Journal of Visualization, Vol. 1, No. 1 (1998) 51-63

Modeling of Complex Flows and Heat Transfer

Chen, C. J.*, Lin, W.*, Haik, Y.*, and Carlson, K. D.*

* Department of Mechanical Engineering, FAMU - FSU College of Engineering, Florida A & M University - Florida State University, 2525 Pottsdamer St., Tallahassee, FL 32310-6046, U.S.A.
E-mail: cjchen@eng.fsu.edu, http://ceb213.eng.fsu.edu

Received 29 July 1997.
Revised 18 November 1997.

Abstract: This paper presents the numerical modeling of complex flows and heat transfer. The Finite Analytic method is used to discretize the transport equations. The diagonal Cartesian method is proposed to model fluid flows and heat transfer over complex geometries. A three-dimensional channel flow with conjugate heat transfer is simulated. By the diagonal Cartesian method and 5-point Finite Analytic scheme, a grooved channel flow and flow in a casting bank at different Reynolds numbers are modeled. Simulations by both the diagonal Cartesian method and the traditional saw-tooth Cartesian method indicates the diagonal Cartesian method improves the modeling of flows, due to the more accurate approximation of complex boundaries. Heat transfer in two-dimensional finned compact heat exchanger is also studied. An improved heat exchanger is proposed based on the numerical prediction of heat transfer.

Keywords: complex flow, Cartesian, diagonal, finite analytic, conjugate heat transfer.

Nomenclature:

Alphabetic

K	Thermal conductivity
L	Characteristic length scale
Nu	Nusselt number ($\partial\theta/\partial n/w$)
P	Dimensionless Pressure (p/U_{ref}^2)
Pe	Peclet number ($Re\ Pr$)
Pr	Prandtl number (ν/α)
Re	Reynolds number ($U_{ref} L / \nu$)
S	Source term
T	Temperature
U	Velocity in the X direction
V	Velocity in the Ydirection
W	Velocity in the Z direction
X	Spatial variable
Y	Spatial variable
Z	Spatial variable
n	Normal direction from the wall
p	Pressure
u	Dimensionless velocity in the x direction (U/U_{ref})
v	Dimensionless velocity in the y direction (V/U_{ref})
w	Dimensionless velocity in the z direction (W/U_{ref})
x	Dimensionless spatial variable (X/L)

y	Dimensionless spatial variable (Y/L)
z	Dimensionless spatial variable (Z/L)

Greek

α	Thermal Diffusivity ($K/\rho C_p$)
Φ	Transport quantity
ν	Kinematic viscosity
ρ	Density

Subscript

bc	Boundary condition
f	Fluid
p	Central node of finite analytic element
ref	Reference
s	Solid
t	First order partial derivative in t
w	Wall
x	First order partial derivative in x
xx	Second order partial derivative in x
y	First order partial derivative in y
yy	Second order partial derivative in y
z	First order partial derivative in z
zz	Second order partial derivative in z

1. Introduction

Due to recent advances in high performance computers, the numerical simulation of fluid flows and heat transfer problems has become common and possible in practical applications. However, the simulation of flows and heat transfer around complex geometries remains a challenging task, since it is difficult to generate a grid system in which the complex boundary conditions can be properly described and the numerical solution is simple, accurate and stable.

The boundary-fitted coordinate system proposed by Thompson (1982) is a popular choice for modeling complex boundaries, since the boundary surface is fitted with a new coordinate line based on the body contour. If the problem has sharp boundaries or complex multi-body systems, it is difficult to achieve automatic grid generation with boundary-fitted coordinates. Overset grids, such as Chimera (Belk, 1995), have great versatility and are especially attractive for multi-body systems with bodies in relative motion. However, concerns have been raised about the accuracy of the interpolation necessary to transfer data between component grids, particularly the lack of conservation at the interfaces of the different grid blocks. Unstructured meshes offer another alternative in numerical grid systems. However, viscous flow analysis using the Reynolds-averaged Navier-Stokes equations are severely hindered for large cases by seemingly insatiable CPU-time and memory requirements, even for steady-state calculations (Mavriplis, 1997).

The Cartesian grid approach is attractive and gaining popularity because of its inherent simplicity and potential for automation. Many studies choose to approximate very complex geometries using only Cartesian coordinate grid lines. This approach is referred to as the saw-tooth Cartesian method in this paper. Currently, this method is widely used for large scale problems in environmental and hydraulic engineering, such as flow modeling in oceans and lakes and heat transfer in engineering devices. One drawback to this method of approximation is that it forms a saw-tooth or stair-like boundary surface, and the boundary will remain rough and full of angles even if the grid is refined.

Another difficulty in simulating flows and heat transfer over complex geometries lies in the proper discretization of the convection-diffusion equation. It should be remarked that the convection term produces an asymmetric phenomenon. In other words, the upstream conditions have a greater influence than the downstream conditions. If a scheme does not represent this phenomenon properly, physically unrealistic solutions will occur. For example, central difference schemes have been found to be stable at low cell Reynolds numbers (Reynolds number based on the local velocity and cell dimension) and unstable at high cell Reynolds numbers, typically larger than a value of two. Chen et al. (1981, 1984, 1995) proposed the Finite Analytic (FA) method to discretize numerically the partial differential equations which govern fluid flow and heat transfer. This is accomplished by dividing the solution domain into small elements and utilizing the local analytic solution in each element to discretize the governing equations into a system of linear algebraic equations which can be solved numerically. The FA scheme provides an automatic upwinding effect of the boundary nodes surrounding an interior node, and it minimizes false diffusion by eliminating the truncation error.

This paper gives an overview of the current state of the FA method. In addition a diagonal Cartesian method is introduced for the modeling of flows with conjugate heat transfer over complex boundaries. The application of the FA method and the diagonal Cartesian method to two and three-dimensional flow and conjugate heat transfer problems involving complex geometries are presented.

