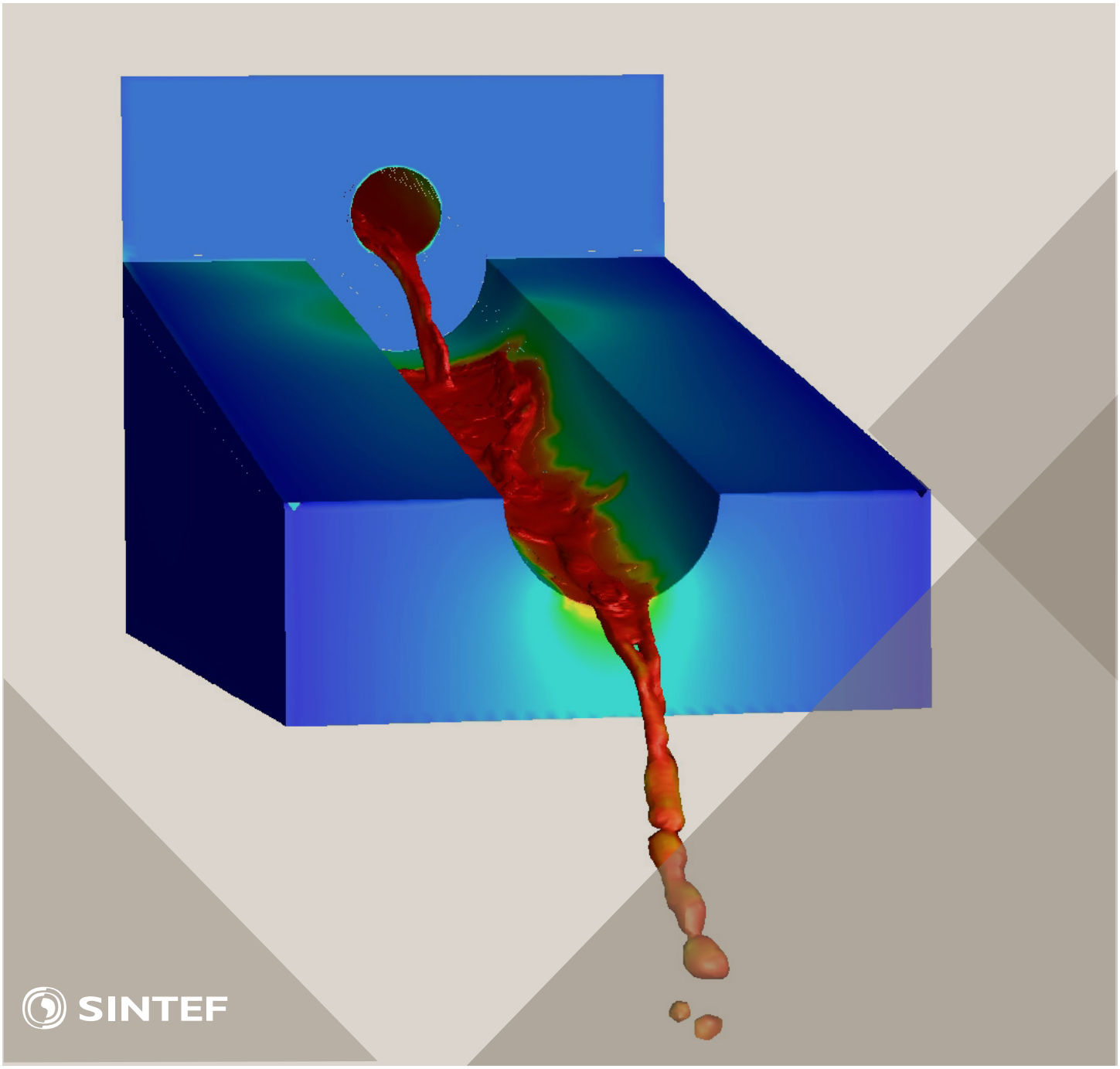
SINTEF
PROCEEDINGS 6

14th International Conference on CFD in
Oil & Gas, Metallurgical and Process Industries
SINTEF, Trondheim, Norway, October 12–14, 2020

Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries



SINTEF Proceedings

Editors:
Jan Erik Olsen, Jan Hendrik Cloete and Stein Tore Johansen

Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries

SINTEF, Trondheim, Norway
October 12–14, 2020

SINTEF Academic Press

SINTEF Proceedings 6
Editors: Jan Erik Olsen, Jan Hendrik Cloete and Stein Tore Johansen

Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries, SINTEF, Trondheim, Norway, October 12–14, 2020

Keywords:
CFD, fluid dynamics, modelling

Cover illustration: Tapping of metal by Jan Erik Olsen

ISSN 2387-4295 (online)
ISBN 978-82-536-1684-1 (pdf)

SINTEF Academic Press
Address: Børrestuveien 3
PO Box 124 Blindern
N-0314 OSLO
Tel: +47 40 00 51 00

www.sintef.no/community
www.sintefbok.no

14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries
SINTEF, Trondheim, NORWAY
October 12-14, 2020

SIMULATION OF ENTRAINMENT OF DROPLETS IN NON-RESOLVED WIND-WAVE FLOWS

Son Tung DANG[1 2 *], **Lim Chi HO**[3 †], **Stein Tore JOHANSEN**[2 ‡]

[1]SINTEF Industry, 7034, Trondheim, Norway

[2]NTNU Department of Energy and Process Engineering, 7491 Trondheim, NORWAY

[3]Delft University of Technology, Mekelweg 5, 2628 CD Delft, Netherlands

* E-mail: son.tung.dang@ntnu.no
† E-mail: limchi_ho@hotmail.com
‡ E-mail: stein.t.johansen@sintef.no

ABSTRACT

In this study we focus on a generic method to model the generation of droplets from a non-resolved simulation of free surface flow. The application is entrainment of droplets from the large-scale interface of waves interacting with a wind field. At first, the droplets generation rate will be calculated based on the local shear stress and other interface properties. Then, a special method is applied to detach liquid droplets from the continuous phase such that the total mass and momentum are conserved. Droplets trajectories will be computed using Lagrangian tracking method. Droplets that collide with the large-scale interface will again become continuous water and are removed from the computation. The interaction between droplets and surrounding gas will be accounted by adding a source term in momentum equations. In order to achieve a stable result, the source term will be handled implicitly. The model suggestions are explored and verified through simulations. The prospects of this modelling approach is discussed.

Keywords: Droplets, interface tracking, Shield number .

NOMENCLATURE

Greek Symbols

ρ	Mass density, $[kg/m^3]$
ν	Kinematic viscosity, $[m^2/s]$
κ	Surface curvature, $[1/m^2]$
λ	Wave length, $[m]$
σ	Surface tension constant, $[J/m^2]$
τ	Shear stress, $[Pa]$
ϕ	Level set function, $[m]$
α	Volume fraction,

Latin Symbols

p	Pressure, $[Pa]$.
$\mathbf{u}$	Velocity vector, $[m/s]$.
$\mathbf{g}$	Gravity vector, $[m^2/s]$.
D	Particle diameter, $[m]$.
c	Wave speed, $[m/s]$.
k	Wave number.

Sub/superscripts

f	fluid.
g	gas.
w	gas.
p	particle.
D	Drag.

