

# REEF3D :: User Guide



REEF3D 21.02

## **Contact**

Hans Bihs

Associate Professor  
Marine Civil Engineering  
NTNU Trondheim

[hans.bihs@ntnu.no](mailto:hans.bihs@ntnu.no)

## **Team**

Arun Kamath  
Csaba Pakozdi  
Tobias Martin  
Weizhi Wang

# Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>Functions</b>	<b>3</b>
2.1	A :: Hydrodynamic models . . . . .	3
2.2	B :: Boundary Conditions . . . . .	11
2.3	C :: Concentration . . . . .	28
2.4	D :: Discretization . . . . .	31
2.5	F :: Free Surface . . . . .	33
2.6	G :: Grid Options . . . . .	39
2.7	H :: Heat Transfer . . . . .	40
2.8	I :: Initialization . . . . .	44
2.9	M :: MPI . . . . .	46
2.10	N :: Numerical Options . . . . .	46
2.11	P :: Printer . . . . .	50
2.12	S :: Sediment . . . . .	59
2.13	T :: Turbulence . . . . .	66
2.14	W :: Water and Material Properties . . . . .	68
2.15	X :: 6DOF . . . . .	72
<b>3</b>	<b>Compiling the Code</b>	<b>81</b>
3.1	Install Required Packages . . . . .	81
3.1.1	Installing GCC GNU compiler on Mac OS X . . . . .	81
3.1.2	Installing OpenMPI . . . . .	81
3.1.3	Installing HYPRE . . . . .	82
3.2	User Installation: Makefile . . . . .	83
3.3	Developer Installation: CodeLite . . . . .	83
3.4	Installing REEF3D on Windows 10 . . . . .	84
3.4.1	Getting Unix Terminal on Windows 10 . . . . .	84
3.4.2	Installing Compilers . . . . .	85
3.4.3	Additional step for Windows for visualization using Paraview . . . . .	85

<b>4</b>	<b>Running the Code</b>	<b>87</b>
4.1	DIVEMesh . . . . .	87
4.2	REEF3D . . . . .	87
<b>5</b>	<b>Post-Processing</b>	<b>89</b>
5.1	Paraview . . . . .	89
5.2	Additional step for Windows users to use paraview . . . . .	89
5.3	Visualising Two-Phase Flow Results in Paraview . . . . .	89
<b>6</b>	<b>Running the Code</b>	<b>93</b>
<b>7</b>	<b>Tutorial   REEF3D::SFLOW</b>	<b>95</b>
7.1	2D linear waves . . . . .	95
<b>8</b>	<b>Tutorial   REEF3D::FNPF</b>	<b>97</b>
<b>9</b>	<b>Tutorial   REEF3D::CFD</b>	<b>99</b>
9.1	2D Dam Break . . . . .	99
9.2	3D Dam Break with Obstacle . . . . .	101
9.3	2D Vortex Shedding . . . . .	103
9.4	Flow through a Narrow Contraction . . . . .	104
9.5	Flow around a Circular Pier . . . . .	106
9.6	Rectangular Wave Tank . . . . .	108
9.7	Rectangular Wave Tank with DWG and AWA . . . . .	110
9.8	Rectangular Wave Tank with Wavemaker . . . . .	112
9.9	Wave Propagation over a Submerged Bar . . . . .	115
9.10	Plunging Breaking Waves over Slope . . . . .	119
9.11	Plunging Breaking Waves over Slope with Non-uniform Grid . . . . .	123
9.12	Shoaling over Irregular Topography . . . . .	124
9.13	Non-Breaking Wave Forces . . . . .	128
9.14	Breaking Wave Forces . . . . .	130

# 1. Introduction

Todays ever increasing computational resources make it possible to compute incredible large flow problems. Every company and research institution interested in performing serious CFD (computational fluid dynamics) simulations has now the real option to buy and maintain supercomputers at reasonable costs. The limiting factor of such simulations becomes less the problem size but rather the time it takes for the engineer to generate grids, start simulations and post-process the results. With the increasing speed of high-performance computer clusters, the amount of grid points is growing. As Peric [2004] writes, in the future the "computational points in a mesh will be counted in billions", the issue of grid quality will become less relevant because the mesh will be very fine in any case.

With these observations in mind, the numerical model REEF3D was designed under the following premises:

1. hydraulic, coastal, marine and environmental engineering
  - level set method for complex free surface flow
  - open channel flow boundary conditions
  - numerical wave tank boundary conditions
  - sediment transport
2. ease of use: grid generation as limiting factor of user productivity
  - immersed boundary
  - STL file input
  - easy natural bathymetry handling
3. increasing computer performance enables larger computations, but can only be exploited in parallel
  - full parallelization based on the domain decomposition strategy and MPI
4. ease of use: stable, accurate and fast numerical simulation
  - staggered grid: tight velocity-pressure coupling
  - very accurate and stable WENO discretization
  - adaptive time-stepping for maximum stability
5. numerical tool should be easy to maintain, changes should be possible
  - C++ Programming
  - object oriented code
  - MPI, an industry standard for high-performance computing

The source code of REEF3D is available at [www.reef3d.com](http://www.reef3d.com) and is published under the GPL license, version 3. Open-source CFD software has several advantages. All code developments have the potential to benefit a large audience, including students, researchers and engineers working on academic or commercial problems. The usage of open-source CFD programs comes at no cost. It gives the people who work with the software the opportunity to gain insights into how the numerical model works and build up valuable competence and experience in this field. This makes contributions to open-source software a sustainable investment in a lot of ways. Open-source also means that REEF3D is more flexible and more open to innovation.

## 2. Functions

A file called “ctrl.txt” is used to give the necessary data to the program. The structure of this file is as follows: a capital letter and a number are used to enable different algorithms followed by the corresponding variables. The “ctrl.txt” file needs to be stored in the same location as the executable REEF3D. In front of the variables in the definition of the algorithms you find either an “int” or a “float”. For “int” an integer needs to be given, for “float” a floating-point number is expected by the program. It is possible to enter comments in the “ctrl.txt” by entering the `#` followed by the comment, but take care not to include capital letters as that will cause the program to look for a valid command option.

Most of the options in this section (A) will determine the discretization and other parameters for the first three models listed above. The validity of the functions for the particular models is denoted by the filled circle beside the function under the columns S (SFLOW), F (FNPF), N (NSEWAVE) and C (CFD) in all the sections for functions in this manual.

Commands with an asterix (\*) can be issued multiple times to create multiple instances of the call.

### 2.1 A :: Hydrodynamic models

The commands in this section are used to select the hydrodynamic model that is to be activated for the simulation. This will determine the governing equations that will be used for the simulations. The different models available in REEF3D are:

- **S** : SFLOW: A depth-averaged shallow water equations model, that solves the non-hydrostatic pressure.
- **N** : NSEWAVE: This model solves the Navier-Stokes in three dimensions, and is suitable for large scale wave modeling with a single-value free surface.
- **F** : FNPF: This model solves the Laplace equation.
- **C** : CFD : This solved the RANS equations.

Valid for:

S F N C



**A 10** [int](#) Hydrodynamic Model

**2** SFLOW

**3** FNPF

**5** NSEWAVE

**6** CFD

**default:** 6



**A 210** [int](#) Time scheme for SFLOW momentum equations

**2** 2nd-order Runge-Kutta

**3** 3rd-order Runge-Kutta

**default:** 3



**A 211** [int](#) Convection scheme for SFLOW momentum equations

**1** FOU

**2** CDS2

**4** WENO FLUX

**5** WENO HJ

**6** High Resolution TVD scheme with Minmod limiter

**7** High Resolution TVD scheme with van Leer limiter

**8** High Resolution scheme with Smart limiter

**default:** 0



**A 212** [int](#) Turn on diffusion for SFLOW velocities

**0** OFF

**1** explicit

**2** implicit

**default:** 0

● ○ ○ ○

**A 214 int** Turn on convection for vertical SFLOW velocity  $w_s$

This option should be turned off for open-channel flow.

**0** OFF

**1** ON

**default:** 1

● ○ ○ ○

**A 217 int** Boundary conditions at walls

**1** slip

**2** no-slip

**default:** 2

● ○ ○ ○

**A 218 int** Turn on roughness

**0** OFF

**1** ON

**default:** 0

● ○ ○ ○

**A 220 int** Pressure scheme for SFLOW dynamics pressure  $w_s$

**0** hydrostatic

**1** linear non-hydrostatic pressure

**2** quadratic non-hydrostatic pressure

**default:** 0

● ○ ○ ○

**A 223 double** Blending of previous and current pressure for gradient calculation

**default:** 0.5

● ○ ○ ○

**A 240 int** Free surface scheme for SFLOW

**0** OFF

**1** ON

**default:** 1

● ○ ○ ○

**A 241** `int` Discretization scheme for SFLOW water levels

**1** FOU

**2** CDS2

**4** WENO FLUX

**default:** 1

● ○ ○ ○

**A 242** `int` Hydrostatic pressure for shallow areas

**0** OFF

**1** ON

**default:** 0

● ○ ○ ○

**A 243** `int` Turn on wetting and drying algorithm

**0** OFF

**1** ON

**default:** 1

● ○ ○ ○

**A 244** `double` Use absolute wetting-drying threshold value

**default:** 0.00005

● ○ ○ ○

**A 245** `double` Use relative wetting-drying threshold value (dx multiplier)

**default:** 0.001

● ○ ○ ○

**A 246** `int` Turn on breaking wave algorithm (turns off dynamics pressure for breaking waves)

**0** OFF

**1** ON

**default:** 1

● ○ ○ ○

**A 247** `double` Breaking wave threshold parameter

**default:** 0.6

● ○ ○ ○

**A 248** `int` Turn on breaking wave persistance algorithm (turns off dynamics pressure for breaking waves)

**0** OFF

**1** ON

**default:** 1

● ○ ○ ○

**A 249** `double` Breaking wave persistence parameter

**default:** 0.3

● ○ ○ ○

**A 260** `int` Turbulence model

**0** OFF

**3** Prandtl length scale model

**4** Parabolic eddy-viscosity model

**default:** 0

● ○ ○ ○

**A 261** `double` Length scale factor for length scale turbulence model

**default:** 0.267

● ○ ○ ○

**A 262** `double` Factor for parabolic eddy-viscosity model

**default:** 0.0667

○ ● ○ ○

**A 310** `int` Time scheme for FNPF algorithm

**3** 3rd-order TVD Runge-Kutta

**4** 4th-order Runge-Kutta

**default:** 3

○ ● ○ ○

**A 311** `int` Spatial discretization for KFSFBC and DFSFBC

**0** OFF

**1** CDS2

**2** CDS4

**4** WENO5

**5** WENO5 with wetting-and-drying

**6** CDS6

**7** WENO7

**default:** 5

○ ● ○ ○

**A 312** `int` Spatial discretization for sigma coordinate system

**2** CDS2

**3** CDS4

**default:** 2

○ ● ○ ○

**A 320** `int` Order of spatial discretization for Laplace equation

**1** CDS2

**2** CDS4

**default:** 1

○ ● ○ ○

**A 321** `int` Order of KBEDBC for 4th-order Laplace equation

**1** 2nd-order

**2** 4th-order

**default:** 1

**A 341** `double` coastline damping distance factor for dxm

**default:** 0.0

**A 342** `double` coastline damping absolute distance

**default:** 0.0

**A 341** `double` coastline relaxation zone factor times mesh size

**default:** 0.0

**A 342** `double` coastline relaxation zone absolute distance

**default:** 0.0

**A 343** `int` Turn of wetting-and-drying

**0** OFF

**1** ON

**default:** 1

**A 344** `double` Use absolute wetting-drying threshold value

**default:** 0.00005

**A 345** `double` Use relative wetting-drying threshold value (dx multiplier)

**default:** 0.001

**A 346** `double` Damping viscosity within the coastline, modulated by the coastline relaxation zone.

**default:** 0.0

**A 350** `int` Breaking wave algorithm

**0** OFF

**1** viscosity based

**2** filter based

**default:** 0

**A 351** `int` Breaking wave detection

**0** OFF

**1** shallow water

**2** deep water

**3** shallow and deep water

**default:** 0

**A 352** `int` Added filtering for breaking when viscosity breaking algorithm is used for additional robustness, for breaking in:

**0** OFF

**1** shallow water

**2** deep water

**3** shallow and deep water

**default:** 1

**A 354** `double` Breaking wave threshold parameter shallow water

**default:** 0.6

**A 355** `double` Breaking wave slope threshold deep water

**default:** 1.25

**A 361** `int` Breaking wave filter algorithm: outer iterations

**default:** 5

**A 362** `int` Breaking wave filter algorithm: inner iterations

**default:** 2

**A 365** `double` Breaking wave viscosity

**default:** 1.86

**A 368** `int` Breaking wave viscosity based algorithm inside numerical beach relaxation zone

**0** OFF

**1** ON

**default:** 0

**A 369** `double` Breaking wave viscosity inside the numerical beach relaxation zone

**default:** 1.86

## 2.2 B :: Boundary Conditions

Valid for:

S F N C

**B 10** `int` Wall functions for the velocities

**0** OFF

**1** ON

**default:** 0

○ ○ ● ●

**B 20** [int](#) Slip or no-slip boundary conditions for velocities

**1** slip

**2** no-slip

**default:** 2

○ ○ ○ ●

**B 26** [int](#) Boundary condition implementation for level set method

**1** Neumann

**2** Extend

**3** Extend at the bed only

**default:** 1

● ○ ● ●

**B 50** [double](#) Global wall roughness

**default:** 0.001

○ ○ ○ ●

**B 51** [double](#) Wall 1 roughness

**default:** nan

○ ○ ○ ●

**B 52** [double](#) Wall 2 roughness

**default:** nan

○ ○ ○ ●

**B 53** [double](#) Wall 3 roughness

**default:** nan

○ ○ ○ ●

**B 54** double Wall 4 roughness

**default:** nan

○ ○ ○ ●

**B 55** double Wall 5 roughness

**default:** nan

○ ○ ○ ●

**B 56** double Wall 6 roughness

**default:** nan

● ○ ● ●

**B 60** int Enable ioFlow for open channel flow

This parameter is important for open channel flow calculations. The ioFlow module makes sure, that the inflow discharge as given in W10 is constant throughout the computation even though the inflow water level may change. This is the counterpart to B90 for open channel flow. When selecting the hydrograph inflow option, make sure that a file with the name "hydrograph.dat" is present in the simulation folder. When selecting the hydrograph outflow option, make sure that a file with the name "hydrograph\_out.dat" is present in the simulation folder. The data format of the hydrograph file consists of two columns, the first one gives the time in seconds and the second column the discharge in  $m^3/s$ .

**0** OFF

**1** constant inflow

**2** hydrograph inflow

**3** hydrograph outflow

**4** hydrograph inflow and outflow

**default:** 0

○ ○ ○ ●

**B 61** int Inflow profile for ioFlow

**1** constant velocity

**2** logarithmic profile bed

**3** constant velocity only for phase 1

**4** logarithmic profile only for phase 1, bed

**5** logarithmic profile only for phase 1, sidewalls and bed

**default:** 2

○ ○ ○ ●

**B 70** \* `double` value, `double` distance, `double` line angle, `double` line x-origin, `double` line y-origin; for use of relaxation method for fixed water level

**default:** na

○ ○ ○ ●

**B 71** \* `double` value, `double` distance, `double` line angle, `double` line x-origin, `double` line y-origin; for use of relaxation method for fixed water level for initialization only

**default:** na

○ ○ ○ ●

**B 75** `int` Outflow boundary condition

**1** zero gradient

**2** convection condition

**default:** 1

● ○ ○ ●

**B 77** `int` Pressure outflow boundary conditions For this parameter to be activated, B 76 needs to be selected in such a way that the pressure outlet boundary condition gets prescribed.

**1** controlled outflow

**2** free stream outflow

**default:** 1

○ ○ ○ ●

**B 81** `double` x-location for focus, y-location for focus, time of wave focusing

(For a 2D simulation set y-location to 0.0)

**default:** 0.0 ;0.0; 0.0

● ● ● ●

**B 82** `int` Type of focusing amplituted calculation

- 1** NewWave
  - 2** Spectrum
  - 3** Fixed wave steepness
  - 4** Constant wave amplitude
- default:** 1

● ● ● ●

**B 83** **double** Gain factor for fixed wave steepness focused waves

**default:** 0.0025

● ● ● ●

**B 85** **int** Wave spectrum for irregular waves In case of the spectrum file, provide a "spectrum-file.dat", with the two columns " $\omega$ " and "S".

- 1** Pierson-Moskowitz
- 2** JONSWAP
- 3** Torsethaugen
- 4** Wavepackets
- 5** Wavepackets for steep waves
- 6** Gaussian Wavepackets
- 10** spectrum file
- 21** Goda JONSWAP

**default:** 1

● ● ● ●

**B 86** **int** Number of regular waves for irregular wave generation

**default:** 10

● ● ● ●

**B 87** **double**  $\omega_s$  and  $\omega_e$  for irregular wave generation When not given, the model will calculate the values automatically.

**default:** 0.0 ; 0.0

● ● ● ●

**B 88** `double`  $\gamma$  for irregular wave generation with JONSWAP spectrum

**default:** 3.3

● ● ● ●

**B 89** `int` Turn on wave generation optimization through space-time decomposition for relaxation method. Currently supported for the following wave theory options:

B 92 5, B 92 3,1 B 92 41, B 92 51

**0** OFF

**1** ON

**default:** 0

● ● ● ●

**B 90** `int` Enable ioWave for numerical wave tank This parameter turns on the numerical wave tank. It is the counterpart to B60 for wave simulations. Different options for generating and dissipating waves exist.

**0** OFF

**1** ON

**default:** 0

● ● ● ●

**B 91** `double` wave height, wave length

The main wave parameters are given here. For regular waves the first values is  $H$ , whereas for irregular waves, the first value is the significant wave height. Alternatively B93 can be used, when the wave period is known.

**default:** 0.0 ; 0.0

● ● ● ●

**B 92** `int` Wave type Different wave theories can be used as input for the numerical wave tank. It is important, to check beforehand whether the selected theory fits the given wave conditions consisting of wave height, wave length/wave period and still water level.

For the piston wavemaker theory with  $\eta$  timeseries input, provide a "wavemaker\_eta.dat" input file with two columns "t" and " $\eta$ "

For the piston and flap wavemaker theories, provide a "wavemaker.dat" input file with two columns "t" and "X" or "t" and " $\beta$ ", depending on input from B 116.

For the double-hinged flap wavemaker, two “X” or two “ $\beta$ ” columns are expected. For giving two “X” columns, this will be the total exitation at the end of each flap. For giving two “ $\beta$ ” columns, the second angle is the additional exitation of the second flap.

For the wave reconstruction, provide a “waverecon.dat” input file with three columns giving “A” and “ $\omega$ ” and phase “ $\phi$ ” for each of the harmonic wave components.

- 1** Shallow Water
  - 2** Intermediate
  - 3** Deep Water
  - 4** 2nd-order Stokes Waves
  - 5** 5th-order Stokes Waves (Fenton)
  - 6** Shallow Water Cnoidal Waves
  - 7** 1st-order Cnoidal Waves
  - 8** 5th-order Cnoidal Waves
  - 9** 1st-order Solitary Wave
  - 10** 3rd-order Solitary Wave
  - 11** 5th-order Stokes Waves (Skjelbreia)
  - 20** Piston Wavemaker based on  $\eta$  timeseries
  - 21** Piston Wavemaker
  - 22** Flap Wavemaker
  - 23** Double-hinged Flap Wavemaker
  - 31** 1st-order Irregular Waves
  - 32** 2nd-order Irregular Waves (Longuet-Higgins)
  - 33** 2nd-order Irregular Waves (Schäffer)
  - 41** 1st-order Focused Waves
  - 42** 2nd-order Focused Waves (Longuet-Higgins)
  - 43** 2nd-order Focused Waves (Schäffer)
  - 51** Wave Reconstruction with 1st-order Irregular Waves
  - 52** Wave Reconstruction with 2nd-order Irregular Waves (Longuet-Higgins)
  - 53** Wave Reconstruction with 2nd-order Irregular Waves (Schäffer)
  - 61** Wave Coupling WCP, FNPF to CFD
- default:** 0



### **B 93** [double](#) wave height, wave period

The main wave parameters are given here. For regular waves the first values is  $H$ , whereas for irregular waves, the first value is the significant wave height. Alternatively B91 can be used, when the wavelength is known.

**default:** 0.0 ; 0.0

● ● ● ●

**B 96** `double` Wave Generation and Beach parameters, dist1, dist2

dist1 ist the distances from the beginning of the wave flume (default, can be changed, see B 106 ), dist3 is the distance measured from the end of the wave flume (default, can be changed, see B 107).

**default:** 0.0 ; 0.0

● ● ● ●

**B 97** `double` Wave Generation Direction in degree °

**default:** 0.0

● ● ● ●

**B 98** `int` Wave Generation Method Relaxation Method 2 ramps up the values for the input wave within one zone following the function by Jacobsen et al. [2012]. Relaxation Method 1 uses two zones to generate the waves Afshar [2010]. For the Dirichlet type, waves are generated by assigning the values for the free surface and the velocity in the inflow boundary only.

**0** OFF

**2** Relaxation Method 2

**3** Dirichlet

**4** Dirichlet with Active Absorption, based on shallow water theory

**default:** 0

● ● ● ●

**B 99** `int` Numerical Beach Method Both relaxation methods dissipate the wave energy by ramping the free surface down to the still water level, the velocity to the zero and the pressure to its hydrostatic distribution for still water conditions. Typically the length of the dissipating relaxation zone should be two wavelengths. The Active Wave Absorption avoids reflections by generating a wave opposite of the reflected wave, thus canceling it out. The method is more efficient in computational terms, as it does not require additional space.

In order to use the Active Wave Absorption beach, keep in mind to use the active beach boundary condition in the C data set in DIVEMesh.

**0** OFF

**1** Relaxation Method 1

- 2** Relaxation Method 2
  - 3** Active Wave Absorption, based on shallow water theory
  - 4** Active Wave Absorption, based on intermediate water theory
  - 5** Active Wave Absorption, using a flap wavemaker (see B 123)
- default:** 0

● ● ● ●

**B 101** `int` Ramp function for wave generation

- 0** OFF
  - 1** On, based on wave period
  - 2** On, based on simulation time
- default:** 0

● ● ● ●

**B 102** `double` Ramp function for duration in wave periods or simulation time (see B 101)

**default:** 1.0

● ● ● ●

**B 105** `double` angle  $\beta$ , x and y coordinates for the distance origin for the generated wave

**default:** 0.0 ; 0.0 ; 0.0

● ● ● ●

**B 106** \* `double` angle  $\beta$ , x and y coordinates for the distance origin of wave generation relaxation zone

**default:** 0.0 ; 0.0 ; 0.0

● ● ● ●

**B 107** \* `double`  $x_{start}, x_{end}, y_{start}, y_{end}$  and acting distance  $d$  of numerical beach relaxation zones.

As default, without any user input for B 107, a relaxation zone for the numerical beach is created at the end of the domain, i.e. at the maximum x-coordinates of the domain, with an orientation perpendicular to the x-axis and with infinite length using the relaxation distance from B 96. If user input is provided for B 107, the default relaxation zone will not be created and the input from B 96 will not be used. Instead the start, end and acting distance of the relaxation zone of the numerical beach are provided through B 107. Multiple numerical beach generation zones are possible.

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

● ● ● ●

**B 108** `double xstart, xend, ystart, yend` and acting distance  $d$  of wave generation relaxation zones.

As default, without any user input for B 108, a relaxation zone for the wave generation is created at the beginning of the domain, i.e. at the minimum x-coordinates of the domain, with an orientation perpendicular to the x-axis and with infinite length using the relaxation distance from B 96. If user input is provided for B 108, the default relaxation zone will not be created and the input from B 96 will not be used. Instead the start, end and acting distance of the relaxation zone of the wave generation are provided through B 108.

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

● ● ● ●

**B 109** `int` use automatic depth field for wave theory velocity profiles If not given, the water depth is calculated based on the assumption that the sea bed level is located at a constant  $z = 0.0$  m.

**0** OFF

**1** ON

**default:** 0

● ● ● ●

**B 110** `double` water depth  $d$  for wave theory If not given, the water depth is calculated based on the assumption that the sea bed level is located at a constant  $z = 0.0$  m.

