Book of abstracts

Programme (*as of 05/07/2022)*

Start time	Tuesday, 5th			
8h30	Authors breakfast - The authors delivering a speech on Tuesday are invited to meet their chairman in order to have their presentations ready.			
9h00	Registration and breakfast			
9h30	A word of welcome			
10h00	**S. Candel**, French Academy of Science			
10h30	Break			
11h00	**Chair: M. Hassan**		**Chair: L. Baranyi / Amphithéâtre 1**	
	BB1	Dimensioning of a solar tracker torque tube for torsional galloping, **Martínez García E., Parrondo Gayo J., Blanco Marigorta E., Navarro Manso A.**	AL1	Impact of the nozzle geometry on the aeroelastic instability of a plate subjected to an air jet, **Tatin A., Cluzel X., Mourlot Y., Hémon P., Ramananarivo S.**
	BB4	Study of the oscillation process and the wake resulting from the accelerated flow over two free-to-rotate tandem cylinders and the effect of a perturbation applied on the flow, **Habowski P., Fiorot G., Neumeister R., Möller S.**	FSI-AL2	The Effects of a Passive Tail on Escape Performance in a Robotic Fast-Start Fish Capable of Rapid Underwater Locomotion, **Currier T., Modarres-Sadeghi Y.**
	BB6	Vortex-induced vibration of a circular cylinder subjected to low-Keulegan-Carpenter-number oscillatory flow, **Dorogi D.**	FSI-AL1	Quantitative Flow Imaging Approach to Unsteady Loading on High-Inertia Oscillating Foils, **Oshkai P., Lee W., Iverson D., Rahimpour M.**
	BB7	Vortex-Induced Vibrations of a One-Degree-of-Freedom Cylinder Transitioning from the Inline to the Crossflow Direction, **Benner B., Modarres-Sadeghi Y.**	FV1	Simultaneous control rod 3D displacement and 3D flow measurements via time resolved 3D3C PTV with one camera only, **Fichet V., Daoudi M., Zimmer L.**
12h30	Lunch			
14h00	Vibration in steam generators - **Prof. M. Hassan**, Guelph University			
15h00	Break			
15h30	**Chair: P. Hémon**		**Chair: M. Hirschberg / Amphithéâtre 1**	
	TV3	Flow-induced vibrations of a flexibly mounted cylinder in the proximity of a stationary parallel cylinder, **Riazat M., Kheiri M., Vermeire B.C.**	FSI3	Developing numerical methods for predicting flow-induced underwater radiated noise from ships, **Mcintyre D., Oshkai P.**
	TV6	GO-VIKING: a HORIZON europe project on flow-induced vibrations, **Zwijsen K., Papukchiev A., Vivaldi D., Hadzic H., Benhamadouche S., Benguigui W., Planquart P.**	FSI2	Articulated beam behaviour under grazing flow, **Abily T., Humbert T., Aurégan Y.**
	TV8	Numerical prediction of Axial-Flow-Induced Vibrations in nuclear fuel rod, **Salachna J., Cioncolini A., Iacovides H.**	FSI1	A Perforated Plate Solution to Mitigate Relief Valve Piping Vibration due to Flow-Excited Acoustic Resonance, **Pontaza J., Menon R.**
	CF2	Passive control of the turbulent flow past a finite circular cylinder fitted with eight peripheral rods, **Carvalho I., Assi G.**	FSI4	Direct measurements of the dynamic lift force acting on rectangular rods in cross-flow during acoustic resonance excitation, **Shoukry A., Mohany A.**
	CF1	Experimental investigation on the optimal control of vortex shedding of a circular cylinder with rotating rods at moderate Reynolds numbers, **Silva P., Assi G.**	FSI9	Using flow to control the damping of a resonant duct, **Humbert T., Aurégan Y.**
17h30	End of the day			

Programme (*as of 05/07/2022*)

Start time	Wednesday, 6th			
8h30	Authors breakfast - The authors delivering a speech on Wednesday are invited to meet their chairman in order to have their presentations ready.			
9h00	**Chair: S. Möller / Auditorium**		**Chair: C. Habchi / Amphithéâtre 1**	
	TA4	On the stability of the rotated square array in two-phase flow using the quasi-steady model, **Darwish S., Mureithi N., Cho M.**	CFD5	FSI simulations of fluid-elastic instabilities of a clamped-clamped cylinder in axial flow, **Delcour L., Dolfen H., Van Langenhove L., Degroote J.**
	TA2	Experimental investigation of cross-flow fluidelastic instability for rotated triangular tube bundles subjected to single-phase and two-phase transverse flows, **Panunzio D., Lagrange R., Piteau P., Delaune X., Antunes J.**	CFD2	A practical approach to using CFD as an early design tool for estimating aerodynamic force coefficients of bridge decks, **Duranovic M., Dempsey T., Meskell C.**
	TA5	Theoretical and experimental study on the fluidelastic instability of rod bundle subjected to jet cross-flow, **Gadelhak I., Mureithi N., Karazis K.**	CFD3	CFD analysis of two-phase flow induced forces on a test flow loop, **Emmerson P., Lewis M., Barton N.**
	BB3	Influence of a control wire on vortex shedding from side-by-side cylinders, **Hammad O., Mohany A.**	CFD1	A discrete forcing method to solve hyperelastic deformation induced by two-phase flow, **Merigoux N., Benguigui W., Baraglia F.**
	BB5	Unsteady wall pressure measurements on a full scale flexible chimney subject to natural wind, **Manal Y., Hémon P.**	CFD7	Multi-scale methodology for the large eddy simulation of steam control valves, **Galpin J., Amice B., Goreaud N., Leconte G., Joly A., Moussou P., Glau A.**
11h00	Break			
11h30	The way of the SDOF – A tribute to R. J. Gibert,			
12h30	Lunch			
14h00	Smart morphing and sensing for aeronautical configurations - **Prof. M. Braza**, Institut de Mécanique des Fluides de Toulouse			
15h00	Break			
15h30	**Chair: J. Antunes / Auditorium**		**Chair: P. Oshkai / Amphithéâtre 1**	
	SM2	Wind Energy Harvesting from Flow-Induced Vibration of Prisms Using Magnetostrictive Material, **Heragy M., Kiwata T., Shima T., Kono T., Hamano T., Ueno T., Ekmekci A.**	FSI5	Flow structure, dynamic lift force, and aeroacoustic response of finned cylinders in cross-flow, **Alziadeh M., Mohany A.**
	SM1	Pressure Driven Soft Vortex Generator, **Khanjian A., Habchi C., Russeil S., Bougeard D., Lemenand T.**	FSI6	Flow-induced tones in a deep periodic cavity, **Golliard J., Aurégan Y.**
	TV5	Fluid structure interaction in a pressure vessel: turbulent forcing, **Kocher M., Moussou P., Panunzio D., Lagrange R., Joly A.**	FSI7	On broad-band noise of thick square-edged orifices in water-pipe flow, **Kottapalli S., Hirschberg A., Waterson N., Smeulders D., Nakiboglu G.**
	TV4	Fluid structure interaction in a pressure vessel: a multipole approach for acoustic analysis, **Moussou P., Kocher M., Panunzio D., Lagrange R., Joly A.**	FS8	The aeroacoustics repsonse of cylindrical cavities in confined flow, **Hanna M., Mohany A.**
17h10	End of the day			
19h30	Conference dinner			

Programme (*as of 05/07/2022)*

Start time	Thursday, 7th			
8h30	Authors breakfast - The authors delivering a speech on Thursday are invited to meet their chairman in order to have their presentations ready.			
9h00	**Chair: G. Assi / Auditorium**		**Chair: A. Mohany / Amphithéâtre 1**	
	TV1	Aspects of vortex-induced in-line vibration at low Reynolds numbers, **Konstantinidis E., Dorogi D., Baranyi L.**	AF1	Aeroelastic effects in a planar flat blade cascade at high Mach number flow, **Šidlof P., Šimurda D., Lepicovsky J., Štěpán M., Vomáčko V.**
	TV9	Numerical simulation of cantilever cylinders in cross-flow: participation to the OECD/NEA fluid-structure interaction benchmark, **Zwijsen K., Hussain M., Roelofs F., Van Zuijlen A.**	CFD6	Modelling vortex induced vibrations in a model of the northern spire bridge, **Duranovic M., Dempsey T., Meskell C.**
	TV10	Numerical simulations of experimental fluid-induced vibrations of cylinders in cross-flow, **Vivaldi D., Ricciardi G.**	MF6	Two-phase flow induced vibration in a tube bundle of steam generators, **Fichet V., Khaddaj Mallat B., Mourgues A., Moulin J., Andrzejewski Q.**
	TV7	In wind tunnel simulation of vortex shedding behind circular cylinders at high reynolds number regimes is incomplete, **Hémon P., Ellingsen O., Amandolese X.**	MF2	Experimental investigation of void fraction distribution behind a cylinder, **Benguigui W., Pinto C., Ries O.**
	TV2	Experimental investigation of vortex-induced vibrations of a circular cylinder under rotary oscillations, **Schmider A., Kerherve F., Cordier L., Spohn A.**	MF1	A new experimental facility for two phase flow characterization in a tube bundle and vibration study, **Spina G., Vivaldi D., Brillant G., Colin C., Benguigui W., Denèfle R., Lelong M.**
11h00	Break			
11h30	Damping in fluids and structures - **Dr. H. G. D. Goyder**, Cranfield University			
12h30	Lunch			
14h00	Free afternoon, with optional tourist tours - The local committee will be pleased to assist attendees in visiting famous and less known places around Paris.			

Programme (*as of 05/07/2022)*

Start time	Friday, 8th			
8h30	Authors breakfast - The authors delivering a speech on Friday are invited to meet their chairman in order to have their presentations ready.			
9h00	Uses of potential flow solutions in fluid-structure interaction - **Prof. Ch. Eloy**, IRPHE Marseilles			
10h00	Break			
10h30	**Chair: C. Habchi / Auditorium**		**Chair: P. Šidlof / Amphithéâtre 1**	
	MF4	High pressure multiphase induced vibrations: influence of pipe orientation, **Belfroid S., Gonzalez-Diez N., Lunde K., Orre S.**	BIO1	Mitigating jet cross-flow induced vibrations using a bio-inspired nozzle, **Gadelhak I, Mureithi N., Karazis K.**
	MF3	Forces and displacements in a bend subjected to an air-water flow, **De Moerloose L., Dolfen H., De Paepe M., Degroote J.**	BIO2	Self-Oscillating Hydrogel-Based Vocal Fold Models for Voice Production Research, **Thomson S., Greenwood T.**
	MF5	Periodic Wake Shedding of Tube Bundles Subjected to Two-Phase Cross Flow, **Taylor C., Pettigrew M.**	TA3	Experimental investigation of in-flow fluidelastic instability for rotated triangular tube bundles subjected to single-phase and two-phase transverse flows, **Antunes J., Piteau P., Delaune X., Panunzio D., Lagrange R.**
	DSS1	Development of an efficient calculation technique for dynamics of mooring lines by using discrete forms of rotation, **Hara K., Shimojima K., Yamaguchi T.**	TA6	Transient vibration phenomenon due to passing of gap vortex street in FSI simulation of tube bundle with eccentricity, **Dolfen H., Degroote J.**
	DSS2	Performance of a Closed Cycle Power Take Off for Mutriku breakwater, **Bellec M., Gurhy C., Gibson L., Meskell C.**	TA1	A new criterion for the instability threshold of a square tube bundle subject to an air-water cross-flow, **Lagrange R., Panunzio D., Piteau P., Delaune X., Antunes J.**
12h30	Lunch			
14h00	**Chair: G. Assi / Auditorium**		**Chair: Y. Aurégan / Amphithéâtre 1**	
	BB8	Wake induced vibration in tandem cylinders: part 1- wake perturbation analysis, **Neumeister R., Ost A., Habowski P., De Paula A., Petry A., Möller S.**	AF2	Axial flow damping investigation by means of 2D CFD, **Berland J., Corre S., Joly A., Martin A.,Moussou P.**
	BB9	Wake induced vibration in tandem cylinders: part 2 - hilbert-huang spectral analysis, **Ost A., Neumeister R., Petry A., Möller S.**	AF3	Dynamics of cantilevered pipes conveying fluid and subjected to reverse annular external flow: experimental investigation of the influence of external flow confinement, **Chehreghani M., Shaaban A., Misra A., Paidoussis M.**
	BB2	Flow-induced vibration of a circular cylinder transverse to oscillatory flow at a high Keulegan-Carpenter number, **Dorogi D., Baranyi L., Konstantinidis E.**	CFD4	Dynamic response of a cantilevered pipe aspirating fluid and subjected to reverse confined external flow: a computational coupled two-way fluid-structure interaction analysis, **Daneshmand F., Liaghat T., Paidoussis M.**
16h00	End of the congress			

Table of contents

Annular and leakage flow

[AL1] Impact of the nozzle geometry on the aeroelastic instability of a plate subjected to an air jet

Antoine Tatin[1,2] Xavier Cluzel[2], Yannick Mourlot[2], Pascal Hémon[1], Sophie Ramananarivo[1]

1 : LadHyX (CNRS - Ecole Polytechnique) 91128 Palaiseau Cedex - France
2 : Fives Stein 108-112 avenue de la Liberté 94700 Maisons-Alfort - France

We study experimentally the flutter instability of a rigid plate supported elastically and impacted by an air jet. The behavior of the plate, modelled as a damped oscillator, is altered by the jet which can generate a negative added damping. As the total damping becomes negative, the plate undergoes growing amplitudes oscillations. We discuss the influence of the geometry of the nozzle, appearing as one of the crucial parameter in the aeroelastic mechanism. The results are compared with the theoretical model of Antoine et al (2008), showing some discrepancies.

