMMIX-1A can account for turbulence and friction, natural convection velocities observed in the experiment are small and did not appear to require these options to be used in the present case.

4.2. *Heat transfer mechanism*

The knowledge of the hot wall temperature (T_h) is not essential to the solution of the energy equation but it is required in order to compare experimental and numerical results. The cold wall temperature (T_c) is defined by the temperature boundary condition of the cooling coil. The calculated temperature of the fluid depends on the heat transfer coefficient between the cool wall and the fluid. Therefore, heat transfer coefficients are important to the solution of this problem.

Convective heat transfer correlations can be obtained from the literature for fully developed flows of various types in conduits or in simple geometries. However, it is difficult to relate the present geometry to any prescribed case since the flow is not necessarily fully developed. To understand the phenomenon and the importance of the model, it is necessary to adopt a more fundamental approach to the problem of heat transfer from a solid structure to a fluid.

Heat transfer from a solid surface to a moving fluid is closely related to the regime of fluid flow. For the simple case of flow parallel to a surface, various forms of the heat transfer solutions have been derived. In the limiting case of fluid flow along a flat plate with a leading edge, the velocity of the fluid varies from 0, at the surface, to u_b, far enough from the plate. The resulting boundary layer increases in thickness away from the leading edge in the direction of the flow. A similar boundary layer exists for the fluid temperature. For laminar flow, the thermal boundary layer thickness is related to the velocity boundary layer thickness and to the thermal fluid properties by expression (18)

$$\delta_T = \frac{\delta\, Pr^{-1/3}}{1.026}$$

where

$$\delta = \frac{5x}{Re^{1/2}}.$$

The heat transfer coefficient, h, defined as

$$h = -\frac{k\,(\mathrm{d}T/\mathrm{d}y)}{T_w - T_b} \quad \text{at } y = 0$$

with the properties evaluated at the film temperature

$$T_f = (T_w + T_b)/2$$

or at the modified film temperature

$$T_f = 0.3T_w + 0.7T_b$$

can be expressed as a function of the thermal boundary layer thickness as

$$h_x = \frac{1.5k}{\delta_T} \quad \text{and} \quad h_{av} = 2h_L.$$

For turbulent flow the solution is similar, leading to a reduced layer thickness because of the enhanced mass and heat transfer. For $Re \geqslant 500\,000$

$$h_{av} = \frac{k}{L} Pr^{1/3}\,(0.036\, Re_L^{0.8} - 836).$$

These correlations provide a simple relationship between the wall temperature and that of the bulk of the fluid, thereby reducing mesh requirements near the surface, where a steep temperature gradient exists. However, in the absence of a well defined flow along the surfaces, a more basic conduction calculation is needed.

Without a given correlation, COMMIX-1A estimates the surface temperature by assuming a constant temperature gradient from the wall to the center of the boundary cell for which an average temperature has been calculated. The actual heat transfer is thus proportional to the slope of the temperature profile, a slope which therefore remains constant for a given rate of heat input. The heat transfer is related to the average cell temperature by

$$q'' = k\frac{T_w - T_1}{\Delta x/2} = \text{constant}.$$

When choosing a boundary cell size which is greater than the actual boundary layer, COMMIX-1A may overpredict T_w. Therefore, it is important to verify the effect of the size of the cell immediately adjacent to

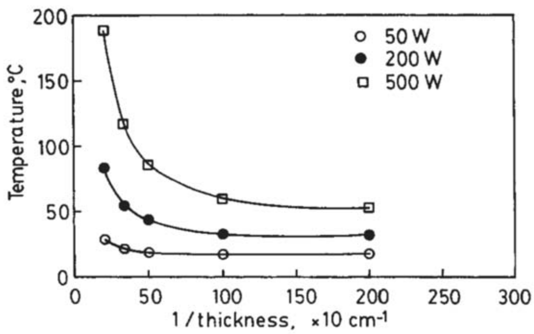

FIG. 3. Effect of cell size near the walls.

the wall on the wall temperature. The problem is the same on the cool wall except that, when the wall temperature is chosen as constant, the temperature of the fluid is the one affected.

5. RESULTS AND DISCUSSION

This section contains the results obtained. Some important preliminary computer results are initially presented. In almost all of the computer runs, numerical convergence is not achieved but energy convergence is. For example, after typically 500 iterations, 0.1% oscillations on the velocity components are still observed. However, resulting fluctuations in the macroscopic energy conservation and in the surface temperatures are of an order less than 0.01%. This is considered sufficient for the following comparison with the experimental data. Thus, a practical convergence is obtained for relevant quantities.

