Computations of laminar and turbulent mixed convection in a
driven cavity using pseudo-compressibility approach

L. Agrawal, J.C. Mandal *, A.G. Marathe

Department of Aerospace Engineering, IIT Bombay, Mumbai 400076, India

Abstract

A driven cavity, for two different thermal boundary conditions, has been chosen as a test case to establish
the suitability of pseudo-compressibility algorithm for mixed convection flow problems for both laminar
and turbulent flow situations. Here shear stress transport eddy viscosity model has been modified to include
the buoyancy effects. The flow is computed inside the cavity under the influence of varying buoyancy and
inertia forces. The computed results demonstrate the ability of the pseudo-compressibility approach and
shear stress transport model to solve complex heat transfer problems.

Keywords: Pseudo-compressibility; SST turbulence model; Mixed convection; Driven cavity

1. Introduction

Driven cavity for isothermal flow constitutes an appealing benchmark problem in numerical
exercise. The geometry is straightforward and the boundary conditions are regular. The intro-
duction of a temperature gradient into the cavity boundary, i.e. thermal non-homogeneity, gives
rise to buoyancy and this, in turn, impacts upon the coupled field of velocity and temperature in
the cavity. An understanding of this mixed convective flow is of value from the standpoint of basic
fluid dynamics as well as in practical engineering applications, such as simulating lubricating
groove between sliding plates, heat exchange between container and laterally flowing stream. A
survey of the relevant literature, however, reveals that studies of mixed convection with a closed cavity are relatively scarce [1–3].

Numerically such problems can be computed by solving incompressible flow equations. The primary difficulty in computing incompressible flows is in finding a satisfactory way to link changes in the velocity field to changes in the pressure field. This interaction must be accomplished in such a manner as to ensure that the divergence of the velocity field vanishes. In two-dimensional (2D) flows, this difficulty is readily overcome using stream function-vorticity formulation. Unfortunately, the stream function-vorticity formulation is less effective and becomes complex for 3D flows, and extension to three dimensions have generally been directed towards alternative formulations.

Among commonly used methods for handling this velocity pressure coupling for 3D problems are pressure based method and the pseudo-compressibility method. The pressure based methods, although widely used in the industry, are complex computationally and in the treatment of boundary conditions [4]. On the other hand, the pseudo-compressibility method, which is originally proposed by Chorin [5], has been proven to be efficient and robust in solving 3D problems [4,6].

In the pseudo-compressibility method an artificial compressibility term is introduced in the continuity equation, which makes the system of equations strongly coupled and hyperbolic. Due to this direct coupling between the governing equations, the pseudo-compressibility method has been found to give better convergence than the pressure based method [4]. This method has been extensively used for wide variety of isothermal flows [7] due to its numerous advantages; namely, the ease with which it can be expressed in body fitted coordinates, the capability for using centered differences for inviscid terms, the use of identical procedure for convective and diffusive terms, and the ease with which physically meaningful boundary conditions may be imposed [8]. This approach, which is widely popular for isothermal flow computations, has been successfully extended and validated for natural convection problems by the present authors [7].

In the present work suitability of pseudo-compressibility approach to compute mixed convection problems has been investigated. The turbulence is modeled using shear stress transport (SST) model [9,10]. This model has been shown to perform superior in terms of performance and model sensitivity compared to other ‘two equation’ models [11,12], for isothermal flow situations. In this work this model has been modified and extended to compute heat transfer problems. To demonstrate applicability of pseudo-compressibility and SST model flow inside a driven cavity under the influence of two different temperature gradients have been computed for laminar and turbulent flow situations.

2. Mathematical model

The governing equations of the flow considered are non-linear time dependent incompressible Reynolds averaged Navier–Stokes equations. Neglecting the adiabatic temperature increase due to friction, the equations governing the flow with constant properties may be written in pseudo-compressibility approach as

\[
\frac{\partial \rho}{\partial t} + \beta_{ps} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) = 0
\]  

(1)
\[
\frac{\partial u}{\partial t} + \frac{\partial (u^2 + p + \frac{\kappa}{\kappa^*} k)}{\partial x} + \frac{\partial u w}{\partial y} = \frac{\partial}{\partial x} \left[ (v + v_t) \left( \frac{\partial u}{\partial x} + \frac{\partial u}{\partial x} + \frac{\partial}{\partial y} \right) \right] + \frac{\partial}{\partial y} \left[ (v + v_t) \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial y} \right) \right]
\]

