SIMULATING UNSTEADY INCOMPRESSIBLE FLOWS USING GALATEA-I, A PARALLEL MULTIGRID FINITE-VOLUME SOLVER

Sotirios S. Sarakinos¹, Georgios N. Lygidakis² and Ioannis K. Nikolos³

¹,²,³ School of Production Engineering and Management
Technical University of Crete
Chania, GR-73100, Greece

³ Contact author, e-mail: jnikolo@dpem.tuc.gr

Keywords: Incompressible Flow, Unsteady RANS, LES, Hybrid Unstructured Grids, Finite-Volume Method, Unsteady Flow.

Abstract. Physical flows are essentially turbulent and unsteady, making the development and utilization of corresponding CFD (Computational Fluid Dynamics) algorithms a prerequisite for real engineering problems. Such an academic code, named Galatea-I, is developed in this study, which employs the Navier-Stokes equations with the Unsteady RANS (Reynolds-Averaged Navier-Stokes) or LES (Large Eddy Simulation) approach, to predict incompressible fluid flows on three-dimensional hybrid unstructured grids, composed of tetrahedral, prismatic and pyramidal elements. The discretization of governing equations is obtained with a node-centered finite-volume scheme, while the artificial compressibility (or pseudo-compressibility) methodology allows for their solution within the framework of a time-marching compressible flow algorithm by adding a temporal derivative of pressure in the continuity equation. Time integration and iterative approximation of each instantaneous pseudo-time steady state is achieved with a second-order accurate in time, four-stage Runge-Kutta scheme. In order to alleviate excessive time-consuming simulations, entailed by the elliptic and transient behavior of incompressible equations, Galatea-I is enhanced with the capability of parallel processing and an agglomeration multigrid scheme. The proposed algorithm is validated against benchmark test cases, concerning unsteady laminar and turbulent flow over a cylinder, while the obtained results, compared with those of corresponding solvers, indicate its potential for such simulations.

1 INTRODUCTION

Most of the real physical flows exhibit unsteadiness, although some turbulent flows can be simulated as quasi-steady ones, using the time-averaging concept of the Navier-Stokes equations and proper turbulence modelling. For example, flow inside blood vessels [1] or air flow around tall buildings [2] are highly unsteady incompressible flows. Therefore, various academic and industrial/commercial CFD codes have been developed during the past decades to simulate unsteady incompressible flow phenomena in diverse types of applications, such as maritime, biomechanics and civil engineering. Despite the significant efforts that have been exerted for the development of such algorithms, succeeding the considerable reduction of the requirements for the excessively time and money consuming experimental studies [3], the continuous need for more accurate and more efficient methods is maintained, as many issues are still open in this scientific field.

First and foremost, various approaches have been developed to model the association between velocity and pressure in such a way that the divergence constraint, entailed by the continuity equation, is satisfied [4]. In general three methodologies have been established, transforming analogously the governing PDEs (Partial Differential Equations); the artificial compressibility or pseudo-compressibility method, the pressure correction method and preconditioning [5, 6]. According to the first one, introduced initially by Chorin [7], the incompressible PDEs are manipulated within the framework of a time-marching compressible flow algorithm by adding a temporal derivative of pressure in continuity equation and completing each physical time step by sub-iterations [8-10]. Alternatively, the pressure-based methodology solving a Poisson equation for pressure can be followed, or preconditioning of the governing PDEs can be implemented [5]. Another issue, affecting strongly both the accuracy and efficiency of such solvers, concerns the approach followed for turbulence modelling [3]. DNS (Direct Numerical Simulation), being certainly the most accurate methodology for turbulence's prediction, cannot yet widely be afforded, as it aims to capture all turbulent whirls in space and time, necessitating, thus, for a very fine spatial resolution along with small time steps [3]; therefore, appropriate modeling has to be implemented in order to account for the effects of turbulence in an approximate manner. Unlike this approach, the Unsteady RANS (Reynolds-Averaged Navier-Stokes) one models all the whirls independently of their magnitude, achieving in that way significant computational savings in expense though of obtained accuracy [5]. A compromise between the aforementioned methodologies is the LES (Large Eddy Simulation) method, according to which the contributions of the large, energy carrying whirls are resolved, while these of the small
ones that cannot be computed by the flow equations are modelled [8]. Besides the aforementioned options quite more issues have to be defined for similar CFD algorithms, such as the type of utilized grids, the method of discretization, the flux computation, the time integration and iterative approximation, etc. [3]. Furthermore, the need for detailed spatial discretization and grid density required especially by LES models, increases the simulation time, necessitating appropriate acceleration techniques, such as parallel processing [3, 9, 11] and multigrid methodology [3, 9, 12-14].

