N

PNL-SA-8070

CONF-800804--35

PREDICTIONS OF CONVECTIVE LOSSES FROM A SOLAR CAVITY RECEIVER

L. L. Eyler

December 1979

Work Support by
the U.S. Department of Energy
under Contract EY-76-C-06-1830

Pacific Northwest Laboratory
Richland, Washington 99352

DISCLAIMER

PREDICTIONS OF CONVECTIVE LOSSES
FROM A SOLAR CAVITY RECEIVER

L. L. Eyler
Member ASME
Pacific Northwest Laboratory
Richland, WA 99352

ABSTRACT

Convective losses arising from buoyancy driven flow were calculated for a two-dimensional model simulating a solar cavity receiver. The TEMPEST code, capable of fully three-dimensional coupled thermal-hydraulic transient calculations, was used for the simulation. Predicted velocity and temperature results for a 2.59 m deep by 2.88 m high rectangular cavity with an aperture opening of 1.72 m were used to determine convective losses for prescribed interior wall temperatures and cavity orientation. Velocity vector and temperature isotherm plots were used to analyze flow characteristics.

NOMENCLATURE

A	area
C_p	specific heat
d	aperture opening
D	interior cavity depth
h_f	forced convection coefficient
h_n	natural convection coefficient
H	interior cavity height
$\dot{Q}$	heat rate
T_w	wall temperature
T_a	ambient temperature
V	velocity
α	inclination angle
ρ_o	reference density

INTRODUCTION

One type of system for generating thermal power from solar energy utilizes a field of two-axis tracking mirrors (called heliostats) to focus sunlight on a central receiver mounted on a tower. Central receivers have been the subject of several feasibility and design optimization research efforts (1,2,3). However, it is apparent from even a brief literature review that estimates of convective losses from a central receiver are, at best, tenuous. The results of a workshop on convective losses from central solar receivers[a] provided additional evidence of difficulties in this area.

Central solar receivers can generally be classified as external or internal (cavity) type, depending upon the geometrical configuration. Neither type is currently amenable to adequate analysis to determine convective losses. External receiver configurations characteristically have Rayleigh numbers beyond experimentally determined heat transfer correlations. In addition, wind variations may cause mixed mode losses, further complicating analysis. Internal cavity receivers have numerous possible geometrical configurations that can lead to complicated flow and, hence, thermal response characteristics. Overall heat transfer coefficients for open cavity flow are virtually nonexistent.

The need to adequately determine convective losses has several justifications. One is economics. Cost-effective minimization of convective, as well as other, losses can only enhance a system's overall operation. Adequately

(a) "Convective Losses from Central Solar Receivers," Workshop sponsored by Sandia-Livermore Laboratory and U.S. Department of Energy, held at Dublin, California, April 1979.

determined convective losses would aid in design analysis and system parameter studies used as guidance for project development. Another justification stems from the guidance that convective loss estimates can provide in the design of experiments. This is especially important for large-scale experiments, which are generally expensive and time consuming.

This paper describes an effort at the Pacific Northwest Laboratory to model relative convective losses for internal cavity-type receivers. Current methods applied to convective loss analyses are briefly reviewed first. Next, the capabilities of the TEMPEST computer code used in the PNL work are described. Results of this simulation are then presented in the form of velocity vector and temperature isotherm plots for a two-dimensional rectangular cavity receiver with prescribed interior wall temperatures.

Relative convective losses were determined for a 2.59 m deep by 2.88 m high cavity with a 1.72-m aperture opening. Results for the aperture oriented vertically, angled 32° to the vertical, and horizontally are presented. These results characterized the nonlinear behavior of the relative convective loss for varied aperture orientation. Similar characterizations are presented for a cavity receiver scaled up by a factor of 5 with two different aperture opening sizes.

BACKGROUND

A review of literature concerned with central receivers has shown that remarkably little sophistication is currently being used to analyze convective losses. In some cases, no account for the loss is made at all. One apparent reason is that the complexity of the problem has necessitated making simplifying assumptions; this results in an analyzable problem, but one which may be oversimplified to the extent of relative uselessness. Until recently, one of the major stumbling blocks seemed to be the lack of experiments to guide design analyses of large-scale receivers.

Consider, for example, the general schematic of a simplified cavity model shown in Figure 1. A heat balance on the system would include:

IN

$\dot{Q}_i$ — solar insolation from a mirror field

OUT

$\dot{Q}_{HX}$ — heat removed by heat exchanger tubes

$\dot{Q}_{cond}$ — heat lost by conduction through cavity walls

$\dot{Q}_{RR}$ — heat lost by reflected radiation

$\dot{Q}_{hL}$ — heat lost by convection flow

To determine the relative efficiency of windowed versus windowless apertures of such a simple cavity, Jarvinen (1) completely neglected convective losses in the windowless cavity. Subject to some stringent window cooling restrictions, he concluded that windowed cavities were (or could be) as thermally efficient as windowless receivers. Indeed, if convective losses for the windowless cavity could be reasonably estimated, relative theoretical windowed cavity efficiencies may look even better.

