# THE USE OF A HIGH LEVEL CFD CODE IN ENGINEERING EDUCATION

J.E.S. Venart, P. Lemieux, A.C.M. Sousa and D. Tatchell*
Fire Science Centre and Department of Mechanical Engineering
University of New Brunswick
Fredericton, NB, Canada E3B 5A3
*Flomerics Ltd., Kingston Upon Thames, UK KT2 5AA

Phone No. (506) 453-4509

**Abstract**

An advanced multipurpose Computational Fluid Dynamics (CFD) code, FloSYS, developed for use in a CAD/CAE environment has been utilized in undergraduate and graduate courses in heat transfer. Problems are introduced at an introductory level in: - steady 2- and 3-dimensional conduction, - transient 1-, 2-, and 3-dimensional conduction, - free convective cavity flow with conjugate heat transfer, and - turbulent flows with heat transfer. More sophisticated problems, related to the use of porous media concepts for modelling heat exchangers, and compartment fire modelling are introduced at a more senior level.

After one year's experimental use, the CFD code is being recommended for use in other courses in fluid mechanics, environmental studies and heating and ventilation.

## 1. Introduction

One of the major challenges facing engineering educators today is the incorporation in course material of some of the wide variety of professional computational tools available. The use of such tools must enhance the students ability to model complex problems in order to gain an understanding of the phenomena and sharpen their ability in problem definition as well as develop self confidence in the critical assessment of solutions.

Graduates must be prepared not only to function in current technologies but have the education, curiosity and vision to innovate in order to meaningful contribute to technical change and development. This requires an individual who can be versatile and adaptable in his creative professional contributions.

In any curriculum it is essential therefore that the subject material permit a meaningful experience in analysis, experiment, and design. The benefits of the incorporation of an advanced computational fluid dynamic (CFD) code in the Mechanical Engineering curriculum at the University of New Brunswick will be discussed in this paper.

The advanced multipurpose CFD code, FloSYS [1,2,3] has been utilized on an experimental basis over the past two years in the courses ME3433 and ME5463. ME3433, Heat Transfer I, is the first undergraduate exposure to heat transfer experienced by Mechanical Engineering students at the University of New Brunswick. ME5463, Heat Transfer II, is a dual level, senior undergraduate - junior graduate, follow on to ME3433, involving the student to correspondingly more sophisticated analysis and some exposure to design. Each course has a 3 credit hour lecture component with one tutorial per week. ME3435 is a companion laboratory course to ME3433 providing exposure of up to six physical or numerical experiments. The laboratory component in ME5463 is a 2 credit hour, 3 hour per week, alternate week,

Supercomputing Symposium '91
Fredericton, NB Canada
June 3-5, 1991

problem and design laboratory.

In both courses FloSYS has been utilized on an experimental basis to illustrate problems in steady and transient one, two, and three dimensional conduction as well as buoyancy driven flows and heat transfers. In ME5463, more advanced modeling is introduced to include porous media concepts applied to baffled heat exchanger simulation, fire simulations and the resulting free convective flows in single and multiple room compartments as well as environmental and heating and ventilation situations.

For the applications to be discussed in this paper the students are requested to perform numerical experiments using the software installed on two PCs for a class of 35 students. Each student group, of up to three individuals, formulates the given problem(s) and executes the problem analysis on the PC providing output in tabular and graphical form. Versions of the code are available which are executable on supercomputers and advanced workstations.

Prior to attempting the problems the level of exposure to numerical modeling is to that required to appreciate basic relaxation techniques [4] with and without heat generation (1 lecture) and explicit as well as implicit techniques in transient conduction [5] (1 lecture) in addition to a basic introduction to the partial differential formulation for forced and free convection including an introduction to turbulence [6] (4 lectures). The very basic techniques are reviewed in a demonstration session at the commencement of each laboratory.

The software is installed on the PCs with hardware keys permitting free access to the students.

## 2. FloSYS

FloSys is a general purpose CFD code whose solver is equipped to perform fluid-flow and heat transfer simulation in rectangular cartesian and cylindrical polar coordinates (non-swirling axi-symmetric flows). The program can solve up to 25 second-order non linear 1-, 2-, or 3-dimensional partial differential equations. The transient formulation is fully implicit.

