A simple and efficient incompressible Navier-Stokes solver for unsteady complex geometry flows on truncated domains

Yann T. Delorme, Kunal Puri, Jan Nordström, Viktor Linders, Suchuan Dong and Steven H. Frankel

Journal Article

N.B.: When citing this work, cite the original article.

Original Publication:

Yann T. Delorme, Kunal Puri, Jan Nordström, Viktor Linders, Suchuan Dong and Steven H. Frankel, A simple and efficient incompressible Navier-Stokes solver for unsteady complex geometry flows on truncated domains, *Computers & Fluids*, 2017. 150, pp.84-94.

<http://dx.doi.org/10.1016/j.compfluid.2017.03.030>

Copyright: Elsevier

<http://www.elsevier.com/>

Postprint available at: Linköping University Electronic Press

<http://urn.kb.se/resolve?urn=urn:nbn:se:liu:diva-136507>

A Simple and Efficient Incompressible Navier-Stokes Solver for Unsteady Complex Geometry Flows on Truncated Domains

Yann T. Delorme^{a,*}, Kunal Puri^a, Jan Nordstrom^b, Viktor Linders^b,
Suchuan Dong^c, Steven H. Frankel^a

^a*Faculty of Mechanical Engineering, Technion-Israel Institute of Technology, Haifa, Israel*

^b*Department of Mathematics, Linköping University, Linköping, Sweden*

^c*Department of Mathematics, Purdue University, West Lafayette, IN, USA*

Abstract

Incompressible Navier-Stokes solvers based on the projection method often require an expensive numerical solution of a Poisson equation for a pressure-like variable. This often involves linear system solvers based on iterative and multigrid methods which may limit the ability to scale to large numbers of processors. The artificial compressibility method (ACM) [6] introduces a time derivative of the pressure into the incompressible form of the continuity equation creating a coupled closed hyperbolic system that does not require a Poisson equation solution and allows for explicit time-marching and localized stencil numerical methods. Such a scheme should theoretically scale well on large numbers of CPUs, GPU's, or hybrid CPU-GPU architectures. The original ACM was only valid for steady flows and dual-time stepping was often used for time-accurate simulations. Recently, Clausen

*Corresponding author

Email address: `delorme.yann@gmail.com` (Yann T. Delorme)

[7] has proposed the entropically damped artificial compressibility (EDAC) method which is applicable to both steady and unsteady flows without the need for dual-time stepping. The EDAC scheme was successfully tested with both a finite-difference MacCormack’s method for the two-dimensional lid driven cavity and periodic double shear layer problem and a finite-element method for flow over a square cylinder, with scaling studies on the latter to large numbers of processors. In this study, we discretize the EDAC formulation with a new optimized high-order centered finite-difference scheme and an explicit fourth-order Runge-Kutta method. This is combined with an immersed boundary method to efficiently treat complex geometries and a new robust outflow boundary condition to enable higher Reynolds number simulations on truncated domains. Validation studies for the Taylor-Green Vortex problem and the lid driven cavity problem in both 2D and 3D are presented. An eddy viscosity subgrid-scale model is used to enable large eddy simulations for the 3D cases. Finally, an application to flow over a sphere is presented to highlight the boundary condition and performance comparisons to a traditional incompressible Navier-Stokes solver is shown for the 3D lid driven cavity. Overall, the combined EDAC formulation and discretization is shown to be both effective and affordable.

Keywords: Artificial compressibility method, EDAC, High-order numerical methods, Large Eddy Simulation

1. Introduction

Fluid flows of practical interest span the low Mach number incompressible regime to the high-speed compressible flow regime. For the latter, the fully

4 compressible Navier-Stokes equations are the only option for prediction. For
 5 low-speed flows, numerical stiffness due to disparities between the relatively
 6 fast acoustic speed and the slow flow speed often results in numerical stability
 7 problems, small time steps, and inefficient calculations. Traditionally, the
 8 most common recourse is to discretize and solve the incompressible form
 9 of the Navier-Stokes equations using a predictor-corrector approach together
 10 with a Poisson equation for a pressure-like variable. The extra cost of solving
 11 the Poisson equation at each time step is somewhat offset by the larger time
 12 step that can be employed. Optimizing the efficiency of the Poisson solver
 13 is crucial to ensure the best performance of the solver. This is because
 14 scalability of the algorithm on large numbers of processors essentially depends
 15 on the parallel scalability of the method used to solve the system of linear
 16 equations. Some applications still require the use of a compressible solver,
 17 even though the flow is mostly low speed. This happens when aeroacoustics
 18 is of interest, or for example in high angle of attack low speed aerodynamics
 19 where localized shocks can occur. In this case, preconditioning methods can
 20 be used [32]. The goal here is to improve the condition number of the system
 21 of equations before starting the computation, allowing a relaxation of the
 22 time step constraint and thus improving the efficiency of the overall solver.
 23 In 1967, Chorin [6] proposed a new method called the Artificial Compress-
 24 ibility Method (ACM) to solve low speed flows. This method is based on a
 25 modified version of the incompressible Navier-Stokes equations where a time
 26 derivative of the pressure is introduced into the continuity equation, hence
 27 obviating the need for a Poisson solver. Upon convergence of the solution
 28 to steady state, the time derivative of the pressure vanishes and the solution