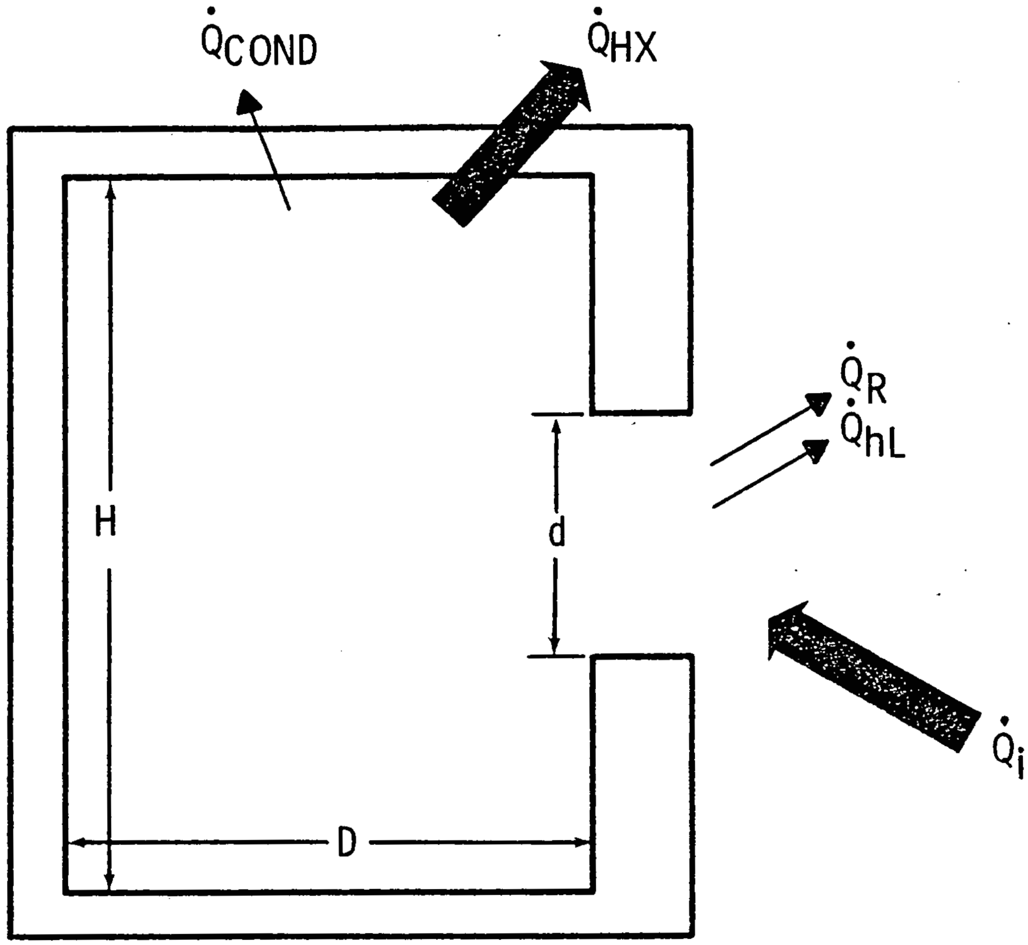


FIGURE 1. Heat Flows in a Simplified Cavity Model

In their analysis of receivers in the 300 to 1300°C range, Wu and Wen (2) utilized grossly simplified assumptions. For example, to evaluate convective losses from a cavity receiver, they determined convected heat loss as

$$\dot{Q}_{hL} = hA(T_w - T_a). \tag{1}$$

They assumed the convection coefficient is the sum of a natural convection coefficient, h_n, and a forced convection coefficient, h_f, as

$$h = h_n + h_f \tag{2}$$

where

$$h_n = 1.3(T_w - T_a)^{1/3} \tag{3}$$

and

$$h_f = 2.0\ V. \tag{4}$$

Equations (3) and (4), obtained by Wu and Wen from Reference (4), are said by them to be applicable to large vertical plates in the turbulent range. The interesting point that must be noted is that the area A in Equation (1), to which the heat loss is applied, is the aperture opening area. The treatment of an opening as a flat plate is somewhat questionable. Further, adding convection coefficients, as in Equation (2), may be incorrect for mixed mode convection.

In another analysis, Tracey, Blake, and Brown (3) evaluated convected losses for an experimental 1 MW_{th} cavity. To do so, they assumed that each surface of their rectangular cavity was at constant (measured) temperature in an ambient temperature fluid. They evaluated a resulting convection loss from each surface individually as if it were in a quiescent ambient fluid. This method is relatively straightforward but does not account for any nonlinear flow interactions that may be caused by mutually perpendicular surfaces. Although it offers a distinctly better approach than the previously noted method of Wu and Wen (2), this method would be very difficult to apply to wind-affected cavities.

