

AN EDGE BASED FINITE VOLUME APPROACH FOR THE SOLUTION OF THE INCOMPRESSIBLE NAVIER-STOKES EQUATIONS ON UNSTRUCTURED TRIANGULAR MESHES

Furkan Oz* and Mehmet Sahin†
Istanbul Technical University
Istanbul, Turkey

ABSTRACT

A face based unstructured finite volume is presented for the solution of the incompressible Navier-Stokes equation on unstructured triangular meshes. The numerical method is based on the stable side-centered arrangement of primitive variables in which the velocity vector components are located at edge mid-points, meanwhile the pressure term is placed at element centroids. A special attention is given to accurately evaluate the viscous terms at the mid-point of the control volume faces. The convected terms are evaluated using the least square upwind interpolations. The resulting algebraic equations are solved in a monolithic manner. The implementation of the preconditioned Krylov subspace algorithm, matrix-matrix multiplication and the restricted additive Schwarz preconditioner are carried out using the PETSc software package in order to improve the parallel performance. The numerical method is validated for the classical benchmark problem of lid-driven cavity in a square enclosure. The numerical results indicate better accuracy for the viscous fluxes on triangular meshes.

INTRODUCTION

A face based unstructured finite volume formulation has been developed for the solution of the incompressible Navier-Stokes equation on unstructured triangular meshes in a fully coupled form, where the velocity vector components are defined at the center of edges, meanwhile the pressure term is defined at the element centroid. This face based approach was initially used on boundary-fitted meshes [Maliska and Raithby, 1984] and later, on unstructured triangular meshes [Hwang, 1995; Rida et al., 1997]. The same approach is employed within the finite element framework by using the stable non-conforming Q_1/Q_0 finite element pair [Rannacher and Turek, 1992]. The present face-centered approach has also been initially implemented on quadrilateral and hexagonal meshes within the Arbitrary Lagrangian-Eulerian (ALE) framework for the solution of the moving boundary problems with large displacements and rotations [Erzincanli and Sahin, 2013].

In the current work, the approach has been extended to unstructured triangular meshes in

*Bsc. in Aeronautical Engineering Department, Email: ozf@itu.edu.tr

†Prof in Astronautical Engineering Department, Email: msahin@itu.edu.tr

two-dimensions. Although the use of triangular elements leads to increase in both edge numbers and element numbers, which leads a larger algebraic equation compared to quadrilateral elements, it allows further flexibility on mesh modifications for large deformations. In addition, the present formulation on triangular meshes leads to a symmetric mass matrix in contrast to quadrilateral elements. Furthermore, the velocity gradients required for the viscous fluxes are exactly computed at the control volume edge mid-points on highly irregular unstructured triangular meshes, which leads a more accurate numerical algorithm. The present algorithm has been implemented in an object oriented framework in C and it allows us to use the full functionality of the PETSc 3.7.7 library [Balay et al., 2018].

MATHEMATICAL AND NUMERICAL FORMULATION

The governing equations for the incompressible fluid flow in the Cartesian coordinate system can be written in the following form: the continuity equation

$$-\nabla \cdot \mathbf{u} = 0 \quad (1)$$

the momentum equations

$$\rho \left[\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right] + \nabla p = \nabla \cdot \mathbf{T} \quad (2)$$

In these equations, ρ represents the fluid density, \mathbf{u} is the velocity vector, p is the pressure and $\mathbf{T} = \mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)$ is the viscous stress tensor. Integrating the differential equations (1) and (2) over an unstructured triangular element Ω_e with boundary $\partial\Omega_e$ gives the following non-dimensional equations:

$$-\oint_{\partial\Omega_e} \mathbf{n} \cdot \mathbf{u} dS = 0 \quad (3)$$

$$Re \int_{\Omega_d} \frac{\partial \mathbf{u}}{\partial t} dV + Re \oint_{\partial\Omega_d} (\mathbf{n} \cdot \mathbf{u}) \mathbf{u} dS + \oint_{\partial\Omega_d} \mathbf{n} p dS = \oint_{\partial\Omega_d} \mathbf{n} \cdot \nabla \mathbf{u} dS \quad (4)$$

The \mathbf{n} represents the outward normal unit vector, V is the control volume, S is the control volume surface area and Re is the non-dimensional Reynolds number. Figure 1-[a] illustrates typical two neighbouring triangular elements with an arbitrary dual control volume for the momentum equation (4) constructed by connecting the element centroids to the common vertices shared by the both triangular elements. The velocity vector components are defined at the mid-point of each cell face while the pressure is defined at the element centroids. The momentum equations along the x - and y -directions are discretized over the dual finite volume shown in Figure 1-[a]. The discrete contribution from the right cell shown in Figure 1-[a] is provided below for each term of the momentum equation along the x -direction.