Axial Flow and thin shells

[AF1] Aeroelastic effects in a planar flat blade cascade at high Mach number flow

Petr Šidlof [1], David Šimurda [2], Jan Lepicovsky[2], Martin Štěpán[1], Václav Vomáčko[1]

1 : Technical University of Liberec, NTI FM Studentska 2, 460 01 Liberec 1 - Czech Republic
2 : Institute of Thermomechanics, Czech Academy of Sciences Dolejškova 5, 182 00 Prague 8 - Czech Republic

The paper reports on measurements of unsteady pressure and aeroelastic response of a simplified turbine blade cascade under kinematic excitation. The flow velocities ranged from high subsonic to transonic regimes and reduced frequencies were between $k = 0 – 0.41$. Unsteady pressures measured by six miniature pressure transducers in the oscillating blade are shown for one representative measurement, and blade oscillation amplitudes compared for the whole measurement matrix. The results show strong pressure oscillations near the leading edge due to switching between subsonic and supersonic airflow. The oscillation amplitudes of the blade sharply increase with the frequency and are weakly influenced by the Mach number.

[AF2] Axial flow damping investigation by means of 2D CFD

Aurélien Joly[1], Sébastien Corre[1], Julien Berland[1], Alexandre Martin[2], Pierre Moussou[1]

1 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA EDF Recherche et Développement 7 boulevard Gaspard Monge, 91120 Palaiseau - France
2 : Laboratoire de Mathématiques Blaise Pascal Centre National de la Recherche Scientifique : UMR6620, Université Clermont Auvergne : UMR6620 Campus universitaire des Cézeaux, 3 place Vasarely, TSA 60026 / CS 60026, 63178 Aubière Cedex - France

Cylinders in axial flow are known to undergo strong damping when vibrating transversely to the flow. The aim of the paper is to assess the ability of a 2D CFD method in predicting accurately the damping coefficient. For this study, the specific geometry of a confined 3x3 cylinder array is chosen, since recent experimental and 3D numerical results are available in this configuration. The numerical method can be understood as a 2D-3C resolution of the fluid dynamics case. The fluid domain has only one layer of cells in the main flow direction and periodic boundary conditions are applied on both sides. This way, the three components of the outlet velocity are reinjected at the inlet. The flow is maintained by means of a momentum source term in each cell that compensates for the friction losses. A harmonic transverse motion is imposed on the central cylinder. The fluid force is recorded; the damping coefficient is obtained from the fluid force component in phase with the motion velocity. Its value is close to the one from experiments and 3D CFD, which validates the method. A parametric study on confinement shows a relatively low sensitivity to that parameter. Imposing a circular motion of the cylinder, i.e. a composition of two harmonic motions (one in each transverse direction), reveals that the damping coefficient is the same in both directions and very similar to that of a simple harmonic motion. The 2D-3C method used here has thus proven useful in characterizing axial flow damping, since its lower computational cost allowed parametric studies, although it requires some subtle tuning of the momentum source term.

[AF3] Dynamics of cantilevered pipes conveying fluid and subjected to reverse annular external flow: experimental investigation of the influence of external flow confinement

Mahdi Chehreghani, Ahmed Shaaban, Arun Misra, Michael Paidoussis

Department of Mechanical Engineering, McGill University 817 Sherbrooke St. West, Montreal, QC H3A 0C3 - Canada

The present paper deals with the dynamics of a hanging cantilevered pipe discharging fluid downwards and subjected to a counter-current partially confined external axial flow over its upper portion. This study is both `curiosity driven' and application-oriented. A bench-top-size apparatus consisting of a pressure vessel filled with water, a hanging

cantilevered pipe and a shorter concentric outer rigid tube was utilized. For various ratios of the external to internal flow velocity, Uo/Ui, the effect of the size of the annular gap between the pipe and the outer rigid tube was investigated. It was found that for the same Uo/Ui, not only does confinement affect the onset of instability, but also its type. For Uo/Ui >0.055 and a wider outer tube, as the flow velocity increased, the pipe underwent first mode oscillations prior to second mode flutter, eventually impacting on the outer rigid tube. For a narrower outer rigid tube, static deformation took place prior to oscillatory instability and impacting.

Bluff bodies Bluff body/near-wake interactions

[BB1] Dimensioning of a solar tracker torque tube for torsional galloping

Antonio Navarro Manso[1], Eva Martínez García[2], Jorge Parrondo Gayo[2], Eduardo Blanco Marigorta[2]

1 : University of Oviedo Oviedo - Spain
2 : University of Oviedo C/Wifredo Ricart, Edificio Este. Campus de Gijón (33204). Gijón - Spain

Solar trackers with a single axis (longitudinally arranged panels on a torque tube, driven by a motor in the central section) have evolved to extremely slender structures, due to the competitiveness and optimization that the market is carrying out. In the present work, an analytical and experimental approach has been developed about the aeroelastic phenomena found in these solar trackers with one degree of freedom. On the one hand, the analytical study has identified the dimensionless parameters governing the differential equation of movement. On the other hand, systematic wind tunnel experiments have been carried out with a 3D full aeroelastic scale model. It has been found that the tests reproduce correctly the aeroelastic phenomena found on a real situation. The main result has been the critical galloping velocity for every tilt angle, and a calculation methodology for the solar tracker shaft.

[BB2] Flow-induced vibration of a circular cylinder transverse to oscillatory flow at a high keulegan-carpenter number

Dániel Dorogi[1], László Baranyi[1], Efstathios Konstantinidis[2]

1 : University of Miskolc Department of Fluid and Heat Engineering, 3515 Miskolc-Egyetemváros - Hungary
2 : University of Western Macedonia Department of Mechanical Engineering, Kozani 50132 - Greece

This work investigates the excitation of flow-induced vibrations of a circular cylinder in a uniform oscillatory flow at the Keulegan-Carpenter number of 500, i.e. in the drag-dominated regime. The cylinder was allowed to vibrate only in the direction transverse to the incident flow. The governing equations for fluid flow and cylinder motion were solved in a non-inertial frame of reference using an in-house finite-difference code. Simulations were conducted for a system with a mass ratio of 2 and zero structural damping. The maximum Reynolds number and the corresponding maximum reduced velocity were set to 150 and 5, respectively. The results show that the cylinder response comprises low-frequency wave-induced oscillations and superimposed high-frequency vortex-induced vibrations (VIV).The most important finding is that the VIV response displays strong hysteresis, as illustrated by plots of the phase-averaged vibration amplitude and magnitude of fluid forces as functions of the time-dependent reduced velocity, despite the fact that the frequency of the oscillatory flow is extremely small compared to the natural frequency of the cylinder. It is further shown that the phase-averaged vibration amplitude over the deceleration stage follows quite closely data points from steady-flow tests, but not so during the acceleration stage. However, the magnitude of the phase-averaged fluid forces that drive the motions as functions of the time-dependent reduced velocity in oscillatory flow are markedly different during most part of both acceleration and deceleration stages than those at the corresponding steady flows.

[BB3] Influence of a control wire on vortex shedding from side-by-side cylinders

Omar Hammad, Atef Mohany

Fluid-Structure Interaction and Noise Control Laboratory, Faculty of Engineering and Applied Science, Ontario Tech University Oshawa, ON - Canada

Vortex-shedding behind bluff bodies can induce noise and vibration problems when it engages in a feedback cycle of oscillations with an acoustic or structural mode. Therefore, several passive and active flow control techniques have been developed to suppress vortex shedding behind a single bluff body. However, less attention was given to the case of multiple bluff bodies in cross-flow. Therefore, in this paper, a numerical investigation is performed to investigate the effect of adding a small wire in the vicinity of two side-by-side cylinders on the vortex shedding characteristics and the hydrodynamic fluid forces. The simulations are conducted at a Reynold's number of Re = 200 and the side-by-side cylinders have a spacing ratio of $T_* = 2.5$. The control wire is placed at a center-to-center ratio of $R' = 1$ with different angular positions relative to the flow stagnation point. The base case of side-by-side cylinders without the control wire is characterized by a synchronized, out-ofphase flow regime. However, when the wire is placed around the cylinders, three flow regimes are observed over the range of wire angles; bistable flow, stable biased flow, and merged wake flow regimes. The stable biased flow regime has resulted in the maximum reduction of the fluctuating lift coefficient, especially at θ =

135°. Adding a second wire to the other cylinder has further reduced the fluctuating lift coefficient. A summary of the results is presented in the paper.

[BB4] Study of the oscillation process and the wake resulting from the accelerated flow over two free-to-rotate tandem cylinders and the effect of a perturbation applied on the flow

Patrick Habowski, Guilherme Fiorot, Roberta Neumeister, Sergio Möller

PROMEC/Universidade Federal do Rio Grande do SUl Rua Sarmento Leite, 425, Porto Alegre, RS CEP: 9050-170 - Brazil

This paper presents the experimental study of the oscillation process of two circular cylinders (D = 25 mm) with a fixed pitch-to-diameter ratio P/D=1.26 attached to a free-to-rotate circular table placed in a wind tunnel submitted to a turbulent crossflow. One of the cylinders is aligned with the circular table axis, while the second one is placed eccentrically. The experimental analysis employed a hot-wire anemometer placed in the cylinders' wake synchronized to a high-speed digital camera. Results of velocity time series and angular displacement of the cylinders were analyzed through Fourier and wavelet transforms. The time series of the cylinders' angular displacements were obtained from the digital movies. When submitted to the uniform flow, the cylinder set aligns with the flow in a tandem arrangement. After that, it starts oscillating with a low steady amplitude. The flow is then disturbed by inserting and removing a cylinder with 32 mm of diameter, parallel to the cylinders and tangent to the channel wall. The main mechanism for initiating the low amplitude oscillation is the shedding of vortices. Applying the disturbance permanently increases the amplitude of the angular oscillation, even after it is removed. This high amplitude oscillation was found to be caused by the flow pattern through the cylinders' gap. Considering the observations made in the present work, the oscillator configuration analyzed may work as a vortex shedding suppressor.

[BB5] Unsteady wall pressure measurements on a full scale flexible chimney subject to natural wind

Yacine Manal[1], Pascal Hémon[2]

1 : Ecole Normale Supérieure Paris-Saclay 4 Avenue des Sciences, 91190, Gif-sur-Yvette - France

2 : LadHyX (CNRS - Ecole Polytechnique) 91128 Palaiseau Cedex - France

A full scale flexible prototype chimney is erected on a natural site in order to study its excitation by wind, especially by the alternate vortex shedding.
For the first time, unsteady wall pressure measurements are performed on the chimney. This experiment combines the two features that cannot be reproduced simultaneously in wind tunnel: a high Reynolds number and a turbulent boundary layer.
Preliminary results are presented, showing notably that the pressure distribution looks like

the one measured in wind tunnel in supercritical conditions, with an additional turbulent noisy component.