5.1. *Effect of boundary cell thickness*

As previously discussed, the size of the boundary cells is expected to be crucial to the determination of surface temperatures. Variations should not significantly affect the overall solution of the energy and momentum equations.

The thickness of the boundary cells is varied between 0.05 and 0.5 cm for the single heater configuration. Figure 3 shows the results for 50–500 W of power output on the average temperature of the heater. As the thickness of the cell is reduced, so is the surface temperature. The latter does not change significantly for values of the thickness less than approximately 0.1 cm. It can also be observed that the threshold thickness for asymptotic behavior decreases as the power increases.

The reason for this behavior is explained in the previous section. As the cell thickness approaches the actual boundary layer thickness, the deviation from linearity decreases. Beyond the value corresponding to the actual thickness, the linear approximation is good and nothing is gained by further reducing the cell size. Since the boundary layer thickness decreases with increasing power (and increasing flows), the threshold size δ^* should also decrease.

Table 1. Effect of mesh on the hot wall temperature (all seven heaters are on; after 200 iterations)

		T_w (°C) Standard mesh	2 ×
Central	900 W	46.2	46.9
	1200 W	53.3	54.6
Outer	900 W	33.4	34.1
	1200 W	37.2	38.9

It is interesting to compare the value of δ^* with the estimated boundary layer thickness obtained from laminar boundary layer theory. According to the proposed analysis, δ_T and δ^* should be of the same magnitude. Based on the results from COMMIX-1A for the 100 W case ($\delta^* \sim 4$ mm)

$$u = 0.03\ \mathrm{m\,s^{-1}}$$

$$T = 15\ ^\circ\mathrm{C}$$

$$Re < 10^4$$

thus $\delta_T \sim 7$ mm. The fact that δ^* is lower than δ_T shows that the linear approximation is only valid within the boundary layer region. Based on these results, a mesh size of 1 mm was used near the walls for all calculations.

5.2. *Effect of mesh density*

The effect of node density is examined by doubling the mesh while keeping the total number of nodes constant. This is done by reducing the angular boundaries from a 1:6 to a 1:12 symmetry. The results are presented in Table 1. As can be seen, doubling the mesh density leads to no significant changes of the surface temperatures for some of the highest power levels studied. The error increases with the power but does not exceed 6% below 1200 W. It should be noted, however, that the mesh density was uniformly increased and no special consideration was given to selectively refining the mesh in regions of increased activity, where it counts most. These are the regions close to the heaters. At high power, when the flow rates are maximum, extra nodes may be required. However, the difference between the results appears low enough so that the 1:6 geometry is deemed to have a sufficiently fine grid for all calculations. The consequence of this conclusion will be seen later.

5.3. *Qualitative description of the results*

The only available experimental means to describe the nature of the flow was visual observation. Other more sophisticated methods were not used, partly because of the difficulty to measure flow fields inside a cylindrical enclosure. However, qualitative observations as well as results from previous work can be used to compare the COMMIX-1A results with the experimental data.

The dominating flow shows fluid moving up along the hot surfaces and down the cool ones. In most

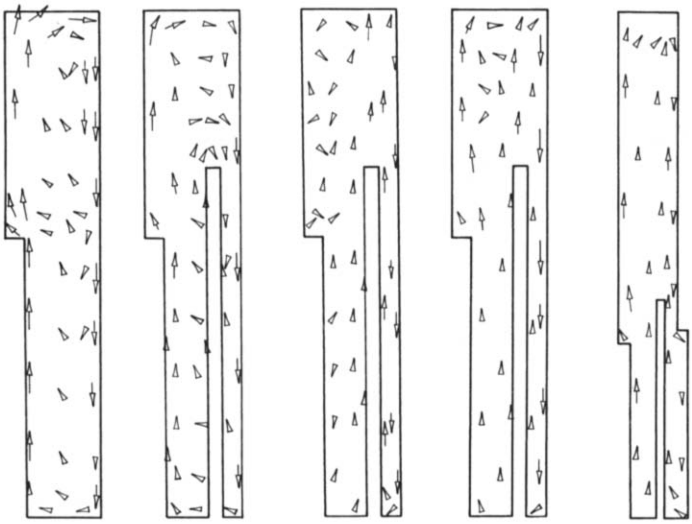

FIG. 4. Flow fields for (a) 50 W, single heater case; (b) 50 W, center heater only; (c) 50 W, outer heaters only; (d) 100 W, all heaters on; (e) 100 W, all heaters on, elevated coil.

