XXXVII Convegno Nazionale di Idraulica e Costruzioni Idrauliche

Reggio Calabria, 7-9 Settembre 2020

MAST-RT0 SOLUTION OF 3D NAVIER STOKES EQUATIONS ON UNSTRUCTURED MESHS. PRELIMINARY RESULTS IN THE LAMINAR CASE

Costanza Aricò*, Marco Sinagra, Calogero Picone, Paolo Picone & Tullio Tucciarelli

Dipartimento di Ingegneria, Università di Palermo

**email: costanza.arico@unipa.it*

KEY POINTS

- *A new numerical solver for the 3D Navier Stokes incompressible laminar problems is proposed*
- *The governing equations are discretized over unstructured tetrahedral meshes*
- *Delaunay mesh condition is not required*
- *The proposed model is suitable for the solution of laminar flows in irregular domains*

1 INTRODUCTION

The 3D Navier-Stokes equations (NSEs) for incompressible and laminar flows in domains with complex boundaries govern many physical and engineering problems, e.g., low speed pumps, low Mach number airfoils and ship propellers, blood in capillary, arteries and veins.

A new numerical method to solve the 3D Navier–Stokes equations for incompressible viscous laminar flows is presented in the present paper. The new procedure is based upon a fractional time step procedure and the finite volume method on unstructured meshes.

Unstructured meshes are suitable to discretize irregular real physical domains. The computational domain is discretized using tetrahedral meshes and the Delaunay mesh conditions is not strictly required.

Preliminary results of the new proposed scheme are provided.

2 THE GOVERNING EQUATIONS AND THE SPATIAL DISCRETIZATION

We solve the following dimensionless 3D Navier - Stokes Equations for a real and incompressible fluid,

$$\frac{\partial \mathbf{U}}{\partial \tilde{t}} + \left(\mathbf{U}\cdot\tilde{\nabla}\right)\mathbf{U} - \frac{1}{Re}\tilde{\nabla}_2\mathbf{U} + \tilde{\nabla}\Psi = 0 \qquad (1,a),$$

$$\nabla\cdot\mathbf{U} = 0 \qquad (1,b),$$

where Eq. (1,a) is the momentum conservation equation, and Eq. (1,b) is the mass conservation equation. Eqq. (1) are non-dimensionalized selecting the appropriate scales, length L, flow velocity $\mathbf{u}^*$, time $L/\mathbf{u}^*$, and pressure $\rho\mathbf{u}^{*2}$ (ρ is the fluid density). The associated dimensionless variables are, spatial coordinates $\tilde{\mathbf{x}}$ = $\mathbf{x}/L$, differential operator $\tilde{\nabla} = \nabla L$, time $\tilde{t}$ = $t/(L/\ \mathbf{u}^*)$, flow velocity $\mathbf{U}$ = $\mathbf{u}/\mathbf{u}^*$, kinematic pressure Ψ = $p/(\rho\mathbf{u}^{*2})$, and the Reynolds number is Re = u*L/ν, with the dimensional variables **x** spatial coordinate vector, t time, ν the kinematic viscosity, **u** the velocity vector and p the fluid pressure.

We solve the Navier - Stokes Equations over general irregular domains, discretized by tetrahedrons. The applied Mixed Hybrid FE scheme assumes the distance among circumcenters to be positive, condition which is always satisfied in the Delaunay meshes. Unfortunately, either bad-quality Delaunay or non-Delaunay meshes are provided by most of the available mesh generators (e.g., see *Li & Teng*, 2001). To cope with this problem, and use good-quality non-Delaunay meshes, we propose the following novel approach. Eq. (1,b) is integrated over each single tetrahedron, but Eq. (1,a) is integrated over clusters of tetrahedrons, such that each external face shared by two different clusters is part of two tetrahedrons whose circumcenters have positive distance. In figures 1,a and 1,b we show two tetrahedrons (T_1 and T_2) which satisfy and not satisfy the Delaunay condition, respectively. In the second case (in figure 1,b), the two tetrahedrons T_1 and T_2 form one cluster. In figure 1,c we show the same cluster as in figure 1,b, with a third tetrahedron (T_3) external to the cluster. To avoid numerical instability problems, if the distance between the circumcentres of two neighbor tetrahedrons T_1 and T_2 satisfying the Delaunay condition is smaller than a given tolerance value, we assume that T_1 and T_2 form a cluster. Generally, a cluster could consist of more than two neighbor tetrahedrons, if the distances of their circumcentres is below the assumed tolerance value.

