Development of artificial compressibility method with elliptic relaxation approach

Md Mizanur Rahman\textsuperscript{1,4}, Feroz Alam\textsuperscript{2}, Zefei Zhu\textsuperscript{1} and AKM Sadrul Islam\textsuperscript{3}

\textsuperscript{1}Hangzhou Dianzi University, School of Mechanical Engineering, 310018 Hangzhou, China
\textsuperscript{2}RMIT University, School of Mechanical Engineering, Melbourne, Australia
\textsuperscript{3}X Professor, Department of Mechanical Engineering, BUET, Dhaka, Bangladesh

\textsuperscript{4}Email: mizanur.rahman@hdu.edu.cn

Abstract. An artificial compressibility (AC) method resorting to the pressure-based algorithm is developed on a non-orthogonal non-staggered grid using a cell-centered finite-volume \( \Delta \)-approximation for incompressible fluid flow problems. A pressure time-derivative is used to perturb the equation of continuity. This artifact provokes a density preconditioning which transforms conservative variables to primitive ones; the physical relevance of density preconditioning signifies to convoke compatible linearizations of residuals in collaboration with an AC parameter. The avoidance of pressure-velocity decoupling is enhanced by a consistently formulated cell-face dissipation approach, preserving an increased accuracy with a greater flexibility analogous to the MUSCL (monotone upstream-centered schemes for conservation laws) approach. An elliptic-relaxation scheme having accountability to promoting anisotropic diffusion coefficients is applied to smooth out the pressure residual. Numerical experiments dictate that pertaining to the pseudo-time integration method, the overall contrivance is benefited with an enhanced robustness having sensible oscillation damping characteristics.

Nomenclature

\begin{align*}
C &= \text{artificial sound speed} \quad & x, y &= \text{Cartesian coordinates} \\
CFL &= \text{Courant number} \quad & \Gamma &= \text{diffusion coefficient} \\
F, G &= \text{flux vector components} \quad & \epsilon &= \text{smoothing coefficient} \\
f &= \text{source term} \quad & \zeta &= \text{invariant of strain-rate or vorticity} \\
Gr &= \text{Grashof number} \quad & \eta &= \text{stability factor of implicitness} \\
L &= \text{characteristic length} \quad & \theta &= \text{dimensionless temperature} \\
p &= \text{static pressure} \quad & \nu &= \text{kinematic viscosity} \\
Pr &= \text{Prandtl number} \quad & \rho &= \text{density} \\
S &= \text{cell-face area} \quad & \phi &= \text{scalar variable} \\
t &= \text{time} \quad & \forall &= \text{cell volume} \\
T &= \text{dimensional temperature} \quad & \text{Subscript} \\
u, v &= \text{dimensional velocity components} \quad & r &= \text{reference condition} \\
U, V &= \text{dimensionless velocity components} \quad & nb &= \text{neighboring grid point} \\
W &= \text{conservative variable vector} \\
\tilde{W} &= \text{primitive variable vector}
\end{align*}
1. Introduction

Numerical algorithms associated with a compressible flow code (CFC) can be used to simulate nearly incompressible flows when preconditionings are involved. The steady-state solution in this case remains unaltered whereas the transient behavior of governing equations is modified [1-5]. Another well-known approach to extending the CFC for applying at appreciably low-Mach number flows emerges from an artificial compressibility (AC) or a pseudo-equation of state [5], signifying that the incompressibility constraint is perturbed to an extent; this perturbation attribute (AC term) is the dominant factor in determining a credible success adhering to a numerical solver.

The viscous incompressible flow in conjunction with an AC is customarily established on a pressure-based algorithm wherein the pressure-velocity decoupling is physically prevented. It is worth mentioning that collocated grid arrangements require artificial damping terms or a cell-face momentum interpolation technique to eradicate grid-scale oscillations due to the pressure-velocity decoupling. A more convenient way to avoid the pressure checker-boarding is to devise a means of convoking proper cell-face dissipations without extenuating the solution accuracy. Several contrivances resorting to primitive-variable formulations have been made to suppress destabilizing effects embedded with non-staggered grids [6-12].