The transport equations solved have the following generic form:

$$\partial(\rho * f)/\partial t + \mathrm{div}(\rho * u * f + D_f) = S_{b,f} + S_{u,f}$$

where $\rho$ is the fluid density, f is the dependent variable of the differential equation, ie. the transported quantity, u is the vector velocity, $S_{b,f}$ is the built-in source term and $S_{u,f}$ denotes sources used for boundary representation. $D_f$ is the diffusional flux vector which flows down the negative gradient of f

$$\text{ie.} \qquad D_f = \Gamma_f * \mathrm{grad}\ (f)$$

where $\Gamma_f$ is the exchange coefficient of the variable f.

In fluid flow problems the mass continuity equation is solved. Laminar and turbulent flows can be modeled. A k-$\varepsilon$ turbulent model is employed which models the Reynolds stress tensor by Boussinesq's eddy viscosity approximation. The turbulent viscosity is calculated by means of the Prandtl-Kolmogorov formula expressed in terms of in-cell values.

The discretized differential equations are solved by the finite-volume method using the upwind-differencing scheme. The momentum equations are formulated using the staggered cell treatment. A variant of the SIMPLE algorithm is used to obtain the velocities via adjustments made to satisfy mass-continuity.

## 3. Numerical Experiments

Four numerical experiments are discussed in the following sections. They are a 2-D steady state

conduction problem, transient conduction in a semi infinite plane, a partitioned heated cavity experiment and the numerical simulation of a simple baffled shell and tube heat exchanger. The first two problems are relatively easy to model while the other two require somewhat more effort in conceptualization and problem formulation.

Two-Dimensional Steady-State Conduction

In this experiment each student group is given a different two-dimensional shape with specified isothermal lines and insulated (adialiatic) boundary conditions. An example is shown in Figure 1. The students are asked to numerically predict the temperature and heat flux distribution and determine the shape factor for the specified geometry. They are also asked to confirm their results, using an analogue field or flux plotter. In the course of the problem they build upon their exposure to the relaxation technique to determine a grid independent solution and comment on the ability of their model to include alternate boundary conditions to those of insulation or constant temperature; ie. convective and/or radiative boundaries.

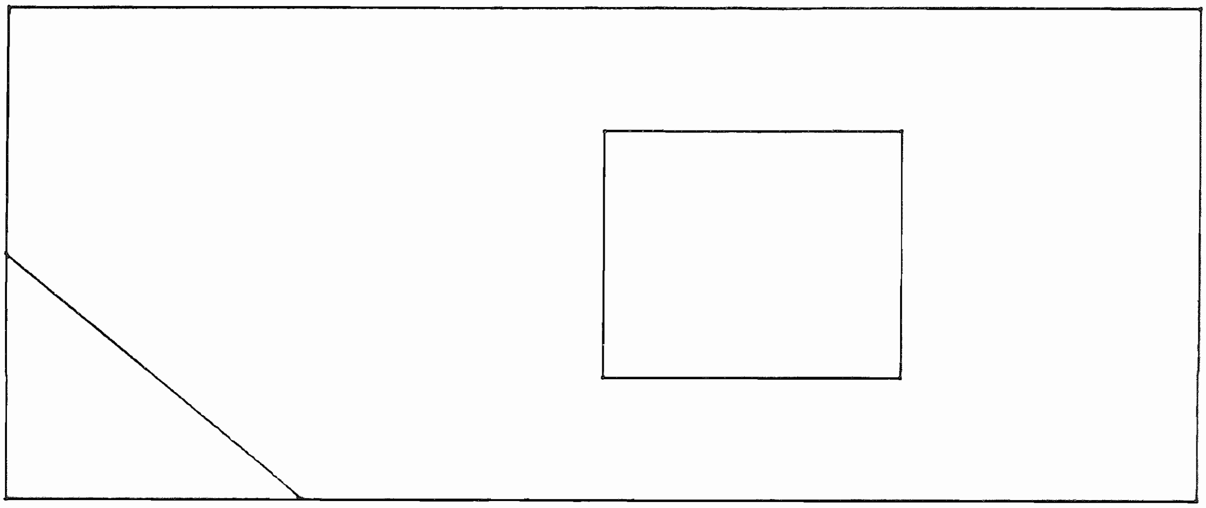

Figure 1(a). Two-dimensional Steady State Conduction; Geometry, $\Delta T = 100°$, 80 . 40 Mesh.