29 satisfies the original incompressible Navier-Stokes equations. As a result of
 30 this formulation, this approach was at first limited to steady flows only. In
 31 1977, Steger *et al* [31] employed this approach to efficiently simulate 2D and
 32 3D vortex wake flows. In 1978, Hafez *et al* [16] demonstrated good perfor-
 33 mance and accuracy for transonic flow over an airfoil using a modified version
 34 of the ACM. In 1985, Choi *et al* [5] compared ACM and low Mach number
 35 approximation to solve steady low speed flows. They showed that in 2D an
 36 optimized preconditioning is necessary for the low Mach number formula-
 37 tion to be as efficient as the ACM in terms of convergence rate. In 1988,
 38 Hartwich *et al* [17, 18] used ACM to study 3D sharp edged delta wings at
 39 different angle of attacks. The algorithm was implemented using high-order
 40 flux reconstruction schemes that allowed for accurate predictions of the per-
 41 formance of the airfoil when compared to experimental data. Similarly, Kwak
 42 *et al* [20] studied 3D space shuttle engine power head using ACM on a 3D
 43 curvi-linear mesh. All the applications cited until now were limited to steady
 44 flows.

45 For application to unsteady flows, a pseudo-time derivative $\partial/\partial\tau$ is added
 46 to the continuity and momentum equations. Hence, there are now two time
 47 derivatives in the momentum equation: the actual time and the pseudo-time.
 48 For each physical time step, the solution is advanced in pseudo-time until con-
 49 vergence to the steady-state, and hence incompressible solution. This results
 50 in an iterative process for each time step until the pseudo-time derivative van-
 51 ishes. This approach was used successfully for many applications over the
 52 past 20 years [26, 28, 4]. In 2006, Madsen *et al* [23] discussed the efficiency
 53 and accuracy of this approach. Indeed, an artificial parameter β was intro-

54 duced in the equations, controlling the rate of convergence of the solution to
 55 steady state. It was then possible to converge faster to the incompressible
 56 solution, but not without affecting the accuracy of the overall solution. It
 57 is then very important to select the value of β carefully to obtain the best
 58 efficiency while maintaining a desired level of accuracy. This approach is still
 59 widely used for simulations of unsteady 3D flows [33].

60 Recently, Clausen [7] proposed an alternative approach to solve the ACM
 61 equations for unsteady flows without using a dual time stepping approach.
 62 This method is called Entropically Damped form of the Artificial Compress-
 63 ibility (EDAC) method and is derived directly from the compressible Navier-
 64 Stokes equations and uses thermodynamic relationships to obtain an equation
 65 for pressure. It assumes an isentropic behavior of the flow and the pressure
 66 equation is obtained by minimizing density fluctuations. In [7], the EDAC
 67 scheme was successfully tested using both a finite-difference MacCormack's
 68 method for the two-dimensional steady lid driven cavity and unsteady peri-
 69 odic double shear layer problem, and a finite-element method for flow over a
 70 square cylinder. Scaling of the finite-element method based solver on a large
 71 number of processors was also shown to be very good due to the absence of
 72 an iterative process, even for unsteady flows.

73 In the present paper, the EDAC formulation is discretized by combining a
 74 new family of optimized high-order finite-difference schemes for spatial dis-
 75 cretization on structured Cartesian grids [22] with the classic fourth-order
 76 Runge-Kutta scheme for explicit time stepping. For non-Cartesian geome-
 77 tries a mirroring immersed boundary method (IBM) [25] is utilized and for
 78 turbulent flows an eddy viscosity based subgrid-scale model (SGS) is used to

79 enable large eddy simulations (LES). In addition, to enable simulations on
 80 highly truncated domains, a new robust outflow boundary condition for in-
 81 compressible flows [10] is implemented and tested. All these numerical meth-
 82 ods were chosen to obtain a simple and efficient incompressible Navier-Stokes
 83 solver. Indeed, the EDAC formulation allows for low-speed flow without the
 84 need of a staggered grid or the use of an iterative method to solve the Poisson
 85 equation for a pressure-like variable. This results in a simpler implementa-
 86 tion of the algorithm as well as a highly scalable solver on a large number of
 87 processors. The use of IBM enable simulations of complex geometries while
 88 maintaining a simple structured Cartesian mesh needed to implement high-
 89 order finite difference schemes. Similarly, the boundary condition by Dong
 90 *et al* [10] enables shorter computational domains, hence reducing the number
 91 of grid points needed for the simulations. In what follows, the mathemat-
 92 ical formulation and numerical discretization of the governing equations is
 93 presented in detail. The solver is then used for classical 2D studies of the
 94 Taylor-Green vortex (TGV), the periodic double-shear layer, and lid-driven
 95 cavity problems. This is followed by extension to the 3D TGV problem and
 96 the 3D lid-driven cavity problem to test the LES model. Finally, an applica-
 97 tion for 3D flow over a sphere at different Reynolds numbers is presented and
 98 efficiency of the solver is assessed by comparisons to results from a traditional
 99 incompressible Navier-Stokes solver.

100 2. Methods

101 2.1. Governing equations

102 The non-dimensional form of the entropically damped artificial compressibil-
103 ity formulation of Clausen [7] is:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial \tau_{ij}}{\partial x_j} \quad (1)$$

104 where u_i is the i^{th} component of the velocity vector, x_i is the i^{th} spatial co-
105 ordinate, p is the pressure, Re is the Reynolds number, and τ_{ij} is the viscous
106 stress tensor. Typical projection methods require solving a Poisson equation
107 for the pressure. EDAC employs an entropy balance to close the system of
108 equations. Assuming an isentropic behavior and minimizing density fluctua-
109 tions, the EDAC equation for pressure is:

$$\frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} = -\frac{1}{M^2} \frac{\partial u_i}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 p}{\partial x_i \partial x_i} \quad (2)$$

110 where M is the Mach number. More details and discussions about the math-
111 ematical derivation of equation 2 can be found in [7]. In all the results
112 presented in this study, the divergence of the velocity was no larger than 0.2
113 locally, with a spatially averaged value lower than 10^{-4} .