(2)

\[
\frac{\partial v}{\partial t} + \frac{\partial u w}{\partial x} + \frac{\partial (v^2 + p + \frac{\kappa}{\kappa^*} k)}{\partial y} = \frac{\partial}{\partial x} \left[ (v + v_t) \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ (v + v_t) \left( \frac{\partial v}{\partial y} + \frac{\partial v}{\partial y} \right) \right] + g_t \beta(T - T_{ref})
\]

(3)

\[
\frac{\partial T}{\partial t} + \frac{\partial u T}{\partial x} + \frac{\partial v T}{\partial y} = \frac{\partial}{\partial x} \left[ (v/Pr + v_t \sigma_T) \frac{\partial T}{\partial x} \right] + \frac{\partial}{\partial y} \left[ (v/Pr + v_t \sigma_T) \frac{\partial T}{\partial y} \right]
\]

(4)

where \( u \) and \( v \) are Cartesian components of velocity, \( p \) is modified pressure (normalized with density) and \( T \) is temperature. Term \( \beta \) is coefficient of thermal expansion, \( g \) is gravitational constant and \( T_{ref} \) is reference temperature. \( \beta_{ps} \) is an arbitrary real positive parameter, which represents the artificial compressibility. This modified continuity equation does not exhibit any physical meaning until steady state is reached. However, at steady state this equation is identical to continuity equation for incompressible flow. \( v \) is kinematic viscosity and \( Pr \) is Prandtl number of fluid. Term \( v_t \) represents turbulent kinematic viscosity and \( \sigma_T \) is inverse of turbulent Prandtl number of the flow. The terms \( v_t \) along with turbulent kinetic energy, \( k \), go to zero for laminar flow. The methodology to evaluate these turbulent terms is discussed next.

2.1. Turbulence model

In the present study \( \kappa-\omega \) based SST eddy viscosity model [9,10] has been used. This SST model, first proposed by Menter [9], is a two layer turbulence model. It works as \( \kappa-\omega \) model in sublayer and logarithmic part of the boundary layer and as \( \kappa-\epsilon \) in wake region. The rationale behind the choice is following.

The \( \kappa-\omega \) model [13] is the model of choice in the sublayer of the boundary layer. Unlike any other two-equation model the \( \kappa-\omega \) model does not involve damping functions and allows Dirichlet boundary conditions to be specified. It is also considered to be numerically stable. It has been shown [13–15] that the behaviour of \( \kappa-\omega \) model in the logarithmic part of boundary layer is superior to \( \kappa-\epsilon \) model in adverse pressure gradient flows and in compressible flows. In the wake region of boundary layer, the \( \kappa-\omega \) model has been abandoned in favor of the \( \kappa-\epsilon \) model. The \( \kappa-\omega \) model has a very strong sensitivity to the free stream values specified for \( \omega \) outside the boundary layer, while \( \kappa-\epsilon \) model does not suffer from this deficiency. Finally, in free shear layers away from surfaces, the standard \( \kappa-\epsilon \) model is utilized. There does not seem to be a model that accurately predicts all free shear flows (wake, jet, mixing layer) and \( \kappa-\epsilon \) seems to be fair compromise.

To achieve the desired features in different region, the \( \kappa-\epsilon \) model (transformed in \( \kappa-\omega \) formulation) is multiplied by a blending function \( (1 - B_1) \) and added to the \( \kappa-\omega \) model multiplied by \( B_1 \). The blending function \( B_1 \) is designed to be one in the sublayer and logarithmic region of the boundary layer and gradually goes to zero in the wake region.
The transport equations of $k$ and $\omega$, for this two layer SST model, can be written as

$$\frac{Dk}{Dt} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta' \omega k + \frac{\partial}{\partial x_j} \left( (\nu + \sigma_k v_i) \frac{\partial k}{\partial x_j} \right)$$

and

$$\frac{D\omega}{Dt} = \frac{\gamma}{\nu} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \omega^2 + \frac{\partial}{\partial x_j} \left( (\nu + \sigma_\omega v_i) \frac{\partial \omega}{\partial x_j} \right) + 2(1 - B_1)\sigma_{o2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}.$$

The model constants are calculated as follows:

Let $\phi_1$ represent any constant in the original $k-\omega$ model (e.g. $\sigma_{k1}, \sigma_{o1}, \ldots$), $\phi_2$ any constant in the transformed $k-\epsilon$ model (e.g. $\sigma_{k2}, \sigma_{o2}, \ldots$) and $\phi$ the corresponding constant of the SST model ($\sigma_k, \sigma_\omega, \ldots$), then the relationship between them is

$$\phi = B_1\phi_1 + (1 - B_1)\phi_2$$

The turbulent viscosity is defined as

$$\nu_1 = \min \left( \frac{k}{\omega}; \frac{a_1 k}{\Omega B_2} \right)$$

where $B_2$ is function which is unity for boundary layer flows and zero for free shear layers, and $\Omega$ is absolute value of vorticity.

