Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 166 (2016) 397 – 404

Procedia Engineering

www.elsevier.com/locate/procedia

2nd International Symposium on Submerged Floating Tunnels and Underwater Tunnel Structures

Analysis of fluid-structure interaction for a submerged floating tunnel

Alberto Mandara[a], Emilio Russo[a], Beatrice Faggiano[b]*, Federico M. Mazzolani[b]

[a]Department of Civil Engineering, Design, Building and Environment, Second University of Naples, Via Roma 9, 81031 Aversa (CE), Italy
[b]Department of Structures for Engineering and Architecture, University of Naples Federico II, P.le V. Tecchio, 80125 Naples, Italy

---

Abstract

The behavior of a submerged floating tunnel (SFT) exposed to a water current of variable velocity is investigated through complex numerical analyses based on the Computational Fluid Dynamics (CFD) and the Finite Element Method (FEM) implemented in the ABAQUS code. An accurate modelling of turbulent phenomena is made, based on both Implicit Large Eddy Simulation and the RANS-based Spalart-Allmaras model, followed by a co-simulation procedure in which the fluid dynamics and the structural analysis are carried out separately and interfaced with each other. Circular and elliptical cross sections are considered, each of them fitted for combined railway and motorway services. The analysis is carried out in both static and dynamic way, by varying the current velocity with a given value of the residual buoyancy of the tunnel. The results emphasize the effect of the main parameters investigated, evidencing the great potentials of the adopted calculation tool for carrying out further investigations aimed at achieving useful elements for the design and optimization of the SFT.

Peer-review under responsibility of the organizing committee of SUFTUS-2016

Keywords: Submerged Floating Tunnel; CFD and Structural FEM Co-simulation Analysis; Water Current; Fluid-Structure Interaction;

---

1. Introduction

Even though not yet exploited in engineering applications, the concept of Submerged Floating Tunnel (SFT) has gained interest in the recent years from both theoretical and practical points of view. As a demonstration of this, a great amount of proposals have been presented all around the world, concerning a number of possible implementations

---

* Beatrice Faggiano. Tel.: +39-081-7682447; fax: +39-081-5934792.
E-mail address: faggiano@unina.it

Peer-review under responsibility of the organizing committee of SUFTUS-2016
doi:10.1016/j.proeng.2016.11.572

of the SFT concept [1-7]. Compared to available strait-crossing techniques, namely bridges, immersed or undersea tunnels, the SFT offers many advantages such as lower environmental impact, less sensitivity to wind and seismic actions, higher adaptability to site morphology, etc [6]. Also, SFT turns to be economically more affordable when long distances are to be covered [8], namely in those cases where a long suspension bridge is not feasible and undersea tunnels have to cope with hard geotechnical problems. Nevertheless, in spite of the recent advances in the study of SFT [9-14], some significant aspects of its behavior are still awaiting proper investigation. In particular, the dynamic interaction between water current and tunnel structure has not yet reached a consolidated position into literature, most of all as far as the characterization of fluid action is concerned [15,16]. This aspect is important when a SFT has to be built in locations susceptible to relatively high-speed water currents, such as sea straits, river estuaries, and so on. In this paper, a sophisticated numerical analyses based on the coupled application of Computational Fluid Dynamics (CFD) and Finite Element Method (FEM), implemented in the ABAQUS 6.13 code [17] is carried out in order to investigate the behavior of SFT exposed to a water current of variable velocity. An accurate modelling of turbulent phenomena is made, solved by means of the Implicit Large Eddy Simulation (ILES) approach and also with the simplified Spalart-Allmaras model. The analysis is carried out in both static and dynamic ways, considering circular and elliptical cross sections with the same transportation layout and capacity, each of them fitted for combined railway and motorway service (Fig. 1). A submerged tunnel positioned at the depth of 50m and anchored to a 150m deep seabed has been selected as case study (Fig. 2). An appropriate volume of water upstream and downstream the tunnel has been considered, in order to reproduce the wake turbulent field as accurately as possible.

