LETTERS TO THE EDITOR

COMMENTS ON "MODERATOR CIRCULATION IN CANDU REACTORS: AN ALTERNATIVE APPROACH FOR THE TUBE MATRIX SIMULATION"

In Ref. 1, Fath and Hussein present some numerical results purportedly relevant to the flow of the moderator fluid in a Canada deuterium uranium (CANDU) reactor vessel. Since the results can be viewed as having relevance to CANDU reactors operated by Ontario Hydro, we feel obliged to comment on both the numerical solutions presented and the method proposed by Fath and Hussein.

THE METHOD

The idea of representing an array of tubes of circular cross section using a different array of square tubes ignores the difference in hydrodynamic properties between the two shapes. How well Fath and Hussein's method allows for the flow resistance of the modeled array of square tubes is not clear, but it is highly unlikely that it would be similar to the flow resistance properties of an actual array of circular tubes, even if the size and spacing of the two arrays were identical (which they are not). On the other hand, the commonly used porous medium approach permits a distributed drag force based on empirical characteristics of flow past circular tubes. Thus, Fath and Hussein's claim to "a more realistic representation of the tube matrix" is not justified.

Equally important is the extremely crude treatment of the curvilinear boundary, which is unsatisfactory in a flow whose character is largely determined by the development of the inlet jets on the curved wall.

Our intention is not to critically analyze all contentious aspects of the methodology presented in Ref. 1. However, the results presented deserve close scrutiny.

THE RESULTS

Fath and Hussein admit that the accuracy of their computational results was severely limited by their computer resources (which had only 28K memory available). While that in itself is not unusual, in this case, the disparity between the task and the resources employed to perform it is not acceptable. We feel that Ref. 1 is severely lacking in its failure to recognize and discuss those features of the presented "solutions" that result from gross approximation errors on a coarse mesh with distorted boundary conditions. These are evident even in the simplest cases presented.

The steady-state isothermal flow pattern without tubes (Fig. 3 of Ref. 1) exposes the inadequacy of the sawtooth approximation of the smooth curved wall. Physics dictates that the inlet jet should remain in the proximity of the wall by virtue of the inertia of the recirculating fluid, and velocities should be the highest close to the wall; in Fig. 3, they are the highest about midway between the wall and the center of the recirculation loop. Near the top of the tank, this loop is separated from the wall by a large region of very sluggish flow. This is not physical.

Similar artificial effects of the rough boundary, in conjunction with a very strong channeling effect of the rows of square tubes, are seen in the flow pattern for the isothermal case with tubes (Fig. 4 of Ref. 1). When this is compared with the experimental data taken from Ref. 2 (Fig. 7 of Ref. 1), Fath and Hussein's contention that "the shape and extent of the circulation velocities and the flow in the core region agree quite well" is in our opinion a misjudgment. Analogous comparisons with computations using the porous medium approach and exact representation of the curved wall (in cylindrical coordinates) presented in Refs. 2 and 4 of Ref. 1, look distinctly better.

Similarly, Fath and Hussein's contention that "a satisfactory degree of confidence in the results may be added by testing the technique with regard to symmetry for a simple circulation problem in a rectangular flat cavity such as shown in Fig. 5" is questionable. It is true that symmetry is maintained, but unless the initial and boundary conditions (which can only be guessed because they are not presented) are very strange, the solution is nonphysical with the temperatures downstream of the obstacle not affected by the surrounding fluid movement or diffusion.

The problems are compounded in the more demanding cases in which the reactor conditions are simulated. An examination of the temperature field in Fig. 9 of Ref. 1, which is supposed to represent (in two dimensions) the temperature distribution in a CANDU-600 calandria under normal operating conditions, reveals its nonphysical character. The field is very flat except for a few cells located between pairs of blocked-off cells near the center of the core where temperatures reach inordinately high values. The temperature difference between two adjacent cells reaches more than 43°C at a location just above the center of the core, which is almost 11 times the temperature differential between the inlet and the outlet. This is clearly a numerical artifact due to the extreme coarseness of the mesh in conjunction with an inadequate level of diffusion in the flow, which is assumed to be laminar. The coarseness of the mesh does not allow any local, secondary flows to be resolved. In a cell with only two vertical faces open to flow, which happens to lie in a region of predominantly vertical velocities, the heat balance is determined by the local heat source and diffusion. If the diffusion coefficient is too small, this must result in exaggerated local temperature gradients and therefore local temperature maximums that are too high. There is little doubt that changing the mesh configuration, e.g., by doubling the number of nodes in each direction, would result in very different local maximum values. In other words, the spatial convergence of the "solution" is highly questionable and is not discussed in Ref. 1.

