**Fluid Mechanics, Coastal and Maritime Engineering**

Department of Mechanical Engineering  
Short name: FVM

**Addresses**  
Type of address: Postal address  
Street: Niels Koppels Alleé  
Building: 403, 006  
Postal code: DK-2800  
City: Kgs. Lyngby  
Country: Denmark

**Phone numbers**  
Phone: (+45) 45 25 13 84  
Fax: (+45) 45 93 14 75

**Web addresses**  
Web: http://www.fvm.mek.dtu.dk/English.aspx

**E-mails**  
E-mail: info@mek.dtu.dk

**Organisation profile**

**Education**  
The Section and the Department offers a Msc in Coastal and Maritime Engineering and other attractive study opportunities, for instance the study to be a Naval Architect.

**Research**  
The research of the section is concentrated on the following subjects:  
Stochastic wave loads on ships and offshore structures

Parametric roll of ships

Propeller flows

Wave dynamics

Collision and grounding

Risk models for navigational safety

Monitoring and decision support

Composite materials
A regularization method for solving the Poisson equation for mixed unbounded-periodic domains

Abstract Regularized Green's functions for mixed unbounded-periodic domains are derived. The regularization of the Green's function removes its singularity by introducing a regularization radius which is related to the discretization length and hence imposes a minimum resolved scale. In this way the regularized unbounded-periodic Green's functions can be implemented in an FFT-based Poisson solver to obtain a convergence rate corresponding to the regularization order of the Green's function. The high order is achieved without any additional computational cost from the conventional FFT-based Poisson solver and enables the calculation of the derivative of the solution to the same high order by direct spectral differentiation. We illustrate an application of the FFT-based Poisson solver by using it with a vortex particle mesh method for the approximation of incompressible flow for a problem with a single periodic and two unbounded directions.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Spietz, H. J. (Intern), Mølholm Hejlesen, M. (Intern), Walther, J. H. (Intern)
Pages: 439–447
Publication date: 2018
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Computational Physics
Volume: 356
ISSN (Print): 0021-9991
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 3.12 SJR 2.034 SNIP 1.822
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 2.098 SNIP 1.988 CiteScore 2.92
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 2.166 SNIP 2.193 CiteScore 3.12
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 2.227 SNIP 2.45 CiteScore 3.3
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 2.161 SNIP 2.052 CiteScore 2.69
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 2.06 SNIP 2.194 CiteScore 2.99
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 2.185 SNIP 2.096
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 2.439 SNIP 2.219
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 2.247 SNIP 2.03
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 2.377 SNIP 2.379
Web of Science (2007): Indexed yes
Scopus rating (2006): SJR 2.182 SNIP 2.285
Web of Science (2006): Indexed yes
Scopus rating (2005): SJR 2.491 SNIP 2.238
Web of Science (2005): Indexed yes
Scopus rating (2004): SJR 2.382 SNIP 2.224
Web of Science (2004): Indexed yes
Scopus rating (2003): SJR 2.335 SNIP 1.958
Web of Science (2003): Indexed yes
Scopus rating (2002): SJR 1.641 SNIP 1.935
Scopus rating (2001): SJR 2.308 SNIP 1.78
Web of Science (2001): Indexed yes
Scopus rating (2000): SJR 2.821 SNIP 1.739
Scopus rating (1999): SJR 1.6 SNIP 1.434
Original language: English
The Poisson equation, Unbounded and periodic domains, Mixed boundary conditions, Regularization methods, Green's function solution, Vortex methods
DOIs: 10.1016/j.jcp.2017.12.018
Source: RIS
Source-ID: urn:C7004E8695A5FDF9BD23F8EEF4D3D502
Publication: Research - peer-review › Journal article – Annual report year: 2018

Evaluation of a Lagrangian Soot Tracking Method for the prediction of primary soot particle size under engine-like conditions

This paper reports the implementation and evaluation of a Lagrangian soot tracking (LST) method for the modeling of soot in diesel engines. The LST model employed here has the tracking capability of a Lagrangian method and the ability to predict primary soot particle sizing. The Moss-Brookes soot model is used here as the Eulerian method to simulate soot formation and oxidation processes. The inception, surface growth and oxidation models are adopted and modified such that the associated reaction rates can be computed using the Lagrangian approach. The soot nuclei are treated as Lagrangian particles when the mass of incipient soot exceeds a designated threshold value. Their trajectories are then computed using the particle momentum equation. The change of primary soot particle size is dependent on the modified Lagrangian surface growth and soot oxidation models. Performance of the LST model in predicting temporal soot cloud development, mean soot diameter and primary soot size distribution is evaluated using measurements of n-heptane and n-dodecane spray combustion obtained under diesel engine-like conditions. In addition, sensitivity studies are carried out to investigate the influence of soot surface ageing and oxidation rates on the primary soot particle size distribution. With the use of surface ageing, the predicted maximum primary soot particle sizes are closer to the experimentally measured maximum primary soot sizes. Also, the associated particle size distribution shows a lognormal shape. A higher rate of soot oxidation due to OH causes the soot particles to be fully oxidized downstream of the flame. In general, the LST model performs better than the Eulerian method in terms of predicting soot sizing and accessing information of individual soot
particles, both of which are shortcomings of the Eulerian method.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy, University of Nottingham, Malaysia Campus
Authors: Cai Ong, J. (Ekstern), Pang, K. M. (Intern), Walther, J. H. (Intern), Ho, J. (Ekstern), Kiat Ng, H. (Ekstern)
Pages: 70-95
Publication date: 2018
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Journal of Aerosol Science
Volume: 70-95
ISSN (Print): 0021-8502

**Ratings:**
- BFI (2018): BFI-level 1
- Web of Science (2018): Indexed yes
- BFI (2017): BFI-level 1
- Web of Science (2017): Indexed Yes
- BFI (2016): BFI-level 1
- Scopus rating (2016): CiteScore 2.21 SJR 0.843 SNIP 1.199
- BFI (2015): BFI-level 1
- Scopus rating (2015): SJR 1.072 SNIP 1.318 CiteScore 2.47
- BFI (2014): BFI-level 1
- Scopus rating (2014): SJR 1.068 SNIP 1.586 CiteScore 2.72
- Web of Science (2014): Indexed yes
- BFI (2013): BFI-level 1
- Scopus rating (2013): SJR 1.187 SNIP 1.858 CiteScore 2.9
- ISI indexed (2013): ISI indexed yes
- BFI (2012): BFI-level 1
- Scopus rating (2012): SJR 1.294 SNIP 1.638 CiteScore 2.64
- ISI indexed (2012): ISI indexed yes
- Web of Science (2012): Indexed yes
- BFI (2011): BFI-level 1
- Scopus rating (2011): SJR 1.137 SNIP 1.623 CiteScore 2.63
- ISI indexed (2011): ISI indexed yes
- BFI (2010): BFI-level 1
- Scopus rating (2010): SJR 1.169 SNIP 1.604
- Web of Science (2010): Indexed yes
- BFI (2009): BFI-level 1
- Scopus rating (2009): SJR 1.457 SNIP 1.782
- Web of Science (2009): Indexed yes
- BFI (2008): BFI-level 2
- Scopus rating (2008): SJR 1.375 SNIP 1.353
- Scopus rating (2007): SJR 0.999 SNIP 0.892
- Scopus rating (2006): SJR 1.044 SNIP 0.929
- Scopus rating (2005): SJR 0.812 SNIP 0.737
- Scopus rating (2004): SJR 1.278 SNIP 1.932
- Scopus rating (2003): SJR 0.616 SNIP 0.554
- Scopus rating (2002): SJR 0.666 SNIP 0.636
- Scopus rating (2001): SJR 0.455 SNIP 0.419
- Scopus rating (2000): SJR 0.457 SNIP 0.687
- Web of Science (2000): Indexed yes
- Scopus rating (1999): SJR 0.52 SNIP 0.349

Original language: English

Soot formation, Soot oxidation, Primary soot particle size, Lagrangian soot tracking, Diesel engine
Near-wellbore modeling of a horizontal well with Computational Fluid Dynamics

The oil production by horizontal wells is a complex phenomenon that involves flow through the porous reservoir, completion interface and the well itself. Conventional reservoir simulators can hardly resolve the flow through the completion into the wellbore. On the contrary, Computational Fluid Dynamics (CFD) is capable of modeling the complex interaction between the creeping reservoir flow and turbulent well flow for single phases, while capturing both the completion geometry and formation damage. A series of single phase steady-state simulations are undertaken, using such fully coupled three dimensional numerical models, to predict the inflow to the well. The present study considers the applicability of CFD for near-wellbore modeling through benchmark cases with available analytical solutions. Moreover, single phase steady-state numerical investigations are performed on a specific perforated horizontal well producing from the Siri field, offshore Denmark. The performance of the well is investigated with an emphasis on the inflow profile and the productivity index for different formation damage scenarios. A considerable redistribution of the inflow profile were found when the filtrate invasion extended beyond the tip of the perforations.
Numerical simulation of condensation of sulfuric acid and water in a large two-stroke marine diesel engine

In the present study, three-dimensional (3D) computational fluid dynamics simulations are performed to examine the process of sulfuric acid (H2SO4) and water (H2O) condensation in a large two-stroke marine diesel engine. A skeletal n-heptane chemical mechanism is coupled with a sulfur (S) subset to simulate the combustion process as well as the formation of sulfuric oxides (SOx) and H2SO4. The condensation process is simulated using a fluid film model which is coupled with the in-cylinder gas phase. Prior to the engine simulations, the fluid film condensation model is validated using the experimental data of sulfuric acid condensation rate in a laminar pipe flow. Next, the engine model is validated against the experimental sulfur dioxide (SO2) to sulfur trioxide (SO3) conversion obtained from the corresponding test engine. Both of the validation studies show a good agreement with the experimental data. The engine model is then utilized to simulate condensation for different operating conditions. The engine simulation results reveal that the fluid film has a significant effect on the total mass of sulfuric acid vapor and a marginal effect on the total mass of water vapor. A close to linear correlation is found between the fuel sulfur content and the total condensed mass of sulfuric acid. The level of humidity of the scavenging air does not affect the condensation of sulfuric acid considerably, relative to the humidity increase, but it has a high impact on water condensation. The study of the scavenging pressure level reveals a counter intuitive behavior where the condensation rates decrease with higher scavenging pressures due to the flow regime and flame size. Next, increasing the cylinder liner temperature decreases significantly the water condensation contrary to the sulfuric acid condensation which is marginally affected. The increase in lubricant film thickness results in a decrease for both the sulfuric acid and water condensation with a more pronounced reduction for water. Finally, a comparison between the high and low load operating conditions reveals a small drop in the total condensed mass of sulfuric acid and water for the low load conditions.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy, MAN B&W Diesel A/S
Authors: Karvounis, N. (Intern), Pang, K. M. (Intern), Mayer, S. (Ekstern), Walther, J. H. (Intern)
Pages: 1009-1020
Publication date: 2018
Main Research Area: Technical/natural sciences

Publication information
Journal: Applied Energy
Volume: 211
ISSN (Print): 0306-2619
Response predictions using the observed autocorrelation function
This article studies a procedure that facilitates short-time, deterministic predictions of the wave-induced motion of a marine vessel, where it is understood that the future motion of the vessel is calculated ahead of time. Such predictions are valuable to assist in the execution of many marine operations (crane lifts, helicopter landings, etc.), as a specic prediction can be used to inform whether it is safe, or not, to carry out the particular operation within the nearest time horizon. The examined prediction procedure relies on observations of the correlation structure of the wave-induced response in study. Thus, predicted (future) values ahead of time for a given time history recording are computed through a mathematical combination of the sample autocorrelation function and previous measurements recorded just prior to the moment of action. Importantly, the procedure does not need input about the exciting wave system, and neither does it rely on o-line training. In the article, the prediction procedure is applied to experimental data obtained through model-scale tests, and the procedure's predictive performance is investigated for various irregular wave scenarios. The presented results show that predictions can be successfully made in a time horizon corresponding to about 8-9 wave periods ahead of current time (the moment of action).

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Norwegian University of Science and Technology
Authors: Nielsen, U. D. (Intern), H. Brodtkorb, A. (Ekstern), Jensen, J. J. (Intern)
Pages: 31–52
Publication date: 2018
Main Research Area: Technical/natural sciences

Publication information
Journal: Marine Structures
Volume: 58
ISSN (Print): 0951-8339
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.49 SJR 1.655 SNIP 2.636
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.618 SNIP 2.602 CiteScore 2.77
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.431 SNIP 3.026 CiteScore 2.18
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.314 SNIP 2.871 CiteScore 2.42
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.836 SNIP 3.464 CiteScore 1.76
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.611 SNIP 2.795 CiteScore 1.82
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 1.136 SNIP 2.518
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 1.165 SNIP 2.795
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 1
The paper presents a review of experiments and calculation procedures for the resistances of ship structural components subjected to impact loadings. The purpose of the paper is to highlight the importance of large-scale collision and grounding experiments and to discuss the technical difficulties and challenges in analytical, empirical and numerical analyses. Experiments on ship structural components are benchmarks and baselines, used to propose analytical or empirical formulae for the structural energy absorptions and/or to validate numerical analyses considering the actual structural and material characteristics. In recent literature, analytical and numerical calculations provide relatively accurate prediction of the purely plastic responses of ship structures under impact loads, but universal approaches have not been found for fracture predictions. The existing formulae for failure criteria still show limitations when evaluating material fracture in various damage patterns. Recently, semi-analytical approaches have been developed to evaluate the relationship between the absorbed energy and the damaged material volume, taking into account the structural arrangements. It seems that these semi-analytical methods often show better accuracy than the numerical simulations when predicting the experimental results.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Wuhan University of Technology, Lloyd's Register EMEA
Authors: Liu, B. (Ekstern), Pedersen, P. T. (Intern), Zhu, L. (Ekstern), Zhang, S. (Ekstern)
Pages: 105-121
Publication date: 2018
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Marine Structures
Volume: 59
ISSN (Print): 0951-8339
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Sea State Estimation Using Vessel Response in Dynamic Positioning

This paper presents a novel method for estimating the sea state parameters based on the heave, roll and pitch response of a vessel in dynamic positioning (DP), i.e., without forward speed. The algorithm finds the wave spectrum estimate from the response measurements by directly solving a set of linear equations, and as a result it is computationally efficient. The main vessel parameters are required as input. Apart from this the method is signal-based, with no assumptions on the
wave spectrum shape. Performance of the proposed algorithm is demonstrated on full-scale experimental DP data of a vessel in three different sea states at head, bow, beam, quartering and following sea waves, respectively.

**General information**

*State:* Published  
*Organisations:* Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Norwegian University of Science and Technology  
*Authors:* H. Brodtkorb, A. (Ekstern), Nielsen, U. D. (Intern), J. Sørensen, A. (Ekstern)  
*Pages:* 76–86  
*Publication date:* 2018  
*Main Research Area:* Technical/natural sciences

**Publication information**

*Journal:* Applied Ocean Research  
*Volume:* 70  
*ISSN (Print):* 0141-1187  
*Ratings:*  
  *BFI (2018):* BFI-level 1  
  *Web of Science (2018):* Indexed yes  
  *BFI (2017):* BFI-level 1  
  *Web of Science (2017):* Indexed Yes  
  *BFI (2016):* BFI-level 1  
  *Scopus rating (2016):* CiteScore 2.16 SJR 1.202 SNIP 1.75  
  *Web of Science (2016):* Indexed yes  
  *BFI (2015):* BFI-level 1  
  *Scopus rating (2015):* SJR 1.245 SNIP 1.999 CiteScore 2.1  
  *Web of Science (2015):* Indexed yes  
  *BFI (2014):* BFI-level 1  
  *Scopus rating (2014):* SJR 1.442 SNIP 2.244 CiteScore 1.91  
  *BFI (2013):* BFI-level 1  
  *Scopus rating (2013):* SJR 1.243 SNIP 2.332 CiteScore 1.84  
  *ISI indexed (2013):* ISI indexed yes  
  *BFI (2012):* BFI-level 1  
  *Scopus rating (2012):* SJR 1.234 SNIP 1.922 CiteScore 1.46  
  *ISI indexed (2012):* ISI indexed yes  
  *Web of Science (2012):* Indexed yes  
  *BFI (2011):* BFI-level 1  
  *Scopus rating (2011):* SJR 1.496 SNIP 1.975 CiteScore 1.73  
  *ISI indexed (2011):* ISI indexed yes  
  *BFI (2010):* BFI-level 1  
  *Scopus rating (2010):* SJR 1.112 SNIP 1.485  
  *Web of Science (2010):* Indexed yes  
  *BFI (2009):* BFI-level 1  
  *Scopus rating (2009):* SJR 1.012 SNIP 1.69  
  *BFI (2008):* BFI-level 1  
  *Scopus rating (2008):* SJR 1.404 SNIP 2.344  
  *Web of Science (2008):* Indexed yes  
  *Scopus rating (2007):* SJR 1.03 SNIP 1.898  
  *Scopus rating (2006):* SJR 1.248 SNIP 2.064  
  *Scopus rating (2005):* SJR 0.495 SNIP 1.45  
  *Scopus rating (2004):* SJR 0.763 SNIP 1.384  
  *Scopus rating (2003):* SJR 1.115 SNIP 1.669  
  *Scopus rating (2002):* SJR 0.398 SNIP 0.709  
  *Scopus rating (2001):* SJR 1.236 SNIP 1.544  
  *Web of Science (2001):* Indexed yes  
  *Scopus rating (2000):* SJR 0.526 SNIP 1.039
Swim and fly: escape strategy in neustonic and planktonic copepods

Copepods can respond to predators by powerful escape jumps that in some surface-dwelling forms may propel the copepod out of the water. We studied the kinematics and energetics of submerged and out-of-water jumps of two neustonic pontellid copepods, Anomalocera patersoni and Pontella mediterranea, and one pelagic calanoid copepod, Calanus helgolandicus (euxinus). We show that jumping out of the water does not happen just by inertia gained during the copepod's acceleration underwater, but also requires the force generated by the thoracic limbs when breaking through the water's surface to overcome surface tension, drag and gravity. The timing of this appears to be necessary for success. At the moment of breaking the water interface, the instantaneous velocity of the two pontellids reached 125 cm s⁻¹, while their maximum underwater speed (115 cm s⁻¹) was close to that of similarly sized C. helgolandicus (106 cm s⁻¹). The average specific power produced by the two pontellids during out-of-water jumps (1700-3300 W kg⁻¹ muscle mass) was close to that during submerged jumps (900-1600 W kg⁻¹ muscle mass) and, in turn, similar to that produced during submerged jumps of C. helgolandicus (1300 W kg⁻¹ muscle mass). The pontellids may shake off water adhering to their body by repeated strokes of the limbs during flight, which leads to a slight acceleration in the air. Our observations suggest that out-of-water jumps of pontellids are not dependent on any exceptional ability to perform this behavior but have the same energetic cost and are based on the same kinematic patterns and contractive capabilities of muscles as those of copepods swimming submerged.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, National Institute of Aquatic Resources, Centre for Ocean Life, National Academy of Sciences of Ukraine
Authors: Svetlichny, L. (Ekstern), Larsen, P. S. (Intern), Kørboe, T. (Intern)
Publication date: 2018
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Experimental Biology
Volume: 221
Issue number: 2
Article number: jeb167262
ISSN (Print): 0022-0949
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.62 SJR 1.722 SNIP 1.279
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.812 SNIP 1.222 CiteScore 2.4
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.722 SNIP 1.331 CiteScore 2.51
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.719 SNIP 1.323 CiteScore 2.75
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.612 SNIP 1.395 CiteScore 2.91
ISI indexed (2012): ISI indexed yes
BFI (2011): BFI-level 2
The aim of this work is to present a fast and viable approach for taking into account turbulence in topology optimization of complex fluid flow systems, without resorting to any simplifying assumptions in the derivation of discrete adjoints. Topology optimization is an iterative gradient-based design process which minimizes an objective and satisfies a set of selected design constraints by distributing material in a design domain. The gradients are obtained using adjoint sensitivity analysis which requires solutions of a forward state problem and an additional adjoint problem. In the presented article the forward solver is based on finite volume discretized Reynolds-averaged Navier–Stokes equations coupled with either one- or two-equation turbulence closure models, and the adjoint solver is obtained via automatic differentiation. The presented approach is demonstrated on the optimization of several 2D and 3D examples including a detailed comparison to designs and sensitivities obtained with different turbulence models and under a frozen turbulence assumption. The results demonstrate the importance of exact sensitivity analysis and open new possibilities for the design of large scale multiphysics problems involving turbulent flows.

General information
State: Published
Organisations: Department of Mechanical Engineering, Solid Mechanics, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Dilgen, C. B. (Intern), Dilgen, S. B. (Intern), Fuhrman, D. R. (Intern), Sigmund, O. (Intern), Lazarov, B. S. (Intern)
Pages: 363–393
Publication date: 2018
Main Research Area: Technical/natural sciences

Publication information
Journal: Computer Methods in Applied Mechanics and Engineering
Water thermophoresis in carbon nanotubes: the interplay between thermophoretic and friction forces

Thermophoresis is the phenomenon wherein particles experience a net drift induced by a thermal gradient. In this work, molecular dynamics simulations are conducted to study with atomistic detail the thermophoresis of water nanodroplets inside carbon nanotubes (CNTs) and its interplay with the retarding liquid-solid friction. Different applied temperatures, thermal gradients, and droplet sizes are used to reveal the dynamics of the two kinetic regimes of the thermophoretic motion in CNTs. The results indicate that during the droplet motion, the thermophoretic force is independent of the velocity of the droplet, whereas the magnitude of the retarding friction force exhibits a linear dependence. In fact, in the initial regime the magnitude of the friction force increases linearly with the droplet velocity, until the thermophoretic force is balanced by the friction force as the droplet reaches its terminal velocity in the final regime. In addition, an increase in the magnitude of the thermophoretic force is found for longer water droplets. These findings provide a deeper understanding of liquid transport driven by temperature gradients in nanoconfined geometries where liquid-solid interfaces govern fluidics.

General information
State: Accepted/In press
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Oyarzua, E. (Ekstern), Walther, J. H. (Intern), Zambrano, H. A. (Ekstern)
Publication date: 2018
Main Research Area: Technical/natural sciences

Publication information
Journal: Physical Chemistry Chemical Physics
ISSN (Print): 1463-9076
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 4.06 SJR 1.678 SNIP 1.117
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.771 SNIP 1.244 CiteScore 4.45
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.772 SNIP 1.253 CiteScore 4.29
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.715 SNIP 1.216 CiteScore 4.05
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.916 SNIP 1.184 CiteScore 3.67
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.697 SNIP 1.203 CiteScore 3.6
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 1.802 SNIP 1.196
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 2.127 SNIP 1.369
Web of Science (2009): Indexed yes
A brute-force spectral approach for wave estimation using measured vessel responses

The article introduces a spectral procedure for sea state estimation based on measurements of motion responses of a ship in a short-crested seaway. The procedure relies fundamentally on the wave buoy analogy, but the wave spectrum estimate is obtained in a direct - brute-force - approach, and the procedure is simple in its mathematical formulation. The actual formulation is extending another recent work by including vessel advance speed and short-crested seas. Due to its simplicity, the procedure is computationally efficient, providing wave spectrum estimates in the order of a few seconds, and the estimation procedure will therefore be appealing to applications related to real-time, onboard control and decision support systems for safe and efficient marine operations. The procedure's performance is evaluated by use of numerical simulation of motion measurements, and it is shown that accurate wave spectrum estimates can be obtained for all wave directions in short-crested waves, taking the wave system to be composed by both wind generated sea and swell.

General information
State: Submitted
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Norwegian University of Science and Technology
Authors: Nielsen, U. D. (Intern), H. Brodtkorb, A. (Ekstern), J. Sørensen, A. (Ekstern)
Number of pages: 37
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Marine Structures
ISSN (Print): 0951-8339
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.49 SJR 1.655 SNIP 2.636
Web of Science (2016): Indexed yes
**A CFD Investigation on the Effect of the Air Entrainment in Breaking Wave Impacts on a Mono-Pile**

In impacts of breaking waves on offshore structures, it is still not well-known how the air entrainment phenomenon affects the exerted loads. In this paper, a developed CFD solver capable of simulating the air entrainment process was employed to reproduce an experimental investigation on the impact of a spilling wave against a circular cylinder. The exerted in-line force was computed with and without the inclusion of dispersed bubbles. Results showed that the magnitude of the computed force was affected when the entrainment of bubbles was simulated.

**General information**
A concise account of techniques available for shipboard sea state estimation
This article gives a review of techniques applied to make sea state estimation on the basis of measured responses on a ship. The general concept of the procedures is similar to that of a classical wave buoy, which exploits a linear assumption between waves and the associated motions. In the frequency domain, this assumption yields the mathematical relation between the measured motion spectra and the directional wave spectrum. The analogy between a buoy and a ship is clear, and the author has worked on this wave buoy analogy for about fifteen years. In the article, available techniques for shipboard sea state estimation are addressed, but with a focus on only the wave buoy analogy. Most of the existing work is based on methods established in the frequency domain but, to counteract disadvantages of the frequency-domain procedures, newer studies are working also on procedures formulated directly in the time domain. Sample results from several studies are included, and the main findings from these are mentioned.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Nielsen, U. D. (Intern)
Pages: 352-362
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Ocean Engineering
Volume: 129
ISSN (Print): 0029-8018
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.46 SJR 1.315 SNIP 2.014
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.172 SNIP 1.989 CiteScore 2.19
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.252 SNIP 2.323 CiteScore 2.11
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.178 SNIP 2.773 CiteScore 2.2
ISI indexed (2013): ISI indexed yes
Adjoint Optimisation of the Turbulent Flow in an Annular Diffuser

In the present study, a numerical optimisation of guide vanes in an annular diffuser, is performed. The optimisation is performed for the purpose of improving the following two parameters simultaneously; the first parameter is the uniformity perpendicular to the flow direction, a 1/3 diameter downstream of the expansion. The second parameter is the pressure loss introduced by these guide vanes. The optimisation yields an improvement of the uniformity of 1.5% and a 28% reduction in the over all pressure loss.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Technical University of Denmark, MAN Diesel and Turbo
Authors: Gotfredsen, E. (Intern), Agular Knudsen, C. (Ekstern), Kunoy, J. D. (Ekstern), Meyer, K. E. (Intern), Walther, J. H. (Intern)
Pages: 71-74
Publication date: 2017

Host publication information
Title of host publication: Proceedings of the 30th Nordic Seminar on Computational Mechanics (NSCM-30)
Editors: Hasberg, J., Pedersen, N.
BFI conference series: Nordic Seminar on Computational Mechanics (5010906)
**A Method for Ship Collision Damage and Energy Absorption Analysis and its Validation**

For design evaluation, there is a need for a method which is fast, practical and yet accurate enough to determine the absorbed energy and collision damage extent in ship collision analysis. The most well-known simplified empirical approach to collision analysis was made probably by Minorsky, and its limitation is also well-recognised. The authors have previously developed simple expressions for the relation between the absorbed energy and the damaged material volume which take into account the structural arrangements, the material properties and the damage modes. The purpose of the present paper is to re-examine this method's validity and accuracy for ship collision damage analysis in ship design assessments by comprehensive validations with experimental results from the public domain. In total, 20 experimental tests have been selected, analysed and compared with the results calculated using the proposed method. It can be concluded that the proposed method has a good accuracy with the mean value of 0.988 and standard deviation of 0.042.

**General information**

**State:** Published

**Organisations:** Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register EMEA

**Authors:** Zhang, S. (Ekstern), Pedersen, P. T. (Intern)

**Pages:** 783–796

**Publication date:** 2017

**Main Research Area:** Technical/natural sciences

**Publication information**

**Journal:** Ships and Offshore Structures

**Volume:** 20

**Issue number:** 5

**ISSN (Print):** 1744-5302

**Ratings:**

- BFI (2018): BFI-level 1
- Web of Science (2018): Indexed yes
- BFI (2017): BFI-level 1
- Web of Science (2017): Indexed yes
- BFI (2016): BFI-level 1
- Scopus rating (2016): CiteScore 1.34 SJR 1.025 SNIP 1.493
- Web of Science (2016): Indexed yes
- BFI (2015): BFI-level 1
- Scopus rating (2015): SJR 0.735 SNIP 1.011 CiteScore 0.89
- Web of Science (2015): Indexed yes
- BFI (2014): BFI-level 1
- Scopus rating (2014): SJR 0.52 SNIP 1.163 CiteScore 0.69
- Web of Science (2014): Indexed yes
- BFI (2013): BFI-level 1
- Scopus rating (2013): SJR 0.651 SNIP 1.14 CiteScore 0.91
- ISI indexed (2013): ISI indexed yes
- BFI (2012): BFI-level 1
- Scopus rating (2012): SJR 0.299 SNIP 0.604 CiteScore 0.41
- ISI indexed (2012): ISI indexed no
- BFI (2011): BFI-level 1
- Scopus rating (2011): SJR 0.47 SNIP 0.776 CiteScore 0.47
- ISI indexed (2011): ISI indexed no
- Web of Science (2011): Indexed yes
- BFI (2010): BFI-level 1
- Scopus rating (2010): SJR 0.483 SNIP 0.727
- BFI (2009): BFI-level 1
A New Volume-Of-Fluid Method in Openfoam

To realise the full potential of Computational Fluid Dynamics (CFD) within marine science and engineering, there is a need for continuous maturing as well as verification and validation of the numerical methods used for free surface and interfacial flows. One of the distinguishing features here is the existence of a water surface undergoing large deformations and topological changes during transient simulations e.g. of a breaking wave hitting an off-shore structure. To date, the most successful method for advecting the water surface in marine applications is the Volume-of-Fluid (VOF) method. While VOF methods have become quite advanced and accurate on structured meshes, there is still room for improvement when it comes to unstructured meshes of the type needed to simulate flows in and around complex geometric structures. We have recently developed a new geometric VOF algorithm called isoAdvector for general meshes and implemented it in the OpenFOAM interfacial flow solver called interFoam. We have previously shown the advantages of isoAdvector for simple pure advection test cases on various mesh types. Here we test the effect of replacing the existing interface advection method in interFoam, based on MULES limited interface compression, with the new isoAdvector method. Our test case is a steady 2D stream function wave propagating in a periodic domain. Based on a series of simulations with different numerical settings, we conclude that the introduction of isoAdvector has a significant effect on wave propagation with interFoam. There are several criteria of success: Preservation of water volume, of interface sharpness and shape, of crest kinematics and celerity, not to mention computational efficiency. We demonstrate how isoAdvector can improve on many of these parameters, but also that the success depends on the solver setup. Thus, we cautiously conclude that isoAdvector is a viable alternative to MULES when set up correctly, especially when interface sharpness, interface smoothness and calculation times are important. There is, however, still potential for improvement in the coupling of isoAdvector with interFoam's PISO based pressure-velocity solution algorithm.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Wind Energy, Fluid Mechanics, University of Zagreb
Authors: Pedersen, J. R. (Intern), Eltard-Larsen, B. (Intern), Bredmose, H. (Intern), Jasak, H. (Ekstern)
Number of pages: 12
Publication date: 2017
Main Research Area: Technical/natural sciences
CFD, Marine Engineering, Interfacial Flows, IsoAdvector, VOF Methods, Surface Gravity Waves
Electronic versions:
Roenbyetal2017.pdf
Publication: Research - peer-review › Paper – Annual report year: 2017

A regularized vortex-particle mesh method for large eddy simulation

We present recent developments of the remeshed vortex particle-mesh method for simulating incompressible fluid flow. The presented method relies on a parallel higher-order FFT based solver for the Poisson equation. Arbitrary high order is achieved through regularization of singular Green's function solutions to the Poisson equation and recently we have derived novel high order solutions for a mixture of open and periodic domains. With this approach the simulated variables may formally be viewed as the approximate solution to the filtered Navier Stokes equations, hence we use the method for Large Eddy Simulation by including a dynamic subfilter-scale model based on test-filters compatible with the aforementioned regularization functions. Further the subfilter-scale model uses Lagrangian averaging, which is a natural candidate in light of the Lagrangian nature of vortex particle methods. A multiresolution variation of the method is applied to simulate the benchmark problem of the flow past a square cylinder at Re = 22000 and the obtained results are compared to results from the literature.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Spietz, H. J. (Intern), Walther, J. H. (Intern), Hejlesen, M. M. (Intern)
Carbon Nanotubes as Thermally Induced Water Pumps

Thermal Brownian motors (TBMs) are nanoscale machines that exploit thermal fluctuations to provide useful work. We introduce a TBM-based nanopump which enables continuous water flow through a carbon nanotube (CNT) by imposing an axial thermal gradient along its surface. We impose spatial asymmetry along the CNT by immobilizing certain points on its surface. We study the performance of this molecular motor using molecular dynamics (MD) simulations. From the MD trajectories, we compute the net water flow and the induced velocity profiles for various imposed thermal gradients. We find that spatial asymmetry modifies the vibrational modes of the CNT induced by the thermal gradient, resulting in a net water flow against the thermal gradient. Moreover, the kinetic energy associated with the thermal oscillations rectifies the Brownian motion of the water molecules, driving the flow in a preferred direction. For imposed thermal gradients of 0.5-3.3 K/nm, we observe continuous net flow with average velocities up to 5 m/s inside CNTs with diameters of 0.94, 1.4, and 2.0 nm. The results indicate that the CNT-based asymmetric thermal motor can provide a controllable and robust system for delivery of continuous water flow with potential applications in integrated nanofluidic devices.
Cavity prediction in sand mould production applying the DISAMATIC process

The sand shot in the DISAMATIC process is simulated by the discrete element method (DEM) taking into account the influence and coupling of the airflow with computational fluid dynamics (CFD). The DEM model is calibrated by a ring shear test, a sand pile experiment and a slump test. Subsequently, the DEM model is used to model the propagation of the green sand inside the mold chamber and the results are compared to experimental video footage. The chamber contains two cavities designed to quantify the deposited mass of green sand. The deposition of green sand in these two cavities is investigated with three cases of different air vent settings which control the ventilation of the chamber. These settings resulted in different air- and particle-velocities as well as different accumulated masses in the cavities, which were successfully simulated by the model.

General information

State: Published
Organisations: Department of Mechanical Engineering, Manufacturing Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DISA Industries A/S
Authors: Hovad, E. (Intern), Larsen, P. (Ekstern), Spangenberg, J. (Intern), Walther, J. H. (Intern), Thorborg, J. (Intern), Hattel, J. H. (Intern)
Pages: 204-217
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information

Journal: Powder Technology
Volume: 321
ISSN (Print): 0032-5910
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 3.16 SJR 0.983 SNIP 1.482
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.965 SNIP 1.598 CiteScore 2.99
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.89 SNIP 1.649 CiteScore 2.67
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.901 SNIP 1.875 CiteScore 2.64
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.854 SNIP 1.826 CiteScore 2.36
CFD Analysis of Scale Effects on Conventional and Tip-Modified Propellers

Full-scale propeller performance is traditionally predicted by scaling model-scale test results, but the traditional scaling methods do not take into account hydrodynamic distinctions of tip-modified propellers in full-scale performance. An open-water CFD analysis is made on scale effects of tip-modified and conventional propellers, which are designed for the same operating condition with identical propeller diameter and expanded area ratio. While model-scale computations are made with a transition model, a fully turbulent flow is modeled in the full-scale computations. The investigation on the effects of the transition model shows that laminar and transitional flow modeling is crucial in model-scale computations. Grid-independent solutions at model and full scale are achieved by grid verification studies.

The CFD analysis of scale effects shows that the efficiency gain of the tip-modified propeller is increased at full scale. The difference of scale effects between the tip-modified and conventional propellers is related to alterations of tip vortex and sectional pressuredistributions by the bent tip and the higher spanwiseloading at the tip region of the tip-modified propeller.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Shin, K. W. (Ekstern), Andersen, P. (Intern)
Number of pages: 8
Publication date: 2017

Host publication Information
Title of host publication: Proceedings of the Fifth International Symposium on Marine Propulsors - SMP’17
Main Research Area: Technical/natural sciences
Conference: Fifth International Symposium on Marine Propulsors, Helsinki, Finland, 12/06/2017 - 12/06/2017
CFD modeling of condensation process of water vapor in supersonic flows

The condensation phenomenon of vapor plays an important role in various industries, such as the steam flow in turbines and refrigeration system. A mathematical model is developed to predict the spontaneous condensing phenomenon in the supersonic flows using the nucleation and droplet growth theories. The numerical approach is validated with the experimental data, which shows a good agreement between them. The condensation characteristics of water vapor in the Laval nozzle are described in detail. The results show that the condensation process is a rapid variation of the vapor-liquid phase change both in the space and in time. The spontaneous condensation of water vapor will not appear immediately when the steam reaches the saturation state. Instead, it occurs further downstream the nozzle throat, where the steam is in the state of supersaturation.
CFD modeling of particle behavior in supersonic flows with strong swirs for gas separation

The supersonic separator is a novel technique to remove the condensable components from gas mixtures. But the particle behavior is not well understood in this complex supersonic flow. The Discrete Particle Method was used here to study the particle motion in supersonic flows with a strong swirl. The results showed that the gas flow was accelerated to supersonic velocity, and created the low pressure and temperature conditions for gas removal. Most of the particles collided with the walls or entered into the liquid-collection space directly, while only a few particles escaped together with the gas flow from the dry gas outlet. The separation efficiency reached over 80%, when the droplet diameter was more than 1.5 μm. The optimum length of the cyclonic separation section was approximate 16–20 times of the nozzle throat diameter to obtain higher collection efficiency for the supersonic separator with a delta wing.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Changzhou University
Authors: Yang, Y. (Ekstern), Wen, C. (Intern)
Pages: 22-28
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Separation and Purification Technology
Volume: 174
ISSN (Print): 1383-5866
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.78 SJR 1.023 SNIP 1.394
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.078 SNIP 1.504 CiteScore 3.75
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.257 SNIP 1.54 CiteScore 3.5
Characterization and Erosion Modeling of a Nozzle-Based Inflow-Control Device

In the petroleum industry, water-and-gas breakthrough in hydrocarbon reservoirs is a common issue that eventually leads to uneconomic production. To extend the economic production lifetime, inflow-control devices (ICDs) are designed to delay the water-and-gas breakthrough. Because the lifetime of a hydrocarbon reservoir commonly exceeds 20 years and it is a harsh environment, the reliability of the ICDs is vital.

**General information**

State: Published
Organisations: Department of Chemistry, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Welltec, Lloyd's Register Consulting
Authors: Olsen, J. J. (Intern), Hemmingsen, C. S. (Intern), Bergmann, L. (Ekstern), Nielsen, K. K. (Ekstern), Glimberg, S. L. (Ekstern), Walther, J. H. (Intern)
Number of pages: 10
Publication date: 2017
Main Research Area: Technical/natural sciences
Computational Fluid Dynamics of Choanoflagellate Filter-Feeding

Choanoflagellates are unicellular aquatic organisms with a single flagellum that drives a feeding current through a funnel-shaped collar filter on which bacteria-sized prey are caught. Using computational fluid dynamics (CFD) we model the beating flagellum and the complex filter flow of the choanoflagellate Diaphanoeca grandis. Our CFD simulations based on the current understanding of the morphology underestimate the experimentally observed clearance rate by more than an order of magnitude: The beating flagellum is simply unable to draw enough water through the fine filter. Our observations motivate us to suggest a radically different filtration mechanism that requires a flagellar vane (sheet), and addition of a wide vane in our CFD model allows us to correctly predict the observed clearance rate.
Continuous versus pulsating flow boiling. Experimental comparison, visualization, and statistical analysis
This experimental study investigates an active method for flow boiling heat transfer enhancement by means of fluid flow pulsation. The hypothesis is that pulsations increase the flow boiling heat transfer by means of better bulk fluid mixing, increased wall wetting, and flow-regime destabilization. The fluid pulsations are introduced by a flow modulating expansion device and are compared with continuous flow by a stepper-motor expansion valve in terms of time-averaged heat transfer coefficient. The cycle time ranges from 1 to 9 s for the pulsations. The time-averaged heat transfer coefficients are reduced from transient measurements immediately downstream of the expansion valves at low vapor qualities. The results show that the pulsations improve the time-averaged heat transfer coefficient by 3.2% on average at low cycle time (1 to 2 s), whereas the pulsations may reduce the time-averaged heat transfer coefficient by as much as 8% at high heat flux (q > 35 kW/m²) and cycle time (8 s). The latter reduction is attributed to a significant dry-out that occurs when the flow modulating expansion valve is closed. Additionally, the effect of fluid flow pulsations is found to be statistically significant, disregarding the lowest heat flux measurements.
Conversion of Measured Turbulence Spectra from Temporal to Spatial Domain

The spatial structure of a turbulent velocity field is of great theoretical interest as its kinematics describe the distribution of spatial scales and its dynamics describe their evolution from large energy carrying scales to smaller scales and finally to dissipation.

However, the overwhelming number of turbulence measurements results in time records from stationary probes, either hot-wire probes (hot-wire anemometers, HWA) or laser beam probes (laser Doppler anemometers, LDA). The spatial structure of the turbulent velocity field is then inferred by "Taylor's hypothesis," as first presented in [1], assuming a "frozen" velocity field carried past the probe with the local mean velocity. However, Taylor's hypothesis breaks down at higher turbulence intensities and can then only be applied with additional corrections, see, for example, [2-4].
Deterministic Predictions of Vessel Responses Based on Past Measurements

The paper deals with a prediction procedure from which global wave-induced responses can be deterministically predicted a short time, 10-50 s, ahead of current time. The procedure relies on the autocorrelation function and takes into account prior measurements only; i.e. knowledge about wave conditions is not needed. In the present study, the procedure is examined on artificially simulated data that represents the measurements. It is shown that predictions, in most cases, can be made fairly accurate up to 20 s ahead of current time; for longer periods ahead the accuracy reduces somewhat. The sensitivity to the amount of prior data is investigated.

Development of a numerical model for fluid-structure interaction analysis of flow through and around an aquaculture net cage

In the present work, we developed a numerical model for fluid-structure interaction analysis of flow through and around an aquaculture net cage. The numerical model is based on the coupling between the porous media model and the lumped mass structural model. A novel interface was implemented to ensure efficient data exchange and element mapping between the fluid and structural solver via random-access memory. The main idea is to apply a static mesh in the fluid model, in case that large deformation of the net structure reduces the quality of the mesh. Then the geometry of the net cage was approximated by a set of dynamic porous zones, where the grid cells were updated at every iteration based on the transferred nodal positions from the structural model. A time stepping procedure was introduced, so the solver is applicable in both steady and unsteady conditions. In order to reduce the computational effort, sub-cycling was applied for the structural solver within each time step, based on the quasi-steady state assumption. The numerical model was validated against experiments in both steady and unsteady conditions. In general, the agreement is satisfactory.
Aquaculture net cage, Coupling scheme, Fluid-structure interaction analysis, Lumped mass model, Porous media model

DOIs:
10.1016/j.oceaneng.2017.07.033

Source: Scopus
Source-ID: 85025064374
Publication: Research - peer-review › Journal article – Annual report year: 2017
**Development of Large Scale Bed Forms in the Sea –2DH Numerical Modeling**

Large repetitive patterns on the sea bed are commonly observed in sandy areas. The formation of the bed forms have been studied extensively in literature using linear stability analyses, commonly conducted analytically and with simplifications in the governing equations. This work presents a shallow water equation model that is used to numerically simulate the morphodynamics of the water-bed system. The model includes separate formulations for bed load and suspended load, featuring bed load correction due to a sloping bed and modelled helical flow effects. Horizontal gradients are computed with spectral accuracy, which proves highly efficient for the analysis. Numerical linear stability analysis is used to identify the likely emergence of dominant finite sized bed forms, as a function of governing parameters. These are then used for interpretation of the results of a long time morphological simulation.

**General information**

State: Published  
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering  
Authors: Margalit, J. (Intern), Fuhrman, D. R. (Intern)  
Number of pages: 11  
Publication date: 2017  
Main Research Area: Technical/natural sciences  
Bed forms, Sediment transport, Morphodynamics, Linear stability analysis  
Electronic versions: 178_margalit_jonatan.pdf  
Publication: Research - peer-review › Paper – Annual report year: 2017

**Drag reduction in silica nanochannels induced by graphitic wall coatings**

Transport of water in hydrophilic nanopores is of significant technological and scientific interest. Water flow through hydrophilic nanochannelis known to experience enormous hydraulic resistance. Therefore, drag reduction is essential for the development of highly efficient nanofluidic devices. In this work, we propose the use of graphitic materials as wall coatings in hydrophlic silica nanopores. Specifically, by conducting atomistic simulations, we investigate the flow inside slit and cylindrical silica channels with walls coated with graphene (GE) layers and carbon nanotubes (CNTs), respectively. We develop realistic force fields to simulate the systems of interest and systematically, compare flow rates in coated and uncoated nanochannels under different pressure gradients. Moreover, we assess the effect that GE and CNT translucencies to wettability have on water hydrodynamics in the nanochannels. The influence of channel size is investigated by systematically varying channel heights and nanopore diameters. In particular, we present the computed water density and velocity profiles, volumetric flow rates, slip lengths and flow enhancements, to clearly demonstrate the drag reduction capabilities of graphitic wall coatings.

**General information**

State: Published  
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion  
Authors: Wagemann, E. (Ekstern), Walther, J. H. (Intern), Zambrano, H. (Ekstern)  
Number of pages: 1  
Publication date: 2017  
Event: Abstract from 70th Annual Meeting of the American Physical Society Division of Fluid Dynamics (DFD17), Denver, United States.  
Main Research Area: Technical/natural sciences  
Electronic versions: MWS_DFD17_2017_000016.pdf  
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2017

**Effects of Nozzle Diameter on Diesel Spray Flames: A numerical study using an Eulerian Stochastic Field Method**

The present numerical study aims to assess the performance of an Eulerian Stochastic Field (ESF) model in simulating spray flames produced by three fuel injectors with different nozzle diameters of 100 μm, 180 μm and 363 μm. A comparison to the measurements shows that although the simulated ignition delay times are consistently overestimated, the relative differences remain below 28%. Furthermore, the change of the averaged pressure rise with respect to the variation of nozzle diameter is captured by the model. The simulated flame lift-off lengths also agree with the measurements, with a maximum relative difference of 13%. The spray flame produced by a larger nozzle diameter has a fuel-richer premixed core region despite the longer lift-off length, indicating that the higher fueling rate used with the larger nozzle diameter is a more dominating factor than the lift-off length is in influencing the air entrainment into the upstream of the spray flames. In addition, the simulated normalised flame lengths are found to decrease when the nozzle diameters increase. These predictions are in good qualitative agreement with the experimental observation. This work proves that the ESF model can serve as an important tool for the simulation of spray flames in marine diesel engines, where fuel injectors with different nozzle diameters are applied for pilot and main injections.
Experimental Investigation of the Performance of Tilt Current Meters in Wave-Dominated Flows

In recent years, tilt current meters (TCMs) have received renewed attention as they provide an inexpensive method for measuring currents in the coastal zone. However, previous studies focused mainly on current dominated flows or the current component of the flow. This study investigates the performance of tilt current meters in wave-dominated flows and capturing the wave orbital velocities. A series of laboratory experiments were performed in which tilt current meters were used to measure flow velocities in pure current, pure wave and combined wave-current flows. Both spherical and cylindrical TCMs were investigated in order to assess the effect of TCM shape on its performance. The measured TCM tilt is compared with the flow velocity measured by conventional methods. Furthermore, the ability of a TCM to measure wave orbital and wave-averaged velocities is discussed.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Hansen, A. B. (Intern), Carstensen, S. (Intern)
Pages: 5473-5481
Filter-Feeding Zoobenthos and Hydrodynamics
This chapter summarizes recent years' studies on zoobenthic filter feeding in the sea. General principles are extracted based on experiments and mathematical modeling, mainly from own studies in shallow temperate Danish waters, in order to present primary characteristics of the sophisticated interplay between benthic filter feeders and hydrodynamics. Starting from the general concept of grazing potential and typical data on benthic population densities its realization is considered, first at the level of the individual organism through the processes of pumping and trapping of food particles for ingestion which relies on hydrodynamics. Studies have shown the importance of biomixing giving increased vertical seston flux due to mixing induced by exhalant jets of filter feeders, particularly in stagnant water but likely also in benthic boundary layers over mussel beds at moderate flow velocities. Mathematical models for such flows are discussed. At the scale of benthic boundary layers, mussels experience flows that are usually turbulent, but at the smaller scale of sublayers, colonies of bryozoans experience viscous-dominated flow that needs modeling. Finally, a case study from a particular shallow water area illustrates the effects of tide, current, and wind on vertical mixing, growth rates, and ecological implications. The main biophysical processes that may allow or prevent dense populations of filter feeders to control the phytoplankton biomass in shallow waters are presented along with remaining challenges for development of improved models for the benthic boundary layers, including effects of wall roughness, biomixing, and oscillating flows caused by waves.

Flow in axisymmetric expansion in a catalytic converter
The flow in an axisymmetric expansion (circular diffusor) is used in many different engineering applications, such as heat exchangers, catalytic converters and filters. These applications require a relatively uniform flow at the inlet. To minimise the pressure loss, an ideal solution would be to use a quite long expansion, but this is often not possible due to space restrictions. Therefore a short expansion combined with e.g. guide vanes is often used. The present study will use a Selective Catalytic Reduction (SCR) system for large marine diesel engines as a case. The catalyst is designed for a specific local flow rate and a non-uniform inflow to the catalyst will severely reduce the efficiency of the process. Since each ship will have a unique design the flow system, it is desirable to be able to design the system using Computational Fluid Dynamics (CFD). However, CFD fails to predict flow separation in many cases and cannot be used as the only design tool [1]. Typically CFD has to be validated against experimental data from representative designs under varying conditions to find trustworthy turbulence modelling, sufficient grid resolution and suitable boundary conditions. Here Particle Image Velocimetry (PIV) is a unique method that resolve the entire cross flow. This type of flow is expected to have a fluctuating ‘jet’-like structure from the smaller inlet pipe into the larger converter. The fluctuations of the jet are difficult, if not impossible, to capture with standard time averaged models, and more expensive methods like Large Eddy
Simulation (LES) could be needed. Here PIV has an advantage compared with other measurement methods, because it captures instantaneous flow fields that are relevant for the catalyst efficiency and thus also for CFD validation. The aim of the present study is to investigate flow phenomena in sudden pipe expansions similar to design used for catalytic converters with different upstream conditions and flow conditioning devices like guide vanes. This is done to provide a set of data that can be used to validate the use of CFD to such flows.

For the present study, a down-scaled model of the catalytic converter is constructed, see figure 1. The experiments are performed at laboratory conditions, with lower pressure, temperature and velocity than the full-scale catalytic converter. The Reynolds number based on the velocity in the inlet pipe and the diameter of the converter is Re = 200000. A preliminary study shows that this Reynolds number is high enough to ensure very small dependence of the Reynolds number. The inlet pipe has a diameter of D = 0.1 m. The catalytic container has a diameter of 2.8D and a length of 8D. The diffusor connecting the pipe and the converter container is expanding abruptly within a length of 0.5D. The inlet section has a length of 20D to give almost fully developed flow conditions before the expansion. Several inlet conditions will be investigated, including a straight pipe, one 90° bend and two out-of-plane bends. A catalyst dummy will also be mounted and tested. For the catalyst dummy different model factors will be tested to insure the corrected pressure resistance. The distanced from the expansion to the dummy will also be varied and tested. Then different guide vane configurations will be mounted to investigate the flow uniformity at the catalyst converter. The investigation is done with Stereoscopic Particle Image Velocimetry (PIV). The measuring plane, a cross plane through the converter pipe, will be transverse along the flow direction (z-axis at figure 1). The cross plane is created with a 200 mJ Nd:YAG double cavity laser. Two 16 MPixel cameras are placed in forward and backward scatter, respectively. Glycerine droplets with a diameter of about 2 μm, are used as tracer particles.

Example results are shown in Figure 2, where the cross plane is placed 5D downstream of the expansion. Here the mean velocity field of 500 snapshots from the empty converter with a straight inlet shows that the flow consist of a fast 'jet' in the middle and negative velocity at the walls. A snapshot been selected to represent a very common flow structure corresponding to the first mode found from a snapshot proper orthogonal decomposition (POD) analysis [2]. The white line indicate the change from positive to negative velocity. In the snapshot, the 'jet' has spread along a line through the center and is in contact with two opposite walls. At the rest of the walls, a recirculation zone is seen. As seen in Figure 2, the wall region is well resolved except at the bottom where velocity vectors are missing due to optical reflections.
Hybrid vortex simulations of wind turbines using a three-dimensional viscous-inviscid panel method

A hybrid filament-mesh vortex method is proposed and validated to predict the aerodynamic performance of wind turbines and to simulate the resulting wake. Its novelty consists of using a hybrid method to accurately simulate the wakedownstream of the wind turbine while reducing the computational time used by the method. The proposed method uses any hybrid approach, where the near wake is resolved by using vortex filaments, which carry the vorticity shed by the trailingedge of the blades. The interaction of the vortex filaments in the near vicinity of the wind turbine is evaluated using a direct calculation, whereas the contribution from the large downstream wake is calculated using a mesh-based method. The hybrid method is first validated in detail against the well-known MEXICO experiment, using the direct filament method as a comparison. The second part of the validation includes a study of the influence of the time-integration scheme used for evolving the wake in time, aerelastic simulations of the National Renewable Energy Laboratory 5 MW wind turbine and an analysis of the central processing unit time showing the gains of using the hybrid filament-mesh method.

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Ramos García, N. (Intern), Hejlesen, M. M. (Intern), Sørensen, J. N. (Intern), Walther, J. H. (Intern)
Pages: 1871-1889
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Wind Energy
Volume: 20
Issue number: 11
ISSN (Print): 1095-4244
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.37 SJR 1.104 SNIP 2.306
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.196 SNIP 2.086 CiteScore 3.06
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.272 SNIP 3.75 CiteScore 3.42
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.275 SNIP 2.464 CiteScore 2.75
Microbial filter feeders are an important group of grazers, significant to the microbial loop, aquatic food webs, and biogeochemical cycling. Our understanding of microbial filter feeding is poor, and, importantly, it is unknown what force microbial filter feeders must generate to process adequate amounts of water. Also, the trade-off in the filter spacing remains unexplored, despite its simple formulation: A filter too coarse will allow suitably sized prey to pass unintercepted, whereas a filter too fine will cause strong flow resistance. We quantify the feeding flow of the filter-feeding choanoflagellate Diaphanoeca grandis using particle tracking, and demonstrate that the current understanding of microbial filter feeding is inconsistent with computational fluid dynamics (CFD) and analytical estimates. Both approaches underestimate observed filtration rates by more than an order of magnitude; the beating flagellum is simply unable to draw enough water through the fine filter. We find similar discrepancies for other choanoflagellate species, highlighting an apparent paradox. Our observations motivate us to suggest a radically different filtration mechanism that requires a flagellar vane (sheet), something notoriously difficult to visualize but sporadically observed in the related choanocytes (sponges). A CFD model with a flagellar vane correctly predicts the filtration rate of D. grandis, and using a simple model we can account for the filtration rates of other microbial filter feeders. We finally predict how optimum filter mesh size increases with cell size in microbial filter feeders, a prediction that accords very well with observations. We expect our results to be of significance for small-scale biophysics and trait-based ecological modeling.

Hydrodynamics of microbial filter feeding
Microbial filter feeders are an important group of grazers, significant to the microbial loop, aquatic food webs, and biogeochemical cycling. Our understanding of microbial filter feeding is poor, and, importantly, it is unknown what force microbial filter feeders must generate to process adequate amounts of water. Also, the trade-off in the filter spacing remains unexplored, despite its simple formulation: A filter too coarse will allow suitably sized prey to pass unintercepted, whereas a filter too fine will cause strong flow resistance. We quantify the feeding flow of the filter-feeding choanoflagellate Diaphanoeca grandis using particle tracking, and demonstrate that the current understanding of microbial filter feeding is inconsistent with computational fluid dynamics (CFD) and analytical estimates. Both approaches underestimate observed filtration rates by more than an order of magnitude; the beating flagellum is simply unable to draw enough water through the fine filter. We find similar discrepancies for other choanoflagellate species, highlighting an apparent paradox. Our observations motivate us to suggest a radically different filtration mechanism that requires a flagellar vane (sheet), something notoriously difficult to visualize but sporadically observed in the related choanocytes (sponges). A CFD model with a flagellar vane correctly predicts the filtration rate of D. grandis, and using a simple model we can account for the filtration rates of other microbial filter feeders. We finally predict how optimum filter mesh size increases with cell size in microbial filter feeders, a prediction that accords very well with observations. We expect our results to be of significance for small-scale biophysics and trait-based ecological modeling.
Impact mechanics of ship collisions and validations with experimental results

Closed-form analytical solutions for the energy released for deforming and crushing of structures and the impact impulse during ship collisions were developed and published in Marine Structures in 1998 [1]. The proposed mathematical models have been used by many engineers and researchers although the methods were only validated with time-domain numerical simulation results at that time. Since then, model and full-scale measurements have been carried out and experimental results are available in the public domain. The purpose of the present paper is to use such experimental results to further analyze the validity and robustness of the closed-form analytical methods as well as to further improve some parameters' accuracy. In total, 60 experimental results have been analyzed and compared with the analytical results and this paper presents the outcome. It can be concluded that the analytical methods give a reasonable agreement with the experimental results. The paper also introduces a simple concept to account for the effective mass of liquids with free surface carried on board of a ship and it is shown how the analytical analysis procedure can be expanded to take into account the effect of ship roll on the energy released for crushing.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register EMEA, Wuhan University
Authors: Zhang, S. (Ekstern), Villavicencio, R. (Ekstern), Zhu, L. (Ekstern), Pedersen, P. T. (Intern)
Pages: 69-81
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Marine Structures
Volume: 52
ISSN (Print): 0951-8339
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.49 SJR 1.655 SNIP 2.636
Web of Science (2016): Indexed yes
Implementation of Generalized Modes in a 3D Finite Difference Based Seakeeping Model

This work is an extension of the finite difference potential flow solver OceanWave3D-Seakeeping developed by Afshar (2014) to include generalized modes. The continuity equation is solved using a fourth-order centered finite difference scheme which requires that the entire fluid domain is discretized as opposed to the more popular panel method where only the body surface - and sometimes the free surface and sea bottom - are discretized. The advantage for the finite difference solver is thought to be found for complex or high-resolution problems where the computational time will scale better due to
Implementation of the far-field method for calculation of added resistance using a high order finite-difference approximation on overlapping grids

The far-field method for calculation of the wave drift force is implemented in the high order finite-difference seakeeping solver. The implementation is based on the Maruo formulation which employs the Kochin function to obtain the complex amplitude of the velocity potential in the far-field. The results are shown both for zero and forward speed for the floating hemisphere and two ship geometries. Comparisons with WAMIT and near-field calculations are also presented.

Improved HPC method for nonlinear wave tank

The recently developed Harmonic Polynomial Cell (HPC) method has been proved to be a promising choice for solving potential-flow Boundary Value Problem (BVP). In this paper, a flux method is proposed to consistently deal with the Neumann boundary condition of the original HPC method and enhance the accuracy. Moreover, fixed mesh algorithm with free surface immersed is developed to improve the computational efficiency. Finally, a two dimensional (2D) multi-block strategy coupling boundary-fitted mesh and fixed mesh is proposed. It limits the computational costs and preserves the accuracy. A fully nonlinear 2D numerical wave tank is developed using the improved HPC method as a verification.
Intact stability analysis of dead ship conditions using FORM

The IMO Weather Criterion has proven to be the governing stability criteria regarding minimum GM for e.g. small ferries and large passenger ships. The formulation of the Weather Criterion is based on some empirical relations derived many years ago for vessels not necessarily representative for current new buildings with large superstructures. Thus it seems reasonable to investigate the possibility of capsizing in beam sea under the joint action of waves and wind using direct time domain simulations. This has already been done in several studies. Here it is combined with the First Order Reliability Method (FORM) to define possible combined critical wave and wind scenarios leading to capsize and corresponding probability of capsize. The FORM results for a fictitious vessel are compared with Monte Carlo simulation and good agreement is found at a much lesser computational effort. Finally, the results for an existing small ferry will be discussed in the light of the current weather criterion.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register EMEA
Authors: Choi, J. H. (Intern), Jensen, J. J. (Intern), Kristensen, H. O. H. (Intern), Nielsen, U. D. (Intern), Erichsen, H. (Ekstern)
Pages: 167-176
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Ship Research
Volume: 61
Issue number: 3
ISSN (Print): 0022-4502
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 1.6 SJR 1.196 SNIP 1.832
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.585 SNIP 1.553 CiteScore 1.12
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.004 SNIP 1.835 CiteScore 1.19
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.576 SNIP 1.648 CiteScore 1.18
ISI indexed (2013): ISI indexed yes
Investigation of journal orbit and flow pattern in a dynamically loaded journal bearing

A hydrodynamic journal bearing has been investigated using both the traditional two-dimensional (2D) Reynolds equation, and the full solution being the three-dimensional (3D) Navier-Stokes equations.

The two approaches are compared by performing an investigation of two inlet groove designs: the axial and the circumferential groove, respectively, on a bearing with length-to-diameter ratio of 0.5 exposed to a sinusoidal load pattern. Pressure distributions, journal orbits and frictional losses are compared. The modelling of grooves by pressure boundary conditions versus geometric conditions is examined. It is investigated if the presence of a groove increases frictional losses and the increase relates to groove dimensions. Furthermore, the influence of the groove design on the flow field is studied using the 3D solution.
Iterative Brinkman penalization for simulation of impulsively started flow past a sphere and a circular disc

We present a Brinkman penalization method for three-dimensional (3D) flows using particle vortex methods, improving the existing technique by means of an iterative process. We perform simulations to study the impulsively started flow past a
sphere at $Re=1000$ and normal to a circular disc at $Re=500$. The simulation results obtained for the flow past a sphere are found in qualitative good agreement with previously published results obtained using respectively a 3D vortex penalization method and a 3D vortex method combined with an accurate boundary element method. From the results obtained for the flow normal to a circular disc it is found that the iterative method enables the use of a time step that is one order of magnitude larger than required by the standard non-iterative Brinkman penalization method.
Mass entrainment rate of an ideal momentum turbulent round jet

We propose a two-phase-fluid model for a full-cone turbulent round jet that describes its dynamics in a simple but comprehensive manner with only the apex angle of the cone being a disposable parameter. The basic assumptions are that (i) the jet is statistically stationary and that (ii) it can be approximated by a mixture of two fluids with their phases in dynamic equilibrium. To derive the model, we impose conservation of the initial volume and total momentum fluxes. Our model equations admit analytical solutions for the composite density and velocity of the two-phase fluid, both as functions of the distance from the nozzle, from which the dynamic pressure and the mass entrainment rate are calculated. Assuming a far-field approximation, we theoretically derive a constant entrainment rate coefficient solely in terms of the cone angle. Moreover, we carry out experiments for a single-phase turbulent air jet and show that the predictions of our model compare well with this and other experimental data of atomizing liquid jets.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Kyushu University
Authors: Medrano, F. F. (Ekstern), Fukumoto, Y. (Ekstern), Velte, C. M. (Intern), Hodzic, A. (Intern)
Number of pages: 10
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Physical Society of Japan. Journal
Volume: 86
Article number: 034401
ISSN (Print): 0031-9015
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 1.1 SJR 0.715 SNIP 0.602
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.719 SNIP 0.649 CiteScore 1.06
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.915 SNIP 0.607 CiteScore 1.04
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.807 SNIP 0.422 CiteScore 0.85
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.353 SNIP 0.733 CiteScore 1.39
A novel exact temporal to spatial mapping for point measurements in turbulence has been developed. The spatial record is obtained based on the instantaneous velocity magnitude, $u = \|u\|$, creating an exact mapping between the sampling interval, $\Delta t$, and the spatial record counterpart, $\Delta s$, through the relation $\Delta s_n = u_n \Delta t_n$. $n$ indicates the sample number in a measurement sequence. Summation of the consecutive streakline elements, $\Delta s$, corresponding to the convection distance of the fluid, results in a spatial "convection record". The exact mapping applies to all flows, since it is based on the instantaneous velocity magnitude, thereby incorporating all relevant aspects of the flow dynamics. Even high intensity non-equilibrium spatial records can be measured using this mapping, which is most straightforwardly applied using laser Doppler anemometry measurements. Computer simulated high intensity LDA data demonstrate the technique. The method will also be demonstrated on measurements in a round turbulent jet in part 2.

**Measurement of turbulent kinetic energy spectrum - Part 1: Convection record method**

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Intarsia Optics
Authors: Buchhave, P. (Ekstern), Velte, C. M. (Intern)
Number of pages: 7
Pages: 163-169
Publication date: 2017

**Host publication information**

Publisher: Springer
Editors: Peinke, J., Kampers, G., Oberlack, M., Waclawczyk, M., Talamelli, A.
ISBN (Print): 978-3-319-57933-7
ISBN (Electronic): 978-3-319-57934-4

Series: Springer Proceedings in Physics
ISSN: 0930-8989
Measurement of turbulent kinetic energy spectrum - Part 2: Convection record measurements

A novel exact temporal to spatial mapping for point measurements in turbulence has been applied to various flow conditions existing in a round turbulent jet. The conditions range between equilibrium and non-equilibrium as well as mid to high turbulence intensities. The exact mapping applies to all flows, including high intensity non-equilibrium flows, since it is based on the instantaneous velocity magnitude, thereby incorporating all relevant aspects of the flow dynamics. Development of the jet turbulence along the stream, from non-equilibrium to equilibrium, is observed. In the developed region of the jet, Taylor’s hypothesis is tested and the spectra using the novel exact mapping is validated with excellent agreement against directly measured spatial spectra in a mapped similarity space using PIV. The method is observed to produce the expected results even at turbulence intensities of the order of 450%.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Intarsia Optics
Authors: Velte, C. M. (Intern), Buchhave, P. (Ekstern), Hodzic, A. (Intern)
Number of pages: 6
Pages: 171-176
Publication date: 2017

Host publication information
Publisher: Springer
Editors: Peinke, J., Kampers, G., Oberlack, M., Waclawczyk, M., Talamelli, A.
ISBN (Print): 978-3-319-57933-7
ISBN (Electronic): 978-3-319-57934-4
Series: Springer Proceedings in Physics
ISSN: 0930-8989
Main Research Area: Technical/natural sciences
DOIs:
10.1007/978-3-319-57934-4_23

Measurement of Turbulent Skin Friction Drag Coefficients Produced by Distributed Surface Roughness of Pristine Marine Coatings

Skin friction drag coefficients are determined for marine antifouling coatings in pristine condition by use of Constant Temperature Anemometry (CTA) with uni-directional hot-wires. Mean flow behaviour for varying surface roughness is analysed in zero pressure gradient, flat plate, turbulent boundary layers for Reynolds numbers from $R_{ex} = 1.91 \times 10^5$ to $R_{ex} = 9.54 \times 10^5$. The measurements were conducted at the Technical University of Denmark in a closed-loop wind tunnel redesigned for investigations as this. Ensemble averages of the boundary layer velocity profiles allowed for determination of skin friction drag coefficients for practically smooth surfaces at low Reynolds numbers compared to normal operation mode for the antifouling coatings. Thus, better estimates for skin friction of rough hulls can be realised using the proposed method to optimise preliminary vessel design.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Technical University of Denmark, Dantec Dynamics A/S
Authors: Zafiryadis, F. (Ekstern), Meyer, K. E. (Intern), Gökhan Ergin, F. (Ekstern)
Number of pages: 6
Publication date: 2017
Event: Paper presented at Tenth International Symposium on Turbulence and Shear Flow Phenomena (TSFP10), Chicago, United States.
Measurement of turbulent spatial structure and kinetic energy spectrum by exact temporal-to-spatial mapping

We present a method for converting a time record of turbulent velocity measured at a point in a flow to a spatial velocity record consisting of consecutive convection elements. The spatial record allows computation of dynamic statistical moments such as turbulent kinetic wavenumber spectra and spatial structure functions in a way that completely bypasses the need for Taylor’s hypothesis. The spatial statistics agree with the classical counterparts, such as the total kinetic energy spectrum, at least for spatial extents up to the Taylor microscale. The requirements for applying the method are access to the instantaneous velocity magnitude, in addition to the desired flow quantity, and a high temporal resolution in comparison to the relevant time scales of the flow. We map, without distortion and bias, notoriously difficult developing turbulent high intensity flows using three main aspects that distinguish these measurements from previous work in the field: (1) The measurements are conducted using laser Doppler anemometry and are therefore not contaminated by directional ambiguity (in contrast to, e.g., frequently employed hot-wire anemometers); (2) the measurement data are extracted using a correctly and transparently functioning processor and are analysed using methods derived from first principles to provide unbiased estimates of the velocity statistics; (3) the exact mapping proposed herein has been applied to the high turbulence intensity flows investigated to avoid the significant distortions caused by Taylor’s hypothesis. The method is first confirmed to produce the correct statistics using computer simulations and later applied to measurements in some of the most difficult regions of a round turbulent jet—the non-equilibrium developing region and the outermost parts of the developed jet. The proposed mapping is successfully validated using corresponding directly measured spatial statistics in the fully developed jet, even in the difficult outer regions of the jet where the average convection velocity is negligible and turbulence intensities increase dramatically. The measurements in the developing region reveal interesting features of an incomplete Richardson-Kolmogorov cascade under development.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Intarsia Optics
Authors: Buchhave, P. (Ekstern), Velte, C. M. (Intern)
Number of pages: 11
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Physics of Fluids
Volume: 29
Article number: 085109
ISSN (Print): 1070-6631
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.16 SJR 1.29 SNIP 1.291
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.366 SNIP 1.278
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.354 SNIP 1.348
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.42 SNIP 1.395
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.215 SNIP 1.356
Modelling of temporal and spatial evolution of sulphur oxides and sulphuric acid under large, two-stroke marine engine-like conditions using integrated CFD-chemical kinetics

In this work, three-dimensional computational fluid dynamics (CFD) studies of sulphur oxides (SOx) and sulphuric acid (H₂SO₄) formation processes in a large, low speed two-stroke marine diesel engine are carried out. The current numerical study aims to investigate the conversion of sulphur dioxide (SO₂) to sulphur trioxide (SO₃) and the possibility of H₂SO₄ condensation which are the prerequisites to better understand the corrosion-induced wear phenomenon. This is achieved with the aid of the implementation of a multicomponent surrogate model, which comprises a skeletal n-heptane mechanism and a reduced sulphur subset mechanism. In the present work, performance of the coupled CFD-chemical kinetic model is evaluated using both qualitative and quantitative methods. The modelling results show that the temporal and spatial evolutions of SOx predicted by the skeletal model are similar to those by the base mechanism. Predictions of the variations of SOx and the associated SO₂ to SO₃ conversion in response to the change of fuel sulphur content, swirl velocity, start of injection, scavenging pressure and humidity qualitatively agree with numerical and experimental results from the literature. The model is further evaluated using the measured SO₂ to SO₃ conversion levels in a low load, low scavenging pressure case and a low load, high scavenging pressure case. The absolute values of simulated and measured conversion levels are close, although the former appear to be higher. The current results show that the flame impingement at the cylinder liner near top dead centre. The gas is cooled rapidly by the wall temperature and H₂SO₄ is produced in the region where the local temperature is less than 600 K. Based on the fluegas correlation, the acid dew point temperature is higher than the wall temperature, suggesting that acid condensation may begin early at the top part of the cylinder liner. The predicted distribution corresponds well with the distribution of corroded parts observed in service engines. The model is expected to serve as an important tool to simulate the rates of SO₂ absorption into lubricating oil film and H₂SO₄.
condensation in this combustion system.