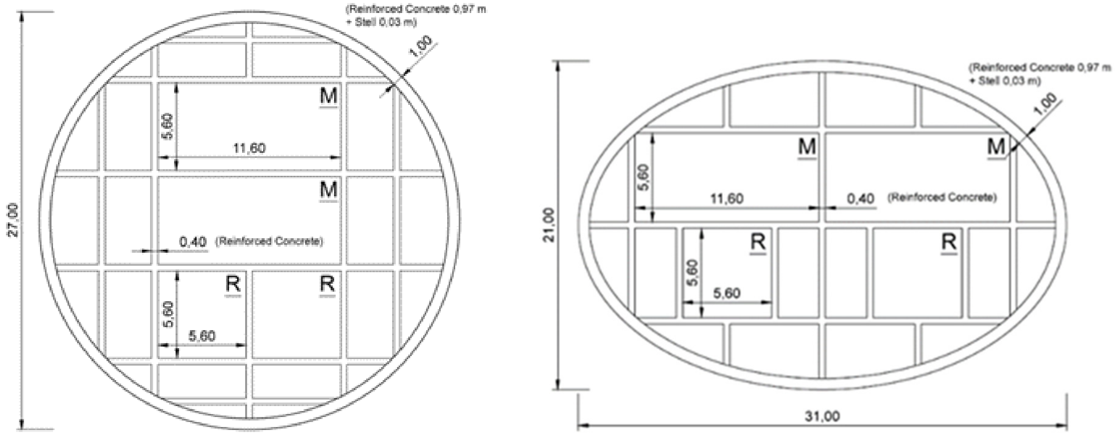

Fig. 1. Circular and elliptical cross-sections of SFT fitted for combined motorway (M) and railway (R) service (lengths in m).

2. Description of the FEM model

The study has been carried out on a 50m-long section, having symmetric boundary conditions at both ends, in such a way to neglect the effect of the tunnel external restraints (Fig. 2a). The whole section is anchored to the seabed by means of single system of retaining cables, according to the scheme shown in Fig. 2b. This corresponds to assume a distance between retaining cable systems equal to 50m. Anchorage cables are made of stainless steel strands with an overall diameter of 30cm and an effective elastic modulus of 140GPa. Cable scheme has been conceived in order to provide effective anchoring resistance along both horizontal and vertical directions. Spherical hinges have been assumed at both ends of the cables. The position of the tunnel in the water volume has been determined in such a way to reproduce in an accurate way the water flow field including the full turbulent phenomenon. This has led to assume a distance from the inlet upstream side equal to 4*D* (*D* diameter of the circular section = 27m). Likewise, a distance of 12*D* has been set so as to fully reproduce the wake turbulence that takes place downstream the tunnel. In order to avoid the effect of surface wave motion, a depth of 50m at tunnel center axis has been set. The cross-section of the tunnel outer shell is shown in Fig. 3a. It is made of a 97cm thick C30/35 concrete wall coated with a 3cm thick S355 steel plate. The internal concrete partitions are 40cm thick. For all materials elastic behavior has been assumed. The corresponding features are shown in Tab. 1. Applied serviceability loads have been determined according to EN1991-1 for both motorway and railway modules (Fig. 3b, 4). In addition to live loads, a ballast load has been applied so to reach a residual buoyancy factor under full loading conditions $R_b = 1.20$. The ABAQUS FEM structural model for both tunnel cross sections under consideration is represented in Fig. 5a,b. S4R reduced integration shell elements have been chosen to model the tunnel, whereas B31 elements have been used for cables. The average size of shell elements is approximately 2 x 2m. Beam elements are 2m long. The ABAQUS Computer Fluid Dynamics (CFD) model is shown in Fig. 5c. Eight-node continuous fluid 3D elements (FC3D8) have been used. Symmetrical boundary conditions have been imposed to the sides of the water volume. A constant load of 0.1 MPa corresponding to the atmospheric pressure has been applied on the top of the water surface. A water flow along the horizontal direction (Y-direction) with different velocity values has been assumed to enter at the inlet surface (Fig. 2a). Conversely, water speed at the seabed has been assumed equal to zero. The mesh represented in Fig. 5c is the result of a long optimization process, in which the best compromise between computational time and solution accuracy has been found.