In this study the recent development of the academic CFD code Galatea-I is reported [9]. It employs the Navier-Stokes equations with the Unsteady RANS methodology coupled with the SST (Shear Stress Transport) turbulence model [9, 15] or LES approach jointed with the Smagorinsky model [16] to predict unsteady incompressible fluid flows on three-dimensional hybrid unstructured grids, composed of tetrahedral, prismatic and pyramidal elements. Discretization of the governing equations, transformed appropriately by the artificial compressibility methodology [6-8], is obtained with a node-centered finite-volume scheme [3]. For the computation of the inviscid fluxes the Roe’s approximate Riemann solver [17] along with a second-order accurate spatial scheme, based on MUSCL (Monotonic Upwind Scheme for Conservation Laws) approach [5, 9], is implemented, while for the viscous ones the required gradients at control volumes’ interfaces are calculated with a nodal-averaging method [5, 9].

The pseudo-time integration of PDEs and unsteady solution is succeeded with an explicit second-order accurate in time four-stage Runge-Kutta method (RK(4)) [18]. As the solution of unsteady flow problems tends to be excessively time and computer-resources consuming, the proposed solver is enhanced by appropriate acceleration techniques. In particular, parallel processing capability, based on domain decomposition approach and MPI (Message Passing Interface) library functions [3, 9, 11] has been introduced. Additionally, an agglomeration multigrid scheme, based on FAS (Full Approximation Scheme) methodology and successively coarser grids constructed via the directional or isotropic fusion of adjacent control volumes [3, 9, 12-14], has been incorporated. Attention in this work is mainly directed towards the evaluation of accuracy of the proposed code, as well as the comparison of the results derived by the RANS and LES approaches; it is validated against benchmark test cases, concerning unsteady laminar and turbulent flow over a cylindrical surface. The obtained results, compared to available in the open literature reference data, reveal Galatea-I’s potential to predict effectively such flows in terms of accuracy and efficiency.

This paper is organized as follows. Section 2 summarizes the employed governing equations, while Section 3 describes in brief the followed numerical method, focusing on spatial discretization, flux computation, time integration and iterative approximation of the solution, and incorporated acceleration techniques. In Section 4 the obtained computational results are presented and compared to those of reference solvers, indicating the capability of Galatea-I for such simulations. Finally Section 5 contains some conclusions and information on ongoing work.

2 GOVERNING EQUATIONS

The incompressible Navier-Stokes equations, modified by the artificial compressibility approach [6-8], are described in integral formulation using dimensionless variables as [9, 19]

\[
\frac{\partial \vec{Q}}{\partial \tau} + \frac{\nabla \cdot \vec{n}}{\partial \Omega} \frac{d \omega}{d \Omega} + \oint_{\partial \Omega} \frac{\vec{f}}{\partial S} dS - \oint_{\partial \Omega} \frac{\vec{g}}{\partial S} dS = \oint_{\partial \Omega} \vec{s} d\Omega
\]

(1)

where \( \Omega \) is the examined control volume, while \( \vec{s} \) is the source term, being equal to zero in this study. The vector of primitive variables \( \vec{Q} \) in the temporal derivative, regarding pseudo-time \( \tau \), is defined as [9, 19]

\[
\vec{Q} = [p, u, v, w]
\]

(2)

where \( p \) denotes the pressure, while \( u, v \) and \( w \) are the Cartesian components of velocity. The second and third LHS terms in eqn (1), denoting inviscid and viscous fluxes, respectively, are described as [9, 19]

\[
\vec{f} = \begin{bmatrix}
\beta \theta \\
u \theta + n_x p \\
w \theta + n_x p
\end{bmatrix}, \quad \vec{g} = \begin{bmatrix}
0 \\
n_x \tau_{xx} + n_y \tau_{xy} + n_z \tau_{xz} \\
n_x \tau_{yx} + n_y \tau_{yy} + n_z \tau_{yz} \\
n_x \tau_{zx} + n_y \tau_{zy} + n_z \tau_{zz}
\end{bmatrix}
\]