114 2.2. Turbulence modeling

115 For 3D studies of transitional or turbulent flows, the low-pass spatially-
116 filtered EDAC equations are solved enabling LES. Hence, only the large-scale
117 flow features are resolved on the grid and effects of the unresolved small-
118 scales on the large-scales are modeled using an eddy viscosity SGS model.
119 The closed filtered EDAC equations are:

$$\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} = -\frac{\partial \tilde{p}}{\partial x_i} + \frac{1}{Re} \frac{\partial \hat{\tau}_{ij}}{\partial x_j} \quad (3)$$

120 where $\hat{\tau}_{ij}$ contains the filtered viscous stresses $\tilde{\tau}_{ij}$ and the SGS tensor $(\tau_{ij})_{SGS}$:

$$\hat{\tau}_{ij} = \tilde{\tau}_{ij} - (\tau_{ij})_{SGS} \quad (4)$$

121 Employing an eddy viscosity based SGS model yields [27]

$$(\tau_{ij})_{SGS} = -2\nu_T S_{ij} \quad (5)$$

122 where

$$S_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \quad (6)$$

123 The eddy viscosity ν_T is specified in terms of gradients of the filtered velocity
 124 field according to the Vreman model [30]. This model is easy to implement,
 125 efficient to use, and applicable to fully inhomogeneous turbulent flows [1, 8,
 126 9, 30]. Following Clausen, the convective derivative in the pressure equation
 127 is neglected as small and the filtered pressure equation is simply:

$$\frac{\partial \tilde{p}}{\partial t} = -\frac{1}{M^2} \frac{\partial \tilde{u}_i}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 \tilde{p}}{\partial x_i \partial x_i} \quad (7)$$

128 2.3. Numerical methods

129 The filtered EDAC equations 3 and 7 are first written in the following con-
 130 servative vector form:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} + \frac{\partial \mathbf{H}}{\partial z} = \mathbf{S} \quad (8)$$

131 where

$$\mathbf{U} = \begin{bmatrix} \tilde{p} \\ \tilde{u} \\ \tilde{v} \\ \tilde{w} \end{bmatrix} \quad (9)$$

$$\mathbf{F} = \begin{bmatrix} -\frac{1}{Re} \frac{\partial \tilde{p}}{\partial x} \\ \tilde{u}^2 + \tilde{p} - \hat{\tau}_{xx} \\ \tilde{u}\tilde{v} - \hat{\tau}_{xy} \\ \tilde{u}\tilde{w} - \hat{\tau}_{xz} \end{bmatrix} \quad (10)$$

$$\mathbf{G} = \begin{bmatrix} -\frac{1}{Re} \frac{\partial \tilde{p}}{\partial y} \\ \tilde{u}\tilde{v} - \hat{\tau}_{yx} \\ \tilde{v}^2 + \tilde{p} - \hat{\tau}_{yy} \\ \tilde{v}\tilde{w} - \hat{\tau}_{yz} \end{bmatrix} \quad (11)$$

$$\mathbf{H} = \begin{bmatrix} -\frac{1}{Re} \frac{\partial \tilde{p}}{\partial z} \\ \tilde{u}\tilde{w} - \hat{\tau}_{zx} \\ \tilde{v}\tilde{w} - \hat{\tau}_{zy} \\ \tilde{w}^2 + \tilde{p} - \hat{\tau}_{zz} \end{bmatrix} \quad (12)$$

$$\mathbf{S} = \begin{bmatrix} \tilde{p} - \frac{1}{M^2} \left(\frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} + \frac{\partial \tilde{w}}{\partial z} \right) \\ \tilde{u} \left(\frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} + \frac{\partial \tilde{w}}{\partial z} \right) \\ \tilde{v} \left(\frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} + \frac{\partial \tilde{w}}{\partial z} \right) \\ \tilde{w} \left(\frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} + \frac{\partial \tilde{w}}{\partial z} \right) \end{bmatrix} \quad (13)$$

132 The semi-discrete form of equation 8 is temporally integrated using the classic
 133 4th–order explicit Runge-Kutta scheme. In order to enable accurate reso-
 134 lution of high-frequencies on relatively coarse grids, while maintaining low
 135 numerical dissipation, the newly developed optimized high-order centered
 136 finite-difference schemes of Linders *et al* [22] are used. According to this
 137 scheme, first derivatives are discretized as:

$$\left(\frac{\partial u}{\partial x}\right)_i + \mathcal{O}(\Delta x^{2p}) = \frac{1}{\Delta x} \sum_{k=1}^{p+n} a_k (u_{i+k} - u_{i-k}) \quad (14)$$

138 In all the results in this paper, the 4th-order accurate version of the family
 139 of schemes is used where $p = 2$, $n = 4$ and the scheme coefficients are:

$$\begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \end{bmatrix} = \begin{bmatrix} 0.896607046646854 \\ -0.320910877852970 \\ 0.119465303396051 \\ -0.037162191039544 \\ 0.008242459236975 \\ -0.000957455525961 \end{bmatrix} \quad (15)$$

140 More details, discussions and proofs regarding these new finite-difference
 141 schemes can be found in [22]. In order to reduce high-frequency spurious
 142 oscillations, a 4th-order spatial filter [2] is applied to the primitive variables
 143 after each time step as shown:

$$(\phi_{filt})_i = \phi_i + 0.1 \left(\sum_{k=1}^6 b_k (\phi_{i+k} + \phi_{i-k}) + b_0 \phi_i \right) \quad (16)$$