INTRODUCTION

Liquid entrainment is a phenomenon by which parts of the liquid are injected into the gas field in the form of droplets. A good example of this phenomenon is shown when waves clash against marine vessels and off-shore structures. Due to the waves breaking, droplets will be generated and most likely end up on the dock of the vessel. Combine this with cold climates and these droplets will freeze onto the structure, which will cause mass accumulation on the vessel. Mass accumulation may cause big damages and even sink the vessel or structure in a small time period. This happens only in the most extreme cases, but still, a good physical understanding of droplet entrainment can help predict the movement of droplets and optimize designs of these vessels and offshore structures. Several experimental studies (Koga; Mestayer and Lefauconnier) have been performed to study wind-wave interaction and address the source of droplets generation. Numerical methods (Richter and Sullivan, a,b) are applied to study droplet particles transport in the turbulent Couette flow. Druzhinin et al. (Druzhinin *et al.*, 2017) examined the effect of droplets movement to turbulent airflow over the surface wave. The droplets are introduced to the simulation by injecting mechanism based on experimental knowledge. All computations are performed using the Direct Numerical Simulation (DNS) method. Tang et al. (Tang *et al.*) to studied the droplets generation due to the wind-wave interaction. In their work, the DNS method is performed to resolve various droplet scales. The droplet production will be handled by the interface tracking method. Therefore, there is no need for the droplet breakup model. However, DNS requires a lot of computational resources due to small grid sizes. As a result, it is not suitable for fast and large-scale simulation. In this study, we will develop a numerical model to generate a droplet spray from a surface wave in a coarse grid. Therefore, there is no need to use the DNS method.

MODEL DESCRIPTION

Governing equation and numerical implementation

The mass and momentum equation for incompressible flow accounting the interaction with paritcles are written as follows

$$\begin{aligned} &\nabla \mathbf{u} = 0, \\ &\frac{\partial \mathbf{u}}{\partial t} + \nabla(\mathbf{u}\mathbf{u}) = -\frac{\nabla p}{\rho}\mathbf{I} + \nu\nabla^2\mathbf{u} + \mathbf{g} + S_{m,p}. \end{aligned} \tag{1}$$

Where, $S_{m,p}$ is the source term which represents the exchange between discrete particles and continuous phase. In our study, we assume that the particle density is much larger than fluid density. Therefore, the source term can be computed as follows,

$$S_{m,p} = \frac{1}{V_f}\sum_{i=1}^{n_p} -\frac{m_{p,i}}{t_{p,i}}|\mathbf{u}_f - \mathbf{u}_{p,i}|[\mathbf{u}_f - \mathbf{u}_{p,i}], \tag{2}$$

where, n_p is the number of particles and

$$t_p = \frac{4}{3}\frac{\rho_p d_p}{\rho_f C_D}, \tag{3}$$

To conserve mass and the momentum equations, the cut-cell method based on the finite volume method (Dang *et al.*) is used. To prevent problems with pressure-velocity coupling, a staggered grid is introduced in this model. The source term is treated implicitly to ensure the stability for the numerical solution. To make the sharp interface between the two phases, a combination of the Level Set(LS) function and the Volume of Fluid(VOF) method was used to represent the free surface(Dang *et al.*; Wang *et al.*, 2009). The advantage of combining these two methods is that it uses the advantages of both methods. The advantage of VOF type models is that mass conversation can be simulated accurately. The disadvantage of using a VOF type model is that the surface normal vector is estimated less precisely based on the spatial derivatives, however, the level-set method can calculate these normal vectors more accurately. Thus by combining these two methods, both the mass conservation as well as the curvature of the free surface can be simulated accurately. The transport equations for the level set function ϕ and volume of fluid α are given as,

$$\begin{aligned} &\frac{\partial \alpha}{\partial t} + \mathbf{u}\cdot\nabla\alpha = S_\alpha, \\ &\frac{\partial \phi}{\partial t} + \mathbf{u}\cdot\nabla\phi = S_\phi, \end{aligned} \tag{4}$$

where, S_α and S_ϕ are source terms which represent the generation of droplets. The source term for volume fraction can be evaluated as.

$$S_\alpha = -\frac{\alpha_p}{\Delta t_d}, \tag{5}$$

where, α_p is a particle volume fraction which indicates the amount of Eulerian liquid that is converted to Lagrangian particles inside a given control volume, and where Δt_d is then time scale for droplet generation. In this work, we assume that the droplets are generated in an one time step. Firstly, we will solve the Eq. 4 without the source term. After extracting liquid volume fraction to form droplets the level set and liquid volume fraction in the continuous field will be updated accordingly. Equation 4 will be solved numerically without source term based on the interface reconstruction method. The details are given in the work of Griebel and Klitz(Griebel and Klitz). After droplets are produced they are considered as Lagrangian particles. The movement of Lagrangian particles is described as follows (Dehghani *et al.*),