(3)

where \( \vec{n} \) is the outward normal vector to the control surface, while \( \theta \) is the normal to the control surface velocity component, defined as [19]

\[
\theta = un_x + vn_y + wn_z
\]

(4)

The stress tensor \( \tau_{ij} \), which is based on Boussinesq hypothesis [5] and is consequently associated linearly to the
Sotirios S. Sarakinos, Georgios N. Lygidakis and Ioannis K. Nikolos.

The mean rate strain, can be computed as \[5, 9, 19\]:

\[
\tau_{ij} = \left( \frac{v}{Re} + \nu_t \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij}, \quad \forall ij = 1,2,3 \text{ with } u_i = (u,v,w) \text{ and } x_i = (x,y,z)
\] (5)

In eqn (5) \(v\) denotes laminar kinematic viscosity, while \(v_t\) the corresponding turbulent one; if RANS approach is selected the SST turbulence model is employed for the computation of \(v_t\) \[9, 15, 19\], while in case of LES methodology it is calculated via the Smagorinsky model \[5, 8, 16, 20\]. No additional model has been incorporated to account for transition phenomena \[19\].

3 NUMERICAL METHOD

3.1 Spatial discretization

Discretization of the governing equations is obtained with a node-centered finite-volume scheme on unstructured hybrid grids, composed of tetrahedral, prismatic and pyramidal elements \[9, 19\]. The control volume of a node is defined by the surfaces, constructed by connecting with lines the mid-points of edges, the barycenters of faces and the barycenters of elements, sharing this node \[3\]. Figure 1 illustrates part of the control volume of a node \(P\), contributed by a tetrahedral and a prismatic element.

![Figure 1. Part of the control volume of a node \(P\), contributed by a tetrahedral and a prismatic element.](image)

Since discretization method is defined, the divergence theorem is employed on eqn (1), transforming it as follows for a node \(i\) \[3, 9, 19\]

\[
\left( \frac{\partial \bar{Q}}{\partial T} \right)_i V_i + \sum_{j \in K_N(i) \text{ and } j \_out} \bar{F}_{invj} - \sum_{j \in K_N(i)} \bar{F}_{vij} = \bar{S}_i V_i
\] (6)

where \(V_i\) is the volume of the control cell of node \(i\), while \(j\) denotes each node connected with an edge \(ij\) to the examined node \(i\). The second and third LHS terms represent the sum of inviscid and viscous fluxes, respectively.

3.2 Numerical fluxes

For the computation of the inviscid fluxes the Roe’s approximate Riemann solver \[17\] is implemented at the interfaces of the control cells, assuming a one-dimensional Riemann problem at the same areas. In order to increase spatial accuracy a second-order accurate scheme is employed, based on MUSCL approach \[5, 9\]. For the calculation of the viscous fluxes, the gradients of the velocity components have to be evaluated at the same interfaces; two methodologies have been included in Galatea-I, an element-based one regarding the construction of additional control volumes around the edges of the grid (edge-dual volumes) \[3, 6\], and a nodal-averaging one, utilizing the corresponding gradients at the endpoints of each edge along with a directional derivative \[3, 5\]. Although the first approach is supposed to be more accurate than the second one, the latter one was selected to be employed in this study due to the significantly lower computation time required \[3\].

Appropriate boundary conditions have to be imposed in order to complete flux balance of nodes, located at these regions. In case of an inviscid flow free-slip conditions are applied implicitly \[3, 5\] at solid-wall surfaces contributing analogously the flux balance of corresponding nodes; the contributions of nodes at symmetry planes are computed in the same way \[19\]. If a viscous flow problem is encountered no-slip conditions are employed explicitly at wall boundaries \[5, 19\]. At inlet/outlet surfaces characteristics-based boundary conditions are applied instead \[9, 19, 21\].