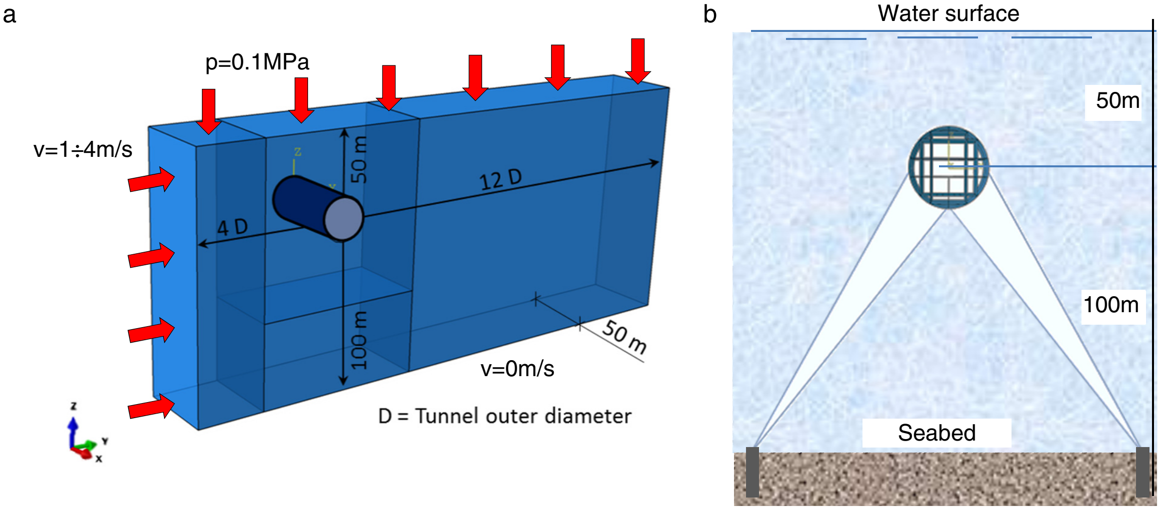

Fig. 2. (a) the modelled water volume with the position of the SFT; (b) the scheme of the retaining cables.

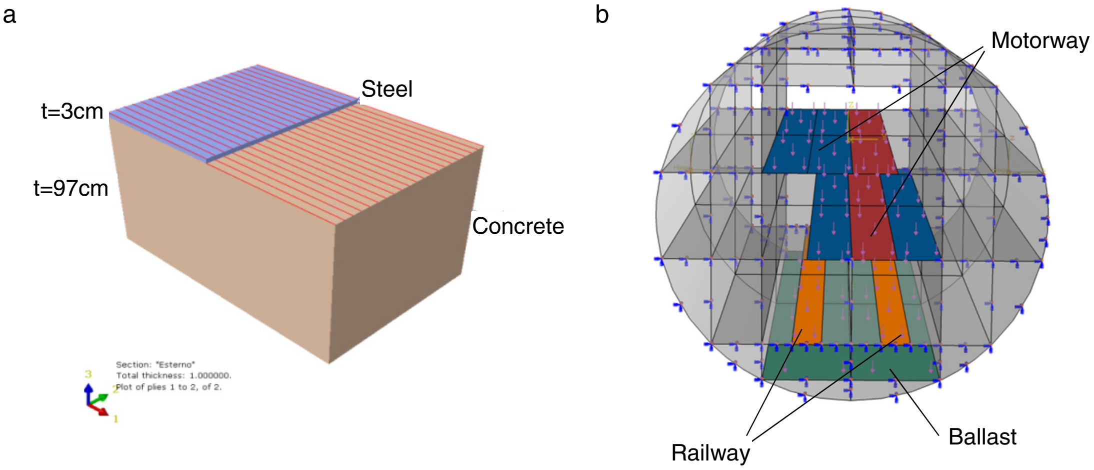

Fig. 3. (a) the cross section of the SFT outer shell; (b) the distribution of loads inside the SFT.

Table 1. Synopsys of material properties assumed in the analysis.

Material	Unit mass ρ [kg/m^3]	Elastic modulus E [GPa]	Poisson's modulus ν	Viscosity coefficient μ [Pa·s]
Sea water	1.021	-	-	0,001
Reinforced Concrete	2.500	33,3	0,1	-
Steel	7.850	210,0	0,3	-
Stainless steel cable	8.500	140,0	0,3	-

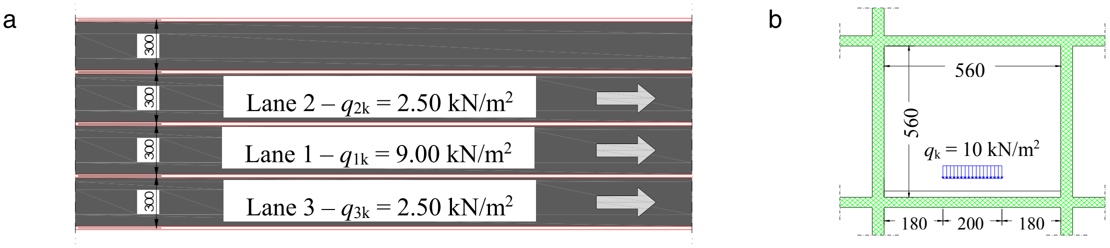

Fig. 4. (a) loading scheme on the motorway lanes and (b) on the railway module.