2. Formulation of the Problem

The fluid is assumed to be an incompressible Newtonian fluid with constant properties, and there is no heat generation in the solid and fluid domains. With these assumptions, the dimensionless equations which govern laminar fluid flow and conjugate heat transfer are:

$$u_x + v_y + w_z = 0 \tag{1}$$

$$u_t + uu_x + vu_y + wu_z = -P_x + \frac{1}{Re}(u_{xx} + u_{yy} + u_{zz}) \tag{2}$$

$$v_t + uv_x + vv_y + wv_z = -P_y + \frac{1}{Re}(v_{xx} + v_{yy} + v_{zz}) \tag{3}$$

$$w_t + uw_x + vw_y + ww_z = -P_z + \frac{1}{Re}(w_{xx} + w_{yy} + w_{zz}) \tag{4}$$

$$\theta_t + u\theta_x + v\theta_y + w\theta_z = \frac{1}{Pe}(\theta_{xx} + \theta_{yy} + \theta_{zz}) \tag{5}$$

solid domain:

$$\theta_t = \frac{\alpha_s}{\alpha_f}\frac{1}{Pe}(\theta_{xx} + \theta_{yy}) \tag{6}$$

where u, v and w are the dimensionless velocity components in the x, y and z directions respectively, P is the dimensionless pressure and θ is the dimensionless temperature. Re is the Reynolds number and Pe is the Peclet number. The subscripts denotes the partial derivative variable.

The variables for time, space, velocity, pressure and temperature have been non-dimensionalized by the characteristic time L/U_{ref}, characteristic length L, characteristic velocity U_{ref}, characteristic dynamics pressure U^2_{ref} and characteristic temperature difference ΔT. The Reynolds number is defined as $Re=U_{ref}L/\nu$, where ν is the kinematics viscosity of the fluid. The Peclet number is defined as $Pe=RePr$, where Pr is the Prandtl number of the fluid. α_s and α_s are the thermal diffusivities of the solid and fluid, respectively. The Dirichlet velocity boundary condition is

$$\vec{V} = \vec{V}_{bc} \tag{7}$$

where v_{bc} is the dimensionless boundary velocity vector and it is zero if the surface is stationary. The temperature boundary condition at the computational (outer) boundary can be specified by either a given temperature or a given heat flux. The temperature boundary condition at the solid/fluid interface (inner boundary) is the conservation of energy flux, given as

$$\theta_{|\text{fluid}} = \frac{k_s}{k_f}(\theta_n)_{|\text{solid}} \tag{8}$$

where k_s and k_f are the fluid and solid thermal conductivities, and n is the direction normal to the interface.

Enforcement of this interface temperature boundary condition provides the necessary coupling between the solid and fluid domains to simulate conjugate heat transfer.

3. Finite Analytic Method

The Finite Analytic (FA) method was first proposed for two-dimensional fluid flow and heat transfer problems (Chen et al., 1981, 1984). It was further developed by Chen et al. (1993, 1995) to solve complex three-dimensional problems, and was also applied to fluid flows over complex geometries (Bravo, 1991; Lin, 1997).

Consider the unsteady, two-dimensional convective transport equation

$$R\,(\Phi_t + u\Phi_x + v\Phi_y) = \Phi_{xx} + \Phi_{yy} + S \qquad (9)$$

where Φ can be any transport quantity such as velocity, vorticity or temperature, R is the Reynolds number or a similar dimensionless parameter, and S is the source term. In the FA method, the computational domain is first decomposed into many small elements, as shown in Fig. 1. Most elements are regular, but some are irregular because they are adjacent to complex boundaries. For regular elements, the governing equations can be discretized at the central node P using the eight neighboring nodes. For irregular elements, the governing equations can be discretized using the four neighboring nodes, as shown in Fig. 1. The governing equation (9) is linearized within each element by setting u and v equal to their respective values at the interior node P, or to the average velocities over the small elemental area. The unsteady term can be approximated by a backward Euler difference scheme. The linearized elliptic equations can be solved analytically by the method of separation of variables on each small element. The corresponding boundary conditions are approximated by a quadratic interpolation function or a piecewise exponential interpolation function. The coefficients associated with these functions can be expressed as a function of the nodal values on the element boundaries. Therefore, one can express the value of the dependent variable Φ_p at the interior node P in terms of the values of the neighboring (nb) nodes Φ_{nb} as

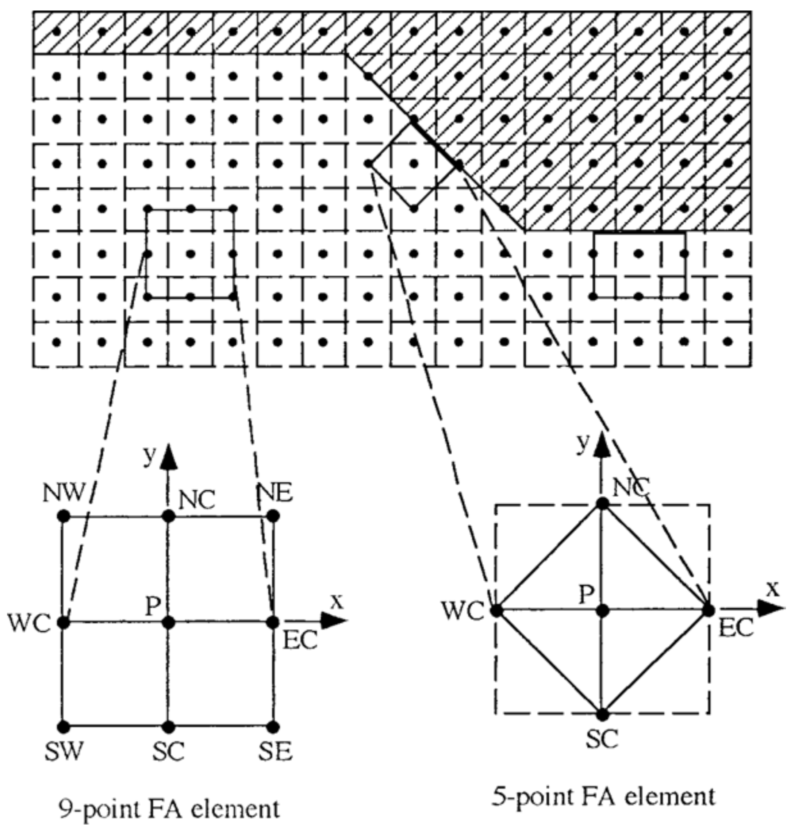

Fig. 1. FA elements for two-dimensional problems.

$$\Phi_p = \sum_{nb=1}^{m} c_{nb}\Phi_{nb} + Rc_p\Phi_p^{n-1}/\Delta t - c_p s_p \tag{10}$$

where m=8 for regular elements and m=4 for irregular elements. c_{nb} and c_p are the FA coefficients. Details of the derivation are given in Chen et al. (1981, 1984, 1987) and are available at http://ceb213.eng.fsu.edu.