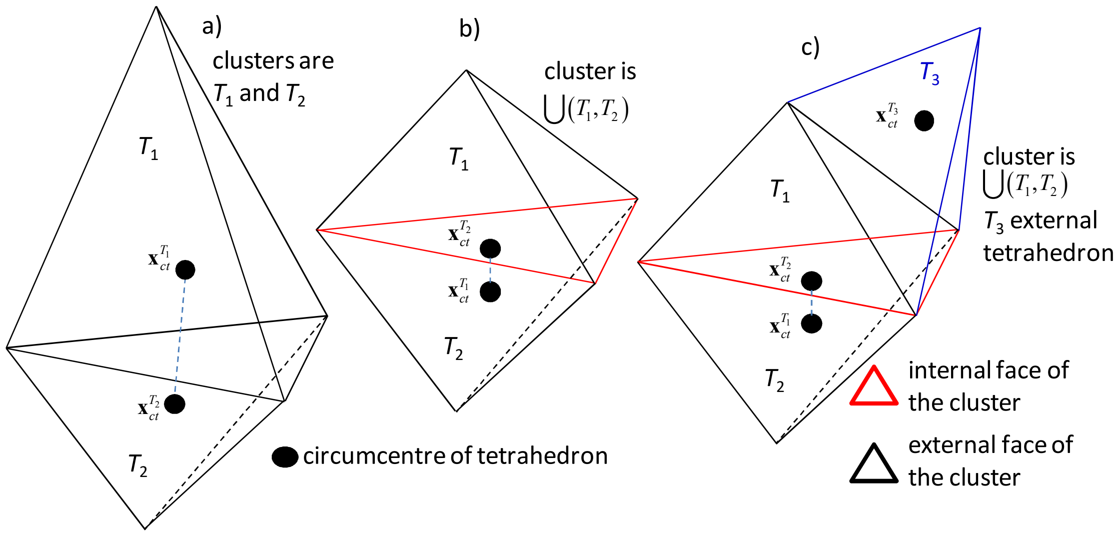

Figure 1. a) Tetrahedrons T_1 and T_2 satisfy the Delaunay condition, b) Tetrahedrons T_1 and T_2 do not satisfy the Delaunay condition and form a cluster, c) tetrahedron T_3 is external to the cluster of T_1 and T_2

To well define velocities inside each cluster with more than one tetrahedron, minimum energy condition is finally enforced for the velocities inside the clusters.

3 THE NUMERICAL PROCEDURE

A projection method is applied for the discretization in time of Eqs. (1). The predictor step deals with the momentum equations and in the corrector step we combine the mass and momentum conservations equations to obtain the divergence-free flow field. The predictor step is solved by applying the MAST (Marching in Space and Time) numerical Eulerian procedure, already proposed for the solution of other physical problems (e.g., *Aricò & Tucciarelli*, 2009, *Aricò et al.*, 2013). The use of the MAST algorithm is another important novelty of the proposed procedure. It explicitly handles the non-linear terms in the momentum equations, and allows numerical stability for Courant numbers greater than one. MAST assumes constant in time values for the gradient of the pressure and viscous terms, computed respectively at the beginning of the time step,

$$\int_W \frac{\mathbf{U}^{k+1/3}-\mathbf{U}^k}{\Delta\tilde{t}}dw+\int_W\left(\mathbf{U}\cdot\tilde{\nabla}\mathbf{U}\right)^{k+1/3}dw+\int_W\tilde{\nabla}\Psi^k dw-\nu\int_W\tilde{\nabla}_2\mathbf{U}^k dw=0 \qquad (2),$$