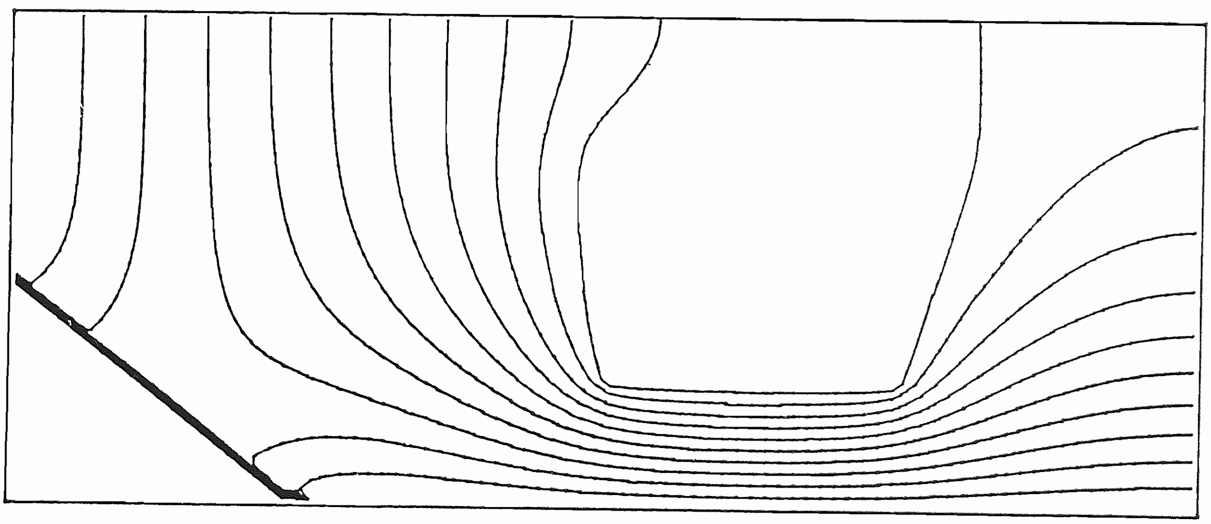

Figure 1(b). Two-dimensional Steady State Conduction; Isotherms, 10° intervals

The results of the students' predictions are generally satisfactory except in those regions of large gradients and inadequate grid resolution. This experience strengthens the students' ability to structure and pose appropriate problems, set necessary boundary conditions define appropriate grids and study differences resulting from the various conditions. The process thus builds confidence in realistic problem formulation and encourages students to use the problems as an opportunity to explore, in a physically meaningful way, the influence of other changes with more complex problems of their own.

Transient conduction in a semi-infinite plane.

A simple geometric configuration for which analytic solutions are available is transient conduction in a semi-infinite solid. This representation approximates to many practical problems and may be used to establish confidence in the student's ability to formulate a variety of different situations.

Closed form solutions have been obtained for three types of changes in surface conditions, instantaneously applied at time t=0. These cases are illustrated in Figure 2 [7].

A further extension to consider 2 and 3D transient conductions provide the opportunity to compare solutions obtained with Heisler charts if desired.