144 with

$$\begin{bmatrix} b_0 \\ b_1 \\ b_2 \\ b_3 \\ b_4 \\ b_5 \\ b_6 \end{bmatrix} = \begin{bmatrix} 0.190899511506 \\ -0.171503832236 \\ 0.123632891797 \\ -0.069975429105 \\ 0.029662754736 \\ -0.008520738659 \\ 0.001254597714 \end{bmatrix} \quad (17)$$

145 This filter removes high frequency oscillations but leaves the lower frequencies
 146 intact. It is 13 points wide, which is the same as the stencil used for the
 147 first derivative. Further, it is optimized to filter wave numbers larger than
 148 $\pi/2$, which is desired since the scheme used to compute the first derivative
 149 accurately resolves wave numbers up to precisely $\pi/2$ but no further. Figure
 150 1 shows the performance of the optimized centered scheme with and without
 151 filtering, and compared to a regular 4^{th} order centered scheme. For this test,
 152 the advection equation is solved:

$$\frac{\partial u}{\partial t} + \frac{\partial u}{\partial x} = 0 \quad (18)$$

153 on a $[0; 5]$ domain with a Gaussian pulse initial condition:

$$u(x, t = 0) = \exp(-3200 \times (x - 0.5)^2) \quad (19)$$

154 We can see that the 4^{th} order scheme (black) shows high oscillations near the
 155 pulse and downstream as well. When using 4^{th} order filtering (green), the os-
 156 cillations are reduced downstream of the pulse, but are still very present near

Figure 1: Comparison of centered schemes for the Gaussian pulse problem ($t=4$).

157 the pulse. On the other hand, the optimized centered scheme (blue) shows a
 158 much better agreement with the exact solution. Only some oscillations are
 159 seen downstream of the pulse. After applying filtering (orange), the oscilla-
 160 tions downstream of the pulse disappear. These 1D results motivate the use
 161 of the optimized centered scheme with filtering for the rest of the current
 162 study. Near the boundaries of the computational domain as well as in the
 163 vicinity of the immersed body for the fluxes, a fourth order one sided finite
 164 difference scheme is used to avoid use of extra layers of ghost points. Finally,
 165 the viscous terms are computed by applying twice the centered scheme for
 166 first order derivatives.

167 2.4. Immersed Boundary Method

168 In order to efficiently simulate flows over or through non-Cartesian geome-
 169 tries a mirroring immersed boundary method (IBM) is used [25]. This IBM

has been widely used and validated in our previous studies for a number of internal and external flows [1, 8, 9].

2.5. Boundary conditions

To further enhance numerical efficiency, in particular for external flows on highly truncated computational domains, a newly developed robust energy-stable outflow boundary condition is implemented [10]. Traditionally the use of homogeneous Neumann boundary conditions for velocity and pressure require the computational domain length to be linearly proportional to the Reynolds number for stability. This results in a more expensive simulation since the flow near the outflow is often damped or discarded. Many previous studies have attempted to address this issue [29]. Following the approach of Dong *et al* [10], the boundary condition for the pressure at the outlet plane is:

$$-\tilde{p}\mathbf{n} + \frac{1}{Re}\mathbf{n}\cdot\nabla\tilde{\mathbf{u}} - \left(\frac{1}{2}|\tilde{\mathbf{u}}|^2 S_0(\mathbf{n}\cdot\tilde{\mathbf{u}})\right)\mathbf{n} = 0 \quad (20)$$

with

$$S_0(\mathbf{n}\cdot\tilde{\mathbf{u}}) = \frac{1}{2}\left(1 - \tanh\left(\frac{\mathbf{n}\cdot\tilde{\mathbf{u}}}{U_0\delta}\right)\right) \quad (21)$$

where $\mathbf{n}$ is the outward pointing normal unit vector, U_0 is the characteristic velocity scale, and δ is a small enough positive constant. In the current study, we used $U_0 = 1$ and $\delta = 1/20$. This formulation compensates for any influx of kinetic energy through the outlet plane. The term $(\frac{1}{2}|\tilde{\mathbf{u}}|^2 S_0(\mathbf{n}\cdot\tilde{\mathbf{u}}))\mathbf{n}$ represents the kinetic energy entering the domain and it is balanced by the

189 effective stress $-\tilde{p}\mathbf{n} + \frac{1}{Re}\mathbf{n}\cdot\nabla\tilde{\mathbf{u}}$ on the outlet plane. Similarly an equation for
 190 the velocity gradient at the outlet plane can be derived:

$$\mathbf{n}\cdot\nabla\mathbf{u} = Re\left(p\mathbf{n} + \frac{1}{2}|\mathbf{u}|^2 S_0(\mathbf{n}\cdot\mathbf{u})\mathbf{n} - \frac{1}{Re}(\nabla\cdot\mathbf{u})\mathbf{n}\right) \quad (22)$$

191 More details about the derivation of the formulation and validation results
 192 can be found in [10].

193 **3. Results**

194 *3.1. 2D Taylor-Green Vortex (2D TGV)*

195 In order to estimate the order of accuracy of the hybrid scheme, predictions
 196 of the 2D TGV are compared to the analytic solution. The initial condition
 197 and exact solution for u , v and p are:

$$\begin{aligned} u(x, y, t) &= 1 - \cos(x - t)\sin(y - t)e^{-2t/Re} \\ v(x, y, t) &= 1 + \sin(x - t)\cos(y - t)e^{-2t/Re} \\ p(x, y, t) &= -\frac{1}{4}(\cos(2(x - t)) + \cos(2(y - t)))e^{-4t/Re} \end{aligned} \quad (23)$$