The time derivation

$$Re \left[\frac{7u_1^{n+1} + u_2^{n+1} + u_3^{n+1}}{9\Delta t} \right] A_{123} - Re \left[\frac{7u_1^n + u_2^n + u_3^n}{9\Delta t} \right] A_{123} \quad (5)$$

The convective term

$$Re [u_{12}^n \Delta y_{12} - v_{12}^n \Delta x_{12}] u_{12}^{n+1} + Re [u_{23}^n \Delta y_{23} - v_{23}^n \Delta x_{23}] u_{23}^{n+1} \quad (6)$$

The pressure term

$$\left[\frac{p_1^{n+1} + p_2^{n+1}}{2} \right] \Delta y_{12} + \left[\frac{p_2^{n+1} + p_3^{n+1}}{2} \right] \Delta y_{23} \quad (7)$$

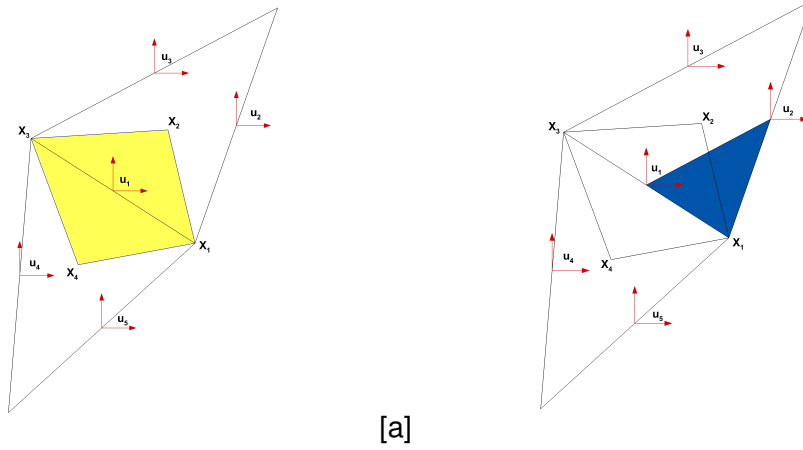


Figure 1: Unstructured triangular mesh with control volume (yellow) [a] and co-volume for viscous gradients (blue) [b].

The viscous term

$$-\left(\frac{\partial u}{\partial x}\right)_{12}^{n+1} \Delta y_{12} - \left(\frac{\partial u}{\partial x}\right)_{23}^{n+1} \Delta y_{23} + \left(\frac{\partial u}{\partial y}\right)_{12}^{n+1} \Delta x_{12} + \left(\frac{\partial u}{\partial y}\right)_{23}^{n+1} \Delta x_{23} \quad (8)$$

In here, A_{123} is the area between the points x_1, x_2 and x_3 , Δt is the time step, $\Delta x_{12} = x_2 - x_1$, $\Delta x_{23} = x_3 - x_2$, $\Delta y_{12} = y_2 - y_1$, $\Delta y_{23} = y_3 - y_2$, the values u_{12}, u_{23}, v_{12} and v_{23} are the velocity vector components defined at the mid-point of each dual volume edge and p_1, p_2 and p_3 are the pressure values at the points x_1, x_2 and x_3 , respectively. The velocity vector components u_{12}, u_{23}, v_{12} and v_{23} are computed using the unweighted least square interpolations [Anderson and Bonhaus, 1994; Barth, 1991]. As an example,

$$u_{12} = \beta [u_1 + \nabla u_1 \mathbf{r}_1] + (1 - \beta) [u_2 + \nabla u_2 \mathbf{r}_2] \quad (9)$$

where β is a weight factor determining the type of convection scheme used, ∇u_1 and ∇u_2 are the gradients of velocity components where the u_1 and u_2 velocity components are defined and \mathbf{r}_1 and \mathbf{r}_2 are the distance vectors from the mid-point of the dual control volume face to the locations where the gradients of velocity components are computed. For evaluating the gradient terms, ∇u_1 and ∇u_2 , a least square procedure is used in which the velocity data is assumed to behave linearly. Referring to Figure 1-[a] as an example, the following system can be constructed for the term ∇u_1