The model constants are

$$\beta'_1 = 0.09; \quad \sigma_{k1} = 0.85; \quad \kappa = 0.41; \quad \gamma_1 = 0.55; \quad \sigma_{o1} = 0.500; \quad \beta_1 = \left( \gamma_1 + \frac{\sigma_{o1} k^2}{\sqrt{\beta'_1}} \right) \beta'_1$$

and

$$\beta'_2 = 0.09; \quad \sigma_{k2} = 1.00; \quad \kappa = 0.41; \quad \gamma_1 = 0.44; \quad \sigma_{o1} = 0.857; \quad \beta_1 = \left( \gamma_2 + \frac{\sigma_{o2} k^2}{\sqrt{\beta'_2}} \right) \beta'_2$$

The blending function $B_1$ is defined as

$$B_1 = \tanh(\text{arg}_1^1); \quad \text{where} \quad \text{arg}_1 = \min \left[ \max \left( \frac{\sqrt{k}}{0.09 \omega y}; \frac{500 \mu}{y^2 \omega} \right) ; \frac{4 \rho \sigma_{o2} k}{CD_{kw} y^2} \right]$$

$y$ is the distance from the nearest wall and

$$CD_{kw} = \max \left( 2 \rho \sigma_{o2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} ; 10^{-20} \right)$$

and the function $B_2$ is defined as:

$$B_2 = \tanh(\text{arg}_2^2); \quad \text{where} \quad \text{arg}_2 = \max \left( \frac{2 \sqrt{k}}{0.09 \omega y}; \frac{500 \mu}{y^2 \omega} \right)$$

This model has been shown to perform superior in terms of model performance evaluation and model sensitivity evaluation compared to other two equation models [11,12]. In this work source
terms of turbulent transport quantities (i.e. of $k$ and $\omega$ equations) have been modified to take account of buoyancy driven turbulence [16–18]. This modification allows the model to be used for buoyant flows involving heat transfer.

The above Eqs. (1)–(4) for mean flow and Eqs. (5) and (6) for turbulent quantities, can be written in compact form as

$$\frac{\partial W}{\partial t} + \frac{\partial (F^c - F^d)}{\partial x} + \frac{\partial (G^c - G^d)}{\partial y} = S$$

(14)

where

$$W = \begin{bmatrix} p \\ u \\ v \\ T \\ k \\ \omega \end{bmatrix}; \quad F^c = \begin{bmatrix} \beta_{ps}u \\ u^2 + p + 2k/3 \\ uv \\ uT \\ uk \\ u\omega \end{bmatrix}; \quad G^c = \begin{bmatrix} \beta_{ps}v \\ uv \\ v^2 + p + 2k/3 \\ vT \\ vk \\ v\omega \end{bmatrix}$$

and

$$F^d = \begin{bmatrix} 0 \\ (v + v_i)(u_x + u_y) \\ (v + v_i)(v_x + u_y) \\ \left(\frac{v}{Pr} + v_i\sigma_T\right)T_x \\ (v + v_i\sigma_k)k_x \\ (v + v_i\sigma_\omega)\omega_x \end{bmatrix}; \quad G^d = \begin{bmatrix} 0 \\ (v + v_i)(u_x + v_y) \\ (v + v_i)(v_x + v_y) \\ \left(\frac{v}{Pr} + v_i\sigma_T\right)T_y \\ (v + v_i\sigma_k)k_y \\ (v + v_i\sigma_\omega)\omega_y \end{bmatrix}; \quad S = \begin{bmatrix} 0 \\ 0 \\ \gamma_i\beta_{ls}(T - T_c) \\ P_k + G_k - \beta^\ast\omega_k \\ P_\omega + G_\omega - \beta^\ast\omega^2 + P_{\omega^2} \end{bmatrix}$$

Pr = \frac{C_m\mu}{K}; \quad P_k = v_i\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right]; \quad G_k = -\frac{v_i g_i\beta}{\sigma_T} \frac{\partial T}{\partial y}$$

$$P_\omega = \frac{\gamma}{v_i}P_k; \quad G_\omega = \frac{\gamma C_3}{v_i}G_k; \quad P_{\omega^2} = 2(1 - B_1)\sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

where superscript $c$ denotes convective part and superscript $d$ denotes diffusive part. The term $G_k$ and $G_\omega$ are added to take the effect of buoyancy on turbulence parameters. The constant $C_3$ has a value of $\tanh |v/u|$ as in Ref. [14].