In principle, discretized Navier-Stokes equations can be numerically solved using either a coupled algorithm in which all discretized equations are solved as one system following best-estimates of other dependent variables or a segregated method, where the system of equations adhering to all variables is solved sequentially. Traditionally, the density-based algorithm invokes a coupled approach, whereas the pressure-based method follows a segregated approach. With the development of SIMPLE method (Semi-Implicit Method for Pressure Linked Equations) for solving incompressible flows, the segregated pressure-based scheme has captured the approval in early 1970s [10]. However, the pressure-velocity correction step associated with the SIMPLE algorithm is eliminated in the current work by incorporating an AC parameter in the continuity constraint.

Literature review and to the authors’ best knowledge have identified that an implicit segregated solver with a recourse to an AC accompanied by an elliptic-relaxation approach has not yet been evaluated where the overall artifact may enhance the avoidance of complicated matrix-inversion with preconditionings. In the current study, a cell-centered finite-volume method on a curvilinear non-staggered grid is devised to solve two-dimensional (2-D) incompressible flows. Unlike Chorin’s artificial density-based scheme, the pressure time-derivative is used to perturb the mass continuity, leading to the density preconditioning/perturbation; it essentially augments the conversion between conservative and primitive variables, rectifying a compatible linearization for residuals. To alleviate non-physical pressure modes, a consistently formulated cell-face dissipation scheme is employed. An ADI (alternating direction implicit) unfactorized pseudo-time integration approach is applied to solve flow equations. Explicit pressure residuals are smoothed out employing an elliptic-relaxation method, inheriting non-isotropic smoothing coefficients. Details of the spatial discretization accompanied by relevant aspects are presented.
2. Governing Equations

An incompressible flow is governed by continuity, momentum and scalar \( \phi \) equations, provided with the Einstein notation as:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (1)
\]

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} + \frac{\partial p}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial u_i}{\partial x_j} \right) \quad (2)
\]

\[
\frac{\partial \rho \phi}{\partial t} + \frac{\partial \rho u_j \phi}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) \quad (3)
\]

where \( t \) the time, \( x_i \) the Cartesian coordinates, \( u_i \) the velocity vector, \( \rho \) the density, \( p \) the pressure and \( \Gamma \) the exchange coefficient (viscosity or conductivity of fluid).

2.1. Discretization with \( \Delta \)-formulation

The \( \phi \)-equation together with 2-D Navier-Stokes equations is solved utilizing a cell-centered finite-volume scheme; the following integral identity is obtained:

\[
dt \int_{V} W \, d\mathbf{v} + \int_{S} \mathbf{F}(W) \cdot d\mathbf{S} = \int_{V} f \, d\mathbf{v} \quad (4)
\]

for an arbitrary region \( V \) with a boundary \( S \): \( W = (\rho, \rho u_i, \rho \phi)^T \) indicate conservative variables and \( f \) represents the source term. Integrating for a computational cell \( i \) comes up with:

\[
\frac{dW_i}{dt} = \sum_{faces} -S \hat{F} + \mathcal{V}_i f_i \quad (5)
\]

where the sum corresponds to computational cell faces. The unit normal vector \( \vec{n} \) regarding each cell face can be defined by

\[
\vec{n} = n_x \vec{i} + n_y \vec{j} = \frac{S_x \vec{x} \vec{i} + S_y \vec{y} \vec{j}}{S} \quad (6)
\]

The correlated cell-face flux complies with

\[
\hat{F} = n_x F + n_y G \quad (7)
\]

where \( F \) and \( G \) represent inviscid and viscous fluxes in the \( x \) and \( y \) directions, respectively, associated with Eqs. (1)-(3). A second-order central difference scheme can be employed to compute viscous fluxes on a curvilinear grid using a thin-layer approximation. On a cell-face \((i + 1/2)\), the inviscid flux is calculated with a rotational matrix which accounts for directional dominance on upwinding schemes [11]. A rotational matrix adapts contravariant momentum-flux components to primitive ones in a Cartesian-coordinate system. Van Leer’s MUSCL scheme [13] is utilized to evaluate primitive variables \( \tilde{W} = (u_i, p, \phi)^T \) at the cell-face.