**General information**

**State**: Published  
**Organisations**: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy, Department of Chemical and Biochemical Engineering, CHEC Research Centre, MAN Diesel & Turbo SE  
**Authors**: Pang, K. M. (Intern), Karvounis, N. (Intern), Walther, J. H. (Intern), Schramm, J. (Intern), Glarborg, P. (Intern), Mayer, S. (Ekstern)

**Pages**: 60-73  
**Publication date**: 2017  
**Main Research Area**: Technical/natural sciences

**Publication information**

**Journal**: Applied Energy  
**Volume**: 193  
**ISSN (Print)**: 0306-2619  
**Ratings**:  
- **BFI (2018)**: BFI-level 2  
- **Web of Science (2018)**: Indexed yes  
- **BFI (2017)**: BFI-level 2  
- **Web of Science (2017)**: Indexed yes  
- **BFI (2016)**: BFI-level 2  
- **Scopus rating (2016)**: CiteScore 7.78 SJR 3.058 SNIP 2.573  
- **Web of Science (2016)**: Indexed yes  
- **BFI (2015)**: BFI-level 2  
- **Scopus rating (2015)**: SJR 2.912 SNIP 2.61 CiteScore 6.4  
- **Web of Science (2015)**: Indexed yes  
- **BFI (2014)**: BFI-level 2  
- **Scopus rating (2014)**: SJR 3.254 SNIP 3.28 CiteScore 6.93  
- **Web of Science (2014)**: Indexed yes  
- **BFI (2013)**: BFI-level 1  
- **Scopus rating (2013)**: SJR 3.164 SNIP 3.377 CiteScore 6.59  
- **ISI indexed (2013)**: ISI indexed yes  
- **Web of Science (2013)**: Indexed yes  
- **BFI (2012)**: BFI-level 1  
- **Scopus rating (2012)**: SJR 2.854 SNIP 3.108 CiteScore 5.69  
- **ISI indexed (2012)**: ISI indexed yes  
- **Web of Science (2012)**: Indexed yes  
- **BFI (2011)**: BFI-level 1  
- **Scopus rating (2011)**: SJR 2.473 SNIP 2.84 CiteScore 5.5  
- **ISI indexed (2011)**: ISI indexed yes  
- **Web of Science (2011)**: Indexed yes  
- **BFI (2010)**: BFI-level 1  
- **Scopus rating (2010)**: SJR 1.516 SNIP 2.25  
- **Web of Science (2010)**: Indexed yes  
- **BFI (2009)**: BFI-level 1  
- **Scopus rating (2009)**: SJR 1.003 SNIP 1.781  
- **Web of Science (2009)**: Indexed yes  
- **BFI (2008)**: BFI-level 2  
- **Scopus rating (2008)**: SJR 0.974 SNIP 1.215  
- **Web of Science (2008)**: Indexed yes  
- **Scopus rating (2007)**: SJR 1.179 SNIP 1.709  
- **Web of Science (2007)**: Indexed yes  
- **Scopus rating (2006)**: SJR 0.979 SNIP 1.293  
- **Scopus rating (2005)**: SJR 1.043 SNIP 0.996  
- **Web of Science (2005)**: Indexed yes
Nominal vs. Effective Wake Fields and their Influence on Propeller Cavitation Performance

Propeller designers often need to base their design on the nominal model scale wake distribution, because the effective full scale distribution is not available. The effects of such incomplete design data on cavitation performance is examined in this paper. The behind-ship cavitation performance of two propellers is evaluated, where the cases considered include propellers operating in the nominal model and full scale wake distributions and in the effective wake distribution, also in model and full scale. The method for the analyses is a combination of RANS for the ship hull and a panel method for the propeller flow, with a coupling of the two for the interaction of ship and propeller flows. The effect on sheet cavitation due to the different wake distributions is examined for a typical full-form ship. Results show considerable differences in cavitation extent, volume, and hull pressure pulses.

Numerical modelling of the erosion and deposition of sand inside a filter layer

This paper treats the numerical modelling of the behaviour of a sand core covered by rocks and exposed to waves. The associated displacement of the rock is also studied. A design that allows for erosion and deposition of the sand core beneath a rock layer in a coastal structure requires an accurate prediction method to assure that the amount of erosion remains within acceptable limits. This work presents a numerical model that is capable of describing the erosion and deposition patterns inside of an open filter of rock on top of sand. The hydraulic loading is that of incident irregular waves and the open filters are surface piercing. Due to the few experimental data sets on sediment transport inside of rock layers, a sediment transport formulation has been proposed based on a matching between the numerical model and experimental data on the profile deformation inside an open filter. The rock layer on top of a sand core introduces a correction term in the Exner equation (the continuity equation for sediment and change in bed level). The correction term originates from the fact that the sand can only be deposited in the pores of the filter material. The numerical model is validated against additional data sets on the erosion and deposition patterns inside of an open filter. A few cases are defined to study the effect of the sinking of the filter into the erosion hole. The numerical model is also applied to several application cases. The response of the core material (sand) to changes in the wave period and wave height is considered. The effect of different layouts of the filter is studied in order to investigate the effect of different filter profiles on the resulting erosion. Finally, it is studied how much the design of a hydraulically closed filter can be relaxed to obtain a reduction in the design requirements of the filter thickness, while the deformation to the sand core remains acceptably small.

General information
Numerical Simulation of Methane Slip in Dual Fuel Marine Engines

The methane slip is the problematic issue for the engines using natural gas (NG). Because methane is more powerful greenhouse gas (GHG) than CO2, understanding of the methane slip during gas exchange process of the engines is essential. In this study, the influence of the gas pipe geometry and the valve timings on the methane slip was investigated. MAN L28/32DF engine was modeled to simulate the gas exchange process of the four stroke NG-diesel dual fuel engines. The mesh size of the model was decided based on the sensitivity study on the peak pressure of the cylinder and the fuel mass estimations. The simulations with various gas pipe geometries were conducted. It seemed that the effect of the change in injection direction is more dominant than the change in the gas hole configuration. The favorable injection direction for minimum amount of methane slip was discovered as the direction which helps developing the flow of methane far from the exhaust ports. The effects of various valve timing settings were also simulated. The advancement of the exhaust valve closing was more efficient than the retardation of the intake valve opening. A little retardation of the intake valve opening even resulted in the increase of the amount of methane slip.

General information
State: Published
Organisations: Department of Mechanical Engineering, Thermal Energy, Fluid Mechanics, Coastal and Maritime Engineering, Korean Advanced Institute of Science and Technology (KAIST)
Authors: Han, J. (Ekstern), Jensen, M. V. (Intern), Pang, K. M. (Intern), Walther, J. H. (Intern), Schramm, J. (Intern), Bae, C. (Ekstern)
Pages: 1475-1480
Publication date: 2017

Host publication information
Title of host publication: 제한기계학회 춘추학술대회
Publisher: The Korean Society of Mechanical Engineers
Main Research Area: Technical/natural sciences
Conference: 2017 KSME Annual Meeting, Jeju, Korea, Republic of, 01/11/2017 - 01/11/2017
Methane Slip, CFD, 4 Stroke dual fuel marine engine, Natural gas, Gas pipe optimization, Valve timing, Un-burned methane emission
Source: PublicationPreSubmission
Source-ID: 141043842
Publication: Research - peer-review › Article in proceedings – Annual report year: 2017

Numerical simulation of scour and backfilling processes around a circular pile in waves

This study continues the investigation of flow and scour around a vertical pile, reported by Roulund et al. (2005). Flow and scour/backfilling around a vertical pile exposed to waves are investigated by using a threedimensional numerical model based on incompressible Reynolds averaged Navier–Stokes equations. The model incorporates (1) k-ω turbulence closure, (2) vortex shedding processes, (3) sediment transport (both bed and suspended load), as well as (4) bed morphology. The numerical simulations are carried out for a selected set of test conditions of the laboratory experiments of Sumer et al. (1997, 2013a), and the numerical results are compared with those of the latter experiments. The simulations are carried out for two kinds of beds: rigid bed and sediment bed. The rigid-bed simulations indicate that the vortex shedding for waves around the pile occurs in a “one-cell” fashion with a uniform shedding frequency over the height of the cylinder, unlike the case for steady current where a two-cell structure prevails. The rigid-bed simulations further show that
the horseshoe vortex flow also undergoes substantial changes in waves. The amplification of the bed shear stress around the pile (including the areas under the horseshoe vortex and the lee wake region) is obtained for various values of the Keulegan-Carpenter number, the principal parameter governing the flow around the pile in waves. The present model incorporated with the morphology component is applied to several scenarios of scour and backfilling around a pile exposed to waves. In the backfilling simulations, the initial scour hole is generated either by a steady current or by waves. The present simulations indicate that the scour and backfilling in waves are solely governed by the lee-wake flow, in agreement with observations. The numerical model has proven successful in predicting the backfilling of scour holes exposed to waves. The results of the numerical tests indicate that the equilibrium depth of scour holes is the same for both the scour and the backfilling for a given Keulegan-Carpenter number, in full agreement with observations.

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Deltares
Authors: Baykal, C. (Intern), Sumer, B. M. (Intern), Fuhrman, D. R. (Intern), Jacobsen, N. G. (Ekstern), Fredsøe, J. (Intern)
Pages: 87-107
Publication date: 2017
Main Research Area: Technical/natural sciences

**Publication information**
Journal: Coastal Engineering
Volume: 122
ISSN (Print): 0378-3839
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.925 SNIP 2.097 CiteScore 2.9
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.785 SNIP 2.123 CiteScore 2.55
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.727 SNIP 2.264 CiteScore 2.58
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.912 SNIP 2.226 CiteScore 2.21
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.616 SNIP 2.502 CiteScore 2.43
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 1.898 SNIP 2.332
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 2.067 SNIP 2.454
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 2
Scopus rating (2008): SJR 1.189 SNIP 2.166
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 1.642 SNIP 2.164
Web of Science (2007): Indexed yes
On phonons and water flow enhancement in carbon nanotubes

The intriguing physics of water transport through carbon nanotubes (CNTs) has motivated numerous studies, reporting flow rates higher than those estimated by continuum models. The quantification of water transport in CNTs remains unresolved, however, with flow rates reported by different experiments and simulations having discrepancies of over three orders of magnitude. Reports of ultrafast and more modest rates conflict with each other. Molecular dynamics (MD) simulations have been used to resolve this puzzle by helping to decipher how the CNT walls interact with water molecules.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Swiss Federal Institute of Technology, National Institute of Chemistry
Authors: Cruz-Chu, E. R. (Ekstern), Papadopoulou, E. (Ekstern), Walther, J. H. (Intern), Popadić, A. (Ekstern), Li, G. (Ekstern), Praprotnik, M. (Ekstern), Koumoutsakos, P. (Ekstern)
Pages: 1106-1107
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Nature Nanotechnology
Volume: 12
ISSN (Print): 1748-3387
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 21.85 SJR 18.746 SNIP 8.299
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 19.08 SNIP 8.617 CiteScore 22.1
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 17.133 SNIP 8.221 CiteScore 21.76
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Performance of a Tilt Current Meter in the Surf Zone

Tilt Current Meters (TCM’s) are relatively simple and inexpensive instruments for measuring currents in rivers and in the sea. Their low cost and easy deployment means that a relatively large number of TCM’s can be deployed compared to more conventional current meters such as Acoustic Doppler Velocimeters (ADV’s) or Acoustic Doppler Current Profiler (ADCP’s). Although, the accuracy of the individual measurements may not be as good as conventional currentmeters, the possibility of deploying many instruments is a great advantage when studying spatial variations in flows. This is especially the case when data is later used for comparison with numerical models whose results are also associated with considerable uncertainty. Previous studies have mainly considered steady current or tidal flows in which velocities were relatively low and the importance of waves limited. The presence of waves adds a number of important challenges to the measurements as the hydrodynamic forcing changes and the oscillations of the TCM cannot necessarily be averaged out as for a steady current. This study addresses some of these challenges by analyzing the performance of a TCM in the surf zone where wave orbital motion is dominant.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Københavns Universitet
Authors: Hansen, A. B. (Intern), Carstensen, S. (Intern), Christensen, D. F. (Forskerdatabase), Aagaard, T. (Forskerdatabase)
Number of pages: 11
Publication date: 2017
Main Research Area: Technical/natural sciences
Coastal hydrodynamics, Surf zone, Observational techniques
Electronic versions: 218_Hansen_etal.pdf
Publication: Research - peer-review › Paper – Annual report year: 2017

PIV measurements of breaking waves

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
PIV measurements of velocities and accelerations under breaking waves on a slope

Understanding the physics of breaking waves is an ongoing research topic, not only due to human curiosity, but also due to the influence breaking waves have on offshore structures. In recent years, the development in experimental methods has facilitated a new insight into the physics of breaking waves. In this study, we have investigated the wave kinematics under steep and breaking waves on a laboratory beach with a slope of 1/25. The velocity field was measured by use of Particle Image Velocimetry (PIV) at a sample rate of 96Hz. The high sample rate allowed for the accelerations to be determined directly from the sampled velocities. It was found that both velocities and accelerations differ from the ones predicted from common wave theories such as streamfunction theory. This was especially evident at the top part of the wave close to the surface. This was not surprising, since the breaking event is a highly non-linear process. The results presented here may facilitate computations of the impact force on offshore structures and furthermore be used for validation of CFD models while altogether shedding light on the mechanisms behind breaking waves.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Vested, M. H. (Intern), Carstensen, S. (Intern), Christensen, E. D. (Intern)
Number of pages: 1
Publication date: 2017
Conference: EGU General Assembly 2017, Vienna, Austria, 24/04/2017 - 24/04/2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Geophysical Research Abstracts
Volume: 19
Article number: 7837
ISSN (Print): 1607-7962
Ratings:
Web of Science (2014): Indexed yes
ISI indexed (2013): ISI indexed no
Web of Science (2013): Indexed yes
ISI indexed (2012): ISI indexed no
Web of Science (2012): Indexed yes
ISI indexed (2011): ISI indexed no
Web of Science (2011): Indexed yes
BFI (2009): BFI-level 1
Original language: English
Electronic versions:
EGU2017_7837.pdf

Relations
Activities:
Oral presentation
Publication: Research - peer-review › Conference abstract in journal – Annual report year: 2017

POD Mode Robustness for the Turbulent Jet Sampled with PIV
An important challenge in the description and simulation of turbulence is the large amount of information that is needed to describe even relatively simple flows in detail. The frequent disagreement between Reynolds averaged Navier–Stokes-based simulations and experiments is well known. Albeit, direct numerical simulations and in certain cases large eddy simulations tend to agree fairly well with experiments, their practical implementation introduces the problem of data storage. The experimentalist, however, experiences the same problem, using highspeed particle image velocimetry (PIV)
systems and even high speed volumetric PIV systems providing fully three dimensional velocity fields. Another challenge is how do we verify simulations against experiments and ensure that we indeed have simulated the same flow that we have measured?

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Hodzic, A. (Intern), Meyer, K. E. (Intern), Velté, C. M. (Intern)
Pages: 163-188
Publication date: 2017

**Host publication information**
Title of host publication: Whither Turbulence and Big Data in the 21st Century?
Publisher: Springer
Editors: Pollard, A., Castillo, L., Danaila, L., Glauser, M.
ISBN (Print): 978-3-319-41215-3
ISBN (Electronic): 978-3-319-41217-7
Chapter: 10
Main Research Area: Technical/natural sciences
DOIs: 10.1007/978-3-319-41217-7_10
Publication: Research - peer-review › Book chapter – Annual report year: 2016

**Pore Pressure Under A Gravity Based Structure Under The Influence Of Waves**
The total wave load on a gravity based foundation for offshore wind turbines is influenced by the pore pressure from beneath the structure. The pore pressure is induced by the wave-structure-seabed interaction. Often the uplift force is included in a simplified way in the design of the gravity based foundation. This leads typically to very conservative designs in order to accommodate the uncertainties in the procedure. The experiments shall lead to better prediction models based on for instance CFD model’s with the direct calculation of pressure variations in the seabed and any erosion protection layer. Herewith, it will be possible to get a direct assessment of wave loads on the foundation, also under the seabed level. The study includes experiments as well as numerical analyses. A good agreement between the experimental results and the numerical analyses was found. In the numerical analyses, it was possible to investigate the effect of air content in the pores, which turned out to have an effect on the distribution of the pore pressure.

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, NIRAS A/S, Rambøll Danmark A/S
Authors: Christensen, E. D. (Intern), Carstensen, S. (Intern), Madsen, M. T. (Ekstern), Allerød Hesselbjerg, P. (Ekstern), Nielsen, C. J. (Ekstern)
Number of pages: 10
Publication date: 2017

**Host publication information**
Title of host publication: Proceedings of the 36st International Conference on Ocean, Offshore, and Arctic Engineering (OMAE36)
Publisher: American Society of Mechanical Engineers
Article number: OMAE2017-62585
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: 36st International Conference on Ocean, Offshore, and Arctic Engineering (OMAE36), Trondheim, Norway, 25/06/2017 - 25/06/2017
DOIs: 10.1115/OMAE2017-62585
Publication: Research - peer-review › Article in proceedings – Annual report year: 2017

**Porous Media and Immersed Boundary Hybrid-Modelling for Simulating Flow in Stone Cover-Layers**
In this paper we present a new numerical modelling approach for coastal and marine applications where a porous media conceptual model was combined with a free surface volume-of-fluid (VOF) model and an immersed boundary method (IBM). The immersed boundary model covers the method of describing a solid object in a simple computational mesh without resolving the object with a conventional body-fitted mesh. This model enables a detailed resolution of some parts of a stone cover layer for erosion protection with the IBM model while other parts are handled with the conceptual porosity model. In this paper, the model is applied to investigate two practical cases in terms of a cover layer of stones on a flat
bed under oscillatory flow at different packing densities, and a rock toe structure at a breakwater.

**Rans-Based Numerical Simulation of Wave-Induced Sheet-Flow Transport of Graded Sediments**

An existing one-dimensional vertical (1DV) turbulence-closure flow model, coupled with sediment transport capabilities, is extended to incorporate graded sediment mixtures. The hydrodynamic model solves the horizontal component of the incompressible Reynolds-averaged Navier–Stokes (RANS) equations coupled with k–ω turbulence closure. In addition to standard bed and suspended load descriptions, the sediment transport model incorporates so-called high-concentration effects (turbulence damping and hindered settling velocities). The sediment transport model treats the bed and suspended load individually for each grain fraction within a mixture, and includes effects associated with increased exposure of larger particles within a mixture. The model also makes use of a modified reference concentration approach, with reference concentrations computed individually for each fraction, and then translated to a common level, which conveniently enables use of a single computational grid for the simulation of suspended sediments. Parametric study shows that these effects combine to help alleviate an otherwise systematic tendency towards over- and under-predicted transport rates for fine and coarse sand fractions, respectively. The sediment transport model is validated against experimental sheet-flow measurements conducted in oscillatory tunnels beneath velocity-skewed wave signals, and demonstrates similar accuracy (predicted transport rates generally within a factor of two of measurements) for both graded mixtures and uniform sands.

**RANS-based simulation of wave-induced sheet-flow transport of graded sediments**

A one-dimensional vertical (1DV) turbulence-closure flow model, coupled with sediment transport capabilities, is extended to incorporate graded sediment mixtures. The hydrodynamic model solves the horizontal component of the incompressible Reynolds-averaged Navier–Stokes (RANS) equations coupled with k–ω turbulence closure. The sediment transport description includes both bed and suspended load descriptions. So-called high-concentration effects (turbulence damping and hindered settling velocities) are likewise included. The sediment transport model treats the bed and suspended load individually for each grain fraction, including effects associated with increased exposure of larger particles within a mixture. The suspended sediment transport model also makes use of modified reference concentration approach, wherein reference concentration computations individually for each fraction are translated to a common level, conveniently enabling use of a single computational grid for the simulation of suspended sediments. Parametric study shows that these two effects combine to help alleviate an otherwise systematic tendency towards over- (under-) predicted transport rates for fine (coarse) sand fractions. The sediment transport model is validated against sheet-flow experimental oscillatory tunnel measurements beneath velocity-skewed wave signals, and demonstrates similar accuracy (transport rates generally within a factor of two) for both graded and uniform sands. The model is likewise validated against an extensive data set involving sheet-flow transport beneath acceleration-skewed wave signals (limited to uniform sands). It is then utilized to study potential effects of gradation on the net transport beneath such flows. The simulations suggest that gradation effects can both increase, as well as decrease, the total transport rate, depending largely on the behavior of the fine sand fraction. The model is implemented within the Matlab environment, and is freely available upon request to the corresponding author.
Sample Applications of the Second Generation Intact Stability Criteria – Robustness and Consistency Analysis

A new Intact Stability Code, the so-called Second Generation of Intact Stability Criteria, is currently under development and validation by the International Maritime Organization (IMO). The criteria are separated into five failure modes, each of which is analyzed by two vulnerability levels and, if needed, a direct numerical simulation. The present paper summarizes results testing the vulnerability levels in these new stability criteria. The calculations are carried out for 17 ships using the full matrix of operational draughts, trims and GM values. Each failure mode criterion is examined individually regarding construction of a GM limit curve for the full range of operational draughts. The consistency of the outcomes has been analyzed, and finally examined whether the new criteria tend to be more or less conservative compared to the present rules by evaluating approved loading conditions.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Knud E. Hansen A/S, HOK Marineconsult ApS, OSK-ShipTech, Odense Maritime Technology A/S, DFDS, University of Southern Denmark, Lloyd’s Register EMEA
Authors: Schrøter, C. (Ekstern), Lützen, M. (Forskerdatabase), Erichsen, H. (Ekstern), Jensen, J. J. (Intern), Kristensen, H. O. (Ekstern), Hagelskjær Lauridsen, P. (Ekstern), Tunccan, O. (Forskerdatabase), Balsersen, J. P. (Ekstern)
Pages: 9-13
Publication date: 2017

Host publication information

Title of host publication: Proceedings of the 16th International Ship Stability Workshop
Main Research Area: Technical/natural sciences
Conference: 16th International Ship Stability Workshop, Belgrade, Serbia, 05/06/2017 - 05/06/2017
IMO, Second generation intact stability criteria, Sample calculations, GM limit curves
Electronic versions:
4_94_Danish_Sample_Ship_Calculation_2016.IMO.pdf
Publication: Research - peer-review • Article in proceedings – Annual report year: 2017

Slip divergence of water flow in graphene nanochannels: the role of chirality

Graphene has attracted considerable attention due to its characteristics as a 2D material and its fascinating properties, providing a potential building block for nanofabrication. In nanochannels the solid-liquid interface plays a non-negligible role in determining the fluid dynamics. Therefore, for an optimal design of nanofluidic devices, a comprehensive understanding of the slippage in a water flow confined between graphene walls is important. In nanoconfinement, experimental and computational studies have found the slip length to increase nonlinearly when the shear rate is larger than a critical value. Here, by conducting molecular dynamics simulations, we study the influence of the graphene crystallographic orientation on the slip boundary conditions inside a nanoslit channel. The flow in channels with heights of 2.0, 2.4 and 2.8 nm is driven parallel to the zig-zag and arm-chair crystallographic directions. We extract flow rates, velocity profiles, slip velocities and slip lengths. The slip velocity displays a linear relationship to the shear stress up to a critical value, which is not size dependent. Moreover, the slip length is found to be shear stress dependent above a critical shear stress value of 0.4 MPa. Furthermore, our results indicate that after this critical shear stress is reached, the flow rates are significantly influenced (up to 10%) by the particular orientation of the graphene topology.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Wagemann, E. (Ekstern), Oyarzua, E. (Ekstern), Walther, J. H. (Intern), Zambrano, H. A. (Ekstern)
Solving the linearized forward-speed radiation problem using a high-order finite difference method on overlapping grids

The linearized potential flow approximation for the forward speed radiation problem is solved in the time domain using a high-order finite difference method. The finite-difference discretization is developed on overlapping, curvilinear body-fitted grids. To ensure numerical stability, the convective derivatives in the free-surface boundary conditions are treated using an upwind-biased stencil. Instead of solving for the radiation impulse response functions, a pseudo-impulsive Gaussian type displacement is employed in order to tailor the frequency-content to the discrete spatial resolution. Frequency-domain results are then obtained from a Fourier transform of the force and motion signals. In order to make a robust Fourier transform, and capture the response around the critical frequency, the tail of the force signal is asymptotically extrapolated assuming a linear decay rate. Fourth-order convergence of the calculations on simple geometries is demonstrated, along with a nearly linear scaling of the solution effort with increasing grid resolution. The code is validated by comparison with analytical and semi-analytical solutions using submerged and floating closed-form geometries. Calculations are also made for a modern bulk carrier, and good agreement is found with experimental measurements.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Amini Afshar, M. (Intern), Bingham, H. B. (Intern)
Pages: 220–244
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Applied Ocean Research
Volume: 69
ISSN (Print): 0141-1187
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.16 SJR 1.202 SNIP 1.75
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.245 SNIP 1.999 CiteScore 2.1
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.442 SNIP 2.244 CiteScore 1.91
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.243 SNIP 2.332 CiteScore 1.84
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.234 SNIP 1.922 CiteScore 1.46
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.496 SNIP 1.975 CiteScore 1.73
Statistical prediction of parametric roll using FORM

Previous research has shown that the First Order Reliability Method (FORM) can be an efficient method for estimation of outcrossing rates and extreme value statistics for stationary stochastic processes. This is so also for bifurcation type of processes like parametric roll of ships. The present paper discusses this solution procedure with a focus on the computational efficiency of FORM as compared with Monte Carlo Simulation (MCS).

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, J. J. (Intern), Choi, J. (Intern), Nielsen, U. D. (Intern)
Pages: 235–242
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Ocean Engineering
Volume: 144
ISSN (Print): 0029-8018
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.46 SJR 1.315 SNIP 2.014
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.172 SNIP 1.989 CiteScore 2.19
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.252 SNIP 2.323 CiteScore 2.11
Web of Science (2014): Indexed yes
Suspended particle transport through constriction channel with Brownian motion

It is well known that translocation events of a polymer or rod through pores or narrower parts of micro- and nanochannels have a stochastic nature due to the Brownian motion. However, it is not clear whether the objects of interest need to have a larger size than the entrance to exhibit the deviation from the dynamics of the surrounding fluid. We show by numerical analysis that the particle injection into the narrower part of the channel is affected by thermal fluctuation, where the particles have spherical symmetry and are smaller than the height of the constriction. The Péclet number (Pe) is the order parameter that governs the phenomena, which clarifies the spatio-temporal significance of Brownian motion compared to hydrodynamics. Furthermore, we find that there exists an optimal condition of Pe to attain the highest flow rate of particles relative to the dispersant fluid flow. Our finding is important in science and technology from nanopore DNA sequencers and lab-on-a-chip devices to filtration by porous materials and chromatography.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Tokyo University of Agriculture and Technology
Authors: Hanasaki, I. (Ekstern), Walther, J. H. (Intern)
Number of pages: 8
Publication date: 2017
The collapse of Tacoma Narrows Bridge: a piece to the puzzle
On Nov. 7th 1940 the newly constructed Tacoma Narrows Bridge collapsed due to excessive torsional oscillations caused by the formation and shedding of large coherent vortices. The subsequent wind tunnel tests conducted on both section- and full bridge models concluded that the bridge should have collapsed at a wind speed corresponding to approximately half of the wind speed at the day of the collapse. This discrepancy questions our understanding of the phenomena responsible for the failure of the bridge. The present study aims at clarifying this "mystery" by considering historical records made available by the US coast guards, and by performing wind tunnel tests and detailed numerical ow simulations. Our findings indicate that the discrepancy is caused by an until now unnoticed yawed wind direction relative to the bridge, which was present at the day of the collapse.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Technical University of Denmark, COWI A/S, FORCE Technology
Authors: Walther, J. H. (Intern), Christensen, D. S. (Ekstern), Malthe, M. G. (Ekstern), Roenne, M. (Ekstern), Spietz, H. J. (Intern), Larsen, A. (Ekstern), Larsen, S. V. (Ekstern)
Number of pages: 1
Publication date: 2017
Event: Abstract from 70th Annual Meeting of the American Physical Society Division of Fluid Dynamics (DFD17), Denver, United States.
Main Research Area: Technical/natural sciences
Electronic versions:
266213.pdf
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2017

Thermophoretic transport of water nanodroplets confined in carbon nanotubes: the role of friction
The development of efficient nanofluidic devices requires driving mechanisms that provide controlled transport of fluids through nanoconduits. Temperature gradients have been proposed as a mechanism to drive particles, fullerenes and nanodroplets inside carbon nanotubes (CNTs). In this work, molecular dynamics (MD) simulations are conducted to study thermophoresis of water nanodroplets inside CNTs. To gain insight into the interplay between the thermophoretic force acting on the droplet and the retarding liquid-solid friction, sets of constrained and unconstrained MD simulations are conducted. The results indicate that the thermophoretic motion of a nanodroplet displays two kinetic regimes: an initial regime characterized by a decreasing acceleration and afterwards a terminal regime with constant velocity. During the initial regime, the magnitude of the friction force increases linearly with the droplet velocity whereas the thermophoretic force has a constant magnitude defined by the magnitude of the thermal gradient and the droplet size. Subsequently, in the terminal regime, the droplet moves at constant velocity due to a dynamic balance between the thermophoretic force and the retarding friction force.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Oyarzua, E. (Ekstern), Walther, J. H. (Intern), Zambrano, H. (Ekstern)
Number of pages: 1
Publication date: 2017
Event: Abstract from 70th Annual Meeting of the American Physical Society Division of Fluid Dynamics (DFD17), Denver, United States.
Main Research Area: Technical/natural sciences
Electronic versions:
MWS_DFD17_2017_000061.pdf
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2017
Transformation of a wave energy spectrum from encounter to absolute domain when observing from an advancing ship

The article presents a practical approach to transform a wave energy spectrum from encounter domain to absolute domain. This problem has its specific relevance, when shipboard sea state estimation is conducted by the buoy analogy; notably for some particular implementation solving for the sea state directly in the encounter domain. The encounter domain is that observed from a ship when it advances in a seaway, whereas the absolute domain is that corresponding to making observations from a fixed point in the inertial frame. Spectrum transformation can be uniquely carried out if the ship sails “against” the waves (beam to head sea) but in following sea conditions there exists no unique solution to the problem. Instead, a reasonable approach valid for practical engineering must be applied, and the article outlines one viable solution that can be used to transform a wave spectrum from encounter to absolute domain. Specifically, two pseudo algorithms are presented, and good performance is achieved with both algorithms when they are tested at different operational scenarios.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Nielsen, U. D. (Intern)
Pages: 160-172
Publication date: 2017
Main Research Area: Technical/natural sciences

Publication information
Journal: Applied Ocean Research
Volume: 69
ISSN (Print): 0141-1187
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.16 SJR 1.202 SNIP 1.75
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.245 SNIP 1.999 CiteScore 2.1
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.442 SNIP 2.244 CiteScore 1.91
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.243 SNIP 2.332 CiteScore 1.84
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.234 SNIP 1.922 CiteScore 1.46
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.496 SNIP 1.975 CiteScore 1.73
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 1.112 SNIP 1.485
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 1.012 SNIP 1.69
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 1.404 SNIP 2.344
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 1.03 SNIP 1.898
Scopus rating (2006): SJR 1.248 SNIP 2.064
Scopus rating (2005): SJR 0.495 SNIP 1.45
Tsunami Induced Scour Around Monopile Foundations

A fully-coupled (hydrodynamic and morphologic) numerical model is presented, and utilized for the simulation of tsunami-induced scour around a monopile structure, representative of those commonly utilized as offshore wind turbine foundations at moderate depths i.e. for depths less than 30 m. The model is based on solutions to Reynolds-averaged Navier-Stokes equations, coupled with two-equation k−ω turbulence closure, with additional bed and suspended load descriptions forming the basis for sea bed morphology. The model is first validated for flow, bed shear stresses, and scour within a steady current, where a generally excellent match with experimentally-based results is found. A methodology for maintaining and assessing hydrodynamic and morphologic similarity between field and (laboratory) model-scale tsunami events is then presented, combining diameter-based Froude number similarity with that based on the dimensionless wave boundary layer thickness-to-monopile diameter ratio. This methodology is utilized directly in the selection of governing tsunami wave parameters (i.e. velocity magnitude and period) used for subsequent simulation within the numerical model, with the tsunami-induced flow modelled as a long sinusoidally-varying current. The flow, sediment transport, and scour processes beneath up to ten tsunami waves are simulated in succession. These illustrate a generally accumulative scour process i.e. a relatively rapid scour induced by the leading wave, with an additional build-up of the scour depth during additional trailing waves. The resulting scour seems to approach an equilibrium value after sufficient time duration, which corresponds reasonably to that predicted by existing steady-current scour depth expressions, after accounting for the finite boundary layer thickness induced by the unsteady tsunami wave, i.e. it is important to incorporate both current-like, as well as wave-like aspects of the long tsunami event. Based on the simulated results, a simple methodology for predicting the tsunami-induced scour depth in engineering practice is finally developed. This methodology is demonstrated to match the predicted scour development for all of the simulated flows considered, ranging from the series of transient tsunami waves to the steady-current limit.
Understanding the formation process of the liquid slug in a hilly-terrain wet natural gas pipeline

In the present work, the liquid slug formation in a hilly-terrain pipeline is simulated using the Volume of Fluid model and RNG k-ε turbulence model. The numerical model is validated by the experimental data of the horizontal slug flow. The influence of the pipe geometric structure and flow condition on the liquid slug formation is discussed including pipe diameter, inclination angle, gas superficial velocity and liquid holdup. The results show that the pipe is blocked by the liquid slug at the moment of slug formed. The pipe pressure suddenly increases, and then decreases gradually in the process of liquid slug formation and motion. The pipe pressure drop and liquid holdup decrease along with the increasing inclination angle of ascending pipe. On the contrary, they rise with the increase of the inclination angle of descending pipe.
Higher gas superficial velocity and liquid holdup result in a larger pressure drop in the formation of a liquid slug, and correspondingly induces a slug flow more rapidly in the hilly-terrain pipelines.

Wake structure and thrust generation of a flapping foil in two-dimensional flow

We present a combined numerical (particle vortex method) and experimental (soap film tunnel) study of a symmetric foil undergoing prescribed oscillations in a two-dimensional free stream. We explore pure pitching and pure heaving, and contrast these two generic types of kinematics. We compare measurements and simulations when the foil is forced with pitching oscillations, and we find a close correspondence between flow visualisations using thickness variations in the soap film and the numerically determined vortex structures. Numerically, we determine wake maps spanned by oscillation frequency and amplitude, and we find qualitatively similar maps for pitching and heaving. We determine the drag–thrust transition for both pitching and heaving numerically, and we discuss it in relation to changes in wake structure. For heaving with low oscillation frequency and high amplitude, we find that the drag–thrust transition occurs in a parameter region with wakes in which two vortex pairs are formed per oscillation period, in contrast to the common transition scenario in regions with inverted von Kármán wakes.
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.82 SJR 1.671 SNIP 1.636
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.912 SNIP 1.676 CiteScore 2.57
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.865 SNIP 1.808 CiteScore 2.66
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.894 SNIP 1.915 CiteScore 2.71
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.731 SNIP 1.88 CiteScore 2.47
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 2.165 SNIP 2.023 CiteScore 2.72
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 2.29 SNIP 2.163
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 2.563 SNIP 1.891
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 2.691 SNIP 2.073
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 2.417 SNIP 1.975
Web of Science (2007): Indexed yes
Scopus rating (2006): SJR 2.641 SNIP 2.181
Web of Science (2006): Indexed yes
Scopus rating (2005): SJR 2.836 SNIP 2.107
Web of Science (2005): Indexed yes
Scopus rating (2004): SJR 2.411 SNIP 2.196
Web of Science (2004): Indexed yes
Scopus rating (2003): SJR 2.896 SNIP 2.059
Web of Science (2003): Indexed yes
Scopus rating (2002): SJR 3.042 SNIP 2.205
Web of Science (2002): Indexed yes
Scopus rating (2001): SJR 3.783 SNIP 2.518
Web of Science (2001): Indexed yes
Scopus rating (2000): SJR 3.66 SNIP 2.242
Web of Science (2000): Indexed yes
Scopus rating (1999): SJR 1.928 SNIP 1.95

Original language: English

Propulsion, Vortex shedding, Wakes/jets
DOIs:
10.1017/jfm.2016.808
Source: FindIt
Source-ID: 2350297490
3D Lagrangian VPM: simulations of the near-wake of an actuator disc and horizontal axis wind turbine

The application of a 3-dimensional Lagrangian vortex particle method has been assessed for modelling the near-wake of an axisymmetrical actuator disc and 3-bladed horizontal axis wind turbine with prescribed circulation from the MEXICO (Model EXperiments InControlled conditions) experiment. The method was developed in the framework of the open-source Parallel Particle-Mesh library for handling the efficient data-parallelism on a CPU (Central Processing Unit) cluster, and utilized a O(N log N)-type fast multipole method for computational acceleration. Simulations with the actuator disc resulted in a wake expansion, velocity deficit profile, and induction factor that showed a close agreement with theoretical, numerical, and experimental results from literature. Also the shear layer expansion was present; the Kelvin-Helmholtz instability in the shear layer was triggered due to the round-off limitations of a numerical method, but this instability was delayed to beyond 1 diameter downstream due to the particle smoothing. Simulations with the 3-bladed turbine demonstrated that a purely 3-dimensional flow representation is challenging to model with particles. The manifestation of local complex flow structures of highly stretched vortices made the simulation unstable, but this was successfully counteracted by the application of a particle strength exchange scheme. The axial and radial velocity profile over the near wake have been compared to that of the original MEXICO experiment, which showed close agreement between results.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Delft University of Technology
Authors: Berdowski, T. (Ekstern), Ferreira, C. M. D. (Ekstern), Walther, J. H. (Intern)
Number of pages: 13
Publication date: 2016
Conference: The Science of Making Torque from Wind, Munich, Germany, 05/10/2016 - 05/10/2016
BFI conference series: European Academy of Wind Energy: The Science of Making Torque from Wind (5010078)
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 753
Article number: 032004
ISSN (Print): 1742-6596
Ratings:
BFI (2018): BFI-level 1
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 0.45 SJR 0.24 SNIP 0.383
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.24 SNIP 0.373 CiteScore 0.35
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.253 SNIP 0.344 CiteScore 0.32
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.231 SNIP 0.272 CiteScore 0.25
ISI indexed (2013): ISI indexed no
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.28 SNIP 0.354 CiteScore 0.33
ISI indexed (2012): ISI indexed no
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 0.292 SNIP 0.352 CiteScore 0.43
ISI indexed (2011): ISI indexed no
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 0.288 SNIP 0.344
Web of Science (2010): Indexed yes
A Coupled VOF-Eulerian Multiphase CFD Model to Simulate Breaking Wave Impacts on Offshore Structures

Breaking wave-induced loads on offshore structures can be extremely severe. The air entrainment mechanism during the breaking process plays a not well-known role in the exerted forces. This paper presents a CFD solver, developed in the Open-FOAM environment, capable of simulating the wave breaking-induced air entrainment. Firstly the model was validated against a bubble column flow. Then it was employed to compute the in-line force exerted by a spilling breaking wave on a vertical cylinder in a 3D domain at a laboratory scale. Results showed that the entrained bubbles affected the magnitude of the force partially. Further analyses on the interaction of the bubble plume with the flow around the cylinder are needed.

Aerodynamically shaped vortex generators

An aerodynamically shaped vortex generator has been proposed, manufactured and tested in a wind tunnel. The effect on the overall performance when applied on a thick airfoil is an increased lift to drag ratio compared with standard vortex generators. Copyright © 2015 John Wiley & Sons, Ltd.
A high order regularisation method for solving the Poisson equation and selected applications using vortex methods

A regularisation method for solving the Poisson equation using Green’s functions is presented. The method is shown to obtain a convergence rate which corresponds to the design of the regularised Green’s function and a spectral-like convergence rate is obtained using a spectrally ideal regularisation. It is shown that the regularised Poisson solver can be extended to handle mixed periodic and free-space boundary conditions. This is done by solving the equation spectrally in the periodic directions which yields a modified Helmholtz equation for the free-space directions which in turn is solved by deriving the appropriate regularised Green’s functions. Using an analogy to the particle-particle particle-mesh method, a framework for calculating multi-resolution solutions using local refinement patches is presented. The regularised Poisson solver is shown to maintain a high order converging solution for different configurations of the refinement patches. The regularised Poisson solver has been implemented in a high order particle-mesh based vortex method for simulating incompressible fluid flow. A re-meshing of the vortex particles is used to ensure the convergence of the method and a re-projection of the vorticity field is included to explicitly fulfil the kinematic constraints of the flow field. The high order, unbounded particle-mesh based vortex method is used to simulate the instability, transition to turbulence and eventual destruction of a single vortex ring. From the simulation data, a novel analysis on the vortex ring dynamics is presented based on the alignment of the vorticity vector with the principal axis of the strain rate tensor. A novel iterative implementation of the Brinkman penalisation method is introduced for the enforcement of a fluid-solid interface in re-meshed vortex methods. The iterative scheme is shown to improve the enforcement of the interface and also allow the simulation to perform significantly larger time steps, than what is customary for the method. The improved accuracy of the iterative implementation is demonstrated by considering challenging benchmark problems such as the impulsively started flow past a cylinder and a flat plate normal or inclined to the flow. The iterative implementation is shown to enhance the quality of the solution by Brinkman penalisation significantly for simulations of highly unsteady flows past complex geometries. A stochastic method of generating a synthetic turbulent flow field is combined with a 2D mesh-free vortex method to simulate the effect of an oncoming turbulent flow on a bridge deck cross-section within the atmospheric boundary layer. The mesh-free vortex method is found to be capable of preserving the a priori specified statistics as well as anisotropic characteristics of the synthesized turbulent flow field. From the simulation, the aerodynamic admittance of four aerodynamically different bridge sections are compared to available wind tunnel data, showing good agreement between the two. A vorticity formulated stochastic turbulence generator is presented which improves the kinetic properties of the generated turbulent field compared to present methods. Additional measures, such as explicit high order smoothing of the flow field, is introduced to ensure that the generated field can be introduced into numerical simulations without an excessive loss of energy due to numerical dissipation.
**A Method for Ship Collision Damage and Energy Absorption Analysis and its Validation**

For design evaluation there is a need for a method which is fast, practical and yet accurate enough to determine the absorbed energy and collision damage extent in ship collision analysis. The most well-known simplified empirical approach to collision analysis was made probably by Minorsky and its limitation is also well recognized. The authors have previously developed simple expressions for the relation between the absorbed energy and the damaged material volume which take into account the structural arrangements, the material properties and the damage modes. The purpose of the present paper is to re-examine this method's validity and accuracy for ship collision damage analysis in ship design assessments by comprehensive validations with the experimental results from the public domain. Twenty experimental tests have been selected, analysed and compared with the results calculated using the proposed method. It can be concluded that the proposed method has a good accuracy with the mean value of 0.988 and standard deviation of 0.042.

**General information**

State: Published  
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd’s Register EMEA  
Authors: Zhang, S. (Ekstern), Pedersen, P. T. (Intern)  
Pages: 10  
Publication date: 2016

**Host publication information**

Title of host publication: Proceedings of the International Conference on Ships and Offshore Structures (ICSOS)  
Main Research Area: Technical/natural sciences  
Conference: The International Conference on Ships and Offshore Structures, Hamburg, Germany, 31/08/2016 - 31/08/2016  
Ship collision, Collision energy, Damage extent and size, Structural arrangement, Material property, Damage mode, Damaged material volume  
Source: PublicationPreSubmission  
Source-ID: 125119738  
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

**A multiresolution method for solving the Poisson equation using high order regularization**

We present a novel high order multiresolution Poisson solver based on regularized Green's function solutions to obtain exact free-space boundary conditions while using fast Fourier transforms for computational efficiency. Multiresolution is achieved through local refinement patches and regularized Green’s functions corresponding to the difference in the spatial resolution between the patches. The full solution is obtained utilizing the linearity of the Poisson equation enabling superposition of solutions. We show that the multiresolution Poisson solver produces convergence rates that correspond to the regularization order of the derived Green's functions.

**General information**

State: Published  
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering  
Authors: Hejlesen, M. M. (Intern), Walther, J. H. (Intern)  
Pages: 188-196  
Publication date: 2016  
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Journal of Computational Physics  
Volume: 326  
ISSN (Print): 0021-9991  
Ratings:  
BFI (2018): BFI-level 1  
Web of Science (2018): Indexed yes  
BFI (2017): BFI-level 1  
Web of Science (2017): Indexed yes  
BFI (2016): BFI-level 1  
Scopus rating (2016): CiteScore 3.12 SJR 2.034 SNIP 1.822  
Web of Science (2016): Indexed yes  
BFI (2015): BFI-level 1
An attempt to define critical wave and wind scenarios leading to capsize in beam sea

The IMO Weather Criterion has proven to be the governing stability criteria regarding minimum GM for e.g. small ferries and large passenger ships. The formulation of the Weather Criterion is based on some empirical relations derived many years ago for vessels not necessary representative for current new buildings with large superstructures. Thus it seems reasonable to investigate the possibility of capsizing in beam sea under the joint action of waves and wind using direct time domain simulations. This has already been done in several studies. Here it is combined with the First Order Reliability Method to define possible combined critical wave and wind scenarios leading to capsize and corresponding probability of capsize. The results for a fictitious vessel are compared with Monte Carlo simulation and good agreement is found at a
much lesser computational effort. Finally, the results for a large container vessel and a small ferry will be discussed in the light of the current weather criterion.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register EMEA, Søfartsstyrelsen
Authors: Jensen, J. J. (Intern), Choi, J. (Intern), Kristensen, H. O. H. (Intern), Nielsen, U. D. (Intern), Erichsen, H. (Ekstern), Tvedt, E. I. (Ekstern)
Number of pages: 7
Publication date: 2016

**Host publication information**

Title of host publication: Proceedings of the 13th International Symposium on Practical Design of Ships and Other Floating Structures (PRADS'2016)
Publisher: Technical University of Denmark (DTU)
Editors: Dam Nielsen, U., Juncher Jensen, J.
Main Research Area: Technical/natural sciences
Conference: 13th International Symposium on Practical Design of Ships and Other Floating Structures (PRADS'2016), Copenhagen, Denmark, 04/09/2016 - 04/09/2016
Weather criteria, Wind loads, Wave loads, FORM, Capsize, Design Load scenarios

Electronic versions:
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

A note on the relative efficiency of methods for computing the transient free-surface Green function

A number of papers have appeared recently on computing the time-domain, free-surface Green function. Two papers in particular, Chuang et al. (2007) and Li et al. (2015) considered the method developed by Clement (1998) who showed that this Green function is the solution to a fourth-order Ordinary Differential Equation (ODE). This ODE has been suggested as a means for speeding up the calculation of the Green function coefficients compared to the standard algorithms developed for example by Newman (1992). Clement solved the ODE using the classical fourth-order, four-step Runge–Kutta scheme (RK44) with a fixed time step size. The two papers mentioned above proposed alternative numerical methods which are claimed to be more efficient. In this note we consider the relative efficiency of these four methods on a representative test case, and conclude that the standard method is the most efficient. Of the ODE-based methods, the method of Chuang et al. (2007) is found to be slightly more efficient than the RK44 method, while the method of Li et al. (2015) is at least an order of magnitude less efficient. It is also pointed out that ODE methods have yet to be extended to include finite water depth.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Bingham, H. B. (Intern)
Pages: 15-20
Publication date: 2016
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Ocean Engineering
Volume: 120
ISSN (Print): 0029-8018
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.46 SJR 1.315 SNIP 2.014
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.172 SNIP 1.989 CiteScore 2.19
Web of Science (2015): Indexed yes
A Review of Sea State Estimation Procedures Based on Measured Vessel Responses

The operation of ships requires careful monitoring of the related costs while, at the same time, ensuring a high level of safety. A ship’s performance with respect to safety and fuel efficiency may be compromised by the encountered waves. Consequently, it is important to estimate the surrounding seastate, and any shipboard decision support system (DSS) needs to have as input information about the encountered waves for the DSS to be the most accurate and reliable. Trustful means for sea state estimation (SSE) include floating wave rider buoys. However, for ships navigating the oceans, wave rider buoys are not practical, as sea state information in real-time and at the actual geographical position of the ship is needed. On the other hand, the analogy between a ship and a floating buoy naturally suggests to using the ship itself as a wave buoy. This paper presents a status on techniques for shipboard SSE using measured vessel responses, resembling the concept of traditional wave rider buoys. Moreover, newly developed ideas for shipboard sea state estimation are introduced. The presented material is all based on the author’s personal experience, developed within extensive work on the subject in the last fifteen years; work conducted alone and together with national as well as international colleagues.
A robust WENO scheme for nonlinear waves in a moving reference frame

For robust nonlinear wave simulation in a moving reference frame, we recast the free surface problem in Hamilton-Jacobi form and propose a Weighted Essentially Non-Oscillatory (WENO) scheme to automatically handle the upwinding of the convective term. A new automatic procedure for deriving the linear WENO weights based on a Taylor series expansion is introduced. A simplified smoothness indicator is proposed and is shown to perform well. The scheme is combined with high-order explicit Runge-Kutta time integration and a dissipative Lax-Friedrichs-type flux to solve for nonlinear wave propagation in a moving frame of reference. The WENO scheme is robust and less dissipative than the equivalent order upwind-biased finite difference scheme for all ratios of frame of reference to wave propagation speed tested. This provides the basis for solving general nonlinear wave-structure interaction problems at forward speed.
A Study on the uncertainty and sensitivity in numerical simulation of parametric roll

Uncertainties related to numerical modelling of parametric roll have been investigated by using a 6-DOFs model with nonlinear damping and roll restoring forces. At first, uncertainty on damping coefficients and its effect on the roll response is evaluated. Secondly, uncertainty due to the “effective (equivalent) wave” concept in calculation of restoring moment is studied. Finally, uncertainty to roll response from different methods of GZ calculation has been checked. It is found that the equivalent wave concept is sufficiently accurate for the purpose of GZ calculation. Two different GZ approximations give a good agreement with direct calculation method if relevant coefficients have been properly found in the fitting.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Choi, J. (Intern), Nielsen, U. D. (Intern), Jensen, J. J. (Intern)
Number of pages: 8
Publication date: 2016

Biomixing in stagnant water above population of blue mussels (Mytilus edulis).

Dense beds of filter-feeding mussels can exert a considerable grazing impact on phytoplankton in many marine areas depending on downmixing promoted by current, wave- and wind action. But downmixing may also be promoted by biomixing caused by the action of the strong exhalent jets of water from the mussels. Here we study the strength of biomixing exerted by large actively filtering blue mussels Mytilus edulis in stagnant water. Vertical concentration profiles of added algal cells (Rhodomonas salina) were measured (as chl a) over a 70 cm high and stagnant water column in an aquarium above a population of 48 ind.m-2 of mussels of shell length 69.5 ± 2.3 mm. Due to the intense agitation (biomixing) generated by exhalant jets of the actively feeding mussels the profiles remained nearly uniform over the full water column while decreasing exponentially with time, reaching a level of about 40% of the initial level after 120 min, which implied a population filtration rate of about 0.3 m3.h-1.m-2 in agreement with prior clearance measurements. Comparing to numerical solutions of a one-dimensional diffusion model, varying the eddy diffusivity, a value of D = 550 × 10-6 m2.s-1 was estimated. This high strength of biomixing far exceeds those of previous similar studies on the filter-feeding polychaete Nereis diversicolor (0.3 × 10-6 m2.s-1) and the ascidian Ciona intestinalis (150 × 10-6 m2.s-1) and suggests that biomixing in moderate benthic boundary layer flows past mussel beds may contribute to the downmixing of phytoplankton.

General information
State: Published
Carbon nanotube-based coatings to induce flow enhancement in hydrophilic nanopores
With the emergence of the field of nanofluidics, the transport of water in hydrophilic nanopores has attracted intensive research due to its many promising applications. Experiments and simulations have found that flow resistance in hydrophilic nanochannels is much higher than those in macrochannels. Indeed, this might be attributed to significant fluid adsorption on the channel walls and to the effect of the increased surface to volume ratio inherent to the nanoconfinement. Therefore, it is desirable to explore strategies for drag reduction in nanopores. Recently, studies have found that carbon nanotubes (CNTs) feature ultrafast waterflow rates which result in flow enhancements of 1 to 5 orders of magnitude compared to Hagen-Poiseuille predictions. In the present study, CNT-based coatings are considered to induce water flow enhancement in silica nanopores with different radius. We conduct atomistic simulations of pressurized water flow inside tubular silica nanopores with and without inner coaxial carbon nanotubes. In particular, we compute water density and velocity profiles, flow enhancement and slip length to understand the drag reduction capabilities of single- and multi-walled carbon nanotubes implemented as coating material in silica nanopores.

CFD modelling of condensation process of water vapor in supersonic flows
The condensation phenomenon of vapor plays an important role in various industries, such as the steam flow in turbines and refrigeration system. A mathematical model is developed to predict the spontaneous condensing phenomenon in the supersonic conditions using the nucleation and droplet growth theories. The numerical approach is validated with the experimental data, which shows a good agreement between them. The condensation characteristics of water vapor in the Laval nozzle are studied numerically in this paper. The results show that the condensation process is a rapid variation of the vapor-liquid phase change both in space and in time. The spontaneous condensation of water vapor will not appear immediately when the steam reaches the saturation state. Instead, it occurs further downstream the nozzle throat, where
the steam is in the state of supersaturation.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Nottingham, Changzhou University
Authors: Wen, C. (Intern), Walther, J. H. (Intern), Yan, Y. (Ekstern), Yang, Y. (Ekstern)
Number of pages: 8
Publication date: 2016

Host publication information
Title of host publication: Proceedings of the International Heat Transfer Symposium 2016
Main Research Area: Technical/natural sciences
Condensation, Water vapor, Laval nozzle
Electronic versions:
ATE_paper.pdf
Source: PublicationPreSubmission
Source-ID: 125067081
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

CFD Study on Effective Wake of Conventional and Tip-modified Propellers
Tip-modified propellers have been developed to improve propulsive efficiency and to lengthen the lifting-surface span. A hydrodynamic distinction of tip-modified propellers consistently shown in model tests is 5-15% higher effective wake fraction compared to conventional propellers. The effective wake fraction is not from direct measurements but from an estimation based on an open-water curve correlation at the same thrust or torque coefficient as in the self-propulsion test. Open-water tests are conducted at 2-4 times higher Reynolds number than that depending on Froude scaling in self-propulsion tests. The effects of different Reynolds number on higher effective wake fraction of tip-modified propellers are investigated by open-water simulations with varying the propeller speed and evaluations of effective wake extracted from self-propulsion simulations on tip-modified and conventional propellers.

Open-water simulations show that the advance ratio at the design thrust is higher at a higher Reynolds number for both propellers and the advance ratio increase is smaller for the tip-modified propeller, which results in a higher effective wake fraction. Effective wake fractions are evaluated by integrating velocity fields at a section 40% of the propeller radius upstream from the propeller plane in self-propulsion simulations. The difference of effective wake fraction from integrating velocity fields between tip-modified and conventional propellers is less than 1%. Based on the open-water simulation result and the effective wake fractions from integrating CFD velocity fields, 5-15% higher effective wake fractions of tip-modified propellers from the existing estimation method based on the open-water correlation at thrust or torque identity can be related mainly to the effects of Reynolds number. However, the effective wake fraction from integrating a total velocity field with excluding a propeller-induced flow is about 10% higher for the tip-modified propeller. The propeller-induced flow is estimated separately by open-water simulations. Further studies are necessary with a more sophisticated way to estimate the propeller-induced flow by taking into account interaction effects between propeller-induced flow and hull wake.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Shin, K. W. (Ekstern), Andersen, P. (Intern)
Number of pages: 16
Publication date: 2016

Host publication information
Title of host publication: Proceedings of the 31st Symposium on Naval Hydrodynamics
Publisher: Office of Naval Research, U.S.A.
Main Research Area: Technical/natural sciences
Source: PublicationPreSubmission
Source-ID: 127556325
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

CNT based thermal Brownian motor to pump water in nanodevices
Brownian molecular motors are nanoscale machines that exploit thermal fluctuations for directional motion by employing mechanisms such as the Feynman-Smoluchowski ratchet. In this study, using Non Equilibrium Molecular Dynamics, we propose a novel thermal Brownian motor for pumping water through Carbon Nanotubes (CNTs). To achieve this we impose a thermal gradient along the axis of a CNT filled with water and impose, in addition, a spatial asymmetry by fixing
specific zones on the CNT in order to modify the vibrational modes of the CNT. We find that the temperature gradient and imposed spatial asymmetry drive the water flow in a preferential direction. We systematically modified the magnitude of the applied thermal gradient and the axial position of the fixed points. The analysis involves measurement of the vibrational modes in the CNTs using a Fast Fourier Transform (FFT) algorithm. We observed water flow in CNTs of 0.94, 1.4 and 2.0 nm in diameter, reaching a maximum velocity of 5 m/s for a thermal gradient of 3.3 K/nm. The proposed thermal motor is capable of delivering a continuous flow throughout a CNT, providing a useful tool for driving liquids in nanofluidic devices by exploiting thermal gradients.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Oyarzua, E. (Ekstern), Zambrano, H. (Ekstern), Walther, J. H. (Intern)
Number of pages: 1
Publication date: 2016

Host publication information
Title of host publication: Bulletin of the American Physical Society
Volume: 61
Publisher: American Physical Society
Article number: A22.00009
Main Research Area: Technical/natural sciences
Electronic versions:
MWS_DFD16_2016_001066.pdf
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2016

Computation of Added Mass and Damping Coefficients of a Horizontal Circular Cylinder in Open Foam
This paper presents numerical computation of added mass and damping coefficients of a slender horizontal cylinder in the free surface zone, which typically serves as a fish cage floater. A fully viscous two phase flow solver in OpenFOAM was employed in the numerical computation. The purpose was to validate the capability of this solver and dynamic mesh functionality. A two dimensional numerical wave tank was set up, and two wave relaxation zones were used to reduce the size of the computational domain. Harmonic forced oscillations of the cylinder were performed at different frequencies and amplitudes. The mesh at free surface zone was refined based on the radiated wave heights at different oscillation frequencies in order to properly resolve the radiated waves. The result shows that in most frequency ranges, the numerical computation agreed well with the experimental data and analytical solution. However at low frequency range for added mass coefficient in heave motion, deviations were observed, and it was due to the effect of finite water depth. In addition for sway motion at high frequency range, the damping coefficient was underestimated comparing with analytical solution. This was believed to be as a result of high steepness of the radiated waves.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Chen, H. (Intern), Christensen, E. D. (Intern)
Number of pages: 8
Publication date: 2016

Host publication information
Title of host publication: Proceedings of the ASME 35th International Conference on Ocean, Offshore and Arctic Engineering (OMAE 2016)
Publisher: American Society of Mechanical Engineers
Article number: OMAE2016-54429
ISBN (Electronic): 978-0-7918-4993-4
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: ASME 2016 35th International Conference on Ocean, Offshore and Arctic Engineering (OMAE2016), Busan, Korea, Republic of, 19/06/2016 - 19/06/2016
DOIs: 10.1115/OMAE2016-54429
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

Continuous vs. pulsating flow boiling. Part 1: Experimental comparison and visualization
This experimental study investigates an active method for flow boiling heat transfer enhancement by means of fluid flow pulsation. The hypothesis is that pulsations increase the flow boiling heat transfer by means of better bulk fluid mixing,
increased wall wetting and flow-regime destabilization. The fluid pulsations are introduced by a flow modulating expansion device and are compared with continuous flow by a stepper-motor expansion valve in terms of time-averaged heat transfer coefficient. The cycle time ranges from 1 s to 9 s for the pulsations. The time-averaged heat transfer coefficients are reduced from transient measurements immediately downstream of the expansion valves at low vapor qualities. The results show that the pulsations improve the time-averaged heat transfer coefficient by 3.2 % on average at low cycle time (1 s to 2) s, whereas the pulsations may reduce the time-averaged heat transfer coefficient by as much as 8 % at high heat flux (q ≥ 35 kW/m²) and cycle time (8 s). The latter reduction is adhered to the significant dry-out when the flow modulating expansion valve is closed.

**Continuous vs. pulsating flow boiling. Part 2: Statistical comparison using response surface methodology**

Response surface methodology is used to investigate an active method for flow boiling heat transfer enhancement by means of fluid flow pulsation. The flow pulsations are introduced by a flow modulating expansion device and compared with the baseline continuous flow provided by a stepper-motor expansion valve. Two experimental designs (data point sets) are generated using a modified Central Composite Design for each valve and their response surfaces are compared using the quadratic model. Statistical information on the significant model terms are used to clarify whether the effect of fluid flow pulsations is statistically significant in terms of the time-averaged flow boiling heat transfer coefficient. The cycle time range from 1 s to 9 s for the pulsations. The results show that the effect of fluid flow pulsations is statistically significant, disregarding the lowest heat flux measurements. The response surface comparison reveals that the flow pulsations improves the time-averaged heat transfer coefficient by as much as 10 % at the smallest cycle time compared with continuous flow. On the other hand, at highest cycle time and heat flux, the reduction may be as much as 20 % due to significant dry-out when the valve is closed. These values are higher than reported in part 1 of the paper, but evaluated more consistently at equal heat flux using the response surfaces.

**Convergence of near-field added resistance calculations using a high order finite-difference method**

Response surface methodology is used to investigate an active method for flow boiling heat transfer enhancement by means of fluid flow pulsation. The flow pulsations are introduced by a flow modulating expansion device and compared with the baseline continuous flow provided by a stepper-motor expansion valve. Two experimental designs (data point sets) are generated using a modified Central Composite Design for each valve and their response surfaces are compared using the quadratic model. Statistical information on the significant model terms are used to clarify whether the effect of fluid flow pulsations is statistically significant in terms of the time-averaged flow boiling heat transfer coefficient. The cycle time range from 1 s to 9 s for the pulsations. The results show that the effect of fluid flow pulsations is statistically significant, disregarding the lowest heat flux measurements. The response surface comparison reveals that the flow pulsations improves the time-averaged heat transfer coefficient by as much as 10 % at the smallest cycle time compared with continuous flow. On the other hand, at highest cycle time and heat flux, the reduction may be as much as 20 % due to significant dry-out when the valve is closed. These values are higher than reported in part 1 of the paper, but evaluated more consistently at equal heat flux using the response surfaces.
DeRisk - Accurate prediction of ULS wave loads. Outlook and first results

Loads from extreme waves can be dimensioning for the substructures of offshore wind turbines. The DeRisk project (2015-2019) aims at an improved load evaluation procedure for extreme waves through application of advanced wave models, laboratory tests of load effects, development of hydrodynamic load models, aero-elastic response calculations and statistical analysis. This first paper from the project outlines the content and philosophy behind DeRisk. Next, the first results from laboratory tests with irregular waves are presented, including results for 2D and 3D focused wave groups. The results of focused wave group tests and a 6-hour (full scale duration) test are reproduced numerically by re-application of the wave paddle signal in a fully nonlinear potential flow wave model. A good match for the free surface elevation and associated exceedance probability curve is obtained. Finally, the utilization of DeRisk's results in practical design is discussed. (C) 2016 Published by Elsevier Ltd.
Derivation and analysis of the analytical velocity and vortex stretching expressions for an $O(N \log N)$-FMM

In the current paper, a method for deriving the analytical expressions for the velocity and vortex stretching terms as a function of the spherical multipole expansion approximation of the vector potential is presented. These terms are essential in the context of 3D Lagrangian vortex particle methods combined with fast summation techniques. The convergence and computational efficiency of this approach is assessed in the framework of an $O(N \log N)$-type Fast Multipole Method (FMM), by using vorticity particles to simulate a system of coaxial vortex rings for which also the exact results are known. It is found that the current implementation converges rapidly to the exact solution with increasing expansion order and acceptance factor. An investigation into the computational efficiency demonstrated that the $O(N \log N)$-type FMM is already viable for a particle size of only several thousands and that this speedup increases significantly with the number of particles. Finally, it is shown that the implementation of the FMM with the current analytical expressions is at least twice as fast as when opting for using even the simplest implementation of finite differences instead.
Development of multi-component diesel surrogate fuel models – Part II: Validation of the integrated mechanisms in 0-D kinetic and 2-D CFD spray combustion simulations

The aim of this study is to develop compact yet comprehensive multi-component diesel surrogate fuel models for computational fluid dynamics (CFD) spray combustion modelling studies. The fuel constituent reduced mechanisms including n-hexadecane (HXN), 2,2,4,4,6,8,8-heptamethylnonane (HMN), cyclohexane(CHX) and toluene developed in Part I are applied in this work. They are combined to produce two different versions of multi-component diesel surrogate models in the form of MCDS1 (HXN + HMN) and MCDS2 (HXN + HMN + toluene + CHX). The integrated mechanisms are then comprehensively validated in zero-dimensional chemical kinetic simulations under a wide range of shock tube and jetstirred reactor conditions. Subsequently, the fidelity of the surrogate models is further evaluated in two-dimensional CFD spray combustion simulations. Simulation results show that ignition delay (ID) prediction corresponds well to the change of fuel constituent mass fraction which is calculated to match the cetane number (CN). In addition, comparisons of the simulation results to the experimental data of #2diesel fuel (D2) in a constant volume combustion chamber show that IDs and lift-off lengths are reasonably well replicated by the models. The MCDS2 model is also found to perform better in the soot formation prediction in D2 fuel combustion as the model contains aromatic and cyclo-alkane components which provide an additional pathway to the formation of rich species such as C2H2 and C6H6. Implementation of MCDS2 predicts an increase of maximum local soot volume fraction by a factor of 2.1 when the ambient temperature increases from 900 K to 1000 K, while the prediction by MCDS1 is lower at 1.6. This trend qualitatively agrees with the experimental observation. This work demonstrates that MCDS1 serves as a potential surrogate fuel model for diesel fuels with CN values ranging from 15 to 100. It also shows that MCDS2 is a more appropriate surrogate model for fuels with aromatics and cyclo-paraffinic contents, particularly when soot calculation is of main interest.
In the present work, development and validation of reduced chemical kinetic mechanisms for several different hydrocarbons are performed. These hydrocarbons are potential representatives for practical diesel fuel constituents. n-Hexadecane (HXN), 2,2,4,4,6,8,8-heptamethylnonane (HMN), cyclohexane (CHX) and toluene are selected to represent straight-alkane, branched-alkane, cyclo-alkane and aromatic compounds in the diesel fuel. A five-stage chemical kinetic mechanism reduction scheme formulated in the previous work is applied to develop the reduced HMN and CHX models based on their respective detailed mechanisms. Alongside with the development of the reduced CHX model, a skeletal toluene sub-mechanism is constructed since the elementary reactions for toluene are subset of the detailed CHX mechanism. The final reduced HMN mechanism comprises 89 species with 319 elementary reactions, while the final reduced CHX mechanism which includes the toluene sub-mechanism consists of 80 species with 287 elementary reactions. Both reduced models are approximately 92% smaller than their respective detailed models in terms of total number of species and elementary reactions. Following that, both the newly developed fuel constituent reduced mechanisms, together with the formerly derived reduced HXN mechanism are comprehensively validated in zero-dimensional chemical kinetic simulations under a wide range of shock tube and jet-stirred reactor (JSR) conditions. Well agreement between the reduced and detailed mechanisms is achieved for ignition delay (ID) and species concentration predictions under both auto-ignition and JSR conditions, with a maximum relative error of 40%. In addition, the reduced models are further validated against the JSR experimental results for each diesel fuel constituents. The surrogate models are able to reasonably reproduce the experimental species concentration profiles in view of their simplified fuel chemistries. Deviations within one order of magnitude in the absolute values are recorded between the computations and measurements. Validation of these reduced models for each diesel fuel constituent in this work serves as a prerequisite for constructing a multicomponent diesel surrogate fuel model. The compact yet accurate chemical models proposed here aid to reduce the chemistry size of the final multi-component diesel surrogate model such that it remains computationally efficient when it is incorporated with multi-dimensional CFD simulations. The reduced mechanism of each fuel constituent can also be used individually for other CFD applications.
Effect of delta wing on the particle flow in a novel gas supersonic separator

The present work presents numerical simulations of the complex particle motion in a supersonic separator with a delta wing located in the supersonic flow. The effect of the delta wing on the strong swirling flow is analysed using the Discrete Particle Method. The results show that the delta wings re-compress the upstream flow and the gas Mach number decreases correspondingly. However, the Mach number does not vary significantly from the small, medium and large delta wing configurations. The small delta wing generates a swirl near its surface, but has minor influences on the flow above it. On the contrary, the use of the large delta wing produces a strong swirling flow in the whole downstream region. For the large delta wing, the collection efficiency reaches 70% with 2 μm particles, indicating a good separation performance of
the proposed supersonic separator.

**General information**

**State:** Published  
**Organisations:** Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy, Changzhou University, CSIRO  
**Authors:** Wen, C. (Intern), Yang, Y. (Ekstern), Walther, J. H. (Intern), Pang, K. M. (Intern), Feng, Y. (Ekstern)  
**Pages:** 261-267  
**Publication date:** 2016  
**Main Research Area:** Technical/natural sciences

**Publication information**

**Journal:** Powder Technology  
**Volume:** 304  
**ISSN (Print):** 0032-5910  
**Ratings:**  
BFI (2018): BFI-level 1  
Web of Science (2018): Indexed yes  
BFI (2017): BFI-level 1  
Web of Science (2017): Indexed yes  
BFI (2016): BFI-level 1  
Scopus rating (2016): CiteScore 3.16 SJR 0.983 SNIP 1.482  
Web of Science (2016): Indexed yes  
BFI (2015): BFI-level 1  
Scopus rating (2015): SJR 0.965 SNIP 1.598 CiteScore 2.99  
Web of Science (2015): Indexed yes  
BFI (2014): BFI-level 1  
Scopus rating (2014): SJR 0.89 SNIP 1.649 CiteScore 2.67  
Web of Science (2014): Indexed yes  
BFI (2013): BFI-level 1  
Scopus rating (2013): SJR 0.901 SNIP 1.875 CiteScore 2.64  
ISI indexed (2013): ISI indexed yes  
BFI (2012): BFI-level 1  
Scopus rating (2012): SJR 0.854 SNIP 1.826 CiteScore 2.36  
ISI indexed (2012): ISI indexed yes  
BFI (2011): BFI-level 1  
Scopus rating (2011): SJR 0.921 SNIP 1.86 CiteScore 2.45  
ISI indexed (2011): ISI indexed yes  
Web of Science (2011): Indexed yes  
BFI (2010): BFI-level 1  
Scopus rating (2010): SJR 0.94 SNIP 1.547  
BFI (2009): BFI-level 1  
Scopus rating (2009): SJR 0.98 SNIP 1.65  
BFI (2008): BFI-level 1  
Scopus rating (2008): SJR 0.911 SNIP 1.597  
Web of Science (2008): Indexed yes  
Scopus rating (2007): SJR 0.854 SNIP 1.316  
Web of Science (2007): Indexed yes  
Scopus rating (2006): SJR 1.118 SNIP 1.324  
Web of Science (2006): Indexed yes  
Scopus rating (2005): SJR 1.253 SNIP 1.399  
Web of Science (2005): Indexed yes  
Scopus rating (2004): SJR 0.867 SNIP 1.341  
Scopus rating (2003): SJR 1.348 SNIP 1.489  
Scopus rating (2002): SJR 1.285 SNIP 1.369  
Scopus rating (2001): SJR 1.11 SNIP 1.292
Effect of meniscus contact angle during early regimes of spontaneous capillarity in nanochannels

In capillary imbibition, the classical Lucas-Washburn equation predicts a singularity as the fluid enters the channel consisting in an anomalous infinite velocity of the capillary meniscus. The Bosanquet's equation overcomes this problem by taking into account fluid inertia predicting an initial imbibition regime with constant velocity. Nevertheless, the initial constant velocity predicted by Bosanquet's equation is much greater than experimentally observed. In the present study, we conduct atomistic simulations to investigate capillary imbibition of water in silica nanochannels with heights between 4 and 18 nm. We also find that the meniscus contact angle remains constant during the inertial regime and its value depends upon the height of the channel. We also find that the meniscus velocity computed at the channel entrance is related to the particular value of the meniscus contact angle. Moreover, after the inertial regime, the meniscus contact angle is found to be time dependent for all the channels under study. We propose an expression for the time evolution of the dynamic contact angle in nanochannels which, when incorporated in Bosanquet's equation, satisfactorily explains the initial capillary rise.

Effect of swirling device on flow behavior in a supersonic separator for natural gas dehydration

The supersonic separator is a revolutionary device to remove the condensable components from gas mixtures. One of the key issues for this novel technology is the complex supersonic swirling flow that is not well understood. A swirling device composed of an ellipsoid and several helical blades is designed for an annular supersonic separator. The supersonic swirling separation flow of natural gas is calculated using the Reynolds Stress model. The results show that the viscous heating and strong swirling flow cause the adverse pressure in the annular channel, which may negatively affect the separation performance. When the swirling flow passes through the annular nozzle, it will damage the expansion characteristics of the annular nozzle. The blade angles and numbers are both optimized by evaluating the swirling and expansion effects for the supersonic separation.
Effect of the meniscus contact angle during early regimes of spontaneous imbibition in nanochannels

Nanoscale capillarity has been extensively investigated; nevertheless, many fundamental questions remain open. In spontaneous imbibition, the classical Lucas-Washburn equation predicts a singularity as the fluid enters the channel consisting of an anomalous infinite velocity of the capillary meniscus. Bosanquet's equation overcomes this problem by taking into account fluid inertia predicting an initial imbibition regime with constant velocity. Nevertheless, the initial constant velocity as predicted by Bosanquet's equation is much greater than those observed experimentally. In the present study, large scale atomistic simulations are conducted to investigate capillary imbibition of water in slit silica nanochannels with heights between 4 and 18 nm. We find that the meniscus contact angle remains constant during the inertial regime and its value depends on the height of the channel. We also find that the meniscus velocity computed at the channel entrance is related to the particular value of the meniscus contact angle. Moreover, during the subsequent visco-inertial regime, as the influence of viscosity increases, the meniscus contact angle is found to be time dependent for all the channels under study. Furthermore, we propose an expression for the time evolution of the dynamic contact angle in nanochannels which, when incorporated into Bosanquet's equation, satisfactorily explains the initial capillary rise.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Karna, N. K. (Ekstern), Oyarzua, E. (Ekstern), Walther, J. H. (Intern), Zambrano, H. A. (Ekstern)
Pages: 31997-32001
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Journal: Physical Chemistry Chemical Physics
Volume: 18
Issue number: 47
ISSN (Print): 1463-9076
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 4.06 SJR 1.678 SNIP 1.117
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.771 SNIP 1.244 CiteScore 4.45
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.772 SNIP 1.253 CiteScore 4.29
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.715 SNIP 1.216 CiteScore 4.05
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.916 SNIP 1.184 CiteScore 3.67
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.697 SNIP 1.203 CiteScore 3.6
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
Erratum for "Simulation of Wave-Plus-Current Scour beneath Submarine Pipelines" by Bjarke Eltard Larsen, David R. Fuhrman, and B. Mutlu Sumer

Errors in reference citations introduced during composition by ASCE are being clarified in this erratum. The citations corresponding to the reference Fredsøe et al. (1992) were inadvertently changed to Fredsøe and Deigaard (1992) in four locations. The citations are being corrected in this paper, in the order of their appearance in the final published version. ASCE regrets these errors.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Eltard-Larsen, B. (Intern), Fuhrman, D. R. (Intern), Sumer, B. M. (Intern)
Number of pages: 1
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Waterway, Port, Coastal, and Ocean Engineering
Volume: 142
Issue number: 5
Article number: 08216001
ISSN (Print): 0733-950x
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Estimation of waves and ship responses using onboard measurements

This thesis focuses on estimation of waves and ship responses using ship-board measurements. This is useful for development of operational safety and performance efficiency in connection with the broader concept of onboard decision support systems.