Governing equations of three-dimensional flows, except the continuity equation, can be written as the following general transport equation

$$D\Phi_t + 2A\Phi_x + 2B\Phi_y + 2C\Phi_z = \Phi_{xx} + \Phi_{yy} + \Phi_{zz} + S \tag{11}$$

where $2A = RU$, $2B = RV$, $2C = RW$, $D = R$ and S is the source term. Similar to the two-dimensional problem, the above equation is locally linearized over each small element as shown in Fig. 2. Here A, B, C, D and S are approximated by their values at the node P or by values averaged over the small elemental volume. Hence, equation (11) becomes

$$D_p\Phi_t + 2A_p\Phi_x + 2B_p\Phi_y + 2C_p\Phi_z = \Phi_{xx} + \Phi_{yy} + \Phi_{zz} + S \tag{12}$$

Based on the superposition of three two-dimensional flows, the following 19-point FA formula for equation (12) is obtained, as shown in Fig.2.

$$\Phi_p = \sum_{nb=1}^{m} F_{nb}^{x}\Phi_{nb}^{x} + \sum F_{nb}^{y}\Phi_{nb}^{y} + \sum_{nb=1}^{m} F_{nb}^{z}\Phi_{nb}^{z} + F_p g_p \tag{13}$$

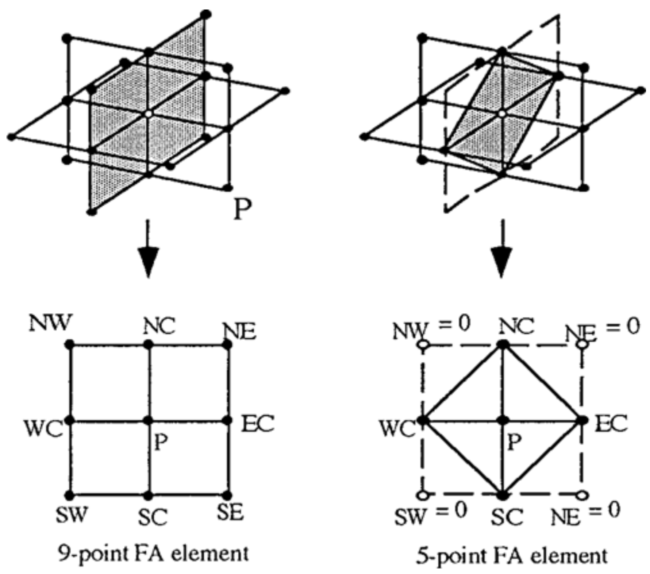

Fig. 2. FA elements for three dimensional problems.

Further details can be found in Chen et al. (1995).

4. Diagonal Cartesian Method and Other Numerical Approaches

The diagonal Cartesian method of computing flow fields involving complex geometries using a Cartesian grid was proposed by Chen et al. (1993, 1997). In this method, complex boundaries are approximated by both Cartesian grid lines and diagonal line segments in Cartesian coordinates, as shown in Fig. 3. A structured Cartesian grid is employed in the approximation for the sake of simplicity. The primary goal of this method is to achieve problem independence and automation of grid generation for complex boundaries.

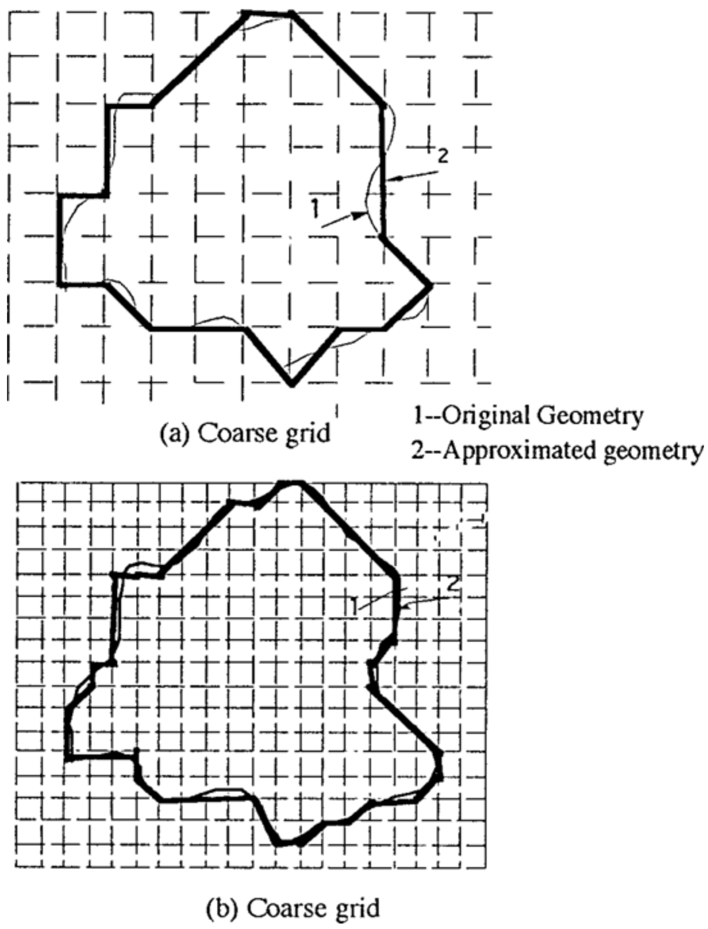

Fig. 3. Diagonal Cartesian method.