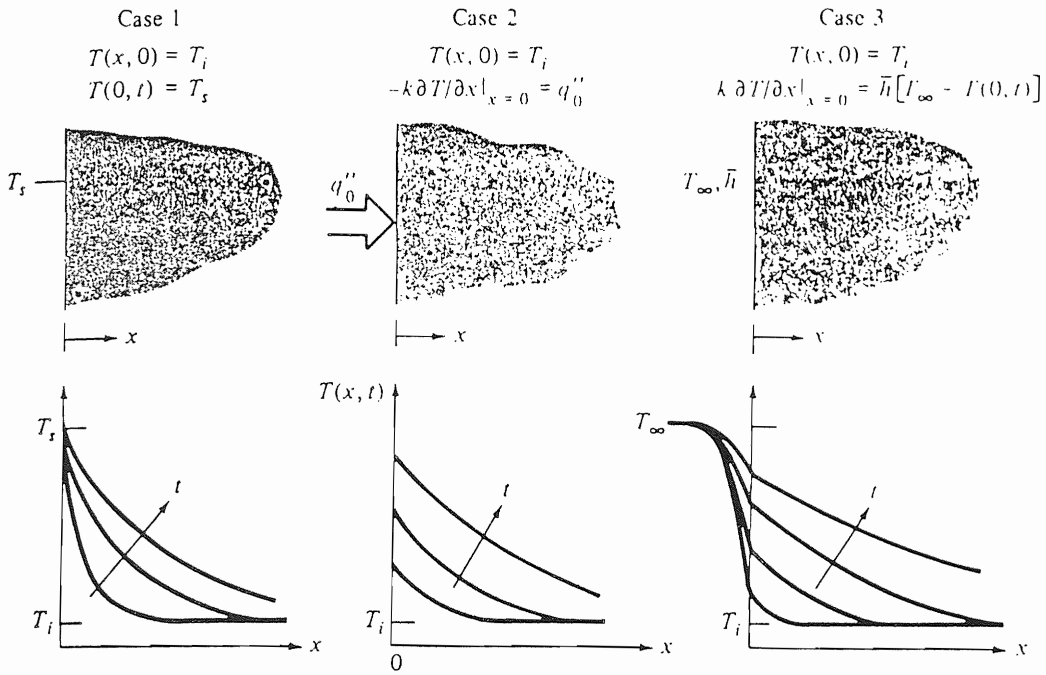


Figure 2. Transient temperature distributions in a semi-infinite solid for three surface conditions: (1) constant surface temperature, (2) constant surface heat flux, and (3) surface convection [5].

Cavity Flows

One major difficulty with instruction in free convection heat transfer is a lack of realistic meaningful scaled experiments by which to demonstrate the physical and engineering importance of the phenomena. A simple cavity problem with one heated and one cooled wall is utilized first to illustrate the physical principles and details of problem formulation, influence of turbulence, and appropriate boundary conditions (Figure 3). The students are next asked to explore the inclusion of partitions and openings on the overall solution results. Problems, similar to those in the latest heat transfer literature [8,9,10], can be examined and extended upon to include room heating and ventilation examples.

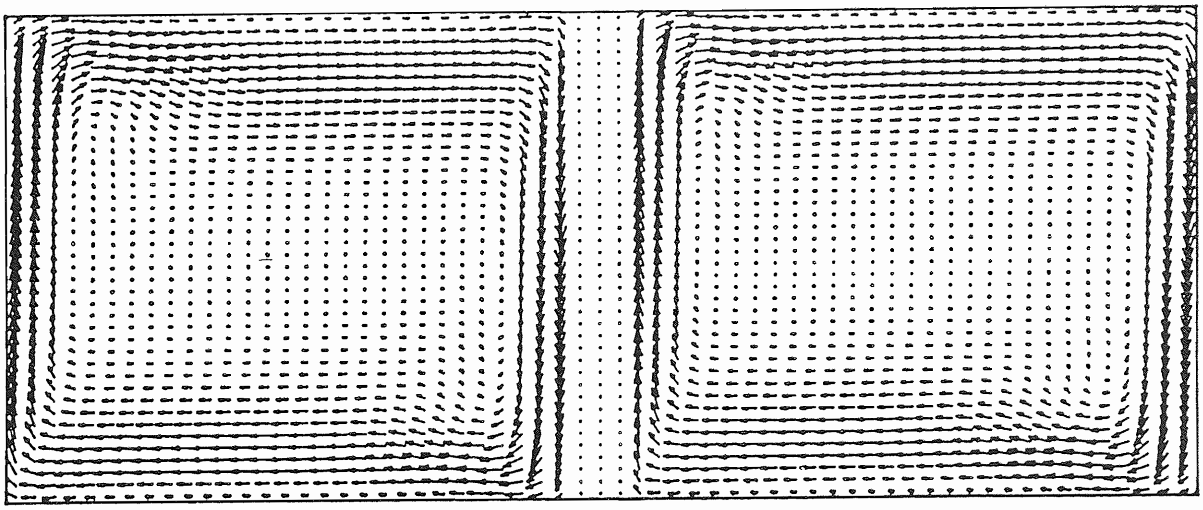

Figure 3a Laminar Free Convection of air in a Partitioned Cavity; Vectors, Grid 58.40, x=0.2, y=0.1, partition 0.01 thick, k=2 W/mK