[BB6] Vortex-induced vibration of a circular cylinder subjected to low-Keulegan-Carpenter-number oscillatory flow

Dániel Dorogi

University of Miskolc Department of Fluid and Heat Engineering, 3515 Miskolc-Egyetemváros - Hungary

In this study vortex-induced vibration of a circular cylinder subjected to a uniform oscillatory stream is investigated. The cylinder is allowed to vibrate only transverse to the main flow. The non-dimensional governing equations of the fluid and cylinder motion are solved in a non-inertial reference frame using an in-house CFD code based on the finite difference method. The Keulegan-Carpenter number and the Reynolds number are fixed at the values of 5 and 100, respectively. Systematic simulations are carried out in the reduced velocity range of 1-12 at fixed mass and structural damping ratio values of 2 and 0, respectively. Two separate lock-in (LI) domains are identified. In the primary LI range the peak oscillation amplitude exceeds one cylinder diameter, and the vibration frequency locks to the flow oscillation frequency. However, in the secondary LI domain the peak amplitude is less than the half of the cylinder diameter and the vibration frequency synchronises with twice the flow oscillation frequency. Using the extended form of the Morison's equation available in the literature it was found that the linear drag component dominates the quadratic drag. This effect is more pronounced close to peak amplitude points.

[BB7] Vortex-Induced Vibrations of a One-Degree-of-Freedom Cylinder Transitioning from the Inline to the Crossflow Direction

Bridget Benner, Yahya Modarres-Sadeghi

University of Massachusetts, Amherst 160 Governors Dr., Amherst MA 01003 - United States

Vortex-Induced Vibrations (VIV) of a cylinder in crossflow (i.e. the direction perpendicular to the incoming flow) is a fundamental problem in fluid-structure interactions (FSI) and several studies exist on different aspects of crossflow VIV. Some studies also exist on the case where the cylinder is allowed to oscillate in the direction of incoming flow (inline). Here, we study VIV of a cylinder with one degree of freedom, which is allowed to oscillate in directions ranging from purely inline to purely crossflow. Experiments are conducted in a recirculating water tunnel using a low mass-damping coefficient system. Force and displacement measurements together with flow visualization of the wake are used to characterize the response of the cylinder over a range of reduced velocities as the single degree of freedom incrementally deviates from the inline direction. It is shown that the

first of two non-zero-amplitude regions, that are observed at low reduced velocities in a purely inline VIV response of the cylinder, slowly disappears as the angle is increased to approximately 45° and the lock-in region, usually observed in a pure crossflow VIV, first appears at angles of approximately 25°. It is also shown that the observed vortex shedding modes change as the degree of freedom deviates from a pure inline direction.

[BB8] Wake induced vibration in tandem cylinders: Part 1- wake perturbation analysis

Roberta Neumeister, Ana Ost, Patrick Habowski, Alexandre De Paula, Adriane Petry, Sergio Möller

PROMEC-Universidade Federal do Rio Grande do Sul Rua Sarmento Leite, 425, Porto Alegre, RS, CEP 90050-170 - Brazil

Flow-induced vibration of a cylinder in the wake of another cylinder in a tandem arrangement in an aerodynamic channel is experimentally investigated in the present study. The first cylinder is rigidly mounted inside the aerodynamic channel, while the second one is free to vibrate transversally to the main flow. Normalized distance L/D between the cylinders ranged from 2.5 to 10. The cylinder free to vibrate presented a mass ratio of 539 and a damping ratio of 0.0169. Hotwires, microphones and accelerometers are the experimental tools applied in this study. The Reynolds number computed with the tube diameter and the main flow velocity ranged around 1.15×10^4. Fourier and Wavelet Transform supported by flow visualizations in a hydraulic channel was applied to analyze the acceleration results as well as velocity and pressure fluctuations. Results show the influence of the wake from the first cylinder on the lift force and vibration regime of the second cylinder. The wake from the first cylinder perturbs the second cylinder even for large pitch ratios and changes the response of force and vibration on the second cylinder. The acceleration in the transversal direction of the cylinder free to vibrate increases under the influence of the wake of the first cylinder for all spacing ratios, mainly values of L/D higher than 5. The perturbation generated by the first cylinder is observed in the acceleration results from the second cylinder.

[BB9] Wake induced vibration in tandem cylinders: Part 2 – Hilbert-Huang spectral analysis

Ana Ost, Roberta Neumeister, Adriane Petry, Sergio Möller

PROMEC-Universidade Federal do Rio Grande do Sul Rua Sarmento Leite, 425, Porto Alegre, RS, CEP 90050-170 - Brazil

Flow induced vibration of a cylinder in the wake of another cylinder in tandem arrangement in an aerodynamic channel is experimentally investigated in the present study. The first cylinder is rigidly mounted inside the channel, while the second one is free to vibrate transversally to the main flow. The cylinder free to vibrate had a mass ratio of 539 and damping ratio of 0.0169. Hot wires, microphones and accelerometers are the

experimental tools applied in this study. The Reynolds number computed with the tube diameter and the main flow velocity ranged between 1.13×10^4 and 2.29×10^4. Hilbert-Huang Transform (HHT) is applied to analyze the acceleration results as well as velocity and pressure fluctuations. According to the literature, the wake induced vibration (WIV) mechanism for tandem is sustained by unsteady vortex–structure interactions that input energy into the system as the second cylinder oscillates across the wake. As the second cylinder is moved farther downstream, vortices coming from the upstream wake have more time to diffuse and the resulting vortex–structure interaction is weakened. This paper presents the experimental study of the flow over two cylinders in tandem with L/D = 2.5 - 10, with the second cylinder free to vibrate. HHT is a tool of analysis for data emerged from non-linear and non-stationary systems and it is a combination of the Empirical Mode Decomposition (EMD) and the Hilbert Spectral Analysis (HSA). To successfully deal with the scale separation and the mode mixing, the Ensemble EMD (EEMD), a noise assisted method, which is based on the statistical properties of white noise was used. The Hilbert Spectral analysis provides an accurate representation of the amplitude-frequency-time distribution of the flow. HHT is used to evaluate the velocity, pressure fluctuations and acceleration results for two tandem cylinders, in an attempt to identify the relationship between the first cylinder perturbation and the second cylinder response.

Bio-mechanical FSI

[BIO1] Mitigating jet cross-flow induced vibrations using a bio-inspired nozzle

Ibrahim Gadelhak[1], Njuki Mureithi[1], Kostas Karazis[2]

1 : Polytechnique Montréal 2500 Chem. de Polytechnique, Montréal, QC H3T 1J4 - Canada

2 : Framatome Inc. 3315 Old Forest Road, Lynchburg, VA 24506 - United States

Sharks discharge a large amount of water flow after the oxygen has been extracted during ram ventilation. The efficient mixing of jet flow exiting the gill openings allows sharks to still have smooth mobility. In many industrial devices, a rapid mixing of jet flows is the important feature to ensure efficient operation. Furthermore, jet flow is critical in the design and operation of specific nuclear pressurized water reactors (PWRs). Jet flows are introduced into these reactors through multiple holes drilled in the baffle plate that surrounds fuel assemblies. The objective of these holes is to release the pressure build-up during a potential loss-of-coolant accident (LOCA). However, during normal operation, the jet flow could induce unwanted fuel rod vibration.

Preventing fuel rods vibration can be achieved by improving the mixing process between jet flow and the surrounding fluid. A biomimetic "shark nozzle" is proposed here to improve mixing between a jet flow and surrounding fluid. Thus, the jet momentum reaching the rods is reduced. The experimental results show that the new shark-inspired biomimetic nozzle has a greater stabilizing effect on rod bundle. Utilizing the biomimetic nozzle delays the critical velocity by 28%. In addition to postponing instability, utilizing the proposed shark-inspired nozzle instead of the circular nozzle resulted in an 88% reduction in post-instability vibration amplitudes.

[BIO2] Self-Oscillating Hydrogel-Based Vocal Fold Models for Voice Production Research

Scott Thomson, Taylor Greenwood

Brigham Young University Provo, Utah - United States

The fabrication and flow-induced vibratory response of a hydrogel-based, multi-layer, self-oscillating vocal fold model is reported. Two hydrogels of differing stiffness, prepared using readily-available materials, were sequentially cast to create two-layer vocal fold models. A thin layer of silicone was applied to the surfaces to represent the epithelium. The models self-oscillated in a manner similar to that of previously-reported silicone models. Evidence of mucosal wave-like motion, including the alternating converging-diverging profile characteristic of human phonation, was seen. The results show the potential for hydrogel and hydrogel-silicone models to be used for voice production research, thereby enabling respective characteristics of hydrogels and silicones to be leveraged for different desired purposes.

Control of FIV and noise

[CF1] Passive control of the turbulent flow past a finite circular cylinder fitted with eight peripheral rods

Icaro Carvalho , Gustavo Assi

University of São Paulo Escola Politécnica - Brazil

Passive technologies devoted to the control of the flow past bluff bodies have been studied widely. However, most of these studies have focused on large aspect ratio, or infinitely long cylinders. We have carried out detached eddy simulations at Reynolds number 10E3 for both infinitely long and low aspect ratio (L = 2D) configurations. These computations were conducted for the bare cylinder (of diameter D and length L, as a reference case) and for the system comprised by the same main body, but now fitted with eight wake-control rods (of diameter d = D/20) uniformly distributed around the main, but distant from it by a gap G = D/100. Our results showed that the low aspect ratio case relative to that of infinitely-long structures i) lowered mean drag and root mean square lift; ii) presented a more organised topology, although did not attain vortex shedding

suppression; iii) the two frontal and the two rear rods concentrated most of the hydrodynamic loads, whereas intermediate rods faced less than 1/10 of these force coefficients – both cases generally following the same tendency; and iv) although both cases developed larger hydrodynamic loads when the main body was fitted with rods, the finite case produced a lower increase.

[CF2] Experimental investigation on the optimal control of vortex shedding of a circular cylinder with rotating rods at moderate Reynolds numbers

Paulo Silva, Gustavo Assi

University of São Paulo Escola Politécnica - Brazil

This paper presents an experimental study of fluid-structure interaction performed with scale models in a water channel at moderate Reynolds numbers. A circular cylinder model was equipped with 8 control rods positioned around its perimeter to interact with the external flow. Each rod could be independently driven to rotate around its own axis. The controlled rotation of the rods interfered with the vortex generation mechanism, mitigating the formation of a coherent wake. The results showed that there is an optimal rotation setting for the 8 rods that drastically reduces vortex formation and minimizes drag and energy consumption. A maximum drag reduction of 22% was obtained in relation to that of a bare cylinder, but at a cost of an increase of 10% in fluctuating lift. This work is a scientific study that clarifies the physical principles involved in the phenomenon, paving the way for the development of technological and innovative solutions.

CFD Techniques

[CFD1] A discrete forcing method to solve hyperelastic deformation induced by two-phase flow

Nicolas Merigoux[1], William Benguigui[1, 2], Federico Baraglia[1]

1 : EDF R&D - Fluid Mechanics Energy and Environment Dept EDF Recherche et Développement
06 quai Watier, 78401 Chatou Cedex, France - France
2 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA EDF Recherche et Développement 7 boulevard Gaspard Monge, 91120 Palaiseau - France

Fluid-structure interaction is an occuring phenomenon in engineering. In power plants for instance, fuel rods, tubes in steam generator or cooling towers may deform under the extreme load caused by the fluid flowing around them. It may also be valuable to study this kind of interaction during pipelines installation or for wind generating structures. If the deformation is sufficiently important to significantly influence the fluid behaviour, the

problem must be tackled as a whole.
Therefore, fluids and structures cannot be processed separatly. In this work, a fluid-structure interaction method taking into account the large deformation of a simple geometry is developped in Neptune_cfd, a 3D multiphase flow code based on Eulerian-Eulerian approach with a single pressure. The solid structures are geometrically non-linear elastic beams in plane strain. The coupling interface is tracked with a discrete forcing method based on a time and space dependent porosity method. The latter is updated several times for each time steps in order to ensure convergence. The presen formulation is validated for a dam break over a flexible plate with experimental confrontation and comparison. This method has then been used for an OECD benchmark on fluid-structure interaction.