**default:** na

● ● ● ●

**B 111** `double`  $z_{start}$   $z_{end}$  for the flap wavemaker

**default:** 0.0 ; 0.0

● ● ● ●

**B 112** `double`  $z_{start}$   $z_{hinge_2}$ ,  $z_{end}$  for the double-hinged flap wavemaker

**default:** 0.0 ; 0.0 : 0.0

○ ○ ○ ●

**B 115** `int` Vertical velocity component for flap wavemaker kinematics

**0** OFF

**1** ON

**default:** 0

● ● ● ●

**B 116** `int` Flap wavemaker kinematics input as paddle motion or angle

**1**  $X(t)[m]$

**2**  $\beta(t)[rad]$

**default:** 1

● ● ● ●

**B 117** `double` Starting time shift for wavemaker kinematics input

**default:** 0.0

● ● ● ●

**B 120** `double` phase angle shift for periodic regular waves

**default:** -90°

● ● ● ●

**B 121** `int` Enable air velocity in wave relaxation zone

**0** OFF

**1** ON

**default:** 1

○ ○ ○ ●

**B 123** `double` Z-coordinate for active wave absorption hinge location

**default:** 0.0

● ● ● ●

**B 130** `int` Enable directional spreading for irregular waves

**0** OFF  
**1** Cosine squared  
**2** Mitsuyasu  
**default:** 0

● ● ● ●

**B 131** `double` Main direction for multidirectional irregular waves

**default:** 0.0°

● ● ● ●

**B 132** `double` Start and end angle for directional spreading

**default:** -90.0° ; 90.0°

● ● ● ●

**B 133** `int` Number of angle intervals for directional spreading

**default:** 1

● ● ● ●

**B 136** `int` Type of multi-directional irregular wave calculation

**1** DSM - double summation method  
**2** PSSM - Pascal's Single Summation Method  
**3** YSSM - Yu's Single Summation Method  
**default:** 1

● ● ● ●

**B 139** `int` Seed number for the random phase generation of irregular waves

**0** time based seed number  
**>0** input based seed number  
**default:** 0

● ● ● ●

**B 140** `double` Numerical Beach: start transition x1, start full damping x2,  $\beta$  damping factor

**default:** 0.0 ; 0.0 ; 0.0

● ○ ○ ○

**B 160** `int` Number of vertical layers for obtaining depth-averaged velocities in the wave generation for REEF3D::SFLOW

**default:** 5

○ ○ ○ ●

**B 180** `int` Enable ioGravity for gravity waves

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**B 181** `double` x-Gravity Wave frequency, x-Gravity Wave amplitude, x-Gravity Wave Phase change

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 182** `double` y-Gravity wave frequency, y-Gravity wave amplitude, y-Gravity wave phase change

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 183** `double` z-Gravity wave frequency, z-Gravity wave amplitude, z-Gravity wave phase change

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 191** `double` Angle for rotation around x-axis, rotation frequency, y-coordinate, z-coordinate

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 192** `double` Angle for rotation around y-axis, rotation frequency, x-coordinate, z-coordinate

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 194** `double` start time rotation, end time rotation

**default:** 0.0 ; 0.0

○ ○ ○ ●

**B 210** `int` Patch boundary condition: inflow or outflow condition

**1** inflow

**2** outflow

**default:** 2

○ ○ ○ ●

**B 240** \* `double` Darcy porous media as rectangular box:  $C, D, x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 241** `int` Porous media in x-direction for B240

**0** OFF

**1** ON

**default:** 1

○ ○ ○ ●

**B 242** `int` Porous media in y-direction for B240

**0** OFF

**1** ON

**default:** 1

○ ○ ○ ●

**B 243** `int` Porous media in z-direction for B240

**0** OFF

**1** ON

**default:** 1

○ ○ ○ ●

**B 260** `double` c factor for VRANS porous media

**default:** 0.0

○ ○ ○ ●

**B 264** `double` KC number VRANS porous media force source terms for the momentum equations

**default:**  $1.0 \cdot 10^{20}$

○ ○ ○ ●

**B 270** \* `double` VRANS porous media as rectangular box  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}, n, d_{50}, \alpha, \beta$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**B 274** \* `double` VRANS porous media as vertical cylinder:  $x_{center}, y_{center}, z_{start}, z_{end}, radius, n, d_{50}, \alpha, \beta$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**B 281** \* `double` VRANS porous media as wedge in x-direction:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}, n, d_{50}, \alpha, \beta$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**B 291** \* `double` VRANS porous media as plate in x-direction:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}, d_{thickness}, n, d_{50}, \alpha, \beta$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**B 295** `int` VRANS source treatment fo turbulence model

- 0** OFF
  - 1** VRANS source terms
  - 2** turn off turbulence inside porous media
- default:** 1

○ ○ ○ ●

**B 308** `int` consider vegetation porosity effects on fluid acceleration

- 0** OFF
  - 1** ON
- default:** 1

○ ○ ○ ●

**B 309** `double`  $C_M$  for vegetation

**default:** 2.0

○ ○ ○ ●

**B 310** \* `double` VRANS vegetation as rectangular box. N is the number of cylinders per unit area, D the diameter of the cylinders and  $C_d$  the drag coefficient.

$x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}, N, D, C_d$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

● ○ ○ ○

**B 411** \* patchBC pressure `int`  $ID$ , `double`  $discharge$

**default:** 0.0

○ ○ ○ ●

**B 412** \* patchBC pressure `int`  $ID$ , `double`  $velocity$

**default:** 0.0

○ ○ ○ ●

**B 414 \*** patchBC velocity perpendicular to face `int ID, double velocity`

**default:** 0.0

○ ○ ○ ●

**B 415 \*** patchBC velocity components `int ID, double U,V,W`

**default:** 0.0 ; 0.0 ; 0.0

● ○ ○ ○

**B 418** `int` free stream outflow

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**B 440 \*** patchBC as line:

`int ID, face, double Xstart,Xend,Ystart,Yend`

The patch boundary condition will convert solid boundaries to inflow or outflow conditions. Patches are addressed with the ID, with which it is possible to prescribe different flow conditions (see B 411 - B 418). When flow values are not prescribed, the default are zero-gradient boundary conditions.

**default:** 0 ; 0; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 441 \*** patchBC as rectangular box:

`int ID, face, double Xstart,Xend,Ystart,Yend,Zstart,Zend`

The patch boundary condition will convert solid boundaries to inflow or outflow conditions. Patches are addressed with the ID, with which it is possible to prescribe different flow conditions (see B 411 - B 417).

**default:** 0 ; 0; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**B 442 \*** patchBC as circle: `int ID, face, double xM,yM,zM, radius`

The patch boundary condition will convert solid boundaries to inflow or outflow conditions. Patches are addressed with the ID, with which it is possible to prescribe different flow conditions (see B 411 - B 417).

**default:** 0 ; 0; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

## 2.3 C :: Concentration

Valid for:

S F N C  
○ ○ ○ ●

**C 1** `double` Additional density from concentration in phase 1

**default:** 10.0

○ ○ ○ ●

**C 2** `double` Additional viscosity from concentration in phase 1

**default:** 0.0

○ ○ ○ ●

**C 3** `double` Additional density from concentration in phase 2

**default:** 10.0

○ ○ ○ ●

**C 4** `double` Additional viscosity from concentration in phase 2

**default:** 0.0

○ ○ ○ ●

**C 5** `double` Additional viscosity from concentration in phase 2

**default:** 0.0

○ ○ ○ ●

**C 10** `int` Time scheme concentration function

**0** OFF

**1** 2nd-order Adams-Bashfort

**2** 2rd-order TVD Runge-Kutta

**3** 3rd-order TVD Runge-Kutta

**default:** 0

○ ○ ○ ●

**C 15** `int` convection discretization for concentration function

**1** FOU

**2** CDS2

**3** QUICK

**4** WENO FLUX

**5** WENO HJ

**6** CDS4

**10** High Resolution TVD scheme with Minmod limiter

**11** High Resolution TVD scheme with van Leer limiter

**12** High Resolution TVD scheme with Umist limiter

**13** High Resolution TVD scheme with Sweby limiter

**14** High Resolution TVD scheme with Superbee limiter

**15** High Resolution scheme with Smart limiter

**16** High Resolution TVD scheme with Limo3 limiter

**42** High Resolution TVD scheme with Weller limiter

**default:** 0

○ ○ ○ ●

**C 20** `int` Diffusion discretization

**0** OFF

**1** explicit

**2** implicit (automatic for implicit convection discretization)

**default:** 0

○ ○ ○ ●

**C 50** `double` Fill ratio concentration area 1 and 2

**default:** 1.0 ; 0.0

○ ○ ○ ●

**C 51** `double` Area 1 start, x-direction

**default:** 0.0 *m*

○ ○ ○ ●

**C 52** double Area 1 start, y-direction

**default:** 0.0 m

○ ○ ○ ●

**C 53** double Area 1 start, z-direction

**default:** 0.0 m

○ ○ ○ ●

**C 54** double Area 1 end, x-direction

**default:**  $1.0 \cdot 10^7$  m

○ ○ ○ ●

**C 55** double Area 1 end, y-direction

**default:**  $1.0 \cdot 10^7$  m

○ ○ ○ ●

**C 56** double Area 1 end, z-direction

**default:**  $1.0 \cdot 10^7$  m

○ ○ ○ ●

**C 57** double Area 1 as 3D-plane:  $ax + by + cz + d = 0$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**C 58** double Area 1 as sphere, center-coordinates:  $x_0, y_0, z_0$ , and radius  $r$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0

## 2.4 D :: Discretization

Here the discretization of the convection terms in the momentum equations is chosen. All methods are implemented in terms of conservative finite differences. The exception is WENO HJ, which is the non-conservative form of the WENO scheme.

Valid for:

S F N C

○ ○ ● ●

**D 10** [int](#) convection discretization

- 0** OFF
  - 1** FOU
  - 2** CDS2
  - 3** QUICK
  - 4** WENO5 FLUX
  - 5** WENO5 HJ
  - 6** CDS4
  - 7** WENO3 FLUX
  - 8** WENO3 HJ
  - 10** High Resolution TVD scheme with Minmod limiter
  - 11** High Resolution TVD scheme with van Leer limiter
  - 12** High Resolution TVD scheme with Umist limiter
  - 13** High Resolution TVD scheme with Sweby limiter
  - 14** High Resolution TVD scheme with Superbee limiter
  - 15** High Resolution scheme with Smart limiter
  - 16** High Resolution TVD scheme with Limo3 limiter
  - 42** High Resolution TVD scheme with Weller limiter
- default:** 4

○ ○ ● ●

**D 11** [int](#) convection velocities for momentum equations

- 0** OFF
  - 1** FOU
  - 2** CDS2
- default:** 2

○ ○ ● ●

**D 20** [int](#) diffusion discretization

Selecting the implicit treatment of the diffusion term in the momentum equations for explicit time stepping has the advantage of removing diffusion from the CFL criterion, which determines the time step size. As the diffusion term enters the CFL criterion in the order of  $1/dx^2$ , effectively larger times steps for D20 2 will be used on especially finer grids. When RANS turbulence models are used, the effective diffusion can be several magnitudes higher than the molecular diffusion. Then implicit diffusion treatment will increase the time step significantly.

**0** OFF

**1** explicit

**2** implicit

**default:** 2

○ ○ ● ●

**D 21** [int](#) print out diffusion solver time and iteration for implicit diffusion and explicit convection discretization

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**D 30** [int](#) pressure algorithm The projection method (PJM) works only together with explicit time advancement schemes, while all SIMPLE type methods work with implicit timestepping.

**0** off

**1** PJM

**2** PJM FSM

**3** PJM CORR

**default:** 1

○ ○ ● ●

**D 39** [int](#) activate 2nd-order pressure correction for PJM CORR

**0** OFF

**1** ON

**default:** 0

## 2.5 F :: Free Surface

Valid for:

S   F   N   C  
○   ○   ○   ●

**F 10** int free surface mode for interface capturing

**1** One-Phase

**2** Two-Phase

**default:** 2

○   ○   ○   ●

**F 30** int time scheme level set

**1** 2nd-order Adams-Bashfort

**2** 2rd-order TVD Runge-Kutta

**3** 3rd-order TVD Runge-Kutta

**default:** 0

○   ○   ○   ●

**F 31** int particle Level Set

**0** off

**1** PLS with tri-linear interpolation

**2** PLS with tri-cubic interpolation

**default:** 0

○   ○   ○   ●

**F 32** int number of particles per cell

**default:** 64

○   ○   ○   ●

**F 33** double factor for particle array allocation

**default:** 0.4

○   ○   ○   ●

**F 34** `int` Printout iteration of particles

**default:** 1000

○ ○ ○ ●

**F 35** `int` convection discretization for level set

- 1** FOU
- 2** CDS2
- 3** QUICK
- 4** WENO5 FLUX
- 5** WENO5 HJ
- 6** CDS4
- 7** WENO3 FLUX
- 8** WENO3 HJ
- 10** High Resolution TVD scheme with Minmod limiter
- 11** High Resolution TVD scheme with van Leer limiter
- 12** High Resolution TVD scheme with Umist limiter
- 13** High Resolution TVD scheme with Sweby limiter
- 14** High Resolution TVD scheme with Superbee limiter
- 15** High Resolution scheme with Smart limiter
- 16** High Resolution TVD scheme with Limo3 limiter
- 42** High Resolution TVD scheme with Weller limiter

**default:** 5

○ ○ ○ ●

**F 39** `double` Relaxation factor for reini volume constraint

**default:** 0.5

○ ○ ○ ●

**F 40** `int` Reinitialization time scheme

- 3** 3rd-order TVD Runge-Kutta
- 5** 3rd-order Runge-Kutta with local volume correction
- 7** 3rd-order TVD Runge-Kutta with volume constraint
- 11** Geometric reinitialization with 3rd-order TVD Runge-Kutta
- 23** 3rd-order TVD Runge-Kutta, reinitialization through solid and topo

**default:** 0

○ ○ ○ ●

**F 42** `double` length for level set initial reinitialization, overrides maximum domain length

**default:** -1.0

○ ○ ○ ●

**F 43** `double` Factor for reinitialization time step size

**default:** 0.55

○ ○ ○ ●

**F 44** `int` Number of Reinitialization time steps

**default:** 3

○ ○ ○ ●

**F 45** `double` Factor for the calculation of the interface thickness  $\epsilon$

**default:** 2.1

○ ○ ○ ●

**F 46** `int` Type of Picard iterations after reinitialization

**0** OFF

- 1** Volume correction for reinitialization step using volume from previous time step
  - 2** Volume correction for level set and reinitialization step using volume from previous time step
  - 3** Volume correction for level set and reinitialization step using total volume balance
- default:** 0

○ ○ ○ ●

**F 47** `int` Number of Picard iterations after reinitialization

**default:** 0

○ ○ ○ ●

**F 49** `int` Reinitialization for interface nodes

**0** OFF

**1** ON

**default:** 1

○ ○ ○ ●

**F 50** **int** Fix Level Set for inflow or outflow

**1** Inflow fixed

**2** Outflow fixed

**3** Fix Both

**4** Fix None

**default:** 2

○ ○ ○ ●

**F 51** **double** Phase 1 start, x-direction

**default:** 0.0 *m*

○ ○ ○ ●

**F 52** **double** Phase 1 start, y-direction

**default:** 0.0 *m*

○ ○ ○ ●

**F 53** **double** Phase 1 start, z-direction

**default:** 0.0 *m*

○ ○ ○ ●

**F 54** **double** Phase 1 end, x-direction

**default:**  $1.0 \cdot 10^7$  *m*

○ ○ ○ ●

**F 55** **double** Phase 1 end, y-direction

**default:**  $1.0 \cdot 10^7$  *m*

○ ○ ○ ●

**F 56** double Phase 1 end, z-direction

**default:**  $1.0 \cdot 10^7 \text{ m}$

○ ○ ○ ●

**F 57** double Interface as 3D-plane:  $ax + by + cy + d = 0$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**F 58** double Interface as sphere, center-coordinates:  $x_m, y_m, z_m$ , and radius  $r$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**F 59** double Interface as vertical cylinder, center-coordinates:  $x_m, y_m, z_{start}, z_{end}$  and radius  $r$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**F 60** double Phase 1 end, z-direction for whole domain

**default:**  $-1.0 \cdot 10^{20} \text{ m}$

○ ○ ○ ●

**F 61** double Phase 1 end, z-direction for inflow boundary

**default:**  $-1.0 \cdot 10^{20} \text{ m}$

○ ○ ○ ●

**F 62** double Phase 1 end, z-direction for outflow boundary

**default:**  $-1.0 \cdot 10^{20} \text{ m}$

○ ○ ○ ●

**F 63** double  $x_{start}$  for level set interpolation

**default:**  $-1.0 \cdot 10^{20} \text{ m}$

○ ○ ○ ●

**F 70** `double*` Phase 1 ini as rectangular box:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**F 71** `double*` Phase 2 ini as rectangular box:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**F 72** `double*` Regions for free surface elevation  $x_{start}, x_{end}, y_{start}, y_{end}, h_{waterlevel}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**F 80** `int` time scheme VOF Currently, the VOF implementation is in beta and for testing purposes only. The algorithms for open channel flow and wave generation only work for the level set method.

**1** 2nd-order Adams-Bashfort

**3** 3rd-order TVD Runge-Kutta

**default:** 0

○ ○ ○ ●

**F 84** `double`  $c_\alpha$  factor for VOF compression

**default:** 1.0

○ ○ ○ ●

**F 85** `int` convection discretization VOF Recommended scheme for VOF convection is HRIC modified.

**1** FOU

**2** CDS2

**3** QUICK

**4** WENO FLUX

**5** WENO HJ

**6** CDS4

**10** High Resolution TVD scheme with Minmod limiter  
**11** High Resolution TVD scheme with van Leer limiter  
**12** High Resolution TVD scheme with Umist limiter  
**13** High Resolution TVD scheme with Sweby limiter  
**14** High Resolution TVD scheme with Superbee limiter  
**15** High Resolution scheme with Smart limiter  
**16** High Resolution TVD scheme with Limo3 limiter  
**42** High Resolution TVD scheme with Weller limiter  
**51** HRIC  
**52** HRIC modified  
**53** CICSAM  
**default:** 0

**F 150** [int](#) Benchmarks This will initialize the level set function and the velocity field according to the benchmark cases

**0** OFF  
**1** Vortex  
**2** Slotted disc  
**3** 3D Deformation  
**4** 3D Taylor-Green vortex  
**default:** 0

## 2.6 G :: Grid Options

Valid for:

S F N C

**G 50** [int](#) look for geodata points in grid file When point coordinates with xyz components exist for the solid boundary representation, such as for example for a river or coastal region, the interpolation for the topography will take place in DIVEMesh. Then the location of the solid boundary is written into each of the grid files.

**0** OFF  
**1** ON  
**default:** 1

**G 52** `int` number of smoothing iterations for geodat points

**default:** 0

**G 60 \*** `double` Topo ini as rectangular box:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0 ; 0

**G 61 \*** `double` Topo ini as wedge in x-direction:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

## 2.7 H :: Heat Transfer

Valid for:

S F N C

**H 1** `double` Thermal Diffusivity Phase 1

**default:**  $1.4 \cdot 10^{-7} \text{ m}^2/\text{s}$  (water)

**H 2** `double` Thermal Diffusivity Phase 2

**default:**  $2.216 \cdot 10^{-5} \text{ m}^2/\text{s}$  (air)

**H 3** `int` Fluid property calculation

**1** interpolation for water and air (density and viscosity)

**2** generic Boussinesq Approximation (density)

**default:** 1

**H 9** `int` Switch Temperature 1 and temperature 2 fluid for the two phases

- 1** 1: water 2: air
  - 2** 1: air 2: water
- default:** 1

○ ○ ○ ●

**H 10** `int` Time Scheme Heat Transfer

- 0** OFF
  - 1** 2nd-order Adams-Bashfort
  - 2** 2nd-order TVD Runge-Kutta
  - 3** 3rd-order TVD Runge-Kutta
- default:** 0

○ ○ ● ●

**H 15** `int` convection discretization for heat transfer

- 0** OFF
  - 1** FOU
  - 2** CDS2
  - 3** QUICK
  - 4** WENO5 FLUX
  - 5** WENO5 HJ
  - 6** CDS4
  - 7** WENO3 FLUX
  - 8** WENO3 HJ
  - 10** High Resolution TVD scheme with Minmod limiter
  - 11** High Resolution TVD scheme with van Leer limiter
  - 12** High Resolution TVD scheme with Umist limiter
  - 13** High Resolution TVD scheme with Sweby limiter
  - 14** High Resolution TVD scheme with Superbee limiter
  - 15** High Resolution scheme with Smart limiter
  - 16** High Resolution TVD scheme with Limo3 limiter
  - 42** High Resolution TVD scheme with Weller limiter
- default:** 5

○ ○ ○ ●

**H 50** double Temperature 1 and Temperature 2

**default:**  $0.0\text{ }C^\circ$  ;  $0.0\text{ }C^\circ$

○ ○ ○ ●

**H 51** double Temperature 1 start, x-direction

**default:**  $0.0\text{ }m$

○ ○ ○ ●

**H 52** double Temperature 1 start, y-direction

**default:**  $0.0\text{ }m$

○ ○ ○ ●

**H 53** double Temperature 1 start, z-direction

**default:**  $0.0\text{ }m$

○ ○ ○ ●

**H 54** double Temperature 1 end, x-direction

**default:**  $1.0 \cdot 10^7\text{ }m$

○ ○ ○ ●

**H 55** double Temperature 1 end, y-direction

**default:**  $1.0 \cdot 10^7\text{ }m$

○ ○ ○ ●

**H 56** double Temperature 1 end, z-direction

**default:**  $1.0 \cdot 10^7\text{ }m$

○ ○ ○ ●

**H 57** double Interface as 3D-plane:  $ax + by + cy + d = 0$

**default:**  $0.0 ; 0.0 ; 0.0 ; 0.0$

○ ○ ○ ●

**H 58** `double` Interface as sphere, center-coordinates:  $x_0, y_0, z_0$ , and radius  $r$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**H 61** `double` Temperature boundary condition on surface 1

**default:** na  $C^\circ$

○ ○ ○ ●

**H 62** `double` Temperature boundary condition on surface 2

**default:** na  $C^\circ$

○ ○ ○ ●

**H 63** `double` Temperature boundary condition on surface 3

**default:** na  $C^\circ$

○ ○ ○ ●

**H 64** `double` Temperature boundary condition on surface 4

**default:** na  $C^\circ$

○ ○ ○ ●

**H 65** `double` Temperature boundary condition on surface 5

**default:** na  $C^\circ$

○ ○ ○ ●

**H 66** `double` Temperature boundary condition on surface 6

**default:** na  $C^\circ$

## 2.8 I :: Initialization

Valid for:

S F N C

**I 10** [int](#) Initialize Everything Turning this parameter on will invoke I11, I12 and I13.

**0** OFF

**1** ON

**2** ON (include free surface for potential flow ini)

**default:** 0

**I 11** [int](#) Initialize Velocities with Potential Flow Solver

**0** OFF

**1** ON

**2** ON (include free surface)

**default:** 0

**I 12** [int](#) Initialize Pressure

**0** OFF

**1** Hydrostatic based on vertical coordinate

**2** Hydrostatic based on level set values

**default:** 0

**I 13** [int](#) Initialize Turbulence Model

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

I 21 int Phase 2 velocities after potential flow initialization

**0** use potential solver result

**1** set to zero

**default:** 0

● ● ● ●

I 30 int Full initialization of Numerical Wave Tank

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

I 40 int Full initialization from state file (hotstart) See option P 40 for state file print out.

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

I 41 int Number of state file (hotstart)

**default:** 0

○ ○ ○ ●

I 56 int pressure above F56 set to zero

**default:** 0

○ ○ ○ ●

I 58 double Initialize Vertical Velocity around Sphere from F 58, double radius for initialization

**default:** 0.0 ; 0.0

○ ○ ○ ●

**I 230** `int` Read FlowFiles for inflow boundary condition. The number gives the ID of the FlowFile gage. For iwave activated, B 98 should be set to 0.

**default:** 0

○ ○ ○ ●

**I 231** `double` Vertical offset of FlowFile inflow.

**default:** 0.0

○ ○ ○ ●

**I 232** `double` Time offset of FlowFile inflow.

**default:** 0.0

## 2.9 M :: MPI

Valid for:

S F N C  
● ● ● ●

**M 10** `int` Number of processors for parallel computations This value needs to be consistent with the grid generation and the console input for starting REEF3D through the mpirun command.