Estimation of sea state is studied using a set of measured ship responses, a parametric description of directional wave spectra (a generalised JONSWAP model) and the transfer functions of the ship responses. The difference between the
spectral moments of the measured ship responses and the corresponding theoretically calculated moments formulates a cost function. A set of wave parameters, characterising the directional wave spectrum, is estimated through an optimisation problem using global search basin with proper constraints. This approach applies a sequential partitioning procedure, which is able to classify swell and wind sea events using wind information.

The model is tested on simulated data based on known unimodal and bimodal wave scenarios. The wave parameters in the output are then compared with the true wave parameters. In addition to the numerical experiments, two sets of full-scale measurements from container ships are analysed. Herein, the validation of the estimation method is assessed by comparing the results with the wave data from other tools, such as wave radar data and hindcast data. The results show that the developed method is reasonably accurate.

Automatic selection of a set of responses to be used for wave estimation is also studied using a sensitivity analysis of the wave parameters. This selection depends on the waves and the operational condition of the ship. Therefore, the method can be utilised based on initial knowledge about the waves and the operational condition in a specific location.

A dynamic trend model is proposed for tracking the evolution of the wave parameters during the voyage. This provides a prediction of the wave parameters, e.g. 20 minutes ahead of the measurements. Given the predicted parameters, a wave spectrum model and the transfer functions, forecasts of different wave-induced responses are made. The predicted variances of the responses are compared with actual measurements. The relatively good agreement in this comparison validates the model and the optimisation method. Finally, an uncertainty analysis of the presented approach is implemented to assess the reliability of the method.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Montazeri, N. (Intern), Nielsen, U. D. (Intern), Jensen, J. J. (Intern)
Number of pages: 153
Publication date: 2016

Publication information
Place of publication: Kgs. Lyngby
Publisher: Technical University of Denmark (DTU)
ISBN (Electronic): 978-87-7475-454-1
Original language: English

Series: DCAMM Special Report
Number: S202
ISSN: 0903-1685
Main Research Area: Technical/natural sciences
Electronic versions:
Thesis_Updated_Final.pdf
S202_Najmeh_Montazeri_PhD_Thesis.pdf

Relations
Projects:
Estimation of waves and ship responses using onboard measurements
Source: PublicationPreSubmission
Source-ID: 124374929
Publication: Research › Ph.D. thesis – Annual report year: 2016

Estimation of wind sea and swell using shipboard measurements – A refined parametric modelling approach
Shipboard wave estimation has been of interest in recent years for the purpose of decision support. In this paper, estimation of sea state is studied using ship responses and a parametric description of directional wave spectra. A set of parameters, characterising a given wave spectrum is estimated through an optimisation problem using global search basin with proper constraints. The cost function is established based on the difference between the energies of a set of measured ship responses and the corresponding theoretical spectral moments. A partitioning procedure is applied, which is able to separate swell components from wind seas.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Montazeri, N. (Intern), Nielsen, U. D. (Intern), Jensen, J. J. (Intern)
Pages: 73-86
Publication date: 2016
Main Research Area: Technical/natural sciences
Evaluation of Shipboard Wave Estimation Techniques through Model-scale Experiments

The paper continues a study on the wave buoy analogy that uses shipboard measurements to estimate sea states. In the present study, the wave buoy analogy is formulated directly in the time domain and relies only partly on wave-vessel...
response amplitude operators (RAOs), which is in contrast to all previous works that either are formulated in the frequency domain and/or depend entirely on RAOs. Specifically, the paper evaluates a novel concept for wave estimation based on combined techniques using a wave frequency estimator, not dependent on RAOs, to detect wave frequency and, respectively, nonlinear least squares fitting to estimate wave amplitude and phase. The concept has been previously tested with only numerical simulations but in this study the techniques are applied to model-scale experiments. It is shown that the techniques successfully can be used to estimate the wave parameters of a regular wave train.

General information
State: Published
Organisations: Department of Electrical Engineering, Automation and Control, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Norwegian University of Science and Technology
Authors: Nielsen, U. D. (Intern), Galeazzi, R. (Intern), H. Brodtkorb, A. (Ekstern)
Number of pages: 8
Publication date: 2016

Host publication information
Title of host publication: Proceedings of OCEANS’16
Publisher: IEEE
ISBN (Electronic): 978-1-4673-9724-7
Main Research Area: Technical/natural sciences
Conference: OCEANS '16 MTS/IEEE, Shanghai, China, 10/04/2016 - 10/04/2016
DOIs: 10.1109/OCEANSAP.2016.7485701
Source: PublicationPreSubmission
Source-ID: 123473638
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

Experimental and numerical study of wave-induced backfilling beneath submarine pipelines
This paper presents results of complementary experimental and numerical studies involving wave-induced backfilling of current-generated scour holes beneath submarine pipelines. The laboratory experiments are conducted in a wave-plus-current flume, utilizing Laser Doppler Anemometry to measure velocities, synchronized flow visualizations using digital image technology, along with live-bed scour and backfilling measurements. Each experiment is based on a two-stage process: (1) initial scour induced by a pure current, followed by; (2) backfilling induced by pure waves (either regular or irregular). The time series of scour depths are closely monitored through video recordings. Systematic analysis of these has resulted in aclosed form expression for the backfilling time scale, which is demonstrated to be a full order of magnitude greater than the well-known time scale of scour (with both governed primarily by the Shields parameter). The developed expression is strictly valid for the current-to-wave backfilling scenarios considered, while likely serving as an upper limit for more general wave-induced backfilling circumstances. The experiments are complemented by similar backfilling simulations utilizing a fully-coupled hydrodynamic and morphodynamic CFD model. The numerical simulations demonstrate the ability of the model to predict backfilling towards expected equilibrium scour depths based on the new wave climate, with time scales reasonably inline with experimental expectations.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Technical University of Denmark
Authors: Bayraktar, D. (Ekstern), Ahmad, J. (Ekstern), Eltard-Larsen, B. (Intern), Carstensen, S. (Intern), Fuhrman, D. R. (Intern)
Pages: 63–75
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Journal: Coastal Engineering
Volume: 118
ISSN (Print): 0378-3839
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Web of Science (2016): Indexed yes
Experimental and Numerical Study of Wave-Induced Backfilling Beneath Submarine Pipelines

Through complementary experimental and numerical efforts, the present paper aims to make a significant contribution to the overall understanding of backfilling processes beneath submarine pipelines. For this purpose, we aim to simplify the experimental backfilling process to an elementary two-stage process: (1) initial scour induced by a pure current, followed by: (2) backfilling induced by pure waves. A steady current is introduced via a re-circulating pump, and is kept constant...
with a cross-sectional velocity of \( V = 0.48 \) m/s until an initial equilibrium scour depth, \( S_0 \), is reached. Then, the current is stopped and waves (characterized by their Kculegan-Carpenter number \( KC \) and Shields parameter \( \theta \)) are introduced to initiate the backfilling process, which is maintained until a new equilibrium scour depth, \( S_f \), is reached. The time at which waves are introduced will be denoted as \( t = 0 \). For the backfilling process both regular and irregular waves are used during the experiments. As a demonstration of the initiated two-stage (scour followed by backfilling) process, bed profiles based on video recordings from a case having \( KC = 9.7 \) and \( \theta = 0.195 \) are depicted at selected stages in Figure 1. Figure 1 (upper left) depicts the current-induced equilibrium scour hole in the near vicinity of the pipe at \( t = 0 \), with the profile approximated as the dashed red line. Similarly, Figure 1 (upper right) depicts the new equilibrium scour profile (approximated as the full blue line) that has developed under wave-induced backfilling, corresponding to \( t = 60 \) min. To ease comparison, the dashed red and full blue lines from these plots are additionally combined onto Figure 1 (bottom). The experimental campaign has additionally been complemented with similar numerical simulations (using regular waves), based on a fully-coupled hydrodynamic and morphodynamic CFD model (Jacobsen et al., 2014), extending previous pipeline scour-related applications of Fuhrman et al. (2014) and Larsen et al. (2016). Comparison of the numerical and experimental results demonstrate the ability of the CFD model to reasonably simulate the current-to-wave backfilling process, both in terms of the achieved new wave induced equilibrium scour depths as well as the corresponding backfilling timescales. Figure 2 depicts a summary of both experimental and numerical backfilling time scale \( T_b \) versus Shields parameter \( \theta \). As can be seen, both experimental as well as numerical results match the regression equation: \( T_b = 0.3 \theta^{-5/3} \) quite closely (solid line in Figure 2).

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Coastal and Maritime Engineering, Technical University of Denmark
Authors: Bayraktar, D. (Ekstern), Ahmad, J. (Ekstern), Eiltard-Larsen, B. (Intern), Carstensen, S. (Intern), Fuhrman, D. R. (Intern)
Number of pages: 1
Publication date: 2016
Event: Abstract from 35th International Conference on Coastal Engineering (ICCE’16), Antalya, Turkey.
Main Research Area: Technical/natural sciences
Electronic versions: 1317.pdf
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2016

**Experimental investigation of the thickness effect for large as-welded SAW S355 steel specimens**

The presented work aims to investigate and establish a precise, thorough and detailed database from series of experimental testing of submerged arc welded (SAW) specimens of various thicknesses typically applied in ships and offshore structures and foundations. Welded structures of all sizes and shapes exhibit fatigue failure primarily in the welded region, rather than in the base material, due to imperfections and flaws relating to the welding procedure. The welded region has therefore received much attention from universities, research institutions along with industry as it is of significant practical importance for all fatigue loaded structures, such as e.g. marine structures.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Ölafsson, Ö. M. (Intern), Jensen, J. J. (Intern), Berggreen, C. (Intern)
Number of pages: 7
Publication date: 2016

**Host publication information**

Title of host publication: Proceedings of the 13th International Symposium on Practical Design of Ships and Other Floating Structures (PRADS'2016)
Publisher: Technical University of Denmark (DTU)
Editors: Dam Nielsen, U., Juncher Jensen, J.
Main Research Area: Technical/natural sciences
Conference: 13th International Symposium on Practical Design of Ships and Other Floating Structures (PRADS'2016), Copenhagen, Denmark, 04/09/2016 - 04/09/2016
Welded steel joints, Submerged arc welding (SAW), Fatigue testing, Thickness effect, S-N curves, Standard recommendations
Electronic versions: Experimental_Investigation_of_the_Thickness_Effect_for_Large_As_Welded_SAW_S355_Steel_Specimens.pdf. Embargo ended: 08/09/2016
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016
A recently developed BioEnergetic Growth (BEG) model for blue mussels (Mytilus edulis), valid for juvenile mussels, has been further developed to an 'extended model' and an alternative 'ad hoc BEG model' valid for post-metamorphic mussels, where the latter accounts for changing ambient chl a concentration. It was used to predict the growth of M. edulis on optimally thinned farm-ropes in Great Belt (Denmark), from newly settled post-metamorphic mussels of an initial shell size of 0.8 mm to marketable juvenile 30–35 mm 'mini-mussels'. Such mussels will presumably in the near future be introduced as a new Danish, smaller-sized consumer product. Field data for actual growth (from Day 0 = 14 June 2011) showed that size of 'mini-mussel' was reached on Day 109 (Oct 1) and length 38 mm on Day 178 (Dec 9) while the corresponding predictions using the extended model were Day 121 (Oct 13) and Day 159 (Nov 20). Similar results were obtained by use of the ad hoc BEG model which also demonstrated the sensitivity of growth prediction to levels of chl a concentration, but less to temperature. The results suggest that it is possible (when the conditions are optimal, i.e., no intraspecific competition ensured by sufficient thinning) to produce 'mini-mussels' in Great Belt during one season, but not the usual marketable 45-mm mussels. We suggest that the prediction model may be used as a practical instrument to evaluate to what degree the actual growth of mussels on farm ropes due to intraspecific competition may deviate from the potential (optimal) growth under specified chl a and temperature conditions, and this implies that the effect of thinning to optimize the individual growth by eliminating intraspecific competition can be rationally evaluated.
geometry. The effective wake field, i.e. the propeller inflow when it is running behind the ship, but excluding the propeller-induced velocities, can not be measured directly and only its mean value can be determined experimentally from self-propulsion tests.

In the present work the effective wake field is computed using a hybrid simulation method, known as RANS-BEM coupling, where the flow around the ship is computed by numerically solving the Reynolds-averaged Navier–Stokes equations, while the flow around the propeller is computed by a Boundary Element Method. The velocities induced by the propeller working behind the ship are known explicitly in such method, which allows to directly compute the complete effective flow field by subtracting the induced velocities from the total velocities. This offers an opportunity for additional insight into hull-propeller interaction and the propeller’s actual operating condition behind the ship, as the actual (effective) inflow is computed.

Self-propulsion simulations at model and full scale were carried out for a bulk carrier, once with a conventional propeller, and once with a Kappel propeller. However, in contrast to the experimental results, neither a significant difference in effective wake fraction nor other notable differences in effective flow were observed in the simulations. It is therefore concluded that the differences observed in model tests are not due to the different radial load distributions of the two propellers. One hypothesis is that the differences are a consequence of the geometry of the vortices shed from the propeller blades. The shape and alignment of these trailing vortices were modeled in a relatively simple way, which presumably does not reflect the differences between the propellers sufficiently.

Obtaining effective wake fields using the hybrid RANS-BEM approach at model and full scale also provides the opportunity to investigate the behind-ship cavitation performance of propellers with comparably low computational effort. The boundary element method for propeller analysis includes a partially nonlinear cavitation model, which is able to predict partial sheet cavitation and supercavitation. The cavitation behaviour of the conventional propeller and the Kappel propeller from the earlier simulations was investigated in the behind-condition using this model, focusing on the influence of the velocity distribution of the inflow field. Generally, the results agree well with experiments and the calculations are able to reproduce the differences between conventional and Kappel propellers seen in previous experiments. Nominal and effective wake fields at model and full scale were uniformly scaled to reach the same axial wake fraction, so that the only difference lies in the distribution of axial of velocities and in-plane velocity components. Calculations show that details of the velocity distribution have a major effect on propeller cavitation, signifying the importance of using the correct inflow, i.e. the effective wake field when evaluating propeller cavitation performance.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Regener, P. B. (Intern), Andersen, P. (Intern)
Number of pages: 111
Publication date: 2016

Impact of groyne fields on the littoral drift: A hybrid morphological modelling study
This paper concerns numerical modelling of the impact on the littoral drift and the shoreline from groynes forming a field of equidistant and identical groyne. The most important effect of a groyne on the shoreline morphology is that the littoral drift is blocked completely or partially. A local reduction in the littoral drift around the groyne introduces longshore gradients in the alongshore sediment transport and sedimentation and erosion around the groyne which will cause re-orientation of the bed contours towards the prevailing wave direction until an equilibrium is reached. A discussion of this mechanism is presented including effects of scales, e.g. the effect of the relative length of the groynes (compared to the width of the surf zone). The model results indicate a strong dependency of the reduction in littoral drift on the initial geometric bypass ratio \( Q_{geo} \), which is defined from the groyne length and the littoral transport on the undisturbed coastline \( Q_{geo} \). The transport occurring outside the tip of the groynes divided by the total transport. It is found that the sensitivity of the littoral drift to variations in groyne spacing and the angle, of the approaching waves, is inversely proportional to \( Q_{geo} \).

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DHI Denmark
Authors: Kristensen, S. E. (Ekstern), Drønen, N. (Ekstern), Deigaard, R. (Ekstern), Fredsøe, J. (Intern)
Improved design bases of welded joints in seawater

The presented work aims to investigate and establish a precise, thorough and detailed database from series of experimental testing of submerged arc welded, SAW, specimens of various thicknesses typically applied in offshore structures and foundations. Additionally, the testing was performed in two different environments, i.e. under in-air conditions and in a corrosion environment. Welded structures of all sizes and shapes exhibit fatigue failure primarily in the welded region, rather than in the base material, due to imperfections and flaws relating to the welding procedure. The welded region has therefore received much attention from universities, research institutions along with industry as it is of significant practical importance for all fatigue loaded structures, such as e.g. marine structures.

As-welded SAW specimens of three different thicknesses, manufactured by Lindoe Welding Technology A/S and Bladt Industries A/S, were subjected to uni-axial tension loading at relatively high R-ratios in order to simulate tensile residual stresses of yield magnitude. The main goal was to confirm the thickness effect for the specific case of large butt joints in the as-welded condition as well as to validate whether the thickness correction factor according to recommendations, codes and guidelines is too conservative when it comes to butt-welded joints. A conservative thickness effect factor results in larger, heavier and more expensive structures. The thickness effect considers the influence of the plate thickness on the fatigue resistance of welded joints and is generally included in design rules by scaling the fatigue strength with a recommended factor. The existing database of experiments that relate to the thickness effect is comprehensive and the effect is well proven experimentally and theoretically for various types of welded joints. However, in the case of large butt-welded joints there is room for improvement as details, quality and precise data which can influence the fatigue life of the welded joint is often lacking and severely lacking in truly thick joints.

Additionally, as-welded SAW specimens were tested in a corrosion environment with cathodic protection. The specimens were subjected to high fatigue loading at the same stress ratio as the tests performed in-air. A direct comparison to the specimens tested under in-air conditions was performed in order to evaluate the effects of the corrosion environment on fatigue resistance.

Furthermore, novelty 25 mm thick steel laser-hybrid welded specimens in the as welded condition were subjected to experimental testing. A fatigue resistance S-N curve was established for the laser hybrid welded joints in addition to a more detailed analysis. The laser hybrid welded joints were thereafter compared directly to the traditional SAW specimens in order to investigate the two different welding techniques. The laser hybrid welding technique offers great potential in lowering the cost of energy associated with offshore structures.
**Improved Wave-vessel Transfer Functions by Uncertainty Modelling**

This paper deals with uncertainty modelling of wave-vessel transfer functions used to calculate or predict wave-induced responses of a ship in a seaway. Although transfer functions, in theory, can be calculated to exactly reflect the behaviour of the ship when exposed to waves, uncertainty in input variables, notably speed, draft and relative wave heading, often compromises results. In this study, uncertainty is applied to improve theoretically calculated transfer functions, so they better fit the corresponding experimental, full-scale ones. Based on a vast amount of full-scale measurements data, it is shown that uncertainty modelling can be successfully used to improve accuracy (and reliability) of theoretical transfer functions.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Technical University of Denmark, Tokyo University of Marine Science and Technology
Authors: Nielsen, U. D. (Intern), Fønss Bach, K. (Ekstern), Iseki, T. (Ekstern)
Pages: 134-137
Publication date: 2016
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Nihon Kokai Gakkai Ronbunshu
Volume: 134
ISSN (Print): 0388-7405

Ratings:
- BFI (2018): BFI-level 1
- BFI (2017): BFI-level 1
- BFI (2016): BFI-level 1
- BFI (2015): BFI-level 1
- BFI (2014): BFI-level 1
- BFI (2013): BFI-level 1
- ISI indexed (2013): ISI indexed no
- BFI (2012): BFI-level 1
- ISI indexed (2012): ISI indexed no
- BFI (2011): BFI-level 1
- ISI indexed (2011): ISI indexed no
- BFI (2010): BFI-level 1
- BFI (2009): BFI-level 1
- BFI (2008): BFI-level 1

Original language: English
Navigational instrument and measurement, Transfer functions, Uncertainty modelling, Frequency-dependent model uncertainty factor, Improved prediction of short-term ship responses

Electronic versions:
- JIN_kouen_Nielsen_et_al._2016_c.pdf

Source: PublicationPreSubmission
Source-ID: 127367396
Publication: Research - peer-review › Journal article – Annual report year: 2016

**Investigations on the porous resistance coefficients for fishing net structures**

The porous media model has been successfully applied to numerical simulation of current and wave interaction with traditional permeable coastal structures such as breakwaters. Recently this model was employed to simulate flow through and around fishing net structures, where the unknown porous resistance coefficients were adjusted by fitting the available experimental data. In the present paper, a new approach was proposed to calculate the porous resistance coefficients based on the transformation of Morison type load model. The transformation follows the principle that the total forces acting on a net panel from Morison type load model should be equal to the forces obtained from the porous media model. In order to account for the interaction effects in-between the twines, two coefficients were introduced, and they were calibrated by minimizing the least square error function. Extensive validation cases were carried out to examine the performance of the numerical model. This includes steady current flow through plane net panels and circular fish cages, and wave interaction with plane net panels. A variety of fishing nets with different solidity ratios were used in the validation...
cases, from which it was seen that the overall agreement between the numerical and experimental results is fair.

**General information**
- **State:** Published
- **Organisations:** Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
- **Authors:** Chen, H. (Intern), Christensen, E. D. (Intern)
- **Pages:** 76-107
- **Publication date:** 2016
- **Main Research Area:** Technical/natural sciences

**Publication information**
- **Journal:** Journal of Fluids and Structures
- **Volume:** 65
- **ISSN (Print):** 0889-9746
- **Ratings:**
  - BFI (2018): BFI-level 1
  - Web of Science (2018): Indexed yes
  - BFI (2017): BFI-level 1
  - Web of Science (2017): Indexed yes
  - BFI (2016): BFI-level 1
  - Scopus rating (2016): SJR 1.316 SNIP 2.051 CiteScore 2.62
  - Web of Science (2016): Indexed yes
  - BFI (2015): BFI-level 1
  - Scopus rating (2015): SJR 1.277 SNIP 1.895 CiteScore 2.33
  - BFI (2014): BFI-level 1
  - Scopus rating (2014): SJR 1.428 SNIP 2.738 CiteScore 2.69
  - BFI (2013): BFI-level 1
  - Scopus rating (2013): SJR 1.582 SNIP 2.841 CiteScore 2.96
  - ISI indexed (2013): ISI indexed yes
  - BFI (2012): BFI-level 1
  - Scopus rating (2012): SJR 1.154 SNIP 2.356 CiteScore 2.14
  - ISI indexed (2012): ISI indexed yes
  - BFI (2011): BFI-level 1
  - Scopus rating (2011): SJR 1.386 SNIP 2.369 CiteScore 2.16
  - ISI indexed (2011): ISI indexed yes
  - Web of Science (2011): Indexed yes
  - BFI (2010): BFI-level 1
  - Scopus rating (2010): SJR 1.534 SNIP 2.711
  - BFI (2009): BFI-level 1
  - Scopus rating (2009): SJR 1.515 SNIP 2.366
  - BFI (2008): BFI-level 1
  - Scopus rating (2008): SJR 1.398 SNIP 2.688
  - Web of Science (2008): Indexed yes
  - Scopus rating (2007): SJR 1.385 SNIP 2.142
  - Web of Science (2007): Indexed yes
  - Scopus rating (2006): SJR 1.285 SNIP 2.934
  - Scopus rating (2005): SJR 1.205 SNIP 1.962
  - Scopus rating (2004): SJR 1.68 SNIP 1.703
  - Scopus rating (2003): SJR 0.766 SNIP 1.521
  - Scopus rating (2002): SJR 0.998 SNIP 1.792
  - Scopus rating (2001): SJR 1.206 SNIP 1.357
  - Web of Science (2001): Indexed yes
  - Scopus rating (2000): SJR 0.668 SNIP 1.377
  - Scopus rating (1999): SJR 1.332 SNIP 1.307
- **Original language:** English
- **Mechanical Engineering, Fishing nets, Navier-Stokes equations, Porous media model, Resistance coefficients**
- **DOIs:**
LES And URANS simulations of the swirling flow in a dynamic model of a uniflow-scavenged cylinder

The turbulent swirling flow in a uniflow-scavenged two-stroke engine cylinder is investigated using computational fluid dynamics. The investigation is based on the flow in a scale model with a moving piston. Two numerical approaches are tested; a large eddy simulation (LES) approach with the wall-adaptive local eddy-viscosity (WALE) model and a Reynolds-Averaged Navier-Stokes approach using the k-ω Shear-Stress Transport model. Combustion and compression are neglected. The simulations are verified by a sensitivity study and the performance of the turbulence models are evaluated by comparison with experimental results. Both turbulence models produce results in good agreement with experimental data. The agreement is particularly good for the LES, immediately after the piston passes the bottom dead center. Furthermore, in the piston standstill period, the LES predicts a tangential profile in agreement with the measurements, whereas the k-ω SST model predicts a solid body rotation. Several instabilities are identified during the scavenging process. The formation of a vortex breakdown with multiple helical vortex structures are observed after the scavenge port opening, along with the shedding of vortex rings with superimposed swirl. The turbulence models predict several flow reversals in the vortex breakdown region through the scavenge process. Flow separations in the scavenge ports lead to a secondary axial flow, in the separated region. The secondary flow exits in the top of the scavenge ports, resulting in large velocity gradients near the cylinder liner above the scavenge ports.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Hemmingsen, C. S. (Intern), Ingvorsen, K. M. (Intern), Mayer, S. (Ekstern), Walther, J. H. (Intern)
Pages: 213-223
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Volume: 62
ISSN (Print): 0142-727X
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.991 SNIP 1.691 CiteScore 2.35
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.175 SNIP 1.788 CiteScore 2.29
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.224 SNIP 2.066 CiteScore 2.3
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.17 SNIP 2.215 CiteScore 2.37
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.375 SNIP 2.174 CiteScore 2.28
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.328 SNIP 2.169 CiteScore 2.6
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 1.414 SNIP 2.108
BFI (2009): BFI-level 1
Longitudinal dispersion of heavy particles in an oscillating tunnel and application to wave boundary layers

The present research aims at getting an understanding of the process of dispersion of surface sediment in an oscillatory boundary layer, which may represent an idealised case of, for example, a stockpile area where excavated sediment is stockpiled temporarily (or permanently). The process is studied numerically, using a random-walk particle model with the input data for the mean and turbulence characteristics of the wave boundary layer picked up from a transitional two-equation k-ω Reynolds averaged Navier–Stokes model and plugged in the random-walk model. First, the flowmodel is validated against experimental data in the literature. Then, the random-walk dispersion model is run for different oscillatory flow cases and for a number of steady flow cases for comparison. The primary sediment grains of concern are fine sediments (with low fall velocity), which would stay in suspension for most of the time. Nevertheless, the dispersion of neutrally buoyant and heavier particles that spend most of their time in close vicinity to the bed are also discussed. The numerical model results are compared with the results of a series of experiments carried out in an oscillating U-tunnel facility. The results are found to be in general agreement both qualitatively and quantitatively. In the last part of the study, an example application of the present model for fine sand dispersing in a wave boundary layer under storm conditions is given.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Istanbul Technical University, Technical University of Denmark
Authors: Kirca, V. S. O. (Ekstern), Sumer, B. M. (Intern), Steffensen, M. (Ekstern), Jensen, K. L. (Intern), Fuhrman, D. R. (Intern)
Number of pages: 25
Pages: 59-83
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Ocean Engineering and Marine Energy
Volume: 2
Issue number: 1
ISSN (Print): 2198-6444
Original language: English
Longitudinal dispersion, Suspended sediment, Oscillatory boundary layer, Random-walk model, RANS modelling, U-tunnel experiments
Electronic versions:
Longitudinal_dispersion.pdf
DOIs:
10.1007/s40722-015-0039-x
Source: FindIt
Measurements of Cloaking Produced by an Array of Circular Cylinders

In the field of water waves, there has been recent interest in a process known as cloaking. According to this concept, the waves scattered by a fixed body are manipulated in the region immediately surrounding the object to eliminate scattering at large distances from the body. Cloaking of a bottom-mounted vertical monopile was first proposed by Porter (2011), who manipulated the bathymetry in the surrounding region to minimise the scattered energy to values approaching zero. Newman (2013) investigated cloaking of a surface-piercing cylinder of finite draft by introducing an array of bodies or an axisymmetric structure around the central object. In this case, a multivariate optimiser was successfully coupled with the boundary-integral equation software WAMIT to identify geometric arrangements resulting in minimal far-field wave scattering at a specified wavelength. The existence of arrays of bodies capable of cloaking has more recently been confirmed numerically by Kashiwagi et al. (2015) using the theory of Kagemoto and Yue (1986) in conjunction with a higher-order, boundary-element method. Kashiwagi et al have furthermore presented experimental results supporting the numerical findings for array-type cloaking. From a practical perspective, the elimination of the meandrift force on a surface-piercing structure associated with the cloaking process may find application in the reduction of second-order mooring loads in large offshore structures. This paper describes a further experimental investigation into the cloaking of a circular cylinder of finite draft using a circumferential array of eight surrounding cylinders. The aims of the work were: to identify a cloaking geometry suitable for investigation in the facilities available at DTU to design and build an apparatus to investigate wave scattering by this and other geometries, to confirm cloaking at one wavelength by measuring the mean drift force and far-field scattering.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Massachusetts Institute of Technology
Authors: Read, R. (Intern), Bingham, H. B. (Intern), Newman, J. N. (Ekstern)
Number of pages: 4
Publication date: 2016

Host publication information
Title of host publication: Proceedings of the 31th International Workshop on Water Waves and Floating Bodies (IWWWFB 2016)
Main Research Area: Technical/natural sciences
Workshop: 31th International Workshop on Water Waves and Floating Bodies (IWWWFB 2016), Plymouth, United States, 03/04/2016 - 03/04/2016
Electronic versions:
read_iwwwfb_31.pdf
Links:
http://www.iwwwfb.org/Workshops/31.htm
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2016

Modelling of Combustion and Pollutant Formation in a Large, Two-Stroke Marine Diesel Engine using Integrated CFD-Skeletal Chemical Mechanism

In this reported work, simulation studies of in-cylinder diesel combustion and pollutant formation processes in a two-stroke, low-speed uniflow-scavenged marine diesel engine are presented. Numerical computation is performed by integrating chemical kinetics into CFD computations. In order to minimize the computational runtime, an in-house skeletal n-heptane chemical mechanism is coupled with the CFD model. This surrogate fuel model comprises 89 reactions with 32 species essential to diesel ignition/combustion processes as well as the formation of soot precursors and nitrogen monoxide (NO). Prior to the marine engine simulation, coupling of the newly developed surrogate fuel model and a revised multi-step soot model [1] is validated on the basis of optical diagnostics measurement obtained at varying ambient pressure levels [2]. It is demonstrated that the variation of ignition delay times, liftoff lengths and averaged soot volume fraction (SVF) with respect to the change of ambient pressure captured using the model agree reasonably well with the measurement, apart from those at the low pressure condition. Numerical models are subsequently validated against experimental combustion characteristics under high load condition in a marine diesel engine. Comparisons to the measurement show that the simulated pressure rise started 1.0 crank angle degree in advance and the calculated peak pressure is 1.7 % lower. The associated flame liftoff length is negligible, yielding high local equivalence ratio and SVF values. In addition, the oxygen availability is found to affect the production of acetylene and hence soot particles. For the current test condition, the averaged NO concentration calculated when soot radiative heat loss is taken into account compared to that when only convective is considered suggests that the former is approximately 7.7 % lower. The findings here aid to gain insight of in-cylinder phenomena in this combustion system. The surrogate fuel model also allows direct couplings of sulfuric oxides formation reactions and a more comprehensive nitrogen oxides mechanism since the surrogate fuel model includes essential radicals such as O, H and OH for these pollutant formation reactions.
Modelling of diesel spray flame under engine-like conditions using an accelerated eulerian stochastic fields method: A convergence study of the number of stochastic fields

The use of transported Probability Density Function (PDF) methods allows a single model to compute the autoignition, premixed mode and diffusion flame of diesel combustion under engine-like conditions [1,2]. The Lagrangian particle based transported PDF models have been validated across a wide range of conditions [2,3]. Alternatively, the transported PDF model can also be formulated in the Eulerian framework[4]. The Eulerian PDF is commonly known as the Eulerian Stochastic Fields (ESF) model. When the same chemical mechanism and micro-mixing model were used, both ESF model and its Lagrangian counterpart generated similar results. The principal motivation for ESF compared to Lagrangian particle based PDF is the relative ease of implementation of the former into Eulerian computational fluid dynamics (CFD) codes [5]. Several works have attempted to implement the ESF model for the simulations of diesel spray combustion under engine-like conditions. The current work aims to further evaluate the performance of the ESF model in this application, with an emphasis on examining the convergence of the number of stochastic fields, $n_{sf}$. Five test conditions, covering both the conventional diesel combustion and low temperature combustion regimes, are used. The associated ambient conditions and injection characteristics are provided in Table 1.

Multiphase flow in porous media using CFD

We present results from a new Navier-Stokes model for multiphase flow in porous media implemented in Ansys Fluent 16.2 [1]. The model includes the Darcy-Forchheimer source terms in the momentum equations and proper account for relative permeability and capillary pressure in the porous media. This approach is widely used for single phase flow, but not for multiphase flow in porous media. This might be due to the complexity of introducing relative permeability and capillary pressure in the CFD solver. The introduction of relative permeability and capillary pressure may cause numerical instabilities as the saturation of a grid cell approaches the residual saturation, i.e. the relative permeability goes towards zero. This means that the viscous resistance in the Darcy-Forchheimer equation approaches infinity. Furthermore, by coupling the Navier-Stokes equation and Darcy-Forchheimer equation it is possible to model both the non-porous and porous media using the same formulation.
Patterns of vorticity in the wake of a single rectangular winglet (vortex generator) embedded in a turbulent boundary layer have been studied using Stereoscopic Particle Image Velocimetry (SPIV). The winglet was mounted normally to a flat surface with an angle to the oncoming flow. A parametric study varying the winglet height (constant aspect ratio) and angle has shown, contrary to the common classical single tip-vortex conception, that the wake generally consists of a complex system of multiple vortex structures. The primary vortex has previously been discovered to contain a direct coupling between the axial and the rotational flow. In the current work, even the longitudinal secondary structures detected from measured streamwise vorticity display similar behavior. A regime map depicting the observed stable far wake states of the multiple vortices as a function of winglet height and angle reveals complex patterns of the flow topologies not only with the primary tip vortex, but with the additional secondary structures as well. A bifurcation diagram shows distinct regimes of the various secondary structures as well as how the primary vortex is in some cases significantly affected by their presence. These data should serve as inspiration in the process of generating longitudinal vortices for enhancement of heat and mass transfer in industrial devices since the multiple vortex regimes can help improve the conditions for these exchanges. Further, these results point to a weakness in existing inviscid models not accounting for the possibility of multiple vortical structures in the wake. © 2015 Elsevier Inc. All rights reserved.
Pulse-jet cleaning and understanding of the complex physics are essential when designing fabric filters used for air pollution control. Today, low-pressure cleaning is of particular interest due to demand for reduced compressed air consumption. Pulse-jet cleaned fabric filters have been studied for many years by experimental investigation and to a limited extent by Computational Fluid Dynamics (CFD). The majority of the studies have focused on high-pressure cleaning systems, and the CFD models presented are so far two-dimensional (2D). In the work presented here, pulse-jet cleaning of low-pressure fabric filters (2 bar) is studied using a full three-dimensional (3D) CFD model. Experimental results obtained in a pilot-scale test filter with 28 bags, in length of 10 m and in general full-scale dimensions of the cleaning system are used to verify the reliability of the present CFD model. The validated CFD model reveals the strong compressible effects, a highly transient behaviour, the formation of compressible vortex rings and the shock cell phenomenon within the overexpanded supersonic jet. The cleaning nozzles and venturi design aid or oppose the pulse-pressure within the bags, and this plays an important role in the resulting efficiency of removing the dust layer from the bags. The CFD simulation shows that the traditional straight-bore nozzles provide substantial misalignment of the jet, and the add-on nozzle design offers only limited improvement. Further, the need for venturis in low-pressure filters and the importance of optimising the venturi design are demonstrated. The working principle of the venturi is to restrict backflow which is detrimental to the pressure rise in the bags. Reducing the venturi throat diameter is shown to reduce backflow and improve the pulse-pressure.

**Numerical and experimental study of pulse-jet cleaning in fabric filters**

Pulse-jet cleaning and understanding of the complex physics are essential when designing fabric filters used for air pollution control. Today, low-pressure cleaning is of particular interest due to demand for reduced compressed air consumption. Pulse-jet cleaned fabric filters have been studied for many years by experimental investigation and to a limited extent by Computational Fluid Dynamics (CFD). The majority of the studies have focused on high-pressure cleaning systems, and the CFD models presented are so far two-dimensional (2D). In the work presented here, pulse-jet cleaning of low-pressure fabric filters (2 bar) is studied using a full three-dimensional (3D) CFD model. Experimental results obtained in a pilot-scale test filter with 28 bags, in length of 10 m and in general full-scale dimensions of the cleaning system are used to verify the reliability of the present CFD model. The validated CFD model reveals the strong compressible effects, a highly transient behaviour, the formation of compressible vortex rings and the shock cell phenomenon within the overexpanded supersonic jet. The cleaning nozzles and venturi design aid or oppose the pulse-pressure within the bags, and this plays an important role in the resulting efficiency of removing the dust layer from the bags. The CFD simulation shows that the traditional straight-bore nozzles provide substantial misalignment of the jet, and the add-on nozzle design offers only limited improvement. Further, the need for venturis in low-pressure filters and the importance of optimising the venturi design are demonstrated. The working principle of the venturi is to restrict backflow which is detrimental to the pressure rise in the bags. Reducing the venturi throat diameter is shown to reduce backflow and improve the pulse-pressure.

**General information**

State: Published  
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, F.L. Smidth A/S  
Authors: O. Andersen, B. (Ekstern), Nielsen, N. F. (Ekstern), Walther, J. H. (Intern)  
Pages: 284-298  
Publication date: 2016  
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Powder Technology  
Volume: 291  
ISSN (Print): 0032-5910  
Ratings:  
BFI (2018): BFI-level 1  
Web of Science (2018): Indexed yes  
BFI (2017): BFI-level 1  
Web of Science (2017): Indexed yes  
BFI (2016): BFI-level 1
Numerical investigation of soot formation and oxidation processes under large two-stroke marine diesel engine-like conditions using integrated CFD-chemical kinetics

In this reported work, multi-dimensional computational fluid dynamics studies of diesel combustion and soot formation processes in a constant volume combustion chamber and a marine diesel engine are carried out. The key interest here is firstly to validate the coupling of a newly developed skeletal n-heptane mechanism and a revised multi-step soot model using laser extinction measurements of diesel soot obtained at different ambient pressure levels in an optical accessible, constant volume chamber experiment. It is revealed that ignition delay times and liftoff lengths generated using the new skeletal model are close to those produced by the larger and more comprehensive chemical mechanisms, apart from those at the low pressure condition. The current study also demonstrates that the variation of averaged soot volume fraction with respect to the change of combustion chamber pressure captured using the revised soot model agrees
reasonably well with the measurements in terms of peak values. The numerical model is subsequently applied to
investigate the flame development, soot/nitrogen monoxide formation and heat transfer in a two-stroke, low-speed uniflow-
scavenged marine diesel engine operating at full load condition, where optical measurements are not available.
Comparisons to the experimental data show that the simulated pressure rise starts 1.0 crank angle degree in advance and
the calculated peak pressure is 1.7% lower. The associated flame liftoff length is negligible, yielding higher local
equivalence ratio and soot volume fraction values as compared to those under similar test condition in the constant
volume chamber. With the use of the revised model, the total heat transfer to the walls calculated when soot radiative heat
loss is taken into account is approximately 30% higher compared to that when only convective heat loss is considered.
The averaged nitrogen monoxide concentration is 7.7% lower when both convective and soot radiative heat losses are
accounted for but the net soot mass production is less sensitive to soot radiation. A sensitivity study reveals that neither
increasing nor decreasing the soot absorption coefficient by 30% from the baseline setup is influential to nitrogen
monoxide formation, soot mass production and heat transfer. The findings here aid to gain insights and provide a better
understanding of the combustion and soot processes in large, uniflow-scavenged marine engines. The numerical model
developed in this work can also be applied to explore different phenomena in this combustion system.
Numerical Simulation of Condensation of Sulfuric Acid and Water in a Large Two-stroke Marine Diesel Engine

We present results from computational fluid dynamics simulations of the condensation of sulfuric acid (H₂SO₄) and water (H₂O) in a large two-stroke marine diesel engine. The model uses a reduced n-heptane skeletal chemical mechanism coupled with a sulfur subset to simulate the combustion process and the formation of SOₓ and H₂SO₄. Condensation is modeled using a fluid film model coupled with the Eulerian in-cylinder gas phase. The fluid film condensation model is validated against both experimental and numerical results. The engine simulations reveal that the fluid film has a significant effect on the sulfuric acid gas phase. A linear correlation is found between the fuel sulfur content and the sulfuric acid condensation rate. The initial in-cylinder water content is found not to affect the sulfuric acid condensation but it has a high impact on water condensation. The scavenging pressure level shows an inverse correlation between pressure and condensation rate due to change in the flame propagation speed. Finally, increasing the cylinder liner temperature significantly decreases water condensation but has a negligible influence on the condensation of sulfuric acid.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy
Authors: Walther, J. H. (Intern), Karvounis, N. (Intern), Pang, K. M. (Intern)
Number of pages: 1
Publication date: 2016

Host publication information
Title of host publication: Bulletin of the American Physical Society
Volume: 61
Publisher: American Physical Society
Article number: E26.00005
Main Research Area: Technical/natural sciences
Electronic versions:
MWS_DFD16_2016_001768.pdf
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2016

Numerical simulation of tsunami-scale wave boundary layers

This paper presents a numerical study of the boundary layer flow and properties induced by tsunami-scale waves. For this purpose, an existing one-dimensional vertical (1DV) boundary layer model, based on the horizontal component of the incompressible Reynolds-averaged Navier–Stokes (RANS) equations, is newly extended to incorporate a transitional
variant of the standard two-equation $k$--$\omega$ turbulence closure. The developed numerical model is successfully validated against recent experimental measurements involving transient solitary wave boundary layers as well as for oscillatory flows, collectively demonstrating the ability to reproduce accurate velocity profiles, turbulence, and bed shear stresses on both smooth and rough beds. The validated model is then employed for the study of transient wave boundary layers at full tsunami scales, covering a wide and realistic geophysical range in terms of the flow duration, bottom roughness, and associated Reynolds numbers. For this purpose, three different “synthetic” (idealised) tsunami wave descriptions are considered i.e., invoking: (1) single wave (solitary-like, but with independent period and wave height),(2) sinusoidal, and (3) N-wave descriptions. The flow, boundary layer thickness, turbulence, and bed shear stresses induced are systematically monitored and parameterised, under both hydraulically smooth and roughbed conditions. The results generally support a notion that the boundary layers induced by tsunami-scale waves are both current-like, due to their long durations, as well as wave-like, in the sense that the boundary layer will not necessarily span the entirety of the water column. The results likewise suggest that there is a continuum connecting wind-wave and tsunami-wave scales, as existing expressions commonly used for characterising boundary layer properties beneath wind-waves maintain reasonable accuracy when extrapolated to full tsunami scales. Boundary layers driven by actual field-measured tsunami signals are likewise simulated, stemming from both the 2004 Indian Ocean as well as the 2011 Tohoku events. These results are reconciled with the various synthetic tsunami signals considered, generally confirming their usefulness as idealised tsunami waves.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Utrecht University
Authors: Williams, I. A. (Ekstern), Fuhrman, D. R. (Intern)
Pages: 17-31
Publication date: 2016
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Coastal Engineering
Volume: 110
ISSN (Print): 0378-3839

Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.925 SNIP 2.097 CiteScore 2.9
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.785 SNIP 2.123 CiteScore 2.55
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.727 SNIP 2.264 CiteScore 2.58
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.912 SNIP 2.226 CiteScore 2.21
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.616 SNIP 2.502 CiteScore 2.43
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 1.898 SNIP 2.332
Web of Science (2010): Indexed yes
On nonlinear wave-structure interaction using an immersed boundary method in 2D

Introduction
We present our progress on the development and preliminary benchmarking results of a new efficient methodology for solving fully non-linear potential flow wave-structure interaction problems. The new model utilises the efficiency of finite difference methods on structured grids. The structure geometry is introduced using an Immersed Boundary Method (IBM) and the body boundary condition (BC) is satisfied with a Weighted Least Squares (WLS) approximation [7]. This allows complex geometries to be represented with high accuracy. The stability of the scheme is ensured by adopting the Weighted Essentially Non-Oscillatory (WENO) scheme [8] together with a Lax-Friedrichs type flux applied to the free surface conditions in Hamilton-Jacobi form. This work can be viewed as a novel extension of the flexible order finite difference potential flow solver OceanWave3D [2] to include the presence of a structure. The method obtains an optimum scaling of the solution effort [2] and has been implemented on massively parallel GPU architectures using the CUDA API [3] making it suitable for high resolution flow simulations. This combination of novel and robust numerical methods aims at creating new efficient tools for non-linear wave-structure interaction problems. The scheme is validated using the forced heaving motion of a two-dimensional (2D) horizontal circular cylinder with promising results, although there are still challenges to be overcome in terms of properly capturing the behavior of the intersection between the body and the free-surface.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Applied Mathematics and Computer Science, Scientific Computing
Authors: Kontos, S. (Intern), Bingham, H. B. (Intern), Lindberg, O. (Intern), Engsig-Karup, A. P. (Intern)
Number of pages: 4
Publication date: 2016

Host publication information
Title of host publication: Proceedings of the 31th International Workshop on Water Waves and Floating Bodies (IWWWFB 2016)
Pipeline–Seabed Interaction

A review of the existing research on the interaction between a pipeline and an erodible bed exposed to waves and/or currents is presented. The review covers three topics, i.e., scour, liquefaction and lateral stability of pipelines. The basic mechanism that leads to scour in two-dimensional (2D) and 3D case is firstly described. The scour processes are deduced from small-scale laboratory experiments. The onset of scour from piping and the developing tunnel erosion are among the processes described. The lateral expansion of the scour hole along the pipe is described, also based primarily on small-scale laboratory experiments. The state of the art of the mathematical/numerical modeling of the scour processes is presented. The associated self-burial of the pipe is described and compared to field observations. In addition to scour, liquefaction may also constitute a risk for pipeline stability. The cause of liquefaction and the resulting consequence for pipeline stability in a natural environment are then discussed. Finally, the lateral stability of pipelines placed on an erodible bed and exposed to waves is briefly described.
Pre-Swirl Stator and Propeller Design for Varying Operating Conditions

Over the last two decades, an increasing number of studies have been conducted to develop and improve energy saving devices (ESDs) in order to increase the propulsive efficiency. One well-known example is the pre-swirl stator (PSS), which consists of an often asymmetric arrangement of fixed stator blades ahead of the propeller. This paper describes the hydrodynamic design of a pre-swirl stator with radially variable pitch, paired with a conventional propeller. The aim is to achieve the highest possible efficiency in various operating conditions, and to avoid efficiency penalties in off-design operation. To investigate the propeller and stator designs and configurations in different operating conditions, the computationally inexpensive vortex-lattice method is used as a first step to optimize the geometry in an initial parameter study. Then the flow over hull, stator and propeller is simulated in a CFD-based approach to confirm the results obtained in the first stage.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Saettone, S. (Intern), Regener, P. B. (Intern), Andersen, P. (Intern)
Number of pages: 8
Publication date: 2016
Reconstruction of 3D flow structures in a cylindrical cavity with a rotating lid

The flow in a cylindrical cavity with a rotating lid has been studied for many years, e.g. by Sørensen et al. (2006). It contains general flow phenomena like vortex breakdown and in some cases the breakdown is accompanied by multihelix vortices (Okulov et al., 2010). This type of flow phenomenon is difficult to capture experimentally since the flow is fully three-dimensional and also varies in time. A measurement in a point or in a plane will by itself not give the full picture of the flow. Measurement with Particle Image Velocimetry (PIV) analyzed with Proper Orthogonal Decomposition (POD) is a promising method of reconstructing the full three-dimensional, time-varying flow structures. This has been attempted in Meyer et al. (2008) and Meyer et al. (2009). The analyzed measurements show both that the vortex breakdown in some cases is asymmetrical (rotating around the cylinder axis) and that the presence of helical vortices can be detected. However, the interpretation of the resulting flow still is done with an element of guessing on whether a specific variation is caused by an actual time variation of a structure or is caused by the rotation of a three-dimensional structure. The present work will also be based on time-resolved stereoscopic PIV measurements in a vertical plane through the cylinder axis as shown in figure 1. Compared to Meyer et al. (2008) the measurements will be expanded by adding measurements in several points outside the PIV data plane with a Laser Doppler Anemometer (LDA). LDA has a very good time resolution and the synchronized PIV and LDA measurements will therefore resolve the ambiguity in the interpretation of PIV data with respect to whether the flow variations are caused by rotation of a three-dimensional structure or is a real transient phenomenon.

Reducing uncertainty of Monte Carlo estimated fatigue damage in offshore wind turbines using FORM

Uncertainties related to fatigue damage estimation of non-linear systems are highly dependent on the tail behaviour and extreme values of the stress range distribution. By using a combination of the First Order Reliability Method (FORM) and Monte Carlo simulations (MCS), the accuracy of the fatigue estimations may be improved for the same computational efforts. The method is applied to a bottom-fixed, monopilesupported large offshore wind turbine, which is a non-linear and dynamically sensitive system. Different curve fitting techniques to the fatigue damage distribution have been used depending on the sea-state dependent response characteristics, and the effect of a bi-linear S-N curve is discussed. Finally, analyses are performed on several environmental conditions to investigate the long-term applicability of this multistep method. Wave loads are calculated using state-of-the-art theory, while wind loads are applied with a simplified model based on rotor thrust coefficients.
Robust Numerical Methods for Nonlinear Wave-Structure Interaction in a Moving Frame of Reference

This project is focused on improving the state of the art for predicting the interaction between nonlinear ocean waves and marine structures. To achieve this goal, a flexible order finite difference potential flow solver has been extended to calculate for fully nonlinear wave-structure interaction problems at forward speed.

The model utilises the efficiency of finite difference methods on structured grids and exploits the flexibility of a novel Immersed Boundary Method (IBM) based on Weighted Least Squares (WLS) for the approximation of the no-flux boundary condition on the body surface. As a result, the grid generation is very simple and the need for regridding when considering moving body problems is avoided. The temporal oscillations related to the IBM method and moving boundaries are minimized by sufficient spatial resolution and an increased time-step size.

The time-dependant physical domain is mapped to a time-invariant computational domain with a sigma transformation. For a smooth and continuous transformation a $C^2$ continuous free surface is required over the entire domain. Thus, an artificial free surface that respects this property is created in the interior of the body using a seventh order polynomial. The forward speed problem is formulated in a moving coordinate system attached to the mean position of the body. Robust approximations for all combinations of forward speed and wave velocity are obtained by expressing the free surface boundary conditions in Hamilton-Jacobi form and using a Weighted Essentially Non-Oscillatory (WENO) scheme for the convective derivatives. The linear WENO weights are derived with a new procedure that is suitable for numerical implementation and avoids the limitations of existing tabulated WENO coefficients. Furthermore, a simplified smoothness indicator that performs as well as the tabulated versions is proposed. Explicit high-order Runge-Kutta time integration and a Lax-Friedrichs-type numerical flux complete the scheme. The solver was tested on the two-dimensional zero speed wave radiation problem and the steady forward speed problem with satisfactory results and thus, the proof of concept for extending the methodology to three dimensions is established.
Selection of the optimum combination of responses for Wave Buoy Analogy - An approach based on local sensitivity analysis

One method to estimate the wave spectrum onboard ships is to use measured ship responses. In this method, known also as Wave Buoy Analogy, amongst various responses that are available from sensor measurements, a couple of responses (at least three) are usually utilized. Selection of the best combination of ship responses is important. Optimally, this selection should not be implemented manually onboard applications. Therefore, availability of an automatic response selection procedure would be a great advantage for decision support. In this paper, a local sensitivity analysis is applied to evaluate the importance of individual responses in sea state estimation. The sensitivity factor is defined by calculation of the partial derivatives of wave parameters with respect to the variance of individual responses.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Norwegian University of Science and Technology
Authors: Montazeri, N. (Intern), Nielsen, U. D. (Intern), Jensen, J. J. (Intern)
Number of pages: 8
Publication date: 2016

Host publication information
Title of host publication: Proceedings of the 13th International Symposium on PRActical Design of Ships and Other Floating Structures (PRADS' 2016)
Publisher: Technical University of Denmark (DTU)
Editors: Dam Nielsen, U., Juncher Jensen, J.
Main Research Area: Technical/natural sciences
Conference: 13th International Symposium on Practical Design of Ships and Other Floating Structures (PRADS'2016), Copenhagen, Denmark, 04/09/2016 - 04/09/2016
Wave Buoy Analogy, Response selection, Sensitivity analysis, Derivatives, Sensitivity Factors
Electronic versions:
Source: PublicationPreSubmission
Source-ID: 124374937
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

Sensitivity analyses of biodiesel thermo-physical properties under diesel engine conditions

This reported work investigates the sensitivities of spray and soot developments to the change of thermo-physical properties for coconut and soybean methyl esters, using two-dimensional computational fluid dynamics fuel spray modelling. The choice of test fuels made was due to their contrasting saturation-unsaturation compositions. The sensitivity analyses for non-reacting and reacting sprays were carried out against a total of 12 thermo-physical properties, at an ambient temperature of 900 K and density of 22.8 kg/m$^3$. For the sensitivity analyses, all the thermo-physical properties were set as the baseline case and each property was individually replaced by that of diesel. The significance of individual thermo-physical property was determined based on the deviations found in predictions such as liquid penetration, ignition delay period and peak soot concentration when compared to those of baseline case. Among all the properties, latent heat of vaporisation produced the greatest effect on the spray and soot developments under the tested conditions, as evidenced by a longer liquid penetration of 35.0% and a reduced peak soot concentration of 22.8%. Besides, coupled effects among the thermo-physical properties were also determined. Meanwhile, the effects of thermo-physical properties were found to vary with the addition of unsaturation levels and chemical kinetics in the simulation.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy, University of Nottingham, Malaysia Campus
Authors: Cheng, X. (Ekstern), Ng, H. K. (Ekstern), Gan, S. (Ekstern), Ho, J. H. (Ekstern), Pang, K. M. (Intern)
Pages: 341-352
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Journal: Energy
Simulating the DISAMATIC process using the discrete element method — a dynamical study of granular flow

The discrete element method (DEM) is applied to simulate the dynamics of the flow of green sand while filling a mould using the DISAMATIC process. The focus is to identify relevant physical experiments that can be used to characterize the material properties of green sand in the numerical model. The DEM parameters describing the static friction coefficients are obtained using a ring shear tester and the rolling resistance and cohesion value is subsequently calibrated with a sand pile experiment. The calibrated DEM model is used to model the sand shot in the DISAMATIC process for three different sand particle flow rates as captured on the corresponding video footage of the interior of the chamber. A mould chamber with three ribs mounted on the fixed pattern plate forming four cavities is chosen as a reference geometry to investigate the conditions found in the real moulding process. The geometry of the cast part and the casting system can make the moulding process complicated due to obstacles such as ribs that deflect the sand flow causing "shadows effects" around the cavities of the mould. These dynamic effects are investigated by the qualitative flow dynamics and quantitative mould filling times captured in the video footage and simulated by the calibrated DEM model. Both two- and three-dimensional DEM models are considered and found to produce results in good agreements with the video footage of the DISAMATIC process.

General information
State: Published
Organisations: Department of Mechanical Engineering, Manufacturing Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DISA Industries A/S
Authors: Hovad, E. (Intern), Spangenberg, J. (Intern), Larsen, P. (Ekstern), Walther, J. H. (Intern), Thorborg, J. (Intern), Hattel, J. H. (Intern)
Pages: 228-240
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Journal: Powder Technology
Volume: 303
ISSN (Print): 0032-5910
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 3.16 SJR 0.983 SNIP 1.482
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.965 SNIP 1.598 CiteScore 2.99
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.89 SNIP 1.649 CiteScore 2.67
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.901 SNIP 1.875 CiteScore 2.64
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.854 SNIP 1.826 CiteScore 2.36
ISI indexed (2012): ISI indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 0.921 SNIP 1.86 CiteScore 2.45
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 0.94 SNIP 1.547
Simulation of bluff-body flows using iterative penalization in a multiresolution particle-mesh vortex method

The ability to predict aerodynamic forces, due to the interaction of a fluid flow with a solid body, is central in many fields of engineering and is necessary to identify error-prone structural designs. In bluff-body flows the aerodynamic forces oscillate due to vortex shedding and variations in the oncoming flow. This may lead to structural instability e.g. when the shedding frequency aligns with the natural frequency of the structure. Fluid structure interaction must especially be considered when designing long span bridges. A three dimensional vortex-in-cell method is applied for the direct numerical simulation of the flow past a bodies of arbitrary shape. Vortex methods use a simple formulation where only the trajectories of discrete vortex particles are simulated. The

Lagrangian formulation eliminates the CFL type condition that Eulerian methods have to satisfy. This allows vortex methods to take significantly larger time steps in convection dominated flows with explicit time integration.

As vorticity is a bounded quantity and the velocity field can be calculated for freespace- or periodic boundary conditions, these method allows for a minimized domain and hence minimize computational efforts.

Pure particle-vortex methods have the disadvantage of being highly costly. The calculation of particle velocities in particle vortex methods has traditionally been done by directly applying the Biot-Savart law yielding an N2-body problem. However the

Poisson equation, that relates the vorticity- to the velocity field, can be solved efficiently using a mesh-based solver with local refinement in the boundary layer regions. We present a higher-order particle-mesh vortex method, where particle velocities are calculated by solving the Poisson equation on several uniform meshes using Fast Fourier Transforms. This we combine with an iterative penalization method, that allows the simulation of external flows past arbitrary geometries in arbitrary motions such as bridge decks in forced heave and pitch motion.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Spietz, H. J. (Intern), Hejlesen, M. M. (Intern), Walther, J. H. (Intern)
Number of pages: 1
Publication date: 2016
Event: Abstract from ECCOMAS Congress 2016, Hersonissos, Greece.
Main Research Area: Technical/natural sciences
Simulation of impulsively started flow past a sphere and a disc using iterative brinkman penalization

We present an iterative Brinkman penalization scheme to enforce the no-slip condition on solid boundaries in three-dimensional flow simulations. We use a high-order particle-mesh-vortex method, where the velocity field is obtained from the vorticity field by solving a Poisson equation on a Cartesian mesh as a convolution of the vorticity field with a regularized Green's function [2]. By doing this we can enforce free-space boundary conditions allowing us to consider a minimal computational domain. The Brinkman penalization method [1] is an immersed body method that allows the treatment of solids having complex geometries on a Cartesian mesh. Thereby we avoid the use of unstructured meshes that conventional flow solvers rely on. In the presented iterative scheme the penalization term is only active in the solid region and in its immediate neighborhood thus the computational costs required for the solution of the penalization problem is kept at a minimum. We apply our method for the simulation of the impulsively started flow past a sphere at Re = 1000 and normal to a circular disc at Re = 500, respectively. Our results for the unsteady sphere flow are found to be in qualitative good agreement with results obtained by Ploumhans et al. [3] using a boundary element method. The flow is illustrated by a volume rendering of the vorticity field cf. Fig. 1. The figure shows that the flow is highly unsteady and a challenging problem for accurate analysis. Furthermore we show that the iterative scheme allows a significantly larger time step than the iterative scheme (more than a factor of 10) in the simulation of the impulsively started flow normal to a circular disc.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Spietz, H. J. (Intern), Hejlesen, M. M. (Intern), Walther, J. H. (Intern)
Number of pages: 2
Publication date: 2016
Event: Abstract from 7th International Conference on Vortex Flows and Vortex Models (ICVFM 2016), Rostock, Germany.
Main Research Area: Technical/natural sciences
Vortex method, 3D flow, Brinkman penalization, Sphere, disc

Simulation of Wave-Plus-Current Induced Scour Beneath Submarine Pipelines

Scour beneath submarine pipelines has been the subject of much past research see eg. Sumer and Fredsøe (2002). To date most research, both numerical and experimental, has focused on scour induced by either pure waves or currents, while comparatively few studies have involved combined wave-plus-current environments. The present study, which is published in Larsen et al. (2016) focuses on the numerical simulation of wave-plus-current induced scour beneath submarine pipelines, based on a model solving Reynolds-averaged Navier-Stokes (RANS) equations, fully coupled with turbulence closure, bed and suspended load sediment transport descriptions, and a seabed morphological model. The model was utilized in simulating breaker bar development by Jacobsen et al. (2014) and has been used in simulating wave induced scour beneath pipelines by Fuhrman et al. (2014).

The model is utilized for the numerical study of combined wave-plus-current scour processes beneath pipelines. The results of 77 simulated wave-plus-current scour cases will be presented and analysed. The cases considered will consist of waves characterized by 10 different Keulegan-Carpenter numbers, KC=U_m T/D and up to eight different values of m=U_m/(U_c+U_m) which defines the relative strength of the current i.e. m=0 corresponds to pure-wave conditions, with m=1 corresponding to pure-current conditions. The resulting equilibrium scour depths are shown to be in accordance with existing experimentally-based expressions. In Figure 1 the time scale of the scour process is compared to the existing empirical formula from Fredsøe et al. (1992) for the time scale governing both the wave and current-induced scour, T*=1/50 θ_w^-5/3. In Figure 1 θ_w can be interpreted as the maximum Shields parameter of the combined wave-current flow. As can be seen the modelled time scales generally agree well for the pure wave and pure current cases whereas the time scales for combined waves and current are generally larger. The time scales are systematically reinvestigated and a new general expression for the time scale is proposed. The expression is fully consistent with existing experimentally based relations at both pure-current and pure-wave limits, and is appropriate for engineering use. In Figure 2 the model results are compared to the new general expression which is of the form T*=F(m) θ_{cw}^-5/3, where F(m) is a relatively simple close form expression. The tight clustering, compared with Figure 1 around the full line in Figure 2, demonstrates that the generalized expression effectively unites the time scales for pure-current, pure-wave, as well as the combined wave-plus-current flows.

General information
Simulation of Wave-Plus-Current Scour beneath Submarine Pipelines

A fully coupled hydrodynamic and morphologic numerical model was utilized for the simulation of wave-plus-current scour beneath submarine pipelines. The model was based on incompressible Reynolds-averaged Navier–Stokes equations, coupled with k-ω turbulence closure, with additional bed and suspended load descriptions forming the basis for seabed morphology. The model was successfully validated against experimental measurements involving scour development and eventual equilibrium in pure-current flows over a range of Shields parameters characteristic of both clear-water and live-bed regimes. This validation complements previously demonstrated accuracy for the same model in simulating pipeline scour processes in pure-wave environments. The model was subsequently utilized to simulate combined wave-plus-current scour over a wide range of combined Keulegan–Carpenter numbers and relative current strengths. The resulting equilibrium scour depths and trends were shown to be in accordance with existing experimentally based expressions from the literature. The variety of scour profile types emerging under various flow conditions is detailed and reconciled with experimental observations. The resulting matrix of scour depth time series was systematically analyzed, resulting in a new generalized expression for the scour time scale in combined wave-plus-current flow environments. This expression is fully consistent with existing experimentally based relations at both pure-current and pure-wave limits and is appropriate for engineering use.
Tangential Inlet supersonic separators: a novel apparatus for gas purification

A novel supersonic separator with a tangential inlet is designed to remove the condensable components from gas mixtures. The dynamic parameters of natural gas in the supersonic separation process are numerically calculated using the Reynolds stress turbulence model with the Peng-Robinson real gas model. The results show that natural gas expands in the supersonic separator to supersonic velocities resulting in low pressures (6 bar, from about 40 bar) and temperatures (-70 °C, from 30 °C), which causes the condensation and nucleation of the condensable components. The tangential velocity can be generated by the tangential inlet, and it increases to the maximum of 200 m/s at the nozzle throat due to decrease of the nozzle area of the converging part. The tangential velocity can maintain the value of about 160 m/s at the nozzle exit, and correspondingly generates the centrifugal acceleration of 3.6×10^6 m/s^2 to remove the condensed droplets from the gas mixtures.
Testing of self-similarity and helical symmetry in vortex generator flow simulations

Vortex generators (VGs) are used increasingly by the wind turbine industry as flow control devices to improve rotor blade performance. According to experimental observations, the vortices generated by VGs have previously been observed to be self-similar for both the axial (uz) and azimuthal (uθ) velocity components. Furthermore, the measured vortices have been observed to obey the criteria for helical symmetry. These are powerful results, as it reduces the highly complex 3-D flow to merely four parameters and therefore significantly facilitates the modeling of this type of flow, which in a larger perspective can assist in parametric studies to increase the total power output of wind turbines. In this study, corresponding computer simulations using Reynolds-averaged Navier–Stokes equations have been carried out and compared with the experimental observations. The main objective is to investigate how well the simulations can reproduce these aspects of the physics of the flow, i.e., investigate if the same analytical model can be applied and therefore significantly facilitate the modeling of this type of flow, which in a larger perspective can assist in parametric studies to increase the total power output of wind turbines. This is especially interesting since these types of flows are notoriously difficult for the turbulence models to predict correctly. Using this model, parametric studies can be significantly reduced, and moreover, reliable simulations can substantially reduce the costs of the parametric studies themselves. Copyright © 2015 John Wiley & Sons, Ltd.
The Effects on the Operating Condition of a Passenger Ship Retro-fitted with a Composite Superstructure

As sustainability and climate change have come on the political agenda, the shipping industry will have to be operating energy efficient ships. An appealing step to achieve this goal is by designing superstructures made out of Fiber Reinforced Plastics (FRP) aiming at the reduction of the ship's lightship weight. The benefits of a light superstructure become most prominent in large passenger ships, as the superstructures constitute a significant percentage of the lightship. Additionally, depending on the size of the ship, the superstructure may tower several decks above the weather deck, affecting the stability of the ship. In this work, the superstructure of a RoPax ferry has been redesigned using composite materials emphasizing the effects on the ship from an operational perspective. The weight reduction has been calculated for a realistic average operating condition quantifying the effects on the stability and the fuel consumption of the retrofitted ship compared to the original design.

General information
State: Published
Organisations: Department of Mechanical Engineering, Solid Mechanics, Fluid Mechanics, Coastal and Maritime Engineering, Niels Hjørnet Yacht Design
Authors: Karatzas, V. (Intern), Hjørnet, N. K. (Ekstern), Kristensen, H. O. H. (Intern), Berggreen, C. (Intern), Jensen, J. J. (Intern)
Number of pages: 5
Publication date: 2016

Host publication information
Title of host publication: Proceedings of the 13th International Symposium on Practical Design of Ships and Other Floating Structures (PRADS'2016)
Publisher: Technical University of Denmark (DTU)
Editors: Dam Nielsen, U., Juncher Jensen, J.
The influence of fully nonlinear wave forces on aero-hydro-elastic calculations of monopile wind turbines

The response of an offshore wind turbine tower and its monopile foundation has been investigated when exposed to linear and fully nonlinear irregular waves on four different water depths. The investigation focuses on the consequences of including full nonlinearity in the wave kinematics. The linear and nonlinear irregular wave realizations are calculated using the fully nonlinear potential flow wave model OceanWave3D [1]. The linear and nonlinear wave realizations are compared using both a static analysis on a fixed monopile and dynamic calculations with the aeroelastic code Flex5 [2]. The conclusion from this analysis is that linear wave theory is generally sufficient for estimating the fatigue loading, but wave nonlinearity is important in determining the ultimate design loads.

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Schløer, S. (Intern), Bredmose, H. (Intern), Bingham, H. B. (Intern)
Pages: 162-188
Publication date: 2016
Main Research Area: Technical/natural sciences

Publication information
Journal: Marine Structures
Volume: 50
ISSN (Print): 0951-8339
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.49 SJR 1.655 SNIP 2.636
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.618 SNIP 2.602 CiteScore 2.77
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.431 SNIP 3.026 CiteScore 2.18
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.314 SNIP 2.871 CiteScore 2.42
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.836 SNIP 3.464 CiteScore 1.76
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.611 SNIP 2.795 CiteScore 1.82
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
Thermophoresis of water droplets inside carbon nanotubes

Carbon Nanotubes (CNTs) offer unique possibilities as fluid conduits with applications ranging from lab on a chip devices to encapsulation media for drug delivery. CNTs feature high mechanical strength, chemical and thermal stability and biocompatibility therefore they are promising candidates for nanodevice fabrication. Thermal gradients have been proposed as mechanism to drive particles, fullerenes and droplets inside CNTs. Here, by conducting Molecular Dynamics (MD) simulations, we study thermophoresis of water droplets inside CNTs. We systematically change the size of the droplets, the axial thermal gradient and CNT chirality. We find that the droplet motion in the armchair CNTs exhibits two clearly delimited stages, a regime wherein the droplet is accelerated and subsequently, a regime wherein the droplet moves with constant velocity. Inside the zig zag CNTs, the droplet accelerates during a very short time and then it moves with constant velocity. We compute the net force during the droplet acceleration and find a correlation between the droplet acceleration and the magnitude of the thermal gradient without any dependence on the droplet size. Moreover, we conduct velocity constrained MD simulations to determine the friction and thermophoretic forces acting on the droplet.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Zambrano, H. (Ekstern), Walther, J. H. (Intern), Oyarzua, E. (Ekstern), Rojano, A. (Ekstern)
Number of pages: 1
Publication date: 2016

Host publication information
Title of host publication: Bulletin of the American Physical Society
Volume: 61
Publisher: American Physical Society
Article number: E22.00006
Main Research Area: Technical/natural sciences
Tsunami Induced Scour Around Monopile Foundations

While the run-up, inundation, and destructive potential of tsunami events has received considerable attention in the literature, the associated interaction with the sea bed i.e. boundary layer dynamics, induced sediment transport, and resultant sea bed morphology, has received relatively little specific attention. The present paper aims to further the understanding of tsunami-induced scour, by numerically investigating tsunami-induced flow and scour processes around a monopile structure, representative of those commonly utilized as offshore wind turbine foundations. The simulations are based on a model solving Reynolds averaged Navier-Stokes (RANS) equations, fully coupled with turbulence closure, bed and suspended load sediment transport descriptions, and a seabed morphological model. The model was developed and utilized in simulating breaker bar morphology by Jacobsen et al. (2014); It has been additionally been used in simulating wave induced scour beneath pipelines by Fuhrman et al. (2014) and Larsen et al. (2016) as well as scour around a monopile by Baykal et al. (2015).

Due to the large computational expenses it is presently only feasible to simulate such scour processes around a monopile at model (laboratory) spatial and temporal scales. Therefore, prior to conducting such numerical simulations involving tsunami-induced scour, it is necessary to first establish a methodology for maintaining similarity of model and full field scales. To achieve hydrodynamic similarity we will select the flow parameters such that we maintain similarity in terms of the diameter-based Froude number, as well as the boundary layer thickness-to-diameter ratio δ/D. Equating the Froude number ensures that the adverse pressure gradient induced by the presence of the structure itself will be similar at both model and field scales, i.e. that the ratio of the excess stagnation pressure head in front of the monopile will be maintained. Similarly, by maintaining similarity in δ/D, we ensure that the relative size of the horseshoe vortex, which is expected to largely govern the scouring process, will be similar at both model and full scales. This strategy also yields reasonable similarity in the expected tsunami period-to-scour time scale ratio.

As an example, three full tsunami periods have been simulated in succession, taking a full scale period of 13 min. Snapshots of the computed scour hole at selected times when the flow is rightward (left sub-plots) as well as leftward (right sub-plots) are depicted Figure 1. These snapshots illustrate the generally stepwise build-up of scour on the two opposing sides of the monopile during each successive half-cycle of the simulated tsunami. A complementary, simple and practical engineering method for predicting tsunami-induced scour is likewise developed, founded upon existing experimentally-based expressions for use in steady current scour, but invoking the boundary layer thickness and Shields parameter expected from tsunami wave events i.e. it effectively combines both current-like and wave-like properties of tsunamis (see e.g. Williams and Fuhrman, 2016).