In this paper, a non-staggered grid is used due to the existence of complex geometries. In order to eliminate the pressure oscillation, the mass conservation is enforced on a smaller control volume so that the pressure values at one node will be coupled to the neighboring nodes. The pressure boundary conditions are avoided by using information from the velocity boundary conditions. The boundary node and ghost boundary node methods on cell-centered nodes of a non-staggered grid proposed by Lin, et al. (1997) are used in the simulation of flows with conjugate heat transfer over complex boundaries. A PISO (Pressure Implicit with Splitting of Operators)-like algorithm is used to obtain an accurate time marching solution for fluid flows with conjugate heat transfer. The overall calculation procedure is:

(1) Specify the initial conditions for the velocity and pressure fields.
(2) Solve the momentum equations (2) to (4) implicitly with an ADI (Alternating Direction Implicit) algorithm.
(3) Return to step (2) until convergent velocities are obtained.
(4) Solve the pressure equation, which is derived from the continuity equation, implicitly with the ADI algorithm.
(5) Update the velocities explicitly from the momentum equations (2) to (4).
(6) Return to step 4 until the pressure and velocity fields are converged.
(7) Return to step 2 for the next time step.

5. Three-Dimensional Channel Flow with Conjugate Heat Transfer

Heat exchangers are used extensively in industry. The objective of these devices is to transfer heat at an optimum rate. Finned heat exchangers are much more compact and perform better than those with tubes. The enhanced performance of the heat exchanger is due to the increased heat transfer area, a disrupted thick boundary layer and cross-stream mixing.

In the present application, the FA method is used to simulate three-dimensional fluid flow and heat transfer for a fin placed inside a square duct. The fin is placed in the square duct from one wall to the opposite wall. All the walls of the duct are heated and maintained at a constant temperature T_w. A cold fluid, with temperature T_o, enters the duct with a fully-developed velocity profile. Energy is transferred by convection from the walls of the duct to the fluid as well as from the fin to the fluid. In addition, heat is transferred along the fin by conduction.

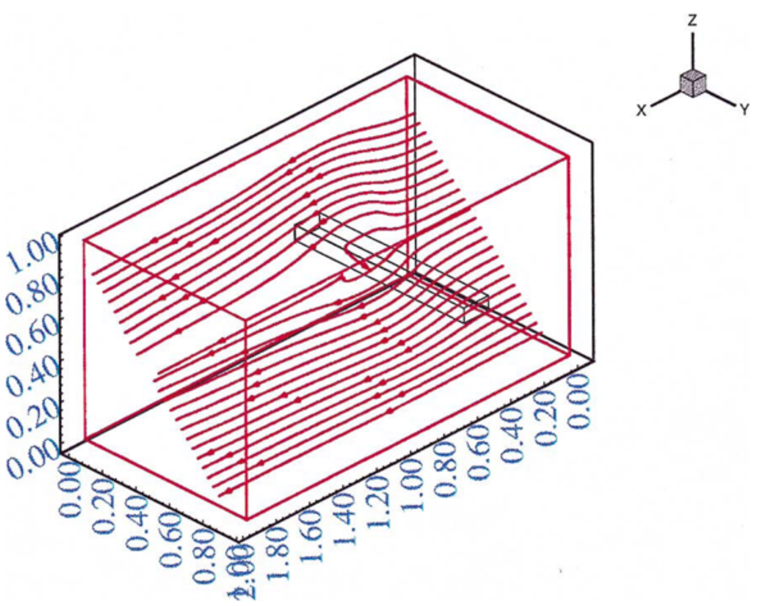

Fig. 4. Stream function for three dimensional finned duct flow.

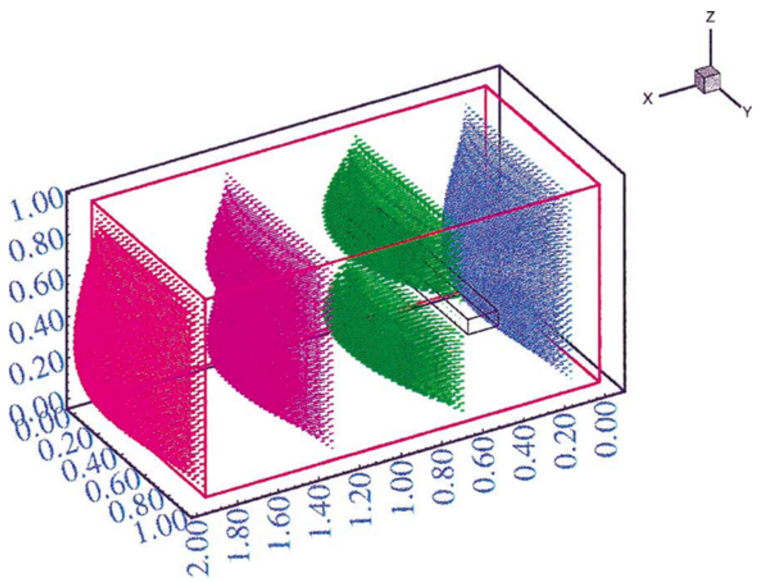

Fig. 5. Velocity profile for three dimensional finned duct flow.

The numerical solution for this problem is calculated using the program FANS-3D developed by Bravo (1991). The code utilizes the 19-point FA method (Chen et al., 1995). The computational grid is 26 × 26 × 26. The Reynolds number is 100 based on the inlet height and maximum inlet velocity. The Prandtl number, the conductivity of the solid (brass) and the conductivity for the fluid (water) are 1.0, 111 W/m-K and 0.6 W/m- K respectively.

The results of the computations are shown in Figs. 4 to 6. Figure 4 shows the streamlines. Initially the bulk of the fluid moves in the center of the duct. However, on reaching the fin, the fluid moves around the fin and enhances the thermal mixing. The flow recovers its original pattern downstream of the fin. Figures 5 and 6 provide velocity and temperature profiles, respectively, at three different locations in the computational domain.