**default:** 0

## 2.10 N :: Numerical Options

Valid for:

S F N C  
○ ● ● ●

**N 10** `int` Iterative solver for the Poisson equation The default BiCGStab solver uses Jacobi Scaling as the default preconditioner, which is implemented in the compressed diagonal storage format (CDS). Jacobi Scaling is currently the most effective preconditioner for the Poisson equation with a jump in the matrix coefficients, as is the case for multiphase flow. SIP is an optimized implementation of ILU.

The fastest combination of HYPRE solvers is the BiCGSTAB struct solver with PFMG preconditioning. From the HYPER AIJ solvers, the combination PCG + BoomerAMG is fastest. For more information on the HYPRE solvers and preconditioners, have a look at the HYPRE user manual Cen [2015].

**3** BiCGStab with Jacobi Preconditioning (internal)

- 11** PCG (HYPRE Struct)
- 12** GMRES (HYPRE Struct)
- 13** LGMRES (HYPRE Struct)
- 14** BiCGStab (HYPRE Struct)
- 15** Hybrid-PCG (HYPRE Struct)
- 16** Hybrid-GMRES (HYPRE Struct)
- 17** Hybrid-BiCGStab (HYPRE Struct)
- 18** PFMG (HYPRE Struct, Geometric Multigrid)
- 19** SMG (HYPRE Struct, Geometric Multigrid)

- 21** PCG (HYPRE AIJ)
- 22** GMRES (HYPRE AIJ)
- 23** LGMRES (HYPRE AIJ)
- 24** BiCGStab (HYPRE AIJ)
- 25** AMG (HYPRE AIJ, BoomerAMG)

- 31** PCG (HYPRE SStruct)
- 32** GMRES (HYPRE SStruct)
- 33** LGMRES (HYPRE SStruct)
- 34** BiCGStab (HYPRE SStruct)
- 35** Hybrid-PCG (HYPRE SStruct)
- 36** Hybrid-GMRES (HYPRE SStruct)
- 37** Hybrid-BiCGStab (HYPRE SStruct)
- 38** PFMG (HYPRE SStruct, Geometric Multigrid)
- 39** SMG (HYPRE SStruct, Geometric Multigrid)

**default:** 14



**N 11** **int** Preconditioner for the Poisson Equations Keep in mind to use only options, that are compatible with N 10.

- 0** OFF
- 11** PFMG (for HYPRE Struct Krylov solvers)
- 12** SMG ( for HYPRE Struct Krylov solvers)
  
- 21** AMG (for HYPRE AIJ Krylov solvers, BoomerAMG)

**31** PFMG (for HYPRE SStruct Krylov solvers)

**32** SMG ( for HYPRE SStruct Krylov solvers)

**default:** 11

○ ● ● ●

**N 21** `int` hypre PFMG relaxation skip

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**N 22** `int` hypre PFMG skip relaxation

**0** Jacobi

**1** Weighted Jacobi

**2** Red/Black Gauss-Seidel (symmetric)

**3** Red/Black Gauss-Seidel (non-symmetric)

**default:** 3

○ ○ ○ ●

**N 23** `int` hypre PFMG RAP type

**0** Galerkin

**1** non-Galerkin

**default:** 0

○ ○ ○ ●

**N 40** `int` Time scheme for the momentum equations The explicit time schemes (option 1-9) require the projection method for the pressure (D 30).

**1** 2nd-order Adams-Bashforth

**2** 2rd-order TVD Runge-Kutta

**3** 3rd-order TVD Runge-Kutta

**4** Implicit-explicit Runge-Kutta

**6** Fractional Step Method based on 3rd-order TVD Runge-Kutta

**default:** 1

● ● ● ●

**N 41** `double` Maximum modeled time Out of N41 and N45, whichever criterion is fulfilled first will finalize the simulations.

**default:**  $1.0 \cdot 10^{19}$

● ● ● ●

**N 44** `double` Stopping criteria iterative solver

**default:**  $1.0 \cdot 10^{-8}$

● ● ● ●

**N 45** `int` Maximum number of outer iterations Out of N41 and N45, whichever criterion is fulfilled first will finalize the simulations

**default:**  $1 \cdot 10^8$

● ● ● ●

**N 46** `int` Maximum number of solver iterations

**default:** 250

● ● ● ●

**N 47** `double` Relaxation factor for time step size This factor is used when determining the time step size based on the CFL criterion for adaptive timestepping.

**default:** 0.3

● ● ● ●

**N 48** `int` Adaptive timestepping

**0** OFF

**1** ON

**2** ON, using velocity around the interface

**default:** 1

● ● ● ●

**N 49** `double` Timestep size for fixed timestepping

**default:** 1.0 sec

● ● ● ●

**N 61** `double` Stopping criterion for critical velocities. The code will exit and write a final .vtu paraview file

**default:** 500.0

## 2.11 P :: Printer

Valid for:

S F N C

● ● ● ●

**P 10** `int` Print ParaView binary format

**0** OFF

**1** ON

**default:** 1

○ ○ ○ ●

**P 11** `int` Log print frequency

**default:** 10

● ● ● ●

**P 12** `int` Terminal print frequency

**default:** 1

● ● ● ●

**P 14** `int` Print files to folders

**0** OFF

**1** ON

**default:** 1

● ● ● ●

**P 15** `int` Print out file numbering

- 1** print out based
  - 2** iteration based
- default:** 1



**P 18** `int` Algorithm type for level set paraview print out

- 1** Standard
  - 2** Node Fill
- default:** 2



**P 20** `int` Print results every  $i^{th}$  iteration Choose between either P20, P30, P34 or P35.

- default:** -10



**P 23** `int` Print test array vtu file

- 0** OFF
  - 1** ON
- default:** 0



**P 24** `int` Print density to vtu file

- 0** OFF
  - 1** ON
- default:** 0



**P 25** `int` Print solid to vtu file

- 0** OFF
  - 1** ON
- default:** 0

**P 26** [int](#) Print cbed and conc to vtu file

**0** OFF

**1** ON

**default:** 0

**P 27** [int](#) Print topo to vtu file

**0** OFF

**1** ON

**default:** 0

**P 28** [int](#) Print fb to vtu file

**0** OFF

**1** ON

**default:** 0

**P 29** [int](#) Print walldist to vtu file

**0** OFF

**1** ON

**default:** 0

**P 30** [double](#) Print Paraview results every  $i^{th}$  second Choose between either P20, P30, P34 or P35.

**default:** -1.0

**P 34** [double](#) Print Paraview results every  $i^{th}$  second based on sediment transport time Choose between either P20, P30, P34 or P35.

**default:** -1.0

○ ● ● ●

**P 35** \* `double` Start print t, `double` End t, `double` results every i<sup>th</sup> second

Choose between either P20, P30, P34 or P35.

**default:** 0.0 ; 0.0 ; 0.0

○ ● ○ ●

**P 40** `int` Print state file for hotstart functionality See I 40 for full initialization from state file. The state file contains the velocities, pressure, level set function, k, eddy viscosity,  $\epsilon$  or  $\omega$ , topo, cbcd and conc.

**0** OFF

**1** print into one file

**2** print into continuous files

**default:** 0

○ ● ○ ●

**P 41** `int` Print state file every i<sup>th</sup> Iteration

**default:** na

○ ● ○ ●

**P 42** `double` Print state file every i<sup>th</sup> second

**default:** na

● ● ● ●

**P 50** \* `double` x-location and `double` y-location of height gauges for wave theory

**default:** na ; na

● ● ● ●

**P 51** \* `double` x-location and `double` y-location of height gauges

**default:** na ; na

● ● ● ●

**P 52** \* `double` y-location of water surface line in x-direction

**default:** na

● ● ● ●

**P 53** `int` for NWT: add theoretical wave to wsflne file

**0** OFF

**1** ON

**default:** 0

● ● ● ●

**P 54** `int` Print wsflne files every  $i^{th}$  iteration

**default:** 10

● ● ● ●

**P 55** `double` Print wsflne files every  $i^{th}$  second

**default:** -1.0

● ● ● ●

**P 56** \* `double` x-location of water surface line in y-direction

**default:** na

○ ○ ○ ●

**P 61** \* `double` x-location, `double` y-location and `double` z-location of point probes

**default:** na

○ ○ ○ ●

**P 62** \* `double`  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$  of line probes

**default:** na

● ○ ○ ○

**P 63** \* `double` x-location and `double` y-location of depth averaged point probes

**default:** na

**P 66** [int](#) print out discharge cross section to console

**0** OFF

**1** ON

**default:** 0

**P 67 \*** [double](#) x-location of discharge in cross section

**default:** na

**P 71** [int](#) print out viscosity to vtu file

**0** OFF

**1** ON

**default:** 0

**P 75** [int](#) print out vorticity to vtu file

**0** OFF

**1** ON

**default:** 0

**P 78** [int](#) print out velocity magnitude as scalar to vtu file

**0** OFF

**1** ON

**default:** 0

**P 79** [int](#) print out bed shear stress to vtu file

**0** OFF

**1** ON

**default:** 0

**P 81** **double** force calculation box:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

**P 85** **double** force calculation box for rectangular objects:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

**P 92** **int** Calculate force from water or water and air combined

**0** force only from water

**1** force from water and air combined

**default:** 0

**P 101** **int** Print sloshing force and moment file

**0** OFF

**1** ON

**default:** 0

**P 121** **double** x-location and **double** y-location of bed level gauges

**default:** na

**P 122** **int** Maximum bedchange gauge

**0** OFF

**1** ON

**default:** 0

**P 123** \* **double** y-location of sediment profile in x-direction

**default:** na

○ ○ ○ ●

**P 124 \*** `double` x-location of sediment profile in y-direction

**default:** na

○ ○ ○ ●

**P 125** `double` x-location and `double` y-location of bed shear stress gauges

**default:** na

○ ○ ○ ●

**P 126** `int` Maximum bedshear stress print out

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**P 150** `int` Print data from DIVEMesh interpolation

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**P 151** `int` Type of data

**1** scalar

**2** signed distance function

**default:** 1

○ ○ ○ ●

**P 152** `int` Type of boundary condition for data array

**1** x-velocity

**2** y-velocity

**3** z-velocity

**4** scalar with Neuman boundary conditions

**default:** 1

**P 180** `int` Print free surface vtp file

**0** OFF

**1** ON

**default:** 0

**P 181** `double`  $i^{th}$  iteration for fsf print out

**default:** 10

**P 182** `double` Print out fsf every  $i^{th}$  second

**default:** -1.0

**P 184** \* `int` Start iteration, `int` End iteration, `int` fsf print results every  $i^{th}$  iteration

Choose between either P181, P182, P184 or P185.

**default:** 0 ; 0 ; 0

**P 185** \* `double` Start t, `double` End t, `double` fsf print results every  $i^{th}$  second

Choose between either P181, P182, P184 or P185.

**default:** 0.0 ; 0.0 ; 0.0

**P 230** `double` \* x-coordinate for FlowFile printing from 2D CFD simulations

**default:** 0

○ ○ ○ ●

**P 240** `double *` x-coordinate for PotentialFile printing from 2D FNPF simulations

**default:** 0

## 2.12 S :: Sediment

Valid for:

S F N C  
● ○ ○ ●

**S 10** `int` Sediment Transport Module

**0** OFF

**1** ON (impermeable sediments)

**2** ON (porous sediment)

**default:** 0

○ ○ ○ ●

**S 11** `int` Bedload Transport Formula

**0** OFF

**1** van Rijn

**2** Meyer-Peter Müller

**3** Engelund and Fredsøe

**default:** 0

○ ○ ○ ●

**S 12** `int` Suspended transport formula

**0** OFF

**1** van Rijn

**default:** 0

○ ○ ○ ●

**S 13** `double` Maximum time step size sediment transport

**default:** 10.0 sec

○ ○ ○ ●

**S 14** `double` Relaxation factor for time step size sediment transport The timestep for the morphodynamic calculations is by default decoupled from the hydrodynamics timestep (see S15), which typically is determined through adaptive timestepping (N48). The morphodynamic timestep is determined by the Courant criterion by analyzing the rate of bed elevation change.

**default:** 0.3

○ ○ ○ ●

**S 15** `int` Sediment timestep selection

**0** adaptive from S13 and S14

**1** from flow solver

**2** fixed from S13

**default:** 0

○ ○ ○ ●

**S 16** `int` Bed shear stress formulation

**1** wall function/velocity based

**2** friction coefficient/velocity based

**3** velocity based

**4** turbulent kinetic energy based

**7** depth-averaged velocities, based on Chezy-formula

**default:** 1

○ ○ ○ ●

**S 17** `int` Inner iterations for topo in order to get to full sediment timestep

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**S 18** `int` Maximum inner iterations for topo

**default:** 100

○ ○ ○ ●

**S 19** `double` Maximum modeled time for sediment transport

**default:**  $1.0 \cdot 10^{19}$

○ ○ ○ ●

**S 20** `double` Sediment  $d_{50}$

**default:** 0.001 *m*

○ ○ ○ ●

**S 21** `double` Factor for  $d_{50}$  in calculation of  $k_s$  in bedshear routine

**default:** 3.0

○ ○ ○ ●

**S 22** `double` Sediment density

**default:** 2650.0 *kg*

○ ○ ○ ●

**S 23** `double` Sediment fall velocity

**default:** 0.000001 *m/s*

○ ○ ○ ●

**S 24** `double` Porosity of the sediment layer

**default:** 0.5

○ ○ ○ ●

**S 25** `double` Angle of repose ( $^{\circ}$ )

**default:** 30.0  $^{\circ}$

○ ○ ○ ●

**S 30** `double` Shields parameter

**default:** 0.047

○ ○ ● ●

**S 32** [int](#) Exner equation discretization

- 1** FOU
  - 2** CDS2
  - 4** WENO5 FLUX
  - 5** WENO5 HJ
- default:** 4

○ ○ ○ ●

**S 37** [int](#) Number of topo reinitialization time steps The mobile sediment bed is represented by a level set method. After erosion or deposition has taken place, it needs to be reinitialized in order keep its signed distance properties.

**default:** 2

○ ○ ○ ●

**S 41** [int](#) Type of sediment start criterion

- 1** iterations
  - 2** flow simulation time
  - 3** t/T
- default:** 1

○ ○ ○ ●

**S 42** [int](#) Type of sediment interval criterion

- 1** iterations
  - 2** flow simulation time
  - 3** t/T
- default:** 1

○ ○ ○ ●

**S 43** [int](#) Number of water iterations, before sediment transport starts

**default:** 1000

○ ○ ○ ●

**S 44** `int` Number of water iterations, between bed calculations

**default:** 10

○ ○ ○ ●

**S 45** `double` Flow simulation time, before sediment transport starts

**default:** 1.0

○ ○ ○ ●

**S 46** `double` Flow simulation time between bed calculation

**default:** 1.0

○ ○ ○ ●

**S 47** `double` t/T, before sediment transport starts

**default:** 1.0

○ ○ ○ ●

**S 48** `double` t/T between bed calculation

**default:** 1.0

○ ○ ○ ●

**S 50** `int` Fix topo level set for inflow or outflow

- 1** Inflow fixed
- 2** Outflow fixed
- 3** Fix Both
- 4** Fix None

**default:** 3

○ ○ ○ ●

**S 57** `double` Sediment end, z-direction for whole domain

**default:**  $-1.0 \cdot 10^2 \text{ m}$

○ ○ ○ ●

**S 60** `int` Time scheme suspended sediments

- 1** 2nd-order Adams-Bashfort
  - 2** 2nd-order TVD Runge-Kutta
  - 3** 3rd-order TVD Runge-Kutta
  - 11** 1st-order Euler Implicit
  - 12** 2nd-order Three Time Level Method
- default:** 0

○ ○ ○ ●

**S 71** `double` Start of erosion in x-direction

**default:** -1.0e20

○ ○ ○ ●

**S 72** `double` End of erosion in x-direction

**default:** 1.0e20

○ ○ ○ ●

**S 73** \* `double` value, `double` distance, `double` line angle, `double` line x-origin, `double` line y-origin; for use of relaxation method for fixed water level

This method is preferred over S 71 and S72 because of the smooth moderation of the sediment bed. It gives also more flexibility as multiple relaxation lines with variable orientation can be chosen.

**default:** na

○ ○ ○ ●

**S 77** `double` Active sediment transport algorithm region in x-direction:  $x_{start}$  and  $x_{end}$

**default:** -1.0e20 ; +1.0e20

○ ○ ○ ●

**S 80** `int` Type of critical shear stress reduction for sloping bed

- 0** off
- 1** Parker and Kovacs

**2** Dey empirical

**3** Dey analytical

**4** Fredsøe

**default:** 0

○ ○ ○ ●

**S 81** `double` Midphi parameter

**default:** 35.0

○ ○ ○ ●

**S 82** `double` Deltaphi parameter

**default:** 5.0

○ ○ ○ ●

**S 90** `int` Sandslide algorithm

**0** off

**1** on

**default:** 0

○ ○ ○ ●

**S 91** `int` Number of sandslide iterations

**default:** 1

○ ○ ○ ●

**S 93** `double` Delta angle for sandslide correction

**default:** 0.0°

○ ○ ○ ●

**S 100** `int` Number of outer spatial filter iterations for the sediment bed for the predictor step  
(larger numbers increase the filter effect, recommended value: 1)

**default:** 0

**S 101** `int` Number of inner spatial filter iterations for the sediment bed for the corrector step (larger numbers move the bed closer to the uncorrected values, recommended value: between 1 and 5)

**default:** 0

**S 102** `int` Number of outer spatial filter iterations for the bed load discharge for the predictor step (larger numbers increase the filter effect, recommended value: 0)

**default:** 0

**S 103** `int` Number of inner spatial filter iterations for the bed load discharge for the corrector step (larger numbers move the bed load discharge closer to the uncorrected values, recommended value: 0)

**default:** 0

## 2.13 T :: Turbulence

Valid for:

S F N C

**T 10** `int` Turbulence Model

**0** OFF

**1** k- $\epsilon$  model

**2** k- $\omega$  model

**21** URANS with k- $\epsilon$  model

**22** URANS with k- $\omega$  model

**31** LES with Smagorinsky SGS model

**default:** 0

**T 11** `int` Time Scheme for 2-equation turbulence models

**11** 1st-order Euler Implicit

**12** 2nd-order Three Time Level Method

**default:** 11

○ ○ ○ ●

**T 12** `int` convection discretization for turbulence model

**0** OFF

**1** FOU

**5** WENO HJ

**default:** 5

○ ○ ○ ●

**T 31** `double` factor for eddy viscosity limiter in phase 1

**default:** 0.816

○ ○ ○ ●

**T 32** `double` factor for eddy viscosity limiter in phase 2

**default:** 0.816

○ ○ ○ ●

**T 35** `double` factor for eddy viscosity limiter near-wall

**default:** 0.816

○ ○ ○ ●

**T 36** `int` FSF boundary condition for turbulent dissipation

**0** OFF

**1** ON

**2** ON, including distance from the nearest wall

**default:** 0

○ ○ ○ ●

**T 37** `double` Virtual origin for distance  $y'$  above the free surface

**default:** 0.07

○ ○ ○ ●

**T 38** `double`  $\epsilon_\delta$  factor for width of free surface turbulence damping

**default:** 1.6

## 2.14 W :: Water and Material Properties

Valid for:

S F N C  
● ● ● ●

**W 1** `double` Density Phase 1

**default:** 998.2  $kg/m^3$

● ● ● ●

**W 2** `double` Viscosity Phase 1

**default:**  $1.004 \cdot 10^{-6} m^2/s$

○ ○ ○ ●

**W 3** `double` Density Phase 2

**default:** 1.205  $kg/m^3$

○ ○ ○ ●

**W 4** `double` Viscosity Phase 2

**default:**  $1.41 \cdot 10^{-5} m^2/s$

○ ○ ○ ●

**W 5** `double` Surface tension

**default:** 0.0  $N/m$

● ○ ● ●

**W 10** `double` Discharge

**default:** 0.0  $m^3/s$

○ ○ ○ ●

**W 11** `double` Inflow velocities U,V,W on surface 1 (requires inflow boundary condition on surface 1 in DIVEMesh)

**default:** 0.0 0.0 0.0 *m/s*

○ ○ ○ ●

**W 12** `double` Inflow velocities U,V,W on surface 2 (requires inflow boundary condition on surface 2 in DIVEMesh)

**default:** 0.0 0.0 0.0 *m/s*

○ ○ ○ ●

**W 13** `double` Inflow velocities U,V,W on surface 3 (requires inflow boundary condition on surface 3 in DIVEMesh)

**default:** 0.0 0.0 0.0 *m/s*

○ ○ ○ ●

**W 14** `double` Inflow velocities U,V,W on surface 4 (requires inflow boundary condition on surface 4 in DIVEMesh)

**default:** 0.0 0.0 0.0 *m/s*

○ ○ ○ ●

**W 15** `double` Inflow velocities U,V,W on surface 5 (requires inflow boundary condition on surface 5 in DIVEMesh)

**default:** 0.0 0.0 0.0 *m/s*

○ ○ ○ ●

**W 16** `double` Inflow velocities U,V,W on surface 6 (requires inflow boundary condition on surface 6 in DIVEMesh)

**default:** 0.0 0.0 0.0 *m/s*

● ● ● ●

**W 20** `double` Acceleration due to gravity, x-component

**default:** 0.0 *m/s<sup>2</sup>*

● ● ● ●

**W 21** `double` Acceleration due to gravity, y-component

**default:** 0.0  $m/s^2$

● ● ● ●

**W 22** `double` Acceleration due to Gravity, z-component

**default:** 0.0  $m/s^2$

○ ○ ○ ●

**W 30** `int` Compressibility for phase 2, using ideal gas theory for constant temperature

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**W 31** `double` Constant gas temperature with compressibility

**default:** 20.0  $C^\circ$

○ ○ ○ ●

**W 41\*** `double` Velocity vertical line source phase 1:  $xcenter, ycenter, zstart, zend, velocity, \beta$

Here,  $\beta$  ist the direction of the flow.

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

● ○ ○ ●

**W 90** `int` Non-Newtonian rheology

**0** OFF

**1** Herschel-Bulkley

● ○ ○ ●

**W 95** `double`  $\nu_0$

**default:** 1.0

● ○ ○ ●

**W 96** double  $\tau_0$

**default:** 1.0

● ○ ○ ●

**W 97** double  $K$  Consistency index

**default:** 0.00001

○ ○ ○ ●

**W 98** double  $n$  Power law index

**default:** 1.0

● ○ ○ ●

**W 101** int Use Mohr-Coulomb

**0** OFF

**1** ON

**default:** 0

○ ○ ○ ●

**W 102** double  $\phi_0, c$

**default:** 30.0 0.0

○ ○ ○ ●

**W 103** double Transition factor for Mohr-Coulomb

**default:** 1.0

○ ○ ○ ●

**W 110** int Numerical treatment of rheology

**1** use viscosity for rheology and yield stress

**2** use viscosity for rheology and momentum source term for yield stress

**3** use momentum source term for rheology and for yield stress

**default:** 1

**W 111** **int** Pressure for Mohr-Coulomb yield stress

- 1** hydrostatic
  - 2** hydrostatic and dynamic pressure
  - 3** hydrostatic pressure away from the interface and dynamic pressure closer to the interface (see W 112)
- default:** 1

**W 112** **double** Threshold factor for W 111 3

**default:** 2.1

## 2.15 X :: 6DOF

Valid for:

S F N C

**X 10** **int** Turn on 6DOF algorithm for floating bodies

- 0** OFF
  - 1** ON
  - 3** (Collar/Morsion) on NetForcing
- default:** 0

**X 11** **int** Turn the individual degrees of freedom of the floating body, the linear and the angular velocities:  $u, v, w, p, q, r$

When degrees of freedom are chosen to be prescribed, then they can be prescribed with either fixed external linear or angular velocities or different kinds of motion files.

- 0** OFF
  - 1** solver 6DOF equation
  - 2** prescribed
- default:** 1 ; 1 ; 1 ; 1 ; 1 ; 1

**X 12** `int` Turn on the force calculation for the floating body

**0** OFF

**1** ON

**default:** 1

**X 13** `int` Rigid body dynamics representation

**0** GCIBM with Euler angles

**1** GCIBM with Euler parameters

**2** Direct Forcing

**default:** 0

**X 18** `int` Relaxation method for flow velocities in the vicinity of the floating body.

**0** OFF

**1** ON

**default:** 0

**X 19** `int` print out interval for 6DOF log files

**default:** 1

**X 21** `double` Floating body homogenous density  $\rho$

Give either X 21 or X 22.

**default:** 900.0

**X 22** `double` Floating body mass  $m$

Give either X 21 or X 22.

**default:** 0.0

**X 23** `double` Floating body center of gravity  $x_G, y_G, z_G$

If not given, it will be calculated automatically. The center of gravity is also the origin of the non-inertial coordinate system.