A consistently formulated cell-face dissipation approach is supplicated to eradicate an odd-even point decoupling [7, 14, 15]:

\[
U_{i+1/2} = \frac{1}{2} (U_i + U_{i+1}) - \frac{1}{4} \left( \frac{1}{(\rho C_i)_{i+1}} - \frac{1}{(\rho C_i)_i} \right) (f_{i+1} - f_i) - \frac{1}{2} \rho C_i \frac{p^R - p^L}{(p^R - p^L)} \quad (8)
\]

where \( U = u_n x + v_n y \) is the contravariant velocity component and \( f = f^u n_x + f^v n_y \); the pseudo-sound (artificial sound) speed \( C \) is approximated later.

Cell-face state variables \((p^L, p^R)\) on the left and right sides are determined with an MUSCL scheme. Since both velocity and pressure fields are inclusively supported by neighboring nodal estimates, Eq. (8) ensures a strong coupling between them, providing a non-linear cell-face interpolation method. Subsequent sections deal with salient features of AC and the construction of elliptic-relaxation model, leading to a pressure-linked equation.
3. Artificial Compressibility and Associated Aspects

The continuity equation includes a time-derivative of pressure, subjected to transforming the incompressible Navier-Stokes equations into an hyperbolic structure. The unsteady mass continuity with a non-conservative form can be introduced within the framework of a 2-D analysis as:

$$\frac{\partial \rho}{\partial t} + \rho \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) = 0 \quad (9)$$

Applying the chain rule with relaxing the incompressibility constraint in the mass continuity yields [5]:

$$\frac{\partial \rho}{\partial p} \frac{\partial p}{\partial t} + \rho \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) = 0 \quad (10)$$

where $\rho = \rho(p)$. Rearranging Eq. (10) provides

$$\frac{\partial p}{\partial t} + \rho C^2 \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) = 0 \quad (11)$$

where $C$ is recovered as a perturbation parameter or an artificial sound speed; it is numerically optimized as:

$$C^2 = \left( \frac{\partial \rho}{\partial p} \right)^{-1} = \beta^2 \max \left( u^2 + v^2; \frac{1}{2} U_r^2 \right) \quad (12)$$

where $\beta$ implies a compressibility parameter and $U_r$ indicates a reference velocity. Conventionally, $\beta$ encompasses values in a range of 1–10 to better enhance the steady-state convergence, enforcing the mass conservation.

An artificial or a differential change in density is invoked to transform conservative variables into primitive variables:

$$\Delta \rho = \frac{\partial \rho}{\partial p} \Delta p = \frac{\Delta p}{C^2} \quad (13)$$

Using Eq. (13), the conversion of conservative variables to primitive ones can be presented with a matrix form, providing obvious interpretations for fluidistic relations:

$$\begin{pmatrix} \Delta \rho \\ \Delta u \\ \Delta v \\ \Delta \phi \end{pmatrix} = \frac{1}{\rho} \begin{pmatrix} \rho C^2 & 0 & 0 & 0 \\ -u & 1 & 0 & 0 \\ -v & 0 & 1 & 0 \\ -\phi & 0 & 0 & 1 \end{pmatrix} \begin{pmatrix} \Delta \rho \\ \Delta \rho u \\ \Delta \rho v \\ \Delta \rho \phi \end{pmatrix} \quad (14)$$

where $\Delta W$, representing conservative explicit changes can be obtained from:

$$\Delta W = (\Delta \rho, \Delta \rho u, \Delta \rho v, \Delta \rho \phi)^T = \frac{\Delta t_i}{\nu_i} R_i \quad (15)$$

with RHS of Eq. (5) can be appointed to evaluate $R_i$ (residuals) and $\Delta t_i$ (spatially varying pseudo-time step) attributed with sound, diffusion and convection speeds is determined from References [16, 17]. The transformation from conservative to primitive variables is executed prior to the implicit stage which envisages $\Delta \rho$ (explicit density residual) induced by tentative velocity fields. The mass imbalance $\Delta \rho$ is apparently beneficial to predict a realistic pressure field as well as to linearize momentum and scalar residuals in conjunction with the perturbation parameter $C$.