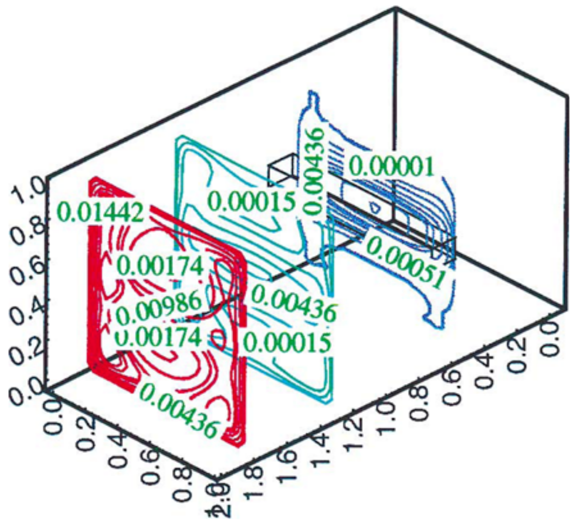

Fig. 6. Temperature field for three dimensional finned duct flow.

6. Modeling of Flows over Complex Boundaries

The application of the diagonal Cartesian method to the modeling of flows in a grooved channel and casting bank are presented. Figures 7 and 8 show the approximation of a grooved channel in Cartesian coordinates. It is seen that the diagonal method gives very good geometry approximation of the original grooved channel, while the saw-tooth Cartesian method fails to accurately approximate the sharp-edge grooved walls. The solutions of a grooved channel flows at Reynolds number of 10 and 600 are shown in Fig. 9. The figure reveals that wave-like stream lines, due to the wave-like grooved walls, are more pronounced at lower Reynolds numbers. At a higher Reynolds number, such as 600, the flow in the straight channel seems almost unaffected by the grooved walls. Physically at low Reynolds number, the diffusion term or elliptic operator dominates the flow and the flow behaves like a creeping flow. The wavy stream lines may diffuse to the upper walls. When Reynolds number increases, the

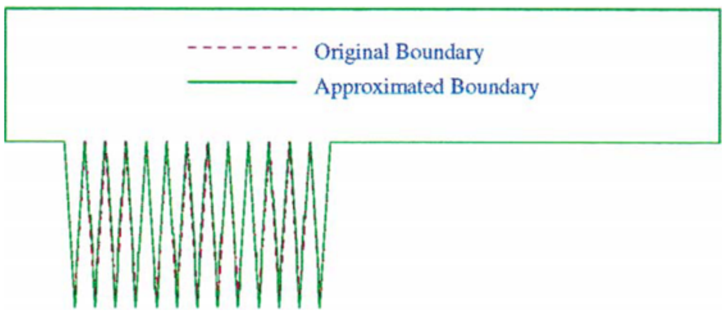

Fig. 7. Diagonal Cartesian approximation of a grooved channel.

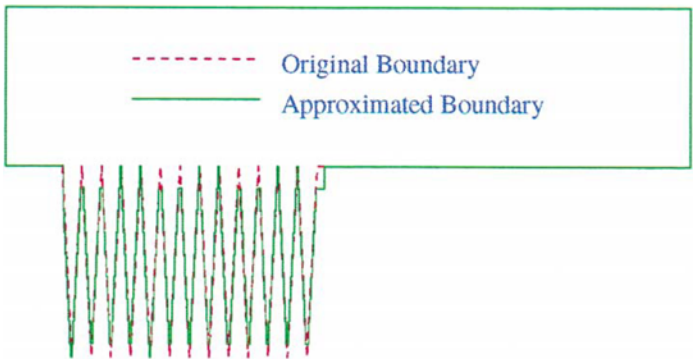

Fig. 8. Saw-tooth Cartesian approximation of a grooved channel.

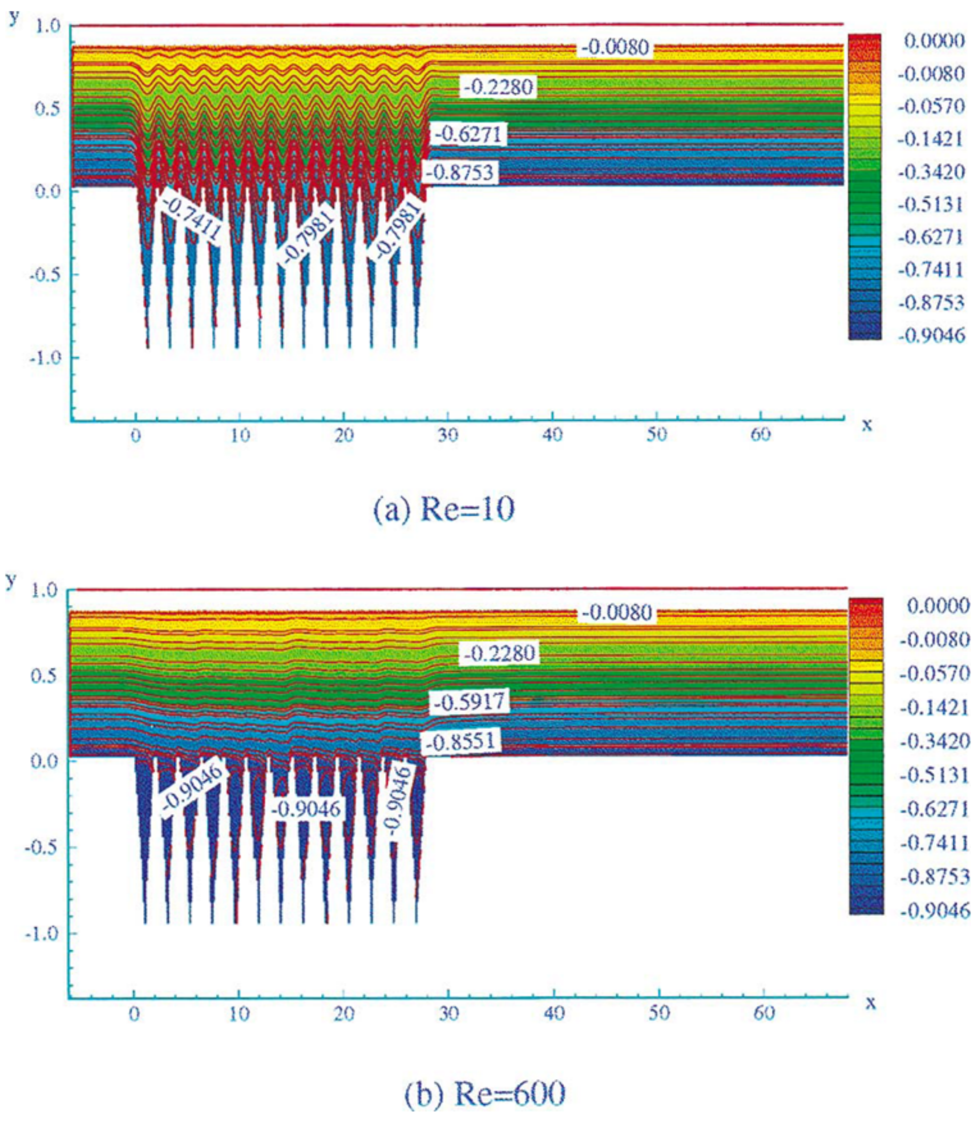

Fig. 9. Stream functions at Reynolds numbers of 10 and 600 for flow in the grooved channel.