**default:** 0.0 ; 0.0 ; 0.0

**X 24** `double` Floating body moments of inertia  $I_x, I_y, I_z$

If not given, it will be calculated automatically.

**default:** 0.0 ; 0.0 ; 0.0

**X 25** `double` Damping coefficient C for rotational motion around the three axes

**default:** 0.0 ; 0.0 ; 0.0

**X 31** `int` Boundary conditions on floating solid for parallel velocities

- 1** slip
- 2** no-slip
- 3** floating body velocities
- 4** 2nd-order extrapolation

**default:** 3

**X 33** `int` Boundary conditions on floating solid for pressure

- 1** Neumann
- 2** Extend

**default:** 1

**X 34** `int` Boundary treatment for new floating solid velocity

- 0** OFF
- 1** ON

**default:** 0

○ ○ ○ ●

**X 40 int** Type of force calculation

- 1 level set surface integration based on a Heaviside function
  - 2 Triangle based integration from the level set function
  - 3 Triangle based integration from the geometrical representation
- default:** 0

○ ○ ○ ●

**X 101 double** Initial Euler angles in [deg]:  $\phi_0, \theta_0, \psi_0$

Positive angles rotate the body clockwise. The option should not be used together with mooring lines. Moored systems should be rotated with X 183 instead (requires use of STL file).

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 102 double** Initial linear floating body velocity:  $u_0, v_0, w_0$

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 103 double** Initial angular floating body velocity:  $p_0, q_0, r_0$

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 110 double** Rectangular box floating body:  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 120 double** Sphere floating body:  $radius, x_{center}, y_{center}, z_{center}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**X 131 double** Cylinder in x-direction floating body:  $radius, height, x_{center}, y_{center}, z_{center}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**X 132 double** Cylinder in y-direction floating body:  $radius, height, x_{center}, y_{center}, z_{center}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**X 133 double** Cylinder in z-direction floating body:  $radius, height, x_{center}, y_{center}, z_{center}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ;

○ ○ ○ ●

**X 153 double** Two-sided wedge in z-direction floating body:  $x_{length}, y_{length}, z_{length}, x_{center}, y_{center}, z_{center}, x_{roll}, y_{pitch}, z_{yaw}$

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 163\* double** wedge floating body, each of the 6 points is given by the coordinates  $x_1, y_1, z_1, x_2, y_2, z_2, x_3, y_3, z_3, x_4, y_4, z_4, x_5, y_5, z_5, x_6, y_6, z_6$

**default:** [6x] 0.0 ; 0.0 ; 0.0

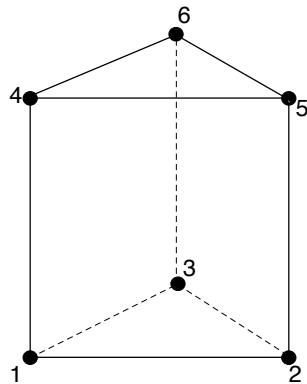


Figure 2.1: Definition of the wedge points for X 163

○ ○ ○ ●

**X 164 \* double** hexahedron floating body, each of the 8 points is given by the coordinates  $x_1, y_1, z_1, x_2, y_2, z_2, x_3, y_3, z_3, x_4, y_4, z_4, x_5, y_5, z_5, x_6, y_6, z_6, x_7, y_7, z_7, x_8, y_8, z_8$

**default:** [8x] 0.0 ; 0.0 ; 0.0

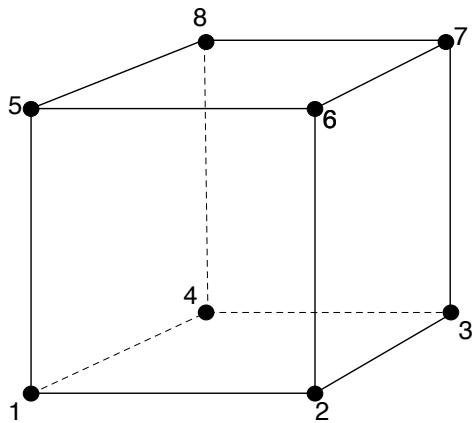


Figure 2.2: Definition of the hexahedron points for X 164

○ ○ ○ ●

**X 180** `int` Read STL file “floating.stl” for floating body geometry

**0** OFF

**1** ON

**default:** 1

○ ○ ○ ●

**X 181** `double` Scale STL geometry

**default:** 1.0

○ ○ ○ ●

**X 182** `double` 3D linear translation of STL object;  $\Delta x, \Delta y, \Delta z$

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 183** `double` 3D rotation;  $x_{origin}, y_{origin}, z_{origin}, \phi, \theta, \psi$   
Positive angles rotate the body clockwise.

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 184** `int` Invert inside/outside for STL geometry

**1** regular

**2** invert

**default:** 1

○ ○ ○ ●

**X 210** `double` Prescribing constant linear velocities of the floating body  $u, v, w$

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 211** `double` Prescribing constant angular velocities of the floating body  $p, q, r$

**default:** 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 221** `double` Vector for vector based motion file  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}$ .

This option requires the 'motionfile\_vec.dat' file to be present in the simulation folder.

In the file, columns with  $t$  and  $S$  need to be given.

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 310** `int` Turn on mooring modelling for floating bodies

**0** off

**1** Static catenary

**2** Quasi-Static FEM

**3** Dynamic DGSEM

**4** Spring (using X 312)

**default:** 0

○ ○ ○ ●

**X 311** \* `double` Specify mooring line configuration  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}, w, \rho_c, EA, d, I, H, P, Q$ .

Each mooring line of length  $l[m]$  and diameter  $d[m]$  has to be specified separately using this option.  $\vec{x}_{start}$  is the mounting point at the bottom,  $\vec{x}_{end}$  the mounting point at the body. The weight of the line is calculated from the specific weight in air

$w$ [kg/m], its density  $\rho_c$ [kg/m<sup>3</sup>] and its elasticity times area  $EA$ .  $H$  represents the number of elements for the numerical approaches,  $P$  is the polynomial order for the dynamic approach. A relaxation technique can be included for this approach using the time factor  $Q$ .

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0 ; 0 ; 0.0

**X 312 \*** **double** Specify spring configuration  $x_{start}, x_{end}, y_{start}, y_{end}, z_{start}, z_{end}, k, T_0$ .

Each spring with stiffness  $k$ [N/m] has to be specified separately using this option.  $\vec{x}_{start}$  is the mounting point at the bottom,  $\vec{x}_{end}$  the mounting point at the body.  $T_0$  is the pre-tension.

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

**X 313** **int** Initial rotation of mooring end points with the floating body (specified with X 101).

**0** off

**1** on

**default:** 0

**X 320** **int** Turn on net modelling. Each net has to be specified separately using this option.

**0** off

**2** Static net cylinder

**3** Static net wall

**4** Static sheets following floating body

**12** Dynamic net cylinder

**13** Dynamic net wall

**default:** 0

**X 321 \*** **double** Specify net material  $S_n, d_t, l_t, d_k, \rho, N_d, N_l$ .

Each net has to be specified separately using this option.  $S_n$  is the solidity,  $d_t$  is the diameter of each twine,  $l_t$  its length,  $d_k$  is the diameter of the knots (0.0 if knotless),  $\rho$  is the density of the net material. The net is discretised in  $N_d$  meshes along  $D$  and  $N_l$  meshes along  $L$  (to be specified in X 322).

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 322 \*** `double` Specify net dimensions  $D, L, x_0, y_0, z_0, \phi, \theta, \psi$ .

Each net has to be specified separately using this option.  $D$  is the diameter or width of the net,  $L$  is the height. The structure is first rotated around the origin with the given angles around the x-, y- and z-axis. Then, the centre point of the bottom of the structure is moved to the position  $(x_0, y_0, z_0)$ .

**default:** 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0 ; 0.0

○ ○ ○ ●

**X 323 \*** `double` mass in air, `double` diameter and `double` length of sinker for dynamic net models.

**default:** 0.0; 0.0; 0.0

○ ○ ○ ●

**X 324 \*** `double` x-location, `double` y-location and `double` z-location of knot probes.

The dynamic net model searches for the nearest knot to track.

**default:** na

○ ○ ○ ●

**X 325 \*** `double` time step, `double` relaxation in x-direction `double` relaxation in y-direction `double` relaxation in z-direction for dynamic net models. Set time step to zero to use the fluid domain time stepping.

**default:** 0.0; 0.0; 0.0; 0.0

# 3. Compiling the Code

REEF3D depends on gcc compilers, OpenMPI, HYPRE and Eigen. If the system has included all prerequisites, section 3.1 can be skipped. Users with Windows 10 systems can follow the instructions in section 3.4.

For a usage only compilation, see section 3.2; for developers see section 3.3.

## 3.1 Install Required Packages

Here, the exemplary installation of REEF3D on Mac OS X is shown. Installations on Linux systems can be done in a similar way.

### 3.1.1 Installing GCC GNU compiler on Mac OS X

This step is Mac only because Linux systems typically have the compilers included already.

- Open the terminal and install the command line tools, which include gcc and g++ using:  
`xcode-select --install`
- Install Homebrew following the instruction on <https://brew.sh/>
- Install gfortran using the following command in the terminal:  
`brew install gfortran`
- Confirm working C, C++ and fortran compilers (in a new terminal, using the commands individually):  
`gcc -v`  
`g++ -v`  
`gfortran -v`

The output of each of these commands is a path. If you do not get any output or an error message, that particular compiler is not installed or has had problems. This needs to be fixed before you proceed to the next step.

### 3.1.2 Installing OpenMPI

REEF3D depends on the third-party library MPI for the parallelization of the code. MPI is very portable and the de-facto standard for the parallelization of high performance computations. Several interesting documents can be found on the official MPI homepage: <http://www.mpi-forum.org/>. Depending on the operation system, several MPI distributions exist. We recommend OpenMPI for Mac OS X and Linux.

- Download source code - <http://www.open-mpi.org/software/> and unzip the archive.

- Open ‘Terminal’ and navigate to the location of the archive and type:  
 (To ensure all the compilers are given MPI support:)  
`./configure CC=gcc CXX=g++ F77=gfortran FC=gfortran --prefix=/usr/local/openmpi`
- Next compile:  
`make all -j N` (with N being the number of available cores on your computer)
- Finally, install by:  
`sudo make install`
- Add the following openmpi locations to the PATH:  
`/usr/local/openmpi/lib`  
`/usr/local/openmpi/bin`
- this can be done for example using the following command:  
`echo 'export PATH=/usr/local/openmpi/bin:/usr/local/openmpi/lib/:$PATH' >>~/.bashrc`

Remember that the file `.bashrc` is a hidden file in your home folder. If you obtain an error message that tells you the file does not exist, you can use the file `.bash_profile`. One of these two files will be available on your system. You can find this by typing the command:

`ls -a`  
when you are in your home folder.

- Close the terminal and open a new terminal to check if the addition to the path has been successful using:  
`echo $PATH` (should return several paths, amongst which the ones you just added are the first to be listed). This should be in order before you proceed to the next step.

### 3.1.3 Installing HYPRE

HYPRE Cen [2015], Falgout et al. [2006] is a library for high-performance linear solvers and preconditioners and can be downloaded here: [http://computation.llnl.gov/project/linear\\_solvers/](http://computation.llnl.gov/project/linear_solvers/). With its underlying rectilinear grid, REEF3D can take advantage of using HYPRE’s geometric multigrid options. The geometric multigrid preconditioner PFMG Ashby and Flagout [1996] together with one of the conjugate gradient accelerators results in extremely small solution times for the Poisson equation for the pressure.

- Download source code and unzip the archive.
- Open ‘Terminal’ and navigate to the “src” folder inside the HYPRE archive and type:  
`./configure --prefix=/usr/local/hypre`

Note: if HYPRE does not find the FORTRAN compiler, use the path returned by the command which gfortran or if you know it to be at a particular location provide this information as follows:

`./configure --prefix=/usr/local/hypre F77=/usr/bin/gfortran`

- Next compile:  
`make all -j N` (with N at least the number of available cores)
- Finally, install by:  
`sudo make install`
- Add the following HYPRE locations to the PATH:  
`/usr/local/hypre/include`  
`/usr/local/hypre/lib`  
The command is the same as that given above for the MPI installation, but with the paths listed above:  
`echo 'export PATH=/usr/local/hypre/include:/usr/local/hypre/lib:$PATH' >>~/ .bashrc`

## 3.2 User Installation: Makefile

For users who are interested in running the code only, the easiest way is the compilation of REEF3D and DIVEMesh with the Makefiles given in the code repository. After installing MPI and HYPRE, open the terminal and go to the main directories of the code.

- For DIVEMesh, type:  
`make -j N` (with N at least the number of available cores)
- For REEF3D, type:  
`make -j N` (with N at least the number of available cores)

The code will compile in the /build directory and the executable is placed in the /bin directory. In case of errors during the compilation, check the Makefile for correct paths to the external libraries.

## 3.3 Developer Installation: CodeLite

REEF3D is written in highly modular C++ in order to provide efficient code development and maintenance. REEF3D is designed with cross-platform usage in mind. The source code can be compiled under Mac OS X, Windows and Linux. Since the user experience should be the same on all platforms when developing the code, CodeLite was chosen for the Integrated Development Environment (IDE). The reasons for this are:

- it is free under the GPL license
- it is cross-platform and works under Mac OS X, Windows and Linux
- it works with a variety of compilers
- it works with the MPI library
- support for parallel builds

CodeLite can be downloaded here. When the REEF3D source is downloaded, a CodeLite project file is included, so all header and source files are sorted in a tree structure. This should make the orientation and navigation in the source code easier.

REEF3D and DiveMESH come with Makefiles to compile the code. CodeLite can utilize these Makefiles by activating:

Workspace > Open Active Project Settings > Customize > Enable custom build

Point the working directory to the source code folder and set the target commands as:

- For DIVEMesh:

Build: `make -f ./Makefile -j N` (with N at least the number of available cores)

Clean: `make -f ./Makefile clean` (with N at least the number of available cores)

Rebuild: `make -f ./Makefile -j N` (with N at least the number of available cores)

- For REEF3D:

Build: `make -f ./Makefile -j N` (with N at least the number of available cores)

Clean: `make -f ./Makefile clean` (with N at least the number of available cores)

Rebuild: `make -f ./Makefile -j N` (with N at least the number of available cores)

Afterwards, the code can be compiled by clicking Build > Build Project or pressing F7.

## 3.4 Installing REEF3D on Windows 10

Windows 10 offers the feature to run a Unix bash terminal, given that you have kept your Windows up-to-date.

- Have administrator privileges on your 64-bit computer
- Check the version of the Windows 10 by navigating to Settings> System> About. You need to have a minimum of version 1809. If you have an earlier version, check for updates and install the update giving you version 1809 or above. Restart the computer when prompted. This can take several restarts. No support is provided for Windows update or for versions below 1809.

### 3.4.1 Getting Unix Terminal on Windows 10

- Look for Ubuntu App in the Microsoft App Store. Ensure that all prerequisites are fulfilled and update the system if necessary.
- Search for 'Ubuntu' in the Start menu and open the bash terminal. You will be prompted to enter a username and password. This can be different from your Windows credentials and has nothing to do with the Windows credentials.
- The Ubuntu terminal starts up at your Ubuntu home folder. To access your Windows C:\Users location, use the command:  
`cd /mnt/c/Users/`

- Your Unix home directory is unfortunately a hidden folder. You will have to activate the option to View hidden and system files in order to access this using the Windows explorer. To do this:
  - Open your Windows explorer.
  - Click on the View tab at the top of the window
  - Click on the last icon on the toolbar that says Options
  - In the pop-up window, click on the View tab
  - Under the title Hidden Files and Folders, Click on the radio button for “Show hidden files, folders and drives”
- Your Ubuntu home folder can now be accessed at C:\Users\<Windows user name>\AppData\Local\Packages\CanonicalGroupLimitedUbuntuonWindows\LocalState\rootfs\home\<your ubuntu username> .

### **3.4.2 Installing Compilers**

- Open the bash terminal and update the apt package (package manager) using:  
`sudo apt update`
- To install the necessary GCC and fortran compilers, type:  
`sudo apt install gcc g++ gfortran cmake libtool libglu1-mesa`
- Use the following commands to find the path to the compilers and note down for later use:  
`which gcc`  
`which g++`  
`which gfortran`
- Get openmpi using:  
`sudo apt install libopenmpi-dev`
- Continue to install Hypre using the steps listed in Sec 3.1.3.

### **3.4.3 Additional step for Windows for visualization using Paraview**

Ubuntu on Windows does not support graphics and the Windows OS treats the Ubuntu system as hidden directory. In order to overcome this hurdle in Windows and visualise the results using Paraview, the following additional steps are necessary.

- Download the same version of Paraview for both Windows and Linux from [paraview.org](http://paraview.org)
- Move the Linux version, the .tar.gz archive, to the Ubuntu home folder by opening an Ubuntu terminal and using:  
`cp /mnt/c/Users/<windows username>/Downloads/<ParaviewLinuxArchiveName> ./`

- Use the following command to unpack the archive:  
`tar -xvf <ParaviewArchiveName>`
- Install Paraview on Windows as well, using the installer that you downloaded. Make sure that you are indeed installing the SAME version of Paraview on both Ubuntu and Windows.

# 4. Running the Code

## 4.1 DIVEMesh

After compilation, copy the DIVEMesh executable into the simulation folder. The simulation folder should be covered by the PATH variable. Make sure that the "control.txt" file is inside the simulation folder, which gives the input for DIVEMesh (see also DIVEMesh User's Guide for more information). Run DIVEMesh by double-clicking the executable or typing in the terminal while being in the path of the folder:

```
./DiveMESH
```

This will generate the grid files. If the program does not execute properly, please make sure that all input commands comply with the definitions in the User's Guide.

## 4.2 REEF3D

After compilation, copy the REEF3D executable into the simulation folder. The simulation folder should be covered by the PATH variable, otherwise the program will not find the MPI libraries. If the program does not execute properly, please make sure that all input commands comply with the definitions in the User's Guide. If a command does not have the required number of input variables, REEF3D will stop at "read ctrl" in the terminal print out.

All REEF3D cases are started with the following terminal command:

```
mpirun -n number of processes reef3d
```

For a computer with four processors, this command then becomes:

```
mpirun -n 4 reef3d
```

Alternatively this command can be used:

```
mpiexec -n 4 reef3d
```



# **5. Post-Processing**

## **5.1 Paraview**

Paraview is an open-source visualization tool. It is cross-platform and works on Mac OSX, Windows and Linux. Paraview supports parallel output files as well as parallel rendering through distributed computing. Because it is very flexible, efficient and free, it is the preferred program for visualizing REEF3D's computed results.

## **5.2 Additional step for Windows users to use paraview**

In order to access the results files created in the Ubuntu system and open them in the Wondows system, the following additonal steps are essential.

- Open a Ubuntu terminal and navigate to the location of the Paraview installation. If you have followed the instructions exactly, then it is:  
`cd <ParaviewFolderName>/bin`
- Start a paraview server using:  
`./pvserver`
- You will get an output in the terminal that will show you a URL of the form:  
`cs://ABCD789: 111111` (This is an example. The actual name will be different, use what you are shown on your screen)  
The first part- ABCD789 is the host name, the second part after the colon (:) is the port number. Make a note of this.
- Open Paraview on your Windows system. Click on the “Connect to server” button (two white towers with a green dot, third icon from the left on the topmost toolbar).
- In the pop-up dialog box, click on Add server. In the dialog box that opens now, enter your host name in the box for Host, and the port number in the box for Port. Use any name you would like in the topmost box for Name.
- Click on configure. And then on save. Click on the saved configuration you are shown on the list and click on Connect.

## **5.3 Visualising Two-Phase Flow Results in Paraview**

REEF3D can write out a range of different result files, depending on the hydrodynamics module used. In this section, the procedure is presented how to visualize the free surface obtained from two-phase flow simulations through the CFD module. For these type of calculations, the level set method is employed which defines the free surface implicitly. The free surface is the zero-contour of the level set function (the variable phi in the paraview

dropdown menu). In the following, the procedure is presented on how to extract the zero level set contour from the .vtu files printed out by the CFD module.

Open the .pvtu fileset in paraview through File> open and selecting the .pvtu file in the popup window. Click on outline in the paraview tool bar as shown in Fig.5.1 to obtain the outline of the wave tank. Click on the Slice filter on the toolbar (circled in Fig.5.1) and select “y-normal” in the properties sidebar to generate a 2D slice along the length of the numerical wave tank as shown in Fig.5.2. The slice can be colored according to the various variables available in the dropdown menu. To obtain the free surface, select the slice in the pipeline browser and click on the Contour filter on the tool bar and select Contour by: phi in the properties sidebar with value range 0 as shown in Fig.5.3. The geometry of the structure used in the wave tank is included by adding the .vtu file generated by DiveMesh to obtain the result shown in Fig.5.4.

To obtain a 3D visualization of the free surface, open the .pvtu file in Paraview and directly use the Contour filter on as shown in Fig.9.44.

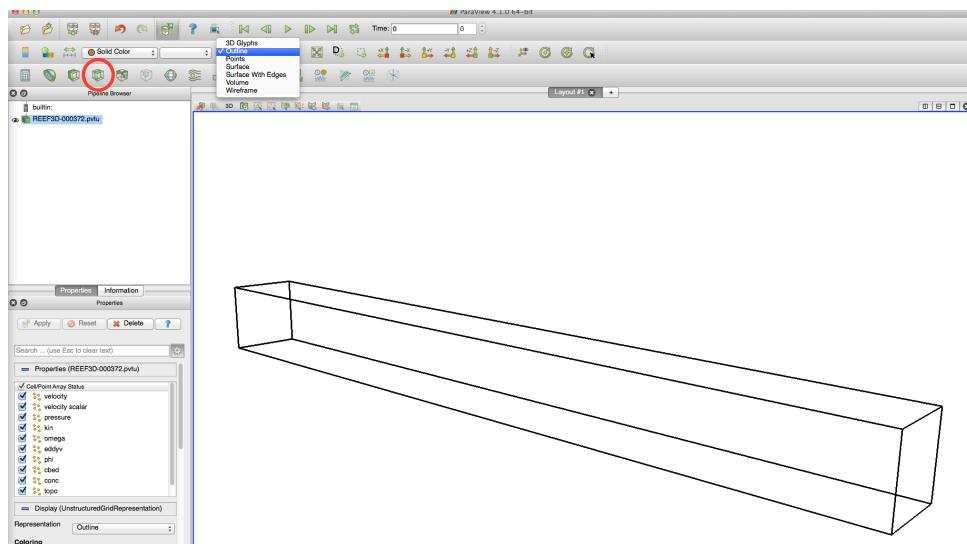


Figure 5.1: Outline of the numerical wave tank

## Generating screenshots and animations from Paraview

Use File> Save Screenshot from the drop down menu, input the desired resolution, click OK and select the file type in the next window to save a screenshot from Paraview.

To generate an animation, use File>Save Animation from the dropdown menu. Enter the desired frame rate (recommended value: 1/P30) and the range of time steps to be included in the animation in Frame range. Select the desired file type in the next window.

