An Analysis of Airflow Effects in Lift Systems

Nishant Singh[1], S. Kaczmarczyk[1], Thomas Ehrl[2]

[1]Faculty of Arts, Science and Technology, The University of Northampton, UK

[2]thyssenkrupp Elevator Innovation GmbH, Berner Feld 60, 78628 Rottweil, Germany

Keywords: Lift Aerodynamics, Computational Fluid Dynamics, Fluid-Structure Interactions, Piston Effect

Abstract. The current trends towards the design of lighter cars for high-speed lift systems and multiple car lift systems have encouraged the design of more aerodynamic efficient car geometries. Lighter lift cars are susceptible to aerodynamic drags and piston effects. The issue of piston phenomena affecting smoke control in traditional lift shaft configurations have been studied extensively. Considering the complexity of multiple car, multidirectional shafts and the susceptibility of lighter cars to aerodynamic drag and piston effects, it is important that relevant analysis is developed to determine the aerodynamic effects arising in those systems. With advances in the field of Computational Fluid Dynamics (CFD), it is now possible to compute 3D compressible large eddy simulation for a multi-car lift systems. A better understanding of piston effect in the context of lighter and faster multi-car systems is necessary to further calculate the impact of these forces on lighter structures. In this paper a coupled Fluid-Structure Interaction (FSI) model is developed based on stiffness and damping of the system and boundary values from transient CFD study. This study will help understand the impact of excitations due to aerodynamic forces and understand the effect of aerodynamic drags and piston forces in the multi-car shaft systems.

1 INTRODUCTION

The operation of lift systems is affected by vibrations and associated vibro-acoustic noise. Aerodynamic loadings due to the airflow around the car result in excessive noise and flow-induced vibrations of the car structure [1]. This affects ride quality and results in a high level of dynamic stresses in elevator components.

2 AERODYNAMIC PHENOMENA

The aerodynamic phenomena affect the performance of lifts. At high speeds the air flow around the car – frame assembly induces excessive vibrations and noise. During the lift travel large air pressure differences between the front and rear of the car are being generated [2]. Furthermore, the effects due to multiple cars running in the same shaft cannot be neglected. Funai et al. [3] conducted a computer simulation case study into these effects when two cars run parallel to and pass each other in a hoistway. The results indicate that the dominant frequency of air pressure fluctuations in the former case is around 3.7 Hz being close to the out-of-phase mode of the car – frame vibration mode. On the other hand, the dominant frequency of air pressure fluctuations in the latter case was 2.2 Hz.

A study to characterize the most important vibro-acoustic energy sources and identify the dominant paths of broad band (100 – 500 Hz) acoustic energy transmission to the car interior in high-rise installations has been carried out by Coffen et al. [4]. It has been identified that lift cars are subject to structure-borne as well as to air-borne noise. Structure-borne noise is caused mainly by the vibration induced by the car roller guides – guide rail interaction and by the hoist rope – rope hitch interface. This structure-borne vibro-acoustic energy is transmitted to the car interior through the car frame structure (and in particular by the uprights).

The air-borne noise is generated by aerodynamic effects during the car travel. It includes shaft noise entering the car through the ventilation openings and the door seals. The wind (flow)-induced vibrations of the car exterior panels generate noise that is transmitted to the car interior.

Finite element modelling, modal analysis and statistical energy analysis (SEA) are used as noise prediction techniques. The latter technique has yielded accurate results and facilitated the identification of the dominant sources and paths of vibro-acoustic energy in the lift car assembly [4]. Namely, it has been concluded that at higher speeds (over 9 m/s) the dominant path was air-borne noise radiating through the acoustic leaks and non-resonant energy transmission. The secondary path was identified as structure-borne noise arising from the car floor. However, at lower velocities (5 m/s) the contributions to interior car noise were the same for both paths.

Lift piston effects have been studied in the context of smoke control [5 – 6] and lift shaft pressurization [7]. A better understanding of piston effect in the context of lighter and faster multi-car systems [8] is required.

CFD has been an effective tool for the flow simulation for last decades. Incompressible CFD solvers reduces the computational effort but that saving in computational time is achieved at the cost of losing acoustic information from the problem, as wave propagation speed is infinite in the incompressible solver. Such acoustic information is often critical to the Fluid-Structure interaction problems. For a compressible solver, acoustic simulations or turbulent flow simulations require highly resolved spatial and temporal resolution and therefore are computationally expensive. However, a better understanding of lift aerodynamics and its interaction with lift structures requires a high fidelity compressible simulation.

3 MODELLING METHODOLOGY

A lift installation can be considered as a multi-body system (MBS) with discrete and continuous (distributed-parameter) components [1]. In a Fluid-Structure interaction scenario, the flow of the problem is solved using the Navier-Stokes equations and the structural part of the problem is solved using Lagrangian system of equations.