$$\frac{\mathrm{d}\mathbf{x}_p}{\mathrm{d}t} = \mathbf{u}_p, \tag{6}$$

$$\begin{aligned} m_p\frac{\mathrm{d}\mathbf{u}_p}{\mathrm{d}t} =& \rho_p V_p \mathbf{g} - C_D\frac{\pi d_p^2}{8}\rho_f|\mathbf{u}_p - \mathbf{u}_f|(\mathbf{u}_p - \mathbf{u}_f) + \\ & \frac{\rho_f V_p}{2}\frac{\mathrm{d}(\mathbf{u}_f - \mathbf{u}_p)}{\mathrm{d}t} + \rho_f V_p\Big(\frac{\mathrm{d}\mathbf{u}_f}{\mathrm{d}t} - \mathbf{g}\Big). \end{aligned} \tag{7}$$

where, $\mathbf{x}_p(x, y)$ is the location of particle, $\mathbf{u}_p$ is the particle velocity vector. ρ_p is particle density, ρ_f is fluid density. d_p is the particle diameter. V_p is the particle volume. u_f is fluid velocity. C_D is drag coefficient

Model for droplets generation

Primary droplets

The Shields number is normally used to estimate the onset of movement in sedimentation in slurry-like streams. This number is based on the shear stress between particles and made dimensionless with the gravitational force by the following equation

$$Sh = \frac{\tau}{(\rho_p - \rho)\, gD}, \tag{8}$$

where, τ denotes the shear force, ρ_p is the particle density, ρ is the fluid density at the location where the particle is located, g is the gravitational constant, and D is particle diameter. The Shield number is modified for droplet entrainment in the gas field as,

$$Sh = \frac{\tau}{2\pi\sqrt{(\rho_w - \rho_g)\, g\sigma}}, \tag{9}$$

where, ρ_w is the water density, ρ_a is the gas density, and σ is the surface tension. The curved Shields number is computed based on shield number by

$$Sh_{curved} = \kappa \cdot Sh. \tag{10}$$

where, κ is surface curvature which can be computed as

$$\kappa = \nabla\cdot\left(\frac{\nabla\phi}{|\nabla\phi|}\right). \tag{11}$$

When the curved Shield number is larger than the critical Shield number ($Sh_c = 1.7.10^{-3}$) droplets will be generated from a wave. The number of droplets will be determined by

$$n_{droplets} = f(x).n_{droplets,max}, \tag{12}$$

where, $n_{droplets,max}$ is the maximum number of droplets generated at one time step and $f(x)$ is the scaling factor which is evaluated by

$$f(x) = \frac{e^{\left(\frac{Sh_{curve} - Sh_c}{Sh_c}\right)^n} - 1}{e^{\left(\frac{Sh_{curve} - Sh_c}{Sh_c}\right)^n} + 1} \tag{13}$$

The droplet diameter can be determined based on the breakup mechanism presented in the work of Marmottant and Villermaunx (MARMOTTANT and VILLERMAUX, 2004).

RESULT AND DISCUSSION

Wave equation

The initial wave geometry is given by the analytical solution of the third-order Stokes wave as

$$\eta(x) = a_0 \cos\left[k(x-ct)\right] + \frac{1}{2}a_0\epsilon_0 \cos\left[2k(x-ct)\right] + \frac{3}{8}a_0\epsilon_0^2 \cos\left[3k(x-ct)\right], \tag{14}$$

where, η is the wave surface height, a_0 is the wave amplitude, $k = 2\pi/\lambda$ is the wave number, $\epsilon_0 = ka_0$ is the initial wave steepness, and $c = \omega/k$ is the wave phase speed. In our study, ω is evaluated by,

$$\omega = \sqrt{gk(1+\epsilon^2)}. \tag{15}$$

The values of parameters are given in table 1. The water velocities corresponding to the wave are defined by

$$u = a_0\omega\frac{\cosh(ky)}{\sinh(ky_{water})}\cos\left[k(x-ct)\right] \text{ for } y \leq y_{water}$$
$$v = a_0\omega\frac{\sinh(ky)}{\sinh(ky_{water})}\sin\left[k(x-ct)\right] \text{ for } y \leq y_{water}, \tag{16}$$

where, y is the water height regarding to the bottom of the computational domain.