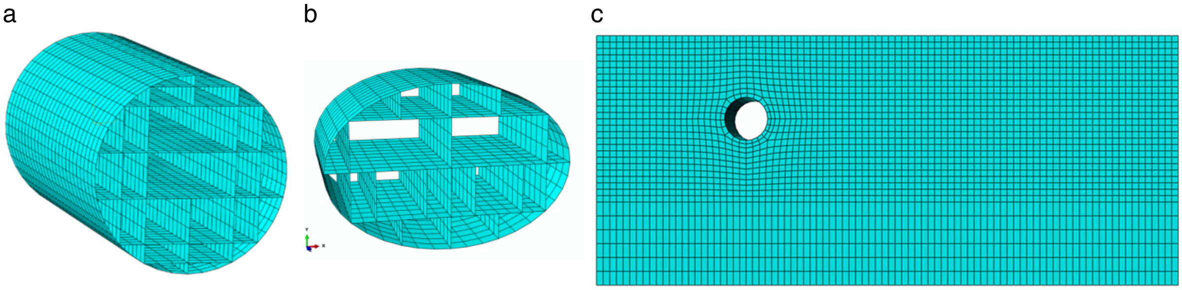

Fig. 5. The ABAQUS FEM model for (a) the circular SFT, (b) the elliptical SFT and (c) the water volume.

3. The CFD Analysis

3.1. The approach to turbulence modelling

The response of a SFT immersed in a water current of a given velocity is potentially affected by turbulent phenomena. The values of the Reynolds number *Re*, in fact, fall in the range of $10^7 \div 10^8$, meaning that fully turbulent flow conditions occur. Thanks to the great development of computational capabilities registered in the last decades, the numerical approach has gained a major interest for the study of turbulence. In case of SFT, analysis of turbulent problems may help to better understand the behavior of the tunnel under both ordinary and extreme working conditions and to trigger appropriate design choices. At the same time it can yield useful information about the influence of relatively small constructional details on the global hydrodynamic performance of the immersed structure. From the computational point of view, the problem of turbulence can be faced in several ways, which are shortly recalled hereafter [18,19,20].

Direct Numerical Solution (DNS). This is the simplest approach from the conceptual point of view. It is based on the direct integration of the Navier-Stokes equations and on the averaging of the flow field results. In the DNS approach all motion scale levels have to be solved, going from the length L of the scale at which the turbulence energy is introduced in the system (approximately 0.1 ÷ 0.3 times the overall size of the body) to the much smaller so-called Kolmogorov micro-scale η, corresponding to the eddy length at which the onset of viscous energy dissipation takes place. Assuming that a suitable calculation grid should have a size equal at least to η and considering that $L/\eta \cong Re^{3/4}$, it results that in a volume L^3 a total number of calculation points $N_{tot} \cong Re^{9/4}$ would be necessary. This makes clear that even for relatively low values of the Reynolds number *Re* the global number of unknowns would be enormously high, even for the very powerful calculation tools available today. This is the main reason why this approach is not usually applied in engineering practice.

A more affordable alternative to DNS is to average the flow governing equations, obtaining the so-called Reynolds-Averaged Navier-Stokes (RANS) approach, in which the mean component of the flow only is simulated, whereas the turbulent fluctuations are modelled. Contrary to DNS, in the RANS approach practically all turbulence scales must be modelled and only time mean quantities are directly computed. The principal problem of the RANS approach is represented by the fact that the derived averaged equations are less than the unknowns they contain, which is why they need to be integrated by additional relationships, usually based on approximate models. This represents the so-called "closure problem", which partially also afflicts LES approach as well as any averaged or filtered approach. One of such approximate models is the $k-\varepsilon$ turbulence model, in which two additional partial differential equations (one for turbulence kinetic energy k and one for turbulence kinetic energy dissipation rate ε) are introduced to characterize the eddy viscosities. When supplemented by suitable models, such as the $k-\varepsilon$ one, RANS-based procedures give results of acceptable accuracy at a fraction of the computational effort of DNS approach. For this reason, RANS-based procedures are widely applied for the solution of engineering problems.

The third family of possible approaches is represented by the Large-Eddy Simulation (LES), which falls at midpoint between DNS and RANS methods [21,22]. In particular, in LES methods direct simulation is made only of large-scale, energy-carrying motion, whereas the small scales are modelled. In particular, the aim of LES procedures is to limit the direct simulation to the so-called inertial subrange of turbulence energy spectrum. This leads to reduce the total number of calculation points to $N_{\mathrm{tot}} \cong Re^{3/2}$ for each time step, which implies a much lower computational cost compared with DNS. This makes LES approach quite affordable for a number of practical problems, in particular for not very high values of the Reynolds number. The large-scale part of the motion is defined by a suitable filtering process of the Navier-Stokes equations, followed by an appropriate modelling of unresolved turbulent stresses at small scales.