198 The computational domain is a $[0, 2\pi]^2$ periodic square and $Re = 100$. To
 199 estimate the order of accuracy of the solver, comparisons between the numer-
 200 ical and analytical solutions were performed after 10 time steps by computing
 201 the L_2 norm of the error as follow:

$$L_2 = \sqrt{\frac{1}{N_x \times N_y} \sum_{j=1}^{N_y} \sum_{i=1}^{N_x} ((u_{ij})_{analytical} - (u_{ij}))^2} \quad (24)$$

202 The kinetic energy decay as a function of time is shown in Figure 2. The
 203 agreement between the analytical and numerical results is excellent. The L_2
 204 norm of the error for different grid sizes is shown in Figure 2-b, where the
 205 slope of the curve is very close to (-4) , confirming the global spatial order
 206 of accuracy of the solution.

(a) Normalized kinetic energy versus time. (b) L_2 norm of the error between numerical and analytical solutions. The 4th order accuracy can be clearly seen.

Figure 2: Validation and accuracy study of the developed solver.

207 3.2. 2D Double Shear Layer (2D DSL)

208 In order to assess the role of numerical dissipation in the solver and its ability
 209 to resolve sharp spatial gradients at low resolutions, the case of the 2D double
 210 shear layer (DSL) is considered. The 2D DSL initial conditions are:

$$\begin{aligned}
u(x, y) &= \tanh(80 \times (y - 0.25)) & (y \leq 0.5) \\
u(x, y) &= \tanh(80 \times (0.75 - y)) & (y > 0.5) \\
v(x, y) &= 0.05 \times \sin(2\pi(x + 0.25))
\end{aligned} \tag{25}$$

211 The computational domain is a $[0, 1]^2$ square periodic domain and $Re =$
 212 10000. Typically, for this case, under-resolved or highly dissipative simu-
 213 lations produce spurious secondary braid vortices which result in an early
 214 breakdown of the shear layer [24]. Contour plots of vorticity are shown in
 215 Figure 3 at $t = 1$ for two different grid resolutions. Secondary vortices in
 216 the braid region are seen to form in Figure 3-a corresponding to the 64×64
 217 grid due to lack of resolution. In contrast, Figure 3-b for the 128×128 grid
 218 shows no evidence of spurious vortices. Hence, the scheme is able to cap-
 219 ture these strong velocity gradients on this relatively coarse grid compared
 220 to previously published studies which required finer grids [7, 19, 24].

221 3.3. 2D Lid-Driven Cavity (2D LDC)

222 In order to consider wall-bounded flows, the classic 2D lid-driven cavity prob-
 223 lem is studied here for a range of different Reynolds numbers Re ($Re =$
 224 400, 1000, 3200, 5000, 10000). For all cases the computational domain is a
 225 unit square and a 256×256 uniformly spaced grid is used. The simulations
 226 are run until steady state is achieved. Steady-state streamlines are shown in
 227 Figure 4 for all cases and demonstrate that the solver captures both primary
 228 and secondary corner vortices well. As the Reynolds number keeps increas-
 229 ing, the flow becomes more chaotic and smaller pockets of recirculating flow
 230 can be seen (Figure 4-e). The results are quantitatively compared with the

(a) 64×64 points.

(b) 128×128 points.

Figure 3: Contour plots of vorticity at $t = 1$ for the 2D Double shear layer.

231 classic numerical data set of Ghia *et al* [15] as shown in Figure 5. Figures
 232 5-a and b show perfect agreement for low Reynolds numbers. Figures 5-c, d
 233 and e show that the flow becomes more and more chaotic resulting in sharp
 234 velocity gradients near the walls and yet excellent agreement is maintained.

235 3.4. 3D Taylor Green Vortex (3D TGV)

236 For the 3D TGV case, the computational domain is a $[-\pi, \pi]^3$ periodic cube
 237 with initial conditions:

$$\begin{aligned}
 u(x, y, z) &= \sin(x)\cos(y)\cos(z) \\
 v(x, y, z) &= \cos(x)\sin(y)\cos(z) \\
 w(x, y, z) &= 0 \\
 p(x, y, z) &= p_0 + \frac{1}{16} (\cos(2x)\cos(2y)\cos(2z) + 2)
 \end{aligned}
 \tag{26}$$

Figure 4: 2D Lid-Driven cavity. u -velocity contour with selected streamlines inside the cavity.

238 For adequate resolution and numerical accuracy, these initial vortices will in-
 239 teract and breakdown with time to form a turbulent flow. Comparisons can
 240 be made to direct numerical simulation (DNS) data from previous published
 241 studies [3]. Figure 6-a shows the initial vortices inside the cubical compu-
 242 tational domain. The simulations are run on a 128^3 grid with $Re = 1600$
 243 and the Vreman SGS model is used enabling LES. As the simulation pro-
 244 gresses, the vortices interact with each other and eventually break down into
 245 smaller scales resulting in a near-homogeneous turbulent state. Figure 6-b
 246 shows smaller-scale vortices at $t = 10$. The kinetic energy decay rate from the
 247 present EDAC-based LES (Figures 7-a and b) compares well to the DNS data
 248 of Brachet [3]. Specifically, on this relatively coarse grid, the results do not
 249 show an over-prediction of the decay, confirming the low level of numerical
 250 dissipation of the solver.

