ICWE 16
August 27-31, 2023
FLORENCE - ITALY
16th International Conference on Wind Engineering

Partitioned numerical coupling for vortex-induced vibrations of a slender chimney equipped with tuned liquid damper

Victor Vîlceanu[1], Samir Chawdhury[2], Guido Morgenthal[3]

[1]*Bauhaus-University, Weimar, Germany, victor.vilceanu@uni-weimar.de*
[2]*Bauhaus-University, Weimar, Germany, samir.chawdhury@uni-weimar.de*
[3]*Bauhaus-University, Weimar, Germany, guido.morgenthal@uni-weimar.de*

SUMMARY:
Flexible and slender structures can be subjected to significant amplitude response due to wind excitation phenomena. Among these types, vortex-induced vibrations can occur at rather low wind speeds, to which undesirable resonant amplifications can produce fatigue or inactivity of services within the structure. One of the ways to mitigate the wind-induced vibrations, especially at resonant behaviour, is the use of passive dampers, such as Tuned Liquid Damper. In this paper, a partitioned CFD coupling approach is proposed between Vortex-Particle Method (wind flow domain) and Smoothed Particle Hydrodynamics (liquid flow domain). To showcase the study, a chimney with circular cross-section is subjected to resonant wind speed, and a comparison numerical analysis between the coupled approach and the uncoupled responses is shown.

Keywords: Vortex-Induced Vibrations (VIV), Tuned Liquid Damper (TLD), Partitioned CFD numerical coupling

1. INTRODUCTION

Vortex-induced vibrations (VIV) occur from wake flow past a bluff body, where vortices are shed at a specific frequency along a vortex street (Simiu and Scanlan, 1996). When the shedding frequency is close to the structural frequency, a lock-in phenomenon is produced in which large across-wind displacement amplitudes of the structure might occur, with subsequent fatigue or structural damage. Supplemental dampers can be installed at low costs to reduce the structural response as an alternative to aerodynamic shape modifications. Among passive dampers, liquid dampers are used with low operation costs, relying on sloshing forces to dissipate the energy (Kwon and Kareem, 2020). However, the VIV phenomenon exhibits nonlinear coupled behavior with unsteady wakes, and semi-analytical models often fail to describe these sophisticated flow characteristics. In this context, computational fluid dynamic (CFD) models can be a reliable alternative to wind tunnel experiments (Chawdhury and Morgenthal, 2021). Accurate modeling of TLD in coupled CFD methods is challenging; however, it allows the accurate prediction of complex nonlinear aerodynamic behavior of structures. A Lagrangian approach based on Smoothed Particle Hydrodynamics (SPH) has been used successfully in applications, proving to be a reliable tool (Green et al., 2021). The SPH can accurately describe the free surface wave behavior following an external excitation and, therefore, is used for sloshing. This paper uses a partitioned CFD numerical coupling methodology between two distinct CFD models to simulate the VIV response of a structure equipped with TLD.

2. METHODOLOGY

The CFD models employed for this study use Navier-Stokes (NS) equations for solving the fluid flow. Although both CFD models use Lagrangian approach to solve NS equations, the methods presented are as following: (i) the fluid-structure-interaction model uses the Vortex-Particles Method (VPM) to discretize the fluid domain (Morgenthal and Walther, 2007), while (ii) the sloshing model uses the Smoothed Particle Hydrodynamics (SPH) to analyze the TLD (Crespo et al., 2015). As an application case study for the proposed coupled numerical model, a slender steel chimney was selected with the dynamic properties described in Fig.1. The selection of TLD was made based on the first structural frequency and geometrical constraints (Faltinsen and Timokha, 2001). To determine the critical wind speed at which VIV response might occur, a static CFD simulation is performed to identify the Strouhal number.