3.2. The FSI analysis procedure

Computer Fluid Dynamics (CFD) capabilities implemented in ABAQUS are manifold. A basic tool for the solution of the incompressible Navier-Stokes equations is provided, to be supplemented with a suitable turbulence model for the complete simulation of the turbulent flow. The following options are available in the code: *Implicit Large Eddy Simulation* (ILES), *Spalart-Allmaras* (S.A.), and *Re-Normalization Group* (RNG) k-ε model. These models cover a wide range of applications, including time-dependent flows and Fluid-Structure Interaction (FSI). In this paper ILES and Spalart-Allmaras models have been used to simulate the effect of water turbulence across the tunnel. Implicit LES is a reliable an effective methodology for modelling high Reynolds number flows [23]. As this model is inherently time-dependent, it requires a time-accurate solution of the incompressible Navier-Stokes equations, whose time scale is approximately that of an eddy-turnover time. The Spalart-Allmaras model [24] is a relatively simple, empirically based, one-equation turbulence model containing the eddy-viscosity $\tilde{\nu}$, with a nonlinear transport equation. When accurately calibrated, the model provides accurate predictions of turbulent flows without requiring a particularly high resolution in boundary layers. It performs effectively even in case of adverse pressure gradients and may be also used for flows where separation occurs. This leads to good accuracy at an acceptable computational cost. The ABAQUS code exploits the Spalart-Allmaras model in the framework of a RANS-based procedure.

The FSI analysis procedure of SFT has been based on a co-simulation technique, namely a process of run-time coupling of ABAQUS with another analysis program. This is an effective technique to perform multiphysics simulations and multidomain (multimodel) coupling, such as FSI. In a co-simulation process both programs run simultaneously and their interaction takes place through a suitable interaction region. In the case under consideration, co-simulation has been applied coupling ABAQUS Standard with ABAQUS CFD, the former modelling the SFT structure and the latter the water current. The outer surface of the tunnel and the inner surface of the water volume have been assumed as interaction region. Both surfaces have been interfaced to each other by means of a node-to-node coupling. The analysis has been performed in both static and dynamic ways. When dynamic analysis is performed, the hydrostatic pressure and water current to the SFT system are applied at the same time and the job is executed until it reaches a steady state condition, that is after not less than 60s. For the sake of numerical lightness, but also in order to highlight the effect of the water current on the tunnel behavior, the FSI between cables and water current has been not considered in the analysis. The cable elastic action on the tunnel buoyancy, however, has been duly accounted for in both static and dynamic analyses.

4. Discussion of results

A static analysis under the nominal serviceability loading conditions has been preliminary carried out. A residual buoyancy factor $R_b = 1.20$ has been assumed. This part of the analysis aimed at assessing the general stress conditions of the tunnel under the most frequent load combination. At the same time, it is useful to compare the result of the global vertical displacement of the tunnel under static load with the value achieved with dynamic CFD analysis under steady state conditions. A typical stress contour coming from static analysis is shown in Fig. 6a.

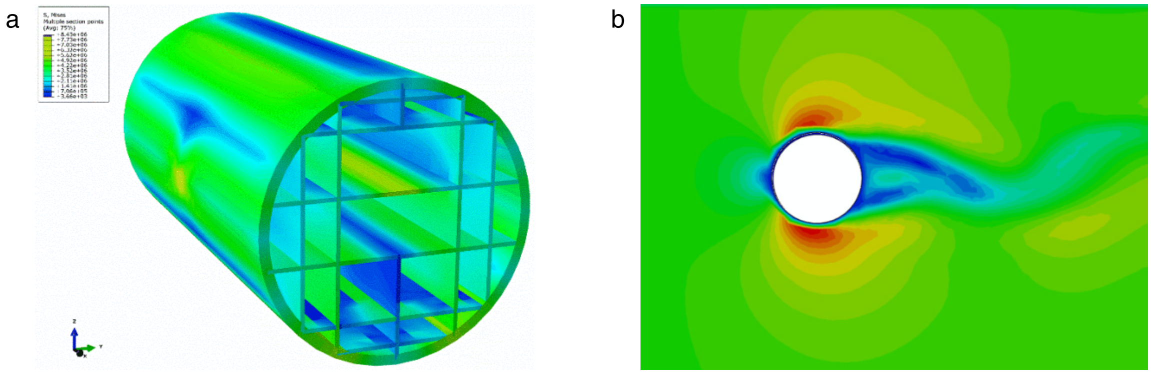

Fig. 6. (a) typical Von Mises stress contour obtained from SFT static analysis; (b) close-up of turbulence wake for circular SFT.