$$\begin{bmatrix} \Delta x_{21} & \Delta y_{21} \\ \Delta x_{31} & \Delta y_{31} \\ \Delta x_{41} & \Delta y_{41} \\ \Delta x_{51} & \Delta y_{51} \\ \vdots & \vdots \end{bmatrix} \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \end{bmatrix} = \begin{bmatrix} u_2 - u_1 \\ u_3 - u_1 \\ u_4 - u_1 \\ u_5 - u_1 \\ \vdots \end{bmatrix} \quad (10)$$

where u_i corresponds to the edges connected to the vertices of the upwind edge u_1 . This overdetermined system of linear equations may be solved in a least square sense using the normal equation approach, in which both sides are multiplied by the transpose. The modified system is solved using the singular value decomposition provided by the Intel Math Kernel Library in order to avoid the numerical difficulties associated with solving linear systems with near rank deficiency. The pressure values at x_1 and x_3 as well as the velocity values at these nodes are computed in a similar manner. Then velocity gradients at the control volume edge mid-points is evaluated from the Green's theorem as shown in Figure 1-[b].

$$\frac{\partial u}{\partial x} = \frac{1}{A} \oint_{\partial \Omega_c} u dy \quad (11)$$

$$\frac{\partial u}{\partial y} = -\frac{1}{A} \oint_{\partial\Omega_c} u dx \quad (12)$$

In here, the evaluated gradient term is exactly located at the control volume edge mid-points in contrast to the gradient evaluation on quadrilateral elements. The contribution from the left cell is also calculated in a similar manner. The continuity equation (3) is integrated within each triangular elements and evaluated using the mid-point rule on each of the element edges.

$$\sum_{j=1}^3 [u_j^{n+1} \Delta y_j - v_j^{n+1} \Delta x_j] = 0 \quad (13)$$

where Δx_j and Δy_j are the element edge lengths along the x - and y -directions, respectively and u_j and v_j are the velocity vector components defined at the mid-point of each triangular element face. The resulting algebraic equations are solved in a monolithic manner [Erzincanli and Sahin, 2013]. The implementation of the preconditioned Krylov subspace algorithm, matrix-matrix multiplication and the restricted additive Schwarz preconditioner are carried out using the PETSc software package in order to improve the parallel performance [Balay et al., 2018].

RESULTS AND DISCUSSION

The numerical method is applied to the classical lid-driven cavity problem in a square enclosure at $Re = 0$. The reason for $Re = 0$ is that the pressure field is almost constant in the lower part of the cavity and any odd-even pressure decoupling can be directly seen from the pressure contours. The computational initial mesh consists of 40,401 vertices and 40,000 quadrilateral elements which leads to 200,800 DOF. The same quadrilateral mesh is split into triangles, which leads to 80,000 triangular elements with 320,800 DOF. The comparison of the computed u -velocity vector components and the pressure contours on both meshes are provided in Figure 2 and 3, respectively. The both numerical results are free from odd-even oscillations due to the employed staggered arrangement of the primitive variables. The calculation wall times are 49.48s and 22.23s on triangular and quadrilateral meshes, respectively. The number of ILU(4) preconditioned FGMRES iterations to reach 1×10^{-8} for triangular and quadrilateral meshes is 197 and 189, respectively.