The instantaneous compressible Navier-Stokes equations can be transformed into an Unsteady Reynolds Average Navier Stokes form (URANS) through time averaging to form a set of equations as summarised below, see [9] for a full description:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}\widehat{u_i}}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial \bar{\rho}\widehat{u_i}}{\partial t} + \frac{\partial \bar{\rho}\widehat{u_i}\widehat{u_j}}{\partial x_j} = -\frac{\partial P}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial \bar{\sigma}_{ij}}{\partial x_j}, \tag{2}$$

$$\frac{\partial \bar{\rho}\hat{E}}{\partial t} + \frac{\partial \bar{\rho}\widehat{u_j}\hat{h}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\bar{\sigma}_{ij}\widehat{u_i} + \overline{\sigma_{ij}u_i'}\right) - \frac{\partial}{\partial x_j}\left(\bar{q}_j + c_p\overline{\rho u'T'} + \bar{\sigma}_{ij}\widehat{u_i}\tau_{ij} + \overline{\rho u'k}\right) \tag{3}$$

where $$P = (\gamma - 1)\{\bar{\rho}\hat{E} - \tfrac{1}{2}\rho(\hat{u}^2 + \hat{v}^2 + \widehat{w}^2) - \bar{\rho}k\}, \tag{4}$$

$\widehat{}$ denotes density weighted Favre-averaged variable, $\bar{}$ denotes averaged variable, $\bar{\sigma}_{ij}$ is usually modelled by the Boussinesq assumption (provides S_{ij}) and

$$\hat{h} = \hat{E} + \frac{\bar{p}}{\bar{\rho}}$$
$$\bar{q}_j = -\frac{c_p \hat{\mu}}{Pr} \frac{\partial \hat{T}}{\partial x_j}.$$

In the Unsteady RANS approach it is assumed that the time averaging process occurs over a period of time sufficiently long to capture the turbulent fluctuations, whilst still short in comparison to large scale temporal changes in the flow field.

In order to derive the differential equations of motion for the structural part of such a system, Lagrange's Equations techniques can be applied [5]. The use of Lagrange equations facilitates the derivation of equations of motion in terms of generalized coordinates, without the need of free body diagrams. Considering the difficulty of mixed mode vibrations in the context of moving mesh systems, a one-way-coupled system is adopted in this work where structure is assumed to have a prescribed vibration. OpenFOAM [10] has been used as a fluid solver and MATLAB module is developed for the structural solver in a Lagrangian framework.