[CFD2] A practical approach to using CFD as an early design tool for estimating aerodynamic force coefficients of bridge decks

Marko Duranovic[1, 2], Tony Dempseya[1], Craig Meskell[2]

1 : Roughan and O'Donovan Dublin - Ireland
2 : Trinity College Dublin Dublin - Ireland

CFD is a relatively new tool in bridge engineering and it is not often used at design stage due to its complexity and uncertainty when used without wind tunnel tests. However, conducting a preliminary CFD analysis prior to committing to a wind tunnel testing campaign can shorten the time spent in the tunnel. The main goal of this paper is to evaluate different methodologies used in CFD bridge aerodynamics and provide advice on future modelling endeavours for estimation of static load coefficients by means of CFD.

The results from a wind tunnel test conducted on a 1:50 scale model of the Northern Spire Bridge in the UK were used to evaluate different modelling choices using ANSYS Fluent. Different bridge barrier configurations are represented with constant solidity ratio. These results suggest that the barrier configuration has a low impact on the drag coefficient but quite a significant effect on lift and moment coefficients. A brief analysis on the effects of different turbulence models led to Transition SST being the turbulence model of choice. A study on the levels of turbulence intensity is outlined showing it has a significant impact on the static lift coefficient values for a range of angles of attack.

It is shown that CFD can yield useful results which can shorten the design cycle, but as discussed, care is needed to achieve practically useful data.

[CFD3] CFD analysis of two-phase flow induced forces on a test flow loop

Paul Emmerson[1], Mike Lewis[1], Neil Barton[2]

1 : Xodus Group Limited Cheapside House, 138 Cheapside, London. EC2V 6BJ - United Kingdom

2 : Xodus Group Limited 180 St Vincent St, Glasgow G2 5SG - United Kingdom

Multiphase flow through pipelines can result in severe piping vibrations and subsequent fatigue damage. Commonly, finite element analyses are used to define the dynamic pipe stresses and to calculate the lifetime of subsea piping. The fluid flow excitation forces can be estimated from measured data, estimated with empirical models or simulated in detail using computational fluid dynamics (CFD). Measurements resulting from multiphase flow in a test loop have been compared to CFD predicted forces.

CFD analysis is shown to be a valuable method for predicting multiphase flow induced forces. This is especially true in subsea piping systems at typical oil and gas production conditions. CFD analysis allows the prediction of the increase of excitation along the pipe due to the pressure drop and the effects of increased operating pressure. Also, when the piping system has phasing between the bends, transient CFD analysis may need to be carried out.

[CFD4] Dynamic response of a cantilevered pipe aspirating fluid and subjected to reverse confined external flow: a computational coupled two-way fluid-structure interaction analysis

Farhang Daneshmand[1], Tahereh Liaghat[2], Michael P. Paidoussis[2]

1 : King's College Department of Engineering, 133 N River St., Wilkes-Barre, PA 18702, USA

2 : McGill University Department of Mechanical Engineering, Montreal, QC, Canada H3A 0C3 - Canada

A computational model is developed for the dynamics of a hanging tubular cantilevered pipe aspirating fluid and simultaneously subjected to an external axial flow in the opposite direction. The external flow is confined over part of the length of the pipe. To gain a better insight into the dynamics of the system, a coupled fluid–structure interaction analysis technique is used, including a finite volume based computational fluid dynamics (CFD) code for the fluid domain and a finite element based computational solid mechanics code for the structural domain. In the numerical simulations, it is seen that limit-cycle motion occurs as the flow velocity exceeds a critical value. The results of the simulations are also compared with those obtained from experimental tests.

[CFD5] FSI simulations of fluid-elastic instabilities of a clamped-clamped cylinder in axial flow

Lucas Delcour[1], Henri Dolfen[1], Lieva Van Langenhove[2], Joris Degroote[1, 3]

1 : Department of Electromechanical, Systems and Metal Engineering, Ghent University Sint-Pietersnieuwstraat 41-B4, 9000 Ghent, Belgium - Belgium
2 : Department of Materials, Textiles and Chemical Engineering, Ghent University Technologiepark Zwijnaarde 125, 9052 Zwijnaarde - Belgium
3 : Flanders Make Belgium - Belgium

A cylinder clamped at both ends and subjected to an axial flow will typically become unstable with increasing axial flow velocity. At low velocity the cylinder will buckle and, as the flow velocity is further increased, flutter will occur. These instabilities can be devastating in some applications such as nuclear reactor cores or can be put to good use in energy harvesting tools. A substantial amount of analytical and experimental work has already been performed on the stability of cylinders in axial flow. However, numerical simulations on the topic are still rather scarce. Mainly due to the associated computational cost and difficulties in dealing with the mesh deformation, especially for cases with very large displacements (e.g. cylinders with a free end) in confined geometries. In this research fluid-structure interaction (FSI) simulations are performed on a clampedclamped cylinder subjected to axial flow for a range of inlet velocities. In the flow solver a Chimera technique is used which allows for large structural deformations without degrading the mesh. The results are compared to experimental data and previous FSI simulations. The goal is to validate the methodology for future use in simulations with cantilevered cylinders and later on investigate the influence of compressiblity on the stability of a flexible cylinder in high-speed air flow.

[CFD6] Modelling vortex induced vibrations in a model of the northern spire bridge

Marko Duranovic[1, 2],Tony Dempsey[2], Craig Meskell[1]

1 : Trinity College Dublin Dublin - Ireland
2 : Roughan and O'Donovan Dublin - Ireland

This paper presents a Fluid-Structure Interaction model used to capture lock-in of vortex shedding phenomenon on a realistic bridge deck model. The results are compared to the wind tunnel results obtained on a 1:50 model of The Northern Spire Bridge in the UK. A set of 2D CFD models was executed in Ansys Fluent allowing free vibration of the rigid bridge deck in the heave direction. Two different turbulence models were used and compared (SST and Transition SST). The computational effort required to execute the CFD models was kept to a level such that the process could realistically be adopted in the bridge design industry. As expected, a clear shift in vortex shedding frequency is exhibited at the lock-in onset confirming the phenomenon. The predicted velocity range of lock-in and velocity for peak response were in a good agreement with the experimental results, although the models significantly overestimate the peak response amplitudes. The paper shows that Transition SST turbulence model provides better quality results when compared to the SST model. Some limited evidence of the effect of Reynolds number on

VIV at this scale was obtained, but it is not conclusive. It is concluded that even a relatively simple CFD model can yield useful information at an early design stage.

[CFD7] Multi-scale methodology for the large eddy simulation of steam control valves

Jeremy Galpin[1], Basile Amice[1], Nicolas Goreaud [1], Guillaume Leconte [2], Aurélien Joly[3], Pierre Moussou[3], Asher Glaun[4]

1 : Framatome 2 rue Professeur Jean Bernard, 69007 Lyon - France
2 : EDF DIPNN/CNEPE EDF 8 rue de Boutteville, 37200 Tours - France
3 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA EDF Recherche et Développement 7 boulevard Gaspard Monge, 91120 Palaiseau - France
4 : Baker Hughes Valves Avon, MA - United States

Steam control valves play a major role in nuclear power plants, but their internal flows featured by large velocities might lead to the occurrence of noise and vibration. The focus is here on the assessment of the steam flow excitation by using Large Eddy Simulation with STAR-CCM+ CFD code. The simulation of the flow inside the control valve faces the challenge of dealing with a large range of geometrical scales from the scale of a hole in the order of magnitude of 1 mm up to the size of the valve body in the order of magnitude of 1 m. This difficulty is tackled by modelling the drilled part with thousands of small holes by using a porous-like approach. On the one hand, the proposed porous model allows to reproduce the flow nozzling occurring at the hole entrances as well as its interaction with the upstream acoustics. On the other hand, large turbulence produced downstream the holes is also using a body-force synthetic turbulence method.

The developed porous-like model has been deployed for performing a Large Eddy Simulation of half an industrial valve. The feasibility of addressing such kind of flow in such complex geometries is thus demonstrated and the associated computational resources and runback time remain compatible with industrial constraints. LES results show that large flow instabilities develop in the examined valve geometry, and the mechanisms leading to flow fluctuations have been identified. Such findings are valuable for mitigating the steam flow excitation by defining more virtuous designs. CFD methods like the ones described in this paper could be implemented on a systematic basis to orient design decisions in the development phase of large steam control valves prior to test rig experiments.

Dynamics of submerged structures

[DSS1] Development of an efficient calculation technique for dynamics of mooring lines by using discrete forms of rotation

Kensuke Hara[1], Kzuaki Shimojima[2], Taihei Yamaguchi[3]

1 : Yokohama National University Yokohama 240-8501, Japan - Japan
2 : Tokyo Institute of Technology 2-12-1 Ookayama, Meguro-ku, Tokyo, 152-8550 - Japan
3 : Ibaraki University Nakanarusawa 4-12-1 Hitachi Ibaraki 316-8511 Japan - Japan

The development of efficient calculation techniques is important for design of deepwater moored floating structures. The deepwater floating structure is comprised of many subsystems, such as a floating vessel, risers and mooring lines. In particular, this study deals with the development of a method for simulating dynamic behavior of the mooring line, which includes a lot of degrees of freedom and geometric nonlinear effects.

In the present model, a modified Morison equation is used for evaluating hydrodynamic forces acting on the mooring lines, that is, the interaction between lines and fluid is evaluated by using the linear potential flow theory. On the other hand, the mooring lines (chain) is modeled as multi-rigid bodies based on the multibody system approach.

In the formulation of the three dimensional motion of rigid body, the treatment of rotation is of key importance for deriving the equations of motion, because the parameterization of rotation complicates the mathematical description of the equation. This study employs the discrete forms of the rotation kinematic compatibility equation for resolving the difficulties arising from the the parameterization of rotation. More specifically, the present method introduces an approximated form of rotation expressed by the discrete form with respect to time with the incremental angular velocity. As a result, it can bypass the parameterization of rotation. It gives discrete form of the equations of motion for the rotation with only incremental angular velocity. Therefore, it can reduce the degrees of freedom for the rotation. Clearly, it can contribute to improvement of computational efficiency.

Furthermore, augmented formulation with constraints for configuration level are introduced for connecting the bodies. Since the configuration constraint can be expressed by information of positions for bodies, it enables us to derive the equations of motion for a whole system easily. On the other hand, other class of constraints (velocity or acceleration) may lead to constraint violation. However, the drawback of the model using the configuration constraints (so-called differential algebraic equations with index-3) involves severe numerical problem arising form the Jacobian matrix depending on the time-step size in the Newton-Raphson iterative processes for the numerical integration. In order to avoid this problem, the equations of motion are scaled based on physical argument derived by the mass and stiffness matrices.

Finally, The present method is evaluated by comparing with a conventional model.

[DSS2] Performance of a Closed Cycle Power Take Off for Mutriku breakwater

Morgane Bellec, Ciara Gurhy, Lee Gibson, Craig Meskell

School of Engineering, Trinity College Dublin College Green, Dublin 2 - Ireland

An oscillating water column (OWC) type wave energy converter was installed in Mutriku breakwater (Basque Country, Spain). The wave power is transferred to an air flow that turns a turbine. The facility currently uses self-rectifying turbines. A closed cycle power take off (CCPTO) arrangement, by forcing the flow to be unidirectional, would allow the use of more efficient turbines than currently employed. In this paper, a reduced order model for such a system is introduced. Using Mutriku breakwater dimensions the expected power performance is assessed. The sensitivity of the system to basic geometric features such as turbine and valves sizes is explored using monochromatic ideal wave. It is found that while there is an opimal size turbine, there is no benefit to increasing the valve size beyond a certain area. More realistic sea states representative of Mutriku are investigated using polychromatic waves as an input. The effect of tide is also considered by comparing extreme high and low tide conditions with the mean tide. It is found that tide height has a significant impact on the power production capacity. The trends uncovered by this work will form a useful basis for the design of the actual power take off including the unidirectional turbine.

Flow sound interaction

[FSI1] A Perforated Plate Solution to Mitigate Relief Valve Piping Vibration due to Flow-Excited Acoustic Resonance

Juan Pontaza, Raghu Menon

Shell International Exploration and Production Inc. Houston, Texas, USA

Piping systems with multiple closed side-branches are particularly liable to pressure pulsation due to acoustic resonances that are excited by flow instabilities in the piping system itself. The pressure pulsation can cause piping vibration problems which may in turn pose a threat to the structural integrity of the piping in terms of fatigue damage. This paper focuses on piping vibration due to flow excitation of acoustic resonant modes of multiple closed side-branches and its mitigation in an industrial setting.