In case of a turbulent flow the required fluxes for SST model are computed much in the same way to these of flow PDEs, except for the inviscid ones, for which a simple first-order spatial accurate upwind scheme is employed instead \[3, 5\]; the viscous terms are the main ones in such turbulence models \[19\]. Regarding the corresponding boundary conditions, Dirichlet ones are imposed to nodes at solid wall and inlet surfaces, while
3.3 Time integration

Since flux balance has been calculated for each computational node, iterative relaxation of the governing equations can be performed utilizing an explicit second-order accurate in time four-stage Runge-Kutta method (RK(4)) after transforming eqn (6) in the following formulation [9, 19]

\[- \frac{V_i \Delta \bar{Q}_i}{\Delta \tau_i} = \bar{R}_i \]  

(7)

where \( \bar{R}_i \) denotes the flux balance of node \( i \), while \( \Delta \tau_i \) is the pseudo-time step, computed with a local time-stepping technique [3, 5, 19]. Nevertheless, the aforementioned formulation is appropriate only for steady-state simulations [9, 19]. In case of unsteady/transient problems a dual time-stepping scheme is employed, according to which an additional temporal term is included in eqn (1) [6, 10], transforming it as follows

\[- \frac{V_i \Delta \bar{Q}_i}{\Delta \tau_i} = \bar{R}_i^* \]  

(8)

where \( t \) denotes real time. The true-derivative term is discretized with a second-order backward difference formula, deriving the corresponding to eqn (7) unsteady relation

\[- \frac{V_i \Delta \bar{Q}_i}{\Delta \tau_i} = \bar{R}_i^* \]  

(9)

which is relaxed at next in pseudo-time for each real time step, exactly in the same way to eqn (7) [6, 10]. The RHS term \( \bar{R}_i^* \) is computed using the values of velocity components of previous real time steps \( n \) and \( n-1 \) and previous pseudo-time step of real time step \( n+1 \) as [6, 10]:

\[ \bar{R}_i^* = \frac{V_i}{\Delta \tau} \left( \frac{3}{2} Q^{n+1} - 2 Q^n + \frac{1}{2} Q^{n-1} \right) + \bar{R}_i \]  

(10)

In that way instead of solving governing PDEs at each time step in the real time domain, the problem is transformed into a sequence of instantaneous pseudo-steady-state computations in the artificial time domain [10].

3.4 Acceleration techniques

As mentioned in Introduction, unsteady flow simulations tend to be excessively time-consuming, requiring thus, appropriate acceleration techniques. Besides local time-stepping method [3, 5] and edge-based data structures [5] utilized in Galatea-I for this reason, parallel processing capability [9, 19] and an agglomeration multigrid scheme have been incorporated [3, 9, 13, 14, 19].

The developed parallelization strategy is based on domain decomposition approach and MPI library functions [11]. The whole procedure begins with METIS application [22], which divides the grid nodes in smaller sets (core nodes) to be processed separately; at next, the elements of the grid are divided accordingly [3, 9, 19]. Nevertheless, the elements located at the interfaces of the sub-domains remain incomplete, calling for further treatment. Their missing nodes are added as additional nodes (ghost nodes) to the corresponding partitions, constructing among them the so-called overlapping region [9, 19]. It is this region that allows for the data exchange (values of nodal variables and gradients) between the adjacent partitions via the MPI communication protocol. At each sub-grid the governing PDEs are relaxed only for core nodes, deriving their variables’ values; the corresponding values of ghost nodes are acquired explicitly from their corresponding core ones at neighboring partitions [3, 9, 19].