During initial testing of the Boeing/EPRI cavity, certain anomalies associated with uneven heat removal in the heat exchanger tubes were noted (5). A subsequent first-order analysis was able to answer qualitatively and, to a limited extent, quantitatively that the uneven heating was due to convective flow-induced velocities of up to 1.5 m/sec through the aperture opening. Corresponding wall temperatures measured in these tests varied from approximately 1300°C on the back wall to typically 700°C on the heat exchanger tubes, and provided the driving force for the buoyancy-induced flow. However, the only conclusions reached in the analysis were that the flow could account for an unexplained loss and that large portions of the aperture opening were active in the convection process (5).

APPROACH

The approach used in this work was to apply state-of-the-art numerical techniques to analyze the complexities of buoyancy-induced flow in cavity-type receiver geometries. The computer code used for this work is TEMPEST. This user-oriented code, under development at PNL, is a fully-coupled, three-dimensional, transient hydrothermal finite difference code. It is capable of modeling geometries using either Cartesian or cylindrical coordinates, and employs a transient approach to a steady-state solution. Solids conduction and flow convection regimes are coupled in the solution scheme. TEMPEST is capable of treating both steady and transient boundary conditions.

During its development, TEMPEST has been applied to numerous test cases, including closed cavity natural convection problems, and has been shown to agree well with data. This work was the first attempted application of TEMPEST to natural convection in an open cavity geometry. Unfortunately, open cavity experimental data are virtually nonexistent. Limited wall temperature data have been obtained for a Boeing/EPRI scaled cavity receiver currently undergoing field tests (5); however, no cavity velocity data [with the exception of a one-point measurement (5)] were available. Consequently, the calculated results presented in this work could not be compared with data, but are instead supported by arguments that the computed results follow physically realizable trends.

Certain aspects of the TEMPEST version used in this work should be noted. First, the Boussinesq approximation was used in writing the discrete form of the conservation equations solved by the code. Second, there is no turbulence model currently incorporated in the code, but a constant turbulent viscosity may be specified as input. It was concluded that sufficient results could be obtained to satisfy the goals of this work by performing constant viscosity calculations.

The Boussinesq approximation is more restrictive in the sense that certain limitations to the problems that can be solved are obtainable from analysis (6). Early in this work, it was determined that the Boussinesq approximation in TEMPEST precluded cavity analyses at back wall temperatures expected in full-scale central receiver systems. However, it was further concluded that low-temperature analyses could be used at this stage to obtain information on general trends and parameter tendencies of cavity convection loss.

The Boeing/EPRI cavity (5) was used as the basis for the model considered in this work. A cross section of the octagonal cavity is shown in Figure 2. In operation, solar insolation enters the aperture opening, reflects off the back wall at temperatures near 1300°C, and is absorbed in the heat exchanger panels. Limited experimental data on the cavity have shown that the high wall temperatures induce buoyant flow with resulting velocities of up to 1.5 m/sec occurring in the aperture.

Using the TEMPEST code, velocities and temperatures were computed for a simplified two-dimensional geometry to demonstrate analysis capabilities. The geometry utilized was a model of the Boeing cavity discussed previously. Although TEMPEST is capable of treating three-dimensional geometries, only a two-dimensional flow model was used for expediency and to simplify interpretation of the results. A two-dimensional model is readily justified in terms of