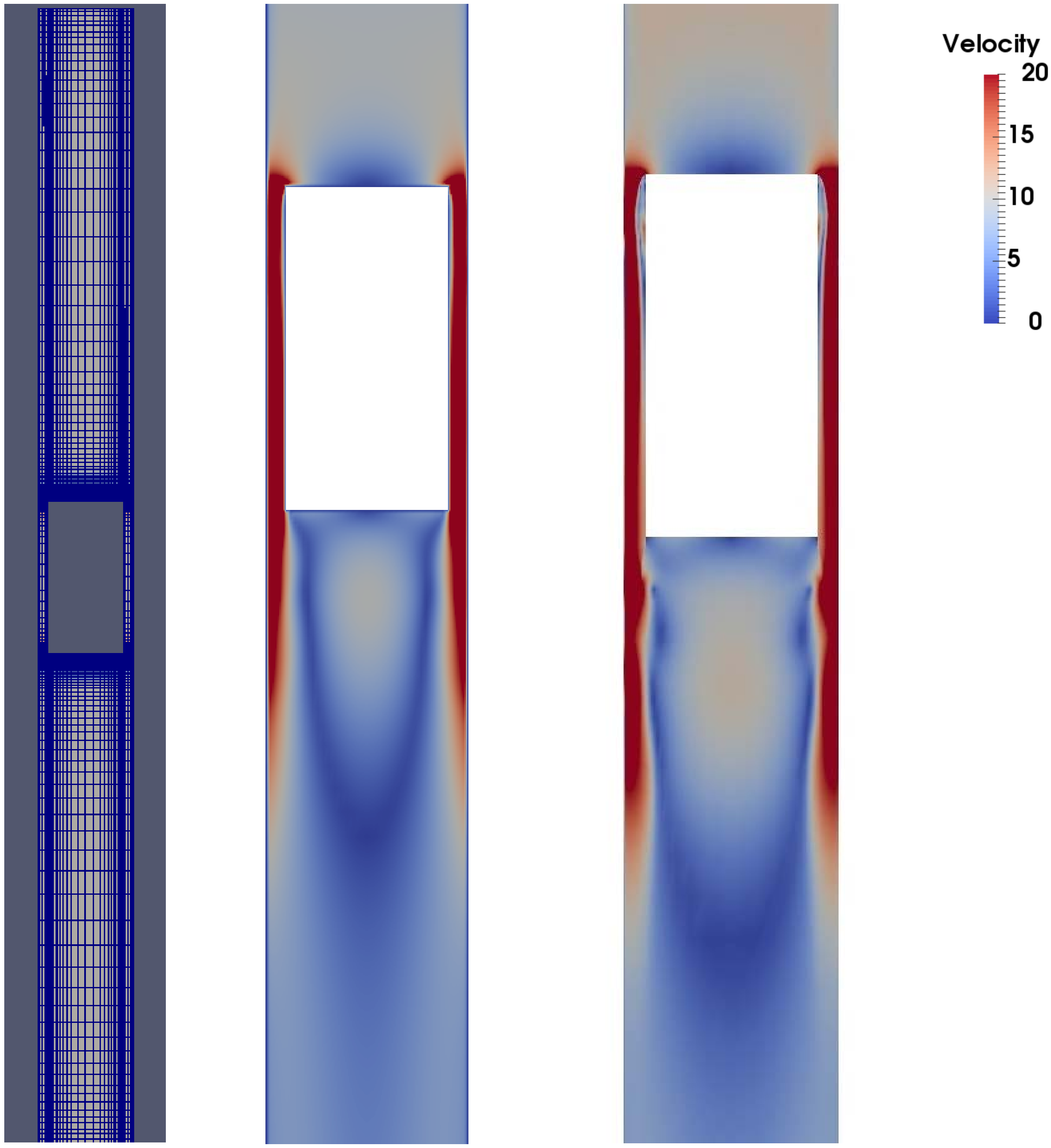
4 COMPUTER SIMULATION AND RESULTS

The fluid flow is coupled to the structure in one-way and the solution scheme is based on Lagrangian formulation for the structure and compressible CFD formulation for the fluid regions. The computer simulations are executed in open source OpenFOAM solver.

Table 1 Fundamental parameters of the system

Parameter	Value	Unit
Car	1000	kg
Frame	400	kg
Lift height	4	m
Hoist Length	30	m

The air properties are considered as density of 1.14 kg/m^3, specific heat ratio of 1.401 at 20 ^{0}C with gas constant (R) of 287 J/ (kg.K). A second order spatial and second order Crank Nicholson scheme is used for the CFD simulation in OpenFoam environment. A mesh of 5 million grid points with adequate near wall thickness is generated as shown in Figure 1(a). A velocity contour of the lift is shown in Figure 1(b). Apart from the RANS simulation, Detached Eddy Simulation (DES) with near wall K-ω SST turbulence model is also conducted on this geometry to obtain an accurate estimation of drag behind the lift body, as shown in Figure 1 (c). Simulation is conducted in parallel cluster on 64 cores at University computing facility.



(a) (b) (c)

Figure 1. *(a) CFD mesh. (b) Velocity contour (RANS). (c) Velocity contour (DES with K-ω SST).*

A structural model is formulated for the investigation of fluid forces on the structure. The general equation of motion for the lift car can be written as follows:

$$m\ddot{y} + c\dot{y} + ky = F(t) \tag{5}$$

Where *m* is the structural mass, *c* is structural damping, *k* is spring constant and *F(t)* is fluid force. Skop and Griffin [10] expressed the time varying force F(t) as:

$$F(t) = \frac{1}{2}\rho U^2 DlC_Y$$

Which can be transformed in terms of the Strouhal number (*St*) and the frequency of vibration (f_s) as follows:

$$F(t) = \frac{C_Y \rho D^3 l f_s^2}{2\, St^2} \tag{6}$$

Where D and l are geometric parameters and C_Y is the fluid dynamics force coefficient which is a function of y and $\dot{y}$. It is important to consider the geometry of the lift installation shown in Figure 1 (a). A simplified setup of current problem has been considered as a rectangular box in the passage. In addition, the surrounded wall facilitates a nozzle structure in the lift installation, as shown in Figure 1(c). Hence, the fluid force can have two components in this case and the Equation (6) can be divided into two different Strouhal number regimes, as below:

$$F(t) = \frac{C_{Y1} \rho D^3 l f_{s1}^2}{2\, St_1^2} + \frac{C_{Y2} \rho d^2 D l f_{s2}^2}{2\, St_2^2} \tag{7}$$

Where d is the clearance between the lift car and the surrounding wall, St_2 is the associated Strouhal number and f_{s2} is the frequency of vibration of the associated geometry. The Strouhal numbers are a known function of flow velocity and their associated structural parameters (Reynolds number). A known correlation of $St = f(U, L, D)$ can provide an analytical expression for the vibration associated with the structure.