Additional acceleration is gained with a multigrid scheme, based on FAS approach and a sequence of coarser resolutions, derived by the fusion of adjacent control volumes [3, 5, 9, 12-14]. The agglomeration strategy considers merging of control cells on a topology preserving framework resembling the advancing front technique [3, 9, 12]; it is limited though by pre-defined rules, concerning mainly boundary and ghost nodes [14]. For example, a boundary node can be associated only with other boundary nodes belonging to the same surface, while internal nodes can be fused only with other internal ones [3, 9, 12]. The ghost nodes aren’t taken into account during the main merging procedure, but they are agglomerated according to the fusion of their corresponding core nodes at neighboring partitions. Considering these constraints the whole process begins at each sub-domain by examining the solid wall boundary nodes and fusing them into supernodes [9]. It continues for these at outlet regions an upwind approach is applied similarly to the internal points [9].
with the examination of the adjacent to these supernodes, non-merged yet, nodes and so forth; the procedure is accomplished with the examination of all the grid nodes [9]. In case a directional agglomeration is selected (along with a hybrid unstructured grid) the nodes of the prismatic layers are examined initially and separately from the rest mesh nodes [9]. If an even coarser resolution is desired, the whole procedure is repeated. The multigrid accelerated iterative solution is obtained with the FAS approach in a V-cycle process, according to which each cycle begins with the relaxation of governing PDEs at the initial finest grid [9]. The obtained values of variables and flux balances are restricted then to the next coarser resolution by means of smoothing [9]. An approximate eqn (7) or (9) for steady- or unsteady-state simulation respectively, is relaxed at this level, while the aforementioned values are restricted similarly to the next coarser grid; the process of solution and restriction are repeated up to the coarsest agglomerated mesh [9]. Since solution is obtained at this level, the corrections of primitive variables are prolonged to the next finer one by means of interpolation, while the same process is repeated up to the initial finest grid at which point the V-cycle is accomplished [9].

4 RESULTS

In this study Galatea-I is validated against a benchmark test case, concerning unsteady quasi-3D laminar and turbulent incompressible flow around a circular cylinder [10, 23-28]. Quite many similar solvers have been evaluated with this problem, mainly because viscous flow over a cylindrical surface is unsteady at even very low Reynolds numbers [10]; as a result it is a very well documented problem in the open literature. The steady-state solution of such a flow can be achieved only with low Reynolds numbers, e.g., around \(40\) [10, 23]. Unsteadiness can be observed at even slightly higher values, e.g., \(100\); the flow becomes oscillatory, shedding vortices at the wake region in a stable frequency and forming a Von Karman Street [10, 23-28]. Turbulent flow over the same geometry holds also unsteady properties with similar to laminar flow vortex shedding phenomena [6]. Considering the previous state, two cases were encountered in this work, at Reynolds numbers \(200\) and \(1E+6\), respectively; the reference length, used for the computation of the aforementioned numbers, is cylinder’s diameter.

The utilized for this paper computational model, including a circular cylinder with diameter \(D=1m\) and length \(L=2m\) is presented in Figure 2. The domain was extended up to \(30D\) at the wake region of the cylinder, in order to capture sufficiently the expected vortex shedding phenomena. At the upstream flow region, as well as over and under the cylindrical surface, the length of the computational filed was set equal to \(10D\). Though both laminar and turbulent flow simulations were performed with this geometry, different grids were generated for each case, as they differentiate significantly in required mesh density mainly at the cylindrical wall region; the second grid layer was set considerably closer to the wall for the turbulent flow problem.

![Figure 2. Geometry of the computational field, including the circular cylinder.](image)

4.1 Laminar unsteady flow over a circular cylinder (Re=200)

The computational mesh generated for this test case was consisted of \(1,361,895\) nodes, \(6,975,754\) tetrahedrons and \(274,770\) prisms; the latter ones were located at the cylindrical wall region. The height of the first layer of prisms was set equal to \(1E-3\) m, while mesh density was increased at the wake region of the cylindrical surface, in order better accuracy to be obtained. In Figure 3 the density of the overall mesh is presented, along with a close-up view at the symmetry/cylinder region. Simulation was performed on a workstation with an AMD FX™ 8150 eight-core processor at 3.62 GHz. For parallel processing the computational domain was divided into eight partitions with the METIS application, while three coarser resolutions were generated for each of them employing the incorporated directional agglomeration strategy in order further acceleration to be gained with multigrid methodology. The artificial compressibility parameter was set equal to \(10\), while integration in pseudo-time was achieved with a CFL number of 0.5; the dimensionless real time step was set to 0.05.
Sotirios S. Sarakinos, Georgios N. Lygidakis and Ioannis K. Nikolos.

Figure 3. Overall mesh density (left) for the circular cylinder test case at Re=200 and a close-up view at the symmetry/cylinder area (right).