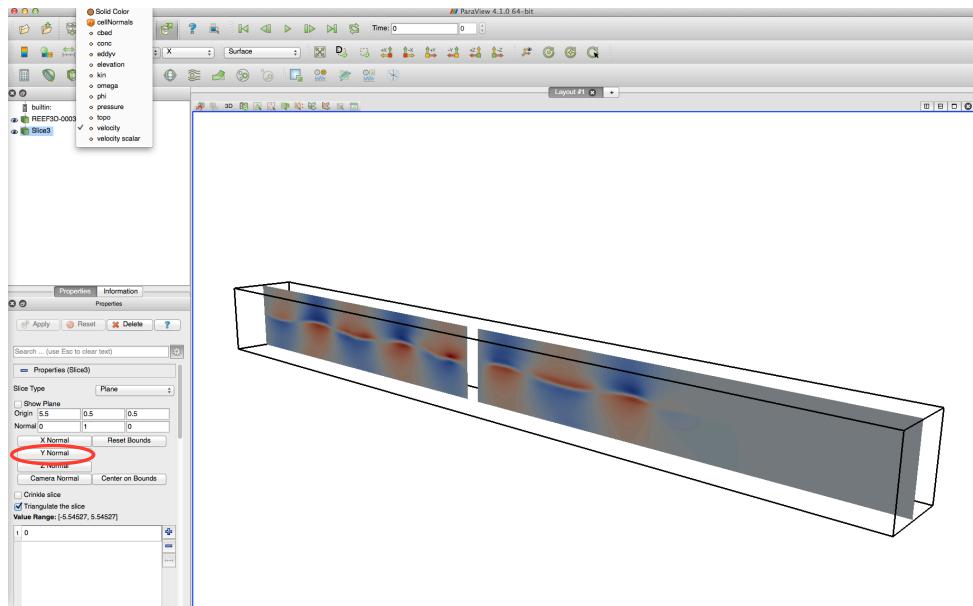


Figure 5.2: Creating a 2D slice

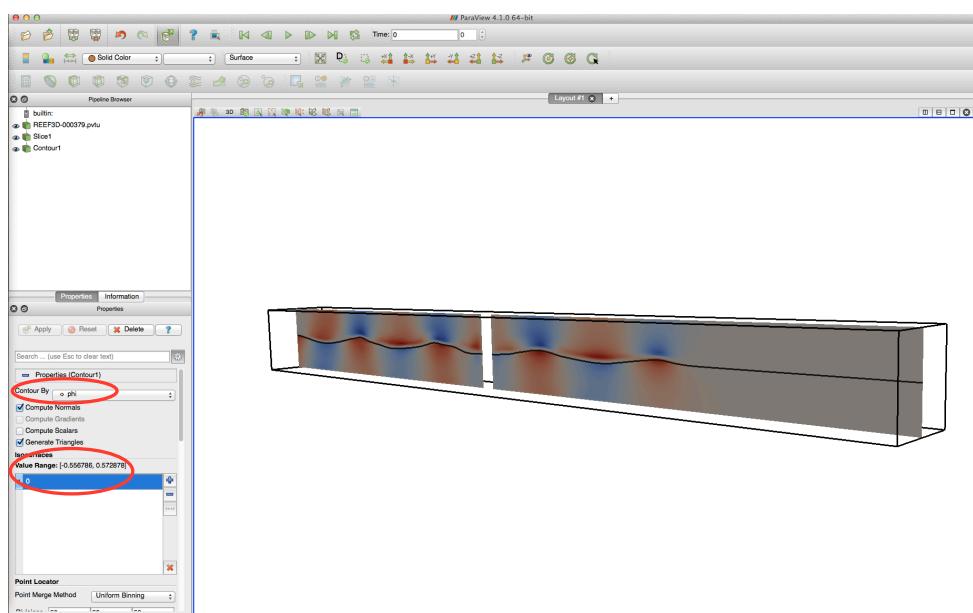


Figure 5.3: Generating the free surface

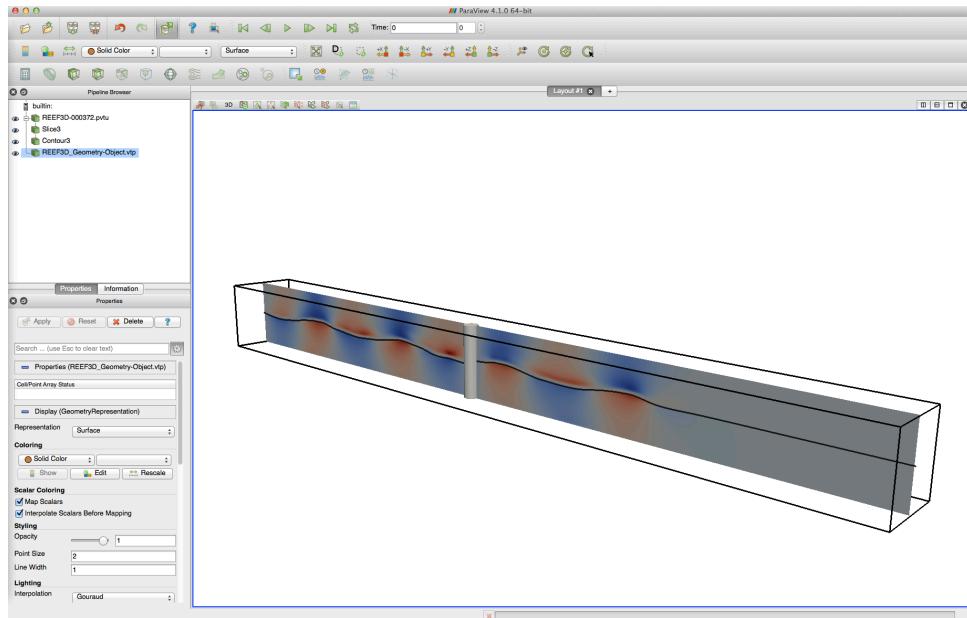


Figure 5.4: Including the geometry of the structure

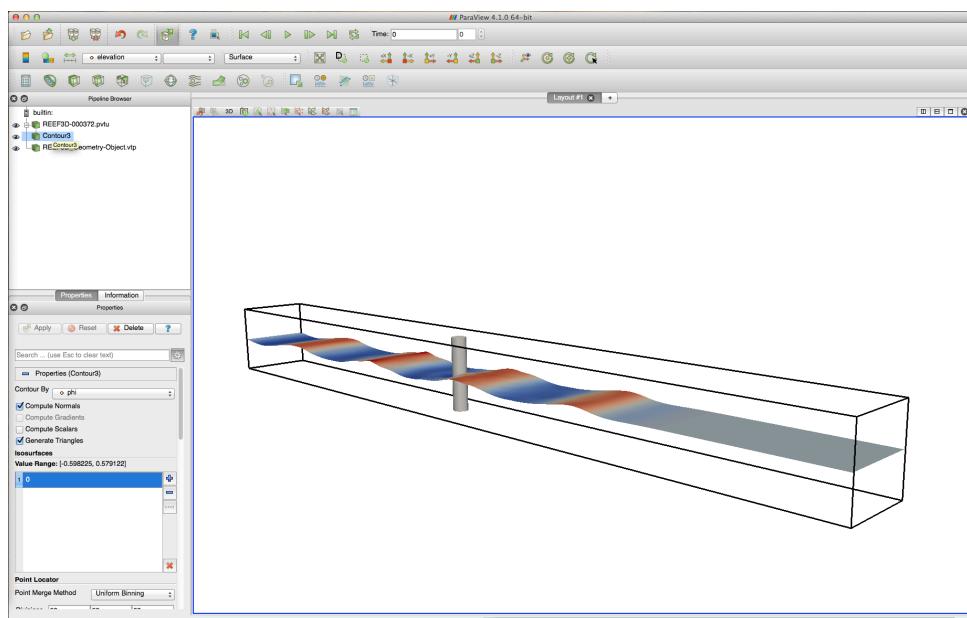


Figure 5.5: Visualizing the free surface in 3D

# 6. Running the Code

In this chapter some examples are given, the raw input files can be downloaded from the REEF3D website. The focus is on helping the user to understand the workflow of REEF3D. The control files are not necessarily optimized, when it comes to grid convergence and resolution. Rather, the mesh size are kept to reasonable sizes in order to be able to run the cases on laptop or desktop machines.

All REEF3D cases are started with the following terminal command:

```
mpirun -n number of processes reef3d
```

For a computer with four processors, this command then becomes:

```
mpirun -n 4 reef3d
```

Alternatively this command can be used:

```
mpiexec -n 4 reef3d
```

The 'ctrl.txt' file contains the input for REEF3D. The idea of the input structure is to use a capital letter for the type of functionality it describes, e.g. 'T' for turbulence. The letter is followed by a number for the individual option. All available options are listed in the previous sections of this document together with the required input, i.e. number of specified values, type of value (int or double) and the default value which will be used by REEF3D in case no input is given by the user.

The 'control.txt' files describe the mesh and are read by DIVEMesh, a separate open-source mesh editor. The structure is the same as in REEF3D and the overview over all available functions is available in the separate DIVEMesh User Guide. Upon executing DIVEMesh, grid files for each of the parallel subprocesses will be generated. The source code for DIVEMesh can also be downloaded from the REEF3D website.



# 7. Tutorial | REEF3D::SFLOW

For visualization, open the REEF3D\_SFLOW\_VTP and REEF3D\_SFLOW\_VTP\_BED folders.

## 7.1 2D linear waves

Linear waves over a flat bed.

### 7.1.1 DIVEMesh: control.txt

```
C 11 6 // left side: wall boundary
C 12 3 // side: symmetry plane
C 13 3 // side: symmetry plane
C 14 7 // right side: wall boundary
C 15 21 // bottom: wall boundary
C 16 3 // top: wall boundary
B 1 0.02 // mesh size dx
B 10 0.0 28.0 0.0 0.02 0.0 1.0 // rectangular domain size
M 10 4 // number of processors
M 20 2 // decomposition method
```

### 7.1.2 REEF3D: ctrl.txt

```
A 10 2 // turn on SFLOW
A 210 3 // 3rd-order Runge-Kutta time discretization
A 211 4 // conservative WENO dizcretizxation for velocities
A 220 2 // non-hydrostatic pressure with quadratic assumption
A 243 1 // absolute wetting criterion value
A 246 1 // turn on wetting and drying
B 90 1 // turn on iowave
B 92 2 // linear wave theory
B 91 0.01 4.0 0.0 // wave amplitude and wave period
```

B 96 4.0 0.0 8.00 // wave generation and absorption relaxation zone lengths  
B 98 2 // use relaxation wave generation  
B 99 2 // use relaxation wave absorption  
B 160 5  
F 42 1.0 F 60 0.5  
N 41 90.0 N 47 0.5  
M 10 8  
P 10 1 P 30 0.1  
P 52 0.01 P 53 1 P 54 100  
W 22 -9.81

## **8. Tutorial | REEF3D::FNPF**



# 9. Tutorial | REEF3D::CFD

## 9.1 2D Dam Break

This a 2D dam break case. Comments to the control files are marked by blue fonts.

### 9.1.1 DIVEMesh: control.txt

```
C 11 21 // left side: wall boundary
C 12 3 // side: symmetry plane
C 13 3 // side: symmetry plane
C 14 21 // right side: wall boundary
C 15 21 // bottom: wall boundary
C 16 21 // top: wall boundary
B 1 0.005 // mesh size dx
B 10 0.0 0.6 0.0 0.005 0.0 0.6 // rectangular domain size
M 10 4 // number of processors
```

### 9.1.2 REEF3D: ctrl.txt

```
D 10 4 // Conservative WENO discretization for velocities
D 20 2 // implicit diffusion treatment
D 30 1 // projection method for the pressure
F 30 3 // 3rd-order Runge-Kutta Scheme for Level Set Time Treatment
F 40 3 // 3rd-order Runge-Kutta Scheme for Reinitialization Time Treatment
F 50 4 // Level set function is not fixed at inlet or outlet
F 54 0.15 // x-coordinate for end fluid phase one
F 56 0.3 // z-coordinate for end fluid phase one
N 10 14 // HYPRE Struct BiCGStab pressure solver
N 11 11 // HYPRE Struct PFMG preconditioner
N 40 1 // 2nd-order Adams-Bashforth time treatment for velocities
N 45 25000 // maximum number of iterations
N 47 0.1 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu print out
P 30 0.01 // print out interval based on simulation time
W 22 -9.81 // gravity
```

### 9.1.3 Results

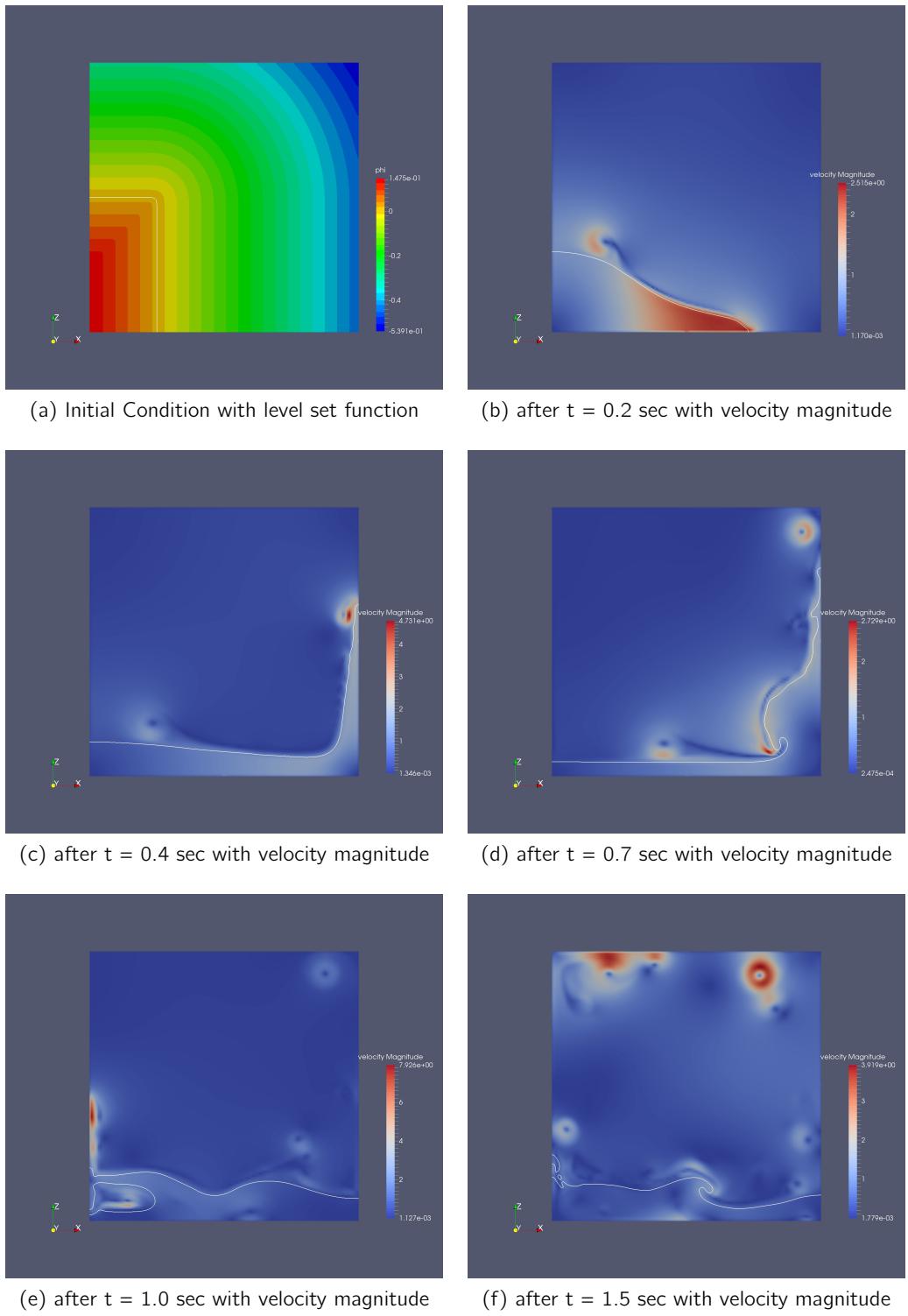


Figure 9.1: Dam Break

## 9.2 3D Dam Break with Obstacle

This case is optimized for fast processing. In order to get a more detailed free surface result, just change B 1 to e.g. 0.0125.

### 9.2.1 DIVEMesh: control.txt

```
C 11 21 // left side: wall boundary
C 12 21 // side: wall boundary
C 13 21 // side: wall boundary
C 14 21 // right side: wall boundary
C 15 21 // bottom: wall boundary
C 16 21 // top: wall boundary
B 1 0.025 // mesh size dx
B 10 0.0 2.0 0.0 1.0 0.0 1.0 // rectangular domain size
O 10 1.2 1.4 0.4 0.6 0.0 1.0 // rectangular obstacle size
M 10 4 // number of processors
```

### 9.2.2 REEF3D: ctrl.txt

```
D 10 4 // Conservative WENO discretization for velocities
D 20 2 // implicit diffusion treatment
D 30 1 // projection method for the pressure
F 30 3 // 3rd-order Runge-Kutta Scheme for Level Set Time Treatment
F 40 3 // 3rd-order Runge-Kutta Scheme for Reinitialization Time Treatment
F 50 4 // Level set function is not fixed at inlet or outlet
F 54 0.5 // x-coordinate for end fluid phase one
F 56 0.7 // z-coordinate for end fluid phase one
N 10 14 // HYPRE Struct BiCGStab pressure solver
N 11 11 // HYPRE Struct PFMG preconditioner
N 40 3 // 3rd-order Runge time treatment for velocities
N 41 25.0 // maximum simulation time
N 45 25000 // maximum number of iterations
N 47 0.1 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu print out
P 30 0.01 // print out interval based on simulation time
T 10 0 // no turbulence model
W 22 -9.81 // gravity
```

### 9.2.3 Results

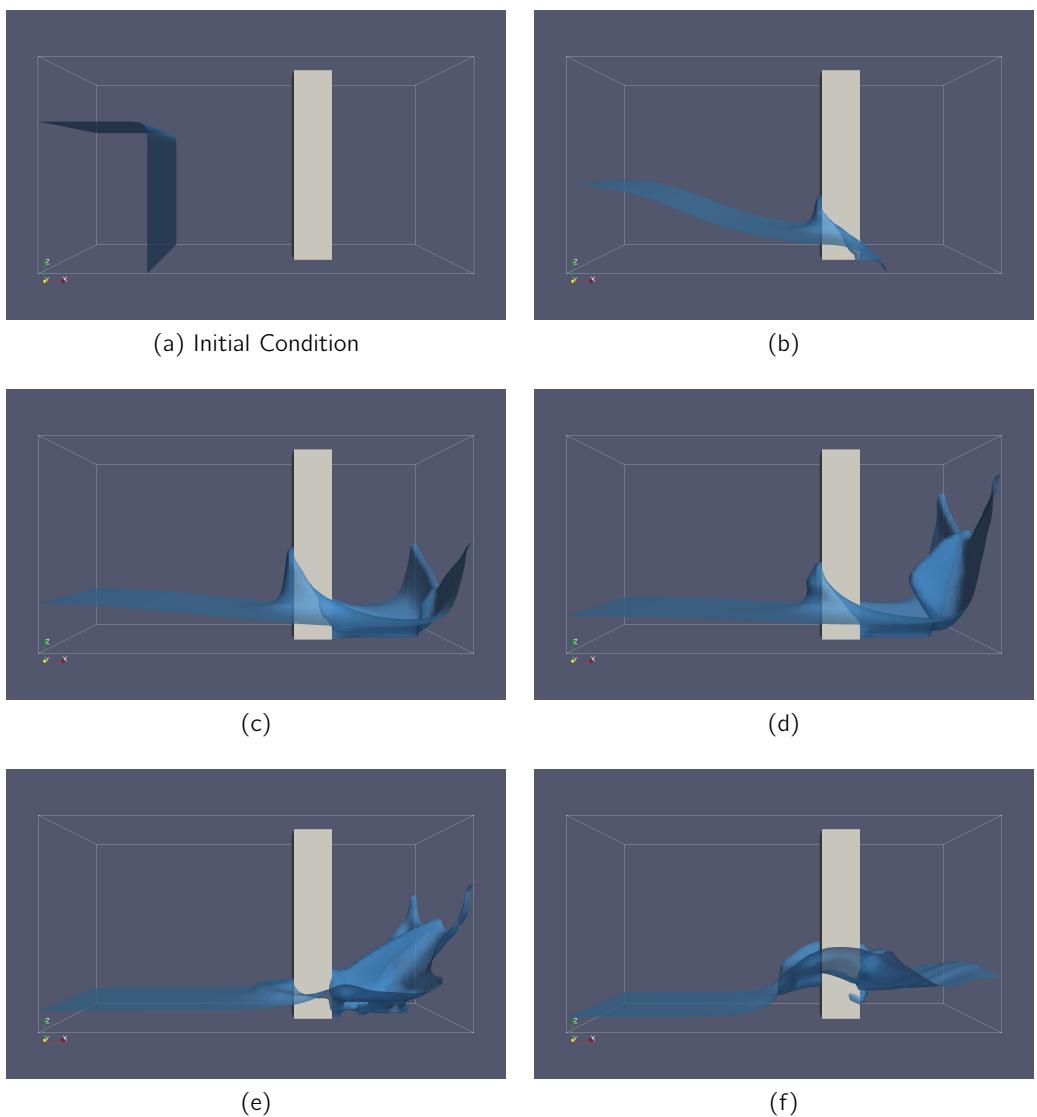


Figure 9.2: 3D Dam Break with Obstacle

## 9.3 2D Vortex Shedding

### 9.3.1 DIVEMesh: control.txt

```
C 11 1 // left side: wall boundary
C 12 3 // side: wall boundary
C 13 3 // side: wall boundary
C 14 2 // right side: wall boundary
C 15 21 // bottom: wall boundary
C 16 21 // top: wall boundary
B 1 0.01 // mesh size dx
B 10 0.0 4.0 0.0 0.01 0.0 1.0 // rectangular domain size
O 32 1.0 0.5 0.1 // cylinder
M 10 4 // number of processors
```

### 9.3.2 REEF3D: ctrl.txt

```
B 10 0 // turn off wall functions for the velocities
B 50 0.0001 // wall roughness  $k_s$ 
B 60 1 // turn on ioflow
D 10 4 // Conservative WENO discretization for velocities
D 20 1 // explicit diffusion treatment
D 30 1 // projection method for the pressure
F 30 0 // turn off level set transport
F 40 0 // turn off level set reinitialization
I 11 1 // Initialize Velocities with potential flow solver
N 40 3 // 3rd-order Runge Runge time treatment for velocities
N 45 10000 // maximum number of iterations
N 47 0.3 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu print out
P 20 10 // print out interval based on iterations
P 62 0.0 0.0 0.005 0.005 0.0 1.0 // line probe
T 10 0 // no turbulence model
W 10 0.000006 // discharge
```

### 9.3.3 Results

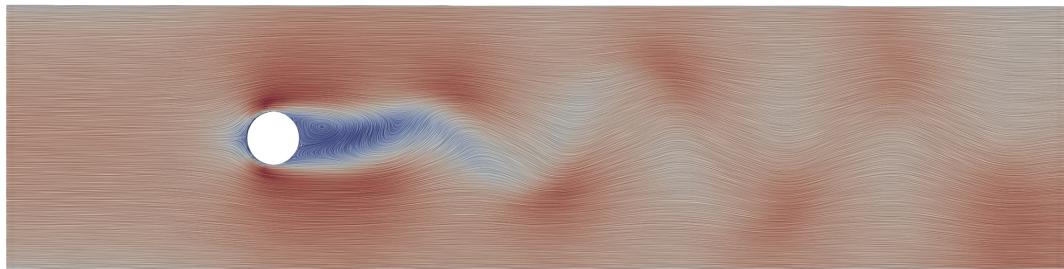


Figure 9.3: 2D vortex shedding with stream lines

## 9.4 Flow through a Narrow Contraction

For the flow through a narrow contraction the open channel flow capabilities in ioFlow are used. Just for information: the grid is rather coarse for this flow situation. The grid can be easily refined by changing the B1 parameter in the DIVEMesh 'control.txt' file, depending on the performance of your computer.