cases, the action is limited to thin layers along the boundaries. As the power is increased, cross flow in the region between the heaters and the cool wall increases. At powers higher than 500 W for the single heater as well as for the seven heater cases, large fluctuations in the temperature readings appear (1–2°C). The flow immediately above the rods is visibly very unstable. The visible thermal layer adjacent to the heaters can also be seen to fluctuate significantly at higher powers. In the seven-heater configuration, there appears to be no clear interaction between the rods. The apparent flow is a combination of individual flow patterns around each heater.

Some of the flow and temperature fields from COMMIX-1A are shown in Figs. 4 and 5, respectively. Only the right half of the vertical cylinder is shown. The natural circulation pattern is clearly discernible in the single heater cases; it is apparent that it varies little with power. In Fig. 4, the flow field is seen to be limited to very thin regions along the surfaces. These results are consistent with the experimental observations.

Figure 4(b) shows the velocity field for a total output of 50 W in the seven-heater assembly with only the central heater turned on. The pattern is almost identical to that found for the previous case (Fig. 4(a)). The only difference (not shown in Fig. 4) lies in the introduction of a small radial flow around the outer heaters. This is not enough to significantly change the nature of the flow which remains practically unaffected by the presence of the extra structures in the enclosure. As in the single heater case,

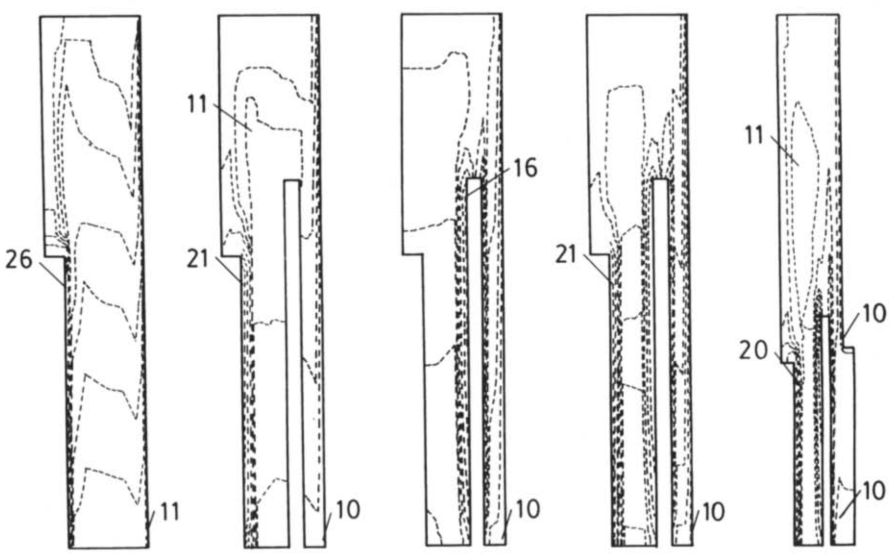

FIG. 5. Temperature fields: same cases as in Fig. 4.

there is no visible difference as power is increased except in the magnitude of the velocities.

The situation is much the same in the seven-heater case with only the outer heaters producing power. However, because in this case the heaters are closer to the cooling wall, most of the circulation occurs outward from the rod. This can be seen in Fig. 4(c). As the fluid rises toward the top of the rod, it splits in two directions, inward and outward. The outward fluid which is cooled faster also moves faster. In the central region, the fluid appears to be cooled by mixing with the larger, more stagnant pool. The central heater creates a minor restriction in the path of the fluid but does not seem to affect the flow pattern significantly. No significant changes are observed at higher power.

The flow becomes more complicated when all seven heaters are generating power. In this case, most of the fluid still follows the general natural circulation pattern with a divergence at the bottom of the enclosure and more mixing at the top. This is shown in Fig. 4(d). It appears that the flow field is nothing more than a superposition of the two preceding cases. If this were so, the radial distance between the rods would be sufficient to prevent boundary layer interaction, which would indicate that the heat transfer characteristics of the multiple heater assembly could be estimated from the characteristics of the individual heaters.