As a second step, the CFD dynamic analysis has been carried out using the ABAQUS FSI co-simulation tool. A water current with velocity ranging between 1,00m/s and 4,00m/s has been considered in the analysis. The results in terms of horizontal displacement obtained using both ILES and Spalart-Allmaras turbulence models are summarized in Tab. 2, whereas Fig. 6b shows a close-up of the turbulent wake downstream the circular tunnel. In general, the whole dynamic process, including the turbulent phenomenon is represented in a quite realistic way, as shown in Fig. 7, in which the time history of horizontal displacement of tunnel axis are plotted for the two cross sections investigated and for $v = 2$m/s. Such diagrams correctly show the initial oscillation due to the application of all acting loads at the same time, including the water current. Such oscillation is mostly the result of cable elasticity prompted by hydrostatic lift.

Table 2. Comparison between ILES and Spalart-Allmaras models in terms of horizontal displacement U_y under steady-state conditions.

Circular cross section				Elliptical cross section			
Velocity (m/s)	$U_{y,\,\mathrm{ILES}}$ (m)	$U_{y,\,\mathrm{S.A.}}$ (m)	Δ	Velocity (m/s)	$U_{y,\,\mathrm{ILES}}$ (m)	$U_{y,\,\mathrm{S.A.}}$ (m)	Δ
1,00	0,00889	0,00913	2,70%	1,00	0,0158	0,0161	1,90%
2,00	0,0143	0,0151	5,59%	2,00	0,0181	0,0190	4,97%
3,00	0,0230	0,0247	7,39%	3,00	0,0219	0,0239	9,13%
4,00	0,0355	0,0389	9,58%	4,00	0,0271	0,0306	12,92%

After a few initial fluctuations the displacement values tend to stabilize to a steady state figure, with some residual alteration due to the turbulence effect. This is observed for vertical displacements, too (Fig. 8a). As expectable, the elliptical section seems to behave slightly better from the hydrodynamic point of view for a given transportation capacity, showing lower values of the horizontal displacement compared with the circular one when loaded by the tunnel dead load and water current, only (Fig. 8b). This conclusion also stands for the global drag exerted on the tunnel by the water current (Fig. 9). In addition, Fig. 9b confirms the better behavior of elliptical section in terms of global drag and, hence, of global load applied to both retaining cables and foundation structures. The obtained plots of drag against water velocity are consistent with the well-known behavior under turbulent conditions, in which the drag depends on the square velocity. Also, shape coefficients derived by drag values (≅ 0,4 and ≅ 0,2 for circular and elliptical section, respectively) are comparable with those available in literature. In general, the effect of turbulence is comparatively more remarkable on horizontal displacements rather than on vertical ones, even though the elliptical section generally exhibits a smoother trend of displacement fluctuations with time. Eventually, the general reliability of obtained results is also indirectly confirmed by the low scattering between results provided using the ILES turbulence model and those obtained from the application of the less accurate Spalart-Allmaras model. In the end, the latter model has shown shorter computational times, which is why it has been used in most of the analyses carried out.

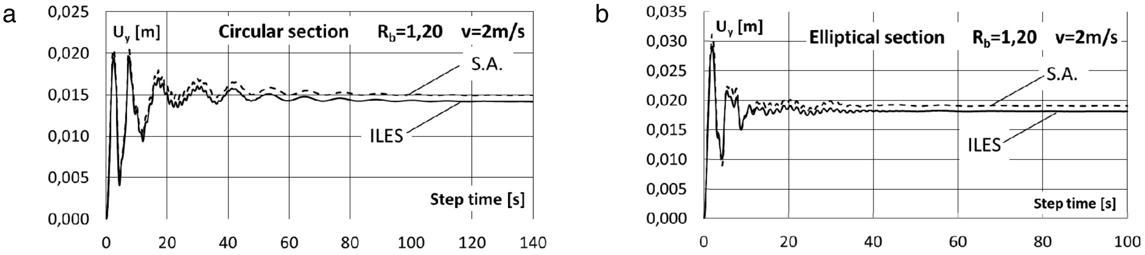

Fig. 7. Time history of the horizontal displacement U_y for circular (a) and elliptical section (b) (v = 2m/s).