In the initial steps (in real time), during which the phenomenon hasn’t been fully developed yet, a pair of symmetrical vortices appears downstream cylinder’s surface with actually steady-state characteristics. As the time advances, unsteadiness arises and the generated vortices are shed in a periodic fashion. The phenomenon is illustrated quantitatively in Figure 4 with the evolution in time of drag and lift coefficients on the cylindrical surface.

Figure 4. Lift and drag coefficient measurements on the cylindrical surface at different real time moments.

It is quite obvious, that both coefficients exhibit an oscillatory attitude from the very beginning of the simulation, almost at dimensionless time zero. These oscillations are amplified approximately up to time step 50, from which moment the phenomenon is developed then periodically. The phenomenon is depicted qualitatively in Figure 5, which presents the derived stream-wise velocity component contours along with the flow streamlines at moment 0 of a period T; on Figure 5 (Right) the obtained vorticity contours are shown for the same moment. Figure 6 includes the corresponding illustrations for four intervals of a full period (0T, 0.25T, 0.5T and 0.75T).

Figure 5. Streamwise velocity contours with flow streamlines (left) and vorticity contours at t=0T.
Figure 6. Streamwise velocity contours and flow streamlines for $t=0T$ (top left), $t=0.25T$ (top right), $t=0.5T$ (bottom left) and $t=0.75T$ (bottom right).

For the quantitative confirmation of the aforementioned qualitative results, the Strouhal number (St) of the vortex shedding phenomenon, as well as the average values and deviations of the lift ($C_L$) and drag ($C_D$) coefficients, were computed; they are included in Table 1, along with various experimental and computed ones, reported in the open literature [10, 23-27]. As one can observe, a sufficient agreement is achieved between the current and reference results.

<table>
<thead>
<tr>
<th></th>
<th>St</th>
<th>$C_L$</th>
<th>$C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Current (Galatea-I)</td>
<td>0.196</td>
<td>0.0±0.63</td>
<td>1.34±0.047</td>
</tr>
<tr>
<td>Tai et al. [10]</td>
<td>0.195</td>
<td>0.0±0.64</td>
<td>1.31±0.041</td>
</tr>
<tr>
<td>Liu et. al. [24]</td>
<td>0.192</td>
<td>0.0±0.69</td>
<td>1.31±0.049</td>
</tr>
<tr>
<td>Chan and Anastasiou [25]</td>
<td>0.183</td>
<td>0.0±0.63</td>
<td>1.48±0.05</td>
</tr>
<tr>
<td>Belov et. al. [26]</td>
<td>0.193</td>
<td>0.0±0.64</td>
<td>1.19±0.042</td>
</tr>
<tr>
<td>Rogers and Kwak [27]</td>
<td>0.185</td>
<td>0.0±0.65</td>
<td>1.23±0.05</td>
</tr>
<tr>
<td>Vrahliotis et al. [23]</td>
<td>0.198</td>
<td>0.0±0.69</td>
<td>1.36±0.046</td>
</tr>
</tbody>
</table>

Table 1. Current and reference Strouhal numbers, average values and deviations of lift and drag coefficients for unsteady laminar flow at $Re=200$.

4.2 Turbulent flow over a circular cylinder at $Re=10^6$

In this section turbulent flow around the circular cylinder is examined, employing both the RANS and LES approaches. A coarser computational mesh, compared to this used in the previously described simulation, was generated to alleviate excessive computation time, consisting of 654,103 nodes, 1,305,744 tetrahedrons and 810,650 prisms. However, the height of the first layer of prisms at the cylindrical surface area was adjusted lower, at $5E-4$ m. The Reynolds number of free-stream flow was assumed $1E+6$ [28, 29], while the artificial compressibility parameter was set equal to 10. Integration in pseudo-time was performed with a CFL number equal to 0.5, while dimensionless real time was advanced with a step of 0.05. Attention was mainly directed towards the comparison of RANS and LES methods, using SST and Smagorinsky turbulence models, respectively. Simulations were accelerated, employing the incorporated parallelization strategy and agglomeration multigrid scheme; the initial mesh was divided into eight subdomains, while two coarser...
resolutions were generated with the full-coarsening directional agglomeration technique in order multigrid scheme to be applied.