Table 1: Computational parameters for the wave equation

λ	c	T	ϵ_0
5	3.19	1.57	0.55

Wind equation

In strong wind conditions, the wind can be described by the logarithmic law (Holmes) as

$$u_a = \frac{u_*}{\kappa}\left[ln\left(\frac{y-d}{y_0}\right)\right] \text{ for } y > y_{water} \tag{17}$$

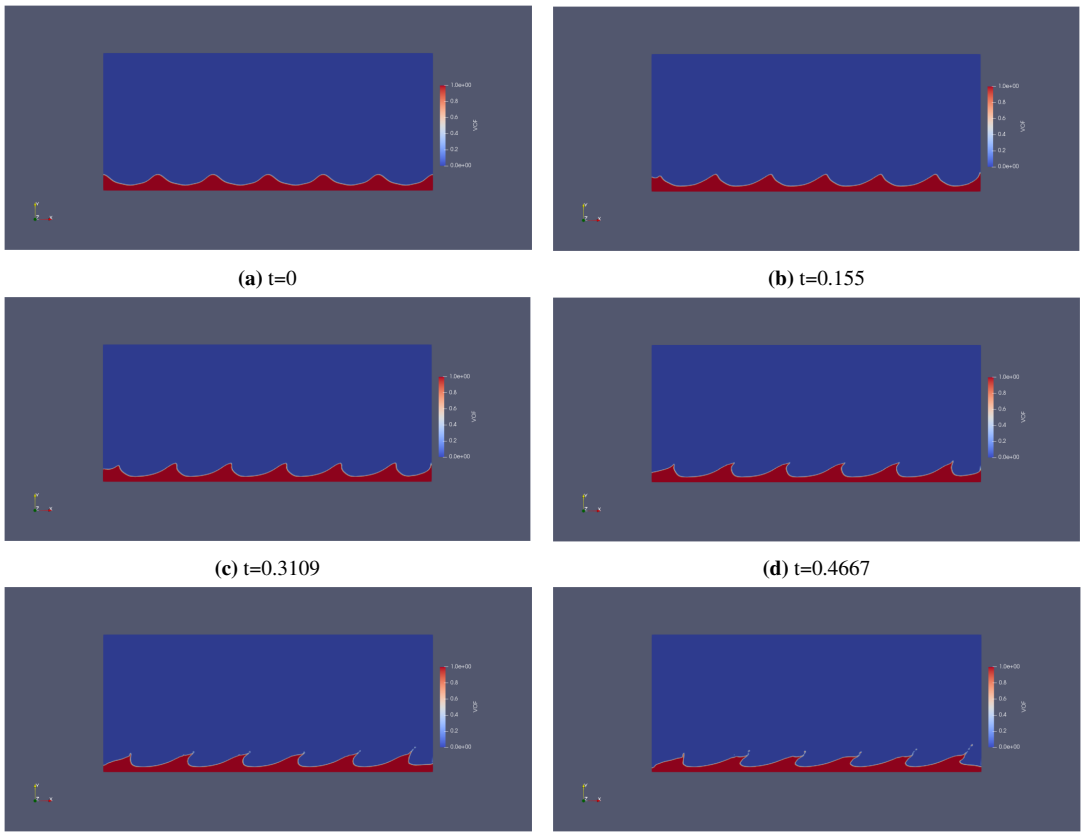
where, u_* = 0.27m/s is the friction velocity, $\kappa \sim 0.41$ is the Von Karman constant, $d = 0$ is the zero plane displacement, and $y_0 = 0.0002$ is the surface roughness (Holmes). The wave interface is depicted in figure 1. The blue and the red interfaces are the air and water phases respectively. Due to blowing wind, the wave tip becomes unstable over time. As result, the interface breaks at the t=0.6015 in figure 2e and t=0.722 in figure 2f. Since this study is only interested in droplet generation at the surface, only grid points at the liquid interface will be given a value for the Shields number. This way, the structure of the wave was visible as well as the values of the Shields number. To prove that the theory of using the Shields number to create droplets would work, the highest values of the Shields number should be at the top of the wave. This is due to the fact that spume droplets are being generated at the wave crest in real scenarios if the velocity difference between the two phases is big enough. Figure 2 shows the highest values of curved Shield number are being created at the wave crest which fits with our assumption. Since the highest values were found at the wave crest, it was fairly easy to determine the critical Shields number ($Sh_c = 0.0017$).The value where the Shields number was higher at the wave crest and lower everywhere else in the domain would be equal to the critical Shields number. As depicted in figure 3, the grid points that exceed the critical Shields number (depicted in white) are indeed at the location of the wave crest. In figure 4, particles are generated at the wave interface. From figures 3 and 4, it can be seen that at the grid points where the local Shields number surpasses the critical Shields number, particles are being generated.