After the implicit treatment, the following relation is involved to update primitive variables $\tilde{W} = (p, u, v, \phi)^T$:

$$\tilde{W}^{n+1} = \tilde{W}^n + (\Delta \rho, \Delta \rho u, \Delta \rho v, \Delta \rho \phi)^T \quad (16)$$
with current estimates at the time level \((n + 1)\Delta t\), indicated by \(\tilde{W}^{n+1}\). To the incompressible limit, \(\Delta \rho\) in Eq. (13) can indeed be recognized as the density perturbation/preconditioning; the artificial density change \(\Delta \rho\) disappears with the iterative convergence.

4. Solution Algorithm
A detailed description of boundary conditions is well found in Reference [11]. Acknowledging a first-order backward-difference to temporal variations in \(\tilde{W}_i\), an implicit pseudo-time integration adhering to Eq. (5) provides a discretized system of algebraic equations [9]:

\[
\left[ 1 + \frac{\Delta t}{V_i} \left( \sum_{nb} A_{nb} + C_i \sqrt{S_i^2 + S_j^2} \right) \right] \Delta \tilde{W}_i = \frac{\Delta t}{V_i} \sum_{nb} A_{nb} \Delta \tilde{W}_{nb} + \Delta \tilde{W}_i^* \tag{17}
\]

where \(\Delta t = \eta \Delta t_i\) and \(\eta = 1.5\) implies a factor of implicitness stability. The influence coefficient \(A\) retains mass and diffusion fluxes; primitive explicit residuals of dependent variables \(\Delta \tilde{W}_i^* = (\Delta u, \Delta v, \Delta \phi)^T\) are represented by Eq. (14). A run over adjacent nodes \((i + 1), (i - 1), (j + 1),\) and \((j - 1)\) is indicated by the subscript \(nb\). The inclusion of \(C\) with LHS of Eq. (17) apparently signifies to augment the nodal-coefficient diagonal-strength. Nonetheless, it is not essential to add the weight due to \(C\) in Eq. (17) when solving the scalar \(\phi\)-equation.

To this end, it can be emphasized that relating to the eigen-structure, the current scheme requires no complicated matrix inversion approach. However, implicit residual smoothing schemes can be employed to enhance the convergence acceleration, implicating essentially the replacement of flow-field residual at one location with weighted average of residuals (smoothed residuals) at neighboring points. This approach has a close resemblance to that of a Helmholtz-type elliptic-relaxation equation; it can be expressed for a node \(i\) as follows:

\[-\epsilon \nabla^2 R_i^* + R_i^* = R_i \tag{18}\]

where \(R\) denotes the non-smooth (original) residual, \(R^*\) is the smoothed residual and \(\epsilon\) stands for a non-dimensional smoothing coefficient with \(\epsilon \leq 1.0\) in order to avoid an excessive diffusion. This is a relevant contrivance to escalate the maximum CFL number by a factor of 2 or 3 in conjunction with a multigrid method. The standard discretization scheme pertaining to Eq. (18) is the central implicit residual smoothing (CIRS), given in 2-D by [18]:

\[-\epsilon R_{i-1}^* + (1 + 2\epsilon) R_i^* - \epsilon R_{i+1}^* = R_i \]
\[-\epsilon R_{j-1}^{**} + (1 + 2\epsilon) R_j^{**} - \epsilon R_{j+1}^{**} = R_j^{**} \tag{19}\]

The implicit system of Eq. (19) can be solved using a simple Gauss-Siedel method or a tridiagonal matrix algorithm (TDMA). Nevertheless, the objective herein is to introduce an appropriate amount of smoothing in every coordinate direction as required to better strengthen stability and damping of high-frequency error components in residuals. In particular, the synthetic UIRS (upwind implicit residual smoothing) method is well suited for high Mach-number flows; the so-called approach has an ability to damp out non-physical errors even at a higher smoothing parameter like \(\epsilon = 500\) in combination with a multigrid scheme [19].