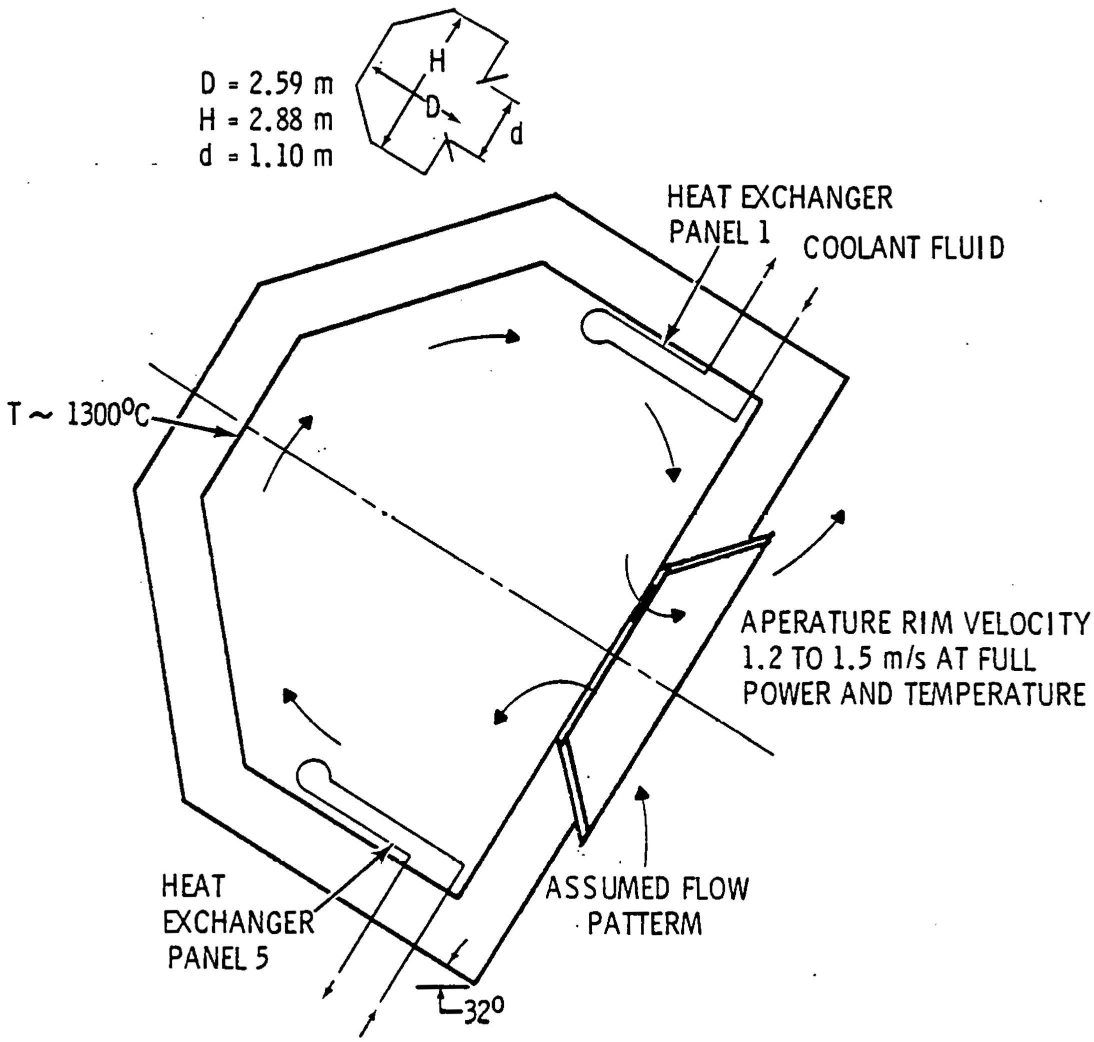


FIGURE 2. EPRI/Boeing Cavity Design Showing Assumed Convection Flow Pattern

reduced computation time for parameterizations. Two-dimensional results are also very useful as a screening process prior to performing a more detailed three-dimensional analysis.

RESULTS

Typical computed velocity and corresponding temperature fields are shown in Figures 3 and 4, respectively, for the open cavity model. For these calculations, a 16 x 10 variable grid spacing was used. In light of the previously discussed Boussinesq approximation, a maximum impressed temperature difference of 100°C was specified by setting the back wall surface temperature, T_3, at 160°C and the exterior ambient temperature at 60°C. A temperature of 110°C was specified for simulated heat exchanger surfaces, as shown in Figures 3 and 4. The other surfaces, designated as 2 in Figures 3 and 4, were treated as adiabatic, modeling highly insulated receiver walls.

As shown in Figures 3 and 4, the flow pattern computed for this case was very steady. With TEMPEST's transient approach to steady state, approximately 60 to 70 sec of CDC 7600 computer time was required for execution. The predicted flow in Figure 3 follows that which would generally be expected. The hot back wall causes a strong buoyancy force that drives a large recirculating flow region in the upper portion of the cavity. Significant spillage or convective loss is readily apparent, as indicated by the outward flow in the upper half of the aperture opening. The inflowing cold air in the lower portion of the aperture is heated in the recirculating lower region.

External to the cavity, the flow is quiescent except for the area near the upper outside wall. In this work, several different exterior boundary