2.1. VPM-Structure-SPH: Two-Dimensional Numerical Coupling

The VPM and SPH equations are coupled through the boundary conditions at the fluid-structure interface. First, the displacement in cartesian coordinates is determined as a result of structural movement from the vorticity calculation using VPM. Afterward, the tank is displaced according to each time step, and the overall sloshing force using the SPH method is fed back into the equation of motion along with aerodynamic force as external forces. The Newmark time integration scheme is used for solving the equation of motion. The mass (m), stiffness (k), and damping (c) are scalar values according to the SDOF dynamic characteristics corresponding to the first mode of vibration of the steel chimney (Fig. 1).

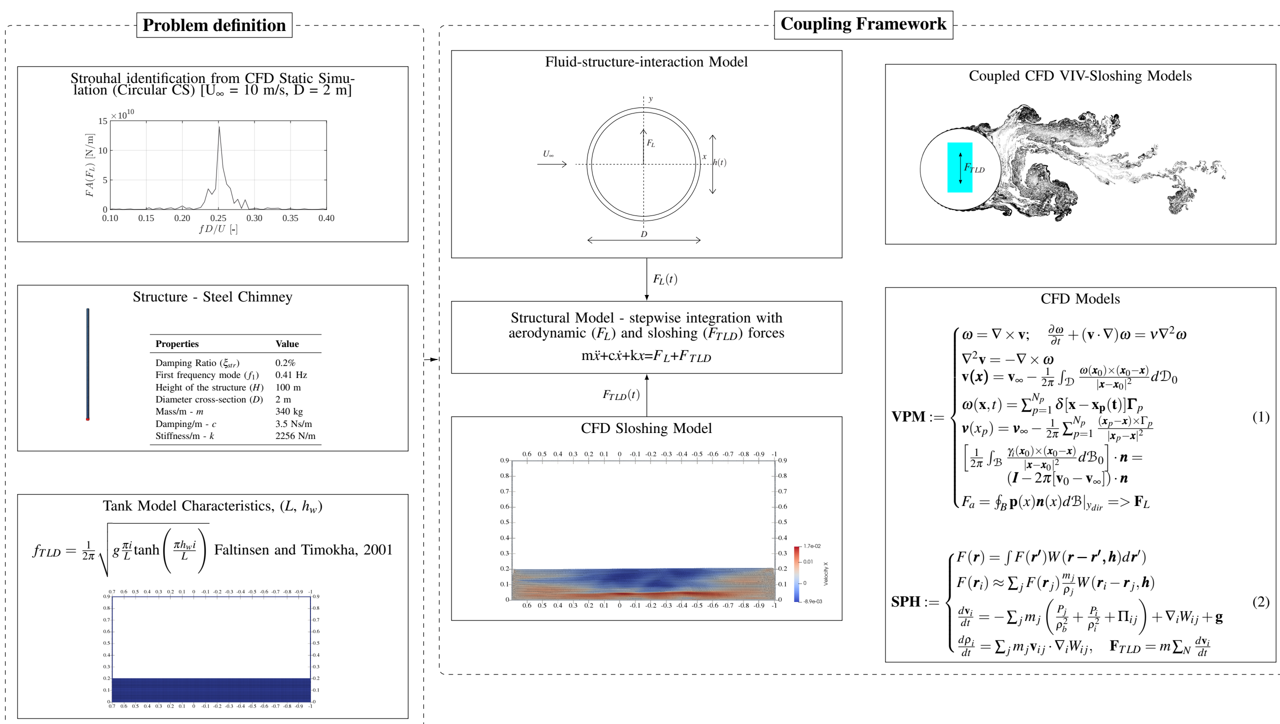

Figure 1. The framework of the two-dimensional partitioned coupling method between the TLD and aerodynamic model under VIV response. Equations from the bottom right block are gradually explained in the subsections below.