The velocity contour in Figure 1(c) provides clear indication of presence of larger wake modes from the lift body and other shorter nozzle modes from the lift-wall clearance. A high-fidelity CFD simulation of a typical lift geometry can provide us with correct estimation of the frequencies in the wake region, either emanating from the lift wake or the wall clearance. A closer consideration of the model provides us with some important information. The Reynolds numbers associated with the key geometrical features of the lift car puts the geometry in the critical or supercritical range of flow features. Some overshoot in Strouhal number have been observed in these critical zones. A better understanding of these overshoots can be better understood with the compressible CFD studies.

5 CONCLUSIONS AND FUTURE WORK

The very fast development of the construction of high-rise buildings raises an essential need for the design of high-speed lifts, which has led to increasing interest in the fluid-structure interaction in such systems. This research describes the application of compressible CFD in development of an analytical formulation for identification of various modes leading to the vibrations in lift systems. CFD can aid to the development of the model either by correct estimation of Strouhal number or by comparing the actual averaged fluid forces or frequency of vortex-induced vibration from CFD to the model fluid forces of the analytical model. The flow field, pressure distribution, velocity, drag forces and the flow patterns have been studied in detail. Furthermore, high fidelity Detached Eddy Simulation with near wall model have been carried out for better estimation of vortices structure and accurate formulation of the model. Refinement of various parameters in the analytical model are facilitated with various CFD flow simulations. The initial results show an acceptable level of Strouhal number estimation, consistent with the analytical estimation for a rectangular bluff body. An acceptable level of one-way coupling between the CFD and structural model is achieved.

REFERENCES

[1] S. Mirhadizadeh, S. Kaczmarczyk, N. Tongue, H. Al-Jelawy, Modelling and Computer Simulation of Aerodynamic Interactions in High-Rise Lift Systems. Proceedings of the 5th Symposium on Lift and Escalator Technologies, Northampton, UK, 23-24 September 2015.

[2] G.X. Shen, H.L. Bai, A.T.P. So, Experiments on Aerodynamics of Super High Speed Elevators. Elevator Technology 15, Proceedings of ELEVCON 2005, June 2005, Beijing, China, pp. 174–184.

[3] K. Funai, H. Katayama, J-I. Higaki, K. Utsonomiya, S. Nakashima, The Development of Active Vibration Damper for Super High-Speed Elevators. Elevator Technology 14, Proceedings of ELEVCON 2004, April 2004, Istanbul, Turkey, pp. 81-89.

[4] C. Coffen, L. Hardin, T. Derwinski, Statistical Energy Analysis of a High Speed Elevator Cab and Frame. Proceedings of the 5th International Congress on Sound and Vibration, December 15-18, 1997, Adelaide, South Australia.

[5] J.H. Klote, An Analysis of the Influence of Piston Effect on Elevator Smoke Control [online]. (NBSIR 88-3751). U.S. Department of Commerce, National Bureau of Standards (1988).

[6] J.H. Klote, G. Tamura, Elevator Piston Effect and the Smoke Problem. Fire Safety Journal, 11 (1986) 227 – 233

[7] J.H. Klote, Elevator Pressurization in Tall Buildings, Journal of High-Rise Buildings, Vol. 2, No. 4, 341-344 (2013).

[8] J. Missler, T. Ehrl, B. Meier, S. Kaczmarczyk, O. Sawodny, Modelling of a rope-free passenger transportation system for active cabin vibration damping. Lift and Escalator Symposium, 2016, 6, pp. 147-158. 2052-7225.

[9] N.K.Singh, (2012). Large Eddy Simulation of Acoustic propagation in Turbulent Flow through Ducts and Mufflers. PhD Thesis, University of Hull.

[10] CFD Direct, (2015). OpenFOAM Features. [Online] Available at: http://www.openfoam.org/features/ [Accessed 15 June 2017].

[11] Skop,R.A. & Griffin, O.M., 1975. "On a theory for the vortex-excited oscillations of flexible cylindrical structures." Journal of Sound and Vibration, 41(3), 263-274.

[12] S. Kaczmarczyk, The Dynamics of Vertical Transportation Systems: from Deep Mine Operations to Modern High-Rise Applications. Proceedings of the 13th International Conference on Dynamical Systems: Theory and Applications, Lodz University of Technology, Poland, 07-10 December 2015.

[13] Y. Bazilevs, K. Takizawa, T.E. Tezduyar, Computational Fluid-Structure Interaction: Methods and Applications. Wiley & Sons Ltd, Chichester, United Kingdom, 2013.