where W is the volume of each cluster. The correction step is split into two parts, CS1 and CS2. In CS1, three linear systems are solved for the three velocity components, to update the viscous terms of the momentum equations,

$$\int_W \frac{\mathbf{U}^{k+2/3}-\mathbf{U}^{k+1/3}}{\Delta\tilde{t}}dw-\nu\int_W\tilde{\nabla}_2\mathbf{U}^{k+2/3}dw=\nu\int_W\tilde{\nabla}_2\mathbf{U}^k dw \qquad (3),$$

Solution of Eq. (3) involves the solution of 3 linear system for the unknowns $\left(\mathbf{U}^{k+2/3}-\mathbf{U}^{k+1/3}\right)$. In CS2, one linear system is solved for the unknown pressure correction,

$$\frac{\mathbf{U}^{k+1}-\mathbf{U}^{k+2/3}}{\Delta\tilde{t}}+\tilde{\nabla}\eta=0 \qquad \text{with} \qquad \eta=\Psi^{k+1}-\Psi^{h} \tag{4}$$

Taking the divergence of Eq. (4) and applying the Green Lemma, we get a 2nd order Poisson equation, which involves the solution of a linear system for the η unknowns.

A RT-0 spatial discretization is adopted for the formulation of the corrective fluxes in Eq. (4) similar to the Mixed Hybrid Finite Element Scheme, lumped in the circumcentres of the tetrahedrons (*Younes et al*., 2006). After solution of Eq. (4) for each cluster, continuity along internal faces is restored by minimizing the kinetic energy of the tetrahedron velocity, subject to the flux continuity.

The Courant limit for the size of the time step, in both PS2 and correction step, is not required. The matrix of all the systems is symmetric and diagonally dominant. The matrix coefficients are computed and factorized at the beginning of the numerical simulation, which allows to save a lot of computational time, compared to other numerical schemes (e.g., the lagrangian schemes).

4 PRELIMINARY MODEL RESULTS

In this test the flow is confined in a rectangular cavity and driven by the displacement of the upper wall, where a tangent and constant in time value of the velocity is imposed (see figure 2). The set-up for this problem is shown in figure 2,a as well as the imposed boundary conditions. We discretize the domain with 87740 tetrahedrons and 17388 nodes, resulting in 84385 clusters (in figure 2,b we show an intersection of the mesh with a cutting plane orthogonal to the $\tilde{y}$ direction). Initial conditions are zero velocity and pressure inside the domain.

In figure 3,a we plot the computed flow velocity vectors obtained setting Re = 100, and in figures 3,b and 3,c we compare for Re = 100, 400 and 1000 the horizontal (U) and vertical (W) velocity components with the reference literature solutions provided by Ghia et al. (1982), after steady state conditions are attained. Generally, the numerical results are in good agreement with the ones provided in literature. In figures 3,d-3,f we plot, for Re = 1000, the velocity streamlines, the vorticity along the $\tilde{y}$ direction and the kinematic pressure. The results in In figures 3,d-3,f refer to the cutting plane $\tilde{y}$ = 0.1 m (i.e., the diametral plane of the domain). The minimum and maximum pressures are computed respectively at the upper-left and upper-right corners, where a discontinuity in the boundary condition occurs. Because of this singularity, no *ad-hoc* handling is required in the model, unlike what is found in Botella and Peyret (1998), Kuhlmann and Romano' (2019), and cited references. The minimum pressure values are associated with the foci of the vortices, due to the high centrifugal acceleration occurring around them.