The effect of cooling wall elevation is shown in Fig. 4(e). In order to obtain an adequate mesh for this case, the angular symmetry was increased to 1 : 12. The most noticeable difference with the previous case seems to be the increased flow in the region between the central and outer heaters which would be due to the higher fluid column and subsequent greater total pressure differential between the top and bottom of the enclosure.

Some isotherm maps for these various cases are shown in Fig. 5. They can be directly related to the flow field characteristics which have just been discussed. In all cases the temperature profiles exhibit signs of good convection near the active heaters with a stratified zone in the stagnant regions, shown by horizontal isotherms. When all seven heaters are on, the stratified zone size is reduced, indicating an increase in the thermally affected region size. This means that more fluid is in movement as a direct consequence. The effect of the elevated cooling wall, as seen in Fig. 5(e), seems to be a significant reduction of the stratification which occurs when the coil is down. This would indicate that the difference in elevation promotes the circulation of the fluid.

5.4. *Quantitative results*

5.4.1. *Experimental data.* Experimental trends and comparison with previous work are shown in Figs. 6 and 7. In Fig. 8, the results obtained with COMMIX-1A are compared with the experimental data.

In Fig. 6, the average data for the outer rods and for the center rod in the seven-rod assembly are presented in a non-dimensional format and compared with the correlation obtained by Keyhani *et al.* [9]. The empirical correlations obtained from a linear regression analysis of the experimental data are given in Table 2. The uncertainty on the coefficients given is typically less than 6%. The present data agrees reasonably well with the power law obtained by Keyhani *et al.* The heat transfer coefficients for the center rod when it is the only active one are significantly higher than the rest. For the remaining cases, Keyhani *et al.*'s correlation is matched closely by the current results. All results obtained for the outer rods are significantly lower than the reference correlation, indicating that the heat transfer coefficient is lower for these rods. In water, Keyhani *et al.*'s results show little variation of the heat transfer characteristics with rod location. Similarly, the results of Sparrow and Charmchi [8] indicate that the average total heat transfer coefficient does not vary with eccentricity. It must be noted that, in the present case, the outer heaters are longer, smaller in diameter and closer to the cool wall than the center one. The rod curvature should be expected to have an effect on the heat transfer characteristics of the rod, although the importance of this effect decreases with the aspect ratio and, for long rods, is not expected to be significant [11]. Generally, the results show a trend which is consistent with equation (3).

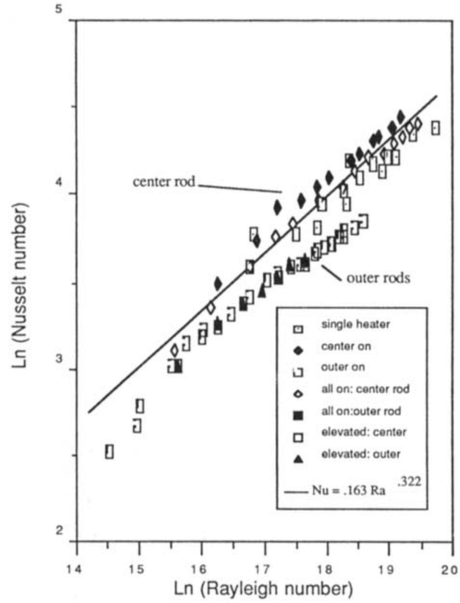

FIG. 6. Nusselt number correlations and comparison with equation (1) in the seven-heater configuration.

From Fig. 6, it is also possible to observe the effect of multiple heat rods and coil elevation on the overall temperature difference. Comparison of the single-heater data with the seven-heater-center-only data shows an increase in the heat transfer coefficient for the latter case. When the outer heaters are also active, the heat transfer coefficient is reduced. It is reduced

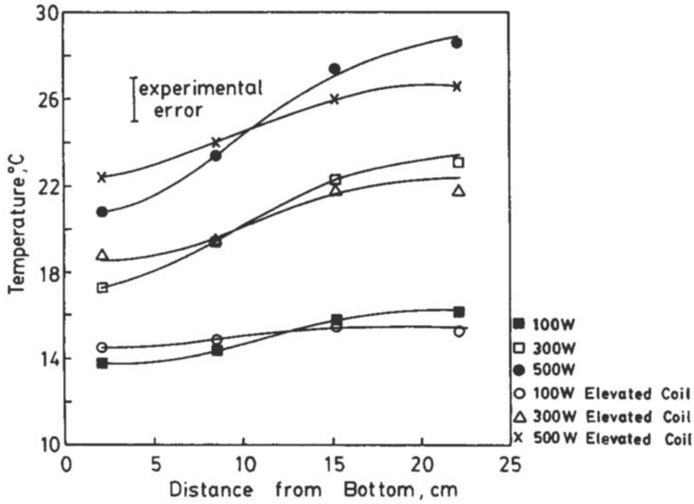

FIG. 7. Effect of cooling coil elevation on the temperature profiles on the outer heaters.