Ultrafast cooling by covalently bonded graphene-carbon nanotube hybrid immersed in water

The increasing power density and the decreasing dimensions of transistors present severe thermal challenges to the design of modern microprocessors. Furthermore, new technologies such as three-dimensional chip-stack architectures require novel cooling solutions for their thermal management. Here, we demonstrate, through transient heat-dissipation simulations, that a covalently bonded graphene-carbon nanotube (G-CNT) hybrid immersed in water is a promising solution for the ultrafast cooling of such high-temperature and high heat-flux surfaces. The G-CNT hybrid offers a unique platform to integrate the superior axial heat transfer capability of individual CNTs via their parallel arrangement. The immersion of the G-CNT in water enables an additional heat dissipation path via the solid-liquid interaction, allowing for the sustainable cooling of the hot surface under a constant power input of up to 10 000 W cm⁻².
Uniform asymptotic approximations for transient waves due to an initial disturbance

In this work, we first present a semi-analytical method for the evolution of linear fully-dispersive transient waves generated by an initial surface displacement and propagating over a constant depth. The procedure starts from Fourier and Hankel transforms and involves a combination of the method of stationary phase, the method of uniform asymptotic approximations and various Airy integral formulations. Secondly, we develop efficient convolution techniques expressed as single and double summations over the source area. These formulations are flexible, extremely fast and highly accurate even for the dispersive tail of the transient waves. To verify the accuracy of the embedded dispersion properties, we consider test cases with sharp-edged disturbances in 1D and 2D. Furthermore, we consider the case of a relatively blunt Gaussian disturbance in 2D. In all cases the agreement between the convolution results and simulations with a high-order Boussinesq model is outstanding. Finally, we make an attempt to extend the convolution methods to geophysical tsunami problems taking into account e.g. uneven bottom effects. Unfortunately, refraction/diffraction effects cannot easily be incorporated, so instead we focus on the incorporation of linear shoaling and its effect on travel time and temporal evolution of the surface elevation. The procedure is tested on data from the 2011 Japan tsunami. Convolution results are likewise compared to model simulations based on the nonlinear shallow water equations and both are compared with field observations from 10 deep water DART buoys. The near-field results are generally satisfactory, while the far-field results leave much to be desired.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Schäffer Waves, Tel Aviv University
Authors: Madsen, P. A. (Intern), Schäffer, H. A. (Ekstern), Fuhrman, D. R. (Intern), Toledo, Y. (Ekstern)
Pages: 60–84
Publication date: 2016
Main Research Area: Technical/natural sciences
基于势流理论和OpenFOAM的耦合模型对多向不规则波浪的模拟

基于势流理论的数值水池可以快速计算波浪的传播及其对建筑物的作用，但是势流理论是基于波浪的无黏性假设的。在现实中，通常需要在固体边界及波浪破碎的区域考虑黏性效应。针对基于求解Navier-Stokes方程的黏性水池计算量较大、速度较慢的缺点，采用耦合的方法模拟波浪的传播，即在外域通过基于势流理论的数值水池产生多向不规则波浪，内域采用求解Navier-Stokes方程和流体体积法（VOF方法）对自由表面进行追踪，通过外域所提供的边界波浪，内域可以在较小区域进行计算，从而达到减少计算量、提高计算效率的目的。

The numerical wave tank based on the potential theory simulates the wave progression and wave-structure interaction quickly, but it is based on the assumption of non-viscous. In reality, viscous effects are always considered at solid boundaries and wave-broken areas. For the viscous numerical wave basin based on solving the Navier-Stokes equation, however, it has the insufficient of large amount of calculation. Currently, the coupling method is used to simulate wave progression to reduce the amount of calculation and enhance the efficiency. In the outer domain, multidirectional irregular waves are generated from potential theory, and in the inner domain, the Navier-Stokes equation is solved in combination the VOF method to track the free surface.
A high-order finite-difference linear seakeeping solver tool for calculation of added resistance in waves

During recent years a computational strategy has been developed at the Technical University of Denmark for numerical simulation of water wave problems based on the high-order finite-difference method, [2],[4]. These methods exhibit a linear scaling of the computational effort as the number of grid points increases. This understanding is being applied to develop a tool for predicting the added resistance (drift force) of ships in ocean waves. We expect that the optimal scaling properties of this solver will allow us to make a convincing demonstration of convergence of the added resistance calculations based on both near-field and far-field methods. The solver has been written inside a C++ library known as Overture [3], which can be used to solve partial differential equations on overlapping grids based on the high-order finite-difference method. The resulting code is able to solve, in the time domain, the linearised potential flow forward-speed hydrodynamic problems; namely the steady, radiation and diffraction problems. The near-field formulation of the wave drift force has also been implemented, and development is under way to include far-field methods. This paper presents validation results based on analytical solutions for exact geometries.
A high order multi-resolution solver for the Poisson equation with application to vortex methods

A high order method is presented for solving the Poisson equation subject to mixed free-space and periodic boundary conditions by using fast Fourier transforms (FFT). The high order convergence is achieved by deriving mollified Green’s functions from a high order regularization function which provides a correspondingly smooth solution to the Poisson equation. The high order regularization function may be obtained analogous to the approximate deconvolution method used in turbulence models and strongly relates to deblurring algorithms used in image processing. At first we show that the regularized solver can be combined with a short range particle-particle correction for evaluating discrete particle interactions in the context of a particle-particle particle-mesh (P3M) method. By a similar approach we extend the regularized solver to handle multi-resolution patches in continuum field simulations by super-positioning an inter-mesh correction. For sufficiently smooth vector fields this multi-resolution correction can be achieved without the loss of convergence rate. An implementation of the multi-resolution solver in a two-dimensional re-meshed particle-mesh based vortex method is presented and validated.

Analyses of Current And Wave Forces on Velocity Caps

Velocity caps are often used in connection with for instance offshore intake sea water for the use of for cooling water for power plants or as a source for desalination plants. The intakes can also be used for river intakes. The velocity cap is placed on top of a vertical pipe. The vertical pipe leads the water into another pipe or tunnel system. A pressure gradient generated by the water level difference between the sea and basin drives the flow through the tunnel system. The tunnel system is often in the order of a couple kilometers long. Based on CFD analyses (computational fluid dynamics) this paper investigates the current and wave forces on the velocity cap and the vertical cylinder. The Morison’s force model was used in the analyses of the extracted force time series in from the CFD model. Further the distribution of the inlet velocities around the velocity cap was also analyzed in detail in the wave case.

Atomistic Simulations of Fluid Flow through Graphene Channels and Carbon Nanotubes

The transport of aqueous solutions in artificial nanopores is of both fundamental and technological interest. Recently, carbon nano-structured materials (fullerenes) have attracted a great deal of attention in nanotechnology. In fact, due to their large specific surface area, high thermal conductivity, extremely low surface friction and superior mechanical...
properties, graphene channels and carbon nanotubes (CNTs) are promising candidates to be implemented as fluid conduits in nanosystems. Performing Non-equilibrium Molecular Dynamics simulations, we study the transport of water-electrolyte solutions inside single and multi-wall graphene channels and inside zig-zag and armchair CNTs of similar cross sectional area. In order to calibrate the force fields, we use dedicated criteria relevant to the hydrodynamics of the systems of interest. Different fluid driving mechanisms such as pressure fields, electro-osmosis and thermal gradients are evaluated. We conduct a detailed analysis of the transport efficiency of each system to impose similar volumetric flow rates. From the simulations, we extract density and velocity profiles to study the liquid structure, wall slippage and flow enhancement in order to compare the hydrodynamic performance of these two novel materials.

Atomistic study of a nanometer-scale pump based on the thermal ratchet concept
In this study, a novel concept of nanoscale pump fabricated using Carbon Nanotubes (CNTs) is presented. The development of nanofluidic systems provides unprecedented possibilities for the control of biology and chemistry at the molecular level with potential applications in low energy cost devices, novel medical tools, and a new generation of sensors. CNTs offer a number of attractive features for the fabrication of fluidic nanodevices including fast flow, useful electronic and thermal properties, high mechanical strength and biocompatibility. Therefore, the transport of liquids in CNTs is now of great interest in nanofluidics. Thermophoresis is the phenomenon observed when a mixture of two or more types of motile objects experience a force induced by a thermal gradient and the different types of objects respond to it differently, inducing a motion and segregation of the objects. Using molecular dynamics simulations, we explore the possibility to design thermophoretic pumping devices fabricated of CNTs for water transport in nanoconduits. The design of the nanopumps is based on the concept of the Feynman-Smoluchowski ratchet.
journal bearing is the converging-diverging geometry making it prone to cavitation which again affects the load carrying capacity of the bearing. In combustion engines the journal bearing plays a vital role especially as main and crosshead bearings transmitting the combustion forces. Those forces vary highly during one combustion cycle which is further influencing the load carrying capacity and ultimately the chances of fatal shaft-sleeve contact. By solving Reynolds equation numerically using finite elements and incorporating a cavitation algorithm, the dynamic coefficients can be used to establish the journal orbit for a given bearing and load pattern. Validation of the results is done against the Ruston and Hornsby 6VEB-X Mk III engine. Besides the numerical investigations a cavitation test rig has been developed. With this rig it is possible to generate cavitation under controlled conditions in terms of load/eccentricity and rotational speed. The development of cavitation in time in terms of position and distribution can be visually recorded.

General information
State: Published
Organisations: Department of Mechanical Engineering, Solid Mechanics, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Christiansen, C. K. (Intern), Klit, P. (Intern), Walther, J. H. (Intern), Vølund, A. (Ekstern)
Pages: 28-35
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Tribologia
Volume: 33
Issue number: 1
ISSN (Print): 0780-2285
Ratings:
Scopus rating (2016): SJR 0.141 SNIP 0.076 CiteScore 0.24
Scopus rating (2015): SJR 0.101 SNIP 0
Scopus rating (2014): SJR 0.316 SNIP 0.126
Scopus rating (2013): SJR 0.118 SNIP 0.019
ISI indexed (2013): ISI indexed no
Scopus rating (2012): SJR 0.24 SNIP 0.312
ISI indexed (2012): ISI indexed no
Scopus rating (2011): SJR 0.237 SNIP 0.103
ISI indexed (2011): ISI indexed no
Scopus rating (2010): SJR 0.125 SNIP 0.055
Scopus rating (2009): SJR 0.459 SNIP 0.37
Scopus rating (2008): SJR 0.13 SNIP 0.228
Scopus rating (2007): SJR 0.126 SNIP 0.186
Scopus rating (2006): SJR 0.159 SNIP 0.438
Scopus rating (2005): SJR 0.181 SNIP 0.429
Scopus rating (2004): SJR 0.104 SNIP 0.479
Scopus rating (2003): SJR 0.227 SNIP 0.277
Scopus rating (2002): SJR 0.101 SNIP 0
Scopus rating (2001): SJR 0.221 SNIP 0
Scopus rating (2000): SJR 0.238
Scopus rating (1999): SJR 0.119
Original language: English
Cavitation, Journal bearing, Finite Element, Journal orbit, Test rig
Electronic versions:
Tribologia_2015_Christiansen_et_al.pdf
Source: PublicationPreSubmission
Source-ID: 110963233
Publication: Research - peer-review › Journal article – Annual report year: 2015

CFD analysis of cascade effects in marine propellers with trailing edge modification
Propeller blades are different from a single hydrofoil in isolation due to cascade effects that blades mutually affect hydrodynamic characteristics of each other in proximity. Propeller design programs based lifting-line theory and blade element momentum theory take into account cascade effect by using cascade correction theory, which has been developed on the basis of wind tunnel tests for a row of evenly spaced airfoils. Cascade effects of marine propellers have been on research by inviscid flow solvers such as boundary element methods and vortex lattice methods, but it has not been investigated intensively by viscous flow solvers, although RANS CFD is prevalent in marine industry nowadays. In
the current work, the cascade effect of a marine propeller is analyzed by CFD simulations on a threedimensional propeller model with varying the number of blades. The influence of trailing-edge configurations on the cascade effect is also investigated by simulating CFD with varying trailingedge thickness and slope. The reason why the trailingedge is handled rather than other parts of blade geometry is that it can be modified without altering overall blade thrust significantly, because the loading on the aft part of a blade section near a trailing edge is relatively low, compared to the other part.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Shin, K. W. (Ekstern), Andersen, P. (Intern)
Number of pages: 5
Publication date: 2015

Host publication information
Title of host publication: Proceedings of the 18th Numerical Towing Tank Symposium 2015 (NuTTS 2015)
Publisher: Curran Associates
Main Research Area: Technical/natural sciences
Publication: Research - peer-review › Article in proceedings – Annual report year: 2016

CFD analysis of cloud cavitation on three tip-modified propellers with systematically varied tip geometry
The blade tip loading is often reduced as an effort to restrain sheet and tip vortex cavitation in the design of marine propellers. This CFD analysis demonstrates that an excessive reduction of the tip loading can cause cloud cavitation responsible for much of noise and surface erosion. Detached eddy simulations (DES) are made for cavitating flows on three tip-modified propellers, of which one is a reference propeller having an experimental result from a cavitation tunnel test with a hull model, and the other two are modified from the reference propeller by altering the blade tip loading. DES results have been validated against the experiment in terms of sheet and cloud cavitation. In DES, non-uniform hull wake is modelled by using the inlet flow and momentum sources instead of including a hull model. A 4-bladed Kappel propeller with a smooth tip bending towards the suction side is used as the reference propeller. For the reference propeller, sheet cavitation extends over a whole chord length in the hull wake peak. As the blade gets out of the wake peak, the rear part of sheet cavity is detached in a form of cloud cavitation. For the reference propeller, the tip pitch reduction from the maximum is about 35%. When decreasing the tip pitch reduction to 10%, tip vortex cavitation is formed and cloud cavitation is significantly weakened. When increasing the tip pitch reduction to 60%, sheet cavitation slightly moves to inner radii and cloud cavitation grows larger.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Shin, K. W. (Ekstern), Andersen, P. (Intern)
Number of pages: 5
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Physics: Conference Series
Volume: 656
Article number: 012139
ISSN (Print): 1742-6596
Ratings:
BFI (2018): BFI-level 1
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 0.45 SJR 0.24 SNIP 0.383
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.24 SNIP 0.373 CiteScore 0.35
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.253 SNIP 0.344 CiteScore 0.32
Web of Science (2014): Indexed yes
Collisions damage assessment of ships and jack-up rigs

Ship collision with offshore installations is one of the key concerns in design and assess of platforms performance and safety. This paper presents an analysis on collision energy and structural damage in ship and offshore platform collisions for various collision scenarios. The platform or rig is treated as either rigid or flexible and its sensitivity on collision energy and structural damage is studied. An application example where an ice-strengthened supply vessel collides against a jack-up rig is analysed and the crushing resistance of the involved thin-walled structures is evaluated.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register EMEA, Lloyd's Register Asia
Authors: Zhang, S. (Ekstern), Pedersen, P. T. (Intern), Ocakli, H. (Ekstern)
Pages: 470-478
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Ships and Offshore Structures
Volume: 10
Issue number: 5
ISSN (Print): 1744-5302
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
Conditional Stochastic Processes Applied to Wave Load Predictions

The concept of conditional stochastic processes provides a powerful tool for evaluation and estimation of wave loads on ships and offshore structures. This article first considers conditional waves with a focus on critical wave episodes. Then the inherent uncertainty in the results is illustrated with an application where measured wave responses are used to predict the future variation in the responses within the next 5-30 seconds. The main part of the article is devoted to the application of the First Order Reliability Method for derivation of critical wave episodes for different nonlinear wave-induced responses. A coupling with Monte Carlo simulations is shown to be able to give uniform accuracy for all exceedance levels with moderate computational time, even for rather complex nonlinear problems. The procedure is illustrated by examples dealing with overturning of jackup rigs, parametric rolling of ships, and slamming and whipping vibrations.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, J. J. (Intern)
Number of pages: 10
Pages: 1-10
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
JOURNAL OF SHIP RESEARCH
Volume: 59
Issue number: 1
ISSN (Print): 0022-4502
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 1.6 SJR 1.196 SNIP 1.832
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.585 SNIP 1.553 CiteScore 1.12
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.004 SNIP 1.835 CiteScore 1.19
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.576 SNIP 1.648 CiteScore 1.18
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.805 SNIP 1.689 CiteScore 0.97
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.146 SNIP 1.79 CiteScore 1.05
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 0.478 SNIP 0.906
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 0.571 SNIP 1.01
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 0.782 SNIP 1.219
Scopus rating (2007): SJR 0.921 SNIP 1.718
Scopus rating (2006): SJR 0.69 SNIP 1.526
Scopus rating (2005): SJR 0.594 SNIP 2.312
Scopus rating (2004): SJR 1.17 SNIP 1.6
Web of Science (2004): Indexed yes
Scopus rating (2003): SJR 1.086 SNIP 1.899
Web of Science (2003): Indexed yes
Scopus rating (2002): SJR 0.307 SNIP 1.1
Web of Science (2002): Indexed yes
Scopus rating (2001): SJR 0.793 SNIP 1.22
Scopus rating (2000): SJR 0.815 SNIP 1.339
Web of Science (2000): Indexed yes
Scopus rating (1999): SJR 0.537 SNIP 1.643
Original language: English
ENGINEERING, JACK-UP UNITS, EXTREME WAVE, RESPONSE PREDICTIONS, DESIGN, RELIABILITY, SIMULATIONS, SHIPS, FORM, wave loads, stochastic processes, Monte Carlo simulations
DOIs:
10.5957/JOSR.59.1.140060
Source: FindIt
Source-ID: 274567883
Publication: Research - peer-review › Journal article – Annual report year: 2015
Continuum Navier-Stokes modelling of water flow past fullerene molecules
We present continuum simulations of water flow past fullerene molecules. The governing Navier-Stokes equations are complemented with the Navier slip boundary condition with a slip length that is extracted from related molecular dynamics simulations. We find that several quantities of interest as computed by the present model are in good agreement with results from atomistic and atomistic-continuum simulations at a fraction of the computational cost. We simulate the flow past a single fullerene and an array of fullerenes and demonstrate that such nanoscale flows can be computed efficiently by continuum flow solvers, allowing for investigations into spatiotemporal scales inaccessible to atomistic simulations.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, National Institute of Chemistry, Swiss Federal Institute of Technology
Authors: Walther, J. H. (Intern), Popadic, A. (Ekstern), Koumoutsakos, P. (Ekstern), Praprotnik, M. (Ekstern)
Number of pages: 1
Publication date: 2015
Event: Abstract from 68th Annual Meeting of the American Physical Society's Division of Fluid Dynamics (DFD), Boston, United States.
Main Research Area: Technical/natural sciences
Source: PublicationPreSubmission
Source-ID: 118265458
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2015

Continuum Navier-Stokes modelling of water ow past fullerene molecules
We present continuum simulations of water flow past fullerene molecules. The governing Navier-Stokes equations are complemented with the Navier slip boundary condition with a slip length that is extracted from related molecular dynamics simulations. We find that several quantities of interest as computed by the present model are in good agreement with results from atomistic and atomistic-continuum simulations at a fraction of the computational cost. We simulate the flow past a single fullerene and an array of fullerenes and demonstrate that such nanoscale flows can be computed efficiently by continuum flow solvers, allowing for investigations into spatiotemporal scales inaccessible to atomistic simulations.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, National Institute of Chemistry, Swiss Federal Institute of Technology
Authors: Popadic, A. (Ekstern), Praprotnik, M. (Ekstern), Koumoutsakos, P. (Ekstern), Walther, J. H. (Intern)
Pages: 2321-2330
Publication date: 2015
Main Research Area: Technical/natural sciences

Continuum simulations of water flow past fullerene molecules
We present continuum simulations of water flow past fullerene molecules. The governing Navier-Stokes equations are complemented with the Navier slip boundary condition with a slip length that is extracted from related molecular dynamics simulations. We find that several quantities of interest as computed by the present model are in good agreement with results from atomistic and atomistic-continuum simulations at a fraction of the cost. We simulate the flow past a single fullerene and an array of fullerenes and demonstrate that such nanoscale flows can be computed efficiently by continuum flow solvers, allowing for investigations into spatiotemporal scales inaccessible to atomistic simulations.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, National Institute of Chemistry, Swiss Federal Institute of Technology
Authors: Popadic, A. (Ekstern), Praprotnik, M. (Ekstern), Koumoutsakos, P. (Ekstern), Walther, J. H. (Intern)
Pages: 2321-2330
Publication date: 2015
Main Research Area: Technical/natural sciences
Covalently Bonded Graphene-Carbon Nanotube Hybrid for High-Performance Thermal Interfaces

The remarkable thermal properties of graphene and carbon nanotubes (CNTs) have been the subject of intensive investigations for the thermal management of integrated circuits. However, the small contact area of CNTs and the large anisotropic heat conduction of graphene have hindered their applications as effective thermal interface materials (TIMs). Here, a covalently bonded graphene–CNT (G-CNT) hybrid is presented that multiplies the axial heat transfer capability of individual CNTs through their parallel arrangement, while at the same time it provides a large contact area for efficient heat extraction. Through computer simulations, it is demonstrated that the G-CNT outperforms few-layer graphene by more than 2 orders of magnitude for the c-axis heat transfer, while its thermal resistance is 3 orders of magnitude lower than the state-of-the-art TIMs. We show that heat can be removed from the G-CNT by immersing it in a liquid. The heat transfer characteristics of G-CNT suggest that it has the potential to revolutionize the design of high-performance TIMs.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, ETH Zurich
Authors: Chen, J. (Ekstern), Walther, J. H. (Intern), Koumoutsakos, P. (Ekstern)
Pages: 7539-7545
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Advanced Functional Materials
Volume: 25
Issue number: 48
ISSN (Print): 1616-301X
Ratings:
Web of Science (2018): Indexed yes
Web of Science (2017): Indexed Yes
Scopus rating (2016): CiteScore 11.56
Web of Science (2016): Indexed yes
Scopus rating (2015): CiteScore 11.93
Web of Science (2015): Indexed yes
Scopus rating (2014): CiteScore 11.32
Web of Science (2014): Indexed yes
Scopus rating (2013): CiteScore 10.6
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
Scopus rating (2012): CiteScore 10.41
ISI indexed (2012): ISI indexed yes
Scopus rating (2011): CiteScore 9.47
ISI indexed (2011): ISI indexed no
Web of Science (2010): Indexed yes
Web of Science (2009): Indexed yes
Web of Science (2008): Indexed yes
Web of Science (2007): Indexed yes
Original language: English
DOIs:
10.1002/adfm.201501593
Source: FindIt
Source-ID: 2289061026
Publication: Research - peer-review › Journal article – Annual report year: 2015

D5.10 - Interaction of the tsunami with the seabed. Implications for wind farms, aquaculture, coastal ecosystems and marine protected areas.

In Chapter 1, a fully-coupled (hydrodynamic and morphologic) numerical model is presented, and utilized for the simulation of tsunami-induced scour around a monopile structure, representative of those commonly utilized as offshore wind turbine foundations at moderate depths i.e. for depths <O(30 m). The model is based on solutions to Reynolds-averaged Navier-Stokes equations, coupled with two-equation k-u turbulence closure, with additional bed and suspended load descriptions forming the basis for sea bed morphology. The model is first validated for flow, bed shear stresses, and scour within a steady current, where a generally excellent match with experimentally-based results is found. A methodology for maintaining and assessing hydrodynamic and morphologic similarity between field and (laboratory) model-scale tsunami events is then presented, combining diameter-based Froude number similarity with that based on the dimensionless wave boundary layer thickness-to-monopile diameter ratio. This methodology is utilized directly in the
selection of governing tsunami wave parameters (i.e. velocity magnitude and period) used for subsequent simulation within the numerical model. The flow, sediment transport, and scour processes beneath three tsunami waves are simulated in succession. These illustrate a generally accumulative scour process i.e. a relatively rapid scour induced by the leading wave, with an additional buildup of the scour depth during additional trailing waves. The resulting scour seems to approach an equilibrium value after sufficient time duration, which corresponds reasonably to that predicted by existing steady-current scour depth, after invoking a boundary layer thickness based on the unsteady tsunami wave, i.e. it is important to incorporate both current-like, as well as wave-like aspects of the long tsunami event. Based on the simulated results, a simple methodology for predicting the scour depth in engineering practice is finally developed. This methodology is demonstrated to match the predicted maximum scour for all of the simulated flows considered i.e. ranging from the series of transient tsunami waves to the steady-current limit. In Chapter 2, the aim is to provide an overview on the tsunami impacts on aquaculture rather than presenting a comprehensive review on the status and trends in aquaculture development. [For such a comprehensive review the reader is referred to the FAO (Food and Agriculture organization of the United Nations) report titled “The State of the World Fisheries and Aquaculture” released in May 2014.] For this purpose, we first briefly provide and introductory summary on aquaculture. This is followed by the section “Vulnerability of Fisheries and Aquaculture Systems” where the main focus is the vulnerability to tsunamis. Next, tsunami Impacts on aquaculture are exemplified based on the major tsunami events that occurred since 2000s. Later, specific case studies highlighting different aspects in aquaculture design are illustrated in the section "Engineering Design of Aquaculture Systems". In Chapter 3, tsunami impact on coastal ecosystems is investigated. Ecosystems along the coast of Portugal are considered and a detailed numerical modelling of tsunami impact is performed for the Ria Formosa lagoon (an important ecosystem located in the southern coast of Portugal). The tsunami modelling is carried out using a validated non-linear shallow water numerical code. A high resolution digital elevation model (50m-resolution) of the zone of interest is used to properly simulate the tsunami hazard. The active earthquake sources of the southwest Iberia Margin (SWIM) region represent the tsunamigenic scenarios in this study. Tsunami impact at the Ria Formosa lagoon is assessed through deriving near-shore tsunami wave heights, inundations, and flow velocities. Numerical results show that the Ria Formosa lagoon can suffer powerful tsunami impact due to the occurrence of a tsunami event in the SWIM region.
D5.3 Interaction between currents, wave, structure and subsoil

This chapter gives an introduction to deliverable D5.3 - Interaction between currents, waves, structure and subsoil – with respect to the MERMAID project. The deliverable focuses on the conditions in European waters such as the four sites that is addressed in the MERMAID project. The most important physical processes will be described and ways to technical address the challenges will be proposed.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Number of pages: 153
Publication date: 2015

D5.4 Guidelines for interaction between seabed and support structure

Geotechnical aspects of MUPs have been researched as part of the MERMAID project, with particular focus on the stability of foundation soils and their necessary scour protection. Details of this work are published in a number of publications as provided in the reference list. Practitioners will be particularly drawn to the book “Liquefaction around Marine Structures” which covers many of the aforementioned subjects researched within MERMAID regarding liquefaction issues of platform designs (Sumer, 2014). The principal guidelines emerging from the complete body of work are:

Sandy soils have a high liquefaction potential, the effect of which must be accounted for in foundation design. The presence of small quantities of fines within a sandy soil can act to increase the potential for wave-induced liquefaction. For the soils studied, this occurred for fine fractions up to 30% of total mass. Sandy soils with larger quantities of fines, or sandy clays, were less susceptible as the clay behaviour dominates above this fraction.

Liquefaction risk under gravity base foundations should be screened properly in the light of in-situ and lab tests of the seabed soil. The preliminary screening methods in the literature (some of which cited here) can be used as a first assessment. If these assessments indicate any risks, detailed analyses and models should be applied for liquefaction assessment.

When it comes to wave-induced liquefaction, standing waves would be seen in the vicinity of reflecting boundaries of platform foundations. Liquefaction under Standing waves, although qualitatively similar, show features different from that caused by progressive waves. The pore pressure accumulation and liquefaction starts at the nodal section, and progresses towards anti-nodal section due to a diffusion mechanism. The rate of liquefaction at the nodal section seems to be the same with the progressive wave case. Thus, mathematical models for progressive waves would also work for standing waves as far as the nodal section is concerned.
Rock berms (protection cover) on pipelines and power transmission cables are essential to protect the structure from floating to the seabed surface due to liquefaction, and strongly suggested to be constructed. However, once the rock berm is constructed, care must be exercised on backfilling the excavated soil into the trench intentionally or due to sediment transport. Studies showed that a cover of stones constructed around the platform foundations for scour protection purpose would also serve for increasing the liquefaction resistance of surrounding soil. However, the stability of such stones against sinkage must also be assured, and studies are ongoing to provide guidelines for this too, which should be ready before the completion of the project (before M48).

**Damping of wind turbine tower vibrations**

Damping of wind turbine vibrations by supplemental dampers is a key ingredient for the continuous use of monopiles as support for offshore wind turbines. The present thesis consists of an extended summary with four parts and appended papers [P1-P4] concerning novel strategies for damping of tower dominated vibrations. The first part of the thesis presents the theoretical framework for implementation of supplemental dampers in wind turbines. It is demonstrated that the feasibility of installing dampers at the bottom of the tower is significantly increased when placing passive or semiactive dampers in a stroke amplifying brace, which amplifies the displacement across the damper and thus reduces the desired level of damper force. For optimal damping of the two lowest tower modes, a novel toggle-brace concept for amplifying the bending deformation of the tower is presented. Numerical examples illustrate that a minimum of three braces in a symmetric circumferential configuration are needed to introduce homogeneous damping in the two lowest vibration modes, independent of the rotor direction. A novel hybrid viscous damper concept is described in the second part. The hybrid damper consists of a viscous dash-pot in series with an actuator and a load cell. The controllable actuator displacement is regulated by an Integral Force Feedback (IFF) with the measured force from the load cell as sensor input. By controlling the actuator displacement exactly 180° out of phase with the damper force, the displacement across the passive viscous dash-pot is increased, thus improving the feasibility of placing dampers at the root of the wind turbine tower. Furthermore, attainable damping can be increased when introducing a filtered version of the proposed IFF control, and explicit design concepts are presented in the thesis. An Active Tuned Mass Damper (ATMD) concept is described in the third part of the thesis. By controlling the supplemental actuator force with absolute tower displacement and relative damper velocity as sensor input a stable control scheme is constructed for effective damping by the ATMD of the two critical tower modes. The frequency response performance of the ATMD is equivalent to that of the passive Tuned Mass Damper, but with a reduced damper mass. Furthermore, it is demonstrated that the active control force can be significantly reduced. Finally, in the last part the performance of the hybrid viscous damper with IFFs validated by a series of real-time hybrid simulations (RTHS). The experimental results illustrate the ability of the hybrid damper concept to increase damper stroke or attainable damping. The results also show that the actuator signal is quite sensitive to drift due an offset in the measured damper force. Thus, for the stroke amplifying IFF control a filtered integration is introduced, which almost retains the desired amplification and reduces drift. For the filtered IFF control, aimed at enhancing attainable damping, an augmented filtering similarly reduces drift, but also deteriorates the damping performance.

**General information**

**State:** Published

**Organisations:** Department of Mechanical Engineering, Solid Mechanics, Fluid Mechanics, Coastal and Maritime Engineering

**Authors:** Brodersen, M. L. (Intern), Høgsberg, J. B. (Intern), Jensen, J. J. (Intern), Pedersen, M. M. (Ekstern)

**Number of pages:** 142

**Publication date:** 2015
Diesel Engine Tribology

Recent years have seen an increase in the wear rate of engine bearings, subsequently followed by bearing failure, for the large two-stroke diesel engines used for ship propulsion. Here, the engine bearings include main, big end and crosshead bearings, with the bearing type used being the journal bearing, belonging to the class of ‘hydrodynamic bearings’. This implies that the load carrying capacity is generated by a relative movement of the involved components, i.e. velocity-driven operation. For the engine application, the velocity stems from the engine RPM. However, to comply with the latest emission requirements as well as attempting to minimise fuel expenses, the engine speed has been lowered together with an increase in the engine mean pressure which in terms lead to larger bearing loads. With worsened operating conditions from two sides, the encountered problems are understandable as the design criteria for the bearings are no longer valid, albeit still not desirable. To come up with a solution, the operating conditions of the bearings have to be understood. The main challenge is to supply sufficient lubricant to avoid metal-metal contact under time-varying combustion load. This project has therefore revolved around the investigation of the tribological performance of the dynamically loaded journal bearing, both theoretically and experimentally. The theoretical work covers two approaches to the modelling of the bearing; a traditional finite element based solver for Reynolds equation, and a more general finite volume discretisation of the Navier-Stokes equations. In this way the influence from assumptions usually made in regards to supply grooves can be verified. A test rig has been constructed for replicating engine-like conditions. In this way the influence from assumptions usually made in regards to supply grooves can be verified. A test rig has been constructed for replicating engine-like conditions. An uni-directional load can be applied in both static and dynamic modes, while another key feature being that of a transparent polymer bearing enabling the study of film rupture and re-forming. Paper [P1] describes the development of a suitable finite volume mesh for dynamic loading, while Paper [P2] contains the perturbation implementation used for the dynamic loading. Resorting to Gümbel boundary conditions, very similar orbits are predicted for a given bearing using the two methods as demonstrated in Paper [P3]. Good agreement is also obtained between the numerical and experimental results. Finally, some suggestions to improvements of the modelling as well as the experimental set-up, are made.
Discussion of "Sediment entrainment probability and threshold of sediment suspension: Exponential-based approach" by Sujit K. Bose and Subhasish Dey

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Sumer, B. M. (Intern)
Number of pages: 1
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Hydraulic Engineering (Reston)
Volume: 141
Issue number: 4
Article number: 07014023
ISSN (Print): 0733-9429
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 2
Scopus rating (2016): SJR 0.856 SNIP 1.612 CiteScore 1.7
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 0.879 SNIP 1.329 CiteScore 1.21
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 0.801 SNIP 1.646 CiteScore 1.43
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 0.781 SNIP 1.399 CiteScore 1.14
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 0.991 SNIP 1.403 CiteScore 1.18
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.088 SNIP 1.485 CiteScore 1.08
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 1.149 SNIP 1.797
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 1.076 SNIP 1.515
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 1.105 SNIP 1.698
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 1.012 SNIP 1.566
Scopus rating (2006): SJR 0.947 SNIP 1.387
Web of Science (2006): Indexed yes
Scopus rating (2005): SJR 0.456 SNIP 1.577
Scopus rating (2004): SJR 0.798 SNIP 1.229
Scopus rating (2003): SJR 0.77 SNIP 1.355
Web of Science (2003): Indexed yes
Scopus rating (2002): SJR 0.542 SNIP 1.179
Early Regimes of Water Capillary Flow in Slit Silica Nanochannels

Molecular dynamics simulations are conducted to investigate the initial stages of spontaneous imbibition of water in slit silica nanochannels surrounded by air. An analysis is performed for the effects of nanoscopic confinement, initial conditions of liquid uptake and air pressurization on the dynamics of capillary filling. The results indicate that the nanoscale imbibition process is divided into three main flow regimes: an initial regime where the capillary force is balanced only by the inertial drag and characterized by a constant velocity and a plug flow profile. In this regime, the meniscus formation process plays a central role in the imbibition rate. Thereafter, a transitional regime takes place, in which, the force balance has significant contributions from both inertia and viscous friction. Subsequently, a regime wherein viscous forces dominate the capillary force balance is attained. Flow velocity profiles identify the passage from an inviscid flow to a developing Poiseuille flow. Gas density profiles ahead of the capillary front indicate a transient accumulation of air on the advancing meniscus. Furthermore, slower capillary filling rates computed for higher air pressures reveal a significant retarding effect of the gas displaced by the advancing meniscus.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Oyarzua, E. (Ekstern), Walther, J. H. (Intern), Mejia, A. (Ekstern), Zambrano, H. (Ekstern)
Pages: 14731-14739
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Physical Chemistry Chemical Physics
Volume: 17
Issue number: 22
ISSN (Print): 1463-9076
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 4.06 SJR 1.678 SNIP 1.117
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.771 SNIP 1.244 CiteScore 4.45
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.772 SNIP 1.253 CiteScore 4.29
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.715 SNIP 1.216 CiteScore 4.05
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.916 SNIP 1.184 CiteScore 3.67
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.697 SNIP 1.203 CiteScore 3.6
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 1.802 SNIP 1.196
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 2.127 SNIP 1.369
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 2
Scopus rating (2008): SJR 2.158 SNIP 1.211
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 1.84 SNIP 1.138
Web of Science (2007): Indexed yes
Scopus rating (2006): SJR 1.467 SNIP 1.128
Web of Science (2006): Indexed yes
Scopus rating (2005): SJR 1.389 SNIP 1.104
Web of Science (2005): Indexed yes
Scopus rating (2004): SJR 1.173 SNIP 1.007
Web of Science (2004): Indexed yes
Scopus rating (2003): SJR 1.093 SNIP 0.925
Web of Science (2003): Indexed yes
Scopus rating (2002): SJR 1.122 SNIP 0.973
Web of Science (2002): Indexed yes
Scopus rating (2001): SJR 1.09 SNIP 0.914
Web of Science (2001): Indexed yes
Scopus rating (2000): SJR 0.948 SNIP 1.068
Web of Science (2000): Indexed yes
Scopus rating (1999): SJR 0.121 SNIP 0
Original language: English
DOIs: 10.1039/c5cp01862e
Source: PublicationPreSubmission
Source-ID: 110629831
Publication: Research - peer-review › Journal article – Annual report year: 2015

Edge scour at scour protections around piles in the marine environment - Laboratory and field investigation

When building offshore wind turbines with monopile foundations, scour protection typically is placed to avoid scouring of the soil close to the monopile. An important aspect is that the scour protection itself causes erosion, inflicted by the local increase in current and/or wave velocities and in turn increased bed shear stresses. Scour of the edge material alongside the scour protection may cause deformations and failure of the scour protection of offshore wind turbine foundations. This can reduce the stability of the stone layer and cause exposure of cables running between the monopiles where they go from buried to the transition piece on the foundation. Although much information is available on the design of scour protection systems around monopiles, little is known on the mechanisms causing edge scour and the equilibrium stages of the edge scour process in steady current, waves and combined waves and current. This paper presents an extensive experimental campaign to explain the edge scour process in current and combined irregular waves and current, as well as tidal current. The three-dimensional flow field around the pile and scour protection is resolved by particle image velocimetry and bed shear stress measurements, showing a local increase in the flow velocities and bed shear stresses leading to increased sediment transport and scour. The governing process in steady current is a pair of symmetrical counter-rotating vortices emerging in the near bed region in the wake of the pile and scour protection, causing a significant downstream scour hole. It is found that the equilibrium scour hole depth and length scales with the pile diameter and the ratio between the thickness- and the width of the scour protection. In the second part of the present paper, the results from the experimental campaign are compared with the edge scour experienced in practice, outlined by a survey program of the offshore wind park Egmond Aan Zee and a published field investigation of Scroby Sands OWF by Whitehouse et al. (2011).
Effects of Mixing on Hopper Sedimentation in Clearing Mixtures

Hopper sedimentation is the result of precipitation of typically fine sediment from a homogenous, high-concentration mixture, which is not completely deficient of turbulence. If hopper sedimentation or loading is accomplished through a single-inflow system, or if the irregularity of the inflow concentrations is pronounced or simply terminated, then the hopper mixture will clear. Whereas turbulent mixing is redundant, when the mixture is homogenous, it may take an active role when the mixture is clearing. The role of turbulence on hopper sedimentation has been the focus of several studies, and a common perception of turbulence (or at least of mixing) is that it delays sedimentation. Existing measurements of sedimentation rates in a closed-flume experiment, engineered to provide input to a hopper sedimentation model, revealed that turbulence in a clearing mixture is not necessarily associated with a delay in sedimentation. The experiment showed that sedimentation was boosted by adding a current to a clearing mixture, which infers that turbulence, under certain conditions, may act as a sedimentation agent on the excess sediment in suspension. Therefore, the interactions between turbulent mixing and settling in high-concentration mixtures were examined theoretically. Analytical solutions for clearance of excess concentrations were derived for the limiting cases of (1) still-water clearance and (2) clearance when the amount of turbulence is abundant. When examining these analytical solutions, a potential for enhanced sedimentation was revealed. It was found that mixing-induced dilution of concentration weakens the hindrance in settling to a degree that enhances sedimentation. The analytical findings prompted a more elaborate analysis of the mechanism using a numerical model, which encompassed time- and depth-varying turbulence. This allowed the experimental setup and the observed settling effects to be simulated. The potential of the enhancing sedimentation mechanism was analyzed under more general conditions with the numerical model.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DHI Denmark
Authors: Hjelmager Jensen, J. (Intern), Saremi, S. (Ekstern)
Number of pages: 10
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication Information

Journal: Journal of Waterway, Port, Coastal, and Ocean Engineering
Volume: 141
Issue number: 2
Article number: 06014002
ISSN (Print): 0733-950X
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 1.48 SJR 0.708 SNIP 1.139
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Whipping can contribute to increased fatigue and extreme loading of container ships, and guidelines have been made available by the leading class societies. Reports concerning the hogging collapse of MSC Napoli and MOL Comfort suggest that whipping contributed. The accidents happened in moderate to small storms. Model tests of three container ships have been carried out in different sea states under realistic assumptions. Preliminary extrapolation of the measured data suggested that moderate storms are dimensioning when whipping is included due to higher maximum speed in moderate storms. This paper considers various extrapolation methods to investigate the uncertainty in the extrapolation methods and to see if all methods confirm that the moderate storms are dimensioning.
Container ships of today are becoming increasingly larger. The increased ship size implies lower natural hull frequencies which influence the hull girder response in waves. The large bow and stern flare of container ships make them vulnerable to whipping-induced loads, and whipping loads may increase the extreme hull girder response significantly. Focus in the present paper is on the influence of the hull girder flexibility on the extreme response amidships, namely the wave-induced vertical bending moment.

Electronic versions:
ISOPE_I_15_419.pdf

Source: FindIt
Source-ID: 2289321392
Publication: Research - peer-review › Conference article – Annual report year: 2015

Extreme value prediction of the wave-induced vertical bending moment in large container ships
Container ships of today are becoming increasingly larger. The increased ship size implies lower natural hull frequencies which influence the hull girder response in waves. The large bow and stern flare of container ships make them vulnerable to whipping-induced loads, and whipping loads may increase the extreme hull girder response significantly. Focus in the present paper is on the influence of the hull girder flexibility on the extreme response amidships, namely the wave-induced vertical bending moment.
vertical bending moment (VBM) in hogging, and the prediction of the extreme value of the same. The analysis in the present paper is based on time series of full scale measurements from three large container ships of 8600, 9400 and 14000 TEU. When carrying out the extreme value estimation the peak-over-threshold (POT) method combined with an appropriate extreme value distribution is applied. The choice of a proper threshold level as well as the statistical correlation between clustered peaks influence the extreme value prediction and are taken into consideration in the present paper.

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Andersen, I. M. V. (Intern), Jensen, J. J. (Intern)
Pages: 389-402
Publication date: 2015

**Host publication information**
Title of host publication: Proceedings of the 7th International Conference on Hydroelasticity in Marine Technology
Main Research Area: Technical/natural sciences
Extreme value prediction, Hull girder flexibility, Vertical bending moment, Wave-induced loads, Hydro-elastic effects
Source: Publication PreSubmission
Source-ID: 116776961
Publication: Research - peer-review › Article in proceedings – Annual report year: 2015

**Fatigue damage estimation in non-linear systems using a combination of Monte Carlo simulation and the First Order Reliability Method**
For non-linear systems the estimation of fatigue damage under stochastic loadings can be rather time-consuming. Usually Monte Carlo simulation (MCS) is applied, but the coefficient-of-variation (COV) can be large if only a small set of simulations can be done due to otherwise excessive CPU time. The reason is that the fatigue damage estimation is very sensitive to the largest values from the simulations. The paper suggests the additional use of the First Order Reliability Method (FORM) to get a better estimation of the tail in the distribution of the estimated fatigue damage and thereby reducing the COV. For a specific example dealing with stresses in a tendon in a tension leg platform the COV is thereby reduced by a factor of three.

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, J. J. (Intern)
Pages: 203-210
Publication date: 2015
Main Research Area: Technical/natural sciences

**Publication information**
Journal: Marine Structures
Volume: 44
ISSN (Print): 0951-8339
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.49 SJR 1.655 SNIP 2.636
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.618 SNIP 2.602 CiteScore 2.77
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.431 SNIP 3.026 CiteScore 2.18
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.314 SNIP 2.871 CiteScore 2.42
ISI indexed (2013): ISI indexed yes
Flow and edge scour in current adjacent to stone covers

This paper presents the results of an experimental investigation on edge scour adjacent to a stone cover laid on a sandy bed. The three-dimensional flow over the edge of the stone layer has been investigated by the use of particle image velocimetry. The flow measurements show a significant amount of turbulence in the primary flow near the junction between the stone layer and the sand bed and the formation of complex secondary-flow structures. The results show that the flow and the edge scour process in a steady current are governed by the size of the roughness elements and to some extent the side slope of the berm. The edge scour is caused by the combined action of the primary flow and the secondary flow. The primary flow stirs up the sediment and puts it into suspension, and the secondary flow carries it away from the junction between the stone layer and the sand bed, resulting in a scour hole forming adjacent to the toe of the stone layer. The measured scour depth attained a constant level of approximately one times the stone size in the live-bed regime; further, the scour depth showed a slight decrease when the side slope of the berm was increased. Design diagrams are presented for the scour depth and the time scale of the scour process.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DHI Denmark, Rambøll Oil and Gas
Flow and Turbulence at Rubble-Mound Breakwater Armor Layers under Solitary Wave

This paper presents the results of an experimental investigation of the flow and turbulence at the armor layer of rubble-mound breakwaters during wave action. The study focused on the details of the flow and turbulence in the armor layer and on the effect of the porous core on flow and stability. To isolate the processes involved with the flow in the porous core, experiments were conducted with increasing complexity. Specifically, three parallel experiments were performed including (1) an impermeable smooth breakwater slope, (2) an impermeable breakwater slope with large roughness elements added to the breakwater, and (3) a porous breakwater where the porous core was added below the breakwater front. One breakwater slope of 1:1.5 was applied. In this paper the focus is on the details of a single sequence of wave approach, run-up, and rundown. To isolate this sequence the experiments were performed applying a solitary wave. The individual sources of turbulence generation were distinguished using Laser Doppler anemometry measurements, and the effect of the armor layer and porous core was described in terms of a reduced impact of the rundown process, production of lee wake turbulence, and less transport of turbulence above the armor layer. The shear stresses were evaluated from the measurements of turbulence, and they were associated with the run-up and rundown phases. The Shields parameter, determined from the shear stresses, was found to be reduced by 30% as a result of the porous core material.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Rambøll Oil and Gas
Authors: Jensen, B. (Intern), Christensen, E. D. (Intern), Sumer, B. M. (Intern), Vistisen, M. (Ekstern)
Number of pages: 14
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication Information
Journal: Journal of Waterway, Port, Coastal, and Ocean Engineering
Volume: 141
Issue number: 6
Article number: 04015006
ISSN (Print): 0733-950X
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 1.48 SJR 0.708 SNIP 1.139
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.915 SNIP 1.172 CiteScore 1.23
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.578 SNIP 1.248 CiteScore 1.05
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.562 SNIP 1.184 CiteScore 1.29
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.632 SNIP 0.964 CiteScore 0.83
ISI indexed (2012): ISI indexed yes
Flow Dynamics of green sand in the DISAMATIC moulding process using Discrete element method (DEM)

The DISAMATIC casting process production of sand moulds is simulated with DEM (discrete element method). The main purpose is to simulate the dynamics of the flow of green sand, during the production of the sand mould with DEM. The sand shot is simulated, which is the first stage of the DISAMATIC casting process. Depending on the actual casting geometry the mould can be geometrically quite complex involving e.g. shadowing effects and this is directly reflected in the sand flow during the moulding process. In the present work a mould chamber with “ribs” at the walls is chosen as a baseline geometry to emulate some of these important conditions found in the real moulding process. The sand flow is simulated with the DEM and compared with corresponding video footages from the interior of the chamber during the moulding process. The effect of the rolling resistance and the static friction coefficient is analysed and discussed in relation to the experimental findings.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Manufacturing Engineering, DISA Industries A/S, Magma Gießereitechnologie GmbH
Authors: Hovad, E. (Intern), Larsen, P. (Ekstern), Walther, J. H. (Intern), Thorborg, J. (Intern), Hattel, J. H. (Intern)
Number of pages: 9
Publication date: 2015
Conference: 14th International Conference on Modelling of Casting, Welding and Advanced Solidification Processes, Awaji island, Hyogo, Japan, 21/06/2015 - 21/06/2015
Main Research Area: Technical/natural sciences

Publication information
Journal: I O P Conference Series: Materials Science and Engineering
Volume: 84
Flow enhancement of water flow through silica slit pores with graphene-coated walls

Nanofluidic devices such as Lab-On-a-Chip often are designed to transport water solutions through hydrophilic nano-conduits. In these systems with narrow confinement, the viscous forces dominate the flow and as a result, the hydrodynamic friction drag is very high. Moreover, the drag and the amount of energy required for pumping a fluid are directly related. Therefore, it is desirable to explore drag reduction strategies in nanoconfined flows. Liquids are known to slip past non-wetting surfaces. Graphene is a single-atom-thick sheet of carbon atoms arranged in a hexagonal honeycomb lattice, which features a unparalleled combination of high specific surface area, chemical stability, mechanical strength and flexibility. Recently, the wettability of water droplets on multilayer graphene sheets deposited on a silica substrate has been investigated. In this study, we investigate the role of graphene coatings to induce flow enhancement in silica channels. We conduct molecular dynamics simulations of pressurized water flow inside silica channels with and without graphene layers covering the walls. In particular, we compute density and velocity profiles, flow enhancement and slip lengths to understand the drag reduction capabilities of multilayer graphene coatings.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Zambrano, H. (Ekstern), Wagemann, E. (Ekstern), Oyarzua, E. (Ekstern), Walther, J. H. (Intern)
Number of pages: 1
Publication date: 2015
Event: Abstract from 68th Annual Meeting of the American Physical Society's Division of Fluid Dynamics (DFD) , Boston, United States.
Main Research Area: Technical/natural sciences
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2015

Go offshore -Combining food and energy production

European oceans will be subject to massive development of marine infrastructure in the near future. The development includes energy facilities, e.g. offshore wind farms, exploitation of wave energy, and also development and implementation of marine aquaculture. This change of infrastructure makes the concept of multi-use offshore platforms particularly interesting. The development of new concepts requires effective marine technology and governance solutions. Simultaneously, both economic costs and environmental impacts have to remain within acceptable limits. These concerns are at the core of the MERMAID project funded under 'The Ocean of Tomorrow' call for proposals. At the end of the project, a set of specific guidelines are produced in order to assist future stakeholders within the offshore industries with a view to planning, establishing and operating their businesses in the most optimal way. The multi-disciplinary and cross-
sectorial approach of this project is very innovative and the EU benefit lies in the case studies that address four EU-regional seas. MERMAID established close links with the other projects, TROPOS and H2OCEAN, funded under the same 'The Ocean of Tomorrow' topic in order to enhance complementarities and synergies.

**General information**

State: Published

Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Cantabria, Wageningen University & Research, Deltares, DHI Denmark, Athens University of Economics and Business, StatOil Petroleum AS, University of Bologna

Authors: Christensen, E. D. (Intern), Stuiver, M. (Ekstern), Guanche, R. (Ekstern), Møhlenberg, F. (Ekstern), Schouten, J. (Ekstern), Svenstrup Pedersen, O. (Ekstern), He, W. (Ekstern), Zanuttigh, B. (Ekstern), Koundouri, P. (Ekstern)

Number of pages: 48

Publication date: 2015

**High order Poisson Solver for unbounded flows**

This paper presents a high order method for solving the unbounded Poisson equation on a regular mesh using a Green’s function solution. The high order convergence was achieved by formulating mollified integration kernels, that were derived from a filter regularisation of the solution field. The method was implemented on a rectangular domain using fast Fourier transforms (FFT) to increase computational efficiency. The Poisson solver was extended to directly solve the derivatives of the solution. This is achieved either by including the differential operator in the integration kernel or by performing the differentiation as a multiplication of the Fourier coefficients. In this way, differential operators such as the divergence or curl of the solution field could be solved to the same high order convergence without additional computational effort. The method was applied and validated using the equations of fluid mechanics as an example, but can be used in many physical problems to solve the Poisson equation on a rectangular unbounded domain. For the two-dimensional case we propose an infinitely smooth test function which allows for arbitrary high order convergence. Using Gaussian smoothing as regularisation we document an increased convergence rate up to tenth order. The method however, can easily be extended well beyond the tenth order. To show the full extent of the method we present the special case of a spectrally ideal regularisation of the velocity formulated integration kernel, which achieves an optimal rate of convergence.
Hydrodynamic analysis of oscillating water column wave energy devices

A 40-chamber I-Beam attenuator-type, oscillating water column, wave energy converter is analyzed numerically based on linearized potential flow theory, and experimentally via model test experiments. The high-order panel method WAMIT by Newman and Lee (WAMIT; a radiation–diffraction panel program for wave-body interactions, 2014, http://www.wamit.com) is used for the basic wave-structure interaction analysis. The damping applied to each chamber by the power take off is modeled in the experiment by forcing the air through a hole with an area of about 1% of the chamber water surface area. In the numerical model, this damping is modeled by an equivalent linearized damping coefficient which extracts the same amount of energy over one cycle as the experimentally measured quadratic damping coefficient. The pressure in each chamber in regular waves of three different height-to-length ratios is measured in the experiments and compared to calculations. The model is considered in both fixed and freely floating, slack-moored conditions. Comparisons are also made to experimental measurements on a single fixed chamber. The capture width ratio in each case is predicted based on the pressures in the chambers. Good agreement is found between the calculations and the experiments.

Implementing Composite Superstructures in Large Passenger Ships

This study focuses on the structural response of the part of the superstructure of a RoPax ferry that has been redesigned using composite materials. The composite superstructure is presented and subsequently compared to the existing steel design considering different loading conditions by the use of FE modelling. Results indicate that it is not the structural response of the superstructure that inhibits the implementation of composites in the superstructures of large passenger ships but the complicated design procedure and the acceptance of such solutions by the regulatory bodies.
Integrated Analysis of the Scavenging Process in Marine Two-Stroke Diesel Engines

Large commercial ships such as container vessels and bulk carriers are propelled by low-speed, uniow scavenged two-stroke diesel engines. An integral-cylinder process in this type of engine is the scavenging process, where the burned gases from the combustion process are evacuated through the exhaust valve and replaced with fresh air for the subsequent compression stroke. The scavenging air enters the cylinder via inlet ports which are uncovered by the piston at bottom dead center (BDC). The exhaust gases are then displaced by the fresh air entering the cylinder. The scavenging ports are cut with an angle to introduce a swirling component to the ow. The in-cylinder swirl is beneow for air-fuel mixture, cooling of the cylinder liner and minimizing recirculation zones where pockets of exhaust gas are trapped. However, a known characteristic of swirling ows is an adverse pressure gradient in the center of the ow, which might lead to a local decline in axial velocity and the formation of central recirculation zones, known as vortex breakdown. Ever more stringent emission legislations over the last 10-15 years have changed the engine lay out diagram in the pursuit of an engine which is both fuel effective and within the current emission legislations. To achieve this goal, a fundamental understanding of the in-cylinder processes, and the interactions between them are needed. This thesis aims at providing in-depth knowledge of the scavenging process and to identify the parameters that governs its performance. This thesis will present a CFD model that is tested and validated with quantitative data obtained from a dedicated test engine and during engine commissioning on location at the shipbuilder. The CFD model comprises the full geometry of a single cylinder from scavenger receiver to the exhaust receiver for a two-stroke diesel engine. Time resolved boundary conditions corresponding to measurements obtained from an operating engine as well as realistic initial conditions are used in the simulations. The CFD model provides a detailed description of the in-cylinder ow from exhaust valve opening (EVO) to exhaust valve closing (EVC). A string of studies are included in this thesis. An engine load sweep is included to evaluate the scavenging process as function of engine load. The engine load sweep follows the propeller curve, where the engine speed varies with the engine load. This implies that the pressure in the scavenge and exhaust receivers increase while the scavenging port exposure time, tscav, decrease. Further the scavenging pressure is varied while the engine speed is kept constant. From the perspective of the scavenging process this will resemble a load sweep following a generator curve. The scavenging port angle is varied to investigate the influence of in-cylinder swirl. A total of 7 port angles is applied; α = 0°, α = 10°, α = 15°, α = 18°, α = 20°, α = 25° and α = 30°. The CFD analysis shows that the bulk purity of air in the cylinder is proportional to the volumetric ow rate (mass ow rate divided by the air density) of scavenge air through the cylinder. The volumetric ow rate decreases with density for a given mass ow rate. When the engine load is increased, both the mass ow rate and the scavenging pressure is increased due to the turbocharger response. It is shown in this thesis that the increased density of the scavenge air, in conjunction with the reduced port exposure time, actually decrease the volume ow rate of air in the cylinders. This impairs the scavenging process at high engine loads. The CFD model also shows that the scavenging process consist of two sub processes. The volumetric scavenge, where the scavenge air displace the exhaust gas. And the push out process, where the piston displace the scavenge air and exhaust gas mixture between inlet port closing, IPC, and exhaust valve closing, EVC. The port angle study shows that the scavenging process is unacted by the changes in the in-cylinder swirl. Visualization of a passive scalar shows some influence of the in-cylinder distribution of scavenge air and exhaust gas, but volumetric displacement is the prime mover in the scavenging process. The CFD simulations is in good agreement with a simple perfect displacement model proposed by Sher (1990). The perfect displacement model is used as the basis for a simplified scavenge model in conjunction with a model
topredict the contribution from the push out process. The model is modied to the CFD results to account for mixing between the scavenge air and the exhaust gases and can therefore only be considered as a preliminary model. However, this model shows that it is possible to obtain a simple model which can be used to ensure adequate scavenging based on turbocharger characteristics and exhaust valve lift proles. The CFD model described in this Ph.D. thesis is used to investigate the response of key parameters on the scavenging process and gives detailed and profound insight to an integral in-cylinder process in the two-stroke diesel engine cycle. Further, the results from the CFD model is a valuable part of the R&D strategy of “full cycle CFD modelling” where the scavenging CFD model shall be coupled together with a combustion CFD model to simulate the complete engine cycle.

Investigation on the Use of a Multiphase Eulerian CFD solver to simulate breaking waves
The main challenge in CFD multiphase simulations of breaking waves is the wide range of interfacial length scales occurring in the flow: from the free surface measurable in meters down to the entrapped air bubbles with size of a fraction of a millimeter. This paper presents a preliminary investigation on a CFD model capable of handling this problem. The model is based on a solver, available in the open-source CFD toolkit OpenFOAM, which combines the Eulerian multi-fluid approach for dispersed flows with a numerical interface sharpening method. The solver, enhanced with additional formulations for mass and momentum transfer among phases, was satisfactorily tested against an experimental bubble column flow. The model was then used to simulate the propagation of a laboratory solitary breaking wave. The motion of the free surface was successfully reproduced up to the breaking point. Further implementations are needed to simulate the air entrainment phenomenon.
Iterative Brinkman penalization for remeshed vortex methods

We introduce an iterative Brinkman penalization method for the enforcement of the no-slip boundary condition in remeshed vortex methods. In the proposed method, the Brinkman penalization is applied iteratively only in the neighborhood of the body. This allows for using significantly larger time steps, than what is customary in the Brinkman penalization, thus reducing its computational cost while maintaining the capability of the method to handle complex geometries. We demonstrate the accuracy of our method by considering challenging benchmark problems such as flow past an impulsively started cylinder and normal to an impulsively started and accelerated flat plate. We find that the present method enhances significantly the accuracy of the Brinkman penalization technique for the simulations of highly unsteady flows past complex geometries.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Eidgenössische Technische Hochschule, California Institute of Technology
Authors: Hejlesen, M. M. (Intern), Koumoutsakos, P. (Ekstern), Leonard, A. (Ekstern), Walther, J. H. (Intern)
Pages: 547-562
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Computational Physics
Volume: 280
ISSN (Print): 0021-9991
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 3.12 SJR 2.034 SNIP 1.822
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 2.098 SNIP 1.988 CiteScore 2.92
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 2.166 SNIP 2.193 CiteScore 3.12
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 2.227 SNIP 2.45 CiteScore 3.3
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 2.161 SNIP 2.052 CiteScore 2.69
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 2.06 SNIP 2.194 CiteScore 2.99
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 2.185 SNIP 2.096
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 2.439 SNIP 2.219
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 2.247 SNIP 2.03
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 2.377 SNIP 2.379
Kapitza Resistance between Few-Layer Graphene and Water: Liquid Layering Effects

The Kapitza resistance ($R_K$) between few-layer graphene (FLG) and water was studied using molecular dynamics simulations. The $R_K$ was found to depend on the number of the layers in the FLG though, surprisingly, not on the water blockthickness. This distinct size dependence is attributed to the large difference in the phonon mean free path between the FLG and water. Remarkably, $R_K$ is strongly dependent on the layering of water adjacent to the FLG, exhibiting an inverse proportionality relationship to the peak density of the first water layer, which is consistent with better acoustic phonon matching between FLG and water. These findings suggest novel ways to engineer the thermal transport properties of solid−liquid interfaces by controlling and regulating the liquid layering at the interface.
This abstract presents an analysis of an attenuator-type Wave Energy Converter (VEC) with 40 Oscillating Water Column (OWC) chambers for the extraction of wave energy. Linearized potential flow calculations are made in the frequency-domain using WAMIT [8]. An equivalent linearized damping coefficient to represent the air turbine Power Take Off (PTO) system is found for each condition by iterating to find the consistent response-damping pair for a given frequency and incident wave amplitude. The absorbed power is estimated based on the pressure in each chamber and the PTO damping coefficient. The calculations are compared to model-scale measurements in a slack-moored condition, and generally good agreement is found. Work is in progress to move the solution to the time-domain and include a more sophisticated PTO model which includes nonlinear and air compressibility effects in the turbine.

**Linearized potential flow analysis of a 40 chamber, oscillating water column wave energy device**

**General information**

- **State:** Published
- **Organisations:** Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
- **Authors:** Bingham, H. B. (Intern), Read, R. (Intern)
- **Number of pages:** 4
- **Publication date:** 2015
- **Event:** Abstract from 30th International Workshop on Water Waves and Floating Bodies, Bristol, United Kingdom.
- **Main Research Area:** Technical/natural sciences

**Electronic versions:**

- Linearized_potential_flow_analysis_of_a_40_chamber_oscillating_water_column_wave_energy_device.pdf
Marine Structures: Future Trends and the Role of Universities
This paper emphasizes some of the challenges and trends associated with the future development of marine structures. Its main focus is on ways to improve the efficiency of energy-consuming ships, and on design challenges related to energy-producing offshore structures. This paper also discusses the analysis tools that are most needed to enable sustainable designs for future ships and offshore structures. The last section of the paper contains thoughts on the role of universities in education, research, and innovation regarding marine structures. It discusses curriculum requirements for maritime-technology education, basic research activities, and international cooperation.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Pedersen, P. T. (Intern)
Number of pages: 10
Pages: 131-138
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Engineering
Volume: 1
Issue number: 1
ISSN (Print): 2095-8099
Ratings:
Web of Science (2018): Indexed yes
Scopus rating (2016): CiteScore 0.34
Original language: English
Marine structures, Ships, Offshore structures, Curriculum, Research activities
Electronic versions:
Marine_Structures_Future_Trends_and_the_Role_of_Universities_1_.pdf
DOIs:
10.15302/J-ENG-2015004

Bibliographical note
This article is published with open access at www.engineering.org.cn
Source: PublicationPreSubmission
Source-ID: 110362595
Publication: Research - peer-review › Journal article – Annual report year: 2015

MARINET experiment KNSWING testing an I-Beam OWC attenuator
The research and results presented in this paper concerns experiments on wave energy conversion carried out in the one meter deep wave tank at the Hydraulic and Maritime Research Center (HMRC) at University College Cork, Ireland in 2013. The purpose is to investigate how much power an attenuator - a ship shaped wave energy converter facing the waves with its bow - can absorb along its sides in a range of regular and irregular wave conditions. The experiments were carried out in model scale 1:50 resembling the wave conditions and water depth of the Danish part of the North Sea and a 150 m long wave energy converter with 20 Oscillating Water Column (OWC) chambers on each side. The damping applied to each chamber by the Power Take Off (PTO) is modeled by forcing the air through a hole with an area of about 1.3% of the chamber water surface area. The results in irregular wave conditions shows that the converter can absorb more than 2.5 MW of wave power that in wave conditions with significant wave heights of 5 m. A capture width ratio between 20% and 25% was measured in the most frequent wave conditions with average periods between 5 and 7 s with significant wave height of 2 m. In short crested waves the energy absorption was slightly better due to the directional spreading of energy. The tests in monochromatic waves indicate capture width ratios up to 35% in waves with a steepness of 2.5%. The capture width ratio decreases somewhat in steeper waves. The experiments with the attenuator wave energy converter demonstrated its seaworthiness and ability to absorb wave energy. The results form a valuable base for the development of a numerical model of the system that will be used for further optimization and development.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Rambøll Danmark A/S
Authors: Nielsen, K. (Ekstern), Bingham, H. B. (Intern)
Number of pages: 21-34
Pages: 21-34
Publication date: 2015
Main Research Area: Technical/natural sciences
Methods for Cavitation Prediction on Tip-Modified Propellers in Ship Wake Fields

Unsteady cavitation simulations on a tip-modified propeller in behind-hull condition are made by both Boundary Element Method (BEM) and Computational Fluid Dynamics (CFD). As the hull geometry typically is not disclosed to the propeller designer and thus cannot be included in the simulation, other measures must be taken to account for the ship's wake field. In CFD, different wake models using non-uniform inlet flow and momentum sources are tested to achieve resulting axial and transverse flows in the propeller plane that resemble the desired wake field. Also, the simulations are carried out with two types of hull wake fields: One originating from model test measurements and the other from a bare hull RANS simulation at the cavitation test Reynolds number. By comparing simulation results, the different numerical approaches are evaluated for accuracy of the unsteady cavitation prediction as a propeller design tool complementing the cavitation tunnel test.
Multi-directional random wave interaction with an array of cylinders

Based on the linear theory of wave interaction with an array of circular bottom-mounted vertical cylinders, systematic calculations are made to investigate the effects of the wave directionality on wave loads in short-crested seas. The multi-directional waves are specified using a discrete form of the Mitsuyasu-type spreading function. The time series of multi-directional wave loads, including both the wave run-up and wave force, can be simulated. The effect of wave directionality on the wave run-up and wave loading on the cylinders is investigated. For multi-directional waves, as the distribution of wave spreading becomes wider, the wave run-up at some points around the cylinders is found to increase. This suggests that multi-directional wave run-up tends to be larger than unidirectional wave run-up. In addition, the wave directionality has a significant influence on the transverse force. The biggest transverse force is found to occur on the rear cylinder rather than the front one. This is quite different from the results in unidirectional waves and should be paid much more attention in the design of offshore structures. At last, the possibility of the near-trapping under the multi-directional random waves is investigated. It is found that the near-trapping also occurs for multi-directional wave conditions. (c) 2015 Elsevier Ltd. All rights reserved.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Dalian University of Technology
Authors: Ji, X. (Ekstern), Liu, S. (Ekstern), Bingham, H. B. (Intern), Li, J. (Ekstern)
Pages: 62-77
Publication date: 2015
Main Research Area: Technical/natural sciences

Publication information
Journal: Ocean Engineering
Volume: 110
ISSN (Print): 0029-8018
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.46 SJR 1.315 SNIP 2.014
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.172 SNIP 1.989 CiteScore 2.19
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.252 SNIP 2.323 CiteScore 2.11
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.178 SNIP 2.773 CiteScore 2.2
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.206 SNIP 2.445 CiteScore 1.71
ISI indexed (2012): ISI indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.055 SNIP 2.528 CiteScore 1.85
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
New Concepts for Shipboard Sea State Estimation

The wave buoy analogy is a tested means for shipboard sea state estimation. Basically, the estimation principle resembles that of a traditional wave rider buoy which relies, fundamentally, on transfer functions used to relate measured wave-induced responses and the unknown wave excitation. This paper addresses however a newly developed concept of the wave buoy analogy but the approach presented herein is, on the contrary, not relying exclusively on transfer functions. Instead, the method combines a signal-based part, estimating wave frequency, and a model-based part, estimating wave amplitude and phase, where only the model-based part depends on transfer functions whereas the signal-based part relies on the measured vessel response alone. Case studies in terms of hypothetical examples show that the method is capable to reconstruct fully the wave elevation process of a sinusoidal regular wave; which include estimation of the wave’s frequency, amplitude and phase. At this stage, the method is far from being a useful means in practical, real-situation applications but the method provides, indeed, a valuable step towards developing new approaches for shipboard sea state estimation.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Electrical Engineering, Automation and Control, Technical University of Denmark, Norwegian University of Science and Technology
Authors: Nielsen, U. D. (Intern), Bjerregård, M. (Ekstern), Galeazzi, R. (Intern), Fossen, T. I. (Ekstern)
Number of pages: 10
Publication date: 2015

Host publication information
Title of host publication: Proceedings of OCEANS '15
Publisher: IEEE
Article number: 7404386
Main Research Area: Technical/natural sciences
Numerical and Experimental Investigation of Heat Flow in Permanent Magnet Brushless DC Hub Motor

This paper investigates the heat dissipation in the hub motor of an electric two-wheeler using lumped parameter (LP), finite element (FE) and computational fluid dynamic (CFD) models. The motor uses external rotor permanent magnet brushless DC topology and nearly all of its losses are generated in the stator. The hub motor construction restricts the available conductive paths for heat dissipation from the stator to the ambient only through the shaft. In contrast to an internal rotor structure, where the stator winding losses are diffused via conduction, here convection plays a major role in loss dissipation. Therefore, a LP thermal model with improved convection modelling has been proposed to calculate the temperature of the components inside the hub motor. The developed model is validated with the FE thermal model and the test data. In addition, CFD tools has been used to accurately model the internal and the external flow as well as the convective heat transfer of the hub motor. Finally, an optimization study of the hub motor has been carried out using the CFD model to improve heat transfer from the stator.

Numerical investigation of flow and scour around a vertical circular cylinder

Flow and scour around a vertical cylinder exposed to current are investigated by using a three-dimensional numerical model based on incompressible Reynolds-averaged Navier–Stokes equations. The model incorporates (i) k-ω turbulence closure, (ii) vortex shedding processes, (iii) sediment transport (both bed and suspended load), as well as (iv) bed morphology. The influence of vortex shedding and suspended load on the scour are specifically investigated. For the selected geometry and flow conditions, it is found that the equilibrium scour depth is decreased by 50% when the suspended sediment transport is not accounted for. Alternatively, the effects of vortex shedding are found to be limited to the very early stage of the scour process. Flow features such as the horseshoe vortex, as well as lee-wake vortices, including their vertical frequency variation, are discussed. Large-scale counter-rotating streamwise phase-averaged vortices in the lee wake are likewise demonstrated via numerical flow visualization. These features are linked to scour around a vertical pile in a steady current.
Numerical Simulation of Wave Interaction with Moving Net Structures

Fluid structure interaction is an important issue in design of many engineering systems including offshore fish cages. As a first step to develop a coupled CFD/FEM model, in the present study a moving porous media model has been developed in the open source library Open FOAM. The net structure was described as a sheet of porous media with prescribed rigid body motion and mesh motion was incorporated to conform the motion of the net. Free surface wave generation and absorption framework was also introduced to simulate wave interaction with moving net structures. The results showed that mesh motion solver was coupled with free surface wave generation and absorption framework and porous media model flawlessly, and the deformed mesh remained acceptable. It was concluded that Laplace mesh motionsolver with inverted quadratic diffusivity model was an appropriate option for simulating wave interaction with moving net structures.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Chen, H. (Intern), Christensen, E. D. (Intern)
Number of pages: 9
Publication date: 2015

On estimating the aerodynamic admittance of bridge sections by a mesh-free vortex method

A stochastic method of generating a synthetic turbulent flow field is combined with a 2D mesh-free vortex method to simulate the effect of an oncoming turbulent flow on a bridge deck cross-section within the atmospheric boundary layer. The mesh-free vortex method is found to be capable of preserving the a priori specified statistics as well as anisotropic characteristics of the synthesised turbulent flow field. From the simulation, the aerodynamic admittance is estimated and the instantaneous effect of a time varying angle of attack is briefly investigated. The obtained aerodynamic admittance of four aerodynamically different bridge sections is compared to available wind tunnel data, showing good agreement between the two.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, COWI A/S
Authors: Heijlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Larsen, A. (Ekstern), Walther, J. H. (Intern)
Pages: 117-127
Publication date: 2015
Main Research Area: Technical/natural sciences
On the modelling of the dynamics of elastically deformable floating structures

In this paper we are reexamining the dynamic equations of an elastically deformable floating structure to identify and evaluate the contribution from the inertia cross coupling terms which commonly have been neglected due to the assumption of small structural deformation. Numerical experiments on two vessels, a flexible barge, and a full scale ultra large container vessel, are designed for revealing the magnitude of errors introduced into the numerical solutions when these inertia cross coupling terms have been ignored. The results shows that in realistic conditions with strong structural dynamics, the errors are less than 1% for the flexible barge and less than 0.5% for the ultra large container vessel.
**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Bureau Veritas
Authors: Seng, S. (Ekstern), Malenica, S. (Ekstern), Jensen, J. J. (Intern), de Lauzon, J. (Ekstern)
Pages: 415-432
Publication date: 2015

**Host publication information**
Title of host publication: Proceedings of the 7th International Conference on Hydroelasticity in Marine Technology
Main Research Area: Technical/natural sciences
6DOF, Deformable body, Floating structures, Hydroelasticity
Publication: Research - peer-review › Article in proceedings – Annual report year: 2015

**Parametric Study of the Scavenging Process in Marine Two-Stroke Diesel Engines**
Large commercial ships such as container vessels and bulk carriers are propelled by low-speed, uniflow scavenged two-stroke diesel engines. The integral in-cylinder process in this type of engine is the scavenging process, where the burned gas from the combustion process is evacuated through the exhaust valve and replaced with fresh air for the subsequent compression stroke. The scavenging air enters the cylinder via inlet ports which are uncovered by the piston at bottom dead center (BDC). The exhaust gas is then displaced by the fresh air. The scavenging ports are angled to introduce a swirling component to the flow. The in-cylinder swirl is beneficial for air-fuel mixture, cooling of the cylinder liner and minimizing dead zones where pockets of exhaust gas are trapped. However, a known characteristic of swirling flows is an adverse pressure gradient in the center of the flow, which might lead to a local deficit in axial velocity and the formation of central recirculation zones, known as vortex breakdown. This paper will present a CFD analysis of the scavenging process in a MAN B&W two-stroke diesel engine. The study includes a parameter sweep where the operating conditions such as air amount, port timing and scavenging pressure are varied. The CFD model comprises the full geometry from scavenger receiver to exhaust receiver. Asymmetric inlet and outlet conditions are included as well as the dynamics of a moving piston and valve. Time resolved boundary conditions corresponding to measurements from an operating, full scale production engine as well as realistic initial conditions are used in the simulations. The CFD model provides a detailed description of the in-cylinder flow from exhaust valve opening (EVO) to exhaust valve closing (EVC). The study reveals a close coupling between the volume flow (delivery ratio) and the in-cylinder bulk purity of air which appears to be independent of operating conditions, rpm, scavenging air pressure, BMEP etc. The bulk purity of air in the cylinder shows good agreement with a simple theoretical perfect displacement model.