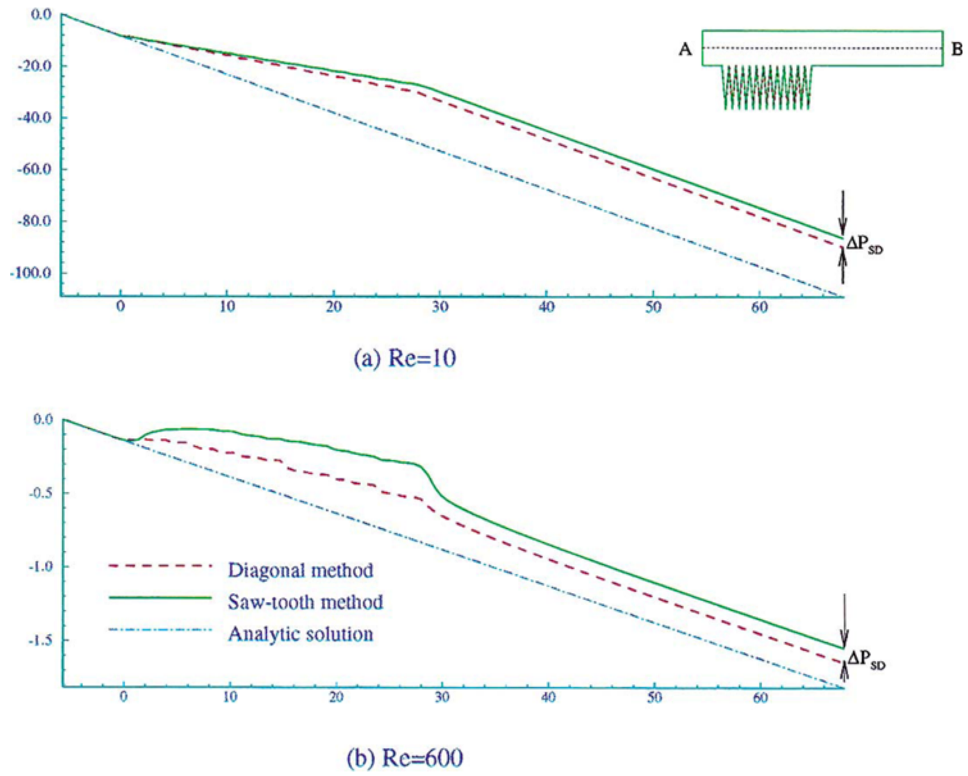

Fig. 10. Pressure distributions along the central line AB for flow in the grooved channel.

diffusion terms are no longer dominant, instead, the importance of convection terms increases. The prediction of pressure by the saw-tooth Cartesian method are given in Fig. 10. It is shown that the saw-tooth method underpredicts the pressure drop along the central line AB for Reynolds of 10, 100 and 600. The differences of pressure coefficients between the two methods, ΔP_{SD}, range from 3.9% at Re=100 to 6.02% at Re=600. This is mainly due to the failure of approximating the sharp edges of grooved walls by the saw-tooth method.

Figures 11 to 12 show the flows in a casting boundary with a Reynolds number of 100. It is seen that large velocities occur in the narrow passage and velocities in the upper right corner are much smaller. Vortices appear due to the sharp edges of the boundaries, as shown in Fig. 11. Figure 12 shows the stream function from the saw-tooth method. It is found that there is some difference between the predictions by the diagonal and saw-tooth

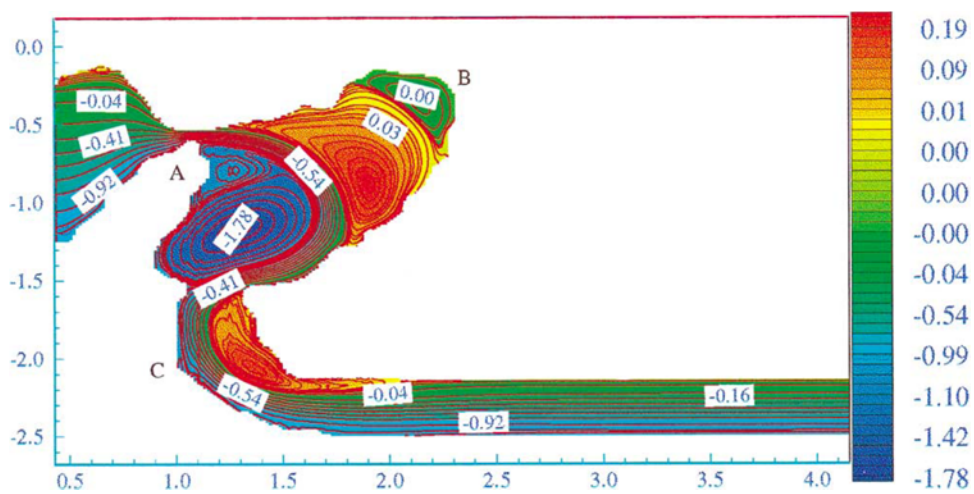

Fig. 11. Stream function of a casting flow by diagonal method (Re =100).