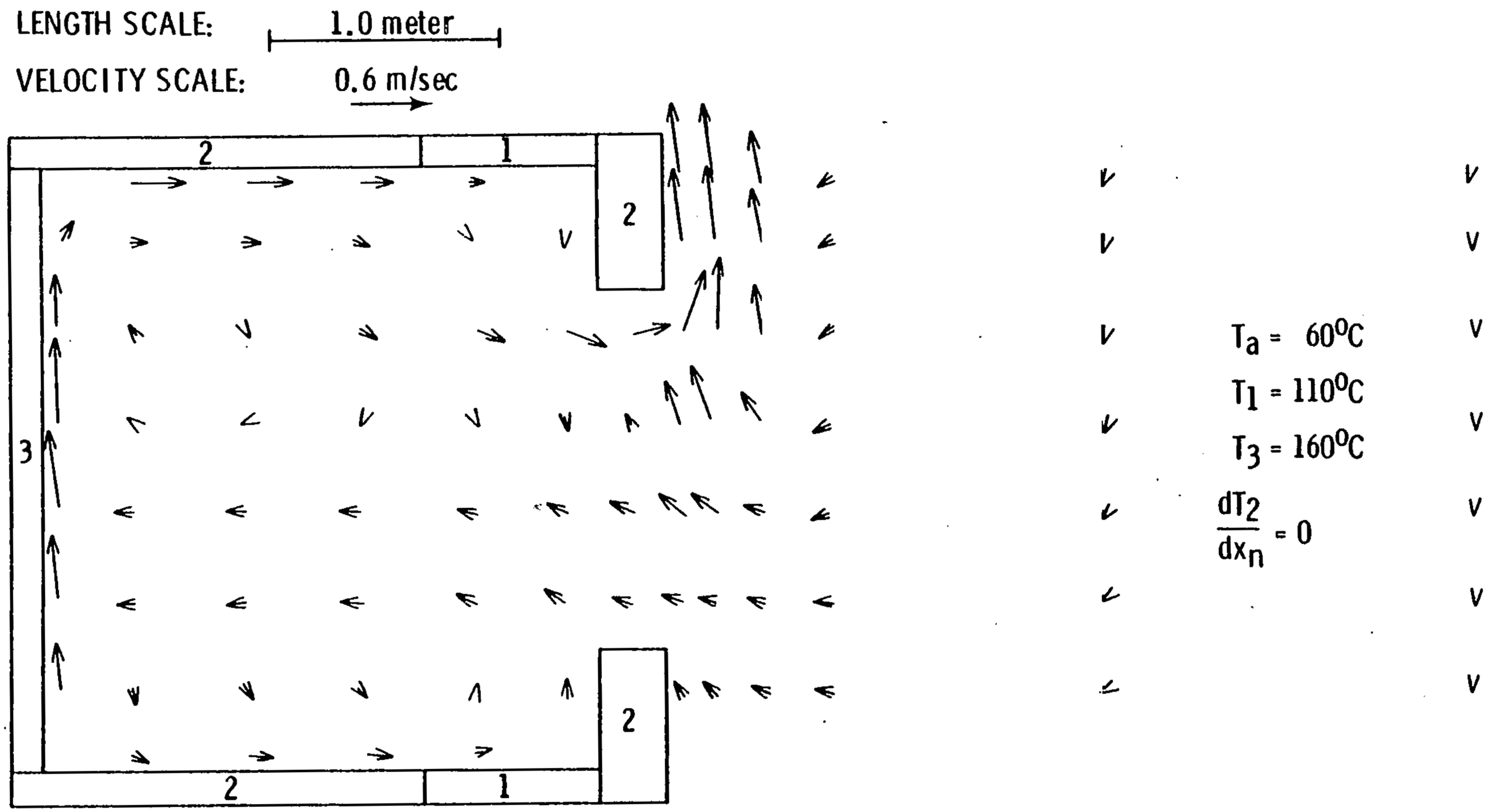
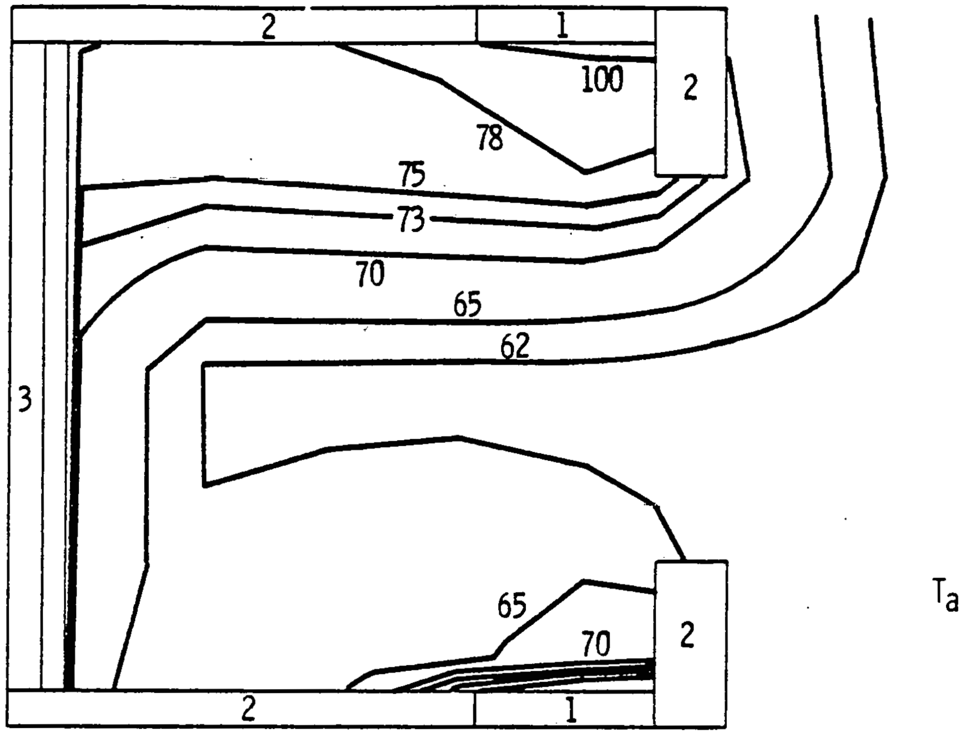