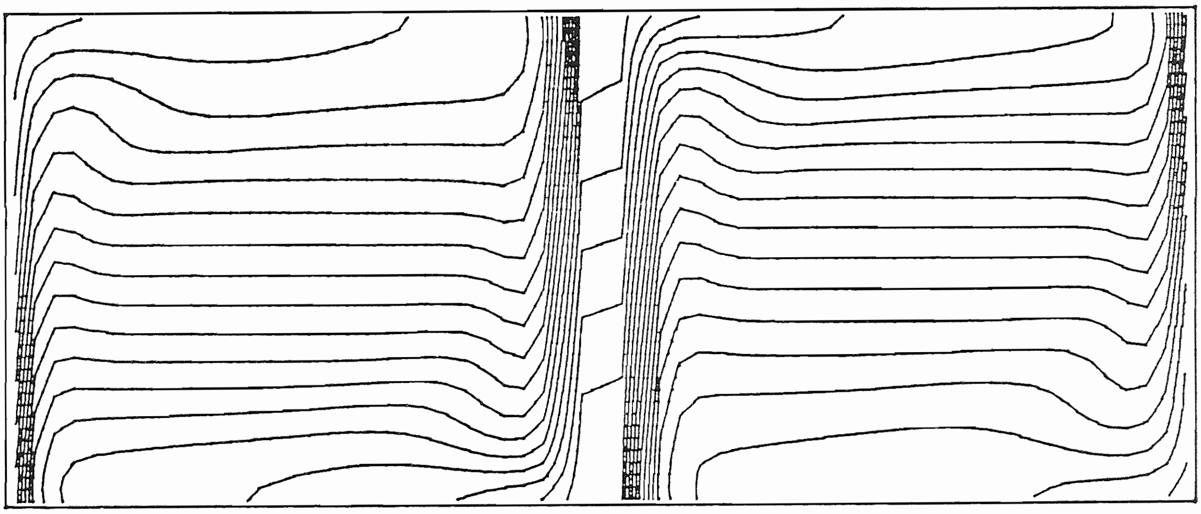

Figure 3b. Laminar Free Convection of Air in a Partitioned Cavity;
Isotherms; $\Delta T = 1.5°$.

Two-Dimensional Heat Exchangers

The last example to be considered simulates a two dimensional baffled shell and tube heat exchanger for which detailed isothermal flow velocity and turbulence data has been obtained [11] and an isothermal numerical solution attempted [1].

The exchanger geometry is shown in Figure 4 with the velocity vector plot obtained shown in Figure 5 for the case of a coarse grid with no tubes in place. Figure 6 shows the velocity vectors in a fine mesh with tubes. Figure 7 illustrates the temperature distribution resulting from the use of a "leaky" baffle plate. Porous media distributed volumetric resistance concepts are used to model the effects of the tube bundle and baffle plate(s). In this case of the resistance offered to the flow is different in each of the three coordinate directions. A loss coefficient must be provided in each direction as well as the ratio of free area normal to the coordinate direction to grid area normal to the direction.