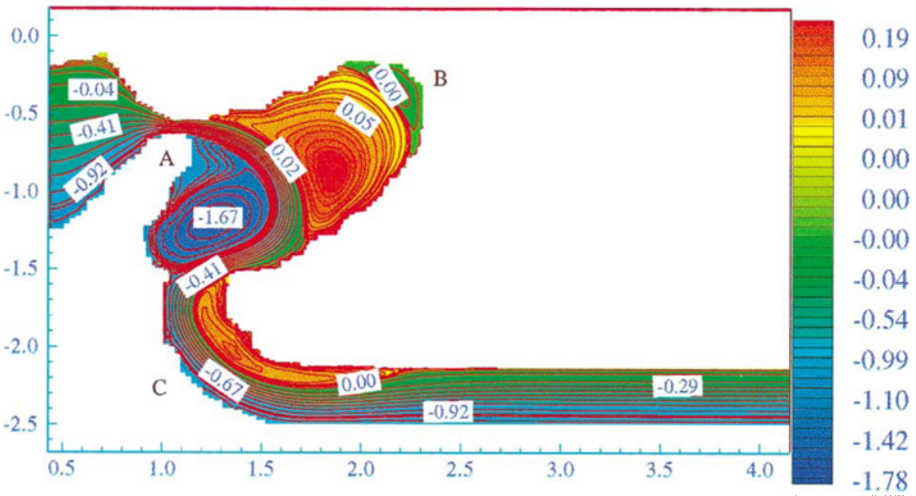

Fig. 12. Stream function of a casting flow by saw-tooth method (*Re* =100).

methods, especially in the separation regions. This indicates that the diagonal Cartesian method can improve the modeling of fluid flows over the saw-tooth method, since it approximates the original boundaries more accurately (Lin, 1997).

7. Conjugate Heat Transfer in a Compact Heat Exchanger

Heat transfer in two-dimensional finned compact heat exchangers has been studied by several researchers (Kelkar and Patankar, 1987; Bravo, 1991). These studies, however, neglected conduction and assumed that the wall and fin temperatures were constant. The present work begins by investigating flow and heat transfer in the heat exchanger shown in Figs. 13 and 14, and includes conduction in the walls and fins. The outer wall temperature of the heat exchanger is held constant at a dimensionless temperature of 1.0, and the inlet has a parabolic velocity profile with a maximum dimensionless velocity of 1.0 and a dimensionless temperature of 0.0. The problem was solved for a Reynolds number of 200, based on the maximum inlet velocity and inlet height. The fluid is water (Pr = 4.0) and the heat exchanger is made of aluminum, giving a solid/fluid conductivity ratio of K = 376. The outlet channel extends 10L downstream of the heat exchanger, and the outlet boundary conditions are taken as fully-developed at this location. The problem was solved on a 196×74 grid.

A plot of the streamlines for this problem is shown in Fig. 13, and the isothermal contour plot is given in Fig. 14. Notice that the temperature of the fins does not remain constant at 1.0, but instead drops below 0.8 in both fins. Thus assuming constant fin temperature, this problem will lead to an overestimation of the heat transfer to the fluid. The average Nusselt number for the problem shown was computed as 7.86, where the average Nusselt number is based on the inner wall surface area of the heat exchanger without fins. This problem was also simulated holding the fin temperature constant at 1.0, yielding an average Nusselt number of 8.78 (a 12% overestimation).

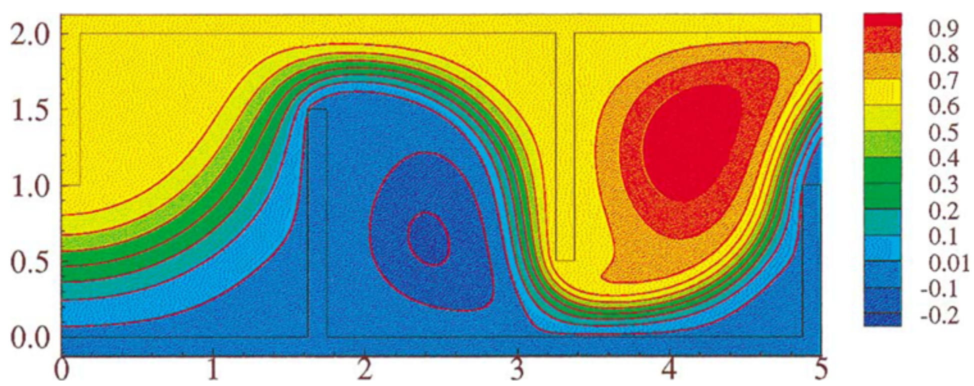

Fig. 13. Streamlines, *Re* =200 for a two dimensional finned heat exchanger.

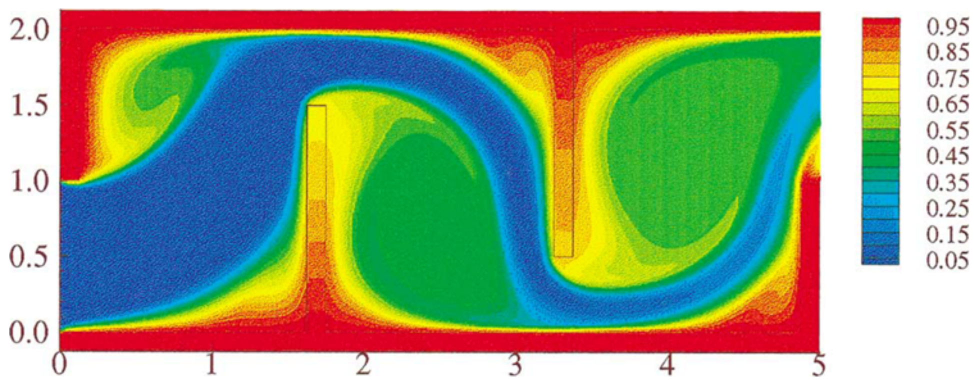

Fig. 14. Isotherms at *Re* =200 and *Pr* =4.0 for a two dimensional finned heat exchanger.

As a basis for comparison, the problem was also solved without fins, giving an average Nusselt number of 3.96. So the addition of fins increases the heat transfer capability by 98%. The price which must be paid for this heat transfer increase is an increase in the pressure coefficient. The heat exchanger without fins experiences a dimensions pressure coefficient of 0.11. The addition of fins increases the pressure drop to 6.08. Thus, the addition of fins to this problem doubles the heat transfer capability, but it increases the pressure coefficient by a factor of fifty. In space-limited applications, however, this penalty may be paid in order to achieve a necessary augmentation in the heat transfer.