**Physiologically regulated valve-closure makes mussels long-term starvation survivors: test of hypothesis**

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Andersen, F. H. (Intern), Mayer, S. (Ekstern)
Number of pages: 9
Publication date: 2015

**Host publication information**
Title of host publication: Proceedings of the ASME 2015 Internal Combustion Engine Division Fall Technical Conference
Volume: 1
Publisher: American Society of Mechanical Engineers
Article number: ICEF2015-1075
ISBN (Print): 978-0-7918-5727-4
Main Research Area: Technical/natural sciences
Publication: Research - peer-review › Article in proceedings – Annual report year: 2015

**Physiologically regulated valve-closure makes mussels long-term starvation survivors: test of hypothesis**

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Southern Denmark
Authors: Riisgård, H. U. (Ekstern), Larsen, P. S. (Intern)
Number of pages: 5
Pages: 303-307
Publication date: 2015
Main Research Area: Technical/natural sciences
Prediction of First-Order Vessel Responses with Applications to Decision Support Systems

The paper presents a practical and simple approach for making vessel response predictions. Features of the procedure include a) predictions which are scaled so to better agree with corresponding true, future values to be measured at the time the predictions apply at; and b) predictions that are assigned an uncertainty measure to reflect a level of confidence. The approach is tested with full-scale data and the obtained results/predictions agree well with measured values. Potentially, the procedure is therefore very useful in future developments of general decision support systems.

General Information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Tokyo University of Marine Science and Technology
Reduction of noise and bias in randomly sampled power spectra

We consider the origin of noise and distortion in power spectral estimates of randomly sampled data, specifically velocity data measured with a burst-mode laser Doppler anemometer. The analysis guides us to new ways of reducing noise and removing spectral bias, e.g., distortions caused by modifications of the ideal Poisson sample rate caused by dead time effects and correlations between velocity and sample rate. The noise and dead time effects for finite records are shown to tend to previous results for infinite time records and ensemble averages. For finite records, we show that the measured sampling function can be used to correct the spectra for noise and dead time effects by a deconvolution process. We also describe a novel version of a power spectral estimator based on a fast slotted autocovariance algorithm.
Response-based sea state estimation for onboard DSS - Safe and Efficient Marine Operations

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Nielsen, U. D. (Intern)
Number of pages: 35
Publication date: 2015

Publication information
Media of output: PowerPoint
Original language: English
Main Research Area: Technical/natural sciences
Electronic versions:
WaveEstim_and_DSS_AMOS_Workshop_.pdf

Relations
Activities:
Response-based sea state estimation for onboard DSS - Safe and Efficient Marine Operations
Publication: Research › Sound/Visual production (digital) – Annual report year: 2015

Retrofitting the Superstructure of a Large Passenger Ship Using Composites – A Demonstration
In this work, the superstructure of a RoPax ferry has been redesigned using composite materials and the new design has been compared to the existing steel superstructure from a structural perspective. To this end, FE models have been developed and the superstructures have been subjected to loading conditions as prescribed from the regulations. Additionally, the effects that the composite superstructure has on the weight of the ship have been calculated. Results indicate that there is a large potential for retrofitting and building new passenger ships with composite superstructures as long as the design procedure and its acceptance by the regulatory bodies are simplified.
Scour at Breakwaters Under Combined Waves and Current

Bed profiles perpendicular to breakwater have been measured for orthogonal combined wave and current scour. The bed profiles show the distinct features of the wave scour forming due to the presence of recirculating cells, namely deposition/scour at the antinodes/nodes depending on whether the sediment transport occurs in the no-suspension or the suspension mode. Also, the current scour which forms by the presence of secondary flow as a result of the corner flow and the change in roughness is noticeable in the bed profiles. The mechanism of scour in both the fundamental cases (waves alone and the current alone) is that the sediment is mobilized by the primary flow while the secondary flow/recirculating cells cause a mean transport of sediment, leading to scour. For the orthogonal combined wave and current the mechanism appears to be similar.

Sea State Estimation Using Model-scale DP Measurements

Complex marine operations are moving further from shore, into deeper waters, and harsher environments. The operating hours of a vessel are weather dependent, and good knowledge of the prevailing weather conditions may ensure cost-efficient and safe operations. This paper considers the estimation of the peak wave frequency of the on-site sea state based on the vessel’s motion in waves. A sea state can be described by significant wave height, peak wave frequency, wave direction, and often wind speed and direction are added as well. The signal-based algorithm presented in this paper is based on Fourier transforms of the vessel response in heave, roll and pitch. The measurements are used directly to obtain an estimate of the peak frequency of the waves. Experimental results from model-scale offshore ship runs at the Marine Cybernetics Laboratory (MCLab) at NTNU demonstrate the performance of the proposed sea state estimation algorithm.
Simulating external flow using vortex method in two- and three dimensions

Vortex methods are numerical methods for simulating fluid flow. They use a simple formulation where only the trajectories of discrete vortex particles are simulated. In our method we combine a high order particle-mesh based vortex method with an iterative penalization method to simulate external flows around arbitrary geometries such as bridge decks. The method only uses a discretized geometry as input and can easily simulate an arbitrary motion of the geometry. As vorticity is a bounded quantity and the velocity field can easily be calculated for a mixture of free-space- and periodic boundary conditions, the method allows for a minimized domain and hence minimal computational resources. However in an external flow problem, vorticity is produced in the boundary layers and transported downstream, consequently the computational domain must grow in time to encapsulate the entire vorticity field. We present a method for truncating this domain by supplementing the free-space- and periodic conditions with an outflow condition. The method is conveniently applied within the field of bridge aerodynamics as it can be used for the calculation of the aerodynamic net forces, which depend highly on the geometry and the wake forming behind it. This is demonstrated in 2D and 3D simulations.

Simulation of external flows using a hybrid particle mesh vortex method

The long-term goal of this project is to develop and apply state-of-the-art simulation software to enable accurate prediction of fluid structure interaction, specifically vortex-induced-vibration and flutter of long-span suspension bridges to avoid error-prone structural designs. In the following a hybrid particle mesh vortex method is applied for the simulation of uniform flow past stationary solid obstacles of arbitrary shapes.

Sinking of armour layer around a vertical cylinder exposed to waves and current

The mechanisms of the sinking of a scour protection adjacent to a monopile are described in this paper, together with the determination of the equilibrium sinking depth in various wave and combined wave and current conditions based on physical model tests. Sinking of the rocks may ultimately lead to failure of the scour protection. It may cause exposure and free-span of cables, and possibly change the natural frequency of the wind turbine in an unfavourable manner. For these reasons it is important to consider the possible effects of sinking in the scour protection design, and to understand the mechanisms that could lead to unacceptable sinking of the scour protection. The study showed that the sinking is controlled by two mechanisms: removal of sediment adjacent to the pile (destabilizing) and infilling of sediment into the...
scour protection from the surrounding seabed (stabilizing). The latter mechanism is found to be the strongest, but it might take some time to fill the pores of the scour protection with sediment and during the time delay considerable sinking might take place. This means that the larger the scour protection, the larger the sinking will be (for a given KC-number smaller than approximately 15). The magnitude of the sinking of the scour protection adjacent to a monopile exposed to waves, and combined waves and current, was found to be similar to the current case.
Small Ro/Pax Vessel stability study

In 2009 new damage stability requirements for passenger ships based on a probabilistic method were adopted by IMO and are now part of the current SOLAS Chapter II-1 regulations (SOLAS 2009). The mandate from IMO was to keep the same safety level as inherent in the old deterministic damage stability regulations in SOLAS (SOLAS 90). During the rule development prior to the adoption, it was argued that the safety level for large passenger ships should be increased, but small ro/pax vessels were only rudimentarily looked at and small vessels with very high attained index were seen as “non-representative”. Currently there is a renewed debate in IMO regarding the required damage stability safety level for passenger ships. The damage stability safety level for small ro/pax vessels has also been discussed outside of the IMO assuming that the damage stability safety level for small ro/pax designs is perhaps not sufficient, i.e. that the current safety level according to SOLAS 2009 is less than the old safety level according to SOLAS 90. In order to establish a solid foundation for the discussion, this study was made possible by a grant from The Danish Maritime Fund. The study focus on small ro/pax vessels in a range from 32 m to 100 m in length and 100 to 600 passengers/persons, and the outcome of this study is described in details in this document.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register EMEA, HOK Marineconsult ApS, Søfartsstyrelsen
Authors: Erichsen, H. (Ekstern), Kristensen, H. O. H. (Intern), Jensen, J. J. (Intern), Tvedt, E. I. (Ekstern)
Number of pages: 20
Publication date: 2015

Host publication information
Title of host publication: Proceedings of the World Maritime Technology Conference
Publisher: Society of Naval Architects and Marine Engineers, The
Main Research Area: Technical/natural sciences
Source: PublicationPreSubmission
Source-ID: 116777291
Publication: Research - peer-review › Article in proceedings – Annual report year: 2015

Study on Short-term Variability of Ship Responses in Waves
Short-term variability of ship responses is investigated by cross-spectrum analysis. In a steady state condition, it is well known that a certain length of sampled data is required for stable results of the spectral analysis. However, the phase lag between responses, in terms of the phase angle of the cross-spectra, has not been discussed in detail. Using long stationary time series, the transition of amplitudes and relative phase angles of the cross-spectra has been investigated by iterative analyzes with a few seconds of time shifting. In the results, the short-term variability of the relative phase angle
was observed. In effect, the variability may compromise the accuracy of the wave buoy analogy.

**Sustaining dry surfaces under water**

Rough surfaces immersed under water remain practically dry if the liquid-solid contact is on roughness peaks, while the roughness valleys are filled with gas. Mechanisms that prevent water from invading the valleys are well studied. However, to remain practically dry under water, additional mechanisms need consideration. This is because trapped gas (e.g. air) in the roughness valleys can dissolve into the water pool, leading to invasion. Additionally, water vapor can also occupy the roughness valleys of immersed surfaces. If water vapor condenses, that too leads to invasion. These effects have not been investigated, and are critically important to maintain surfaces dry under water. In this work, we identify the critical roughness scale, below which it is possible to sustain the vapor phase of water and/or trapped gases in roughness valleys – thus keeping the immersed surface dry. Theoretical predictions are consistent with molecular dynamics simulations and experiments.
The mussel filter–pump – present understanding, with a re-examination of gill preparations

Filter feeding in mussels is a secondary adaptation where the gills have become W-shaped and greatly enlarged, acting as the mussel filter–pump. Water pumping and particle capture in the blue mussel, Mytilus edulis, have been studied over many years. Here, we give a short status of the present understanding of ciliary structure and function of the mussel filter–pump, supplemented with new photo-microscope and scanning electron microscopy (SEM) pictures of gill preparations. Pumping rate (filtration) and pressure to maintain flow have been extensively studied so the power delivered by the mussel pump to the water flow is known (1.1% of total respiratory power), but the actual cost based on gill respiration is much higher (19%), implying that the cost of maintaining of the large gill pump is considerable and that only relatively little energy can be saved by stopping or reducing the activity of the water-pumping cilia so that continuous feeding with a ‘minimal scaled’ pump is cheaper than discontinuous feeding with a correspondingly larger pump.

According to the present view, the pump proper is the beating lateral cilia (lc) on the gill filaments and particle capture is accomplished by the action of laterofrontal cirri (lfc) transferring particles from the main water current to the frontal gill filament currents driven by frontal cilia (fc). Unexplained aspects include retention efficiency according to particle size and the role of pro-laterofrontal cirri (p-lfc) placed between the lfc and fc. The structure of cilia and the mode of ciliary beating have been re-examined in this study by new high-resolution light and scanning electron microscopy of isolated gill preparations exposed to serotonin (5-HT) stimulation which can activate the lc and lfc at low concentrations (10^{-6} M), but removes the lfc from the interfilament canals at higher concentrations (10^{-5} M).

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Aarhus University, University of Southern Denmark
Authors: Riisgård, H. U. (Ekstern), Funch, P. (Ekstern), Larsen, P. S. (Intern)
Pages: 273-282
Trend modelling of wave parameters and application in onboard prediction of ship responses

This paper presents a trend analysis for prediction of sea state parameters onboard ships during voyages. Given those parameters, a JONSWAP model and also the transfer functions, prediction of wave induced ship responses are thus made. The procedure is tested with full-scale data of an in-service container ship. Comparison between predictions and the actual measurements, implies a good agreement in general. This method can be an efficient way to improve decision support on board ships.
Uncertainties in ship-based estimation of waves and responses
Real time estimation of waves and ship responses using onboard measurements has been under investigation in recent years. This has been done using different methods, including parametric and non-parametric models. Since none of the methods are believed to be fully accurate, it is important to assign an uncertainty measure to the waves and responses that are being estimated. In this paper, a parametric model approach based on moments of responses is considered for wave estimation. A method based on linear error propagation is introduced to assess the uncertainty of wave estimations. The uncertainty of response calculation based on the estimated wave is also quantified.

Water transport in graphene nano-channels
The transport of water in nanopores is of both fundamental and practical interest. Graphene Channels (GCs) are potential building blocks for nanofluidic devices due to their molecularly smooth walls and exceptional mechanical properties. Numerous studies have found a significant flow rate enhancement, defined as the ratio of the computed flow rate to that predicted from the classical Poiseuille model. Moreover, these studies point to the fact that the flow enhancement is a function of channel height and the fluid-wall physical-chemistry. In spite of the intensive research, an explicit relation between the chirality of the graphene walls and the slip length has not been established. In this study, we perform non-equilibrium molecular dynamics simulations of water flow in single- and multi-walled GCs. We examine the influence on the flow rates of dissipating the viscous heat produced by connecting the thermostat to the water molecules, the CNT wall atoms or both of them. From the atomic trajectories, we compute the fluid flow rates in GCs with zig-zag and armchair walls, heights from 1 to 4 nm and different number of graphene layers on the walls. A relation between the chirality, slip length, and flow enhancement is found.
A high-order finite-difference solver for the linearised potential flow wave resistance problem on curvilinear overset grids

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Amini Afshar, M. (Intern), Bingham, H. B. (Intern), Read, R. (Intern)
Number of pages: 4
Publication date: 2014

Host publication information
Title of host publication: Proceedings of the International Workshop on Water Waves and Floating Bodies 2014
Publisher: IWWWFB
Main Research Area: Technical/natural sciences
Conference: 29th International Workshop on Water Waves and Floating Bodies, Osaka, Japan, 30/03/2014 - 30/03/2014
Electronic versions:
A_high_order_finite_difference_solver.pdf
Links:
http://www.iwwwfb.org/Workshops/29.htm

Allometric equations for maximum filtration rate in blue mussels Mytilus edulis and importance of condition index
The relationship between body dry weight (W) and shell length (L) of blue mussels, Mytilus edulis, can be expressed by the condition index (CI = W/L^3) which varies from population to population and during the year. Here, we examine the influence of CI on the relationships between maximum filtration rate (F, l h^{-1}), W (g), and L (mm) as described by the equations: FW = aW^b and FL = cL^d, respectively. This is done by using available and new experimental laboratory data on M. edulis obtained by members of the same research team using different methods and controlled diets of cultivated algal cells. For all data, it was found that FW = 6.773W^{0.678} and FL = 0.00135L^{2.088} which are very similar to equations for mussels with ‘medium condition’ (CI = 4–6 mg cm^{-3}): FW = 6.567W^{0.681} and FL = 0.00150L^{2.051}, with b- and d-values within a few percent of the theoretically expected of 2/3 and 2, respectively. Further, based on the present data, we propose a correction factor expressed by the empirical relation FW/FL = 0.3562CI^{2/3} which implies that FW tends to underestimate the actual filtration rate (FL) when CI<4.70 and to overestimate the filtration rate when CI>4.70.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Southern Denmark, City University of Hong Kong
Authors: Riisgård, H. U. (Ekstern), Larsen, P. S. (Intern), Pleissner, D. (Ekstern)
Pages: 193-198
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Helgoland Marine Research
Volume: 68
Issue number: 1
ISSN (Print): 1438-387X
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.611 SNIP 0.73 CiteScore 1.34
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.79 SNIP 0.877 CiteScore 1.43
Analyses of hydraulic performance of velocity caps

The hydraulic performance of a velocity cap has been investigated. Velocity caps are often used in connection with offshore intakes. CFD (computational fluid dynamics) examined the flow through the cap openings and further down into the intake pipes. This was combined with dimension analyses in order to analyse the effect of different layouts on the flow characteristics. In particular, flow configurations going all the way through the structure were revealed. A couple of suggestions to minimize the risk for flow through have been tested.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Rambøll Oil and Gas
Authors: Christensen, E. D. (Intern), Degn Eskesen, M. C. (Ekstern), Buhrkall, J. (Ekstern), Jensen, B. (Intern)
Number of pages: 10
Publication date: 2014

Host publication information
Title of host publication: Book of Proceedings
Publisher: IAHR
Main Research Area: Technical/natural sciences
Conference: 3rd IAHR Europe Congress, Porto, Portugal, 14/04/2014 - 14/04/2014
Electronic versions:
ANALYSES_OF_HYDRAULIC_PERFORMANCE.pdf
An efficient domain decomposition strategy for wave loads on surface piercing circular cylinders

A fully nonlinear domain decomposed solver is proposed for efficient computations of wave loads on surface piercing structures in the time domain. A fully nonlinear potential flow solver was combined with a fully nonlinear Navier–Stokes/VOF solver via generalized coupling zones of arbitrary shape. Sensitivity tests of the extent of the inner Navier–Stokes/VOF domain were carried out. Numerical computations of wave loads on surface piercing circular cylinders at intermediate water depths are presented. Four different test cases of increasing complexity were considered: 1) weakly nonlinear regular waves on a sloping bed, 2) phase-focused irregular waves on a flat bed, 3) irregular waves on a sloping bed and 4) multidirectional irregular waves on a sloping bed. For all cases, the free surface elevation and the inline force were successfully compared against experimental measurements.
A non-linear wave decomposition model for efficient wave–structure interaction. Part A: Formulation, validations and analysis

This paper deals with the development of an enhanced model for solving wave–wave and wave–structure interaction problems. We describe the application of a non-linear splitting method originally suggested by Di Mascio et al. [1], to the high-order finite difference model developed by Bingham et al. [2] and extended by Engsig-Karup et al. [3] and [4]. The enhanced strategy is based on splitting all solution variables into incident and scattered fields, where the incident field is assumed to be known and only the scattered field needs to be computed by the numerical model. Although this splitting technique has been applied to both potential flow and Navier–Stokes solvers in the past, it has not been thoroughly described and analyzed, nor has it been presented in widely read journals. Here we describe the method in detail and carefully analyze its performance using several 2D linear and non-linear test cases. In particular, we consider the extreme case of non-linear waves up to the point of breaking reflecting from a vertical wall; and conclude that no limitations are imposed by adopting this splitting. The advantages of this strategy in terms of robustness, accuracy and efficiency are also demonstrated by comparison with the more common strategy of solving the incident and scattered fields together.
A numerical and experimental study of the scavenging process in a two-stroke marine diesel engine

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Hemmingsen, C. S. (Intern), Ingvorsen, K. M. (Intern), Walther, J. H. (Intern), Meyer, K. E. (Intern)
Number of pages: 1
Publication date: 2014
Event: Abstract from 10th European Fluid Mechanics Conference (EUROMECH), KGs. Lyngby, Denmark.
Main Research Area: Technical/natural sciences
Source: PublicationPreSubmission
Source-ID: 101143626
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2014

Application of CFD based wave loads in aeroelastic calculations
Two fully nonlinear irregular wave realizations with different significant wave heights are considered. The wave realizations are both calculated in the potential flow solver Ocean-Wave3D and in a coupled domain decomposed potential-flow CFD solver. The surface elevations of the calculated wave realizations compare well with corresponding surface elevations from laboratory experiments.

In aeroelastic calculations of an offshore wind turbine on a monopile foundation the hydrodynamic loads due to the potential flow solver and Morison's equation and the hydrodynamic loads calculated by the coupled domain decomposed potentialflow CFD solver result in different dynamic forces in the tower and monopile, despite that the static forces on a fixed monopile are similar. The changes are due to differences in the force profiles and wave steepness in the two solvers. The results indicate that an accurate description of the wave loads is very important in aeroelastic calculations especially in cases where the aerodynamic loads and damping are insignificant.

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Schløer, S. (Intern), Paulsen, B. T. (Intern), Bredmose, H. (Intern)
Number of pages: 11
Publication date: 2014

Host publication information
Title of host publication: Proceedings of the ASME 33rd 2014 International Conference on Ocean, Offshore and Arctic Engineering
Publisher: American Society of Mechanical Engineers
Article number: OMAE2014-24684
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: 33rd International Conference on Ocean, Offshore and Arctic Engineering, San Francisco, CA, United States, 08/06/2014 - 08/06/2014
Source: PublicationPreSubmission
Source-ID: 97207575
Publication: Research - peer-review › Article in proceedings – Annual report year: 2014

Application of finite elements and computational fluid dynamics to predict and improve the filling ratio in journal bearings under dynamic loading

General information
State: Published
Organisations: Department of Mechanical Engineering, Solid Mechanics, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Christiansen, C. K. (Intern), Klit, P. (Intern), Walther, J. H. (Intern), Vølund, A. (Ekstern)
Number of pages: 4
Publication date: 2014

Host publication information
Title of host publication: Proceedings of NSCM-27: the 27th Nordic Seminar on Computational Mechanics
Editors: Eriksson, A., Kulachenko, A., Mihaescu, M., Tibert, G.
Aquaculture as a part of a multi-use platform

European oceans will be subject to massive development of marine infrastructure in the near future. The most obvious is the energy facilities e.g. offshore wind farms, exploitation of wave energy, expansion of electricity connections, and also further development and implementation of marine aquaculture. These developments urgently require effective marine technology and governance solutions to facilitate installation, operation and maintenance of these novel offshore activities. Simultaneously, both economic costs and environmental impact have to remain within acceptable limits, in order to increase the feasibility of the use of ocean space. Aquaculture can play an important role in the multi-use of ocean space. This idea is tested on four different sites around Europe, where this paper focus on the one in the Baltic Sea.

A review of recent advances in numerical modelling of local scour problems

A review is presented of recent advances in numerical modelling of local scour problems. The review is organized in five sections: Highlights of numerical modelling of local scour; Influence of turbulence on scour; Backfilling of scour holes; Scour around complex structures; and Scour protection around offshore wind turbine foundations.

Cavitation analysis of a journal bearing - Finite Element modelling and experimental studies

State: Published
Organisations: Department of Mechanical Engineering, Solid Mechanics, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Christiansen, C. K. (Intern), Klit, P. (Intern), Walther, J. H. (Intern), Vølund, A. (Ekstern)
Number of pages: 1
CFD analysis of the scavenging process in marine two-stroke diesel engines
The scavenging process is an integral part of any two-stroke internal combustion engine regardless of being spark ignited (SI) or compression ignited (CI). The scavenging process is responsible for replacing the burned gas from the combustion process from the previous working stroke with fresh air/charge before the subsequent compression stroke. This implies that the scavenging process is integral to engine performance as it influence the initial condition for the combustion process, thus affecting the fuel economy, power output and emission of hazardous gases. Two-stroke diesel engines for marine propulsion normally operates by the uniflow scavenging method, where the scavenge air enters the cylinder via inlet ports located near the bottom dead center and exits through one or several exhaust valves located in the cylinder head. This arrangement concentrates the airflow in one direction through the cylinder thus giving the method its name. The inlet ports are angled with respect to the local radius which will introduce a tangential velocity component to the airflow. The air moves axially through the cylinder in a swirling motion that favors mixing of fuel and air as the injected fuel is transported with the swirling air in the combustion chamber during fuel injection. A known characteristic of swirling flows is an adverse pressure gradient in the center of the rotating flow which might lead to a local deficit in axial velocity and the formation of central recirculation zones, known as vortex breakdown. Optimal scavenging is achieved when the gas exchange is done by displacement, the local deficit in axial velocity will increase the mixing of burned gas and scavenge air thus decreasing the amount of pure displacement.

CFD investigation of a transonic pulse-jet in a fabric filter applicaton
The scavenging process is an integral part of any two-stroke internal combustion engine regardless of being spark ignited (SI) or compression ignited (CI). The scavenging process is responsible for replacing the burned gas from the combustion process from the previous working stroke with fresh air/charge before the subsequent compression stroke. This implies that the scavenging process is integral to engine performance as it influence the initial condition for the combustion process, thus affecting the fuel economy, power output and emission of hazardous gases. Two-stroke diesel engines for marine propulsion normally operates by the uniflow scavenging method, where the scavenge air enters the cylinder via inlet ports located near the bottom dead center and exits through one or several exhaust valves located in the cylinder head. This arrangement concentrates the airflow in one direction through the cylinder thus giving the method its name. The inlet ports are angled with respect to the local radius which will introduce a tangential velocity component to the airflow. The air moves axially through the cylinder in a swirling motion that favors mixing of fuel and air as the injected fuel is transported with the swirling air in the combustion chamber during fuel injection. A known characteristic of swirling flows is an adverse pressure gradient in the center of the rotating flow which might lead to a local deficit in axial velocity and the formation of central recirculation zones, known as vortex breakdown. Optimal scavenging is achieved when the gas exchange is done by displacement, the local deficit in axial velocity will increase the mixing of burned gas and scavenge air thus decreasing the amount of pure displacement.
Collision analysis of ships and jack-up rigs

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register Asia, Lloyd's Register EMEA
Authors: Zhang, S. (Ekstern), Pedersen, P. T. (Intern), Ocakli, H. (Ekstern)
Number of pages: 9
Publication date: 2014

Host publication information
Title of host publication: Proceedings of the ICTWS 2014 7th International Conference on Thin-Walled Structures
Article number: ICTWS2014-1404
Main Research Area: Technical/natural sciences
Conference: 7th International Conference on Thin-Walled Structures, Busan, Korea, Republic of, 28/09/2014 - 28/09/2014

Continuum simulations of water flow in carbon nanotube membranes
We propose the use of the Navier–Stokes equations subject to partial-slip boundary conditions to simulate water flows in Carbon NanoTube (CNT) membranes. The finite volume discretizations of the Navier–Stokes equations are combined with slip lengths extracted from molecular dynamics (MD) simulations to predict the pressure losses at the CNT entrance as well as the enhancement of the flow rate in the CNT. The flow quantities calculated from the present hybrid approach are in excellent agreement with pure MD results while they are obtained at a fraction of the computational cost. The method enables simulations of system sizes and times well beyond the present capabilities of MD simulations. Our simulations provide an asymptotic flow rate enhancement and indicate that the pressure losses at the CNT ends can be reduced by reducing their curvature. More importantly, our results suggest that flows at nanoscale channels can be described by continuum solvers with proper boundary conditions that reflect the molecular interactions of the liquid with the walls of the nanochannel.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, National Institute of Chemistry, Swiss Federal Institute of Technology
Authors: Popadić, A. (Ekstern), Walther, J. H. (Intern), Koumoutsakos, P. (Ekstern), Praprotnik, M. (Ekstern)
Number of pages: 11
Publication date: 2014
Main Research Area: Technical/natural sciences

Journal: New Journal of Physics
Volume: 16
Article number: 082001
ISSN (Print): 1367-2630
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed Yes  
BFI (2016): BFI-level 2  
Scopus rating (2016): CiteScore 2.97 SJR 1.788 SNIP 1.031  
Web of Science (2016): Indexed yes  
BFI (2015): BFI-level 2  
Scopus rating (2015): SJR 1.938 SNIP 1.047 CiteScore 2.8  
Web of Science (2015): Indexed yes  
BFI (2014): BFI-level 2  
Scopus rating (2014): SJR 2.806 SNIP 1.307 CiteScore 2.89  
Web of Science (2014): Indexed yes  
BFI (2013): BFI-level 2  
Scopus rating (2013): SJR 2.871 SNIP 1.372 CiteScore 2.77  
ISI indexed (2013): ISI indexed yes  
Web of Science (2013): Indexed yes  
BFI (2012): BFI-level 2  
Scopus rating (2012): SJR 3.352 SNIP 1.533 CiteScore 3.4  
ISI indexed (2012): ISI indexed yes  
Web of Science (2012): Indexed yes  
BFI (2011): BFI-level 2  
Scopus rating (2011): SJR 3.47 SNIP 1.634 CiteScore 3.99  
ISI indexed (2011): ISI indexed yes  
Web of Science (2011): Indexed yes  
BFI (2010): BFI-level 2  
Scopus rating (2010): SJR 3.395 SNIP 1.421  
Web of Science (2010): Indexed yes  
BFI (2009): BFI-level 2  
Scopus rating (2009): SJR 3.215 SNIP 1.503  
Web of Science (2009): Indexed yes  
BFI (2008): BFI-level 2  
Scopus rating (2008): SJR 2.913 SNIP 1.396  
Web of Science (2008): Indexed yes  
Scopus rating (2007): SJR 2.825 SNIP 1.354  
Web of Science (2007): Indexed yes  
Scopus rating (2006): SJR 2.2 SNIP 1.296  
Web of Science (2006): Indexed yes  
Scopus rating (2005): SJR 1.641 SNIP 1.116  
Web of Science (2005): Indexed yes  
Scopus rating (2004): SJR 1.211 SNIP 1.009  
Web of Science (2004): Indexed yes  
Scopus rating (2003): SJR 1.057 SNIP 0.75  
Scopus rating (2002): SJR 0.77 SNIP 0.666  
Web of Science (2002): Indexed yes  
Scopus rating (2001): SJR 1.033 SNIP 0.843  
Web of Science (2001): Indexed yes  
Scopus rating (2000): SJR 1.326 SNIP 1.307  
Scopus rating (1999): SJR 0.737 SNIP 0.26

Original language: English  
Electronic versions:  
Popadic_2014.pdf  
DOIs:  
doi:10.1088/1367-2630/16/8/082001  
Source: PublicationPreSubmission  
Source-ID: 98137819  
Publication: Research - peer-review › Journal article – Annual report year: 2014
Continuum Simulations of Water Flow in Carbon Nanotube Membranes

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, National Institute of Chemistry, Swiss Federal Institute of Technology
Authors: Walther, J. H. (Intern), Popadic, A. (Ekstern), Koumoutsakos, P. (Ekstern), Praprotnik, M. (Ekstern)
Number of pages: 1
Publication date: 2014
Event: Abstract from 67th Annual Meeting of the APS Division of Fluid Dynamics, San Francisco, CA, United States.
Main Research Area: Technical/natural sciences

Corrigendum to "Second-order theory for coupling 2D numerical and physical wave tanks-Derivation, evaluation and experimental validation" [Coast. Eng. 71 (2013) 37–51]

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Dalian University of Technology
Authors: Yang, Z. (Ekstern), Liu, S. (Ekstern), Bingham, H. B. (Intern), Li, J. (Ekstern)
Pages: 87-88
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Coastal Engineering
Volume: 85
ISSN (Print): 0378-3839
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.925 SNIP 2.097 CiteScore 2.9
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.785 SNIP 2.123 CiteScore 2.55
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.727 SNIP 2.264 CiteScore 2.58
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.912 SNIP 2.226 CiteScore 2.21
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.616 SNIP 2.502 CiteScore 2.43
Cross-shore redistribution of nourished sand near a breaker bar

This paper focuses on the optimal location for dumping nourished sand on a barred coastline. This is done by investigating the short-term behavior of the cross-shore redistribution of nourished sediment on a breaker-bar profile in a two-dimensional vertical plane. This is achieved by the use of a complete numerical description of the surf-zone processes with respect to both hydrodynamics and sediment transport. The numerical model is based on the finite-volume approach with a free surface-tracking method, also known as the volume of fluid (VOF), and the sediment transport is calculated applying the Engelund and Fredsøe deterministic concept. The methodology is as follows: a Dean/Bruun equilibrium profile is exposed to regular waves (chosen as H = 1.3 m and T = 4.8 s). These waves will form bars on the Dean/Bruun equilibrium profile, and the wave impact is continued until a quasi-steady behavior of the bars has been obtained, i.e., bars that have stopped growing in height and only migrate slowly offshore. Next, the profile is nourished. The resulting continued morphologic developments are compared with the development of the unnourished (reference) profile. Also, the effect of a net shoreward current over the profile is considered as a means of imposing the net recirculation resulting from longshore nonuniformities or the impact of wind-induced shear stresses. The impact of wave height, irregularity in incident waves, and the size of the sediment grains on the cross-shore distribution of suspended sediment transport rates are described. In addition, all profiles are seen to lose sediment seaward when no net shoreward current is enforced over the outermost breaker bar. Alternative strategies for nourishment are investigated, and it is seen that the shape on the nourishment seaward of the outermost bar can have both destructive and protective impacts on the original shoreward-located cross-shore profile.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Deltares
Authors: Jacobsen, N. G. (Ekstern), Fredsøe, J. (Intern)
Pages: 125-134
Publication date: 2014
Data-driven Vessel Performance Monitoring

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Applied Mathematics and Computer Science, Cognitive Systems, FORCE Technology
Authors: Pedersen, B. P. (Intern), Andersen, P. (Intern), Larsen, J. (Intern), Sinding, P. (Ekstern)
Number of pages: 272
Publication date: 2014

Publication information
Publisher: DTU Mechanical Engineering
ISBN (Electronic): 978-87-7475-408-4
Original language: English
Series: DCAMM Special Report
Number: S180
ISSN: 0903-1685
Main Research Area: Technical/natural sciences
Electronic versions:
S180_Benjamin_Pjedsted_Pedersen.pdf
Source: PublicationPreSubmission
Source-ID: 107670293
Publication: Research › Ph.D. thesis – Annual report year: 2015

Dead time effects in laser Doppler anemometry measurements
We present velocity power spectra computed by the so-called direct method from burst-type laser Doppler anemometer (LDA) data, both measured in a turbulent round jet and generated in a computer. Using today’s powerful computers, we have been able to study more properties of the computed spectra than was previously possible, and we noted some unexpected features of the spectra that we now attribute to the unavoidable influence of a finite measurement volume (MV). The most prominent effect, which initially triggered these studies, was the appearance of damped oscillations in the higher frequency range, starting around the cutoff frequency due to the finite size of the MV. Using computer-generated data mimicking the LDA data, these effects have previously been shown to appear due to the effect of dead time, i.e., the finite time during which the system is not able to acquire new measurements. These dead times can be traced back to the fact that the burst-mode LDA cannot measure more than one signal burst at a time. Since the dead time is approximately equal to the residence time for a particle traversing a measurement volume, we are dealing with widely varying dead times, which, however, are assumed to be measured for each data point. In addition, the detector and processor used in the current study introduce a certain amount of fixed processing and data transfer times, which further contribute to the distortion of the computed spectrum. However, we show an excellent agreement between a measured spectrum and our modeled LDA data, thereby confirming the validity of our model for the LDA burst processor.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Intarsia Optics, Princeton University
Authors: Velte, C. M. (Intern), Buchhave, P. (Ekstern), George, W. K. (Ekstern)
Number of pages: 10
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Experiments in Fluids
Volume: 55
Issue number: 11
Article number: 1836
ISSN (Print): 0723-4864
Ratings:
BFI (2018): BFI-level 1
Design of cooling water intake structure CFD modelling case study

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DHI Denmark
Authors: Hjelmager Jensen, J. (Intern), Jensen, B. (Intern), Pedersen, C. (Ekstern), Fredsøe, J. (Intern)
Number of pages: 1

Original language: English
DOIs:
10.1007/s00348-014-1836-z
Publication: Research - peer-review › Journal article – Annual report year: 2014
Development of a numerical 2-dimensional beach evolution model
This paper presents the description of a 2-dimensional numerical model constructed for the simulation of beach evolution under the action of wind waves only over the arbitrary land and sea topographies around existing coastal structures and formations. The developed beach evolution numerical model is composed of 4 submodels: a nearshore spectral wave transformation model based on an energy balance equation including random wave breaking and diffraction terms to compute the nearshore wave characteristics, a nearshore wave-induced circulation model based on the nonlinear shallow water equations to compute the nearshore depth-averaged wave-induced current velocities and mean water level changes, a sediment transport model to compute the local total sediment transport rates occurring under the action of wind waves, and a bottom evolution model to compute the bed level changes in time based on the gradients of sediment transport rates in cross-shore and longshore directions. The developed models are applied successfully to the SANDYDUCK field experiments and to some conceptual benchmark cases including simulation of rip currents around beach cusps, beach evolution around a single shore perpendicular groin, and a series of offshore breakwaters. The numerical model gave results in agreement with the measurements both qualitatively and quantitatively and reflected the physical concepts well for the selected conceptual cases.
Development of Smoothed Particle Hydrodynamics for Flow in Complex Geometries and Application of Open Source Software for the Simulation of Turbulent Flow

Turbulence modelling is a key issue in many industrial application, as the computational power of direct numerical simulation (DNS) is insufficient to deal with complex flow structures with high Reynolds number. Also in industrial applications often involve turbulent flow in complex geometries. Thus developing a computational method which can deal with complex fluid structure, simulate complex geometries that change topology is particularly challenging as the connectivity of the computational domain may change dynamically, and still efficient is important.

In this thesis we are presenting a remeshed particle-mesh method, the method involves three-dimensional compressible turbulent flow modelling, and coupled with an immersed boundary technique to deal with the complex solid obstacles. This dissertation is composed of three parts.

In combustion engines the scavenging process in two-stroke marine diesel engines removes combustion gases from the engine cylinder and fills up the cylinder with the fresh air charge for the next cycle. Understanding the scavenging flow is crucial for the development of such engines, since it affects fuel consumption, engine cooling and production of pollutants. We consider a state-of-the art eulerian methods to study the turbulent flow in a model diesel engine. the goals of this study include validation of large eddy simulations (LES) turbulence models.

Dwarfism of blue mussels in the low saline Baltic Sea — growth to the lower salinity limit
Mussels within the Baltic Mytilus edulis × M. trossulus hybrid zone have adapted to the low salinities in the Baltic Sea which, however, results in slow-growing dwarfed mussels. To get a better understanding of the nature of dwarfism, we studied the ability of M. trossulus to feed and grow at low salinity (7 psu) compared with its performance at relatively high-salinity (20 psu) in controlled laboratory experiments, supplemented with field (Great Belt) growth experiments with M. trossulus and M. edulis in net-bags. Subsequently, the growth of M. trossulus transplanted in cages to various localities in
the northern Baltic Sea was used to evaluate the effect of very low salinities, down to 3.4 psu. The laboratory feeding experiments with M. trossulus at 7 psu showed that the growth in shell length was negligible, whereas the body dry weight nearly doubled during the 15 d experiment, with a weight-specific growth rate of 3.7% d⁻¹. The same parameters measured at 20 psu showed a pronounced growth in both shell length and body dry weight, with a weight-specific growth rate of 2.2% d⁻¹. The growth rates of M. trossulus and M. edulis in suspended net-bags in the Great Belt (22 psu) were similar: 5.6 and 6.8% d⁻¹, respectively. M. trossulus in cage experiments had positive growth rates at locations with salinities above 4.5 psu, up to 2.60% d⁻¹, but negligible increase in the shell length, and at sites with salinities below about 4.5 psu, the somatic growth was negative, around −0.3% d⁻¹, which indicates valve closure and respiratory weight loss. A trend line in a plot of all available growth data for both mussel species as a function of salinity indicates that the growth of mussels is steadily hampered by reduced salinities from 30 psu down to about 10 psu, below which the growth is rapidly reduced to become negative below 4.5 psu. We suggest that reduced ability to produce shell material at extremely low salinity may explain dwarfism of mussels in the Baltic Sea. Reduced bio-calcification at low salinity, however, may impede shell growth, but not somatic growth, and this may at first result in an increased condition index, as seen in the benthic Baltic Sea mussels transferred to cages suspended in the water column.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Finnish Environment Institute, University of Southern Denmark
Authors: Riisgård, H. U. (Ekstern), Larsen, P. S. (Intern), Turja, R. (Ekstern), Lundgreen, K. (Ekstern)
Pages: 181–192
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Marine Ecology - Progress Series
Volume: 517
ISSN (Print): 0171-8630
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.4
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): CiteScore 2.56
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): CiteScore 2.75
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): CiteScore 2.79
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): CiteScore 2.9
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): CiteScore 2.85
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 2
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 2
Early regimes of water imbibition in nanoslit silica channels

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Oyarzua, E. (Ekstern), Zambrano, H. (Ekstern), Walther, J. H. (Intern), Mejia, A. (Ekstern)
Number of pages: 1
Publication date: 2014
Event: Abstract from 67th Annual Meeting of the APS Division of Fluid Dynamics, San Francisco, CA, United States.
Main Research Area: Technical/natural sciences
Electronic versions:
Abstract
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2014

Edge scour at scour protections around offshore wind turbine foundations
When building offshore wind turbines with mono-pile foundations, scour protection is typically placed to avoid scouring of the soil close to the mono-pile. An important aspect is that the scour protection itself causes erosion, inflicted by the local increase in the hydrodynamic field and in turn increased bed shear stresses. Scour of the edge material alongside the scour protection may cause deformations and failure of the scour protection. This can reduce the stability of the stone layer and cause exposure of cables running between the mono-piles. The early stage results of an extensive experimental campaign to explain the edge sour process in steady current is presented. The three-dimensional flow field around the pile and scour protection is resolved by particle image velocimetry, showing a local increase in the hydrodynamic field leading to increased sediment transport and scour. The governing process in steady current is a pair of symmetrical counter-rotating vortices emerging in the near bed region in the wake of the pile and scour protection, causing a significant downstream scour hole.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Petersen, T. U. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern)
Pages: 587-592
Publication date: 2014

Host publication information
Title of host publication: Scour and Erosion
Publisher: C R C Press LLC
Editors: Cheng, L., Draper, S., An, H.
ISBN (Print): 978-1-138-02732-9
Main Research Area: Technical/natural sciences
Conference: 7th International Conference on Scour and Erosion, Scarborough, Perth, Australia, 02/12/2014 - 02/12/2014
Publication: Research - peer-review › Article in proceedings – Annual report year: 2014
Estimation of burst-mode LDA power spectra
The estimation of power spectra from LDA data provides signal processing challenges for fluid dynamicists for several reasons: acquisition is dictated by randomly arriving particles, the registered particle velocities tend to be biased toward higher values, and the signal is highly intermittent. The signal can be interpreted correctly by applying residence time weighting to all statistics and using the residence time-weighted discrete Fourier transform to compute the Fourier transform. A new spectral algorithm using the latter is applied to two experiments: a cylinder wake and an axisymmetric turbulent jet. These are compared with corresponding hot-wire spectra as well as to alternative algorithms for LDA signals such as the time-slot correlation method, sample-and-hold and common weighting schemes.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Princeton University, Intarsia Optics
Authors: Velte, C. M. (Intern), George, W. K. (Ekstern), Buchhave, P. (Ekstern)
Number of pages: 20
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Experiments in Fluids
Volume: 55
Issue number: 3
Article number: 1674
ISSN (Print): 0723-4864
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.994 SNIP 1.324 CiteScore 2.18
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.193 SNIP 1.592 CiteScore 2.04
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.235 SNIP 1.721 CiteScore 2.21
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.425 SNIP 1.927 CiteScore 2.41
ISI indexed (2013): ISI indexed yes
Evaluation of the DeepWind concept
The report describes the DeepWind 5 MW conceptual design as a baseline for results obtained in the scientific and technical work packages of the DeepWind project. A comparison of DeepWind with existing VAWTs and paper projects are carried out and the evaluation of the concept in terms of cost, as well as the technical and scientific recommendations are performed. The work is a result of the contributions within the DeepWind project which is supported by the European Commission, Grant 256769 FP7 Energy 2010 - Future emerging technologies, and by the DeepWind beneficiaries: DTU(DK), AAU(DK), TUDELFT(NL), TUTRENTO(I), DHI(DK), SINTEF(N), MARINTEK(N), MARIN(NL), NREL(USA), STATOIL(N), VESTAS(DK) and NENUPHAR(F).

General information
State: Published
Organisations: Department of Electrical Engineering, Department of Wind Energy, Test and Measurements, Aeroelastic Design, Wind Turbines, Fluid Mechanics, Department of Mechanical Engineering, Manufacturing Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Nenuphar, Delft University of Technology, SINTEF, University of Trento, National Renewable Energy Laboratory, Aalborg University, DHI Denmark, Norwegian Marine Technology Research Institute, MARIN, Statool ASA
Number of pages: 26
Publication date: 2014
Flow and sediment transport induced by plunging waves

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Sumer, B. M. (Intern), Güner, A. (Intern), Hansen, N. M. (Intern), Fuhrman, D. R. (Intern), Fredsøe, J. (Intern)
Number of pages: 1
Publication date: 2014

Host publication information
Title of host publication: Abstract Book of the 34th International Conference on Coastal Engineering
BFI conference series: Proceedings of the International Conference on Coastal Engineering (5010056)
Main Research Area: Technical/natural sciences
Conference: 34th International Conference on Coastal Engineering, Seoul, Korea, Republic of, 15/06/2014 - 15/06/2014
Electronic versions:
FLOW_AND_SEDIMENT_TRANSPORT.pdf
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2014

Flow-structure-seabed interactions in coastal and marine environments
Flow-structure-seabed interaction in coastal and marine environments is a rapidly growing area of research and applications. In this vision paper, this area is discussed with a view of identifying its state of the art and current research challenges. The discussion draws attention to key issues related to structures such as marine pipelines, offshore windfarms, and multiuse offshore platforms. Tsunamis, which received considerable attention after two recent extreme events (2004 Indonesia tsunami and 2011 Japan tsunami) are also included in the discussion. Marine hydrogeomechanics is highlighted, among other areas, as an emerging branch of Marine Civil Engineering. Predictions of the field development for the forthcoming years are also briefly outlined.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Sumer, B. M. (Intern)
Pages: 1-13
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Hydraulic Research
Volume: 52
Issue number: 1
ISSN (Print): 0022-1686
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.809 SNIP 1.659 CiteScore 1.85
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.052 SNIP 1.592 CiteScore 1.41
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.909 SNIP 1.459 CiteScore 1.63
Web of Science (2014): Indexed yes
Forcing of a bottom-mounted circular cylinder by steep regular water waves at finite depth

Forcing by steep regular water waves on a vertical circular cylinder at finite depth was investigated numerically by solving the two-phase incompressible Navier–Stokes equations. Consistently with potential flow theory, boundary layer effects were neglected at the sea bed and at the cylinder surface, but the strong nonlinear motion of the free surface was included. The numerical model was verified and validated by grid convergence and by comparison to relevant experimental measurements. First-order convergence towards an analytical solution was demonstrated and an excellent agreement with the experimental data was found. Time-domain computations of the normalized inline force history on the cylinder were analysed as a function of dimensionless wave height, water depth and wavelength. Here the dependence on depth was weak, while an increase in wavelength or wave height both lead to the formation of secondary load cycles. Special attention was paid to this secondary load cycle and the flow features that cause it. By visual observation and a simplified analytical model it was shown that the secondary load cycle was caused by the strong nonlinear motion of the free surface which drives a return flow at the back of the cylinder following the passage of the wave crest. The numerical computations were further analysed in the frequency domain. For a representative example, the secondary load cycle was found to be associated with frequencies above the fifth- and sixth-harmonic force component. For the third-harmonic force, a good agreement with the perturbation theories of Faltinsen, Newman & Vinje (J. Fluid Mech., vol. 289, 1995, pp. 179–198) and Malenica & Molin (J. Fluid Mech., vol. 302, 1995, pp. 203–229) was found. It was shown that the third-harmonic forces were estimated well by a Morison force formulation in deep water but start to deviate at decreasing depth.

General information
State: Published
Formation and Development of a Breaker Bar under Regular Waves. Part 1: Model Description and Hydrodynamics

In this work a detailed hydrodynamic model is presented, which is used for the study of cross-shore sediment transport and morphodynamics in two dimensions. The model is described in the framework of the generally unstructured, finite volume method. Considerable emphasis is put on those subtleties in the morphological formulation, which are required to achieve mass conservation for the amount of sediment in the bed and in suspension. In this first part of two, the hydrodynamic description over the cross-shore profile is presented. The model is validated against an experiment with detailed measurements of the free surface and turbulence over a fixed breaker bar profile. A test matrix covering a large interval of the surf similarity parameter is simulated, and the phase lag between the breakpoint and the initiation of the setup is described. The relation of this phase lag to a cross-shore delay in dissipation of organised energy into turbulence is described. The relation of this phase lag to the distribution of the location of maxima in bed shear stresses and magnitude of the undertow is also described. Furthermore, processes in the hydrodynamics, which will have a smoothing effect on the mean cross-shore sediment transport and morphodynamic response are considered.

All simulations are presented for regular waves and for values of the deep-water surf similarity parameter, $\zeta_0$, in the range from 0.08 to 1.19, i.e. covering both spilling and plunging breakers. © 2014 Elsevier B.V. All rights reserved.
Formation and development of a breaker bar under regular waves. Part 2: Sediment transport and morphology

In Part 2 of this work, the hydrodynamic model described in Part 1 is applied for the simulation of sediment transport and the associated morphological development of breaker bars. The sediment description is split into bed load and suspended load, and like the hydrodynamics the sediment transport is phase-resolved in order to get on- and offshore directed contributions to the sediment transport from phase lags between the suspended sediment and the hydrodynamics. First, the sediment transport over a morphologically fixed bed of a constant slope is considered, and the transport rates are discussed in terms of three dimensionless parameters: the surf similarity parameters (ζ0) and two variants of Dean's parameter (ΩD and ΩHK). These cover the ranges ζ0∈[0.08, 1.19], ΩD∈[1.1, 27.7] and ΩHK∈[0.01, 0.52]. Secondly, the morphological development of the cross-shore profile is considered, where the initial condition is a profile of constant slope. One reference simulation is discussed in detail and the morphological development due to changes in the hydrodynamic forcing (wave period, wave height, wave irregularity and a net cross-shore current) and in other properties (sediment size and beach slope) is discussed relative to the reference simulation. The coupling between sediment transport rate and morphology is analysed and discussed. © 2014 Elsevier B.V.
Full Scale Measurements of the Hydro-Elastic Response of Large Container Ships for Decision Support

The overall topic of this thesis is decision support for operation of ships and several aspects are covered herein. However, the main focus is on the wave-induced hydro-elastic response of large container ships and its implications on the structural response.

The analyses are based mainly on full scale measurements from four container ships of 4,400 TEU, 8,600 TEU, 9,400 TEU and 14,000 TEU. Primarily, strains measured near the deck amidships are used. Furthermore, measurements of motions and the encountered sea state are available for one of the ships. The smallest ship is in operation on the North Atlantic, while the three largest ships are operated on the Europe - Asia route.

In the design rules of the classification societies for container ships the minimum design sagging bending moment amidships is larger than the hogging bending moment. Due to their design (full midship section and slender bow and stern sections) and their normal cargo loading condition, container ships are typically operated in a still-water hogging condition (tension in deck and compression in the bottom structure). The wave-induced bending moment is added to the still-water bending moment, which, together with the smaller design hogging bending moment, generally makes the wave-induced hogging bending moment more critical than the sagging bending moment in the operation of container ships.

As container ships of today become larger, their natural vibration frequencies become lower and approach the typical encounter frequency with the waves. Together with the relatively high design speed and often pronounced bow flare this makes large container ship more sensitive to slamming and, consequently, the effects of wave-induced hull girder vibrations. From full scale strain measurements of individual, measured hull girder responses in the four container ships, the wave-induced hull girder vibrations are found to increase the vertical bending moment amidships by 100% or more. From the full scale measurements the amplification, due to the hull girder flexibility, is found to be largest for the 8,600 TEU and the 9,400 TEU ships, but, in addition to ship size, speed and bow flare angle are also believed to be important factors contributing to the hull girder vibrations.

The hull girder vibrational response is found to be dominated by the 2-node vertical bending mode. No torsional vibrations are found but torsion may, however, still be a concern for ultra large container ships. The damping of the 2-node vertical bending mode is estimated from full scale measurements for the four ships to 1.3-2.5% of the critical damping. No effect of ship size on the damping is identified.

In some cases the hogging bending moment is more amplified by the effect of the hull girder flexibility than the sagging bending moment. No general trend in the amplification of the response is found from the full scale measurements. In some cases, the rigid-body hogging bending moment, found from full scale measurements and model tests, is considerably larger than the corresponding sagging bending moment. Generally, the difference between the design sagging and hogging bending moments is not reflected in the full scale measurements considered here.

The extreme value of the vertical hogging bending moment, as estimated from full scale measurements, is investigated using the peak-over-threshold method for different periods. The tails of the peak distributions for the four different ships are found to be very different from case to case. The irregularity of the tail behaviour makes it difficult to determine an appropriate extreme value distribution for the hogging bending moment. The Gumbel distribution is believed to be the appropriate extreme value distribution, but it may be necessary to fit other types of extreme value distributions to the largest peaks.

From the full scale measurements it is difficult to assess the influence of operational parameters (ship speed, heading relative to the waves and wave height) on the extreme response because these data are not readily available in all cases. Model tests indicate that bow-quartering sea may induce larger structural loads on the ship than direct head sea and that the amplification of the response due to the hull girder flexibility is largest in bow-quartering waves. However, this fact is not necessarily reflected in the behaviour of ship masters who seemingly tend to prefer bow-quartering sea to head sea.
when encountering adverse weather.

Numerical design tools are widely used in ship design, but may not be able to fully capture the effect of the hull girder flexibility and are here found to significantly underestimate the effect compared to model tests and full scale measurements. Hence, full scale measurements from ships are highly valuable in the evaluation of existing designs and may reveal effects that cannot be assessed numerically.

For decision support, accurate knowledge of the encountered sea state parameters (wave height, period and relative wave direction) is crucial. One means to estimate the on-site sea state from an advancing ship is to use the wave buoy analogy, i.e. use the motions of the ship and the associated motion transfer functions to derive the sea state parameters. The method is promising but needs further refinement before it can be implemented in decision support systems on board ships.

Fatigue damage is estimated from full scale strain measurements from two of the container ships with focus on the assessment of the influence of the wave-induced high-frequency hull girder vibrations. In several cases, the high-frequency contribution to the fatigue damage is dominating the estimated fatigue damage. Spectral formulations for estimating fatigue damage from bi-modal processes are explored and found to yield results fairly similar to the outcome of classical fatigue damage estimation from rainflow counting. However, in a few cases higher fatigue damage rates were estimated from rainflow counting than from narrow-band approximations.

In summary and only considering larger container ships, the new and original contributions of the thesis are believed to be:

- From full scale measurements the hull girder vibrational response is generally found to be dominated by the 2-node vertical bending mode even when the ship is sailing in oblique seas.
- The vertical bending moment in hogging and sagging is amplified considerably by the effect of the hull girder flexibility and the wave-induced hull girder vibrations are found capable of increasing the vertical wave bending moment amidships by 100% or more.
- The vertical hogging bending moment can be as critical as the sagging bending moment in design.
- From comparison of models tests and numerical methods, it seems that the numerical methods are not capable of fully capturing the effect of hull girder flexibility seen in model tests.
- The peak-over-threshold method is found to be the most useful method for extreme value prediction of the vertical bending moment in combination with an appropriate asymptotic extreme value distribution.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Andersen, I. M. V. (Intern), Jensen, J. J. (Intern), Nielsen, U. D. (Intern)
Number of pages: 416
Publication date: 2014

Publication information
Place of publication: Kgs. Lyngby
Publisher: Technical University of Denmark (DTU)
ISBN (Print): 978-87-7475-385-8
Original language: English

Series: DCAMM Special Report
Number: S166
ISSN: 0903-1685
Main Research Area: Technical/natural sciences
Electronic versions: S166_Ingrid_Marie_Vincent_Andersen.pdf

Relations
Projects:
Full Scale Measurements of the Hydro-Elastic Response of Large Container Ships for Decision Support
Publication: Research › Ph.D. thesis – Annual report year: 2014

Fully Coupled Three-Dimensional Dynamic Response of a Tension-Leg Platform Floating Wind Turbine in Waves and Wind
A dynamic model for a tension-leg platform (TLP) floating offshore wind turbine is proposed. The model includes three-dimensional wind and wave loads and the associated structural response. The total system is formulated using 17 degrees of freedom (DOF), 6 for the platform motions and 11 for the wind turbine. Three-dimensional hydrodynamic loads have been formulated using a frequency-and direction-dependent spectrum. While wave loads are computed from the wave kinematics using Morison's equation, the aerodynamic loads are modeled by means of unsteady blade-element-momentum (BEM) theory, including Glauert correction for high values of the axial induction factor, dynamic stall, dynamic
wake, and dynamic yaw. The aerodynamic model takes into account the wind shear and turbulence effects. For a representative geographical location, platform responses are obtained for a set of wind and wave climatic conditions. The platform responses show an influence from the aerodynamic loads, most clearly through quasi-steady mean surge and pitch responses associated with the mean wind. Further, the aerodynamic loads show an influence from the platform motion through a fluctuating rotor load contribution, which is a consequence of the wave-induced rotor dynamics. Loads and coupled responses are predicted for a set of load cases with different wave headings. Further, an advanced aeroelastic code, Flex5, is extended for the TLP wind turbine configuration and the response comparison with the simpler model shows a generally good agreement, except for the yaw motion. This deviation is found to be a result of the missing lateral tower flexibility in the simpler model.

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Kumari Ramachandran, G. K. V. (Intern), Bredmose, H. (Intern), Serensen, J. N. (Intern), Jensen, J. J. (Intern)
Number of pages: 12
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Offshore Mechanics and Arctic Engineering
Volume: 136
Issue number: 2
Article number: 020901
ISSN (Print): 0892-7219
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 0.89 SJR 0.668 SNIP 0.749
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.643 SNIP 1.015 CiteScore 0.97
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.609 SNIP 1.038 CiteScore 0.69
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.462 SNIP 0.856 CiteScore 0.66
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.787 SNIP 1.206 CiteScore 0.58
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 0.625 SNIP 1.497 CiteScore 0.83
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 0.471 SNIP 1.191
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 0.593 SNIP 1.453
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 2
Scopus rating (2008): SJR 0.704 SNIP 1.52
Scopus rating (2007): SJR 0.732 SNIP 1.383
Scopus rating (2006): SJR 0.807 SNIP 1.582
Global hydroelastic model for springing and whipping based on a free-surface CFD code (OpenFOAM)

The theoretical background and a numerical solution procedure for a time domain hydroelastic code are presented in this paper. The code combines a VOF-based free surface flow solver with a flexible body motion solver where the body linear elastic deformation is described by a modal superposition of dry mode shapes expressed in a local floating frame of reference. These mode shapes can be obtained from any finite element code. The floating frame undergoes a pseudo rigid-body motion which allows for a large rigid body translation and rotation and fully preserves the coupling with the local structural deformation. The formulation relies on the ability of the flow solver to provide the total fluid action on the body including e.g. the viscous forces, hydrostatic and hydrodynamic forces, slamming forces and the fluid damping. A numerical simulation of a flexible barge is provided and compared to experiments to show that the VOF-based flow solver has this ability and the code has the potential to predict the global hydroelastic responses accurately.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Bureau Veritas
Authors: Seng, S. (Intern), Jensen, J. J. (Intern), Malenica, S. (Ekstern)
Number of pages: 17
Pages: 1024-1040
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: International Journal of Naval Architecture and Ocean Engineering
Volume: 6
Issue number: 4
ISSN (Print): 2092-6782
Ratings:
Web of Science (2018): Indexed yes
Web of Science (2017): Indexed Yes
Scopus rating (2016): CiteScore 0.98 SJR 0.771 SNIP 1.097
Scopus rating (2015): SJR 0.654 SNIP 0.997 CiteScore 0.84
Scopus rating (2014): SJR 0.446 SNIP 1.047 CiteScore 0.6
Web of Science (2014): Indexed yes
Scopus rating (2013): SJR 0.316 SNIP 0.767 CiteScore 0.41
ISI indexed (2013): ISI indexed yes
Scopus rating (2012): SJR 0.35 SNIP 1.017 CiteScore 0.53
Scopus rating (2011): SJR 0.182 SNIP 0.948 CiteScore 0.36
Scopus rating (2010): SJR 0.426 SNIP 1.097
Original language: English
ENGINEERING, FLUID, MESHES, VOLUME, FLOW, Hydroelasticity, Fluid-structure interaction (FSI), Volume of fluid (VOF), CFD, OpenFOAM, Modal superposition
Electronic versions:
Global_hydroelastic_model.pdf
DOIs:
10.2478/IJNAOE-2013-0229
Impact of an offshore wind farm on wave conditions and shoreline development

The influence of offshore wind farms on the wave conditions and impact on shoreline development is studied in a generic set-up of a coast and a shoreline. The objective was to estimate the impact of a typical sized offshore wind farm on a shoreline in a high wave energetic environment. Especially the shoreline’s sensitivity to the distance from the OWF to the shoreline was studied. The effect of the reduced wind speed inside and on the lee side of the offshore wind farm was incorporated in a parameterized way in a spectral wind wave model. The shoreline impact was studied with a one-line model.
Influence of clay content on residual liquefaction in waves

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Istanbul Technical University
Authors: Kirca, V. O. (Ekstern), Sumer, B. M. (Intern), Fredsøe, J. (Intern)
Number of pages: 1
Publication date: 2014

Host publication information
Title of host publication: Abstract Book of the 34th International Conference on Coastal Engineering
BFI conference series: Proceedings of the International Conference on Coastal Engineering (5010056)
Main Research Area: Technical/natural sciences
Conference: 34th International Conference on Coastal Engineering, Seoul, Korea, Republic of, 15/06/2014 - 15/06/2014
Electronic versions:
INFLUENCE_OF_CLAY_CONTENT.pdf
Source: PublicationPreSubmission
Source-ID: 93654509
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2014

Influence of clay content on wave-induced liquefaction

This paper presents the results of an experimental study of the influence of clay content (CC) on liquefaction of seabed beneath progressive waves. Experiments were, for the most part, conducted with silt and silt-clay mixtures; in supplementary tests, sand-clay mixtures were used. Two types of measurements were carried out: (1) pore-water pressure measurements across the soil depth and (2) water-surface elevation measurements. These measurements were
synchronized with video recordings of the liquefaction process from the side. The ranges of the various quantities in the experiments were wave height $H = 7.62 \pm 2.18$ cm, wave period $T = 51.6$ s, and water depth $h = 55$ cm. The experiments showed that the influence of wave-induced liquefaction is very significant. Susceptibility of silt to liquefaction was increased with increasing $C_{C}$ up to $C_{C} = 30\%$ (which is clay-specific), beyond which the mixture of silt and clay was not liquefied. Sand may become prone to liquefaction with the introduction of clay, contrary to the general perception that this type of sediment is normally liquefaction-resistant under waves. For instance, sand with $d_{50} = 4.0$ mm was liquefied with $C_{C} = 10.8\%$, whereas sand with $d_{50} = 4.17$ mm was partially liquefied with $C_{C}$ as small as 2.9\%. Remarks are made as to how to check for liquefaction of clayey soils exposed to waves in real-life situations.
Stereoscopic particle image velocimetry (SPIV) is nowadays a well-established measurement technique for turbulent flows. However, the accuracy and the spatial resolution are still highly questionable in the presence of complex flow with both strong gradients and out-of-plane motions. To give guidelines for both setup and measurements of such flow configurations, a large region of overlap between two SPIV systems on the same laser light sheet is acquired in a plane normal to the streamwise direction of a high Reynolds turbulent boundary layer flow. A simple separation of the two light sheets is used to improve the accuracy of the measurements by increasing the velocity dynamic range especially. It also presents the enhancement of accuracy due to the light sheet separation for characterizing streamwise vortices (i.e. perpendicular to the sheet). The proposed technique has been demonstrated in the Laboratoire de Mecanique de Lille wind tunnel facility which has been specially designed to study fully developed turbulent boundary layers at high Reynolds numbers. The outlook is to study in detail the physics of the streamwise vortices generated from vortex generators taking advantage of the large scales of this turbulent boundary layer.
This paper considers the flow in porous media that occurs in coastal and offshore engineering problems. Over the past decades numerous formulations of flow equations for porous media have been presented. The present work re-examines the porous media equations of the most recent form and corrects some shortcomings which were identified. The applied type of porosity models relies on empirical resistance coefficients which often need to be measured or calibrated. Only few examples of calibration for numerical models which are present in the literature often applied the same experimental results. In this study new calibration cases were introduced to the calibration procedure in order to achieve a better understanding of the variation of the resistance coefficients. Hereby the coefficients were determined with a better description over the entire parameter space for the resistance coefficients than previously found in the literature. Constant
values for the resistance coefficients for a broad range of flow conditions were recommended based on the new calibrations. The model was validated for the main physical processes that occur in wave–structure interaction in coastal structures including three-dimensional wave–structure interaction, run-up, run-down and pressure damping, regular and irregular wave conditions and evaluation of overtopping. Simple two and three dimensional uniform caisson structures and breakwater layouts were investigated. The model was implemented in the open source CFD library OpenFOAM® and has been made publicly available to the engineering community as part of the wave generation framework waves2Foam.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, B. (Intern), Jacobsen, N. G. (Intern), Christensen, E. D. (Intern)
Pages: 56-72
Publication date: 2014
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Coastal Engineering
Volume: 84
ISSN (Print): 0378-3839
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.925 SNIP 2.097 CiteScore 2.9
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.785 SNIP 2.123 CiteScore 2.55
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.727 SNIP 2.264 CiteScore 2.58
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.912 SNIP 2.226 CiteScore 2.21
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.616 SNIP 2.502 CiteScore 2.43
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 1.898 SNIP 2.332
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 2.067 SNIP 2.454
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 2
Scopus rating (2008): SJR 1.189 SNIP 2.166
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 1.642 SNIP 2.164
Web of Science (2007): Indexed yes
Scopus rating (2006): SJR 1.249 SNIP 2.2
Web of Science (2006): Indexed yes
Large eddy simulations of the influence of piston position on the swirling flow in a model two-stroke diesel engine

Purpose – The purpose of this paper is to study the effect of piston position on the in-cylinder swirling flow in a simplified model of a large two-stroke marine diesel engine. Design/methodology/approach – Large eddy simulations with four different models for the turbulent flow are used: a one-equation model, a dynamic one-equation model, a localized dynamic one-equation model and a mixed-scale model. Simulations are carried out for two different geometries corresponding to 100 and 50 percent open scavenge ports. Findings – It is found that the mean tangential profile inside the cylinder changes qualitatively with port closure from a Lamb-Oseen vortex profile to a solid body rotation, while the axial velocity changes from a wake-like profile to a jet-like profile. The numerical results are compared with particle image velocimetry measurements, and in general, the authors find a good agreement. Research limitations/implications – Considering the complexity of the real engine, the authors designed the engine model using the simplest configuration possible. The setup contains no moving parts, the combustion is neglected and the exhaust valve is discarded. Originality/value – Studying the flow in a simplified engine model, the setup allows studies of fundamental aspects of swirling flow in a uniform scavenged engine. Comparing the four turbulence models, the local dynamic one-equation model is found to give the best agreement with the experimental results.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy, MAN Diesel & Turbo SE
Pages: 325-341
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Volume: 24
Issue number: 2
ISSN (Print): 0961-5539
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.559 SNIP 0.961 CiteScore 1.62
BFI (2015): BFI-level 1
Lift of a Rotating Circular Cylinder in Unsteady Flows

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Wind Energy, Test and Measurements
Authors: Carstensen, S. (Intern), Mandviwalla, X. (Ekstern), Vita, L. (Intern), Schmidt Paulsen, U. (Intern)
Pages: 41-49
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Ocean and Wind Energy
Volume: 1
Issue number: 1
ISSN (Print): 2310-3604
Ratings:
BFI (2018): BFI-level 1
BFI (2017): BFI-level 1
BFI (2016): BFI-level 1
BFI (2015): BFI-level 1
Loads for use in the design of ships and offshore structures

The evaluation of structural responses is key element in the design of ships and offshore structures. Fundamental to this is the determination of the design loads to support the Rule requirements and for application in direct calculations. To date, the current design philosophy for the prediction of motions and wave-induced loads has been driven by empirical or first-principles calculation procedures based on well-proven applications such as ship motion prediction programs. In recent years, the software, engineering and computer technology available to predict the design loads imposed on ships and offshore structures has improved dramatically. Notwithstanding, with the stepwise increase in the size and structural complexity of ships and floating offshore installations and the advances in the framework of Rules and Standards it has become necessary to utilise the latest technologies to assess the design loads on new designs. Along the lines of the recommendations from the International Ship and Offshore Structures Committee (ISSC) I.2 on Loads this paper reviews some of the recent advances in the assessment of loads for ships and offshore structures with the aim to draw the overall technological landscape available for further understanding, validation and implementation by the academic and industrial communities. Particular emphasis is attributed on methodologies applicable for the prediction of environmental and operational loads from waves, wind, current, ice, slamming, sloshing and operational factors. Consideration is also given to deterministic and statistical load predictions based on model experiments, full-scale measurements and theoretical methods.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Lloyd's Register Group Ltd., INSEAN/CNR Maritime Research Centre, China Ship Scientific Research Center, Marine Technology Centre, Delft University of Technology, Osaka University, University of Zagreb, Technical University of Lisbon, Germanischer Lloyd SE, Istanbul Technical University, National University of Singapore, National Technical University of Athens, University of Strathclyde
Pages: 131–174
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Ocean Engineering
Volume: 78
Measurements in a container ship of wave-induced hull girder stresses in excess of design values

This paper describes full-scale measurements of the wave-induced vertical bending moment amidships a 9400 TEU container carrier and focuses on the effect of the hydro-elastic high-frequency vibration on the extreme hogging wave bending moment. One extreme event, where the vertical wave-induced hogging bending moment amidships slightly exceeds the design value, is analysed and the measurements are verified by the use of the relationship between measurements of accelerations and strains and simple beam theory. The measurements are found to be reliable. In the extreme case, the high-frequency vibrations caused by impulsive loads are observed to be of the same magnitude as the rigid-body wave-induced response and thus acts to double the total vertical bending moment amidships. It was also found that even though the ship is sailing in bow quartering seas, only the 2-node vertical vibration mode is apparently excited. Following the extreme event analysis and verification, three hours of strain measurements are used for establishing a Gumbel distribution for the extreme value prediction, and it is found that the probability of exceeding the rule design wave bending moment by 50% in the given sea state is quite significant. Finally, the hydro-elastic behaviour of the hull girder is assessed by simple approximations using the measured statistical properties and closed-form expressions and the agreement with the actual measurements is found to be good. © 2014 Elsevier Ltd.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Andersen, I. M. V. (Intern), Jensen, J. J. (Intern)
Pages: 54-85
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Marine Structures
Volume: 37
ISSN (Print): 0951-8339
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.49 SJR 1.655 SNIP 2.636
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.618 SNIP 2.602 CiteScore 2.77
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.431 SNIP 3.026 CiteScore 2.18
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.314 SNIP 2.871 CiteScore 2.42
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.836 SNIP 3.464 CiteScore 1.76
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.611 SNIP 2.795 CiteScore 1.82
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 1.136 SNIP 2.518
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 1
Molecular dynamics simulations of water on a hydrophilic silica surface at high air pressures

We present a force field for Molecular Dynamics (MD) simulations of water and air in contact with an amorphous silica surface. We calibrate the interactions of each species present in the system using dedicated criteria such as the contact angle of a water droplet on a silica surface, and the solubility of air in water at different pressures. Using the calibrated force field, we conduct MD simulations to study the interface between a hydrophilic silica substrate and water surrounded by air at different pressures. We find that the static water contact angle is independent of the air pressure imposed on the system. Our simulations reveal the presence of a nanometer thick layer of gas at the water-silica interface. We believe that this gas layer could promote nucleation and stabilization of surface nanobubbles at amorphous silica surfaces. © 2014 Elsevier B.V. All rights reserved.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion, NASA Ames Research Center
Authors: Zambrano, H. (Ekstern), Walther, J. H. (Intern), Jaffe, R. (Ekstern)
Pages: 107–113
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Molecular Liquids
Volume: 198
ISSN (Print): 0167-7322
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Moving least squares simulation of free surface flows

In this paper a Moving Least Squares method (MLS) for the simulation of 2D free surface flows is presented. The emphasis is on the governing equations, the boundary conditions, and the numerical implementation. The compressible viscous isothermal Navier–Stokes equations are taken as the starting point. Then a boundary condition for pressure (or density) is developed. This condition is applicable at interfaces between different media such as fluid–solid or fluid–void. The effect of surface tension is included. The equations are discretized by a moving least squares method for the spatial derivatives and a Runge–Kutta method for the time derivatives. The computational frame is Lagrangian, which means that the computational nodes are convected with the flow. The method proposed here is benchmarked using the standard lid driven cavity problem, a rotating free surface problem, and the simulation of drop oscillations. A new exact solution to the unsteady incompressible Navier–Stokes equations is introduced for the rotating free surface problem. © 2013 Elsevier Ltd. All rights reserved.
Multiphase CFD modeling of nearfield fate of sediment plumes

Disposal of dredged material and the overflow discharge during the dredging activities is a matter of concern due to the potential risks imposed by the plumes on surrounding marine environment. This gives rise to accurately prediction of the fate of the sediment plumes released in ambient waters. The two-phase mixture solution based on the drift-flux method is evaluated for 3D simulation of material disposal and overflow discharge from the hoppers. The model takes into account the hindrance and resistance mechanisms in the mixture and is capable of describing the flow details within the plumes and gives excellent results when compared to experimental data.