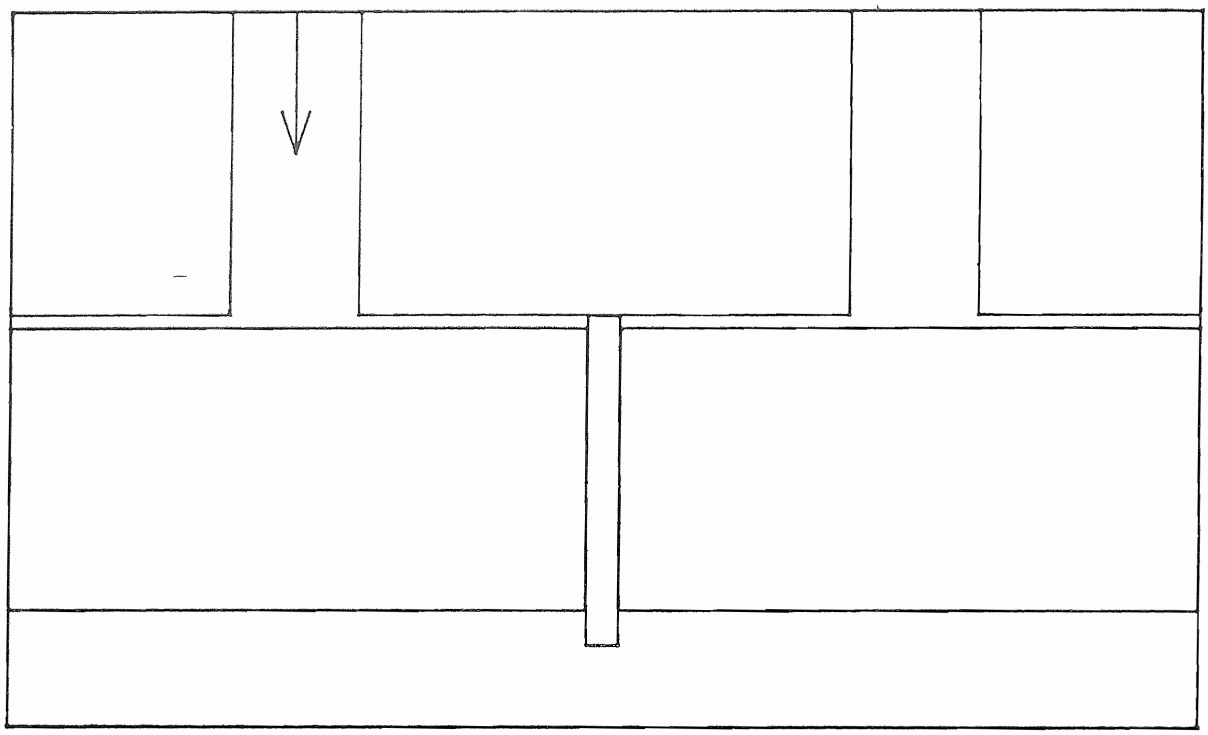

Figure 4. 2D Heat Exchanger: Geometry showing tube bundles and baffle

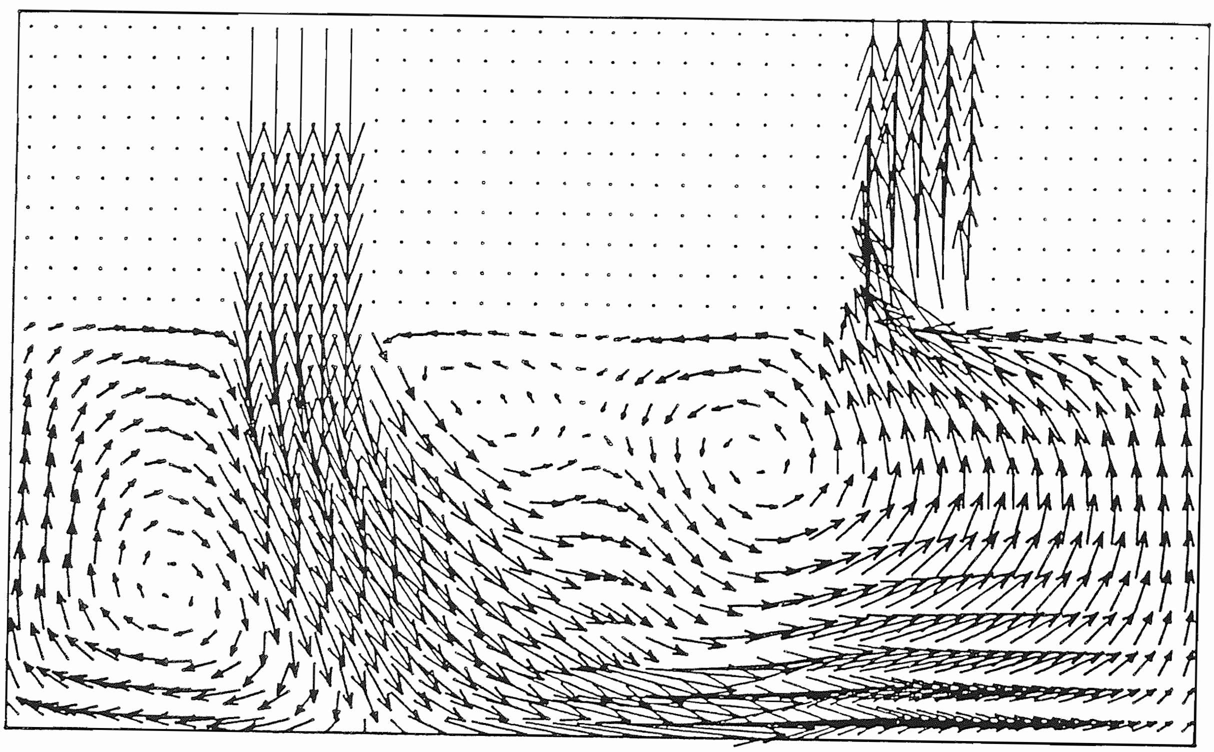

Figure 5. 2D Heat Exchanger: Velocity Vectors; no tubes with baffle (coarse grid)