251 3.5. 3D Lid-Driven Cavity (3D LDC)

252 Here the 3D LDC case at $Re = 12000$ is considered. A unit cube compu-
 253 tational domain is discretized with an 80^3 grid. The Vreman SGS model is
 254 used enabling LES. No-slip boundary conditions are used on all but the top
 255 face where the wall velocity is specified as [21]:

$$u(x, 1, z) = (1 - (x - 1)^{18})^2 (1 - (z - 1)^{18})^2 \quad (27)$$

256 The LES results are compared to DNS data from Leriche *et al* [21] and
 257 are used to assess accuracy of the numerical scheme, as well as the SGS
 258 turbulence model. Figure 8-a depicts a snapshot of vortical structures within
 259 the cavity at $t = 100$. The solver is clearly capable of capturing a wide range

260 of vortical structures with high vorticity magnitude near the top wall and
 261 near the left surface where the impact of the moving wall is the strongest.
 262 Figure 8-b shows very good agreement for mean velocity profiles between the
 263 LES (averaged over 1000 time units) and the DNS data.

264 3.6. 2D Flow over a Cylinder

265 To test the solver using the IBM and the energy-stable outflow boundary
 266 condition [10], 2D flow over a cylinder at $Re = 1500$ on a severely truncated
 267 domain is first considered. The computational domain is $[-2D, 2.5D] \times$
 268 $[-2.5D, 2.5D]$ and is discretized using a uniform 90×128 grid. The cylinder
 269 is placed inside a channel, resulting in no slip boundary condition on the
 270 top and bottom boundaries. Instantaneous vorticity contour plots at several
 271 times are shown in Figure 9. The results shown on the left column were
 272 obtained using a standard homogeneous Neumann boundary condition at
 273 the outlet whereas the results on the right column were obtained using the
 274 energy-stable outflow boundary condition from Dong *et al.* At the beginning
 275 of the simulations (Figures 9-a to d), the two methods show similar solutions,
 276 the outlet boundary not affecting the upstream solution at this point since no
 277 vortex has crossed the outlet plane yet. At a later time (Figures 9-e and f),
 278 the two solutions are very different. The results obtained using the energy-
 279 stable outflow boundary condition show a typical vortex shedding without
 280 any re-circulation or inflow coming from the outlet plane. In contrast, the
 281 case with the Neumann boundary condition, clearly shows some back-flow
 282 at the outlet, which affects the upstream flow and considerably alters the
 283 solution. This phenomenon only gets worse with time resulting in additional
 284 back-flow at the outlet (Figure 9-g) and eventual numerical instability of the

285 solver. This is not observed with the other boundary condition (Figure 9-h):
 286 the vortices that are generated by the presence of the cylinder are leaving
 287 the computational domain without affecting the rest of the flow, keeping the
 288 solution stable. These two simulations validate the implementation of the
 289 energy stable outflow boundary condition, and show its positive effect on the
 290 stability of the solution. This feature is critical when trying to improve the
 291 computational efficiency of the solver.

292 3.7. 3D Flow over a Sphere

293 Finally, 3D flow over a sphere at $Re = 200$ and at $Re = 1500$ are considered.
 294 For both cases, the computational domain size is $[-4D, 25D] \times [-3D, 3D] \times$
 295 $[-3D, 3D]$ and is discretized using a $200 \times 100 \times 100$ uniform grid. The results
 296 are shown in Figure 10. Figure 10-a shows the out-of-plane vorticity along
 297 the $y = 0$ slice for the $Re = 200$ case where the flow is laminar and steady and
 298 the vorticity shows top-bottom symmetry in this plane. Figure 10-b shows
 299 very good agreement between the current numerical predictions for C_p at the
 300 surface of the sphere with the DNS data of Fadlun *et al* [11]. This validates
 301 the implementation of the IBM in the context of the EDAC formulation.
 302 Figure 10-c shows an iso-surface of λ_2 colored by vorticity magnitude for the
 303 sphere case at $Re = 1500$. The solver captures the complex wake structure
 304 downstream of the sphere including hairpin-like structures near the sphere
 305 which break down further downstream.

306 3.8. Algorithm Efficiency Assessment

307 In order to assess the computational efficiency of the overall solver, several
 308 tests were run. First, the scalability of the solver was studied for the 3D LDC

309 problem. It is parallelized using Message Passing Interface (MPI). Figure
 310 11-a shows the scalability for two different tests: in black we can see the
 311 scalability of the solver when the number of total grid points is maintained
 312 constant (51.2 millions) and the number of processors varies. It can be seen
 313 that the solver scales linearly with performances close to the ideal scalability.
 314 In red we can see that scalability of the solver when the load of each processor
 315 is maintained constant. Once again it can be seen that the performance of
 316 the solver stays very good, even as the number of processors increases up to
 317 1000.

318 Now, comparisons are also made to a traditional predictor/corrector incom-
 319 pressible Navier-Stokes code. The exact same numerical methods are used
 320 in this new solver as in the EDAC solver. Both solvers are parallelized
 321 using Message Passing Interface (MPI) and the Poisson equation in the
 322 predictor/corrector incompressible code is solved using the Hypre library
 323 [12, 13, 14]. Both solvers showed linear scaling on up to 800 processors. The
 324 efficiency comparison between the two solvers is carried out for the 3D LDC
 325 problem on 512 processors. The simulation speed up is defined as:

$$\text{speedup} = \frac{(t_{1s})_{pred/correc} - (t_{1s})_{EDAC}}{(t_{1s})_{pred/correc}} \quad (28)$$