Numerical analysis of the scavenge flow and convective heat transfer in large two-stroke marine diesel engines

A novel computational fluid dynamics (CFD) model is presented for the study of the scavenging process and convective heat transfer in a large two-stroke low-speed uniflow-scavenged marine diesel engine. The engine is modeled using a fully resolved 12 sector, corresponding to one scavenge port, with cyclic boundaries in the tangential direction. The CFD model is strongly coupled to experiments and effectively provides a high order “interpolation” of the engine processes through the solution of the Reynolds-Averaged Navier–Stokes (RANS) equations subject to boundary conditions obtained through experiments. The imposed experimental data includes time histories of the pressure difference across the engine and the heat release during combustion. The model is validated by a numerical sensitivity analysis and through comparison of model predictions and experimental data, which shows a good agreement. The results show an effective scavenging and a low convective heat loss in agreement with experimental data for large marine diesel engines. © 2014 Elsevier Ltd. All rights reserved.
Numerical and Experimental Investigation of Heat Flow in Permanent Magnet Brushless DC Hub Motor

This paper investigates the heat dissipation in the hub motor of an electric two-wheeler using lumped parameter (LP), finite element (FE) and computational fluid dynamic (CFD) models. The motor uses external rotor permanent magnet brushless DC topology and nearly all of its losses are generated in the stator. The hub motor construction restricts the available conductive paths for heat dissipation from the stator to the ambient only through the shaft. In contrast to an internal rotor structure, where the stator winding losses are diffused via conduction, here convection plays a major role in loss dissipation. Therefore, a LP thermal model with improved convection modelling has been proposed to calculate the temperature of the components inside the hub motor. The developed model is validated with the FE thermal model and the test data. In addition, CFD tools has been used to accurately model the internal and the external flow as well as the convective heat transfer of the hub motor. Finally, an optimization study of the hub motor has been carried out using the CFD model to improve heat transfer from the stator.
Numerical calculation of backfilling of scour holes
A fully-coupled hydrodynamic and morphologic CFD model is presented for simulating backfilling processes around structures. The hydrodynamic model is based on Reynolds-averaged Navier-Stokes equations, coupled with two-equation \( k-\omega \) turbulence closure. The sediment transport model consists of separate bed and suspended load descriptions, the latter based on a turbulent diffusion equation coupled with a reference concentration function near the sea bed boundary. Bed morphology is based on the sediment continuity (Exner) equation. The present simulations have utilized continuous morphologic updating in time, both the hydrodynamic and morphologic solutions being advanced with the same time increment. In this way, the simulations illustrate the ability to simulate fully-coupled hydrodynamic and morphologic developments based on continuous feedback. The model has been implemented for two kinds of structures: piles, and pipelines. Initial scour holes are generated by the same model. The numerical results appear to be in accord with the existing experimental information.

Numerical modeling of backfilling process around monopiles

Numerical modeling of backfilling process around monopiles
Numerical simulation of wave-induced scour and backfilling below submarine pipelines

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Deltares
Authors: Fuhrman, D. R. (Intern), Baykal, C. (Intern), Sumer, B. M. (Intern), Jacobsen, N. G. (Ekstern), Fredsøe, J. (Intern)
Number of pages: 1
Publication date: 2014

Host publication information
Title of host publication: Abstract Book of the 34th International Conference on Coastal Engineering
BFI conference series: Proceedings of the International Conference on Coastal Engineering (5010056)
Main Research Area: Technical/natural sciences
Conference: 34th International Conference on Coastal Engineering, Seoul, Korea, Republic of, 15/06/2014 - 15/06/2014
Electronic versions:
Numerical_simulation.pdf

Relations
Projects:
Numerical simulation of wave-induced scour and backfilling below submarine pipelines
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2014

A fully-coupled hydrodynamic/morphodynamic numerical model is presented and utilized for the simulation of wave-induced scour and backfilling processes beneath submarine pipelines. The model is based on solutions to Reynolds-averaged Navier–Stokes equations, coupled with k−ω turbulence closure, with additional bed and suspended load descriptions forming the basis for sea bed morphology. The morphological evolution is updated continuously, rather than being based e.g. on period- or other time-averaging techniques. Simulations involving wave-induced scour over the range of Keulegan–Carpenter number 5.6≤KC≤30 demonstrate reasonable match with previous experiments, both in terms of the equilibrium scour depth as well as the scour time scale. Wave-induced backfilling processes are additionally studied by subjecting initial conditions taken from scour simulations with larger KC to new wave climates characterized by lower KC values. The simulations considered demonstrate the ability of the model to predict backfilling toward expected equilibrium scour depths based on the new wave climate, in line with experimental expectations. The simulated backfilling process is characterized by two stages: (1) An initial re-distribution phase involving re-organization of sediments in the immediate vicinity of the pipeline, potentially followed by (2) a more lengthy backfilling evolution toward equilibrium scour depth. The simulated backfilling time scales are of the same order of magnitude as in experiments, though the multi-stage process complicates a more systematic characterization. The simulated sequences of scour and backfilling achieved within the present work are estimated to represent temporal durations of up to approximately 12h at full practical scales.
Offshore wind energy developments

This chapter will give a brief overview of a few of the activities within offshore wind energy research, specifically 1) Support structure optimization, 2) Blade coatings for wind turbines; 3) Scour protection of foundations, 4) Offshore HVDC and 5)
Overflow Concentration and Sedimentation in Hoppers

Sediment spillage from hopper overflow constitutes a source for sediment plumes and can also impact the turbidity of aquatic environments. The overflowing mixture is often different from the mixture pumped into the hopper (the inflow), because the mixture undergoes compositional transformation as a result of different timescales in the segregation of the various sediment fractions. The heavier constituents in a mixture will have had time to settle, and overflowing sediments are therefore primarily composed of the finer and lighter constituents, whose concentrations potentially exceed those at the inflow. The hopper constitutes a complex system despite its geometrical regularity; the complexities are largely from the settling processes in concentrated polydisperse mixtures. These settling processes can, however, be captured by employing available settling formulas applicable for multifractional sediment mixtures (i.e., polydispersions). Strictly speaking, these formulas have been validated for homogeneous and unenergetic mixtures only, but the hopper system fulfills these criteria reasonably well. A proper description of the compositional transformation during filling and subsequent overflow stages can be captured using a sediment budget approach, i.e., by using continuity equations for water and sediment phases. In this study, the compositional transformation and the bed height inside the hopper are obtained by solving these equations, considering monodisperse, bidisperse, and polydisperse mixtures, the former analytically. Although assumptions tied to the mathematical model are fulfilled best for hoppers rigged with a multiple-inflow system, the model accurately predicts measured concentrations in the final stage of overflow for single-inflow systems. The model can be used as a preprocessing tool for engineering plume models, providing source specifications for overflow spill and for the subsequent dumping of hopper loads.
Parametric estimation in the wave buoy analogy - an elaborated approach based on energy considerations

An accurate estimation of the ocean wave directional spectrum at the location of an advancing ship is very useful for the ship master to improve operation and safety in a seaway. Research has been conducted to obtain sea state estimates by the Wave Buoy Analogy. The method deals with processing the ship’s wave-induced responses based on different statistical inferences including parametric and non-parametric approaches.
This paper considers a concept to improve the estimate obtained by the parametric method for sea state estimation. The idea is illustrated by an analysis made on full-scale response measurements carried out on a 9400 TEU container ship. The process of fitting a standard JONSWAP spectrum takes into account an analysis of residuals formed by the difference between measured and estimated amount of energy of the response(s). A couple of motion responses and also bending moment are considered as the input of the estimation process. A comparison is made between the results and also with some in-hand outputs from other estimation sources, e.g., wave radar measurements and sea surface elevation by microwave sensors. The discussed and analyzed procedure could also lead to an automatic selection of response combinations for sea state estimation.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Montazeri, N. (Intern), Nielsen, U. D. (Intern)
Number of pages: 7
Publication date: 2014

**Host publication information**

Title of host publication: Proceedings of the ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering (OMAE 2014)
Publisher: American Society of Mechanical Engineers
Article number: OMAE2014-24646
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: 33rd International Conference on Ocean, Offshore and Arctic Engineering, San Francisco, CA, United States, 08/06/2014 - 08/06/2014
DOI's: 10.1115/omae2014-24646
Source: PublicationPreSubmission
Source-ID: 93452829
Publication: Research - peer-review › Article in proceedings – Annual report year: 2014

**Particle motions in oscillatory flow over a smooth bed**

This study investigates particle motions near the bed in an oscillating tunnel with a smooth bed. Trajectories of a heavy particle were recorded in two dimensions (horizontal and vertical) and in time. The wave boundary layer Reynolds number is Re = 520000. Kinematical quantities such as the probability distribution of particle position in the vertical, and the horizontal and vertical particle velocities, among others, are determined. The particle is observed to reach heights of 2.5-3d, similar to that characterizing a typical bedload particle in sediment transport.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, K. L. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern), Hjelmager Jensen, J. (Intern)
Number of pages: 10
Publication date: 2014

**Host publication information**

Title of host publication: Book of Proceedings
Publisher: IAHR
Main Research Area: Technical/natural sciences
Conference: 3rd IAHR Europe Congress, Porto, Portugal, 14/04/2014 - 14/04/2014
Oscillatory flow, Particle tracking, Sediment transport, single particle turbulence
Electronic versions: PARTICLE_MOTIONS.pdf
Source: PublicationPreSubmission
Source-ID: 92681449
Publication: Research - peer-review › Article in proceedings – Annual report year: 2014

**Particle motions in oscillatory flow over a smooth bed**

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, K. L. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern), Hjelmager Jensen, J. (Intern)
Number of pages: 1
Publication date: 2014
Power Spectrum Estimation of Randomly Sampled Signals

The random, but velocity dependent, sampling of the LDA presents non-trivial signal processing challenges due to the high velocity bias and the arbitrariness of particle path through the measuring volume, among other factors. To obtain the desired non-biased statistics, it has previously been shown analytically as well as empirically that residence time weighting is the suitable choice (Buchhave 1979, Buchhave et al. 1979, Velte et al. 2014). Unfortunately, due to technical problems related to the processors providing erroneous measurements of the residence times, this previously widely accepted theory has been questioned and instead a wide spectrum of alternative methods have been invented and tested (c.f. Albrecht et al. 2003). The objective of the current study is to create a simple computer generated signal for baseline testing of residence time weighting and some of the most commonly proposed algorithms; sample and-hold and the direct spectral estimator without residence time weighting. The computer generated signal is a Poisson process with a sample rate proportional to velocity magnitude that consist of well-defined frequency content, which makes bias easy to spot. The idea is that if the algorithms are not able to produce correct statistics from this simple signal, then they will certainly not be able to function well for a more complex measured LDA signal. This is, of course, true also for other methods that are based on the tested algorithms. The extremes are tested by increasing, e.g., the ‘turbulence intensity’ and the ‘shear’. In the paper, we have generated and tested these algorithms on synthetic data generated for the following signals: a Gaussian pulse, a single and multiple sine waves and a Gaussian random signal. In the abstract we have chosen to show results only for the sum of five sine waves. The primary signal and the corresponding power spectrum are shown in Figure 1. The conventional spectrum shows multiple erroneous mixing frequencies and the peak values are too low. The residence time weighted spectrum is correct. The sample-and-hold spectrum has lower power than the correct spectrum, and the f^-2-filtering effect appearing for low data densities is evident (Adrian and Yao 1987). The remaining tests also show that sample-and-hold and the free-running processor perform well only under very particular circumstances with high data rate and low inherent bias, respectively. Residence time weighting provides non-biased estimates regardless of setting. The free-running processor was also tested and compared to residence time weighting using actual LDA measurements in a turbulent round jet. Power spectra from measurements on the jet centerline and the outer part of the jet illustrate a distinct difference between the residence time weighted and the non-weighted spectra, in particular for positions far off the jet center axis where the bias increases.
Power Spectrum Estimation of Randomly Sampled Signals

The random, but velocity dependent, sampling of the LDA presents non-trivial signal processing challenges due to the high velocity bias and the arbitrariness of particle path through the measuring volume, among other factors. To obtain the desired non-biased statistics, it has previously been shown analytically as well as empirically that residence time weighting is the suitable choice. Unfortunately, due to technical problems related to the processors providing erroneous measurements of the residence times, this previously widely accepted theory has been questioned and instead a wide spectrum of alternative methods attempting to produce correct power spectra have been invented and tested. The objective of the current study is to create a simple computer generated signal for baseline testing of residence time weighting and some of the most commonly proposed algorithms (or algorithms which most modern algorithms ultimately are based on), sample-and-hold and the direct spectral estimator without residence time weighting, and compare how they perform in relation to power spectra based on the equidistantly sampled reference signal. The computer generated signal is a Poisson process with a sample rate proportional to velocity magnitude that consist of well-defined frequency content, which makes bias easy to spot. The idea is that if the algorithms are not able to produce correct statistics from this simple signal, then they will certainly not be able to function well for a more complex measured LDA signal. This is, of course, true also for other methods that are based on the tested algorithms. The extremes are tested by increasing, e.g., the ‘turbulence intensity’ and the ‘shear’. It is observed that sample and hold and the free-running processor perform well only under very particular circumstances with high data rate and low inherent bias, respectively, while residence time weighting provides non-biased estimates regardless of setting. The free-running processor was also tested and compared to residence time weighting using actual LDA measurements in a turbulent round jet. Power spectra from measurements on the jet centerline and the outer part of the jet illustrate a distinct difference between the residence time weighted and the non-weighted spectra, in particular for positions far off the jet center axis where the bias increases.
Reducing roll motion by passive free surface tanks

Roll stabilisation of motorised vessels plays an important part in reducing passenger discomfort and increasing safety and cargo capacity. Passive free surface tanks are considered a low-cost stabilising method, which is efficient at all speeds without increasing hull resistance.

In this study, a mathematical model for evaluating the performance of a passive free surface tank is established. This is done by coupling a roll model to a fluid flow model. As a numerical example, the seakeeping abilities of a container vessel are evaluated. The necessary methods for performing the simulation are presented and the design of a free surface tank is explained. The effects of the passive free surface tank are evaluated and a significant damping effect is observed, particularly in cases with resonant roll.

Risk assessment for ship collisions against offshore structures

Offshore installations served by in-field vessels and/or situated in the vicinity of ship traffic lanes are exposed to collision hazards such as risk of loss of life, economic loss, and environmental damage. Therefore, one of the many performance goals in the design phase of such structures is to ensure that the risk for major accidents and service disruptions is low enough to be acceptable to users, the public and those responsible for public safety. The collision risk hazards must also be re-evaluated at proper time intervals during the operational phase in order to update the risk estimate due to changes in ship traffic in the vicinity of the offshore structures and new developments within navigational equipment and procedures. The paper highlights some of the available analytical elements in collision frequency estimation and response calculations for different types of offshore installations and indicates how these tools can be applied to evaluate relevant risk control options.
Role of the vertical pressure gradient in wave boundary layers

By direct numerical simulation (DNS) of the flow in an oscillatory boundary layer, it is possible to obtain the pressure field. From the latter, the vertical pressure gradient is determined. Turbulent spots are detected by a criterion involving the vertical pressure gradient. The vertical pressure gradient is also treated as any other turbulence quantity like velocity fluctuations and statistical properties of the vertical pressure gradient are calculated from the DNS data. The presence of a vertical pressure gradient in the near bed region has significant implications for sediment transport.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Genoa
Authors: Jensen, K. L. (Intern), Sumer, B. M. (Intern), Vittori, G. (Ekstern), Blondeaux, P. (Ekstern)
Number of pages: 13
Publication date: 2014

Scour around Offshore Wind Turbine Foundations

Over the last decades several offshore wind farms have been installed and commissioned in the European waters. Typically the foundations of the wind turbines are protected against scour at the base by installing scour protection with rock dump. The Egmond aan Zee offshore wind farm located about 10 – 18 km off the coast of the Netherlands constructed in 2006, utilized such a scour protection system to prevent removal of the sediment base. Following the installation of the scour protection system and subsequently on a yearly basis an extensive survey campaign has been carried out, to evaluate the performance (stability) of the scour protection and to quantify the edge scour development at the circumference of the scour protection. The survey campaign showed considerable edge scour of up to 2.7 m, which was expected from design considerations. However, no clear information exists on the mechanisms causing the edge scour development around scour protections at offshore wind turbine foundations. The purpose of the present thesis is to investigate and explain the development of the edge scour in such applications, and describe the flow mechanism causing the scour. Furthermore, the dissertation also focuses on the case where the foundation is placed without scour protection. Here the continuously changing flow climate, comprising currents, waves and combined waves and current forces the scour depth around the foundation to change over time experiencing scour and backfill in an alternating fashion. The scour mechanism and the time development of a scour process in waves, current and steady current has received a vast amount of research over the last decades revealing various features of scour processes. While the scour process seems well documented, relatively few studies have been reported on backfilling and to the authors knowledge no study is yet available, investigating in a systematic manner the backfilling process around monopiles. The thesis therefore also aims at explaining the mechanism and time development of backfill. In Chapter 2, flow and edge scour adjacent to stone covers in steady current propagating in-line with a stone cover layer is studied. The results indicate that the exposure to steady current scouring the bed at the side edge of the cover structure, the edge scour. The model tests show that the edge scour is caused by the combined action of the following two effects: (1) Primary flow; and (2) Secondary flow. The primary flow stirs up the sediment and entrains the sediment into the main body of the flow by, and the secondary flow carries the sediment away from the junction between the sediment bed and the cover stones, resulting in the edge scour. With this, stones at the edge of the cover structure are “undermined”, and as a result, slump down into the scour hole. The maximum scour is about 0.8 times the equivalent stone diameter. Abstract XIII In Chapter 3, flow and edge scour around scour protections at offshore wind turbine foundations are studied by means of particle image velocimetry, hot-film measurements and actual scour tests. The experimental results are supported by a field investigation of edge scour at the offshore wind park Egmond aan Zee and Scroby Sands offshore wind farm. It is found that the edge scour is caused by the local increase in the hydrodynamic field leading to increased sediment transport and scour. The governing flow processes are the horseshoe vortex generated in front of the scour protection bern, the contraction and acceleration flow
at the side edge of the scour protection, and a pair of counter-rotating vortices emerging in the near bed wake region of the pile and scour protection. The latter mentioned flow features show a significant potential to scour the adjacent sea bed, and in turn cause loss of stability of the scour protection. Design guidelines are provided to determine the equilibrium stage scour in current, tidal current and combined waves and current. In Chapter 4, a description of the backfilling process around both slender and large piles exposed to waves and combined waves and current, is given, based on results of physical model tests. Using the results it is possible to numerically model and estimate the changing scour depth over time. It is shown that the scour depth corresponding to the equilibrium stage of the backfilling process is the same as that corresponding to the equilibrium stage of the scour process for the same wave or combined waves and current climate. The time scale of the backfilling process has been determined as a function of three parameters namely: (1) the Keulegan-Carpenter number of the initial wave or the current (which generate the initial scour hole); (2) that of the subsequent wave which backfills the scour hole; and (3) the Shields parameter associated with the latter wave for live-bed conditions. In the case of combined waves and current, the current-to-wave-velocity ratio is also involved. The time scale of the backfilling process is completely different from that of scour. The time scale of backfilling is much larger than that of scour when the Keulegan-Carpenter number associated with the backfilling is 퐾퐶퐹 < 픰(10) (typical wind farm application), while the trend is opposite when 퐾퐶퐹 ≥ 픰(10). In Chapter 5, the time scale of scour around a slender vertical pile in combined waves and current is studied. The time scale of scour in combined waves and current is found to depend on three parameters: (1) the Keulegan-Carpenter number; (2) the Shields parameters of the wave component of the flow; and (3) the current-to-wave-velocity ratio. The study indicates that the time scale of scour increase significantly when superimposing even a slight current on a wave. The results further show that the time scale of scour approaches the wave-alone time scale in the wave-dominated regime, and it approaches the current values in the current-dominated regime, as anticipated.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Petersen, T. U. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern), Fuhrman, D. R. (Intern), Christensen, E. D. (Intern)
Number of pages: 195
Publication date: 2014

Publication information
Publisher: Technical University of Denmark. Department of Mechanical Engineering
Original language: English
Main Research Area: Technical/natural sciences
Edge scour, Cover stones, Scour protection, Backfill, Time scale, Steady current, Waves, Offshore wind farms, Monopiles , Tidal current
Electronic versions:
ThorUgelvigPetersenPhDThesis_rev1.pdf
Publication: Research › Ph.D. thesis – Annual report year: 2014

Second-order coupling of numerical and physical wave tanks for 2D irregular waves. Part I: Formulation, implementation and numerical properties
In this series of two papers, we report on the irregular wave extension of the second-order coupling theory of numerical and physical wave model described in [Z. Yang, S. Liu, H.B. Bingham and J. Li. Second-order theory for coupling numerical and physical wave tanks: Derivation, evaluation and experimental validation. Coast. Eng. 71,37-51]. We also correct several errors which unfortunately appeared in that manuscript. In the present part I, the full second-order coupling theory for irregular wave is described in detail. The new second-order coupling signal is presented including both superharmonics and subharmonics and covering wavemaker configurations of the piston- and flap-types. The second-order dispersive correction allows for an improved nonlinear transfer of wave information between the two models. For practical implementation, the coupling equations are solved by a combined five-point Lagrange interpolation and the fourth-order Runge-Kutta scheme, with a numerical velocity time series which is decomposed by the Newton-Raphson iterative method. Analytical evaluations on the suppression of spurious free waves and the relative errors of the resultant bound waves have been conducted by considering a 2nd-order, bi-chromatic wave over a range of dimensionless water depth and oscillation frequency combinations, indicating that the resultant wave quality is significantly improved using the second-order coupling theory. A separate verification combining numerical and experimental model of the theory will be presented in Part II by the same authors. (C) 2014 Elsevier B.V. All rights reserved.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Dalian University of Technology
Authors: Yang, Z. (Ekstern), Liu, S. (Ekstern), Bingham, H. B. (Intern), Li, J. (Ekstern)
Pages: 48-60
Publication date: 2014
Main Research Area: Technical/natural sciences
Second-order coupling of numerical and physical wave tanks for 2D irregular waves. Part II: Experimental validation in two-dimensions

This paper provides an experimental validation of the second-order coupling theory outlined by Yang et al. (Z. Yang, S. Liu, H.B. Bingham and J. Li., 2013. Second-order coupling of numerical and physical wave tanks for 2D irregular waves. Part I: Formulation, implementation and numerical properties, submitted for publication) using 2D irregular waves. This work provides a second-order dispersive correction for the physical wavemaker signal which improves the nonlinear transfer of information between the numerical and physical models compared to the first-order method of Zhang et al. (2007). The important nonlinear parameters and numerical performance were theoretically investigated in Part I. In the present Part II, careful experimental validation is carried out using a sequence of progressively more complex analytical and numerical target waves. The results demonstrate clearly that improved performance is achieved by using the second-order correction. When controlling with a second-order coupling signal, two key points are notable: (i) The higher harmonics underlying the numerical waves are accurately captured and transferred into the physical model. (ii) The second-order behavior leads to an unwanted spurious freely propagating second harmonic that is substantially reduced when compared to an identical wave paddle operating with a first-order coupling signal. Using nonlinear regular (monochromatic), bichromatic and irregular wave cases as well as varying coupled wave tank bathymetries, both these aspects are verified over a broad range of wave frequencies and shown to be extensively applicable to physical wave tanks. (C) 2014 Elsevier B.V. All rights reserved.
SELF-SIMILARITY AND HELICAL SYMMETRY IN VORTEX GENERATOR FLOW SIMULATIONS

According to experimental observations, the vortices generated by vortex generators have previously been observed to be self-similar for both the axial ($u_z$) and azimuthal ($u_Ө$) velocity profiles. Further, the measured vortices have been observed to obey the criteria for helical symmetry. This is a powerful result, since it reduces the highly complex flow to merely four parameters. In the present work, corresponding computer simulations using Reynolds-Averaged Navier-Stokes equations have been carried out and compared to the experimental observations. The main objective of this study is to investigate how well the simulations can reproduce the physics of the flow and if the same analytical model can be applied. Using this model, parametric studies can be significantly reduced and, further, reliable simulations can substantially reduce the costs of the parametric studies themselves.
Simulation of extreme events of oblique wave interaction with porous breakwater structures

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Deltares
Authors: Jensen, B. (Intern), Christensen, E. D. (Intern), Jacobsen, N. G. (Ekstern)
Number of pages: 1
Publication date: 2014

Host publication information
Title of host publication: Abstract Book of the 34th International Conference on Coastal Engineering
BFI conference series: Proceedings of the International Conference on Coastal Engineering (5010056)
Main Research Area: Technical/natural sciences
Conference: 34th International Conference on Coastal Engineering, Seoul, Korea, Republic of, 15/06/2014 - 15/06/2014
Electronic versions:
SIMULATION_OF_EXTREME_EVENTS.pdf
Source: PublicationPreSubmission
Source-ID: 92775972
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2014

Simulation of extreme events of oblique wave interaction with porous breakwater structures

Simulations of a single turbulent vortex ring using a regularized particle-mesh based vortex method

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Eidgenössische Technische Hochschule
Authors: Hejlesen, M. M. (Intern), Walther, J. H. (Intern)
Publication date: 2014
Event: Poster session presented at 11th World Congress on Computational Mechanics, 5th European Conference on Computational Mechanics, 6th European Conference on Computational Fluid Dynamics, Barcelona, Spain.
Main Research Area: Technical/natural sciences
Vortex rings, Turbulence, Particle-mesh methods, Regularized vortex methods
Electronic versions:
Abstract
Simulations of a single vortex ring using an unbounded, regularized particle-mesh based vortex method

In recent work we have developed a new FFT based Poisson solver, which uses regularized Greens functions to obtain arbitrary high order convergence to the unbounded Poisson equation. The high order Poisson solver has been implemented in an unbounded particle-mesh based vortex method which uses a re-meshing of the vortex particles to ensure the convergence of the method. Furthermore, we use a re-projection of the vorticity field to include the constraint of a divergence-free stream function which is essential for the underlying Helmholtz decomposition and ensures a divergence free vorticity field. The high order, unbounded particle-mesh based vortex method is used to simulate the instability, transition to turbulence and eventual destruction of a single vortex ring. From the simulation data a novel method on analyzing the dynamics of the enstrophy is presented based on the alignment of the vorticity vector with the principal axis of the strain rate tensor. We find that the dynamics of the enstrophy density is dominated by the local flow deformation and axis of rotation, which is used to infer some concrete tendencies related to the topology of the vorticity field.

Simulations of vortical flow using an unbounded, regularized particle-mesh based vortex method

Stable finite difference discretizations of the forward speed seakeeping problem
Stochastic procedures for extreme wave induced responses in flexible ships

Different procedures for estimation of the extreme global wave hydroelastic responses in ships are discussed. Firstly, stochastic procedures for application in detailed numerical studies (CFD) are outlined. The use of the First Order Reliability Method (FORM) to generate critical wave episodes of short duration, less than 1 minute, with prescribed probability content is discussed for use in extreme response predictions including hydroelastic behaviour and slamming load events. The possibility of combining FORM results with Monte Carlo simulations is discussed for faster but still very accurate estimation of extreme responses. Secondly, stochastic procedures using measured time series of responses as input are considered. The Peak-over-Threshold procedure and the Weibull fitting are applied and discussed for the extreme value predictions including possible corrections for clustering effects.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, J. J. (Intern), Andersen, I. M. V. (Intern), Seng, S. (Intern)
Number of pages: 12
Pages: 1148-1159
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: International Journal of Naval Architecture and Ocean Engineering
Volume: 6
Issue number: 4
ISSN (Print): 2092-6782
Ratings:
Web of Science (2018): Indexed yes
Web of Science (2017): Indexed Yes
Scopus rating (2016): CiteScore 0.98 SJR 0.771 SNIP 1.097
Scopus rating (2015): SJR 0.654 SNIP 0.997 CiteScore 0.84
Scopus rating (2014): SJR 0.446 SNIP 1.047 CiteScore 0.6
Web of Science (2014): Indexed yes
Scopus rating (2013): SJR 0.316 SNIP 0.767 CiteScore 0.41
ISI indexed (2013): ISI indexed yes
Scopus rating (2012): SJR 0.35 SNIP 1.017 CiteScore 0.53
Scopus rating (2011): SJR 0.182 SNIP 0.948 CiteScore 0.36
Scopus rating (2010): SJR 0.426 SNIP 1.097
Original language: English
ENGINEERING, PREDICTIONS, VALUES, TIME, FORM, Extreme value predictions, Stochastic processes, First order reliability method (FORM), Peak-over-threshold, Monte carlo simulations
Electronic versions:
Stochastic_procedures_for_extreme_wave.pdf
DOIs:
10.2478/IJNAOE-2013-0236
Source: FindIt
Source-ID: 273889131
Publication: Research - peer-review › Journal article – Annual report year: 2015
Strain Engineering of Kapitza Resistance in Few-Layer Graphene

We demonstrate through molecular dynamics simulations that the Kapitza resistance in few-layer graphene (FLG) can be controlled by applying mechanical strain. For unstrained FLG, the Kapitza resistance decreases with the increase of thickness and reaches an asymptotic value of $6 \times 10^{-10} \text{m}^2\text{K}/\text{W}$ at a thickness about 16 nm. Uniaxial cross-plane strain is found to increase the Kapitza resistance in FLG monotonically, when the applied strain varies from compressive to tensile. Moreover, uniaxial strain couples the in-plane and out-of-plane strain/stress when the surface of FLG is buckled. We find that with a compressive cross-plane stress of 2 GPa, the Kapitza resistance is reduced by about 50%. On the other hand it is almost tripled with a tensile cross-plane stress of 1 GPa. Remarkably, compressive in-plane strain can either increase or reduce the Kapitza resistance, depending on the specific way it is applied. Our study suggests that graphene can be exploited for both heat dissipation and insulation through strain engineering.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Swiss Federal Institute of Technology
Authors: Chen, J. (Ekstern), Walther, J. H. (Intern), Koumoutsakos, P. (Ekstern)
Pages: 819–825
Publication date: 2014
Main Research Area: Technical/natural sciences

Publication information
Journal: Nano Letters
Volume: 14
Issue number: 2
ISSN (Print): 1530-6984
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 13.4
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): CiteScore 14.76
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): CiteScore 14.04
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): CiteScore 14.23
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): CiteScore 13.78
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): CiteScore 13.83
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 2
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 2
Web of Science (2008): Indexed yes
Web of Science (2007): Indexed yes
The Effect of a Finite Measurement Volume on Power Spectra from a Burst Type LDA

We analyze the effects of a finite size measurement volume on the power spectrum computed from data acquired with a burst-type laser Doppler anemometer. The finite measurement volume causes temporal distortions in acquisition of the data resulting in phenomena such as finite processing time and dead time. We compare analytical expressions for the bias and distortion of the velocity power spectrum computed from computer-generated data. We then compare the spectrum from the computer-generated data and a power spectrum from a measurement on a free turbulent jet in air and conclude that we have a valid understanding of the effects of the finite measurement volume on the measured velocity power spectrum.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Princeton University, Intarsia Optics
Authors: Buchhave, P. (Exter), Velte, C. M. (Intern), K. George, W. (Exter)
Number of pages: 9
Publication date: 2014

Host publication information
Title of host publication: Proceedings of the 17th International Symposium on Applications of Laser Techniques to Fluid Mechanics
Main Research Area: Technical/natural sciences
The effect of dead time on randomly sampled power spectral estimates

We investigate power spectra of a randomly sampled stationary stochastic signal, e.g., a spatial component of a turbulent velocity. We extend the methods of previous authors that basically assumed point or delta function sampling by including features characteristic of real measurement systems. We consider both the effect on the measured spectrum of a finite sampling time, i.e., a finite time during which the signal is acquired, and a finite dead time, that is a time in which the signal processor is busy evaluating a data point and therefore unable to measure a subsequent data point arriving within the dead time delay.
The New Maritime Engineering Education at the Technical University of Denmark

Until 2010, the maritime engineering education at the Technical University of Denmark (DTU) followed the rather classical naval architecture approach with the main focus on marine hydrodynamics and strength of ship structures. The number of students was rather modest and constant. However, at that time the last major ship yard in Denmark was closing down and ship operation, together with ship design, became the main working area for the students after graduation. It was then decided to broaden the naval architecture education to a maritime engineering education taking marine logistics, management, transport optimization and engine system design into to the curriculum. Furthermore, the concept of green shipping was introduced wherever relevant in teaching modules at DTU and two new maritime engineering courses were introduced: (1) Sailing practice in a merchant vessel or DTU’s research vessel (guided by the experience gained from Australian Maritime College in a similar course) and, (2) Green transportation dealing holistically with global ship transport. Furthermore, study trips to Asia visiting ship yards have been made possible by support from the various private funds. This new maritime engineering education has so far been very successful with the number of students increased by a factor of two and with very good job opportunities in the Danish maritime industry. A spin-off of this change is DTU’s participation in a dual MSc degree engineering program: Nordic Master in Maritime Engineering where DTU is responsible for the study track Ship Operations. This change has also led to the creation of the centre Maritime DTU as a one-point entry for the industry and maritime authorities regarding R&D related maritime issues. The paper will discuss the process and the way ahead for further strengthening the interaction between maritime industries, ship owners, maritime authorities and universities dealing with maritime engineering in Denmark and internationally.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, J. J. (Intern), Andersen, I. M. V. (Intern), Nielsen, U. D. (Intern), Andersen, P. (Intern)
Pages: 353-359
Publication date: 2014

Host publication information
Title of host publication: Proceedings of the 15th IAMU Annual General Assembly : Looking Ahead - Innovation in Maritime Education, Training and Research
Publisher: International Association of Maritime Universities
Editors: Ranmuthugala, D., Lewarn, B.
ISBN (Print): 978-0-9806391-4-4
Main Research Area: Technical/natural sciences
Electronic versions:
The_New_Maritime_Engineering_Education.pdf
Source: PublicationPreSubmission
Source-ID: 101856519
Theoretical and experimental investigation of cavitation in main bearings for large two-stroke marine diesel engines

General information
State: Published
Organisations: Department of Mechanical Engineering, Solid Mechanics, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Christiansen, C. K. (Intern), Klit, P. (Intern), Walther, J. H. (Intern), Vølund, A. (Ekstern)
Number of pages: 1
Publication date: 2014
Event: Abstract from 10th European Fluid Mechanics Conference (EUROMECH), KGs. Lyngby, Denmark.
Main Research Area: Technical/natural sciences

Towards Predicting the Added Resistance of Slow Ships in Waves
The objective of this project was to develop a calculation tool for the added resistance of ships in ocean waves. To this end a linear potential flow time-domain numerical seakeeping solver has been developed. The solver is based on higher order finite-difference schemes on overlapping grids and has been implemented using the Overture framework for solving partial differential equations on overset, boundary-fitted grids. This library includes support for parallel processing and a variety of direct and iterative system solvers. The non-linear water wave problem is linearised about two base flows namely: the uniform stream, and the double body flow. The resulting linearised initial boundary value problem has been solved in the time domain. In order to march the free surface in time, the fourth-order Runge-Kutta integration scheme has been used to integrate the kinematic and dynamic free-surface boundary conditions. The field continuity equation has been discretised by a centered fourth-order finite difference scheme which also includes ghost layers at the boundaries. For the zero-speed hydrodynamic problem, the same centered scheme can be utilised to calculate the freesurface derivatives. In the case of the forward-speed problem however, the convective terms in the free-surface conditions have been calculated using an upwind biased scheme, where the stencil is weighted in the upwind direction. As an alternative to using the biased scheme, a flexible filtering scheme has been implemented which can be applied to the solution after each time step. The filtering scheme can be used with the centered finite difference scheme. Both of these strategies introduce numerical diffusion into the model to ensure the stability in the case of the forward-speed hydrodynamic problems. The developed computational strategy has been applied to solve three hydrodynamic problems: the wave resistance problem, the radiation problem, and the diffraction problem. The main objective was to find the first-order velocity potentials, free-surface elevation and the body motions that are required to calculate the wave drift force or the added resistance. Instead of solving the time-domain water wave problem by the impulse response function approach, a pseudo-impulsive Gaussian motion is used in this project. In the case of the diffraction problem the pseudo-impulse describes the elevation of the incident waves. In the radiation problem this is the displacement which will be applied to the body in the time domain. The time-domain solutions of the hydrodynamic problems are then Fourier transformed to get the frequency-domain solutions. In the case of the radiation problem these are the added mass and damping coefficients. For the diffraction problem we obtain the wave exciting forces in the frequency domain. By solving the equation of motion the response amplitude operators for six degrees of freedom are also calculated. For each hydrodynamic problem, the free-surface elevation along the waterline, the velocity potential and its gradients on the body surface, are obtained in the frequency domain via Fourier transform of the transient solutions. All this frequency-domain data is then used to calculate...
the added resistance in the frequency domain. This has been implemented using the near-field formulation. The solver has been validated against analytical solutions for simple exact geometries like a cylinder and a sphere. The solver is now ready to be exercised on real ship geometries.

**General information**
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Amini Afshar, M. (Intern), Bingham, H. B. (Intern), Andersen, P. (Intern)
Number of pages: 156
Publication date: 2014

**Publication Information**
Publisher: DTU Mechanical Engineering
ISBN (Electronic): 978-87-7475-393-3
Original language: English

Series: DCAMM Special Report
Number: S171
ISSN: 0903-1685
Main Research Area: Technical/natural sciences
Electronic versions: S171_Mostafa_Amini_Afshar.pdf
Publication: Research › Ph.D. thesis – Annual report year: 2015


**General information**
State: Published
Organisations: Department of Applied Mathematics and Computer Science, Scientific Computing, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, FORCE Technology
Authors: Lindberg, O. (Ekstern), Bingham, H. B. (Intern), Engsig-Karup, A. P. (Intern)
Number of pages: 4
Publication date: 2014

**Host publication Information**
Title of host publication: Proceedings of the International Workshop on Water Waves and Floating Bodies 2014
Publisher: IWWFB
Main Research Area: Technical/natural sciences
Conference: 29th International Workshop on Water Waves and Floating Bodies, Osaka, Japan, 30/03/2014 - 30/03/2014
Electronic versions: iwwfb29_34.pdf
Links:
http://www.iwwfb.org/Workshops/29.htm
Source: PublicationPreSubmission
Source-ID: 118278406
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2015

Turbulent swirling flow in a dynamic model of a uniflow-scavenged two-stroke engine

It is desirable to use computational fluid dynamics for optimization of the in-cylinder processes in low-speed two-stroke uniflow-scavenged marine diesel engines. However, the complex nature of the turbulent swirling in-cylinder flow necessitates experimental data for validation of the used turbulence models. In the present work, the flow in a dynamic scale model of a uniflow-scavenged cylinder is investigated experimentally. The model has a transparent cylinder and a moving piston driven by a linear motor. The flow is investigated using phase-locked stereoscopic particle image velocimetry (PIV) and time-resolved laser Doppler anemometry (LDA). Radial profiles of the phase-locked mean and rms velocities are computed from the velocity fields recorded with PIV, and the accuracy of the obtained profiles is demonstrated by comparison with reference LDA measurements. Measurements are carried out at five axial positions for 15 different times during the engine cycle and show the temporal and spatial development of the swirling in-cylinder flow. The tangential velocity profiles in the bottom of the cylinder near the end of the scavenge process are characterized by a concentrated swirl resulting in wake-like axial velocity profiles and the occurrence of a vortex breakdown. After scavenge port closing, the axial velocity profiles indicate that large transient swirl-induced structures exist in the cylinder. Comparison with profiles obtained under steady-flow conditions shows that the scavenge flow cannot be assumed to be quasi-steady. The temporal development of the swirl strength is investigated by computing the angular momentum. The swirl strength shows an exponential decay from scavenge port closing to scavenge port opening corresponding to a reduction of 34%, which is in good agreement with theoretical predictions.
Two-Dimensional Depth-Averaged Beach Evolution Modeling: Case Study of the Kizilirmak River Mouth, Turkey

This study presents an application of a two-dimensional beach evolution model to a shoreline change problem at the Kizilirmak River mouth, which has been facing severe coastal erosion problems for more than 20 years. The shoreline changes at the Kizilirmak River mouth have been thus far investigated by satellite images, physical model tests, and one-dimensional numerical models. The current study uses a two-dimensional depth-averaged numerical beach evolution model, developed based on existing methodologies. This model is mainly composed of four main submodels: a phase-averaged spectral wave transformation model, a two-dimensional depth-averaged numerical waveinduced circulation model, a sediment transport model, and a bottom evolution model. To validate and verify the numerical model, it is applied to several cases of laboratory experiments. Later, the model is applied to a shoreline change problem at the groin field to the east of the Kizilirmak River mouth at the Bafra alluvial plain. The results of the numerical model agree with the existing laboratory and field measurements. © 2014 American Society of Civil Engineers.
Vertical pressure gradient and particle motions in wave boundary layers

The present study covers both a numerical and experimental investigation of the processes in the oscillatory boundary layer. In the first part a direct numerical simulation (DNS) is conducted to study the vertical pressure gradient, and its role in relation to laminar to turbulent transition and its role in the fully turbulent boundary layer. The pressure in the flow is obtained from the flow fields of the oscillatory boundary layer. What differs, the vertical pressure gradient, from other turbulent quantities, like e.g. velocity fluctuations is that it can detect newly generated turbulence. This is in contrast to velocity fluctuations that are diffusive, so they can also contain residual turbulence from the previous half cycle until they are dissipated. Furthermore, the magnitude of the mean value of conditionally averaged vertical pressure gradient (for $-\partial p/\partial x > 0$) is compared to the submerged weight of sediment. This revels that the upward directed vertical pressure gradient on average has a magnitude that yields in a contribution to the force needed to overcome the submerged weight of the water-sediment mixture.

Secondly particle motion in the oscillatory boundary layer is investigated. The experiment is conducted in a oscillating water tunnel, for both smooth bed and rough bed. The particle motion is determined by utilizing particle tracking base on a video recording of the particle motion in the flow. In the oscillatory flow, in contrast to steady current, the particle motion is a function of phase. Therefore the particle will settle towards the end of each half period, and after flow reversal, when the turbulent intensity becomes large enough it can be suspended. If the particle is light enough it can be maintained in suspension, otherwise it will settle before it is resuspended. The governing parameter for the particle motions after it is brought into suspension, the Rouse parameter ($\beta = \omega_s/(\kappa U_f)$, $\omega_s$ the settling velocity, $\kappa = 0.4$ Karmans constant). For large values of the Rouse parameter, the particle tend to stay near the bed while for smaller values the particle spends more time away from the bed.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, K. L. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern), Hjelmager Jensen, J. (Intern)
Number of pages: 242
Publication date: 2014
Vortex Generator Induced Flow in a High Re Boundary Layer

Steroscopic Particle Image Velocimetry measurements have been conducted in cross-planes behind three different geometries of Vortex Generators (VGs) in a high Reynolds number boundary layer. The VGs have been mounted in a cascade producing counter-rotating vortices and the downstream flow development was examined. Three VG geometries were investigated: rectangular, triangular and cambered. The various VG geometries tested are seen to produce different impacts on the boundary layer flow. Helical symmetry of the generated vortices is confirmed for all investigated VG geometries in this high Reynolds number boundary layer. From the parameters resulting from this analysis, it is observed at the most upstream measurement position that the rectangular and triangular VGs produce vortices of similar size, strength and velocity induction whilst the cambered VGs produce smaller and weaker vortices. Studying the downstream development in the ensemble and spanwise averaged measurements, it is observed that the impact from the rectangular and triangular VGs differs. For the rectangular VGs, self-similarity in the streamwise component was confirmed.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Université Lille Nord de France
Authors: Velte, C. M. (Intern), Braud, C. (Ekstern), Coudert, S. (Ekstern), Foucaut, J. (Ekstern)
Number of pages: 11
Publication date: 2014
Conference: The science of Making Torque from Wind 2012, Oldenburg, Germany, 09/10/2012 - 09/10/2012
BFI conference series: European Academy of Wind Energy : The Science of Making Torque from Wind (5010078)
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 555
Article number: 012102
ISSN (Print): 1742-6596
Ratings:
BFI (2018): BFI-level 1
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 0.45 SJR 0.24 SNIP 0.383
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.24 SNIP 0.373 CiteScore 0.35
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.253 SNIP 0.344 CiteScore 0.32
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.231 SNIP 0.272 CiteScore 0.25
ISI indexed (2013): ISI indexed no
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.28 SNIP 0.354 CiteScore 0.33
ISI indexed (2012): ISI indexed no
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 0.292 SNIP 0.352 CiteScore 0.43
Advances in Seabed Liquefaction and its Implications for Marine Structures

A review is presented of recent advances in seabed liquefaction and its implications for marine structures. The review is organized in seven sections: Residual liquefaction, including the sequence of liquefaction, mathematical modelling, centrifuge modelling and comparison with standard wave-flume results; Momentary liquefaction; Floatation of buried pipelines; Sinking of pipelines and marine objects; Liquefaction at gravity structures; Stability of rock berms in liquefied soils; and Impact of seismic-induced liquefaction.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Sumer, B. M. (Intern)
Number of pages: 14
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Geotechnical Engineering
Volume: 45
Issue number: 4
ISSN (Print): 0046-5828
Ratings:
Scopus rating (2016): SJR 0.259 SNIP 0.444 CiteScore 0.35
Scopus rating (2015): SJR 0.221 SNIP 0.371 CiteScore 0.37
Scopus rating (2014): SJR 0.369 SNIP 0.927 CiteScore 0.49
Scopus rating (2013): SJR 0.269 SNIP 0.321 CiteScore 0.25
ISI indexed (2013): ISI indexed no
Scopus rating (2012): SJR 0.325 SNIP 0.452 CiteScore 0.25
ISI indexed (2012): ISI indexed no
Scopus rating (2011): SJR 0.119 SNIP 0.204 CiteScore 0.04
ISI indexed (2011): ISI indexed no
Scopus rating (2010): SJR 0.108 SNIP 0.175
Scopus rating (2009): SJR 0.227 SNIP 0.634
Scopus rating (2008): SJR 0.125 SNIP 0.08
Scopus rating (2007): SJR 0.1 SNIP 0.097
Scopus rating (2006): SJR 0.113 SNIP 0.098
Scopus rating (2005): SJR 0.194 SNIP 0.839
A high order solver for the unbounded Poisson equation

In mesh-free particle methods a high order solution to the unbounded Poisson equation is usually achieved by constructing regularised integration kernels for the Biot-Savart law. Here the singular, point particles are regularised using smoothed particles to obtain an accurate solution with an order of convergence consistent with the moments conserved by the applied smoothing function. In the hybrid particle-mesh method of Hockney and Eastwood (HE) the particles are interpolated onto a regular mesh where the unbounded Poisson equation is solved by a discrete non-cyclic convolution of the mesh values and the integration kernel. In this work we show an implementation of high order regularised integration kernels in the HE algorithm for the unbounded Poisson equation to formally achieve an arbitrary high order convergence. We further present a quantitative study of the convergence rate to give further insight in the convergence of particle methods.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universite Catholique de Louvain
Authors: Hejlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Chatelain, P. (Ekstern), Walther, J. H. (Intern)
Number of pages: 1
Publication date: 2013
Event: Abstract from Workshop on Hybrid Particle-Continuum Methods in Computational Materials Physics, Jülich, Germany.
Main Research Area: Technical/natural sciences
Source: dtu
Source-ID: u::7359
Publication: Research - peer-review » Conference abstract for conference – Annual report year: 2013

A high order solver for the unbounded Poisson equation

A high order converging Poisson solver is presented, based on the Green’s function solution to Poisson’s equation subject to free-space boundary conditions. The high order convergence is achieved by formulating regularised integration kernels, analogous to a smoothing of the solution field. The method is extended to directly solve the derivatives of the solution to Poisson’s equation. In this way differential operators such as the divergence or curl of the solution field can be solved to the same high order convergence without additional computational effort. The method, is applied and validated, however not restricted, to the equations of fluid mechanics, and can be used in many applications to solve Poisson’s equation on a rectangular unbounded domain.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universite Catholique de Louvain
Authors: Hejlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Chatelain, P. (Ekstern), Walther, J. H. (Intern)
Pages: 458-467
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Computational Physics
Volume: 252
ISSN (Print): 0021-9991
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
An improved interface penalisation for vortex methods

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Swiss Federal Institute of Technology, California Institute of Technology
Authors: Hejlesen, M. M. (Intern), Koumoutsakos, P. (Ekstern), Leonard, A. (Ekstern), Walther, J. H. (Intern)
Number of pages: 1
Publication date: 2013
Event: Abstract from 3rd International Conference on Particle-Based methods, Stuttgart, Germany.
Main Research Area: Technical/natural sciences
Electronic versions:
An_improved_interface_penalisation_abstract.pdf

Bibliographical note
bibtex entry:

@INPROCEEDINGS{Hejlesen:2013c,
AUTHOR = "Hejlesen, M. M. and Walther, J. H",
TITLE = "An improved interface penalisation for vortex methods",
BOOKTITLE = "III International Conference on Particle-Based methods. Fundamentals and Applications",
PUBLISHER = "",
EDITOR = "",
YEAR = "2013",
MONTH = "September",
ADDRESS = "Stuttgart, Germany",
ORGANIZATION = "",
KEYWORD = "",
PAGES = ""},

dtu authors on this are:

jhwa and mmhej
mvh jens walther

Source: dtu
Source-ID: u::8771
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2013

An Iterative Brinkman penalization for particle vortex methods

We present an iterative Brinkman penalization method for the enforcement of the no-slip boundary condition in vortex particle methods. This is achieved by implementing a penalization of the velocity field using iteration of the penalized vorticity. We show that using the conventional Brinkman penalization method can result in an insufficient enforcement of solid boundaries. The specific problems of the conventional penalization method is discussed and three examples are presented by which the method in its current form has shown to be insufficient to consistently enforce the no-slip boundary condition. These are: the impulsively started flow past a cylinder, the impulsively started flow normal to a flat plate, and the uniformly accelerated flow normal to a flat plate. The iterative penalization algorithm is shown to give significantly improved results compared to the conventional penalization method for each of the presented flow cases.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, California Institute of Technology, Swiss Federal Institute of Technology
Authors: Walther, J. H. (Intern), Hejlesen, M. M. (Intern), Leonard, A. (Ekstern), Koumoutsakos, P. (Ekstern)
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Volume: 58
Issue number: 8
A note on added resistance for slow ships

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Bingham, H. B. (Intern), Amini Afshar, M. (Intern)
Number of pages: 4
Publication date: 2013

Bibliographical note
The authors wish to thank the EU 7th Framework Programme (grant # 266030, ULYSSES) for funding, and the Danish Center for Scientific Computing for supercomputing resources.
Main Research Area: Technical/natural sciences
Source: dtu
Source-ID: u::7047
Publication: Research › Working paper – Annual report year: 2013

A numerical shoreline model for shorelines with large curvature
This paper presents a new numerical model for shoreline change which can be used to model the evolution of shorelines with large curvature. The model is based on a one-line formulation in terms of coordinates which follow the shape of the shoreline, instead of the more common approach where the two orthogonal horizontal directions are used. The volume error in the sediment continuity equation which is thereby introduced is removed through an iterative procedure. The model treats the shoreline changes by computing the sediment transport in a 2D coastal area model, and then integrating the sediment transport field across the coastal profile to obtain the longshore sediment transport variation along the shoreline. The model is used to compute the evolution of a shoreline with a 90° change in shoreline orientation; due to this drastic change in orientation a migrating shoreline spit develops in the model. The dimensions of the spits evolving in the model compare favorably to previous model results and to field observation of the Skaw Spit in the north of Denmark.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Kærgaard, K. H. (Intern), Fredsøe, J. (Intern)
Pages: 19-32
Publication date: 2013
Main Research Area: Technical/natural sciences
Publication information
Journal: Coastal Engineering
Volume: 74
ISSN (Print): 0378-3839
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Backfilling of a Scour Hole around a Pile in Waves and Current

This paper presents the results of an experimental investigation of the backfilling of scour holes around circular piles. Scour holes around a pile are generated either by a current or a wave. Subsequently, the flow climate is changed from
current to wave, combined waves and current, or wave to a smaller wave, leading to the backfilling of the scour hole. The investigation has shed light onto the mechanism behind the backfilling process. The results show that the scour depth corresponding to the equilibrium state of backfilling is the same as that corresponding to the equilibrium state of scour around the pile for the same wave (or combined waves and current) climate. The time scale of backfilling has been determined as a function of three parameters, namely, (1) the Keulegan-Carpenter number of the initial wave or current (which generates the initial scour hole); (2) that of the subsequent wave, which backfills the scour hole; and (3) the Shields parameter associated with the latter wave, for live-bed conditions. In the case of the combined waves and current, the current-to-wave-velocity ratio is also involved. The time scale of the backfilling process is completely different from that of scour. The time scale of backfilling is much larger than that of scour when the Keulegan-Carpenter number associated with the backfilling is \( KC_f < O(10) \) (typical wind farm application), while the time scale of backfilling can be smaller than that of scour when \( KC_f >> O(10) \). DOI: 10.1061/(ASCE)WW.1943-5460.0000161. (C) 2013 American Society of Civil Engineers.
Barriers to Superfast Water Transport in Carbon Nanotube Membranes

Carbon nanotube (CNT) membranes hold the promise of extraordinary fast water transport for applications such as energy efficient filtration and molecular level drug delivery. However, experiments and computations have reported flow rate enhancements over continuum hydrodynamics that contradict each other by orders of magnitude. We perform large scale molecular dynamics simulations emulating for the first time the micrometer thick CNTs membranes used in experiments. We find transport enhancement rates that are length dependent due to entrance and exit losses but asymptote to 2 orders of magnitude over the continuum predictions. These rates are far below those reported experimentally. The results suggest that the reported superfast water transport rates cannot be attributed to interactions of water with pristine CNTs alone.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Strathclyde, Swiss Federal Institute of Technology, University of Illinois at Chicago
Authors: Walther, J. H. (Intern), Ritos, K. (Ekstern), Cruz-Chu, E. R. (Ekstern), Megaridis, C. M. (Ekstern), Koumoutsakos, P. (Ekstern)
Pages: 1910-1914
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Nano Letters
Volume: 13
ISSN (Print): 1530-6984
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 13.4
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): CiteScore 14.76
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
CFD simulation on Kappel propeller with a hull wake field

Marine propellers are designed not for the open-water operation, but for the operation behind a hull due to the inhomogeneous hull wake and thrust deduction. The adaptation for the hull wake is important for the propulsive efficiency and cavitation risk especially on single-screw ships.

CFD simulations for a propeller with a hull model have showed acceptable agreement with a model test result in the thrust and torque (Larsson et al. 2010). In the current work, a measured hull wake is applied to the simulation instead of modelling a hull, because the hull geometry is mostly not available for propeller designers and the computational effort can be reduced by excluding the hull. The CFD simulation of a propeller flow with a hull wake is verified in order to use CFD as a propeller design tool.

A Kappel propeller, which is an innovative tip-modified propeller, is handled. Kappel propellers are characterized by non-planar lifting surfaces and blade tips smoothly curved towards the suction side. An accurate analysis of the Kappel propeller using CFD is important due to the more complicating blade shape than the conventional propellers.

The propeller model is verified by comparing the open-water characteristics. The hull wake field is simulated without the propeller flow to check whether it is preserved at the propeller plane or not.

Propeller flow simulations are made with mean axial wake varying only along the radius (i.e. circumferentially uniform), whole axial wake and upstream transverse wake. It is investigated how the accuracy is improved, as the wake field is modelled more precisely. The thrust variation and pressure distribution on the blade from the CFD simulation with the hull wake model are also analyzed.
Comparisons of Means for Estimating Sea States from an Advancing Large Container Ship

The paper deals with sea state estimation from a container carrier (9,400 TEU) en-route. Knowledge of the on-site sea state is fundamental input to any kind of in-service decision support system that evaluates performance of, e.g., accelerations, fuel efficiency, and hull girder strength, related to ship-wave interactions in a seaway. In the paper, sea state estimates are produced by three means: the wave buoy analogy, relying on shipboard response measurements, a wave radar system, and a system providing the instantaneous wave height. The presented results show that for the given data, recorded on five different days of continuous operation, the agreement between the estimating means is reasonable; in terms of both absolute (mean) values and hourly trends of integrated sea state parameters.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MARIN
Authors: Nielsen, U. D. (Intern), Andersen, I. M. V. (Intern), Koning, J. (Ekstern)
Number of pages: 9
Publication date: 2013

Coupling of theory and practice through inductive learning in experimental fluid mechanics education: A practical study

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Office for Study Programmes and Student Affairs
Authors: Velte, C. M. (Intern), Andersson, P. H. (Intern), Meyer, K. E. (Intern)
Number of pages: 8
Publication date: 2013
Educating Maritime Engineers for a Globalised Industry

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Nielsen, U. D. (Intern)
Number of pages: 15
Publication date: 2013

Effect of air on water capillary flow in silica nanochannels
Capillarity is a classical topic in fluid dynamics. The fundamental relationship between capillarity and surface tension is solidly established. Nevertheless, capillarity is an active research area especially as the miniaturization of devices is reaching the molecular scale. Currently, with the fabrication of microsystems integrated by nanochannels, a thorough understanding of the transport of fluids in nanoconfinement is required for a successful operation of the functional parts of such devices. In this work, Molecular Dynamics simulations are conducted to study the spontaneous imbibition of water in sub 10 nm silica channels. The capillary filling speed is computed in channels subjected to different air pressures. In order to describe the interactions between the species, an effective force field is developed, which is calibrated by reproducing the water contact angle. The results show that the capillary filling speed qualitatively follows the classical Washburn model, however, quantitatively it is lower than expected. Furthermore, it is observed that the deviations increase as air pressure is higher. We attribute the deviations to amounts of air trapped at the silica-water interface which leads to changes in the dynamics contact angle of the water meniscus.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universidad de Concepcion
Authors: Zambrano, H. (Ekstern), Walther, J. H. (Intern), Oyarzua, E. (Ekstern)
Publication date: 2013
Main Research Area: Technical/natural sciences

Effect of air on water capillary flow in silica nanochannels

Electronic versions:
Presentation_NMU13_udn.pdf
Source: dtu
Source-ID: u::6841
Publication: Research › Sound/Visual production (digital) – Annual report year: 2013
Effects from fully nonlinear irregular wave forcing on the fatigue life of an offshore wind turbine and its monopile foundation

The effect from fully nonlinear irregular wave forcing on the fatigue life of the foundation and tower of an offshore wind turbine is investigated through aeroelastic calculations. Five representative sea states with increasing significant wave height are considered in a water depth of 40 m. The waves are both linear and fully nonlinear irregular 2D waves. The wind turbine is the NREL 5-MW reference wind turbine. Fatigue analysis is performed in relation to analysis of the sectional forces in the tower and monopile.

Impulsive excitation of the sectional force at the bottom of the tower is seen when the waves are large and nonlinear and most notably for small wind speeds. In case of strong velocities and turbulent wind, the excitation is damped out. In the monopile no excitation of the force is seen, but even for turbulent strong wind the wave affects the forces in the pile significantly. The analysis indicates that the nonlinearity of the waves can change the fatigue damage level significantly in particular when the wave and wind direction is misaligned.

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Aeroelastic Design, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Schløer, S. (Intern), Bredmose, H. (Intern), Bingham, H. B. (Intern), Larsen, T. J. (Intern)
Pages: 393-402
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the ASME 31th 2012 International Conference on Ocean, Offshore and Arctic Engineering
Volume: 7
Publisher: American Society of Mechanical Engineers
Chapter: OMAE2012-83477
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Conference: 31st ASME International Conference on Ocean, Offshore and Arctic Engineering, Rio de Janeiro, Brazil, 01/07/2012 - 01/07/2012
Source: du
Source-ID: u::5920
Publication: Research - peer-review › Article in proceedings – Annual report year: 2012

Efficient computations of wave loads on offshore structures
The present thesis considers numerical computations of fully nonlinear wave impacts on bottom mounted surface piercing circular cylinders at intermediate water depths. The aim of the thesis is to provide new knowledge regarding wave loads on foundations for offshore wind turbines. Hence, the dimensions of the cylinders and the chosen wave parameters were inspired by typical monopile foundations for offshore wind turbines.

The numerical computations are carried out using three numerical solvers. That is, the fully nonlinear Navier-Stokes/VOF solver provided as a part of the open-source CFD-toolbox OpenFoam R, the fully nonlinear potential flow solver OceanWave3D and finally a fully nonlinear domain decomposed solver, which was developed as part of this project. In the domain decomposed solver, the outer wave field is described by the potential flow solver, whereas the inner wave field, in the vicinity of a given structure, is described by the Navier-Stokes/VOF solver. All numerical models are carefully validated either in terms of convergence by grid refinement or by comparisons to experimental measurements. Special attention is paid to the newly developed domain decomposed solver, which is carefully validated against experimental measurements of regular-, irregular- and multi-directional irregular waves. The ability of the numerical model to accurately reproduce experiments is also investigated.

Wave impacts on a bottom mounted circular cylinder from steep regular waves are presented. Here, the inline forces and the motion of the free surface is described as a function of the non-dimensional wave steepness, the relative water depth, the relative cylinder diameter and a co-existing current. From the computations, higher harmonic forces are determined and compared against the Morison equation and established analytical force formulations accurate to the third order in wave steepness.

The physics related to the strongly nonlinear load phenomena “secondary load cycles” is described and an explanation of the wave load phenomena is provided. To further support the explanation a simple inviscid kinematic model flow is derived.

The discussion of wave impacts on circular cylinders is further extended to uni- and bi-directional phase-focused waves. Here, the influence of the nondimensional wave steepness and wave directionality is discussed. For the steepest wave impacts “secondary load cycles” are observed and the physics of the impact and the mechanisms related to the “secondary load cycle” are discussed and compared to the observations made for regular waves.

Additionally, attention is paid to experimental determination of hydrodynamic forces. Significant differences between experimentally measured and computed higher harmonic forces are observed and the differences are explained in terms of the eigenmotion of the test setup. Finally, the application of the domain decomposed solver is discussed in an engineering context. Here, a simple and robust way of identifying forces, which may be inaccurately estimated by the Morison equation, is presented. It is suggested that these impacts are recomputed by the domain decomposed solver.
Efficient Hybrid-Spectral Model for Fully Nonlinear Numerical Wave Tank
A new hybrid-spectral solution strategy is proposed for the simulation of the fully nonlinear free surface equations based on potential flow theory. A Fourier collocation method is adopted horizontally for the discretization of the free surface equations. This is combined with a modal Chebyshev Tau method in the vertical for the discretization of the Laplace equation in the fluid domain, which yields a sparse and spectrally accurate Dirichletto-Neumann operator. The Laplace problem is solved with an efficient Defect Correction method preconditioned with a spectral discretization of the linearised wave problem, ensuring fast convergence and optimal scaling with the problem size. Preliminary results for very nonlinear waves show expected convergence rates and a clear advantage of using spectral schemes.

Evaluation of Response Prediction Procedures using Full Scale Measurements for a Container Ship
This paper deals with the analysis of recent full-scale strain measurements in the hull of a large container carrier covering several months of operation. The focus is on the real-time prediction accuracy of responses 5-15 seconds ahead of the measurements. Such results are less applicable in the operation of container carriers but are important in e.g. loading/unloading operations at sea or helicopter landings.
Three different procedures are discussed: Conditional processes with analytical estimates of the mean values and standard deviations, the autoregressive predictor method and a method based on superposition of sinusoidal components. The conditional processes do not need offline training and will be applied to measured time series in order to evaluate the accuracy of response predictions within the next 1-30 seconds. The number of measured points and the time distances between them are varied to determine the best solutions. A procedure based on 11 measured points spaced 1 sec, covering the last 10 sec of the instantaneous measured signal seems generally able to give fair predictions up to 5-10 sec ahead of the current time.
The full-scale data is provided through the EU FP7 project Tools for Ultra Large Container Ships (TULCS) project no. 234146.
Flow and bed shear stresses in scour protections around a pile in a current

Transport of bed sediment inside and beneath the scour protection may cause deformation and sinking of the scour protection for pile foundations. This may reduce the stability of the mono pile and change the natural frequency of the dynamic response of an offshore wind turbine installed on it in an unfavourable manner. Using physical models and 3D computational fluid dynamic (CFD) numerical simulations, the velocity and bed shear stresses are investigated in complex scour protections around mono piles in steady current. In the physical model the scour protections consisted of an upper cover layer with uniformly distributed coarse stones and a lower filter layer with finer stones. For the numerical simulations, the Flow-3D software was used. The scour protection layers were simulated with different numerical approaches, namely regularly arranged spheres, porous media, or their combinations (hybrid models). Numerical simulations with one or four layers of cover stones without filter layer were first computed. Three additional simulations were then made for a scour protection with a cover layer and a single filter layer. Finally, a simulation of a full scale foundation and scour protection was made with porous media approach. Based on the physical and numerical results, a method to determine the critical stones size to prevent motion of the base sediment is established and compared to a full scale case with sinking of scour protection (Horns Rev I Offshore Wind Farm, Denmark). It is also found that the CFD simulations are capable of calculating the flow velocities when the scour protection is represented by regular arranged spheres, while the turbulence in general is underestimated. The velocity can also be calculated using porous media flow approach, but the accuracy is not as good as for spheres. The deviation is more severe for more complex scour protections. In general, computational models provide valuable information for the prediction and design of scour protections for offshore wind farms.
Fluid Dynamic Characterization of Vortex Generators and Two-dimensional Turbulent Wakes

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Polytechnic University of Catalonia
Authors: Fernández-Gámiz, U. (Ekstern), Estevez, E. E. (Ekstern), Velte, C. M. (Intern)
Number of pages: 127
Publication date: 2013
Fully Coupled Three-Dimensional Dynamic Response of a TLP Floating Wind Turbine in Waves and Wind

A dynamic model for a tension-leg platform (TLP) floating offshore wind turbine is proposed. The model includes three-dimensional wind and wave loads and the associated structural response. The total system is formulated using 17 degrees of freedom (DOF), 6 for the platform motions and 11 for the wind turbine. Three-dimensional hydrodynamic loads have been formulated using a frequency- and direction-dependent spectrum. While wave loads are computed from the wave kinematics using Morison’s equation, aerodynamic loads are modelled by means of unsteady Blade-Element-Momentum (BEM) theory, including Glauert correction for high values of axial induction factor, dynamic stall, dynamic wake and dynamic yaw. The aerodynamic model takes into account the wind shear and turbulence effects. For a representative geographic location, platform responses are obtained for a set of wind and wave climatic conditions. The platform responses show an influence from the aerodynamic loads, most clearly through a quasi-steady mean surge and pitch response associated with the mean wind. Further, the aerodynamic loads show an influence from the platform motion through more fluctuating rotor loads, which is a consequence of the wave-induced rotor dynamics. In the absence of a controller scheme for the wind turbine, the rotor torque fluctuates considerably, which induces a growing roll response especially when the wind turbine is operated nearly at the rated wind speed. This can be eliminated either by appropriately adjusting the controller so as to regulate the torque or by optimizing the floater or tendon dimensions, thereby limiting the roll motion. Loads and coupled responses are predicted for a set of load cases with different wave headings. Based on the results, critical load cases are identified and discussed. As a next step (which is not presented here), the dynamic model for the substructure is therefore being coupled to an advanced aero-elastic code Flex5, Øye (1996), which has a higher number of DOFs and a controller module.
Heat Transfer in Large Two-Stroke Marine Diesel Engines

Heat transfer between the cylinder gas and the piston surface during combustion in large two-stroke uniflow scavenged marine diesel engines has been investigated in the present work. The piston surface experiences a severe thermal load during combustion due to the close proximity of the combustion zone to the surface. At the same time, cooling of the piston crown is relatively complicated. This can cause large thermal stresses in the piston crown and weakening of the material strength, which may be critical as it can lead to formation of cracks. Information about the piston surface heat transfer is thus important for the engine manufacturers.

The piston surface heat transfer was studied in the event of impingement of hot combustion products on the piston during combustion, and an estimate was obtained of the peak heat flux level experienced on the piston surface. The investigation was carried out numerically by performing simulations with a CFD code of the heat transfer between gas and wall in a jet impingement configuration where a hot round turbulent gas jet impinged normally onto a wall under conditions approximating the in-cylinder conditions in the engine during combustion.

A jet impingement reference case was first established based on estimations of the incylinder conditions during combustion. Subsequently, variations of different jet impingement parameters were performed and the influence on the wall heat transfer was observed. In all the cases, the ratio between the jet inlet to wall distance, H, and the jet diameter at the inlet, D, was H/D = 2. The jet Reynolds number, Re, varied between 1.10•10^5 and 6.64•10^5. The resulting Nusselt numbers along the wall were calculated for dimensionless radial distances from the stagnation point, r/D, between 0 and 6. The maximum Nusselt number was located in the stagnation point in most of the investigated cases, and an analysis was performed of the variation of the stagnation point Nusselt number, Nu0, with the jet Reynolds number and the jet turbulence intensity at the jet inlet, TI. Based on the observed relations, a correlation between Nu0, Re and TI is suggested for high jet Reynolds number cases. A satisfactory validation of the correlation was not possible to perform due to insufficient available experimental data. A comparison of the correlation predictions to existing experimental data indicated however an overprediction of Nu0 in the magnitude of 50% – 100%. The overprediction is considered to be caused primarily by incorrect numerical model predictions. Based on the performed jet impingement heat transfer investigations, an estimate is provided of the peak convective piston surface heat flux level experienced in the considered large marine diesel engines. The contribution from thermal radiation to the piston surface heat flux was not investigated in the present work, but a coarse estimation of the magnitude was performed. The obtained estimations indicate a peak piston surface heat flux level in the interval from about 1 MW/m^2 and up to 9.5 MW/m^2 with the actual value probably being in the lower part of this interval. This is about the same magnitude as that previously reported for automotive size diesel engines. The obtained interval is relatively large, but a more accurate prediction is difficult to achieve with the applied method due to limited knowledge about the actual local in-cylinder conditions during combustion. Therefore, further research in this area is encouraged.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy
Authors: Jensen, M. V. (Intern), Walther, J. H. (Intern), Carlsen, H. (Intern), Schramm, J. (Intern)
Number of pages: 196
Publication date: 2013

Publication information

Publisher: DTU Mechanical Engineering
ISBN (Print): 978-87-7475-364-3
Original language: English

Series: DCAMM Special Report
Number: S157
ISSN: 0903-1685
Main Research Area: Technical/natural sciences

Hull Girder Fatigue Damage Estimations of a Large Container Vessel by Spectral Analysis

This paper deals with fatigue damage estimation from the analysis of full-scale stress measurements in the hull of a large container vessel (9,400 TEU) covering several months of operation. For onboard decision support and hull monitoring systems, there is a need for a fast reliable method for estimation of fatigue damage in the ship hull.