In an attempt to increase the heat transfer by maintaining a higher fin temperature, the fin shape can be altered. The streamlines and isotherms for this geometry are shown in Figs. 15 and 16, respectively. Notice in Fig.

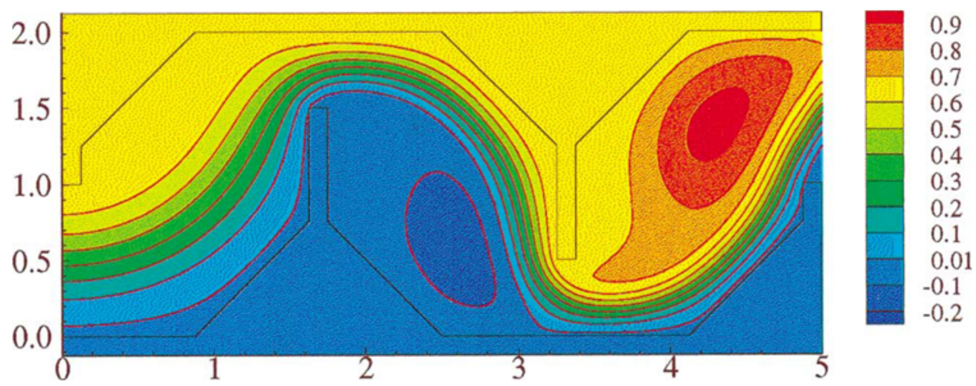

Fig. 15. Streamlines at *Re* =200 for a two dimensional finned heat exchanger.

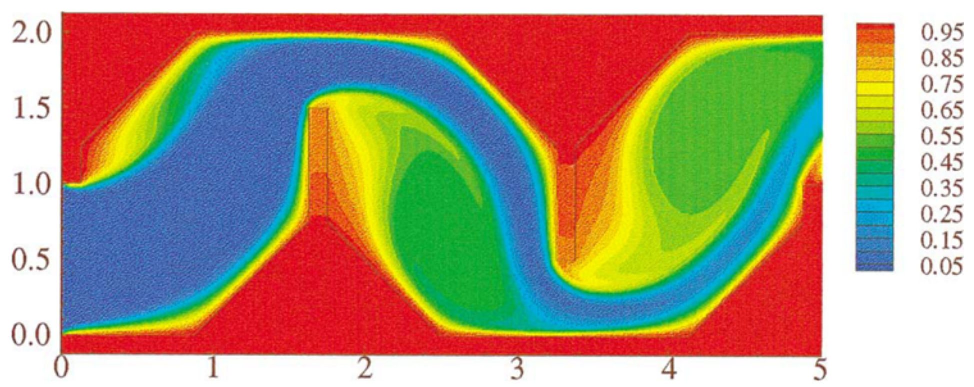

Fig. 16. Isotherms at *Re* =200 and *Pr*=4.0 for a two dimensional finned heat exchanger.

16 that the fin temperatures are indeed higher than in the previous geometry. The average Nusselt number for this case is 8.43, which is 113% larger than the case without fins. The pressure coefficient is 6.03. The complex fin geometry, then, increases the heat transfer another 15% over the rectangular fin geometry, while slightly decreasing the pressure coefficient.

Acknowledgments

We would like to thank SCRI (Supercomputer Computations Research Institute) at Florida State University for the use of their computer facilities.

References

Belk, D. M., "The Role of Overset Grids in the Development of the General Purpose CFD Code," In Proceedings of the Surface Modeling, Grid Generation and Related Issues in Computational Fluid Dynamics Workshop, page p.193, NASA Lewis Research Center, Cleveland, OH, May 1995. NASA Conference Publication 3291.

Bravo, R. H., "Development of the Three Dimensional Finite Analytic Method for Simulation of Fluid Flow and Conjugate Heat Transfer," Ph.D. thesis, University of Iowa, December 1991.

Chen, C. J., Naseri-Neshat H. and Ho, K. S., "Finite Analytic Numerical Solution of Heat Transfer in Two-Dimensional Cavity Flow", Numerical Heat Transfer, Vol. 4, pp. 179-197, 1981.

Chen C. J. and Chen, H. C., "Development of Finite Analytic Numerical Method for Unsteady Two-Dimensional Navier-Stokes Equations", J. of Computational Physics, Vol. 53, No. 2, pp. 209-226, 1984.

Chen, C. J., Bravo R. H., Chen H. C. and Xu Z., "Accurate Discretization of Incompressible Three-Dimensional Navier-Stokes Equations", J. of Numerical Heat Transfer, Vol. 27 (4) , pp. 371-392, 1995.

Carlson, K. D., Tien H. C. and Chen, C. J., " Finite Analytical Numerical Simulation of Fluid Dynamics Problems Involving Irregular Geometries", Proceeding of the 5th International Symposium on Refined Flow Modeling and Turbulence Measurements, Paris, France, pp. 199-206, September 1993.

Lin W. L., "Diagonal Cartesian Method for Modeling of Complex Boundaries," Ph.D. Thesis, Department of Mechanical Engineering, Florida State University, 1997.

Lin, W. L., Carlson K. and Chen, C. J., "Pressure Boundary Conditions of Incompressible Flows with Conjugate Heat Transfer on Non-staggered Grids, Part I: Methods," To appear in Journal of Numerical Heat Transfer, 1997.

Mavriplis, D. J., "Unstructured Grid Techniques," Annu. Rev. Fluid Mech., 29-473-514, 1997.

Kelkar, K. M. and Patankar, S. V., "Numerical Prediction of Flow and Heat Transfer in a Parallel Plate Channel With Staggered Fins," Journal of Heat Transfer, Vol. 109, pp. 25-30, 1987.

Thompson, J. F., Warsi, Z. U. A. and Mastin, C. W., "Boundary-Fitted Coordinate System for Numerical Solution of Partial Deferential Equations -- A Review," J. of Computational Physics, Vol. 47, pp. 1-108, 1982.