FIGURE 3. Velocity Vectors Computed for a Cavity Inclination of 0° with the Horizontal



$T_a = 60^0C$

$T_1 = 110^0C$

$T_3 = 160^0C$

$$\frac{dT_2}{dx_n} = 0$$

CONTOUR LINES SHOWN
ARE AT SPECIFIED ^0C

FIGURE 4. Temperatures Contours Computed for a Cavity Inclination of 0° with the Horizontal

conditions were used to investigate the effect on the interior flow. It was determined that, as long as the exterior modeled region extended beyond a distance about equal to the interior cavity depth, the boundary condition of the modeled exterior region had no apparent effect. It should be noted that TEMPEST has the capability of treating user specified transient boundary conditions on the exterior modeled region. However, in this work, no attempt was made to model an external wind and its effect on the cavity flow or convective losses.

The isotherms shown in Figure 4 depict a temperature field that is in good agreement with the physically expected field. The characteristic inflow of cold ambient air, the hot and relatively stagnant fluid in the upper frontal area, and the steep gradient near the rear wall are all visible. The stratification evidenced by the gradually changing, horizontally-oriented contour lines in the central region is quite apparent. The isotherms shown in Figure 4 indicate that the 16 x 10 grid spacing used does not provide sufficiently fine resolution. This modeling aspect deserves further attention in future work.

The Boeing cavity central receiver concept is, in practice, designed to be inclined at a 32° angle with the horizontal. This inclination uses the mirror field most effectively and, heuristically, decreases convective loss. In Figure 5, velocity vector and temperature field plots are shown for three orientations. Figures 5a and 5b repeat the results shown in Figures 3 and 4. Figures 5c and 5d are results for the 32° incline. The results for a fully rotated, down-facing model are illustrated by Figures 5e and 5f.