The objective of the study is to investigate whether the higher frequency contributions from the hydroelastic responses (springing and whipping) can satisfactorily be included in the fatigue damage estimation by only a few parameters derived from the stress power spectrum as an alternative to rainflow counting. The outcome of rainflow counting is compared with the results of a narrow-band spectral analysis as well as the outcome of three approaches considering the process as bimodal in order to take the hydroelastic responses into account. In most cases, the spectral analysis show satisfactory agreement with the results from rainflow counting.
Hybrid morphological modelling of shoreline response to a detached breakwater

We present a new type of model for calculating morphological changes induced by the presence of breakwaters. The model combines a process based area model, used to calculate the sediment transport field in the two horizontal dimensions, with a simplified morphological updating scheme where the evolving cross-shore profile is described by a limited number of parameters. The hybrid morphological model is a strong tool for medium and long term modelling because it is cost effective while containing important features of the sediment transport description. Two versions of the model are developed in order to study the evolution of beach morphology: one suited for offshore breakwaters (1D model) and one mainly dedicated to coastal breakwaters ("1.5D" model). The version for offshore breakwaters is first presented and tested against field observations of salient evolution. The model is then applied to a model study of the principle correlations between evolving salients (spatial and temporal scales), the characteristic dimensions of the breakwater (distance to shore and alongshore length) and wave climate (wave height, normal and oblique wave incidence). The second version is applied to investigate in more detail the evolving morphology behind coastal breakwaters. It is demonstrated how the model is able to calculate the evolution of either salient or tombolo planforms, and furthermore it is shown that the results are in reasonable agreement with existing rules.
Hybrid-Spectral Model for Fully Nonlinear Numerical Wave Tank

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Applied Mathematics and Computer Science, Scientific Computing
Authors: Christiansen, T. R. B. (Intern), Engsig-Karup, A. P. (Intern), Bingham, H. B. (Intern)
Number of pages: 3
Pages: 37-40
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the 28th International Workshop on Water Waves and Floating Bodies
Main Research Area: Technical/natural sciences
Innovative multi-purpose offshore platforms: Planning, design and operation

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Flanders Marine Institute, University of Bologna
Authors: Claus, S. (Ekstern), Seys, J. (Ekstern), Pirlet, H. (Ekstern), Mees, J. (Ekstern), Christensen, E. D. (Intern), Zanitigh, B. (Ekstern)
Number of pages: 2
Publication date: 2013
Event: Abstract from LITTORAL 2012, Oostende, Belgium.
Main Research Area: Technical/natural sciences
Source: dtu
Source-ID: u::8344
Publication: Research - peer-review » Conference abstract for conference – Annual report year: 2013

Laboratory observations of flow and sediment transport induced by plunging regular waves

Two parallel experiments involving the evolution and runup induced by plunging regular waves near the shoreline of a sloping bed are considered: (1) a rigid-bed experiment, allowing direct (hot film) measurements of bed shear stresses and (2) a sediment-bed experiment, allowing for the measurement of pore-water pressures as well as observation of sediment suspension and bed morphological changes. Both experiments utilize the same initial bed profile and wave forcing. The experiments show that the mean bed shear stresses experienced onshore of incipient breaking are amplified by nearly a factor of 2 relative to prebreaking conditions, whereas their corresponding turbulent fluctuations are amplified even more strongly, by a factor of 5–6. The plunging processes lead to a series of vortices, whose formation may be explained as the result of shear layer instability. Measurements show that these vortices can significantly enhance peaks in the offshoredirected bed shear stresses. Moreover, near-bed pore pressure measurements indicate that these vortices cause large upward-directed pressure gradients, which in turn produce a corresponding series of suspended sediment plumes shoreward of the initial breaking event. These findings are related to the induced morphological changes over both short and long time scales. The present results are also compared and contrasted with previous experiments utilizing a similar methodology, but involving plunging solitary waves.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Sumer, B. M. (Intern), Güner, A. (Intern), Hansen, N. M. (Intern), Fuhrman, D. R. (Intern), Fredsøe, J. (Intern)
Pages: 6161–6182
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Geophysical Research
Volume: 118
Issue number: 11
ISSN (Print): 0148-0227
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.36 SJR 1.996 SNIP 1.313
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 2.288 SNIP 1.362 CiteScore 3.39
Loads and response from steep and breaking waves on monopiles

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Aeroelastic Design, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Marine and Coastal Morphology: medium term and long-term area modelling

This thesis documents development and application of a modelling concept developed in collaboration between DTU and DHI. The modelling concept is used in morphological modelling in coastal areas where the governing sediment transport processes are due to wave action. The modelling concept is defined: Hybrid morphological modelling and it is based on coupling calculated sediment transport fields from a traditional process based coastal area model with a parametrised morphological evolution model. The focus of this study is to explore possible parametric formulations of the morphological evolution model and apply them to problems concerning coastal protection strategies (both hard and soft measures). The applied coastal protection strategies involve morphological impact of detached shore parallel segmented breakwaters and shore normal impermeable groynes in groyne fields, and morphological evolution around seawalls and response to beach and shoreface nourishment.

The hybrid morphological modelling concept is introduced and is put into context with existing models used for medium term and long-term morphological simulation. The modelling concept is intended to improve the long-term predictive capabilities of process based area models thereby bridging the gap between short term models and long-term models. A number of different implementations within the hybrid morphological modelling concept have been developed and tested in order to ascertain the usefulness of the concept. The implementations are grouped into shoreline models and models used to develop local features on the coastal profile such as alongshore migration of a bar front or redistribution of a shoreface nourishment. The implementations are tested on a variety of problems (idealised cases and case studies at specific locations). Applications of the modelling concept have been collected into 5 papers (1 accepted journal paper, 3 published conference papers and 1 unpublished journal manuscript) which each contain an introduction to the implementation, the motivation for the implementation and the overall results from the application. The 5 papers are enclosed as separate chapters of this thesis.

The overall conclusions to each of the model implementations are given as follows.

1D shoreline model
Implementing the hybrid morphological modelling concept with a 1D morphological model is a strong engineering tool. The model is robust and computationally efficient and it may be adapted to real engineering problems. The results of the 1D shoreline model are however greatly affected by the imposed freedom of the model, and exaggerated alongshore smoothing of the calculated shoreline may occur if the 1D model is applied to problems in which the true solution has a two dimensional nature.

1.5D shoreline model
A so-called “1.5D” implementation which introduces redistribution of sediment on the shoreface in response to horizontal 2D currents makes it possible to simulate the morphological development in areas where 2D evolution occurs. The coastal profiles tend however to drift due to the fact that the response of an entire profile is coupled thereby in some cases leading to morphological activity in inactive areas. Diffusion of the coastal profile is therefore introduced in order to weakly force the profile towards an equilibrium form. The 1.5D model is seen to produce reasonable results when subject to cases with detached breakwaters and groynes. The computational efficiency of the model is however reduced compared to the 1D model, because the increased freedom of the model reduces the maximum stable morphological time step.

Bar models
The models for redistributing sediment on the shoreface have been applied with limited success. The models can only be applied to cases where cross-shore transport processes are of minor importance, due to inaccuracies in the calculated cross-shore transport in the applied 2D coastal model. Stabilisation of the bar parameters using behaviour-oriented methods may be possible but will require a strong forcing which according to the author reduces the significance of the modelling concept.
Measurement of the rotor wake using PIV on a scaled turbine rotor in a water flume

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Russian Academy of Sciences
Authors: Mikkelsen, R. F. (Intern), Okulov, V. (Intern), Meyer, K. E. (Intern), Naumov, I. (Ekstern), Karbardin, I. (Ekstern), Sørensen, J. N. (Intern)
Number of pages: 26
Publication date: 2013

Publication information
Media of output: PowerPoint
Original language: English
Main Research Area: Technical/natural sciences
Electronic versions:
Publication: Research › Ph.D. thesis – Annual report year: 2013

Measuring Sandy Bottom Dynamics by Exploiting Depth from Stereo Video Sequences

In this paper an imaging system for measuring sandy bottom dynamics is proposed. The system exploits stereo sequences and projected laser beams to build the 3D shape of the sandy bottom during time. The reconstruction is used by experts of the field to perform accurate measurements and analysis in the study of the final equilibrium conditions of sea bottoms in the presence of water flows. Results obtained by processing data acquired in hydraulic laboratory confirm the effectiveness of the system which makes simple and fast the understanding of the sandy bottom dynamics and the related equilibrium phenomena. © 2013 Springer-Verlag.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Catania
Authors: Musumeci, R. E. (Ekstern), Farinella, G. M. (Ekstern), Foti, E. (Ekstern), Battiato, S. (Ekstern), Petersen, T. U. (Intern), Sumer, B. M. (Intern)
Number of pages: 11
Pages: 420-430
Publication date: 2013

Host publication information
Title of host publication: Image Analysis and Processing – ICIAP 2013
Volume: 8156
Publisher: Springer
ISBN (Print): 978-3-642-41180-9
ISBN (Electronic): 978-3-642-41181-6

Series: Lecture Notes in Computer Science
Volume: 8156
ISSN: 0302-9743
Main Research Area: Technical/natural sciences
Image analysis, Dynamics, Accurate measurement, Depth from stereo, Equilibrium conditions, Equilibrium phenomena, Presence of water, Sea bottom, Stereo system, Stereosequence, Sandy bottom dynamics
DOIs: 10.1007/978-3-642-41181-6_43

Bibliographical note
The research described in this manuscript was supported by the EC project HYDRA LAB IV (contract no. 261520) and by the Italian Ministry for Education, University and Research funded PRIN 2010-2011 HYDROCAR (Prot. no. 20104J2Y8M
Numerical analysis of jet impingement heat transfer at high jet Reynolds number and large temperature difference

Jet impingement heat transfer from a round gas jet to a flat wall was investigated numerically for a ratio of 2 between the jet inlet to wall distance and the jet inlet diameter. The influence of turbulence intensity at the jet inlet and choice of turbulence model on the wall heat transfer was investigated at a jet Reynolds number of $1.66 \times 10^5$ and a temperature difference between jet inlet and wall of $1600$ K. The focus was on the convective heat transfer contribution as thermal radiation was not included in the investigation. A considerable influence of the turbulence intensity at the jet inlet was observed in the stagnation region, where the wall heat flux increased by a factor of almost 3 when increasing the turbulence intensity from 1.5% to 10%. The choice of turbulence model also influenced the heat transfer predictions significantly, especially in the stagnation region, where differences of up to about 100% were observed. Furthermore, the variation in stagnation point heat transfer was examined for jet Reynolds numbers in the range from $1.10 \times 10^5$ to $6.64 \times 10^5$. Based on the investigations, a correlation is suggested between the stagnation point Nusselt number, the jet Reynolds number, and the turbulence intensity at the jet inlet for impinging jet flows at high jet Reynolds numbers.

Copyright © 2013 Taylor and Francis Group, LLC.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, M. V. (Intern), Walther, J. H. (Intern)
Pages: 801-809
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Heat Transfer Engineering
Volume: 34
Issue number: 10
ISSN (Print): 0145-7632
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.594 SNIP 0.847 CiteScore 1.25
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.841 SNIP 0.984 CiteScore 1.23
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.579 SNIP 0.961 CiteScore 0.98
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.51 SNIP 0.888 CiteScore 1.03
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.613 SNIP 0.868 CiteScore 0.87
ISI indexed (2012): ISI indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 0.624 SNIP 0.991 CiteScore 1.13
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 0.7 SNIP 1.121
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 0.835 SNIP 0.935
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 0.572 SNIP 0.734
The paper describes the results of a numerical and experimental investigation of flow and scour around a half-buried sphere exposed to a steady current. Hot-film bed shear stress and Laser Doppler Anemometer measurements were made with a half sphere mounted on the smooth bed in an open channel. The hydrodynamic model is a 3-D general purpose N–S flow solver. The k-omega SST turbulence model was used for closure. The flow model was used to study the horseshoe vortex and lee-wake vortex flow processes around the sphere. The flow model was coupled with a morphologic model to calculate scour around the half-buried sphere in currents. The morphologic model includes a sediment-transport description, and a description of surface-layer sand slides for bed slopes exceeding the angle of repose. The sediment transport description includes, for the first time, the effect of externally-generated turbulence (induced by the horseshoe-vortex flow and the lee-wake flow processes) on sediment transport. The results show that the scour depth increases and time scale decreases when the effect of externally-generated turbulence is incorporated in the calculations. Empirical expressions representing the numerically obtained data on the equilibrium scour depth and the time scale are presented. The results show that the equilibrium maximum scour depth in the live-bed regime can be approximated by 0.5 D in which D is the sphere diameter.
Numerical Investigation of the Scavenging Process in Marine Two-Stroke Diesel Engines

The scavenging process is an integral part of any two-stroke internal combustion engine cycle whether it is spark ignited or compression ignited. The scavenging process is responsible for transporting the burned gases from the previous working stroke out of the combustion chamber to allow for the fresh charge or fresh air to enter for the next combustion/working stroke. This implies that the scavenging process is responsible for setting the initial condition for the
combustion process, consequently affecting fuel economy, power output and emission of hazardous gases. Two-stroke diesel engines for marine propulsion are usually uniflow scavenged cross-head engines. In uniflow scavenged engines the scavenge air enters the cylinder via ports located near the bottom dead center and exits through an exhaust valve located in the cylinder head. The in cylinder flow is therefore concentrated in one direction which gives the method its name. In this study a CFD analysis of the scavenging process in the 4T50ME-X test engine at MAN Diesel & Turbo is presented. The CFD model uses the full engine geometry including a moving piston and valve combined with time resolved measurement data as boundary and initial conditions. The results are then validated with experimental data from the engine. Optical velocity measurements have been performed through a specially designed engine cover with optical access.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Andersen, F. H. (Intern), Hult, J. (Ekstern), Nogenmyr, K. (Ekstern), Mayer, S. (Ekstern)
Number of pages: 12
Publication date: 2013

**Host publication information**

Title of host publication: Technical papers presented at the SAE/KSAE 2013 International Powertrains, Fuels & Lubricants Meeting
Publisher: Society of Automotive Engineers, Incorporated

Series: S A E Technical Papers
Number: 2013-01-2647
ISSN: 0148-7191
Main Research Area: Technical/natural sciences
DOIs: 10.4271/2013-01-2647
Source: dtu
Source-ID: u::9248
Publication: Research - peer-review › Article in proceedings – Annual report year: 2013

**Numerical modeling of shoreline undulations part 1: Constant wave climate**

This paper presents a numerical study of the non-linear development of alongshore undulations up to fully developed quasi-steady equilibrium. A numerical model which describes the longshore sediment transport along arbitrarily shaped shorelines is applied, based on a spectral wave model, a depth integrated flow model, a wave-phase resolving sediment transport description and a one-line shoreline model. First the length of the shoreline undulations is determined in the linear regime using a stability analysis. Next the further evolution from the linear to the fully non-linear regime is described. In the fully non-linear regime down-drift spits and migrating shoreline undulations are described. Three different shoreline shapes are found depending on the wave conditions: undulations with no spits, undulations with shore parallel spit and undulations with reconnecting spits. © 2012 Published by Elsevier B.V.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Coastal, Maritime and Structural Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Kærgaard, K. H. (Intern), Fredsøe, J. (Intern)
Pages: 64-76
Publication date: 2013
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Coastal Engineering
Volume: 75
ISSN (Print): 0378-3839
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Shoreline undulations, Shoreline spit, Numerical shoreline model, Longshore sediment transport, Area model

Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.925 SNIP 2.097 CiteScore 2.9
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.785 SNIP 2.123 CiteScore 2.55
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.727 SNIP 2.264 CiteScore 2.58
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.912 SNIP 2.226 CiteScore 2.21
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.616 SNIP 2.502 CiteScore 2.43
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 1.898 SNIP 2.332
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 2.067 SNIP 2.454
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 2
Scopus rating (2008): SJR 1.189 SNIP 2.166
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 1.642 SNIP 2.164
Web of Science (2007): Indexed yes
Scopus rating (2006): SJR 1.249 SNIP 2.2
Web of Science (2006): Indexed yes
Scopus rating (2005): SJR 1.22 SNIP 1.966
Web of Science (2005): Indexed yes
Scopus rating (2004): SJR 1.143 SNIP 2.273
Web of Science (2004): Indexed yes
Scopus rating (2003): SJR 0.997 SNIP 1.873
Web of Science (2003): Indexed yes
Scopus rating (2002): SJR 0.729 SNIP 1.104
Web of Science (2002): Indexed yes
Scopus rating (2001): SJR 0.864 SNIP 1.127
Web of Science (2001): Indexed yes
Scopus rating (2000): SJR 0.832 SNIP 1.273
Web of Science (2000): Indexed yes
Scopus rating (1999): SJR 0.534 SNIP 1.096
Original language: English
Shoreline undulations, Shoreline spit, Numerical shoreline model, Longshore sediment transport, Area model

DOIs:
10.1016/j.coastaleng.2012.11.006
Source: dtu
Source-ID: n:oai:DTIC-ART:elsevier/384054105::27398
Publication: Research - peer-review › Journal article – Annual report year: 2013
Numerical modeling of shoreline undulations part 2: Varying wave climate and comparison with observations

The present work applies the shoreline model from part 1 to a real environment. In part 1, a numerical shoreline model which could handle the development of arbitrarily shaped shorelines was applied to consider the development of shoreline undulations on an unstable shoreline exposed to incoming waves with a directional spreading. In this paper, these findings are extended to Firstly include the effect of a varying wave climate on the shoreline morphology and secondly, to tune the model to two naturally occurring shorelines. It is found that the effect of a variable wave climate is to slow down the development of the morphology and in some cases to inhibit the formation of shore-parallel spits at the crest of the undulations. On one of the natural shorelines, the west coast of Namibia, the shore is exposed to very obliquely waves from one main direction. Here, the shoreline model is able to describe the observed shoreline features qualitatively and quantitatively. The model slightly over-predicts the scale of the feature and, associated with this, slightly under-predicts the migration speeds of the features. On the second shoreline, the west coast of Denmark, the shore is exposed to waves with an angle close to the critical around 45°, and here the existence of undulations is discussed in detail. © 2012 Published by Elsevier B.V.
Numerical simulations on a twin-plate wake

In this work, a detailed numerical analysis of two-dimensional mean velocity profiles downstream of two parallel flat plates was carried out at a Reynolds number of $3.2 \times 10^4$ (based on the plate length and free stream velocity) using Reynolds Averaged Navier-Stokes (RANS) and have been compared with experimental data. Furthermore, a self-similar study of the wake behind the twin plate was carried out based on the computer simulations. The main objective is to investigate how well the simulations can reproduce the physics of the flow behind a twin plate.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of the Basque Country, Polytechnic University of Catalonia
Authors: Fernández-Gámiz, U. (Ekstern), Velte, C. M. (Intern), Egusquiza, E. (Ekstern)
Number of pages: 6
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the 5th International Congress of Energy and Environment Engineering and Management (CIIEM)
Main Research Area: Technical/natural sciences
Conference: 5th International Congress of Energy and Environment Engineering and Management (CIIEM), Lisbon, Portugal, 17/07/2013 - 17/07/2013

Electronic versions:
Source-ID: u::10572
Publication: Research - peer-review › Book chapter – Annual report year: 2014

Phase-locked stereoscopic PIV measurements of the turbulent swirling flow in a dynamic model of a uniflow-scavenged two-stroke engine cylinder

It is desirable to use computational fluid dynamics for the optimization of in-cylinder processes in large two-stroke low-speed uniflow-scavenged marine diesel engines. However, the complex nature of the turbulent swirling in-cylinder flow
necessitates experimental data for validation of the used turbulence models. In the present work, the flow in a dynamic scale model of a uniflow-scavenged cylinder is investigated experimentally. The model has a transparent cylinder and a movable piston driven by a linear motor. The flow is investigated using phase-locked stereoscopic particle image velocimetry (PIV) and time resolved laser Doppler anemometry (LDA). Radial profiles of the phase-averaged mean velocities are computed from the velocity fields recorded with PIV and the validity of the obtained profiles is demonstrated by comparison with reference LDA measurements. Radial profiles are measured at five axial positions for 15 different times during the engine cycle and shows the temporal and spatial development of the swirling in-cylinder flow. The tangential velocity profiles in the bottom of the cylinder near the end of the scavenging process are characterized by a concentrated swirl resulting in wake-like axial velocity profiles and the occurrence of a vortex breakdown. After scavenging port closing the axial velocity profiles indicate that large transient swirl-induced structures exists in the cylinder. Comparison with profiles obtained under steady-flow conditions shows that the steady profiles in general will not be representative for the dynamic conditions. The temporal development of the swirl strength is investigated by computing the angular momentum. The swirl strength shows an exponential decay from scavenging port closing to scavenging port opening corresponding to a reduction of 34%.

**PIV in a model wind turbine rotor wake**

Stereoscopic particle image velocimetry (PIV) measurements of the flow in the wake of scale model of a horizontal axis wind turbine is presented Near the rotor, measurements are made in vertical planes intersecting the rotor axis These planes capture flow effect from the tip and root vortices The stability of the tip vortices as a function of different tip speed ratios are demonstrated An instability seems to occur after a time corresponding to four blade passages The result is vortex pairing and in some cases grouping of three vortices Further downstream in the wake, measurements in planes perpendicular to the rotor axis is used to investigate the dynamics in the far wake Here, a precessing core is found and data indicate that the Strouhal number of the precessing is independent of the rotor speed

**PIV study of the effect of piston position on the in-cylinder swirling flow during the scavenging process in large two-stroke marine diesel engines**
A simplified model of a low speed large two-stroke marine diesel engine cylinder is developed. The effect of piston position on the in-cylinder swirling flow during the scavenging process is studied using the stereoscopic particle image velocimetry technique. The measurements are conducted at different cross-sectional planes along the cylinder length and at piston positions covering the air intake port by 0, 25, 50 and 75%. When the intake port is fully open, the tangential velocity profile is similar to a Burgers vortex, whereas the axial velocity has a wake-like profile. Due to internal wall friction, the swirl decays downstream, and the size of the vortex core increases. For increasing port closures, the tangential velocity profile changes from a Burgers vortex to a forced vortex, and the axial velocity changes correspondingly from a wake-like profile to a jet-like profile. For piston position with 75% intake port closure, the jet-like axial velocity profile at a cross-sectional plane close to the intake port changes back to a wake-like profile at the adjacent downstream cross-sectional plane. This is characteristic of a vortex breakdown. The non-dimensional velocity profiles show no significant variation with the variation in Reynolds number.

**General information**

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy, Department of Wind Energy, Fluid Mechanics, MAN Diesel & Turbo SE
Authors: Haider, S. (Intern), Schnipper, T. (Intern), Obeidat, A. (Intern), Meyer, K. E. (Intern), Okulov, V. (Intern), Walther, J. H. (Intern), Mayer, S. (Ekstern)
Pages: 133-143
Publication date: 2013
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Journal of Marine Science and Technology
Volume: 18
Issue number: 1
ISSN (Print): 0948-4280
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 1.043 SNIP 1.56 CiteScore 1.45
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.82 SNIP 1.71 CiteScore 1.28
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.04 SNIP 1.586 CiteScore 1.16
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.569 SNIP 1.788 CiteScore 1.13
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.582 SNIP 1.35 CiteScore 1.01
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 0.623 SNIP 1.457 CiteScore 1.03
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 0.785 SNIP 1.454
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 0.853 SNIP 1.844
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 0.465 SNIP 0.839
Scopus rating (2007): SJR 0.944 SNIP 1.867
Web of Science (2007): Indexed yes
Scopus rating (2006): SJR 0.693 SNIP 1.342
Scopus rating (2005): SJR 0.425 SNIP 1.834
Planar laser-induced fluorescence fuel imaging during gas-turbine relight

This experimental study investigates the influence of fuel distribution on ignition outcome during high-altitude relight of a gas turbine. Planar laser-induced fluorescence is used to image fuel inside a lean direct-injection combustor under realistic conditions. A novel apparatus is developed to permit planar laser-induced fluorescence imaging, in which large quantities of poorly atomized fuel impinges on the internal surfaces of the combustor. Results reveal high variability in atomization quality. In the absence of flame, small droplets are confined to areas of recirculating flow, whereas large droplets impact on the walls. All fuel is introduced through a pilot air-blast atomizer close to the injector centerline. However, comparatively little fuel is apparent near the igniter tip because the outside swirlers of the fuel injector create a fast-moving stream of fuel-free air that flows directly below the upper combustor wall. The droplet size and fuel concentration in the main recirculation zone do not differ radically at test conditions with markedly different fuel-to-air ratios, suggesting that turbulent straining is a more important factor than equivalence ratio in the failure of ignition when the airflow rate is high. In the presence of flame, medium-sized burning droplets are observed close to the injector centerline. Flame interference resulting from fluorescence of polycyclic aromatic hydrocarbons is apparent, but small, suggesting that kerosene planar laser-induced fluorescence is a useful tool for the analysis of all stages of altitude relight. Copyright © 2013 by R. W. Read, J. W. Rogerson, and S. Hochgreb.
RANS-based simulation of turbulent wave boundary layer and sheet-flow sediment transport processes

A numerical model coupling the horizontal component of the incompressible Reynolds-averaged Navier–Stokes (RANS) equations with two-equation $k-\omega$ turbulence closure is presented and used to simulate a variety of turbulent wave boundary layer processes. The hydrodynamic model is additionally coupled with bed and suspended load descriptions, the latter based on an unsteady turbulent-diffusion equation, for simulation of sheet-flow sediment transport processes. In addition to standard features common within such RANS-based approaches, the present model includes: (1) hindered settling velocities at high suspended sediment concentrations, (2) turbulence suppression due to density gradients in the water–sand mixture, (3) boundary layer streaming due to convective terms, and (4) converging–diverging effects due to a sloping bed. The present model therefore provides a framework for simultaneous inclusion of a number of local factors important within cross-shore wave boundary layer and sediment transport dynamics. The hydrodynamic model is validated for both hydraulically smooth and rough conditions, based on wave friction factor diagrams and boundary layer streaming profiles, with the results in excellent agreement with experimental and/or previous numerical work. The sediment transport model is likewise validated against oscillatory tunnel experiments involving both velocity-skewed and acceleration-skewed flows, as well as against measurements beneath real progressive waves. Model capabilities are exploited to investigate the importance of boundary layer streaming effects on sediment transport in selected velocity-skewed conditions. For the medium sand grain conditions considered, the model results suggest that streaming effects can enhance onshore sediment transport rates by as much as a factor of two. Moreover, for fine sand conditions streaming (and related convective) effects are demonstrated to potentially reverse the direction of net transport (i.e. from offshore to onshore) relative to predictions of oscillatory tunnel conditions. The developed model is implemented within the popular Matlab environment, and hence may be attractive for both research and educational purposes.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Wind Energy, Fluid Mechanics, Technical University of Denmark
Authors: Fuhrman, D. R. (Intern), Schløer, S. (Intern), Sterner, J. (Ekstern)
Pages: 151-166
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Coastal Engineering
Real-Time Simulation of Ship-Structure and Ship-Ship Interaction
This paper gives the status of the development of a ship-hydrodynamic model for real-time ship-wave calculation and ship-structure and ship-ship interaction in a full mission marine simulator. The hydrodynamic model is based on potential flow theory, linear or non-linear free surface boundary condition and higher-order accurate numerical approximations. The equations presented facilitate both Neumann-Kelvin and double-body linearizations. The body boundary condition on the ship hull is approximated by a static and dynamic moving pressure distribution. The pressure distribution method is used, because it is simple, easy to implement and computationally efficient. Multiple many-core graphical processing units (GPUs) are used for parallel execution and the model is implemented using a combination of C/C++, CUDA and MPI. Two ship hydrodynamic cases are presented: Kriso Container Carrier at steady forward speed and lock entrance of a TEU 12.000 Container Carrier. These calculations reveal that the pressure distribution model is a too simple approximation of the body boundary condition and that it has the limitations of a flat-ship approximation. It is necessary to investigate more accurate approximations of the body boundary condition, which does not compromise the overall computational efficiency.

Residual liquefaction of seabed under standing waves
This paper presents the results of an experimental study of the seabed liquefaction beneath standing waves. Silt (with $d_{50}=0.070\text{mm}$) was used in the experiments. Two kinds of measurements were carried out: pore water pressure measurements and water surface elevation measurements. These measurements were synchronized with video recording of the liquefaction process from the side. The ranges of the various quantities in the experiments were wave height $H=5.9-12.0\text{ cm}$, wave period $T=1.09\text{s}$, and water depth $h=30\text{ cm}$. The experiments show that the seabed liquefaction under standing waves, although qualitatively similar, show features different from that caused by progressive waves. The pore water pressure builds up (or accumulated) in the areas around the node and subsequently spreads out toward the antinodes. The experimental results imply that this transport is caused by a diffusion mechanism with a diffusion coefficient equal to the coefficient of consolidation. The experiments further show that the number of waves to cause liquefaction at the nodal section appears to be equal to that experienced in progressive waves for the same wave height. © 2013 American Society of Civil Engineers.
This study was partially funded by the Danish Council for Strategic Research through the research program "Seabed and Wind Farm Interaction" and the Seventh Framework Program of EU through the project "Innovative Multi-purpose offshore platforms: planning, design and operation, MERMAID."

Source: dtu
Source-ID: n:oai:DTIC-ART:compendex/424172610::33694
Publication: Research - peer-review › Journal article – Annual report year: 2013

Second-order theory for coupling 2D numerical and physical wave tanks: Derivation, evaluation and experimental validation

A full second-order theory for coupling numerical and physical wave tanks is presented. The ad hoc unified wave generation approach developed by Zhang et al. [Zhang, H., Schäffer, H.A., Jakobsen, K.P., 2007. Deterministic combination of numerical and physical coastal wave models. Coast. Eng. 54, 171–186] is extended to include the second-order dispersive correction. The new formulation is presented in a unified form that includes both progressive and evanescent modes and covers wavemaker configurations of the piston- and flap-type. The second order paddle stroke correction allows for improved nonlinear wave generation in the physical wave tank based on target numerical solutions. The performance and efficiency of the new model is first evaluated theoretically based on second order Stokes waves. Due to the complexity of the problem, the proposed method has been truncated at 2D and the treatment of regular waves, and the re-reflection control on the wave paddle is also not included. In order to validate the solution methodology further, a series of nonlinear, periodic waves based on stream function theory are generated in a physical wave tank using a piston-type wavemaker. These experiments show that the new second-order coupling theory provides an improvement in the quality of nonlinear wave generation when compared to existing techniques.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Dalian University of Technology
Authors: Yang, Z. (Ekstern), Liu, S. (Ekstern), Bingham, H. B. (Intern), Li, J. (Ekstern)
Pages: 37-51
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Coastal Engineering
Volume: 71
ISSN (Print): 0378-3839
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 3.44 SJR 1.98 SNIP 2.252
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.925 SNIP 2.097 CiteScore 2.9
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.785 SNIP 2.123 CiteScore 2.55
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.727 SNIP 2.264 CiteScore 2.58
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.912 SNIP 2.226 CiteScore 2.21
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 1.616 SNIP 2.502 CiteScore 2.43
Simulation of the Initial 3-D Instability of an Impacting Drop Vortex Ring

Computational vortex particle method simulations of a perturbed vortex ring are performed to recreate and understand the instability seen in impacting water drop experiments. Three fundamentally different initial vorticity distributions are used to attempt to trigger a Widnall instability, a Rayleigh centrifugal instability, or a vortex breakdown-type instability. Simulations which simply have a perturbed solitary ring result in an instability similar to that seen experimentally. Waviness of the core which would be expected from a Widnall instability is not visible. Adding an opposite-signed secondary vortex ring or an image vortex ring to the initial conditions, to trigger a Rayleigh or breakdown respectively, does not appear to significantly change the instability from what is seen with a solitary ring. This suggests that a Rayleigh or vortex breakdown-type instability are not likely at work, though tests are not conclusive. Perhaps the opposite-signed secondary vortex was not strong enough or placed appropriately. Elliptical streamlines, as expected, are visible in the core of the solitary ring at early times. Support from the Canadian Natural Sciences and Engineering Research Council grant 41747 is gratefully acknowledged.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Alberta
Authors: Sigurdson, L. (Ekstern), Wiwchar, J. (Ekstern), Walther, J. H. (Intern)
Publication date: 2013
Sinking failure of scour protection at wind turbine foundation

This paper summarises the results of an experimental study on scour protection around offshore wind turbine foundations, with special emphasis on the sinking failure of the scour protection work in Horns Rev 1 offshore wind farm (Denmark). The paper reviews previous results obtained by the author (AN), and is organised as follows. Section 2 addresses flow around a pile with a scour protection. Section 3 looks at the initiation of sand motion beneath scour protection. Section 4 discusses sediment motion beneath scour protection and resulting sinking. Section 5 investigates the Horns Rev 1 case. A brief account is also given of filter criteria and their application to the Horns Rev 1 case, whereby the present results and the filter criteria results are linked.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DHI Denmark
Authors: Sumer, B. M. (Intern), Nielsen, A. W. (Ekstern)
Pages: 170-188
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Institution of Civil Engineers. Proceedings. Energy
Volume: 166
Issue number: EN4
ISSN (Print): 1751-4223
Ratings:
Web of Science (2018): Indexed yes
Web of Science (2017): Indexed Yes
Scopus rating (2016): SJR 0.27 SNIP 0.734 CiteScore 0.68
Scopus rating (2015): SJR 0.452 SNIP 0.903 CiteScore 0.82
Scopus rating (2014): SJR 0.244 SNIP 0.84 CiteScore 0.45
Scopus rating (2013): SJR 0.39 SNIP 0.922 CiteScore 0.52
ISI indexed (2013): ISI indexed no
Scopus rating (2012): SJR 0.547 SNIP 0.843 CiteScore 0.72
ISI indexed (2012): ISI indexed no
Scopus rating (2011): SJR 0.472 SNIP 1.096 CiteScore 0.76
ISI indexed (2011): ISI indexed no
Scopus rating (2010): SJR 0.294 SNIP 1.115
Scopus rating (2009): SJR 0.132 SNIP 0.1
Original language: English
Coastal engineering, Granular materials, Offshore engineering
DOIs:
10.1680/ener.12.00006
Source: dtu
Source-ID: u::9759
Publication: Research - peer-review › Journal article – Annual report year: 2013
The sinking of initially buried irregular blocks into the seabed under wave-induced liquefaction was investigated by experimental methods. Pore-water pressure in the soil, water surface elevation time series and block displacements were measured. Results indicated that initiation of sinking coincides with the instant at which the accumulated pore-water pressure at the bottom level of the block reaches the initial mean normal effective stress. The drag forces and drag coefficients on steadily sinking irregular shaped blocks, as well as spherical and cubical ones, were calculated from the obtained data and compared with the available data in the literature. The results show that the shape of sinking block is of minor importance as far as the kinematics and dynamics of the sinking block is concerned. The conditions at which the sinking terminates are discussed in the light of experiments. Using the approach presented here, the ultimate sinking depths are calculated for the tested cases and compared with the experimental results. The calculated and measured values showed a reasonable agreement when compared. Finally a summary and remarks are presented to calculate the ultimate sinking depth of irregular shaped blocks for practical applications. © 2013 Elsevier B.V.
SPIV investigations of correlation between streamwise vorticity and velocity in the wake of a vortex generator in a boundary layer

The current work describes the experimental parametric study of streamwise vortices generated in a boundary layer by a rectangular vane (commonly named vortex generator) mounted perpendicularly to the wall and at an angle to the oncoming flow. Stereoscopic Particle Image Velocimetry measurements have been conducted in cross-planes to obtain a full picture of the wake for each measured case. It is observed that this seemingly simple configuration produces a complicated vortex system consisting of 4 vortices: the tip vortex, a horseshoe vortex system consisting of two sleeves and a secondary vortex generated by the interaction between the tip vortex and the wall. Depending on the height and angle of the vane, some structures are enhanced and others are suppressed. Comparing the corresponding vorticity and velocity fields, a strong correlation between the two is found. Stream-wise vorticity induced velocity with the observed strong correlation found is not at all evident. This has previously been shown to be true for the primary (tip) vortex, but not for the remaining secondary structures.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Wind Energy, Fluid Mechanics
Authors: Velte, C. M. (Intern), Okulov, V. (Intern), Hansen, M. O. L. (Intern)
Number of pages: 6
Publication date: 2013
Steep wave loads from irregular waves on an offshore wind turbine foundation: Computation and experiment

Two-dimensional irregular waves on a sloping bed and their impact on a bottom mounted circular cylinder is modeled by three different numerical methods and the results are validated against laboratory experiments. We here consider the performance of a linear-, a fully nonlinear potential flow solver and a fully nonlinear Navier-Stokes/VOF solver. The validation is carried out in terms of both the free surface elevation and the inline force. Special attention is paid to the ultimate load in case of a single wave event and the general ability of the numerical models to capture the higher harmonic forcing. The test case is representative for monopole foundations at intermediate water depths. The potential flow computations are carried out in a two-dimensional vertical plane and the inline force on the cylinder is evaluated by the Morison equation. The Navier-Stokes/VOF computations are carried out in three-dimensions and the force is obtained by spatial pressure integration over the wetted area of the cylinder. In terms of both the free surface elevation and inline force, the linear potential flow model is shown to be of limited accuracy and large deviations are generally seen when compared to the experimental measurements. The fully nonlinear Navier-Stokes/VOF computations are accurately predicting both the free surface elevation and the inline force. However, the computational cost is high relative to the potential flow solvers. Despite the fact that the nonlinear potential flow model is carried out in two-dimensions it is shown to perform just as good as the three-dimensional Navier-Stokes/VOF solver. This is observed for both the free surface elevation and the inline force, where both the ultimate load and the higher harmonic forces are accurately predicted. This shows that for moderately steep irregular waves a Morison equation combined with a fully nonlinear two-dimensional potential flow solver can be a good approximation.

General information
State: Published
Organisations: Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering, Department of Wind Energy, Fluid Mechanics
Authors: Paulsen, B. T. (Intern), Bingham, H. B. (Intern), Bredmose, H. (Intern), Schlaer, S. (Intern)
Number of pages: 10
Publication date: 2013

Study on a method for estimating fuel consumption in a seaway

On-board measurement of fuel consumption of a ship has been carried out in a relatively severe sea condition. In the full scale experiment, the ship traveled on several courses to investigate the change of fuel consumption relative to the encounter wave angle. The result shows that the wave direction has a great influence on the main engine horse power and fuel consumption, and also shows a possibility of fuel efficiency prediction. In order to develop an eco-friendly navigation support system, results of Bayesian wave estimation are applied to fuel efficiency prediction. The Bayesian method does not require wave measurements but needs only ship motion data as input and the method is suitable for on-site wave estimation.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Tokyo University of Marine Science and Technology
Authors: Iseki, T. (Ekstern), Nielsen, U. D. (Intern)
Number of pages: 6
Publication date: 2013
**Swirling flow in a two-stroke marine diesel engine**

Computational fluid dynamic simulations are performed for the turbulent swirling flow in a scale model of a low-speed two-stroke diesel engine with a moving piston. The purpose of the work is to investigate the accuracy of different turbulence models including two-equation Reynolds-Averaged Navier-Stokes models and large eddy simulations. The numerical model represents the full three-dimensional geometry and the piston motion is modeled by compressing cells in the axial direction. The CFD predictions are compared to experimental results and a reasonable agreement is found.

**The second-order decomposition model of nonlinear irregular waves**

A new method to decompose the nonlinear irregular waves is proposed. The second-order potential flow theory is employed to construct the relation of the second-order items solution by deriving the transfer function between the first- and the second-order components. Target waves are decomposed into the first- and the second-order super-harmonic as well as the second-order sub-harmonic components by transferring them into an identical Fourier frequency-space and using a Newton-Raphson iteration method. In order to evaluate the present model, a variety of monochromatic waves and the second-order nonlinear irregular waves over a broad range of frequencies have been analyzed, and the effects on wave nonlinearity are analyzed. The experimental results show that the present method is reasonably effective for the wave decomposition.
The Wave Loads project

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Aeroelastic Design, DHI Denmark
Authors: Bredmose, H. (Intern), Mariegaard, J. (Ekstern), Paulsen, B. T. (Intern), Jensen, B. (Intern), Schløer, S. (Intern), Larsen, T. J. (Intern), Kim, T. (Intern), Hansen, A. M. (Intern)
Number of pages: 120
Publication date: 2013

Publication information
Publisher: DTU Wind Energy
ISBN (Electronic): 978-87-92896-77-3
Original language: English
Series: DTU Wind Energy E
Number: 0045
Main Research Area: Technical/natural sciences
DTUWind Energy E–0045, DTUWind Energy E–45
Electronic versions:
DTUWind_Energy_E_0045.pdf
Publication: Research › Report – Annual report year: 2014

Towards real time simulation of ship-ship interaction - Part II: double body flow linearization and GPU implementation

General information
State: Published
Organisations: Department of Applied Mathematics and Computer Science, Scientific Computing, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, FORCE Technology
Authors: Lindberg, O. (Intern), Glimberg, S. L. (Intern), Bingham, H. B. (Intern), Engsig-Karup, A. P. (Intern), Schjeldahl, P. J. (Ekstern)
Number of pages: 4
Pages: 125-128
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the 28th International Workshop on Water Waves and Floating Bodies
Main Research Area: Technical/natural sciences
Conference: 28th International Workshop on Water Waves and Floating Bodies (IWWWFB 2013), Marseille, France, 07/04/2013 - 07/04/2013
Electronic versions:
iwwwfb28_32.pdf
Publication: Research › Article in proceedings – Annual report year: 2013

Transmission of wave energy through an offshore wind turbine farm
The transmission of wave energy passing an offshore wind farm is studied. Three effects that can change the wave field are analysed, which is the A) energy dissipation due to drag resistance, B) wave reflection/diffraction from structures, and C) the effect of a modified wind field inside and on the lee side of the wind farm. The drag dissipation, A), is quantified by a quadratic resistance law. The effect of B) is parameterised based on 1st order potential theory. A method to find the amount of reflected and transmitted wave energy is developed based on the panel method WAMIT™ and a radiation condition at infinity. From airborne and Satellite SAR (Synthetic Aperture Radar) a model has been derived for the change of the water surface friction C) inside and on the lee side of the offshore wind farm. The effects have been implemented in a spectral wind wave model, MIKE21 SW, and a parametric study to compare the 3 different processes has been carried out. The method to study reflection/diffraction can be used for any type of offshore structure, vessel or a number of structures, as long as the assumptions for the use of potential wave theory are valid, and the effect of the modified wind...
field on the water surface friction is known.

© 2013 Elsevier B.V. All rights reserved.
Turbulent swirling flow in a model of a uniflow-scavenged two-stroke engine

The turbulent and swirling flow of a uniflow-scavenged two-stroke engine cylinder is investigated using a scale model with a static geometry and a transparent cylinder. The swirl is generated by 30 equally spaced ports with angles of 0°, 10°, 20°, and 30°. A detailed characterization of the flow field is performed using stereoscopic particle image velocimetry. Mean fields are calculated using both a fixed coordinate system and a coordinate system based on the instantaneous flow topology. Time-resolved measurements of axial velocity are performed with laser Doppler anemometry, and power spectra are calculated in order to determine vortex core precession frequencies. The results show a very different flow dynamics for cases with weak and strong swirl. In the strongly swirling cases, a vortex breakdown is observed. Downstream of the breakdown, the vortex becomes highly concentrated and the vortex core precesses around the exhaust valve, resulting in an axial suction effect at the vortex center. Mean fields based on the instantaneous flow topology are shown to be more representative than mean fields based on a fixed coordinate system in cases with significant variations in the swirl center location.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, MAN Diesel & Turbo SE
Authors: Ingvorsen, K. M. (Intern), Meyer, K. E. (Intern), Walther, J. H. (Intern), Mayer, S. (Ekstern)
Number of pages: 17
Publication date: 2013
Main Research Area: Technical/natural sciences

Publication information
Journal: Experiments in Fluids
Volume: 54
Issue number: 3 art. no. 1494
ISSN (Print): 0723-4864
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 0.994 SNIP 1.324 CiteScore 2.18
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.193 SNIP 1.592 CiteScore 2.04
Wind loads on post-panamax container ship
An investigation of the influence of the container configuration on the deck of a 9000+ TEU container ship on wind forces has been carried out through a series of wind tunnel tests with a 1:450 scale model. An analysis of earlier studies was used to select a series of appropriate loading conditions for the tests. The wind tunnel tests were carried out in the naturally existing boundary layer of the wind tunnel. The longitudinal and transverse forces and the yaw moment were measured and the measurements were corrected for the effects of boundary layer and blockage in the wind tunnel. The results are presented as two different types of non-dimensional coefficients. It was apparent, as expected, that the measured forces and moment depend on the container configuration on deck. The test results may provide a general idea of how the magnitude of the wind forces is affected by a given container configuration.
1st DeepWind 5 MW Baseline design

The first 5MW baseline design of the DeepWind concept is presented for a Darrieus type floating wind turbine system for water depths of more than 150 m. This design will be used as design reference to test the next technological improvements of sub-component level, being based as much as possible on existing technology. The iterative design process involves all sub-components and the potential constraints, and the most important dependencies are highlighted and the selected design presented. The blades are designed with constraints to minimize the gravitational loads and to be produced in a controlled pulltrusion process. The floating platform is a slender cylindrical structure (i.e. spar buoy) rotating along with the rotor. The stability of the platform is achieved by adding counter weight at the bottom of the structure. During operations, the rotor is tilted and acts as a gyro, describing an elliptical trajectory on the water plane. The generator is placed at the bottom of the platform and uses 5MW direct drive technology. The conceptual design is evaluated with numerical simulations in the time domain using the aero-elastic code HAWC2. In order to investigate the concept, a double-disc blade element momentum (BEM) code for VAWTs has been included in the numerical solver through a dll. The analysis of the design is carried out in two different steps: 1) to estimate natural frequencies of the platform in order to avoid major resonance problems, 2) to evaluate the baseline concept for certain load cases. A site has been chosen for the floating turbine off Norway as representative for external conditions. The structure is verified according to an ultimate strength analysis, including loads from wind, waves and currents. The stability of the platform is investigated, considering the displacements of the spar buoy and the maximum inclination angle, which is kept lower than 15 degrees.

General information
State: Published
Organisations: Department of Wind Energy, Test and Measurements, Aeroelastic Design, Department of Mechanical Engineering, Manufacturing Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Aalborg University, Marine Technology Centre
Authors: Schmidt Paulsen, U. (Intern), Vita, L. (Intern), Aagaard Madsen, H. (Intern), Hattel, J. H. (Intern), Ritchie, E. (Forskerdatabase), Leban, K. M. (Forskerdatabase), Berthelsen, P. A. (Ekstern), Carstensen, S. (Intern)
Pages: 27-35
Publication date: 2012
Conference: 9th Deep Sea Offshore Wind R&D Seminar, Trondheim, Norway, 19/01/2012 - 19/01/2012
Main Research Area: Technical/natural sciences

Publication information
Journal: Energy Procedia
Volume: 24
ISSN (Print): 1876-6102
Ratings:
BFI (2018): BFI-level 1
BFI (2017): BFI-level 1
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 1.16 SJR 0.467 SNIP 0.586 Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.365 SNIP 0.561 CiteScore 0.92
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.433 SNIP 0.81 CiteScore 1.09
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.425 SNIP 0.785 CiteScore 1.02
ISI indexed (2013): ISI indexed no
Web of Science (2013): Indexed yes
Scopus rating (2012): SJR 0.425 SNIP 0.563 CiteScore 1.08
ISI indexed (2012): ISI indexed no
Web of Science (2012): Indexed yes
Scopus rating (2011): SJR 0.918 SNIP 1.505 CiteScore 2.42
ISI indexed (2011): ISI indexed no
Accurate computation of wave loads on a bottom fixed circular cylinder
This abstract describes recent progress in the development of a fast and accurate tool for computations of wave-structure interactions of realistic sea states that include breaking waves.

The practical motivation is extreme wave loads on offshore wind turbine foundations, but the tool is applicable to a range of other problems.

The central idea is to drive an inner CFD model that resolves the flow around the structure with an outer wave model that is based on potential flow theory. By letting the potential flow solver describe the waves in the outer flow domain and the Navier-Stokes solver describe the flow in the inner domain a fast and accurate description of wave loads on offshore structures is obtained, even for breaking waves.

Engsig-Karup et. al [1] have recently developed a fully nonlinear potential flow solver (OceanWave3D) to represent propagation and development of fully nonlinear three-dimensional water waves up to the point of breaking.

The CFD solver is the open source CFD toolbox OpenFOAMR in combination with the newly developed waves2Foam utility, which in [5] has been successfully applied to calculations of free surface flows. The numerical solution is obtained by solving the incompressible Navier-Stokes equations in combination with a surface tracking scheme. The CFD solver has been thoroughly tested for stability and first order grid convergence has been shown for the propagation of stream function waves.

Here we present results for the magnitudes, of the third-harmonic forces on a vertical circular cylinder from steep waves. This partly serves as a validation and further brings insight into third-harmonic wave loads on cylinders which are relevant for ringing. Next, preliminary results for the coupled model are presented in terms of irregular waves propagation and the associated forces on a cylinder.

A comparative study of two fast nonlinear free-surface water wave models
This paper presents a comparison in terms of accuracy and efficiency between two fully nonlinear potential flow solvers for the solution of gravity wave propagation. One model is based on the high-order spectral (HOS) method, whereas the second model is the high-order finite difference model OceanWave3D. Although both models solve the nonlinear potential flow problem, they make use of two different approaches. The HOS model uses a modal expansion in the vertical direction to collapse the numerical solution to the two-dimensional horizontal plane. On the other hand, the finite difference model simply directly solves the three-dimensional problem. Both models have been well validated on standard test cases and shown to exhibit attractive convergence properties and an optimal scaling of the computational effort with increasing problem size. These two models are compared for solution of a typical problem: propagation of highly nonlinear periodic waves on a finite constant-depth domain. The HOS model is found to be more efficient than OceanWave3D with a difference dependent on the level of accuracy needed as well as the wave steepness. Also, the higher the order of the finite difference schemes used in OceanWave3D, the closer the results come to the HOS model.
Ocean-wave3D, Hydrodynamics, Water waves, High-order finite differences, High-order spectral, Numerical comparisons
A Comparison of Methods for Computing the Added Resistance

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, FORCE Technology
Authors: Jonquez, S. (Ekstern), Andersen, P. (Intern), Bingham, H. B. (Intern)
Pages: 106-119
Publication date: 2012
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Ship Research
Volume: 56
Issue number: 2
ISSN (Print): 0022-4502
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 1.6 SJR 1.196 SNIP 1.832
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.585 SNIP 1.553 CiteScore 1.12
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.004 SNIP 1.835 CiteScore 1.19
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.576 SNIP 1.648 CiteScore 1.18
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.805 SNIP 1.689 CiteScore 0.97
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.146 SNIP 1.79 CiteScore 1.05
ISI indexed (2011): ISI indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 0.478 SNIP 0.906
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 0.571 SNIP 1.01
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 0.782 SNIP 1.219
Scopus rating (2007): SJR 0.921 SNIP 1.718
Scopus rating (2006): SJR 0.69 SNIP 1.526
Scopus rating (2005): SJR 0.594 SNIP 2.312
Scopus rating (2004): SJR 1.17 SNIP 1.6
Web of Science (2004): Indexed yes
Scopus rating (2003): SJR 1.086 SNIP 1.899
Web of Science (2003): Indexed yes
A Coupled Finite Difference and Moving Least Squares Simulation of Violent Breaking Wave Impact

Two models for simulation of free surface flow are presented. The first model is a finite difference based potential flow model with non-linear kinematic and dynamic free surface boundary conditions. The second model is a weighted least squares based incompressible and inviscid flow model. A special feature of this model is a generalized finite point set method which is applied to the solution of the Poisson equation on an unstructured point distribution. The presented finite point set method is generalized to arbitrary order of approximation. The two models are applied to simulation of steep and overturning wave impacts on a vertical breakwater. Wave groups with five different wave heights are propagated from offshore to the vicinity of the breakwater, where the waves are steep, but still smooth and non-overturning. These waves are used as initial condition for the weighted least squares based incompressible and inviscid model and the wave impacts on the vertical breakwater are simulated in this model. The resulting maximum pressures and forces on the breakwater are relatively high when compared with other studies and this is due to the incompressible nature of the present model.

General information
State: Published
Organisations: Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering, Department of Informatics and Mathematical Modeling, Scientific Computing
Authors: Lindberg, O. (Intern), Bingham, H. B. (Intern), Engsig-Karup, A. P. (Intern)
Number of pages: 12
Publication date: 2012

Host publication information
Title of host publication: Proceedings of 31st International Conference on Ocean, Offshore and Arctic Engineering
Publisher: American Society of Mechanical Engineers
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: 31st ASME International Conference on Ocean, Offshore and Arctic Engineering, Rio de Janeiro, Brazil, 01/07/2012 - 01/07/2012
Source: dtu
Source-ID: u::6058
Publication: Research - peer-review › Article in proceedings – Annual report year: 2012

A high order solver for the unbounded Poisson equation

This work improves upon Hockney and Eastwood's Fourier-based algorithm for the unbounded Poisson equation to formally achieve arbitrary high order of convergence without any additional computational cost. We assess the methodology on the kinematic relations between the velocity and vorticity fields.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Université Catholique de Louvain
Authors: Hejlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Chatelain, P. (Ekstern), Walther, J. H. (Intern)
Number of pages: 4
Publication date: 2012

Host publication information
Title of host publication: Proceedings of the 25th Nordic Seminar on Computational Mechanics
Publisher: Lund University
Editors: Persson, K., Revstedt, J., Sandberg, G., Wallin, M.
Main Research Area: Technical/natural sciences
A high order solver for the unbounded Poisson equation with specific application to the equations of fluid kinematics

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Universite Catholique de Louvain
Authors: Hejlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Walther, J. H. (Intern), Chatelain, P. (Ekstern)
Publication date: 2012
Conference: 65th Annual Meeting of the APS Division of Fluid Dynamics, San Diego, CA, United States, 18/11/2012 - 18/11/2012
Main Research Area: Technical/natural sciences

A High-Order WENO Finite Difference Water Wave Model for Interactive Ship-Wave Simulation

General information
State: Published
Organisations: Department of Informatics and Mathematical Modeling, Scientific Computing, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Engsig-Karup, A. P. (Intern), Lindberg, O. (Intern), Glimberg, S. L. (Intern), Dammann, B. (Intern), Bingham, H. B. (Intern), Madsen, P. A. (Intern)
Number of pages: 2
Publication date: 2012
Main Research Area: Technical/natural sciences

Airflow characteristics in the breathing zone of a seated person using desk incorporated pair of confluent jets as personalized ventilation - effect of supply velocities

A workplace with desk, desk incorporated personalized ventilation (PV) and a dressed thermal manikin with realistic body and surface temperature distribution were set in a test room (4.70 m x 1.62 m x 2.6 m). 15 L/s were supplied from a ceiling diffuser to ventilate the room at 26 oC air temperature. The PV consisted of two plane jets placed beside each other (confluent jets) and along the front edge of the desk. The slots had dimensions: 0.06 m x 0.5 m (W x L). The manikin was seated upright with abdomen pressed against the front edge of the desk. The airflow supplied isothermally and upwards from the inner jet (closest to manikin) was the same, twice bigger or twice lower compared to that of the outer jet. The mean velocity field at the breathing zone was measured by Particle Image Velocimetry: a dual cavity laser (λ = 532 nm) and a CCD camera - 35 mm lenses. Glycerol droplets (seeding) were added to the total volume air supply. The maximum absolute mean velocity measured near the manikin’s mouth was 0.25 m/s, when the two confluent jets supplied 8 L/s each. Same velocity was measured when the inner jet was supplying 8 L/s and the outer 4 L/s. The opposite combination, i.e. outer jet 8 L/s and inner 4 L/s, resulted in lower velocity (0.13 m/s) compared to that of the free convection layer alone:
0.20 m/s. The increased velocity at the face allowed more clean air to be inhaled.

A Model for Prediction of Propulsion Power and Emissions – Tankers and Bulk Carriers
To get an idea of the reduction in propulsion power and associated emissions by varying the speed and other ship design
main parameters, a generic model for parameter studies of tankers and bulk carriers has been developed.
With only a few input parameters of which the maximum deadweight capacity is the primary input a proposal for the main
dimensions is made. Based on these dimensions and other ship particulars which are determined by the program the
necessary installed propulsion power can be calculated. By adjusting the vessel design, i.e. the suggested main
dimensions, and varying the speed it is possible to estimate the influence of the different parameters on the power
demand. The model is based on previously well-established power prediction methods which have been updated and
verified by model test results and full-scale data, meaning that the predictions are up to date according to modern ship
design standards.
The IHS Fairplay World Fleet Statistics for vessels built in the period 1990 – 2010 is used as a basis for the modeling of
the main dimensions.
The model can be used to calculate exhaust gas emissions, including emissions of carbon dioxide (CO2), from bulk
carriers and tankers. A calculation procedure for estimating the Energy Efficiency Design Index (EEDI) which is presently
being developed by the International Maritime Organization (IMO) is also included in the model. Different ship design
parameters have been varied to see the influence of these parameters on the EEDI. The paper will focus on the technical
and the design measures which can improve the environmental performance and will not take into account operational
measures.

An ALE Weighted Least Squares Method for Simulation of Violent Water Wave Impact

General information
State: Published
Organisations: Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering, Department
of Informatics and Mathematical Modeling, Scientific Computing
Authors: Lindberg, O. (Intern), Engsig-Karup, A. P. (Intern), Bingham, H. B. (Intern)
Number of pages: 2
Publication date: 2012
Event: Paper presented at International Conference on Spectral and High Order Methods (ICOSAHOM 2012), Gammarth,
Tunisia.
Main Research Area: Technical/natural sciences
Source: dtu
Source-ID: u::6060
Publication: Research - peer-review › Paper – Annual report year: 2012
A note on turbulent spots over a rough bed in wave boundary layers
This study is a continuation of the investigation of turbulent spots in wave boundary layers over a smooth wall reported by Carstensen et al. [J. Fluid Mech. 646, 169–206 (2010)]. The present paper summarises the results of an experimental investigation of turbulent spots in wave boundary layers over a rough wall. Two kinds of experiments were made: flow visualization tests and velocity profile measurements. The experiments show that turbulent spots as observed in the smooth-wall wave boundary layers are present irrespective of wall roughness conditions. As in the smooth-wall case the turbulent spots over the rough wall appear locally in time and space and grow in size as time progresses.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Carstensen, S. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern)
Number of pages: 13
Pages: Paper 115104
Publication date: 2012
Main Research Area: Technical/natural sciences

Publication information
Journal: Physics of Fluids
Volume: 24
ISSN (Print): 1070-6631
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.16 SJR 1.29 SNIP 1.291
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.366 SNIP 1.278
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.354 SNIP 1.348
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.42 SNIP 1.395
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.215 SNIP 1.356
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.445 SNIP 1.474
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
Scopus rating (2010): SJR 1.38 SNIP 1.388
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 1.64 SNIP 1.36
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 1.776 SNIP 1.362
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 1.72 SNIP 1.362
Web of Science (2007): Indexed yes
An overset grid approach to linear wave-structure interaction

A finite-difference based approach to wave-structure interaction is reported that employs the overset approach to grid generation. A two-dimensional code that utilizes the Overture C++ library has been developed to solve the linear radiation problem for a floating body of arbitrary form. This software implementation has been validated by performing time-domain simulations to evaluate the dynamic forces applied to a half-submerged cylinder and a rectangular barge in response to a prescribed motion. A Gaussian displacement is used to introduce a range of wave frequencies, thereby allowing the measurement of the body response over the frequency range of interest. The radiation addedmass and damping coefficients of both bodies have been evaluated and compared to exact analytical solutions. The numerical and analytical results show good agreement when the modes of excitation and response are the same. The cross-coupled results are in qualitative agreement, but show some quantitative variations that may be related to slight differences in the fluid domain geometry. For both the cylinder and the barge, the effects of bottom slope on the coefficients are found to be minimal.

Bibliographical note
© 2012 American Institute of Physics. This article may be downloaded for personal use only. Any other use requires prior permission of the author and the American Institute of Physics. The following article appeared in Phys. Fluids 24, 115104 (2012) and may be found at http://pof.aip.org/resource/1/phfle6/v24/i11/p115104_s1?ver=pdfcov

An overset grid approach to linear wave-structure interaction

A finite-difference based approach to wave-structure interaction is reported that employs the overset approach to grid generation. A two-dimensional code that utilizes the Overture C++ library has been developed to solve the linear radiation problem for a floating body of arbitrary form. This software implementation has been validated by performing time-domain simulations to evaluate the dynamic forces applied to a half-submerged cylinder and a rectangular barge in response to a prescribed motion. A Gaussian displacement is used to introduce a range of wave frequencies, thereby allowing the measurement of the body response over the frequency range of interest. The radiation addedmass and damping coefficients of both bodies have been evaluated and compared to exact analytical solutions. The numerical and analytical results show good agreement when the modes of excitation and response are the same. The cross-coupled results are in qualitative agreement, but show some quantitative variations that may be related to slight differences in the fluid domain geometry. For both the cylinder and the barge, the effects of bottom slope on the coefficients are found to be minimal.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Read, R. (Intern), Bingham, H. B. (Intern)
Number of pages: 10
Publication date: 2012

Host publication information
Title of host publication: Proceedings of the ASME 2012 31st International Conference on Ocean, Offshore and Arctic Engineering
Publisher: American Society of Mechanical Engineers
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: 31st ASME International Conference on Ocean, Offshore and Arctic Engineering, Rio de Janeiro, Brazil, 01/07/2012 - 01/07/2012
OMAE2012-83568
Source: dtu
Source-ID: u::7052
Publication: Research - peer-review › Article in proceedings – Annual report year: 2012
A numerical study of the aerodynamic admittance of bridge deck sections by a two-dimensional mesh-free vortex method

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, COWI Consultants A/S
Authors: Hejlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Larsen, A. (Ekstern), Walther, J. H. (Intern)
Publication date: 2012
Main Research Area: Technical/natural sciences
Electronic versions:
iutam2012.pdf
Source: dtu
Source-ID: u::5705
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2012

A wave generation toolbox for the open-source CFD library: OpenFoam
The open-source CFD library OpenFoam® contains a method for solving free surface Newtonian flows using the Reynolds averaged Navier–Stokes equations coupled with a volume of fluid method. In this paper, it is demonstrated how this has been extended with a generic wave generation and absorption method termed ‘wave relaxation zones’, on which a detailed account is given. The ability to use OpenFoam for the modelling of waves is demonstrated using two benchmark test cases, which show the ability to model wave propagation and wave breaking. Furthermore, the reflection coefficient from outlet relaxation zones is considered for a range of parameters. The toolbox is implemented in C++, and the flexibility in deriving new relaxation methods and implementing new wave theories along with other shapes of the relaxation zone is outlined. Subsequent to the publication of this paper, the toolbox has been made freely available through the OpenFoam-Extend Community.

General information
State: Published
Organisations: Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering
Authors: Jacobsen, N. G. (Intern), Fuhrman, D. R. (Intern), Fredsøe, J. (Intern)
Pages: 1073-1088
Publication date: 2012
Main Research Area: Technical/natural sciences

Publication information
Volume: 70
Issue number: 9
ISSN (Print): 0271-2091
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): SJR 1.398 SNIP 1.491 CiteScore 2.26
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.122 SNIP 1.346 CiteScore 1.69
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.017 SNIP 1.303 CiteScore 1.85
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.754 SNIP 1.374 CiteScore 1.66
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.954 SNIP 1.248 CiteScore 1.58
ISI indexed (2012): ISI indexed yes
Backfilling of scour holes around piles

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Environmental Engineering, Urban Water Engineering, Water Resources Engineering, University of Cantania
Authors: Sumer, B. M. (Intern), Petersen, T. U. (Intern), Locatelli, L. (Intern), Fredsøe, J. (Intern), Musumeci, R. E. (Ekstern), Foti, E. (Ekstern)
Publication date: 2012

Host publication information
Title of host publication: Proceedings of the 33rd International Conference on Coastal Engineering
Publisher: American Society of Civil Engineers
Main Research Area: Technical/natural sciences
Conference: ICCE 2012, Santander, Spain, 01/07/2012 - 01/07/2012
Electronic versions:
65.pdf
Source: dtu
Source-ID: u::5874
Publication: Research - peer-review › Conference abstract in proceedings – Annual report year: 2012
Blind estimation of a ship’s relative wave heading

This article proposes a method to estimate a ship's relative heading against the waves. The procedure relies purely on ship-board measurements of global responses such as motion components, accelerations and the bending moment amidships. There is no particular (mathematical) model connected to the estimate, and therefore it is called a 'blind estimate'. The approach is in this introductory study tested by analysing simulated data. The analysis reveals that it is possible to estimate a ship’s relative heading on the basis of shipboard measurements only.
Breaking Wave on a Slender Cylinder: Comparison of Experimental Data and Numerical Results

CFD models are promising in predicting non-linear wave loads on fixed and floating offshore structures. The NS3 model is described in this paper and it has been validated by means of model test such as wave run-up on monopiles in regular waves. The goal for the use of the NS3 model is to make a detailed investigation of the effect of 2D waves on the run-up height. The focused wave is designed by choosing the phases of the linear harmonic components such that they are in phase at a certain target location. The aim of this paper is the approach on the ability of NS3 model to reproduce these focused wave groups and compare with the results of the experimental tests carried out at Grossen WellenKanal (G WK).

CFD modelling of combined blast and contact cooling for whole fish

Dynamic selection of ship responses for estimation of on-site directional wave spectra

Knowledge of the wave environment in which a ship is operating is crucial for most on-board decision support systems. Previous research has shown that the directional wave spectrum can be estimated by the use of measured global ship responses and a set of transfer functions determined for the specific ship. The approach can either be based on parametric or Bayesian (non-parametric) modelling, and in both cases a set of three ship responses usually provides the best estimation. The optimal response combination of three responses at any time depends on the environmental conditions and the operation of the ship. Since measurements of more than three responses are usually available, a quick, dynamic selection procedure of the three signals best suited for the wave spectrum estimation procedure is essential. In the present paper the concept of a selection method based on a simple pre-estimate of the wave spectrum is suggested. The selection method needs to be robust for what reason a parameterised uni-directional, two-parameter wave spectrum is treated. The parameters included are the zero up-crossing period, the significant wave height and the main wave direction relative to the ship's heading. The procedure basically seeks to minimise the difference between a set of spectral moments derived from a measured response spectrum and the corresponding response spectrum calculated from the parameterised wave spectrum and the transfer function for any given response. Subsequently, the three responses with the best overall agreement are selected for the actual estimation of the directional wave spectrum. The transfer functions for the ship responses can be determined using different computational methods such as striptheory, 3D panel codes, closed form expressions or model tests. The uncertainty associated with transfer functions depends on the computational method used, relative heading, frequency and type of response. This uncertainty is conceptually taken into account in the selection procedure using the First Order Reliability Method (FORM). However, specific assessment of the uncertainties has not been carried out, but is subject to later studies.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Det Norske Veritas
Authors: Andersen, I. M. V. (Intern), Storhaug, G. (Ekstern)
Number of pages: 6
Publication date: 2012

Host publication information
Title of host publication: Proceedings of 31st International Conference on Ocean, Offshore and Arctic Engineering
Publisher: American Society of Mechanical Engineers
Chapter: OMAE2012-83206
Main Research Area: Technical/natural sciences
Conference: 31st ASME International Conference on Ocean, Offshore and Arctic Engineering, Rio de Janeiro, Brazil, 01/07/2012 - 01/07/2012
Source: dtu
Source-ID: u::7180
Publication: Research - peer-review › Article in proceedings – Annual report year: 2012
Educating Maritime Engineers for a Globalised Industry - Bridging the Gap Between Industry and Universities

In Denmark, the maritime engineering competences requested by the industry have changed in the past one to two decades. The typical naval architects do no longer find themselves working in the ship-building industry but rather in the industry of ship operators, consultancies, class societies, etc. This means that universities educating maritime engineers need to reflect the changes in the curricula for their maritime engineering students. Topics and issues regarding this matter have recently been addressed in a survey made in the Danish maritime industry. The survey concludes that the demand for maritime engineers in the industry is considerably larger than the output from the technical universities. Moreover, it sets forth a series of recommendations to the industry as well as to the universities to facilitate meeting the demand for maritime engineers in Denmark in the future. The recommendations are outlined together with work commenced at the Technical University of Denmark (DTU) to update the curricula for DTU’s maritime engineering students. Thus, DTU offers an education reflecting a large share of the recommendations in the curricula.

Efficient Pseudo-Spectral Model for Free Surface Nonlinear Water Waves

Efficient psudo-spectral model for nonlinear water waves
Environmental Performance Evaluation of Ro-Ro Passenger Ferry Transportation: Danish information paper for the work in the MEPC committee under The International Maritime Organisation, IMO in London, January 2012

With increasing focus on the environmental performance of different transport modes (for example trucks, trains, ships and aircraft) it is of utmost importance that the different transport modes are compared on an equal basis so that the environmental impact, defined as energy demand and/or emissions per transport unit, is related to the same unit for the different transport forms.

For Ro-Ro passenger ferries it can be difficult to find a suitable common transport unit, as they often transport a mix of cargo, such as passengers, passenger cars, trucks, lorries, busses and other rolling transport units. In this paper a method for determination of a common transport unit for Ro-Ro passenger ships will be described.

European Research in Marine Structures

An overview is presented of the results obtained in Europe by a network with a large number of research groups in the field of Marine Structures during a period of 6 years. The European Union has funded a project aimed at improving the collaboration among European research groups specialized in marine structures, which has led, among other results to a number of benchmark studies organized in 6 main topical areas, namely, Methods and Tools for Loads and Load Effects, Methods and Tools for Strength Assessment, Experimental Analysis of Structures, Materials and Fabrication of Structures, Methods and Tools for Structural Design and Optimization and Structural Reliability, Safety and Environmental Protection. This paper presents an overview of various studies performed, which helps identifying the level of consistency and robustness of different numeric tools used in this field.

European Research in Marine Structures

An overview is presented of the results obtained in Europe by a network with a large number of research groups in the field of Marine Structures during a period of 6 years. The European Union has funded a project aimed at improving the collaboration among European research groups specialized in marine structures, which has led, among other results to a number of benchmark studies organized in 6 main topical areas, namely, Methods and Tools for Loads and Load Effects, Methods and Tools for Strength Assessment, Experimental Analysis of Structures, Materials and Fabrication of Structures, Methods and Tools for Structural Design and Optimization and Structural Reliability, Safety and Environmental Protection. This paper presents an overview of various studies performed, which helps identifying the level of consistency and robustness of different numeric tools used in this field.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Department of Informatics and Mathematical Modeling, Scientific Computing
Authors: Christiansen, T. (Ekstern), Engsig-Karup, A. P. (Intern), Bingham, H. B. (Intern)
Publication date: 2012
Evaluation of nourishment schemes based on long-term morphological modeling

A recently developed long-term morphological modeling concept is applied to evaluate the impact of nourishment schemes. The concept combines detailed two-dimensional morphological models and simple one-line models for the coastline evolution and is particularly well suited for long-term simulation. This hybrid concept is here applied to study the decadal morphological evolution of several nourishment scenarios in Dunkirk, France. The morphological simulations successfully allowed identifying the impact of beach versus shoreface nourishment scenarios on the background morphological behavior of the study site. This study strongly indicates that the hybrid model may be used as an engineering tool to predict shoreline response following the implementation of a nourishment project.

General information
State: Published
Organisations: Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering, DHI Denmark, DHI, Grand Port Maritime de Dunkerque
Authors: Grunnet, N. (Forskerdatabase), Kristensen, S. E. (Intern), Drønen, N. (Ekstern), Deigaard, R. (Ekstern), Tessier, C. (Ekstern), Forain, N. (Ekstern)
Publication date: 2012

Host publication information
Title of host publication: Proceedings of the International Conference on Coastal Engineering 2012
Publisher: American Society of Civil Engineers
ISBN (Print): 9780989661119
Main Research Area: Technical/natural sciences
Conference: ICCE 2012, Santander, Spain, 01/07/2012 - 01/07/2012
Existing Design Trends for Tankers and Bulk Carriers - Design Changes for Improvement of the EEDI in the Future

To get an idea of the reduction in propulsion power and associated emissions by varying the speed and other ship design main parameters, a generic model for parameter studies has been developed. With only a few input parameters of which the maximum deadweight capacity is the primary one, a proposal for the main dimensions and the necessary installed power is calculated by the model. By adjusting the vessel design, i.e. the main parameters, and varying the speed it is possible to observe the influence of the different parameters on the power demand. The model can be used to calculate exhaust gas emissions from bulk carriers and tankers, including emissions of carbon dioxide (CO2). A calculation procedure for estimating the Energy Efficiency Design Index (EEDI) is also included in the model. The IHS Fairplay World Fleet Statistics for vessels built in the period 1990–2010 are used as a basis for the generic modelling. A comprehensive regression analysis has been carried out to find the formulas to be used as a basis for the model. Furthermore, it was found during the analysis that the design trend of bulk carriers and tankers has moved in a wrong direction seen from an energy saving point of view. The block coefficient has increased during the last twenty years while the length displacement ratio (L/displ.volume1/3) has decreased over the same period. These two design changes have resulted in an increased EEDI. This development must be changed in the coming years when the EEDI shall be reduced gradually, ending in a 30 per cent reduction in 2025. An overview of the historical development and the necessary design changes will be documented here, including a complete list of the formulas for the main dimensions found by the regression analysis.

Experimental study on the scour around a monopile in breaking waves

The scour process around monopiles caused by breaking waves is studied experimentally using regular waves. The use of regular waves is conservative, which made it possible to avoid scour phenomena caused by nonbreaking waves such as scour generation and backfilling. The waves were breaking on a flat sand section after shoaling on a mildly sloping ramp. Various monopiles were exposed to plunging breakers that were breaking at various distances from the pile. It was found that the scour was caused by turbulence generated by the breaking and was diverted toward the bottom by the pile. The maximum scour depth found was approximately 0.60D. This was smaller than the scour observed around piles exposed to current; however, in some cases it was an order of magnitude larger than the scour caused by nonbreaking waves. This is apparently especially true for larger piles.
Experimental Validation of a Mathematical Model for Seabed Liquefaction Under Waves

This paper summarizes the results of an experimental study directed towards the validation of a mathematical model for the buildup of pore water pressure and resulting liquefaction of marine soils under progressive waves. Experiments were conducted under controlled conditions with silt (d(50) = 0.070 mm) in a wave flume with a soil pit. Waves with wave heights in the range of 7.7-18 cm, 55-cm water depth and 1.6-s wave period enabled us to study both the liquefaction and no-liquefaction regime pore water pressure buildup. The experimental data were used to validate the model. A numerical
example is also included in the paper to demonstrate the implementation of the model for real-life scenarios.