In Figure 7 the obtained with RANS methodology streamwise velocity contours along with velocity streamlines are presented (left); in Figure 7 (Right) the corresponding kinematic viscosity contours at the wake region of the cylinder are illustrated; a Von Karman Street is clearly developed. Similarly to the laminar flow test case, the lift coefficients were computed during the RANS and LES simulations, deriving the corresponding graph in Figure 8. The RANS approach exhibits a periodic oscillatory behavior analogously to laminar flow; unlike them, the results obtained with the LES methodology aren’t exactly periodic, similarly to those reported in [29].

![Figure 7. Streamwise velocity contours with flow streamlines (left) and kinematic viscosity contours at t=0T.](image)

![Figure 8. CL evolution in time for the simulations with LES and RANS methods.](image)

![Figure 9. Vorticity contours, obtained with RANS (top) and LES (bottom).](image)
The vorticity contours, obtained with the RANS (top) and LES (bottom) approaches, at the same time instant, are illustrated in Figure 9. The vortex shedding phenomenon is clearer at the RANS results, where separated vortices appear to move along the streamwise direction, with a decreasing though intensity as they move away from the cylindrical surface. The obtained with LES vorticity contours at the same instant appear less coherent, with connected vortex regions not well organized as at RANS results. The aforementioned differentiations between RANS and LES methodologies explain the alteration, observed in the lift coefficient evolutions in Figure 8.

The quantitative evaluation of both RANS and LES simulations is presented in Table 2, where the Strouhal number for the vortex shedding intensity is presented, along with the mean drag coefficient around the cylindrical surface. The obtained results are compared with the experimental and numerical reference ones [28, 29]. While the results for the Strouhal number are comparable with the experimental ones, a relatively large differentiation is observed between the drag coefficient values. This is mainly attributed to the coarse grid used for these simulations, in order to reduce the required computation time.

<table>
<thead>
<tr>
<th>Method</th>
<th>St</th>
<th>C_D</th>
</tr>
</thead>
<tbody>
<tr>
<td>RANS (Current)</td>
<td>0.222</td>
<td>1.18</td>
</tr>
<tr>
<td>LES (Current)</td>
<td>0.190</td>
<td>1.21</td>
</tr>
<tr>
<td>Shih et. al. (experimental) [28]</td>
<td>0.203</td>
<td>0.24</td>
</tr>
<tr>
<td>Catalano (URANS) [29]</td>
<td>0.31</td>
<td>0.40</td>
</tr>
<tr>
<td>Catalano (LES) [29]</td>
<td>0.35</td>
<td>0.31</td>
</tr>
</tbody>
</table>

Table 2. Comparison of results for unsteady turbulent flow around circular cylinder at Re=10⁶.

5 CONCLUSIONS

In this study the recently developed academic CFD code Galatea-I was presented in brief, while it was validated against a benchmark test case. In particular, laminar and turbulent flows over cylindrical surfaces were examined, while the obtained results were compared to those of similar reference solvers, achieving a sufficient agreement and indicating consequently the potential of the proposed solver for such simulations. Moreover, LES and RANS methodologies, employing Smagorinsky and SST turbulence models respectively, were applied for the same test case and their results were compared to each other. Both of them succeeded in portraying with detail the vortex shedding phenomena occurring at such high Reynolds numbers. The Strouhal number of the vortex shedding phenomenon was well predicted with both methodologies, compared to experimental and numerical reference test cases found in the open literature. However, as it remains a challenging flow test case, especially at high Reynolds numbers [29], a more systematic investigation is required, including for example, grid independency study and examination of more sophisticated LES models.

ACKNOWLEDGMENTS

This study was funded by Wacker Bauwerksaerodynamik GmbH, DENCOD Developments & Engineering Consultants S.A. (DENCOD S.A.) and the EUREKA-EUROSTARS Project No E!7987 “Model-Based Aeroelastic Analysis of Long-Span Bridges on the HPC Cloud (BridgeCloud)”. 

REFERENCES

Sotirios S. Sarakinos, Georgios N. Lygidakis and Ioannis K. Nikolos.