Piping vibrations had been present in the relief valve (RV) take-off piping of the mixed refrigerant (MR) axial compressor discharge line in a liquefied natural gas (LNG) plant. The vibrations were present in three LNG trains with identical piping layout. Re-supporting and bracing of the piping did not mitigate vibration levels, which were observed to grow as the MR flow rate in the main line increased.

Fluid flow and acoustic numerical analyses were carried out to determine the root-cause of the piping vibrations and confirmed to be flow-excited acoustic resonance of the RV piping, revealing the acoustic coupling of multiple closed side-branches in series. A special design perforated plate solution was developed to mitigate the piping vibrations with no piping layout modifications. A digital evaluation, by means of computational fluid dynamics (CFD), was performed to verify that the proposed solution would mitigate the piping vibrations and to quantify the expected level of vibration mitigation.

Installation of the developed solution took place first in one LNG train and verified by measured vibration levels to work as intended, i.e., it mitigated the RV piping vibrations. The vibration mitigation solution was replicated in the other two LNG trains, with successful outcomes.

The solution involved the installation of a special design perforated plate above the tee-fitting throat at each of the RV take-offs to weaken the excitation velocity into the RV piping and attenuate acoustic pressure waves propagating into and out of the RV piping. The design is such that relief capacity is not compromised. A patent was granted for this 'industry-first' solution.

[FSI2] Articulated beam behaviour under grazing flow

Thibault Abily, Thomas Humbert, Yves Aurégan

Laboratoire d'Acoustique de l'Université du Mans (LAUM), UMR 6613, Institut d'Acoustique - Graduate School (IA-GS) - CNRS Avenue Olivier Messiaen, 72000 Le Mans - France

In recent years, new concepts of efficient low-frequency acoustic absorbers have been developed. In particular, treatments using moving parts are being studied.
Among several types of structures, those with fine slits are investigated in this paper. At the top of a cavity, a flexible beam is surrounded by a fine slit with a high resistance to air flow. This system produces a strong acoustic effect in the vicinity of the resonance frequency of the beam, which can be adjusted to very low frequencies.
In this paper, the system is mechanically and acoustically characterised under normal incidence in order to be compared with an analytical model. An array of these beams is mounted in the wall of a rectangular duct. The effect of grazing flow on the acoustic behaviour is studied as a function of Mach number.

[FSI3] Developing numerical methods for predicting flow-induced underwater radiated noise from ships

Duncan Mcintyre, Peter Oshkai

University of Victoria (UVIC) 3800 Finnerty Road, Victoria, BC, V8P 5C2 - Canada

Shipping is the largest anthropogenic source of Underwater Radiated Noise (URN), which represents a significant threat to the health of marine ecosystems. Propellers generate much of the URN from ships through a combination of wake vorticity and cavitation, two phenomena that interact extensively. Predictive models for this noise source are needed to develop intelligent mitigation strategies; however, the complexity of the physical phenomena and difficulty in obtaining measurements suitable for validation has so far limited model development. We present progress toward a generalized numerical framework for predicting URN from marine propellers. We detail two approaches, the fist using Unsteady Reynolds Averaged Navier-Stokes (URANS) solutions for a cavitating propeller and the second an ongoing study employing Delayed Dethatched-Eddy Simulation (DDES) to study non-cavitating flow around a ship hull and propeller combination. We highlight shortcomings of the RANS approach and detail the shortcomings are addressed by moving to a more rigorous approach employing DDES as well as those that must be addressed in continuing and future work.

[FSI4] Direct measurements of the dynamic lift force acting on rectangular rods in cross-flow during acoustic resonance excitation

Ahmed Shoukry, Atef Mohany

Fluid-Structure Interaction and Noise Control Laboratory, Faculty of Engineering and Applied Science, Ontario Tech University Oshawa, ON - Canada

The effect of self-excited acoustic resonance on the dynamic lift force acting on rectangular rods with three different aspect ratios is investigated experimentally. The rod has three different aspect ratios of l/h = 0.5, l/h = 1, and l/h = 2, where l is the rod length in the flow direction and h is the rod height perpendicular to the flow. The acoustic resonance was found to drastically affect the dynamic lift force coefficient of the rod. For l/h = 1 and l/h = 0.5, only the first acoustic cross-mode of the duct was excited. The excitation of this acoustic mode is triggered by the vortex shedding process and a jump in the dynamic lift force occurs. However, the jump in the dynamic lift force coefficient for the aspect ratio of l/h = 1 was significantly lower than that for l/h = 0.5. This is because the generated acoustic pressure for the case of l/h = 1 during resonance is weaker than that of l/h = 0.5. For l/h = 2, an early excitation of the third acoustic mode occurs with significant reduction in the dynamic lift force coefficient compared with that before the onset of resonance. This is because the early excitation

mechanism of the third mode is found to be dependent on the shear layer separation at the upstream corners of the rod rather than the vortex shedding process downstream of the rod. A summary of the results is presented in the paper.

[FSI5] Flow structure, dynamic lift force, and aeroacoustic response of finned cylinders in cross-flow

Mohammed Alziadeh, Atef Mohany

Fluid-Structure Interaction and Noise Control Laboratory, Faculty of Engineering and Applied Science, Ontario Tech University Oshawa, ON - Canada

The flow structure, dynamic lift force, and aeroacoustic response of single finned cylinders are experimentally investigated. Two different fin types are studied with varying fin pitch (p). The results of the finned cylinders are compared with bare circular cylinders that have the same equivalent diameter (Deq) as identified in the literature. It is revealed that the fins introduce fundamental changes to the flow structure around the finned cylinders that cannot be captured using bare cylinders with the equivalent diameter approach. During self-excitation of acoustic resonance, an abrupt increase in the dynamic lift force is observed, reaching a value of approximately 8 times its amplitude before the onset of resonance excitation. This is almost double what was reported for a single bare cylinder. Moreover, phase-locked PIV measurements revealed that the spike in the dynamic lift force and acoustic pressure for the case of the finned cylinders is due to a well organized vorticity field in comparison with that of the bare cylinder. A summary of the results is presented in this paper.

[FSI6] Flow-induced tones in a deep periodic cavity

Joachim Golliard[1, 2], Yves Aurégan[2]

1 : CTTM Centre de Transfert de Technologie du Mans 20 rue Thalès de Milet 72000 Le Mans - France
2 : Laboratoire d'acoustique de l'université du Maine CNRS : UMR6613, Université du Maine UFR Sciences Avenue Olivier Messiaen 72085 LE MANS CEDEX 9 - France

This paper reports a set of experiments with a chamber made of a succession of 10 deep cavities. This periodic geometry creates a bandgap of forbidden frequencies for which transmission of sound is not possible. This behaviour is first investigated with some measurements and simulations of the transmission coefficients. When a low-Mach number flow is added through the chamber, small changes on the transmission coefficients are observed. Particularly, the transmission coefficient can become larger than one at some frequencies, indicating a possible whistling of the cavities when installed in favourable conditions. The frequency range of amplification depends on the flow velocity following a nearly constant Strouhal number for the lowest velocities, but when reaching the range of

forbidden frequencies, the amplification stops. A second set of measurements is reported where the cavity, installed between two anechoic terminations, is submitted to a mean flow of increasing velocity. Whistling tones are observed, once again following the Strouhal number dependency until the bandgap, but a tone is also observed right in the middle of the bandgap. This unexpected behaviour is related to rotational modes in the cavities.

[FSI7] On broad-band noise of thick square-edged orifices in water-pipe flow

Shravan Kottapalli[1], Avraham Hirschberg[1], Nicholas Waterson[1], David Smeulders[1], Gunes Nakiboglu[1, 2]

1 : TU/e Wb, Postbus 513, 5600 MB Eindhoven - The Netherlands
2 : ASML De Run 6501, Veldhoven - The Netherlands

Broadband noise of water flow through orifices with a thickness equal to the orifice radius has been investigated experimentally. Incompressible large-eddy simulations (LES) of water flow through sharp 90° edged are used to predict the near-field hydrodynamic pressure fluctuations at low frequencies and the associated axial dipole sound source. Incorporating this lumped source model in an acoustic model provides a fair prediction of the acoustic field as detailed in an earlier publication. One observes a maximum of the Power Spectral Density (PSD) of the axial-dipole sound source for Strouhal numbers corresponding to varicose hydrodynamic instability modes as predicted for square-edged thick orifices in the literature. This sound source is larger than reported for thin orifices in the literature.

[FSI8] The aeroacoustic response of cylindrical cavities in confined flow

Marc Hanna, Atef Mohany

Fluid-Structure Interaction and Noise Control Laboratory, Faculty of Engineering and Applied Science, Ontario Tech University Oshawa, ON - Canada

Cavities of various aspect ratios exposed to grazing, low Mach number flow ($M < 0.4$) are responsible for flow-excited acoustic oscillations. Shear layer modes inherent to the flow oscillation across the opening of a cavity mouth may couple with acoustic modes of the given system and give rise to large acoustic pressure amplitudes when their frequencies coincide. This current investigation focuses on the aeroacoustics response of deep and shallow cylindrical cavities subject to a confined flow by flush mounting the cavity to the wall of a duct. It is shown that the response of a cavity in a confined duct pro-motes the acoustic resonance at multiple peak frequencies when interacting with the first three shear layer hydrodynamic modes. The identification of the acoustic modes for deep and shallow aspect ratios as well as the nature of the shear layer impingement are investigated by

virtue of acoustic pressure measurements at select locations in the system. The Strouhal numbers for aspect ratios h/D = 1 and 1.5 are re-ported and agree well with values documented in literature, whereas aspect ratios of h/D = 0.5 exhibit significantly lower Strouhal numbers.

[FSI9] Using flow to control the damping of a resonant duct

Thomas Humbert, Yves Aurégan

Laboratoire d'Acoustique de l'Université du Maine (LAUM) - CNRS : UMR6613, Université du Maine Avenue Olivier Messiaen 72085 Le Mans cedex 09 - France

This paper describes a resonant duct created by a corrugated tube submitted to a flow and placed between two chambers. The sound-flow interactions in the corrugated tube are used as an amplifier for frequencies determined by the flow velocity. This gain modulus is used to control the resonant duct damping. For certain modes, loss compensation is close to be achieved. However, when gain over compensates losses, whistling arrises.

Flow visualization

[FV1] Simultaneous control rod 3D displacement and 3D flow measurements via time resolved 3D3C PTV with one camera only

Vincent Fichet [1], Moaud Daoudi [2], Laurent Zimmer [3]

1 : Technical Center Framatome 71200 Le Creusot - France

2 : Physicochimie des Processus de Combustion et de l'Atmosphère - UMR 8522

CNRS, Université de Lille- 59655 Villeneuve d'Ascq Cedex - France

3 : Laboratoire d'Énergétique Moléculaire et Macroscopique, Combustion UPR288 CNRS, CentraleSupélec, Université Paris-Saclay, 3, rue Joliot Curie - 91192 Gif-sur-Yvette - France

In the framework of Flow-Induced Vibration (FIV), the measuring capability of a time-resolved 3D-3C Particle Tracking Velocimetry (PTV) with one camera only is addressed in this paper. The study deals with a representative turbulent flow (Re = 3.10^4) around guide plates and simulated control rods as parts of a nuclear CRGA (Control Rod Guide Assembly). Based on image defocus, this 3D3C PTV transforms the scattered light of laser-illuminated seeding particles into circular patterns on the defocused images. A patented annular pupil device is used in this scope. These particle patterns are detected and tracked over time by PTV in order to provide instantaneous 3D velocity fields. 3D3C

PTV velocities are validated by comparison with LDV data for both spatial (mean and standard deviation of velocity) and spectral (PSD) domains to confirm the accurate capture of the flow dynamics. The measures are found to be consistent in both shape and amplitude. The 3D3C PTV measurements are even more refined than LDV near the rods for the main axial flow. An easy, non-intrusive in-situ calibration based on a laser sheet sweep along the camera's in-depth direction completes the technique. The time required to post-process 20 000 images (time points) is less than 24 h on an HP Z820 workstation with 24 cores, and more than 1000 particles can be tracked per image over time to create trajectories.

Simultaneously, the 3D displacement of moving rods is recorded on these images (one camera only). Tracer particles are indeed fixed on the moving rods to track the rod FIV. Two moving rods are scrutinized in the upstream vicinity of a guide plate: one "free rod" is submitted to flexion with pinned-pinned boundary conditions, and another is held by strings in the highest cross-flow velocity region. With the same set of defocused images, the technique describes then both the excitation (flow) and the rod response (displacement) at the same time, leading to the whole Fluid-Structure Interaction (FSI) or the structure transfer function (response/excitation) with one camera only.