2.1.1. Fluid-Structure Interaction Model: Vortex-Particle Method (VPM)

The VPM describes the fluid flow dynamics through the vorticity field (Eq.1 in Fig. 1). This way, a gridless numerical scheme and particle discretization in a Lagrangian manner can be used. The third-row expression from Eq. (1) represents the velocity field by solving the Poisson equation with the use of Green's function, where ν is the kinematic viscosity, $\mathbf{v}_\infty$ is the free stream velocity, and $\mathcal{D}_0$ is the boundary domain attributed to the vortex strength integration. The vorticity field is discretized by vortex particles ($\boldsymbol{p}^{th}$), characterized by their location, $\mathbf{x}_p(t)$, and strength, $\boldsymbol{\Gamma}_p$. The vorticity field equation can be determined through the discrete form for particle velocity as described in fourth- and fifth- row, respectively (δ is the Dirac delta function, κ is the velocity kernel, N_p is the number of particles). In the sixth- and seventh- rows, the surface vorticity (γ_i) is discretized as sheets of linearly varying vorticity along the panels ($\gamma_i = \int_0^B \omega dB$). The aerodynamic forces, F_a, acting on the body are obtained by integrating the pressure over the boundary perimeter (B) based on surface vorticity, and the subsequent lift force is determined for the across-wind response (F_L).

2.1.2. Sloshing Model: Smoothed Particle Hydrodynamics Method (SPH)

The SPH solves the NS equations from Eq.2 in Fig. 1 using a set of material points (particles) in a Lagrangian approach. The fluid dynamics continuum is solved through a set of integral equations based on an interpolation function, referred to as kernel function (W). The smoothing kernel uses a function defined by the distance (r) between any two given particles. The discrete form can be interpolated at a particle i, with a summation of the support region, defined by the smoothing length, h, as shown in the first and second rows. To include the effects of dissipation in the fluid flow simulation, the artificial viscosity scheme is used, which was proposed by Monaghan (third row), where P_n and ρ_n are the pressure and density corresponding to particle n. The viscosity term Π_{ij} is expressed function of mean speed of sound ($\overline{c_{ij}}$), and the dynamic viscosity, μ_{ij} based on smoothing length, h, and α, the dissipation coefficient. The continuity equation for weakly-compressible SPH simulation assumes the mass of each particle, m_n, to remain constant, only their corresponding density fluctuating (fourth row). The overall sloshing force (F_{TLD}) is expressed by integrating the masses with their corresponding accelerations.

3. RESULTS

The VIV response of a steel chimney with TLD was numerically investigated in terms of displacement response for the first mode of vibration under laminar flow conditions. Based on the Strouhal number for the selected cross-section, the critical wind speed was identified as 3.28 m/s. However, larger displacements were noticed in the dynamic case with an incoming wind speed of 3.5 m/s. The critical case was selected for the numerical coupling; thus, the efficiency of TLD was determined (Table 1). The displacement time-history of the coupled CFD simulation is shown in fig.2. The efficiency of TLD is displayed through the out-of-phase sloshing forces compared to lift forces in fig.3.

Table 1. Left side: Displacement response for CFD dynamic simulations (no TLD, and with TLD) due to VIV response. Right side: CFD simulation characteristics. The time step is similar for both CFD models.

U_∞ [m/s]	3.5	D [m]	2	L_{tank} [m]	1.7
Disp - NO TLD [m]	0.497	$Re = DU/\nu[-]$	$5{\cdot}10^5$	h_{water} [m]	0.2
Disp - TLD [m]	0.011	Δt [s]	0.0107	$f_{sloshing}$ [Hz]	0.403

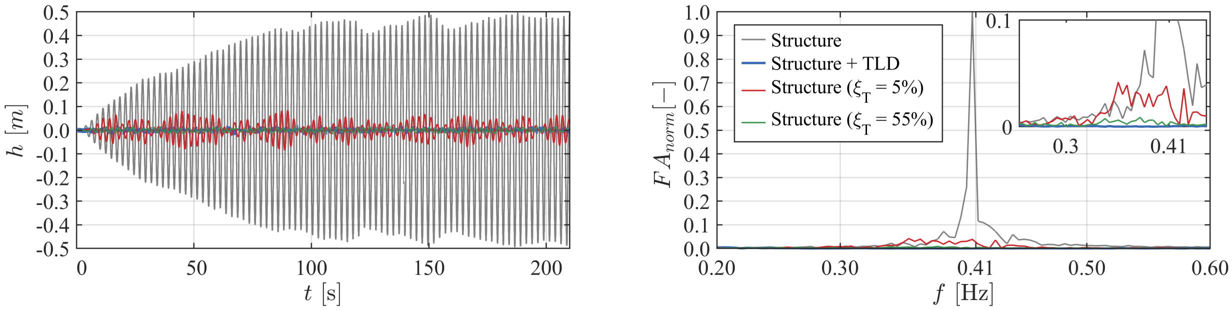

Figure 2. Across-wind displacement of the circular cross-section CFD simulations, with dynamic properties of the structure presented in Fig.1, and various structural damping scenarios. Laminar incoming wind speed - 3.5 m/s.