### 9.4.1 DIVEMesh: control.txt

```
C 11 1 // inflow boundary
C 12 21 // wall boundary
C 13 21 // wall boundary
C 14 2 // outflow boundary
C 15 21 // bottom: wall boundary
C 16 3 // top: symmetry plane
B 1 0.05 // mesh size dx
F 3 0.0 0.5 // Freecoor: vertical extension of channel,  $z_s$  and  $z_e$ 
F 11 0.0 0.8 0.6 0.6 // Freecoor:  $x_s$ ,  $x_e$ ,  $y_s$ ,  $y_e$  for first point of left side in flow direction
F 11 0.8 1.1 0.6 0.4
F 11 1.1 1.4 0.4 0.4
F 11 1.4 1.7 0.4 0.6
F 11 1.7 2.5 0.6 0.6
F 21 0.0 0.8 0.0 0.0 // Freecoor:  $x_s$ ,  $x_e$ ,  $y_s$ ,  $y_e$  for first point of right side in flow direction
F 21 0.8 1.1 0.0 0.2
F 21 1.1 1.4 0.2 0.2
F 21 1.4 1.7 0.2 0.0
F 21 1.7 2.5 0.0 0.0
M 10 4
```

#### 9.4.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities
B 11 1 // use wall functions for the turbulence model
B 50 0.0001 // wall roughness  $k_s$ 
B 60 1 // turn on ioFlow
D 10 4 // Conservative WENO for velocities
D 20 2 // implicit diffusion treatment
D 30 1 // projection method for the pressure
F 30 3 // 3rd-order Runge-Kutta scheme for the level set method
F 40 3 // 3rd-order Runge-Kutta scheme for reinitialization time treatment
F 42 0.5 // length for level set initialization, instead of maximum length in domain
F 50 2 // keep outflow water level fixed
F 60 0.15 // initial free surface location
I 10 1 // initialize everything, including velocities with potential flow solver
N 10 14 // HYPRE Struct BiCGStab pressure solver
N 11 11 // HYPRE Struct PFMG preconditioner
N 40 6 // 3rd-order fractional step method for the velocities
N 45 12000 // maximum number of iterations
N 47 0.3 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu printout
P 20 10 // print out interval based on iterations
T 10 2 //  $k-\omega$  turbulence model
W 10 0.05 // inflow discharge
W 22 -9.81 // gravity
```

#### 9.4.3 Results

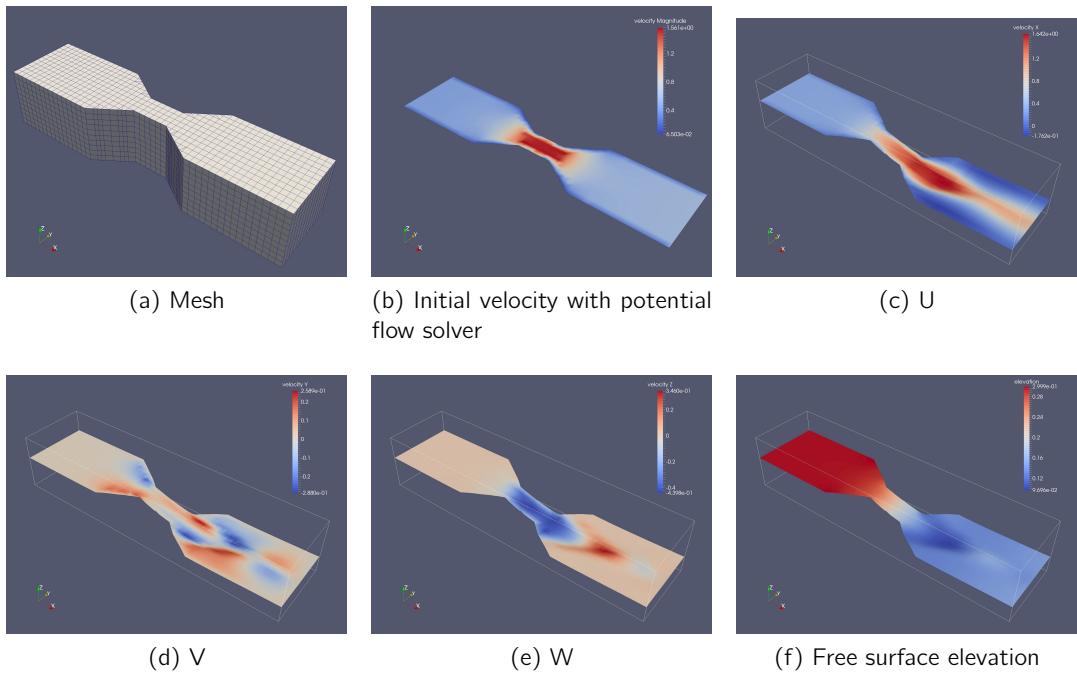


Figure 9.4: Narrow Contraction

## 9.5 Flow around a Circular Pier

### 9.5.1 DIVEMesh: control.txt

C 11 1 // inflow boundary

C 12 21 // wall boundary

C 13 21 // wall boundary

C 14 2 // outflow boundary

C 15 21 // bottom: wall boundary

C 16 3 // top: symmetry plane

B 1 0.025 // mesh size dx

B 10 0.0 1.5 0.0 1.0 0.0 0.4 // rectangular domain size

O 33 0.5 0.5 0.075 // vertical circular cylinder

M 10 4 // number of parallel processes

### 9.5.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions on the velocities
B 11 1 // use wall functions for the turbulence model
B 50 0.0001 // wall roughness  $k_s$ 
B 60 1 // turn on ioFlow
D 10 4 // Conservative WENO discretization for the velocities
D 20 2 // implicit diffusion treatment
D 30 1 // SIMPLE method for the pressure
F 30 3 // second-order implicit time treatment for the level seth method
F 40 3 // 3rd-order Runge-Kutta scheme for reinitialization time treatment
F 50 2 // keep outflow water level fixed
F 60 0.2 // initial free surface location
I 10 1 // initialize everything, including velocities with potential flow solver
N 10 14 // HYPRE Struct BiCGStab pressure solver
N 11 11 // HYPRE Struct PFMG preconditioner
N 40 6 // 3rd-order fractional step method for the velocities
N 45 100000 // maximum number of iterations
N 47 0.3 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu print out
P 20 10 // print out interval based on iterations
T 10 2 //  $k - \omega$  turbulence model
W 10 0.1 // inflow discharge
W 22 -9.81 // gravity
```

### 9.5.3 Results

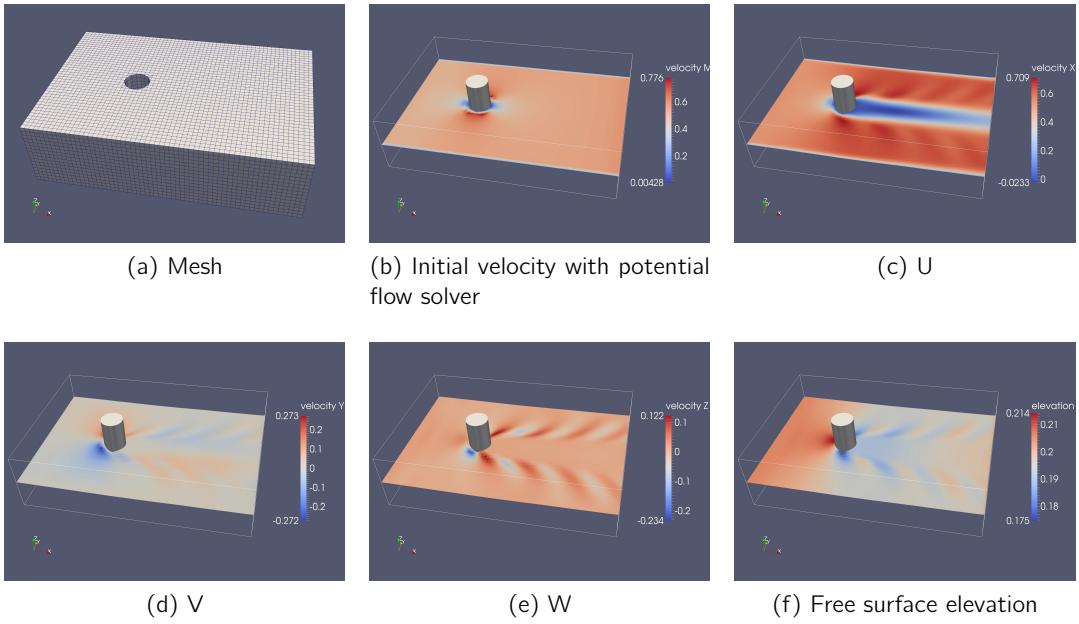


Figure 9.5: Flow around Circular Cylinder

## 9.6 Rectangular Wave Tank

For the case of a simple rectangular wave tank without obstacles, the simulation can be run in 2D. Linear waves are generated, and the propagating waves are compared with the theoretical solution throughout the wave tank. The idea is, that if the wave propagation is represented well by the numerical model, the numerical results will match the theoretical free surface profile in the tank. In this example, the wave has a relatively low steepness, and good convergence can be reached for  $dx = 0.05m$  already, which improves further on finer grids. For steeper waves, finer grids may be required.

### 9.6.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave relaxation zone
C 12 3 // side: symmetry plane
C 13 3 // side: symmetry plane
C 14 7 // right side: numerical relaxation beach
C 15 21 // bottom: wall boundary
C 16 21 // top: symmetry plane
B 1 0.05 // mesh size dx
B 10 0.0 30.0 0.0 0.05 0.0 1.0 // rectangular domain size
M 10 4 // number of parallel processes
```

### 9.6.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities
B 50 0.0001 // wall roughness  $k_s$ 
B 90 1 // turn on the numerical wave tank
B 92 2 // use linear waves
B 91 0.02 4.0 // wave height, wave length
B 96 4.0 8.0 // wavegen relaxation length, numerical beach length
B 98 2 // use relaxation method 2 for wave generation
B 99 2 // use relaxation method 2 for numerical beach
D 10 4 // Conservative WENO discretization for velocity convection
D 20 2 // Implicit diffusion for velocities
D 30 1 // Projection Method for the Pressure
F 30 3 // 3rd-order Runge-Kutta scheme for level set time treatment
F 40 3 // 3rd-order Runge-Kutta scheme for reinitialization time treatment
F 42 1.0 // length for level set initialization, instead of maximum length of domain
F 60 0.5 // still water level
I 12 1 // hydrostatic pressure initialization
N 10 14 // HYPRE Struct BiCGStab pressure solver
N 11 11 // HYPRE Struct PFMG preconditioner
N 40 3 // 3rd-order Runge-Kutta Scheme for velocity time treatment
N 41 90.0 // Maximum simulation time
N 47 0.25 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu printout
P 30 0.05 // print out .vtu files interval based on simulation time
W 22 -9.81 // gravity
P 52 0.025 // y-coordinate and print out water surface line (wfsline) in x-direction
P 53 1 // add theoretical wsflne to file
P 54 500 // print out interval based on iterations for wsflines
```

### 9.6.3 Results

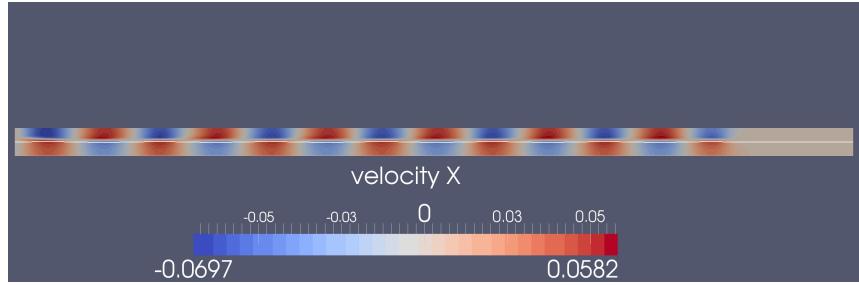


Figure 9.6: Wave tank with free surface and U

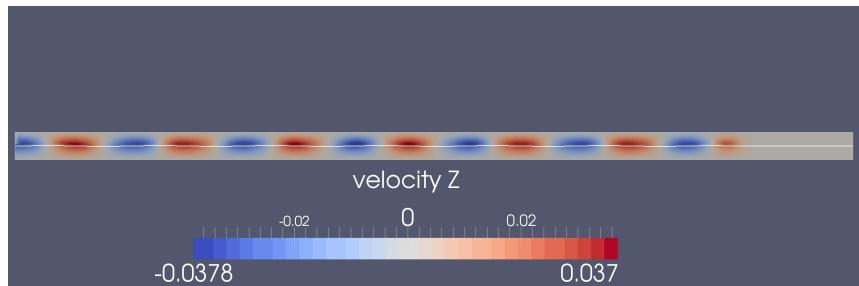


Figure 9.7: Wave tank with free surface and W

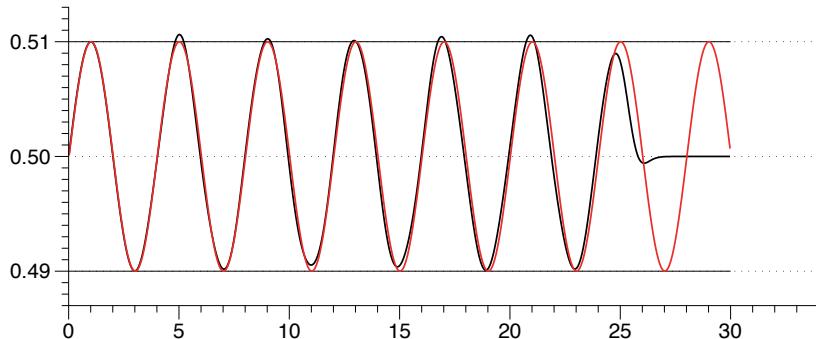


Figure 9.8: Free surface along the tank, numerical result (black line) vs theory (red line)

## 9.7 Rectangular Wave Tank with DWG and AWA

Similar to the previous chapter, a two-dimensional rectangular wave tank is used. Now, different wave generation and absorption methods are used, namely Dirichlet wave generation (DWG) and active wave absorption (AWA). With DWG, the values for the horizontal and vertical velocities are prescribed at the inlet boundary without further modulation. The level set function is allowed to move freely, but due to the correct flow momentum it will follow the prescribed wave theory. AWA generates a wave opposite to the reflected one at the outlet boundary, thus cancelling out the unwanted reflections. AWA is based on shallow water theory, which also has consequences for type of applications it should be used for, i.e. shallow to intermediate water with preferably longer waves. Overall, the AWA-DWG combination requires smaller tank or basin compared to the relaxation methods (RM). AWA, DWG and RM can be used for generation and absorption in any combination.

### 9.7.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave generation  
C 12 3 // side: symmetry plane  
C 13 3 // side: symmetry plane  
C 14 8 // right side: numerical beach AWA  
C 15 21 // bottom: wall boundary  
C 16 21 // top: symmetry plane  
B 1 0.025 // mesh size dx  
B 10 0.0 15.0 0.0 0.025 0.0 1.0 // rectangular domain size  
M 10 4 // number of parallel processes  
M 20 2 // advanced domain decomposition
```

### 9.7.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities  
B 50 0.0001 // wall roughness  $k_s$   
B 90 1 // turn on the numerical wave tank  
B 92 5 // 5th-order stokes waves  
B 91 0.1 2.0 // wave height, wave length  
B 98 3 // Dirichlet Wave Generation (DWG)  
B 99 3 // Active Wave Absorption (AWA)  
D 10 4 // Conservative WENO discretization for velocity convection  
D 20 2 // Implicit diffusion for velocities  
D 30 1 // Projection Method for the Pressure  
F 30 3 // 3rd-order Runge-Kutta scheme for level set time treatment  
F 40 3 // 3rd-order Runge-Kutta scheme for reinitialization time treatment  
F 42 1.0 // length for level set initialization, instead of maximum length of domain  
F 60 0.5 // still water level  
I 12 1 // hydrostatic pressure initialization  
N 40 3 // 3rd-order Runge-Kutta Scheme for velocity time treatment  
N 41 60.0 // Maximum simulation time
```

```

N 47 0.25 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu printout
P 30 0.1 // print out .vtu files interval based on simulation time
P 40 1 // turn state file print out for hot-start
P 42 1.0 // state file print out interval
T 10 0 // no turbulence model
W 22 -9.81 // gravity
P 52 0.0125 // y-coordinate and print out water surface line (wsline) in x-direction
P 53 1 // add theoretical wsline to file
P 55 0.50 // print out wsline files interval based on simulation time
P 51 2.50 0.0125 // wave gage with x- and y-coordinates
P 51 5.00 0.0125 // wave gage with x- and y-coordinates
P 51 15.0 0.0125 // wave gage with x- and y-coordinates
P 51 20.0 0.0125 // wave gage with x- and y-coordinates

```

### 9.7.3 Results

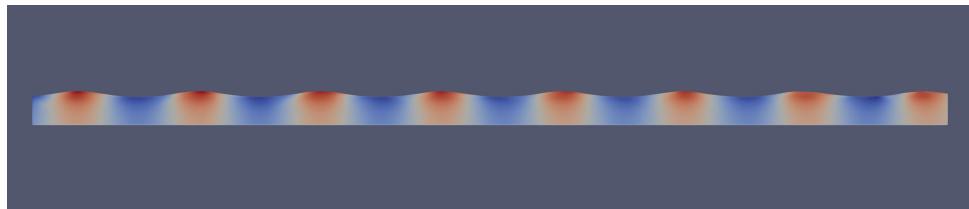


Figure 9.9: DWG-AWA Wave tank with free surface and U

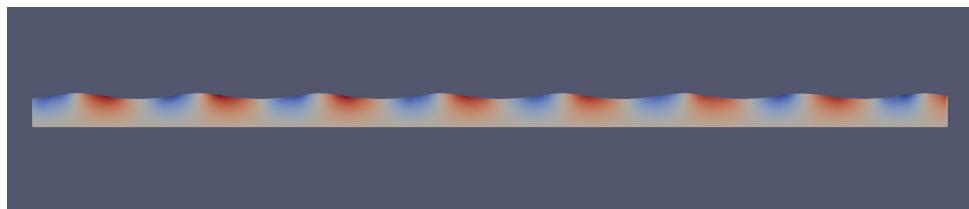


Figure 9.10: DWG-AWA Wave tank with free surface and W

## 9.8 Rectangular Wave Tank with Wavemaker

Following the NWT examples in the previous examples, now the waves are generated with wavemaker kinematics input, see e.g. Aggarwal et al. [2018a] for more information.

### 9.8.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave generation  
C 12 3 // side: symmetry plane  
C 13 3 // side: symmetry plane  
C 14 8 // right side: numerical beach AWA  
C 15 21 // bottom: wall boundary  
C 16 21 // top: symmetry plane  
B 1 0.025 // mesh size dx  
B 10 0.0 15.0 0.0 0.025 0.0 1.5 // rectangular domain size  
M 10 4 // number of parallel processes  
M 20 2 // advanced domain decomposition
```

### 9.8.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities  
B 50 0.0001 // wall roughness  $k_s$   
B 90 1 // turn on the numerical wave tank  
B 92 22 // flap wavemaker  
B 98 3 // Dirichlet Wave Generation (DWG)  
B 99 3 // Active Wave Absorption (AWA)  
B 111 -0.25 1.46 //  $z_{start}$  and  $z_{end}$  for flap wavemaker  
B 116 2 // flap wavemaker input as angle  
B 117 4.0 // time shift for wavemaker input  
D 10 4 // Conservative WENO discretization for velocity convection  
D 20 2 // Implicit diffusion for velocities  
D 30 1 // Projection Method for the Pressure  
F 30 3 // 3rd-order Runge-Kutta scheme for level set time treatment  
F 40 3 // 3rd-order Runge-Kutta scheme for reinitialization time treatment  
F 42 1.5 // length for level set initialization, instead of maximum length of domain  
F 60 0.75 // still water level  
I 12 1 // hydrostatic pressure initialization
```

```

N 40 3 // 3rd-order Runge-Kutta Scheme for velocity time treatment
N 41 60.0 // Maximum simulation time
N 47 0.25 // factor for CFL criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu printout
P 30 0.1 // print out .vtu files interval based on simulation time
P 40 1 // turn state file print out for hot-start
P 42 1.0 // state file print out interval
T 10 0 // no turbulence model
W 22 -9.81 // gravity
P 52 0.0125 // y-coordinate and print out water surface line (wsline) in x-direction
P 55 0.50 // print out wsline files interval based on simulation time
P 51 5.00 0.0125 // wave gage with x- and y-coordinates
P 51 10.0 0.0125 // wave gage with x- and y-coordinates

```

### 9.8.3 REEF3D: wavemaker.dat

The 'wavemaker.dat' file contains the time dependent kinematics of the wavemaker.

### 9.8.4 Results



Figure 9.11: Wavemaker generated waves with free surface and U



Figure 9.12: Wavemaker generated waves with free surface and W

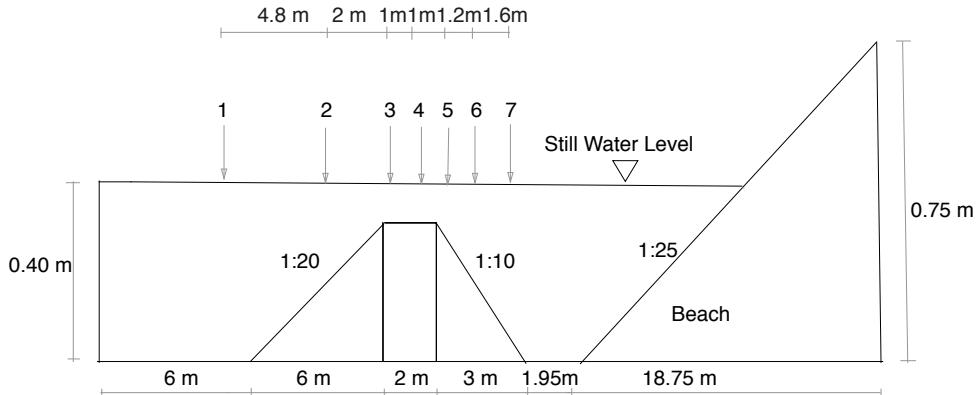


Figure 9.13: Experimental setup submerged bar

## 9.9 Wave Propagation over a Submerged Bar

The well-known benchmark case for wave propagation over a submerged bar is presented here. The description of the experimental setup can be found in the original paper by Beji and Battjes [1993]. The simulations are run in 2D with wave gages at several locations along the wave tank.

### 9.9.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave relaxation zone
C 12 3 // side: symmetry plane
C 13 3 // side: symmetry plane
C 14 7 // numerical relaxation beach
C 15 21 // bottom: wall boundary
C 16 3 // top symmetry plane
B 1 0.01 // mesh size dx
B 10 0.0 24.0 0.0 0.01 0.0 0.8 // rectangular domain size
O 61 6.0 12.0 0.0 0.01 0.0 0.3 // front wedge of the bar
O 10 12.0 14.0 0.0 0.01 0.0 0.3 // middle section of the bar
O 61 14.0 17.0 0.0 0.01 0.3 0.0 // back wedge of the bar
M 10 4 // number of parallel processes
```

### 9.9.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities
B 50 0.0001 // wall roughness  $k_s$ 
B 90 1 // turn on the numerical wave tank
B 92 2 // use linear waves
B 91 0.02 3.73 // wave height, wave length
B 96 3.73 3.73 // wavegen relaxation length, numerical beach length
B 98 2 // use relaxation method 2 for wave generation
B 99 2 // use relaxation method 2 for numerical beach
D 10 4 // Conservative WENO for velocity convection
D 20 2 // Implicit diffusion for velocities
D 30 1 // Projection Method for the pressure
F 30 3 // 3rd-order Runge-Kutta scheme for Level Set time treatment
F 40 3 // 3rd-order Runge-Kutta scheme for Reinitialization time treatment
F 42 0.8 // length for level set initialization, instead of maximum length of domain
F 60 0.4 // still water level
I 12 1 // hydrostatic pressure initialization
N 10 14 // HYPRE Struct BiCGStab pressure solver
N 11 11 // HYPRE Struct PFMG preconditioner
N 40 3 // 3rd-order Runge-Kutta scheme for velocity time treatment
N 41 60.0 // Maximum simulation time
N 47 0.25 // factor for CFD criterion
M 10 4 // number of parallel processes
P 10 1 // turn on .vtu printout
P 30 0.5 // print out interval for .vtu files based on simulation time
W 22 -9.81 // gravity
P 52 0.005 // y-coordinate and print out water surface line wsflne in x-direction
P 53 1 // add theoretical wsflne to file
P 54 10 // print out interval for wsflines based on iterations
P 51 2.0 0.005 // Wave gage 1 location, will printed in the order given here
P 51 4.0 0.005 // Wave gage 2
P 51 5.2 0.005 // Wave gage 3
P 51 10.5 0.005 // Wave gage 4
P 51 12.5 0.005 // Wave gage 5
P 51 13.5 0.005 // Wave gage 6
P 51 14.5 0.005 // Wave gage 7
```