In order to generate anisotropic smoothing coefficients \(\epsilon\), the pressure-residual smoothing is carried out by a Helmholtz-type elliptic-relaxation equation, representing a general ellipticity, adhering to \(\Delta \tilde{p}\):

\[-L^2 \nabla^2 \Delta \tilde{p} + \Delta \tilde{p} = \Delta \rho C^2 \tag{20}\]

where \(\Delta \rho\) is an explicit mass imbalance (residual) induced by the tentative velocity fields in Eq. (14). Remarkably, the compatible formulation of characteristic length scale \(L\) is a prerequisite
to controlling the magnitude of $\epsilon$; it is evaluated at the nodal point based on the computational experience:

$$L_i = \max \left[ \frac{C_i}{\zeta_i} \sqrt{\frac{\Delta t_i C_i}{L_r}} \left( 1 + \frac{\Delta t_i \zeta_i}{\nu_i} \right) \right]$$  \hspace{1cm} (21)

where $\zeta_i = \sqrt{\max(S_i^2; W_i^2) + 2/3(C_i/L_r)^2}$, containing invariants $\hat{S} = \sqrt{2\hat{S}_{ij}\hat{S}_{ij}}$ and $\hat{W} = \sqrt{2\hat{W}_{ij}\hat{W}_{ij}}$. The strain-rate and vorticity tensors $\hat{S}_{ij}$ and $\hat{W}_{ij}$, respectively are given by

$$\hat{S}_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad \hat{W}_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$$  \hspace{1cm} (22)

Note-worthily, the construction of $\zeta$ better matches viscous flows retaining a substantial low-speed region such as shear and thick boundary layers, preserving a non-zero value even at the free-stream regime. The dignity of Eq. (20) is that it necessitates no particular numerical treatment unlike the Poisson equation. The solution to elliptic-relaxation equation can be progressed with an initial guess $\Delta \tilde{p} = 0$. The pressure-linked Eq. (20) can be discretized as

$$\left( 1 + L_i^2 \sum_{nb} B_{nb} \right) \Delta \tilde{p}_i = L_i^2 \sum_{nb} B_{nb} \Delta \tilde{p}_{nb} + \Delta \rho_i C_i^2 \left( 1 + \frac{\Delta t_i}{\nu_i} C_i \sqrt{S_i^2 + S_j^2} \right) \hspace{1cm} (23)$$

where $B_{nb}$ are neighboring influence coefficients and the term $\left( 1 + \frac{\Delta t_i}{\nu_i} C_i \sqrt{S_i^2 + S_j^2} \right)$ is an additional limiting factor for the explicit pressure residual.

Discretized flow equations are solved with a TDMA. The boundary conditions are treated explicitly. Solution sequences are given as follows:
1. Make initial guesses for primitive fields (pressure, velocity and scalar).
2. Evaluate the cell-face mass flux from Eq. (8).
3. Estimate primitive explicit residuals from Eq. (14).
4. Use the above-mentioned procedure to solve the system of equations.
5. Provide physical fields by updating primitive variables using Eq. (16).
6. Continue steps 2–5 until the solver converges.