While the existence of high local temperature maximums between the square "tubes" is explainable and on a very coarse mesh seems inevitable, an isolated maximum of 64.1°C in a wide-open cell near the edge of the core at about the 3 o'clock position appears to have no explanation other than possibly a fluke in the heat source distribution data or a coding error.

Incidentally, the distribution and relative magnitude of the local temperature extremes in the mockup solution is very different from the full-scale solution, which does not support the claim of "thermal similarity" between the two. The mockup solution also contains an unexplainable fluke value (in this case a minimum) of 45.0, which strongly suggests a coding error.

A separate issue of importance regarding the relevance of the results to CANDU reactors is the temperature difference between the inlet and the outlet in the full-scale solutions (Figs. 9 and 12 of Ref. 1). The design moderator flow rate for CANDU-600 (not divulged in Ref. 1) is ~0.94 m^3/s. If a total heat load of 118 MW (as in Table I of Ref. 1) is assumed, the heat balance requires an inlet-outlet temperature difference of 27°C. In the solutions presented in Ref. 1, this difference varies from 4.0°C in the case with the design inlet arrangement (Fig. 9 of Ref. 1) to 4.9°C in the case with the modified inlet arrangement. This raises two questions. First, the >20% difference between the two solutions is not explained. It may signify either a failure of the numerical scheme to conserve energy or a failure of the transient calculation in at least one of the cases to approach a time-independent solution (no criterion of convergence to a steady-state solution is mentioned). Another possibility is a lack of consistency in some assumptions or input data between the two cases. The second problem with the inlet-outlet temperature difference is that it is five or six times less in both solutions than in a CANDU-600. Fath and Hussein mention that difficulties with inlet boundary conditions resulted in too high a flow rate. However, the ratio of the total heat load to the third power of the total flow rate is an important similarity criterion for this type of flow. If the flow rate is five times too large, this ratio is two orders of magnitude too small. This alone would have made the results of Fath and Hussein's computations totally irrelevant to CANDU reactors.

Obviously, any conclusions based on these irrelevant results, such as the assessment of designing details of inlet nozzle placement, bear no weight.

Jacek Szymanski
W. I. Midvidy

Ontario Hydro
Nuclear Safety Department
700 University Avenue
Toronto, Ontario M5G 1X6, Canada

July 24, 1990

REFERENCES

1. H. E. S. FATH and M. A. HUSSEIN, "Moderator Circulation in CANDU Reactors: An Alternative Approach for the Tube Matrix Simulation," *Nucl. Technol.*, **88**, 307 (1989).

RESPONSE TO "COMMENTS ON 'MODERATOR CIRCULATION IN CANDU REACTORS: AN ALTERNATIVE APPROACH FOR THE TUBE MATRIX SIMULATION'"

In response to the comments of Szymanski and Midvidy[1] concerning Ref. 2, I begin by quoting the following from Ref. 2:

> ". . . The results presented here are to be considered as an illustration of the code's capabilities and the effectiveness of the developed approach. A complete simulation can be done whenever enough computer storage is available."

Szymanski and Midvidy's careful investigation drew my attention to two important typographical errors that were used as a basis for Szymanski and Midvidy's criticism. In Fig. 9 of Ref. 2, the temperature spots near the edge of the core at about the 3 o'clock position should read 46.1°C instead of 64.1. Also, in the mockup solution of Fig. 14 of Ref. 2, the temperature spot of 45.0 should read 49°C. All of the arguments based on these typographical errors are not relevant in view of these corrections.

In Szymanski and Midvidy's opinion, the introduced method is not accepted in simulating moderator flow in Canada deuterium uranium (CANDU) reactors because it ignores the effect of the drag forces due to the presence of the tube matrix and because of the crude treatment of the calandria curvilinear boundary. The claim that drag forces are not accounted for is not true. Drag forces are already included[3] in the formulation through boundary conditions and pressure variation around the immersed object. In principle, one can calculate the drag on a blocked cell from the pressure variation around this cell. In the maximum credible accident method, cell pressure is the driving force behind adjusting the divergence in the computational region. Our approximation of the smooth curvilinear calandria surface as a stepped surface is not a reason for rejecting the approach as a whole. In fact, if one has a large computer storage capacity, the consequences of the stepped boundary are highly reduced. On the other hand, Viecelli[4] proposed a method that treats the fluid boundary at an arbitrary curved wall or obstacle that can be