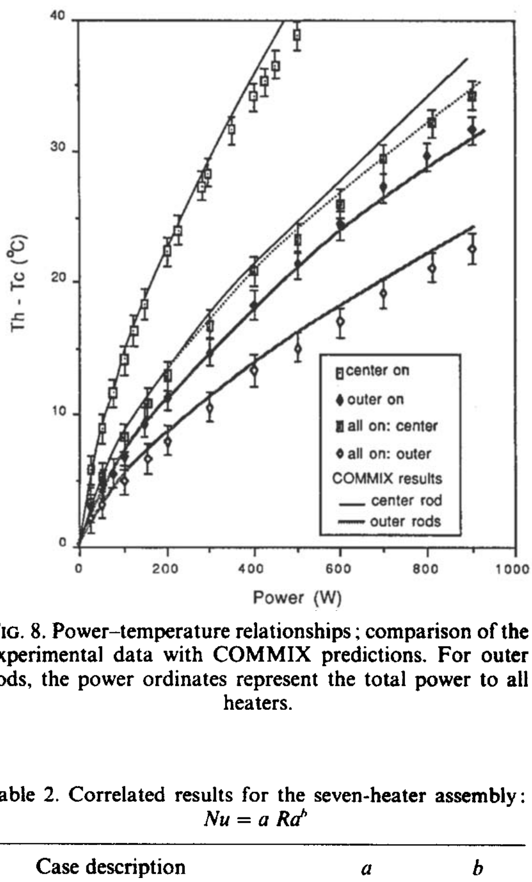

FIG. 8. Power–temperature relationships; comparison of the experimental data with COMMIX predictions. For outer rods, the power ordinates represent the total power to all heaters.

Table 2. Correlated results for the seven-heater assembly: $Nu = a\,Ra^b$

Case description	a	b
Keyhani *et al.* [9]	0.163	0.322
Center heater:		
Center heater on	0.225	0.309
All heaters on	0.159	0.322
Elevated coil	0.223	0.300
Outer heater:		
Outer heaters on	0.227	0.289
All heaters on	0.141	0.317
Elevated coil	0.183	0.304

even further as the cooling coil is elevated. The differences are less obvious for the outer heaters for which the heat transfer coefficient seems to vary little with the configuration.

Figure 7 shows the temperature profiles on the outer rods when all seven heaters are active. It can be seen that the main effect of coil elevation is to flatten the profile along the length of the rod. The average temperature remains practically unchanged. This observation is due to two factors. On one hand, as it was mentioned in the previous section, coil elevation promotes the natural circulation flow and increases the active volume of water which circulates. On the other hand, the water is now cooled in the upper region of the tube and is partially heated by mixing and heat transfer from the rods while flowing downward. This latter factor explains the higher temperature at the bottom of the rod. The former agreement explains why the average temperature gradient along the rod is smaller when the coil is elevated.

5.4.2. *Comparison with COMMIX-1A.* Temperature–power relationships for four cases are presented in Fig. 8. Excellent agreement is shown for the single and for the seven-heater cases when only the center rod or only the outer rods are active. In the case where all heaters are turned on, COMMIX-1A predictions are excellent at low power but diverge slightly at high power. The results obtained with the elevated coil vary little from the latter. However, the deviation is less than 10% below 1000 W. A possible explanation for this behavior, which is only observed when all seven heaters are on, is that the node density used is not adequate to describe the increased action which results from the higher heat production density in the enclosure. To verify this, the radial mesh number was increased to allow for a higher node density immediately next to the central heater. The penalty on the solution time is considerable and to increase the node density everywhere in the enclosure would not justify the gain in accuracy.

These results are shown as the dashed line in Fig. 8. The improved agreement indicates that the code can adequately predict even complicated problems in a tight geometry, as long as the mesh is sufficiently refined.