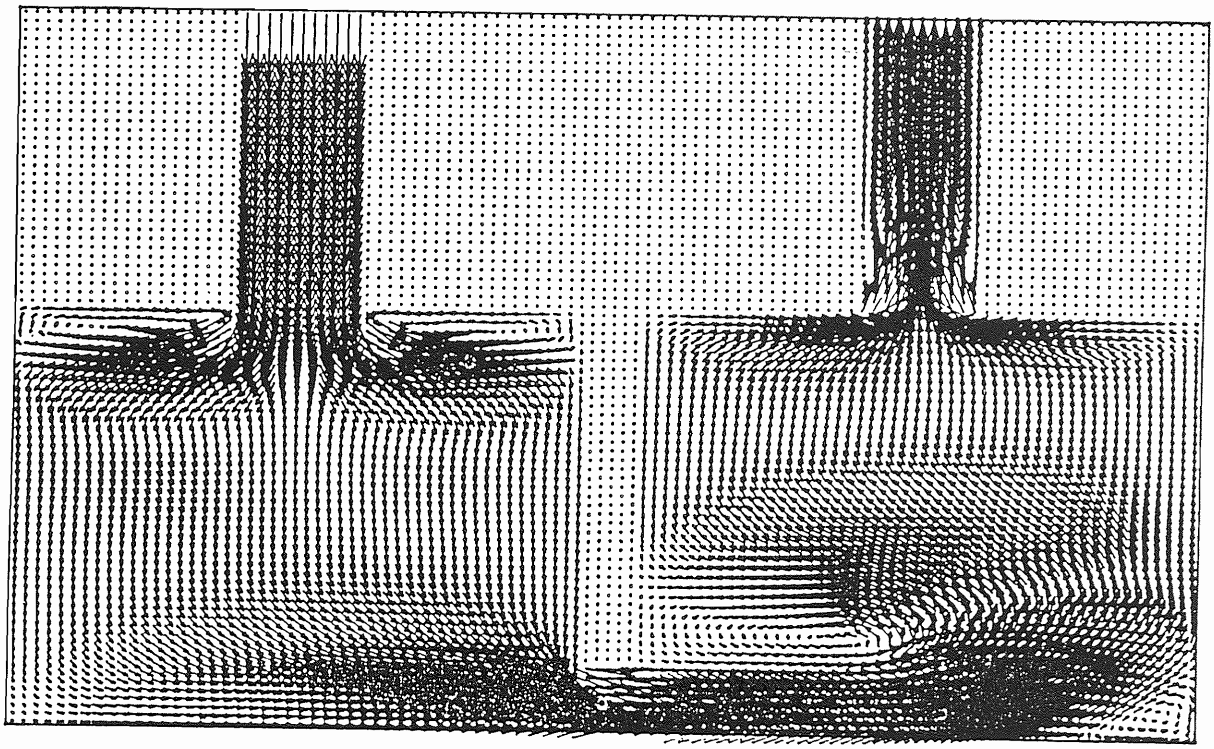

Figure 6. 2D Heat Exchanger: Velocity vectors with tubes (fine grid).