Fluid-structure interactions of animal locomotion

[FSI-AL1] Quantitative Flow Imaging Approach to Unsteady Loading on High-Inertia Oscillating Foils

Peter Oshkai, Waltfred Lee, Dylan Iverson, Mostafa Rahimpour

University of Victoria Department of Mechanical Engineering, University of Victoria, PO Box 1700 STN CSC, Victoria, BC, Canada V8W2Y2 - Canada

This work studies the effects of chordwise flexibility, inertia and kinematic parameters of an oscillating foil on the unsteady flow-induced loading on the foil. Three low-aspect-ratio foils with different flexibilities were undergoing pitch and heave motions in a uniform flow at the Reynolds number of 80000. Forces exerted on the foil were directly measured using a load cell and were used to calculate the propulsive thrust and efficiency values. The phase-averaged flow velocity and out-of-plane vorticity in the wake of the foil were obtained using particle image velocimetry (PIV). The circulation in the wake was related to the loading on the foil by calculating the moments of vorticity with respect to the pitching axis of the foil. The transverse deflection of the trailing edge, which defined the effective instantaneous angle of attack, determined the timing of the trailing edge vortex shedding and, in turn, the generated thrust and efficiency. The deformation of the foil resulted in an increased wake width, leading to larger amplitudes of the instantaneous loading on the foil and higher thrust coefficient compared to the reference case of a rigid foil.

[FSI-AL2] The Effects of a Passive Tail on Escape Performance in a Robotic Fast-Start Fish Capable of Rapid Underwater Locomotion

Todd Currier, Yahya Modarres-Sadeghi

University of Massachusetts, Amherst 160 Governors Dr., Amherst MA 01003 - United States

An experimental study is conducted on a robotic fish designed to emulate the fast-start response. The fish body is constructed of 3D materials and a light spring steel spine. The body is actuated using pressurized pistons. A total of two pistons are supplied with pressure through lightweight high-pressure service lines. The source of pressure is carbon dioxide with a 4.82 MPa peak operating pressure resulting in a body response that can cycle a C-start maneuver in milliseconds. The motion of the fish is controlled using large bandwidth solenoids with a control signal produced by a programmable microprocessor. The buckling modes of a slender column in compression are used to produce organic movements in the body with only two sources of actuation. The interaction of the fluid with the underactuated structure results in a travelling wave in the body of the robotic fish that is kinematically comparable to the live fish. The classical question of which tail stiffness is optimal in the fast-start is explored in a complete free floating model of the fish.

Multiphase Flows

[MF1] A new experimental facility for two phase flow characterization in a tube bundle and vibration study

Giuseppe Spina [1], Daniele Vivaldi [1], Guillaume Brillant [1], Catherine Colin [2], William Benguigui [3,4], Romain Denèfle [3], Muriel Lelong [3]

1 : Institut de Radioprotection et de Sûreté Nucléaire (IRSN) Cadarache, St Paul-lez-Durance 13115 - France

2 : Institut de mécanique des fluides de Toulouse

Centre National de la Recherche Scientifique : UMR5502, Université Toulouse III - Paul Sabatier,

Allée du Professeur Camille Soula 31400 Toulouse - France

3 IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA EDF Recherche et Développement 7 boulevard Gaspard Monge, 91120 Palaiseau - France

Vibration mechanisms caused by cross flow is a major issue of shell and tube heat exchangers. In particular, concerning the nuclear industry, in U-tube Steam Generators (SGs), two-phase cross flow occurs within the U-bend zone, which can be particularly

severe for vibration issues. Within this context, a new experimental apparatus has been designed and is in construction in Cadarache at IRSN. With the goals of characterizing the air-water two-phase flows across the tube bundle and of studying the vibration behavior of one tube. The facility contains a test section of 5x5 in-line square tube bundle. The pitch-to-diameter ratio is 1.44, the outer diameter (D) of the tubes is 0.03 meters and the tube length is 10D. The foregone experimental campaigns are going to investigate air superficial velocity range of 0.01 - 1.2 m/s and water superficial velocity range of 0.07 - 0.12 m/s. The present communication details the design of the facility and the preparation of the experimental work.

[MF2] Experimental investigation of void fraction distribution behind a cylinder

William Benguigui [1, 2], Christophe Pinto [1], Olivier Ries [1]

1 : EDF R&D - Fluid Mechanics Energy and Environment Dept EDF Recherche et Développement
06 quai Watier, 78401 Chatou Cedex - France
2 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA EDF Recherche et Développement 7 boulevard Gaspard Monge, 91120 Palaiseau - France

The numerical simulation of interaction between cylindrical structures and two-phase flows is a major concern for industrial applications, especially where cross-flows may cause damages, in heat ex-changers for example. In order to understand the phenomenon of vibrations induced by two-phase flows, many experiments were conducted with reduced-scale models using several simulant two-phase mixtures. In single-phase flow, the phenomenon has been characterized with dimensionless numbers. In 1986, Inoue et al. [1] pointed out different properties of a dispersed air/water flow around a single rigid cylinder for various inlet void fractions, velocities, and bubble to cylinder size ratios. In the present work, the authors are interest-ed in the behavior of two-phase flow from 30% to 90% of void fraction around a square cylinder.

[MF3] Forces and displacements in a bend subjected to an air-water flow

Laurent De Moerloose [1], Henri Dolfen [1], Michel De Paepe [1, 2], Joris Degroote [1, 2]

1 : Department of Electromechanical, Systems and Metal Engineering, Ghent University Sint-Pietersnieuwstraat 41-B4, 9000 Ghent - Belgium
2 : Flanders Make - Belgium

Understanding the vibration of tubes subjected to an internal liquid-gas mixture is of primary importance to improve the design of heat exchangers. Consequently, a large amount of experimental research has been performed in the past decades. Although an

extensive number of numerical investigations have been published more recently, most of these papers do not focus on the effect of conditions applied at the fluid domain boundaries. In the current paper it is shown that these boundary conditions can have an important effect on both the stability of the obtained flow solution and on the force exerted on the surrounding tube. The current work aims at correctly predicting the forces on a horizontal 90° bend that was investigated experimentally by Belfroid et al. Belfroid (2016). The forces exerted on the bend are directly related to the presence of liquid slugs coming into the bend. In the last part of this research, a finite element model for the structure of the pipe is presented. This model will be coupled to the flow solver in a partitioned fluid-structure interaction (FSI) simulation, but currently only some preliminary results are reported.

[MF4] High pressure multiphase induced vibrations : influence of pipe orientation

Stefan Belfroid [1], Nestor Gonzalez-Diez [1], Knud Lunde [2], Steinar Orre [2]

1 : TNO Delft - The Netherlands

2 : Equinor Stavanger - Norway

Three experimental campaigns were done at an industrial-scale test site to measure the influence of pressure on the forces induced by multiphase flow in process piping. For these different campaigns, measurements were done at 10, 25, 45 and 80 bar, involving different fluids (natural gas, MEG, oil, CO_2) and different orientation of the test section with regards to gravity (horizontal and vertical). During the experiments, the vibrations were measured at a large range of flow conditions and regimes.

In this paper, the results obtained for the latest campaign are described. The main goal of this campaign was to study the sensitivity of the measurements to the orientation of the test section with regards to gravity. It was observed that the vibration amplitude at 80 bar is much lower than at 10 bar at similar gas and liquid rates. This reduction can be attributed to a generally lower hold-up and lower unsteady hold-up variations at higher pressures, due to the higher gas shear forces and the resulting lower slip. At the test conditions, no effect of pipe orientation was observed for in-plane vibrations. The out-of-plane vibrations were much lower in vertical orientation compared to the horizontal orientation at lower pressures. At higher pressures, these differences disappeared, which is connected to a better mixing of the flow.

[MF5] Periodic Wake Shedding of Tube Bundles Subjected to Two-Phase Cross Flow

Colette Taylor [1], Michel Pettigrew [2]

1 : Canadian Nuclear Laboratories 1249 Victoria St., Petawawa, ON - Canada

2 : Canadian Nuclear Laboratories Chalk River, ON - Canada

Some periodic wake shedding has been observed in tube bundles subjected to two-phase cross flow, albeit, the consequences are not as severe as fluidelastic instability. The results of four different experimental programs are reviewed in this paper. Data on periodic forces and frequencies was deduced from the vibration response. Dynamic lift coefficients and Strouhal Numbers were obtained. This information was used to develop design guidelines to avoid vibration problems in process system components.

[MF6] Two-phase flow induced vibration in a tube bundle of steam generators

Vincent Fichet, Bachar Khaddaj Mallat, Alejandro Mourgues, Jérôme Moulin, Quentin Andrzejewski,

Département de mécanique des fluides, des structures et essais sur composants
Centre technique Framatome 30 boulevard de l'industrie 71200 Le Creusot - France

Tests were carried out at the Framatome Technical center in Le Creusot to study two-phases flows with two gas/liquid couples (Water/Air and an Im-proved Simulating Liquid - ISL/Air) in a tube bundle of Steam Generators (SG) at ambient temperature and pressure. High-speed video sequences were recorded to identify the different flow regimes and the differences in flow topologies related to the type of simulating liquid used. These differences were then quantified by means of fluctuating force measurements on the SG tube as well as by measurements of the temporal local void fractions in the vicinity of the tube by a dual optical probe sensor mounted on the tube. The results obtained underline the differences between simulating liquids and emphasize the im-portance of using representative fluids to better capture two-phase FIV in SG tube bundle in order to gain operational margins and to improve SG de-signs. A way to characterize two-phase flow regimes locally and its interaction with vibrating structure via response PSD (amplitude) and first mode “peak” analysis (added mass, fluid damping) is also pro-posed.

Smart materials in FSI/FIV

[SM1] Pressure Driven Soft Vortex Generator

Assadour Khanjian [1], Charbel Habchi [2], Serge Russeil [3], Daniel Bougeard [3], Thierry Lemenand [4]

1 : Lebanese International University LIU Campus Beirut - Lebanon PO Box: 146404 Mazraa - Lebanon

2 : Institut des Sciences de la mécanique et Applications industrielles EDF R&D

828 bd des maréchaux 91762 Palaiseau cedex France anciennement LAMSID UMR 8193 - France

3 : Ecole nationale supérieure Mines-Télécom Lille Douai Institut Mines-Télécom [Paris]

Cité Scientifique, Rue Guglielmo Marconi BP 20145 Villeneuve d'Ascq - France

4 : Laboratoire Angevin de Recherche en Ingénierie des Systèmes Université d'Angers : EA7315

Université d'Angers, LARIS l Polytech Angers, 62 Avenue Notre Dame du Lac - 49000 Angers - France

Vortex generators are used in heat exchangers and static mixers for the enhancement of heat and mass transfer processes. This enhancement is caused by the generation of streamwise vortices, and it is generally accompanied by an increase in the pumping power. Thus, active, and passive control techniques would be adopted to achieve the required rates of heat and mass transfer while maintaining low pressure losses. This could be done by adjusting the angle of attack of the vortex generators for instance.

In the present study, different types of soft vortex generators are proposed which can deform passively due to the flow forces. Hence, for low Reynolds number, i.e., low flow speed, the angle of attack is higher inducing more energetic vortices for higher transfer rates with higher pumping power. For higher Reynolds numbers, the vortex generator would deflect in the streamwise direction, lowering the vortices momentum and decreasing the pumping power.

For this end, experimental studies on soft vortex generators inside a wind tunnel are performed to quantify the angle of attack dependence on the flow speed and on the vortex generator material. In addition of concept validation, the results would be useful for validating numerical results.