### 9.9.3 Results

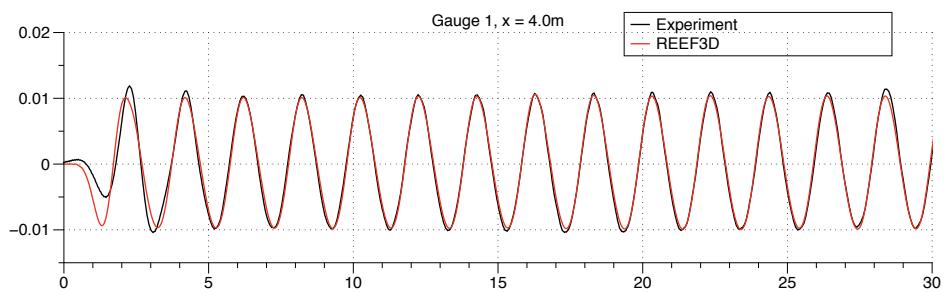


Figure 9.14: Wave gage 1

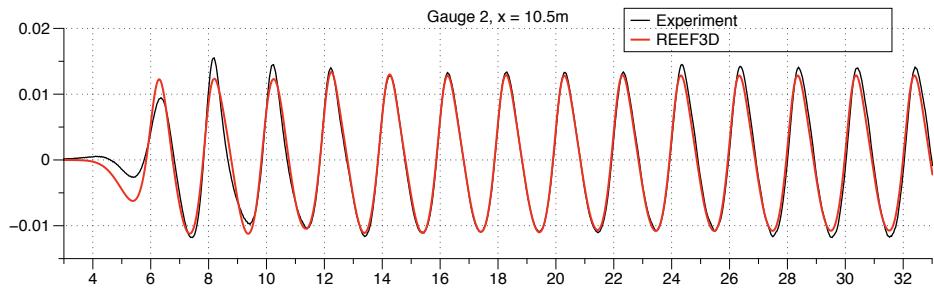


Figure 9.15: Wave gage 2

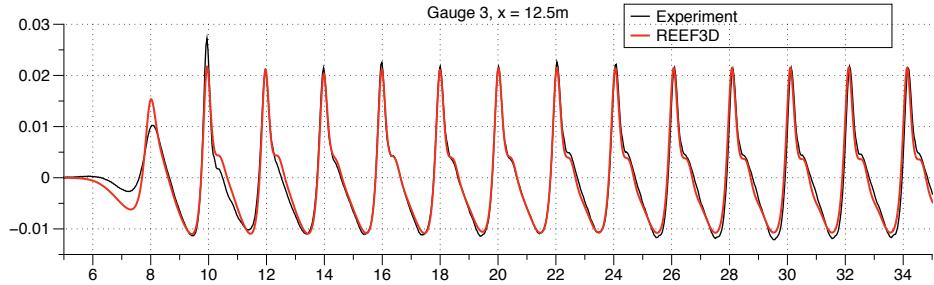
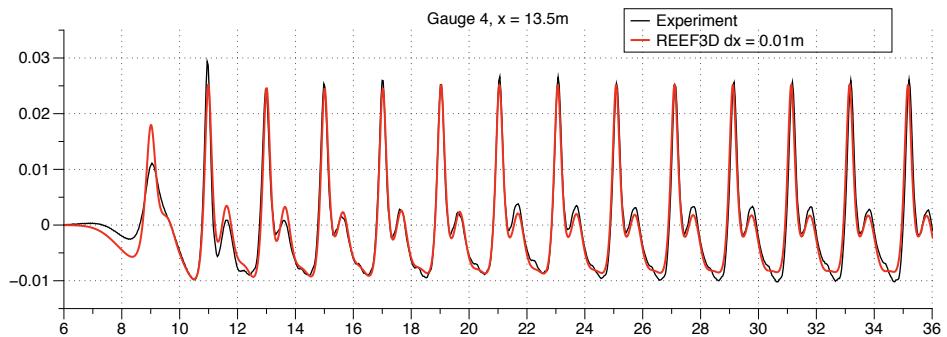


Figure 9.16: Wave gage 3



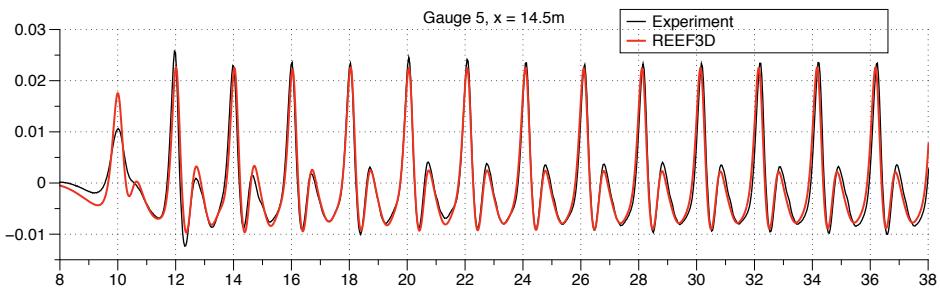


Figure 9.18: Wave gage 5

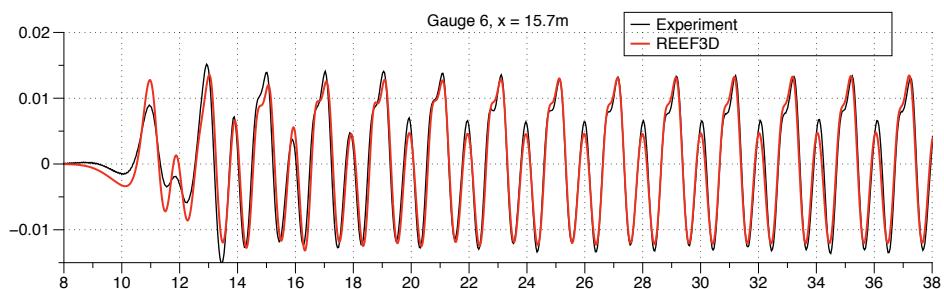
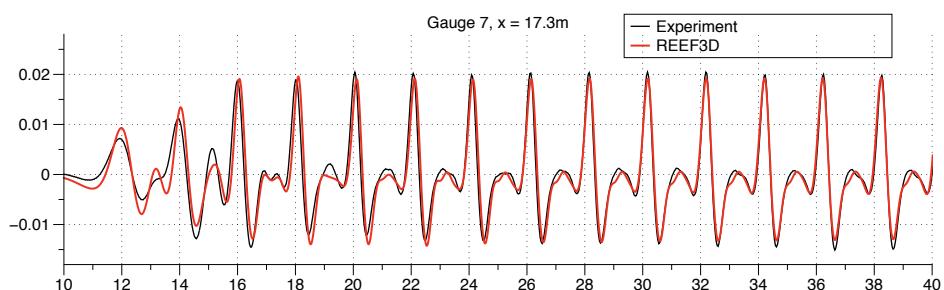


Figure 9.19: Wave gage 6



## 9.10 Plunging Breaking Waves over Slope

The well known benchmark case for plunging breaking waves over a submerged bar by Ting and Kirby [1995] is shown here. The breaking waves require a relatively finer grid, thus 128 cores on NOTUR's supercomputer facilities were used to simulate the case. For more information on this case and breaking waves over slopes in general, please have a look at these in-depth studies: Alagan Chella et al. [2015b], Alagan Chella et al. [2015c] and Alagan Chella et al. [2015a].

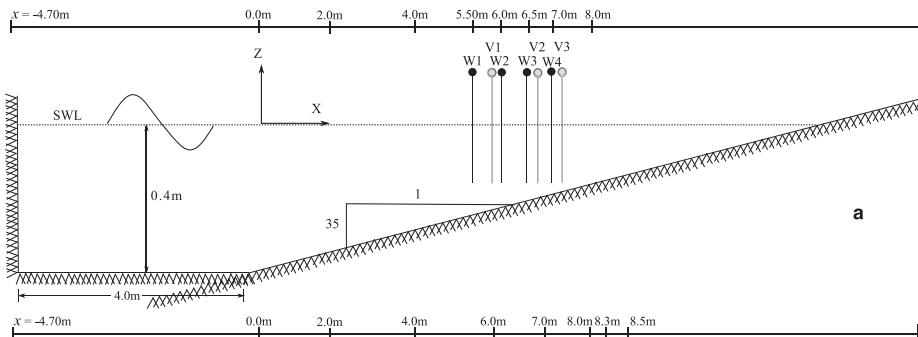


Figure 9.21: Experimental setup plunging breaking waves over slope

### 9.10.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave relaxation zone
C 12 3 // side: symmetry plane
C 13 3 // side: symmetry plane
C 14 7 // numerical relaxation beach
C 15 21 // bottom: wall boundary
C 16 3 // top symmetry plane
B 1 0.01 // mesh size dx
B 10 0.0 24.0 0.0 0.01 0.0 0.8 // rectangular domain size
O 61 6.0 12.0 0.0 0.01 0.0 0.3 // front wedge of the bar
O 10 12.0 14.0 0.0 0.01 0.0 0.3 // middle section of the bar
O 61 14.0 17.0 0.0 0.01 0.3 0.0 // back wedge of the bar
M 10 128 // number of parallel processes
M 20 2 // advanced domain decomposition
```

### 9.10.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities
B 10 1 // use wall functions for the turbulence model
B 50 0.0001 // wall roughness  $k_s$ 
B 90 1 // turn on the numerical wave tank
B 92 8 // use 5th-order cnoidal waves
B 93 0.128 5.0 // wave height, wave period
B 96 9.8 0.0 0.0 // wavegen relaxation length, numerical beach length
B 98 2 // use relaxation method 2 for wave generation
B 99 2 // no beach
D 10 4 // Conservative WENO for velocity convection
D 20 2 // Implicit diffusion for velocities
D 30 1 // Projection Method for the pressure
F 30 3 // 3rd-order Runge-Kutta scheme for Level Set time treatment
F 40 3 // 3rd-order Runge-Kutta scheme for Reinitialization time treatment
F 42 1.0 // length for level set initialization, instead of maximum length of domain
F 60 0.4 // still water level
I 12 1 // hydrostatic pressure initialization
N 40 3 // 3rd-order Runge-Kutta Scheme for velocity time treatment
N 41 60.0 // Maximum simulation time
N 47 0.25 // factor for CFD criterion
M 10 128 // number of parallel processes
P 10 1 // turn on .vtu printout
P 30 0.5 // print out .vtu files based on simulation time
P 40 1 // turn state file print out for hot-start
P 42 0.5 // state file print out interval
T 10 2 // k- $\omega$  turbulence model
T 31 0.816 // factor for eddy viscosity limiter in phase 1
T 32 0.212 // factor for eddy viscosity limiter in phase 2
T 36 2 // FSF boundary condition for turbulent dissipation
```

W 22 -9.81 // gravity

P 51 19.8 0.0025 // wave gage

P 51 20.8 0.0025 // wave gage

P 51 21.8 0.0025 // wave gage

P 51 22.1 0.0025 // wave gage

P 52 0.0025 // wsline in x-direction

P 61 21.095 0.0025 0.35 // point probe

P 61 21.095 0.0025 0.25 // point probe

P 61 21.095 0.0025 0.4

P 61 22.025 0.0025 0.35 // point probe

P 61 22.025 0.0025 0.26 // point probe

P 61 22.025 0.0025 0.4 // point probe

P 61 21.595 0.0025 0.4 // point probe

P 61 22.145 0.0025 0.35 // point probe

P 61 22.145 0.0025 0.28 // point probe

P 61 22.145 0.0025 0.4 // point probe

### 9.10.3 Results

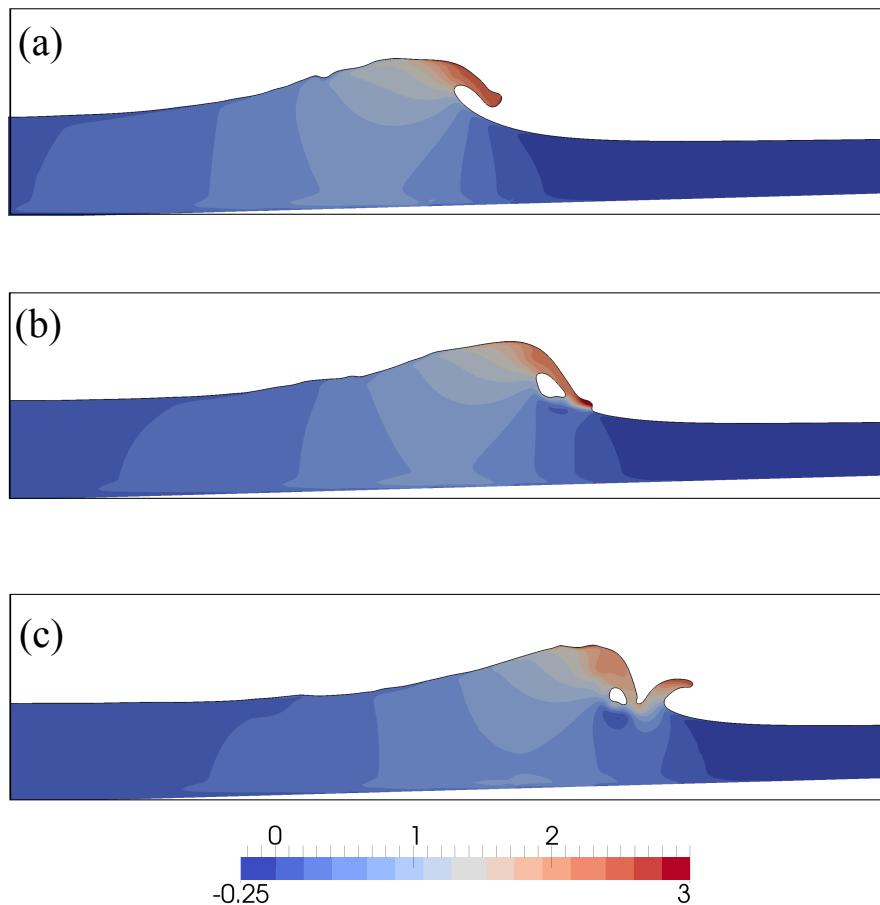


Figure 9.22: Variation of horizontal velocity,  $U$  (m/s) under the plunging breaker at  $t = 10.85\text{s}$  (a),  $10.95\text{s}$  (b) and  $11.05\text{s}$  (c)

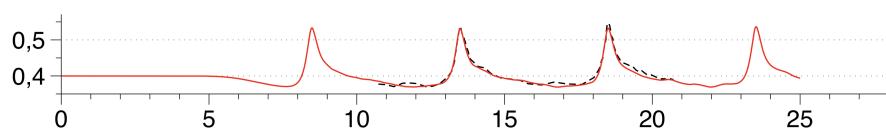


Figure 9.23: Wave gage 1, black experiment Ting and Kirby [1995], red REEF3D

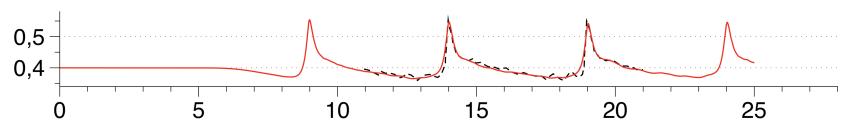


Figure 9.24: Wave gage 2, black experiment Ting and Kirby [1995], red REEF3D

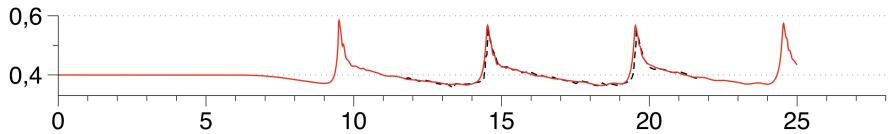


Figure 9.25: Wave gage 3, black experiment Ting and Kirby [1995], red REEF3D

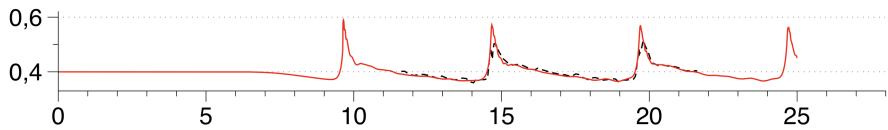


Figure 9.26: Wave gage 4, black experiment Ting and Kirby [1995], red REEF3D

## 9.11 Plunging Breaking Waves over Slope with Non-uniform Grid

The simulation of plunging breaking waves over a slope from the previous section by Ting and Kirby [1995] is shown here with a non-uniform grid (NUG) setup. The ctrl.txt is identical to the one above, with the exception of the M10 parameter.

### 9.11.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave relaxation zone
C 12 3 // side: symmetry plane
C 13 3 // side: symmetry plane
C 14 7 // numerical relaxation beach
C 15 21 // bottom: wall boundary
C 16 3 // top symmetry plane
B 1 0.005
B 2 3000 1 80 // mesh size: Nx,Ny,Nz
B 10 0.0 30.0 0.0 0.005 0.0 1.0 // rectangular domain size
S 61 13.8 30.0 0.0 0.005 0.0 0.463 // solid slope
M 10 12 // number of parallel processes
M 20 2 // advanced domain decomposition
B 101 5 // point focus using sinh stretching function in x-direction
B 111 2.5 // stretching factor in x-direction
B 114 20.0 // focus point for stretching in x-direction
B 103 5 // point focus using sinh stretching function in x-direction
B 113 2.5 // stretching factor in z-direction
B 116 0.40 // focus point for stretching in z-direction
```

## 9.12 Shoaling over Irregular Topography

In this case, waves propagation irregular topography is calculated using phase accurate irregular waves based on Boers' experiment Boers [1996]. Here, the bed topography is generated using a 'geo.dat' file with xyz-point data. These coordinates are interpolated using a local inverse distance algorithm in DIVEMesh. The waves are generated using wave reconstruction, are very convenient way to create phase accurate irregular waves based on a single wave gage Aggarwal et al. [2018b].

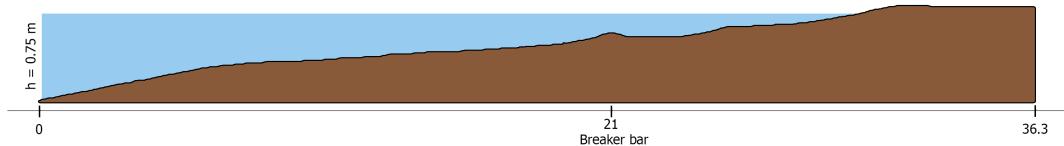


Figure 9.27: Experimental setup plunging breaking waves over slope Elakel [2018]

### 9.12.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave relaxation zone  
C 12 3 // side: symmetry plane  
C 13 3 // side: symmetry plane  
C 14 7 // numerical relaxation beach  
C 15 21 // bottom: wall boundary  
C 16 3 // top symmetry plane  
B 1 0.01 // mesh size dx  
B 10 0.0 36.3 0.0 0.01 0.0 1.00 // rectangular domain size  
G 10 1 // turn on geodat interpolation  
G 15 2 // use local inverse distance interpolation  
G 31 14 // number of smoothing iterations  
M 10 128 // number of parallel processes  
M 20 2 // advanced domain decomposition
```

### 9.12.2 DIVEMesh: geo.dat

In the 'geo.dat' file all bathymetry coordinates are given in ASCII format and are then used to interpolate into the bed by DIVEMesh. The bed can be visualized with the zero contour of 'topo' in the vtu files read by ParaView.

### 9.12.3 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities
B 10 1 // use wall functions for the turbulence model
B 50 0.0001 // wall roughness  $k_s$ 
B 90 1 // turn on the numerical wave tank
B 92 51 // wave reconstruction based on first-order irregular waves
B 98 3 // use Dirichlet wave generation
B 99 0 // no beach
D 10 4 // Conservative WENO for velocity convection
D 20 2 // Implicit diffusion for velocities
D 30 1 // Projection Method for the pressure
F 30 3 // 3rd-order Runge-Kutta scheme for Level Set time treatment
F 40 3 // 3rd-order Runge-Kutta scheme for Reinitialization time treatment
F 42 1.0 // length for level set initialization, instead of maximum length of domain
F 60 0.75 // still water level
I 12 1 // hydrostatic pressure initialization
N 40 3 // 3rd-order Runge-Kutta Scheme for velocity time treatment
N 41 200.0 // Maximum simulation time
N 47 0.25 // factor for CFD criterion
M 10 128 // number of parallel processes
P 10 1 // turn on .vtu printout
P 30 0.5 // print out .vtu files based on simulation time
P 40 1 // turn state file print out for hot-start
P 42 0.5 // state file print out interval
T 10 2 // k- $\omega$  turbulence model
T 31 0.816 // factor for eddy viscosity limiter in phase 1
T 32 0.212 // factor for eddy viscosity limiter in phase 2
T 36 2 // FSF boundary condition for turbulent dissipation
G 50 1 // look for geodata points in grid file
W 22 -9.81 // gravity
P 51 0.0 0.005 // wave gage
P 51 4.0 0.005 // wave gage
P 51 8.0 0.005 // wave gage
P 51 12.0 0.005 // wave gage
P 51 16.0 0.005 // wave gage
P 51 20.0 0.005 // wave gage
P 61 14.37 0.005 0.74 // point probe
```

#### 9.12.4 REEF3D: waverecon.dat

The 'waverecon.dat' files contains the individual wave components including amplitude, frequency and phase shift in order to generate phase accurate irregular waves.

#### 9.12.5 Results

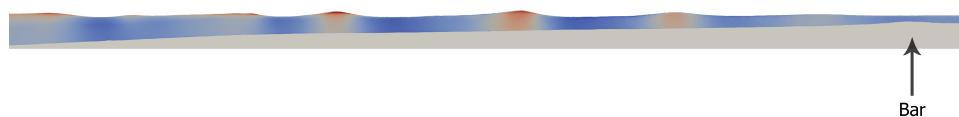


Figure 9.28: Shoaling of irregular waves for the Boers case Elakel [2018]

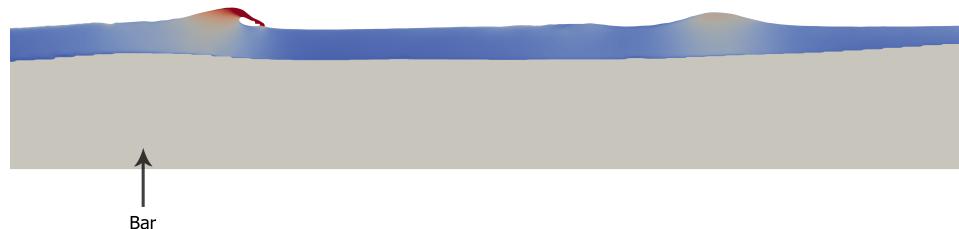


Figure 9.29: Breaking waves on the bar for the Boers case Elakel [2018]

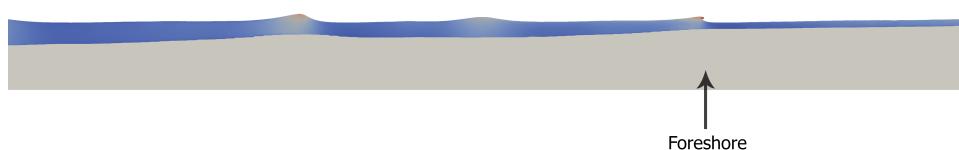


Figure 9.30: Breaking waves on the foreshore for the Boers case Elakel [2018]

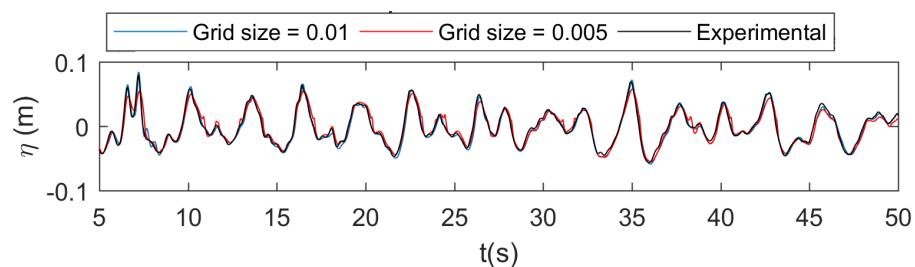


Figure 9.31: Wave gage 1

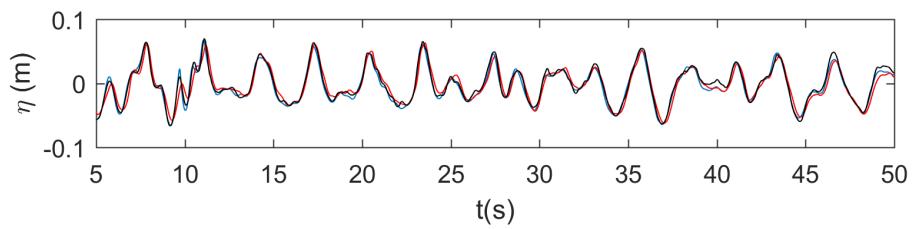


Figure 9.32: Wave gage 2

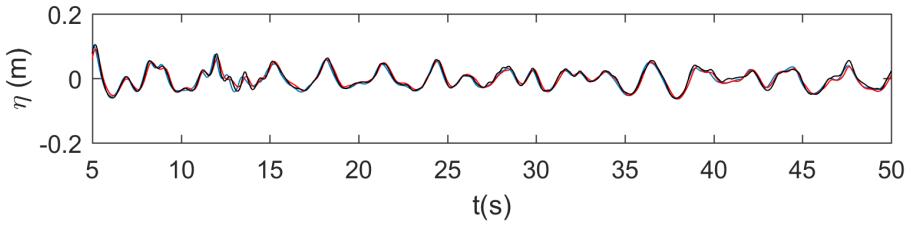


Figure 9.33: Wave gage 3

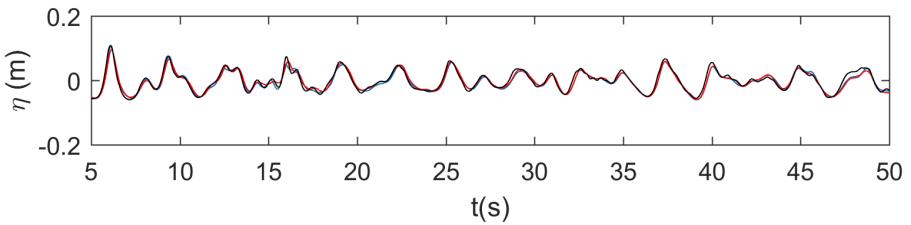


Figure 9.34: Wave gage 4

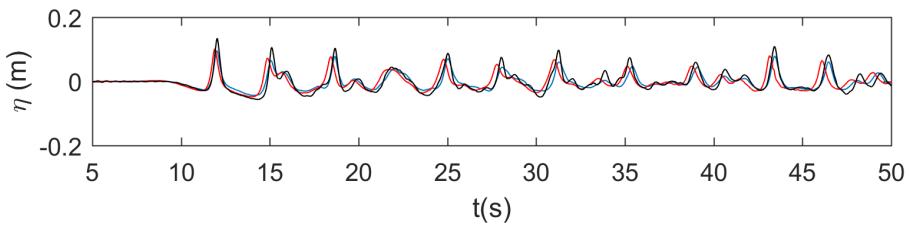


Figure 9.35: Wave gage 5

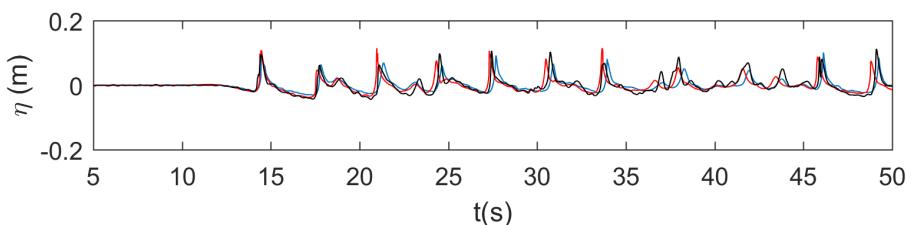


Figure 9.36: Wave gage 6

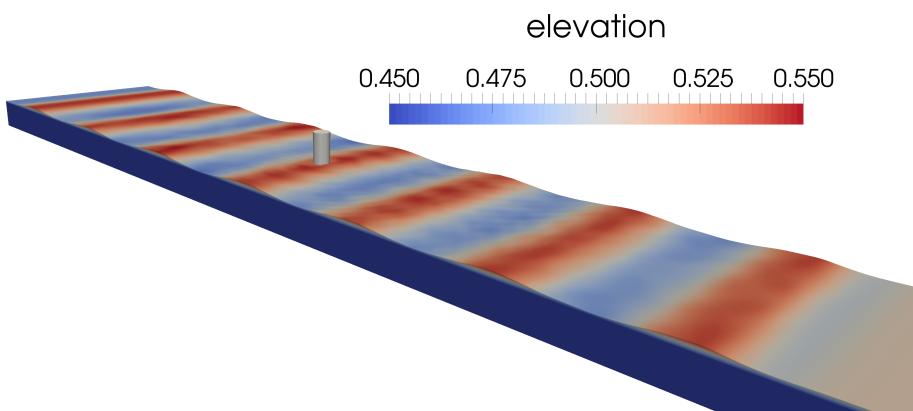


Figure 9.37: Non-breaking wave forces

## 9.13 Non-Breaking Wave Forces

In many scenarios, wave forces need to be computed. Based on the experiments by Chen et al. [2014], numerical benchmark results were produced, see Bihs et al. [2016a] for more detail. In addition to previous NWT examples, the use of the force box is shown in this example. The forces on a solid within the force box is calculated and printed out. It is important to give some margin around the solid in order capture all surface facets in the pressure integration algorithm. More REEF3D related literature on non-breaking wave forces can be found here: Kamath et al. [2015a], Kamath et al. [2016b] and Kamath et al. [2015b].

### 9.13.1 DIVEMesh: control.txt