CONCLUSIONS

Numerical simulations were carried out by using an in-house CLS-VOF model (Simcoflow) to describe droplet entrainment in breaking-wave scenarios. The wave in this study follows the analytical of the third-order Stokes wave. The shear force will play a significant role when the velocity difference between the air- and water stream is sufficient enough. This causes the wave to deform and liquid sheets will be developed. These liquid sheets will split up in smaller liquid fragments that are called ligaments. If the shear force is big enough, small globes of liquid will be sheared off the ligament top and droplets are being generated. The criterion for primary droplet entrainment was based on the curved Shields number. Currently, the critical Shields number is a constant value, however, with new insights, this could or maybe should be altered. A good starting point could be to develop a function that is affected by different variables, such as the turbulent properties (fluctuating velocities at local disturbances), wave properties (wavelength, wave height, or amplitude), the velocity at the wave tip, or what is used now in the model, implementing a curvature in the Shields number. When particles are being generated, the liquid fractions that are sheared off will turn into Lagrangian particles, where it will use the two-way coupling model until the particles are out of the computational domain or back in the liquid stream.

ACKNOWLEDGEMENT

We would like to thank the Norwegian Research Council for funding our works through researcher program SprayIce (project 244386) and ICELOAD (308780)

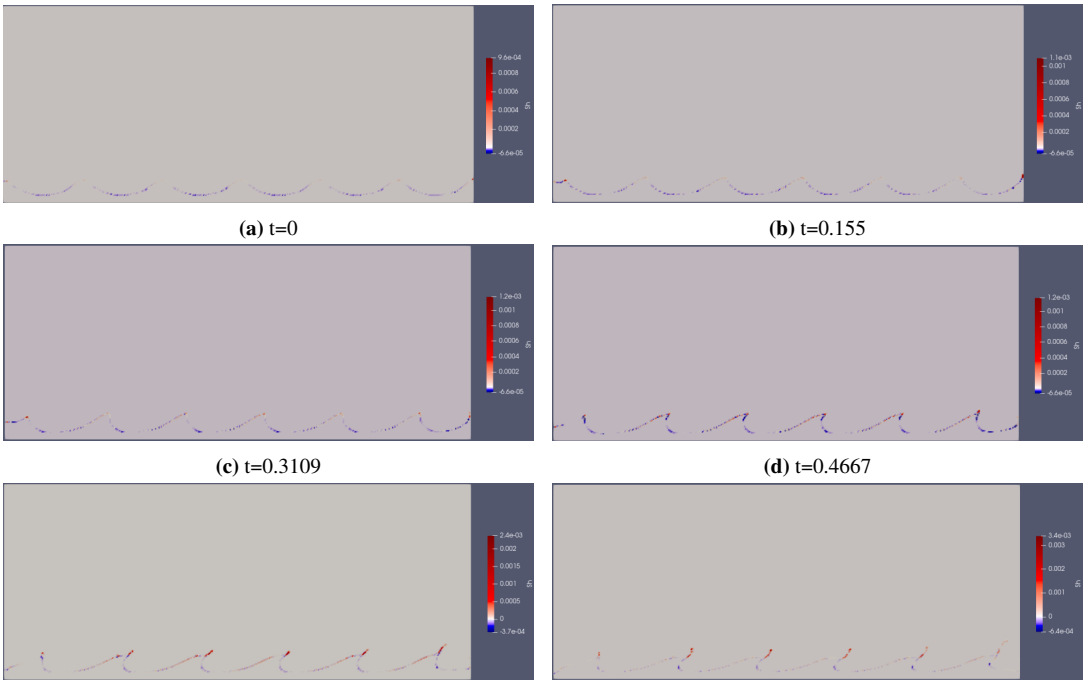


(a) t=0 **(b)** t=0.155

(c) t=0.3109 **(d)** t=0.4667

(e) t=0.6015 **(f)** t=0.722

Figure 1: Wave interface at different time steps

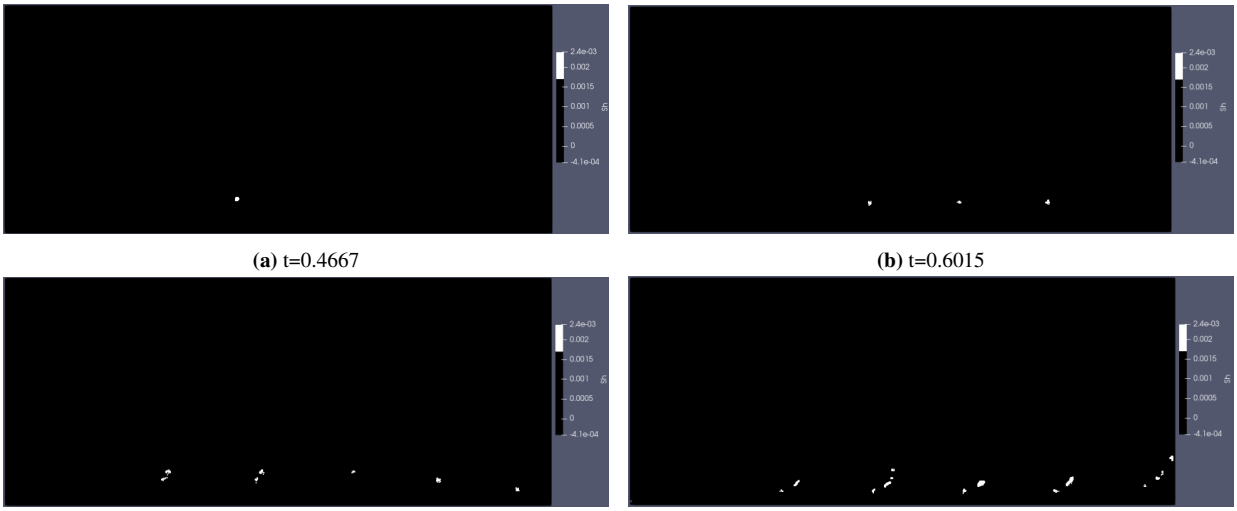


(a) t=0 **(b)** t=0.155

(c) t=0.3109 **(d)** t=0.4667

(e) t=0.6015 **(f)** t=0.722

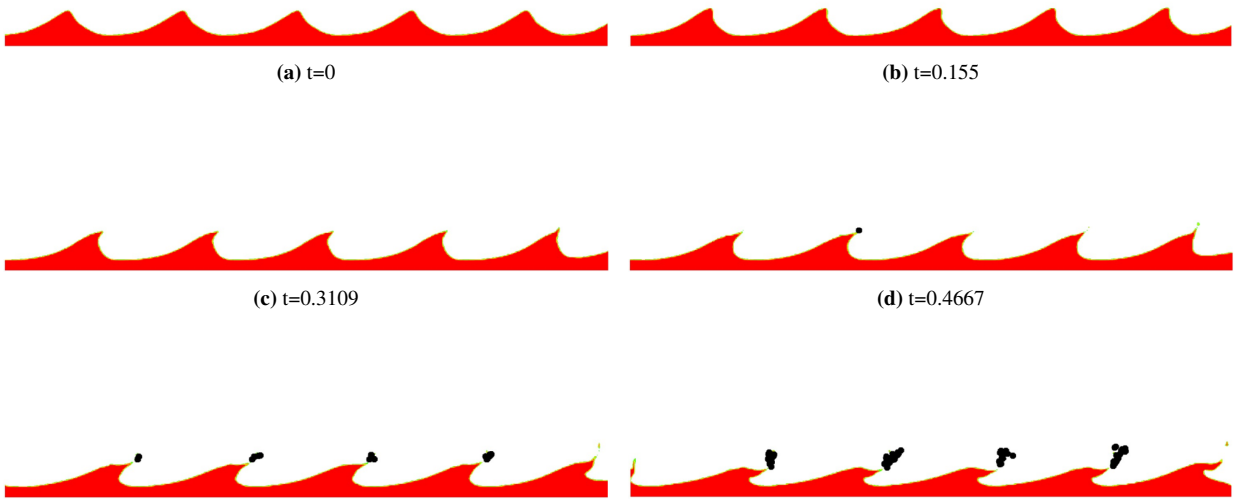
Figure 2: The values of the curved Shields number



(a) t=0.4667 **(b)** t=0.6015

(c) t=0.722 **(d)** t=0.835

Figure 3: The values that are above the critical Shields number



(a) t=0 **(b)** t=0.155

(c) t=0.3109 **(d)** t=0.4667

(e) t=0.6015 **(f)** t=0.722

Figure 4: Particle generation at different time steps

REFERENCES

DANG, S.T., MEESE, E.A., MORUD, J.C. and JOHANSEN, S.T. (). "Numerical approach for generic three-phase flow based on cut-cell and ghost fluid methods". **91(9)**, 419–447.

DEHGHANI, S.R., MUZYCHKA, Y.S. and NATERER, G.F. (). "Droplet trajectories of wave-impact sea spray on a marine vessel". **127**, 1–9.

DRUZHININ, O., TROITSKAYA, Y. and ZILITINKEVICH, S. (2017). "The study of droplet-laden turbulent airflow over waved water surface by direct numerical simulation". *Journal of Geophysical Research : Oceans*, **122(3)**, 1789–1807.