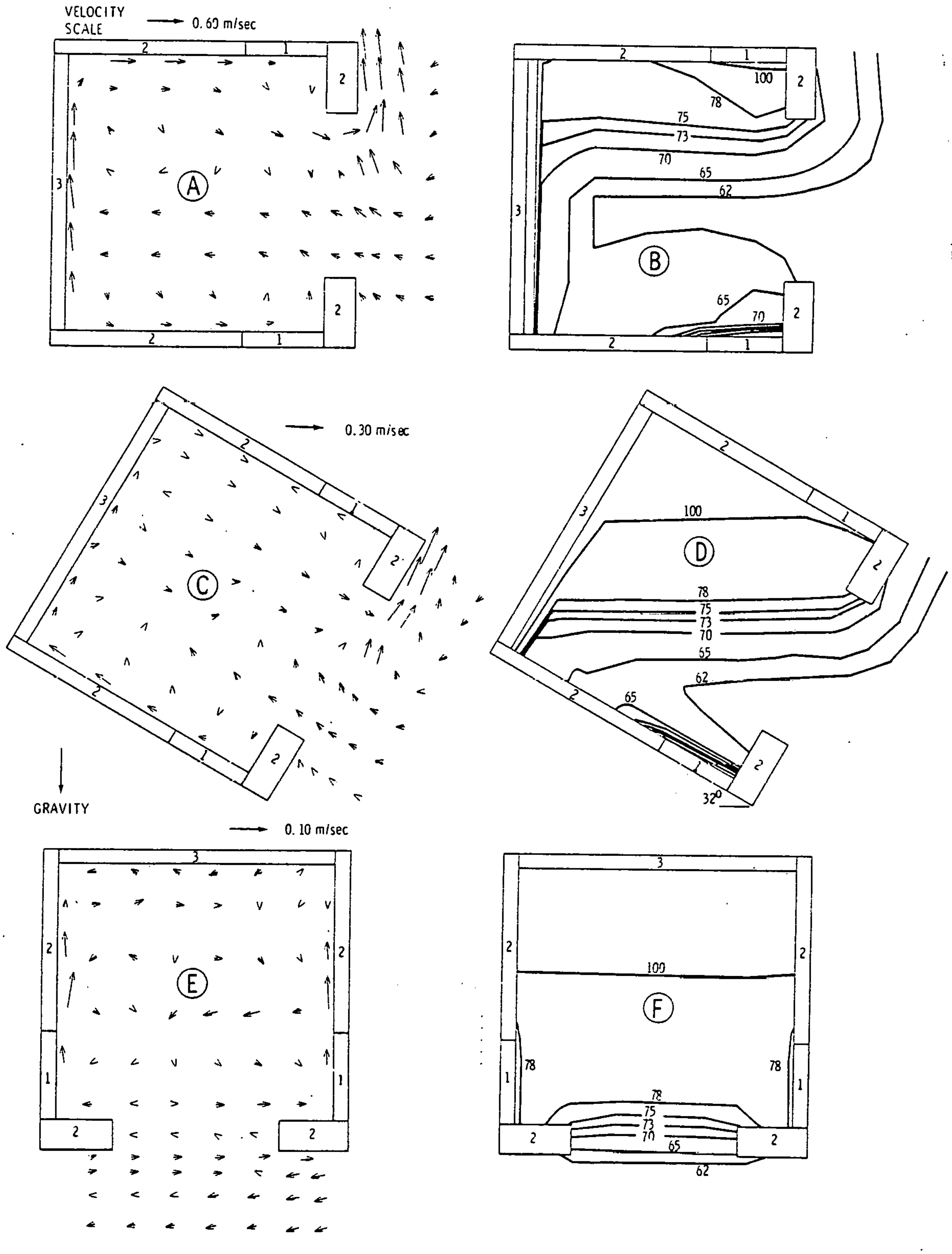


FIGURE 5. Computed Velocity Vectors and Temperature Contour Lines at Inclination Angles of 0°, 32° and 90°

It should be noted that the velocity vectors (arrow lengths) in Figure 5 are normalized to the maximum velocity occurring in each case. While the maximum is defined for each case, caution must be exercised in comparing absolute velocities represented by arrow lengths from one case to the other. For comparison, the results in Figure 5 are shown at the same time, t = 200 sec, into the simulated transient approach to steady state. It was previously noted that the horizontal orientation (A) was very steady. At t = 200 sec, the 32° angled cavity was showing some oscillatory behavior of the convective flow through the aperture opening. The typical oscillations were periodic, lasting approximately 11 sec with variations of ±5%, about a mean outflow velocity. This observation may indicate that either steady state had not yet been reached or that a chugging effect was being predicted. Continuation of the simulated transient could be used to verify this. The flow for the 90° rotated cavity had definitely not reached steady state. Due to the stable flow situation of the hottest wall being on the top, the cavity flow should eventually reach a virtual stably-stratified condition. As mentioned previously, however, the grid structure used for this case was not optimal. Hence, no attempt was made to continue the calculations beyond the 200-sec point shown in Figures 5e and 5f.

The velocity vector plot of the 32° rotated cavity shows a fairly significant decrease in outflowing velocity when compared to the vertical opening. This indicates that the convective loss from the angled cavity is lessened. The vector plot also shows that the large recirculation flow assumed for the analysis of the angled Boeing cavity (compare Figure 2 with Figure 5c) is not necessarily correct. A stably-stratified upper triangular region is indicated by the isotherms. As a result, the overall buoyant driving force has been lessened and, hence, the convective loss decreased.

The actual convective loss from cavities can be determined from the computed velocities and temperatures as

$$\dot{Q}_{hL} = \rho_o C_p \sum_i U_i T_i A_i \tag{5}$$

where the summation is carried out across the computed flow cells in the aperture opening. Q_{hL} represents a net flow of energy out of the cavity by convection. This loss is dependent on many factors such as cavity size, aperture-to-cavity size ratio, and surface temperatures. To determine these effects, TEMPEST output was used to calculate the convective loss, subject to variation in several parameters. The results are presented in relative form in Table 1. The loss calculated for the previously discussed, two-dimensional rectangular model is used as a baseline for comparison.

TABLE 1. Relative Convective Losses

Case	Depth D	Height H	Aperture d	Inclination Angle, α	Relative Loss $\dot{Q}_{hL}/\dot{Q}_{hL,base}$
1 - base case	2.59 m	2.88 m	1.72 m	0°	1.0
2	2.59	2.88	1.72	32°	0.64
3	12.95	14.35	8.58	0°	5.31 (1.0)
4	12.95	14.35	8.58	32°	2.81 (0.53)
5	12.95	14.35	4.29	0°	4.09 (0.77)

The results are presented as relative losses for several reasons. First, radiation transfer from the back wall to the heat exchanger surfaces was not included in this work, so actual magnitudes of convective losses cannot be related to a percentage-of-total value. Second, it is somewhat more convenient to identify parameter trends in relative form.