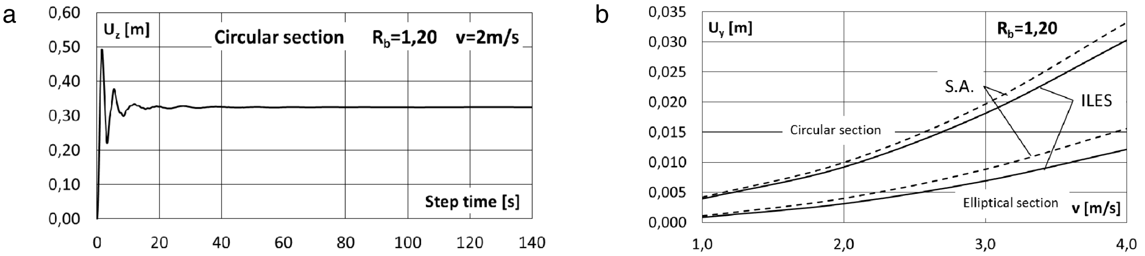

Fig. 8. (a) typical time history of vertical displacement for circular section (S.A. model); (b) U_y variation with velocity for both sections.

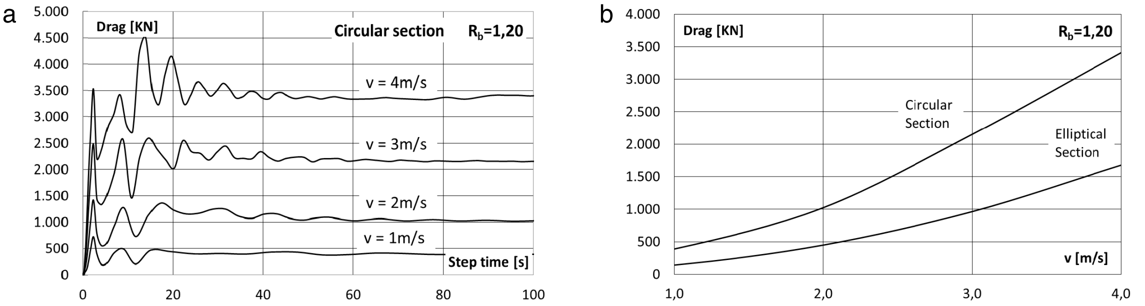

Fig. 9. (a) time history of the global drag exerted on the tunnel and (b) corresponding variation with velocity for both sections (S.A. model).

5. Conclusions

The study reported in this paper represents a preliminary step of a wider research plan dealing with both conception and design of SFT in the perspective of fluid-structure interaction. To this purpose, a refined numerical analyses based on the coupled application of Computational Fluid Dynamics (CFD) and Finite Element Method (FEM) implemented in the ABAQUS 6.13 code has been performed, aiming at pointing out the behavior of SFT exposed to a water current of constant velocity. Until now, in fact, most of research on SFT has been mainly devoted to the structural design of the tunnel and its anchoring system, without paying enough attention to the dynamic interaction with possible water currents.

This aspect, which can be of concern in some environmental and/or geographical conditions, has been faced herein exploiting the powerful capabilities of the numerical co-simulation tool embedded into the ABAQUS FEM code, which enabled the faithful reproduction of the complete fluid-structure interaction (FSI). Results of the analysis, carried out in static and dynamic way, show a very good physical consistence for both considered cases. In particular, the advantageous hydrodynamic behavior of the more streamlined elliptical cross section has been confirmed for the same internal transportation layout. Turbulent phenomena arising as a consequence of the water flow along the tunnel contour have also been investigated, using the Implicit Large Eddy Simulation (ILES) approach and the more affordable RANS-based Spalart-Allmaras single-equation turbulence model. In this context, relatively small differences have been registered between ILES and Spalart-Allmaras turbulence approaches, showing that for this class of problems the less demanding averaged RANS approach provides acceptable results at a lower computational cost.

In conclusion, the procedure illustrated in this study has demonstrated a good capability to cope with the complex phenomenon of the fluid-structure interaction for a SFT. The obtained values of horizontal displacements are fully compatible with structural serviceability requirements. At the same time, for the investigated values of the water velocity, the turbulent water flow should not represent a great issue, at least for the tunnel structure. Nevertheless, further research should be addressed to check the compatibility of turbulence-induced vibrations with specific motorway and railway requirements, in order to evaluate the use of possible damping devices on the tunnel retaining system. Likewise, as a future research step, the FEM model and the corresponding analysis procedure here described could be used to investigate the influence of details such as cables, anchorages, and other ancillary components, in order to get a complete understanding of the hydrodynamic response of SFT under all possible working conditions.

References

[1] L. Skorpa. Developing new methods to cross wide and deep Norwegian fjords. Procedia Engineering, 4, 2010, 81-89.

[2] S. Kanie. Feasibility studies on various SFT in Japan and their technological evaluation, Procedia Engineering, 4, 2010, 13-20.