[SM2] Wind Energy Harvesting from Flow-Induced Vibration of Prisms Using Magnetostrictive Material

Mohamed Heragy, Takahiro Kiwata, Takaaki Kono, Takuma Shima, Takahito Hamano, Toshiyuki Ueno, Alis Ekmekci

1 : Kanazawa University Kakuma-machi, Kanazawa 920-1192 - Japan

As an energy harvesting technology, power generation from ambient vibration has attracted interest because of the development of low power consumption sensors and wireless communication systems such as Internet of Things (IoT) devices. The flow-induced vibration by the wind combines with magnetostrictive material, i.e., iron-gallium alloy, to develop a vibration-power generator, which can generate electrical energy from a light wind. This new technology is expected to supply power to IoT devices as a substitute for batteries. The vibration-power generator is composed of a wind-receiving

unit and a power-generating unit. In the present research, we attempt to develop a magnetostrictive wind vibration-power generator by low-speed galloping and vortex-induced vibrations. The purpose of the present study is to investigate the most suitable cross section of a cantilevered prism for the transverse flow-induced vibration and the performance of a wind vibration-power generator using the magnetostrictive material through wind tunnel experiments.

The experiments were performed in a wind tunnel with a rectangular working section having a height of 1200 mm, a width of 300 mm, and a length of 2000 mm. We focus on transverse galloping for a rectangular prism with a critical depth section of less than $D/H \approx 0.6$ (where D: depth of the prism in the flow direction, H: height of the prism), a V-section prism, and a filleted triangle prism, and the vortex-induce vibration for a circular cylinder. The wind velocity U was varied from 1.0 to 6.2 m/s. The variations of the non-dimensional amplitude of displacement and the power of vibration-power generator with the reduced velocity Vr ($=U/fc \cdot H$; fc, Characteristic frequency of a prism; H, Height of a prism) was shown.

The maximum power generations for a circular cylinder, rectangular, filleted triangle and V-section prisms with a height of 50, 50, 60, and 50 mm are 1.28, 3.5, 8.24 and 5.02 mW, respectively. The power coefficient of a filleted triangle prism with 60 mm height is Cp = 0.22%, which is larger than the other test models. These values of peak power generation are enough to run a wireless sensor. The wind vibration-power generator using a circular cylinder is safe for the strong wind because it does not vibrate over Vr = 10.

Tube Arrays

[TA1] A New criterion for the instability threshold of a square tube bundle subject to an air-water cross-flow

Romain Lagrange[1], Domenico Panunzio[1], Philippe Piteau[1], Xavier Delaune[1], Jose Antunes[2]

1 : Commissariat à l'énergie atomique et aux énergies alternatives
CEA Centre de Saclay - France

2 : Centro de Ciências e Tecnologias Nucleares (C2TN), Instituto Superior Técnico (IST), Universidade de Lisboa 2695-066 Bobadela LRS - Portugal

In this paper, we investigate the fluid-elastic instability of a square tube bundle subject to a two-phase cross-flow. A dimensional analysis is carried out, leading to a new criterion of stability. This criterion establishes a direct link with the stability thresholds in single-phase flows and as such constitutes an extension to the Connors equation for two-phase flows. In parallel to the dimensional analysis, an experimental work is conducted to test the validity of the criterion in air-water cross-flow. The bundle consists on 5 rows of 3 tubes (plus two end-rows of half-tubes), with reduced pitch P/D = 1.5, tubes diameter and length D = 30 mm, L=300 mm. The central tube is mounted on two flexible blades

allowing a vibration in the transverse direction, whereas all the other tubes are rigid. Three sets of blades with different rigidities are tested to investigate the stability of the central tube, considering, for an homogeneous void fraction in the range 0% to 100%. We show that the criterion of stability derived from the dimensional analysis is in very good agreement with the air-water experiments. This new criterion is of theoretical interest for the understanding of complex two-phase flow excitations, as well as of practical significance for the predictive analysis of industrial components.

[TA2] Experimental investigation of cross-flow fluidelastic instability for rotated triangular tube bundles subjected to single-phase and two-phase transverse flows

Domenico Panunzio [1], Romain Lagrange [1], Philippe Piteau [1], Xavier Delaune [1], Jose Antunes[2]

1 : Commissariat à l'énergie atomique et aux énergies alternatives CEA
Centre de Saclay - France
2 : Centro de Ciencias e Tecnologias Nucleares (C2TN), Instituto Superior Tecnico (IST), Universidade de Lisboa 2695-066 Bobadela LRS - Portugal

An experimental program is conducted to study the vibration response of a rotated triangular tube bundle with a pitch ratio of 1.44 and an aspect ratio of 10. The tube bundle is subject to a single-phase (air or water) or a two-phase air-water cross-flow, with an homogeneous void fraction in the range [0.1,0.95]. The bundle is free to vibrate only in the cross-flow, i.e. lift direction. Four bundle configurations, with one or multiple flexible tubes, are tested by increasing the flow velocity up to the onset of a fluidelastic instability. From the experimental observations and a modal analysis we show that, when $\alpha > 0.3$, instability can only materialize for flexible cells with two or more tubes, i.e., two or more degrees of freedom. In such cases, instability arises for a higher-order mode. Finally, we show that the instability is characterized by a Connors's constant greater than the one generally observed in the literature.

[TA3] Experimental investigation of in-flow fluidelastic instability for rotated triangular tube bundles subjected to single-phase and two-phase transverse flows

Jose Antunes [1], Philippe Piteau [2], Xavier Delaune [2], Domenico Panunzio [2], Romain Lagrange [2]

1 : Centro de Ciências e Tecnologias Nucleares (C2TN), IST, Universidade de Lisboa,
2695-066 Bobadela LRS - Portugal
2 : Université Paris-Saclay, CEA, Service d'Études Mécaniques et Thermiques, F-91191, Gif-sur-Yvette - France

The in-flow fluidelastic instability of tube bundles prompted renewed interest since the recent unanticipated failure of the replacement steam generators at the San Onofre nuclear power station. A literature review on the topic discloses contrasting views, depending on the tube bundle and flow configuration addressed. In a recent paper, the authors reported experiments using square bundles, subjected to single and two-phase flows. No streamwise instability was observed, for the tested bundle configurations and the flow velocity ranges explored. In the present paper, experimental results obtained at CEA-Saclay for a rotated triangular tube bundle are presented, providing new in-flow fluidelastic data for both single-phase and two-phase transverse flows. The bundle consists of 50 tubes, with reduced pitch P/D=1.44 and tube diameter D=30mm. It was subjected to single-phase (air or water) and two-phase air-water (with homogeneous void fraction in the range 40% to 98%) transverse flows. In the upper region of the bundle, several different flexibility configurations were tested, with up to 15 flexible tubes, mounted using anisotropic supports, which allow for in-flow vibrations. Results presented in the paper include in-flow fluidelastic stability data obtained for both single-phase and two-phase transverse flows. Moreover, local void fraction and identified flow regimes are also presented. These results are discussed and compared to those obtained by previous authors, for similar tested configurations.

[TA4] On the stability of the rotated square array in two-phase flow using the quasi-steady model

Sameh Darwish[1] Njuki Mureithi[1], Minki Cho[2]

1 : Ecole Polytechnique de Montréal Campus de lÚniversité de Montréal 2500, chemin de Polytechnique Montréal (Québec) H3T 1J4 - Canada

2 : Doosan Heavy Industries and Construction 22, Doosan Volvo-ro, Seongsan-gu, Changwon-si, Gyeongnam, 51711 - South Korea

Despite the extensive data available in the literature for fluidelastic instability modelling, no data was published for the rotated square layout instability prediction in single and two-phase flow. The complex nature of the rotated square array creates a barrier to predicting fluidelastic behaviour. In the present work, the quasi-static fluid forces are measured in single-phase (water) and over a wide range of void fractions of two-phase flow. This array has a relatively wide spacing of P/D=1.64.
The test section is designed to measure the fluid forces for an instrumented tube, and cross-coupling forces of the surrounding tubes in order to incorporate the measurements into single and multiple degrees-of-freedom quasi-steady model. The present experimental data uncovers the stable behaviour of the single tube for this rotated square array due to the significantly small lift force variation with transverse displacement (CL_y). In the streamwise direction, no fluidelastic instability was found for a single-degree-of-freedom model.

[TA5] Theoretical and experimental study on the fluidelastic instability of rod bundle subjected to jet cross-flow

Ibrahim Gadelhak [1], Njuki Mureithi [1], Kostas Karazis [2]

1 : Polytechnique Montréal 2500 Chem. de Polytechnique, Montréal, QC H3T 1J4 - Canada

2 : Framatome Inc. 3315 Old Forest Road, Lynchburg, VA 24506 - United States

Jet cross-flow induced vibration has been recently found in some designs of PWRs that have fail-safe features in the event of a loss of coolant accident (LOCA). In these reactors, LOCA holes and slots are machined in the baffle plates to release the pressure build up during a LOCA event. During normal operation, however, the transverse flow from these holes and slots entering the peripheral fuel assembly can reach the fuel rods before being mixed with the axial flow. The localized jet cross-flow can cause fuel rod vibrations, which can contribute to grid-to-rod fretting (GTRF) and cladding wear.

This paper presents an experimental and theoretical framework to model the instability of a flexible rod bundle subjected to jet cross-flow. The work reported here is part of a test plan set up by Framatome and Polytechnique Montreal. A few basic models based on the Connors equation were developed in the literature for baffle jetting (i.e. 2D jet) problem in PWRs. A fluidelastic instability model is developed here based on quasi-steady approach developed by Price and Paidoussis in the early eighties. In their work, the authors introduced a time delay parameter to account for the phase lag between tube motion and flow-induced (fluidelastic) forces. Unlike the uniform flow case, a new formulation of fluidelastic forces as a function of the flow area derivative is proposed to account variation of the projected area through LOCA holes. Model results are compared and validated against the experimental results.

[TA6] Transient vibration phenomenon due to passing of gap vortex street in FSI simulation of tube bundle with eccentricity

Henri Dolfen [1], Joris Degroote[1,2]

1 : Department of Electromechanical, Systems and Metal Engineering, Ghent University Sint-Pietersnieuwstraat 41-B4, 9000 Ghent - Belgium

2 : Flanders Make Belgium - Belgium

Gap vortex streets in tube bundles subject to axial flow have been found for several geometries and fluids, both experimentally as numerically. An important parameter influencing its occurrence and strength is the size of the gaps between the tubes. It has also been shown that the associated pressure fluctuations are strong enough to give rise to flow-induced vibrations (FIV).

In this work the existence of a gap vortex street is demonstrated to trigger vibrations in a large pitch-over-diameter ratio (P/D) rod bundle with eccentric positioning of the central tube for a flow of liquid lead-bismuth-eutectic (LBE). This is done using fluid-structure interaction (FSI) simulations. In the first half of the simulation a gap vortex street arises, then triggers an asymmetric transient vibration while being advected downstream, after which the vibration decays. In the second half the flow pulsations weaken globally and are only present close to the pressure outlet. They trigger very small amplitude vibrations in a beating fashion.

The overall behavior described in this paper raises some new questions: firstly why no new vortex train arises at an upstream position nor any associated (large) vibration and secondly whether this is due to the influence of the boundary conditions and thus mainly a numerical artifact. Another question to be answered is what lies at the root of the asymmetry of the vibration.

Turbulence, vortex and wave-induced vibrations

[TV1] Aspects of vortex-induced in-line vibration at low Reynolds numbers

Efstathios Konstantinidis [1], Dániel Dorogi [2], László Baranyi [2]

1 : University of Western Macedonia Department of Mechanical Engineering, Kozani 50132 - Greece

2 : University of Miskolc Department of Fluid and Heat Engineering, 3515 Miskolc-Egyetemváros - Hungary

In this work, we investigate the vortex-induced vibration of a circular cylinder purely in-line with a free stream by means of two-dimensional simulations at low Reynolds numbers. The equations of fluid motion are solved using an in-house CFD code based on the finite-difference method in tandem with integration of the equation of cylinder motion. Results are reported for a mass ratio of 10 and null structural damping ratio at Reynolds numbers of 100 and 200. At these Reynolds numbers, the response amplitude as a function of the reduced velocity displays a single excitation region that is associated with alternating vortex shedding. The peak amplitude increases by a factor of six between Re=100 and 200 reaching a value of 1.4% of the cylinder diameter in the latter case. We employ the phasing of the unsteady lift force with respect to the cylinder displacement in order to decipher the vortex dynamics in the wake. Unlike the phase of the unsteady force acting along the direction of cylinder oscillation, which is restricted per the equation of cylinder motion at null damping to values of 0° and 180°, the phase of the unsteady lift force displays a gradual increase with reduced velocity. We interpret this variation as a shift in the timing of vortex shedding in the wake with the oscillation frequency. We propose an extension of the well-known Morison's equation that includes a separate term for the

vortex drag force, which captures the effect of vortex dynamics on the phasing of the driving force required to satisfy the equation of in-line motion. This new model uncouples the fluid and structural dynamics and provides physical insight not possible heretofore.