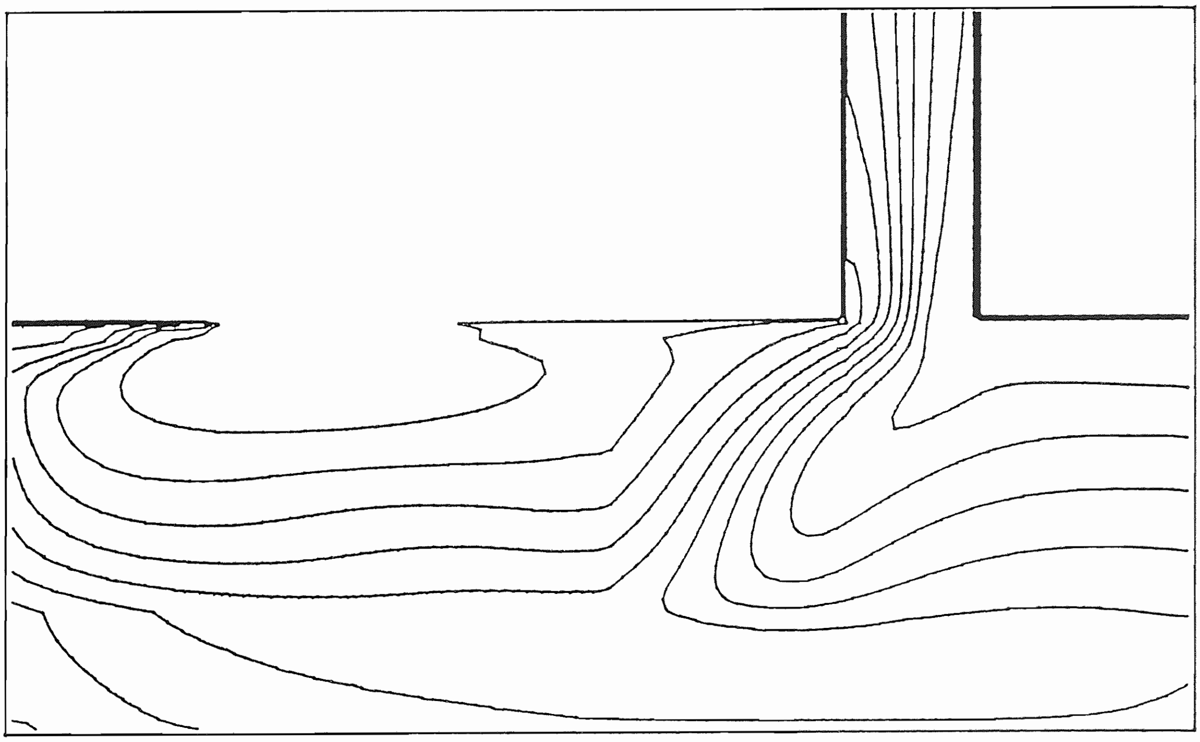

Figure 7. 2D Heat Exchanger: Temperature Distribution

From a consideration of the pressure distribution, turbulent terms, and temperatures distributions important design considerations relative to tube clearances in the baffle plates, tube arrangements (in line or staggered) can be easily illustrated.

## 4. Conclusion

The use of an advanced CFD code, FloSYS, at the undergraduate and junior level for four representative types of problems has been described. Overall the experience has been excellent.

- The students gain confidence in their ability to understand physical phenomena, formulate, and solve realistic problems.

- They are able to explore the influence of changing boundary and initial conditions and see the effects immediately.

- They obtain experience into details of numerical modeling, such as grid size, time steps, etc.

- They are able to gain experience with a high level commercial code which is currently utilized in industry.

## 5. References


[1] FloSYS Users Guide (1989), Flomerics Ltd., Kingston Upon Thames, Surrey England.

[2] FloSYS Reference Manual (1989), Flomerics Ltd., Kingston Upon Thames, Surrey, England.

[3] FloSYS System Manual (1990), Flomerics Ltd., Kingston Upon Thames, Surrey, England.

[4] Kreith, F. and Black, W.Z., Basic Heat Transfer, Harper & Row, New York (1980), p. 94-100.

[5] Kreith, F. and Bohn, M.S., Principles of Heat Transfer, 4th Ed. Harper & Row, New York (1986), pp. 184-197.

[6] Ibid. p. 149-176.

[7] Ibid. p. 190-204, p. 225-231.

[8] Ciofalo, M. and Karayiannis, T.G., Natural convection heat transfer in a partially or completely-partitioned vertical rectangular enclosure, Int. J. Heat Mass Transfer 34, (1991) 1, p. 167-179.

[9] Chen, K.S. and Ko, P.W., Natural convection in a partially divided rectangular enclosure with an opening in the partition plate and isoflux side walls, Int. J. Heat Mass Transfer, 34 (1991) 1, p. 237-246.

[10] Galea, E.R. and Markatos, N.C., The mathematical modeling and computer simulation of fire development in aircraft, Int. J. Heat Mass Transfer 34 (1991) 1, p. 181-197.

[11] Elphick, I.G. and Currie, I.G., Flow distribution measurements in a model heat exchanger, Phase I, (March 1982-110 p.), Phase II (April 1982-31 p.), LDAL, Dept. of Mech. Eng., University of Toronto.

[12] Rhodes, D.B. and Carlucci, L.N., Predicted and measured velocity distributions in a model heat exchanger, AECL-8241, Chalk River Nuclear Labs., Jan. 1984, 14 pages.