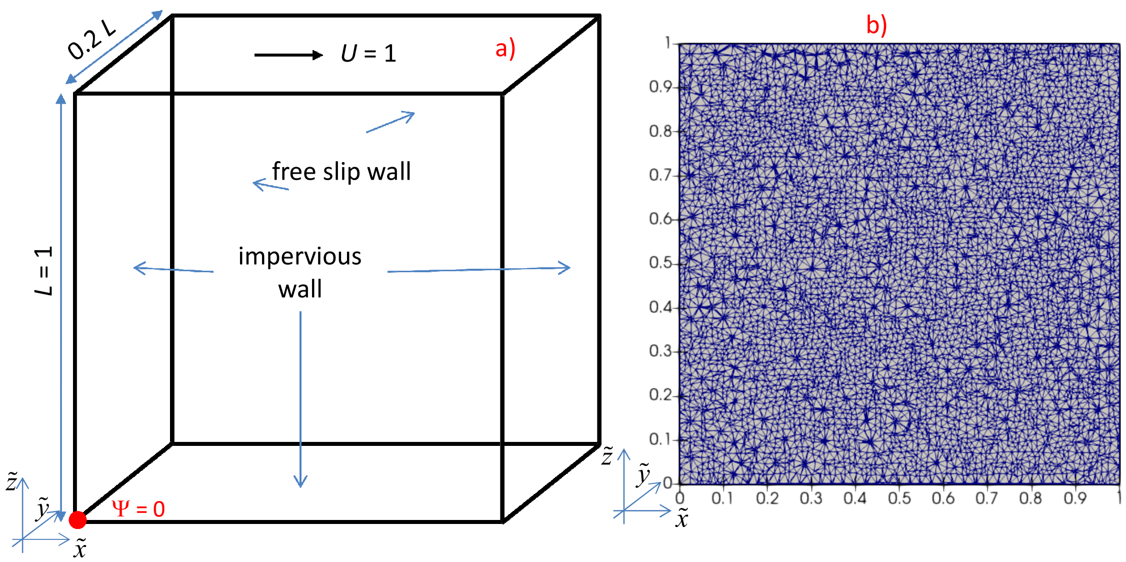

Figure 2. a) computational domain and boundary conditions, b) section of the tetrahedral mesh with a plane orthogonal to the $\tilde{y}$ axis