[3] G. Martire, B. Faggiano, F.M. Mazzolani, A. Zollo, T.A. Stabile. Seismic analysis of a SFT solution for the Messina Strait crossing, Procedia Engineering, 4, 2010, 303-310.

[4] F.M. Mazzolani, B. Faggiano, G. Martire. Design aspects of the AB prototype in the Qiandao Lake. Procedia Engineering, 4, 2010, 21-33.

[5] B.Faggiano, G. Martire, F.M. Mazzolani. Cable supported immersed inversed bridge: A challenging proposal, Procedia Eng. 4, 2010, 283-291.

[6] H. Østlid. When is SFT competitive? Procedia Engineering, 4, 2010, 3-11.

[7] B. Faggiano, J. Panduro, M. T. Mendoza Rosas, F.M. Mazzolani The conceptual design of a roadway SFT in Baja California, Mexico. Procedia Engineering, in press.

[8] G. Martire, B. Faggiano, F.M. Mazzolani. Compared cost evaluation among traditional versus innovative strait crossing solutions, Procedia Engineering, 4, 2010, 293-301.

[9] B. Faggiano, F.M. Mazzolani, R. Landolfo. Design and modelling aspects concerning the submerged floating tunnels: an application to the Messina Strait crossing. 4th Symposium on Strait Crossing, Bergen, Norway, Sep 02-05, 2001 Ed. Krokeborg, J, 2001, 511-519.

[10] F.M. Mazzolani, R. Landolfo, B. Faggiano, M. Esposto, F. Perotti, G. Barbella. Structural analyses of the Submerged Floating Tunnel prototype in Qiandao Lake (PR of China), In International Journal Advances in Structural Engineering, 11, 4, 2008, 439-454.

[11] F. Perotti, G. Barbella, M. Di Pilato. The dynamic behaviour of Archimede's Bridges: Numerical simulation and design implications, Procedia Engineering, 4, 2010, 91-98.

[12] X. Hong, F. Ge, Y. Hong. Feasibility study on buoyancy–weight ratios of a submerged floating tunnel prototype subjected to hydrodynamic loads Acta Mech. Sin. (2015) 31: 750.

[13] F. Ge, W. Lu, X. Wu, Y. Hong. Fluid-structure interaction of submerged floating tunnel in wave field, Procedia Eng., 4, 2010, 263-271.

[14] X. Yiqiang, C. Chunfeng. Vortex-induced dynamic response analysis for the submerged floating tunnel system under the effect of currents. Journal of Waterway, Port, Coastal and Ocean Engineering, 139, 3, (2013) 183-189.

[15] G. Martire, B. Faggiano, M. Esposto, F.M. Mazzolani, R. Landolfo, A. Zollo, T.A. Stabile. The structural response of Submerged Floating Tunnel to multi-support seismic excitations, Proc. Int. Conf. STESSA09 “Seismic Behaviour of Steel Structure in Seismic Areas”, 2009, 19-25.

[16] G. Martire, B. Faggiano, F. M. Mazzolani, A. Zollo, T.A. Stabile. A comprehensive study on the performance of Submerged Floating Tunnels during severe seismic events. Proc. Int. Conf. STESSA2012 “Seismic Behaviour of Steel Structure in Seismic Areas”, 2012, 523-529.

[17] 3DS SIMULIA - ABAQUS 6.13 Analysis User’s Manual, 2013.

[18] J. M. McDonough. Introductory Lectures on Turbulence: Physics, Mathematics and Modeling. Departments of Mechanical Engineering and Mathematics, University of Kentucky. www.engr.uky.edu/~acfd/lctr-notes634.pdf, 2007.

[19] P. A. Davidson. Turbulence - An Introduction for Scientists and Engineers. Oxford University Press, 2004.

[20] F.T.M. Nieuwstadt, B.J. Boersma, J. Westerweel. Turbulence-Introduction to Theory and Applications of Turbulent Flows. Springer, 2016.

[21] M. Lesieur, O. M´etais. New trends in large-eddy simulations of turbulence, Annual Rev. Fluid Mech. 28, 1996, 45–82.

[22] P. Sagaut. Large Eddy Simulation for Incompressible Flows, an Introduction, Springer, Berlin, 2001.

[23] C. Fureby, F. F. Grinstein. Large eddy simulation of high-Reynolds-number free and wall-bounded flows, J. Comput. Physics, 181, (2002) 68–97.

[24] P. R. Spalart, S. R. Allmaras. A one-equation turbulence model for aerodynamic flows, AIAA Journal 92 (1992) 439. Physics 181 (2002) 68–97.