6. CONCLUSIONS

The study of natural convection trends in water in a vertical cylindrical assembly and the simulation using the three-dimensional thermal–hydraulic computer code COMMIX-1A were carried out as part of a design process to examine the possibility of using sodium in a similar assembly as the heart of a small nuclear reactor concept. The first step of this process was to examine the nature of the natural convection phenomenon and to evaluate the capability of the code to deal with such geometries.

Results obtained experimentally indicate that:

(1) in a one-heater arrangement, natural convection flow patterns are established and remain relatively unperturbed for the range of power studied;

(2) in a seven-heater assembly, the overall flow pattern can be estimated from a superposition of the individual flow circulation schemes;

(3) adding heating rods in the enclosure results in a decrease of the heat transfer coefficient from the single heater case;

(4) the heat transfer coefficient is lower for outer rods;

(5) the main effect related to cooling wall elevation is a flattening of the temperature profile on individual rods, leaving the average temperature approximately unchanged for a given rate of heat exchange;

(6) cooling wall elevation has a greater effect on the centrally located heater than on the outer rods.

The results obtained in this study are generally in good agreement with the correlations previously suggested by other workers. For the range of heat fluxes and the number of heaters which were examined, the heat transfer characteristics are found to be reasonably well predicted by single heater correlations.

Simulated results from COMMIX-1A are shown to be in excellent agreement with experimental results for most cases. At high power levels and higher power density, a slight divergence occurred ($\sim 10\%$) which can be remedied by increasing the node density in regions of complex flow. COMMIX-1A has shown the ability to accurately predict natural convection parameters in a small enclosure with simple geometry but complex heat transfer and flow mechanisms.

Acknowledgements—The numerical portion of this work was performed at Argonne National Laboratory, Illinois. The invaluable help and cooperation of Don Malloy and Pat Gardner, from Applied Physics Division, are gratefully acknowledged.

REFERENCES

1. J. W. Hilborn and G. A. Burbidge, Slowpoke: the first decade and beyond, AECL 8252 (1983).
2. *Proc. Second Specialists' Meeting on Decay Heat Removal and Natural Convection in LMFBRs*, New York (1985).
3. J. F. Lafortune, Conceptual design of a small nuclear reactor for electricity generation, Ph.D. Thesis, University of New Brunswick, Fredericton (August 1988).
4. Ostrach, Natural convection in enclosures. In *Advances in Heat Transfer*, Vol. 8, pp. 161–223, Academic Press, New York (1982).
5. T. H. Schwab and K. J. de Witt, Numerical investigation of free convection between two coaxial cylinders, *A.I.Ch.E. Jl* **16**(6), 1005–1010 (1970).
6. M. A. Havstad and P. J. Burns, Convective heat transfer in vertical cylindrical annuli filled with porous medium, *Int. J. Heat Mass Transfer* **26**, 1755–1765 (1982).
7. M. Charmchi and E. M. Sparrow, Analysis of natural convection in the space between concentric cylinders of different height and diameter, *Numer. Heat Transfer* **5**, 119–144 (1982).
8. E. M. Sparrow and M. Charmchi, Natural convection experiments in an enclosure between eccentric or concentric vertical cylinders of different height and diameter, *Int. J. Heat Mass Transfer* **26**, 133–142 (1983).
9. M. Keyhani, F. A. Kulacki and R. N. Christensen, Free convection in a vertical annulus with constant heat flux on the inner wall, *Trans. ASME* **105**, 454–459 (1983).
10. M. Keyhani, F. A. Kulacki and R. N. Christensen, Experimental investigation of free convection in a vertical rod bundle—a general correlation for Nusselt numbers, *J. Heat Transfer* **107**, 611–623 (1985).
11. L. E. Wiles and J. R. Welty, An experimental investigation of natural convection with vertical cylinders in mercury, *ASME J. Heat Transfer* **96**, 455–458 (November 1974).
12. N. Sheriff and N. W. Davies, Liquid metal natural convection from plane surfaces: a review including recent sodium measurements, *Int. J. Heat Fluid Flow* **1**(4), 149–154 (1979).
13. R. G. Colwell and J. R. Welty, An experimental investigation of natural convection with a low Prandtl number fluid in a vertical channel with uniform wall heat flux, *ASME J. Heat Transfer* **96**, 448–454 (November 1974).
14. R. Viskanta, D. M. Kim and C. Gau, Three-dimensional natural convection heat transfer of a liquid metal in a cavity, *Int. J. Heat Mass Transfer* **29**, 475–485 (1986).
15. V. G. Kubair and G. R. V. Simha, Free convection heat transfer to mercury in vertical annuli, *Int. J. Heat Mass Transfer* **25**, 399–407 (1982).
16. R. K. MacGregor and A. F. Emery, Free convection through vertical plane layers—moderate and high Prandtl number fluids, *ASME J. Heat Transfer* **91**, 391–403 (1969).
17. H. M. Domanus, R. C. Schmidt, W. T. Sha and V. L. Shah, *COMMIX-1A: A Three-dimensional Transient Single-phase Computer Program for Thermal–Hydraulic Analysis of Single and Multicomponent Systems*; Vol. 1: *Users Manual*, NUREG/CR-2896, Vol. I (1983).
18. B. Gebhart, *Heat Transfer* (2nd Edn), p. 377. McGraw-Hill, New York (1961).
19. E. M. Sparrow and J. L. Gregg, The variable fluid-property problem in free convection, *Trans. ASME* **80**, 879–886 (1958).
20. H. M. Domanus, R. C. Schmidt, W. T. Sha and V. L. Shah, *COMMIX-1A: A Three-dimensional Transient Single-phase Computer Program for Thermal–Hydraulic Analysis of Single and Multicomponent Systems*; Vol. 2: *Assessment and Verification*, NUREG/CR-2896, Vol. II (1983).