2.2. Finite volume discretization

The Navier–Stokes equations together with turbulent transport equations, given in Eq. (14), can be written, in the integral form in an arbitrary 2D domain, as

$$\int_V \frac{\partial W}{\partial t} dV + \oint_A (F n_x + G n_y) dA = \int_V S dV$$

(15)

where $F = F^c - F^d$ and $G = G^c - G^d$, $n_x$ and $n_y$ are direction cosines for the face of area $A$, and $V$ is the cell volume. The computational domain is divided into quadrilateral cells denoted by subscripts $i, j$. Assuming that the dependent variables are known at the center of each cell, a
system of ordinary differential equations is obtained by applying finite volume discretization to the integral equation (15) separately to each cell. The discrete form of the Navier–Stokes equations together with turbulence equations, then can be written as

$$\frac{d}{dt}(V_{i,j}W_{i,j}) + Q_{i,j} = S_{i,j}V_{i,j}$$

(16)

where $Q_{i,j}$ is the net flux out of the cell, which can be evaluated as

$$Q_{i,j} = \sum_{m=1}^{4} \left[ (F_{m} + G_{m}) A_{m} \right]$$

(17)

where subscript $m$ denotes values on the $m$th face of area $A_{m}$ and the sum is over the four faces of the cell.

The convective part of fluxes, denoted by superscript ‘c’, are evaluated by taking the average of the values in the cells on either side of each face. For example,

$$Q_{i+\frac{1}{2},j} = \frac{1}{2}(Q_{i+1,j} + Q_{i,j})$$

(18)

This averaging that operates like second-order central differencing in regular smooth grid, requires artificial dissipative term ($D$) to inhibit odd and even point decoupling in the solution.

The viscous fluxes ($Q^{l}$, and the net flux $Q = Q^{c} + Q^{l}$) are evaluated based on an approach similar to auxiliary cell method. The details of the discretization of the viscous fluxes are given in Ref. [19].

With the addition of artificial dissipative terms Eq. (16) takes the form

$$\frac{d}{dt}(V_{i,j}W_{i,j}) + Q_{i,j} - D_{i,j} = S_{i,j}V_{i,j}$$

(19)

where $Q_{i,j}$ is defined by Eq. (17). To preserve conservation, the artificial dissipative terms are generated by artificial dissipative fluxes. The artificial dissipative fluxes ($D_{i,j}$) are calculated by fourth differences of the dependent variables. Since these terms are fourth order accurate in space, they do not compromise the second order accuracy of the basic scheme. A five stage Runge–Kutta type of scheme [20] is used to advance the solution in pseudo-time towards steady state, explicitly.

3. Results and discussion

Earlier, an extensive validation of the basic pseudo-compressibility method algorithm has been done for inviscid [19], viscous [19], natural convection [7]. The SST turbulence model, also has been validated both for isothermal, i.e. non-buoyant flow, and for natural convection cases [7,21]. In the present work, the algorithm has been applied to compute mixed convection problems. Two different thermal boundary cases are computed.

The problem considered is a 2D square cavity whose top lid is moving at constant velocity. In the first case, case (a), side walls are adiabatic, bottom wall is maintained at ambient temperature and top lid is maintained at constant temperature higher than the ambient temperature, as shown in Fig. 1(a). In the second case, case (b), the side and bottom walls are kept at constant temperature while the top moving lid is kept at ambient temperature, as shown in Fig. 1(b).
For both the cases, two main forces dictate the nature of flow, namely buoyancy driven natural convection (represented by Grashof number \(Gr\) in non-dimensional form) and the lid driven forced convection (represented by Reynolds number \(Re\) in non-dimensional form). The relative effect of these forces can be demonstrated using different combinations of Grashof number and Reynolds number. The computations are performed for Grashof number of \(10^5\) and \(10^6\), while keeping Reynolds number fixed at \(10^3\), for laminar flow situations [1,2]. In another case computations are performed for Grashof number of \(10^7\) and \(10^8\) at a fixed Reynolds number of \(10^4\), for turbulent flow situations [3]. The fluid considered for all the computations is air of Prandtl number of 0.71. The non-dimensional parameters Grashof number \(Gr\) and Reynolds number \(Re\) are defined as

\[
Gr = \frac{\beta \rho g H^3 \Delta T}{v^2}, \quad Re = \frac{V_{ref} H}{v}
\]

where \(H\) is the height of cavity, \(\Delta T\) is unity and \(V_{ref}\) is the lid speed.