5. Numerical Computations
Reference [9] schematically shows physical models for buoyancy-driven flows in a square cavity and in a half-concentric annulus, respectively with a characteristic length of $L$, having adiabatic (insulated) bottom and top walls. Cold and hot walls have constant temperatures of $T_c$ and $T_h$, respectively. Dimensional parameters are non-dimensionalized with reference scales; non-dimensional variables reduce computer round-off errors, avoiding restrictions on a particular system of units. From Reference [7], 2-D dimensionless governing equations can be achieved. Two non-dimensional parameters obtained from the normalization dominate the test-case: Grashoff number $Gr = g\gamma(T_h - T_c)L^3/\nu^2$ and Prandtl number $Pr = \nu/\alpha$ with $\gamma$, $\alpha$ and $g$ are the thermal-expansion coefficient, thermal diffusivity and gravitational acceleration, respectively. Results are plotted in the appearance of $V = vL/\nu$ and $\theta = (T - T_c)/(T_h - T_c)$ versus $X = x/L$ with $Pr = 1.0$. For this test problem, a dimensional reference velocity $u_r = \sqrt{g\gamma(T_h - T_c)L} = \nu \sqrt{Gr}/L$ can be obtained and therefore, $U_r = u_r L/\nu = \sqrt{Gr}$ is extracted as a non-dimensional reference velocity [11].
Figure 1. Square-cavity buoyancy-driven flow: (a) velocity; (b) temperature; (c) local Nusselt number and (d) artificial density convergence.

The test-case retains both highly diffusive central and convective near-wall flow regimes; since no analytic solution to this problem is available, literature data [20], results of the SIMPLE algorithm [9] and CVFEM (control-volume based finite-element method) with stream-function vorticity formulation [21] are considered to estimate the precision of current simulations. The artificial mass imbalance with the root-mean-square identity

$$\Delta \rho^* = \sqrt{\frac{\sum_{i=1}^{NP} |(\Delta p/C^2)_i|^2}{NP}} \leq \eta_t$$

is used to judge the convergence, where $\eta_t$ indicates a user-defined tolerance limit and NP stands for the number of computational cells. A non-uniform grid spacing is employed with numerical problems to capture intense gradients in the wall-vicinity. Numerical errors due to the grid resolution is evaluated as less than 2% for all flow-field variables based on grid dependence tests. A finite-difference formula with three-point one-sided stencils is used to approximate $\partial \theta / \partial X$ associated with the local Nusselt number $Nu = -(\partial \theta / \partial X)|_{X=0}$. All walls use no-slip boundary conditions; an adiabatic boundary condition is applied to top and bottom walls. The source term $f = (0, 0, Gr \theta, 0)^T$ and temperature boundary conditions: $\theta_l = 1.0$ and $\theta_r = 0.0$ are on left and right vertical walls, respectively. An FUS (fully upwinded second-order) scheme is implemented to determine inviscid fluxes on the cell-face. Current computations exercise a compressibility
parameter of $\beta = 1$. Since buoyancy-driven flows contain sources with positive magnitudes in momentum equations, the linearization of which is not straightforward. Nevertheless, the rigidity caused by a positive source can be minimized by applying the pseudo-linearization [11].

![Graphs showing velocity, temperature, local Nusselt number, and artificial density convergence.]

**Figure 2.** Annular-cavity buoyancy-driven flow: (a) velocity; (b) temperature; (c) local Nusselt number and (d) artificial density convergence.

### 5.1. Buoyancy-driven flow in square cavity

A non-uniform grid $32 \times 32$ for $Gr = 10^5$, $40 \times 40$ for $Gr = 10^6$ and $60 \times 60$ for $Gr = 10^7$ are appointed for numerical simulations; grids are heavily clustered in near-wall regions than in the core. Figures 1(a) and 1(b) present profiles of vertical velocity and temperature, respectively on the horizontal mid-plane at $Gr = 10^7$. SIMPLE and CVFEM computational results together with literature data produced by Massarrotti et al. [20] are also plotted for comparisons. Apparently, a motivated qualitative consensus is obtained from comparisons. Figure 1(c) depicts distributions of the local Nusselt number across the hot vertical wall at $Gr = 10^7$. A careful inspection shows that with CVFEM computations, the present method maintains a fairly good agreement. Figure 1(d) exhibits the convergence of artificial density from the square-cavity buoyancy-driven flow at various Gr numbers with similar initial conditions. Modes of performances with regards to CFL numbers ascertain that the choice of elliptic relaxation length scale in Eq. (21) accommodates better convergence characteristics with all simulations. Above all, the interesting fact is that the square-cavity buoyancy-driven flow maintains a good balance with $CFL = 1.0$ for all Gr’s.
5.2. Buoyancy-driven flow in annular cavity