21. R. C. Amar and J. C. Mills, COMMIX code validation studies with phenomenological sodium natural convection experiments, Second Specialists' Meeting on Decay Heat Removal and Natural Convection in LMFBRs, pp. 399–407 (1982).
22. F. E. Dunn, D. J. Malloy and D. Mohr, LMR thermal hydraulic calculations in the U.S., Int. Meeting on Advances in Reactor Physics, Mathematics and Computation Conf., Paris, 27–30 April (1987).
23. D. J. Malloy, Stratification and mixing effects in an LMFBR cold pool following loss of heat sink transients, Int. Meeting on Advances in Reactor Physics, Mathematics and Computation Conf., Paris, 27–30 April (1987).

CONVECTION NATURELLE DANS UN CYLINDRE VERTICAL: COMPARAISON DES PREVISIONS DE COMMIX-1A AVEC L'EXPERIENCE

Résumé—On conduit une expérimentation sur la convection naturelle dans l'eau pour étudier l'effet de plusieurs tiges chauffantes verticales et de la profondeur d'immersion de la surface chauffante sur les caractéristiques globales du transfert thermique de l'assemblage à l'intérieur d'un cylindre vertical. Le code de calcul tridimensionnel COMMIX-1A est utilisé pour prédire les profils de température et les coefficients globaux de transfert de chaleur. L'accord entre les données expérimentales et les prédictions numériques est excellent.

NATÜRLICHE KONVEKTION IN EINEM SENKRECHTEN ZYLINDER: VERGLEICH VON BERECHNETEN (COMMIX-1A) UND EXPERIMENTELLEN ERGEBNISSEN

Zusammenfassung—Es wird die natürliche Konvektion in einem senkrechten, wassergefüllten Zylinder untersucht, der eine Anzahl senkrechter Heizstäbe und eine Kühlfläche unterschiedlicher Höhe enthält. Mit Hilfe des dreidimensionalen Rechenmodells COMMIX-1A wird das Temperaturprofil und der Gesamt-Wärmeübergangskoeffizient berechnet, wobei sich eine hervorragende Übereinstimmung zwischen Meßwerten und numerischer Berechnung ergibt.

ЕСТЕСТВЕННАЯ КОНВЕКЦИЯ В ВЕРТИКАЛЬНОМ ЦИЛИНДРЕ: СРАВНЕНИЕ РАСЧЕТОВ, ПОЛУЧЕННЫХ С ИСПОЛЬЗОВАНИЕМ ПРОГРАММЫ COMMIX-1A, И ЭКСПЕРИМЕНТАЛЬНЫХ ДАННЫХ

Аннотация—Проведен эксперимент по естественной конвекции в воде с целью исследования этого явления для набора вертикальных нагреваемых стержней и влияния высоты подъема охлаждающей поверхности на характеристики суммарного теплопереноса блока внутри вертикального цилиндра. Для определения температурных профилей и коэффициентов суммарного теплопереноса используется трехмерная компьютерная программа COMMIX-1A. Получено очень хорошее соответствие межбу экспериментальными данными и численными расчетами.