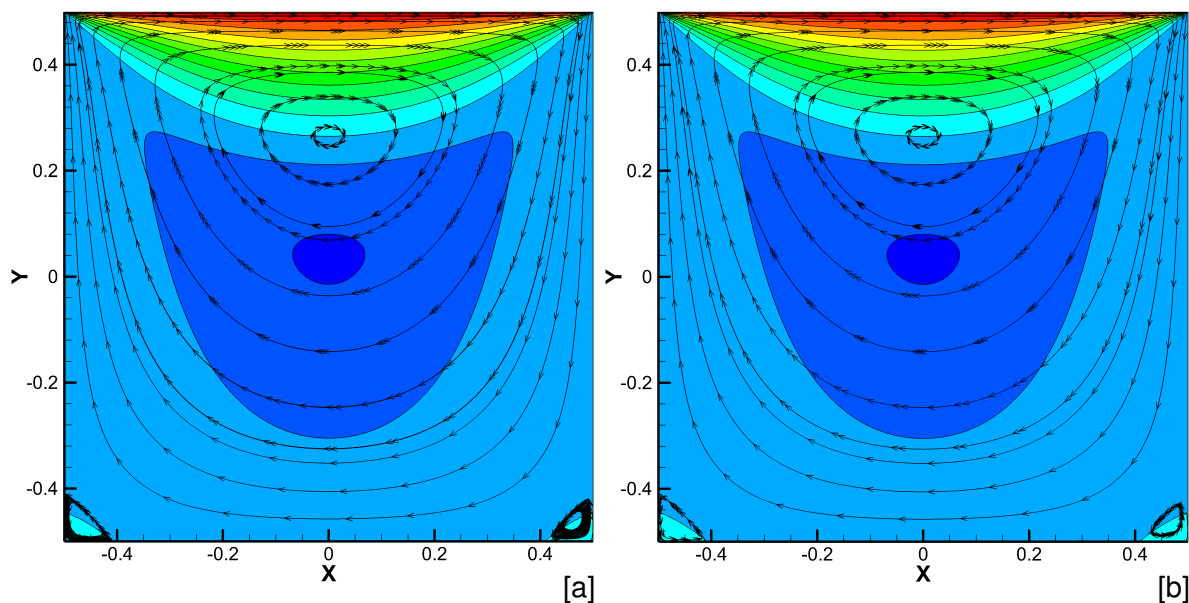


Figure 2: The computed u -velocity components on triangular [a] and quadrilateral [b] meshes at $Re = 0$.

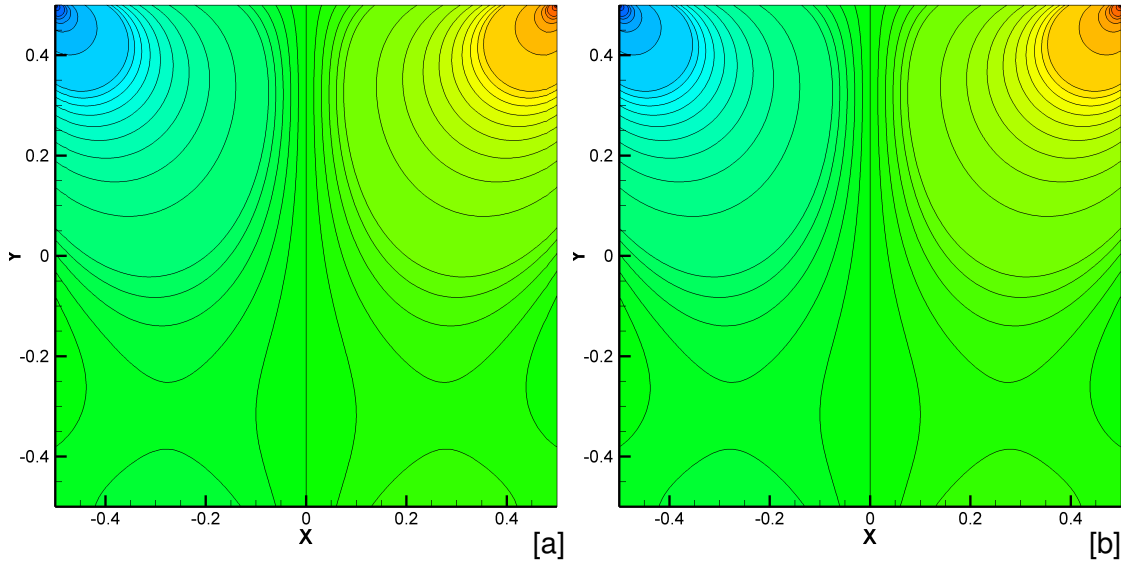


Figure 3: The computed pressure contours on triangular [a] and quadrilateral [b] meshes at $Re = 0$.

The numerical method is applied to the classical lid-driven cavity problem in a square enclosure one more time at $Re = 1000$ in order to compare results with values taken from related paper [Botella and Peyret, 1998]. Corresponding u -velocity profile at $x = 0$ line and v -velocity profile at $y = 0$ can be seen in Figure 4-[a] and Figure 4-[b], respectively. The results show that velocity profiles taken from the algorithm are very close to the ones taken from paper. The mean percentage errors are 0.28% for $x = 0$ velocity profile and 0.39% for $y = 0$ velocity profile.