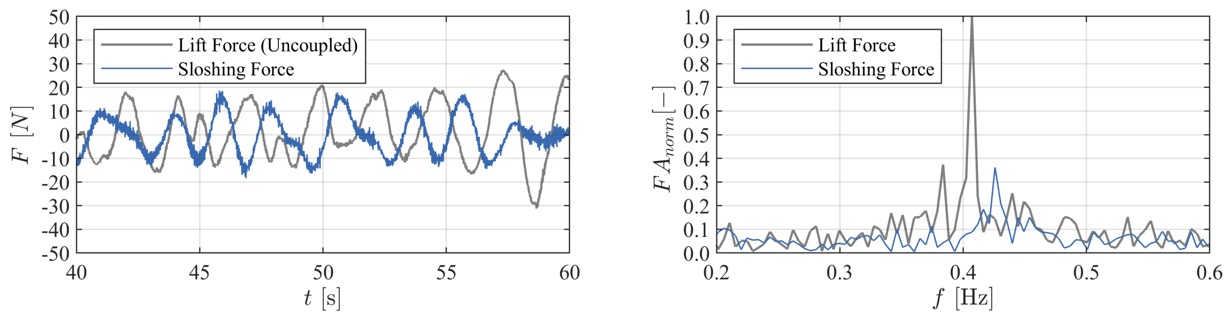

Figure 3. Out-of-phase of sloshing force coupled to the VIV response, and the VIV Lift Force of uncoupled system.

4. CONCLUSIONS

The efficiency of TLD for the slender structure was shown, in which a case of a slender steel chimney was showcased for VIV response. A numerical CFD partitioned coupling was performed, in which (i) the VPM was used for wind-structure-interaction, and (ii) the SPH method was used for determining the sloshing behavior. The CFD simulations were performed for a 2D case, with dynamic properties associated with the first mode of vibration. Across-wind displacements were observed, mainly around the critical wind speed (U=3.5 m/s), with large structural displacements caused by VIV. The results show a difference of 95% between the cases without and with TLD.

REFERENCES

Chawdhury, S. and Morgenthal, G., 2021. An extension of pseudo-3D vortex particle methods for aeroelastic interactions of thin-walled structures. Journal of Wind Engineering and Industrial Aerodynamics 208, 104391.

Crespo, A., Domínguez, J. M., Rogers, B. D., Gómez-Gesteira, M., Longshaw, S., Canelas, R., Vacondio, R., Barreiro, A., and García-Feal, O., 2015. DualSPHysics: Open-source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH). Computer Physics Communications 187, 204–216.

Faltinsen, O. M. and Timokha, A. N., 2001. An adaptive multimodal approach to nonlinear sloshing in a rectangular tank. Journal of Fluid Mechanics 432, 167–200.

Green, M. D., Zhou, Y., Dominguez, J. M., Gesteira, M. G., and Peiró, J., 2021. Smooth particle hydrodynamics simulations of long-duration violent three-dimensional sloshing in tanks. Ocean Engineering 229, 108925.

Kwon, D. K. and Kareem, A., 2020. Hybrid simulation of a tall building with a double–decker tuned sloshing damper system under wind loads. The Structural Design of Tall and Special Buildings.

Morgenthal, G. and Walther, J. H., 2007. An immersed interface method for the Vortex-In-Cell algorithm. Computers & Structures 85, 712–726.

Simiu, E. and Scanlan, R. H., 1996. Wind effects on structures: fundamentals and applications to design. 3rd ed. John Wiley & Sons, New York, USA.