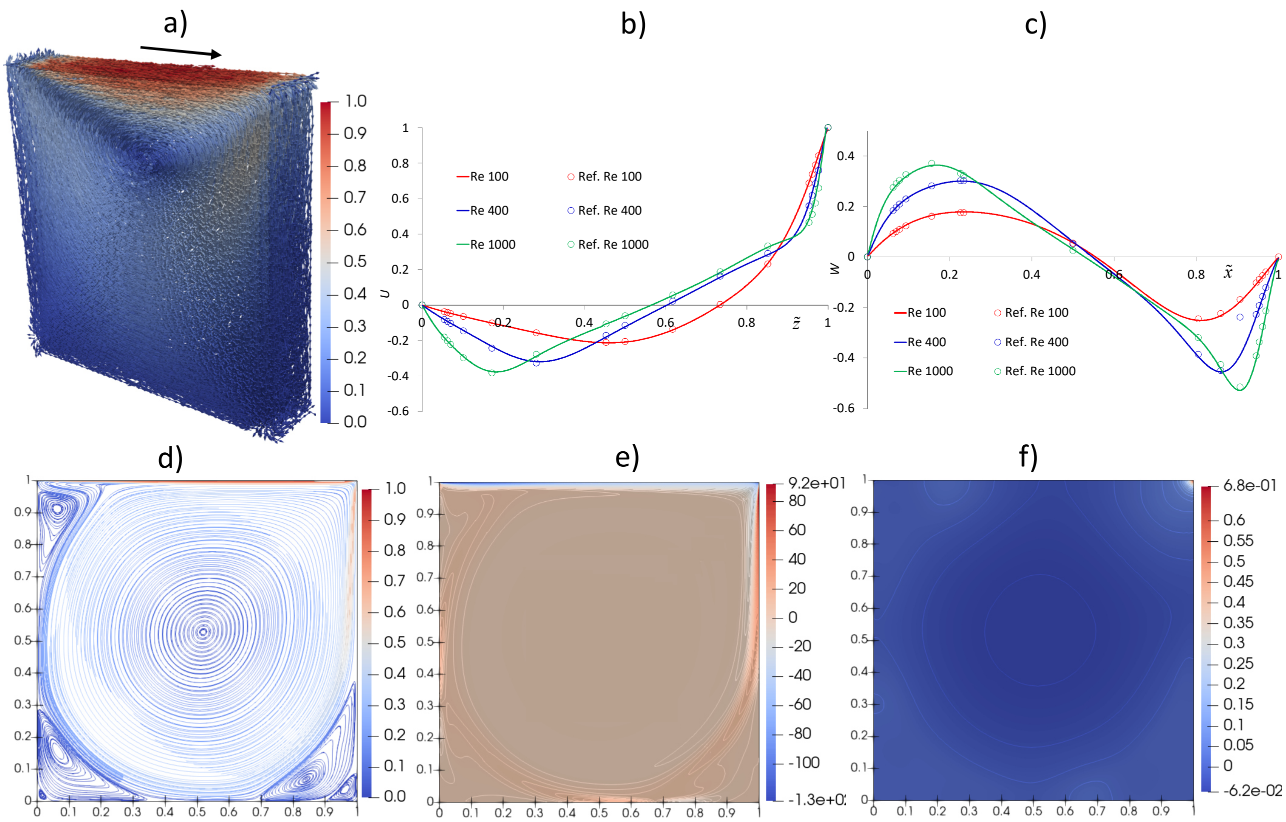

Figure 3. a) computed 3D velocity vector for Re = 100 (the black arrow shows the moving lid direction), b) comparison of the horizontal velocity component with the reference ones (Re = 100, 400, 1000, plane $\tilde{y} = 0.1$, "Ref." results by Ghia et al., 1982), c) comparison of the vertical velocity component with the reference ones (Re = 100, 400, 1000, plane $\tilde{y} = 0.1$, "Ref." results by Ghia et al., 1982), d) flow velocity streamlines (Re = 1000, plane $\tilde{y} = 0.1$), e) $\tilde{y}$ -vorticity (Re = 1000, plane $\tilde{y} = 0.1$), f) kinematic pressure (Re = 1000, plane $\tilde{y} = 0.1$)

5 CONCLUSIONS

The proposed model has proved, in the previous study case and in other ones here not shown for brevity, a remarkable robustness for the solution of flow problems in domains discretized with 3D irregular meshes. A research activity is in progress for the application of the present model in the framework of hemodynamic and bio-fluid mechanics simulations, characterized by very irregular shapes of the computational domains (e.g., aortic or cerebral aneurysms).

REFERENCES

Li, X.,Y. & Teng, S.H. Generating well-shaped Delaunay meshes in 3D. Conference: Proceedings of the Twelfth Annual Symposium on Discrete Algorithms, January 7-9, 2001, Washington, DC, USA

Aricò, C. & Tucciarelli, T. The MAST-FV/FE scheme for the simulation of thermohaline processes in variable density saturated porous media. J. Comp. Phys., 2009, 228(4), 1234-1274

Aricò, C., Sinagra, M. & Tucciarelli, T. Anisotropic potential of velocity fields in real fluids: Application to the MAST solution of shallow water equations. Adv. Wat. Res., 2013, 62, 13-36

Younes, A,, Ackerer, P. & Lehmann, F. A new mass lumping scheme for the mixed hybrid finite element method. Int. J. Num. Meth. Eng., 2006, 67(1), 89-107

Ghia, U., Ghia, K.N., and Shin, C.T. (1982). High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, J. Comp. Phys., 48:387-411

Botella, O., and Peyret, R. (1998). Benchmark spectral results on the lid driven cavity flow, Comp. & Fluids, 27:421-433

Kuhlmann, H.C., and Romanò, F. (2019). The lid-driven cavity, Computational Methods in Applied Sciences Volume, 50:233-309