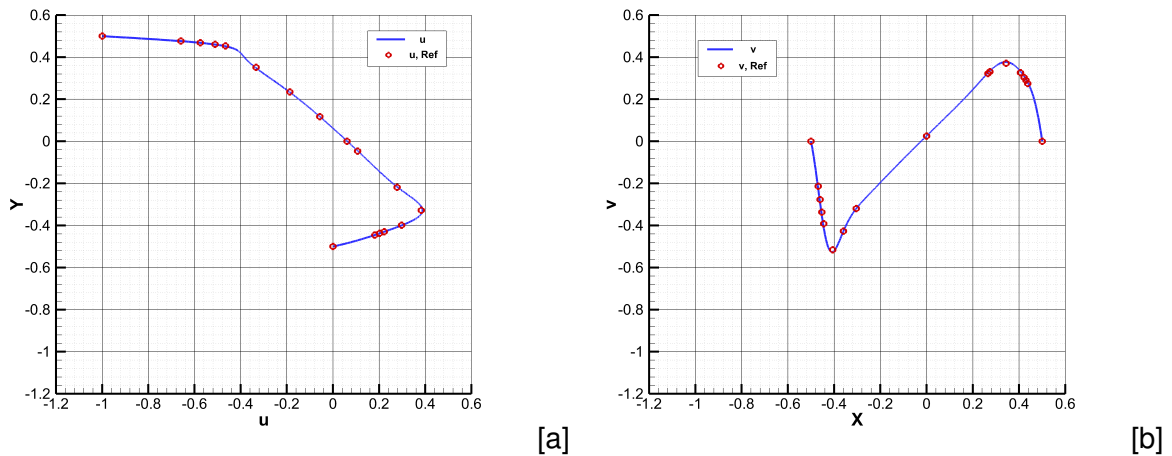


Figure 4: u -velocity profiles at $x = 0$ [a] and v -velocity profiles at $y = 0$ [b] in $Re = 1000$.

CONCLUSIONS

A faced based unstructured finite volume has been developed for the solution of the incompressible Navier-Stokes equation on unstructured triangular meshes. The numerical method is based on the stable side-centered arrangement of primitive variables in which the velocity vector components are located at edge mid-points, meanwhile the pressure term is placed at element centroids. A special attention is given in order to accurately evaluate the viscous terms at the mid-point of the control volume edges. The resulting algebraic equations are solved in

a fully coupled (monolithic) manner. The numerical results indicates a significant improvement on the numerical accuracy, meanwhile the computational time is increased in comparison to quadrilateral elements.

References

- Anderson, W. K. and Bonhaus, D. L. (1994) An implicit upwind algorithm for computing turbulent flows on unstructured grids. *Comp. & Fluids*, **23**:1–21.
- Balay, S., Abhyankar, S., Adams, M. F., et al. (2018) PETSc Users Manual. <http://www.mcs.anl.gov/petsc>
- Barth, T. J. (1991) A 3-D upwind Euler solver for unstructured meshes. *AIAA Paper 91-1548-CP*.
- Botella, O. and Peyret, R. (1997) Benchmark spectral results on the lid-driven cavity flow. *Comp. & Fluids*, **27**, 4:421-433.
- Erzincanli, B. and Sahin, M. (2013) An arbitrary Lagrangian-Eulerian formulation for solving moving boundary problems with large displacement and rotations. *J. Comput. Physics*, 255:660-679.
- Hwang, Y. H. (1995) Calculations of incompressible flow on a staggered triangle grid, Part I: Mathematical formulation. *Numer. Heat Transfer B*, **27**:323–1995.
- Maliska, C. R. and Raithby, G. D. (1984) A method for computing three dimensional flows using non-orthogonal boundary-fitted co-ordinates. *Int. J. Numer. Meth. Fluids*, **4**:519–537.
- Rannacher, R. and Turek, S. (1992) A simple nonconforming element. *Numer. Meth. PDEs*, **8**:97–111.
- Rida, S., McKenty, F., Meng, F. L. and Reggio, M. (1997) A staggered control volume scheme for unstructured triangular grids. *Int. J. Numer. Meth. Fluids*, **25**:697–717.