```
C 11 6 // left side: wave relaxation zone
C 12 21 // side: wall boundary
C 13 21 // side: wall boundary
C 14 7 // numerical relaxation beach
C 15 21 // bottom: wall boundary
C 16 3 // top symmetry plane
B 1 0.025 // mesh size dx
B 10 0.0 18.0 0.0 3.0 0.0 1.0 // rectangular domain size
S 33 7.50 1.50 0.125 // solid vertical cylinder
M 10 12 // number of parallel processes
M 20 2 // advanced domain decomposition
```

### 9.13.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities
B 10 1 // use wall functions for the turbulence model
B 50 0.0001 // wall roughness  $k_s$ 
B 90 1 // turn on the numerical wave tank
B 92 4 // use 2nd-order Stokes waves
B 93 0.07 1.22 // wave height, wave period
B 96 2.11 4.22 // wavegen relaxation length, numerical beach length
B 98 2 // use relaxation method 2 for wave generation
B 99 2 // relaxation beach
D 10 4 // Conservative WENO for velocity convection
D 20 2 // Implicit diffusion for velocities
D 30 1 // Projection Method for the pressure
F 30 3 // 3rd-order Runge-Kutta scheme for Level Set time treatment
F 40 3 // 3rd-order Runge-Kutta scheme for Reinitialization time treatment
F 42 1.0 // length for level set initialization, instead of maximum length of domain
F 60 0.505 // still water level
I 12 1 // hydrostatic pressure initialization
N 40 3 // 3rd-order Runge-Kutta Scheme for velocity time treatment
N 41 50.0 // Maximum simulation time
N 47 0.25 // factor for CFD criterion
M 10 12 // number of parallel processes
P 10 1 // turn on .vtu printout
P 35 15.0 30.0 0.05 // print out interval for .vtu files based on simulation time
P 40 1 // turn state file print out for hot-start
P 42 1.0 // state file print out interval
T 10 2 // k- $\omega$  turbulence model
T 36 2 // FSF boundary condition for turbulent dissipation
W 22 -9.81 // gravity
P 51 5.00 1.50 // wave gage
P 55 1.0 // print out wave gage based on simulation time
P 81 7.325 7.675 1.325 1.675 0.0 1.0 // force box coordinates
```

### 9.13.3 Results

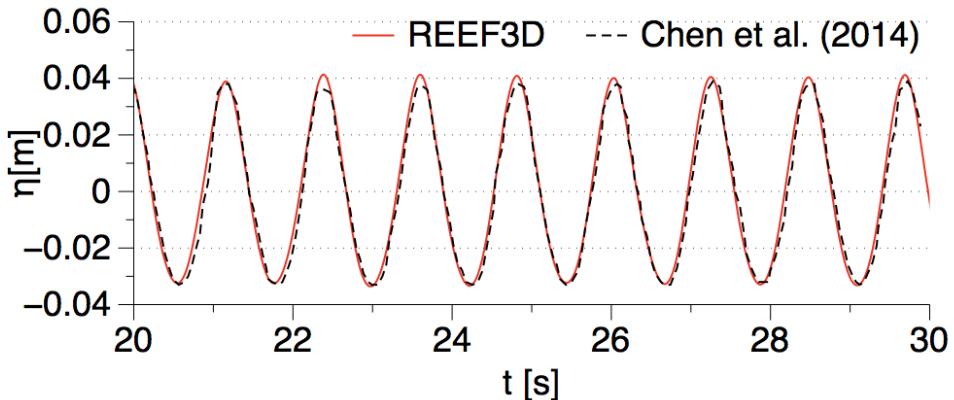


Figure 9.38: Free surface comparison between REEF3D and the experimental data Chen et al. [2014]

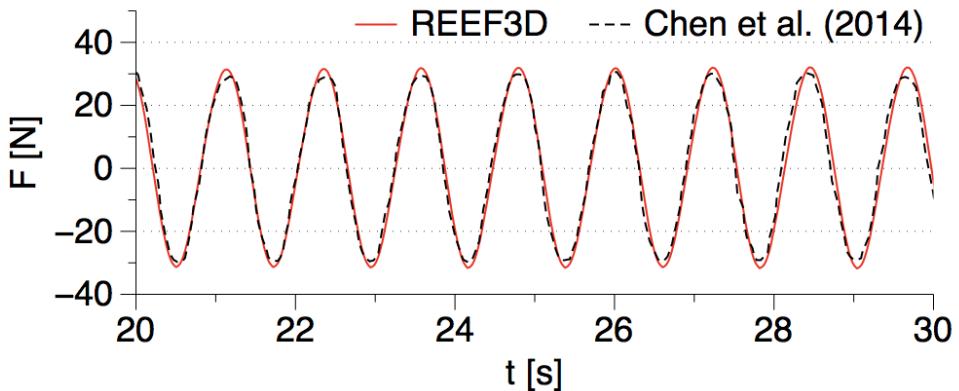


Figure 9.39: Wave Force comparison between REEF3D and the experimental data Chen et al. [2014]

## 9.14 Breaking Wave Forces

Extending the previous scenario, breaking wave forces are simulated by using the case from Kamath et al. [2016a] based on an experimental study in the GWK in Hannover Irschik et al. [2002]. The waves are shoaling on a 1:10 slope and are breaking directly at the vertical cylinder. The slamming forces are calculated using the force box approach. REEF3D has been extensively validated and applied for breaking wave forces: Bihs et al. [2016b], Alagan Chella et al. [2019c], Alagan Chella et al. [2019b], Alagan Chella et al. [2019a]

### 9.14.1 DIVEMesh: control.txt

C 11 6 // left side: wave relaxation zone

C 12 21 // side: wall boundary

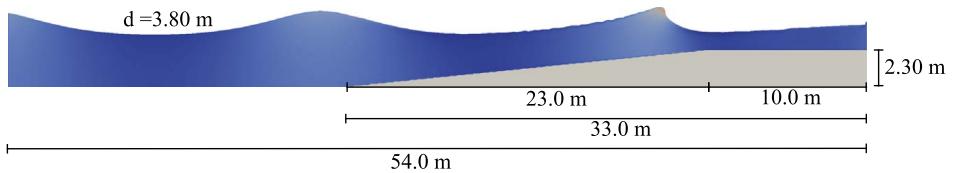


Figure 9.40: Breaking wave forces setup

```
C 13 21 // side: wall boundary
C 14 8 // numerical AWA beach
C 15 21 // bottom: wall boundary
C 16 3 // top symmetry plane
B 1 0.05 // mesh size dx
B 10 0.0 54.0 0.0 5.0 0.0 7.0 // rectangular domain size
S 61 21.0 44.0 0.0 5.0 0.0 2.30 // solid slope
S 10 44.0 54.0 0.0 5.0 0.0 2.30 // solid flat
O 33 44.0 2.50 0.35 // vertical cylinder
M 10 256 // number of parallel processes
M 20 2 // advanced domain decomposition
```

### 9.14.2 REEF3D: ctrl.txt

```
B 10 1 // use wall functions for the velocities
B 10 1 // use wall functions for the turbulence model
B 50 0.0001 // wall roughness  $k_s$ 
B 90 1 // turn on the numerical wave tank
B 92 5 // use 5th-order Stokes waves
B 93 1.3 4.0 // wave height, wave period
B 96 21.0 0.0 // wavegen relaxation length, numerical beach length
B 98 2 // use relaxation method 2 for wave generation
B 99 3 // AWA beach
D 10 4 // Conservative WENO for velocity convection
D 20 2 // Implicit diffusion for velocities
D 30 1 // Projection Method for the pressure
```

```

F 30 3 // 3rd-order Runge-Kutta scheme for Level Set time treatment
F 40 3 // 3rd-order Runge-Kutta scheme for Reinitialization time treatment
F 42 7.0 // length for level set initialization, instead of maximum length of domain
F 60 3.8 // still water level
I 12 1 // hydrostatic pressure initialization
N 40 3 // 3rd-order Runge-Kutta Scheme for velocity time treatment
N 41 50.0 // Maximum simulation time
N 47 0.25 // factor for CFD criterion
M 10 256 // number of parallel processes
P 10 1 // turn on .vtu printout
P 35 23.5 25.2 0.05 // print out interval for .vtu files based on simulation time
P 40 1 // turn state file print out for hot-start
P 42 1.0 // state file print out interval
T 10 2 // k- $\omega$  turbulence model
T 36 2 // FSF boundary condition for turbulent dissipation
T 31 0.816 // factor for eddy viscosity limiter in phase 1
T 32 0.212 // factor for eddy viscosity limiter in phase 2
W 22 -9.81 // gravity
P 81 43.30 44.70 1.80 3.20 2.30 8.0 // force box coordinates
P 10 1 // print .vtu file
P 35 23.5 25.2 0.05 // print .vtu file in interval
P 180 1 // print fsf files
P 182 0.1 // print out fsf files based on simulation time
P 51 22.0 2.50 // wave gage
P 51 30.0 2.50 // wave gage
P 51 35.0 2.50 // wave gage
P 51 40.0 2.50 // wave gage
P 51 43.65 4.80 // wave gage
P 51 43.65 2.50 // wave gage
P 51 44.5 2.505 // wave gage
P 51 47.0 2.5 // wave gage
P 51 50.5 2.5 // wave gage
P 51 52.0 2.5 // wave gage
P 51 53.0 2.5 // wave gage

```

### 9.14.3 Results

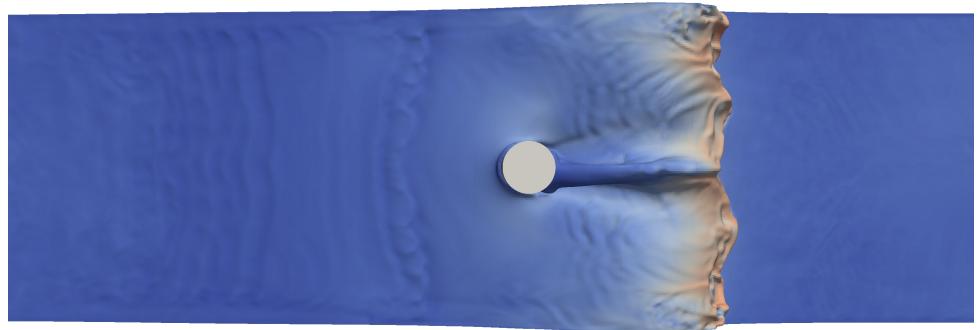


Figure 9.41: Breaking wave impact with REEF3D, plane view

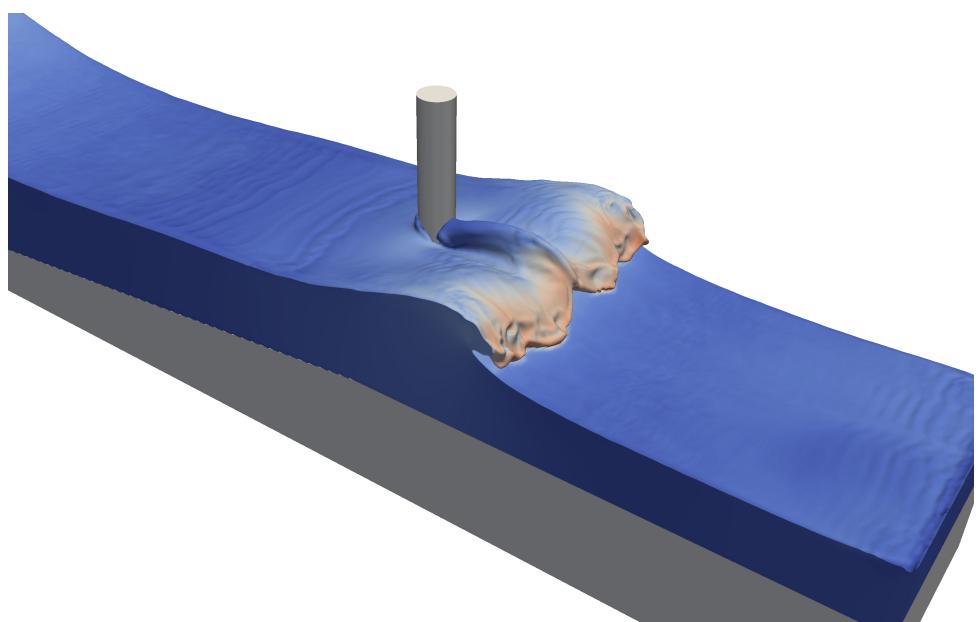


Figure 9.42: Breaking wave impact with REEF3D, side view

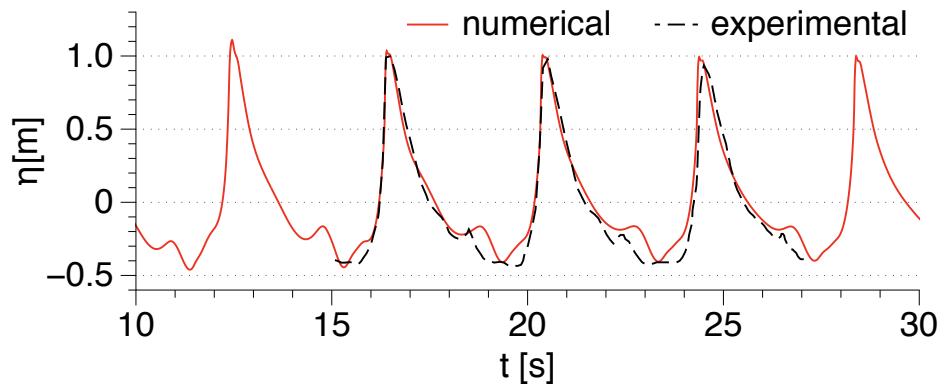


Figure 9.43: Wave Force comparison between REEF3D and the experimental data Irschik et al. [2002]

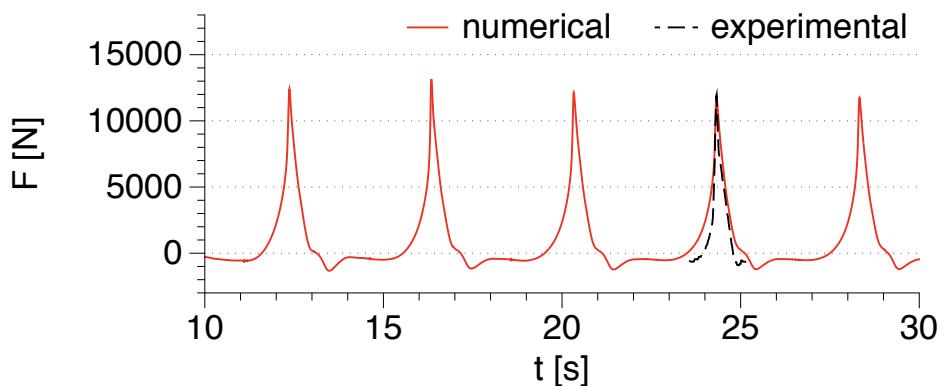


Figure 9.44: Free surface comparison between REEF3D and the experimental data Irschik et al. [2002]

# Bibliography

- M. A. Afshar. Numerical wave generation in OpenFOAM. Master's thesis, Chalmers University of Technology, 2010.
- A. Aggarwal, M. Alagan Chella, H. Bihs, C. Pakozdi, P. A. Berthelsen, and Ø. A Arntsen. CFD-based study of steep irregular waves for extreme wave spectra. *International Journal of Offshore and Polar Engineering*, 28(2):164–170, 2018a.
- A. Aggarwal, C. Pakozdi, H. Bihs, D. Myrhaug, and M. Alagan Chella. Free surface reconstruction for phase accurate irregular wave generation. *Journal of Marine Science and Engineering*, 6(105):1–23, 2018b.
- M. Alagan Chella, H. Bihs, and D. Myrhaug. Characteristics and profile asymmetry properties of waves breaking over an impermeable submerged reef. *Coastal Engineering*, 100:26–36, 2015a.
- M. Alagan Chella, H. Bihs, D. Myrhaug, and M. Muskulus. Breaking characteristics and geometric properties of spilling breakers over slopes. *Coastal Engineering*, 95:4–19, 2015b.
- M. Alagan Chella, Hans Bihs, Dag Myrhaug, and Michael Muskulus. Hydrodynamic characteristics and geometric properties of plunging and spilling breakers over impermeable slopes. *Ocean Modelling, Virtual Special Issue: Ocean Surface Waves*, pages 1–20, 2015c.
- M. Alagan Chella, H. Bihs, and Myrhau D. Numerical modeling of breaking wave kinematics and wave impact pressures on a vertical slender cylinder,. *Journal of Fluids and Structures*, 86:94–123, 2019a.
- M. Alagan Chella, H. Bihs, Myrhau D., and Ø. A Arntsen. Numerical modeling of breaking wave kinematics and wave impact pressures on a vertical slender cylinder,. *Journal of Offshore Mechanics and Arctic Engineering*, 141(5):1–10, 2019b.
- M. Alagan Chella, H. Bihs, A. Kamath, Myrhau D., and Ø. A Arntsen. Breaking wave interaction with a group of four vertical slender cylinders in two square arrangements. *Journal of Offshore Mechanics and Arctic Engineering*, 141(6):1–10, 2019c.
- S. F. Ashby and R. D. Flagout. A parallel multigrid preconditioned conjugate gradient algorithm for groundwater flow simulations. *Nuclear Science and Engineering*, 124(1):145–159, 1996.
- S. Beji and J. A. Battjes. Experimental investigation of wave propagation over a bar. *Coastal Engineering*, 19:151–162, 1993.
- H. Bihs, A. Kamath, M. Alagan Chella, A. Aggarwal, and Ø. A. Arntsen. A new level set numerical wave tank with improved density interpolation for complex wave hydrodynamics. *Computers & Fluids*, 140:191–208, 2016a.
- H. Bihs, A. Kamath, M. Alagan Chella, and Ø. A Arntsen. Breaking-wave interaction with tandem cylinders under different impact scenarios. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, 2016b.

- M. Boers. *Simulation of a surf zone with a barred beach. Report 1. Wave heights and wave breaking*. PhD thesis, Report, Department of Civil Engineering, Delft Technical University, 1996.
- hyper high performance preconditioners - User's Manual*. Center for Applied Scientific Computing, Lawrence Livermore National Laboratory, 2015.
- L. F. Chen, J. Zang, A. J. Hillis, G. C. J. Morgan, and A. R. Plummer. Numerical investigation of wave–structure interaction using openFOAM. *Ocean Engineering*, 88:91–109, 2014.
- M. Elakel. *Investigation of wave transformation and breaking processes in the coastal zone using REEF3D*. PhD thesis, Master Thesis, Marine Civil Engineering, NTNU Trondheim, 2018.
- R. D. Falgout, J. E. Jones, and U. M. Yang. *Numerical Solution of Partial Differential Equations of Parallel Computers*, chapter The Design and Implementation of hypre, a Library of Parallel High Performance Preconditioners. Lecture Notes in Computational Science and Engineering. Springer, 2006.
- K. Irschik, U. Sparboom, and H. Oumeraci. Breaking wave characteristics for the loading of a slender pile. In *Proc. 28th International Conference on Coastal Engineering, Cardiff, Wales*, 2002.
- N. G. Jacobsen, D. R. Fuhrman, and J. Fredsøe. A wave generation toolbox for the open-source CFD library: OpenFOAM. *International Journal for Numerical Methods in Fluids*, 70(9):1073–1088, 2012.
- A. Kamath, M. Alagan Chella, H. Bihs, and Ø. A. Arntsen. Evaluating wave forces on groups of three and nine cylinders using a 3D numerical wave tank. *Engineering Applications of Computational Fluid Mechanics*, 2015b.
- A. Kamath, M. Alagan Chella, H. Bihs, and Ø. A Arntsen. Cfd investigations of wave interaction with a pair of large tandem cylinders. *Ocean Engineering*, 108:738–748, 2015a.
- A. Kamath, M. Alagan Chella, H. Bihs, and Ø. A Arntsen. Breaking wave interaction with a vertical cylinder and the effect of breaker location. *Ocean Engineering*, 128:105–115, 2016a.
- A. Kamath, H. Bihs, M. Alagan Chella, and Øivind A Arntsen. Upstream-cylinder and downstream-cylinder influence on the hydrodynamics of a four-cylinder group. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, 2016b. doi: 10.1061/(ASCE)WW.1943-5460.0000339.
- M. Peric. Why STAR-CCM+ is the next step for cfd. *dynamics*, 23:3–4, 2004.
- F. C. K. Ting and J. T. Kirby. Dynamics of surf-zone turbulence in a strong plunging breaker. *Coastal Engineering*, 24:177–204, 1995.