For example, using the modeled Boeing cavity as a base case, several observations can be made from the results in Table 1. Rotating the cavity by 32° caused a decrease in convective loss of about 36% when compared to the vertical base case. When the cavity was geometrically scaled up by a factor of 5, the convective loss was multiplied by nearly 5.3 times. This could indicate a nonlinear geometry effect. When the larger cavity was rotated through an angle of 32°, there was a 47% decrease in convective loss. Here again is apparent evidence of nonlinear geometry effects.

To determine the effect on convective loss of decreasing the aperture opening, the opening size was decreased by a factor of 2, while holding all other parameters the same as the horizontal case. For this case, the convective loss was decreased by 23% from the same size base case.

The convective losses shown in Table 1 can be treated as only typical, not absolute, results. They represent the outcomes of code calculations applied to a simplified two-dimensional model and should be treated as such. Nevertheless, results of this type do demonstrate the capabilities of TEMPEST in analyzing convection losses in buoyancy-induced cavity flows.

CONCLUSIONS AND RECOMMENDATIONS

Convective losses for a simplified model of a large, cavity-type, central solar receiver were calculated. The results show that, within limitations of approximations in the TEMPEST code, cavity thermal-hydraulic characteristics can be determined. The results also indicate that the TEMPEST code is a sophisticated tool for analyzing convective losses, and could be applied to system design analysis, parameterization, and experimental design.

Velocity vector plots and temperature contour plots were calculated in this work for a simplified two-dimensional model of the Boeing/EPRI cavity receiver. These results showed very clearly expected physical trends for the flow field and corresponding temperature field. At an aperture inclination angle of 32° from the vertical, buoyancy driving forces are lessened and convective losses reduced when compared to losses from a cavity with a vertically oriented aperture. For a five-fold scale-up in cavity size, predicted results showed a slightly greater than five-fold increase in convective losses. A similar prediction for a relative decrease in aperture size also showed a considerable decrease in convective losses.

Two shortcomings in the current version of TEMPEST hindered more complete cavity analysis--the Boussinsq approximation used in formulating the conservation equations, and a constant (user-input) turbulent viscosity limitation. The Boussinesq approximation is restrictive in that it limits the range of density variations that may be treated and, hence, currently limits TEMPEST's gas flow applicability to low-temperature differences. Relaxation of this restricton through rewriting of the governing equations to account for position-dependent density will be required to be able to treat cavities with realistic back wall temperatures.

A user-input constant turbulent viscosity is perhaps sufficient to conduct parameterizations. However, to more adequately treat turbulent effects on cavity convection, TEMPEST will have to be modified to include a turbulence model capable of treating buoyancy-dominated and stably-stratified flows.

Relaxation of the Boussinesq approximation and incorporation of a turbulence model into the TEMPEST code require further analysis. Plans for future work include:

- investigation of the effect of noding structure and spacing on the calculation of convective losses--This could be accomplished by varying both the number of nodes used to model a cavity and their spacing.
- investigation of transient external conditions, such as wind variation--This could be performed utilizing TEMPEST's capabilities to accept transient boundary conditions.

These recommended investigations could readily be completed with a two-dimensional model. Subsequently, more sophisticated noding structure with a three-dimensional model could be examined.

ACKNOWLEDGMENTS

Invaluable support leading to the completion of this project was provided by D. S. Trent, responsible for TEMPEST code development, and M. J. Budden. Funding support for TEMPEST code development was provided by the U.S. Department of Energy, Division of Reactor Research and Technology. The work described in this paper was supported by a U.S. Department of Energy Special Studies Project.

REFERENCES

1 Jarvinen, P. O. "Windowed versus Windowless Solar Energy Cavity Receivers." Eleventh Intersociety Energy Conversion Engineering Conference Proceedings. American Institute of Chemical Engineers, New York, New York, 1976.

2 Wu, Y. C., and Wen, L. C. Solar Receiver Performance in the Temperature Range of 300 to 1300°C. JPL No. 5102-82, Jet Propulsion Laboratory, Pasadena, California, 1978.

3 Tracey, T. R., Blake, F. A., and Brown, C. T., "1 MW_{th} Solar Cavity Steam Generator Solar Test Program." Proceedings of the 12th Intersociety Energy Conversion Engineering Conference. American Nuclear Society, New York, New York, 1977.

4 ASHRAE Handbook of Fundamentals, published by ASHRAE, 1967.

5 Zenter, R. C., "Evidence of Free Convection Effects in EPRI/Boeing Solar Receiver Tests." Paper presented at the Sandia Laboratory/Department of Energy Workshop on Convective Losses from Central Solar Receivers, April 17-18, 1979, Dublin, California.

6 Gray, D. D., and Giorgini, A. "The Validity of the Boussinesq Approximation for Liquids and Gases." International Journal of Heat and Mass Transfer. 19:545-551, 1976.