326 where $(t_{1s})_{EDAC}$ is the computational time to reach 1s of simulation using
 327 the EDAC code and $(t_{1s})_{pred/correc}$ is the computational time to reach 1s
 328 of simulation using the predictor/corrector code. The results are shown in
 329 Figure 11-b where speed up versus grid size is compared. It is clear that on
 330 a 64^3 grid, the EDAC code is about 30% faster than the predictor/corrector
 331 solver. As the grid size increases, the speed up of the solver decreases slightly

332 to around 27% for the 128^3 grid, 23% for the 256^3 grid, and 15% for the
 333 1024^3 . The improved performance of the EDAC code is primarily due to
 334 avoidance of the Poisson solver, which makes the computation time per time
 335 step much smaller even though the global time step has to be slightly reduced
 336 for stability reasons. As the number of grid points increases, the time step
 337 used in the EDAC solver had to be reduced faster than for the incompressible
 338 solver, which affects the overall speed up. But the drop in efficiency is only
 339 seen for very large grid (around a billion point mesh), making this approach
 340 suitable for practical applications. Moreover, the advantage of the EDAC
 341 formulation for simulations of incompressible flows is the fact that no elliptic
 342 equations need to be solved, making parallel implementation easier. The
 343 use of hybrid grids such as Adaptive Mesh Refinement (AMR), curvilinear
 344 meshes or multiple resolution by block is also facilitated. In conclusion, the
 345 EDAC formulation when combined with numerical methods selected for this
 346 study demonstrate a relatively simple, efficient, and accurate incompressible
 347 flow solver.

348 **4. Conclusion**

349 An efficient algorithm was implemented for the simulation of laminar and tur-
 350 bulent incompressible internal and external flows. An inherently unsteady
 351 and stable version of the ACM, from Clausen [7] was used. This avoids
 352 the need for a Poisson solver or dual-time stepping resulting in a very effi-
 353 cient numerical scheme. Spatial discretization on structured Cartesian grids
 354 was achieved using recent high-order centered finite-difference schemes op-
 355 timized for simulating flows featuring a wide range of wave numbers. For

356 non-Cartesian geometries a mirroring immersed boundary method was used.
 357 For turbulent flows, an eddy viscosity based subgrid-scale modeled was used
 358 to enable large eddy simulations. For external flows, a recent energy stable
 359 outflow boundary condition was used enabling stable high Reynolds number
 360 simulations on highly truncated domains. Both 2D and 3D periodic, inter-
 361 nal, and external flows were simulated and compared to previously published
 362 data for validation. The results show excellent high-resolution capability on
 363 relatively coarse grids.

364 **Acknowledgment**

365 The authors would like to acknowledge Dr. Jonathan Clausen for the dis-
 366 cussions related to the EDAC method. The authors would also like to ac-
 367 knowledge financial support for this work from the Rosenblatt Chair within
 368 the faculty of Mechanical Engineering and Zeff Fellowship Trust.

369 **References**

- 370 [1] Anupindi K, Delorme Y, Shetty D and Frankel S 2013, *A novel multi-*
 371 *block immersed boundary method for large eddy simulation of complex*
 372 *arterial hemodynamics*, Journal of Computational Physics, 254:200–218
- 373 [2] Bogey C and Bailly C 2004, *A family of low dispersive and low dis-*
 374 *sipative explicit schemes for flow and noise computations*, Journal of
 375 Computational Physics, 194:194–214
- 376 [3] Brachet M 1991, *Direct numerical simulation of three dimensional tur-*
 377 *bulence in the Taylor Green vortex*, Fluid Dynamic Research, 8:1–9

- 378 [4] Chen K and Pletcher R 1993, *Simulation of three dimensional liquid*
 379 *sloshing flows using a strongly implicit calculation procedure*, AIAA
 380 Journal, 31:901–910
- 381 [5] Choi D and Merkle C 1985, *Application of time iterative schemes to*
 382 *incompressible flow*, AIAA Journal, 23:1518–1524
- 383 [6] Chorin A 1967, *A numerical method for solving incompressible viscous*
 384 *flow problems*, Journal of Computational Physics, 135:118–125
- 385 [7] Clausen J 2013, *Entropically damped form of artificial compressibility for*
 386 *explicit simulation of incompressible Flow*, Physical Review E, 87:13309–
 387 13321
- 388 [8] Delorme Y, Anupindi K, Kerlo A.E, Shetty D, Rodefeld M, Chen J and
 389 Frankel S 2013, *Large eddy simulation of powered Fontan hemodynamics*,
 390 Journal of Biomechanics, 46:408–422
- 391 [9] Delorme Y, Anupindi K and Frankel S 2013, *Large eddy simulation of*
 392 *FDA’s idealized medical device*, Cardiovascular Engineering and Tech-
 393 nology, 4:4
- 394 [10] Dong S, Karniadakis G and Chrysosostomidis C 2014, *A robust and ac-*
 395 *curate outflow boundary condition for incompressible flow simulations*
 396 *on several truncated unbounded domains*, Journal of Computational
 397 Physics, 261:83–105
- 398 [11] Fadlun E, Verzicco R, Orlandi P and Mohd-Yusof J 2000, *Combined im-*
 399 *mersed boundary finite difference methods for three dimensional complex*
 400 *flow simulations*, Journal of Computational Physics, 161:35–60