For this test-case, radii of outer and inner surfaces are respectively, \( R_O \) and \( R_i \) with \((R_O/R_i) = 3\). The Cartesian-coordinate system origin is placed at the center of circles, having a characteristic length of \( L = (R_o - R_i) \) with adiabatic straight bottom and top walls. The outer curved surface is cold with a temperature \( T_c \) and the inner one is hot with a temperature \( T_h \). Governing equations and boundary conditions resemble those of the square-cavity buoyancy-driven flow.

The non-uniform computational grid consists of 30 circumferential and 40 radial line segments; this grid resolution is presumably adequate to capture characteristic flow and transport properties. On the horizontal mid-plane at \( Gr = 10^5 \), Figures 2(a) and 2(b) illustrate profiles of vertical velocity and temperature, respectively; \( X \) is exactly counted from the inner surface. As is observed, the present method makes good correspondence with the SIMPLE and CVFEM; remarkably, the proposed algorithm replicates impressive agreement with CVFEM data. Ostensibly, distributions of the local Nusselt number in Figure 2(c) reproduce satisfactory solutions compared with the CVFEM. However, some disparity between CVFEM and current data is distinguishable in the inner wall-vicinity (e.g., lower section). This discrepancy appears due to the methodological differences; note that a weighted nodal temperature is computed by the CVFEM. The inconsistency associated with numerical results can be minimized to some extent using a higher-order formulation and a finer grid resolution with the present algorithm. Alike the square-cavity buoyancy-driven flow, similar conclusions can be drawn for the convergence of artificial mass imbalance shown in Figure 2(c); however, this flow appreciates \( CFL < 1.0 \) with increasing \( Gr \) number.

![Figure 3. Average smoothing coefficients: (a) buoyancy-driven flow in square cavity and (b) buoyancy-driven flow in annulus.](image)

5.3. Average smoothing coefficients

Plots of average smoothing coefficients on the horizontal mid-plane are demonstrated in Figure 3 for both square and annular cavities at different \( Gr \)'s. Average smoothing coefficients are estimated as \( \epsilon_i = \frac{1}{4L_i^2} \sum_{nb=1}^{4} B_{nb} \), where \( nb \) refers to neighboring nodal points. Since the elliptic-relaxation length scale \( L_i \) is dependent on CFL numbers, magnitudes of \( \epsilon \) are reduced with decreasing CFL and accordingly, the solver needs more iterations to converge. It seems likely that within the near-wall (i.e., boundary layer) regions \( \epsilon \) experiences reduced magnitudes because of the non-local (e.g., wall-blocking) impacts of the elliptic-relaxation model. Conspicuously, higher values of \( \epsilon \) augment better convergence as reflected in convergence plots [see Figures 1(d)].
and 2(d)]. The convergence acceleration could be further improved when the present scheme is coupled with a multigrid cycle since it may induce an augmentation in the maximum CFL number by a factor of 2 or 3 [18].

6. Conclusions
With an adherence to the artificial compressibility (AC), a pressure-based algorithm is formulated on a curvilinear collocated grid for two-dimensional incompressible flows; the involved pseudo-time integration approach utilizes a cell-centered finite-volume $\Delta$-formulation. The AC-generated sound speed provides just enough cell-face damping with the velocity, requiring no complicated matrix inversion regarding the eigen-structure. The current algorithm favorably matches CVFEM and literature data for a specified degree of grid refinement as demonstrated by computational results. The primitive formulation with a non-linear cell-face dissipation scheme succeeds satisfactorily in preventing non-physical modes and leads to an effective stabilization for the iteration procedure as dictated by convergence studies. The prospective significance of elliptic-relaxation approach is obvious.