The grid used for all the computations has following grid point distribution [14],

\[
x_i = \frac{1}{2} \left( 1 + \frac{\tanh[\alpha(i/i_{max} - 1/2)]}{\tanh(\alpha/2)} \right) \quad i = 1, 2, \ldots, i_{max}
\]

where \(\alpha = 3.5\). Similar grid point distribution has been used in the \(y\) direction of the cavity also.

### 3.1. Case (a)

Figs. 2 and 3 present the temperature contours and velocity vectors for laminar flow situations. The flow structure for lower Grashof number, \(10^2\), resembles that of forced convection situation. The main circulation fills the almost entire cavity. The isotherms are clustered close to the bottom wall, i.e. strong gradient in the bottom region and weak gradient in the bulk of the cavity. This implies that, due to the vigorous mechanical driven circulations, fluids are well mixed. On the other hand for higher Grashof number, \(10^6\), the circulation is restricted to a small zone close to the sliding top lid, much of the fluid remains stagnant in the middle and bottom region. The isotherms show that the heat transfer is mostly conductive in the middle and bottom parts of the cavity. Only in a small region in the top portion of the cavity fluids are well mixed. Fig. 4 shows variation
of $x$-component of velocity along the vertical mid-line and $y$-component of velocity along the horizontal mid-line of the cavity for laminar flow situations. The results compare very well with the computed result, on grid size of $257 \times 257$, of Iwatsu et al. [1], while the grid size used for the present computation is $129 \times 129$.

For turbulent flow situations also the same trends in flow circulation, as of laminar flow situations, are observed (Figs. 5 and 6). As expected, the flow being turbulent at the higher Reynolds number, it is able to penetrate more inside the cavity thus enhancing the heat transfer. In the Fig. 7, variation of $x$-component of velocity along the vertical mid-line and $y$-component of velocity along the horizontal mid-line of the cavity, for turbulent flow situations, are shown.
3.2. Case (b)

Figs. 8 and 9 present the temperature contours and velocity vectors for laminar flow situations. Again the flow structure is of forced convection type for low Grashof number, $10^2$. However, for high Grashof number, $10^6$, the thermal boundary conditions are making significant impact on the flow structure. The strong buoyancy force, generated from the three heated walls, prevents the downward movement of cold fluid of the sliding lid, making cold stream to circulate in the upper portion of the cavity. A strong secondary circulation is observed in the rest of the cavity.
Fig. 6. Plots of temperature and velocity, \( Re = 1.0 \times 10^4 \) and \( Gr = 1.0 \times 10^5 \), Case (a).

Fig. 7. Velocity distribution along the centerline for turbulent flow, Case (a).

For turbulent flow situations also the same trends in flow circulation, as of laminar flow situations, are observed (Figs. 10 and 11). For Grashof number of \( 10^7 \) the flow is able to penetrate inside the cavity. In this case the inertia force is dominating thereby setting the flow of forced convection nature. However, for Grashof number of \( 10^8 \) buoyancy force is not allowing the flow to penetrate inside the cavity. The effect of lid movement is confined to a very small domain in the top and two prominent secondary circulation are formed inside the cavity.

Fig. 12 shows variation of \( x \)-component of velocity along the vertical mid-line and \( y \)-component of velocity along the horizontal mid-line of the cavity for both laminar and turbulent flow situations.
Fig. 8. Plots of temperature and velocity, $Re = 1.0 \times 10^3$ and $Gr = 1.0 \times 10^5$, Case (b).

Fig. 9. Plots of temperature and velocity, $Re = 1.0 \times 10^3$ and $Gr = 1.0 \times 10^6$, Case (b).

Since no benchmark results were available for this case, grid independence studies is conducted. Three different grids of sizes $65 \times 65$, $97 \times 97$ and $129 \times 129$ have been used. Fig. 13 shows the variation of $x$-component of velocity along the vertical mid-line and $y$-component of velocity along the horizontal mid-line of the cavity for the case of Reynolds number $10^4$ and Grashof number $10^8$. It can be seen that the results are almost grid independent. Grid independent results have also been obtained for other combination of Reynolds and Grashof numbers.
4. Concluding remarks

The pseudo-compressibility approach, extensively used for isothermal cases, has been successfully applied to compute mixed convection flow inside a driven cavity for two different thermal boundary conditions. This approach is able to demonstrate all the salient features of the flow inside the cavity as the influence of buoyancy and inertia forces are altered for both laminar flow situations and turbulent flow situations. The turbulence is modeled using the modified, to take account of buoyancy, SST eddy viscosity model.
Fig. 12. Velocity distribution along the centerline, Case (b).

Fig. 13. Effect of grids on velocities for Case (b) at $Re = 10^4$ and $Gr = 10^5$.

Acknowledgements

The authors are grateful to BRNS for the financial support.

References