**General information**
State: Published  
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering  
Authors: Sumer, B. M. (Intern), Kirca, Ö. (Intern), Fredsøe, J. (Intern)  
Pages: 133-141  
Publication date: 2012  
Main Research Area: Technical/natural sciences

**Publication information**
Volume: 22  
Issue number: 2  
ISSN (Print): 1053-5381  
Ratings:  
BFI (2018): BFI-level 1  
Web of Science (2018): Indexed yes  
BFI (2017): BFI-level 1  
Web of Science (2017): Indexed Yes  
BFI (2016): BFI-level 1  
Scopus rating (2016): SJR 0.502 SNIP 1.008 CiteScore 0.97  
Web of Science (2016): Indexed yes  
BFI (2015): BFI-level 1  
Scopus rating (2015): SJR 0.456 SNIP 0.886 CiteScore 0.73  
BFI (2014): BFI-level 1  
Scopus rating (2014): SJR 0.695 SNIP 1.281 CiteScore 0.98  
BFI (2013): BFI-level 1  
Scopus rating (2013): SJR 0.389 SNIP 0.704 CiteScore 0.61  
ISI indexed (2013): ISI indexed yes  
BFI (2012): BFI-level 1  
Scopus rating (2012): SJR 0.511 SNIP 1.002 CiteScore 0.76  
ISI indexed (2012): ISI indexed yes  
Web of Science (2012): Indexed yes  
BFI (2011): BFI-level 2  
Scopus rating (2011): SJR 0.617 SNIP 1.074 CiteScore 0.71  
ISI indexed (2011): ISI indexed yes  
BFI (2010): BFI-level 1  
Scopus rating (2010): SJR 0.787 SNIP 1.546  
BFI (2009): BFI-level 1  
Scopus rating (2009): SJR 0.387 SNIP 1.23  
BFI (2008): BFI-level 1  
Scopus rating (2008): SJR 0.293 SNIP 0.713  
Web of Science (2008): Indexed yes  
Scopus rating (2007): SJR 0.392 SNIP 1.317  
Scopus rating (2006): SJR 0.457 SNIP 0.786  
Scopus rating (2005): SJR 0.815 SNIP 1.277  
Web of Science (2005): Indexed yes  
Scopus rating (2004): SJR 0.497 SNIP 0.752  
Web of Science (2004): Indexed yes  
Scopus rating (2003): SJR 0.704 SNIP 1.243  
Scopus rating (2002): SJR 1.152 SNIP 0.85  
Scopus rating (2001): SJR 0.731 SNIP 1.132  
Scopus rating (2000): SJR 1.036 SNIP 0.849  
Scopus rating (1999): SJR 0.296 SNIP 0.682  
Original language: English  
Engineering, Pore pressure, Water-waves, Soil, Approximation, Sediment
Flow diagnostics downstream of a tribladed rotor model

This paper presents results of a study of vortex wake structures and measurements of instantaneous 3D velocity fields downstream of a three-bladed turbine model. Two operation modes of flow around the rotor with different tip speed ratios were tested. Initially the wake structures were visualized and subsequently quantitative data were recorded through velocity field restoration from particle tracks using a stereo PIV system. The study supplied flow diagnostics and recovered the instantaneous 3D velocity fields in the longitudinal cross section behind a tribladed rotor at different values of tip speed ratio. This set of data provided a basis for testing and validating assumptions and hypothesis regarding classical theories of rotors.

General information
State: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Russian Academy of Sciences, Novosibirsk State University
Authors: Naumov, I. V. (Ekstern), Rahmanov, V. V. (Ekstern), Okulov, V. (Intern), Velte, C. M. (Intern), Meyer, K. E. (Intern), Mikkelsen, R. F. (Intern)
Pages: 171-181
Publication date: 2012
Main Research Area: Technical/natural sciences

Publication information
Journal: Thermophysics and Aeromechanics
Volume: 19
Issue number: 2
ISSN (Print): 0869-8643

Bibliographical note
The research was partially supported in the framework of projects of the Ministry of Science and Education RF (Project No. 2.1.2/1270, SC No. 14.740.11.0144 and SC No. 11.519.11.6022), RFBR Grants (No. 10-08-01093) and Danish Energy Technology Development and Demonstration Program (No. 64011-0094)

Source: dtu
Source-ID: n:oai:DTIC-ART:isr/366090069::17368
Publication: Research - peer-review › Journal article – Annual report year: 2012
High-order Finite Difference Solution of Euler Equations for Nonlinear Water Waves

The incompressible Euler equations are solved with a free surface, the position of which is captured by applying an Eulerian kinematic boundary condition. The solution strategy follows that of [1, 2], applying a coordinate-transformation to obtain a time-constant spatial computational domain which is discretized using arbitrary-order finite difference schemes on a staggered grid with one optional stretching in each coordinate direction. The momentum equations and kinematic free surface condition are integrated in time using the classic fourth-order Runge-Kutta scheme. Mass conservation is satisfied implicitly, at the end of each time stage, by constructing the pressure from a discrete Poisson equation, derived from the discrete continuity and momentum equations and taking the time-dependent physical domain into account. An efficient preconditioned Defect Correction (DC) solution of the discrete Poisson equation for the pressure is presented, in which the preconditioning step is based on an order-multigrid formulation with a direct solution on the lowest order-level. This ensures fast convergence of the DC method with a computational effort which scales linearly with the problem size. Results obtained with a two-dimensional implementation of the model are compared with highly accurate stream function solutions to the nonlinear wave problem, which show the approximately expected convergence rates and a clear advantage of using high-order finite difference schemes in combination with the Euler equations.

High Order Poisson Solver for Unbounded Flows

General information
State: Published
Organisations: Department of Informatics and Mathematical Modeling, Scientific Computing, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Christiansen, T. R. B. (Intern), Bingham, H. B. (Intern), Engsig-Karup, A. P. (Intern)
Number of pages: 10
Publication date: 2012
Main Research Area: Technical/natural sciences
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2012
Implementation of the Spalart-Allmaras turbulence model in the two-dimensional vortex-in-cell method

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, COWI Consultants A/S
Authors: Hejlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Larsen, A. (Ekstern), Walther, J. H. (Intern)
Publication date: 2012

Host publication information
Title of host publication: Proceedings of the 6th European Congress on Computational Methods in Applied Sciences and Engineering
Publisher: Vienna University of Technology
Editors: Eberhardsteiner, J., Böhm, H., Rammerstorfer, F.
Main Research Area: Technical/natural sciences
Electronic versions:
eccomas2012.pdf
Source: dtu
Source-ID: u::5691
Publication: Research › Article in proceedings – Annual report year: 2012

Influence of piston displacement on the scavenging and swirling flow in two-stroke diesel engines
We study the effect of piston motion on the in-cylinder swirling flow in a low speed, large two-stroke marine diesel engine. The work involves experimental, and numerical simulation using OpenFOAM platform, Large Eddy Simulation was used with three different models, One equation Eddy, Dynamic One equation Eddy, and Ta Phouc Loc model, to study the transient phenomena of the flow. The results are conducted at six cross sectional planes along the axis of the cylinder and with the piston displaced at four fixed piston positions covering the air intake ports by 0%, 25%, 50%, and 75% respectively, for the fully opened case LES model with 8/12 million mesh points were used. We find that the flow inside the cylinder changes as the ports are closing, from a Rankine/Burger vortex profile to a solid body rotation while the axial velocity profiles change from a wake-like to a jet-like profile.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Thermal Energy
Authors: Obeidat, A. (Intern), Haider, S. (Intern), Ingvorsen, K. M. (Intern), Meyer, K. E. (Intern), Walther, J. H. (Intern)
Number of pages: 5
Publication date: 2012
Main Research Area: Technical/natural sciences
Diesel engine, Swirl, Scavenging, LES simulation
Electronic versions:
Obeidat:2010.pdf
Source: dtu
Source-ID: u::5268
Publication: Research › Conference abstract for conference – Annual report year: 2012

Interaction between Seabed Soil and Offshore Wind Turbine Foundations
Today, monopiles are widely used as foundation to support offshore wind turbines (OWT) in shallow waters. The stiffness of monopiles is one of the important design aspects. Field observations show that some monopiles, already installed in the field, behaves more stiff than predicted by the current design recommendations. The present study addresses the pile/seabed interaction problem, related to the stiffness of the monopile by means of a numerical model and experimental investigations. The numerical model is a 3D model. COMSOL Multiphysics, a finiteelement software, is used to calculate the soil response. The model is based on the Biot consolidation theory which involves a set of four equations, the first three equations describing the equilibrium conditions for a stress field, and the fourth one, the so-called storage equation, describing the conservation of mass of pore water with the seepage velocity given by Darcy’s law (Sumer and Fredsøe, 2002, chap. 10). The constitutive equation for the soil considered in the model is the familiar stress-strain relationship for linear poro-elastic soils. The so-called no-slip boundary condition is adopted on the surface of the rocking pile. The numerical model is validated against the laboratory experiments. The experimental setup includes a container (a circular tank with a diameter of 2 m and a height of 2.5 m), and a stainless steel model pile (with a diameter of 20 cm). Coarse sand ($d_{50} = 0.64$ mm) is used in the experiments. Pore-water pressures, pile displacements and forces on the pile are
measured in the experiments. The pore-water pressure is measured at 12 points over a mesh extending 0.75 m in the vertical and 0.10 m in the radial direction (the measurement points closest to the pile being at 2 cm from the edge of the pile), using Honeywell pressure transducers. The pile displacement is measured, using a conventional potentiometer, while the force is measured with a tension/compression S-Beam load cell. The model, validated and tested, is used to calculate the soil response for a set of conditions, normally encountered in the field. The results are presented in terms of non-dimensional $p-y$ curves, obtained from the numerical simulations. A parametric study is undertaken to observe the influence of various parameters on the latter. The parametric study shows that, for a given displacement, $y$, the soil resistance $p$, increases with increasing $S$, a non-dimensional parameter responsible for generation and further dissipation of the pore-water pressure. The parametric study also shows that, again for a given $y$, the soil resistance increases with decreasing bending stiffness of the pile, expressed in terms of a non-dimensional quantity $s$. Finally, the soil resistance increases, when the non-dimensional foundation depth decreases.

**General information**

State: Published
Organisations: Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering
Authors: Hansen, N. M. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern)
Number of pages: 154
Publication date: 2012

**Publication information**

Place of publication: Kgs.Lyngby
Publisher: DTU Mechanical Engineering
ISBN (Print): 978-87-90416-80-5
Original language: English

Series: DCAMM Special Report
Number: S143
ISSN: 0903-1685
Main Research Area: Technical/natural sciences
Electronic versions:
S143_Nilas_Mandrup_Hansen.pdf
Publication: Research › Ph.D. thesis – Annual report year: 2012

**LES of turbulent jet in cross-flow: Part 1 – A numerical validation study**

The paper presents results of a LES based numerical simulation of the turbulent jet-in-cross-flow (JICF) flowfield, with Reynolds number based on cross-flow velocity and jet diameter $Re=2400$ and jet-to-cross-flow velocity ratio of $R=3.3$. The JICF flow case has been investigated in great detail, involving conduction of two independent precursor simulations, prior to the main JICF simulation, as the considered case has turbulent inflow conditions on both jet and cross-stream side. The LES results are directly compared to pointwise Laser Doppler Anemometry (LDA) measurements, showing a very good agreement on the level of various statistical quantities in all flow regions but the immediate jet-to-cross-flow exhaustion zone. Several LES computations involving grids of up to 15million grid points have been conducted, showing no improvement in the agreement between numerical results and measurements, possibly indicating a LDA measurement problem in this particular region.

**General information**

State: Published
Organisations: Department of Wind Energy, Meteorology, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Cavar, D. (Intern), Meyer, K. E. (Intern)
Pages: 18-34
Publication date: 2012
Main Research Area: Technical/natural sciences

**Publication information**

Volume: 36
ISSN (Print): 0142-727X
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
The paper presents results of a Proper Orthogonal Decomposition (POD) investigation of the LES based numerical simulation of the jet-in-crossflow (JICF) flowfield with Reynolds number based on the cross flow velocity and jet diameter $Re=2400$ and the velocity ratio of $R=3.3$. LES results are validated against pointwise time averaged Laser Doppler Anemometry (LDA) measurements in PART1 of this study. In PART2 of the presented study – a planar (2D) LES based snapshot POD analysis is first conducted on two mutually perpendicular planes located in the jet-to-crossflow entrance zone. The obtained results are directly compared and found to be in close agreement with results of a Particle Image Velocimetry (PIV) based planar (2D) snapshot POD analysis by Meyer et al. (JFM 583, p. 199-227, 2007), indicating that LES is able to predict the same large scale flow dynamics as that captured by PIV. Some differences are also observed, but appear to be directly connected to the differences in levels of the resolved turbulent kinetic energy (TKE) between LES and PIV datasets. Those differences proved to be linked to the process of filtering out the small-scale fluctuations implicit to the PIV measurement technique. Comparisons of TKE captured by the first POD modes showed that they are not affected by this implicit filtering. The LES based POD analysis was also conducted in 3D. The 3D POD analysis results based on the first two POD modes show a full ability to directly visualize details of the relationship between the counter-rotating vortex pair (CVP), the hanging vortex and the wake vortices. POD reconstruction shows that the CVP originates from the hanging vortex formed at the lateral sides of the jet. It also shows that the shedding process involving oscillation of the jet core is responsible for the creation of wake vortices and that the wake vortex originates from the hanging vortex, but grows quickly by "sucking up" the wall boundary layer fluid and vorticity.
Method to Predict Long Time Span of Scour Around Offshore Wind Turbine Foundations

A new method to predict scour development around offshore structures has been developed. The method has been tested on a monopile. The method consists of table of scour rates, which is used to predict the scour development around the structure at different stages of the scour hole. The scour rate tables have been made based on full 3D numerical simulations of the flow and sediment transport for fixed configurations of the scour hole. When changing the governing parameters which are causing the scour development around the structure, the erosion rate or backfilling rate can be calculated from the mass balance of the sediment. This leads to the scour rates tables that are used to analyse the development of the scour hole under different wave and current conditions. The method has been tested against experimental scour data and showed very promising results.

Morphological modelling of the response to a shipwreck - A case study at Cape Town

A simulation of the morphological development and degrade of a salient behind a shipwreck located north of Cape Town, South Africa is presented. The morphological model is based on a hybrid morphological model concept which combines a 2D coastal model for calculating sediment transport with a simplified 1D morphological evolution model for the coastline. The model concept is applied to the case study in order to show how the modelling concept may be applied to real coastlines with general bathymetric features. The results show that the model captures the overall morphological response fairly well without the need for extensive calibration which is often required by traditional 2D morphological models. This is attributed by the authors to the fact that the sediment transport description is based on a process based model that captures the most important features, while neglecting the often challenging description of the cross-shore sediment transport.

On the Effect of Hull Girder Flexibility on the Vertical Wave Bending Moment for Ultra Large Container Vessels

Currently, a number of very large container ships are being built and more are on order, and some concerns have been expressed about the importance of the reduced hull girder stiffness to the wave-induced loads. The main concern is related to the fatigue life, but also a possible increase in the global hull girder loads as consequence of the increased hull flexibility must be considered. This is especially so as the rules of the classification societies do not explicitly account for
the effect of hull flexibility on the global loads. In the present paper an analysis has been carried out for the 9,400 TEU container ship used as case-ship in the EU project TULCS (Tools for Ultra Large Container Ships). A non-linear time-domain strip theory is used for the hydrodynamic analysis of the vertical bending moment amidships in sagging and hogging conditions for a flexible and a rigid modelling of the ship. The theory takes into account non-linear radiation forces (memory effects) through the use of a set of higher order differential equations. The non-linear hydrostatic restoring forces and non-linear Froude-Krylov forces are determined accurately at the instantaneous position of the ship in the waves. Slamming forces are determined by a standard momentum formulation. The hull flexibility is modelled as a nonprismatic Timoshenko beam. Generally, good agreement with experimental results and more accurate numerical predictions has previously been obtained in a number of studies. The statistical analysis is done using the First Order Reliability Method (FORM) supplemented with Monte Carlo simulations. Furthermore, strip-theory calculations are compared to model tests in regular waves of different wave lengths using a segmented, flexible model of the case-ship and good agreement is obtained for the longest of the waves. For the shorter waves the agreement is less good. The discrepancy in the amplitudes of the bending moment can most probably be explained by an underestimation on the effect of momentum slamming in the strip-theory applied.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Andersen, I. M. V. (Intern), Jensen, J. J. (Intern)
Number of pages: 8
Pages: Paper OMAE2012-83043
Publication date: 2012

Host publication information
Title of host publication: Proceedings of the ASME 2012 31st International Conference on Ocean, Offshore and Arctic Engineering
Publisher: American Society of Mechanical Engineers
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: 31st ASME International Conference on Ocean, Offshore and Arctic Engineering, Rio de Janeiro, Brazil, 01/07/2012 - 01/07/2012
Source: dtu
Source-ID: u::5627
Publication: Research - peer-review › Article in proceedings – Annual report year: 2012

On the Influence of Hull Girder Flexibility on the Wave
Numerical predictions and model test results of the wave induced bending moments in a 9,400 TEU post-Panamax container vessel are presented for two regular wave scenarios. Different numerical procedures have been applied: a linear and non-linear time-domain strip theory and a direct calculation (CFD) solving the Navier-Stokes equations with the free surface captured by a volume-of-fluid (VOF) method. In all procedures the flexibility of the hull girder is modelled as a non-uniform Timoshenko beam. It is observed that the non-linear models agree well with the model tests and as there is no occurrence of severe slamming in the cases considered, the inexpensive non-linear strip theory is as accurate as the direct CFD calculation method. In a comparison with the results using the rigid body assumption, the increase in the vertical bending moment (VBM) amidships due to the flexibility of the hull girder is found to be approximately 7% (peak-to-peak amplitude) in general. The non-linear responses, however, contain over-harmonic frequencies which may coincide with the natural frequency of the two-node vertical bending mode inducing resonance. In that case the hull girder flexibility causes the responses to increase as much as 22% (peak-to-peak amplitude) in one of the present cases.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Seng, S. (Intern), Andersen, I. M. V. (Intern), Jensen, J. J. (Intern)
Pages: 341-353
Publication date: 2012

Host publication information
Title of host publication: Proceedings of the 6th International Conference on Hydroelasticity in Marine Technology
Publisher: University of Tokyo Press
Editors: Takagi, K., Ogawa, Y.
ISBN (Print): 978-4-9906321-0-6
Main Research Area: Technical/natural sciences
Conference: 6th International Conference Hydroelasticity in Marine Technology, Tokyo, Japan, 19/09/2012 - 19/09/2012
Electronic versions:
Ontogenetic differentiation of swimming performance and behaviour in relation to habitat availability in the endangered North Sea houting (Coregonus oxyrinchus)

The survival of the highly endangered, anadromous fish species North Sea houting (Coregonus oxyrinchus) depends on the correct timing of downstream dispersal during its early ontogenetic stages. To date, however, no studies have investigated the ontogenetic differentiation of swimming performance and behaviour, including the potential of habitat complexity to influence dispersal rates. By testing larval and juvenile North Sea houting in a laboratory, we examined (1) swimming performance measured as maximum swimming performance (Umax) and routine swimming speed (Uroutine) and (2) the potential of habitat complexity (i.e., cover providing shade) to influence dispersal behaviour in an indoor stream channel. The Umax and the Uroutine were 9.4 and 4.6 cm s⁻¹, respectively, in the larvae [body length (BL) s⁻¹: 7.3 and 3.5, respectively], and 25.2 and 16.3 cm s⁻¹ in the juveniles (BL s⁻¹: 7.0 and 5.2, respectively). We compared laboratory swimming performance data with water speeds in North Sea houting spawning areas in the Danish River Vidaa. Results showed that the water speeds present in 95% and 85% of the water column caused downstream displacement of larvae and juveniles, respectively. However, areas with slow-flowing water near river banks and river beds could function as nursery habitats. Stream channel experiments showed that cover providing shade caused delayed dispersal in both larvae and juveniles, but the larvae dispersed later and spent less time under cover than the juveniles, a finding that implies ontogenetic effects. Finally, the larvae refused to cross an upstream-positioned cover, a behaviour that was not observed in the juveniles. Therefore, habitat complexity may have the potential to influence dispersal behaviour in both larval and juvenile North Sea houting. Overall, we provided the first evidence of ontogenetic differentiation in the North Sea houting. These findings will be valuable for the development and dissemination of science-based conservation strategies.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, National Institute of Aquatic Resources, Fisheries and Maritime Museum, Christian-Albrechts-Universität zu Kiel, Ribe Environmental Centre
Number of pages: 9
Pages: 241-249
Publication date: 2012
Main Research Area: Technical/natural sciences

Publication information

Journal: Aquatic Living Resources
Volume: 25
Issue number: 3
ISSN (Print): 0990-7440
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed Yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 1.41 SJR 0.59 SNIP 0.743
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 0.747 SNIP 0.848 CiteScore 1.39
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 0.641 SNIP 0.905 CiteScore 1.25
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 0.547 SNIP 0.68 CiteScore 1.15
ISI indexed (2013): ISI indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 0.554 SNIP 0.618 CiteScore 1.19
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
The purpose of the research project is to establish a reliable index in the performance evaluation of ships. During operation the ship will experience added resistance due to fouling of hull and propeller. The added resistance will lead to increased fuel consumption and thus increased emissions to the environment. The monitoring of the ship’s performance can be used as decision support in determining when actions to improve performance should be taken. The performance evaluation is based on a model of the ship and the added resistance from wind and waves during operation. Logged data on board the ship is used as input to the system and by comparing model and ship behaviour, an index describing the ship’s performance is generated.

The work in this thesis is based on data logged through the automation system on board a PostPanmax container ship where data have been logged through a year. A routine handling drift in time series, spikes and outliers have been suggested for the purpose of introducing an automatic logging system.

The performance system is modelled in software based on the Bond Graph method. The system is described by bond graph elements which describe the characteristics of each component and several ships are modelled in the system. A simple model is used as initial model and several elements are added to improve the estimate of the performance. Several resistance models are compared in order to determine which is giving the best estimate of the performance. Constraints in the models have been identified. The models used in this work are based on empirical relations or based on regression analyses of model tests and full-scale trials. In order to achieve valid results the conditions where performance is estimated have to be inside the boundaries of the model. Filters have been determined to establish cases where the ship is in steady state conditions and where these conditions are inside the boundaries of the constraints of the model. Several indexes have been used in the evaluation. Two indexes relate to the ship’s logged speed and one relates to the measured torque on the propeller shaft. Further, an index based on the properties of the ship’s propeller, is used. The different indexes are described in a case study where the performance of a container ship is evaluated over one year. The reliability of the performance index is measured from the scatter in results and the ability of identifying the events that improves performance e.g. propeller and hull clean.

Performance Monitoring of Ships

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Hansen, S. V. (Intern), Petersen, J. B. (Ekstern), Jensen, J. J. (Intern), Lützen, M. (Intern)
RANS-based simulation of turbulent wave boundary layers and sediment transport

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Fuhrman, D. R. (Intern)
Publication date: 2012
Event: Abstract from ICCE 2012, Santander, Spain.
Main Research Area: Technical/natural sciences
Electronic versions:
Abstract_in_Depth.pdf
Source: dtu
Source-ID: u::5750
Publication: Research › Conference abstract for conference – Annual report year: 2012

Residual Liquefaction under Standing Waves
This paper summarizes the results of an experimental study which deals with the residual liquefaction of seabed under standing waves. It is shown that the seabed liquefaction under standing waves, although qualitatively similar, exhibits features different from that caused by progressive waves. The experimental results show that the buildup of pore-water pressure and the resulting liquefaction first starts at the nodal section and spreads towards the antinodal section. The number of waves to cause liquefaction at the nodal section appears to be equal to that experienced in progressive waves for the same wave height. Recommendations are made as to how to assess liquefaction potential in standing waves.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Istanbul Technical University
Authors: Kirca, V. O. (Ekstern), Sumer, B. M. (Intern), Fredsøe, J. (Intern)
Pages: 1392-1398
Publication date: 2012

River Mouth Management In Malaysia– An Overview of Issues
Dredging and structural interventions have been applied for centuries in attempts to train and regulate river mouths and tidal inlet systems. The notoriously dynamic nature of river mouths and tidal inlets complicate any intervention and applied intervention schemes are frequently seen to display limited and sometimes even defective functionality. The latter is often due to lack of proper river mouth management which rely on solid assessment tools and a regard for implications of derived and external impacts. Inevitably, interventions will create an array of secondary problems (derived impacts) of which some are often unforeseen. The key parameter for success and effectiveness of interventions including adopted
mitigation measures for secondary problems depends on a detailed understanding of physical conditions at the river mouth as much as on the functionality of the layout, its design and reliable risk assessments. In many cases an optimal solution is achieved by a compromise between many environmental, social and economical aspects. This paper provides an overview of the physical characteristics of river mouths and challenges and issues in river mouth management in a Malaysian context. Numerical models have been used in the past to obtain qualitative and quantitative understanding of physical conditions at river mouths which is required as part of the design of interventions, as baseline for successful management as well as to test potential intervention schemes for various design and optimization phases. Examples demonstrating the use of numerical modeling as an engineering tool for previous river mouth improvement works are highlighted to reiterate its value in river mouth engineering and hopefully serve as motivation for future usage.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, DHI Denmark
Authors: Hjelmager Jensen, J. (Intern), Pedersen, C. (Ekstern)
Number of pages: 12
Publication date: 2012

Host publication information
Title of host publication: Proceedings of COPEDEC 2012
Main Research Area: Technical/natural sciences
Conference: 8th International Conference on Coastal and Port Engineering In Developing Countries (COPEDEC 2012), Chennai, India, 20/02/2012 - 20/02/2012
Source: dtu
Source-ID: u::5307
Publication: Research - peer-review › Article in proceedings – Annual report year: 2012

Sea state estimation from an advancing ship - The wave buoy analogy: Presentation at Skibsteknisk Selskab

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Nielsen, U. D. (Intern), Andersen, I. M. V. (Intern)
Number of pages: 23
Publication date: 2012

Publication information
Original language: English
Publisher: DTU Mechanical Engineering
Main Research Area: Technical/natural sciences
Electronic versions:
WaveEstim_SkibstekniskSelskab.pdf
Source: dtu
Source-ID: u::4039
Publication: Communication › Sound/Visual production (digital) – Annual report year: 2012

Slamming And Whipping Analysis Of Ships

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Seng, S. (Intern), Pedersen, P. T. (Intern), Jensen, J. J. (Intern)
Number of pages: 196
Publication date: 2012

Publication information
Publisher: DTU Mechanical Engineering
Original language: English
Series: DCAMM Special Report
Number: S151
ISSN: 0903-1685
Main Research Area: Technical/natural sciences
Electronic versions:
Sopheak_Seng_PhD_Thesis.pdf
A study of slamming events in conditional waves is presented in this paper. The ship is sailing in head sea and the motion is solved for under the assumption of rigid body motion constrained to two degree-of-freedom i.e. heave and pitch. Based on a time domain non-linear strip theory most probable conditional waves are generated to induce short term extreme responses of 4500 MNm sagging and hogging vertical bending moment (VBM) amidships on a modern 9,400-TEU post-Panamax container ship and 3000 MNm (sag) on a Panamax container ship. The results of the strip theory are compared to the results of free surface NS/VOF CFD simulations under the same wave conditions. In moderate seas and no occurrence of slamming the structural responses predicted by the methods agree well. When slamming occurs the strip theory overpredicts VBM but the peak values of VBM occurs at approximately the same time as predicted by the CFD method implying the possibility to use the more accurate CFD results to improve the estimation of slamming loads in the strip theory through a rational correction coefficient.
Staying dry under water

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Northwestern University, Swiss Federal Institute of Technology, University of Illinois
Authors: Jones, P. (Extern), Cruzchu, E. (Extern), Megaridis, C. (Extern), Walther, J. H. (Intern), Koumoutsakos, P. (Extern), Patankar, N. (Extern)
Pages: Paper DFD12-2012-001193
Publication date: 2012
Conference: 65th Annual Meeting of the APS Division of Fluid Dynamics, San Diego, CA, United States, 18/11/2012 - 18/11/2012
Main Research Area: Technical/natural sciences

Suction removal of sediment from between armor blocks. III: Breaking waves

When a sediment beach covered by stones or an armor layer is exposed to breaking waves, the turbulence generated by the breaking waves can cause mobilization and removal of the sediment underneath the stones. In two earlier studies by the Technical University of Denmark (DTU)-group, the suction removal of sediments from between armor blocks has dealt with the case of steady current and nonbreaking waves. The present study is an extension of these studies and the results will be presented in a similar way. The critical conditions for removal of sediment are determined. It is found that the onset of removal of sediment is governed by three parameters: (1) the sediment mobility (based on the sediment size, wave height, and wave period), (2) the ratio between the sediment size and the stone size, d=D, and (3) surf similarity parameter, ξ = tan(β)/L0. The variation of the critical mobility number for removal of sediment as function of d=D and ξ is determined for the range 0.001 < d=D ≤ 1.0 and 0.15 < ξ < 6.00. The experiments were made on 1:2, 1:14, and 1:30 slopes. Spilling, plunging, and surging breakers were used in the experiments. Both one and two layers of armor stones and rectangular blocks were studied.
Swirling flow in model of large two-stroke diesel engine

A scale model of a simplified cylinder in a uniflow scavenged large two-stroke marine diesel engine is constructed to investigate the scavenging process. Angled ports near the bottom of the cylinder liner are uncovered as the piston reaches the bottom dead center. Fresh air enters through the ports forcing the gas in the cylinder to leave through an exhaust valve located in the cylinder head. The scavenging flow is a transient (opening/closing ports) confined port-generated turbulent swirl flow, with complex phenomena such as central recirculation zones, vortex breakdown and vortex precession. The model has a transparent cylinder five diameters long and is fitted with a static valve with a simplified geometry. The piston motion is controlled by a linear motor. The flow in the experiment has a Reynolds number of 50000 based on cylinder diameter and bulk velocity. Stereoscopic Particle Image Velocimetry (PIV) is used to investigate the scavenging flow for cases with both static and moving piston. Measurements are carried out for several cross-sectional planes covering the majority of the cylinder length. The effect of swirl intensity is investigated using four different port angles going from 0 – 30 degree. Although the flow has a relatively low swirl number of around 0.4, a central recirculation zone is observed indicating a vortex breakdown. The steady flow is also analyzed with Proper Orthogonal Decomposition (POD). The analysis reveals systematic variations in the shape and location of the vortex core. Transient measurements using phase-locked PIV are carried out with moving piston. The transient measurements reveal a violent change in flow topology as a central recirculation zone is rapidly formed, resulting in a change from large positive to negative velocities of the axial component.
The Maritime Engineering Education: meeting industry demands

This article describes the outcome of a survey initiated by the Danish Maritime Fund (DMF). The survey resulted in a report that describes the engineering competencies requested by the Danish maritime industry. This is of interest since the desired competencies have changed in the past one to two decades, where Denmark no longer has a considerable ship-building industry. Furthermore, the DMF initiated report concludes that the demand for maritime engineers in the industry is larger than the output. The report sets forth a series of recommendations to the industry as well as the universities to enable meeting the demand for maritime engineers in Denmark. The recommendations are outlined together with the work commenced at the Technical University of Denmark (DTU) and the University of Southern Denmark (SDU) to follow up on the recommendations in the report.

Three dimensional remeshed smoothed particle hydrodynamics for the simulation of turbulent flow in complex geometries

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Obeidat, A. (Intern), Walther, J. H. (Intern)
Publication date: 2012
Main Research Area: Technical/natural sciences

Time scale of scour around a pile in combined waves and current

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Obeidat, A. (Intern), Walther, J. H. (Intern)
Publication date: 2012
Main Research Area: Technical/natural sciences

Time scale of scour around a pile in combined waves and current

The time scale of the scour process around a circular vertical pile is studied in combined waves and current. A series of tests were carried out in a flume with pile diameters 40 mm and 75 mm, in both steady current, waves and combined waves and current. In the combined wave and current flow regime the waves and the current were co-directional. All the tests were conducted in the live bed regime.
The time scale of scour in combined waves and current is governed by three parameters, namely the current-velocity-to-wave-velocity ratio ($U_{cw}$), the Keulegan–Carpenter number ($KC$) and Shields parameter ($\Theta_w$). The time scale of scour increases significantly when even a slight current is superimposing on a wave. The KC dependence of the time scale $T$ is mainly observed for low values of $U_{cw}$ in the wave dominated regime. For $U_{cw}$ values larger than 0.4 no clear KC dependency was observed. The time scale decreases with increasing $\Theta_w$ over the entire range of $U_{cw}$.

**Towards fault-tolerant decision support systems for ship operator guidance**

Fault detection and isolation are very important elements in the design of fault-tolerant decision support systems for ship operator guidance. This study outlines remedies that can be applied for fault diagnosis, when the ship responses are assumed to be linear in the wave excitation. A novel numerical procedure is described for the calculation of residuals using the ship’s transfer functions which correlate the wave excitation and the ship responses. As tests, multiplicative faults have artificially been imposed to full-scale motion measurements and it is shown that the developed model is able to detect and isolate all faults.
We present recent and preliminary work directed towards the development of a simplified, physics-based model for improved simulation of ship-ship interaction that can be used for both analysis and real-time computing (i.e. with real-time constraints due to visualization). The goal is to implement the model into a large maritime simulator for training of naval officers, in particular tug boat helmsmen. Tug boat simulators are used for training of communication and situation awareness during manoeuvre involved with towing of large vessels. A main objective of the work is to improve and enable more accurate (realistic) and much faster ship-wave and ship-ship simulations than are currently possible. The coupling of simulation with visualization should improve the visual experience such that it can be perceived as more realistic in training. Today the state-of-art in real-time ship-ship interaction is for efficiency reasons and time-constraints in visualization based on model experiments in towing tanks and precomputed force tables. We anticipate that the fast, and highly parallel, algorithm described by Engsig-Karup et al. [2011] for execution on affordable modern high-throughput Graphics Processing Units (GPUs) can provide the basis for efficient simulations in combination with an accurate free-surface model for Ship-Ship simulation. Another area of application is the determination of wave disturbances from a ship in a coastal environment, channels and harbours. The model proposed in the following can in a simple and efficient way calculate the wave field from a ship sailing in a finite depth sea, even with variations in the height of sea bed. The generated wave field can be applied as an input to other models that simulate the marine environment on a larger scale.
Transient waves generated by a moving bottom obstacle: a new near-field solution

We consider the classical problem of a single-layer homogeneous fluid at rest and a low, slowly varying, long and positive bottom obstacle, which is abruptly started from rest to move with a constant speed $V$. As a result a system of transient waves will develop, and we assume that locally in the region over the obstacle dispersion can be ignored while nonlinearity cannot. The relevant governing equations for the near-field solution are therefore the nonlinear shallow water (NSW) equations. These are bidirectional and can be formulated in terms of a two-family system of characteristics. We analytically integrate and eliminate the backward-going family and achieve a versatile unidirectional single-family formulation, which covers subcritical, transcritical and supercritical conditions with relatively high accuracy. The formulation accounts for the temporal and spatial evolution of the bound waves in the vicinity of the obstacle as well as the development of the transient free waves generated at the onset of the motion. At some distance from the obstacle, dispersion starts to play a role and undular bores develop, but up to this point the new formulation agrees very well with numerical simulations based on a high-order Boussinesq formulation. Finally, we derive analytical asymptotic solutions to the new equations, providing estimates of the asymptotic surface levels in the vicinity of the obstacle as well as the crest levels of the leading non-dispersive free waves. These estimates can be used to predict the height and speed of the leading waves in the undular bores. The numerical and analytical solutions to the new single-family formulation of the NSW equations are compared to results based on the forced Korteweg–de Vries/Hopf equation and to numerical Boussinesq simulations.
Turbulence modelling in the two-dimensional vortex-in-cell method

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, COWI Consultants A/S
Authors: Hejlesen, M. M. (Intern), Rasmussen, J. T. (Intern), Larsen, A. (Ekstern), Walther, J. H. (Intern)
Publication date: 2012
Introduction

Particle-fluid interactions in supersonic flows are relevant in many different applications e.g. the cold gas-dynamic spray process. The optimal application of the process is hindered by a lack of understanding of the particle-fluid interactions. To obtain detailed information on the particle-fluid interactions in these high-speed flows special high performance techniques are required. The present work is an investigation into the applicability of magnified digital in-line holography with ultrahigh-speed recording for the study of three-dimensional supersonic particle-laden flows. An optical setup for magnified digital in-line holography is created, using an ultra-high-speed camera capable of frame rates of up to 1.0MHz. To test the new technique an axisymmetric supersonic underexpanded particle-laden jet is investigated. The results show that the new technique allows for the acquisition of time resolved qualitative and quantitative data on the fluid and particle dynamics. The potential for determining the three-dimensional particle positions using holographic reconstruction is also investigated. Five different focus functions used for depthwise location are tested using two different planar targets, a 10 × 10mm calibration grid and 120 μm particles on a glass plate. In the case with the calibration grid it is found that accurate determination of the depthwise position is possible. However, when applying the same technique to the particle target, significant problems are encountered. © 2012 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Ingvorsen, K. M. (Intern), Buchmann, N. A. (Ekstern), Soria, J. (Ekstern)
Wave interaction with large roughness elements on an impermeable sloping bed

The present paper presents the results of an experimental and numerical investigation of the flow between large roughness elements on a steep sloping impermeable bed during wave action. The setup is designed to resemble a breakwater structure. The work is part of a study where the focus is on the details in the porous core flow and the armour layer flow i.e. the interaction between the two flow domains and the effect on the armour layer stability. In order to isolate the processes involved with the flow in the porous core the investigations are first carried out with a completely impermeable bed and successively repeated with a porous bed. In this paper the focus is on the impermeable bed.

Results are obtained experimentally for flow and turbulence between the roughness elements on the sloping bed. Numerical simulations have reproduced the experimental results with good agreements and can hereby add more details to the understanding of the fluid-structure interaction.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jensen, B. (Intern), Christensen, E. D. (Intern), Sumer, B. M. (Intern)
Number of pages: 15
Publication date: 2012

Wave liquefaction in soils with clay content

The paper presents the results of an experimental study of the influence of clay content (in silt-clay and sand-clay mixtures) on liquefaction beneath progressive waves. The experiments showed that the influence of clay content is very significant. Susceptibility of silt to liquefaction is increased with increasing clay content, up to 30%, beyond which the mixture of silt and clay is not liquefied. Sand may become prone to liquefaction with the introduction of clay, contrary to the general perception that this type of sediment is normally liquefaction resistant under waves.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Kirca, Ö. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern)
Number of pages: 8
Publication date: 2012
Wind Forces on Container Ships

An investigation of the wind forces acting on a 9,000+ TEU container ship has been carried out through a series of wind tunnel tests. It was investigated how the wind forces depend on the container configuration on the deck using a 1:450 scale model and a series of appropriate container configurations. The wind tunnel tests were carried out in the naturally existing boundary layer of the wind tunnel. The longitudinal and transverse forces and the yaw moment were measured and the measurements were corrected for the effects of the boundary layer and blockage in the wind tunnel. The results are presented as nondimensional coefficients. It is concluded, that the measured forces and moment depend on the container configuration on deck, and the results may provide a general idea of how the magnitude of the wind forces is affected by a given container stacking configuration on a similar container ship.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Andersen, I. M. V. (Intern)
Number of pages: 12
Publication date: 2012
Main Research Area: Technical/natural sciences

Publication information
Journal: Mercator
Issue number: Marts
ISSN (Print): 1902-164X
Ratings:
ISI indexed (2013): ISI indexed no
ISI indexed (2012): ISI indexed no
ISI indexed (2011): ISI indexed no
Original language: English
Electronic versions:
Mercator_Wind_kort_endelig.pdf
Source: dtu
Source-ID: u::4067
Publication: Research › Journal article – Annual report year: 2012

Wind-Tunnel Investigation of Wind Loads on a Post-Panamax Container Ship as a Function of the Container Configuration on Deck

An investigation of the wind forces acting on a 9,000+ TEU container ship has been carried out through a series of wind tunnel tests. It was investigated how the wind forces depend on the container configuration on the deck using a 1:450 scale model and a series of appropriate container configurations. The wind tunnel tests were carried out in the naturally existing boundary layer of the wind tunnel. The longitudinal and transverse forces and the yaw moment were measured and the measurements were corrected for the effects of the boundary layer and blockage in the wind tunnel. The results are presented as nondimensional coefficients. It is concluded, that the measured forces and moment depend on the container configuration on deck, and the results may provide a general idea of how the magnitude of the wind forces is affected by a given container stacking configuration on a similar container ship.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Andersen, I. M. V. (Intern)
Number of pages: 18
Publication date: 2012

Host publication information
Title of host publication: 11th International Marine Design Conference
Main Research Area: Technical/natural sciences
Conference: IMDC2012, Galsgow, United Kingdom, 11/06/2012 - 11/06/2012
Wind loads, Container ship, Container stacking, Wind resistance, Wind tunnel tests
Electronic versions:
Scour Protection of Offshore Wind Farms

One of the first large offshore wind farms is the Horns Rev 1 Wind Farm in the Danish part of the North Sea. It is located around 20 km from the coast in relatively shallow water. The wind farm was installed and commissioned during 2002. In 2005 a control survey of the scour protections around the foundations showed that the scour protections adjacent to the mono piles sank by up to 1.5 m. This was unexpected and shortly after the survey in 2005 the holes were repaired by adding additional stones. The aim of the thesis is to give an explanation of the sinking at Horns Rev 1 Wind Farm and to describe the processes causing the sinking.

In Chapters 2 and 3, a description of the main processes causing the sinking of a scour protection around a pile if exposed to a current is given, based on results of physical and numerical models. Using the results it is possible to make designs of the scour protection to prevent or estimate sinking of scour protections around mono piles. It is shown that the horseshoe vortex at the base of the pile will penetrate into the scour protection and if strong enough mobilize the base sediment and transport it out of the scour protection where it will be removed by the outer ow. The equilibrium sinking of the scour protection is found for various conditions. It is also found that a fine filter layer can prevent the mobilization of the sediment and therefore the sinking.

In Chapter 4, the scour around mono piles in breaking waves is studied. The scour is found to depend on two parameters: (1) The distance between the breaking point and the pile normalized by the wave length and (2) the breaking wave height normalized by the pile diameter. The maximum scour is found to be approximately 0.65 times the pile diameter. It can be larger than the scour generated by non-breaking waves especially for small KC-numbers. The main reason for the increased scour is found to be turbulence generated by the breaking and is forced to the bottom by the pile.

In Chapter 5, the onset of suction from between armour stones under breaking waves is studied. The critical conditions for onset of suction are determined for several different conditions, regarding wave height and period, slope of the bed, sediment and cover stone size and number of cover layers. The oblique descending eddies generated by the breaking waves are found to be the main mechanism regarding suction of the base sediment.

In Chapter 6, the ow in and the bed shear stresses under a stone cover under an oscillatory ow is described. The ow velocities and the turbulence are measured in case of one, two and three layers of stones in several pores. The tests showed that the horizontal velocities in the pores are affected by the outer ow approximately one stone diameter into the stone cover. Under this level the horizontal velocities become constant. The bed shear stresses are much smaller than at a smooth bed without stone cover and large variations are observed.

In Chapter 7, the effect of waves on sinking of the scour protection around a mono pile is studied. It is found that the sinking will increase for increasing KC-numbers for a given diffraction parameter. The magnitude of the sinking is found to be comparable with the scour observed around an unprotected pile exposed the same wave conditions.
Environmental performance evaluation of RoPax ferries

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Coastal, Maritime and Structural Engineering, University of Applied Sciences Bremen
Authors: Hagemeister, C. (Ekstern), Kristensen, H. O. H. (Intern)
Pages: 10-14
Publication date: 2011
Main Research Area: Technical/natural sciences

Publication information
Journal: Ship & Offshore
Issue number: 3
ISSN (Print): 2191-0057
Original language: English
Links:
http://digimagazin.shipandoffshore.net/2011-03/magazine.html

Environmental Performance Evaluation of Ro-Ro Passenger Ferry Transportation

General information
State: Published
Organisations: Fluid Mechanics, Coastal and Maritime Engineering, Department of Mechanical Engineering, Nordic Yards
Authors: Kristensen, H. O. H. (Intern), Hagemeister, C. (Ekstern)
Number of pages: 12
Publication date: 2011
Main Research Area: Technical/natural sciences

Publication information
Journal: Selected Proceedings from the Annual Transport Conference at Aalborg University
ISSN (Print): 1603-9696
Ratings:
BFI (2018): BFI-level 1
BFI (2017): BFI-level 1
BFI (2016): BFI-level 1
BFI (2015): BFI-level 1
BFI (2014): BFI-level 1
BFI (2013): BFI-level 1
ISI indexed (2013): ISI indexed no
BFI (2012): BFI-level 1
ISI indexed (2012): ISI indexed no
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
ISI indexed (2011): ISI indexed no
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 1
BFI (2009): BFI-level 1
BFI (2008): BFI-level 1
Original language: English
Electronic versions:
Environmental Performance Evaluation of Ro-Ro Passenger Ferry Transportation-FINAL.pdf
Source: dtu
Source-ID: u::3642
Publication: Research - peer-review › Conference article – Annual report year: 2012
High Spatial Resolution Imaging of a Supersonic Underexpanded Jet Impinging on a Flat Plate

Influence of piston position on the scavenging and swirling flow in two-stoke diesel engines

Influence of piston position on the scavenging and swirling flow in two-stoke diesel engines

Influence of piston position on the scavenging and swirling flow in two-stoke diesel engines

Influence of piston position on the scavenging and swirling flow in two-stoke diesel engines
Molecular Dynamics Studies of Nanofluidic Devices

Nanotechnology and fluid mechanics are two scientific areas where recent progress has disclosed a variety of new possibilities. The advances in both fields established the grounds for interdisciplinary approaches and recent findings promise novel applications that are leading to a technological revolution. Novel nanofabrication techniques have opened up possibilities for the development of small-scale integrated devices, such as lab-on-a-chip for biochemical synthesis and analysis, the integration is achieved by miniaturization of the functional elements e.g., of the channels transporting the fluid and of the sensors performing the analysis, and as the size of these devices reaches the sub-micron range we enter the field of nanofluidics. Nanofluidics is defined as the study of flows in and around nanosized objects. Modeling of transport in nanofluidic systems differs from microfluidic systems because changes in transport caused by the walls become more dominant and the fluid consists of fewer molecules. Carbon nanotubes are tubular graphite molecules which can be imagined to function as nanoscale pipes or conduits. Another important material for nanofluidics applications is silica.

Nowadays, silica nanochannels are produced in nanometer scale using different nanofabrication techniques. Silica nanochannels are being implemented in several nanotechnology applications such as nanosensor devices, nano separators, nanofilters and a plethora of devices for nanobiological and biochemical applications. Experiments at the nanoscale are expensive and time consuming moreover the time scale associated to several nanoscale phenomena requires a very high time resolution of the devices performing nanoscale measurements. Computational nanofluidics is the enabling technology for fundamental studies, development, and design of such devices. Computational nanofluidics complements experimental studies by providing detailed spatial and temporal information of the nanosystem. In this thesis, we conduct molecular dynamics simulations to study basic nanoscale devices. We focus our studies on the understanding of transport mechanism to drive fluids and solids at the nanoscale. Specifically, we present the results of three different research projects. Throughout the first part of this thesis, we include a comprehensive introduction to computational nanofluidics and to molecular simulations, and describe the molecular dynamics methodology. In the second part of this thesis, we present the results of three different research projects. Firstly, we present a computational study of thermophoresis as a suitable mechanism to drive water droplets confined in different types of carbon nanotubes. We observe a motion of the water droplet in opposite direction to the imposed thermal gradient also we measure higher velocities as higher thermal gradients are imposed. Secondly, we present an atomistic analysis of a molecular linear motor fabricated of coaxial carbon nanotubes and powered by thermal gradients. The MD simulation results indicate that the motion of the capsule (inner carbon nanotube) can be controlled by thermophoretic forces induced by thermal gradients. The simulations find large terminal velocities of 100 to 400 nmns\(^{-1}\) for imposed thermal gradients in the range of 1 to 3 Knm\(^{-1}\). Moreover, the results indicate that the thermophoretic force is velocity dependent and its magnitude decreases for increasing velocity. Finally, we present an extensive computational study of nanoscale systems including silica substrates and channels, water and air. This study includes the calibration of a force field to describe the silica-water-air interactions. Moreover, In this study we perform very long simulations of nanoscale systems containing silica, water and air. We investigate the solubility of air at different pressures in silica-water systems. From our simulations we infer a layer with high air density close to silica surface. Furthermore, we conduct simulations to analyze the earlier stage of the capillary filling process of silica nanochannels, we focus this study on the role of air in this system. We find that air at high pressures can affect the capillarity in silica channels below 10 nm height.
Numerical Modeling of Shoreline Undulations

The present thesis considers undulations on sandy shorelines. The aim of the study is to determine the physical mechanisms which govern the morphologic evolution of shoreline undulations, and thereby to be able to predict their shape, dimensions and evolution in time. In order to do so a numerical model has been developed which describes the longshore sediment transport along arbitrarily shaped shorelines. The numerical model is based on a spectral wave model, a depth integrated flow model, a wave-phase resolving sediment transport description and a one-line shoreline model. First the theoretical length of the shoreline undulations is determined in the linear regime using a shoreline stability analysis based on the numerical model. The analysis shows that the length of the undulations in the linear regime depends on the incoming wave conditions and on the coastal profile. For larger waves and flatter profiles the length of the undulations increases. Secondly the evolution of the shoreline undulations from the linear regime to the fully non-linear regime is described using the numerical shoreline evolution model. In the fully non-linear regime down drift spits and migrating shoreline undulations are described by the model. The shoreline evolution is considered for both constant and varying wave forcing and both periodic model domains with a single undulation as well as periodic model domains with multiple undulation are considered. Three different shoreline shapes are found depending on the wave conditions and the coastal profile: undulations with no spits, undulations with flying spits and undulations with reconnecting spits. It is further shown that the evolution of the shoreline undulations is governed mainly by the angle between the shoreline and the incoming waves and the curvature of the shoreline. Thirdly the shoreline evolution model is tuned to two naturally occurring shorelines. On one of the shorelines, the west coast of Namibia, the shoreline model is able to describe the observed shoreline features in both a qualitative and quantitative way. The model over-predicts the scale of the feature and under predicts the migration speeds of the features. On the second shoreline, the shoreline model predicts undulations lengths which are longer than the observed undulations. Lastly the thesis considers field measurements of undulations of the bottom bathymetry along an otherwise straight coast at the DanishWest Coast. Two bathymetric datasets and two time series of wave measurements are used in order to determine the following properties: The offshore extent of shoreline undulations, the amount of sediment transported alongshore in the shoreline undulations, the relationship between the shoreline undulations and longshore bars and the relationship between the morphology and the hydrodynamics. In one of the data sets the shoreline undulations are well correlated with undulations on the depth contours between -5 m and +2 m relative to mean sea level. An analysis of the wave climate shows that this shoreline is right at the limit between a stable and an unstable shoreline.

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Kærgaard, K. H. (Intern), Fredsøe, J. (Intern)
Number of pages: 284
Publication date: 2011
Planar measurements of velocity and concentration of turbulent mixing in a T-junction
Turbulent mixing of two isothermal air streams in a T-junction of square ducts are investigated. Three dimensional velocity fields and turbulent kinetic energy are measured with stereoscopic Particle Image Velocimetry (PIV). The concentration field is obtained with a planar Mie scattering technique using the stereoscopic PIV setup. The concentration measurement method is developed in the present study and the accuracy of the technique is investigated. The resulting data are two dimensional concentration fields taken at 4Hz. The combination of velocity, turbulence and concentration fields give valuable insight into the mixing process, e.g. by showing large scale flow instabilities. The present technique is well suited for easy testing of mixing devices and for validation of computational models.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, F.L. Smidt A/S
Authors: Ingvorsen, K. M. (Intern), Meyer, K. E. (Intern), Nielsen, N. F. (Ekstern)
Number of pages: 5
Publication date: 2011
Event: Abstract from 9th International Symposium On Particle Image Velocimetry, Kobe, Japan.
Main Research Area: Technical/natural sciences
Electronic versions: 2_136_552921293279324.pdf
Source: dtu
Source-ID: u::3679
Publication: Research - peer-review › Conference abstract for conference – Annual report year: 2012

Coherent structures in wave boundary layers, Part 1. Oscillatory motion
This work concerns oscillatory boundary layers over smooth beds. It comprises combined visual and quantitative techniques including bed shear stress measurements. The experiments were carried out in an oscillating water tunnel. The experiments reveal two significant coherent flow structures: (i) Vortex tubes, essentially two-dimensional vortices close to the bed extending across the width of the boundary-layer flow, caused by an inflectional-point shear layer instability. The imprint of these vortices in the bed shear stress is a series of small, insignificant kinks and dips. (ii) Turbulent spots, isolated arrowhead-shaped areas close to the bed in an otherwise laminar boundary layer where the flow ‘bursts’ with violent oscillations. The emergence of the turbulent spots marks the onset of turbulence. Turbulent spots cause single or multiple violent spikes in the bed shear stress signal, which has profound implications for sediment transport (in both the laboratory and the field). The experiments also show that similar coherent flow structures exist in the case of combined oscillatory flow and current.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Coastal, Maritime and Structural Engineering
Authors: Carstensen, S. (Intern), Sumer, B. M. (Intern), Fredsøe, J. (Intern)
Pages: 169-206
Publication date: 2010
Main Research Area: Technical/natural sciences

Publication information
Journal: Journal of Fluid Mechanics
Volume: 646
ISSN (Print): 0022-1120
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.82 SJR 1.671 SNIP 1.636
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.912 SNIP 1.676 CiteScore 2.57
Web of Science (2015): Indexed yes
Coherent structures in wave boundary layers. Part 2. Solitary motion

This study continues the investigation of wave boundary layers reported by Carstensen, Sumer & Fredsøe (J. Fluid Mech., 2010, part 1 of this paper). The present paper summarizes the results of an experimental investigation of turbulent solitary wave boundary layers, simulated by solitary motion in an oscillating water tunnel. Two kinds of measurements were made: bed shear stress measurements and velocity measurements. The experiments show that the solitary-motion boundary layer experiences three kinds of flow regimes as the Reynolds number is increased: (i) laminar regime; (ii) laminar regime where the boundary-layer flow experiences a regular array of vortex tubes near the bed over a short period of time during the deceleration stage; and (iii) transitional regime characterized with turbulent spots, revealed by single/multiple, or,
sometimes, quite dense spikes in the bed shear stress traces. Supplementary synchronized flow visualization tests confirmed the presence of the previously mentioned flow features. Information related to flow resistance are also given in the paper.

**General information**

State: Published
Organisations: Coastal, Maritime and Structural Engineering, Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Technical University of Denmark
Authors: Sumer, B. M. (Intern), Jensen, P. M. (Intern), Sørensen, L. B. (Ekstern), Fredsøe, J. (Intern), Liu, P. (Ekstern), Carstensen, S. (Intern)
Pages: 207-231
Publication date: 2010
Main Research Area: Technical/natural sciences

**Publication information**

Journal: Journal of Fluid Mechanics
Volume: 646
ISSN (Print): 0022-1120
Ratings:
BFI (2018): BFI-level 2
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 2
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 2
Scopus rating (2016): CiteScore 2.82 SJR 1.671 SNIP 1.636
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 2
Scopus rating (2015): SJR 1.912 SNIP 1.676 CiteScore 2.57
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 2
Scopus rating (2014): SJR 1.865 SNIP 1.808 CiteScore 2.66
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 2
Scopus rating (2013): SJR 1.894 SNIP 1.915 CiteScore 2.71
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 2
Scopus rating (2012): SJR 1.731 SNIP 1.88 CiteScore 2.47
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 2
Scopus rating (2011): SJR 2.165 SNIP 2.023 CiteScore 2.72
ISI indexed (2011): ISI indexed yes
Web of Science (2011): Indexed yes
BFI (2010): BFI-level 2
Scopus rating (2010): SJR 2.29 SNIP 2.163
Web of Science (2010): Indexed yes
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 2.563 SNIP 1.891
Web of Science (2009): Indexed yes
BFI (2008): BFI-level 1
Scopus rating (2008): SJR 2.691 SNIP 2.073
Web of Science (2008): Indexed yes
Scopus rating (2007): SJR 2.417 SNIP 1.975
Web of Science (2007): Indexed yes
Scopus rating (2006): SJR 2.641 SNIP 2.181
Web of Science (2006): Indexed yes
The effectiveness of moving masses in reducing the roll motion of floating vessels

Dynamic motions of Ships in severe conditions of sea maybe undesired and should be controlled by some devices. The roll motion is much more significant than the other oscillations which can affect comfort, safety and efficiency of navigation at sea. This motion is controlled by some common stabilizers such as rudder, fins, etc. They may be divided into two categories of passive and active stabilizers or controllers. However controlling devices may be high in cost. Moving masses are employed to reduce the motions of floating vessels or offshore structures. This type of stabilizers consists of a mass with one or more springs and dampers. The movement of the mass produces an inertia force and moment to absorb wave excited motions. In this study the effectiveness of moving masses in vessels with large roll motion is investigated. A rolling vessel equipped with a moving mass can be modeled as a two-degree of freedom system. As in large amplitude roll motion the nonlinearities cannot be neglected, a mathematical model with second-order damping and fifth-order restoring moment is implemented as roll equation in beam waves. Applying to a Trawler, this equation is solved simultaneously with the equation of movement of the mass in both frequency-domain and time-domain by analytical and numerical approaches. The computation shows that with an optimum passive control, a moving mass with maximum 2% of the displacement of the ship is an effective device in roll minimization and enhance the vessel’s stability in high load conditions.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, Amirkabir University of Technology
Authors: Montazeri, N. (Intern), Mousavizadegan, S. (Ekstern), Bakhtiarinejad, F. (Ekstern)
Number of pages: 6
Publication date: 2010

Host publication information
Title of host publication: Proceedings of the ASME 2010 International Mechanical Engineering Congress & Exposition
Publisher: American Society of Mechanical Engineers
Article number: IMECE2010-38574
Main Research Area: Technical/natural sciences
Source: PublicationPreSubmission
Source-ID: 118269161
Publication: Research - peer-review Article in proceedings – Annual report year: 2010

Multi Vessel interaction in shallow water

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Hansen, H. F. (Ekstern), Carstensen, S. (Intern), Christensen, E. D. (Intern), Kirkegaard, J. (Ekstern)
Number of pages: 964
Filter-feeding zoobenthos and importance of hydrodynamics in the shallow Odense Fjord (Denmark) - earlier and recent studies, perspectives and modelling

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering, University of Southern Denmark, University of Rostock, Danish Ministry of the Environment
Authors: Riisgård, H. U. (Ekstern), Lassen, J. (Ekstern), Kortegård, M. (Ekstern), Møller, L. F. (Intern), Friederichs, M. (Ekstern), Jensen, M. (Ekstern), Larsen, P. S. (Intern)
Pages: 281-295
Publication date: 2007
Main Research Area: Technical/natural sciences

Publication information
Journal: Estuarine, Coastal and Shelf Science
Volume: 75
ISSN (Print): 0272-7714
Ratings:
BFI (2018): BFI-level 1
Web of Science (2018): Indexed yes
BFI (2017): BFI-level 1
Web of Science (2017): Indexed yes
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 2.43 SJR 0.997 SNIP 1.127
Web of Science (2016): Indexed yes
BFI (2015): BFI-level 1
Scopus rating (2015): SJR 1.107 SNIP 1.186 CiteScore 2.44
Web of Science (2015): Indexed yes
BFI (2014): BFI-level 1
Scopus rating (2014): SJR 1.067 SNIP 1.257 CiteScore 2.28
Web of Science (2014): Indexed yes
BFI (2013): BFI-level 1
Scopus rating (2013): SJR 1.323 SNIP 1.439 CiteScore 2.64
ISI indexed (2013): ISI indexed yes
Web of Science (2013): Indexed yes
BFI (2012): BFI-level 1
Scopus rating (2012): SJR 1.256 SNIP 1.419 CiteScore 2.52
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
BFI (2011): BFI-level 1
Scopus rating (2011): SJR 1.383 SNIP 1.325 CiteScore 2.52
Shape and Dimensions of Ripples

In this work, the flow and bed load transport over ripple profiles under the influence of oscillatory will be investigated. The investigation is made through a parametric study, where the bed shape and the ripple steepness are varied. For the sediment transport, the Shields parameter relative to the critical one is varied in the range 1.31-3.45. A detailed flow description in terms of phase resolved quantities and period averaged quantities are given. The former are phase lag between the free stream flow and the separation and a description of the lee side vortices behind vortex ripples. The latter are the recirculation zone and its penetration into the main flow. This will be compared to the period averaged bed shear stress. An analysis of the bed load transport, both as a transport capacity and an analysis of possible stable ripples is conducted. This leads to some stable ripples and the general conclusion is that sharp crested ripples is thought unlikely to become stable under oscillatory flow due to the large increase in bed shear stresses because of the rapid contraction of the flow around the crest. Further, a dynamically moving ripple crest is investigated. The key focus is to investigate how a perturbation on top of the ripple influences the flow. It will be seen that both the shape and the velocity of the crest are affecting the period averaged flow globally.

General information
State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Jacobsen, N. G. (Intern), Fredsøe, J. (Intern)
Number of pages: 84
Publication date: 2007

Publication information
Publisher: DTU Mechanical Engineering
Original language: English
Main Research Area: Technical/natural sciences
Electronic versions:
Numerical Study on the Effect of Buffer Bow Structure in Ship-to-ship Collisions

A disastrous oil spill from a struck oil tanker has become one of the major problems in view of conservation of maritime environment. So far double hulls (D/H) have been introduced to reduce the consequences of collision and grounding events in order to further reduce the oil spill from struck oil tankers, the introduction of buffer bulbous bows has been proposed. Relatively soft buffer bows absorb part of the kinetic energy of the striking ship before penetrating the inner hull of the struck vessel. The purpose of the present paper is to verify the effectiveness of a prototype buffer bulbous bow structure in ship-ship collisions as compared with that of standard bulbous bows. This is demonstrated by conducting a series of large-scale finite element analyses. The finite element analyses are conducted with the general-purpose nonlinear structural code “LS-DYNA”. The applied scenario is one where a very large crude oil carrier (VLCC) in ballast condition collides with the midship region of a D/H VLCC in a laden condition. Fracture of fillet welds, elastic-plastic material properties and strain rate effects, are taken into account in the simulations. The effect of the equivalent failure strain (FS) and the forward velocity of the struck ship on the collapse mode of the bow of the striking vessel are investigated. Collapse modes, contact forces and energy absorption capabilities of the buffer bows are compared with those of conventional bows.
State-of-the-art laser Doppler systems development for turbulence measurements
Testing and development of improved laser Doppler anemometry methods

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Period: 01/06/2015 → 31/05/2018
Number of participants: 4
Project participant:
Yaacob, Mohd Rusdy (Ekstern)
Velte, Clara Marika (Intern)
Meyer, Knud Erik (Intern)
Buchhave, Preben (Intern)

Experimental and theoretical investigations of turbulent axi-symmetric jets
Fundamental turbulence study for studying the development of the jet for creating an analytical model. The results will be useful for studying the dependence upon initial/upstream condition and the development of turbulence.

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Imperial College London
Period: 15/04/2015 → 14/04/2018
Number of participants: 4
Number of related Ph.D. students: 1
Project participant:
Hodzic, Azur (Intern)
Velte, Clara Marika (Intern)
Meyer, Knud Erik (Intern)
George, William K (Intern)

Dynamic Propeller Shaft Speed Control

Department of Electrical Engineering
Automation and Control
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Maersk Maritime Technology
Lyngsøe Marine A/S
Propelco
Period: 01/04/2015 → 01/05/2017
Number of participants: 2
Project participant:
Galeazzi, Roberto (Intern)
Andersen, Poul (Intern)
Development of Improved Cavitation and Noise Radiation Prediction Methods for Marine Propellers

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Period: 01/03/2015 → 31/03/2018
Number of participants: 4
Phd Student:
Mirsadraee, Yasaman (Intern)
Supervisor:
Schöön, Johannes (Ekstern)
Walther, Jens Honore (Intern)
Main Supervisor:
Andersen, Poul (Intern)

Financing sources
Source: Internal funding (public)
Name of research programme: Industrial PhD
Project: PhD

Assessment, STrategy And Risk Reduction for Tsunamis in Europe
ASTARTE is organized to foster tsunami resilience in Europe, through innovative research on scientific problems critical to enhance forecast skills in terms of sources, propagation and impact. ASTARTE will employ lessons on coastal resilience learned from disaster surveys following tsunamis and hurricane surges. Within ASTARTE, we will acquire new information to complete the existing European knowledge base, and we will benefit from a stronger integration than ever attempted previously in the field. This will involve close cooperation with coastal populations, civil protection, emergency management and other local organizations.

ASTARTE is a collaborative project within the FP7-ENV2013 6.4-3

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Period: 01/11/2013 → 01/11/2016
Number of participants: 3
tsunamis, risk reduction
Acronym: ASTARTE
Project ID: 603839
Number of related Ph.D. students: 1
Project participant:
Fuhrman, David R. (Intern)
Sumer, B. Mutlu (Intern)
Eltard-Larsen, Bjarke (Intern)

Relations
Publications:
D5.10 - Interaction of the tsunami with the seabed. Implications for wind farms, aquaculture, coastal ecosystems and marine protected areas.

State-of-the-art laser Doppler system for turbulence measurements

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Intarsia Optics
Period: 23/10/2013 → 31/12/2015
Number of participants: 2
Project ID: 76503
Project participant:
Velte, Clara Marika (Intern)
Buchhave, Preben (Intern)

Project
Innovative Multi-purpose offshore platforms: planning, design and operation
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Administration
Period: 01/01/2012 → 31/12/2015
Number of participants: 8
Acronym: MERMAID
Project ID: FP7 OCEAN.2011-1 Grant Agreement no.: 288710
Project participant:
Sumer, B. Mutlu (Intern)
Hjelmager Jensen, Jacob (Intern)
Jensen, Bjarne (Intern)
Saremi, Sina (Intern)
Chen, Hao (Intern)
Tomaselli, Pietro (Intern)
Project Manager, organisational:
Carlberg, Lena Kristina (Intern)
Project Coordinator:
Christensen, Erik Damgaard (Intern)

Characterization of vortex generator induced structures in high Reynolds number wall bounded flow
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Ecole Centrale de Lille
Period: 01/04/2011 → 30/06/2012
Number of participants: 4
Project ID: 76177
Project participant:
Velte, Clara Marika (Intern)
Braud, Caroline (Ekstern)
Foucaut, Jean Marc (Ekstern)
Cuvier, Christophe (Ekstern)

Fluid Dynamic Characterization of Vortex Generators and Two-dimensional Turbulent Wakes
A numerical investigation of the physics of the wake induced by vortex generators.
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Period: 01/10/2010 → 30/09/2013
Number of participants: 3
Number of related Ph.D. students: 1
Project participant:
Fernandez Gamiz, Unai (Ekstern)
Egusquiza Estevez, Eduard (Ekstern)
Velte, Clara Marika (Intern)

Relations
Publications:
Fluid Dynamic Characterization of Vortex Generators and Two-dimensional Turbulent Wakes
Documents:
Thesis_190913
Project
Optimization of vortex generators on wind turbine blades
Experimental/theoretical optimization and model construction for the wake induced by vortex generators.

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Department of Wind Energy
Fluid Mechanics
Aeroelastic Design

LM Glasfiber A/S
Period: 01/02/2010 → 31/07/2013
Number of participants: 5
Project ID: 76031
Project participant:
Velte, Clara Marika (Intern)
Hansen, Martin Otto Laver (Intern)
Okulov, Valery (Intern)
Sørensen, Niels N. (Intern)
Fuglsang, Peter (Intern)

Activities:

Ocean Dynamics (Journal)
Period: 2018
David R. Fuhrman (Reviewer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Description
Guest Editor (Special Issue on Coastal Dynamics)
Degree of recognition: International

Related journal
Ocean Dynamics
1616-7341
Central database
Activity: Research › Journal editor

Coastal Engineering (Journal)
Period: 2017 → …
David R. Fuhrman (Reviewer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Description
Advisory Editorial Board
Degree of recognition: International

Related journal
Coastal Engineering
0378-3839
Journal of Waterway, Port, Coastal, and Ocean Engineering (Journal)
Period: 2017 → …
David R. Fuhrman (Reviewer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Description
Associate Editor
Degree of recognition: International

Related journal
Journal of Waterway, Port, Coastal, and Ocean Engineering
0733-950X
Central database
Activity: Research › Journal editor

Response prediction of vessel motions and sea state estimation from ships
Period: 17 Nov 2017
Ulrik Dam Nielsen (Guest lecturer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Description
Seminar at University of California - Berkeley @ Ocean Engineering.
Documents:
Response prediction and SSE (UCB Nov. 2017)

Related external organisation
University of California at Berkeley
United States
Activity: Talks and presentations › Conference presentations

Shipboard sea state estimation based on wave-induced response measurements
Period: 26 Sep 2017
Ulrik Dam Nielsen (Guest lecturer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Degree of recognition: International
Documents:
WaveEstim and DSS (MIT Sep. 2017)

Related external organisation
Massachusetts Institute of Technology
Cambridge, United States
Activity: Talks and presentations › Conference presentations

Complex Motion in Fluids Summer School
Period: 24 Sep 2017 → 29 Sep 2017
The school will consist of 16 lectures in total, given by 8 speakers (90'+60' each), contributed talks, poster sessions and other activities.

Degree of recognition: International

Related event

Complex Motion in Fluids Summer School
24/09/2017 → 30/09/2017
Cambridge, United Kingdom
Activity: Attending an event › Participating in or organising workshops, courses, seminars etc.

Coastal Dynamics 2017
Period: 12 Jun 2017 → 16 Jun 2017
David R. Fuhrman (Organizer)

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering

Description
Local Organizing Committee
Degree of recognition: International

Related event

Coastal Dynamics 2017
12/06/2017 → 16/06/2017
Helsingør, Denmark
Activity: Attending an event › Participating in or organising a conference

Oral presentation
Period: 17 May 2017
Malene Hovgaard Vested (Guest lecturer)

Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Degree of recognition: National
Documents:
AbstractDANSIS2017

Related event

Dansis Research Seminar 2017
Oral presentation
Period: 24 Apr 2017 → 28 Apr 2017
Malene Hovgaard Vested (Guest lecturer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Degree of recognition: International
Documents: EGU2017-7837

Related event
EGU General Assembly 2017: European GEosciences Union 2017
24/04/2017 → 28/04/2017
Vienna, Austria
Activity: Talks and presentations › Conference presentations

Harmonic Polynomial Cell method with Immersed Boundaries
Period: 3 Apr 2017 → 7 Apr 2017
Yanlin Shao (Speaker)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Degree of recognition: International
Documents: HYWEC2017_Shao

Related event
BCAM WORKSHOP HYDRODYNAMICS OF WAVE ENERGY CONVERTERS
03/04/2017 → 07/04/2017
Basque, Spain
Activity: Talks and presentations › Guest lectures, external teaching and course activities at other universities

New Concepts for Shipboard Sea State Estimation
Period: 1 Mar 2016
Ulrik Dam Nielsen (Speaker)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Description
Seminar talk/presentation given at University of California, Santa Barbara, USA.
Documents: SeaStateEstimation_UCSB

Related external organisation
Unknown external organisation
Activity: Talks and presentations › Conference presentations

Applied Ocean Research (Journal)
Period: 2015 → …
David R. Fuhrman (Reviewer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering

**Description**
**Editorial Board**
Degree of recognition: International

**Related journal**
**Applied Ocean Research**
0141-1187
BFI (2018): BFI-level 1, Scopus rating (2016): CiteScore 2.16 SJR 1.202 SNIP 1.75, ISI indexed (2013): ISI indexed yes,
Web of Science (2018): Indexed yes
Central database
Activity: Research › Journal editor

**Journal of Offshore Mechanics and Arctic Engineering (Journal)**
Period: 2015 → …
David R. Fuhrman (Reviewer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering

**Description**
**Associate Editor**
Degree of recognition: International

**Related journal**
**Journal of Offshore Mechanics and Arctic Engineering**
0892-7219
BFI (2018): BFI-level 1, Scopus rating (2016): CiteScore 0.89 SJR 0.668 SNIP 0.749, ISI indexed (2013): ISI indexed yes,
Web of Science (2018): Indexed yes
Central database
Activity: Research › Journal editor

**Maritime Engineering Education (at DTU): What are the challenges?**
Period: 9 Oct 2015
Ulrik Dam Nielsen (Lecturer)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering
Documents:
Maritime education - What are the challenges

**Related event**
**Danish Maritime Days: ATV Seminar**
05/10/2015 → 09/10/2015
Denmark
Activity: Talks and presentations › Conference presentations

**Response-based sea state estimation for onboard DSS - Safe and Efficient Marine Operations**
Period: 13 Nov 2014
Ulrik Dam Nielsen (Invited speaker)
Department of Mechanical Engineering
Fluid Mechanics, Coastal and Maritime Engineering

**Description**
Guest lecture given at NTNU, Trondheim, Norway, November, 2014.
WaveEstim and DSS (AMOS Workshop)

**Related external organisation**

**Unknown external organisation**

Activity: Talks and presentations › Conference presentations

**European Maritime Day**

Period: 19 May 2014 → 20 May 2014

Erik Damgaard Christensen (Speaker)

Department of Mechanical Engineering

**Fluid Mechanics, Coastal and Maritime Engineering**

**Description**

Koordinator for MERMAID.

**Documents:**

EMD Workshop Flyer

**Links:**


**Related event**

**European Maritime Day: Innovation driving Blue Growth**

19/05/2014 → 20/05/2014

Bremen, Germany

Activity: Talks and presentations › Conference presentations

**Prizes:**

**Green Challenge 2014**

Malene Hovgaard Vested (Recipient)

Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering

**Description**

First prize

**Details**

Awarded date: 27 Jun 2014

Degree of recognition: Local event: Green Challenge (Grøn Dyst) 2014

Prize: Prizes, scholarships, distinctions

**Outstanding Reviewer Award**

David R. Fuhrman (Recipient)

Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering

**Description**

ASCE Journal of Waterway, Port, Coastal, and Ocean Engineering

**Details**

Awarded date: 2015

Degree of recognition: International

Prize: Prizes, scholarships, distinctions