- 401 [12] Falgout R and Jones J, *Multigrid on massively parallel architectures*
- 402 [13] Falgout R and Yang 2002, *Hypre: a library of high performance preconditioners*
- 403
- 404 [14] <http://acts.nersc.gov/hypre/>
- 405 [15] Ghia U, Ghia K and Shin C 1982, *High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method*, Journal of Computational Physics, 48:387–411
- 406
- 407
- 408 [16] Hafez M, South J and Murman E, *Artificial compressibility methods for numerical solutions of transonic Full Potential Equation*, AIAA Journal, 17:838–844
- 409
- 410
- 411 [17] Hartwich P and Hsu C 1988, *High resolution upwind schemes for the three dimensional incompressible Navier Stokes equations*, AIAA Journal, 26:1321–1328
- 412
- 413
- 414 [18] Hartwich P, Hsu C and Liu C 1988, *Vectorizable implicit algorithms for the flux difference split three dimensional Navier Stokes equations*, Journal of Fluids Engineering, 110:297–305
- 415
- 416
- 417 [19] Hashimoto T, Tanno I, Yasuda T, Tanaka Y, Morinishi K and Satofuka N 2015, *Higher order numerical simulation of unsteady viscous incompressible flows using kinetically reduced local Navier Stokes equations on a GPU*, Computers and Fluids, 110:108-113
- 418
- 419
- 420
- 421 [20] Kwak D, Chang J, Shanks P and Chakravarthy S 1986, *A three dimensional*

- 422 *sional incompressible Navier Stokes flow solver using primitive variables,*
 423 AIAA Journal, 24:390–396
- 424 [21] Leriche E and Gavrilakis S 2000, *Direct numerical simulation of the flow*
 425 *in a lid-driven cubical cavity*, Physics of Fluids, 12:1363-1376.
- 426 [22] Linders V and Nordstrom J 2015, *Uniformly best wave number approxi-*
 427 *mations by spatial central difference operators*, Journal of Computational
 428 Physics, *In press*
- 429 [23] Madsen P and Shaffer H 2006, *A discussion of artificial compressibility,*
 430 Coastal Engineering, 53:93–98
- 431 [24] Minion M and Brown D 1997, *Performance of under-resolved two-*
 432 *dimensional incompressible flow simulations, II*, Journal of Computa-
 433 tional Physics, 138:734–765
- 434 [25] Mark A and van Wachem B 2008, *Derivation and validation of a novel*
 435 *implicit second order accurate immersed boundary method*, Journal of
 436 Computational Physics, 227:6660–6680
- 437 [26] Pan D and Chakravarthy S 1989, *Unified formulation for incompressible*
 438 *Flows*, AIAA
- 439 [27] Pope S 2000, *Turbulent flows*, Cambridge University Press
- 440 [28] Rogers S, Kwak D and Kiris C 1989, *Numerical solution of the incom-*
 441 *pressible Navier Stokes equations for steady state and time dependent*
 442 *problems*, AIAA

- 443 [29] Tsynkov S 1998, *Numerical solution of problems on unbounded domains:*
444 *A review*, Applied Numerical Mathematics, 27:465–532
- 445 [30] Shetty D, Fisher T, Chunekar A and Frankel S 2010, *High order incom-*
446 *pressible large eddy simulation of fully inhomogeneous turbulent flows*,
447 Journal of Computational Physics, 229:8802–8822
- 448 [31] Steger J and Kulkarni P 1977, *Implicit finite difference procedures for the*
449 *computation of vortex wakes*, AIAA Journal, 15:581–590
- 450 [32] Turkel E 1993, *Review of preconditioning methods for fluid dynamics*,
451 Applied Numerical Mathematics, 12:257–284
- 452 [33] Vrahliotis S, Pappou T and Tsangaris S 2012, *Artificial compressibil-*
453 *ity 3D Navier Stokes solver for unsteady incompressible flows with hy-*
454 *brid grids*, Engineering Applications of Computational Fluid Mechanics,
455 6:248–270

(a) $Re = 400$.

(b) $Re = 1000$.

(c) $Re = 3200$.

(d) $Re = 5000$.

(e) $Re = 10000$.

Figure 5: 2D Lid-Driven cavity. Comparisons between current results and DNS data from Ghia *et al.*

(a) Iso-surface of λ_2 colored by vorticity magnitude ($t = 0$). (b) Iso-surface of λ_2 colored by vorticity magnitude ($t = 10$).

Figure 6: 3D Taylor-Green vortices. Vortical structures.

(a) Decay of kinetic energy compared to (b) Kinetic energy decay rate compared to DNS data of Brachet.

Figure 7: 3D Taylor-Green vortices. Comparison of kinetic energy decay predicted by the current EDAC solver (black line) with DNS data of Brachet (Red dots).

(a) Iso-surface of λ_2 colored by vorticity magnitude (t=100). (b) Comparison between current predictions and DNS data.

Figure 8: 3D Lid-Driven cavity. Vortical structures and comparisons with DNS data (Leriche *et al* [21]).

(a) Homogeneous Neumann (b) Dong *et al* BC outflow BC ($t = t_0$).

(c) Homogeneous Neumann (d) Dong *et al* BC outflow BC ($t = t_1$).

(e) Homogeneous Neumann (f) Dong *et al* BC outflow BC ($t = t_2$).

(g) Homogeneous Neumann (h) Dong *et al* BC outflow BC ($t = t_3$).

Figure 9: 2D cylinder on a truncated domain. Out of plane vorticity.

(a) Out of plane vorticity at a centerplane of the domain ($y=0$). $Re = 200$.

(b) Comparison between current predictions and DNS data from Fadlun *et al.* $Re = 200$.

(c) Iso-surface of λ_2 colored by vorticity magnitude. $Re = 1500$.

Figure 10: 3D flow over sphere (laminar and turbulent).

(a) Scaling of the solver with a constant number of grid points or a constant load. (b) Comparison of the efficiency between the developed EDAC solver and a traditional predictor / corrector approach.

Figure 11: Performances of the developed solver.