GRIEBEL, M. and KLITZ, M. (). "CLSVOF as a fast and mass-conserving extension of the level-set method for the simulation of two-phase flow problems". **71(1)**, 1–36.

HOLMES, J.D. (). *Wind Loading of Structures*. CRC Press.

KOGA, M. (). "Direct production of droplets from breaking wind-waves -its observation by a multi-colored overlapping exposure photographing technique". **33(6)**, 552–563.

MARMOTTANT, P. and VILLERMAUX, E. (2004). "On spray formation". *Journal of Fluid Mechanics*, **498**, 73â€"111.

MESTAYER, P. and LEFAUCONNIER, C. (). "Spray droplet generation, transport, and evaporation in a wind wave tunnel during the humidity exchange over the sea experiments in the simulation tunnel". **93(C1)**, 572–586.

RICHTER, D.H. and SULLIVAN, P.P. (a). "Momentum transfer in a turbulent, particle-laden Couette flow". **25(5)**, 053304.

RICHTER, D.H. and SULLIVAN, P.P. (b). "Sea surface drag and the role of spray". **40(3)**, 656–660.

TANG, S., YANG, Z., LIU, C., DONG, Y.H. and SHEN, L. (). "Numerical Study on the Generation and Transport of Spume Droplets in Wind over Breaking Waves". **8(12)**, 248.

WANG, Z., YANG, J., KOO, B. and STERN, F. (2009). "A coupled level set and volume-of-fluid method for sharp interface simulation of plunging breaking waves". *International Journal of Multiphase Flow*, **35(3)**, 227–246.