References
[1] Choi Y-H, Merkle CL 1993: The Application of preconditioning to viscous flows. J. Comput. Phys., 105:207-223.
[2] Turkel E, Radespiel R, Kroll N 1997: Assessment of preconditioning methods for multidimensional aerodynamics. Comput. Fluids, 26:613-634.
[3] Weiss JM, Smith WA: Preconditioning applied to variable and constant density flows. AIAA J., 33:2050-2057:1995.
[4] Rahman MM, Rautaheimo P, Siikonen T 1997: Numerical study of turbulent heat transfer from a confined impinging jet using a pseudo-compressibility method, Second Int. Symposium on Turbulence, Heat and Mass transfer, Delft, The Netherlands, Hanjalic K and Peeters TWJ, (eds), Delft University Press: Delft, 511-520.
[5] Chorin AJ 1967: A numerical method for solving incompressible viscous flow problems. J. Comput. Physics, 2:12-26.
[6] Rhie CM, Chow WL 1983: Numerical study of the turbulent flow past an airfoil with trailing edge separation. AIAA J., 21:1525-1532.
[7] Rahman MM, Miettinen A, Siikonen T 1996: Modified SIMPLE formulation on a collocated grid with an assessment of the simplified Quick scheme. Numer. Heat Transfer Part B, 30:291-314.
[8] Rahman MM, Siikonen T, Miettinen A 1997: A pressure correction method for solving fluid flow problems on a collocated grid. Numer. Heat Transfer, Part B, 32:63-84.
[9] Rahman MM, Siikonen T 2000: An improved SIMPLE method on a collocated grid. Numer. Heat Transfer Part B, 38:177-201.
[10] Patankar SV, Spalding DB 1972: A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. Int. J. Heat Mass Transfer, 15:17871806.
[11] Rahman MM 2020: Avoiding under-relaxations in SIMPLE algorithm. Numer. Heat Transfer Part B, 52(5). doi.org/10.1080/10407790.2020.1787043.
[12] Qin J, Pan HC, Rahman MM, Tian XQ, Zhu Z 2021: Introducing compressibility with SIMPLE algorithm. Math. Comput. Simul., 180:328-353. doi.org/10.1016/j.matcom.2020.09.010.
[13] Anderson WK, Thomas JL, Van Leer B 1983: Comparison of finite volume flux vector splittings for the Euler equations. AIAA J., 24:649-664.
[14] Rahman MM, Siikonen T 2001: An artificial compressibility method for incompressible flows. *Numer. Heat Transfer Part B*, 40:391-409.

[15] Rahman MM, Siikonen T 2002: A dual-dissipation scheme for pressure-velocity coupling. *Numer. Heat Transfer Part B*, 42, 231-242.

[16] Rahman MM, Siikonen T 2008: An artificial compressibility method for viscous incompressible and low Mach number flows. *Int. J. Numer. Meth. Enng.*, 75:1320-1340.

[17] Rahman MM 2021: Compromising with corrector step of SIMPLE algorithm. *Math. Comput. Simul.*, 188:135-163.

[18] Enander R, Karlsson AR 1995: Implicit explicit residual smoothing in multigrid cycle. *AIAA Paper*, 95-0204.

[19] Blazek J, Kroll N, Radespiel R, Rossow C-C 1991: Upwind implicit residual smoothing method for multi-stage schemes. *AIAA Paper*, 91-1533.

[20] Massarotti N, Arpino F, Lewis RW, Nithiarasu P 2006: Explicit and semi-implicit CBS procedures for incompressible viscous flows. *Int. J. Numer. Methods in Enng.*, 66:1618-1640.

[21] Rahman MM, Lampinen MJ 1995: Numerical study of natural convection from a vertical surface due to combined buoyancies. *Numer. Heat Transfer A*, 28:409-429.