[TV2] Experimental investigation of vortex-induced vibrations of a circular cylinder under rotary oscillations

Agathe Schmider , Franck Kerherve, Laurent Cordier, Andreas Spohn

Département Fluides, Thermique et Combustion Institut PPRIME Téléport 2 11 Boulevard Marie et Pierre CurieBP 30179F86962 FUTUROSCOPE Chasseneuil Cedex - France

A new experimental device to study vortex-induced vibrations (VIV) of a cylinder in a free-surface water channel is presented. The cylinder is elastically-mounted on an air-bearing platform and is free to oscillate in the cross-flow direction. Two DC motors are coupled to the oscillating mass and allow : (i) to simulate for positive or negative damping through forcing, (ii) to measure the position and speed of the cylinder, (iii) to force the cylinder into a rotative motion. The dynamical response of the system has a typical two or three branch response amplitude depending on the damping ratio. One case of intensification of VIV oscillations with rotary oscillations is detailled. Flow velocity surveys obtained with Particle Image Velocimetry (PIV) are reported to compare the different VIV wake modes.

[TV3] Flow-induced vibrations of a flexibly mounted cylinder in the proximity of a stationary parallel cylinder

Mahdi Riazat, Mojtaba Kheiri, Brian C. Vermeire

Concordia University 1455 de Maisonneuve Blvd. W., Montreal, Quebec H3G 1M8, Canada - Canada

This paper presents the results from a computational study on the flow-induced vibrations (FIV) of a flexibly mounted cylinder (with diameter d) placed in the proximity of a stationary side-by-side (SBS) cylinder (with diameter D) at Reynolds number 100. The flexibly mounted cylinder has the combined mass-damping ratio 0.1 and can freely vibrate in the transverse direction with respect to the oncoming flow. Two gap ratios $g^*=g/d=0.75$ and 1.75 and five diameter ratios $d^*=D/d=0.5$, 1, 1.25, 1.5, and 2.5 are examined. The arbitrary Lagrangian-Eulerian form of the Navier-Stokes equations are solved using the Discontinuous Galerkin method for the fluid, and a mass-damper-spring system is used to model the structural dynamics.

The effects of d^* and g^* on the FIV characteristics of the flexibly mounted cylinder and fluid dynamic forces acting on the two cylinders are investigated. Numerical results showed that compared to an isolated cylinder, the lock-in region for the vibrating cylinder in the SBS arrangement begins at a lower reduced velocity. However, depending on the gap ratio, the lock-in region may become narrower or wider. Moreover, wake patterns

behind the two cylinders in the lock-in region are presented. Pairing up of the like-signed vortices and development of 'binary' vortices is a common feature of the near wake.

[TV4] Fluid structure interaction in a pressure vessel : a multipole approach for acoustic analysis

Pierre Moussou [1], Maud Kocher[1], Domenico Panunzio[2], Romain Lagrange[2], Aurélien Joly[1]

1 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA, EDF Recherche et Développement Palaiseau - France
2 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA, CEA/ DRT/LIST Palaiseau - France

Pressure vessels submitted to turbulent flows are prone to fluid-structure interactions and vibrations. In the case of heavy fluids like water, the inertia of the fluid and the flow coupling generate supplementary forces and energy exchanges. A consequence of these interactions is that, seen from the surrounding pipes, a pressure vessel behaves like the association of an unsteady pressure force and of an acoustic resonator, with a resonance frequency partly determined by the structural modes. The present paper aims at shedding some light on this effect, on the basis of a simplified mechanical model.

[TV5] Fluid structure interaction in a pressure vessel : turbulent forcing

Maud Kocher[1], Pierre Moussou[1], Domenico Panunzio[2], Romain Lagrange[2], Aurélien Joly[1]

1 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA, EDF Recherche et Développement Palaiseau - France
2 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA, CEA/ DRT/LIST Palaiseau - France

Pressure vessels submitted to turbulent flows are prone to fluid-structure interactions and vibrations. Scale testing is a useful tool to predict these effects and assess computational approaches. The present study focuses on the vibrational design of a small test rig intended as an experimental reference case. The test rig consists in a flexible structure submitted to an external axial flow, coming from two lateral perpendicular branch pipes. Introducing Au-Yang's general purpose pressure spectrum and correlation lengths, and focusing the attention on a balancing mode, the modal force PSD can be theoretically predicted. Combining this force with the mechanical parameters of the structure, the response of the inner cylinder can be determined in the frequency domain. Finally, the expression of the RMS vibration amplitude of the flexible structure is provided, and a dramatic influence of the product of the structure natural frequency with the turbulence time scale is observed.

[TV6] GO-VIKING : a Horizon Europe project on flow-induced vibrations

Kevin Zwijsen[1], Angel Papukchiev[2], Daniele Vivaldi[3], Hidajet Hadzic[4], Sofiane Benhamadouche[5], William Benguigui[5,6], Philippe Planquart[7]

1 : Nuclear Research and consultancy Group (NRG) Westerduinweg 3, 1755 LE, Petten - The Netherlands

2 : Gesellschaft für Anlagen und Reaktorsicherheit gGmbH (GRS) Boltzmannstr 14, 85748, Garching - Germany

3 : Institut de Radioprotection et de Sûreté Nucléaire (IRSN) Cadarache, St Paul-lez-Durance 13115, France

4 : Framatome GmbH Paul-Gossen-Str. 100, 91052, Erlangen - Germany

5 : EDF R&D - Fluid Mechanics Energy and Environment Dept
EDF Recherche et Développement 06 quai Watier, 78401 Chatou Cedex - France

6 : IMSIA UMR 9219 CNRS-EDF-CEA-ENSTA, EDF R&D 7 boulevard Gaspard Monge 91120 Palaiseau - France

7 : von Karman Institute for Fluid Dynamics (VKI) Waterloosesteenweg 72, B-1640 Sint-Genesius-Rode - Belgium

Flow-induced vibrations (FIV) appearing as a result of Fluid-Structure Interaction (FSI) remain an area of concern in nuclear power plants. If not properly addressed, FIV can have large consequences to the operation and safety of these plants. With the increase in computational power, the use of numerical simulation tools to predict vibrations induced by the surrounding flow is rapidly increasing.

In order to further advance the knowledge of solving FIV problems with the help of numerical tools, the Horizon Europe GO-VIKING (**G**athering Expertise **O**n **V**ibration **I**mpa**K**t **I**n **N**uclear Power **G**eneration) project was launched in mid-2022 in which 18 partners agreed to collaborate in this field for four years. The GO-VIKING project addresses issues with vibrations induced by the primary and secondary coolant in the nuclear power plants. Focus is put on two types of FIV issues, i.e. fuel rod fretting in fuel assemblies and fluid elastic instability occurring in steam generators, for both, single- and two-phase flows. The main objective of the GO-VIKING project is to develop, improve and validate FSI methods, and to provide guidelines for the prediction and assessment of FIV phenomena in nuclear reactors.

The current paper gives an overview of the GO-VIKING project, with a focus on its rationale and the experimental and numerical work to be performed during its four years.

[TV7] In wind tunnel, simulation of vortex shedding behind circular cylinders at high Reynolds number regimes is incomplete

Pascal Hémon[1], Oyvind Ellingsen[2], Xavier Amandolese[3]

1 : LadHyX, CNRS-Ecole polytechnique, IP Paris CNRS UMR 7646
Ecole Polytechnique, 91128 Palaiseau - France
2 : LadHyX & CSTB Centre Scientifique et Technique du Bâtiment (CSTB)
LadHyX, Ecole polytechnique, 91120 Palaiseau - France
3 : LMSSC, CNAM Conservatoire National des Arts et Métiers (CNAM)
CNAM, LMSSC, 2 rue Conté - 75141 Paris cedex 03 - France

Wind tunnel tests of 2D rough cylinders are presented. The goal is to simulate the alternate vortex shedding in flow regimes encountered in wind engineering applications, where the full scale Reynolds number is larger than the one that can be reproduced in wind tunnel with small scaled models.

Measurements are mainly the synchronized unsteady wall pressures on the cylinder which are post processed using bi-orthogonal decompositions.

By comparing the small scale results with those from a previous large scale experiment, we show that the technique of rough cylinder is incomplete and can approach roughly global parameters only.

[TV8] Numerical prediction of Axial-Flow-Induced Vibrations in nuclear fuel rod

Justyna Salachna , Andrea Cioncolini, Hector Iacovides

Thermo-Fluids Research Group, Department of Mechanical, Aerospace and Civil Engineering
The University of Manchester, Manchester, M13 9PL - United Kingdom

This study investigates the computational modelling of flow-induced vibrations of cantilever rods subjected to turbulent axial flow at operating conditions relevant to those of fuel rods of pressurized-water-cooled (PWR) nuclear reactors. The aim is to assemble all the modelling elements needed for a cost-effective and thus URANS-based modelling strategy, employing high-Reynolds-number turbulence models and using existing experimental data for validation.

The study involves:

1. Assessment and evaluation of URANS models (namely, the effective viscosity models, k-omega, k-epsilon, k-omega SST and Reynolds Stress Models LRR and SSG) to effectively reproduce fluctuating forces which can induce vibrations on a suitable representative 2D test case with rigid walls of Camussi et al. (2008)

2. Two-way FSI simulations validated against the experimental data of Cioncolini et al. (2018).

The resulting comparisons show that for the first time, to our knowledge, both the frequency and the amplitude of the flow-induced vibrations of this case, have been successfully reproduced.

[TV9] Numerical simulation of cantilever cylinders in cross-flow: participation to the OECD/NEA fluid-structure interaction benchmark

Kevin Zwijsen[1], Mohammed Hussain[2], Ferry Roelofs[1], Alexander Van Zuijlen[2]

1 : Nuclear Research and consultancy Group (NRG) Westerduinweg 3, 1755 LE, Petten - The Netherlands

2 : Delft University of Technology 2629 HS, Delft - The Netherlands

Flow-induced vibrations as a result of Fluid-Structure Interaction (FSI) remain an area of concern in nuclear power plants. If not properly addressed, they can have large consequences to the functioning and operability of these plants. With the increase in computational power, the use of numerical tools to predict vibrations induced by the surrounding flow is rapidly increasing.

The current paper presents some of the results obtained at Nuclear Research and consultancy Group during its participation to the OECD/NEA benchmark on FSI. This benchmark consists of an experiment comprising two in-line cantilever cylinders in cross flow and was started to further increase the knowledge and predictive capabilities of cross-flow induced vibrations.

In order to reduce the computational resources needed, a URANS approach was used for the fluid. A comprehensive description is given of the used approach and the setup of the model, including mesh sensitivity tests. Finally, results are presented for one of the tests performed in the benchmark.

Results obtained indicate a good match with experimental data, showing that flow-induced vibrations of cylinders in cross-flow can be quite accurately predicted using a URANS-based solver when proper choices are made for the mesh and numerical settings.

[TV10] Numerical simulations of experimental fluid-induced vibrations of cylinders in cross-flow

Daniele Vivaldi[1], Guillaume Ricciardi[2]

1 : Institut de Radioprotection et de Sûreté Nucléaire (IRSN) Cadarache, St Paul-lez-Durance 13115 - France

2 : Commissariat à l'Energie Atomique et aux Energies Alternatives CEA Cadarache

F-13108 Saint-Paul-Lez-Durance - France

This paper presents fluid-structure interaction (FSI) simulations of a published experimental campaign dedicated to the study of fluid-induced vibrations o cylinders. The experimental configuration consists in two in-line cylinders subjected to water cross-flow. This experimental campaign is relevant for numerical FSI validation purposes, since it accesses both fluid and structure measurements. This paper presents the numerical simulations of one of the different water flow rates tested experimentally. The FSI simulations are run with the CFD code code Saturne: a two-way coupling between the fluid and the structure is realized thanks to the implementation of an Euler-Bernoulli finite element beam model inside code Saturne. This paper describes the FSI approach and presents its application to the experimental configuration considered. Numerical results are compared to the experimental ones, in terms of velocity fluctuations behind the cylinders and vibrations of the cylinders. Fluid simulations are realized retaining both a URANS approach and the Scale Adaptive Simulation (SAS) hybrid URANS/LES approach.

NOTES

NOTES